

WANDA 4.5

User Manual



WANDA, Version 4.5

Copyright © 2000-2017 by Deltares (the former WL | Delft Hydraulics), Delft, The Netherlands. All rights reserved. Printed in The Netherlands. This book, or parts thereof, may not be reproduced in any form without permission of the publisher.

The material contained herein is supplied without representation of warranty of any kind. The authors and publisher assume no liability, consequential or otherwise, of any kind, arising from the use of these Programs or any parts thereof.

The software herein may be updated, extended, or otherwise revised by the authors and the publisher at their sole discretion. Purchase of this edition, whether in book or disk form, in no way obliges the authors and the publisher to provide the purchaser with future editions at a cost below the publisher's suggested retail price.

Table of Contents

1. Introduction	11
1.1 What is WANDA 4?.....	11
1.2 WANDA domains	11
1.2.1 WANDA 4 Liquid.....	12
1.2.2 WANDA 4 Heat.....	12
1.2.3 Control	12
1.3 Modelling aspects	13
1.3.1 From real world to mathematical model	13
1.3.2 Hydraulic diagram editor.....	13
1.3.3 Components and galleries.....	13
1.4 Calculation modes.....	15
1.4.1 Time scales.....	15
1.4.2 Engineering mode.....	15
1.4.3 Transient mode.....	15
1.5 Validation.....	16
1.6 WANDA 4 main features.....	17
1.7 To whom is WANDA 4 a useful tool?.....	17
1.8 How to use this manual.....	17
1.9 System requirements	18
1.10 About Deltares	18
1.11 Technical Support	18
2. User Interface	19
2.1 Overview of WANDA 4 features.....	19
2.1.1 MSOffice look and feel.....	19
2.1.2 Diagrams.....	19
2.1.3 Routes in diagram.....	20
2.1.4 Automatic name generation.....	20
2.1.5 Property text boxes.....	20
2.1.6 Property list windows	20
2.1.7 Table editor	21
2.1.8 Spreadsheet.....	21
2.1.9 Calculations	22
2.1.10 Chart engine	22
2.1.11 Time navigator	23
2.1.12 Unit groups.....	23
2.1.13 Selection builder	23
2.1.14 'On demand' data loading / Smart saving.....	23
2.1.15 Instant window/view refresh	23
2.2 Getting Started	24
2.2.1 Hydraulic analysis - Functional steps	24
2.2.2 WANDA 4 - User Interface	26
2.2.3 WANDA 4 in 10 steps	30
2.3 Building the diagram	32
2.3.1 Shape Library / Gallery.....	32
2.3.2 Adding components.....	33
2.3.3 Looking closer at components in the diagram.....	33

2.3.4	Connecting components	34
2.3.5	Rules for creating a hydraulic model	37
2.3.6	Advanced drawing a scheme	38
2.3.7	Vector based schematisation	39
2.3.8	Aligning routes	40
2.3.9	Changing the flow definition	40
2.3.10	Navigating large diagrams	41
2.3.11	Splitting a diagram	42
2.4	Property Window	43
2.4.1	Properties of a Component	43
2.4.2	Visibility of properties	44
2.4.3	How to use Property Windows	46
2.4.4	Example	47
2.5	Selection Builder	62
2.5.1	How to use the Selection Builder	62
2.6	Using Charts	64
2.6.1	Kinds of charts	64
2.6.2	How to use charts	64
2.6.3	Chart with more series	64
2.6.4	Time series	65
2.6.5	Location series for a single pipe	66
2.6.6	Location series for a route	68
2.6.7	Location axis	71
2.6.8	Selection charts	72
2.6.9	Chart templates	73
2.7	Time Navigator	75
2.7.1	How to use the Time Navigator	75
2.7.2	Playing Movies	75
2.7.3	AVI-creator	76
2.8	Creating your own database system	77
2.8.1	Property templates	77
2.9	File Management	79
2.9.1	File management in WANDA 4	79
2.9.2	The File Menu	81
2.10	Arrange Menu	83
2.11	Selection Menu	87
2.11.1	Add to/ Remove keyword from selection	88
2.11.2	Select invalid components	88
2.11.3	Select components and/or connections	89
2.11.4	Deselect components and/or connections	89
2.11.5	Select route between selected components	89
2.12	Compare menu	92
2.12.1	Objects different in other case	92
2.12.2	Objects not found in other case	94
2.13	Units Menu	96
2.13.1	The Units Menu	96
2.14	Model Menu	100
2.14.1	Model Menu	100
2.14.2	Mode and options	102
2.14.3	Property labels	103
2.14.4	Change Log (Shift+F4)	105
2.14.5	Hydraulic input of your model	105
2.14.6	Initial values	108
2.14.7	System characteristic chart	109
2.14.8	Calculating your model	115
2.14.9	Run Parameter Script	119

2.15	Reports Menu.....	123
2.15.1	Input selected components.....	123
2.15.2	Input all	123
2.15.3	Output steady state.....	124
2.15.4	Output extremes	125
2.16	Tools+ menu	125
2.16.1	Knowledge management.....	125
2.17	Pump Energy	127
2.17.1	Overview Pump Energy module.....	127
2.17.2	Schematic aspects.....	128
2.18	Tips for WANDA 4 users.....	132
2.18.1	General tips.....	132
2.18.2	Shape libraries.....	132
2.18.3	Diagram general	133
2.18.4	Diagram editing.....	133
2.18.5	Diagram layers.....	134
2.18.6	Diagram routes	135
2.18.7	Property windows	135
2.18.8	Tables and tabular views.....	136
2.18.9	Charts	136
2.18.10	Time navigator	137
2.18.11	Selection builder	137
2.19	Frequently asked questions.....	138
2.19.1	Manage files.....	138
2.19.2	Gallery/Shape library	138
2.19.3	Route	138
2.19.4	Diagram / Layers	139
2.19.5	Property window	139
2.19.6	Charts	140
2.20	Important keys and shortcuts.....	141
2.20.1	Windows general	141
2.20.2	IGrafx Diagram.....	142
2.20.3	Wanda diagram specific	143
2.20.4	Property window	143
2.20.5	Table Windows	144
2.20.6	Spreadsheet windows.....	145
3.	Theoretical and mathematical background	147
3.1	Definitions.....	147
3.1.1	Nomenclature.....	147
3.1.2	Bernoulli's theorem	148
3.1.3	General definitions.....	149
3.1.4	Examples of systems.....	150
3.2	Modelling aspects	151
3.2.1	WANDA Architecture	151
3.2.2	Hydraulic components	151
3.2.3	Connect point variables	152
3.2.4	Component phases.....	152
3.2.5	Time dependant actions and control	153
3.2.6	Calculation types	153
3.2.7	Solution procedure.....	155
3.3	Implementation.....	157
3.3.1	Steady state	157
3.3.2	Unsteady state	159
3.3.3	Start Vector	162
3.3.4	Linearisation of H-models.....	163

3.4	Friction models.....	166
3.4.1	Theoretical friction models (Darcy-Weisbach).....	166
3.4.2	Non-Newtonian Fluids	169
3.4.3	Empirical models	172
3.5	Wave propagation speed.....	173
3.5.1	Circular cross-section	173
3.5.2	Rectangular cross-section	173
3.5.3	Reduction of wave speed due to free gas	174
3.5.4	Number of pipe elements	175
3.6	Cavitation	176
3.6.1	Conceptual Model.....	176
3.6.2	Algorithmic Implementation	176
3.6.3	Software Implementation.....	178
3.7	Pipe models	179
3.7.1	Pressurised pipe	179
3.7.2	Free surface flow conduit	184
3.8	References.....	186

4. WANDA Liquid 187

4.1	Liquid components and nodes.....	187
4.2	Overview of Liquid components.....	187
4.3	Modular concept of H-component properties	191
4.3.1	Specify actions.....	191
4.4	Hydraulic nodes	193
4.4.1	General	193
4.4.1	Pressure definition for H-node.....	193
4.4.2	Cavitation in H-node	194
4.4.3	Standard output of H-node types.....	195
4.4.4	Available H-node types.....	195
4.4.5	H-node	196
4.4.6	H-node with initial head	196
4.4.7	H-node with conditional initial head.....	197
4.5	Air Vessel.....	199
4.5.1	AIRVha (class).....	199
4.5.2	Airvessel horizontal vented.....	200
4.5.3	AIRVhn (class).....	201
4.5.4	Airvessel horizontal non-vented	202
4.5.5	AIRVva (class).....	204
4.5.6	Airvessel vertical vented.....	205
4.5.7	AIRVvn (class).....	207
4.5.8	Airvessel vertical non-vented.....	208
4.5.9	Airvessel Vertical Hybrid (class).....	211
4.5.10	Airvessel vertical hybrid properties.....	213
4.6	Inclined Air Vessel.....	216
4.6.1	Inclined air vessel (class).....	216
4.6.2	Airvessel inclined non-vented.....	217
4.7	Boundary Conditions.....	222
4.7.1	BOUNDH (class).....	222
4.7.2	BOUNDH (reservoir).....	223
4.7.3	BOUNDQ (class)	224
4.7.4	BOUNDQ (reservoir).....	225
4.8	Channel.....	226
4.8.1	Channel (class).....	226
4.8.2	Channel.....	229
4.9	Checkvalve.....	231
4.9.1	CHECKV (class)	231

4.9.2	Checkvalve (ideal)	237
4.9.3	Checkvalve (deltaP for reopen)	238
4.9.4	CHECKV - undamped	239
4.9.5	CHECKV - undamped (dimensionless)	241
4.10	Collector	244
4.10.1	Collector (class)	244
4.10.2	Collector	248
4.11	Conduit	250
4.11.1	Free surface flow conduit	250
4.12	Control valve	254
4.12.1	FCV	254
4.12.2	PdCV	262
4.12.3	PuCV	271
4.13	Damper, Surge vessel	281
4.13.1	Damper, Surge vessel	281
4.14	Infinite Pipe	284
4.14.1	INFPIP (class)	284
4.14.2	Infinite Pipe (H-bound)	286
4.14.3	Infinite Pipe (Q-bound)	287
4.15	T-JUNCTION	288
4.15.1	Mathematical model	288
4.15.2	T-Junction properties	296
4.16	Y-JUNCTION	301
4.16.1	Mathematical model	301
4.16.2	Y-Junction properties	310
4.17	X-JUNCTION	313
4.17.1	Mathematical model	313
4.17.2	X-Junction properties	319
4.18	Merger	321
4.18.1	Mathematical model	321
4.18.2	Merger properties	323
4.19	ORIFICE	326
4.19.1	ORIFICE (class)	326
4.19.2	Orifice plate	329
4.20	PIPE	331
4.20.1	Pressurised pipe	331
4.20.2	Pressurised pipe Properties	332
4.21	Fast filling pipe	341
4.21.1	PIPE FFP (class)	341
4.21.2	Fast filling pipe properties	343
4.22	Pressure relief valve	346
4.22.1	PRV (class)	346
4.22.2	PRV (simplified char.)	348
4.22.3	PRV (Ideal)	349
4.23	Pump	350
4.23.1	PUMP (class)	350
4.23.2	Pump properties	361
4.24	Pump Energy	367
4.24.1	Pump Energy components	367
4.24.2	PUMPSCEN (Pump Scenario)	368
4.24.3	Q_PATTRN (discharge pattern)	370
4.24.4	Q_FREQ (discharge frequency)	371
4.24.5	SYSCHAR (System characteristic)	372
4.25	PUMPSS	380
4.25.1	PUMPSS (class)	380
4.25.2	PUMPSS (Piston Pump)	383

4.26	QSCHAR	385
4.26.1	QSCHAR (Class)	385
4.26.2	QSCHAR (BoundQ for system characteristic)	386
4.27	Reservoir	387
4.27.1	RSVOIR (class)	387
4.27.2	RSVOIR (reservoir)	388
4.28	Resistance	390
4.28.1	RESIST (class)	390
4.28.2	Resist quadr. xi	391
4.28.3	Resist quadr. C	392
4.28.4	Resist linear C	392
4.28.5	Resist quadr. Initial Q	394
4.28.6	Resist 2-way quadr.xi	394
4.28.7	Resistance polynomial	395
4.29	Shaft	399
4.29.1	Mathematical model	399
4.29.2	Hydraulic specifications	400
4.30	Splitter	401
4.30.1	Mathematical model	401
4.30.2	Splitter properties	403
4.31	Surge tower, surge shaft	406
4.31.1	SURGTW (class)	406
4.31.2	Surgetower constant area	407
4.31.3	Surgetower height dep. area	408
4.32	TANK	409
4.32.1	TANK (class)	409
4.32.2	TANK (constant pressure)	410
4.33	Tap point / distribution area	411
4.33.1	TAP (class)	411
4.33.2	Tap return flow	412
4.33.3	Tap non-return flow	413
4.33.4	Tap non-return flow with DP reopen	413
4.34	Turbine	415
4.34.1	Turbine (class)	415
4.34.2	Turbine model	418
4.34.3	Examples	421
4.35	VALVE	427
4.35.1	Valve (class)	427
4.35.2	Valve properties	431
4.36	Vent, Air valve	434
4.36.1	VENT (class)	434
4.36.2	VENT (discharge coeff.)	440
4.36.3	VENT (inflow/outflow char.)	445
4.37	Vent, Air valve with filling pipe section	449
4.37.1	VENT2LEG (class)	449
4.37.2	VENT2LEG properties	454
4.38	Venturi	457
4.38.1	Venturi (class)	457
4.38.2	Venturi_open	459
4.39	Vee notch Weir	460
4.39.1	V-weir (class)	460
4.39.2	Vee notch Weir	462
4.40	Weir	464
4.40.1	WEIR (class)	464
4.40.2	WEIR (two way)	469
4.40.3	WEIR_S (class)	471

4.40.4	Outlet WEIR (incl. storage)	472
5.	WANDA Control	475
5.1	Control Components	475
5.2	Overview Control components	476
5.3	Modeling aspects	478
5.3.1	Measured signals	478
5.3.2	Controlled hydraulic components	479
5.3.3	Data flow Unsteady	480
5.3.4	Control Diagram aspects	480
5.3.5	Control Output properties	482
5.4	Definitions	483
5.4.1	Control diagram remarks	485
5.5	SENSOR	486
5.6	CONST	488
5.7	TAB	489
5.8	TAB2	491
5.9	FUN	493
5.10	WAVE	495
5.11	TIME	500
5.12	LtoN	502
5.13	MSP	504
5.14	COND	509
5.15	COND2	511
5.16	2COND	513
5.17	LCS	516
5.18	LCS2	518
5.19	SELECT	520
5.20	PID	521
5.21	PID_N	525
5.22	PIDSS	530
5.23	PIDSS_N	536
5.24	AND	541
5.25	OR	543
5.26	XOR	545
5.27	NOT	546
5.28	SUM	547
5.29	MUL	548
5.30	SUB	549
5.31	DIV	551
5.32	SFUN	552
5.33	STAB	554
5.34	MIN	556
5.35	MAX	557
5.36	MOVAVG	559
5.37	ABS	562
5.38	SIGN	563
5.39	BIAS	564
5.40	GAIN	565
5.41	RAMP	566
5.42	TRUNC	567
5.43	INTGRT	568
5.44	MOD	570
5.45	DELAY	571
5.46	LDELAY	572
5.47	LCON	573

5.48	LTAB	574
5.49	RDFILE.....	576
5.50	WRFILE.....	578
5.51	Examples.....	581
5.51.1	Tripping pump.....	581
5.51.2	Loading Arm.....	582
5.51.3	Flow control valve	583
5.51.4	Flow control valve with start/stop conditions	584
5.51.5	Control of a pumping station.....	586
6. WANDA Heat		589
6.1	HEAT domain.....	589
6.1.1	Core quantities.....	589
6.1.2	Fluid properties	590
6.1.3	Overview of Heat Components.....	591
6.1.4	Component properties	593
6.1.5	Hydraulic and thermal action tables	593
6.2	Heat nodes.....	594
6.2.1	Mathematical model.....	594
6.2.2	Heat node	595
6.2.3	Heat node init pT	596
6.2.4	Heat node init p.....	597
6.2.5	Heat node init T	598
6.2.6	Heat node with demand.....	599
6.2.7	Heat node conditional init pT	600
6.3	Heat air vessel	601
6.3.1	Mathematical model.....	601
6.3.2	Airvessel properties	603
6.4	Heat boundary.....	605
6.4.1	Mathematical model.....	605
6.4.2	Heat BoundPT	606
6.4.3	Heat BoundMT.....	607
6.4.4	Heat BoundT(PM).....	607
6.5	Check valve / Non-return valve.....	609
6.5.1	Mathematical model.....	609
6.5.2	Heat Check valve properties	610
6.6	Heat exchanger	611
6.6.1	Heat supply QHeat in.....	612
6.6.2	Heat supply Tdown	614
6.6.3	Heat supply QHeat in limit on T.....	616
6.6.4	Heat exchanger	618
6.6.5	Heat demand	621
6.6.6	Gas boiler.....	624
6.6.7	Solar collector	627
6.7	Surge tower, surge shaft.....	630
6.7.1	Mathematical model.....	630
6.7.2	Surge tower properties	630
6.8	4-Way Heat exchanger	632
6.8.1	Mathematical model.....	632
6.8.2	4-Way heat exchanger.....	635
6.9	Heat resist.....	637
6.9.1	Mathematical model.....	637
6.9.2	Heat Resist	638
6.10	Pipe	639
6.10.1	Mathematical model.....	639
6.10.2	Heat pipe.....	649

6.11	Pump	653
6.11.1	Mathematical model.....	653
6.11.2	Pump properties	663
6.12	Surge tower, surge shaft.....	669
6.12.1	Mathematical model.....	669
6.12.2	Surge tower properties	669
6.13	Heat pump.....	671
6.13.1	Heat pump (class).....	671
6.13.2	Evaporating section	673
6.14	Valve	679
6.14.1	Mathematical model.....	679
6.14.2	Valve properties	683
6.15	Converters.....	687
6.15.1	Mathematical model.....	687
6.15.2	Liquid to heat converter	688

7. WANDA Locks 689

7.1	Locks domain	689
7.1.1	Core quantities.....	689
7.1.2	Overview of lock components.....	691
7.1.3	Component properties	692
7.1.4	Action tables	693
7.1.5	Lock nodes.....	693
7.1.6	Mathematical model.....	693
7.2	Salinity boundary.....	695
7.2.1	ZL_BOUND.....	695
7.2.2	Mathematical model.....	695
7.3	Lock chamber.....	697
7.3.1	Mathematical model.....	697
7.3.2	LCKCMB2	698
7.4	Locks valve.....	700
7.4.1	Mathematical model.....	700
7.4.2	Valve properties	703
7.5	Ideal pump.....	706
7.5.1	Mathematical model.....	706
7.5.2	IDPUMP	706
7.6	Salt pipe	708
7.6.1	Mathematical model.....	708
7.6.2	Salt pipe	713
7.7	Lock Exchange.....	717
7.8	Converters.....	726
7.8.1	Mathematical model.....	726
7.8.2	Liquid to lock converter	727

1. Introduction

1.1 What is WANDA 4?

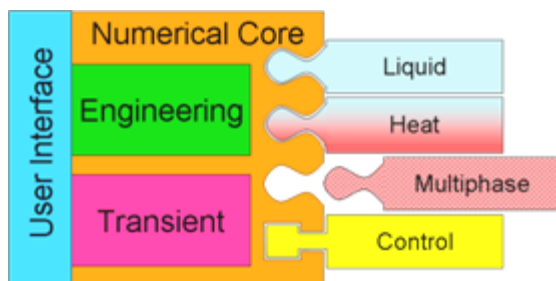
WANDA 4 is a powerful and user-friendly program for the hydraulic design and optimization of pipeline systems. Both engineers and operators use WANDA to study the steady and dynamic behaviour of liquid, heat, and slurry flows in arbitrarily configured pipeline networks. The software can be easily linked to other applications, e.g. for operator training systems or advanced real-time control. WANDA has been extensively tested, validated and designed for engineers by engineers.

WANDA can be used throughout the lifetime of the pipeline system: from initial design, component optimization, control procedure evaluation up to commissioning and operator training. Detailed information of the flow can be viewed at any location in the system at any moment in time. WANDA's intelligent components and standard-setting GUI enable you to carry out any hydraulics analysis in the most efficient way.

WANDA can model a rich variety of flow processes such as liquid, heat transport, cavitation and variable fluid properties. Flows from less than a few hundred litres per hour to well over 100,000 m³/h have been accounted for.

1.2 WANDA domains

The sophisticated software architecture of WANDA 4 enables the user to access physical domains such as heat transfer (density, viscosity, temperature) and multiphase systems. The Liquid module incorporates the former WANDA 4 version; the well-known User Interface and other successful parts are basically unchanged.



1.2.1 WANDA 4 Liquid

WANDA Liquid provides a wide variety of components from which most hydraulic systems can be built for each type of fluid, no matter what size or operational conditions of the network. WANDA Liquid includes amongst others:

- Accurate head loss of T-, Y- and X-junctions (Idelchik)
- Newtonian and non-Newtonian fluids (slurry friction model)
- Active components: pumps, (control) valves and air vessels
- Filling of pipelines
- Free surface pipe/channel component
- Transition of partial filled pipelines into a completely filled, "waterhammer" sensitive system
- Advanced components: e.g. turbine and dynamic check valve

1.2.2 WANDA 4 Heat

Heat transfer and monitoring the temperatures in a liquid system is realised with Wanda Heat, making it an interesting module for urban district heating networks and process industry. WANDA Heat calculates mass flow rates, pressures and temperatures.

- Full temperature dependent fluid properties
- Heat transport due to flow
- Heat transfer to and from the surroundings
- Heat transfer by dedicated components (heat exchangers)
- Heat generation by friction

1.2.3 Control

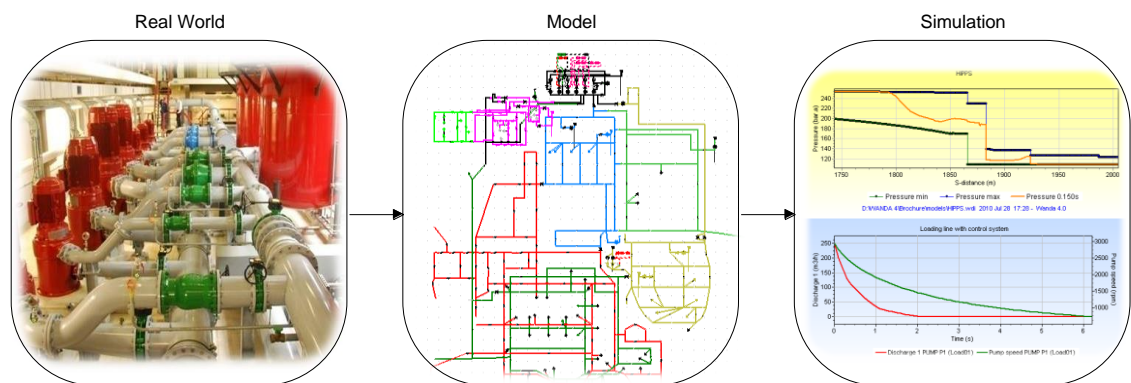
WANDA Control is a very powerful module supporting all other modules with the simulation of the complex operation of systems. The control module allows the hydraulic system to be linked to a control system. This allows the user to evaluate the effectiveness of different control philosophies on the hydraulic system. Notable features include:

- Advanced pumping station controllers including parameters for ramp-up time, minimum speed etc.
- Activation of valves, pumps, etc
- Varying of boundary conditions (pressure head or discharge)
- Sensor and generators (logical or numerical output)
- Conditional block (IF-statements and switches)
- Logic operators (AND, OR, exclusive OR, NOT)
- Numeric operators (add, multiply, functions, etc.)
- Continuous controllers (PID-controllers incl. start/stop)

1.3 Modelling aspects

1.3.1 From real world to mathematical model

In (numerical) models such as WANDA, only separate parts of the real world are included that can be considered more or less independent of external influences. At the boundaries of a model interaction with the outside world takes place. The modeller must decide based on experience what level of detail to include in the model. This process of schematization of the real world results in a model of the parts of the real world that are of interest to the user. If the modeller makes the right decisions, the model can be used to simulate events that (can) occur in the real world. In this way phenomena in the real world can be studied without the need of expensive or even impossible changes to the (pipeline) system.



A simulation is an event in the model that corresponds to the real world

WANDA helps the modeller to build a solid model for pipeline systems quickly. The modeller can build a model by connecting symbols of common hydraulic components to each other. The result is a scheme of symbols and lines that represent the part of interest of the real world. The symbols represent mathematical equations in WANDA and the lines exchange information

1.3.2 Hydraulic diagram editor

WANDA uses iGrafx® FlowCharter® as diagram editor. This program is part of the delivery and fully integrated in the WANDA software. iGrafx is a division of Corel®.

iGrafx® FlowCharter® is a full-featured, easiest to use process analysis and modelling tool to help organizations understand and improve (business) processes. A graphical representation of processes allows people to easily comprehend information and quickly focus on the bottlenecks and issues in a process.

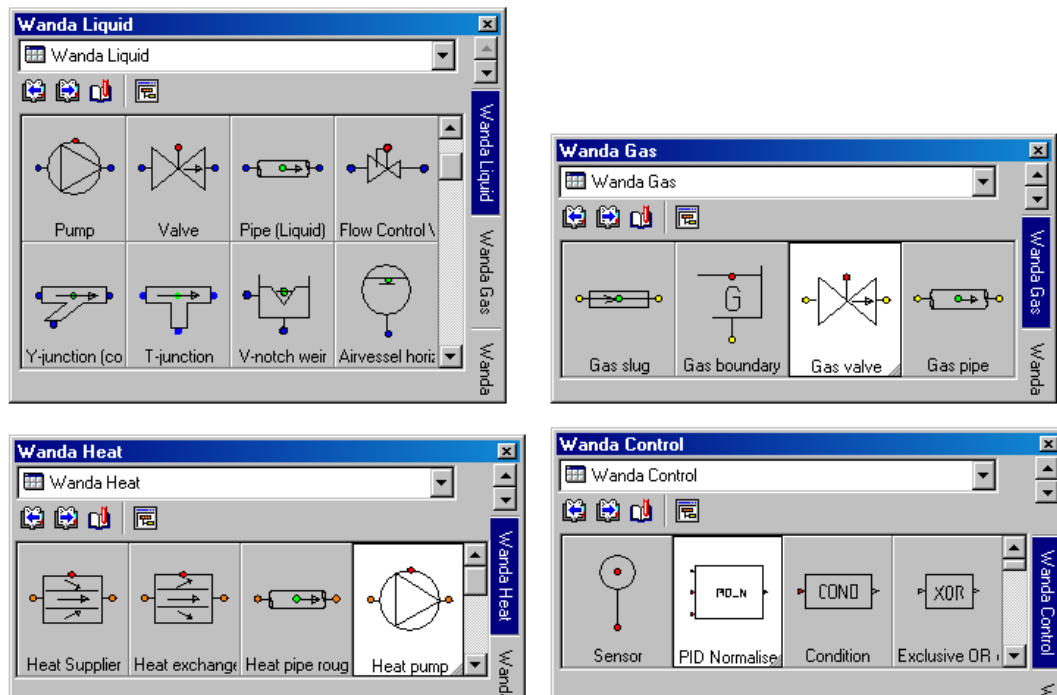
1.3.3 Components and galleries

The symbols in the WANDA diagram are called components. The components contain mathematical models which are expressed in equations. Combining the mathematical models creates a hydraulic model.

For each WANDA domain there is a library of components that can be applied to build the hydraulic model. This library is called the gallery. Each domain has its own gallery.

For each domain this manual contains a separate chapter where all available components are described.

The figure below shows some examples of the galleries.



Some components in different domains are very similar but the equations will be different. In the Liquid domain the main variables are H (head) [m] and Q (discharge) [m^3/s], in the Heat domain the main variables are p (pressure) [N/m^2], M (massflow) [kg/s] and T (temperature [$^{\circ}\text{C}$]).

Each component has 1 or more connect points. For each connect point the main variables are calculated. The components are connected to each other by connection lines. To prevent the user for connecting components from different domains each domain has his unique connect point colour (Liquid: blue, Heat: orange).

NOTE: Since the introduction of the WANDA software the group of hydraulic components and nodes are called H-components and H-nodes, With the introduction of new domains like Heat, each domain has his own group of components and nodes. Therefore it is better to use the general name of physical components and physical nodes which can be split into liquid and heat components and nodes. The annotation H-component and H-node as many times used in this manual must be read as pPhysical component and pPhysical node

1.4 Calculation modes

1.4.1 Time scales

A distinction can be made between several hydraulic aspects considering time scales. Therefore, we will distinguish two different modes: the Engineering mode and the Transient mode.

1.4.2 Engineering mode

In Engineering mode only steady state calculations are done, assuming that there are no changes in time and all variables are constant. The fluid is considered incompressible.

WANDA Engineering has been developed to support the most common activities in the hydraulic design of any piping system:

- Design of arbitrary hydraulic systems using a wide variety of available hydraulic components.
- Quick design using flow-prescribing components (pumps, valves, taps, etc.). Pump speeds, valve positions, tap head losses, etc are calculated. Flow and pressure balancing.
- Reliable design using component parameters from your own (company) database.
- Evaluation of pipe diameters, hydraulic gradients, flow velocities.
- Calculation of the system characteristic to support pump selection.
- Calculation of the energy consumption and efficiency of a pumping station

1.4.3 Transient mode

The Transient mode allows unsteady flow conditions to be investigated.

Wanda computes all different kind of unsteady phenomena, varying from extended time simulations until fast waterhammer events, including cavitation.

To change from steady to unsteady flow regimes it is necessary to change the behaviour of one or more components. Therefore some components are active which means that certain settings can be changed in time by the user or by control components.

Quasi-steady (extended time simulation)

Quasi-steady calculations can be considered as a number of independent steady state calculations, which happen to be following each other time. Although the conditions may change with time (e.g. different discharge patterns or valve positions for each time step), each calculation is independent from earlier solutions. For instance engineering tools like Epanet use the quasi-steady approach to determine the water levels and residence time in reservoirs during a certain period of time (day or week).

WANDA uses the quasi-steady approach to calculate the pipe friction at the new time step. WANDA Control uses the quasi-steady approach as it 'measures' at the old time step to determine the control-actions at the new time step.

Waterhammer (Pressure transients)

Due to abrupt changes in the flow conditions (valve closure, filling of pipeline or reservoir) pressure waves develop, which travel through the pipeline (surges). As the fluid and containing pipeline are compressible, the pressure wave needs time to travel from one end of the pipeline to the other end. These pressure waves are very quick phenomena (transients) with wave speeds up to over 1000 m/s.

In Transient mode WANDA calculates the pressure waves in all waterhammer pipes. The fluid in rigid column pipes is considered incompressible, resulting in an infinite wave speed.

Cavitation

Pressure waves and abrupt variations in the pipe profile (e.g. a partially closed valve) can result in substantial changes in pressure. When the fluid pressure drops below the fluid vapour pressure, cavitation can occur. Liquid becomes gas and as gas is more compressible than liquid, the amount of gas in the pipeline changes the flow conditions.

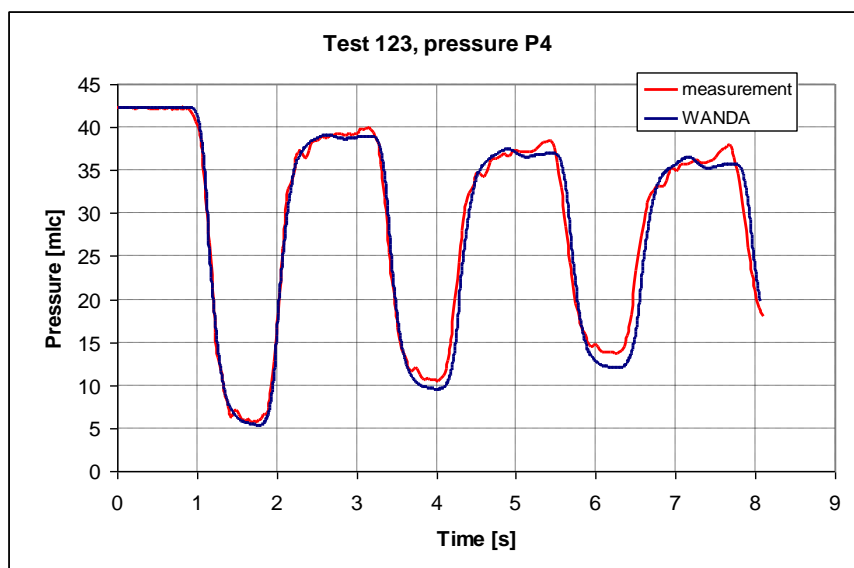
In WANDA the effect of cavitation is included as it recalculates the flow conditions at each time step depending on the pressure and fluid properties.

1.5 Validation

The numerical models in WANDA and its predecessors have been validated since the seventies. For these validations special test rigs were build (e.g. 1000 m long cavitation test rig) or field measurements are used. Component modelling is based on physical performance testing in the hydraulic flow laboratory of Deltares as well as on site testing. Deltares' reputation and decades long experience of fluid dynamics are a guarantee for the high quality of this powerful simulation software package.

A benchmark analysis has been conducted in a European project between 1998 and 2002, in which WANDA and three other commercially available transient simulation software tools have been benchmarked against a selected set of field and lab data.

The validation report is available via <http://wanda.deltares.nl>



Validation results from measurement and WANDA simulation

1.6 WANDA 4 main features

- Branched and looped systems,
- Steady flow,
- Transient flow,
- Free surface flow (optional),
- Cavitation,
- Control module (optional),
- System optimisation module,
- Over 100 different components, supporting rapid design calculations,
- Powerful pipe model with several friction models including local losses,
- Interrupting and resuming of simulations,
- Graphical user interface with intuitive look and feel
- Simple and flexible operating principles,
- Choice of different unit systems (SI, UK, US, user units),
- Your own (company) database of hydraulic component data
- Graphical and tabular presentation of time and distance functions of all hydraulic output results
- Animation of the dynamic simulation results

1.7 To whom is WANDA 4 a useful tool?

In the years following the introduction of WANDA in 1994, it has been licensed to many well-known companies and it has been used for diverse applications related to:

- (Process) plant piping,
- Sewage transport pipelines,
- LPG pipelines,
- Long distance crude oil pipeline transport,
- White product conveyance systems,
- Oil loading and off-loading pipelines,
- Fire fighting systems,
- Drinking water networks,
- Water treatment facilities
- Cooling water systems,
- Heat transportation systems,
- Gathering and distribution networks,
- Chemical pipelines,
- Slurry transport and sludge handling

1.8 How to use this manual

In chapter 2 the WANDA user interface is described. This part is based on the Wanda 4 Liquid module. The basic user actions like creating a hydraulic model, specifying data and retrieving result in charts and tables applicable for all other modules.

Chapter 3 contains the main theory necessary to understand the numerical background of the program.

For each physical domain a separate chapter is used where all individual components are described.

1.9 System requirements

In order to run the Wanda software, your computer must meet the following requirements:

- Operating system: Windows XP, Vista, 7, 8, 10
- At least 1 GB of memory
- At least 512 MB free space on hard drive
- The user account must have sufficient privileges to read from the Wanda installation directory
- The user account must have sufficient privileges to write to the document directories, user profile and Windows TEMP directory

1.10 About Deltares

Deltares owns a flexible large-scale, Multi-phase Flow Facility (alpha-loop) in order to enable investigations in a wide range of academic and industrial multi-phase research topics. This includes studies of gas blow out, pressure relief valves and air valves. The alpha-loop allows for single-phase (water), two-phase (water, air) and three-phase (water, air and sediment) studies in a pipeline system of arbitrary shape.

1.11 Technical Support

Deltares

Boussinesqweg 1

P.O. box 177

2600 MH Delft

The Netherlands

Wanda community Wiki: wanda.deltares.nl

internet: www.deltaressystems.com

Software Sales team:

telephone: +31 88 335 8188

e-mail: Sales@deltaressystems.nl

Software Support team:

telephone: +31 88 335 8100

e-mail: Support@deltaressystems.nl

2. User Interface

2.1 Overview of WANDA 4 features

2.1.1 MSOffice look and feel

- Multiple documents support.
- Full drag and drop support.
- Full clipboard support.
- MS office like tool bars.
- MS office like menus.

2.1.2 Diagrams

- Wanda 4 uses iGrafx FlowCharter 2013 as diagram engine.
- Full drag and drop support.
- Copy and move objects and their data between one or multiple documents.
- Clipboard support to Word/Excel.
- Export a diagram to HTML.
- Export a diagram to AutoCAD.
- Export a diagram to Visio.
- Enhance a diagram with your own text and drawings.
- Multiple layer support (you can hide and protect layers easily).
- Use your own pictures/maps as background (bmp, gif, autocad, coreldraw, wmf).
- Different line styles / weights / colours supported.
- Zoom support.
- Quick zoom to selection.
- Auto fit your printed diagram to m-by-n pages.
- Direct feedback to the user. The diagram is real-time validated.
- Components with empty/incorrect properties can be easily selected. The diagram window scrolls automatically to the invalid component if the component is out of side.
- Vector based and object based schematisation.
- Components including connection lines are automatically rotated when you add them in vector based mode.
- Hydraulic nodes are automatically recognised. Multiple lines that are connected to each other are recognised as one single calculation node.
- Custom-made shape libraries. Deltares can design shape libraries with customer specific components. You can use different shape libraries simultaneously.
- Protect your diagrams and properties by a password.
- Optionally visible grid.
- Optionally snap objects automatically to grid.

2.1.3 Routes in diagram

- You can quickly select a route/path through your diagram by selecting the first and last component in the route.
- You can quickly 'align to line'/'rotate'/'reverse a route'.
- Selections that contain an unambiguous sequence of pipes are automatically recognised as route.
- Selections and routes can be stored using a keyword.
- You can see simultaneously an unlimited number of (moving) chart series of different routes.
- You can see the overall extremes and total length of a route.
- You can build routes with different pipe orientations (results are automatically 'flipped')

2.1.4 Automatic name generation

- Components and connections are automatically named.
- If you assign a name that already exists or if you copy an existing component a new unique name based on the existing name is assigned to the new component. If the name ends with a number, this number is increased, otherwise a character is added to the name.
- You can rename a selection of components / connections with a single mouse click.

2.1.5 Property text boxes

- View properties (input and output) of hydraulic components directly in text boxes in the diagram.
- You can visualise output directly in the diagram for the active time step (see "How to use the Time Navigator" on page 75).
- You can visualise output extremes directly in the diagram.
- You can specify the properties that you want to see in the diagram.
- You can easily move property text boxes in the diagram.
- Property text boxes are always attached to a component or connection. Text boxes are automatically moved/deleted/copied when the component or connections moved/deleted/copied.
- Property text boxes are placed in a separate layer. This layer can be made hidden. When this text layer is active you can easily move and change the font attributes of those property text boxes.
- Property text boxes are also visible when you print the diagram.
- The properties 'Comment', 'Model name' and 'Reference ID' (for pipes only) have been included to support a traceable modelling in accordance with ISO 9001.

2.1.6 Property list windows

- The main property window displays the properties of all components and connections in diagram selection.
- You don't have to open a separate dialog every time you want to inspect/edit an object. The main property window is instantly adjusted when the diagram selection changes.
- You can simply select one or more objects and view/edit this objects at once.
- You can open an unlimited number of extra property windows that show a single object.

- Input and output together in one list.
- You can choose to see the output for the active time step (see "How to use the Time Navigator" on page 75) or to see the extremes in the simulated time period.
- Drag and drop support between two property lists.
- Easily select properties in the same way you select files in a file manager/explorer. A property that you edit is automatically selected.
- Quick synchronisation of selected properties between objects.
- Easily compare properties (and tables) from two or more hydraulic components (different values are highlighted in red).
- Direct access to all tables from the property list of a hydraulic component with one mouse click.
- Write the selected properties to a template file.
- Easily change properties by loading a template file (or dropping a template file in the property list window). The loaded properties are automatically selected/highlighted. If you have loaded the wrong template file you can simply press the undo button.
- Easily connect you own programs/databases to the property list by using the clipboard or template files.
- Opening a time or location chart of one property by pressing the chart button of a property.
- Add a property to a chart by dragging a chart button to any chart you like.
- Enter values with other units than the active unit group.
- View the property value in every available unit for the property dimension at once.
- Optionally hide the input or output data.
- Overall view of all the selected components. In this view you see the (extreme) values of all the properties available in the selected components.
- Change the component type without losing the property values and tables.
- Rollback/Undo support.
- User name property. The name of the user that made a change to a component is added to the user name property of the changed component.
- Modified property. This property shows the last date/time the selected component was changed.

2.1.7 Table editor

- The table data can be visualised in a chart next to the table. This chart is automatically updated while you are typing.
- Easy adding or removing (multiple) rows.
- Clipboard support within one table editor.
- Clipboard support between two table editors.
- Clipboard support between one table editor and Excel/Word.
- Simple numerical operations (+, -, x, /) to a range of table values.
- Auto fill missing values by linear interpolation.
- Print management.

2.1.8 Spreadsheet

- Spreadsheet can be used to edit or view properties of several components simultaneously.

- Create a spreadsheet of the diagram selection and selected properties in the property window by one single mouse click.
- Unlimited number of different spreadsheets from one or more documents.
- Components are sorted by type and name.
- Column width is automatically adjusted to fit all values.
- You can transpose the spreadsheet. This way you can display components side by side as well as above each other.
- You can display extreme values instead of the values of the active time step (see "How to use the Time Navigator" on page 75).
- When you print a spreadsheet you can automatically fit to m-by-n pages (portrait and landscape).
- Print headers and footers are automatically updated.
- You can copy a selection to Excel using the clipboard.
- Navigation coupling between spreadsheet and diagram.

2.1.9 Calculations

- Fast calculations.
- Change something and see the impact of that change in all open windows.
- Stop and resume transient calculations.
- You can stop a transient calculation before the simulation has ended. When the calculation is stopped you can see the results so far. You can change action tables to interact with the results before you continue the calculation.
- Background calculations. While a calculation is in progress you can inspect the running document or continue with another document.
- You can calculate multiple documents sequentially (like printer jobs).
- Create child cases from parent case
- Repeat calculations with varying some input parameters as defined in a simple parameter script; specified output summarised in compact table
- You can disuse components temporarily instead of deleting them from the diagram. It is possible to hide parts of the diagram for the calculation kernel. Disused components are greyed out in the diagram. You need one look to see the differences between variants, if you disuse components instead of deleting them.

2.1.10 Chart engine

- Drop multiple series (from different documents) into the chart.
- Moving location series supported (picture for active time step, see "How to use the Time Navigator" on page 75).
- Frozen location series supported. Frozen series show one fixed time step.
- Chart time cursor. A vertical blue line gives the active time step in time charts (see "How to use the Time Navigator" on page 75).
- View as many charts simultaneously as you want
- All open charts support moving series and are automatically updated when input or output is changed.
- Create your own custom dedicated chart templates.
- Export the chart data to a spreadsheet using the clipboard.
- Export the chart to a WMF/JPG/BMP/GIF picture.
- Paste the chart in Word using the clipboard.

- Series are automatically removed when the (belonging) hydraulic component is deleted or the document is closed.
- The chart is automatically closed when it is empty.
- Zoom and scroll supported by mouse.
- Chart headers and chart footers are automatically updated.

2.1.11 Time navigator

- The time navigator controls the *active time step* at once for all open documents. This makes it easy to compare different documents.
- Same look and feel as the multi-media-player of windows.
- Fast navigation (less than one second) to every time step in your model output.
- The time navigator controls all open property windows, tabular views, charts and diagrams at once.
- Auto walk forward/backward to the model output at every speed you want.
- Auto walk and repeat trough a selected time-period.

2.1.12 Unit groups

- Quickly switch between different unit groups.
- All property lists, tables, tabular views and charts are immediately adjusted to the active unit group.
- User customised unit group.

2.1.13 Selection builder

- Quickly (de)select components/connections by a property, value and operator (<, <=, =, <>, >=, >, like).
- Existing keywords, component types, model names and user names are listed by a drop down list.
- Add/Remove keywords to/from a selection.
- Smart zoom/scroll. If not every selected object is visible the view extend is adjusted to see every selected object.

2.1.14 'On demand' data loading / Smart saving

- Quick access to output data of existing documents.
- Complicated documents with large output data (100Mb) are loaded within seconds.
- Changes are only saved at the moment you save explicitly. If your computer goes down your last saved documents are unchanged/undamaged.
- Only changes are saved. This makes it possible to save slightly changed large documents within a second.

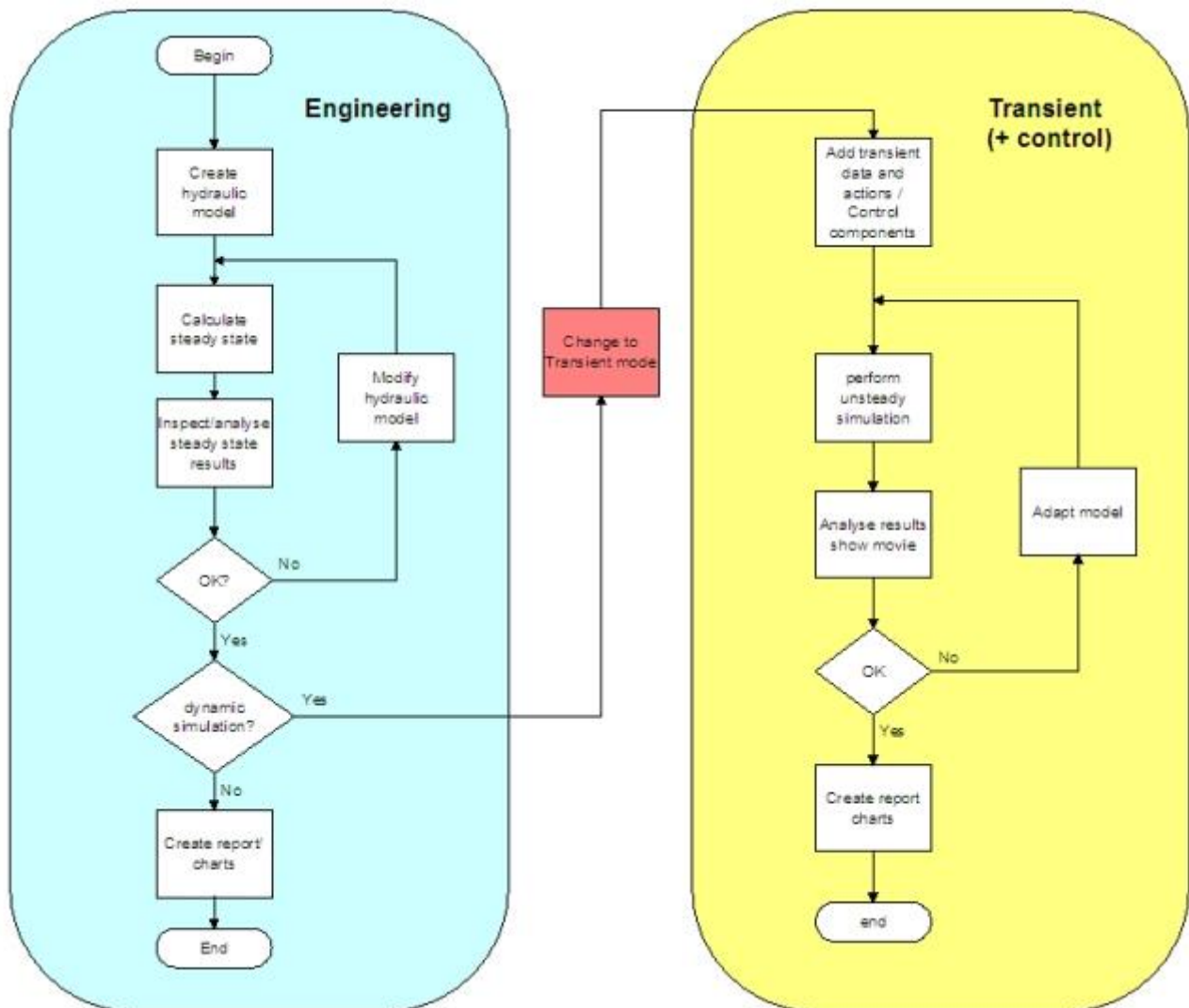
2.1.15 Instant window/view refresh

- Every (implicit) change is directly reflected in all open views (diagrams, property lists, tables, tabular views and charts).
Any recalculation is directly reflected in all open views. You do not have to reselect your output after recalculation.

2.2 Getting Started

2.2.1 Hydraulic analysis - Functional steps

Examining a hydraulic study of a pipeline system depends on the objectives of the study. A typical sequence is shown in the diagram beneath.



In Engineering mode only so-called steady state calculations can be performed. You have no access to the unsteady state simulation to carry out dynamic phenomena. Only input parameters necessary for the steady state (resistance) calculations must be entered. Changing to Transient mode (menu bar: model/change mode & options) all additional data for the unsteady state becomes accessible.

Opening a new document; it always start in Engineering mode

2.2.1.1 Create the hydraulic model

The first step is to create the schematic diagram of the pipe system and enter component specific hydraulic input data such as geometry, size and other characteristics. This is called “the hydraulic model”. It must be completed successfully before the user may proceed with the actual computation of steady and transient flow.

The most essential function of this part is the conception of the hydraulic model and input of the component specific hydraulic data.

The hydraulic model is in fact a schematic drawing of the actual pipe system, using lines and easy-to-read symbols that represent different elements in the system. It defines all the components in the pipe system with respect to their class and the way they are connected. The conception of the model is realised in the **diagram user interface** and is fully mouse-operated. WANDA has been designed in such a way that a model can be created with minimum efforts.

The numerical hydraulic data specific to each component are entered via the **property windows**.

2.2.1.2 Calculate steady state

WANDA computes the steady state. The results are used as the initial conditions for the computation of the unsteady state. STEADY has no dialogue with the user.

2.2.1.3 Specify hydraulic actions

(Applicable to WANDA Transient only; change to Transient mode using menu model/mode & options)

In WANDA one can specify the total simulation time and hydraulic actions. Hydraulic actions, such as manipulation of a valve, pump trip or prescribed changes in pressure head or delivery rates, cause hydraulic transients.

Actions are specified in the property window of the particular H-component.

2.2.1.4 Calculate transient

(Applicable to WANDA Transient only; change to Transient mode using menu model/mode & options)

WANDA also computes the transient flow. This is a separate task. If you unhide this task from the taskbar, the progress in computation including logging messages can be monitored. The estimated time for the simulation is displayed in the header (caption) of the diagram window. The user can interrupt the computation at any time. An interrupted computation can be resumed later on using the same simulation.

During the unsteady calculation the “real time” results can be followed. The user opens the selected graphic windows before starting a calculation. The graphics in these windows are regularly refreshed during the calculation (refresh time is set in model/time window). Looking to the progressing results, you can interrupt the simulation when you find them unsatisfactory. For example, when you evaluate a control system the

behaviour of the control can be followed during the simulation. This way it is possible to reduce the simulation time.

Editing data after a calculation all old results remain visible, however coloured differently, and with a strike-through value.

2.2.1.5 Get results

WANDA has powerful utilities to show results. Selected data can be printed in standard report format. Location and/or time series can be put into charts and the dynamic flow process can be visualised using an internal movie feature. The data report and the charts can be displayed on screen, saved on disk or directly sent to the printer.

Printed data report

Printed data are useful for inspection of numerical values of calculated variables. The input and output report are adequately organised in a spreadsheet view, such that it can be included in any formal documentation of a waterhammer simulation.

Charts

These are useful for examining how the pressure or discharges vary as a function of time or along the pipes. Pressure waves travelling along the pipe system are easily recognised from a graph. Selection of the variable and settings of the chart are menu operated and require minimum effort.

The movie feature

The movie is used to visualise the dynamic behaviour of pressures and discharges along selected routes in the pipe system (see "Time Navigator" on page 75).

When the steady and transient state computations are completed, the user might sometimes not be interested in the absolute numbers of calculated variables. He may first want to get some insight into the physical flow process. Animated view of the flow variables using the movie feature will prove to be quite helpful for that purpose. Furthermore the extreme values are displayed in the movie chart. The movie can be recorded and save as AVI-file.

2.2.2 WANDA 4 - User Interface

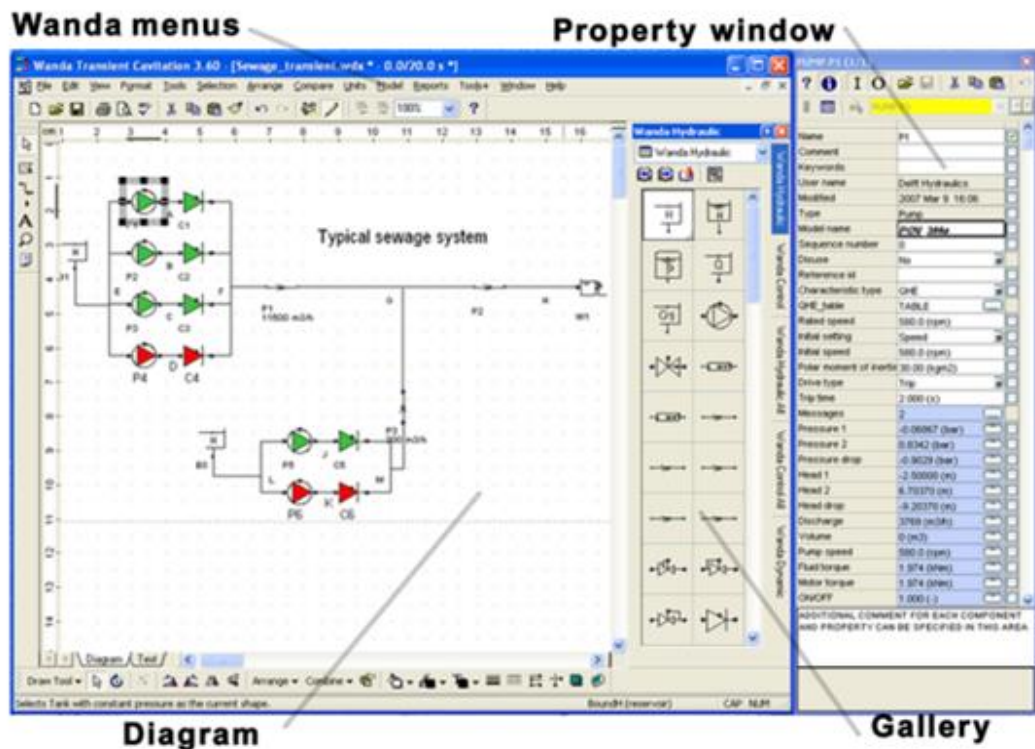
In this section we will briefly explain how to use together the Component Gallery, the Property Window and the Diagram, in the construction of a hydraulic model.

See also:

- Manual/Help iGrafx FlowCharter 2013
- Help on "Liquid components" on page 187.

2.2.2.1 Introduction

When you start the application you see a screen that can be divided in three major parts.



The Diagram in your first session will be empty of course.

The **Diagram** is the area where you will actually build the hydraulic model that you wish to calculate (see "Building the diagram" on page 32). You will build this model adding components from the "Shape Library / Gallery"; see page 32.

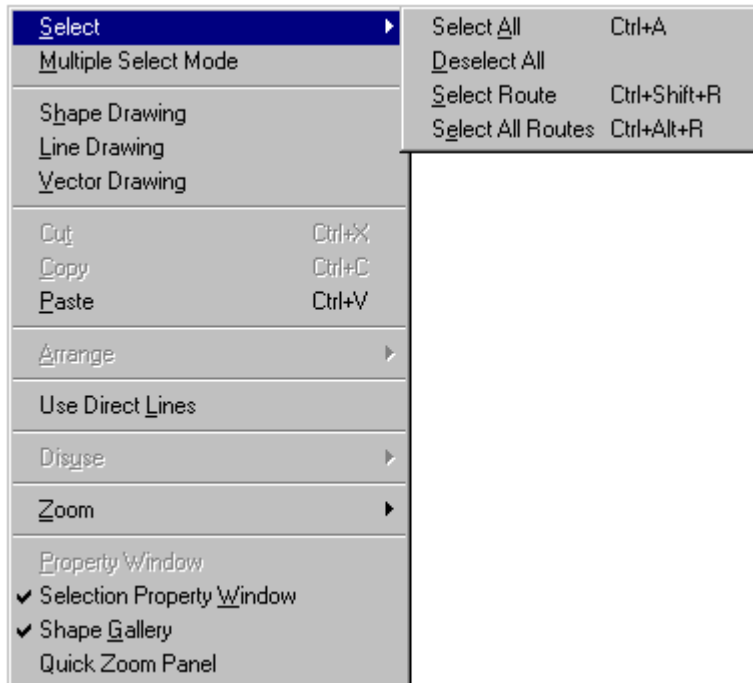
The menu bar contains:

- iGrafx Flowcharter menus (file/edit/view/insert/ format/tools/arrange/window) and
- WANDA menus (Selection/Compare/Units/Model/ Report/Help).

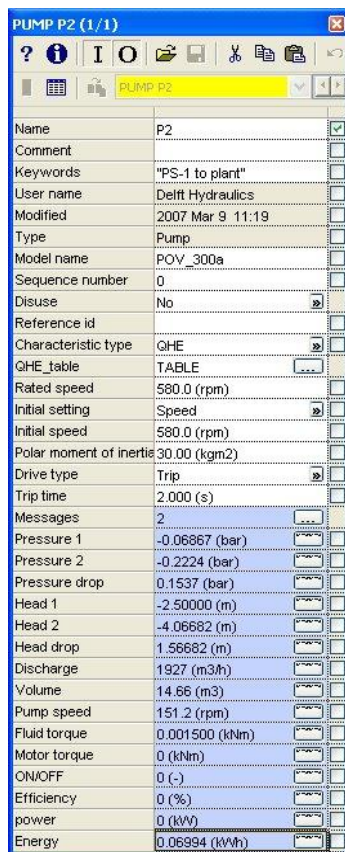
The various **WANDA Menus** are used to:

- quick selection of components to view or edit (see "Selection Menu" on page 87)
- compare contents and show differences of two cases (see "Compare menu" on page 92)
- choose the desired unit for view/edit (see "Units Menu" on page 96)
- specify your other input data (see "Model Menu" on page 100)
- view and report the results of your hydraulic model (see "Reports Menu" on page 123).

The diagram short cut menu (right mouse click) gives you fast access to the most important diagram features



At the right side of your screen, the **Property Window** is displayed.

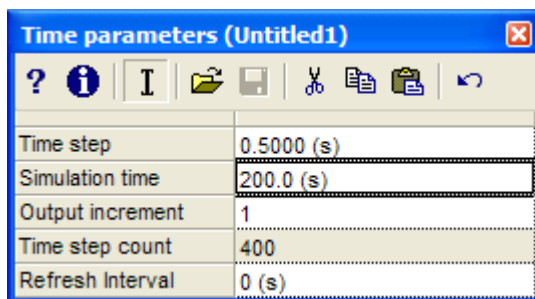


In this window you can see input and output of a (group of) selected component(s). The Property Window is one of the most important features. It helps you to navigate through your data. The short cut menu (right mouse click) gives you access to the most important property window features.

Select	Space
Deselect All	Ctrl+A
Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V
Edit	F2
Edit Comment	Shift+F2
Open Spreadsheet	Ctrl+Shift+S
Help	F1
User Defined Help...	Shift+F1
Copy to Diagram Selection	Ctrl+Enter
Active Time Values	
Extreme Values	
Extreme Locations	
Extreme Times	
✓ Locked	Ctrl+L

See for further detail: "Property Window" on page 43.

2.2.2.2 Dialog windows - General



The appearance of all WANDA dialog windows the proceeding is the same, so that the user will get familiar with the program immediately.

All dialog windows contain:

- a toolbar
- a number of input boxes (white background)
- a number of output boxes (blue background)

Toolbar



Most icons are self-explaining (copy, past, save, open, etc.). Read the tool tip to understand the feature.

Input, Output and Units

We would like to draw attention on the following three buttons in the toolbar:

- Displays the input of a selected component
- Displays the output of a selected component
- Displays the possible units for a selected property of a selected component.

Input boxes

For all characteristics, default values are supplied. If an input box is grey, no values can be entered. The values in the white boxes can be edited.

Output boxes

For all type of components calculated output values are displayed in boxes with a blue background. These values cannot be edited (read only).

2.2.2.3 *Tooltips*

ToolTips are short messages that appear in bubble text. These messages help to explain the name of the tool or button and, in some cases, what the tool or button does.

In WANDA all functions, buttons, icons, etc. have their own ToolTip. It is advisable to look at these tooltips first before selecting a WANDA 4 option.

2.2.3 WANDA 4 in 10 steps

2.2.3.1 *Step 1. Add Components*

See:

- "Shape Library / Gallery" on page 32
- "Adding components" on page 33

2.2.3.2 *Step 2. Connect Components*

See:

- "Looking closer at components in the diagram" on page 33
- "Connecting components" on page 34
- "Aligning routes" on page 40

2.2.3.3 *Step 3. Set Component properties*

See:

- "Property Window" on page 43
- "Changing the flow definition" on page 40

2.2.3.4 *Step 4. Set other inputs*

See:

- "Units Menu" on page 96
- "Fluid..." on page 106
- "Physical constants ..." on page 107

- "Time parameters..." on page 107
- "Accuracy..." on page 108

2.2.3.5 *Step 5. Use advanced features*

See:

- "Add to/ Remove keyword from selection..." on page 88
- "Selection Builder" on page 62
- "Selection Menu" on page 87
- Creating your own database system
- "File Management" on page 79

2.2.3.6 *Step 6. Calculate steady state*

See:

- "Model Menu" on page 100

2.2.3.7 *Step 7. Specify hydraulic actions*

See:

- Help on "Liquid components" on page 187.

2.2.3.8 *Step 8. Calculate transient*

See:

- "Model Menu" on page 100

2.2.3.9 *Step 9. View output*

See:

- "Time Navigator" on page 75
- "How to use charts" on page 64

2.2.3.10 *Step 10. Report results*

See:

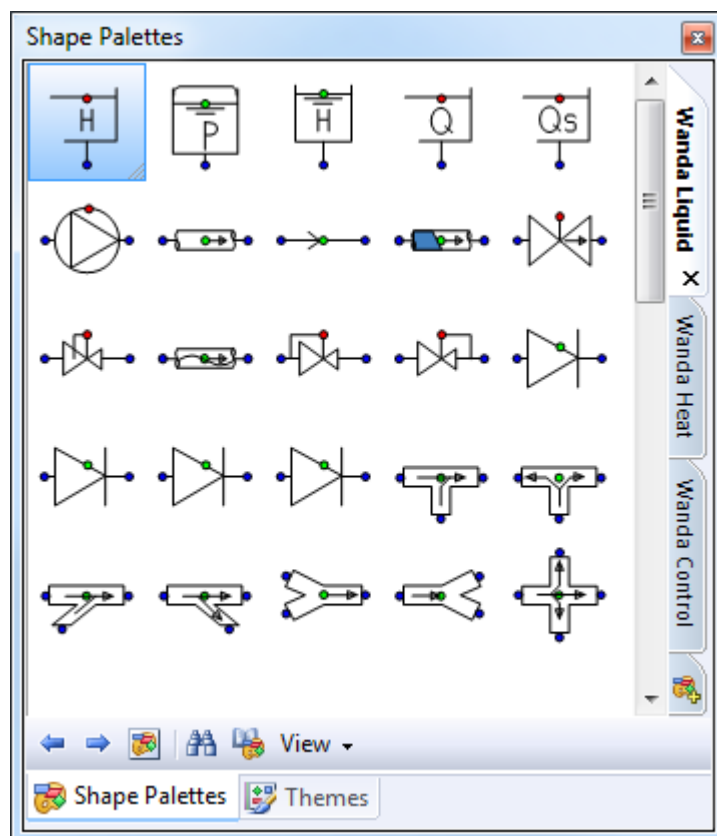
- "Reports Menu" on page 123

2.3 Building the diagram

For the diagram the incorporated program iGrafx Flowcharter is used. All features about diagram editing is explained in the separate Flowcharter Manual and Flowcharter Online Help.

In this chapter the WANDA related topics are explained.

2.3.1 Shape Library / Gallery



The default shape Libraries WANDA 4 shows are:

- WANDA Liquid
- WANDA Heat
- WANDA Control

In the future new galleries will be introduced for other physical domains or hybrid components. Special made components for particular clients are stored in the “WANDA special” gallery.

To show the status of some active components, like PUMP and VALVE, WANDA uses an extra gallery “WANDA Dynamic” but this gallery will be hidden.

It is possible to assemble your own gallery which contains a subset of the standard galleries. This can be easy to use only one gallery with the frequent used components.

To create your own gallery, consult the iGrafx Help (Shape pallet, F9 – Media manager)

If the gallery is not visible, go to menu view /show gallery or click right in the diagram to activate short cut menu and check Gallery.

By dragging the components in the library you can change their order, or edit the size of the icons.

Default only the shape is visible in the gallery. The tooltip informs you about the particular type. To display this type in the gallery, use the right mouse menu

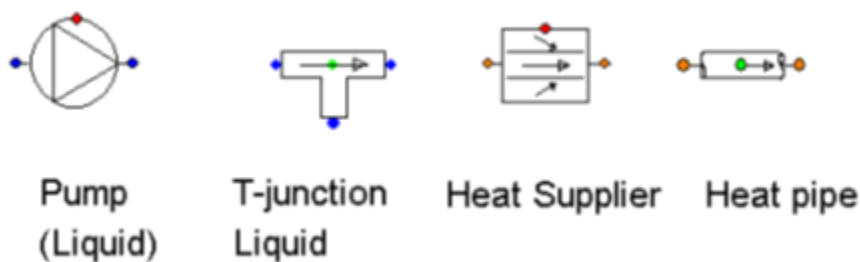
2.3.2 Adding components

You can add components in the diagram by selecting them in the gallery and dragging them into the Drawing Area.

For details on this subject we also refer to the iGrafx Flowcharter Help.

Each component gets automatically a name starting with the first character of the component type followed by a sequence number. This name can be changed in the property window. The combination “Component type + local Name” must be unique.

2.3.3 Looking closer at components in the diagram



When we take a closer look at the components we see that they have several connect points in different colours. We distinguish physical connect points and control connect points. The physical connect points are used to connect other components which belong to the same physical domain. Therefore each domain has his own connect point colour: blue the Liquid domain, and orange for the heat domain.

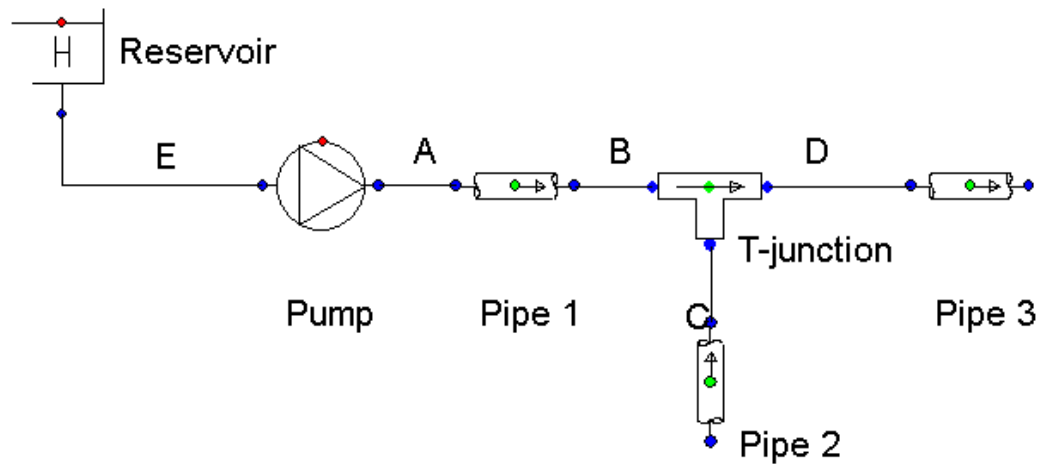
It is not possible to connect different coloured connect points to each other

The red and green connect points are reserved for the control module

Note that the shape of a component indicates the positive flow direction if applicable (e.g. P1 and V1 in the picture above). You can change the flow direction by flipping or rotating the component in the Arrange Menu.

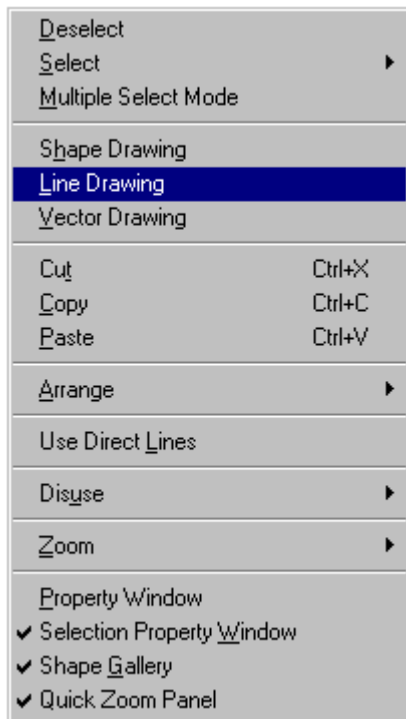
For details on this subject we also refer to the iGrafx Flowcharter Help.

2.3.4 Connecting components



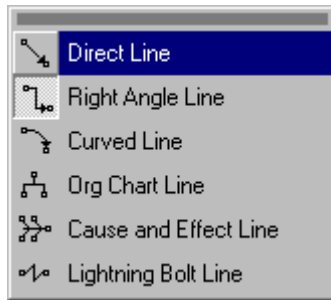
Connecting components is as simple as adding them. All you've got to do is select a component and drag the mouse pointer to another component.

To switch to line mode, activate the right mouse click menu and choose "Line drawing mode"



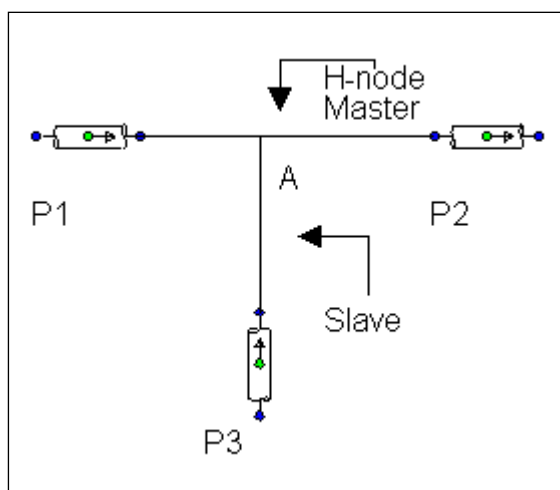
You can also use the Connector Line Tool from the Toolbox, left of the Diagram.

For WANDA 4 diagrams, it is recommended to use only the first three types from the menu below.



See also iGrafx Flowcharter Help for details on drawing and manipulating connections.

2.3.4.1 Master and Slave connections

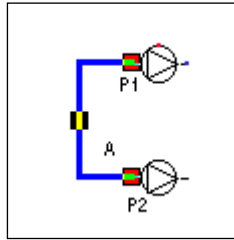


The H-node is represented by at least one single (master) connection line that can connect none, one or two H-components. With extra slave lines it's possible to connect more than two H-components to one H-node

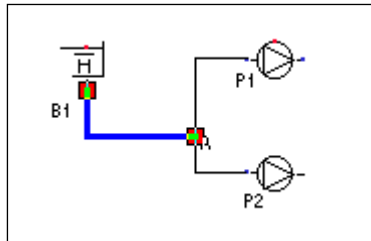
A connection line that is NOT connected to any other line is called a master line. The first connection line you draw is always a master line. Slave lines are connected by ONE end to another master or other slave line. It is not allowed to draw a line between two other lines.

If you select a connection line both ends are marked with a square. The colour of this square is very important. When the square is red it tells you that the line is connected to another line or component. If the square is black the line end is connected to nothing. It's possible that a line is visually connected but is not connected for Wanda. In this case you see a black square at the fake connected end of the line. When you press the pause button while the property window is active you can see the connectivity of h-components and h-nodes used by Wanda.

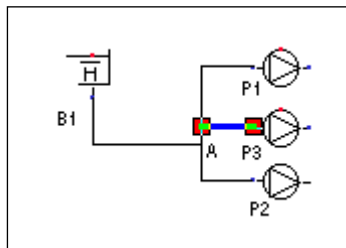
The easiest way to fully understand connections is to draw some arbitrary connection lines. If you draw something that is not allowed (schematically) Wanda popup a message and deletes the illegal lines”.



Step 1. Adding connection A (master) between P1 and P2



Step 2. Adding connection (slave) from B1 on to connection A



Step 3. Adding connection (slave) from P3 on to connection A

The pictures above show the steps to create a “complex” H-node.

The master and the slave connection line belong to the same H-node. So it doesn't matter which line you select to edit the properties.

Strings of lines

If you want to connect two components over a long distance it is easier to draw the connection with several shorter lines

First draw the master line that is connected to none or one H-component. Then draw a slave line from the free line end to a second (grid) point. You can attach a second slave line from the free end of the first slave line (and so on).

Deleting master lines

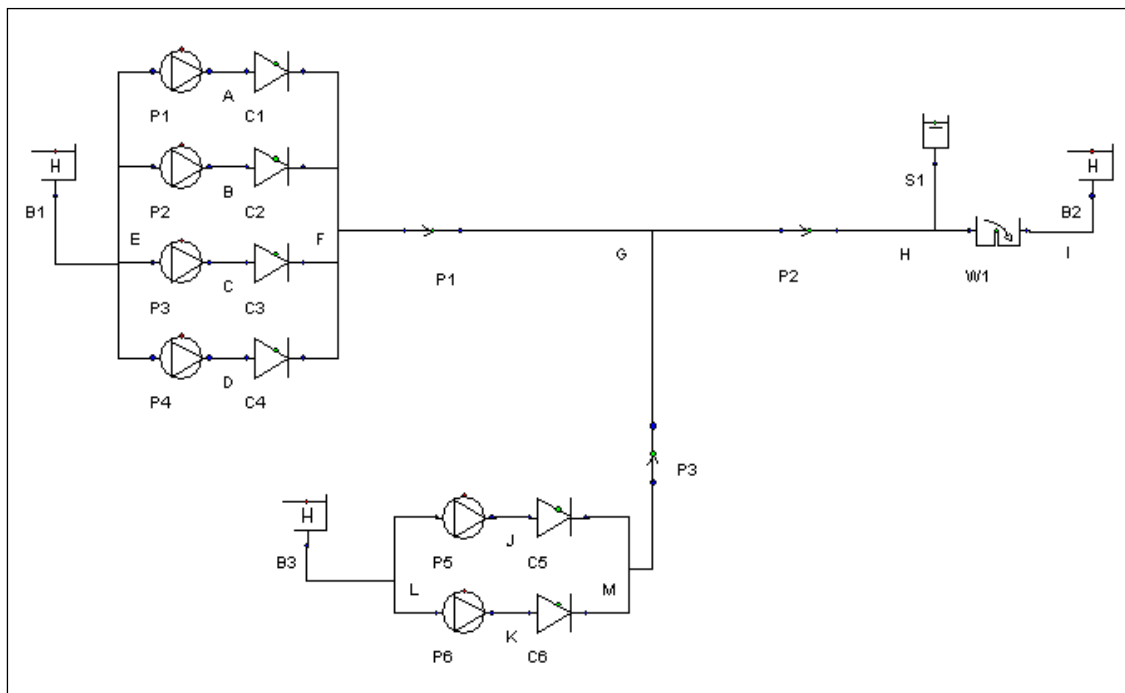
If a line is deleted automatically all lines attached to this line are also deleted. If you do not want this, disconnect the attached lines before delete.

Combining / splitting H-nodes

When one end of a master line is connected to another line, the master line is automatically converted to a slave line. When you disconnect a slave line from another line it is automatically converted to a master line.

2.3.5 Rules for creating a hydraulic model

A hydraulic model usually consists of several H-components of different classes connected in a specific way. An example of a hydraulic model is given below.



A hydraulic model is built according to a few simple rules. If possible each rule is validated directly and in case of incorrectness a message explains the violation.

Rule 1

A fall-type component has two hydraulic connections: a "FROM" connect point (1) and a "TO" connect point (2) (also called the left-hand side (lhs) and right hand side (rhs)).

The relative position of the two connect points also defines the positive flow direction through the component. This direction is from left to right (FROM --> TO).

Rule 2

A supplier has 1 hydraulic connect point (the FROM).

Note that the positive flow direction at the supplier is from the supplier to the attached H-node.

Rule 3

The number of BOUNDH H-components in the model must be at least one; the number of BOUNDH H-components connected to one connection should be no more than one.

Rule 4

A hydraulic connect point has only 1 incoming or outgoing connection line

A good hydraulic model must be valid (satisfying the four rules above), consistent with the actual piping system and readable. The readability helps to check the consistency

of a hydraulic model. For complex systems, however, symbols and straight lines are readily entangled, which makes the model confusing.

As a final remark, it should be noted that building a hydraulic model is by no means a simple "translation" of a real piping system into a computer model following the above-mentioned rules. The rules introduced in this section serve only as a guide for building such a model. Practical skills and engineering judgement are necessary when making a hydraulic model. For example, many details of a real system are actually unimportant for the flow process simulated and can therefore be omitted in the model. This not only saves labour and computer time, but is sometimes also necessary when the system is large and the computer capacities are limited.

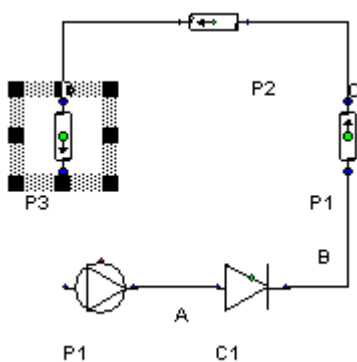
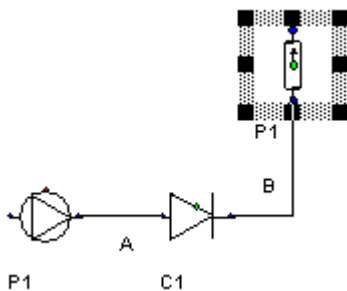
An experienced user of WANDA, combined with adequate practical skills and engineering judgement, will find in WANDA a most useful and powerful tool for solving various problems related to transient flow in complex network systems.

2.3.6 Advanced drawing a scheme

It is not necessary to place the shape one by one in the diagram and after that to make a connection between them. You can do it in one simple action.

Select the required component from the shape gallery and move the mouse to the existing component. The mouse cursor changes into the connect icon. Now, **drag** the mouse to the required location and release the button. The program will determine the direction of the new component based on the position of the connected component.

The scheme below has been drawn using the **drag** and **drop** feature, and not using flip or rotate.

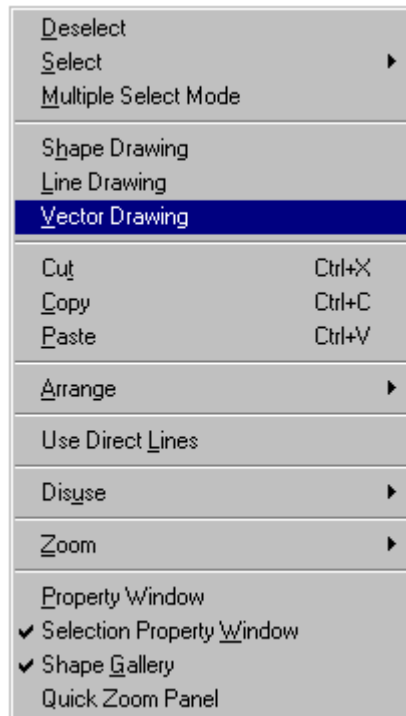


2.3.7 Vector based schematisation

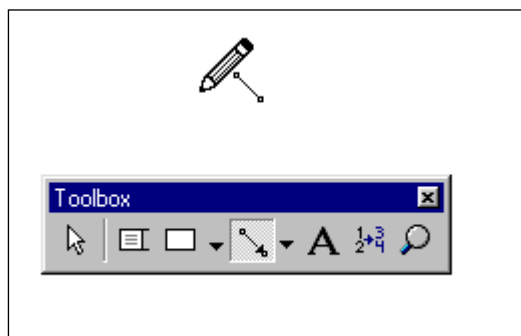
The default way to create your scheme is point based: the component is placed in the diagram as defined in the gallery. For GIS oriented diagrams you have to rotate this component. This may be a lot of work.

There is a way to draw the diagram vector based.

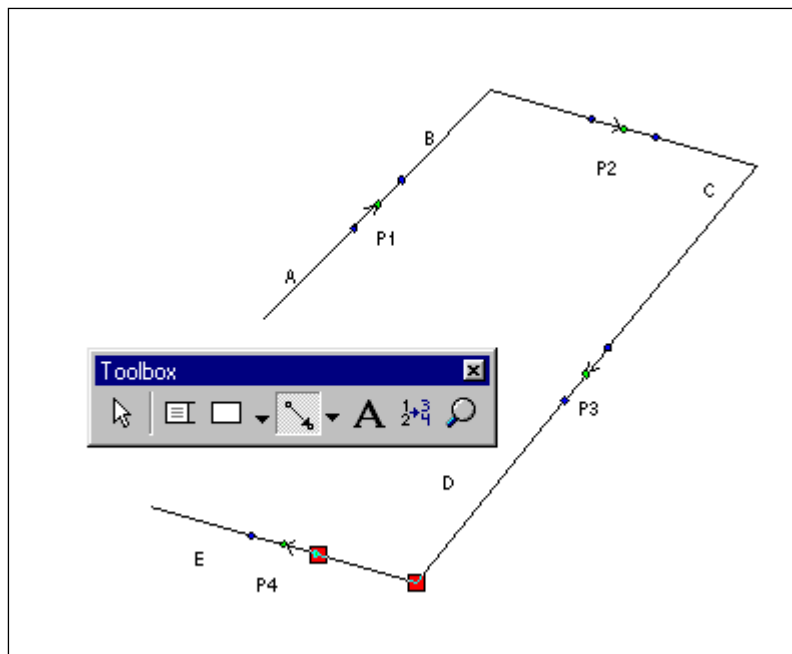
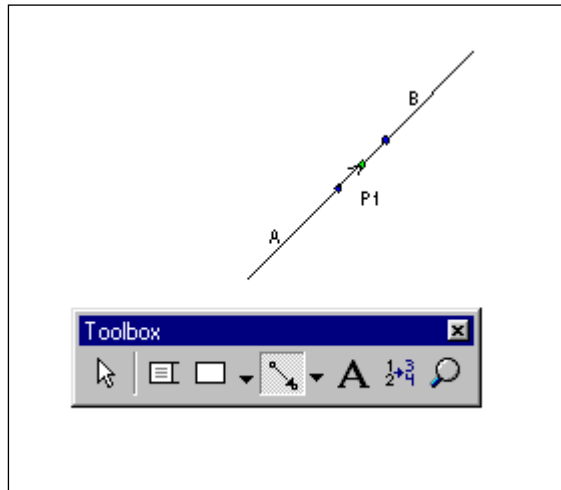
First select the component at the shape library you want to draw as vector. Switch on to Vector drawing mode, using the short cut menu.



The mouse cursor is changed to a pencil with a line symbol.



Now you can draw the line. The component selected in the shape library is now automatically inserted in the middle of the new line. The component is automatically rotated to the direction of the line. Click in the shape gallery to select another component. Switch of the vector drawing mode when you are finished.



PS: If the component is a supplier it is attached to a free end of the new line.

2.3.8 Aligning routes

WANDA 4 offers the possibility to align certain routes or parts of it. This can be useful, e.g. to match the diagram with the actual geographical situation.

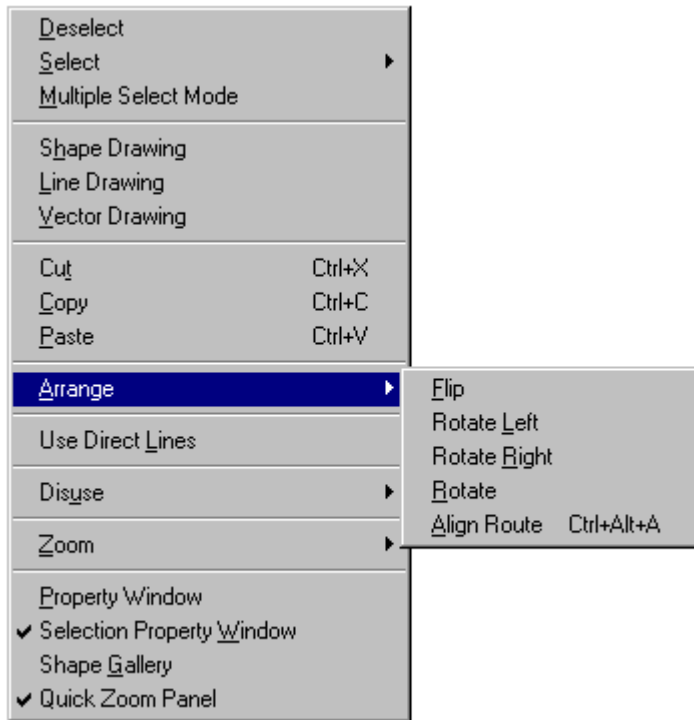
See for further detail the "Arrange Menu" on page 83.

2.3.9 Changing the flow definition

The direction of the positive flow is depicted in the H-component symbol. The sign of the calculated quantity corresponds with this direction.

If you want to change the positive flow definition, you have to "flip" or rotate the symbol.

Use the menu option Arrange/Flip positive flow direction” to reverse the symbol or the short cut menu.

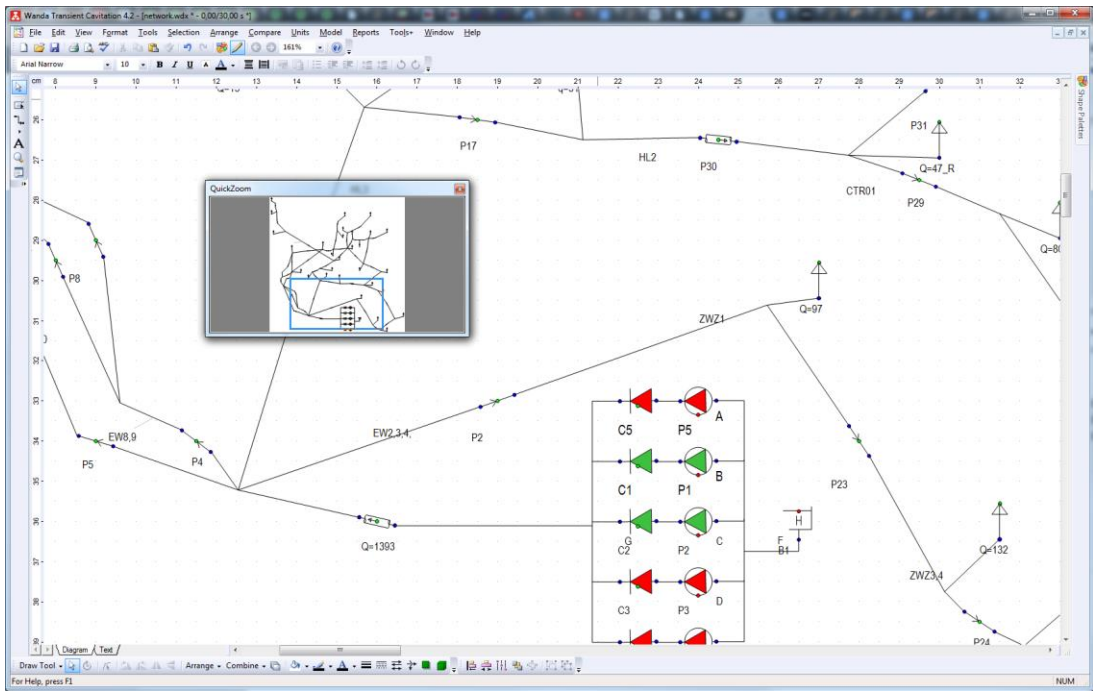


See for further detail the "Arrange Menu" on page 83.

2.3.10 Navigating large diagrams

If the diagram contains a lot of components, it will be quite difficult to fit them in one screen in a readable way. If you increase the zoom factor you have to scroll using the horizontal and vertical scroll bars. You lose the overview about the screen.

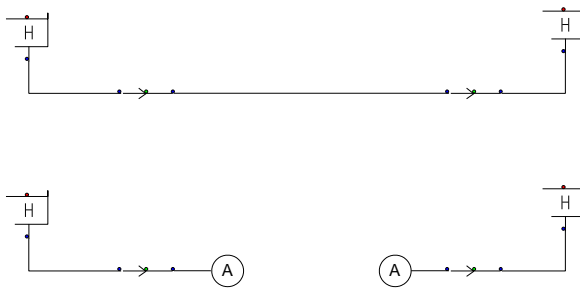
This can be solved using the quick zoom panel. Choose “Quick Zoom” from the View menu.



A New dialog will appear. The whole system is displayed in the new Quick Zoom panel. Select in this part a component and the focus of the main window is automatically re-positioned. The zoom factor of the main window can still be adjusted.

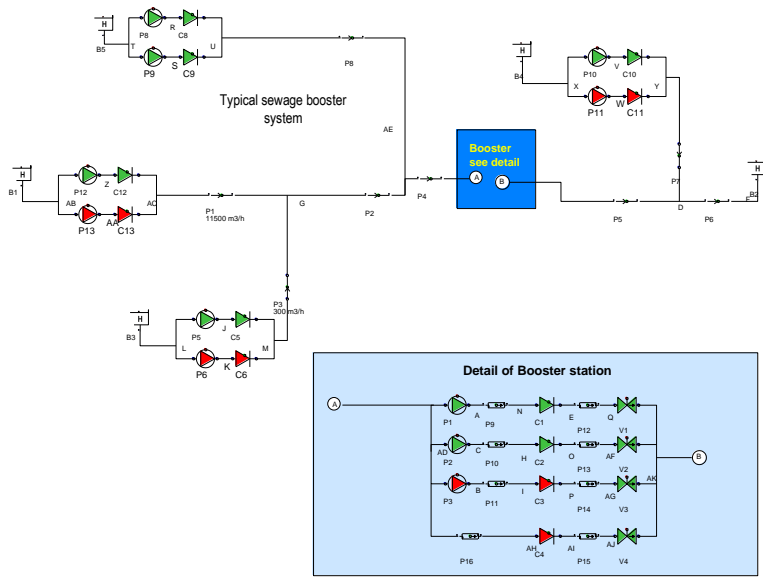
2.3.11 Splitting a diagram

To enhance readability of a complex diagram one can use sub-diagrams. A connection line can be split in two using connectors as indicated in the figure below.



To split the selected master connection line go to “Line and Border” in the Format menu. In the “Format Line” windows select the tab “Arrows and Crossovers”. In the corner left below select the checkbox **Connectors**. The line is split using connectors with a ‘gap’ in between.

Slave connection lines formerly connecting to the split master line should probably be relocated as they now appear to float freely in the gap between the connectors.



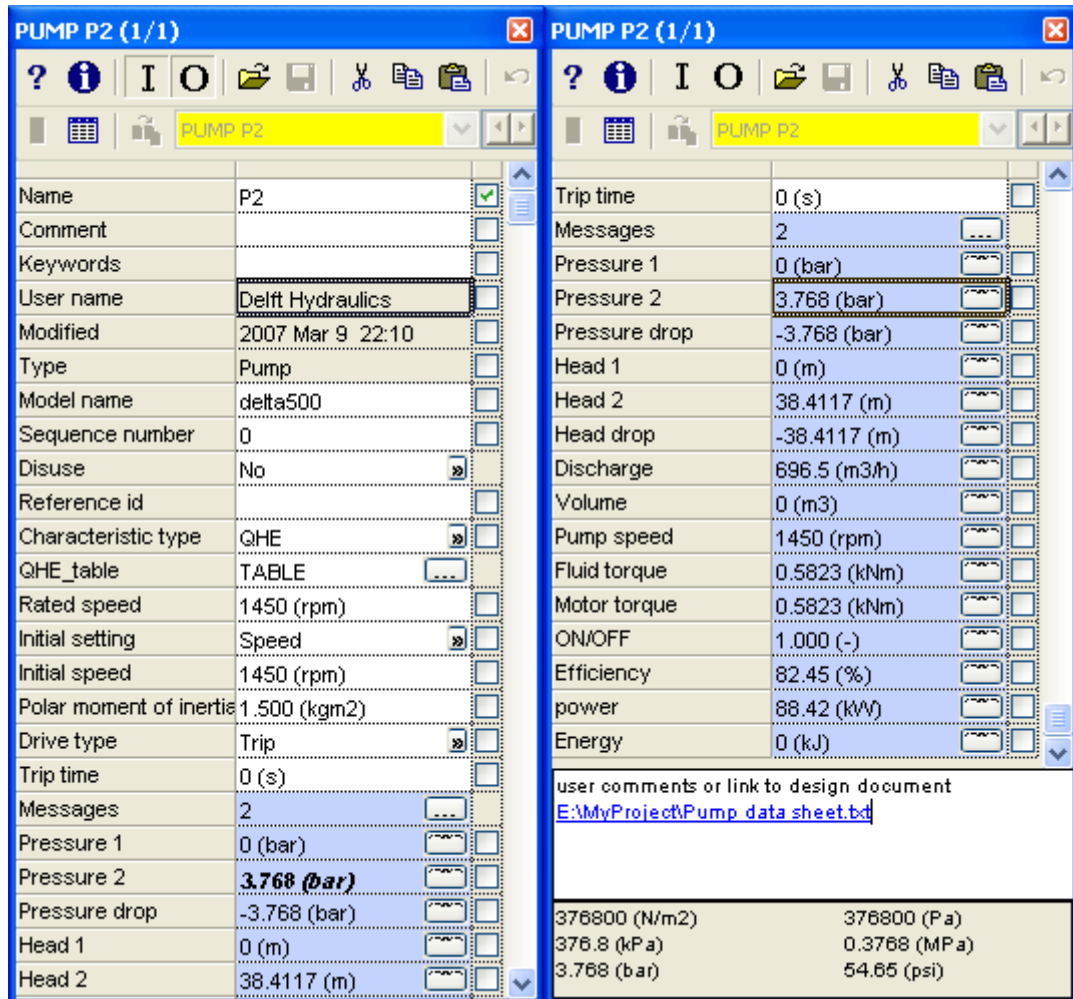
2.4 Property Window

2.4.1 Properties of a Component

A hydraulic component has several kinds of input and output properties (see Help on "Liquid components" on page 187). For editing the input properties and viewing the output properties we use the so-called property window.

The Property window applies to individual and groups of selected components. See also: "How to use Property Windows" on page 46.

You see an example of all input and output values of PUMP P1 below.



The property window has his own context sensitive short cut menu which gives you access to common used acts.

Select	Space
Deselect <u>A</u> ll	Ctrl+A
<u>C</u> ut	Ctrl+X
<u>C</u> opy	Ctrl+C
<u>P</u> aste	Ctrl+V
<u>E</u> dit	F2
<u>O</u> pen Spreadsheet	Ctrl+Shift+S
<u>C</u> opy to Diagram Selection	Ctrl+Enter
Extreme Values	
Extreme <u>L</u> ocations	
✓ Locked	Ctrl+L

2.4.2 Visibility of properties

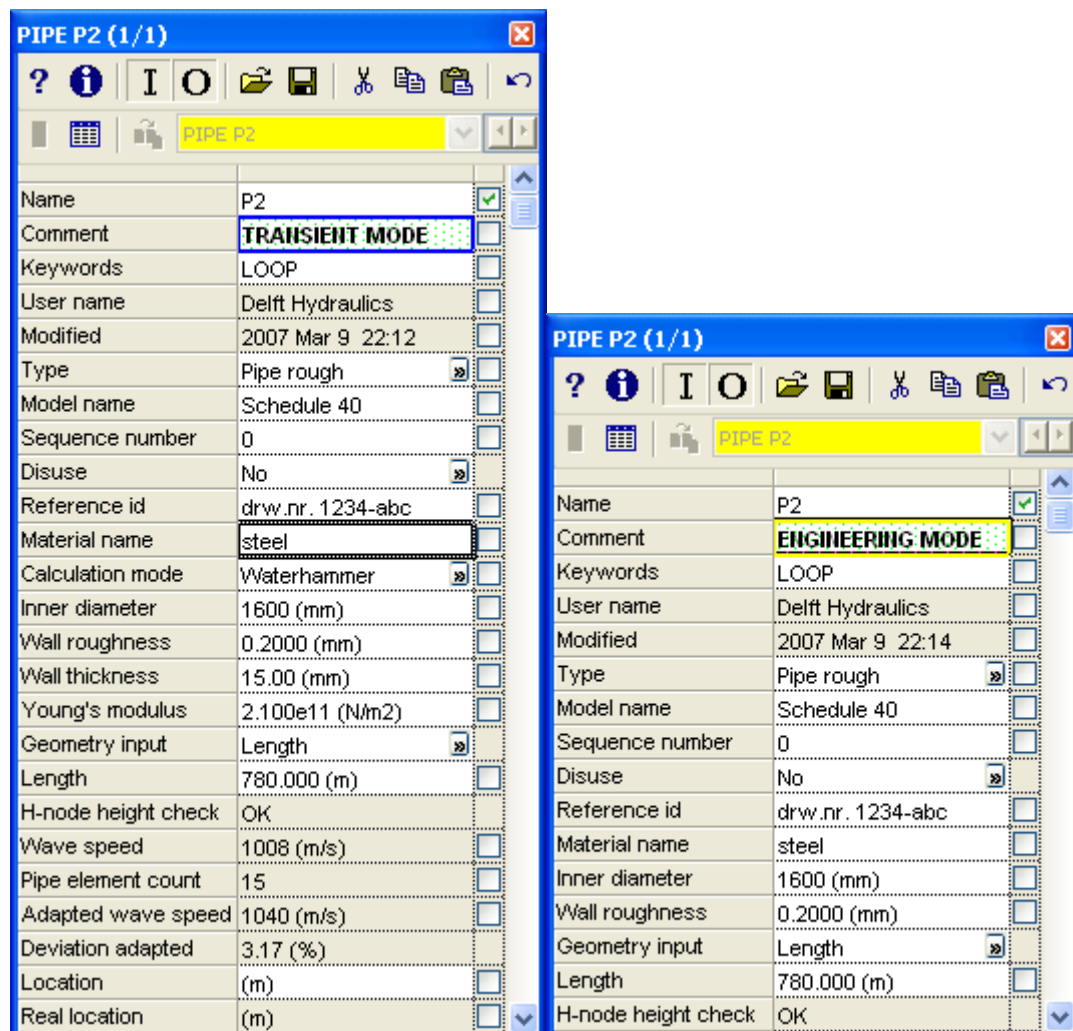
The content of the property window is for each component different.

See also

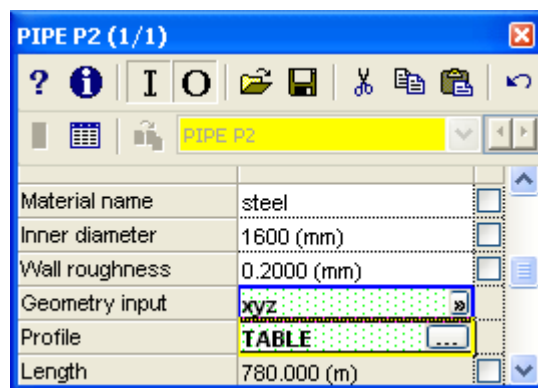
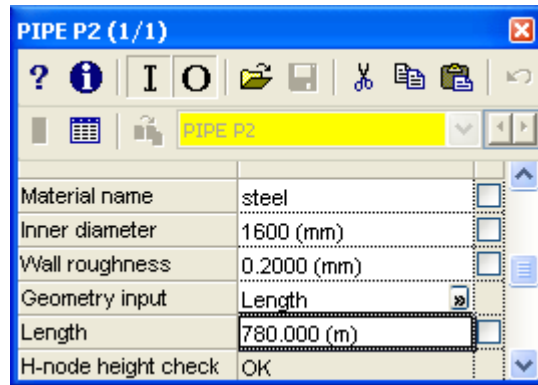
- "Liquid components" on page 187,
- "Control Components" on page 475.

The content of the Property Window depends on the operation mode or input values of other properties. An example of mode dependent properties is the component PIPE. In Engineering mode only the length, diameter and wall roughness are required and must be specified; other properties are hidden. In Transient mode the Young's modulus and wall thickness must be specified in addition. These dependencies reduce the user input to the required minimum number of properties.

The following examples show the Pipe input properties for engineering mode and transient mode:



The contents of the Property Window may also depend on the value of a particular property. As an example the geometry must be specified for the Pipe. The geometry may be specified in several ways: by means of the length, a longitudinal profile or a 3D-geometry. The type of geometry is specified using a list box. The appearance of the Property Window depends on the choice in the listbox 'Geometry input'. See examples below.



2.4.3 How to use Property Windows

The Property Window is one of the most important features in WANDA 4. With this feature it is very easy to view the characteristics of one or more components.

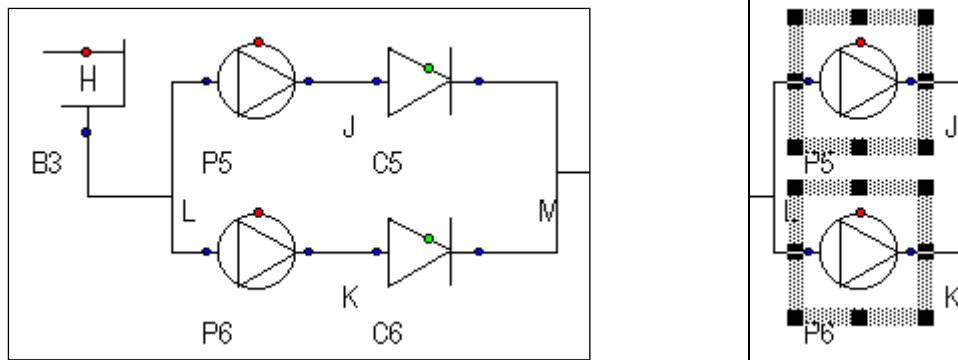
It enables the user to view these characteristics

- individually or
- in relation to other selected components.

When you select components in your model, its properties and values are automatically displayed in the Property Window. Therefore it is recommended to keep it always open. In this way you can see model, components and properties at a glance.

2.4.4 Example

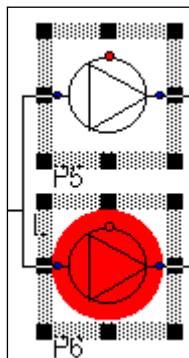
2.4.4.1 Individual components



If you select pumps P5 and P6 in the model then the property values for P5 and P6 (individual) are listed in the Property Window:

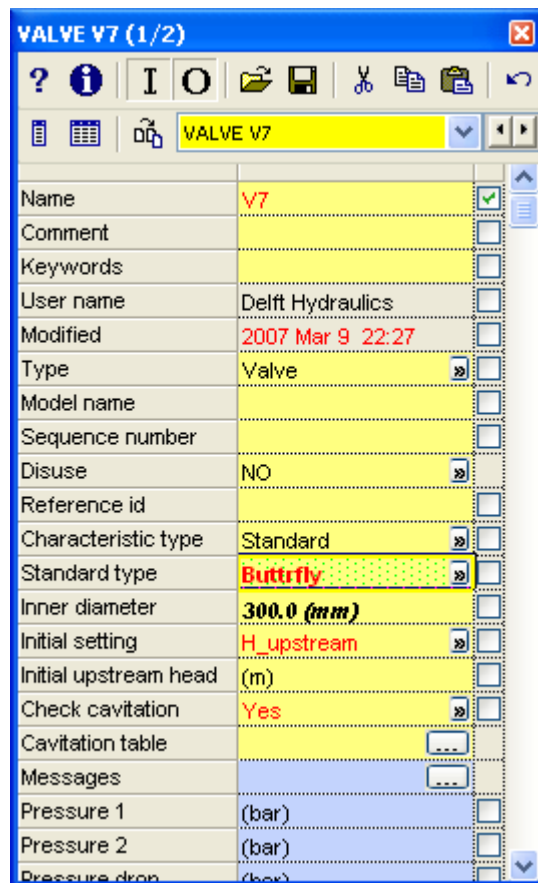
PUMP P5 (1/2)		PUMP P6 (2/2)	
Name	P5	Name	P6
Comment	[]	Comment	stand-by
Keywords	"PS-2 to plant"	Keywords	[]
User name	Delft Hydraulics	User name	Delft Hydraulics
Modified	2007 Mar 9 16:06	Modified	2007 Mar 9 16:06
Type	Pump	Type	Pump
Model name	DMD#250	Model name	DMD#250
Sequence number	0	Sequence number	0
Disuse	No	Disuse	No
Reference id		Reference id	
Characteristic type	QHE	Characteristic type	QHE
QHE_table	TABLE	QHE_table	TABLE
Rated speed	735.0 (rpm)	Rated speed	735.0 (rpm)
Initial setting	Speed	Initial setting	Speed
Initial speed	735.0 (rpm)	Initial speed	0 (rpm)
Polar moment of inertia	9.160 (kgm2)	Polar moment of inertia	9.160 (kgm2)
Drive type	Trip	Drive type	Running
Trip time	2.000 (s)	Messages	1
Pressure 1	-0.01962 (bar)	Pressure 1	-0.01962 (bar)
Pressure 2	1.426 (bar)	Pressure 2	-0.01962 (bar)
Pressure drop	-1.446 (bar)	Pressure drop	0 (bar)
Head 1	-1.90000 (m)	Head 1	-1.90000 (m)
Head 2	12.8386 (m)	Head 2	-1.90000 (m)
Head drop	-14.7386 (m)	Head drop	0 (m)
Discharge	330.1 (m3/h)	Discharge	0 (m3/h)
Volume	0 (m3)	Volume	0 (m3)
Pump speed	735.0 (rpm)	Pump speed	0 (rpm)
Fluid torque	0.2878 (kNm)	Fluid torque	0 (kNm)
Motor torque	0.2878 (kNm)	Motor torque	0 (kNm)
ON/OFF	1.000 (-)	ON/OFF	0 (-)
Efficiency	59.85 (%)	Efficiency	78.83 (%)
power	22.15 (kW)	power	0 (kW)
Energy	0 (kWh)	Energy	0 (kWh)

Inputs and Outputs of P5 and P6 are displayed in the two Property Windows above. The caption shows which component is on top. This is indicated in the diagram by a red circle.



In this way you know at which component you are looking.

2.4.4.2 Font Colours



The colours used in the property window have a special meaning.

Values in **black-coloured** text are **identical** for the selected components. The edit field "Type" is the same for both valves for instance.

Values in italic font are linked to extra information. In this example some information is linked to the field "Inner diameter". For more information on the linking of extra information see "Toolbars" below.

Values in **red-coloured** text are **different** for the selected components. Name and Initial setting for V4 and V7 differ from each other.

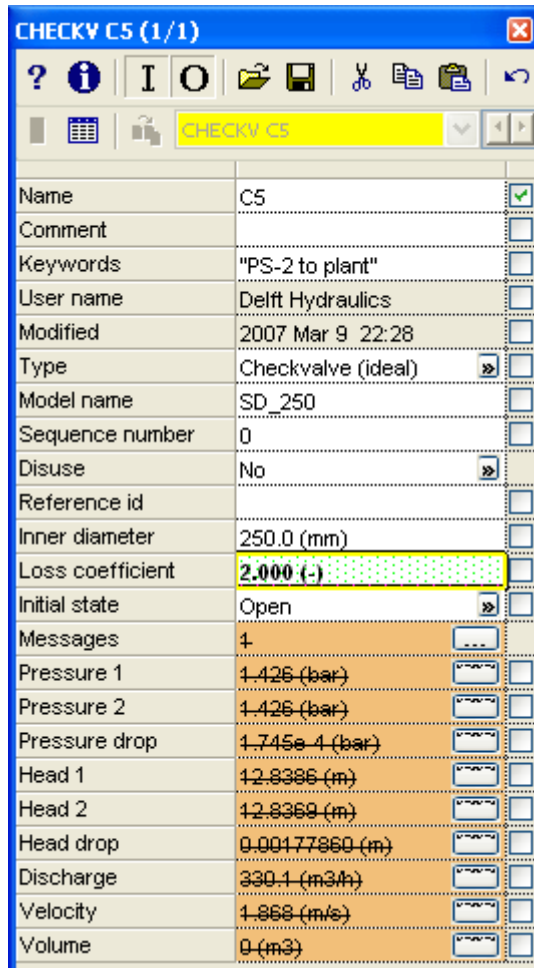
Values in **blue-coloured** text are **selected or edited**. You can (de)select values using the space bar, dragging the mouse in the input area or click in the description area.

Properties with a **yellow background** are **editable** properties.

Properties with a **grey background** are **read-only inputs** and can't be modified.

Properties with a **lilac background** are **read-only outputs** (calculated) and can't be modified.

If the output properties have an **orange background** and a **strike-through font**, this means that one or more properties in the model are changed and the output is no longer valid anymore. The results are still accessible until you close the changed case.



CHECKV C5 (1/1)		
Name	C5	<input checked="" type="checkbox"/>
Comment		<input type="checkbox"/>
Keywords	"PS-2 to plant"	<input type="checkbox"/>
User name	Delft Hydraulics	<input type="checkbox"/>
Modified	2007 Mar 9 22:28	<input type="checkbox"/>
Type	Checkvalve (ideal)	<input type="checkbox"/>
Model name	SD_250	<input type="checkbox"/>
Sequence number	0	<input type="checkbox"/>
Disuse	No	<input type="checkbox"/>
Reference id		<input type="checkbox"/>
Inner diameter	250.0 (mm)	<input type="checkbox"/>
Loss coefficient	2.000 (-)	<input type="checkbox"/>
Initial state	Open	<input type="checkbox"/>
Messages	4	<input type="checkbox"/>
Pressure 1	1.426 (bar)	<input type="checkbox"/>
Pressure 2	1.426 (bar)	<input type="checkbox"/>
Pressure drop	1.745e-4 (bar)	<input type="checkbox"/>
Head 1	12.8386 (m)	<input type="checkbox"/>
Head 2	12.8369 (m)	<input type="checkbox"/>
Head drop	0.00177860 (m)	<input type="checkbox"/>
Discharge	330.1 (m3/s)	<input type="checkbox"/>
Velocity	1.868 (m/s)	<input type="checkbox"/>
Volume	0 (m3)	<input type="checkbox"/>

2.4.4.3 Toolbar



The buttons in the Toolbar enables you to display the desired information by toggling on or off the buttons and to manage the data in an efficient manner.

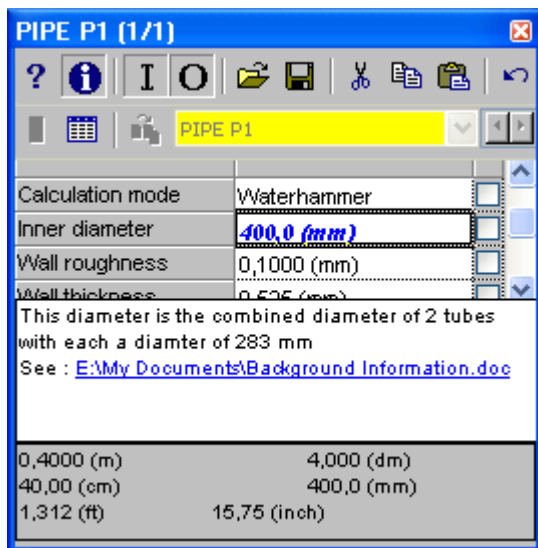
All buttons are explained below:



Toggling on **Help** (or key F1) shows the Wanda Help relevant to the active component. With Shift+F1 the user can add a so-called user-defined help entry in a WordPad file. The question mark gets a yellow background to show that a user defined help entry has been made. In this way the user can extend the standard WANDA Help. It offers the possibility to add tips or tricks related to this component. If the file is saved on a network drive, all other WANDA users in the company can access this information.

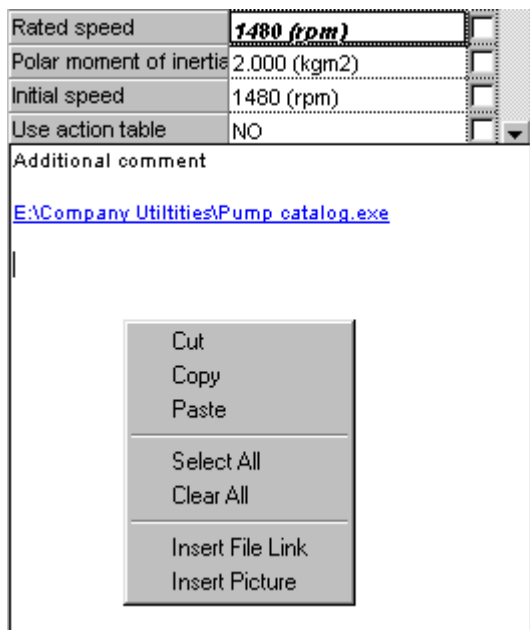
i Information

toggling on Information shows two extra information fields. One field that shows what units can be used for this property (see also "Units Menu" on page 96). In this example it shows the available diameter units:



Furthermore, it shows a field with additional information on the selected property. In this case it shows how the Inner diameter of this pipe is calculated: You can also put links to Word-documents or other files in this extra information window. Even pictures are possible. Properties with additional comments can be recognised by a different font (bold+italic). All additional information is saved in the Wanda wdi-file.

Click the right mouse button, for an additional edit menu, to modify your comment:



This comment feature is very powerful for quality assurance at project level. Now it is possible to add essential information to each component property. This linked

information allows easy access for project reviews, (external) audits, and simple follow-up of projects by colleagues.

Show/Hide Input

This button shows or hides the input properties of the selected component.

Show/Hide Output

This button shows or hides the output properties of the selected component.

Property Template File Management

Open a property template file and apply it to this component, if relevant (see also: "Creating your own database system" on page 77). The retrieved data are displayed in blue.

Save the selected (blue) values to a property template file.

Note: All values are stored in SI units.

Edit

These buttons enable you to:

- Cut the selected (blue) values to the clipboard
- Copy the selected (blue) values to the clipboard
- Paste the content of the clipboard in this window and thus use it for this component, if applicable (see also: "Editing or Synchronising values" on page 58)

Range View

Activating this button shows an overview of the value ranges of all properties of all selected components. This is useful for inspecting the hydraulic model.

Spreadsheet View

Activating this button shows a tabular view of all selected properties of all selected components. This is useful for inspecting and editing into this selection.

Copy / Synchronise

This button enables you to Copy (synchronise) the blue-coloured properties to the selected objects in the diagram. See "Editing or Synchronising values" on page 58.

Checkboxes and buttons

Check boxes on the right of the Property Window are used to display or hide the value in the diagram. Component names are displayed by default. You can add other relevant properties to your diagram by checking the boxes. The displayed info remains correct because of the link between the Property Window and the diagram. The property values are put in a text box in the text layer.

For an explanation of the buttons in the edit and output fields, see:



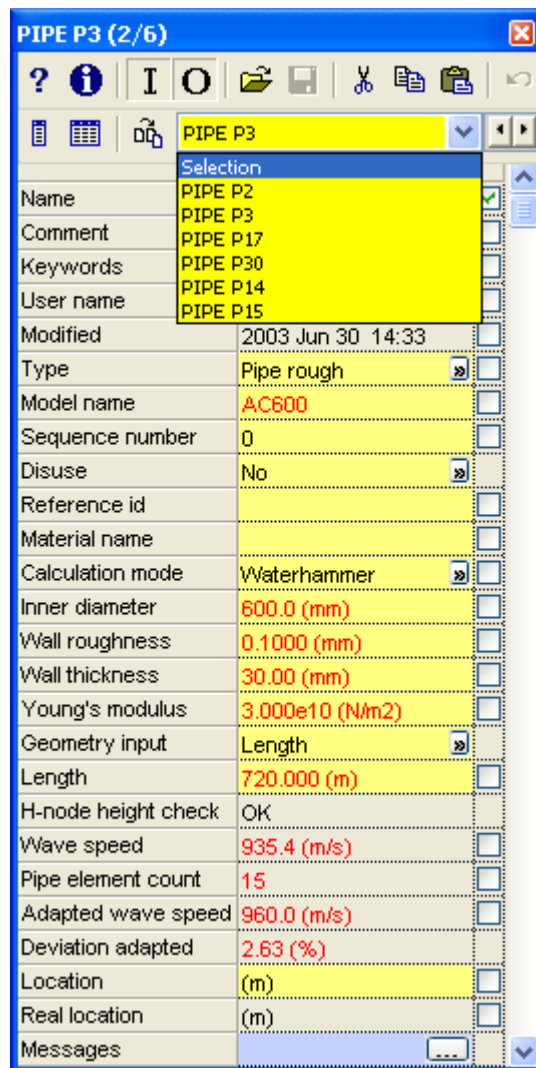
(Use Table): see "Table" on page 59,



(Show Graph): see "How to use charts" on page 64.

2.4.4.4 A group of components

The Property window enables you also to see significant values for a group of selected components, by choosing 'Selection' in the drop down menu below the toolbar:



After choosing 'Selection' you see the properties in the selection of the components (read-only). If values differ the range of values is shown.

Selection (6)	
Name	P2 P3 P17 P30 P14 P15
Comment	
Keywords	LOOP
User name	"WL\Delft Hydraulics"
Modified	[]
Type	"Pipe rough"
Model name	PVC360 AC600 Gij450
Sequence number	
Disuse	No
Reference id	
Material name	
Calculation mode	Waterhammer
Inner diameter	380.0 .. 600.0 (mm)
Wall roughness	0.05000 .. 2.000 (mm)
Wall thickness	4.000 .. 30.00 (mm)
Young's modulus	3.000e9 .. 1.800e11 (N/m ²)
Geometry input	Length
Profile	TABLE
Length	360.000 .. 1450.00 (m)
Total Length	4330.00 (m)
H-node height check	OK
Wave speed	288.8 .. 952.9 (m/s)
Pipe element count	8 .. 100
Total Pipe element count	248
Adapted wave speed	284.6 .. 960.0 (m/s)
Deviation adapted	-5.56 .. 2.63 (%)

In this example only two components of the same type are shown. Therefore the selection of values is rather transparent. The situation is more complex, if you select more components of various types.

2.4.4.5 Spreadsheet

With a spreadsheet you can edit multiple properties of multiple components/nodes.

	Rated speed (rpm)	Polar moment of inertia (kgm2)	Operational speed (rpm)
PUMP P1	580.0	30.00	580.0
PUMP P2	580.0	30.00	580.0
PUMP P3	580.0	30.00	580.0
PUMP P5	735.0	9.160	735.0

You can copy easily copy/paste selections to Excel/Word.

You can create a spreadsheet by pressing the "grid" button at the Property Window.

PUMP P5 (4/4)

Name: Open Spreadsheet (Ctrl+Shift+S)

Comment:

Keywords: "PS-2 to plant"

You get a view with the selected objects in the diagram and the selected properties in the Property window.

	Inner diameter (mm)	Discharge (m3/h)	Velocity (m/s)	Pump speed (rpm)	Efficiency (%)
CHECKV C1	500.0	3769	5.332		
CHECKV C2	500.0	3769	5.332		
H-node D					
H-node E					
H-node F					
PUMP P1		3769		580.0	78.83
PUMP P2		3769		580.0	78.83

To rotate the spreadsheet press the left button; columns and rows will be exchanged.

	CHECKV C1	CHECKV C2	H-node D	H-node E	H-node F	PUMP P1	PUMP P2
Inner diameter (mm)	500.0	500.0					
Discharge (m3/h)	3769	3769				3769	3769
Velocity (m/s)	5.332	5.332					
Pump speed (rpm)						580.0	580.0
Efficiency (%)						78.83	78.83

	PUMP P1	PUMP P5
Print (Ctrl+P)		
Pressure 2.range (kPa)	-44.59 .. 83.42	-3.202 .. 142.6
Head 2.range (m)	-6.346 .. 6.704	-2.026 .. 12.84
Discharge.range (m3/h)	1896 .. 3769	67.18 .. 330.2
Pump speed.range (rpm)	149.4 .. 580.0	91.41 .. 735.0

The spreadsheet has his own context sensitive short cut menu. For example, this menu can be used to sort the spreadsheet on an arbitrary column, the selection sequence or a user-defined sequence.

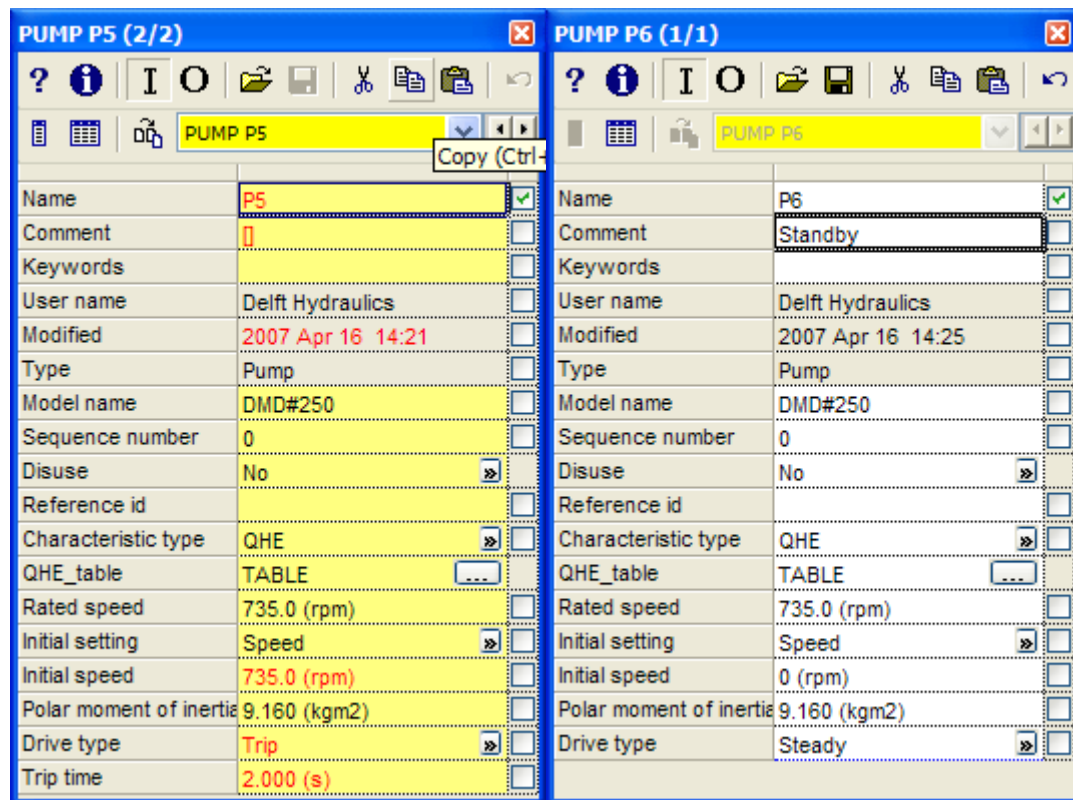
C <u>u</u> t	Ctrl+X
C <u>o</u> py	Ctrl+C
P <u>a</u> ste	Ctrl+V
E <u>d</u> it Cell	
Active Time <u>V</u> alues	
E <u>x</u> treme Values	
Extreme <u>L</u> ocations	
Extreme <u>T</u> imes	
Min and Max <u>S</u> eparated	
Sort by Sequence Number	
Sort by Name	
<input checked="" type="checkbox"/> Sort by Inner diameter	
Refresh Sorting	Ctrl+Shift+R
F <u>r</u> ozen	
Always <u>O</u> n Top	
O <u>p</u> en Location Chart	
O <u>p</u> en Time Chart	
O <u>p</u> en Table	

Output ranges, including the corresponding time and location can be viewed in an easy way. The contents of the spreadsheet can be frozen during the edit and simulation session to compare the results with other calculation made within the same session

The tabular view is automatically updated when input or output is changed. When the selection of objects and properties is changed, an existing tabular view keeps unchanged. You can create multiple spreadsheets with different selections.

With the drop down list near the print button, you can set up and create print reports of the spreadsheet.

2.4.4.6 Editing or Synchronising values

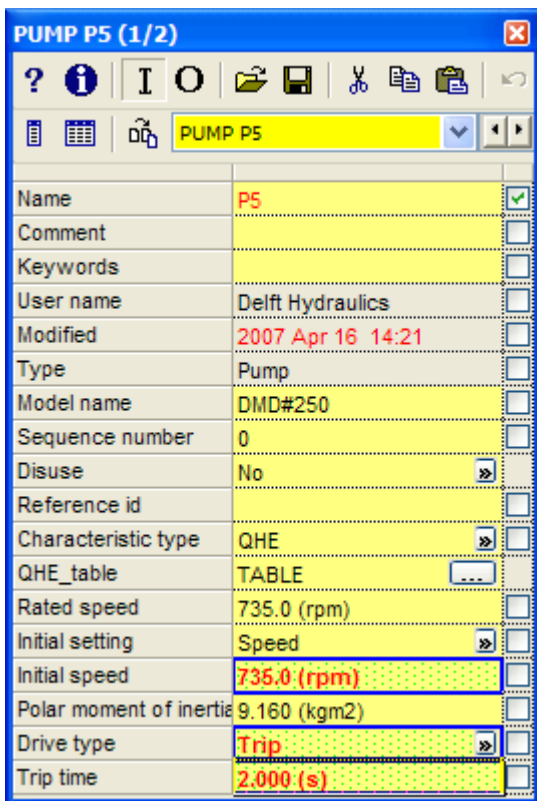



As discussed the Property Window is the feature to have a detailed look at the components in your model. You can see differences between components immediately, indicated by red text.

Sometimes these differences are unintended. The Synchronise (or copy) feature allows you to solve this problem.

In the property window above you see the properties of pump P5, in a selection with another pump P6. You can see that the (red-coloured) values for the properties differ from P6.


To synchronise one or more altered values for both pumps, you only have to select the values you want to synchronise. To select, drag the mouse or press the space button. Bold text together with a dithered background and a blue outline, indicates which properties have been selected.



When you click  (or shortcut key Ctrl+Enter) the selected values are copied to pump P6.

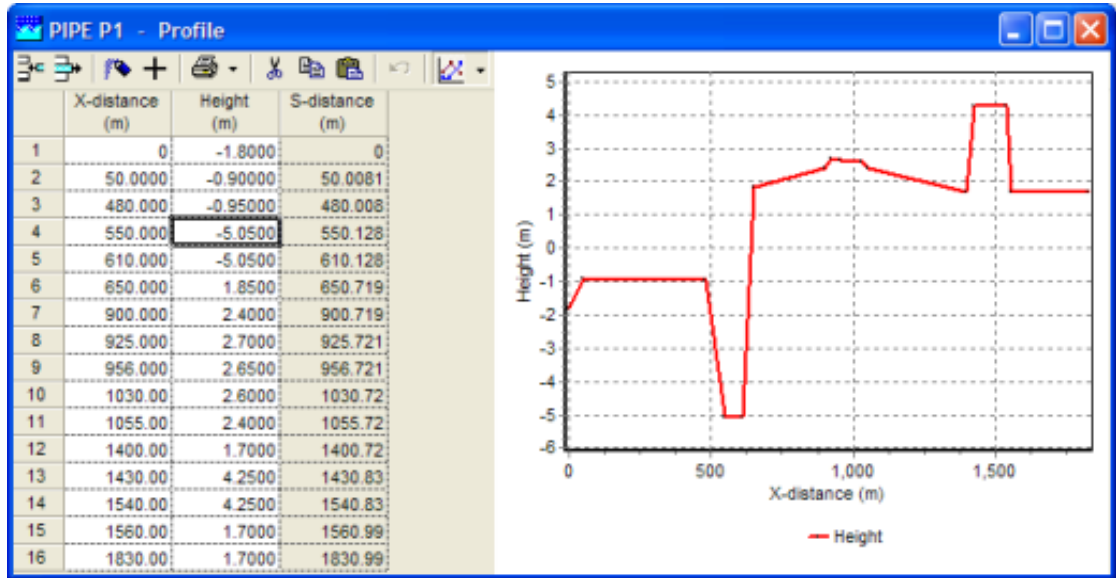
Note that the output has disappeared from the Property Window after applying Synchronise. The output has to be recalculated, due to a change in input parameters.

2.4.4.7 Tables

Pressing  opens a table. Some properties, like pump characteristics, pipe profile or action table, are non-scalars but multi-column data series.

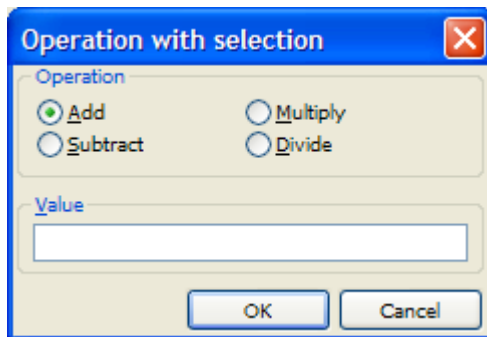
The table displays 2 or more input columns. In case of a pump, a read-only column shows you a derived quantity. The edit grid acts as a spreadsheet.

Some type of tables must satisfies certain rules, e.g. a pump capacity curve (QH) is monotonously decreasing; values in first column must be in successive order. If the input doesn't satisfy the rule the colour becomes red.



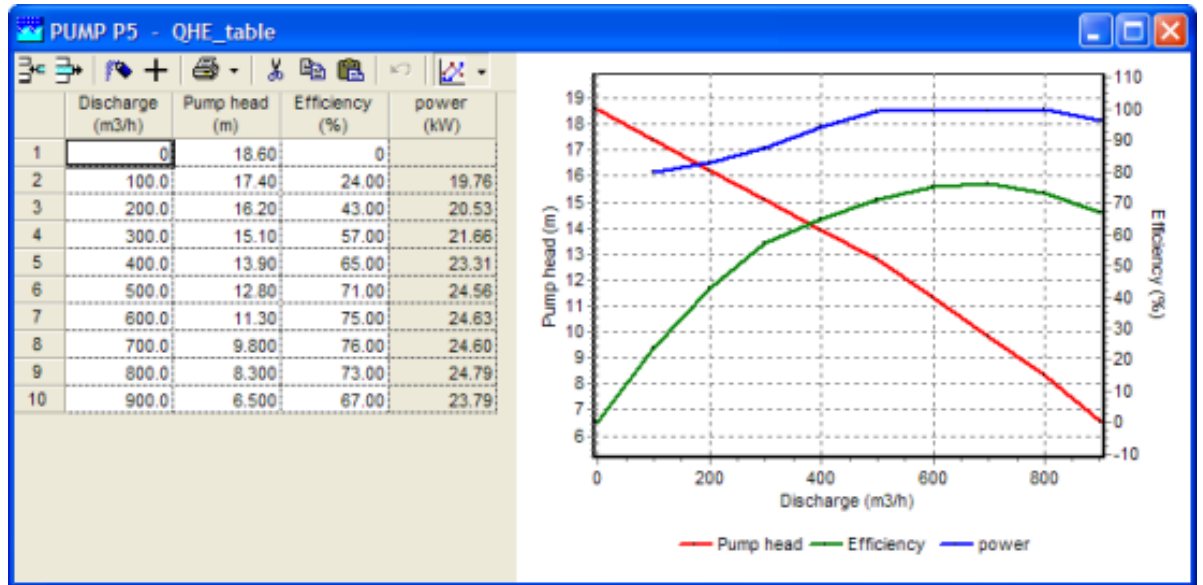
Taskbar

The first two toolbar buttons above, allow you to insert or delete rows in the action table
 The 'plus' button activates a new window to carry out some simple numerical operations at the selected cells.

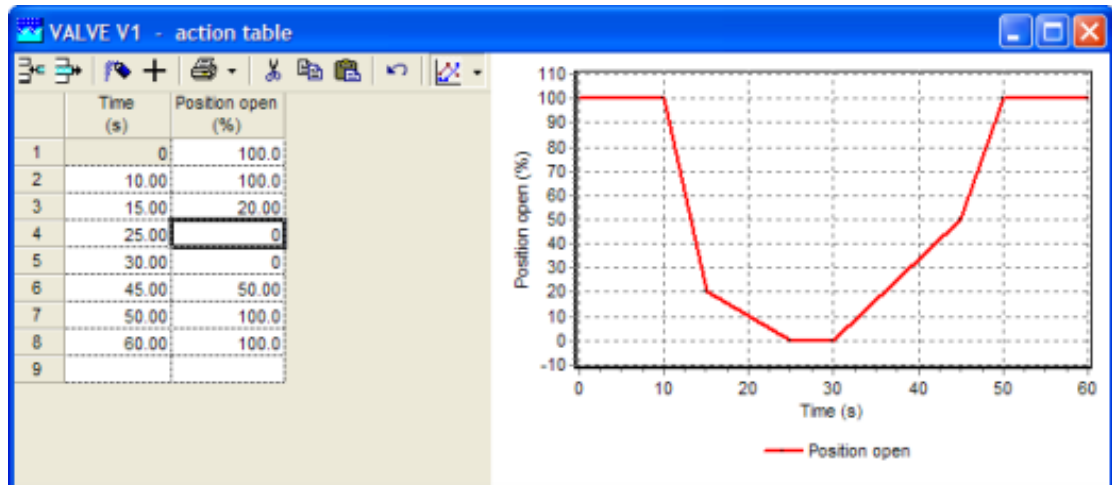


'Auto fill' can be used to fill empty cells between two values, using linear interpolation.
 The button on the right displays a graph of the table.
 See: "How to use charts" on page 64.

Example of pump characteristic table:

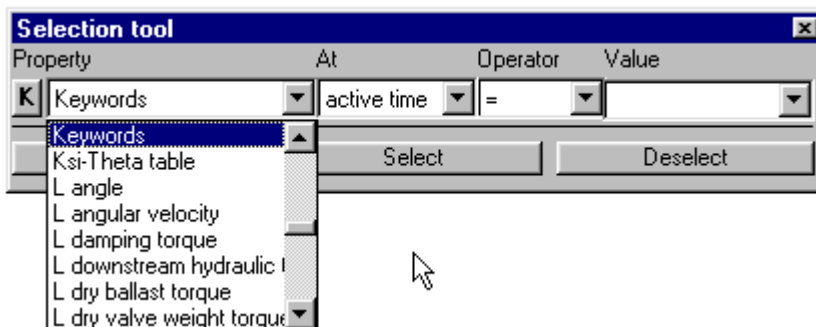


Example of Valve action table



2.5 Selection Builder

2.5.1 How to use the Selection Builder



In WANDA 4 it is very easy to select single (or groups of) elements. A tool that is very helpful is the Selection Builder (accessed via menu Selection/Selection Builder Window) With this tool you find elements, selected on:

- Properties (input and output properties)
- At a given time step or during all calculated time steps
- Applying a certain operator (=, <, >, <>, like)
- With a given value for the properties.

In this way you can select elements by keywords, inputs and/or outputs. Keywords can apply to any selection (see "Add to/ Remove keyword from selection..." on page 88). You can even build complex queries by applying the Selection Builder several times in succession and by choosing Select or Deselect.

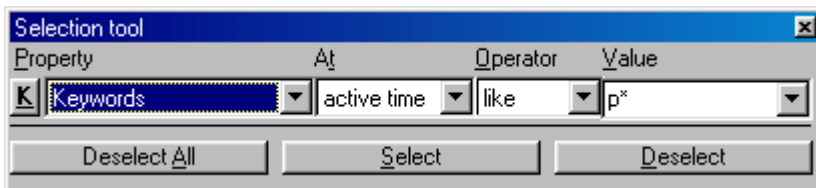
The first 4 operators speak for themselves. The 'like' operator is a powerful string-matching operator. The edit field value contains the pattern that is to be matched. This pattern may be a normal keyword, component name etc. Furthermore the pattern may contain one or more of the following special characters:

Character	Matches
?	Any single character
*	Zero or more characters
#	Single digit (0-9)
[charlist]	Any single character in charlist
[!charlist]	Any single character not in charlist

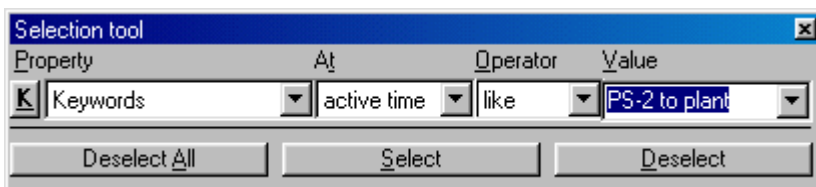
The set of characters in *charlist*, may contain a hyphen ('-') to specify a range of characters, e.g. [a-k].

Examples

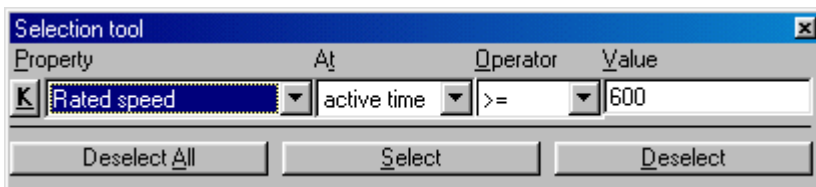
In the screen below we are looking for elements in the model that start with a 'p' for the property keyword. (Note that you can use wildcards.)



If you want to search on keywords, a dropdown list of available keywords is displayed in the edit value box. We have selected the keyword PS-2 to plant, at the active time, in the example below.



If you want to select all components that have a rated speed of 600 (rpm) or more, you could apply the following selection query. In the diagram, two pumps are now selected.



If you click the 'K' button, you will go back to searching by keyword.

Tip

Pressing Alt +F7 activates the Selection Builder.

If you press Alt +F7 while a property is selected in the Property Window, this property will be entered in the Selection Builder.

2.6 Using Charts


2.6.1 Kinds of charts

In WANDA we distinguish three kinds of charts

- A **chart** of an input table: (pump curve, pipe profile, action table etc). The quantity and unit of the X-axis and Y-axis depends on the kind of table.
- **Time series**: This chart contains one or more quantities as a function of time ($H(t)$, $Q(t)$, $v(t)$, etc.). See also "Time series" on page 65.
- **Location series**: Output properties of a pipe can be visualised in a chart as a function of the location ($P(x)$, $H(x)$, $Q(x)$, etc). See also "Location series for a single pipe" on page 66.

Each chart type uses its own default chart template. In these templates the layout of the chart is defined. The user may adapt the templates to personal preferences. More information on managing these templates, see Help function of Deltares Graph Server.

2.6.2 How to use charts

If you click the chart button  in the Property Window (see "How to use Property Windows" on page 46), a chart of the corresponding output property is displayed. The graph button is part of the output property boxes.

The most important features of charts in WANDA 4 are:

- **Zoom** in on the graph, by drawing a rectangle from left-up to right-down.
- **Scroll** through a zoomed graph by holding the right mouse button.
- Change the **layout** and **presentation** of the graph by double clicking on the graph or through File menu 'graph properties'.
- **Save graph settings** in a template, for later use, using the template menu (see also Help Chart server).
- **Export charts** as a Bitmap, Windows Metafile or other formats for use in other applications or documents.
- **Copy** graph data for use in other applications (tab-delimited data, e.g. spreadsheets).

2.6.3 Chart with more series

Each time you click the chart-button a new window is opened. If you want to add a series to an existing graph, just press and drag the button into an existing graph. The series is added to this graph.

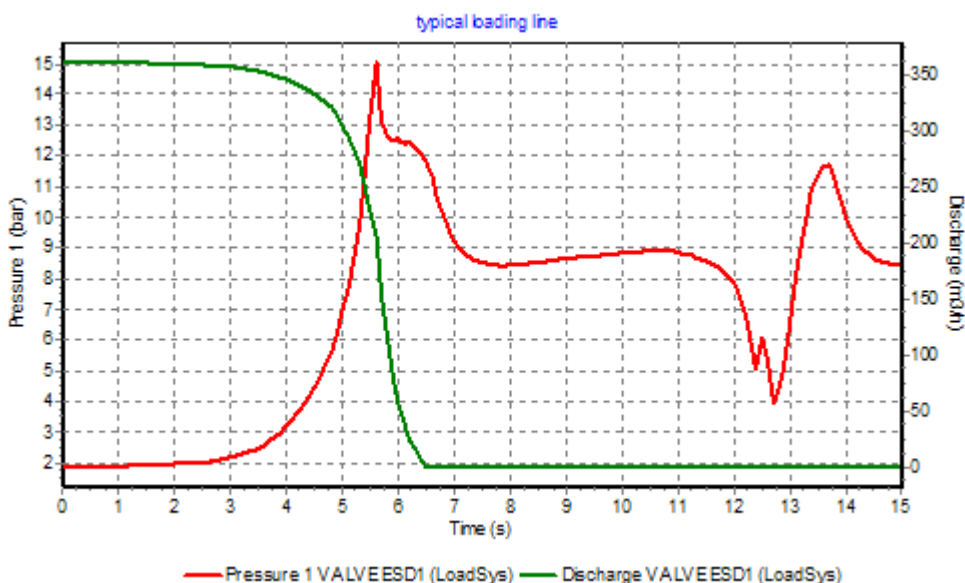
WANDA allows multiple documents; that means that another case can be opened and that output of more than one case can be drawn in one chart to compare the results of different cases.

2.6.4 Time series

Time series are applicable in transient mode only!

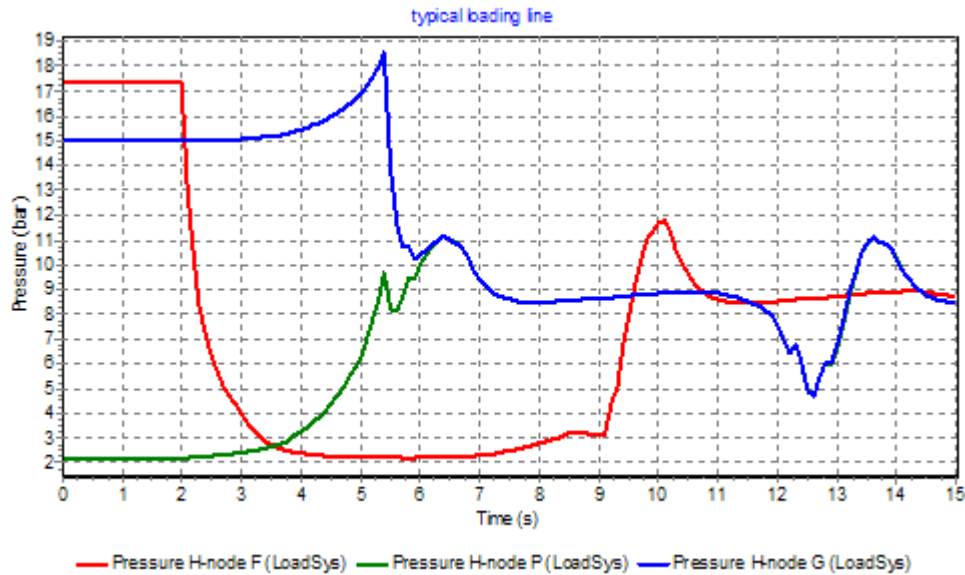
All output properties of WANDA objects can be visualised in a chart. This chart contains one or more quantities as a function of time (H(t), Q(t), v(t), etc.).

To create a chart with two or more different output properties, for example the upstream pressure P1 and the discharge Q of a valve, first press the chart button of Pressure 1, then drag the chart button of Discharge into the Pressure chart. The result is displayed below.



In this way you can also combine different properties of different components.

To create a time series chart with two or more time series of the same output property, first select the objects in the diagram, then press the chart button of one of the selected components. The result is that the output property of all selected components is displayed.

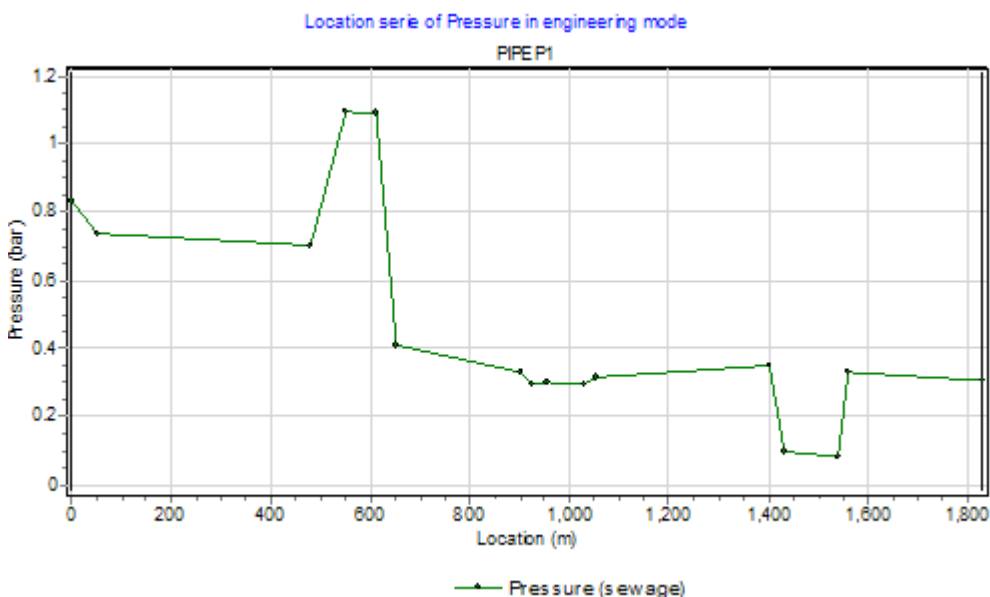
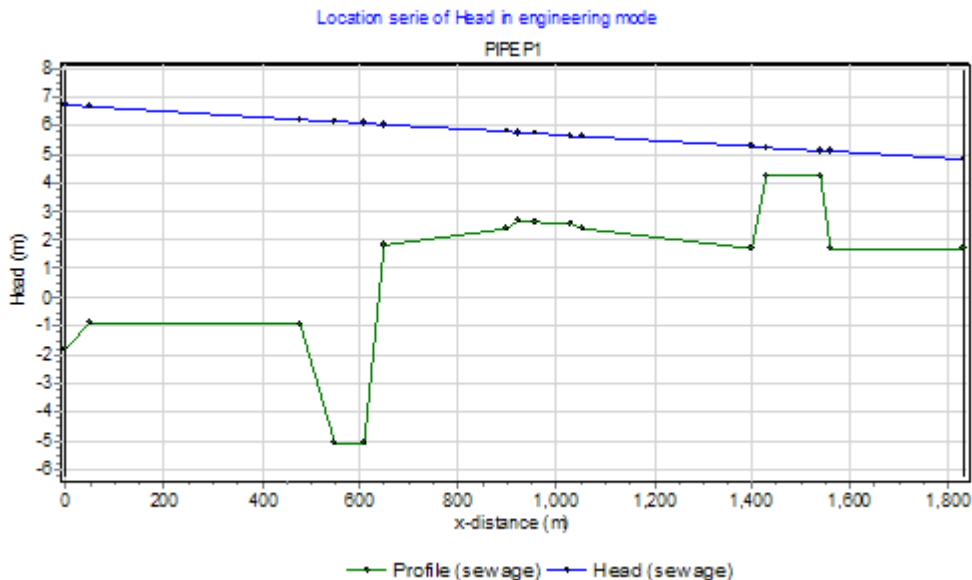


To create a time series at a certain location in a pipe, you have to specify a value in the property field "Location". The program adapts this location to the nearest calculation node, which is shown in the property field "Real location". If the Location field is empty, pressing the chart button results in a location chart of the pipe. See also "Location series for a single pipe" on page 66.

2.6.5 Location series for a single pipe

Output properties of a pipe can be displayed in a chart as a function of the location (X-distance). All routes are displayed with the horizontally projected length on the X-axis. If you create a chart of the Head, also the profile of the pipe is displayed (same quantity).

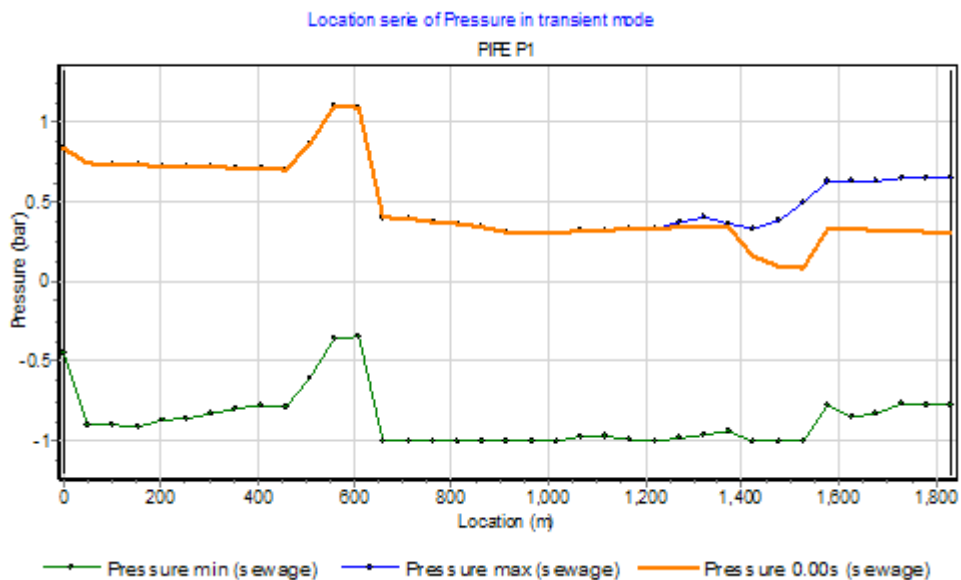
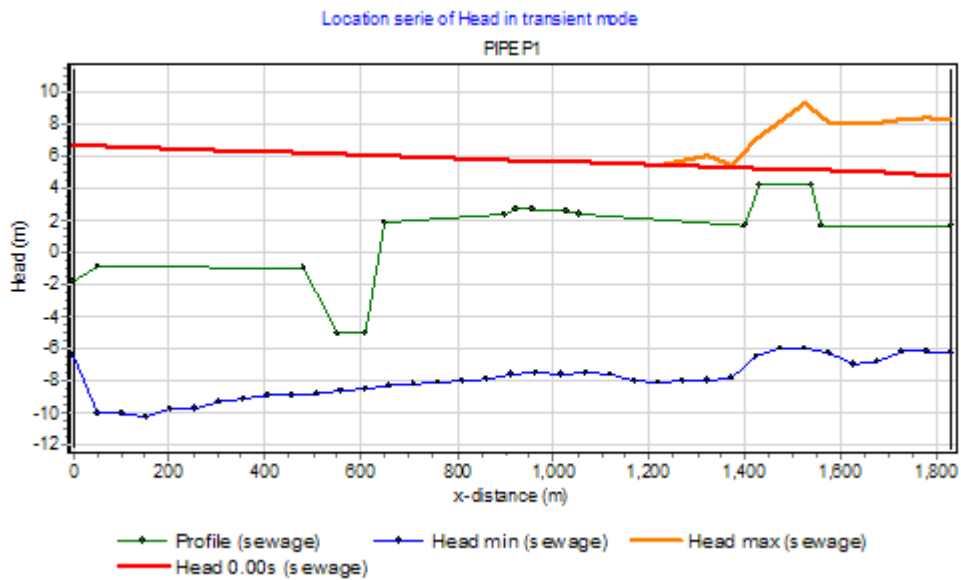
The representation of the location chart depends on the mode you are running. In engineering mode the entered profile points are displayed as internal points (see below):



In transient mode the pipe is divided in one or more elements of equal length (based on the wave propagation speed and time step). The output is available in the internal calculation points only (the so-called waterhammer nodes). These waterhammer nodes do not match the profile points. The waterhammer nodes are displayed on the minimum values. The user is strongly recommended to check that the waterhammer nodes cover all relevant high points; decrease the time step if necessary.

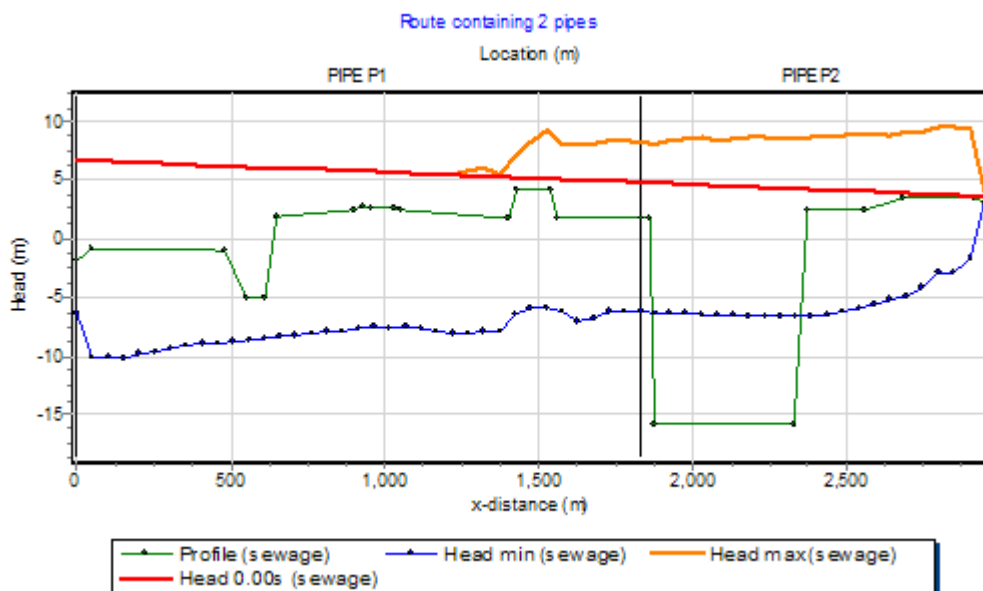
In transient mode the chart shows the minimum and maximum values (the envelope) and the values at the current time step. Use the time navigator to analyse the dynamic behaviour (see "How to use the Time Navigator" on page 75). The actual time is displayed in the caption.

The following charts show the same charts as shown in the previous chapter, but now for transient mode:



2.6.6 Location series for a route

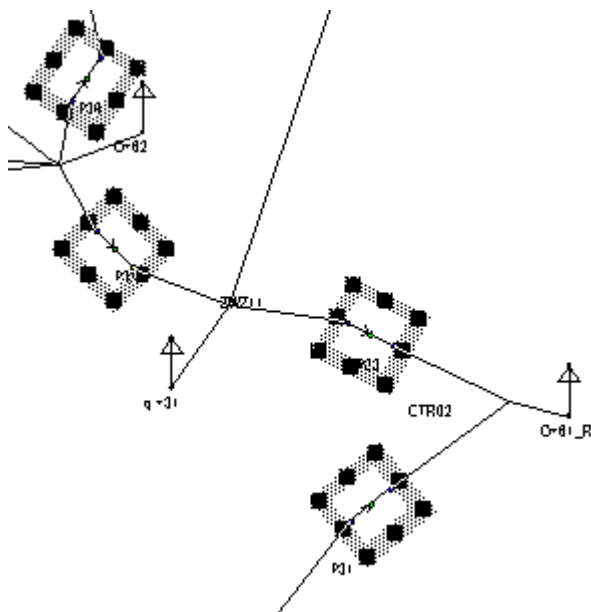
A route is a sequence (string) of selected hydraulic components without a gap. It is not necessary to select the H-components in a route one by one (extend the selection with Shift+Click). The program automatically selects a route by clicking the first component and Ctrl+Shift+Click the last component. For more information of selecting routes see "Diagram routes" on page 135.

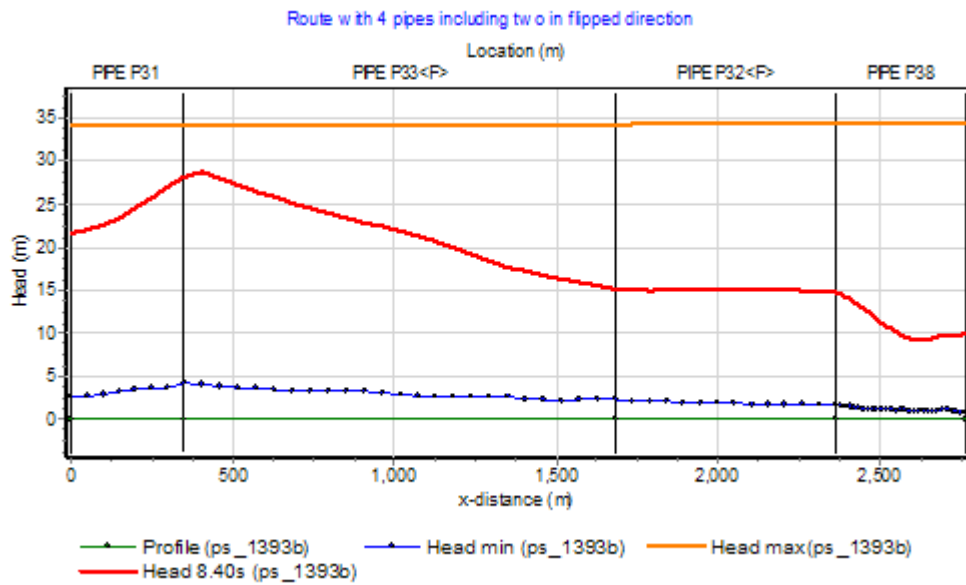


To check if a selection satisfies the route definition, just choose the “Selection” view in the property window (press the A-button). If a chart button is visible at the output properties, the selection defines a route. Only pipe components have a length, other H-components have no length and are not recognised in the chart.

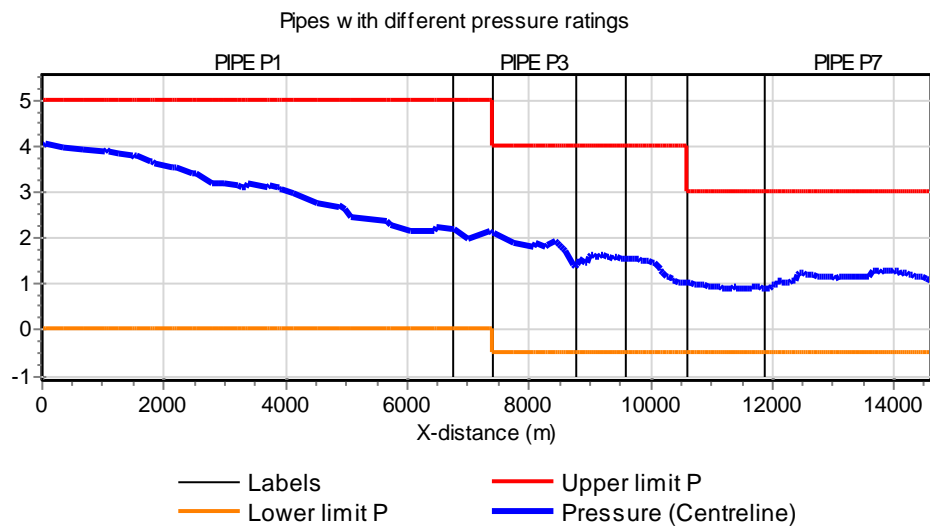
The route function is drawn for the actual time, defined by the Time Navigator (see “How to use the Time Navigator” on page 75). The route creation is the basis for displaying and analysing pressure surge Movies. The actual time is displayed in the caption.

In the chart all pipes are represented in the same flow direction. In a large distribution network it is possible that the positive flow definition is opposite to the current flow. In that case the pipe is represented in flipped position, indicated by a <F>.





It is possible to display the pressure rating of the pipe in the chart. In the Mode & options menu (see "Mode and options" on page 102) a visibility checkbox is available to manage this option. The upper and lower limit values of the pipe are part of the hydraulic input properties of the pipe.



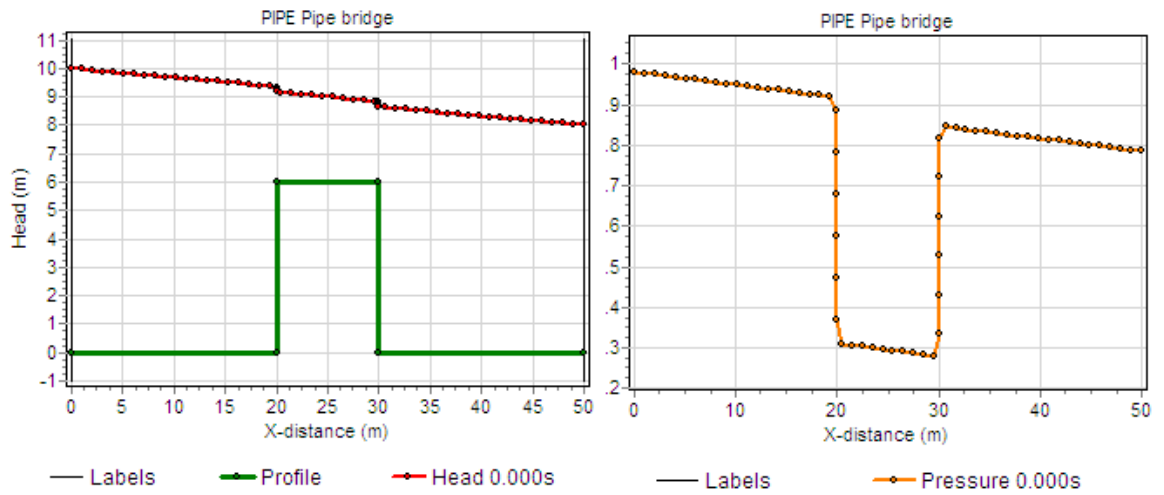
2.6.7 Location axis

For a location series, default the horizontal length is used. The user can switch this horizontal length (X-distance) to the spatial distance (S-distance, real length).

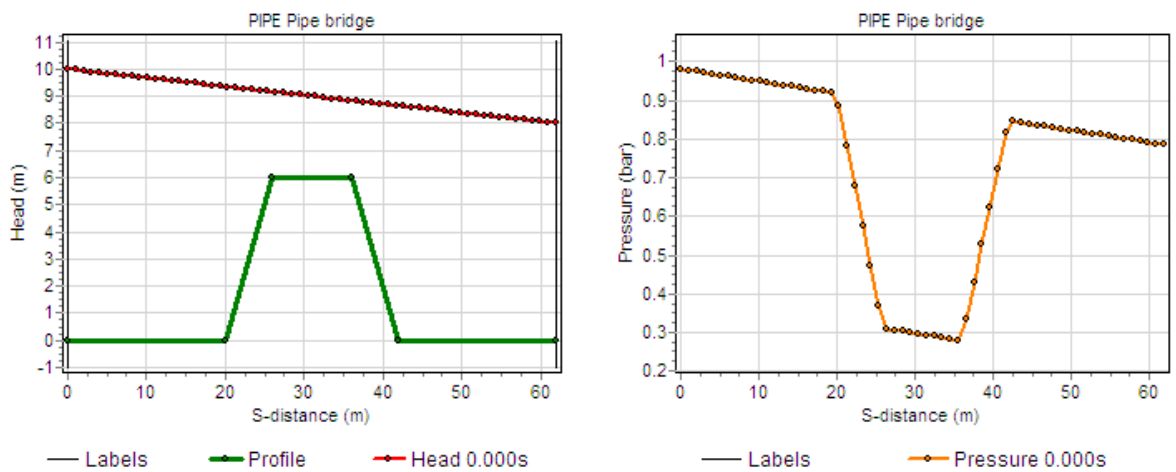
Especially for vertical pipe sections this may be useful. To change this mode, go to the mode & options menu (see "Mode and options" on page 102).

To show the difference between both representations the Pressure Head and Pressure charts are given for both axes in case of a pipe bridge (crossing width 10 m, height 6 m).

Bottom axis as X-distance (horizontal):



Bottom axis as S-distance (Spatial):



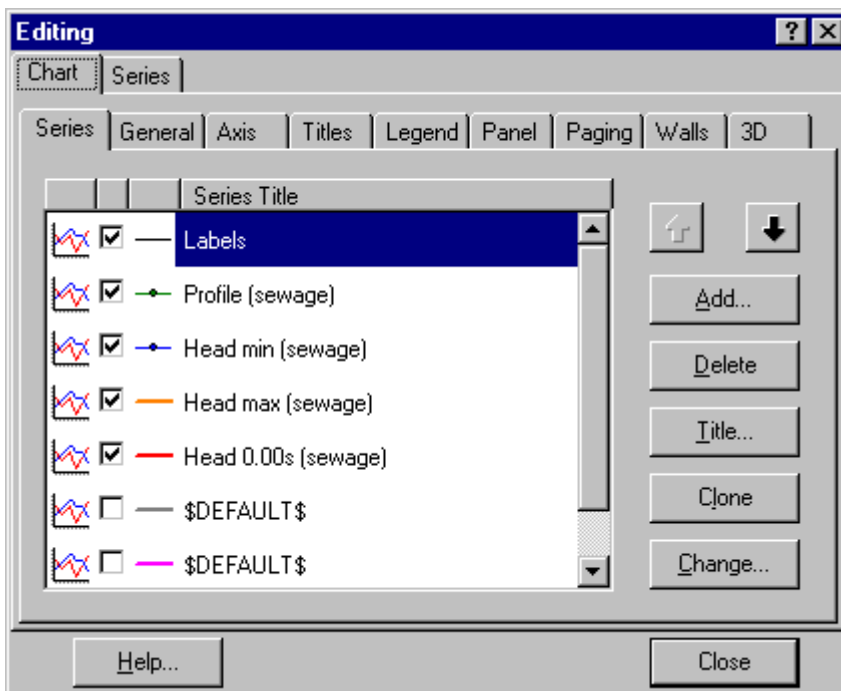
2.6.8 Selection charts

Each time you press the chart button in the property window, a new chart window opens. Without closing these windows your screen becomes full within a few moments. It is possible to open one common chart window for a location and one common chart for a time series. Now, when you select an output value, the series is automatically drawn in the chart window. These so-called selection charts are activated via the context sensitive short cut menu (press right mouse button in the property window).

Select	Space
Deselect <u>A</u> ll	Ctrl+A
<u>C</u> ut	Ctrl+X
<u>C</u> opy	Ctrl+C
<u>P</u> aste	Ctrl+V
<u>E</u> dit	F2
<u>O</u> pen Spreadsheet	Ctrl+Shift+S
<u>C</u> opy to Diagram Selection	Ctrl+Enter
Active Time <u>V</u> alues	
Extre <u>m</u> e Values	
Extre <u>m</u> e <u>L</u> ocations	
Extre <u>m</u> e <u>T</u> imes	
Open <u>L</u> ocation Chart	
Open <u>T</u> ime Chart	Enter
Open <u>L</u> ocation <u>S</u> election Chart	
Open <u>T</u> ime <u>S</u> election Chart	
<input checked="" type="checkbox"/> <u>L</u> ocked	Ctrl+L

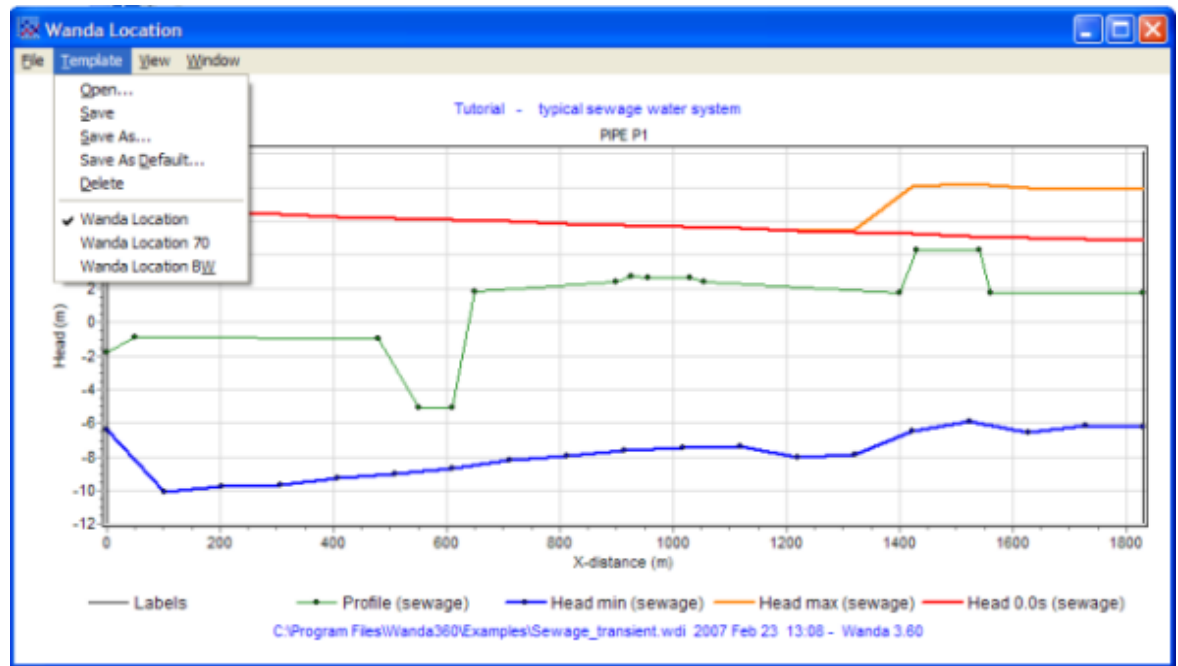
2.6.9 Chart templates

The layout of the several kinds of charts are defined in the so-called chart templates. These templates are part of the WANDA delivery. The chart template contains the chart properties, which describe the layout of the chart. If you are not satisfied about the layout, you may edit the chart properties. The modification can be stored in the default template or in a user defined template.



To change the chart properties, double click in the chart or use the menu “file”/chart properties”. All changes are applied directly. To store these settings you must save it

into a template. Be aware that the axis title, legend and text headers are variable (text depends on case and kind of property). To maintain these variable text items, the template must be saved in design mode. Choose from the Chart menu “Captions/Absolute”. Then save the template via menu “Template/Save” or activate the Template manager.

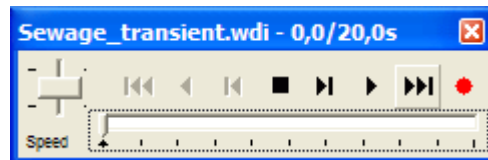


2.7 Time Navigator

2.7.1 How to use the Time Navigator

The Time Navigator allows you to scroll in time through the calculated results of your model, and look at specific locations and instances in your model.

You can only use the Navigator after you have calculated the model in Transient mode and thus generated output. Activate the time navigator via the menu Model/Time Navigator or press Ctrl+T



By moving the horizontal slider you can see the output (e.g. head or pressure) of a selected component at a selected moment in time. The selected time is visible in the caption of the WANDA window and in the caption of the Time Navigator.

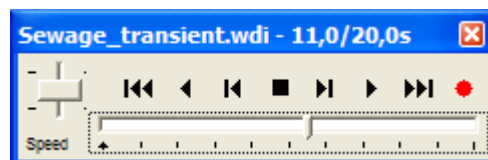
By moving the vertical slider, you can control the speed of the viewed output.

It is also possible to view the output of a component at:

- a given time and
- a given location.

Example

If we want to see the head for PIPE P1 at $t = 11.100s$ and at location 1500m, we first put the Time navigator at that particular point in time:

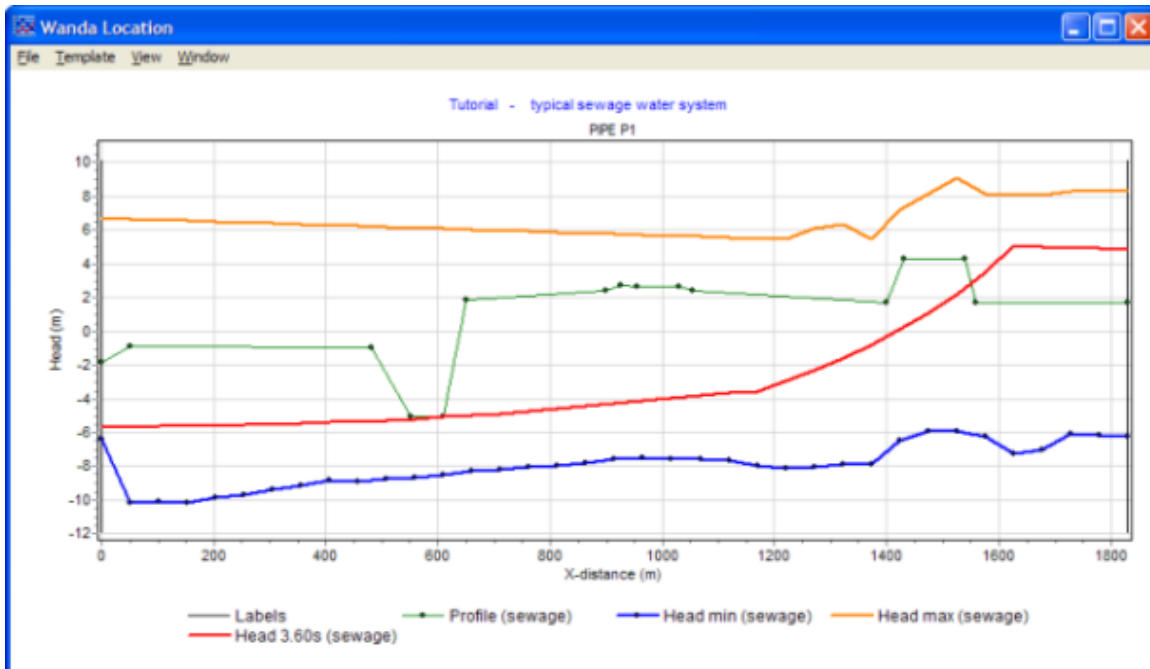


After we specify the location in the Property Window of PIPE P1 (see “Properties of a Component” on page 43), WANDA 4 uses the closest calculation point (1525m) as the real location for PIPE P1, which has a total length of 1831m (see pipe length).

2.7.2 Playing Movies

WANDA allows you to play movies of the output of certain components. The transient pressure in a pipe is a wave phenomenon that propagates through the system.

If you want to see the changes in time of a simulation, we first display the graph for the pressure of a certain pipe, by clicking the graph button. The edit field ‘location’ in the Property Window (see “Properties of a Component” on page 43) must be empty in order to view the route consisting of all selected components; see “How to use charts” on page 64 for more info on the creation of routes.



The blue line is the minimum pressure, orange the maximum and red the pressure at a moment in time, along the total length of the pipe (location). The calculation grid is indicated on the minimum series. If the head along the selected pipelines is displayed, then the pipeline profile is displayed as well. The head graph above reflects the dynamic situation after 3.6 s (indicated in the legend).

By shifting the time to another moment, the chart shows the new situation automatically:

If we click the play button on the Time Navigator the graph will turn into a movie, displaying the pressure in this pipe, in a linear time mode, depending on the simulation time and time steps you have specified for this model (see also "Time parameters..." on page 107). You can view multiple movies in a single graph or in several charts simultaneously, for example movies of the head and flow.

2.7.3 AVI-creator

You can record the movie in a separated file (AVI format) to use it for presentations or to include it in your digital report. The AVI-file can be shown with any media player which support AVI-files.



To record the movie, simply press the red record button in the "movie navigation window". A "Save Dialogue" window will ask for a file name of the AVI -file.

Note that only one location chart window may be active. It is not possible to record two or more location chart windows together.

2.8 Creating your own database system

2.8.1 Property templates

WANDA 4 allows you to create your own set of frequently used settings for all kinds of data. This applies to component properties and specific settings for fluids, accuracy or time parameters.

In this way you can use WANDA 4 the way you want, and adapt the application to your specific use.

Frequently used properties can be stored on disk and re-used. WANDA uses the file system to manage these property templates. An example of such a directory tree is part of the WANDA installation.

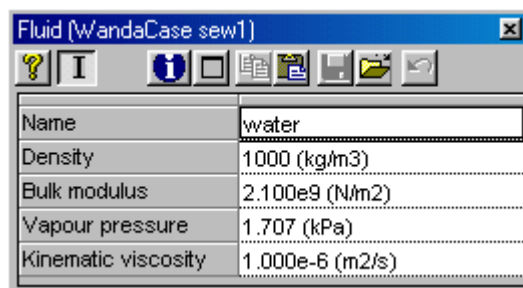
For example:

- map Fluids contains templates of several kind of fluids
- map Pumps contains templates of several kind of pumps
- map Pipe sizes contains templates with pipe dimensions for several materials and pressure ranges
- map Roughness contains only the roughness property of several kinds of pipe materials.

Important: the delivered property templates are only provided as examples. Deltares not be held responsible for the correctness of the contents.

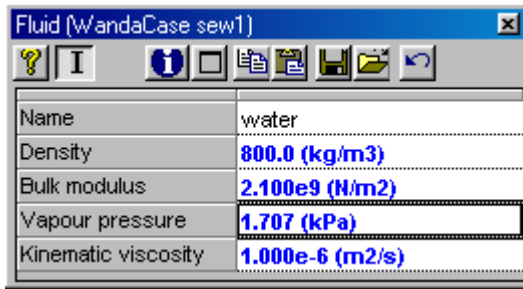
2.8.1.1 Example: Property window

The default fluid window (for Liquid) in WANDA 4, looks like this:

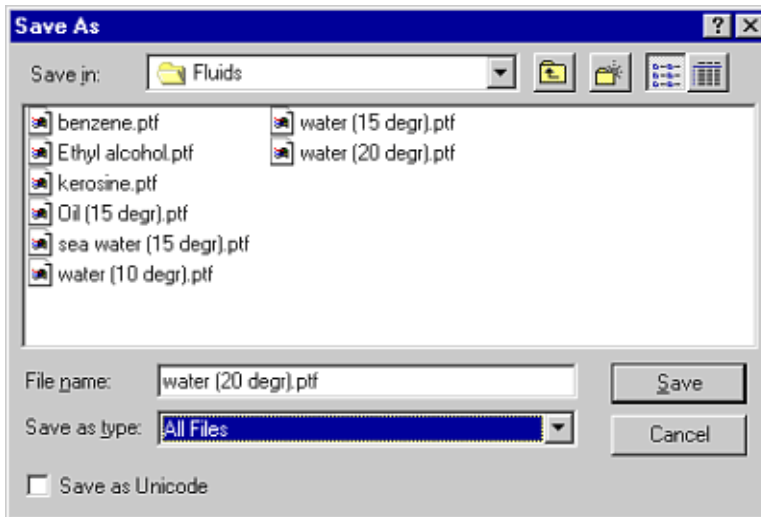


In the model you use, it is possible that you frequently calculate schemes for water of 20 degrees. This means that every time you build a model, you have to adapt the characteristics for water.

In WANDA this problem doesn't exist because you can create your own template for water of 20 degrees, and store it for later use. All you have to do is give in the characteristics and save them:



When you click the save button, WANDA asks you for a file name:



If we open this template in Notepad, it looks like this.

```

* Example Fluid property template file
*
* Fresh water (20 degr)
*
*****
Bulk modulus      2.10000e9
Vapour pressure  2338.00
Kinematic viscosity  1.00200e-6
Density  998.21|
Name    fresh water
    
```

Data is always stored in SI units, but automatically converted if you select another unit system in WANDA.

It is obvious that you can make templates for all kinds of fluids in this way and create your own database system.

The properties are stored in ASCII (text) files with extension “ptf” which stands for **Property Template File**. We recommend you to extend these files with useful comments.

See next example:

```
* Source: Catalog Manufacturer XXX
*       page II-32a
* Last updated: 1999-09-25      S.O. Mebody Deltares*
```

```
Model name      HPE 125x110.8
Comment         0.6 MPa  SDR176
Material name   HPE
Inner diameter  1.108
Wall thickness  0.00710000
Young's modulus 8.00000e8
```

If you save component properties to a template, only the selected properties are saved (the blue ones). For the fluid, physical, initial values, user units, all properties shown are saved.

2.9 File Management

2.9.1 File management in WANDA 4

A complete case study using WANDA produces a set of data files, called case files. The case study is identified by a case name, defined by the user. Case files have the same name as case name but differ by extension. File extensions are administrated by WANDA. This section presents the filename conventions and the way to deal with the data files.

2.9.1.1 Conventions for WANDA file names

One set of case files is related to an unique WANDA computation/simulation. To distinguish between different simulations, the user should give a unique case name to each simulation. Case files use the case name as the filename, so that they can be easily identified in the working directory.

The maximum length of a case name including the path (C:\...\...) is 260 characters (Windows limit). The path and file name may contain blanks.

The general format of filenames is then: "xxxxxxx.ext" where "xxxxxxx" is the case name (user dependent), and ".ext" is the extension generated by WANDA. The extensions used, are:

extension	file type
wdi	file for input data
wdx	file for iGrafx diagram
wdo	file for data generated in both steady and transient computations
__i	interface file between the modules STEADY and TRANSIENT
_sm	messages (log file) from the module steady-state
__r	interface file between the modules TRANSIENT and RESULTS
_um	messages (log file) from the module transient-state
wir	ascii file containing the input report
wor	ascii file containing the output report
wps	ascii input file used for parameter run
wcr	ascii output file with case compare results
rtf	log file, written by closure of case file
csv	ascii output file (column separated) with summary output parameter run
wdd	ascii file containing the case description text; handy for preview in a file manager
wmf	file containing the diagram picture; handy for preview in a file manager

It is recommended to choose a case name, which contains as much information about the system and process simulated as possible. One may think of, for example, "oilload" or "pumptrip".

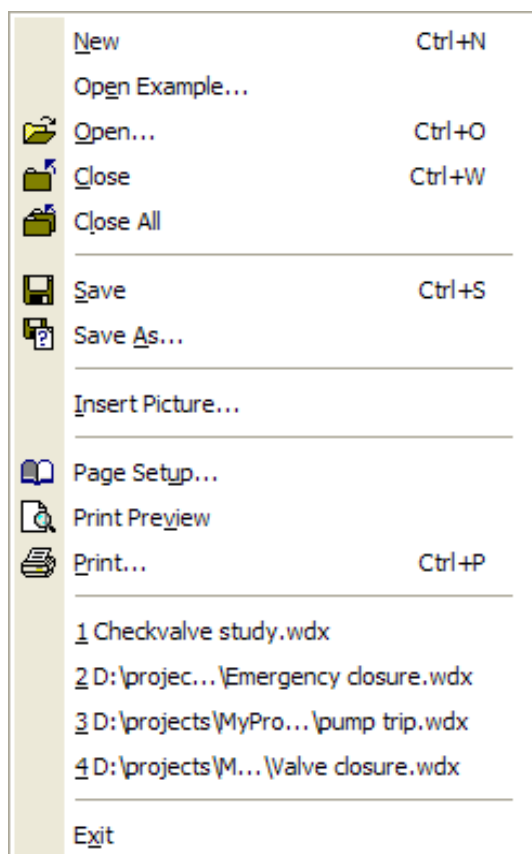
One often needs to make several simulations based on the same hydraulic model. This is possible provided that each simulation has a unique case name. It is advised to administrate the simulations by having serial integer numbers included in the case name, for examples, "pumptrip01", "pumptrip02", etc.

2.9.1.2 *Important files for backup*

For backup purposes it is recommended to store at least the WANDA input files in a save place. It is not necessary to store all files because the output files can be generated easily again.

To store the complete input, you need only the "wdi" and "wdx: file.

2.9.2 The File Menu



The WANDA File Menu has, next to the traditional, menu options, a few specific ones (see for details Help on iGrafx Professional).

If you choose:

- New: WANDA 4 opens a file called Untitled.wdx. In the caption you also see "(no output)".
- Open: opens *.wdi-files
- Save as: saves *.wdi-files
- Close: When closing, WANDA 4 asks "Save changes to *.wdx?"

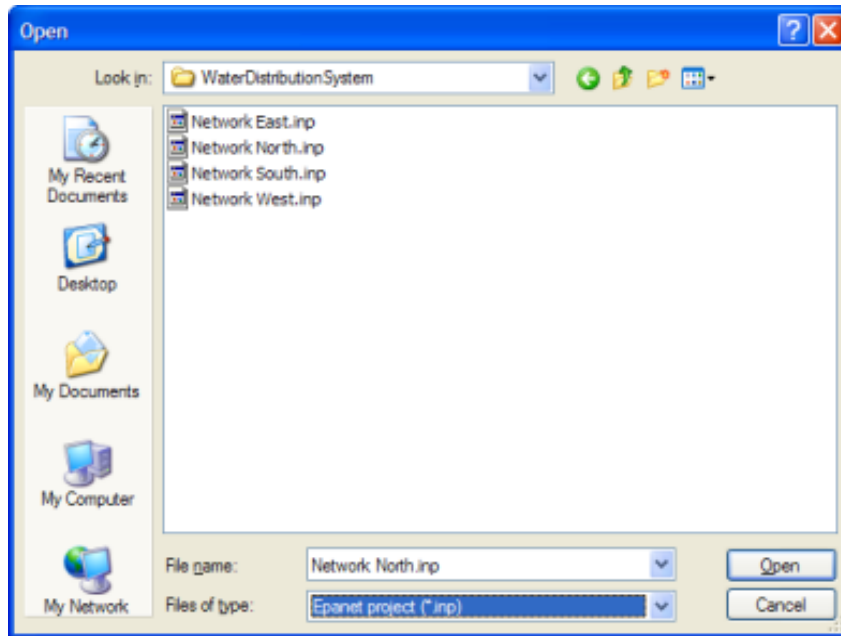
At the bottom of the menu you see projects and files, that were used in previous sessions.

Note: You can open and view diagrams, inputs and results of all kinds of WANDA cases in WANDA 4, including cases that contain components that are not covered in your license agreement. You can not save cases that are not covered by your license agreement; the Save and Save as.... items are disabled.

2.9.2.1 EPANET / ALEID import

WANDA supports the import of EPANET and ALEID models. ALEID is a Dutch program used for drinkwater distribution (based on Epanet scheme).

To import an EPANET or ALEID model, choose from the file menu "Open" and select from "Files of Type" EPANET project (*.inp) or ALEID project (*.pro).



From the distribution model, the next data is converted: pipe length, pipe diameter, wall roughness, H-node elevations, delivery rate (H-component TAP).

Converting these files takes some time dependant of the network size. Be aware that the network size match your WANDA size license.

2.10 Arrange Menu



The Arrange Menu allows you to edit the layout of the diagram. Most features are also accessible using the context sensitive right mouse click menu.

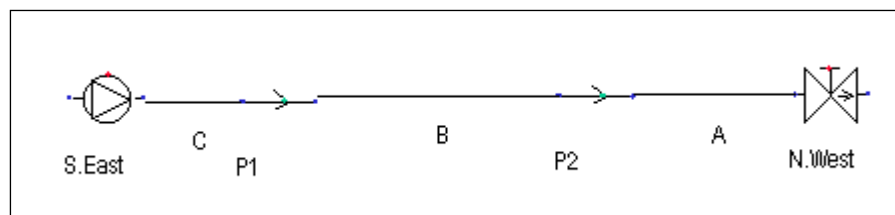
In addition to the iGrafx Arrange Menu, WANDA 4 offers two additional features, especially for use in Hydraulic schemes.

2.10.1.1 Align route

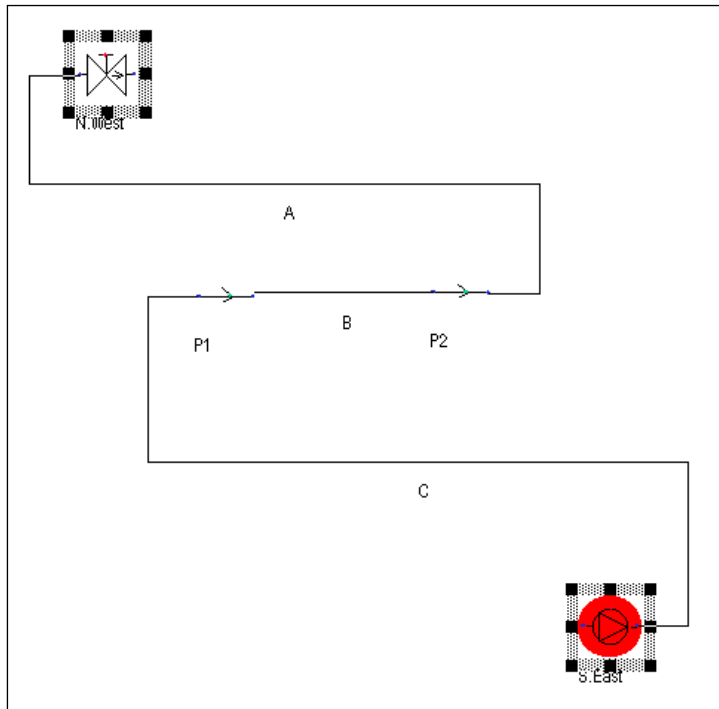
Aligns a selected route between several hydraulic components, for the purpose of reporting.

Note: When aligning routes it is important that you have used **direct connection** lines.

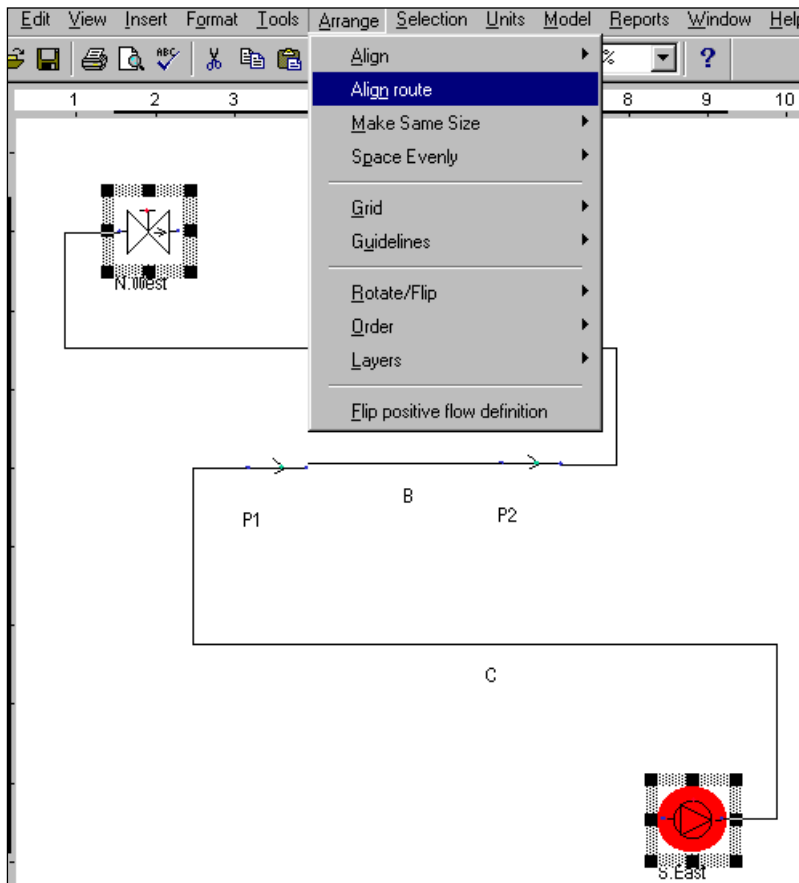
The pictures below show how to align a route of components:



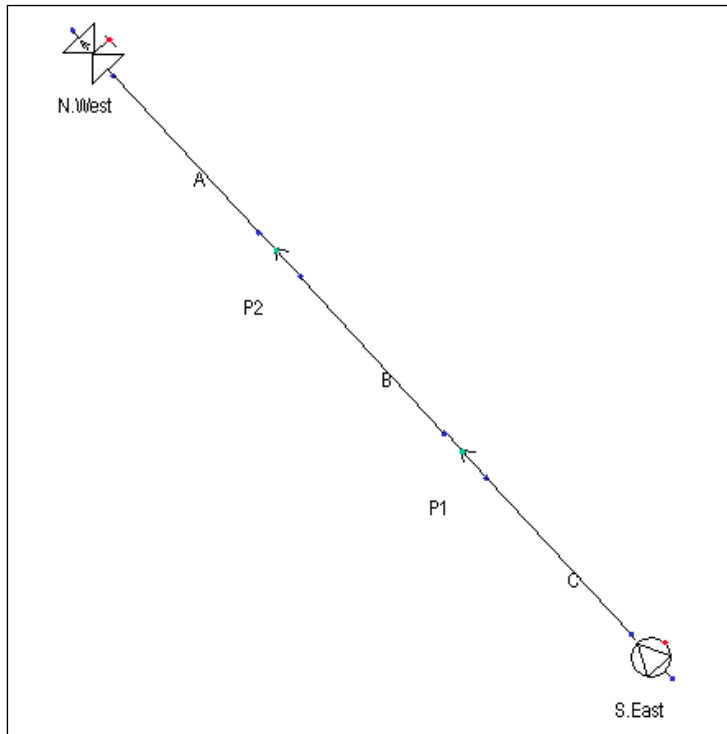
1. Schematic situation: A flow from South East to North West



2. Real situation: North West (top-left), South East (bottom-right)



3. Aligning real situation



4. Result

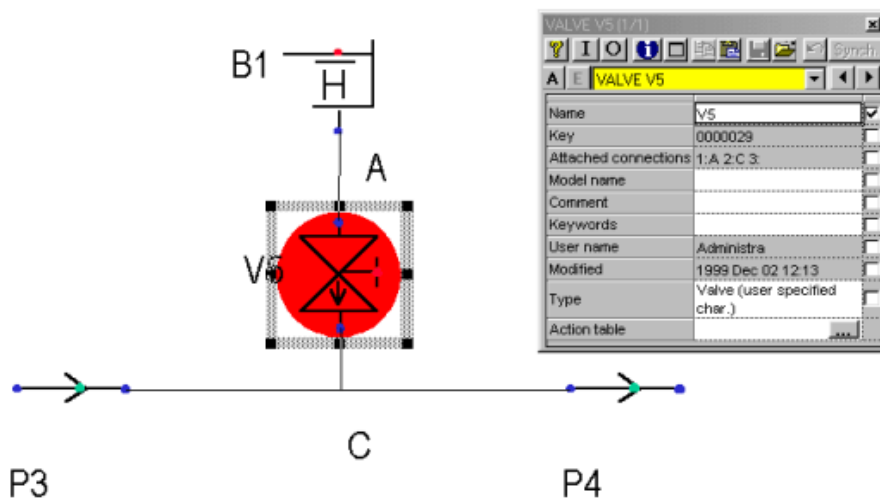
Tip

It is also possible to draw a series of components in a particular direction using the "Vector based schematisation" on page 39.

2.10.1.2 Flip positive flow definition

Allows you to change the flow direction (definition) of selected components.

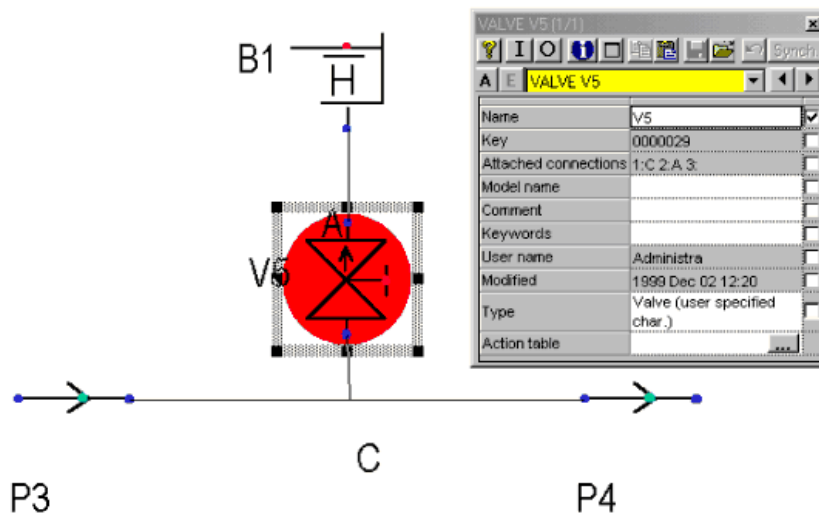
The pictures below give an example:



1. Valve V5: flow direction from A to C. Note the property 'Attached Connections' in the Property Window.



2. Choosing 'Flip positive flow definition' in the Arrange Menu



3. The Flow definition of Valve V5 is flipped (see also Attached Connection)

The flip feature is also supported by the flip button in the draw toolbar.



If the draw toolbar is not visible, select Menu/View/Toolbars.

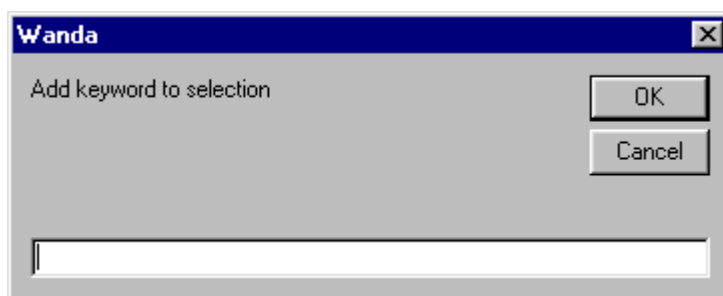
2.11 Selection Menu

Selection Builder Window	Alt+E7
Add Keyword to Selection...	Ctrl+Alt+Num+
Remove Keyword from Selection...	Ctrl+Alt+Num--
Select Objects with Keyword...	Ctrl+Num+
Deselect Objects with Keyword...	Ctrl+Num--
Select All Objects with Property Comments	Ctrl+Shift+F11
Select First Invalid Object	F10
Select All Invalid Objects	
Select All Objects with Warnings/Errors	Ctrl+Shift+F10
Select All H-Components	
Select All Pipes	
Select All None Pipes	
Select All C-Components	
Select All H-Nodes	
Select All Signal Lines	
Deselect All H-Components	
Deselect All Pipes	
Deselect All None Pipes	
Deselect All C-Components	
Deselect All H-Nodes	
Deselect All Signal Lines	
Select Next Object	▶
Add Next Object to Selection	▶
Select Next Object And Keep First Object Selected	▶
Select Route Between Selected Components	Ctrl+Shift+R
Select All Routes Between Selected Components	Alt+Shift+R

In the Selection Menu you can view and edit all kinds of characteristics of components in your model, by simply selecting one, a group or all the components.

See also: "Selection Builder" on page 62.

2.11.1 Add to/ Remove keyword from selection...



WANDA 4 offers you the possibility to shape and build the hydraulic model to your preferences.

Each H-component and H-node has a keyword property. This is a text string up to 50 characters. The string may contain more than one keyword. Use quotes (e.g. "from A to B") if the keyword contains spaces.

Keywords are used to select a certain group of objects with a special meaning, e.g. a pump station, a network extension, a route etc. This means that an object can contain more than one keyword. To add or remove a keyword from a selection you need this feature.

Note: if you use the synchronise button in the property window, each object gets the same keyword(s). Now only the specified one is added/removed.

In this way it is easy to find a group of selected elements, by using the "Selection Builder" on page 62.

2.11.2 Select invalid components

In some cases the calculation of your model is not performed. This means that one (or more) component(s) in the model are invalid. They can be of the wrong type, in the wrong place or may have the wrong input. All these faults depend of course on the type of model you want to calculate.

In the selection menu it is possible to select these invalid components by choosing:

- **Select first invalid component:** selects first invalid component in the flow (press F10)
- Select all invalid components.

If there are no invalid components this option is grey and cannot be used.

2.11.3 Select components and/or connections

The function of these options is self-explanatory.

It is advisable not to select an element before using one of the options. E.g. if you have selected all pipes, and afterwards you select all connections the selected pipes will stay in the total selection.

It is possible to select:

- all H-components
- all pipes
- all none pipes
- all H-nodes
- all C-components
- all signal lines

2.11.4 Deselect components and/or connections

This function is the opposite of *Select components and/or connections* and therefore self-explanatory.

Note that you can use both features in addition to each other. You can for instance select all H-components and delete all pipes from this selection by using 'deselect all pipes'.

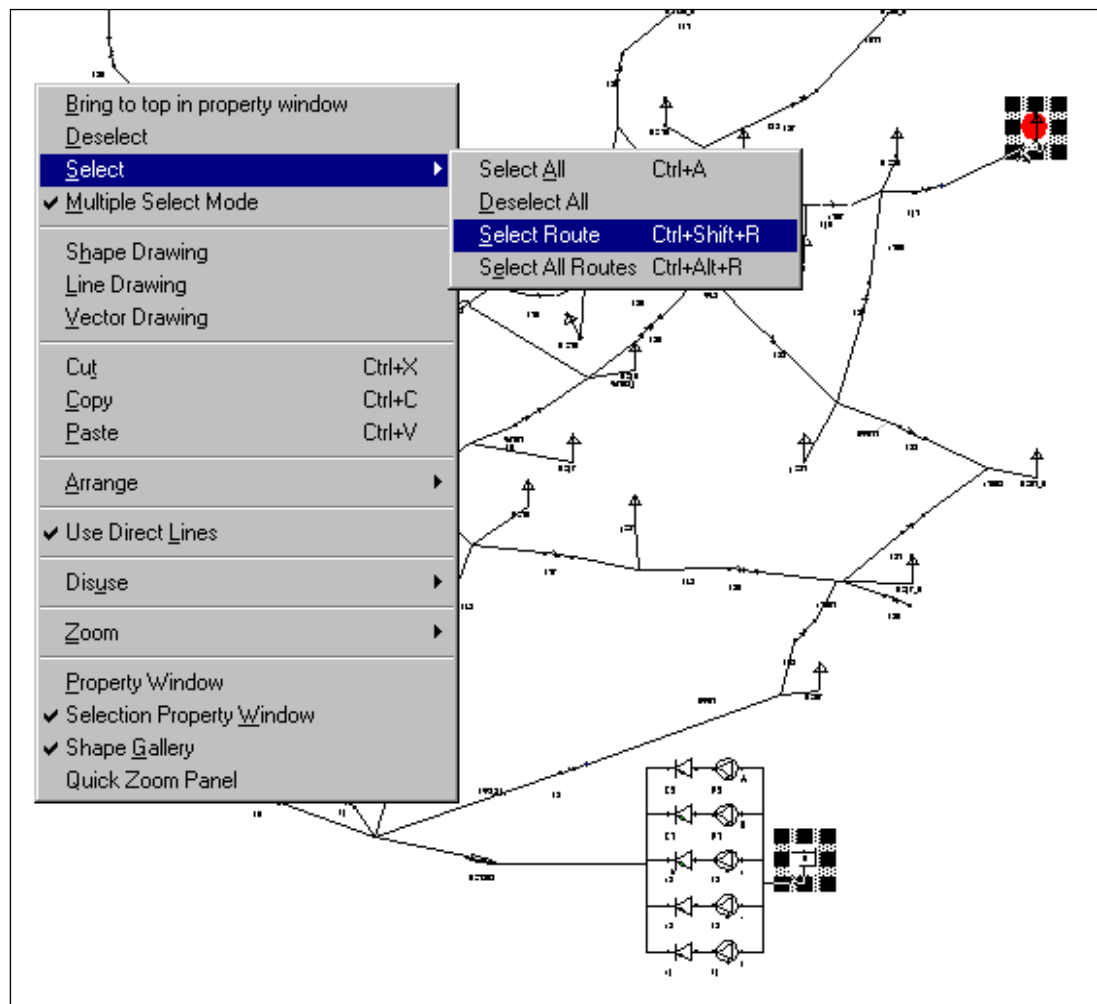
2.11.5 Select route between selected components

When you have selected two components, this feature selects all components and connections (H-nodes) on the shortest route between the two selected components.

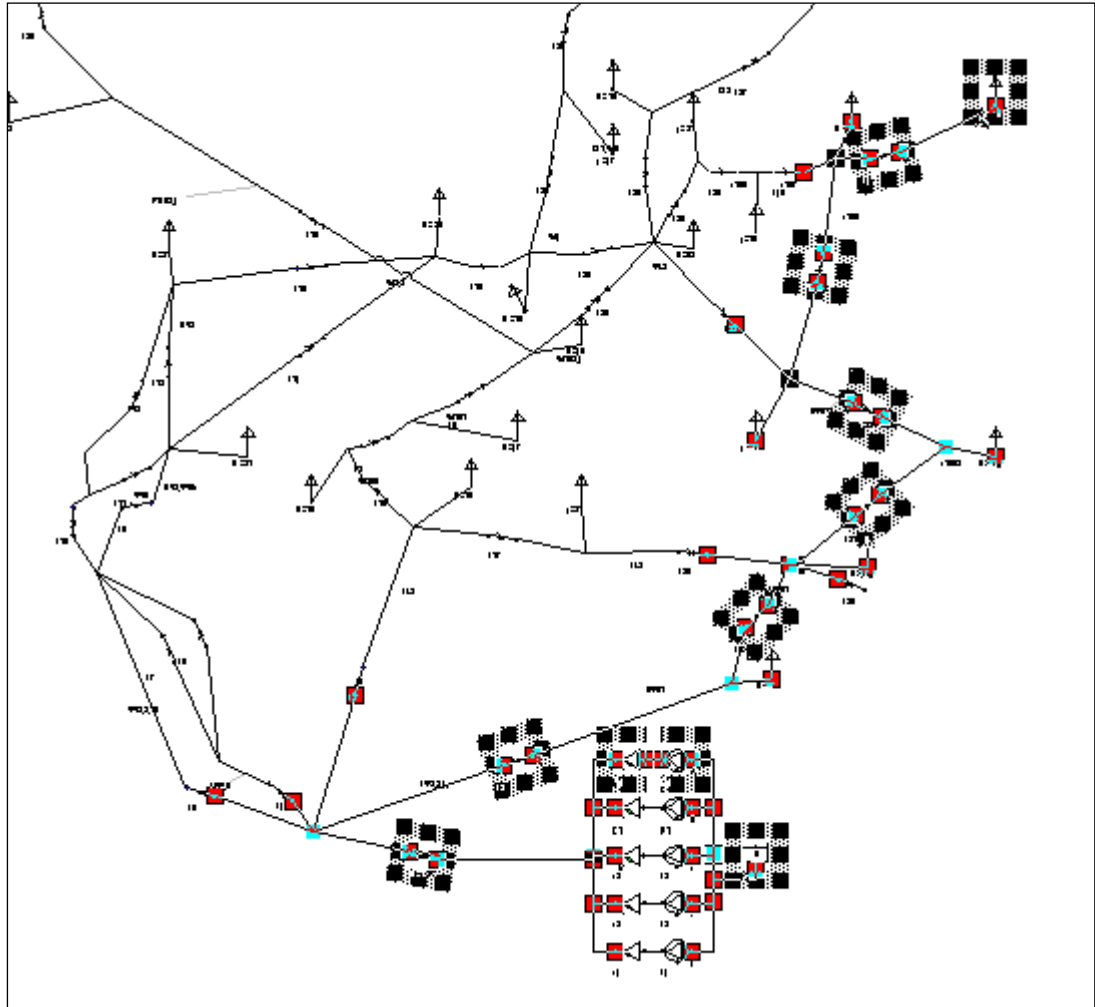
In this way you can easily compare or synchronise certain components or store the selected route under a keyword.

It is now possible to look specifically at the output of the selected route. When the graph button is displayed in the Property Window (focussed on selection – press 'A') you have created a valid route.

The pictures below show how to select a route:



1. Two components in the model are selected. By choosing "Route between selected components" from the Selection Menu all components along the route are selected.



2. All components along the route are selected in the diagram.

A very fast way to select a route is done by a combination of mouse and keyboard.

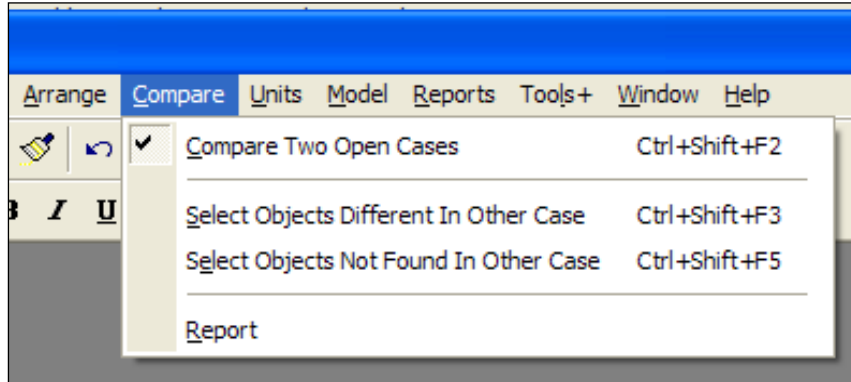
First click (select) the beginning of the route, then click the end of the route with the Shift key pressed down.

The select route option is also accessible via the right mouse click menu.

2.12 Compare menu

The compare menu allows the user to compare two cases that are opened simultaneously. If more than two cases are opened, this menu cannot be used. In this way the user can easily find which objects are changed, and which objects are added. This can be very useful in a study trajectory, in which more than one version of the model may be created.

Below the Compare menu is shown:

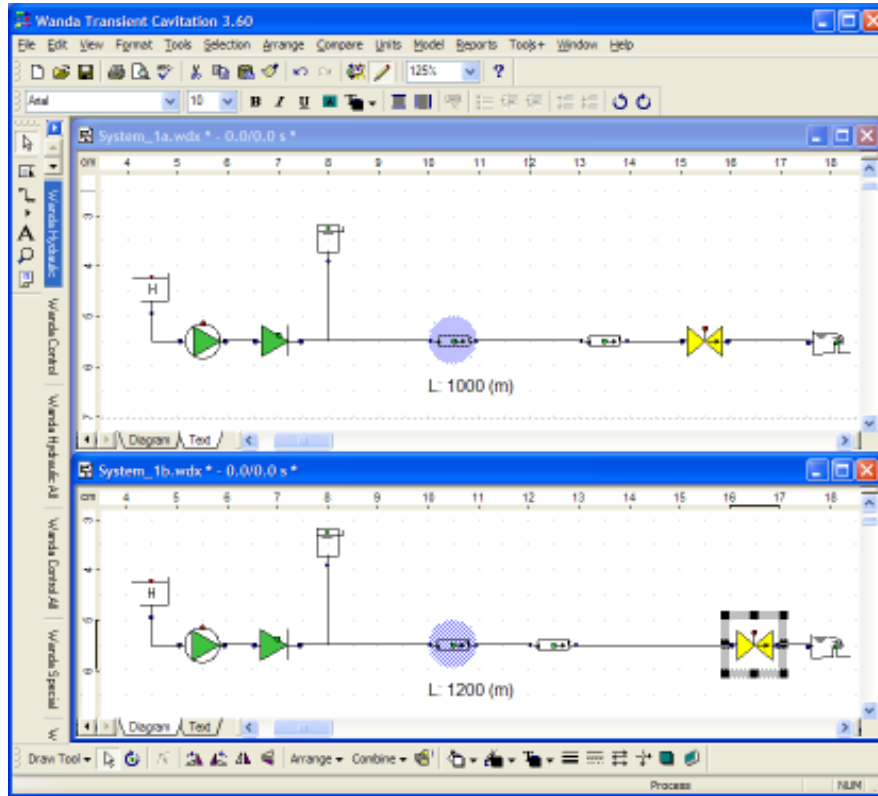


Comparison is primary based on the unique key each object has (like H-component, H-node, C-component, signal line). This key is normally hidden but can be made visible by pressing the pause key with the property window active. The graphical location in the diagram is not taken into account during the comparison, neither other graphical attributes.

Checking the "Compare Two Open Cases" marks all objects with different input properties with a coloured hatch. Differences in property values itself (input and output) are marked in red.

2.12.1 Objects different in other case

When we compare System_1a with System_1b, an object is highlighted with blue hatches. In this situation the key of the object is equal for both cases, but its properties are not. E.g., in the case shown in figure below, the length of the pipe is 1000 m for System_1a and 1200 m for System_1b

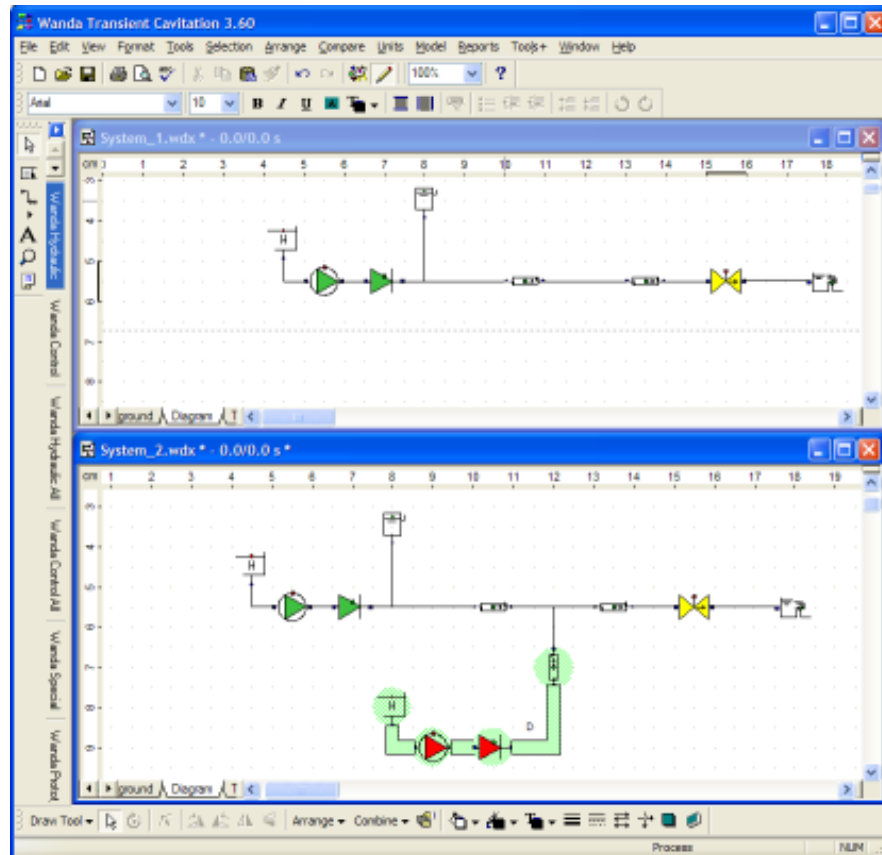


When selecting the object, the property that is different is shown in red in the property window. If we move the mouse to this value, the value of this property for both cases is shown in a light yellow box, see figure 2 in which we have selected the pipe from System_1a: the pipe length of System_1a is printed in front of the '<>' and the pipe length of System_1b after the '<>'. Note that every input property that the user can change, may trigger that an object is found which is different in the other case, although the hydraulic behaviour may be identical (i.e. when adding some comments, or changing the name of an object).

Geometry input	Length	
Length	1000.00 (m)	<input checked="" type="checkbox"/>
H-node height	OK	<input type="checkbox"/>
Wave speed	L: 1000.00 (m) <> 1200.00 (m)	<input type="checkbox"/>
Pipe element count	12	<input type="checkbox"/>
Adapted wave speed	833.3 (m/s)	<input type="checkbox"/>

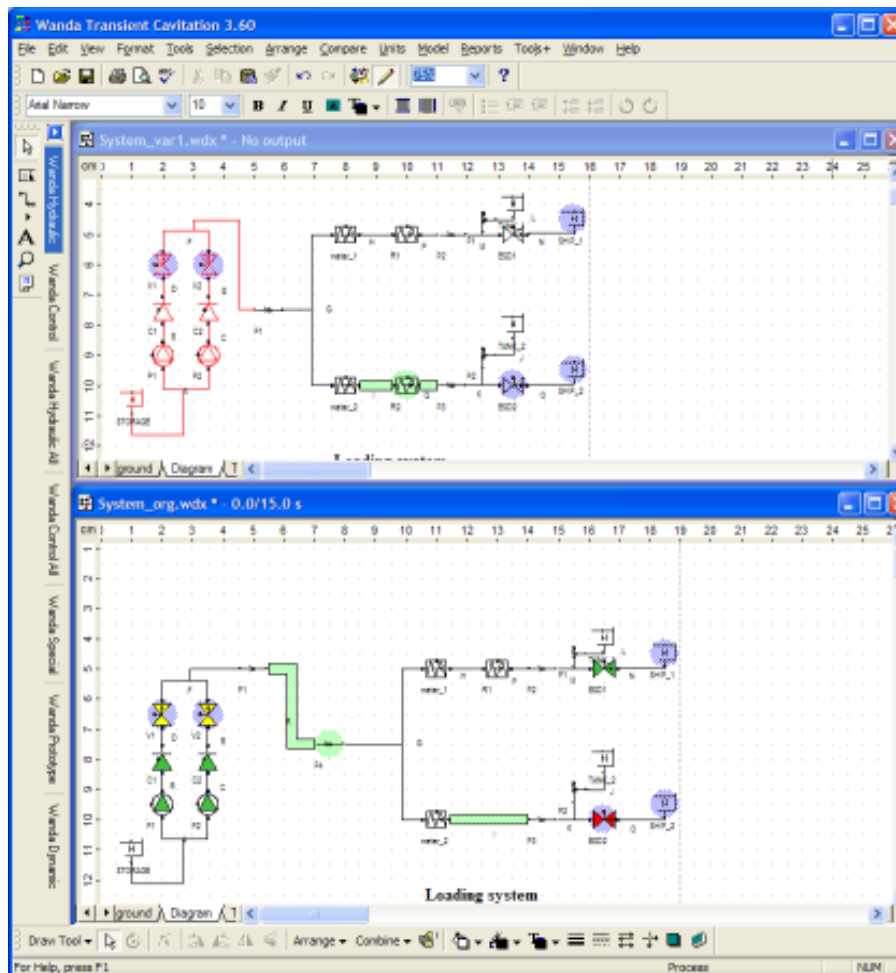
2.12.2 Objects not found in other case

When we compare System_2 with System_1, using 'Compare Two Open Cases' (Ctrl+Shift+F2), some objects are highlighted with green hatches, see figure below. This is because these objects do only exists in System_2 and not in System_1. Here we note that Wanda uses only the key of the components when it is looking for objects that do not exist in both cases.



With the compare menu it is also possible to select all objects that are non-existent in the other case (highlighted green): Ctrl+Shift+F5, or to select all objects that are different in the other case (highlighted blue): Ctrl+Shift+F3.

With the option report, a report is printed on screen and saved as a wcr-file.
See example below



```

System_org.wcr - WordPad
File Edit View Insert Format Help

Wanda Compare Report
Case: System_org <> Case: System_var1

*** DIFFERENT COMPONENT PROPERTIES ***

BOUNDH SHIP_1
Pressure head at t = 0 [s]: 20.2000 (m) <> 19.2000 (m)

BOUNDH SHIP_2
Pressure head at t = 0 [s]: 20.2000 (m) <> 19.2000 (m)

VALVE ESD2
Initial position (open): 0 (%) <> 100.0 (%)
action table: TABLE <> TABLE

VALVE V1
Initial position (open): 80.00 (%) <> 100.0 (%)

VALVE V2
Initial position (open): 80.00 (%) <> 100.0 (%)

*** EXTRA COMPONENTS IN System_org ***

PIPE P4

*** EXTRA COMPONENTS IN System_var1 ***

RESIST R2

*** EXTRA H-NODES / SIGNAL LINES IN System_org ***

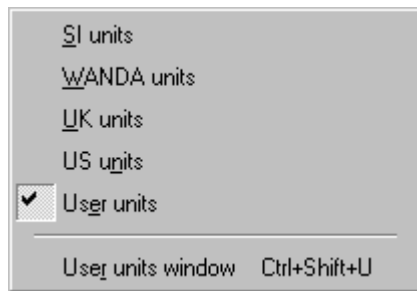
H-node I
H-node R

For Help, press F1
NUM

```

2.13 Units Menu

2.13.1 The Units Menu



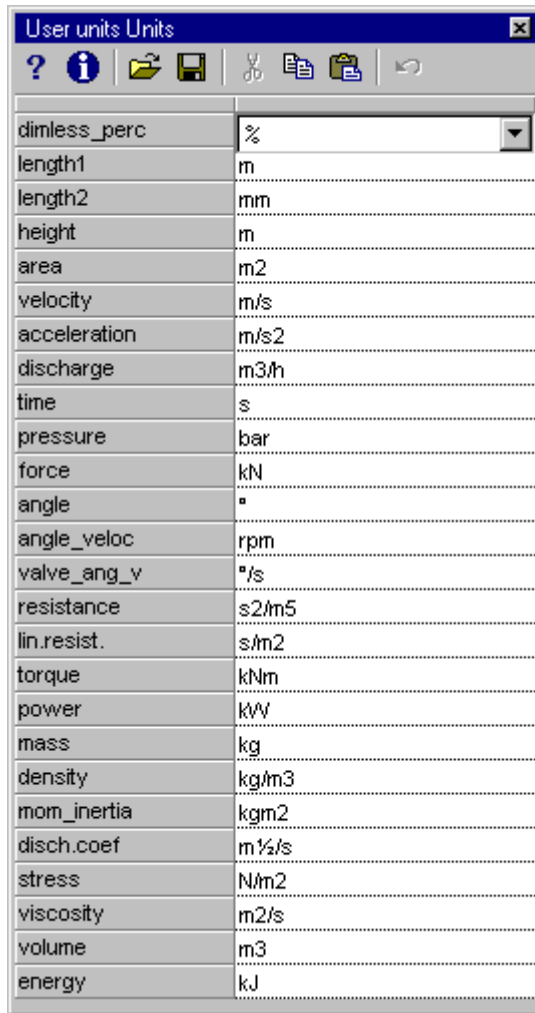
Several unit systems are supported in WANDA. The user can choose among the following systems:

- SI units (SI) (System International)
- WANDA units (WD)
- Imperial units UK)
- ANSI units (US)
- User units (User)

Default units are user units. The user unit table can be configured with your own preferences.

The selected unit system is used for the input and output of data. However, WANDA uses the SI units as the internal system. Conversion between the internal unit system and the system selected by the user is made automatically via the built-in system interface. The user is completely shielded from the internal unit system.

Within a WANDA session with multiple cases active, there is only one unit table active. The last opened case defines the actual unit system, which is used for all open cases.



Next tables shows the contents of the available unit groups and the conversion factor used.

The factor is defined as:

- value [user units] = value [SI-units] * factor.

Dimension	SI	WD	UK	US	User *)
dimless	-	-	-	-	-
length1	m	m	ft	ft	m
length2	m	mm	inch	inch	mm
height	m	m	ft	ft	m
area	m ²	m ²	ft ²	ft ²	m ²
velocity	m/s	m/s	ft/s	ft/s	m/s
acceleration	m/s ²	m/s ²	ft/s ²	ft/s ²	m/s ²
discharge	m ³ /s	m ³ /h	gpm	gpm(US)	m ³ /s
time	s	s	s	s	s
pressure	N/m ²	kPa	psi	psi	barg
pressure_abs	N/m ² .a	kPa.a	psi.a	psi.a	bar.a

pressure_drop	N/m ²	kPa	psi	psi	bar
force	N	N	lbf	lbf	N
angle	rad	□	□	□	□
angle_velocity	rad/s	rpm	rpm	rpm	rpm
valve_angle_velocity	rad/s	°/s	°/s	°/s	°/s
resistance	s ² /m ⁵	s ² /m ⁵	s ² /ft ⁵	s ² /ft ⁵	s ² /m ⁵
linear resistance	s/m ²	s/m ²	s/ft ²	s/ft ²	s/m ²
torque	Nm	Nm	lbf.ft	lbf.ft	kNm
power	W	KW	hp	hp	kW
mass	kg	kg	lb	lb	kg
density	kg/m ³	kg/m ³	lb/ft ³	lb/ft ³	kg/m ³
moment of inertia	kgm ²	kgm ²	lb.ft ²	lb.ft ²	kgm ²
discharge coefficient	m ^{1/2} /s	m ^{1/2} /s	ft ^{1/2} /s	ft ^{1/2} /s	m ^{1/2} /s
stress	N/m ²	N/m ²	psi	psi	N/m ²
viscosity	m ² /s	m ² /s	St	St	m ² /s
volume	m ³	m ³	ft ³	ft ³	m ³
dimless percentage	-	%	%	%	%
energy	J	kJ	kJ	kJ	kJ
angle velocity acceleration	rad/s ²	rpm/s	rpm/s	rpm/s	rpm/s

*) depends on individual user settings; table shows standard setup contents

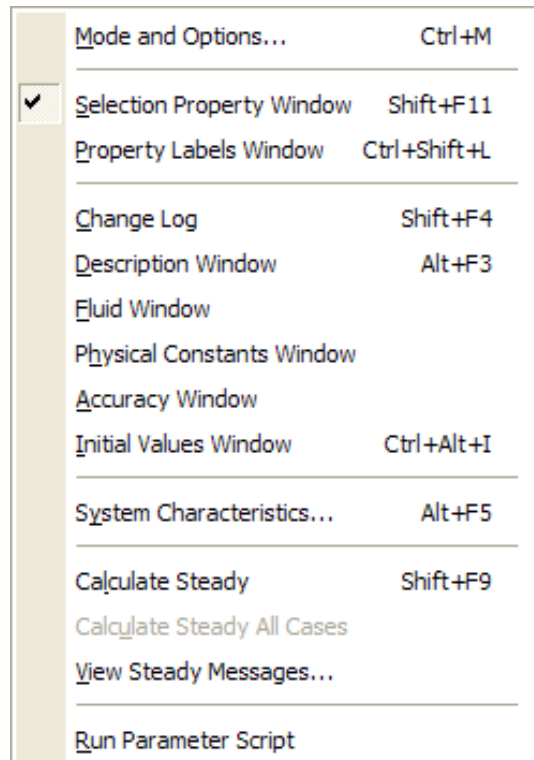
Dimension	Symbol	Factor
mass	lb	2.204622476
length1	ft	3.280839895
length2	mm	1000
length2	inch	39.37007874
height	ft	3.280839895
area	ft ²	10.76391042
volume	ft ³	35.31466672
volume	gallon	219.99
volume	US gallon	264.20
velocity	ft/s	3.280839895
acceleration	ft/s ²	3.280839895
discharge	m ³ /h	3600
discharge	gpm	13199.4
discharge	USgpm	15852.0
discharge	MCMD	0.0864
discharge	l/min	60000
density	lb/ft ³	6.242691e-2
power	kW	1.e-3

power	hp	1.341e-3
pressure	kPa	1.e-3
pressure	bar	1.e-5
pressure	barg	1.e-5
pressure	psi	1.4503768e-4
stress	psi	1.4503768e-4
torque	kNm	1.e-3
torque	lbf.ft	0.7373
viscosity	St	1.e+4
viscosity	CSt	1.e+6
angle_veloc	rpm	9.5492966
resistance	s ² /ft ⁵	2.630721e-3
lin.resist.	s/ft ²	9.290304e-2
mom_inertia	lb.ft ²	23.72996
disch.coef	ft ³ /s	1.811309
diml_percentag e	%	100
energy	kJ	0.001
energy	kWh	0.277778e-6
angle vel. acc	ft ³ /s	9.5492966

2.14 Model Menu

2.14.1 Model Menu

The model menu depends on the mode in which you are operating. Below the Model menu in engineering mode:



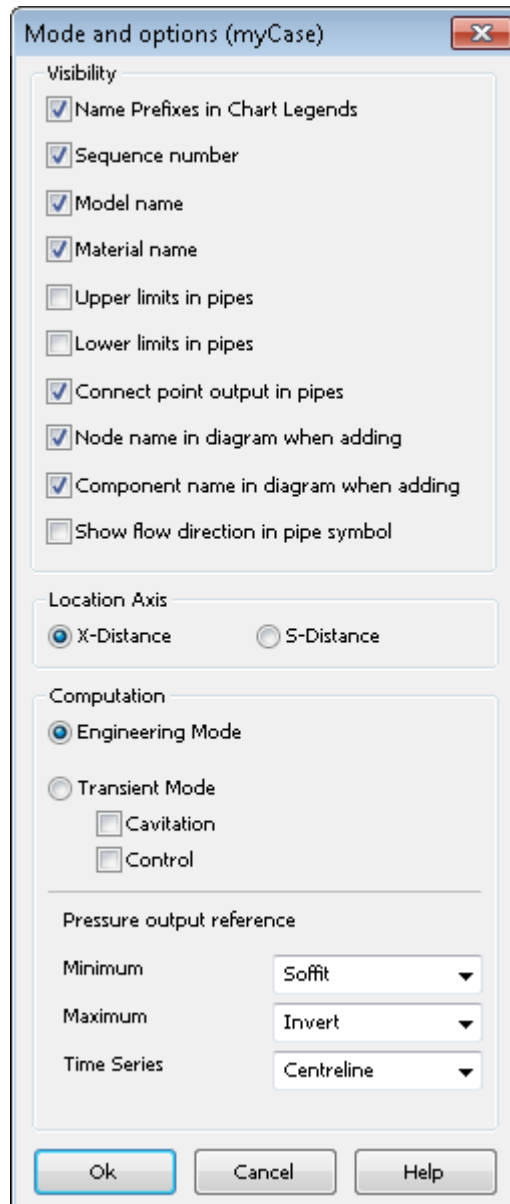
In Transient mode the model menu appears as follows:

	Mode and Options...	Ctrl+M
✓	Selection Property Window	Shift+F11
	Property Labels Window	Ctrl+Shift+L
	Change Log	Shift+F4
	Description Window	Alt+F3
	Fluid Window	
	Physical Constants Window	
	Time Parameters Window	Ctrl+Alt+T
	Accuracy Window	
	Initial Values Window	Ctrl+Alt+I
	System Characteristics...	Alt+F5
	Calculate Steady	Shift+F9
	Calculate Steady All Cases	
	View Steady Messages...	
	Calculate Transient	Alt+F9
	Stop Transient	Ctrl+Alt+F9
	Resume Transient	Ctrl+Alt+F9
	Calculate Transient All Cases	
	View Transient Messages...	
	Time Navigator	Ctrl+T
	Run Parameter Script	

See also

- "Property Window" on page 43
- "Time Navigator" on page 75.

2.14.2 Mode and options



This dialog allows you to select for this case the visibility of some properties and the calculation mode of Wanda.

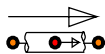
Default all boxes of the visibility properties in the upper part of this window are checked and thus visible in the Property Window (or chart). By un-checking the boxes you can make the properties invisible.

The “Upper limit pressure” and “Lower limit pressure” fields are only applicable for the PIPE’s. if these fields are checked, the PIPE input properties are extended with extra input fields and become visible in the pressure and head location charts.

With the field "connect point output in pipes" checked, the pipe output is extended with the quantities in the begin and end H-node. The advantage is that the time series in these points are available directly without specifying a value in the "location" field.

Adding a new H-component or H-node automatically the name of the new object is displayed in the diagram (the display box in the property window is “checked”). This display function during adding can be controlled by the two “.. when adding” boxes.

The checkbox “Show flow direction in pipe symbol” can be used to show arrows on top of pipes (see figure) in the flow direction. This can be helpful to determine how the flow is distributed in complex networks. For water hammer pipe the flow direction is the average of the velocity at the connect points. To enable or disable this, the simulation has to be rerun. When the absolute velocity is lower than the convergence criteria, no arrow is shown.



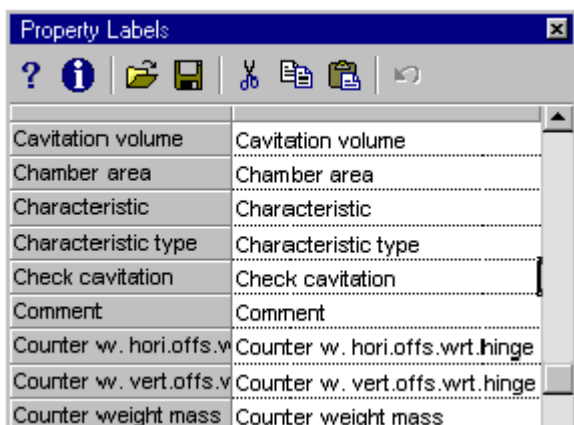
The lower part of this window contains mode and options related to the computations. The radio button allows you to switch between Engineering mode and Transient mode. In Transient mode you can activate three options by checking the boxes.

If “Cavitation” is checked, cavitation is taken into account during the transient calculation. See “Theoretical and mathematical background” on page 147. If “Control” is checked (must be part of your license), the control scheme is part of the transient calculation. If not checked only the hydraulic components are calculated.

The “Pressure output reference” had to do with elevation reference is used for the pressure computation. For more information about this subject see the PIPE component: "PIPE" on page 331.

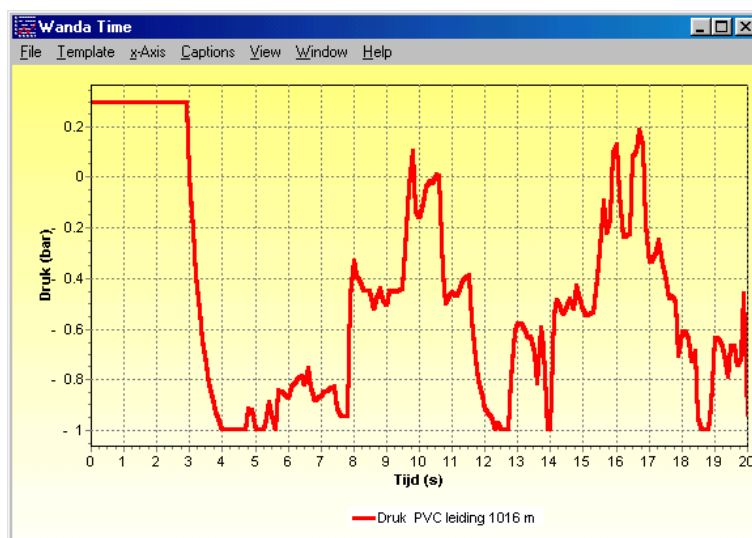
2.14.3 Property labels

The WANDA user-interface is in (UK)-English. For all output the English terms are used, such as “cavitation volume” and “discharge”. This can be a problem when reporting to non-English clients, or preferring other (US)-English words. To solve this ‘language problem’, the user can define another description for any component property. The user definitions become part of the case and can be also saved in template files to build your own “property library”. All output tables and figures can be represented with property labels that best correspond to the hydraulic vocabulary of the user or user-clients. To change back to the default value, delete the user specified one.



Property	Label
Pressure	Druk
Pressure 1	Druk bovenstrooms
Pressure 2	Druk benedenstrooms
Pressure at t = 0 [s]	Druk op t = 0 [s]
Pressure drop	Druk daling
Pressure head at t = 0	Drukhoogte op t = 0 [s]

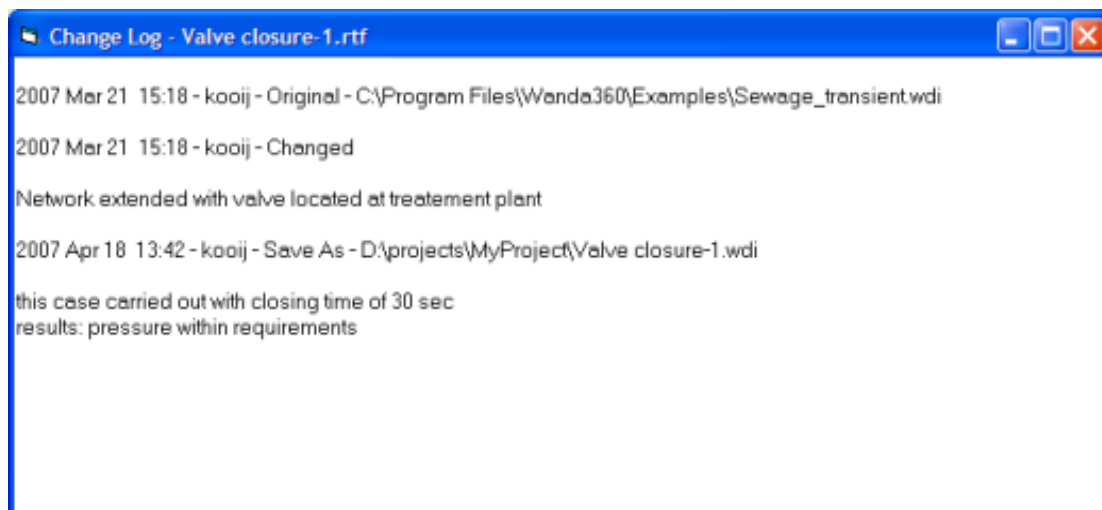
Property	Value
Locatie	1000.00 (m)
Werkelijke locatie	1016.50 (m)
Mededelingen	
Druk	0.2988 (bar)
Druk daling	0.5260 (bar)
Drukhoogte	5.65471 (m)
Drukhoogte verval	1.86211 (m)
Debiet	11310 (m ³ /h)
Snelheid	1.562 (m/s)



- The property labels window show all property labels which are available. It is recommended to translate it once and store this translation in a property template file.
- The next time you open a new case, you only have to read these labels from file.
- The translated labels are stored in the case. If you have more than one case open, the latest case defines the current label setting, because the userinterface can handle only one translation.

2.14.4 Change Log (Shift+F4)

For every case, a log-file is created in a rtf-file. This log-file can be modified via the Model menu. The log-file also keeps track of the 'save-as'-actions, such that the complete history of the case can be seen. Upon closing the case, the user can make some extra comments in this log-file.



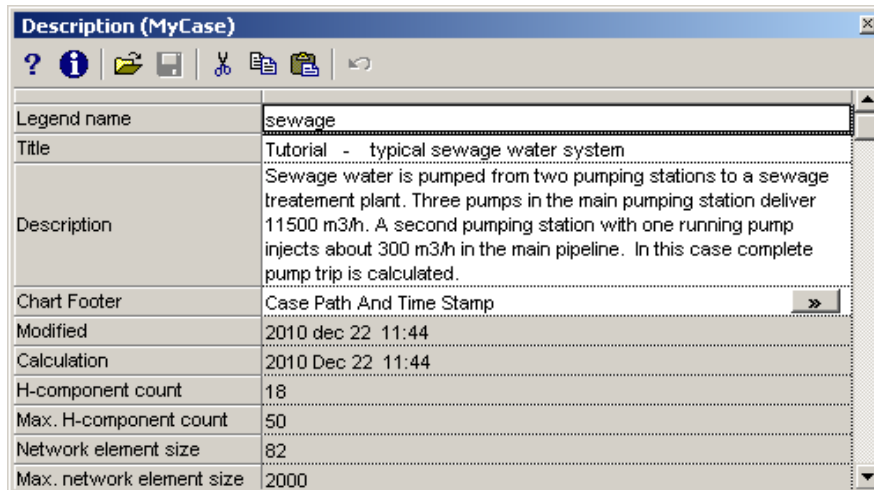
2.14.5 Hydraulic input of your model

The hydraulic model consists of the diagram with its H-component and H-nodes. Properties of these objects are specified using the property window.

To run steady and transient, the model requires some general data. This data is divided in 5 parts:

- Description: enter a description of the current case
- Fluid: specify various physical properties of the fluid
- Physical constants
- Time Parameters: enter the time-step, simulations time, etc, necessary for the computation of steady- and transient-state flow
- Accuracy

2.14.5.1 Description...

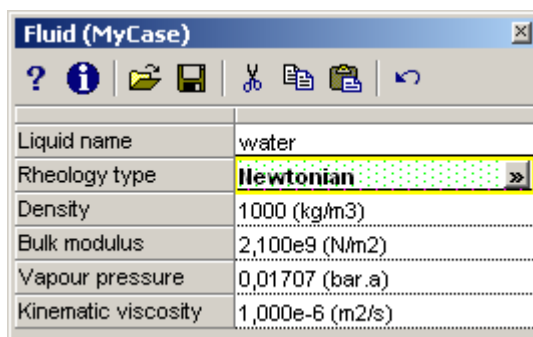


The case name can contain certain information about a case simulation. However, the length of the case name is too short to allow a general description. The **description** window in the model menu can be used to describe the simulation in more detail. The user may give an elaborate description of the system simulated, such as the piping network, the flow process, the objective of the simulation, and the main results.

The “**legend name**” field is used in the chart legend (max 8 char. long). The “**Title**” field (60 char. long) is used as header in charts and input and output reports. The “**Description**” field (1000 char. long) is for additional comment. You create a new line by pressing alt+enter (like Excel).

The “**Chart Footer**” field is used for printing an identification string in time and location charts. There are several options for the content of this footer text varying from a full path and case name including simulation date/time to only the case name. With option “None” the footer text is empty.

2.14.5.2 Fluid...



In the fluid window the physical properties of the fluid are specified. The contents of this property window depend on the mode you are working (Engineering/ Transient) and the domains you are authorized for (Liquid, Heat). For instance, the “bulk modulus” is applicable in transient mode only.

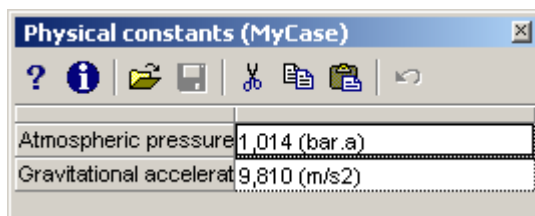
By default the values valid for water at ambient temperature are given in the table. The fluid name is purely documentary and has no effects on the computation. In case other

fluids or water at a different pressure/temperature rating are simulated, the values in the table have to be adapted accordingly.

Choosing Rheology type = Slurry (Optional licence feature) shows the required input parameters for the Herschel-Bulkley model.

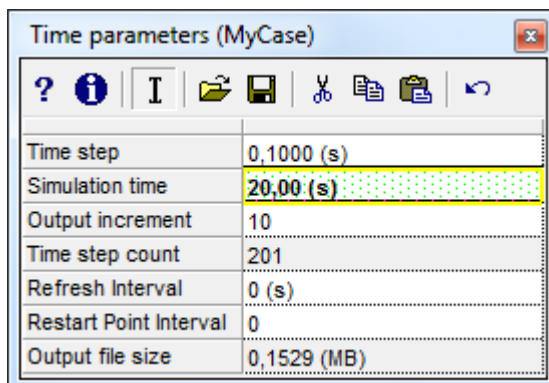
In the Heat domain the fluid properties are temperature dependant. Therefore a table has to be specified. A default table can be retrieved from the property templates subdirectory (part of the Wanda program directory)

2.14.5.3 Physical constants ...



In the window above you can specify atmospheric pressure and gravitational acceleration for your hydraulic model.

2.14.5.4 Time parameters...



The time parameter window is applicable in transient mode only.

Time parameters must be entered for each transient calculation.

The choice of the time-step demands careful consideration. A large time-step generally saves computation time, but the accuracy in the computed results could be low. Choosing a small time-step has the opposite effect. The choice should be based on the dynamic process simulated and the properties of the pipes and the fluid in the system.

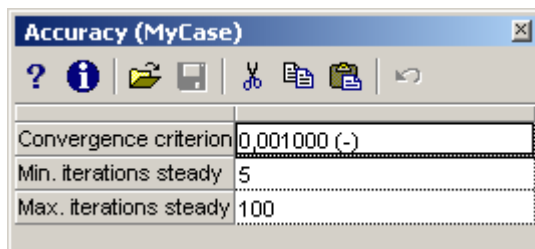
The size of the output file depends on the size of the network (number of elements) and the number of time steps. The estimated size is mentioned in this window. Note that the maximum file size is 2 GB. You will get a warning if this size will be exceeded.

You can reduce the size of the output file by setting the Output increment to a higher value. Setting the Output increment to 10, for example, reduces your output file with a factor 10. If you play Movies or view time series only every 10th time step is displayed. The extreme values charts do remain correct (i.e. based on all simulated time steps).

It is possible to show the results during the simulation. The “Refresh Output Interval” is the refresh rate in real seconds; all open charts and property output windows are updated automatically with this interval. Looking to the progressing results, you can interrupt the simulation when you find them unsatisfactory.

The Restart point interval controls how often a restart point is created during the unsteady simulation. The Wanda simulation can be restarted from these points in the simulation. This greatly reduces simulation times when the user changes an action table in a large model. The restart point interval must be larger than the output increment and must be a multiple of the output increment.

2.14.5.5 Accuracy...



These parameters are used for determining the end of the iteration process to solve the non-linear hydraulic equations in the steady state and at the boundary conditions in the unsteady state. If the relative difference for each element between the last solution vector and the previous iteration is smaller than the convergence criterion, the solution is assumed to be converged.

For steady state calculation it can be necessary to prescribe the minimum number of iterations to get a satisfied and fully converged solution.

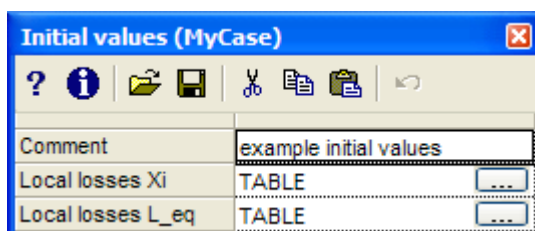
To prevent the program for looping if no convergence can be obtained, a maximum number of iterations is introduced.

Normally the default values has not to be adapted.

2.14.6 Initial values

The PIPE component (Liquid module) use a table with local losses.

You can define your own default values for the local losses. These values will be used as default settings for each pipe.



	local losses ()	Loss coef Xi (-)
1	Elbow 90 degr; R=1.5D	0.2000
2	Elbow 45 degr; R=1.5D	0.1200
3	Ball valve 100% open	0.2500
4	Diffusor 1:4	0.1000
5	Tee; flow through run	0.3000
6	Tee; flow through branch	0.9000

	local losses ()	L_equiv #D (-)
1	Elbow 90 degr; R=1.5D	15.00
2	Elbow 45 degr; R=1.5D	8.000
3	Ball valve 100% open	16.00
4	Diffusor 1:4	6.000
5	Tee; flow through run	20.00
6	Tee; flow through branch	60.00

Wanda is delivered with settings as given above. These settings are saved in a template file “default.ptf” in subdirectory “Initial values”. If these settings do not satisfy your company settings, change the contents and save it (using the save button) to the default.ptf-file.

Each time a new case is opened, the contents of the default.ptf file is copied to the internal data structure of the case. Changing these case settings does not affect the global settings in the template file.

Each time a Pipe with local losses is added to the diagram, the current case settings are copied to the property data structure of the Pipe. Editing the Pipe properties does not affect the case settings.

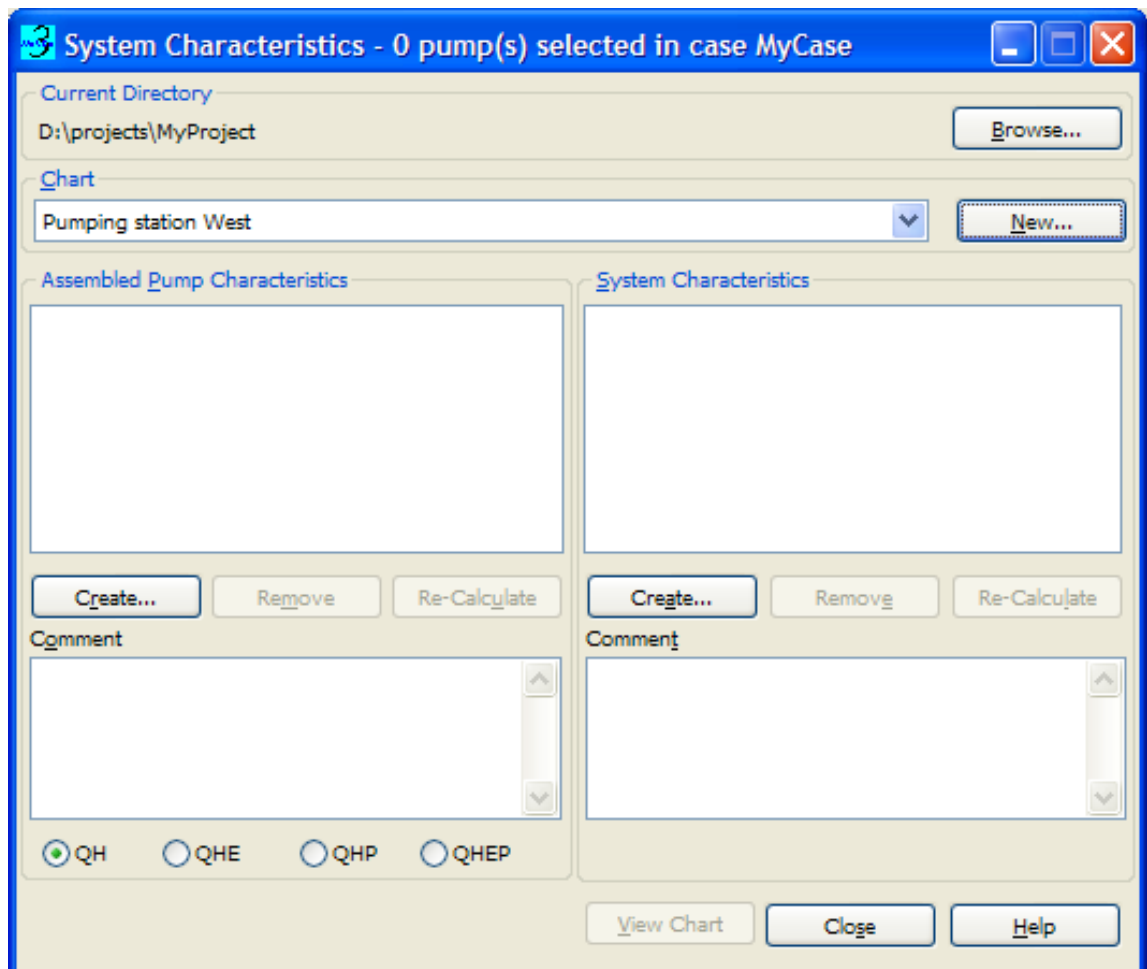
2.14.7 System characteristic chart

This option creates a chart containing several assembled pump characteristics and several system characteristics

This is done in three steps:

- create an assembled pump characteristic
- create a system characteristic
- combine pump and system characteristics in one chart

These steps are carried out using the window as depicted below.



By default the directory name of the current case is shown. It is possible to choose another directory using the browse button.

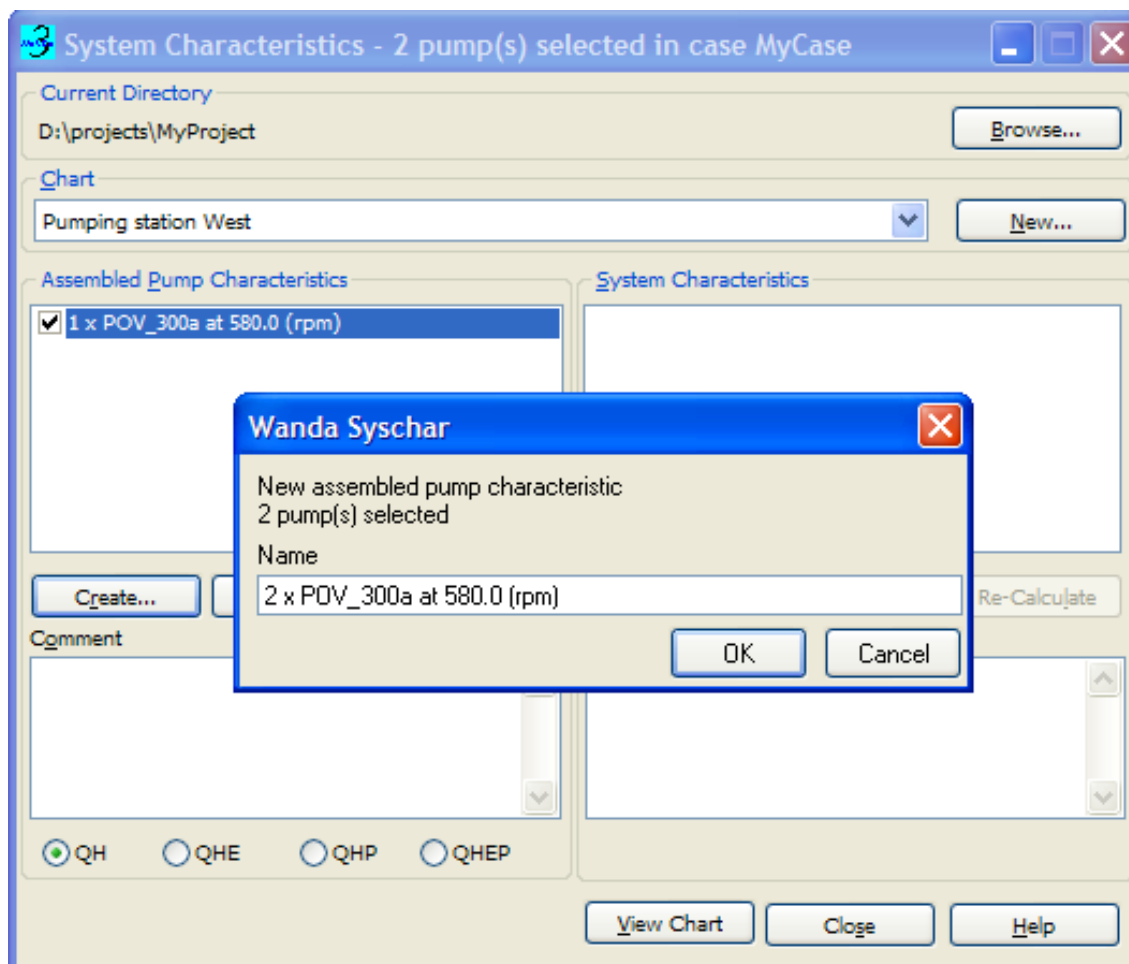
If you have never defined a chart before, you first have to specify a new one. Press the New button and specify a chart title. The dropdown box shows all available chart titles created before.

The two large list boxes show existing characteristics. If they are empty it means that no curves have been generated yet.

All characteristics are stored in ASCII files with fixed extension (.wpc for pump characteristics and .wsc for system characteristics) These files are stored in a subdirectory of the current case directory. This subdirectory name is specified by the user and this name is used as title for the chart to be assembled.

2.14.7.1 Assembled pump characteristic

To create an assembled pump characteristic you first have to select one or more pumps in the diagram. The caption of this window shows the number of selected pumps. Then press the Create button in the assembled pump characteristic box and the program will generate the assembled curve. To identify this curve you have to specify a unique label. The program suggests a label based on the model name of selected pumps, which may be changed by the user. In the chart this label will be used as legend text. The label is also used as file name (filename.wpc in subdirectory case\chart label)

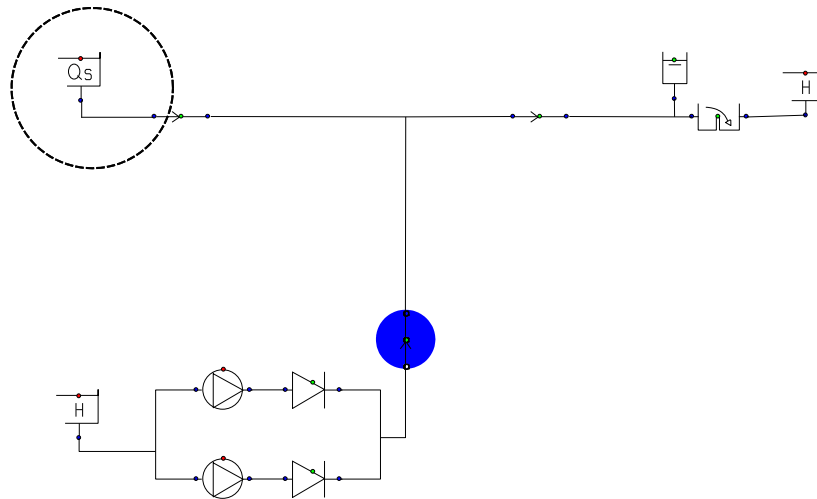


To delete an existing curve, just select the curve label in the box and press the remove button.

2.14.7.2 System characteristic

The system characteristic describes the head as a function of discharge at a certain location. To determine this function the H-component QSCHAR must be defined in the diagram. The QSCHAR represents a set of pumps.

QSCHAR represents the pump station



Tip:

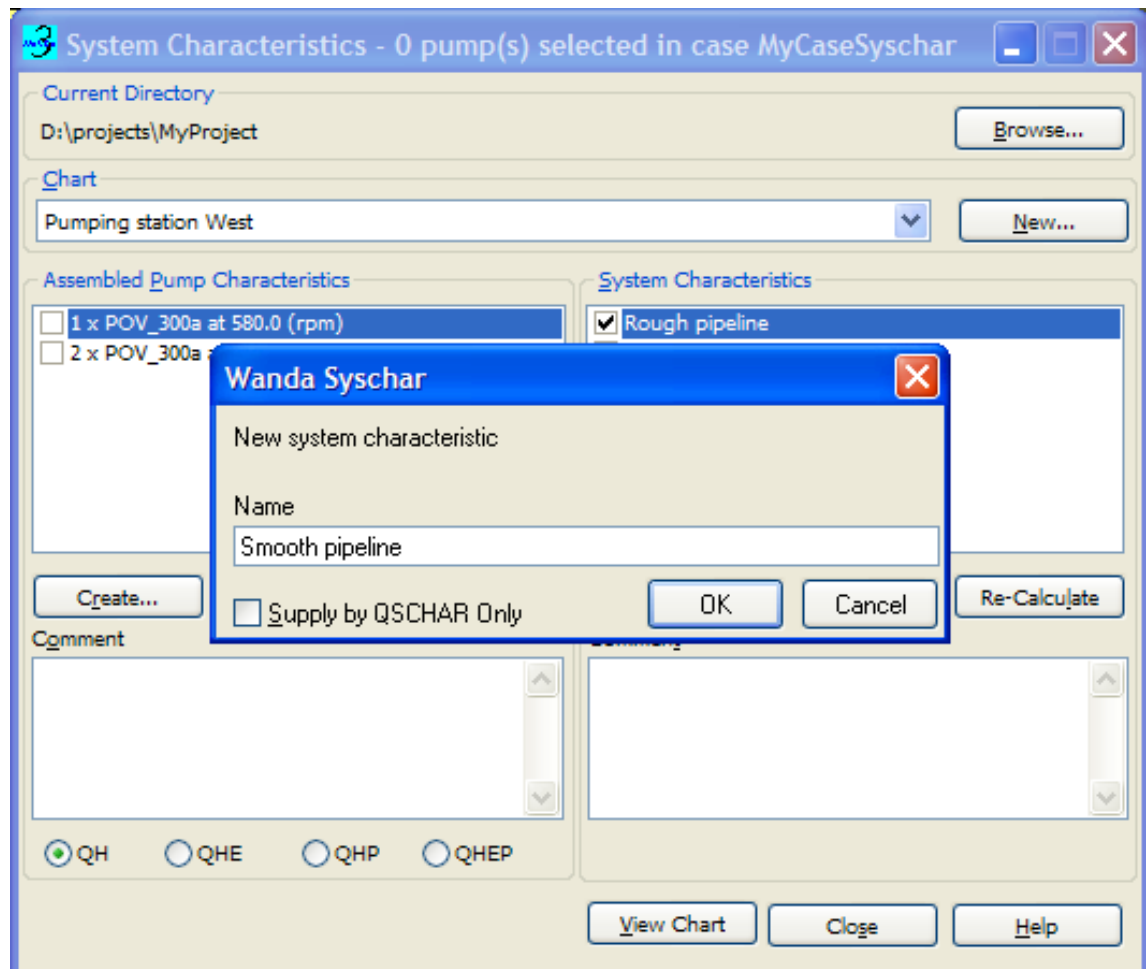
In systems in which PUMP's are already defined in the scheme, use the 'Disuse' property to inactivate the pumps. The pumps are replaced by the QSCHAR component.

Pressing the Create button in the system characteristic box activates the calculation. In fact this is a repeating calculation of Steady with different discharge steps. Steady takes 25 equidistant steps beginning with 0 m³/s until Q_{max} m³/s. Q_{max} is an input property of QSCHAR. To activate this calculation all input of the hydraulic network must be complete and satisfy the validation rules. If the "Create" button is disabled, check the input using the selection menu item "select first (all) invalid components".

In contrast to the normal Steady calculation, it is not necessary to specify the status of checkvalves (or other solution dependent properties of other H-components) in the right way. During calculating the Q-range the checkvalve can change from "procedure phase". This change is automatically carried out.

See H-component QSCHAR for more information.

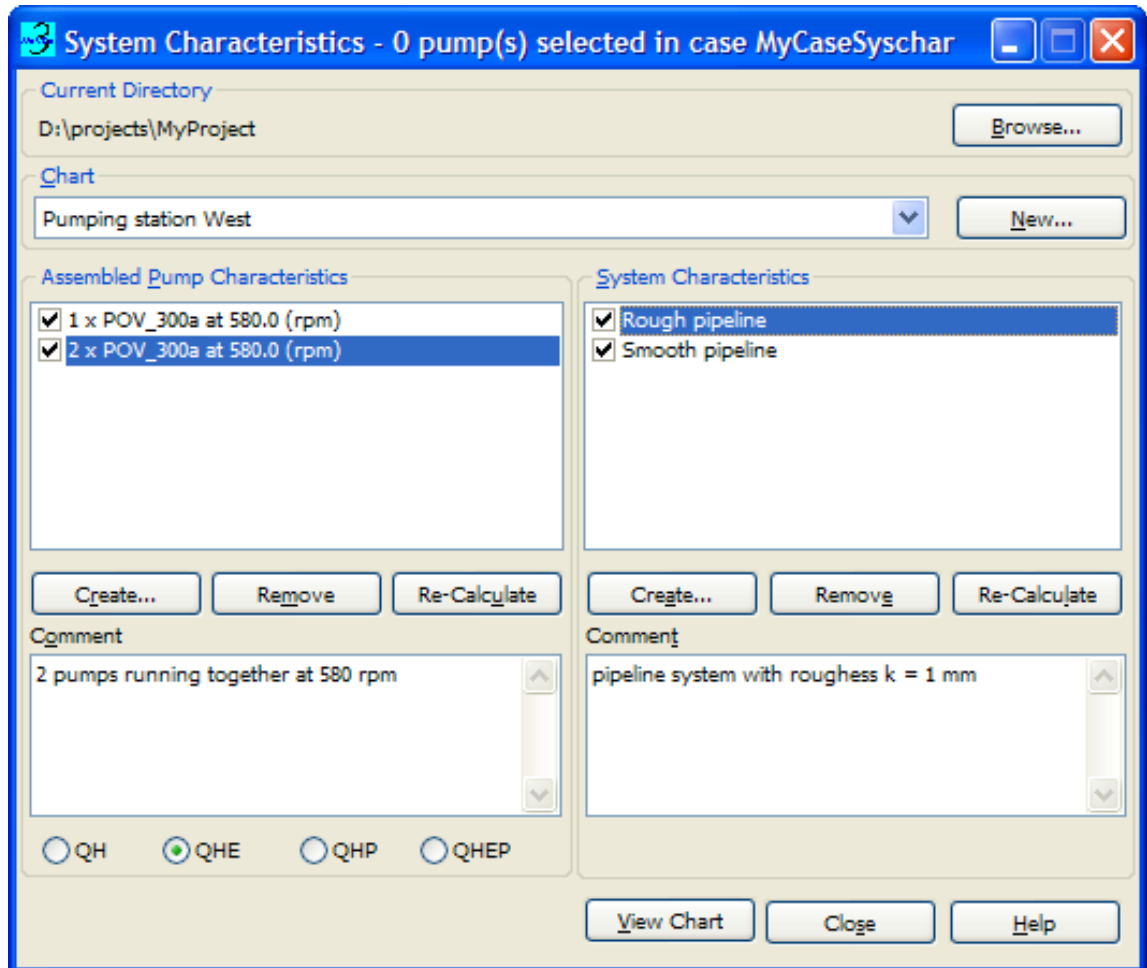
To identify this curve you have to specify a unique label. In the chart this label will be used as legend text, so use a meaningful name. The label is also used as file name (filename.wsc in subdirectory case\chart label).



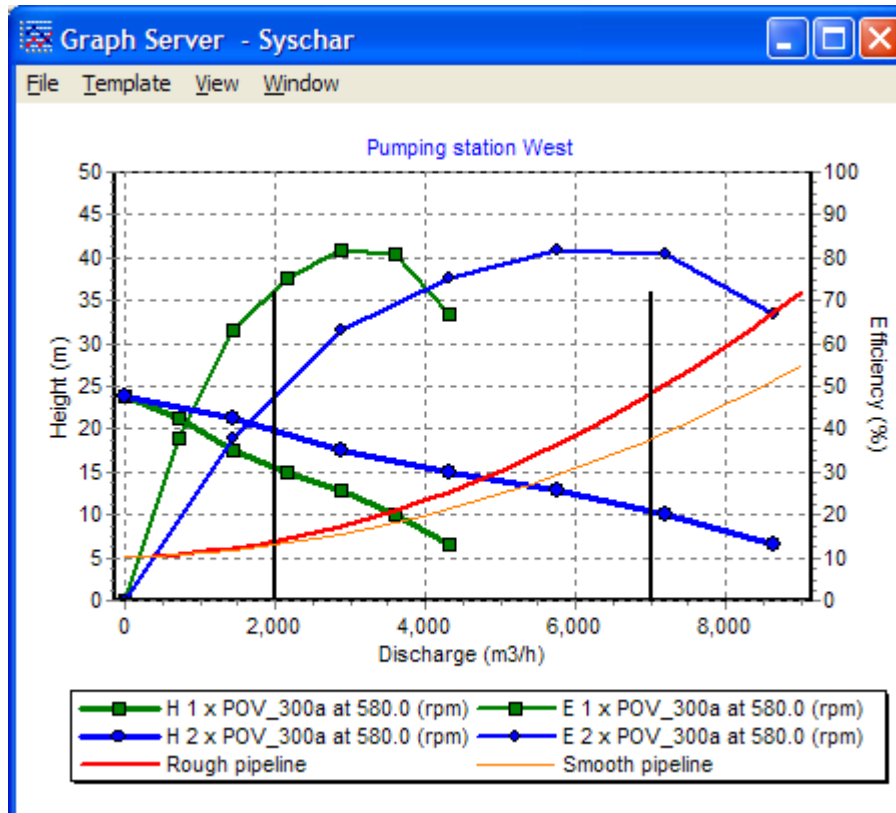
To delete an existing curve just select the curve in the box and press the remove button.

2.14.7.3 Create combined chart

To create a chart with pump characteristics and system characteristics select the applicable characteristics in the list boxes.



Then press the "View chart" button.



The chart appears in a separate window. To create another chart, first deselect the items by clicking on them a second time and select a new set. Press the "View chart" button again and a new window appears.

2.14.8 Calculating your model

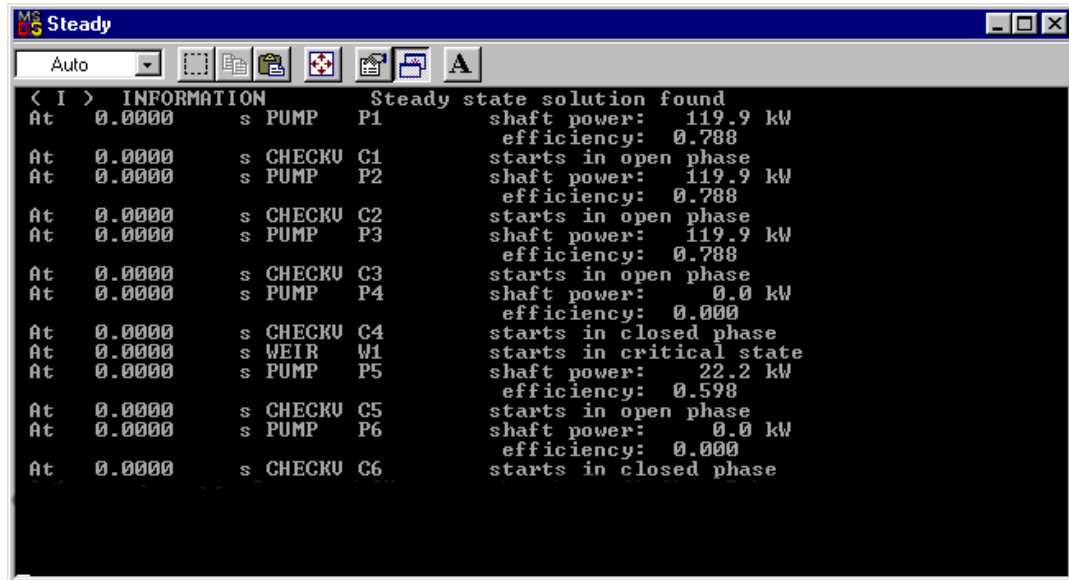
2.14.8.1 Calculate steady

When a hydraulic model is completed and the hydraulic specifications are given, the user tends to go immediately for a run of the steady-state and transient flow computations. However, input errors are often inevitable. Causes for the errors can be associated with the complexity of the system modelled, incidental errors made by the user or lack of experience with WANDA. Therefore, it is useful to check the correctness of the hydraulic model and the input hydraulic data before make any computation.

See also:

- "Select invalid components" on page 88
- "Reports Menu" on page 123

Calculate steady in WANDA is used to compute the steady-state flow in the pipeline system. When you choose this option WANDA starts to calculate the steady state in a separate console program:

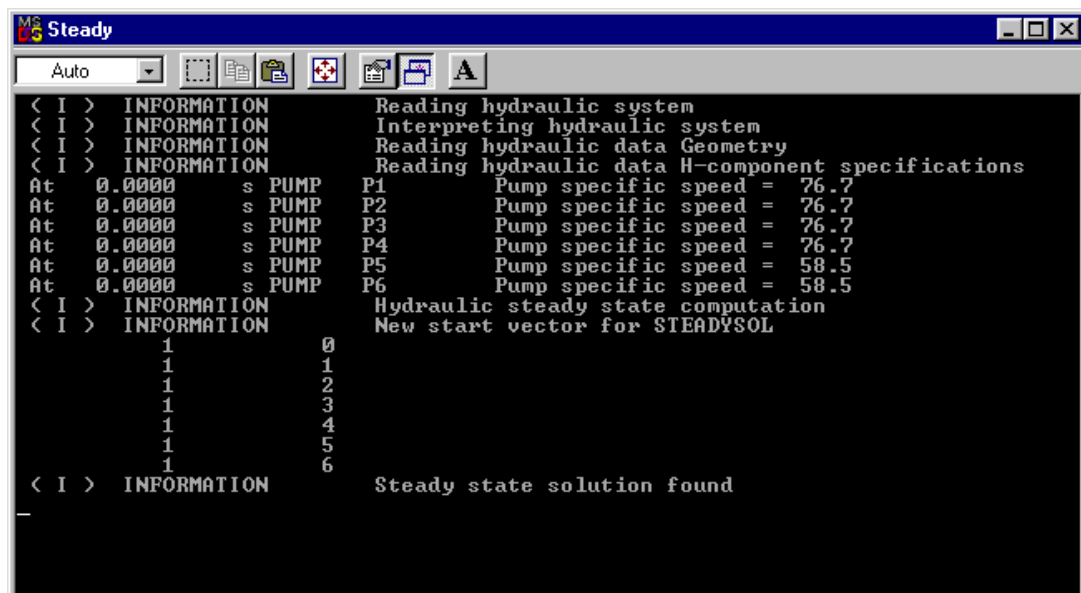


```

MC Steady
Auto
< I > INFORMATION      Steady state solution found
At  0.0000    s PUMP    P1      shaft power:  119.9 kW
                                efficiency:  0.788
At  0.0000    s CHECKU  C1      starts in open phase
At  0.0000    s PUMP    P2      shaft power:  119.9 kW
                                efficiency:  0.788
At  0.0000    s CHECKU  C2      starts in open phase
At  0.0000    s PUMP    P3      shaft power:  119.9 kW
                                efficiency:  0.788
At  0.0000    s CHECKU  C3      starts in open phase
At  0.0000    s PUMP    P4      shaft power:    0.0 kW
                                efficiency:  0.000
At  0.0000    s CHECKU  C4      starts in closed phase
At  0.0000    s WEIR    W1      starts in critical state
At  0.0000    s PUMP    P5      shaft power:   22.2 kW
                                efficiency:  0.598
At  0.0000    s CHECKU  C5      starts in open phase
At  0.0000    s PUMP    P6      shaft power:    0.0 kW
                                efficiency:  0.000
At  0.0000    s CHECKU  C6      starts in closed phase

```

The progress of the steady-state computation is displayed. The computation is successful when the message is **Steady-state solution found** is displayed:



```

MC Steady
Auto
< I > INFORMATION      Reading hydraulic system
< I > INFORMATION      Interpreting hydraulic system
< I > INFORMATION      Reading hydraulic data Geometry
< I > INFORMATION      Reading hydraulic data H-component specifications
At  0.0000    s PUMP    P1      Pump specific speed = 76.7
At  0.0000    s PUMP    P2      Pump specific speed = 76.7
At  0.0000    s PUMP    P3      Pump specific speed = 76.7
At  0.0000    s PUMP    P4      Pump specific speed = 76.7
At  0.0000    s PUMP    P5      Pump specific speed = 58.5
At  0.0000    s PUMP    P6      Pump specific speed = 58.5
< I > INFORMATION      Hydraulic steady state computation
< I > INFORMATION      New start vector for STEADYSOL
                                0
                                1
                                2
                                3
                                4
                                5
                                6
< I > INFORMATION      Steady state solution found

```

After computation these messages can be viewed using *View steady messages...* from the Model menu.

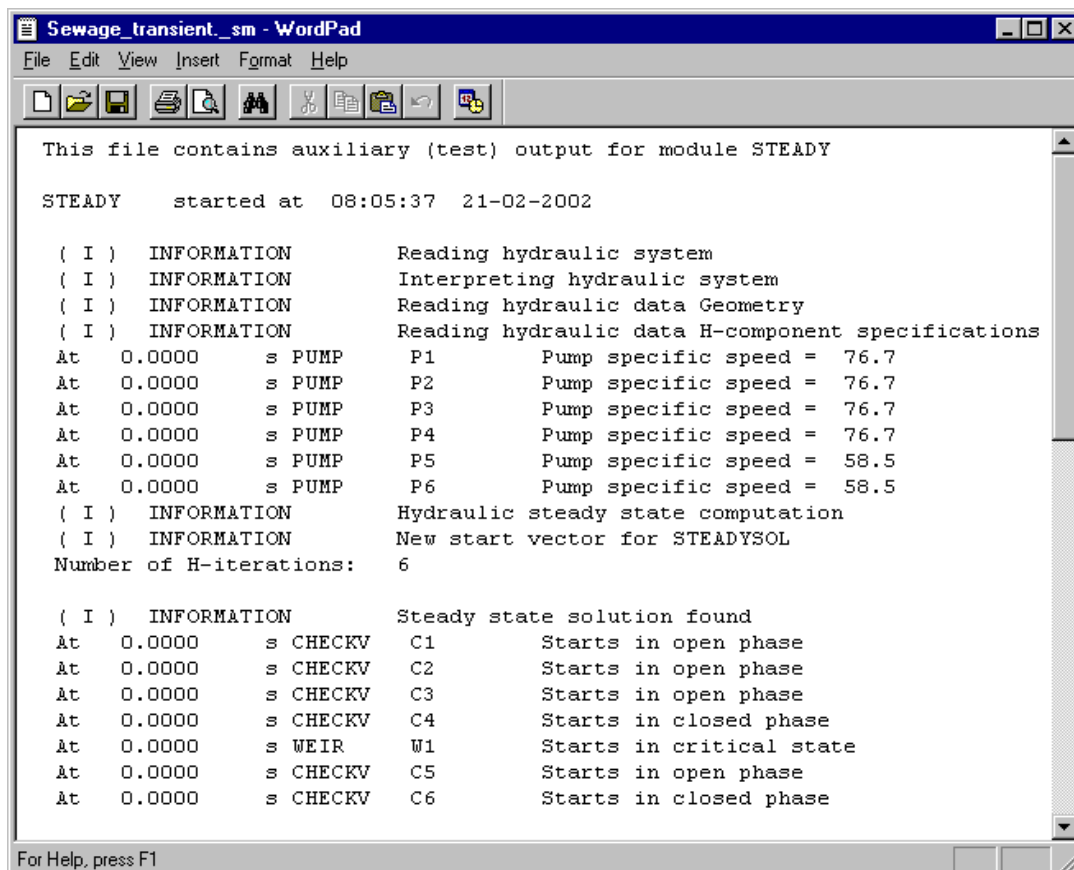
Steady-state flow equations are non-linear equations, and are solved iteratively. A set of initial values of the flow variables (a start vector) is used for the iteration. When no convergence is found using one vector, the computation continues with a new start vector. A total number of 9 start vectors are designed in the module. In case they are exhausted and no convergence is found yet, WANDA terminates the computation and issues an error message. Details about the mathematical model and numerical methods used to solve non-linear equations can be found in the section Theory behind WANDA 4 (General form of the equations).

Messages generated by the module can be classified into five categories. The computation is aborted in case of a terminating error message. Certain categories of messages can be suppressed in the display window using the message threshold in the general computation data.

2.14.8.2 Calculate steady all cases

It is possible to open several different cases. By choosing this option all open cases will be calculated.

2.14.8.3 View steady messages...



```

This file contains auxiliary (test) output for module STEADY

STEADY   started at  08:05:37  21-02-2002

( I )   INFORMATION           Reading hydraulic system
( I )   INFORMATION           Interpreting hydraulic system
( I )   INFORMATION           Reading hydraulic data Geometry
( I )   INFORMATION           Reading hydraulic data H-component specifications
At  0.0000   s PUMP           P1           Pump specific speed = 76.7
At  0.0000   s PUMP           P2           Pump specific speed = 76.7
At  0.0000   s PUMP           P3           Pump specific speed = 76.7
At  0.0000   s PUMP           P4           Pump specific speed = 76.7
At  0.0000   s PUMP           P5           Pump specific speed = 58.5
At  0.0000   s PUMP           P6           Pump specific speed = 58.5
( I )   INFORMATION           Hydraulic steady state computation
( I )   INFORMATION           New start vector for STEADYSOL
Number of H-iterations:      6

( I )   INFORMATION           Steady state solution found
At  0.0000   s CHECKV         C1           Starts in open phase
At  0.0000   s CHECKV         C2           Starts in open phase
At  0.0000   s CHECKV         C3           Starts in open phase
At  0.0000   s CHECKV         C4           Starts in closed phase
At  0.0000   s WEIR           W1           Starts in critical state
At  0.0000   s CHECKV         C5           Starts in open phase
At  0.0000   s CHECKV         C6           Starts in closed phase
  
```

Above you see an example of the messages after a steady computation of your model. WANDA 4 shows these messages in the Notepad. In this way it is very simple to save the messages as a text document and use them for reporting purposes.

2.14.8.4 Calculate transient

Calculate transient in WANDA is used to compute the unsteady state in the pipeline system.

The progress in the transient state computation is shown. Program messages are also displayed.

Messages generated by the module can be classified into five categories. The computation is aborted in case of a terminating error message. Certain categories of messages can be suppressed in the display window using the message threshold in the general computation data.

Calculate transient generates two files. One of the files has the extension "__R", and is an interface file between the module STEADY and TRANSIENT. The other file, with

extension "_UM", is the message file where all messages generated by module TRANSIENT and the component procedures are stored.

The computation can be stopped at any time and resumed later (see "Resume transient" on page 118).

2.14.8.5 Stop transient

This option interrupts the transient calculation. Results of the simulation so far can be analysed. The user may resume an interrupted transient calculation using the menu option Resume transient.

2.14.8.6 Resume transient

This option continues the transient calculation.

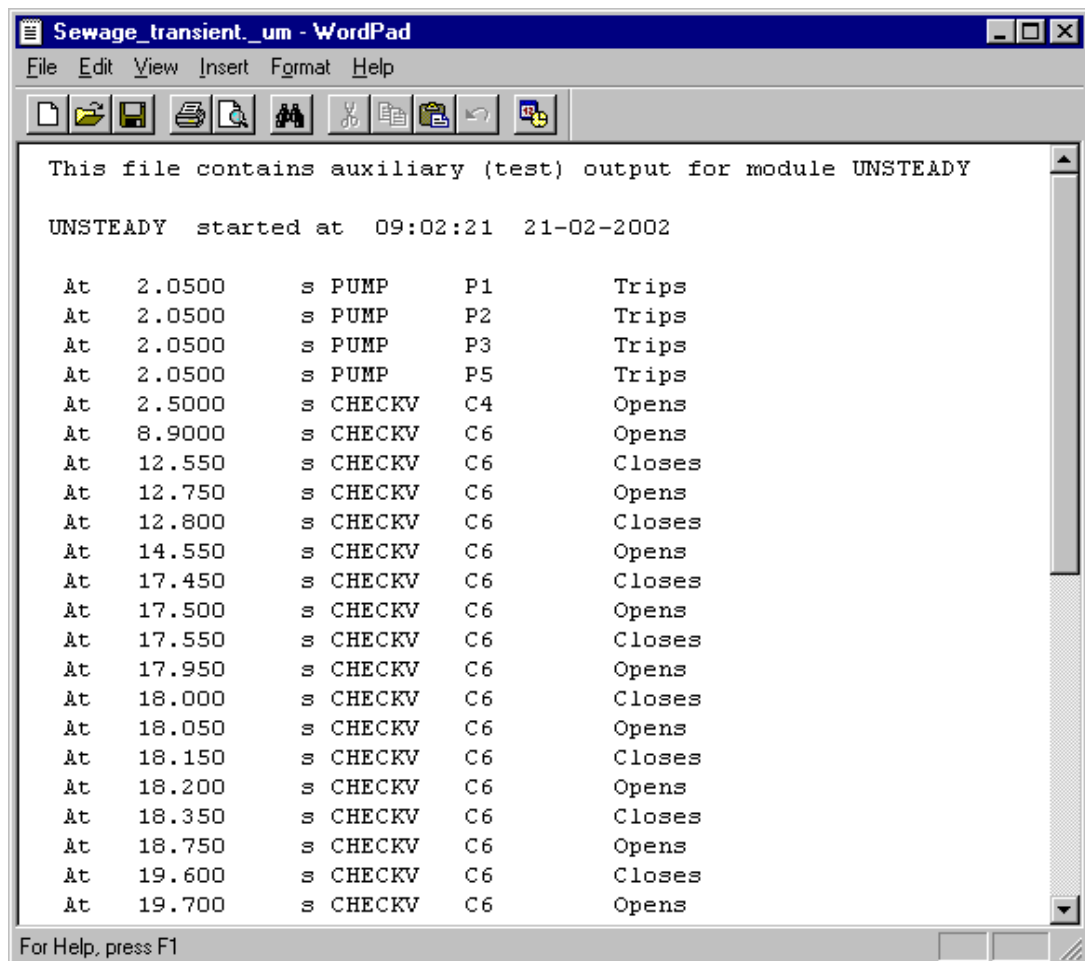
2.14.8.7 Restart transient from

Allows the user to choose a restart point and restart the transient simulation from that point.

2.14.8.8 Calculate transient all cases

It is possible to open several different cases in WANDA 4. By choosing this option, all open case will be calculated. This option is useful if you want to calculate several large models (e.g. by night). Verify that you have sufficient disk space available. You can adjust the size of the output file without losing calculation accuracy by setting a time output increment in the Time parameters window.

2.14.8.9 View transient messages...



```

This file contains auxiliary (test) output for module UNSTEADY

UNSTEADY started at 09:02:21 21-02-2002

At 2.0500 s PUMP P1 Trips
At 2.0500 s PUMP P2 Trips
At 2.0500 s PUMP P3 Trips
At 2.0500 s PUMP P5 Trips
At 2.5000 s CHECKV C4 Opens
At 8.9000 s CHECKV C6 Opens
At 12.550 s CHECKV C6 Closes
At 12.750 s CHECKV C6 Opens
At 12.800 s CHECKV C6 Closes
At 14.550 s CHECKV C6 Opens
At 17.450 s CHECKV C6 Closes
At 17.500 s CHECKV C6 Opens
At 17.550 s CHECKV C6 Closes
At 17.950 s CHECKV C6 Opens
At 18.000 s CHECKV C6 Closes
At 18.050 s CHECKV C6 Opens
At 18.150 s CHECKV C6 Closes
At 18.200 s CHECKV C6 Opens
At 18.350 s CHECKV C6 Closes
At 18.750 s CHECKV C6 Opens
At 19.600 s CHECKV C6 Closes
At 19.700 s CHECKV C6 Opens

```

Above you see an example of the messages after a transient computation of your model. WANDA 4 shows these messages in Notepad. In this way it is very simple to save the messages as a text document and use them for reporting purposes.

2.14.9 Run Parameter Script

2.14.9.1 Concept

In many projects, different scenarios (e.g. flow rate, operating pressure etc.) and settings (water level in surge vessel, valve closure speeds etc.) need to be simulated. Often it is useful to start from the same original (mother) case. The Parameter Script in Wanda is the tool to use for this kind of projects.

The concept of the Parameter Script is that there is one (original) mother case, from which several child cases are created and run. For every child case the user defines the input which needs to be modified. To enable quick review of the results, extreme output values for every child case can be saved in a CSV-file. For further details, the user can open the child cases of interest. Since the Wanda 4.2 release, an Excel tool is included to simplify the use of the Parameter Script. In this Excel file, all input values and required output results are listed. The advantage of using this Excel file is that:

1. It is easy to distinguish between different cases and manage a sensitivity analysis.
2. It is easy to open the different child cases, since for every child case a shortcut is added.
3. All internal Excel functions can be used to create the values for the input parameters.

Although the use of Excel is very convenient, the Parameter Script can also be used with any text editor.

2.14.9.2 Main steps

In the Wanda example folder (“File -> Open Example -> Parameterscript”) an example of the Parameter Script can be found (“Sewage_transient_WPS.wdi”).

To use Parameter Script the following steps need to be taken:

- Create a mother case using the Wanda user interface.
- Create a Parameter Script file: “MyCase.wps”. This can be done with help of the parameter Excel file, which can be found in Wanda4/tools folder, or by manually creating a “Mycase.wps” file by using a text editor like Notepad. Note: the name of the Parameter Script must be equal to that of the mother case file.
- Start the Parameter Script via the pull-down menu in Wanda (Model menu -> ‘Run Parameter Script’).
- Review the results in the CSV-file. Child case can be opened in the Wanda user interface, when the option for saving the child cases is selected.

2.14.9.3 WPS file creation

The WPS file can be created via an Excel file. The Excel file consists of three sheets: *Cases*, *Output* and *Readme*. The latter contains the same information as given here and is added as quick reference. Below an explanation of the other two sheets is given.

Cases sheet

The Cases sheet contains all input information, like the parameters to change and the value of these parameters.

The top three rows are the header rows. From row four downward, information for every child case is given.

First column “Case number” and second column “Case description”: These rows can be used to support organize the Parameter Script. Any text string or number can be used but it can also be left empty.

Third column “Include”: It is used to select the case which should be (1) included in the Parameter Script run and which not (0).

In column four “*Name*”, the mother case can be saved and the extension for every child case is given (the child case name becomes: mothercase_childcase).

In the columns after the Name column, the input parameters are given. In the top row the Name of the component (example “*CHECKV C5*”) or a Keyword is given. In the second row the Property to change (“*Initial state*”) is given. This is the English Property and case sensitive.

The set-up of the parameter Script excel file gives the user a lot of freedom for extensions (and deletions). Almost all columns may be deleted, moved and copied. However, the two columns “Include” and “Name” may not be deleted.

Please note the following:

1. All input value should be given in the unit system of the mother case.
2. It is also possible to use Excel functions to calculated values or to get them from different sheets (e.g. VLOOKUP function) if required.
3. Components can only be disused and not used. Meaning: all components which are listed in the in- or output should be used in the mother case.
4. Be careful in using disuse, since this can cause the Wanda model to be invalid and resulting into a crash of the calculations.
5. The length of a pipeline should not be changed, since this will result in unexpected output and possibly a crash of Igrafix.
6. Using the keyword: GENERAL, the fluid and physical properties can be assessed, just as the simulation time parameters, and the simulation accuracy (see also: Model menu – Fluid Window / Physical Constants Window / Time parameters Window / Accuracy Window).
7. Drop down list (e.g. “*Initial state*”) can also be changed. The number of the item in list requested should be given as input value (e.g. “4” should be used if the valve should be set to initial discharge).
8. Tables (local ksi losses, pipe profile, action table, QHE-characteristics) cannot be adjusted with the ‘Parameter Script’.
9. To run Parameter Script again, check whether the result-file (csv) is closed.

Output sheet

The Output sheet contains three columns. The first column gives the name of the component or Keyword (e.g. “*PG1*”) for which an output should be saved. The second column contains the Property for which the output should be saved. The third row states whether the minimum (“*MIN*”) or maximum (“*MAX*”) should be saved. Next to this the following SAVE keywords can be given:

- **SAVECASE**: for each child case the .wdi-, .wdo-, .wdx--file is saved. These child-cases can be opened with Wanda.
- **SAVEMSG**: for each child case the ._sm-file (see: Model menu – View Steady Messages), and the ._um file (see: Model menu – View Transient Messages) is saved.
- **SAVEALL**: combination of SAVECASE, SAVEMSG.
- **REBUILD_CSV**: The CSV file is rebuilt from the existing child cases. This is useful when other output is required after the simulations have been performed.

Creation of the WPS file

After the Excel sheet is filled, click on the button "Create wps" in the Cases sheet. Please note that macros need to be enabled. Follow the instructions. At the end a WPS file is created. This file consists of 3 blocks:

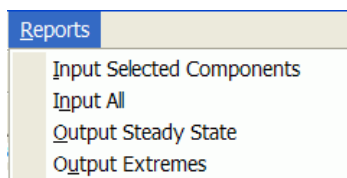
- **Input parameter definition:** defining the parameters that the user wants to change; via the Keywords of the objects and the Property description.
- **Output results definition:** defining the results of each individual child case that the user wants to have saved; via the keywords of the objects and the property description. Using the SAVE-keywords the child cases themselves can be stored.
- **Run definition:** lists for each child case: The name of the child case and the input values of the input parameters as defined in the Input parameter definition.

Every group starts with the number of in-, outputs or cases which are given. The example below shows this syntax.

The final step is starting the parameter script from the Wanda User interface (Model -> Run Parameter Script) After the Parameter Script runs are finished, the csv-file with the results is opened automatically. Depending on the chosen option for saving of simulations, the child cases can be inspected (or not).

```
* Input definitions:
9
"GENERAL","Simulation time"
"PG1","Initial setting"
"PG1","Initial speed"
"PG1","Initial discharge"
"PG2","Initial speed"
"CHECKV C5","Initial state"
"Signal ESD","Disuse"
"FPT","Disuse"
"MSP C1","Ramp 1"
* Output definitions:
6
"SAVEALL"
"PG1","Pump speed","MAX"
"PS-1 to plant","Pressure","MIN"
"PS-1 to plant","Pressure","MAX"
"PS-2 to plant","Pressure","MIN"
"PS-2 to plant","Pressure","MAX"
* Case/run definitions:
6
'case1',20,1,580,0,735,1,0,1,0.01
'case2',20,4,580,3000,0,2,0,1,0.01
'case3',200,1,580,0,735,1,1,0,0.01
'case4',200,4,580,3000,0,2,1,0,0.01
'case5',20,1,580,0,735,1,1,0,0.1
'case6',20,4,580,3000,0,2,1,0,0.1
```


2.15 Reports Menu



The Reports Menu allows you to save and edit all input and output of your hydraulic model, for the purpose of reporting.

After the hydraulic model has been created and the input data have been entered, the user may need to have a full report of the input data. The report can be used either for inspection of the numerical values of the input data, or for documentary purposes.

Likewise the user wants a summarized report of the output after a successful calculation

WANDA generates the reports in a compact spreadsheet format (tab separated ASCII file). After generating WANDA starts automatically the default spreadsheet program associated with "csv" extension. The spreadsheet output can be further arranged outside the Wanda program.

The components are ordered by name. Each row contains all input or output parameters. Because a cell can contain only scalars, the tables with input items are displayed separately. A unique table identification is used for reference.

2.15.1 Input selected components

The input report contains only the input specifications of the components selected by the user. This means, that if you want a report on a special group of components you have to select them by using the selection builder (ALT+F7) or the selection menu.

All other input data such as fluid properties, network connectivity is ignored.

2.15.2 Input all

A full input report contains the following information:

- case descriptions
- general computational data
- fluid data
- hydraulic properties of all components, including tables
- network connectivity

This option provides an overview of the input of all data in the model. This report can be used for backup purposes (printed hardcopy of your model input).

Pump	User name	Modified	Model name	Inner diam (mm)	Loss coefficient (-)	Initial state	Keywords	Comment
P1	Deltares	2008 Dec 30 9:27	POV_300a	500	0.01	Open		
P2	Deltares	2008 Dec 30 9:27	POV_300a	500	0.01	Open	PS-1 to plant	
P3	Deltares	2008 Dec 30 9:27	POV_300a	500	0.01	Open		
P4	Deltares	2008 Dec 30 9:27	POV_300a	500	0.01	Closed	close	
P5	Deltares	2008 Dec 30 9:27	DMD#250	250	0.01	Open	PS-2 to plant	
P6	Deltares	2008 Dec 30 9:27	DMD#250	250	0.01	Closed	close	

2.15.3 Output steady state

This output report shows all calculated properties of all components of the steady state computation.

Because the components are ordered by type, you get a compact table of standard output (depends on the physical domain) and component specific output.

Because the component messages are text tables, they are displayed separately. If you want a chronological list with messages, access the Steady messages report (model menu).

Checkvalve	Messages	Discharge (m3/h)	Head 1 (m)	Head 2 (m)	Pressure 1 (bar)	Pressure 2 (bar)	Velocity 1 (m/s)	Velocity 2 (m/s)	Head loss (m)	Pressure loss (bar)	Volume (m3)
C1	TABLE 5	3769	6.70364	6.68915	0.6921	0.6907	5.332	5.332	0.014488	0.001421	0
C2	TABLE 5	3769	6.70364	6.68915	0.6921	0.6907	5.332	5.332	0.014488	0.001421	0
C3	TABLE 5	3769	6.70364	6.68915	0.6921	0.6907	5.332	5.332	0.014488	0.001421	0
C4	TABLE 6	0	-2.5	6.68915	-0.06867	0.8328	0	0	-9.18915	-0.9015	0
C5	TABLE 5	330	12.8399	12.8381	1.409	1.409	1.867	1.867	0.001778	1.75E-04	0
C6	TABLE 7	0	-1.9	12.8381	-0.01962	1.426	0	0	-14.7381	-1.446	0

2.15.4 Output extremes

This output extremes report shows all minimum and maximum calculated properties of all components after the simulation has been executed. Also the point of time when the extreme value occurs, is included

Because the components are ordered by type, you get a compact table of standard output (depends on the physical domain) and component specific output.

Because the component messages are text tables, they are displayed separately. If you want a chronological list with messages, access the Transient messages report (model menu).

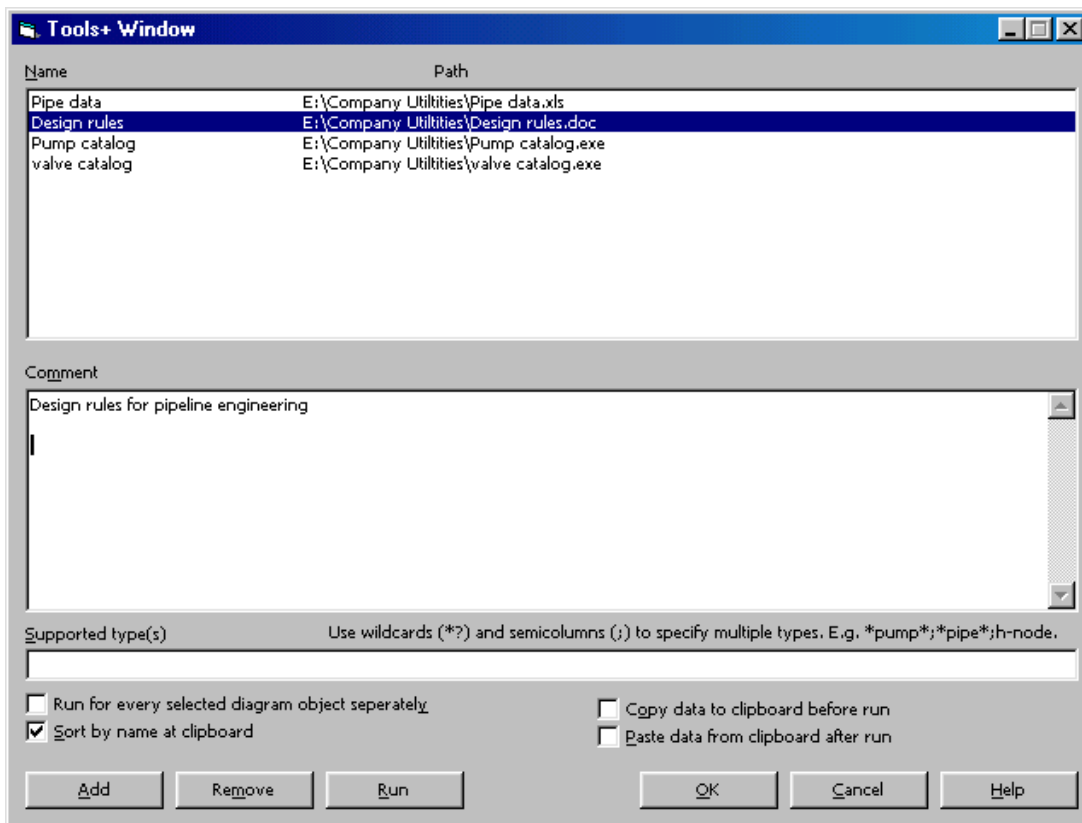
	A	B	C	D	E	F	G	H	I	J	K	L	M	N	
1	WANDA Transient 4.0.954														
2	Licensee:														
3	Pump	Messages	Discharge 1				Discharge 2				Head 1				
4			min	max	T min	T max	min	max	T min	T max	min	max	T min	T max	
5			(m3/h)	(m3/h)	(s)	(s)	(m3/h)	(m3/h)	(s)	(s)	(m)	(m)	(s)	(s)	
6	P1	TABLE 1	1953	3769	0	0	1953	3769	0	0	0	-2.5	-2.5	0	0
7	P2	TABLE 1	1953	3769	0	0	1953	3769	0	0	0	-2.5	-2.5	0	0
8	P3	TABLE 1	1953	3769	0	0	1953	3769	0	0	0	-2.5	-2.5	0	0
9	P4	TABLE 2	0	1723	5.6	5.6	0	1723	5.6	5.6	0	-2.5	-2.5	0	0
10	P5	TABLE 3	77.26	330	0	0	77.26	330	0	0	0	-1.9	-1.9	0	0
11	P6	TABLE 4	0	80.64	17.3	17.3	0	80.64	17.3	17.3	0	-1.9	-1.9	0	0
12															
13	BoundH (res)	Discharge					Head				Pressure				Volume
14			min	max	T min	T max	min	max	T min	T max	min	max	T min	T max	min
15			(m3/h)	(m3/h)	(s)	(s)	(m)	(m)	(s)	(s)	(bar)	(bar)	(s)	(s)	(m3)
16	B1		6919	11310	0	0	-2.5	-2.5	0	0	-0.06867	-0.06867	0	0	0
17	B3		77.26	330	0	0	-1.9	-1.9	0	0	-0.01962	-0.01962	0	0	0
18															
41	TABLE 1 Messages														
42	Type	Time	Message												
43		(s)													
44	Info	0	Pump specific speed = 76.7												
45	Info	2.1	tripping												
46															

2.16 Tools+ menu

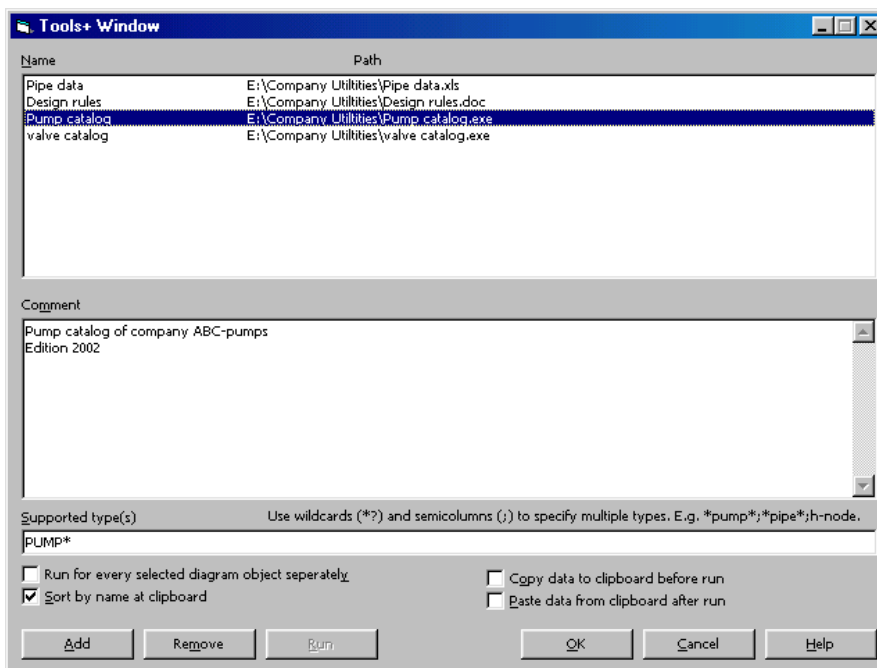
2.16.1 Knowledge management

Knowledge management is a hot topic. Disclosure of knowledge is of significance also to the WANDA user. WANDA is often used by several engineers, all having their own field of expertise. This expert knowledge of the use of WANDA or, in general, of pipeline-hydrodynamics can now be saved directly in the WANDA environment. This way it is available to everybody at all times. The new menu *Tools+* make access possible to all available (company) tools to link information to a specific (group of) component(s). A tool can be a 'simple' text file in Word or HTML, but also a complete spreadsheet in Excel or a program developed in-house. For example, an online databank with information on material properties can be accessed from WANDA. This way supporting information or input is easily found by another engineer. Furthermore, with a small programming effort, it is possible to exchange data between WANDA and external applications. Results from a home-made Excel spreadsheet, for example, can form the input for WANDA, and vice versa.

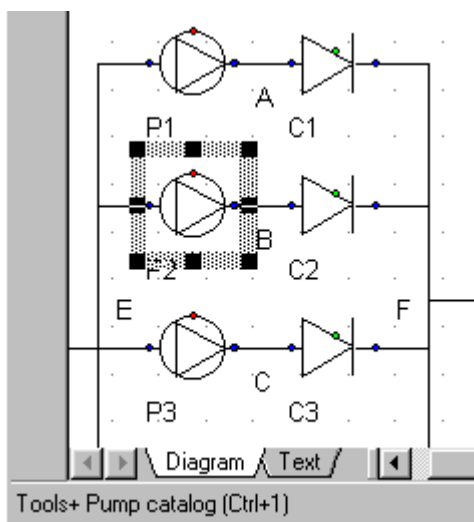
Via menu Tools+ you can open the Tools Window. In this window you can add certain tools to certain (groups of) components. In the comment field you can, for example, give an additional comment on the tool you have linked.



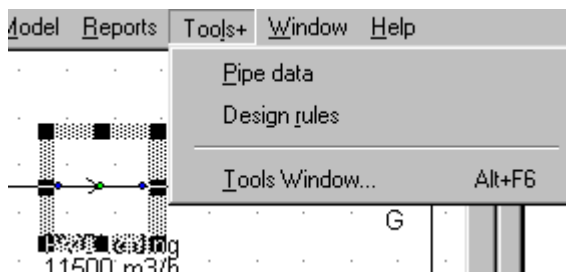
In the field “Supported type(s)” you can specify a (group of) components to which this tool is related.



In the diagram taskbar the available tools, dependant on the selected components, are shown. With the displayed hot-key combination the tool is activated.



The tool can also be accessed via the Tool+ menu



2.17 Pump Energy

2.17.1 Overview Pump Energy module

The Pump Energy Module is part of WANDA Engineering. With this module you can calculate the most efficient use of pumps in a pumping station based on a specified delivery pattern and the system characteristic. The program gives you the total energy consumption in a specified period regarding the specified pump operation.

The Pump Energy module is not a separate window but is a combination of a few components. These components are explained in detail in the Help on "Liquid components" on page 187.

Hydraulics components that must/can be used in Pump Energy are:

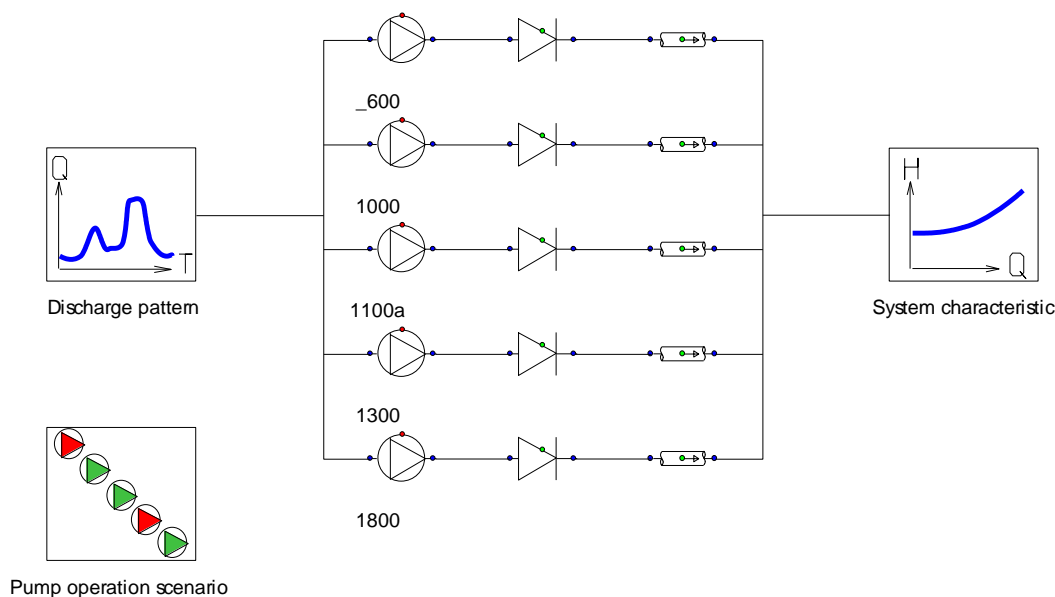
- PUMP the pump model
- PUMPSZEN Pump scenario to specify the operation of pumps
- SYSCHAR System characteristic

- Q_FREQ Discharge Frequency
- Q-PATTRN Discharge Pattern

One or more of these components are placed in the diagram. The module needs only one SYSCHAR and one or more PUMPSCENs and PUMPs. The use of Q_FREQ and Q_PATTRN is optional. These components define the time; without one of these components no energy consumption can be calculated. The system characteristic is a Q-H table defined by the user; it may be retrieved using the System Characteristic module. In that case the curve data must be copied (using the clipboard) from the system characteristic chart to the SYSCHAR input table.

2.17.2 Schematic aspects

The figure below shows an example of a diagram to perform a Pump Energy computation.



The group of pumps represents the pumping station; other components like valves, checkvalves and pipes may occur in the system but doesn't have any influence.

Only one (not-disused) SYSCHAR must be present; More than one PUMPSCEN is allowed, so you can compare different scenarios at once. The connections between the different components are not necessary; the PUMPSCEN component carries out the computation based on the specified data of the active PUMP's, SYSCHAR and (optionally) Q_FREQ/QPATTRN. The computation will be done directly after an input is done; that means that changing a parameter, the new results are directly available.

Inconsistency in the diagram or input is communicated by the message property of PUMPSCEN.

The PumpEnergy components PUMPSCEN, Q_PATTRN, Q_FREQ and SYSCHAR are ignored by the WANDA processes "Steady" and "Unsteady"

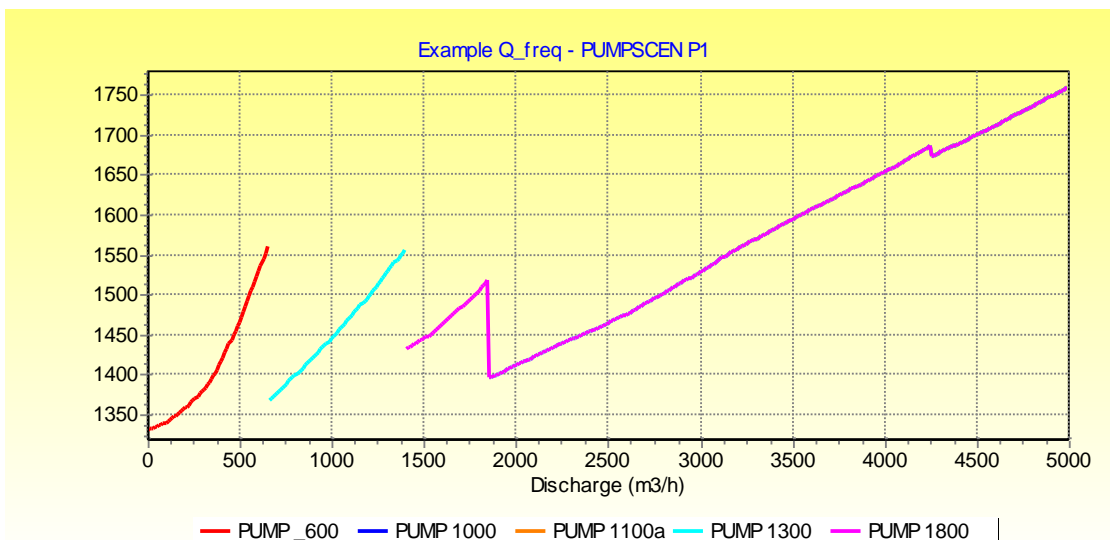
The dialog between user and program is done by the diagram: the pumps that must be evaluated are located in the scheme. Only the input fields QHE table and rated speed are necessary, the other values may remain empty.

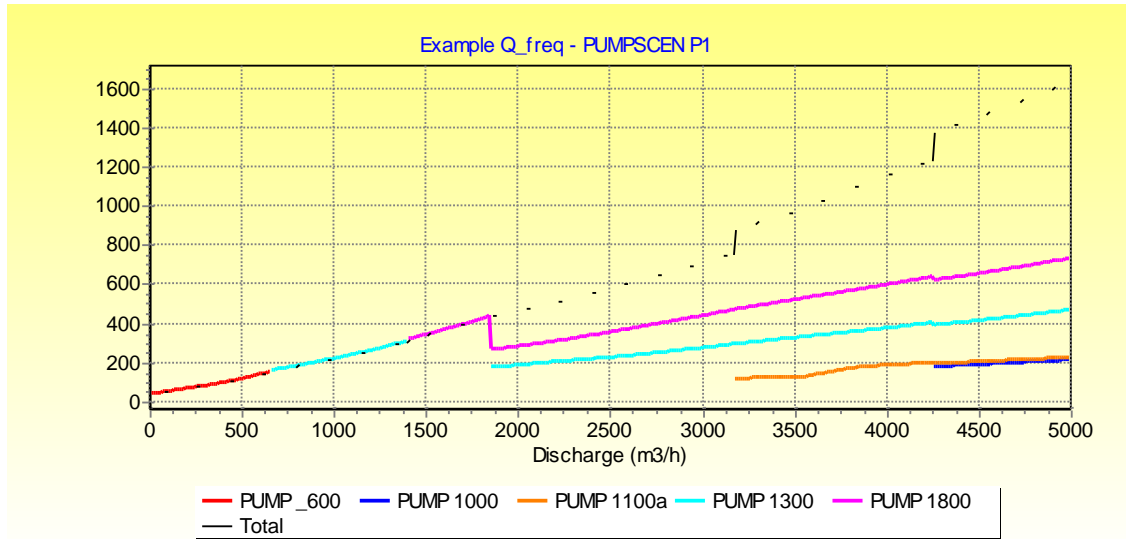
In the PUMPSZEN component a scenario of operation is defined. The scenario table contains all (not disused) pumps.

	Q from (m3/h)	Q to (m3/h)	PUMP 1000 (rpm)	PUMP_600 (rpm)	PUMP 1100a (rpm)	PUMP 1300 (rpm)	PUMP 1800 (rpm)
1	0	650.0	0				
2	650.0	1400				0	
3	1400	1850					0
4	1850	3150				1500	0
5	3150	4252			1500	1500	0

The user has to specify in which discharge range a pump or group of pumps are in operation. The pumps are speed controlled (a "0" is specified) or runs at a fixed speed (the concerning speed is specified). In above example the first 3 stages are operated with variable speed pumps, 1 different pump for each different case. The next stages are operated with one or two pumps at fixed speed and one at variable speed.

The PumpEnergy module calculates for the whole range of discharges the efficiency and power of each pump. In case of variable speed pumps, the necessary speed is calculated to satisfy the duty point according the system characteristic. All important pump properties, like discharge, head, speed, efficiency and power, can be visualised.

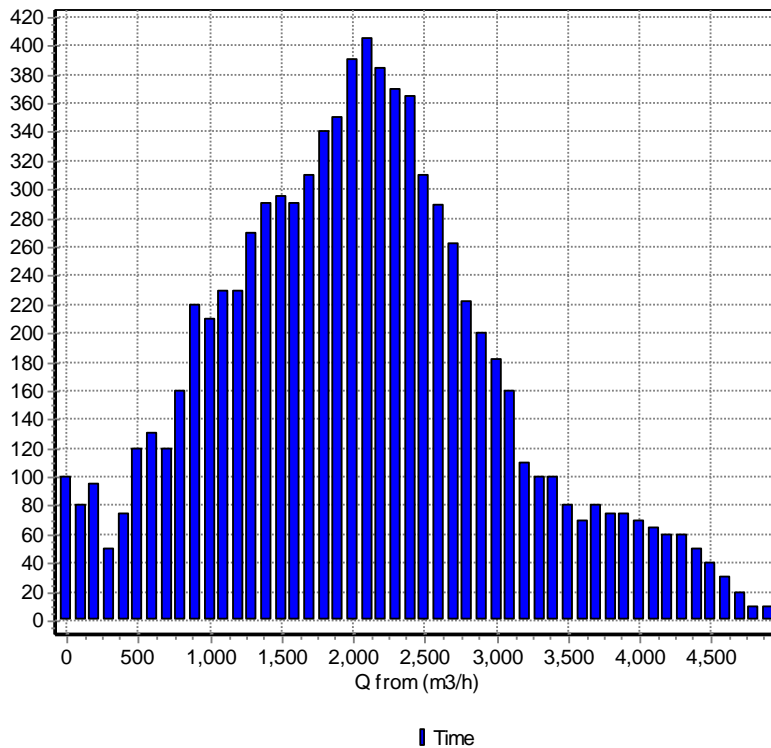




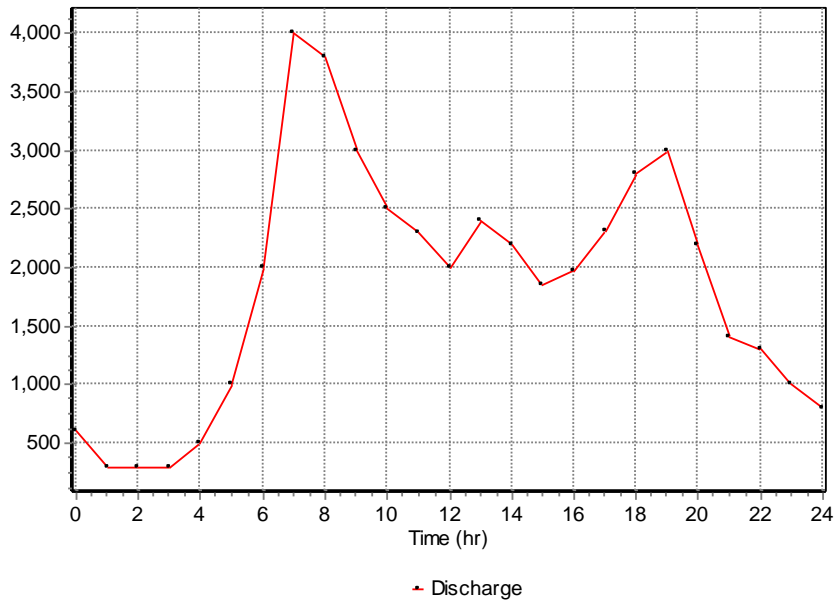
If we know how many hours the pumps operate, the energy consumption can be determined.

The time aspect can be specified in two different ways:

- a histogram annotation that tell us how many hours a certain discharge is delivered in a certain period (Q_FREQ)



- a discharge pattern as function of time (Q_PATTRN)



The energy consumption is displayed in the scenario table and can be visualised in a chart

	Q from (m3/h)	Q to (m3/h)	PUMP_600 (rpm)	PUMP 1000 (rpm)	PUMP 1100a (rpm)	PUMP 1300 (rpm)	PUMP 1800 (rpm)	Energy (MWh)	Energy ratio (%)
1	0	650.0	0					53.20	1.18
2	650.0	1400				0		354.3	7.88
3	1400	1850					0	516.0	11.5
4	1850	3150				0	0	2242	49.9
5	3150	4250			0	0	0	961.9	21.4
6	4250	5000		0	0	0	0	367.1	8.17

More information about the individual Pump Energy components is explained in the Help on "Liquid components" on page 187.

2.18 Tips for WANDA 4 users

2.18.1 General tips

- Read the manual.
- **Always** make a (temporary) local copy of a Wanda case stored on a network server or removable drive. Data is loaded dynamically so the access time determines the speed of Wanda. The access time of a local drive is about 100 times faster.
- Use the context sensitive menu (right mouse click) in the diagram and property window to get fast access to frequently used options.
- You can open a Wanda file from a file manager by double clicking the wdi-file or dragging a wdx-file into the Wanda diagram window.
- You also can drag a file to the windows taskbar. Keep the mouse button down until Wanda becomes visible and move the dragged file in the diagram window.
- You can use your favourite text editor (Messages, Reports) by associating the txt-extension to your favourite text editor. ('View.Options.File types' menu item in the windows file explorer).
- Use the same model name for components that share the same specifications (properties).
- Use keywords to group a set of Components.
- The information in every window reflects the current situation. You don't have to re-open windows to update them.
- If you are not interested in the output at every single time step check the 'Show Extremes' option in the Model menu.
- Press F11 to switch the focus between the property window and the diagram window.
- Wanda works best at a resolution of 1280x1024 and a 19" monitor or better.
- Sometimes the diagram doesn't become visible when you click the taskbar. Right click the Wanda button at the taskbar and choose restore.
- You can resume transient with a different output increment.
- Make property templates with fluids that you use frequently.
- You can delete the wdo-files of cases to create more disk space. When you delete the wdo file of a case only the output is deleted. If you send a case by mail create a zip file with the wdi and wdx file. These three files are the only files needed.

- Learning the keyboard and mouse shortcuts will save time when you are using Wanda frequently.
- If you have additional tips lets us know.

2.18.2 Shape libraries

- You can customise the Wanda Hydraulic and Wanda Control shape library.
- You can add extra custom shape libraries.
- DON'T DRAG WANDA COMPONENTS FROM THE DIAGRAM TO THE SHAPE LIBRARY.
- You can copy shapes from one shape library to another. Use the right mouse button to cut/paste.

- Copy shapes between shape libraries **one by one**. If you don't, Wanda will crash due a bug in iGrafx.
- It's easier to collapse the gallery than closing the gallery. Use the button '>' left from the 'X'
- You can resize the buttons in the gallery (shape libraries) by moving the right-down corner of a single button.

2.18.3 Diagram general

- The mouse cursor indicates what happens when you click (Selecting, Moving, Dragging a line, Inserting a new component).
- Hold the SHIFT-key down while clicking an object to extend a selection by a single object.
- Hold the SHIFT-key down while clicking a **selected** object to shrink the selection by a single object.
- You can drag a selection rectangle to select multiple objects at once.
- You can extend a selection by multiple objects at once by dragging a rectangle while the SHIFT-key is down.
- If you change the line style / weight / colour while no objects are selected the style is applied for new lines and shapes.
- You can visualise a property in the diagram by clicking its checkbox in the property window.
- If you toggle the menu item 'show extremes' the output properties in the diagram are also changed (delay 3 seconds).
- If you move the slider in the time navigator the output properties in the diagram are also changed (delay 3 seconds).
- Use a postscript printer driver when the diagram or chart is not printed correctly. (HP delivers also a postscript driver for all it's laser printers).
- Right click at a component or connection to popup extra property windows.
- Deselect all objects before printing to prevent red circles behind objects.
- Before you copy the diagram selection to another application (e.g. MS Word) first press the 'A' button in the property window. This prevents that a red circle behind a component is also copied.
- Some Nvidia video drivers don't work properly when using iGrafx professional. Wanda crashes during starting or directly after adding a component. Download the newest video driver from the Internet. It's still possible that background pictures hide the diagram partly. Use a separate layer for the pictures and hide this layer during editing.
- By pressing CTRL+F6 you can quickly switch between open diagrams.
- To navigate in large scheme's, use the split window mode. The upper part is used for navigation, the lower part is used for editing.

2.18.4 Diagram editing

- If you are dragging a selection you can cancel this operation by pressing the escape button before releasing the mouse button.
- Hold the CTRL-key down if you want to copy a selection instead of moving a selection.

- You can draw components like vectors (Wanda 2 functionality). Select from the shape library the component type you want to draw. Press the shift key and keep it down. Right click the diagram background to switch to the line draw mode. Draw a line. The selected component in the shape library is now automatically inserted in the middle of the new line. The H-nodes (connection lines) at both sides (only upstream side in case of a supplier type) are automatically generated. When you are finished with vector drawings you can release the shift button.
- You can hide/unhide the grid dots with the sub menu "Grid" in the "Arrange" menu.
- Editing is easier when you first 'snap to grid' all components. If you convert a Wanda 2 case the diagram size is reduced with 25 percent. Due to this reducing the objects are not 'snapped to grid'.
- If you want to connect a connection line in the middle of another connection line which is very short (less than 1 cm), the green blinking connect point snaps automatically to the end points. Accept this location while the WANDA program moves it to the centre of the line.
- If you want to move a text (e.g. name) of a component/node first select the tab "Text" in the diagram window.
- Don't move a selection to *another* document.
- Copy the selection first and delete the original selection afterwards.
- You can align components by using the blue rectangle. When you move a component across another component this component is marked with a blue rectangle. Keep the mouse button down and move the component to the wanted location. When you release the mouse button the component is snapped to the position that's exactly aligned with the blue marked component.
- You can shift + right-mouse-click at the diagram background instead of pressing the line tool button. Press twice to toggle between *direct line* and *right angle line*
- You can ctrl + right-mouse-click at the diagram background instead of pressing the last selected shape at the shape library.
- By adding extra vertexes to a connection line you can route the line to another path. You can add extra vertexes to connection lines by pressing the right-mouse button while you hold the SHIFT-key down.
- If the diagram isn't displayed correctly or the selection isn't the same as in the property window press F5 (Refresh) while the diagram is active.

2.18.5 Diagram layers

- Use a separate layer for pictures (backgrounds). You can quickly hide this layer to increase the scroll/zoom speed and reduce flickering (layer manager, activated by right click at layer tab). The background must be the first in the list (use move up button), otherwise the components are not visible.
- Move (a group of) disused components to a different layer. You can toggle the printable option when you right-click the tab. When the option is unchecked the layer is not visible when you print the diagram.
- If you want to edit multiple layers choose edit all layers and lock the layers you don't want to edit.
- Right click at a layer tab to see a context menu. With this menu it is easy, among other things, to hide and unhide layers. It's important that the layer-tab that you want to hide is not active.

2.18.6 Diagram routes

- A route is an unambiguous and continues sequence of H-components; this route can be specified with a keyword.
- You don't have to select all the component in your route one by one. Select the first component in your route. Now select the last component in your route while holding the Ctrl and Shift key down. The other components and connections in the route are now automatically added to the selection. It's important that the components are connected before you select the route.
- If the route Wanda chooses isn't the way you had in mind you can select extra components before you select the last component by Ctrl+Shift+Click. Now Wanda is forced to choose a certain path. Keep the Shift key down while you select the extra components.
- Use the 'align route' menu option or (CTRL+ALT+A) to align a route in a fast way to another direction. First move component A to the wanted position, then move component Z to the wanted position. Choose 'align route' to redirect all components between A and Z in a straight way.

2.18.7 Property windows

- If you select multiple components/nodes you can inspect multiple properties by pressing the "range" or "spreadsheet" button in the property window.
- To select/deselect properties in a property window, press the space bar, click at the description or drag the mouse in the input field.
- You can drag/drop selected properties from one property window to another property window.
- Every component/node also has hidden properties. With the hidden properties you can see the connectivity used by steady and transient. To unhide the invisible properties press the pause key while a property window is active (may be useful in case of 'strange' problems).
- If you are editing a text property you can insert a line break by pressing ALT+ENTER (same way as in Excel).
- You can quickly move to a property in the property window by pressing CTRL+ALT+"First character property name".
- If you want to edit a part of a property value first press F2 or enter to become in edit mode.
- You can open a table or chart window by pressing enter at the particular property.
- Use the menu items 'Add keyword to selection' and 'Remove keyword from selection' to update the keyword list when you have a multiple selection.
- Export your frequently used properties of components to property templates. Now you can easily reuse components between cases (or colleagues).
- You can load a property template file from the file manager by dragging the file into the info/unit section of a property window (I-button must be pressed).
- You can see the Wanda help while editing the properties by setting the online help 'always on top'.

2.18.8 Tables and tabular views

- To show a tabular view, select one or more properties (marked as blue) in the property window and press the "spreadsheet" button.
- You can print a table or tabular view by a right double click at the grid. An Excel like window appears. Choose 'Page setup.' in the file menu to fit the print to n by m pages (landscape or portrait).
- While you are defining the pipe profiles of the pipes in a route you can open the Head chart to see the profile of the complete route. This chart is updated while you are changing a pipe profile in the route.
- The contents of the spreadsheet can be copied to Excel using the clipboard.
- If copying data between Wanda and Excel fails, check the regional settings. The decimal symbol must be set to "." (dot).
- You can open multiple spreadsheets at the same or at different cases.
- To change rows and columns, press the rotate button.
- When you press CTRL+F4 the window closes and the focus returns to the property window.

2.18.9 Charts

- Use the most recent printer driver when charts are not printed correctly.
- You can drag/drop chart-series buttons (also from different cases) to a single chart window (also when a movie is playing).
- Use an applicable short name if you have a multiple case chart. The short name is added between brackets to the series title. You can change the short name of a case in the description window.
- Make you own chart templates if you are not satisfied with the default graph settings.
- There are four groups of templates: Time series templates, Location templates, table templates and system characteristic templates. Templates can be shared or copied to another group by copying a local template to a *global* template.
- Before saving a chart template first check the 'captions.absolute' menu item. You see now the quotes that are not converted to keywords. These quotes are saved absolute to the template. You can choose 'Reset to default' to remove all absolute quotes.
- You can see multiple location series at once (Route) when clicking a chart button. First make a multiple selection of pipes. All the pipes in a selection must have the same flow direction. Also select components between selected pipes.
- You can see multiple time chart series lines at once when you click a chart button. First make a multiple selection of components with the time series you want to see.
- Fill the location property if you want to see pipe time chart series instead of pipe location chart series.
- Use the right mouse click to open a "selection chart window". This is a global window wherein each selected property chart is drawn. The advantage is that now only one chart window is used for all time series and one for all location series.
- You can drag a chart button to a chart that already has that location property. This location series is frozen at the active time step. Also frozen series are automatically updated when you run steady/transient again.

- You can quickly zoom into a chart by dragging a rectangle from left to right.
- You can quickly undo zoom by dragging from right to left.
- You can scroll the chart down/up and left/right by moving the mouse while the right mouse button is pressed.
- Line patterns don't work fine if the series points are close together.

2.18.10 Time navigator

- You can open multiple charts and multiple property windows when you are using the time navigator.
- The time navigator controls every open case/document, so you can easily compare cases with respect to the time.
- When you activate the time slider you can use the arrow keys to change the active time step.
- You don't have to stop the movie to move the time slider.
- You can select a continuous playing loop by holding the CTRL-key down while moving the time slider.
- You can deselect a continuous playing loop by clicking the time slider while the ALT-key is down.
- If you don't want to miss any time step while playing a movie, decrease the speed until you see '0%' below the speed slider.
- You can increase the width of the time navigator to increase the accuracy of the time slider.

2.18.11 Selection builder

- If you don't specify a value and using the '=' operator Wanda searches for empty/missing values.
- If you specify a '*' as value and using the 'like' or '=' operator Wanda searches for every value.
- You can use the hidden colour property to select all objects with the same colour.

2.19 Frequently asked questions

2.19.1 Manage files

Questions

Which files are necessary for backup purposes

I can not save my WANDA case

Answers

All input of the case is stored in the wdi and wdx file; all other files can be generated again performing the calculation.

You are probably viewing a WANDA case with components that are not part of your license. You can only save cases that can be built with your license.

Press F10 to select the unlicensed components.

2.19.2 Gallery/Shape library

Questions

The gallery is not visible

One of the shape libraries is not available in the gallery Hydraulic (no tab visible)

The component tooltip text is partially not visible, it disappears behind the property window

Answers

Go to menu "view" and mark "Gallery" as visible, or activate the right mouse click menu and check "shape gallery"

At the gallery tabs, click the right mouse button; choose "open palette" and select from the group "Wanda components" the required subject

The property window is always "on top". Move the gallery window to the left side of the diagram.

2.19.3 Route

Questions

How can I create routes?

Answers

There are several ways to select a route (a route is a concatenation of 2 or more H-components). A very fast way is click first H-component followed by CTRL-click of last required H-component. WANDA selects the shortest route. If a particular H-component must be part of the route, add them to the selection before creating the route.

See also: "**Select route between selected components**" on page 89 or use the short cut menu

2.19.4 Diagram / Layers

Questions

It is not possible to move the text labels in the diagram.

How can I put a map as wallpaper to my diagram?

How can I rotate or flip a component

I want to draw a H-component in another direction than from left to right. Is that possible without using the flip/rotate?

Can I change the connection line type afterwards?

Answers

The model is built in different layers. Choose the "text" layer tab at the bottom of your WANDA window to change the position/font/size of the text.

On the Arrange menu, point to Layers, and click Add Layer.

The new layer is added in front of the current layer and becomes the new current layer. The wallpaper layer must be moved up to the top

On the Insert menu, point to Picture and specify a bitmap picture from file

Resize the picture to the desired size

Press the right mouse button in the diagram and choose "Arrange", or use the "Arrange" option from the menu bar. Flip and rotate buttons can be made visible via "view/toolbars" menu, check the draw toolbar.

In vector line mode you can draw a H-component including its connection lines (H-nodes) in one movement. From the diagram short cut menu choose "Vector drawing". Draw a line and the active H-component is placed in the centre of the line. Press Esc-key to leave the vector mode or choose from the short cut menu another mode.

Another way to draw a component in a other (straight) connection is as follows:

Draw the new component starting at an existing component by dragging the mouse. The new component is automatically arranged in the direction of the connection line drawn.

Yes, a direct line can be changed to right angle line or just the other way round.

A curved line can changed in a direct line or right angle line only

2.19.5 Property window

Questions

What is the difference between the properties "Model name", "Comment" and "Keyword"?

Answers

In fact all of them are additional strings to describe the H-component. Model name and Keyword have particular intentions.

Model name is introduced to define a particular identification of the H-component, e.g.: serial number, product ID, manufacturer name. In the next WANDA version this name will be used as default name for the property template file, when properties are exported or imported.

Cases built with WANDA 1 and 2 uses the so-called common spec labels. These labels (used for data sharing) are replaced by a more generic data sharing concept; see also "Properties of a Component" on page 43. Common labels are

assigned to the property "Model name".

The "Comment" field has no special meaning; use this for additional remarks.

The "Keywords" field is used for quick selection. A H-component may have more than 1 keyword.

Keywords are added and removed using the "Selection" menu; see "How to use the Selection Builder" on page 62.

Use keywords for Route definitions, groups of components (Pump station, High Pressure System etc); see "How to use the Selection Builder" on page 62.

2.19.6 Charts

Questions

How can I show more time series of the same quantity in 1 chart?

Is it possible to show more series of different quantities in 1 chart?

Is it possible to show more series of different calculations?

How can I remove or change format from the footer text from the chart (in the footer text the case file and edit/run time is displayed)

How can I change the layout of the chart?

How can I save the changed layout settings of a chart?

Answers

Select first the wanted components (or H-nodes), then push the chart button of the wanted quantity. All series of the selected components are drawn in 1 time

If you want to add new series, press the chart button and drag it into the existing chart. Release the mouse button and the new series will be added into the chart.

Yes, create a new chart by pressing the chart button. This property quantity is drawn at the left axis. Drag the chart button of the second quantity into the chart. This quantity is drawn at the right axis.

It is not possible to have more than 2 different quantities in 1 chart.

Yes, create a new chart by pressing the chart button in case 1. Open case 2 and select the wanted component. Drag the chart button of the wanted property into the chart. To distinguish both series, use "legend name" from the menu "model/description window"

Choose from the model menu "Description window" Select form field "Chart footer" the required display text

Double click in the chart or choose menu "file/chart properties". You become in the edit window at which all elements of the chart can be changed. The changed settings can be changed. See next question.

To store these settings you must save it into a template. Choose from the Chart menu "Template". followed by Save, Save As. or Save As Default.

Using this Template menu you can switch between several layouts

2.20 Important keys and shortcuts

2.20.1 Windows general

2.20.1.1 Keyboard

Ctrl + F4	Close document window
Ctrl + F6	Next document window
Alt + F4	Exit application
Ctrl + Shift + N	New
Ctrl + O	Open
Ctrl + S	Save
F12	Save as
Ctrl + P	Print
Ctrl + A	Select all
Ctrl + X	Cut
Ctrl + C	Copy
Ctrl + V	Paste
Ctrl + Z	Undo
Ctrl + Y	Redo
F2	Edit Text
Shift + F10	Popup context menu (right click menu)

You can also the special button between the right alt and ctrl keys

2.20.1.2 Mouse

Ctrl + drag	Copy
Shift + drag rectangle	Select and keep old selection
Click	Select
Shift + click	Select and keep old selection
Right Click	Popup Context Sensitive Menu

2.20.2 IGrafX Diagram

2.20.2.1 Keyboard

Ctrl + Shift + 4	Align Top
Ctrl + Shift + 6	Align Middle
Ctrl + Shift + 8	Align Bottom
Ctrl + Shift + 3	Align Left
Ctrl + Shift + 5	Align Center
Ctrl + Shift + 7	Align Right
F5	Redraw diagram
F2	Start (editing) text
F4	Focus window on the currently selected object
F6	Note window
F7	Spelling
F9	Open shape library
Ctrl + U	Underline
Ctrl + I	Italic
Ctrl + B	Bold
Ctrl + D	Format text
Ctrl + F9	Move (selected objects) back one layer
Ctrl + F10	Move (selected objects) forward one layer

2.20.2.2 Mouse (*click at background, not at object*)

Click	Deselect all objects
Ctrl + right click	Activate shape insert mode
Shift + right click	Activate connect line insert mode. Change connect line mode (direct, rectangle) when line mode is already active
Shift + line draw	Add active shape including 2 connect lines (keep down while drawing a line). This is called vector-based schematisation.
Drag rectangle	Main Panel: Select objects in rectangle Quick Zoom Panel: Zoom to rectangle
Shift + Drag rectangle	Select objects in rectangle and keep selection
Alt + Right click	Best fit all (Zoom out)

2.20.3 Wanda diagram specific

2.20.3.1 Keyboard

F10	Select first invalid component
Tab	Toggle focus between diagram and selection property window
Ctrl + M	Activate mode / option window
Ctrl + T	Activate time navigator
Shift + F9	Run steady
Shift + Tab	Hide/unhide selection property window
Shift + F11	
Alt + F3	Activate description window
Alt + F5	Show system characteristics dialog
Alt + F7	Activate selection builder window
Alt + F9	Run transient (and steady when required)
Ctrl+Shift+F10	Select all components with warnings or errors
Ctrl+Shift+L	Activate Property Labels Window
Ctrl + Alt + F	Flip positive flow definition
Ctrl + Alt + I	Show initial values window
Ctrl + Alt + T	Activate time parameters window
Ctrl + Alt + F9	Stop / resume transient

2.20.3.2 Mouse (click at shape/line)

Drag rectangle	Main Diagram Panel: Select objects in rectangle Quick Zoom Panel: Zoom to objects in rectangle
Ctrl + Shift + click	Select shape and select route between selected shapes
Right click	Activate relevant menu options

2.20.4 Property window

2.20.4.1 Keyboard

F10	Select first invalid component
F11	Toggle focus between diagram and selection property window
Ctrl + S	Save selected properties to file
Ctrl + O	Open properties file
Ctrl + C	Copy selected properties to clipboard
Ctrl + V	Paste selected properties from clipboard
Ctrl + R	Toggle Overall view of selection
Ctrl + I	Show/Hide Info/Unit Pane
Ctrl + Shift + I	Show/Hide Input
Ctrl + Shift + O	Show/Hide Output
Ctrl + Shift + S	Activate Spreadsheet Window
Ctrl + Enter	Synchronise button

Page Down	Jump to next selected component in selection/route
Page Up	Jump to previous selected component in selection/route
F4	Drop down yellow selection list box
Space	Select/deselect property
Shift + arrow down	Select/deselect property
Alt + Ctrl + letter	Quick search for a property name (starting with specified letter)
Letter	Start editing property (overwrite existing value). Dropdown list and jump to value starting with specified letter
F2/enter	Start editing property (change text instead of overwrite text) / Open a table / Open a chart / Dropdown value list
Enter	Stop editing a (text) property / Apply dropdown value
Esc	Cancel editing a property
Alt + enter	Insert manual line break when editing a text property
Esc	Hide special property window (e.g. Fluid)
+ / *	Arithmetic operation with active cell
Pause	Show/Hide invisible properties

2.20.4.2 Mouse

Double click	Edit / Open table / Open chart
Click at property name	Select / deselect property
Drag rectangle	Select / deselect properties /checkboxes in rectangle
Drag button	You can drag a chart button to an existing chart to add series
Right click	Activate relevant menu options

2.20.5 Table Windows

2.20.5.1 Keyboard

Ctrl + Enter	Insert Row
Ctrl + Delete	Delete Row
+ * /	Arithmetic operation with selected cells
Ctrl + P	Popup print menu
Ctrl + G	Show/Hide chart
Ctrl + E	Chart Setting
Ctrl + T	Chart templates
Ctrl + Shift + Space	Select column
Shift + Space	Select row
Ctrl + A	Select All / Deselect All
Esc	Close table window Cancel editing a cell
Shift + Arrow keys	Select cells
F2	Edit cell

2.20.5.2 *Mouse*

Double click	Edit cell / Open table / Open chart
Drag rectangle	Select cells
Right click	Activate relevant menu options

2.20.6 Spreadsheet windows

2.20.6.1 *Keyboard*

Ctrl + R	Transpose, Swap rows and columns
Enter	Open table / Open chart
+ * /	Arithmetic operation with selected cells
Ctrl + P	Popup print menu
Ctrl + Shift + Space	Select column
Shift + Space	Select row
Ctrl + A	Select All / Deselect All
Esc	Close spreadsheet window Cancel editing a cell
Shift + Arrow keys	Select cells
F2	Edit cell

2.20.6.2 *Mouse*

Double click	Edit cell / Open table / Open chart
Drag rectangle	Select cells
Right click	Activate relevant menu options

3. Theoretical and mathematical background

3.1 Definitions

3.1.1 Nomenclature

A	=	cross-sectional area	[m ²]
c	=	wave propagation speed	[m/s]
D	=	inner diameter of the pipe	[m]
E	=	Young's modulus	[N/m ²]
e	=	wall thickness	[m]
f	=	friction factor	[-]
f	=	frequency	[Hz]
g	=	gravitational acceleration	[m/s ²]
H	=	Total head	[m]
ΔH	=	head loss across the valve	[m]
h	=	elevation of the pipe	[m]
K	=	fluid bulk modulus	[N/m ²]
L	=	pipe length, element length	[m]
m	=	mass of the pipe, valve	[kg]
P	=	pressure	[N/m ²]
Q	=	discharge	[m ³ /s]
R	=	inner radius of the pipe	[m]
R	=	hydraulic radius	[m]
s	=	co-ordinate along the pipe	[m]
t	=	time	[s]
u_R	=	radial displacement of pipe wall	[m]
V	=	mean fluid velocity	[m/s]
γ	=	elevation angle	[rad]
ν	=	Poisson's ratio	[-]
ξ	=	loss coefficient	[-]
ρ	=	density	[kg/m ³]
σ	=	stress	[N/m ²]
σ_ϕ	=	hoop stress	[N/m ²]

3.1.1.1 *Superscripts*

n	=	time step
-----	---	-----------

3.1.1.2 Subscripts

atm	=	atmosphere
f, F	=	fluid
l	=	place indicator
o, oo	=	steady state
t	=	(pipe), tube
V	=	valve

3.1.2 Bernoulli's theorem

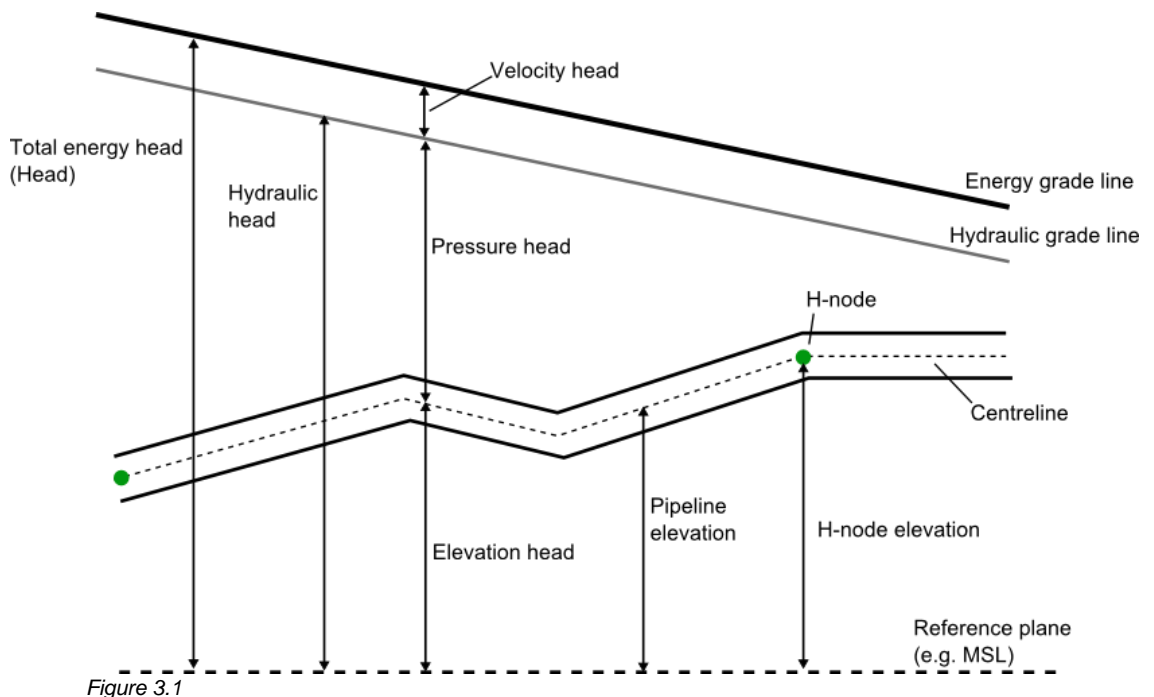
An important formula to understand the energy terms is Bernoulli's theorem: it states that the total energy of each particle of a body of fluid is the same provided that no energy enters or leaves the system at a point. The division of this energy between potential, pressure and kinetic energy may vary, but the total remains constant.

Total head = potential head + pressure head + velocity head

In symbols:

$$H = z + \frac{p}{\rho g} + \frac{v^2}{2g} \quad (3.1)$$

In the figure below these terms are explained illustratively (elevation head = potential head) :



3.1.3 General definitions

Some definitions are necessary in order to understand the functionality of WANDA. These definitions are used throughout this document and are also important for a proper use of the program.

First we give a summary of general definitions. The general definitions are explained by some examples, which already reveal some of the functionality.

BRANCHED PIPE SYSTEM:

A system consisting of pipes and components connected in an arbitrary way.

HYDRAULIC SYSTEM (H-system):

The pipe system with respect to the hydraulic components and pipes.

HYDRAULIC MODEL (H-model):

Mathematical model of an H-component with respect to its hydraulic features.

HYDRAULIC NODE (H-node):

Node that connects H-components via the connect points. The H-node height is the reference level for the pressure in the H-node.

HYDRAULIC COMPONENT (H-component):

Component defined by its hydraulic properties.

PIPE:

Pipe defined by constant hydraulic properties. The fluid in the pipe is considered as compressible. The pipe is in fact a fall type with internal waterhammer nodes.

FALL TYPE:

Type of H-component, which can be represented by a 2-connect point H-model. The fluid in the fall type component is considered as incompressible.

SUPPLIER TYPE:

Type of H-component, which can be represented by a 1-connect point H-model. The fluid in the supplier type component is considered as incompressible.

A H-node is represented by one or more connection lines in the diagram. A connection line connects two H-components or a H-component with another connection line.

WATERHAMMER NODE (W-NODE):

Node in a pipe (not on the extremities). Calculation point of pressure and discharge.

NODAL SET:

Clusters of H-components (fall types and supplier types) at the extremities of the pipes.

WATER LEVEL NODE (S-NODE):

Even node (0, 2,..) in a free surface pipe in which the water levels are calculated in the staggered grid of the free surface algorithm.

VELOCITY NODE (U-NODE):

Odd node (1, 3,..) in a free surface pipe in which the fluid velocities are calculated in the staggered grid of the free surface algorithm.

LEVEL, HEIGHT AND OFFSET

In Wanda there are several different inputs used for calculation of elevations. These are:

- Level: This is always relative the reference plane of the case
- Height: This depends upon the component and the manual should be checked to determine relative to which reference the height is taken.
- Offset: This is relative to the h-node elevation unless otherwise specified in the manual.

3.1.4 Examples of systems

In figure 3.2.1 a schematic representation of a pipe system is given. Such a pipe system is often called a 'branched pipe system'. Several pipes are connected to the same H-node.

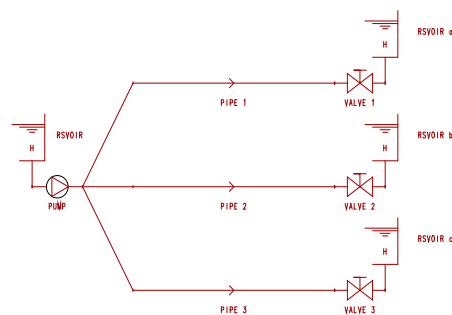


Figure 3.2.1: Schematic representation of a pipe system

In figure 3.2.2 a symbolic representation of a simple H-system is given. The system has four H-components: two reservoirs, a pump and a check valve, and one pipe. The H-nodes are emphasised by small squares in the figure.



Figure 3.2.2: Schematic representation of a simple pipe system, the H-system

The two reservoirs are supplier types. They are connected to H-node 1 and to H-node 4. The pump and checkvalve are fall types. They are located between H-node 1 and 2, respectively H-node 2 and 3. The difference between supplier and fall types is clear. The supplier types provide the H-system with fluid. The fall types cause a pressure fall between the H-nodes.

The pipe has constant hydraulic properties (constant diameter and friction coefficient). There is only one pipe in this system and there are two nodal sets. The first nodal set contains three H-components. The H-nodes 1, 2 and 3 all belong to this nodal set. The second nodal set contains only one H-component and one H-node.

To be able to perform a transient analysis on the H-system the pipe must be divided into elements. Suppose we divide the pipe into five elements (figure 3.2.3).



Figure 3.2.3: Schematic representation of the H-system including the W-nodes

Then four W-nodes are introduced. Note that the nodes at the extremities of the pipe are H-nodes. Once the W-nodes are defined the H-system is completely specified.

If the proper data are provided we can perform a transient analysis (steady state and unsteady state).

3.2 Modelling aspects

3.2.1 WANDA Architecture

WANDA is designed such that different domains are covered by one computational core. The architecture is schematically represented in figure 3.3:

- Liquid domain for isothermal fluid flow
- Heat domain for fluid flows with difference in temperature
- Also other domains such as multi-phase flow and mechanics are possible

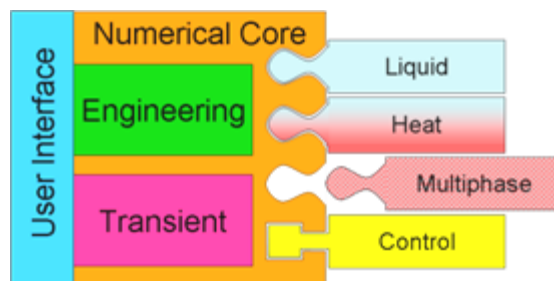


Figure 3.3: WANDA architecture

Throughout the following chapters the main focus will be on the Liquid-domain, because the other domains in general show much resemblance to Liquid.

3.2.2 Hydraulic components

The symbols in the WANDA diagram are called hydraulic components. The components contain hydraulic models (H-models) which are expressed in mathematical equations. The H-models define how the variables are related. The number of equations in each H-model depends on the number of connected nodes and the number of core variables in the actual WANDA domain.

For each WANDA domain there is a library of components that can be applied to build the H-model. Some components in different domains are very similar, such as the pipe or valve in the Liquid or Heat domain. For each domain this manual contains a separate chapter where you can find an extensive physical description of the components.

Each domain contains some components that can be used to define a boundary condition. The component BOUNDH and BOUNDQ (Liquid) represent a prescribed head and discharge, whilst the BOUND PT and BOUND MT represent a prescribed and temperature-related pressure and mass flow rate (Heat).

3.2.3 Connect point variables

Each Hydraulic component has one or more connect points. For each connect point the main variables (also called the core variables) are calculated. These main variables differs for each domain. Therefore each domain has his own colour of connect point, see the figure below.

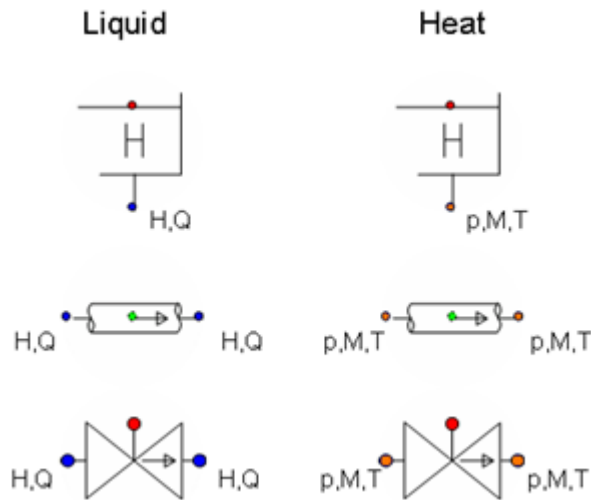


Figure 3.4: Similarity between domains

The main variables depends on the actual WANDA domain:

Liquid:

- H (head) [m];
- Q (discharge) [m³/s];

Heat:

- p (pressure) [N/m²];
- M (mass flow) [kg/s];
- T (temperature) [°C];

WANDA uses the main variables to calculate the flow in the pipeline system. Other variables (like for instance fluid velocity or pressure in Liquid models) are derived indirectly from the main variables after the calculation has finished.

The calculations are performed internally in SI-units, but the user can chose the units to use in the Graphical User Interface. WANDA converts the units to SI before calculation and converts it back to the unit system chosen by the user afterwards.

3.2.4 Component phases

Every component can have more than one phase ('component phase' or 'state'). To clarify this concept we take the ideal check valve as an example.

The check valve is a fall type component. It can be in open phase (1st phase) and in closed phase (2nd phase). Both phases are governed by different equations:

phase 1:	check valve is open; $Q_1 > 0$	$f_1 : H_1 = H_2$ $f_2 : Q_1 = Q_2$
phase 2:	check valve is closed; $H_1 < H_2$	$f_1 : Q_1 = 0$ $f_2 : Q_1 = Q_2$

As a result, depending on the component phase, different equations are needed to describe the flow through the check valve. This might seem very trivial, but it is not from the viewpoint of programming, because we do not know a priori when the check valve will close.

The number of phases can be 3, 4 or more depending on the model of the specific component. A pump, for instance, can be in the states: initialisation, running, tripping, starting, turbining etc.

3.2.5 Time dependant actions and control

Some components are **active**. This means that such components can be activated by the user or by control components. For instance one can close a valve or have a pump trip. One could think that a check valve is also active. This is not true. A check valve closes due to the fluid flow in transient conditions, it cannot be activated by the user in an 'operative' sense.

Access to the action tables is only possible in Transient Mode. In an action table the behaviour of that particular component is prescribed in time by the user, e.g. the level in a reservoir, the position of a valve.

With WANDA Control it is possible to simulate operational control loops. As WANDA Control consists of more than 30 logical controllers and numerical functions, there is almost no limitation to the Control circuits in WANDA models.

3.2.6 Calculation types

A distinction can be made between several hydraulic aspects considering time scales. Some component equations have a different form for the steady state and the unsteady state. Therefore, we will distinguish the steady state and the unsteady state.

3.2.6.1 Steady state

Steady state calculations assume that there are no changes in time and all variables are constant. The fluid is considered incompressible. This is the case for instance with computing the appropriate diameter of a pipeline for a given design discharge.

In Engineering Mode WANDA uses only steady state computations.

3.2.6.2 Unsteady state

Unsteady state calculations can be only done in the Transient mode. To start with an unsteady state computation, a correct steady state calculation must be performed first.

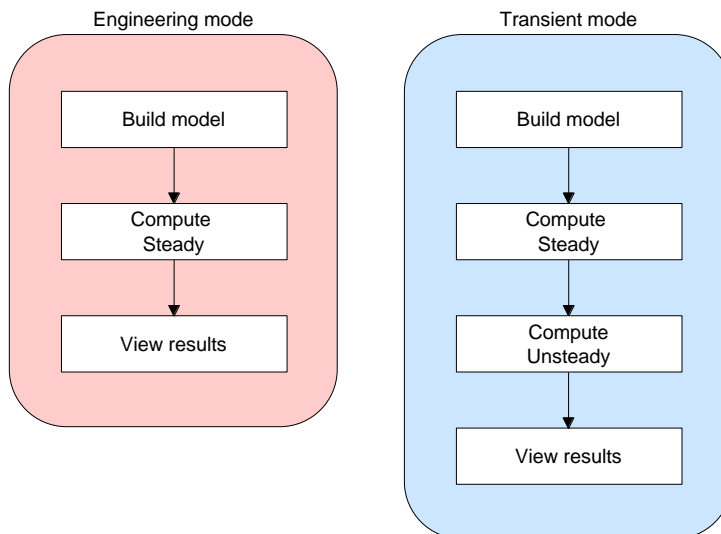


Figure 3.5: Schematically layout of the WANDA modelling process

Quasi-steady / extended time simulations

Quasi-steady calculations can be considered as a number of independent steady state calculations, which happen to be following each other time. Although the conditions may change with time (e.g. different discharge patterns or valve positions for each time step), each calculation is independent from earlier solutions. For instance engineering tools like Epanet use the quasi-steady approach to determine the water levels and residence time in reservoirs during a certain period of time (day or week).

WANDA uses the quasi-steady approach to calculate the pipe friction at the new time step. WANDA Control uses the quasi-steady approach as it 'measures' at the old time step to determine the control-actions at the new time step.

Pressure waves

Due to abrupt changes in the flow conditions (valve closure, filling of pipeline or reservoir) pressure waves develop, which travel through the pipeline (surges). As the fluid and containing pipeline are compressible, the pressure wave needs time to travel from one end of the pipeline to the other end. These pressure waves are very quick phenomena (transients) with wave speeds up to over 1000 m/s.

In Transient mode WANDA calculates the pressure waves in all waterhammer pipes. The fluid in non-waterhammer pipes is considered incompressible, resulting in an infinite wave speed.

Cavitation

Pressure waves and abrupt variations in the pipe profile (e.g. a partially closed valve) can result in substantial changes in pressure. When the fluid pressure drops below the fluid vapour pressure, cavitation can occur. Liquid becomes gas and as gas is more compressible than liquid, the amount of gas in the pipeline changes the flow conditions.

In WANDA the effect of cavitation is included as it recalculates the flow conditions at each time step depending on the pressure and fluid properties.

3.2.7 Solution procedure

The WANDA calculation starts with initialisation and validation of the input. Once the input is validated (e.g. position of check valve is correct), the steady calculation can start (engineering). After the steady calculation has obtained convergence, an unsteady calculation can be performed (transient). All calculations are performed in SI-units. Post processing and conversion to the unit system chosen by the user are part of the Grafical User Interface (GUI) and take place after the calculation of steady (engineering) or the last time step of unsteady (transient). The different steps in the solution procedure of WANDA calculations are schematically depicted in figure 3.6 below.

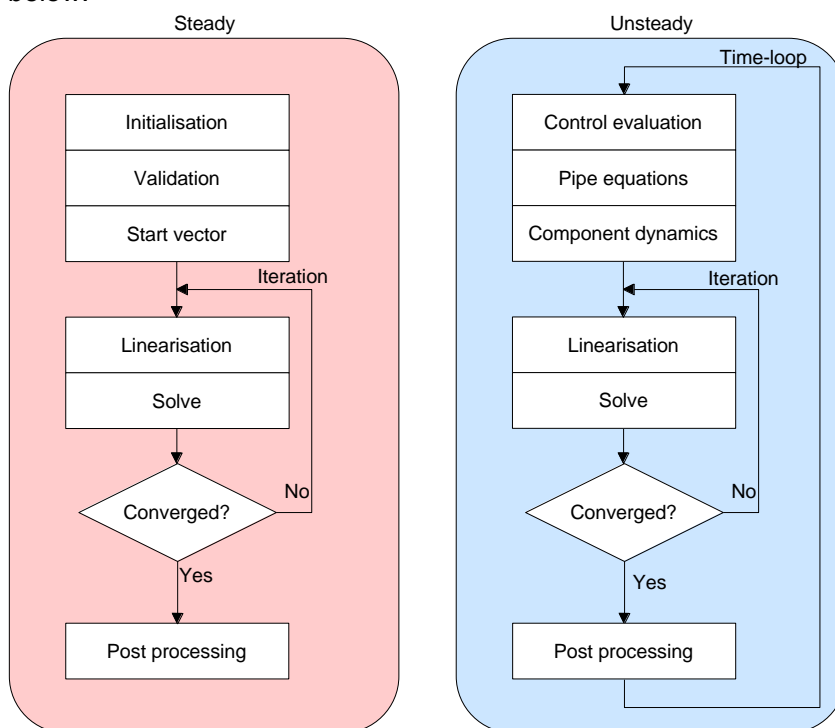


Figure 3.6: Schematically layout of WANDA Steady and Unsteady solution procedures

In steady state (engineering mode) a start vector is built to start the iteration process. The start vector contains values of the core variables estimated from the input. The friction in all pipes is updated at each iteration.

In unsteady state (transient mode) WANDA uses the solution of the old time step to speed up the calculation process. Dynamic friction and control values are based on the quantities of the core variables at the old time step (quasi-steady).

From a computational point of view the main difference between Engineering and Transient mode is the compressibility of the fluid. As no time-effects are included in Engineering mode, the fluid is considered incompressible and the complete hydraulic set is solved as one big system of equations.

In Transient mode however there is a time-effect due to travel times in the pipes. The non-pipe components (nodal sets) are decoupled, as the wave needs time to travel

through the pipe. The decoupled nodal sets are solved as independent systems of equations.

3.2.7.1 Pipe equations

Waterhammer pipes are essential components in WANDA. Due to the combined compressibility of the water and containing pipe, there is a delay in exchange of information between the extremities of the pipe. WANDA uses the method of characteristics (see section "Pipes: Method Of Characteristics" on page 159) to calculate the (pressure) waves in the waterhammer pipes. The waterhammer pipes contain internal nodes (W-nodes) where the values of core variables is calculated at each time step.

The total hydraulic set is solved for each time step in the following way. First all waterhammer and free surface pipes are handled such that in every internal node the core variables are computed at $t=t+\Delta t$. This results in new values of the core variables at the pipe extremities.

Then all other components are handled as one system of equations. Also 'rigid column pipes' are computed here, as the incompressibility causes the information at the extremities to be exchanged instantaneously.

3.2.7.2 System of equations

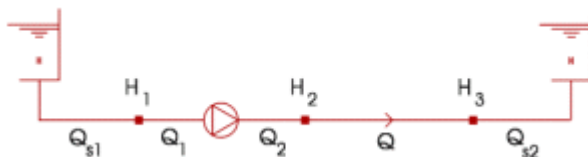
All equations and variables can be considered together as one system of equations, based on the scheme as build by the modeller. Each component (equation) is therefore related with other components (equations). The model can be represented mathematically by a matrix representing all coefficients of all equations involved. WANDA solves this system of equations for the core variables. This requires the system to have exactly the same number of equations as variables. The components and nodes supply the equations:

- Node equations:
- Mass balance
- Energy balance
- Momentum balance
- Additional (ideal gas law)
- Component equations
- Boundary conditions
- Energy loss
- Wave translation (pipe characteristics)

Since most equations are non-linear (head loss for instance is generally quadratically related to discharge), the equations are linearised in order to solve the system of equations.

3.2.7.3 Number of unknown variables versus number of equations, an example

Suppose we have the following Q-H system:



The number of unknown variables is 8: Q_{s1} , Q_1 , Q_2 , Q , Q_{s2} , H_1 , H_2 , H_3

The number of equations is 8: 2 * 1 boundary condition + 1 * 2 fall eq. + 1 pipe eq. + 3 cont. eq.

The number of unknowns equals the number of equations. Hence, the hydraulic set is completely defined.

3.3 Implementation

Throughout the following the main focus will be on the Liquid-domain, because the other domains in general show much resemblance to Liquid. The core variables are therefore H (head) and Q (discharge).

In steady state the hydraulic set is different in form than in unsteady state. This is caused by the fact that the method of characteristics (MOC) is used to solve the pipe equations, cancelling the direct exchange of information between the extremities of the pipe.

3.3.1 Steady state

3.3.1.1 Supplier type boundary conditions

The basic equation, which governs the steady state of a boundary condition (supplier type) has the general form:

$$f (H, Q) = 0 \tag{3.2}$$

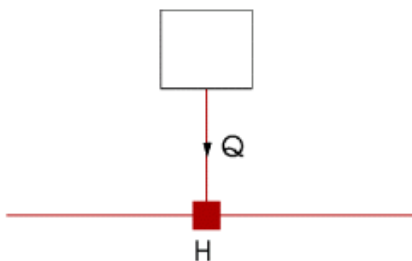


Figure 3.6: Definition of variables for supplier type components, *flow into the system is positive, flow out of the system is negative

H is the head in the node to which the boundary condition is attached. The discharge Q is the discharge that is supplied to the system. The sign convention is: flow into the system is positive, flow out of the system is negative. Most suppliers (air vessel, surge tank) do not supply discharge in steady state. In that case the equation is simply $Q = 0$.

A BOUNDH is a supplier that may supply fluid in steady state as well. The basic equation in steady state is:

$$H = \text{constant} \quad (3.3)$$

A supplier type may be non-linear, but in steady state the basic equation is usually linear.

3.3.1.2 Fall type

The basic equations, which govern the steady state flow through a fall type have the general form (here as an example for a 2-node component):

$$f_1 (H_1, H_2, Q_1, Q_2) = 0 \quad (3.4)$$

$$f_2 (H_1, H_2, Q_1, Q_2) = 0 \quad (3.5)$$

H_1 and H_2 are the pressure heads at the extremities of the 2-node component (see figure 3.7).

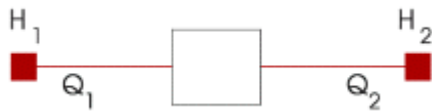


Figure 3.7: Definition of variables for 2-node fall type components

Here we give the valve as an example:

$$f_1 : H_1 - H_2 = c |Q_1|/Q_1 \quad (3.6)$$

$$f_2 : Q_1 = Q_2$$

Q_1 is the discharge in the upstream part of the fall type, Q_2 is the discharge in the downstream part of the fall type. Note that the equations for the valve are non-linear in Q_1 .

Fall type components are usually non-linear. The sign-convention is depending on the symbol representing the component in the WANDA diagram. An arrow, generally from left to right, indicates the positive direction.

3.3.1.3 Pipes: special form of 2-node component

Due to the incompressible behaviour of the fluid in steady state, the discharge is constant along the pipe component. The basic equations, which govern the steady state flow through a pipe are therefore given by the same equations as the other fall-type components. Besides the constant discharge (inflow = outflow) the following equation for the head loss holds:

$$H_1 - H_2 - \frac{fL}{D} \frac{Q|Q|}{2g A_f^2} = 0 \quad (3.7)$$

The general form of this equation is denoted by:

$$f (H_1, H_2, Q) = 0 \quad (3.8)$$

Q is the discharge through the pipe. Note that Eq. (3.8) is non-linear in Q .

H_1 and H_2 are the heads at the H-nodes of the pipe (see figure 3.8). The Darcy-Weisbach friction factor f in 3.7 above is given by the friction model depending on the flow velocity and properties of the fluid (Newtonian or non-Newtonian), the pipe (diameter, wall roughness). See "Friction models" on page 166 for more information on the friction models in WANDA

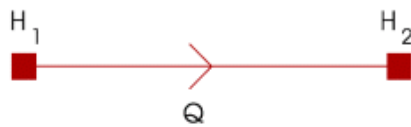


Figure 3.8: Definition of variables for pipes

3.3.1.4 Nodes: Continuity equations

To complete the total hydraulic set, the continuity equations, one for each H-node, are added. Hence:

$$\sum_{\forall H\text{-node}} Q = 0 \quad (3.9)$$

It is obvious that these equations are linear. The sign convention is the same as for the boundary conditions. Flow into the H-node is positive, flow out of the H-node is negative.

3.3.2 Unsteady state

Boundary conditions are needed to solve the pipe equations using the Method Of Characteristics (MOC). The boundary conditions have to be prescribed at the extremities of each pipe. These boundary conditions are formed by all models in the nodal sets. Therefore, it is necessary to distinguish the equations of the pipes and the so-called nodal set equations.

3.3.2.1 Pipes: Method Of Characteristics

The basic equations, which govern the unsteady state flow through a pipe are usually called the characteristic equations. These equations result from the MOC. A characteristic equation has the general form:

$$C_H H + C_Q Q = C_C \quad (3.10)$$

and is valid along a characteristic line with either an angle of $1/\alpha$ (C+) or $-1/\alpha$ (C-) in the s,t -plane (α is the propagation speed of the pressure waves).

Internal waterhammer nodes

If a pipe is divided into 6 elements, the pipe has 5 W-nodes. The computational grid of the pipe in the s,t plane is shown in figure 3.9.

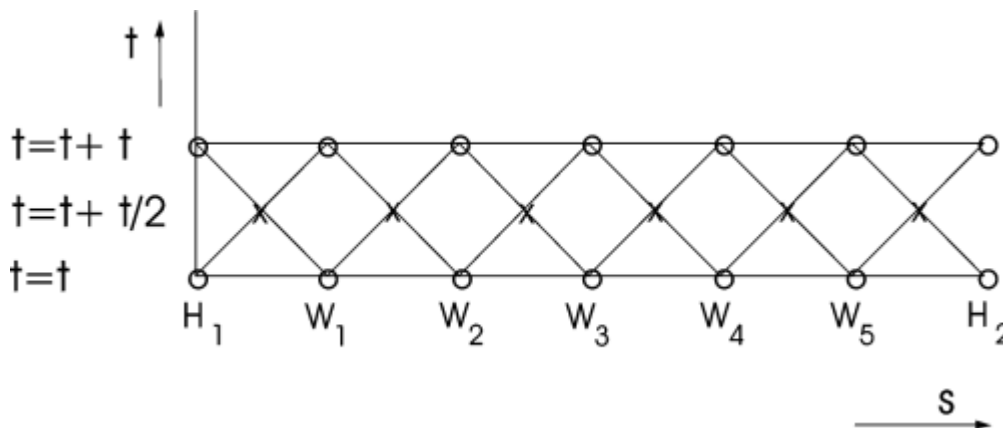


Figure 3.9: Computational grid in transient calculations for a pipe divided into 6 elements. The elements have a constant length $\Delta s = c_f \cdot \Delta t$. The characteristic lines cross at nodes denoted with x.

This pipe is described by a total of 12 characteristic equations, one equation for every line. The characteristic lines cross at nodes x. These nodes are neither W-nodes nor H-nodes but must be seen as assistance nodes. Suppose that the hydraulic state (H, Q) is known at $t=t$ in every node. First the new H and Q are calculated in the assistance nodes at $t=t+(1/2)\Delta t$. Once this is done the new H and Q are calculated in the W-nodes $W_1 - W_5$ at $t=t+\Delta t$.

As mentioned before, it is not possible to calculate the H and Q at $t=t+\Delta t$ in H_1 and H_2 by only using the characteristic equations from the specific lines. We also have to know the boundary conditions at H_1 and H_2 . These boundary conditions are given by the H-models in the nodal sets. The characteristic equations, which belong to the characteristic lines to the end points of a pipe will be part of the nodal set equations.

The use of assistance nodes is necessary to avoid uncoupled characteristic lines. Such a grid is known as 'leap frog'. The coefficients for a C^+ equation are:

$$\begin{aligned} CH &= 1 \\ CQ &= R \end{aligned} \tag{3.11}$$

$$CC = H_i^n + R Q_i^n - S Q_i^n / Q_i^n /$$

and for a C^- equation:

$$\begin{aligned} CH &= 1 \\ CQ &= -R \end{aligned} \tag{3.12}$$

$$CC = H_{i+1}^n - R Q_{i+1}^n + S Q_{i+1}^n / Q_{i+1}^n /$$

The upper index denotes time, the lower index denotes place.

3.3.2.2 Other components: Nodal set equations

Every nodal set yields so-called nodal set equations. The nodal set equations consists of:

1. Equations for the fall type components in the nodal set;
2. Equations for the supplier type components in the nodal set;
3. Continuity equations for every H-node in the nodal set;
4. Characteristic equations corresponding to the characteristic lines of pipes adjacent to the nodal set.

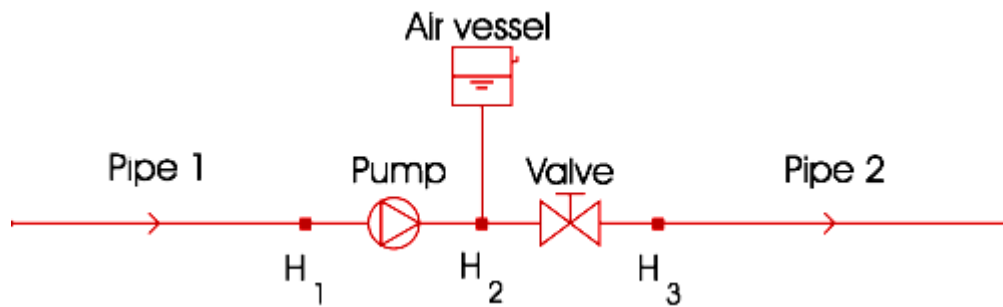


Figure 3.10: An example of a nodal set configuration

In figure 3.10 an example of a nodal set is given. The nodal set consists of two fall types (pump, valve), one supplier (air vessel) and 3 H-nodes. This nodal set is described by the following equations:

- one characteristic equation for pipe 1
- one characteristic equation for pipe 2
- two equations for the pump
- two equations for the valve
- one equation for the air vessel
- three continuity equations.

Hence the nodal set is governed by 10 equations. Of course the total number of unknown variables is 10 as well:

- discharge of pipe 1
- discharge of pipe 2
- two discharges of the pump
- two discharges of the valve
- one discharge of the airvessel
- three heads at H_1 , H_2 and H_3 .

As a result the hydraulic behaviour in the nodal set is completely determined. We will discuss now the general forms of all equations except the pipe equations, which we have discussed already.

Fall types

The basic equations, which govern the unsteady state flow through a fall type have the general form (an example for a 2-node component is given):

$$\begin{aligned} f_1 (H_1, H_2, Q_1, Q_2, t) &= 0 \\ f_2 (H_1, H_2, Q_1, Q_2, t) &= 0 \end{aligned} \quad (3.13)$$

H_1 and H_2 are the heads at the extremities of the fall type. The discharge Q_1 is defined upstream in the fall type, Q_2 is the discharge downstream in the fall type. The parameter t denotes that other variables in the equation can be a function of time. Here we give an open valve as an example:

$$\begin{aligned} f_1 : H_1 - H_2 &= c(t) Q_1/Q_1 \\ f_2 : Q_1 &= Q_2 \end{aligned} \quad (3.14)$$

The parameter $c(t)$ is a measure of the discharge coefficient of the valve. This coefficient varies in time when the valve is closed or opened. Note that the equations for the valve are non-linear in Q_1 . The fall types are usually non-linear.

Supplier types

The basic equation, which governs the unsteady state condition of a supplier type has the general form:

$$f(H, Q, t) = 0 \quad (3.15)$$

H is the head in the H-node to which the supplier is attached. Q is the discharge that is supplied to the system. The sign convention is: flow into the system is positive, flow out of the system is negative.

Most suppliers (air vessel, surge tank) supply discharge in unsteady state. A fixed surge tank is governed by the following differential equation:

$$Q = A_{st} \frac{dH}{dt} \quad (3.16)$$

in which A_{st} denotes the area of the surge tank. Such an equation cannot be solved straightforward, it must be discretised as:

$$Q(t + \Delta t) = A_{st} \frac{H(t + \Delta t) - H(t)}{\Delta t} \quad (3.17)$$

in which Δt is the time step. This equation is linear in the unknown variables Q and H at $t = t + \Delta t$. H at $t = t$ is known. In general a supplier type can be non-linear as well.

Continuity equations

To complete the nodal point equations the continuity equations must be added for every H-node in the nodal point. Hence:

$$\sum_{\forall H\text{-node}} Q = 0 \quad (3.18)$$

It is obvious that these equations are linear. The sign convention is the same as for the suppliers. Flow into the H-node is positive, flow out of the H-node is negative.

3.3.3 Start Vector

3.3.3.1 Steady state

To compute the steady state of the hydraulic set, all equations of the appropriate component are assembled in one big matrix and one big right hand side vector.

To start the iteration a so-called start vector must be created by the processor and/or components.

The minimum and maximum number of iterations are defined in the user interface dialogue window "Accuracy window".

If after the maximum number of iterations the solution has still not been obtained something is badly wrong.

3.3.3.2 Unsteady state

Suppose we have to calculate the first time step. In this case the steady state solution is used as start vector. To solve the next time step we can use the old values of the former time step as start vector. However to speed up the convergence, the procedure after the first time step is slightly different.

In unsteady stat the start vector is filled with new values obtained by extrapolation of the values of the two former time steps, thus for every H-node in the nodal set:

$$H(t + \Delta t) = \frac{H(t) - H(t - \Delta t)}{\Delta t} \Delta t + H(t) \rightarrow H(t + \Delta t) = 2H(t) - H(t - \Delta t) \quad (3.19)$$

and for every discharge in the nodal point:

$$Q(t + \Delta t) = \frac{Q(t) - Q(t - \Delta t)}{\Delta t} \Delta t + Q(t) \rightarrow Q(t + \Delta t) = 2Q(t) - Q(t - \Delta t) \quad (3.20)$$

In this way the start vector for all nodal sets is built.

3.3.4 Linearisation of H-models

During the discussion of the hydraulic set, it appeared that some of the component-models could be non-linear. A summary of the models with respect to their linear or non-linear character is given in table 3.3. Again the steady state and unsteady state are distinguished.

Component	Steady state	Unsteady state
Pipe	non-linear	linear
Fall-type	non-linear/linear	non-linear/linear
Supplier-type	non-linear/linear	non-linear/linear

Table 3.3: H-models classified to linearity

The problem with non-linear equations is that they cannot be solved by a straightforward algorithm. There are no 'solvers' for non-linear equations. That is why non-linear equations have to be linearised.

As we can see the component-model of the pipe in steady state is always non-linear, whilst in unsteady state the model is always linear. Fall and supplier types can be linear or non-linear. The linearisation of non-linear equations is carried out with the Newton-Raphson method.

3.3.4.1 Linearisation of pipes in steady state

The general form is given by: $f(H_1, H_2, Q) = 0$. Linearisation of this equation results in:

$$f^{i+1}(H_1, H_2, Q) = f^i(H_1, H_2, Q) + \frac{\partial f^i}{\partial H_1} \Delta H_1^i + \frac{\partial f^i}{\partial H_2} \Delta H_2^i + \frac{\partial f^i}{\partial Q} \Delta Q^i = 0 \quad (3.21)$$

in which:

- $\Delta H_1^i = H_1^{i+1} - H_1^i$
- $\Delta H_2^i = H_2^{i+1} - H_2^i$
- $\Delta Q^i = Q^{i+1} - Q^i$

In this equation i denotes the iteration step. The values subscripted by i are known while the values subscripted by $i+1$ are to be calculated. Eq. (3.21) can be written in matrix notation in the following way.

Denote:

$$\frac{\partial f^i}{\partial H_1} = DH_1, \quad \frac{\partial f^i}{\partial H_2} = DH_2, \quad \frac{\partial f^i}{\partial Q} = DQ \quad (3.22)$$

Then Eq. (3.21) can be written as:

$$\begin{bmatrix} DH_1 & DH_2 & DQ \end{bmatrix}^i \begin{bmatrix} H_1 \\ H_2 \\ Q \end{bmatrix}^{i+1} = \begin{bmatrix} DH_1 H_1 + DH_2 H_2 + DQ Q - f \end{bmatrix}^i \quad (3.23)$$

This equation can be written more conveniently by writing the right hand side as CON:

$$\begin{bmatrix} DH_1 & DH_2 & DQ \end{bmatrix}^i \begin{bmatrix} H_1 \\ H_2 \\ Q \end{bmatrix}^{i+1} = \begin{bmatrix} CON \end{bmatrix}^i \quad (3.24)$$

This linear equation is actually to be solved by the computer. Because of the linearisation an iterative procedure is applied. This procedure is called the hydraulic iteration (H-iteration).

In unsteady state the pipe equations do not have to be linearised. The characteristic equations are already linear. The solution procedure of these equations is completely different from the nodal set equations.

3.3.4.2 Linearisation of fall-types

The linearisation of the fall-type equations is the same in steady and unsteady state. The general form is given by:

$$\begin{aligned} f_1(H_1, H_2, Q_1, Q_2, t) &= 0 \\ f_2(H_1, H_2, Q_1, Q_2, t) &= 0 \end{aligned} \quad (3.25)$$

Linearisation of these equations results in:

$$f_1^{i+1} (H_1, H_2, Q_1, Q_2, t) = f_1^i (H_1, H_2, Q_1, Q_2, t) + \quad (3.26)$$

$$\frac{\partial f_1^i}{\partial H_1} \Delta H_1^i + \frac{\partial f_1^i}{\partial H_2} \Delta H_2^i + \frac{\partial f_1^i}{\partial Q_1} \Delta Q_1^i + \frac{\partial f_1^i}{\partial Q_2} \Delta Q_2^i = 0$$

And

$$f_2^{i+1} (H_1, H_2, Q_1, Q_2, t) = f_2^i (H_1, H_2, Q_1, Q_2, t) + \quad (3.27)$$

$$\frac{\partial f_2^i}{\partial H_1} \Delta H_1^i + \frac{\partial f_2^i}{\partial H_2} \Delta H_2^i + \frac{\partial f_2^i}{\partial Q_1} \Delta Q_1^i + \frac{\partial f_2^i}{\partial Q_2} \Delta Q_2^i = 0$$

Using the same convention for the notation as for the pipe, the linearised equations can be written as:

$$\begin{bmatrix} D1H_1 & D1H_2 & D1Q_1 & D1Q_2 \\ D2H_1 & D2H_2 & D2Q_1 & D2Q_2 \end{bmatrix}^i \begin{bmatrix} H_1 \\ H_2 \\ Q_1 \\ Q_2 \end{bmatrix}^{i+1} = \begin{bmatrix} CON_1 \\ CON_2 \end{bmatrix}^i \quad (3.28)$$

in which:

$$D1H_1 = \frac{\partial f_1^i}{\partial H_1}, \quad D1H_2 = \frac{\partial f_1^i}{\partial H_2}, \quad D1Q_1 = \frac{\partial f_1^i}{\partial Q_1}, \quad \dots \text{etc.}$$

and:

$$CON_1 = D1H_1 H_1^i + D1H_2 H_2^i + D1Q_1 Q_1^i + D1Q_2 Q_2^i - f_1^i$$

$$CON_2 = D2H_1 H_1^i + D2H_2 H_2^i + D2Q_1 Q_1^i + D2Q_2 Q_2^i - f_2^i$$

The linear equation (3.27) is actually to be solved by the computer. Because of the linearisation the equations are part of the hydraulic iteration (H-iteration).

3.3.4.3 Linearisation of supplier-types

The linearisation of the supplier-types is the same in steady and unsteady state. The general form is given by $f (H, Q, t) = 0$. Linearisation results in:

$$f^{i+1} (H, Q, t) = f^i (H, Q, t) + \frac{\partial f^i}{\partial H} \Delta H^i + \frac{\partial f^i}{\partial Q} \Delta Q^i = 0 \quad (3.29)$$

In matrix notation Eq. (3.29) reads:

$$[DH \quad DQ]^i \begin{bmatrix} H \\ Q \end{bmatrix}^{i+1} = [CON]^i \quad (3.30)$$

in which:

$$\bullet \quad \text{DH} = \frac{\partial f^i}{\partial H}, \quad \text{DQ} = \frac{\partial f^i}{\partial Q}$$

$$\bullet \quad \text{CON} = \text{DH} H^i + \text{DQ} Q^i - f^i$$

The linear equation (3.29) is actually to be solved by the computer. The equations are updated during the hydraulic iteration (H-iteration).

3.4 Friction models

Several correlations exist for the modelling of the turbulent wall friction. Each correlation has its own background in a particular field. WANDA 4 supports a wide range of turbulent wall friction correlations for both Newtonian and non-Newtonian fluids. The Newtonian fluid is defined by a constant kinematic viscosity. For laminar wall friction a theoretical wall friction function can be derived: the well-known Poiseuille equation.

WANDA supports the following wall friction correlations:

- Theoretical friction according to Darcy-Weisbach
- for Newtonian fluids: Colebrook-White
- for non-Newtonian fluids: Herschel-Bulkley
- Empirical models
- Hazen-Williams
- Chezy-Manning

3.4.1 Theoretical friction models (Darcy-Weisbach)

The steady state pressure drop over a pipe is computed with the Darcy-Weisbach equation and is a function of the pipe dimensions (L and D), velocity U and wall friction f .

$$H_2 - H_1 = \frac{f L}{D} \frac{Q|Q|}{2gA_f^2} \quad (3.31)$$

The user has the flexibility to specify a set of local losses (ξ or $n \cdot D$ values) and the number of fittings for each tabulated local loss. These local losses are included in the hydraulic grade line:

$$H_2 - H_1 = \left(\frac{f L}{D} + \sum_i \xi_i \right) \frac{Q|Q|}{2gA_f^2} \quad (3.32)$$

$$H_2 - H_1 = \frac{f (L + nD)}{D} \frac{Q|Q|}{2gA_f^2} \quad (3.33)$$

in which:

Q	=	fluid discharge	$[m^3/s]$
A_f	=	fluid cross-sectional area	$[m^2]$
D	=	internal diameter of the pipe	$[m]$
H_1	=	upstream head	$[m]$
H_2	=	downstream head	$[m]$
f	=	Darcy-Weisbach friction factor	$[s^2/m^5]$
g	=	gravitational acceleration	$[m/s^2]$
n	=	Equivalent length local loss specification	$[-]$
ξ_i	=	Local loss coefficient i	$[-]$

3.4.1.1 Newtonian fluids: Colebrook-White

The friction Darcy-Weisbach factor f is dependent of the wall roughness k and the Reynolds number Re and is computed with one of the following equations:

$$f = 0.64, \text{ for } 0 < Re < 100, \quad (3.34)$$

$$f = \frac{64}{Re} \text{ for } 100 < Re < 2000, \quad (3.35)$$

and for turbulent flow, when $Re > 4000$, with the Colebrook-White equation:

$$\frac{1}{\sqrt{f}} = -2 \log \left(\frac{k/D}{3.7} + \frac{2.51}{Re \sqrt{f}} \right) \quad (3.36)$$

in which:

$$Re = \frac{QD}{\nu A_f} = \frac{Q}{\nu \pi D} \quad (3.37)$$

with:

- ν = kinematic viscosity $[m^2/s]$

For $2000 < Re < 4000$ f is calculated with a linear interpolation between f calculated with (3.35) for $Re=2000$ and f calculated with (3.36) and $Re=4000$.

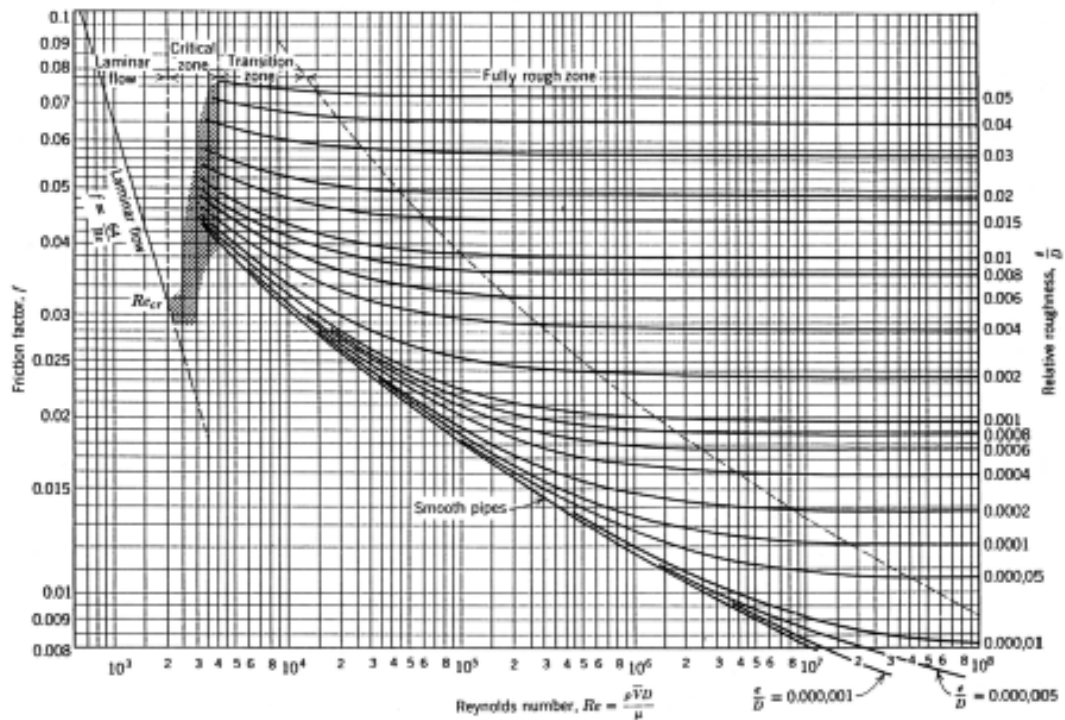


Figure 3.11: Moody-Diagram

Since the Reynolds number is dependent on the fluid discharge, which is to be computed by solving equation (3.33), in which f is present, which in its turn is dependent on the Reynolds number, cyclic dependency occurs. To solve this implicit problem the solution procedure has to be iterative.

The Reynolds numbers of the pipes for which k has been specified (the k -pipes) will initially be computed by substituting the estimated discharges Q (in the start vector) in the Colebrook-White equation (3.36). Subsequently the f -values of those pipes can be computed by solving equation (3.34), (3.35) or (3.36). Then the steady state computation can be carried out, solving equation (3.33) for all pipes (together with the equations describing the mathematical models of the other H-components and the continuity equations for all H-nodes).

After the computation of the steady state, the Reynolds numbers of all k -pipes are re-computed based on the updated discharges Q and new f -values are computed. The f -values of the pipes for which f has been specified by the user (the f -pipes) are left unaffected. The steady state computation is then repeated with the current solution vector used as start vector. This procedure is repeated until convergence is obtained.

The f - and k -values of all pipes are output.

3.4.1.2 Wall roughness k -pipes

The k -values are user specified and are generally in the order of 0.1 to 10 mm. They are dependent of material and age of the pipe. A database of k -values belonging to pipe materials may be created.

When the option “Dynamic Friction” is set to “Quasi-steady”, the friction factor f is recalculated for every time step in all calculation points. This calculation is based on the local flow velocity of the previous time step and the input concerning the wall roughness

of the pipe. For k -pipes with “Dynamic Friction” set to “none” the f -value and for the other friction models is left unaffected as is the case with f -pipes.

3.4.1.3 Friction factor f -pipes

The friction factor f is user-specified. After finding a converged steady state solution, the k -values of the f -pipes are computed with the following equation:

$$k = 3.7 D \left(10 \left(\frac{1}{2 f^{1/2}} \right) - \frac{2.51}{N_R \sqrt{f}} \right) \quad (3.38)$$

Caution should be taken when using f -pipes, because the f -value depends on the flow and cannot be known beforehand. Other pipe models where f is taken constant during unsteady calculations (k -pipe without dynamic friction and the empirical models) are limited in a similar way. Large values for the wall roughness k are an indication of erroneous application of the friction model. The user is advised to apply another friction model.

3.4.2 Non-Newtonian Fluids

WANDA supports wall friction factors for all homogeneous non-Newtonian fluids with time-independent behaviour. Non-Newtonian fluids are categorized as having time-independent, time-dependent or visco-elastic rheological behaviour. The non-Newtonian fluids with time-independent rheology include:

- dilatant, shear thickening, fluids like sand or a starch suspension,
- pseudoplastic, shear thinning, fluids like polymer solutions, many colloidal suspensions, paper pulp,
- Bingham plastics with a yield stress, like tooth paste or drilling muds and
- Herschel-Bulkley fluid or Generalised Bingham Plastic (GBP), like some mine tailings.

$$\text{Dilatant, shear thickening} \quad \tau_{yx} = K \left(\frac{du}{dy} \right)^n, n > 1 \quad (3.39)$$

$$\text{Pseudoplastic, shear thinning} \quad \tau_{yx} = K \left(\frac{du}{dy} \right)^n, n < 1 \quad (3.40)$$

$$\text{Bingham plastic} \quad \tau_{yx} = \tau_y + K \left(\frac{du}{dy} \right) \quad (3.41)$$

$$\text{Herschel-Bulkley} \quad \tau_{yx} = \tau_y + K \left(\frac{du}{dy} \right)^n, n > 0 \quad (3.42)$$

Equations (39) to (41) show that the Herschel-Bulkley rheological definition generalizes the other non-Newtonian fluids and even Newtonian fluids ($n = 1$, $K = \mu$, $\tau_y = 0$). Therefore, the Herschel-Bulkley rheological model is made available in WANDA. The fluid properties that characterise the non-Newtonian fluid (slurry) are specified in the fluid window:

Yield stress, τ_y

Viscosity coefficient, K

Exponent, n

The following steady state relation for the laminar flow regime can be derived analytically:

$$Q = \frac{\pi D^3}{32} \left(\frac{4n}{3n+1} \right) \left(\frac{\tau_w [1-x]}{K} \right)^{1/n} [1 - a \cdot x - b \cdot x^2 - c \cdot x^3] \quad (3.43)$$

where

$$x = \frac{\tau_y}{\tau_w} = \frac{4\tau_y}{\rho g D \frac{dH}{dx}}$$

$$a = \frac{1}{2n+1} \quad (3.44)$$

$$b = \frac{2n}{(n+1)(2n+1)}$$

$$c = n \cdot b$$

The laminar friction factor of a Herschel-Bulkley fluid can be derived analytically as well, using the effective viscosity concept:

$$f_{s,lam} = \frac{64}{\text{Re}_s}$$

$$\text{Re}_s = \frac{v \cdot D}{\nu_{eq}} \quad (3.45)$$

$$\nu_{eq} = \frac{gD}{4} \frac{dH}{dx} \left(\frac{4n}{3n+1} \right)^{-1} \left(\frac{\tau_w [1-x]}{K} \right)^{-1/n} [1 - a \cdot x - b \cdot x^2 - c \cdot x^3]^{-1}$$

The laminar friction factor does not exceed 6400 for reasons of numerical stability.

If the slurry Reynolds number is greater than 1000, then both the laminar and turbulent friction factor are determined and the maximum of both friction factors is applied. The turbulent wall friction of a Herschel-Bulkley fluid is based on the secant viscosity at the wall and the approach from Wilson and Thomas (1985). Since Wilson and Thomas' approach is based on a comparison with an equivalent Newtonian fluid, we will use subscripts s for slurry and n for Newtonian properties. The following computations are performed to obtain the turbulent friction factor of non-Newtonian fluids $f_{s,t}$.

$$\left(\frac{8}{f_{s,t}}\right)^{0.5} = \left(\frac{8}{f_{n,t}}\right)^{0.5} + 11.6(\alpha - 1) - 2.5 \ln \alpha - \Omega \quad (3.46)$$

Where $f_{n,t}$ denotes the turbulent friction factor of a Newtonian fluid at the same wall shear stress. Hence the following relation holds between the non-Newtonian and Newtonian friction factors and pipe velocities:

$$\tau_{w,s} = \tau_{w,n} \Leftrightarrow f_s \cdot v_s^2 = f_n \cdot v_n^2 \quad (3.47)$$

The two friction factors are computed from the Colebrook-White correlation (see "Theoretical friction models (Darcy-Weisbach)" on page 166) at the wall viscosity μ_w and Reynolds numbers.

$$\begin{aligned} \mu_w &= \frac{\tau_{w,s}}{dv(r)/dr|_w} = \tau_{w,s} \left(\frac{K}{\tau_{w,s} - \tau_y} \right)^{1/n} \\ \text{Re}_s &= \frac{\rho v_s D}{\mu_w} \\ \text{Re}_n &= \frac{\rho v_n D}{\mu_w} \end{aligned} \quad (3.48)$$

The variable α in equation (46) is the ratio of viscous sublayers of the non-Newtonian and equivalent Newtonian fluid. The variable Ω models the effect of plug flow in the core, due to a non-trivial yield stress τ_y . Since the shear stress drops linearly towards the pipe centreline for any fluid, the local shear stress always becomes smaller than the yield stress $\tau_y (>0)$ within a certain core region.

$$\begin{aligned} \alpha &= 2 \frac{(1+n \cdot x)}{(1+n)} \\ \Omega &= -2.5 \ln(1-x) - 2.5x(1+0.5x) \end{aligned} \quad (3.49)$$

Since the variables x , α , Ω , μ_w are all functions of the wall shear stress $\tau_{w,s}$ and thus functions of the resulting slurry friction factor, an iterative procedure is required to compute the slurry friction factor. The iterative procedure stops if the relative error in the friction factor has dropped to 1e-6.

As opposed to Newtonian fluids, there is no transition region between the laminar and turbulent flow regime, because in a non-Newtonian fluid the viscous sublayer is relatively thick at the beginning of the turbulent flow regime (Wilson, Thomas 1985). Therefore, the friction factor at these turbulent flows is relatively small, which explains the absence of the transition region. The applicable friction factor is simply the maximum of the laminar and turbulent friction factor:

$$f_s = \max \{ f_{s,lam}, f_{s,t} \} \quad (3.50)$$

If the user has specified a non-trivial yield stress τ_y , then a hydraulic gradient may be present in a pipe without any flow. This situation occurs if the following condition is satisfied:

$$\left| \frac{dH}{dx} \right| < \frac{4\tau_y}{\rho g D} \quad (3.51)$$

The fluid remains static until the hydraulic gradient exceeds this yield stress criterion.

During a transient event, flow reversal may occur locally in a pipe. The flow reversal may lead to fluid stiffening and activation of the yield stress, if the hydraulic gradient satisfies the yield stress criterion.

3.4.3 Empirical models

The implicit expression of the Darcy-Weisbach friction factor requires an extensive iterative calculation. Therefore empirical expressions such as the Hazen-Williams and Chezy-Manning formulae have become very popular. However the lack of theoretical basis restricts the validity. In WANDA they are included mainly because of their wide application in engineering practice. As the Darcy-Weisbach model is generally considered more accurate and the fast iteration in WANDA cancels its main drawback of implicitness, it is advised to use the Darcy-Weisbach model instead of these empirical models. The Darcy-Weisbach model is valid for all flow regimes and liquids.

The Hazen-Williams formula is the most commonly used headloss formula in the US. It cannot be used for liquids other than water and was originally developed for turbulent flow only.

The Chezy-Manning formula is more commonly used for open channel flow. Each formula uses a different pipe roughness coefficient that must be determined empirically. Table 3.2 lists general ranges of these coefficients for different types of new pipe materials. Be aware that a pipe's roughness coefficient can change considerably with age.

Material	Hazen-Williams C (unitless)	Manning's n (unitless)
Cast Iron	130 - 140	0.012 - 0.015
Concrete or Concrete Lined	120 - 140	0.012 - 0.017
Galvanized Iron	120	0.015 - 0.017
Plastic	140 - 150	0.011 - 0.015
Steel	140 - 150	0.015 - 0.017
<i>Vitrified Clay</i>	110	0.013 - 0.015

3.4.3.1 Hazen-Williams

The Hazen-Williams constants are based on "normal" conditions with flow velocities of approximately 1 m/s. This method is only valid for water at ordinary temperatures (4 to 25 °C) and turbulent flow with Reynolds numbers above 10^5 .

$$H_2 - H_1 = \frac{4.727 L Q^{1.852}}{C^{1.852} d^{4.871}} \quad (3.52)$$

3.4.3.2 Chezy-Manning

The Chezy-Manning friction model was mainly derived for uniform steady state flow (open channel flow) and therefore the validity for pipe systems is limited.

$$H_2 - H_1 = \frac{4.66LQ^2}{d^{5.33}} \quad (3.53)$$

3.5 Wave propagation speed

The velocity of the pressure wave in the pipeline depends on the compressibility of the fluid and pipeline itself. The general form of the equation for propagation speed can be written as:

$$c = \sqrt{\frac{1}{\rho \left[\frac{1}{K} + \frac{1}{A} \frac{\partial A}{\partial p} \right]}} \quad (3.54)$$

In which the bulk modulus of the liquid, K , is related to the density change of the liquid by:

$$K = \frac{\Delta p}{\Delta \rho / \rho} \quad (3.55)$$

3.5.1 Circular cross-section

For thin-wall pipes of internal diameter D and wall thickness e (i.e. $D/e > 10$), the propagation speed can be calculated from:

$$c = \sqrt{\frac{1}{\rho \left[\frac{1}{K} + \frac{D}{Ee} \right]}} \quad (3.56)$$

The Young's Modulus of Elasticity, E , depends on the material of the pipe wall. Here, the expression is given for an idealised case where the restraint factor is taken as unity, assuming the Young's modulus to be large.

3.5.2 Rectangular cross-section

For non-circular cross sections theoretical wave speeds can be derived from equation 3.57 if an expression can be found for the last term in the denominator. For a rectangular cross-section with width w and height h this term can be written as (ref. E.B. Wylie and V.L. Streeter, Fluid Transients, 1978, McGraw-Hill inc.):

$$\frac{\partial A}{A \partial p} = \frac{w}{eE} + \frac{w^4 R}{15e^3 E h} \quad (3.57)$$

In (3.55) the first term is the contribution of extension of the pipe wall. The second term is the contribution of the bending of the pipe wall. The 'rectangular factor', R , is given by

$$R = \frac{6-5\alpha}{2} + \frac{1}{2} \left(\frac{h}{w} \right)^5 \left[6-5\alpha \left(\frac{w}{h} \right)^2 \right] \quad (3.58)$$

with

$$\alpha = \frac{1 + (h/w)^3}{1 + h/w} \quad (3.59)$$

In practise, however, rectangular pipes may contain fillets (filled corners) to increase the strength and flow properties. In WANDA it is possible to include the effect of these fillets in the wave speed by indicating the size of the fillet (distance to wall). For the bending contribution the width w and height h will be reduced with this fillet-size in the equations above. The fillets are also included in calculation of the velocity via the reduction of the cross-sectional area and wet perimeter. As the fillets will only reduce the flexibility (the fillets themselves are also flexible), the reduction of h and w will not be 100% of the fillet size. Therefore, it is possible to specify the structural contribution of the fillets as a percentage of this h/w reduction. The influence of the fillets on the wave speed can be reduced by lowering the structural contribution.

3.5.3 Reduction of wave speed due to free gas

If there is homogenous distributed free gas in the pipeline, the wave speed is reduced because the bulk modulus is reduced. Equation is then written as:

$$c = \sqrt{\frac{1}{\rho_m \left[\frac{1-\alpha}{K} + \frac{\alpha}{p \cdot \gamma} + \frac{1}{A} \frac{\partial A}{\partial p} \right]}}$$

Where α is the volume fraction of gas, γ is the polytropic coefficient of the mixture, and p is the absolute pressure of the gas. ρ_m is the density of the mixture, which is:

$$\rho_m = \alpha \rho_g + (1-\alpha) \rho_l$$

Where ρ_l and ρ_g are the liquid and gas densities, respectively.

Although the effect of the free gas can be significant, the free gas is not taken into account in Wanda. In practice the flow velocity in most systems is not high enough to keep the air evenly distributed. The air bubbles will usually collapse into larger air pockets, which cause different behaviour (multiphase flow).

It should be noted that this only concerns "free" gas in the liquid. Dissolved gas doesn't influence the wave speed. Due to sudden changes in the pressure, dissolved gas can come out of solution (becomes "free" gas) after a certain incubation period. However, the passage time of the pressure surge is much smaller than the incubation period, which ensures that the pressure surge has passed before the gas can affect the wave speed.

3.5.4 Number of pipe elements

The number of elements in the pipe, n , is calculated by the following equation:

$$n = \frac{L}{c\Delta t} \quad (3.60)$$

Where L is the length of the pipeline, c is the wave speed and Δt is the time step. If the wave speed is lowered, the number of pipe elements rises.

Entering too low a Young's modulus can cause the number of pipe elements to exceed the maximum number of pipe elements and Wanda will not allow you to run any computations when this happens. Increasing the time step or the Young's modulus may remedy this.

3.6 Cavitation

3.6.1 Conceptual Model

The standard water hammer equations are used during the calculation. The absolute pressure P_{abs} may not drop below the vapour pressure P_v . The vapour pressure is dependent on the fluid and the temperature. However, during a computation it is kept constant.

When cavitation occurs at some point in the pipe, the point becomes a boundary condition for the water hammer equations in the adjacent parts of the pipe. In the spatial distribution of the discharges a discontinuity exists at this boundary. The vapour volume enters in the continuity equation. When this volume again becomes zero, the cavity collapses and the boundary condition vanishes.

3.6.2 Algorithmic Implementation

The water hammer computation is executed in all internal computational points in the pipes (the so-called W-nodes). Sets of two equations in the unknown H_P and Q_P in point P (see figure below). In the equations the H_A , Q_A , H_B and Q_B in points A and B respectively are known. The coefficients R and S are determined by pipe and fluid (liquid) properties.

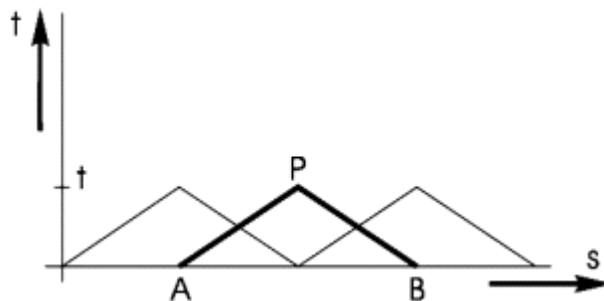


Figure 3.12: Sets of two equations in point P

The equations are:

$$H_P - H_A + R(Q_P - Q_A) + S Q_A |Q_A| = 0 \quad (3.61)$$

$$H_P - H_B - R(Q_P - Q_B) - S Q_B |Q_B| = 0 \quad (3.62)$$

with:

$$R = \frac{c_f}{g A_f}; \quad S = \frac{\lambda c_f \Delta t}{2gD A_f^2} \quad (3.63)$$

and:

$$c_f = \frac{1}{\sqrt{\rho_f \left(\frac{1}{K} + \frac{D}{Ee} \right)}} \quad (3.64)$$

The following variables are used:

A_f = fluid cross-sectional area [m²]

D = internal diameter of the pipe [m]

E	=	Young's modulus of the pipe	[N/m ²]
H	=	head	[m]
K	=	fluid bulk modulus	[N/m ²]
Q	=	fluid discharge	[m ³ /s]
c _f	=	water hammer wave speed	[m/s]
e	=	pipe wall thickness	[m]
g	=	gravitational acceleration	[m/s ²]
Δt	=	computational time step	[s]
λ	=	Darcy-Weisbach friction factor	[s ² /m ⁵]
ρ _f	=	fluid mass density	[kg/m ³]

When at a computational point, H is calculated to be less than H_v , it is simply set equal to H_v . This leads to the following boundary condition:

$$H = H_v \quad (3.65)$$

where:

$$(3.66)$$

$$H_v = \frac{P_v - P_{atm}}{\rho_f g} + \frac{v^2}{2g} + h$$

is the relative vapour head.

- h = height of computational point relative to the horizontal reference plane
- v = velocity in computational point
- P_{atm} = atmospheric pressure

Condition (3.63) and the two equations (3.61) and (3.62) now constitute an undetermined system of three equations in the two unknown variables H_P and Q_P . Instead of introducing the void fraction as the third unknown variable, the fluid velocity is assumed to be discontinuous at the computational point. The unknown Q_P is replaced by two differing discharges Q_1 and Q_2 up- and downstream of the "cavitating" computational point. A cavity is now formed, the volume of which is governed by:

$$\frac{\partial V}{\partial t} = Q_2 - Q_1 \quad (3.67)$$

or, when numerically integrated:

$$V(t) = V(t - \Delta t) + (Q_2(t) - Q_1(t))\Delta t \quad (3.68)$$

The cavity disappears when its volume is calculated to be negative. Then, to satisfy the mass balance, the last positive cavity volume is filled with liquid, according to

$$Q_1(t) - Q_2(t) = \frac{V(t - \Delta t)}{\Delta t} \quad (3.69)$$

which is equation (3.68) with $V(t) = 0$.

In the following figure examples are given for the naming and dependencies of two grid-points: one with cavitation (P) and one without cavitation (Q).

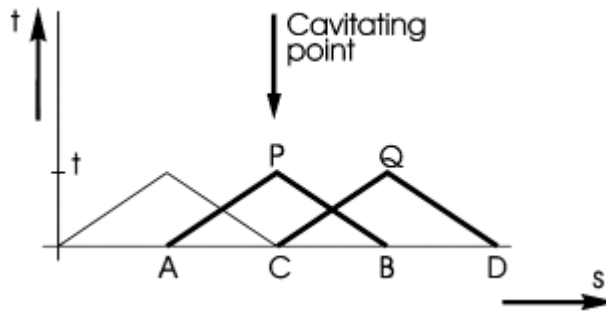


Figure 3.13: Point with (P) and without cavitation (Q).

The system of (uncoupled) equations to be solved in grid-point P is:

$$H_v - H_A + R(Q_{P1} - Q_A) + S Q_A | Q_A | = 0 \quad (3.70)$$

$$H_v - H_B - R(Q_{P2} - Q_B) - S Q_B | Q_B | = 0 \quad (3.71)$$

The system of equations to be solved in grid-point Q is:

$$H_Q - H_C + R(Q_Q - Q_{C2}) + S Q_{C2} | Q_{C2} | = 0 \quad (3.72)$$

$$H_Q - H_D - R(Q_Q - Q_D) - S Q_D | Q_D | = 0 \quad (3.73)$$

Characteristic equations in nodal sets

As long as point P in the above example is an (internal) W-node there are no problems. However, when P is at a pipe extremity (end points) it is an H-node. H and Q in these H-nodes are solved in the systems of nodal set equations. After solution of the nodal set equations it may turn out (e.g. in calculation phase 4) that condition (3.65) is not fulfilled. This may be regarded as a change in "procedure phase" of the characteristic equation of the adjacent pipe. The characteristic equation is replaced by: $H = H_v$ thus decoupling the pipe from the nodal set. The Q into that pipe is then separately computed. The cavitation volume V at the H-node must of course being kept track of. The pipe ends are thus regarded as H-procedures with procedure phases "not-cavitating" and "cavitating".

3.6.3 Software Implementation

The vapour pressure P_v is user specified. The cavitation volume V is component specific output, so it can be selected and plotted. When cavitation occurs within a pipe, the complete vector of cavitation volumes from that pipe is output, so that an animation of the cavitation volumes as function from a route can be produced.

3.7 Pipe models

3.7.1 Pressurised pipe

Compressible fluid: continuity equation is now more complicated for pipes. Fluid in other components is still assumed incompressible as the length scale is much smaller.

Waterhammer pipes are essential components in WANDA. Therefore we give a full description of the theory and solution methods.

For the description of an isothermal fluid flow through a pipe two equations are of importance:

- Momentum equation
- Continuity equation.

In water hammer theory both equations are applied on the mean flow. In other words, the flow is considered as one-dimensional.

Derivation of these one-dimensional equations can be found in any textbook on transient flow [1, 2, 3].

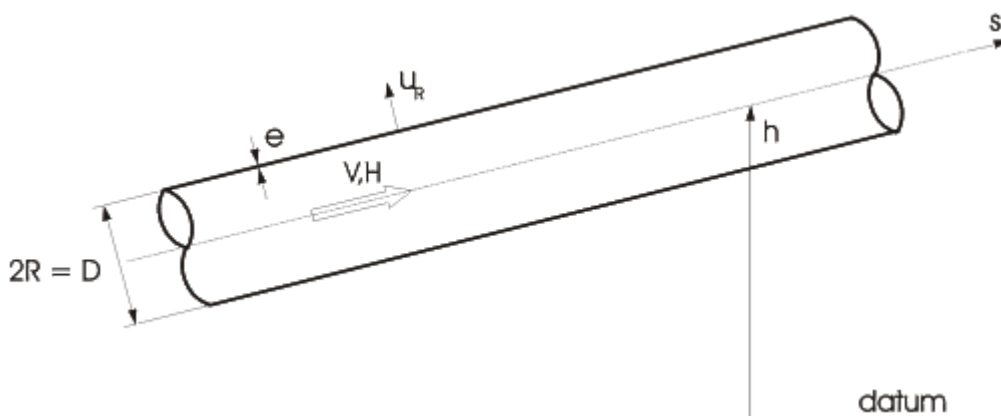


Figure 3.14: Definition of variables

The equations are (see also figure 3.14)

$$\text{Momentum: } \frac{\partial V}{\partial t} + g \frac{\partial H}{\partial s} + \frac{f V|V|}{2D} = 0 \tag{3.74}$$

$$\text{Continuity: } \frac{\rho_f}{K_f} \frac{g}{\partial t} \frac{\partial H}{\partial t} + \frac{\partial V}{\partial s} + \frac{2}{R} \frac{\partial u_R}{\partial t} = 0 \tag{3.75}$$

in which:

V	=	mean velocity	[m/s]
H	=	head	[m]
s	=	axial coordinate (along the pipe)	[m]
g	=	gravitational acceleration	[m/s ²]
f	=	friction factor	[-]
D=2R	=	internal diameter of the pipe	[m]
ρ_f	=	fluid density	[kg/m ³]
K_f	=	fluid bulk modulus	[N/m ²]
u_R	=	radial displacement of pipe wall	[m]

The head H is defined as:

$$H = \frac{P}{\rho_f g} + \frac{v^2}{2g} + h \quad (3.76)$$

In which P denotes the mean fluid pressure and h the elevation of the centreline of the pipe relative to an arbitrary datum.

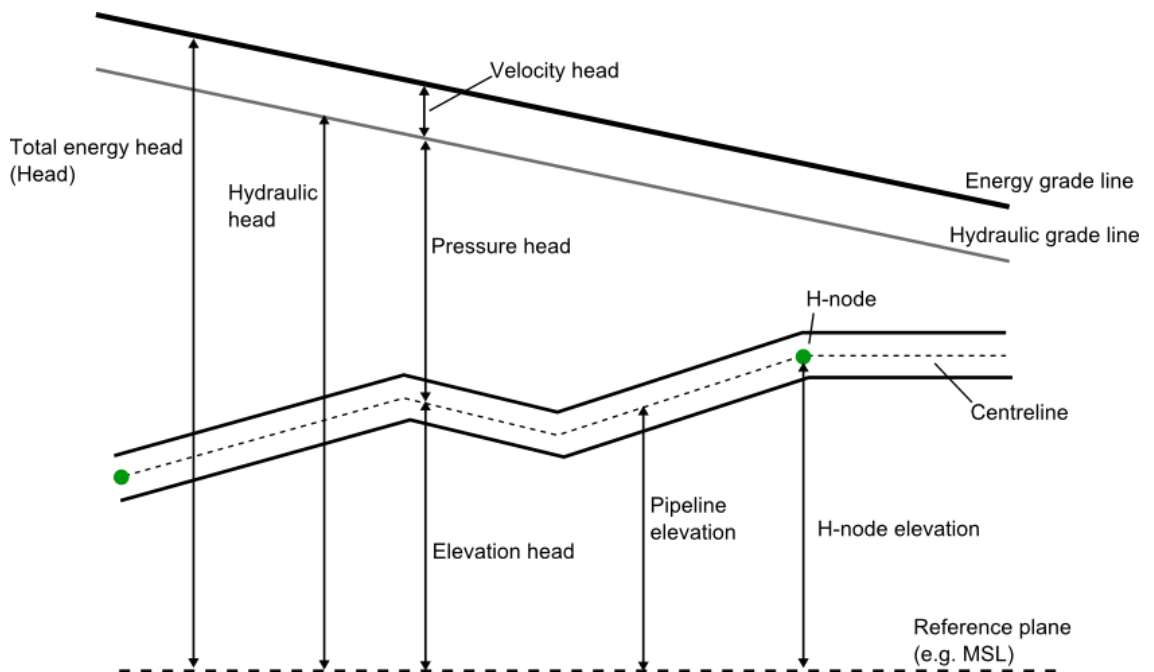


Figure 3.15: Hydraulic gradient, pipe elevation, Head and pressure.

Assumptions

- The third term in the momentum equation accounts for the friction between fluid and pipe wall. The friction is modelled the same as if the velocity were steady. This is usually done in transient flow calculations and works out well.
- In the equations (3.74) and (3.75) convective terms like $V \partial V / \partial s$ are neglected. This is allowed only when V is much smaller than the wave speed of the pressure waves. This assumption is known in literature as the acoustic approximation.
- The one-dimensional description of the fluid flow is only valid for pressure waves with low frequency. One can imagine that if the wavelength of the pressure waves is of the same magnitude as the diameter of the pipe, radial fluid inertia becomes of importance. In transient flow long wavelengths are dominant and a one-dimensional approach is valid. This assumption is known in literature as the long wavelength approximation [4].

Continuity equation

The continuity equation (3.75) can be transformed in a more convenient form. Therefore we use the stress-strain relations for the pipe:

$$\frac{u_R}{R} = \frac{1}{E} (\sigma_\phi - \nu \sigma_s) \quad (3.77)$$

$$\frac{\partial u_s}{\partial s} = \frac{1}{E} (\sigma_s - \nu \sigma_\phi)$$

or:

$$\sigma_s = \frac{E}{1-\nu^2} \left(\frac{\partial u_s}{\partial s} + \nu \frac{u_R}{R} \right) \quad (3.78)$$

$$\sigma_\phi = \frac{E}{1-\nu^2} \left(\frac{u_R}{R} + \nu \frac{\partial u_s}{\partial s} \right)$$

in which:

E	=	Young's modulus	[N/m ²]
ν	=	Poisson's ratio	[-]
σ_s	=	axial pipe stress	[N/m ²]
σ_ϕ	=	hoop stress	[N/m ²]

By means of the stress-strain relations the continuity equation can be written as:

$$\frac{\rho_f}{K_f} \frac{\partial H}{\partial t} + \frac{1}{g} \frac{\partial V}{\partial s} + \frac{2}{gE} \frac{\partial \sigma_\phi}{\partial t} - \frac{2\nu}{gE} \frac{\partial \sigma_s}{\partial t} = 0 \quad (3.79)$$

This equation can be simplified by applying again the long wavelength approximation.

This approximation states that the radial inertia of the pipe wall can be neglected. If this approximation is applied one can derive an important equation:

$$\sigma_\phi = \frac{PR}{e} \quad (3.80)$$

In fact this equation states that forces in radial direction are always in equilibrium. There are no dynamic effects involved.

Substitution of equation (3.78) in (3.79) yields:

$$\frac{\rho_f}{K_f} \frac{\partial H}{\partial t} + \frac{1}{g} \frac{\partial V}{\partial s} + \frac{2R}{gEe} \frac{\partial P}{\partial t} - \frac{2\nu}{gE} \frac{\partial \sigma_s}{\partial t} = 0 \quad (3.81)$$

The pressure P can be expressed in terms of H by means of equation (3.76):

$$P = \rho_f g (H - h)$$

Hence:

$$\frac{\partial P}{\partial t} = \rho_f g \left(\frac{\partial H}{\partial t} - \frac{\partial h}{\partial t} \right)$$

In this equation we will neglect the influence of $\partial h / \partial t$ on $\partial P / \partial t$. This can be done if we consider displacements of the pipe small compared to its diameter.

So it appears:

$$\frac{\partial P}{\partial t} = \rho_f g \frac{\partial H}{\partial t} \quad (3.82)$$

Substitution of (3.82) in (3.81) yields the continuity equation:

$$\frac{1}{c_f^2} \frac{\partial H}{\partial t} + \frac{1}{g} \frac{\partial V}{\partial s} - \frac{2\nu}{gE} \frac{\partial \sigma_s}{\partial t} = 0 \quad (3.83)$$

With:

$$c_f^2 = \frac{K_f}{\rho_f} \left(1 + \frac{D K_f}{E e} \right)^{-1} \quad (3.84)$$

where:

c_f	=	Wave propagation speed	[m/s]
K_f	=	Fluid compressibility	[N/m ²]
ρ_f	=	Fluid density	[kg/m ³]
D	=	Internal pipe diameter	[m]
E	=	Young's modulus (pipe elasticity)	[N/m ²]
e	=	Pipe wall thickness	[m]

Of course V can be replaced by Q/A_f again to yield the equations in terms of discharge Q and head H . The last term in the continuity equation accounts for the Poisson effect. This term as well as all inertia effects (radial and axial) of the pipe wall is omitted in classical water hammer theory.

Summary

The two basic equations, which describe the compressible flow in a pipe are (in terms of Q and H):

Momentum:

$$\frac{1}{A_f} \frac{\partial Q}{\partial t} + g \frac{\partial H}{\partial s} + \frac{f Q |Q|}{2D A_f^2} = 0$$

Continuity:

$$\frac{1}{c_f^2} \frac{\partial H}{\partial t} + \frac{1}{g A_f} \frac{\partial Q}{\partial s} = 0$$

with:

$$c_f^2 = \frac{K_f}{\rho_f} \left(1 + \frac{D K_f}{E e} \right)^{-1}$$

c_f represents the propagation speed of the pressure waves through the pipe, as we will see.

The method of characteristics (MOC)

If we consider H and Q as the dependent variables, the basic equations for the pipe form a set of hyperbolic partial differential equations. This hyperbolic set can be transformed into two sets of ordinary differential equations, which we will denote as C^+ and C^- equations:

$$\left. \begin{aligned} \frac{dH}{dt} + \frac{c_f}{g A_f} \frac{dQ}{dt} + \frac{f c_f}{2gD A_f^2} Q|Q| = 0 \end{aligned} \right\} C^+ \quad (3.85)$$

$$\frac{ds}{dt} = + c_f \quad (3.86)$$

$$\left. \begin{aligned} \frac{dH}{dt} - \frac{c_f}{g A_f} \frac{dQ}{dt} - \frac{f c_f}{2gD A_f^2} Q|Q| = 0 \end{aligned} \right\} C^- \quad (3.87)$$

$$\frac{ds}{dt} = - c_f \quad (3.88)$$

These equations are usually called the compatibility equations. It must be stated that every solution of this set will be a solution of the original set.

No mathematical approximations have been made.

Finite Difference Equations

To derive the finite difference equations, a "pipe" is divided into equal reaches of ds in length. The time step is chosen equal to $dt = ds / c_f$.

The equations (3.86) and (3.88) can be represented by straight lines as shown in figure 3.16.

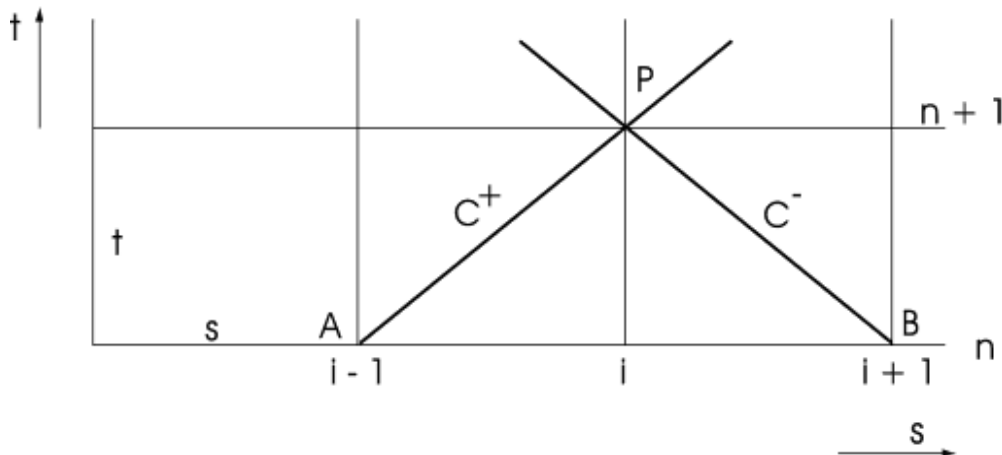


Figure 3.16: The s - t plane with its characteristic lines C^+ and C^-

If H and Q are known at point A and B, we can integrate the equations (3.85) and (3.87) along the characteristic lines to point P. After integration the unknowns H and Q in point P can be solved in principle.

Integration yields:

$$C^+ : \int_A^P \frac{dH}{dt} dt + \frac{c_f}{g A_f} \int_A^P \frac{dQ}{dt} dt + \frac{f c_f}{2gD A_f^2} \int_A^P Q|Q| dt = 0 \quad (3.89)$$

$$C^- : \int_B^P \frac{dH}{dt} dt - \frac{c_f}{g A_f} \int_B^P \frac{dQ}{dt} dt - \frac{f c_f}{2gD A_f^2} \int_B^P Q|Q| dt = 0 \quad (3.90)$$

The first two terms in the equations can easily be integrated. However, in the last term Q is not known a priori. For this friction term a first order approximation is used.

Integration yields (upper index time, lower index place):

$$H_i^{n+1} - H_{i-1}^n + R(Q_i^{n+1} - Q_{i-1}^n) + S Q_{i-1}^n / Q_{i-1}^n = 0 \quad \} C^+ \quad (3.91)$$

$$H_i^{n+1} - H_{i+1}^n - R(Q_i^{n+1} - Q_{i+1}^n) - S Q_{i+1}^n / Q_{i+1}^n = 0 \quad \} C^- \quad (3.92)$$

in which:

$$R = \frac{c_f}{g A_f} ; S = \frac{f c_f \Delta t}{2 g D A_f^2}$$

Then H_i^{n+1} can be expressed in terms of all other variables. Hence:

$$H_i^{n+1} = C_{i-1} - R Q_i^{n+1} \quad \} C^+ \quad (3.93)$$

$$H_i^{n+1} = C_{i+1} + R Q_i^{n+1} \quad \} C^- \quad (3.94)$$

in which:

$$C_{i-1} = H_{i-1}^n + R Q_{i-1}^n - S Q_{i-1}^n / Q_{i-1}^n \quad (3.95)$$

$$C_{i+1} = H_{i+1}^n - R Q_{i+1}^n + S Q_{i+1}^n / Q_{i+1}^n \quad (3.96)$$

C_{i-1} and C_{i+1} both are functions of variables at time t_n .

3.7.2 Free surface flow conduit

The free surface conduit is capable of simulating the behaviour of free surface phenomena. The application of this component is useful for simulation of slow filling and draining procedures.

The free surface conduit is capable of handling arbitrary cross sections defined by a table. The cross section table may have a real width (greater than 0) at the top. In this case it is assumed that the cross section is extended with the last entered width.

Waterhammer is not calculated in this component. Consequently the entered time step equals the free surface time step. The grid length of this component is calculated directly from this time step.

$$\Delta x_{FS} = 2 \cdot \Delta x = 5 \sqrt{gh} \cdot \Delta t_{FS} \quad (3.97)$$

A free Surface Conduit pipe may be modelled with a closed cross section; a circular cross section for example. This is handled by the free surface algorithm by adding a slot with a width equal to 0.001 times the cross section height and length equal to the grid length. The storage of this slot is used to relate the pressure fluctuation to the flow fluctuation in the completely filled open free surface pipe nodes. Hence the slot storage area of the open free surface pipe is NOT related to the waterhammer wave propagation velocity. Therefore the application of this open free surface pipe in a system with completely filled parts is limited. One should be aware of the way the pressure fluctuations in the completely filled parts are modelled.

The dynamic behaviour of a free surface pipe is described by the continuity and momentum equation.

$$\frac{\partial A(\zeta)}{\partial t} + \frac{\partial}{\partial t} (VA(\zeta)) = 0 \quad (3.98)$$

$$\frac{\partial}{\partial t} (AV) + \frac{\partial}{\partial x} (AV^2) + gA \frac{\partial H}{\partial x} + g \frac{AV|V|}{C^2 R} = 0 \quad (3.99)$$

in which the Chézy coefficient is related to the pipe friction factor.

$$C = \sqrt{\frac{8g}{\lambda}} \quad (3.100)$$

It is assumed that the gas (air) above the free surface in this pipe or channel can enter or leave the pipe freely without causing pressure surges.

The applied numerical algorithm to solve the free surface behaviour, described by equations (3.98) and (3.99), handles all possible free surface flow regimes: sub-critical, super-critical and transitions between both flow types. The numerical algorithm is based on a staggered grid with an upwind computation scheme. The staggered grid has been explicitly implemented in WANDA: the algorithm calculates heads in the even S-nodes and velocities in the odd U-nodes. The remaining heads, discharges and other dependent variables are interpolated values. The upwind condition in the numerical scheme guarantees a robust solution, which simulates transitions between sub-critical and super-critical flow regimes.

Slope

The applicability of the continuity and momentum equations for free-surface flow (Saint-Venant equations) is limited to slopes of 1:7 (8 degrees or 14%).

$$\frac{|b(i+1) - b(i)|}{\Delta x_{FS}} \leq 0.14$$

where $b(i+1)$, and $b(i)$ denote the bottom in water level nodes $i+1$, and i . This criterion is checked in the water level calculation nodes; the boundaries and the even internal calculation nodes are water level nodes.

Slope change

In addition to the maximum slope, the slope change in two consecutive free surface elements is limited. The maximum slope change should be less than 0.14. Formally:

$$\frac{|b(i+1) - 2b(i) + b(i-1)|}{\Delta x_{FS}} \leq 0.14$$

where $b(i+1)$, $b(i)$ resp. $b(i-1)$ denote the bottom in water level nodes $i+1$, i and $i-1$. This criterion implies that after an element with maximum slope, the slope in the following element may not be negative. This criterion is checked in the water level calculation nodes.

Element length

The element length of free surface pipes is calculated from the time step. However in Engineering mode the time step is not present. Therefore the user should specify explicitly a maximum element length, which is used in Engineering mode. It is recommended to set this property to $200 \cdot D$ (diameter) or less. In Transient mode the element length is calculated from the time step and it is checked whether the calculated element length is less than the maximum element length. It is recommended to set the maximum element length to $200 \cdot D$ or less, in order to obtain sufficient accuracy.

3.8 References

- Wilson, K.C., Thomas, A.D., 1985, A new analysis of the turbulent flow of non-Newtonian fluids, *Can. J. Chem. Eng.*, vol. 63 (pp. 539 – 546).
- Streeter, V.L., Wylie, E.B., 1978: *Fluid Transients*. McGraw-Hill, New York.
- Fox, J.A., 1977: *Hydraulic Analysis of Unsteady Flow in Pipe Networks*. MacMillan Press Ltd., London.
- Chaudhry, M.H., Yevjevich, V., 1981: *Closed-Conduit Flow*. Water Resources Publications.
- Lin, T.C., Morgan, G.W., 1956: Wave propagation through fluid contained in a cylindrical elastic shell. *Journal of the Acoustic Society of America*, Vol. 28, no. 6, pp. 1165-1176.
- Stelling, G.S., *Numerical methods for flooding of dry land*, *Journal of ASCE Hydraulics engineering*, 1998
- Ven Te Chow, Ph.D., 1973: *Open Channel Hydraulics*. McGraw-hill book Company, New York. ISBN 0-07-085906-X
- M. Hnif Chaudhry, *Open-Channel Flow*, 1993, Prentice Hall, Inc. ISBN 0-13-637141-8

4. WANDA Liquid

4.1 Liquid components and nodes

In this part of the Help documentation all available hydraulic-components and nodes for the liquid domain are described. Each class description of the H-component contains the symbol, the list of types and the mathematical model. Each type description consists of the input list (with descriptions, dimensions and ranges), the H-component specific output list, the H-actions list and a list of error messages and explanations.

Please note that not all classes and types have to be present in your version of WANDA since some of them are optional.

With the introduction of Wanda 4 there are several types of liquid nodes available. They are explained in this chapter as well.

Note:

The general annotation H-component and H-node must be read as Liquid component and Liquid node.

4.2 Overview of Liquid components

Currently, the following components as listed below are available in WANDA. Your license specifies if you are authorised to use them.

Class	Type	Short description
Air Valve	In/outlet	See Vent, Air valve.
AIRVha (class)	Airvessel horizontal vented	Horizontally positioned, cylindrical, vented air vessel
AIRVhn (class)	Airvessel horizontal non-vented	Horizontally positioned, cylindrical, non-vented air vessel
AIRVva (class)	Airvessel vertical vented	Vertically positioned, prismatic (constant area), vented air vessel
AIRVvn (class)	Airvessel vertical non-vented	Vertically positioned, prismatic (constant area), non-vented air vessel
Airvessel Vertical Hybrid (class)	Airvessel vertical non-vented	Vertically oriented air vessel with an air valve
Inclined air vessel (class)	Inclined air vessel (class)	Air vessel with a cylindrical volume oriented at an angle with horizontal plane
BOUNDH (class)	BOUNDH (reservoir)	Constant or time dependent head
BOUNDQ (class)	BOUNDQ (reservoir)	Constant or time dependent discharge
Channel (class)	Channel	Short open channel with constant slope
CHECKV (class)	Checkvalve (ideal)	ideal check valve; closes if $Q < 0.0$, opens if $H1 > H2$
	Checkvalve (deltaP for reopen)	ideal check valve; closes if $Q < 0.0$, opens if $P1-P2 > \Delta P$
	CHECKV -	Check valve using the undamped dynamic

Class	Type	Short description
	undamped	characteristic (with dimensions)
	CHECKV - undamped (dimensionless)	Check valve using the undamped dynamic characteristic, dimensionless
Collector (class)	Collector	Short open channel with constant slope and lateral inflow
Conduit	Conduit	Pipe with free surface flow (atmospherical) and pressurised flow without waterhammer
Control valve	FCV	Flow control valve
	PdCV	Pressure control valve downstream side
	PuCV	Pressure control valve upstream side
Damper, Surge vessel	Damper, Surge vessel	Vertically positioned, bladder or piston type damper
INFPIP (class)	Infinet Pipe (H-bound)	Infinite pipe with initial pressure head
	Infinet Pipe (Q-bound)	Infinite pipe with initial discharge
Junctions	T-JUNCTION	3-node resistance junction, 90 degree
	Y-JUNCTION	3-node resistance junction, sharp angle
	X-JUNCTION	4-node resistance junction, 90 degree
Merger	Merger	3 node junction which merges 2 separate branch flows into a combined flow
ORIFICE (class)	Orifice plate	Orifice plate complying with ISO 5167-2:2003
PIPE	Pressurised pipe	Pressurized pipe with different friction models
PIPE FFP (class)	Fast filling pipe	The fast filling pipe can be used to simulate the fast filling of a single pipeline
PRV (class)	PRV (simplified char.)	This PRV monitors the system pressure and opens upon exceedance of a set pressure. The opening pressure, as well as the full lift pressure and discharge, must be specified. (accumulation is taken into account). Opening time, friction, choking flow and valve dynamics are not taken into account.
	PRV (Ideal)	This PRV monitors the system pressure and opens upon exceedance of a set pressure. Opening time, friction, accumulation, choking flow and valve dynamics are not taken into account.
PUMP (class)	Pump	Complete pump model with various data and drives; positive and negative speed.
Pump Energy	PUMPSCEN (Pump Scenario)	To calculate the energy consumption of a pumping station, (belongs to module PumpEnergy)
	Q_PATRN (discharge pattern)	Discharge pattern in a certain period, to calculate the energy consumption of a pumping station, (belongs to module PumpEnergy)
	Q_FREQ (discharge frequency)	Discharge frequency in a certain period, to calculate the energy consumption of a pumping station, (belongs to module PumpEnergy)

Class	Type	Short description
	SYSCHAR (System characteristic)	System characteristic to calculate the duty point of pump(s) in a pump scenario; (belongs to module PumpEnergy)
PUMPSS (class)	PUMPSS (Piston Pump)	Single cylinder, single-acting pump with mechanically driven piston
QSCHAR (Class)	QSCHAR (BoundQ for system characteristic)	Varying discharge to determine the system characteristic
RSVOIR (class)	RSVOIR (reservoir)	Reservoir with initial level and finite size, constant or variable area
RESIST (class)	Resist quadr. xi	Dimensionless ξ -value from loss equation $\Delta H = \xi v v / 2g$
	Resist quadr. C	C-value from loss equation $\Delta H = C Q Q $
	Resist linear C	C-value from loss equation $\Delta H = C Q$
	Resist quadr. Initial Q	Initial Q is specified, C from loss equation $\Delta H = C Q Q $ is computed
	Resist 2-way quadr.xi	Dimensionless ξ -value from loss equation $\Delta H = \xi v v / 2g$ ξ -value depends on flow direction
	Resistance polynomial	Resistance according user defined polynomial
Shaft	Shaft	Shaft with inflow
Splitter	Splitter	3-node junction which splits up a combined flow into 2 separate branch flows
SURGTW (class)	Surgetower constant area	Storage area is independent of liquid height (pressure head)
	Surgetower height dep. area	Storage area is a function of liquid height (pressure head)
TANK (class)	TANK (constant pressure)	Pressurised tank with initial pressure and finite size
TAP (class)	Tap return flow	Calculated constant resistance based on delivery rate and downstream pressure. If system head drops below downstream head, TAP supplies back to system
	Tap non-return flow	Calculated constant resistance based on delivery rate and downstream pressure. If system head drops below downstream head, TAP does not supply back to system
	Tap non-return flow with DP reopen	If system pressure drops below downstream pressure, TAP does not supply back to system. Supply to system (re)starts when a minimum required pressure is exceeded.
Turbine	Turbine	Turbine whose speed is controlled by its vane position, absorbed power and up/downstream conditions
VALVE	VALVE	Control or block valve with choice out of four predefined Deltares standard head loss characteristics or user characteristics (Cv Kv, Xi)
VENT (class)	VENT (discharge coeff.)	Vent with the possibility of air in/outlet, air inlet only or air outlet only.

Class	Type	Short description
		Capacity specified with discharge coefficients
	VENT (inflow/outflow char.)	Vent with the possibility of air in/outlet, air inlet only or air outlet only. Capacity specified by inflow/outflow characteristics
VENT2LEG (class)	Filling pipe section with airvalve	Vent with the possibility of air in/outlet, air inlet only or air outlet only. Has two connections points, models the filling of the pipeline accurately
Venturi (class)	Venturi_open	Venturi for discharge measurement in open channel
V-weir (class)	Vee notch Weir	Thin plate weir with one or more triangular shaped notches
WEIR (class)	WEIR (two way)	Sharp edged weir, capable of simulating discharge in positive and negative direction. Storage must be modelled separately.
WEIR_S (class)	Outlet WEIR (incl. storage)	Sharp edged weir, only be used for boundary conditions. Only capable of simulating discharge in positive direction. Upstream storage is included.

4.3 Modular concept of H-component properties

In the property window all input and output properties are displayed. The property window can be divided in 5 parts:

General input data	Name, comment, keywords etc
Component specific input data	Type dependent
Action table	Type dependent for active types only (H-actions)
General output data	Messages, Pressure, Head, Discharge and Velocity
Component specific output data	Type dependent

The visibility of some properties can be managed using the checkboxes in the upper part of the Mode & Options window in the "Model" menu.

The type dependent properties are described separately for each component type. Some details about the action table are described in next chapter.

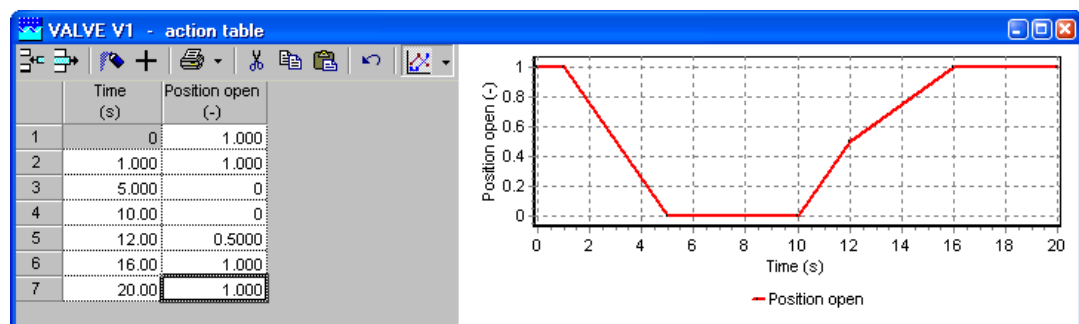
4.3.1 Specify actions

Hydraulic transients in a piping system are caused by changes in the flow conditions. These changes are initiated by manoeuvring H-components in the system, called actions. Not all components in a pipeline system can be activated. A component is categorised as either an active or a passive component, depending on its abilities.

An action is described using a control parameter specified as a function of time. The controllable parameter varies by component. g198

Notes

1. The interpretation of an action into an action table is illustrated below using the example of a control valve. Note that values of the control parameter at times between two successive points in the action table are obtained using linear interpolation.



The valve is fully open at $t = 0$ s;

- it begins to close at a constant rate;
- the closure is complete after 5 s;
- it remains closed until $t = 10$ s;

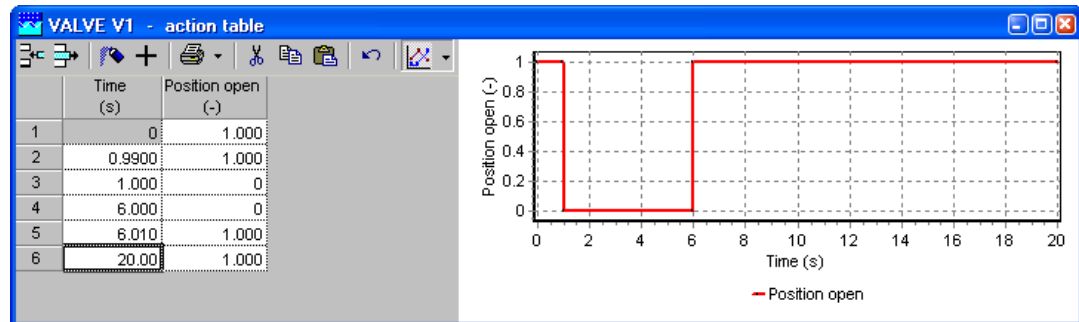
It turns open again

- the opening rate is slower between $t = 12$ s and $t = 16$ s;
- the valve is fully open after 16 s;
- the valve remains open until the end of the computation;

2. Sudden actions can be modelled by reducing the time steps in the action table to one calculation time step. If necessary the calculation time step should be reduced.

Example

The following action table describes that the valve is open at $t = 0$ s. It is suddenly closed at $t = 1.0$ s, remains closed for 5 seconds, then suddenly opens and remains open until $t = 20.0$ s.



Theoretically, a mathematical step-function can be used to describe an instantaneous action, for example, the instantaneous closure of a valve. However, such an action is unrealistic as it always takes some time in practice. The approximation above is more realistic than a mathematical step-function.

3. The last point in the action table is not necessarily the end time of the computation. WANDA uses the last entry in the table and assumes it does not change until the end of computation. Consequently the last entry in the table above may be removed.

4.4 Hydraulic nodes

4.4.1 General

An H-node connects two or more H-components which each other. In the diagram this H-node is represented by a connection line, connected to the hydraulic connect point of a H-component (blue dot). In case more than two connect points have to be connected the H-node is compounded of a master and one or more slave connection lines, thus forming a tree. For more information about the connection line, see "Connecting components" on page 34.

4.4.1 Pressure definition for H-node

In the H-node the main variable H (Head) is calculated from a set of equations as explained in "Implementation" on page 157. The pressure P is derived from this head according Bernoulli (see "Bernoulli's theorem" on page 148).

The total pressure is derived from this total head, according to:

$$P_{Total} = \rho g(H - z)$$

In which

P_{Total}	= total pressure [N/m ²]
ρ	= density of fluid [kg/m ³]
g	= gravitational acceleration [m/s ²]
H	= total head [m]
z	= elevation of H-node [m]

The pressure in the connect points of the adjacent H-components are derived from the pressure of the H-node. If the adjacent H-component has a diameter property (e.g: VALVE, PIPE), the velocity v can be determined and thus, the velocity head can be taken into account. Please note that only the velocity head of pipes which are flowing into the node is taken into account, according:

$$P = \rho g(H - z - \frac{v^2}{2g})$$

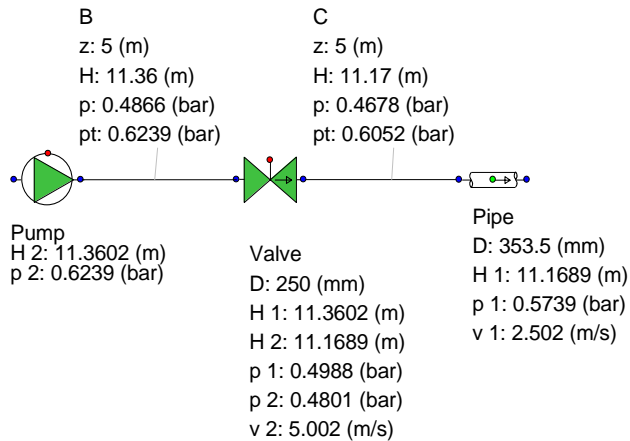
Because the diameters of the adjacent H-components may differ or the elevation of the centreline (in case of PIPE) may differ, the local pressure in the individual connect points may differ. From a mathematical view the connect points and the H-node is the same location.

The H-node model knows the elevation and diameters of the adjacent H-components. For all adjacent components the lowest pressure is computed at the soffit.

The pressure shown for the H-node is the lowest connect point pressure.

Example:

In the system below the discharge $Q = 884 \text{ m}^3/\text{h}$. In the Mode & Options menu the pressure output reference (only used for PIPE output) is set to "centreline".



Consider H-node C and adjacent H-components:

The energy head (or total head) = 11.17 m.

Elevation of H-node and elevation begin of pipe = 5 m

$$\text{Velocity head of Valve: } \frac{v^2}{2g} = \frac{5 \times 5}{2 \times 9.81} = 1.27 \text{ m}$$

$$\text{Velocity head in Pipe: } \frac{v^2}{2g} = \frac{2.5 \times 2.5}{2 \times 9.81} = 0.32 \text{ m}$$

$$\text{Pressure downstream side Valve: } P = 1000 \times 9.81 \times (11.17 - 5.00 - 1.27) = 0.480 \text{ barg}$$

$$\text{Pressure upstream side Pipe at CL: } P = 1000 \times 9.81 \times (11.17 - 5.00 - 0.32) = 0.574 \text{ barg}$$

$$\text{Total Pressure in H-node C: } P = 1000 \times 9.81 \times (11.17 - 5.00) = 0.605$$

Lowest pressure in H-node C based on soffit location: Valve:

$$P = 1000 \times 9.81 \times (11.17 - 5.00 - 1.27 - 0.125) = 0.468 \text{ barg}$$

$$\text{Pipe: } P = 1000 \times 9.81 \times (11.17 - 5.00 - 0.32 - 0.177) = 0.556 \text{ barg}$$

Lowest pressure at Valve is shown in the property window

4.4.2 Cavitation in H-node

If the pressure drops below the vapour pressure, cavitation occurs (see "Cavitation" on page 176). Cavitation in an H-node is only taken into account if the H-node is connected to a PIPE.

The H-node model knows the elevation and diameters of the adjacent H-components. For all adjacent components the lowest pressure is computed at the soffit. If this lowest pressure drops below the vapour pressure, cavitation occurs.

The lowest pressure in an H-node is computed according:

$$P_{lowest} = \text{MIN}_i \left(\rho g \left(H - z_{soffit\ i} - \frac{v_i^2}{2g} \right) \right)$$

In wich

- P_{lowest} = lowest pressure [N/m²]
- ρ = density of fluid [kg/m³]
- g = gravitational acceleration [m/s²]
- H = total head [m]
- $Z_{soffit\ i}$ = soffit of “diameter-defined” H-component_i [m]
- v_i = velocity at begin or end of of H-component_i [m/s]

The void fraction is expressed as the cavitation vapour volume divided by the sum of the half-element volumes of the adjacent pipes.

4.4.3 Standard output of H-node types

Standard output for each type of H-node is:

- Head [m]
- Pressure [N/m²] (the lowest pressure of the connected H-components)

Node specific output for the H-node is:

- Total Pressure [N/m²]
- Void fraction [-]

4.4.4 Available H-node types

The following types of H-nodes are available and are described in next sections:

- H-node
- H-node with initial head
- H-node with conditional initial head

4.4.5 H-node

This H-node is the standard hydraulic node, by default generated after connecting of two H-components.

The output (H and P) can be calculated if the H-node is part of a fully connected (“open”) hydraulic network with one or more a boundary conditions for the Head (e.g. BOUNDH, TANK). “Open” means that each H-component has a certain relation between head and discharge and that there is at least one component prescribing the head or pressure.

Input properties

Description	Input	unit	remarks
Elevation	Real	[m]	

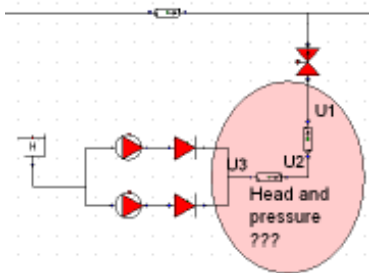
Messages

Message	Message	Explanation
Cavitation in steady state not allowed	error	The lowest pressure in the H-node is less than the vapour pressure
Pressure < P _{vapour} ; Cavitation not supported for H-node without PIPE connections	warning	In this H-node there is no PIPE component connected. The cavitation model needs at least 1 PIPE to compute the vapour volume. This warning is only given the first time the pressure condition is true; the rest of the simulation is physically invalid.
Cavitates	informative	H-node starts cavitating; that means that the lowest pressure becomes less than the vapour pressure
Cavitation collapses	informative	Void fraction becomes 0
Change Type to Hydraulic node with initial head	error	H-node is part of an isolated part of the network in which no information is available for the head; the user must define the head in this part of the model. See "H-node with initial head" on page 196.

4.4.6 H-node with initial head

If the H-node is part of a “closed” or “isolated” hydraulic network, there is no connection with a boundary condition like a BOUNDH and the actual head in the steady state computation can not be determined.

This is explained in the figure below: the head in the H-nodes U1, U2 and U3 can not be calculated.



It is therefore necessary to define the head for this area in one of the H-nodes. It is up to the user which H-node head will be prescribed (changed into the type: H-node with initial head).

Note: if this “isolated network” occurs during the transient computation, the head of the previous time step will be used.

The type of the H-node can be changed via the property window (drop down list in “Type” field)

Input properties

Description	Input	unit	remarks
Elevation	Real	[m]	
Initial head	Real	[m]	

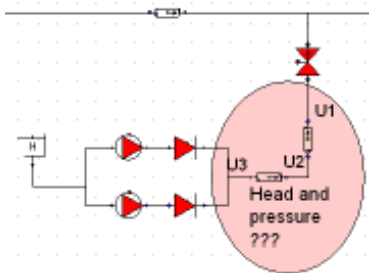
Messages

Message	Message	Explanation
Cavitation in steady state not allowed	error	The lowest pressure in the H-node is less than the vapour pressure
Pressure < Pvapour; Cavitation not supported for H-node without PIPE connections	warning	In this H-node there is no PIPE component connected. The cavitation model needs at least 1 PIPE to compute the vapour volume. This warning is only given the first time the pressure condition is true; the rest of the simulation is physically invalid.
Cavitates	informative	H-node starts cavitating; that means that the lowest pressure becomes less than the vapour pressure
Cavitation collapses	informative	Void fraction becomes 0

4.4.7 H-node with conditional initial head

If the H-node is part of a “closed” or “isolated” hydraulic network, there is no connection with a boundary condition like a BOUNDH and the actual head in the steady state computation cannot be determined.

This is explained in the figure below: the head in the H-nodes U1, U2 and U3 cannot be calculated.



It is therefore necessary to define the head for this area in one of the H-nodes. It is up to the user which H-node head will be prescribed (changed into the type: H-node with initial head). However when using for example the parameter script it can be useful that the node only describes the pressure and temperature when they are part of an isolated system. For this purpose the conditional init H node can be used. This node only pre describes its head when it is part of an isolated system. Note: if this “isolated network” occurs during the transient computation, the head of the previous time step will be used. The type of the H-node can be changed via the property window (drop down list in “Type” field)

Input properties

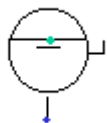
Description	Input	unit	remarks
Elevation	Real	[m]	
Initial head	Real	[m]	

Messages

Message	Message	Explanation
Cavitation in steady state not allowed	error	The lowest pressure in the H-node is less than the vapour pressure
Pressure < P _{vapour} ; Cavitation not supported for H-node without PIPE connections	warning	In this H-node there is no PIPE component connected. The cavitation model needs at least 1 PIPE to compute the vapour volume. This warning is only given the first time the pressure condition is true; the rest of the simulation is physically invalid.
Cavitates	informative	H-node starts cavitating; that means that the lowest pressure becomes less than the vapour pressure
Cavitation collapses	informative	Void fraction becomes 0

4.5 Air Vessel

4.5.1 AIRVha (class)



Horizontal vented air vessel

Supplier type

Type label	Description	active
Airvessel horizontal vented	Horizontally positioned, cylindrical, vented air vessel	No

4.5.1.1 Mathematical model

A horizontal air vessel or air chamber with air inlet (vented air chamber) can be modelled by the same equations as the air vessel without air inlet (see "AIRVhn (class)" on page 201) as long as the fluid level is higher than the air inlet level. When the fluid level is below the air inlet level the air vessel behaves like the 'surge tower' and the air pressure remains atmospheric.

As long as the fluid level is above the air inlet level the governing equations are:

$$PV^k = C \tag{1}$$

and:

$$Q = dV / dt \tag{2}$$

in which:

P	=	absolute air pressure	[N/m ²]
V	=	air volume	[m ³]
k	=	Laplace coefficient	[-]
C	=	constant	[Nm] if k = 1

The Laplace coefficient depends on the thermodynamic behaviour of the air. Isothermal expansion is described by $k = 1$. Adiabatic expansion is described by $k = 1.4$.

If the fluid level is below the air inlet level the behaviour is governed by equation (2) solely. In steady state the air vessel does not supply fluid. The governing equation then is:

$$Q = 0 \tag{3}$$

4.5.2 Airvessel horizontal vented

4.5.2.1 Hydraulic specifications

description	input	unit	range	default	remarks
top level	real	[m]			
diameter	real	[-]	(0-100]		
length	real	[m]	(0-100]		See remarks
air inlet level	real	[m]			See remarks
Laplace coefficient	Real	[-]	[1-1.4]		

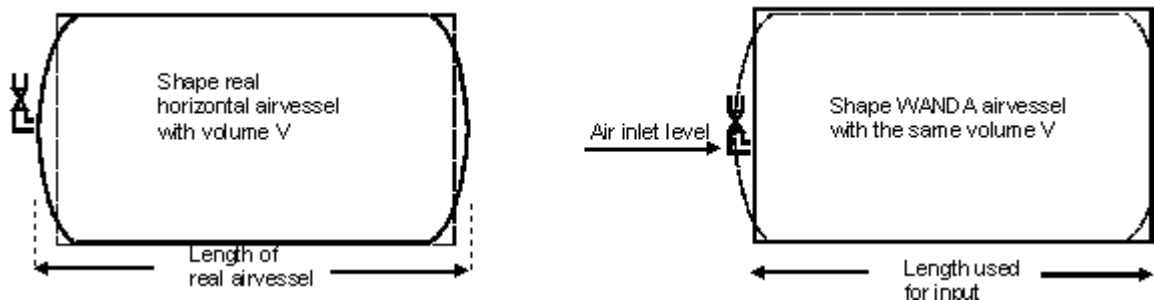
See also: "Mathematical model" on page 199.

Remarks

"Top level" and "air inlet level" are related to the horizontal reference plane.

The air inlet level depicts the level where the air inlet enters the airvessel.

An airvessel in WANDA is modelled with a straight instead of rounded sides. The volume of the real airvessel and the WANDA airvessel should be the same. The length used for input should therefore be smaller than the length of the real airvessel, depending on its shape.



4.5.2.2 Component specific output

Fluid level [m]

Air pressure (absolute) [N/m².abs]

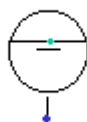
4.5.2.3 H-actions

None

4.5.2.4 Component messages

Message	Type	Explanation
Air inlet level not in between top and bottom level of air chamber	Error	The air inlet level must be in between the top and bottom level, in order to have a defined amount of air in the vessel.
Empty air chamber	Error	The fluid level has dropped below the bottom. The storage area becomes zero and further calculations can not be carried out.
Air inlet closes	Info	The fluid level has risen above the air inlet level. Air compression starts to be taken into account.
Air inlet opens	Info	The fluid level has dropped below the air inlet level. Air decompression is no longer taken into account.
Air inlet is closed	Steady state	The initial fluid level is above the air inlet level.
Air inlet is open	Steady state	The initial fluid level is below the air inlet level.

4.5.3 AIRVhn (class)



Horizontal non-vented air vessel

Supplier type

type label	description	active
Airvessel horizontal non-vented	Horizontally positioned, cylindrical, non-vented air vessel	No

4.5.3.1 Mathematical model

A horizontal air vessel or air chamber without air inlet can be modelled by two equations just as the vertical chamber.

The first equation describes the behaviour of the enclosed air:

$$PV^k = C \quad (1)$$

in which:

P	=	absolute air pressure	[N/m ²]
V	=	air volume	[m ³]

k	=	Laplace coefficient	[-]
C	=	constant	[Nm] if k = 1

The Laplace coefficient depends on the thermodynamic behaviour of the air. Isothermal expansion is described by $k = 1$. Adiabatic expansion is described by $k = 1.4$.

The second equation governs the amount of supplying discharge Q :

$$Q = -dV_f / dt \quad (2)$$

in which V_f denotes the fluid volume. Note that this equation is different from the equation (2) for vertical chambers. This is due to the fact the storage surface of a horizontal chamber is not constant.

The minus sign indicates that the chamber supplies fluid to the system when the fluid level is decreasing in time.

Equation (2) can also be formulated as:

$$Q = dV / dt \quad (3)$$

where variable V refers to the air volume. In steady state the air vessel does not supply fluid. Hence in steady state the governing equation is:

$$Q = 0 \quad (4)$$

4.5.4 Airvessel horizontal non-vented

4.5.4.1 Hydraulic specifications

description	Input	unit	range	default	remarks
top level	real	[m]			
diameter	real	[m]	(0 100]		
length	real	[m]	(0-100]		See remarks
air quantity by	Fluid Level/ Air volume/ Constant C				See remarks
initial fluid level	real	[m]			If "air quantity by" = Fluid Level
initial air volume	real	[m ³]			If "air quantity by" = Air volume
initial C in P*V = C	real	[J]			If "air quantity by" = Constant C
Laplace coefficient	real	[-]	[1-1.4]		

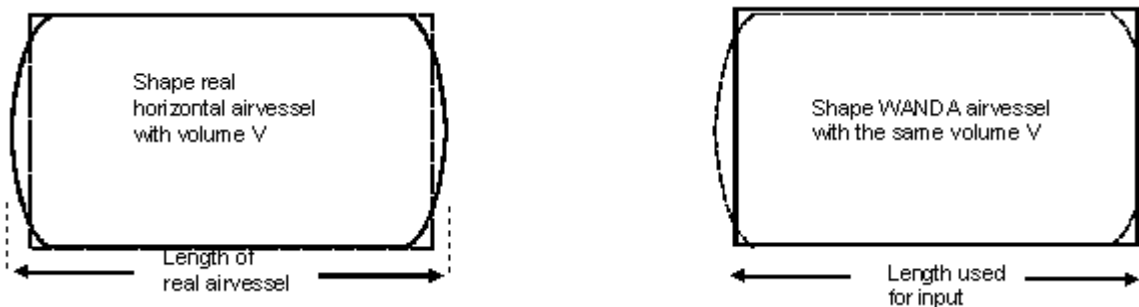
See also: "Mathematical model" on page 201

Remarks

“Top level” and “air inlet level” are related to the horizontal reference plane.

The air volume and the fluid level in the airvessel are determined by either the initial fluid level, the initial air volume or the initial constant C. One quantity has to be given by the user, the other two quantities are calculated by Wanda. Isothermal behaviour is assumed during the steady state computation.

An airvessel in WANDA is modelled with a straight instead of rounded sides. The volume of the real airvessel and the WANDA airvessel should be the same. The length used for input should therefore be smaller than the length of the real airvessel, depending on its shape.



4.5.4.2 Component specific output

Fluid level [m]

Air pressure (absolute) [N/m².abs]

Air volume [m³]

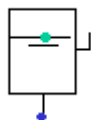
4.5.4.3 H-actions

None

4.5.4.4 Component messages

Message	Type	Explanation
Initial fluid level not in between top and bottom level of air chamber	Error	This is an input error.
Empty air chamber	Error	The fluid level has dropped below the bottom. The storage area becomes zero and further calculations can not be carried out.
Accuracy not obtained in computing fluid level as function of volume with bisection method	Warning	

4.5.5 AIRVva (class)



Vertical vented air vessel

Supplier type

type label	description	active
Airvessel vertical vented	Vertically positioned, prismatic (constant area), vented air vessel	No

4.5.5.1 Mathematical model

A vertical air vessel or air chamber with air inlet (vented air chamber) can be modelled by the same equations as the air vessel without air inlet (see "AIRVhn (class)" on page 201) as long as the fluid level is higher than the air inlet level. However when the fluid level is below the air inlet level the air vessel behaves like the 'surge tower' and the air pressure remains atmospheric.

As long as the fluid is above the air inlet level the governing equations are:

$$PV^k = C \quad (1)$$

in which:

P	=	absolute air pressure	[N/m ²]
V	=	air volume	[m ³]
k	=	Laplace coefficient	[-]
C	=	constant	[Nm] if k = 1

and:

$$Q = -A_{av} \frac{dx_f}{dt} \quad (2)$$

in which A_{av} denotes the storage surface and x_f the fluid level in the air chamber with respect to the global co-ordinate system. The minus sign indicates that the chamber supplies fluid to the system when the fluid level is decreasing in time.

When the fluid level is below the air inlet level the governing equation is simply:

$$Q = -A_{av} \frac{dH}{dt} \quad (3)$$

For steady state calculations the air vessel does not supply fluid and hence the equation for steady state is simply:

$$Q = 0 \tag{4}$$

4.5.6 Airvessel vertical vented

4.5.6.1 Hydraulic specifications

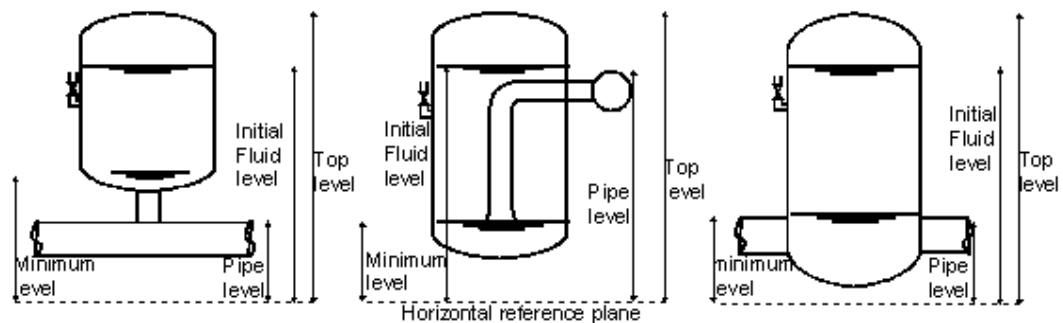
description	Input	unit	range	default	remarks
top level	real	[m]			See remarks
bottom level	real	[m]			See remarks
air inlet level	real	[m]			See remarks
chamber area	real	[m ²]	(0-100)		
Laplace coefficient	real	[-]	[1-1.4]		

See also "Mathematical model" on page 204

Remarks

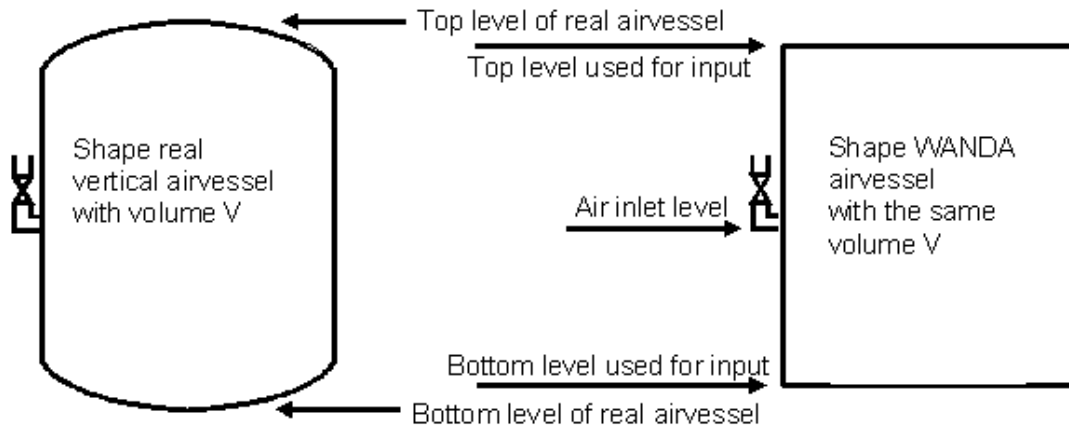
The "Bottom level" is used to check if the vessel becomes empty during the simulation; that means that "Bottom level" is the allowed minimum fluid level.

The meaning depends on the constructive design of the vessel; see pictures below



The air inlet level depicts the level where the air inlet enters the airvessel.

An airvessel in WANDA is modelled with a straight instead of a rounded top and bottom. The volume of the real airvessel and the WANDA airvessel should be the same. The top and bottom level should therefore be taken respectively lower and higher than the level of the real airvessel. How much lower and higher depends on the shape of the real airvessel.



Meaning of Pressure in property window Airvessel:

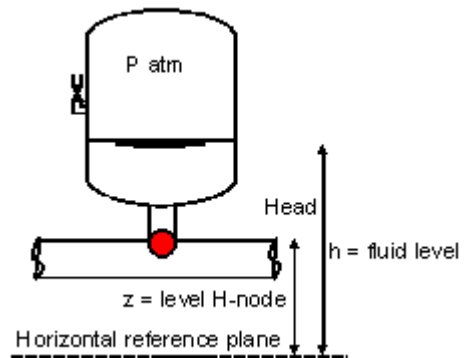
Head = Head in H-node of airvessel

Pressure P (H-node) = $\rho \cdot g \cdot (H-z)$ (relative to atmosphere)

z = height specified by particular H-node

Air pressure (absolute) = $\rho \cdot g \cdot (H-h) + P_{atm}$

h = fluid level



Component specific output

Fluid level [m]

Air pressure (absolute) [N/m².abs]

H-actions

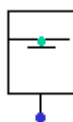
None

4.5.6.2 Component messages

Message	Type	Explanation
Air inlet level not in between top and bottom level of air chamber	Error	The air inlet level must be in between the top and bottom level, in order to have a defined amount of air in the vessel.
In computation of initial fluid	Error	has been violated somehow.

level. Check input data		
Empty airchamber	Warning	Computation can continue, since storage area is constant independent of the height. The results will not be physically correct, but may assist in the further design of the air vessel.
Air inlet closes	Info	The fluid level has risen above the air inlet level. Air compression starts to be taken into account.
Air inlet opens	Info	The fluid level has dropped below the air inlet level. Air decompression is no longer taken into account.

4.5.7 AIRVvn (class)



Vertical non-vented air vessel

Supplier type

Type label	description	active
Airvessel vertical non-vented	Vertically positioned, prismatic (constant area), non-vented air vessel	No

4.5.7.1 Mathematical model

A vertical air vessel or air chamber without air inlet can be modelled by two equations. The first equation describes the behaviour of the enclosed air:

$$PV^k = C \quad (1)$$

in which:

P	=	absolute air pressure	[N/m ²]
V	=	air volume	[m ³]
k	=	Laplace coefficient	[-]
C	=	constant	[Nm] if k = 1

The Laplace coefficient depends on the thermodynamic behaviour of the air. Isothermal expansion is described by $k = 1$. Adiabatic expansion is described by $k = 1.4$.

The second equation governs the amount of supplying discharge Q:

$$Q = -A_{av} \frac{dx_f}{dt} \quad (2)$$

in which A_{av} denotes the storage surface and x_f the fluid level in the air chamber with respect to the global co-ordinate system. The minus sign indicates that the chamber supplies fluid to the system when the fluid level is decreasing in time.

In steady state the air vessel does not supply fluid. Hence in steady state the governing equation is:

$$Q = 0 \tag{3}$$

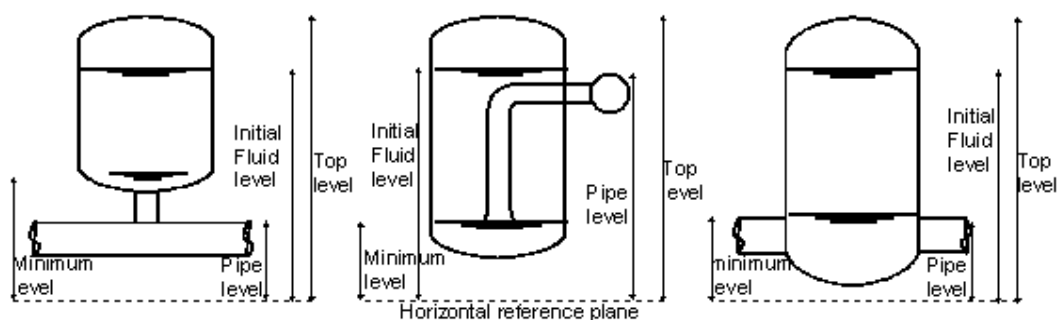
4.5.8 Airvessel vertical non-vented

4.5.8.1 Hydraulic specifications

description	Input	unit	range	default	remarks
top level	real	[m]			See remarks
bottom level	real	[m]			See remarks
chamber area	real	[m ²]	(0-100)		
air quantity by	Fluid Level/ Air volume/ Constant C				See remarks
initial fluid level	real	[m]			If "air quantity by" = Fluid Level
initial air volume	real	[m ³]			If "air quantity by" = Air volume
initial C in P*V = C	real	[J]			If "air quantity by" = Constant C
Laplace coefficient	real	[-]	[1-1.4]		

See also "Mathematical model" on page 207.

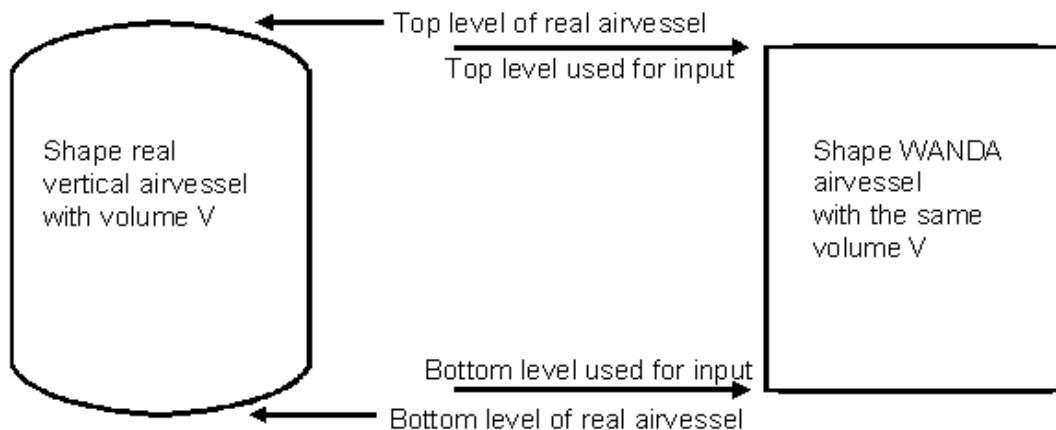
Remarks



The air volume and the fluid level in the airvessel are determined by either the initial fluid level, the initial air volume or the initial constant C. One quantity has to be given by

the user, the other two quantities are calculated by Wanda. Isothermal behaviour is assumed during the steady state computation.

An airvessel in WANDA is modelled with a straight instead of a rounded top and bottom. The volume of the real airvessel and the WANDA airvessel should be the same. The top and bottom level should therefore be taken respectively lower and higher than the level of the real airvessel. How much lower and higher depends on the shape of the real airvessel.



Meaning of Pressure in property window Airvessel:

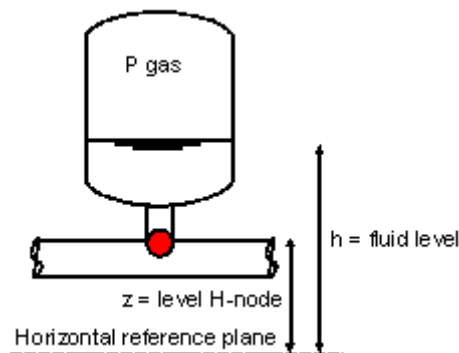
Head = Head in H-node of airvessel

Pressure P (H-node) = $\rho \cdot g \cdot (H-z)$ (relative to atmosphere)

z = height specified by particular H-node

Air pressure (absolute) = $\rho \cdot g \cdot (H-h) + P_{atm}$

h = fluid level



4.5.8.2 Component specific output

Fluid level [m]

Air pressure (absolute) [N/m².abs]

Air volume [m³]

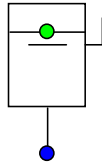
4.5.8.3 H-actions

None

4.5.8.4 Component messages

Message	Type	Explanation
Top level below bottom level	Error	This is obviously an input error.
Initial fluid level not in between top and bottom level of air chamber	Error	This is an input error.
Empty airchamber	Warning	Computation can continue, since storage area is constant independent of the height. The results will not be physically correct, but may assist in the further design of the air vessel.

4.5.9 Airvessel Vertical Hybrid (class)



Fall type

type label	description	active
Airvessel vertical hybrid	Vertically oriented air vessel with an air valve	No

4.5.9.1 Mathematical model

The hybrid air vessel is a combination of a non-vented air vessel and an air valve at a strategic elevation. When the fluid level is above the air valve level, the component is governed by the same equations as the non-vented air vessel (see "AIRVvn (class)" on page 207). However, when the fluid reaches the air valve level, the air is expelled via the air valve. The user specifies the initial fluid level/air volume/constant C in steady state, which means that the air valve level can be reached when the air pressure is still greater than the atmospheric pressure. The capacity of the air valve is modelled by the discharge coefficient and in/outflow area. The hybrid air vessel is especially useful in long distance pipelines, in which the initial stage of the transient is the most critical stage. The initial wave front is much flatter, because of the large air volume. The air valve prevents the air expanding into system in the later stages of the transient scenario.

As long as the fluid is above the air valve level, the governing equations are:

$$PV^k = C \quad (1)$$

in which:

P	=	absolute air pressure	[N/m ²]
V	=	air volume	[m ³]
k	=	Laplace coefficient	[-]
C	=	constant	[Nm] if $k = 1$

The Laplace coefficient depends on the thermodynamic behaviour of the air. Isothermal expansion is described by $k = 1$. Adiabatic expansion is described by $k = 1.4$.

The second equation governs the amount of supplying discharge Q:

$$Q = -A_{av} \frac{dx_f}{dt} \quad (2)$$

in which A_{av} denotes the storage surface and x_f the fluid level in the air chamber with respect to the global coordinate system. The minus sign indicates that the chamber supplies fluid to the system when the fluid level is decreasing in time.

When the fluid level is below the air valve level the change of air volume in time is dependent of two phenomena. First, the compression/expansion of the air, secondly the amount of air leaving/entering the system. This state is modelled as the air valve with a certain residual volume (see "VENT (class)" on page 434). The residual volume equals the volume between the air valve elevation and the top of the air vessel. The air leaving/entering the system is determined by equations (3) to (6).

1. Subsonic air flow in: $0,53 < \frac{P}{P_0} < 1,0$

$$Q_{air} = C \cdot A \cdot \sqrt{7RT_0} \sqrt{\left(\frac{P}{P_0}\right)^{1,4286} - \left(\frac{P}{P_0}\right)^{1,714}} \quad (3)$$

in which:

A	=	air discharge area	[m ²]
C	=	air discharge coefficient	[-]
P	=	abs. internal pressure on fluid level	[Pa]
P_0	=	atmospheric pressure	[Pa]
R	=	gas constant	[J/kg·K]
T_0	=	ambient air temperature	[K]
Q_{air}	=	air flow (positive if into system: supplier!)	[m ³ /s]

2. Critical flow in: $\frac{P}{P_0} < 0,53$

$$Q_{air} = C \cdot A \cdot \sqrt{7RT_0} \cdot 0,259 \quad (4)$$

3. Subsonic air flow out: $1,0 < \frac{P}{P_0} < \frac{1}{0,53}$

$$Q_{air} = C \cdot A \cdot \sqrt{7RT_0} \cdot \left(\frac{P_0}{P}\right)^{\frac{k+1}{2k}} \sqrt{\left(\frac{P_0}{P}\right)^{1,4286} - \left(\frac{P_0}{P}\right)^{1,714}} \quad (5)$$

in which:

k	=	Laplace coefficient (ratio of specific heats)	[-]
-----	---	---	-----

$$4. \text{ Critical flow out: } \frac{P}{P_0} > \frac{1}{0,53}$$

$$Q_{air} = -C \cdot A \cdot \sqrt{7RT_0} \cdot \left(\frac{P_0}{P}\right)^{\frac{k+1}{2k}} \cdot 0,259 \quad (6)$$

For steady state calculations the air vessel does not supply fluid and hence the equation for steady state is simply:

$$Q = 0 \quad (4)$$

4.5.10 Airvessel vertical hybrid properties

4.5.10.1 Hydraulic specifications

description	Input	unit	range	default	remarks
Top level	real	[m]			
Bottom level	real	[m]	(0 100)		
Air inlet level	real	[m]	(0-100)		
Chamber area	real	[-]	(0-100)		
Air quantity by	Fluid Level/ Air volume/ Constant C				See remarks
Initial fluid level	real	[m]			If "air quantity by" = Fluid Level
Initial air volume	real	[m ³]			If "air quantity by" = Air volume
Initial C in P*V = C	real	[J]			If "air quantity by" = Constant C
Laplace coefficient	real	[-]	[1-1.4]		
Ambient air temperature	real	[°C]			
Air discharge coeff.	real	[-]			
Air discharge area	real	[m ²]			

See also help on using the Property Window.

Remarks

"Top level", "Bottom level", "Air inlet level" and "initial fluid level" are related to the horizontal reference plane.

The air volume and the fluid level in the air vessel are determined by the initial fluid level, the initial air volume or the initial constant C. The user specifies either one of these properties. The other two quantities are calculated by WANDA.

If the user specifies the initial fluid level/air volume/constant C such that the air inlet level is reached when the pressure inside the air chamber is higher than the atmospheric pressure, the air will be expelled via the air inlet. This results in a decreasing C (see equation 1).

4.5.10.2 Component specific output

Fluid level	[m]	(relative to datum)
Air volume	[m ³]	-
Air temperature	[°C]	-
Air pressure	[Pa]	-

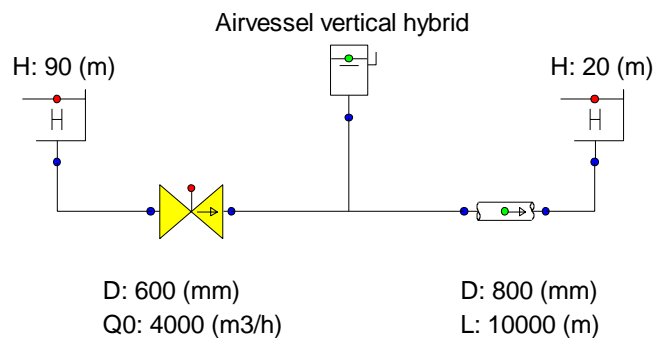
4.5.10.3 H-actions

None

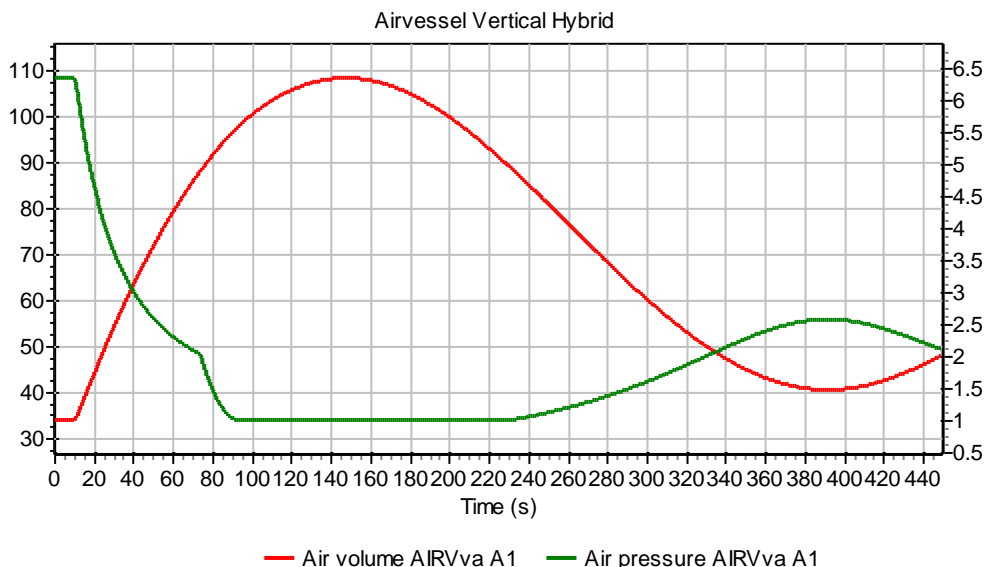
4.5.10.4 Example

The hybrid air vessel in the example below has the following specifications:

Property	Value
Top level	20 m
Bottom level	6 m
Air inlet level	9 m
Chamber area	8 m ²
Air quantity by:	Constant C. $C = 21600$ kJ
Laplace coefficient	1.20
Ambient temperature	15 °C
Air discharge coeff.	0.9
Air discharge area	0.01770 m ²



The valve closes in 3 seconds.



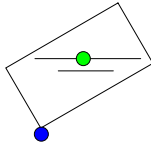
The air vessel dampens the initial pressure wave, such that the air pressure drops from 6.4 bara to 2.0 bara in 65 s. After 75 s of simulation the air valve level is reached and air is being expelled from the vessel to prevent draining of the air vessel. The graph shows that the air pressure reaches atmospheric pressure at $t = 95$ s. The downstream boundary condition is large enough to smoothly shut the air valve after 340 s.

4.5.10.5 Component messages

message	explanation
ERROR: Initial fluid level not in between top and bottom level of air chamber	Input error. Keep in mind that the air vessel is inclined and that the fluid level is still relative to the horizontal reference plane.
ERROR: Initial fluid level inconsistent with steady head	The initial fluid level is below the air inlet level and inconsistent with the pressure head.
WARNING: empty air chamber	The fluid level has dropped below the bottom. The storage area becomes zero and further calculations cannot be carried out.
WARNING: Accuracy not obtained in computing fluid level as function of volume with bisection method	This warning should not normally occur and is intended to warn the programmer.
WARNING: Initial fluid level below air valve level. Air pressure is atmospheric.	
INFO: Air inlet is closed	
INFO: Air inlet is open	
INFO: Air inlet opens	
INFO: Air inlet closes	

4.6 Inclined Air Vessel

4.6.1 Inclined air vessel (class)



Inclined air vessel

Supplier type

type label	description	active
Inclined air vessel	Air vessel with a cylindrical volume oriented at an angle with the horizontal plane	No

4.6.1.1 Mathematical model

An inclined air vessel or air chamber without air inlet can be modelled by two equations.

The first equation describes the behaviour of the enclosed air:

$$PV^k = C \quad (1)$$

in which:

P	=	absolute air pressure	[N/m ²]
V	=	air volume	[m ³]
k	=	Laplace coefficient	[-]
C	=	constant	[Nm] if $k = 1$

The Laplace coefficient depends on the thermodynamic behaviour of the air. Isothermal expansion is described by $k = 1$. Adiabatic expansion is described by $k = 1.4$.

The second equation governs the amount of supplying discharge Q :

$$Q = -dV_f / dt \quad (2)$$

in which V_f denotes the fluid volume. Note that this equation is different from the equation (2) for vertical chambers. This is due to the fact the storage surface of an inclined chamber is not constant (it's a function of the fluid level).

The minus sign indicates that the chamber supplies fluid to the system when the fluid level is decreasing in time.

Equation (2) can also be formulated as:

$$Q = dV / dt \quad (3)$$

where variable V refers to the air volume. In steady state the air vessel does not supply fluid. Hence in steady state the governing equation is:

$$Q = 0 \quad (4)$$

Note: for an inclination angle of 0 degr, the model coincides with the horizontal non-vented airvessel; for an inclination angle of 90 degr, the model coincides with the vertical non-vented airvessel.

4.6.2 Airvessel inclined non-vented

4.6.2.1 Hydraulic specifications

description	Input	unit	range	default	remarks
Bottom level	real	[m]			
Inner diameter	real	[m]	(0 100)		
Inner length	real	[m]	(0-100)		See remarks
Relative head depth	real	[-]	(0-1)		See remarks
Inclination	real	[°]	(0-90)		
Number of vessels	integer	[-]	(1-10)		See remarks
Air quantity by	Fluid Level/ Air volume/ Constant C				See remarks
Initial fluid level	real	[m]			If "air quantity by" = Fluid Level
Initial air volume*	real	[m ³]			If "air quantity by" = Air volume
Initial C in P*V = C	real	[J]			If "air quantity by" = Constant C
Laplace coefficient	real	[-]	[1-1.4]		

See also help on using the Property Window.

* All inclined air vessel input is specified per vessel. Some output is specified for all vessels combined.

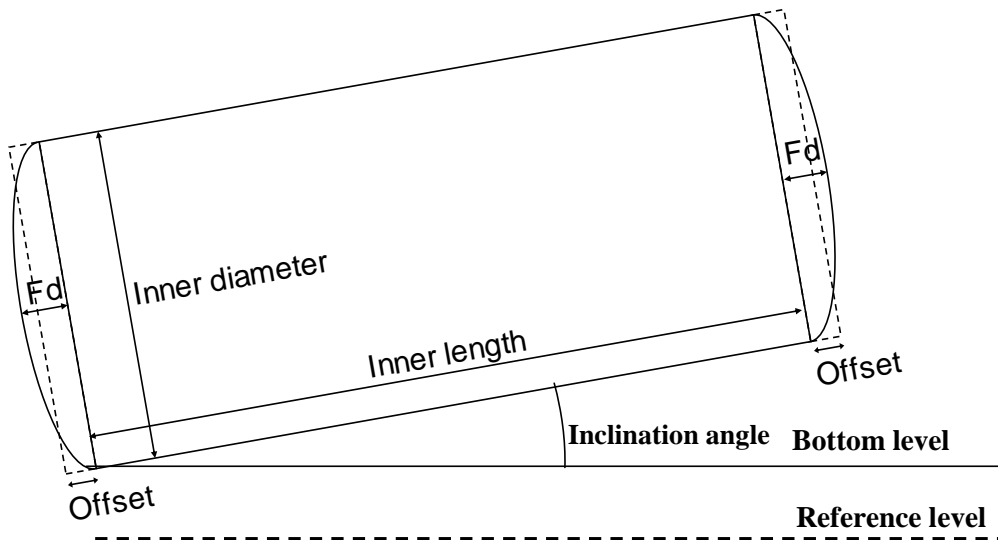
Remarks

"Bottom level" and "initial fluid level" are related to the horizontal reference plane.

The air volume and the fluid level in the air vessel are determined by the initial fluid level, the initial air volume or the initial constant C. The user specifies either one of these properties. The other two quantities are calculated by WANDA. Isothermal behaviour is assumed during the steady state computation.

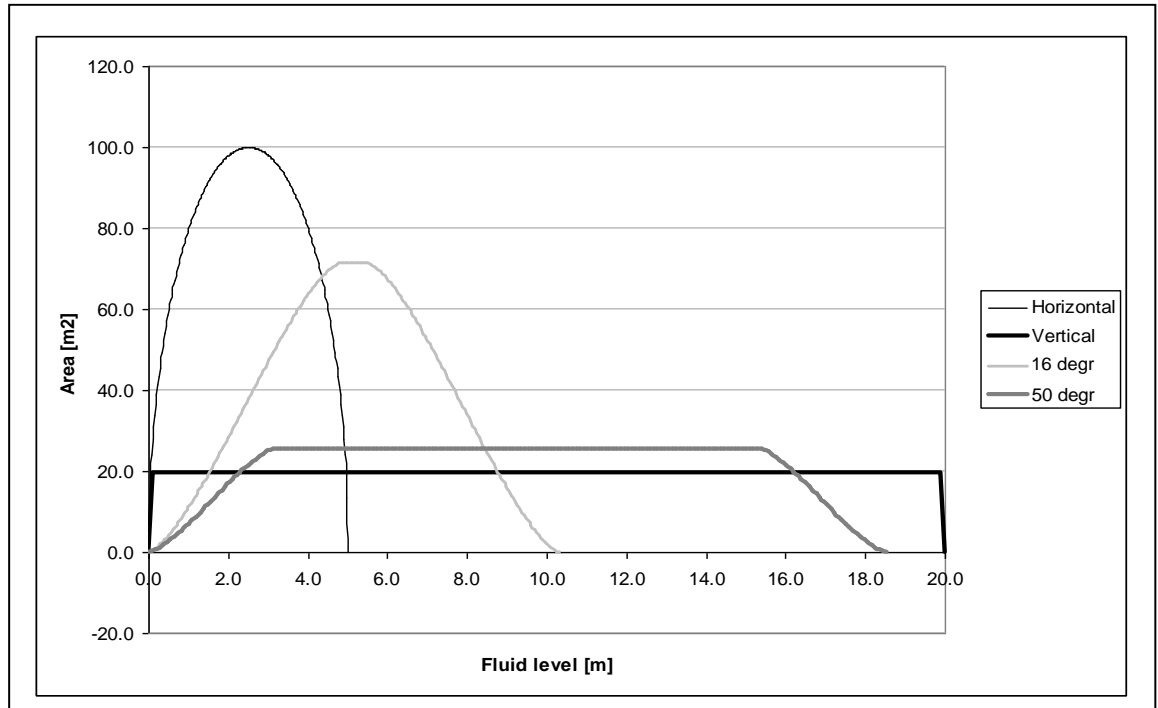
The inclined air vessel is modelled in WANDA as a cylindrical element. The length is specified by the length of the cylindrical part of the vessel combined with two offsets to

account for the volume in the spherical ends of the vessel (See figure below). The offsets are calculated using the ratio between the inner radius (R) and the depth of the ellipsoidal vessel head (F_d). The relative head depth is specified by (F_d/R) . The effective length of the vessel is $L + 2 \times 0.666 \times F_d$. The equivalent length (offset) of a cylinder with the same volume as the vessel head is $0.666 \times F_d$.



It is possible to specify the number of air vessels at this location. The air vessels are treated as one large vessel in WANDA. It may however facilitate the user input. For horizontal vessels this could be achieved by multiplying the length with the number of vessels and for vertical vessels by multiplying the storage area with the number of vessels.

The inclination determines the storage area as a function of the fluid level in the vessel. Several different curves may exist depending on the dimensions of the vessel. Some examples are shown in the graph below ($L = 20$ m, $D = 5$ m). The different curves may result in bends in the air vessel dynamic behaviour.



4.6.2.2 Component specific output

Liquid level	[m]	(relative to datum!)
Air volume	[m ³]	All vessels combined
Storage area	[m ²]	All vessels combined
Air pressure	[Pa]	-

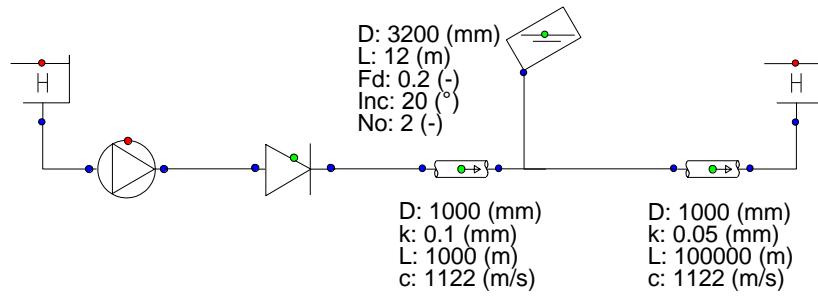
Note: The air volume and storage area output are shown for all vessels together. The air pressure differs from the H-node pressure because of the difference in elevation between the H-node and the liquid level.

4.6.2.3 H-actions

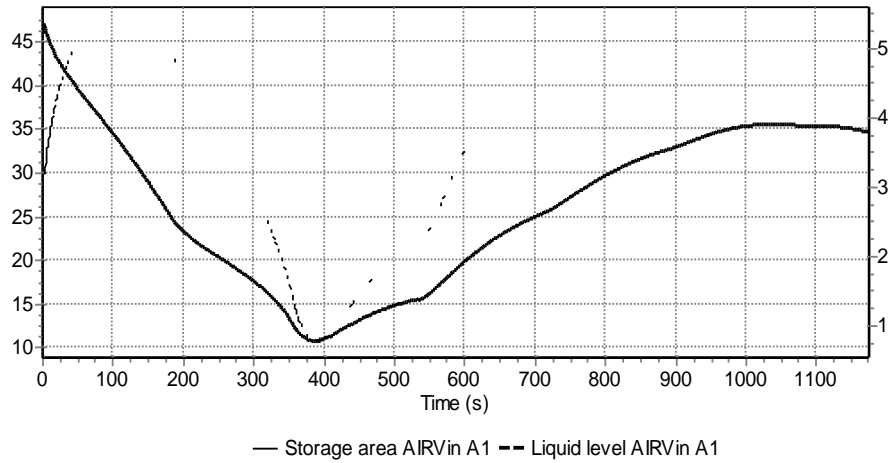
None

4.6.2.4 Example

For the surge protection of a 100 km pipeline 3 inclined air vessels are simulated using the following model. The air vessels are represented in the model by one component. The vessel properties are shown in the figure below. The inclination of the vessel relative to the horizontal plane is 20 degrees. The front to depth ratio is 0.2.



The figure below shows the fluid level and the storage area in the inclined air vessel. The storage initially increases with decreasing fluid level, remains constant for a short period and ultimately decreases at the lowest fluid levels.

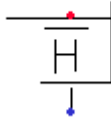


4.6.2.5 Component messages

message	explanation
ERROR: Initial fluid level not in between top and bottom level of air chamber	Input error. Keep in mind that the air vessel is inclined and that the fluid level is still relative to the horizontal reference plane.
ERROR: empty air chamber	The fluid level has dropped below the bottom. The storage area becomes zero and further calculations can not be carried out.
WARNING: Accuracy not obtained in computing fluid level as function of volume with bisection method	This warning should not normally occur and is intended to warn the programmer.

4.7 Boundary Conditions

4.7.1 BOUNDH (class)



Pressure head boundary condition

Supplier type

type label	description	active
BOUNDH (reservoir)	Constant or time dependent head	Yes

4.7.1.1 Mathematical model

A BOUNDH prescribes the head in a certain point of the system.

$$H = f(t) \quad (1)$$

The head specified is the total energy head and, because the velocity in the reservoir is zero, equal to the hydraulic head and equal to the level in the reservoir. The head is constant during the steady state calculation. During the unsteady state calculation the head can vary in time according to a table specified by the user or specified by a Control Component. To achieve the specified head the BOUNDH has to supply or consume fluid to or from the system. The discharge will be determined by the continuity equation for the H-node the BOUNDH is connected to. When a BOUNDH is located between two pipes it actually decouples the system: the pipes are behaving independently of each other and waterhammer waves will reflect fully (reflection factor - 1).

4.7.2 BOUNDH (reservoir)

4.7.2.1 Hydraulic specifications

description	input	unit	range	default	remarks
Head at t = 0	Real	[m]			

See also "Mathematical model" on page 222.

4.7.2.2 Component specific output

None

4.7.2.3 H-actions

In unsteady state the head can vary in time using the action table. An example is given below:

BOUNDH Action table HTIME

Time [s]	Head [m]
0.0	10.0
0.99	10.0
1.0	20.0
1.99	20.0
2.00	0.0
2.99	0.0
3.00	20.0
10.00	20.0

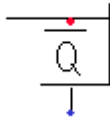
The table must always start at t = 0. The value of the property "head at t = 0" is synchronized automatically by the user interface with the steady state value (value at t = 0) specified in the action table.

In this example the head remains at the steady state value until 0.99 s. At t = 1.0 s the head suddenly rises to 20.0 m and keeps that value until 1.99 s. At t = 2.00 s the head falls stepwise to 0 m etc. Wanda will interpolate the values in the action table to find the correct value for the current timestep.

4.7.2.4 Component messages

None

4.7.3 BOUNDQ (class)



Discharge (or flow) boundary condition

Supplier type

type label	description	active
BOUNDQ (reservoir)	Constant or time dependent discharge	yes

4.7.3.1 Mathematical model

A BOUNDQ prescribes the discharge in a certain point of the system.

$$Q = f(t) \quad (1)$$

Sign convention:

+ into the system

- out of the system.

The component BOUNDQ supplies discharge (called the delivery rate) to the system. For the steady state calculation the discharge is constant. For the unsteady state calculation the discharge can vary in time according to a user-specified table or a Control Component. The component BOUNDQ can be very helpful in modelling a branched system or a reciprocating pump.

4.7.4 BOUNDQ (reservoir)

4.7.4.1 Hydraulic specifications

Description	input	unit	range	default	remarks
Discharge at t = 0	real	[m ³ /s]			

See also "Mathematical model" on page 224.

4.7.4.2 Component specific output

None

4.7.4.3 H-actions

In unsteady state the discharge can vary in time. The time variation must be defined in menu 'Hydraulic actions' via a specified table.

An example is given below:

BOUNDQ Action table

QTIME

Time [s]	Discharge [m ³ /h]
0.0	0.0
1.00	200.0
10.00	20.0

The table must always start at t = 0. The value of the head at t = 0 must match with the steady state value given in the template of component BOUNDQ.

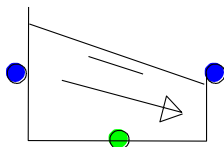
In the example the rate is zero at steady state. It rises linearly in 1 s to 200 m³/h. After 1 s the rate decreases linearly in 9 s to 20 m³/h.

4.7.4.4 Component messages

None

4.8 Channel

4.8.1 Channel (class)



Short open channel with constant slope.

The Channel component is introduced to model short open channel in waste water treatment plants (wwtp). This component is only applicable in situation with a upstream discharge boundary condition. For more complicated flow regimes, use the Free Surface Flow Conduit (PIPE conduit $Q > 0$).

Fall type

type label	description	active
Channel	Short open channel with constant slope	no

Note: In transient mode, this component computes a steady-state profile for each time-step.

4.8.1.1 Mathematical model

The channel model is used for continuous flow cases. It computes the upstream and downstream flow depths and velocities in a channel with constant slope (-8° to $+8^\circ$, or -14% to $+14\%$). Figure 1 presents the definition sketch of the channel model. The flow in the channel can be shooting (supercritical) or tranquil (subcritical). For shooting flow the hydraulic control is upstream (with a depth equal or smaller to the critical depth). For tranquil flow the hydraulic control is downstream (with a depth equal to or greater than the critical depth). The channel does not account for the occurrence of a hydraulic jump (which might occur when the hydraulic control is between the upstream and downstream boundaries and the downstream boundary has a subcritical flow depth).

The geometry of the channel cross-section can be specified by the user using a depth-width table or simply by selecting a rectangular or half-round shape. Furthermore, the channel cannot overflow when the calculated flow depth exceeds the last entered depth-width table entry, since the model uses the last depth-width entry for such situations. Finally the model assumes that the flow depth is at least 1mm.

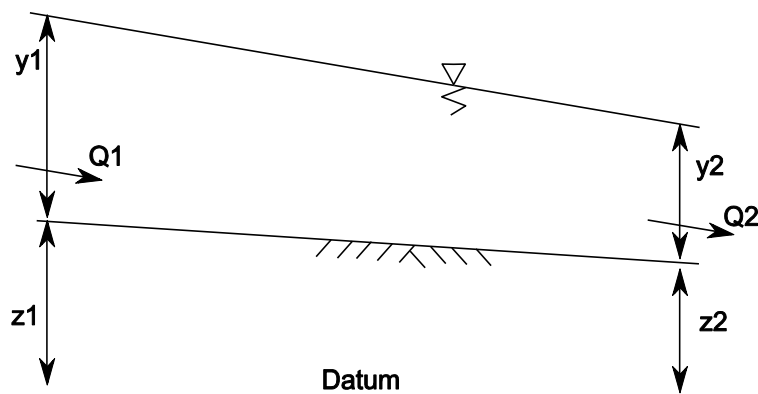


Figure 1: Definition sketch of channel

Flow profile computation

The flow profile (and ultimately the upstream flow depth) is computed from

$$\frac{dy}{dx} = \frac{S_0 - S_f}{1 - \frac{\alpha Q^2}{gA^2 D}} \quad (1)$$

where

S_0 = bed slope [-]

S_f = friction slope [-]

α = energy coefficient [-]

Q = discharge [m^3/s]

g = gravity acceleration [m/s^2]

A = cross-sectional area [m^2]

D = hydraulic depth [m].

Equation (1) is solved using the direct-step method. For tranquil (subcritical) flow the computation starts at a known downstream depth, while for shooting (supercritical) flow the computation starts at a known upstream depth. A default depth is assumed for the following adjacent point and its slope is computed from equation (1). If the slopes of the upstream and downstream points do not differ by more than 1%, then the computation proceeds to the next adjacent point. If the two slopes differ by more than 1% then the default depth of the upstream point is adjusted. Its slope is recomputed and again compared again.

Hydraulic control

In order to use equation (1) a known flow depth is required. From this known depth the computation proceeds upstream or downstream (depending on whether the flow is sub- or super-critical). The known flow depth is determined as follows:

For subcritical flow the flow depth at the downstream boundary is greater or equal to the critical depth, while for supercritical flow the flow depth at the upstream boundary is less or equal to the critical depth.

The critical flow depth y_c is determined from

$$y_c = \frac{Q^2}{A^2(y_c) \cdot g} \quad Fr = \frac{Q}{A \sqrt{g \cdot y_c}} \quad (2)$$

where

y_c = critical flow depth [m]

Q = discharge [m³/s]

g = gravity acceleration [m/s²]

A = cross-sectional area [m²]

Fr = Froud number [-]

Equation (2) is solved iteratively.

Examples of allowed flows:	
	Subcritical flow with free outflow
	Critical flow (with submerged outflow)
	Critical flow (with free outflow)
	Critical flow in a long channel (normal depth is achieved)

Examples of disallowed flows:	
	Transitions from sub-critical to critical flow
	Transitions from critical to sub-critical flow (hydraulic jump)

4.8.2 Channel

4.8.2.1 Hydraulic specifications

Description	Input	Unit	range	default	remarks
Length	Real	[m]			
Wall roughness	Real	[mm]			
Cross section	rectangular / semi-circular / table			rectangular	
Width	Real	[m]			If cross-section is specified as rectangular or semi-circular
Depth-width table	table				If cross-section is specified as table
Hydraulic control	Upstream/ Downstream	[m]			Upstream implies supercritical flow; Downstream implies subcritical flow

See also "Mathematical model" on page 226.

Remarks

The upstream and downstream bed levels and hence channel slope are determined by the heights of the upstream and downstream H-nodes. The slope of the channel must be less than 14% due to the constraints of the mathematical model.

The component upstream of the channel should specify a discharge (a BOUNDQ or WEIR component, for example) and not a head. The discharge through the channel should be greater than zero (i.e. the channel cannot be drained).

If the computation time takes a lot of time to solve the equation, this indicates that a wrong hydraulic control has been chosen.

4.8.2.2 Component specific output

Description	Unit	remarks
Velocity upstream	[m/s]	
Velocity downstream	[m/s]	
Depth upstream	[m]	
Depth downstream	[m]	
Normal depth downstream	[m]	
Critical depth	[m]	
Normal depth	[m]	
Froude upstream	[-]	
Froude downstream	[-]	

4.8.2.3 H-actions

None

4.8.2.4 Component messages

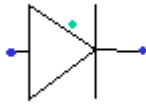
Message	explanation
Warning: Negative discharge not allowed.	Flow is assumed to move in positive direction (from node 1 to node 2)
Error: Discharge may not be zero.	It is assumed that there is always flow in the channel.
Error: Hydraulic head below channel bed level	Hydraulic head cannot be lower than the channel bed level.
Info: Slope horizontal/adverse; normal depth infinite.	Normal depth cannot be computed since the channel slope is horizontal or adverse.
Info: Downstream Froude ≤ 1 : Tranquil flow	Flow is subcritical throughout the channel, except at the downstream end where it is critical or subcritical.
Info: Downstream Froude > 1 : Shooting flow	Flow is supercritical throughout the channel, except at the upstream end where it is critical or supercritical.
Terminating Error: Slope $> 14\%$	Slope exceeds 14%; Check elevations of connected H-nodes.

Literature

- V.T. Chow, Open channel hydraulics, Int. ed., McGraw-Hill, Singapore, 1973.
- H. Chanson, 'The hydraulics of open channel flow: an introduction', Elsevier, Amsterdam, 2004

4.9 Checkvalve

4.9.1 CHECKV (class)



Check valve / Non-return valve

Fall type

type label	description	active
Checkvalve (ideal)	ideal check valve; closes if $Q < 0.0$, opens if $H1 > H2$	No
Checkvalve (deltaP for reopen)	ideal check valve; closes if $Q < 0.0$, opens if $P1 - P2 > \Delta P$	No
CHECKV - undamped	Check valve using the undamped dynamic characteristic (with dimensions)	No
CHECKV - undamped (dimensionless)	Check valve using the undamped dynamic characteristic, dimensionless	No

Within the component **CHECKV** four check valves models are available. The choice of the model will often depend on the information (data) which is available and on the degree of accuracy desired. If no data are available at all, one may be forced to model the check valve as an **ideal** check valve, which closes without reverse flow. However, in practice a check valve does not behave ideally, but closes after flow reversal. The closure is accompanied with a certain amount of reverse flow and pressure surges, depending on the valve type. A better way to model **undamped** as well as **damped** check valves is by means of so-called dynamic characteristics. At this moment WANDA supports only the undamped dynamic characteristic. For several valve types these characteristics (dimensionless only) are available in the check valve property templates.

Alternatively, the closure behaviour of check valves may be modelled with the component **VALVE**. The valve position is then prescribed as function of (closure) time. In the case of damped check valves the valve may be closed in two stages: A stage of passive and a stage of active damping.

4.9.1.1 Mathematical models

w. Ideal check valve

An open check valve is modelled as a resistance by the following equations:

$$H_1 - H_2 = a \xi |Q_1| Q_1 \quad (1)$$

$$Q_1 = Q_2$$

In which:

- H_1 = upstream head [m]
- H_2 = downstream head [m]
- $a = 1/(2gA^2)$ [s²/m⁵]
- ξ = valve loss coefficient [-]
- Q_1 = upstream discharge [m³/s]
- Q_2 = downstream discharge [m³/s]
- A = valve inlet area [m²]

The discharge area of the check valve may differ from the cross sectional area of the adjacent pipes. The area is calculated from the check valve inlet diameter D according to:

$$A = \pi D^2 / 4 \quad (2)$$

Once the check valve is closed, the valve is modelled by:

$$Q_1 = 0 \quad (3)$$

$$Q_1 = Q_2$$

Now the criteria for opening and closure must be defined.

Closure criteria

An ideal check valve closes when:

$$Q_1 < 0 \quad (4)$$

Opening criteria

A closed valve (re)opens when the pressure difference across the valve exceeds a specified value:

$$P_1 - P_2 > \Delta P \quad (5)$$

The default value of $\Delta P = 0$.

2. Undamped check valve (standard)

An open check valve is modelled as a resistance by:

$$\begin{aligned} H_1 - H_2 &= a \xi |Q_1| Q_1 \\ Q_1 &= Q_2 \end{aligned} \quad \bullet \quad (6)$$

In which:

H_1	=	upstream head	[m]
H_2	=	downstream head	[m]
a	=	$1/(2gA^2)$	[s ² /m ⁵]
ξ	=	valve loss coefficient	[-]
Q_1	=	upstream discharge	[m ³ /s]
Q_2	=	downstream discharge	[m ³ /s]
A	=	valve inlet area	[m ²]

Once the check valve is closed, the valve is modelled by:

$$\begin{aligned} Q_1 &= 0 \\ Q_1 &= Q_2 \end{aligned} \quad (7)$$

Now the criteria for opening and closure must be defined.

Closure criteria

The check valve (re)closes when:

$$Q_1 < Q_R = \frac{\pi}{4} D^2 v_R \quad (8)$$

$$v_R = f_1(\theta_i) \cdot f_2\left(\frac{dv}{dt}\right) \quad (9)$$

Where:

$$f_1(\theta_i) = \frac{v_i}{v_o} \quad (10)$$

$$\frac{dv}{dt} = \frac{v - v_i}{t - t_i} \quad (11)$$

If: $v_i \geq v_o$ then $v_i = v_o$ and $t_i = t_o$

If: $v_i < v_o$ then $v_i = v_i$ and $t_i = t_i$

In which:

D	=	valve inlet diameter	[m]
v_R	=	maximum reverse flow velocity	[m/s]
v_o	=	critical velocity (valve just 100% open)	[m/s]
v_i	=	initial (steady) velocity before (re)closure	[m/s]
dv/dt	=	(mean) flow deceleration	[m/s ²]

The function f_1 accounts for the effect of an initially, partly opened valve on the maximum reverse flow velocity. The function f_2 represents the dynamic characteristic for undamped check valves, for the initially, fully opened valve.

For several valve types dynamic characteristics are available in the check valve property templates

Opening criteria

A closed valve (re)opens when the pressure difference across the valve exceeds a specified value:

$$P_1 - P_2 > \Delta P \quad (12)$$

The default value of $\Delta P = 0$.

3. Undamped check valve (dimensionless)

An open check valve is modelled as a resistance by:

$$\begin{aligned} H_1 - H_2 &= a \xi |Q_1| Q_1 \\ Q_1 &= Q_2 \end{aligned} \quad (13)$$

In which:

H_1	=	upstream head	[m]
H_2	=	downstream head	[m]
a	=	$1/(2gA^2)$	[s ² /m ⁵]
ξ	=	valve loss coefficient	[-]
Q_1	=	upstream discharge	[m ³ /s]
Q_2	=	downstream discharge	[m ³ /s]
A	=	valve inlet area	[m ²]

Once the check valve is closed, the valve is modelled by:

$$\begin{aligned} Q_1 &= 0 \\ Q_1 &= Q_2 \end{aligned} \quad (14)$$

Now the criteria for opening and closure must be defined.

Closure criteria

The check valve (re)closes when:

$$Q_1 < Q_R = \frac{\pi}{4} D^2 v_R \quad (15)$$

The maximum reverse flow Q_R (< 0) is determined from:

$$\frac{v_R}{v_o} = f_1 \left(\frac{\theta_i - \theta_c}{\theta_o - \theta_c} \right) \cdot f_2 \left(\frac{D}{v_o^2} \frac{dv}{dt} \right) \quad (16)$$

Where:

$$f_1 \left(\frac{\theta_i - \theta_c}{\theta_o - \theta_c} \right) = \frac{v_i}{v_o} \quad (17)$$

$$\frac{dv}{dt} = \frac{v - v_i}{t - t_i} \quad (18)$$

If: $v_i \geq v_o$ then $v_i = v_o$ and $t_i = t_o$

If: $v_i < v_o$ then $v_i = v_i$ and $t_i = t_i$

In which:

D	=	valve inlet diameter	[m]
v_R	=	maximum reverse flow velocity	[m/s]
v_o	=	critical velocity (valve just 100% open)	[m/s]
v_i	=	initial (steady) velocity before (re)closure	[m/s]
dv/dt	=	(mean) flow deceleration	[m/s ²]

The function f_1 accounts for the effect of an initially, partly opened valve on the maximum reverse flow velocity. The function f_2 represents the dynamic characteristic for undamped check valves, for the initially, fully opened valve.

For several valve types dimensionless dynamic characteristics are available in the property template files.

Opening criteria

A closed valve (re)opens when the pressure difference across the valve exceeds a specified value:

$$P_1 - P_2 > \Delta P \quad (19)$$

The default value of $\Delta P = 0$.

4.9.2 Checkvalve (ideal)

4.9.2.1 Hydraulic specifications

description	input	unit	range	default	remarks
inner diameter	real	[m]	(0-10)		
loss coefficient	real	[-]	[0-10]		
initial state	open/closed			open	

See also “Mathematical models” on page 232.

4.9.2.2 Component specific output

None

4.9.2.3 H-actions

None

4.9.2.4 Component messages

Message	Type	Explanation
Starts in closed phase	Info	
Starts in open phase	Info	
Closes	Info	
Opens	Info	

4.9.3 Checkvalve (deltaP for reopen)

4.9.3.1 Hydraulic specifications

description	input	unit	range	default	remarks
inner diameter	real	[m]	(0-10)		
loss coefficient	real	[-]	[0-10]		
delta_p for reopen	real	[N/m ²]	(0-10 ⁶)		
initial state	open/closed			open	

See also “Mathematical models” on page 232.

4.9.3.2 Component specific output

None

4.9.3.3 H-actions

None

4.9.3.4 Component messages

Message	Type	Explanation
Starts in closed phase	Info	
Starts in open phase	Info	
Closes	Info	
Opens	Info	

4.9.4 CHECKV - undamped

4.9.4.1 Hydraulic specifications

description	input	unit	range	default	remarks
inner diameter	real	[m]	(0-10)		
loss coefficient	real	[-]	[0-10]		
delta_p for reopen	real	[N/m ²]	(0-10 ⁶)		
Dynamic characteristic	table				
Critical velocity	real	[m/s]	(0-10)		
initial state	open/closed			open	

See also “Mathematical models” on page 232.

4.9.4.2 Component specific output

Dv/dt: [m/s²]

Anchor force: [N]

4.9.4.3 H-actions

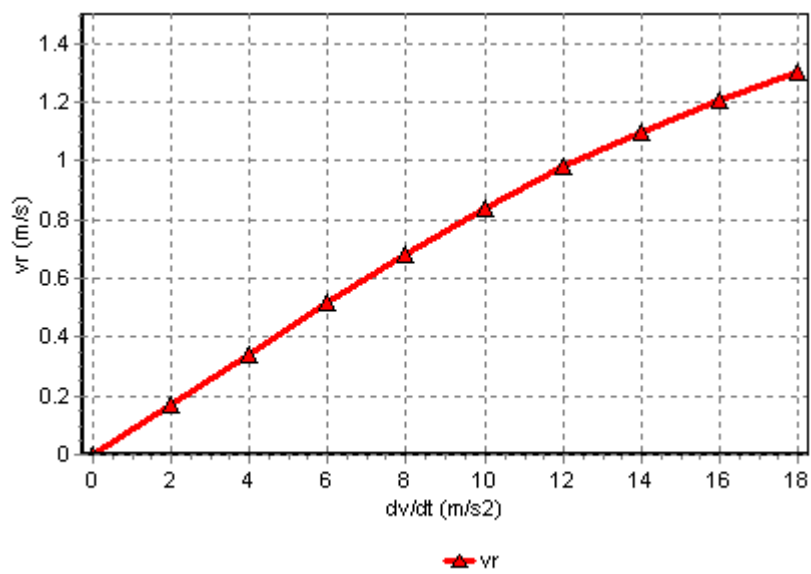
None

4.9.4.4 Component messages

Message	Type	Explanation
starts in closed phase	Info	
starts in open phase	Info	
closes	Info	
opens	Info	

4.9.4.5 Example

Type	Checkvalve (undamped)
Comment: horizontal position; spring loaded	
Model name	Split disc
Inner diameter (mm)	200.0
Loss coefficient (-)	2.400
Delta_p for reopen (kPa)	5.000
Critical velocity V_0 (m/s)	1.750



Consult your manufacturer for the dynamic characteristic of his product.

4.9.5 CHECKV - undamped (dimensionless)

4.9.5.1 Hydraulic specifications

description	input	unit	range	default	remarks
inner diameter	real	[m]	(0-10)		
loss coefficient	real	[-]	[0-10]		
delta_p for reopen	real	[N/m ²]	(0-10 ⁶)		
Dimensionless dynamic characteristic	table				
Critical velocity	real	[m/s]	(0-10)		
initial state	open/closed			open	

See also "Mathematical models" on page 232.

4.9.5.2 Component specific output

Dv/dt: [m/s²]

Anchor force: [N]

4.9.5.3 H-actions

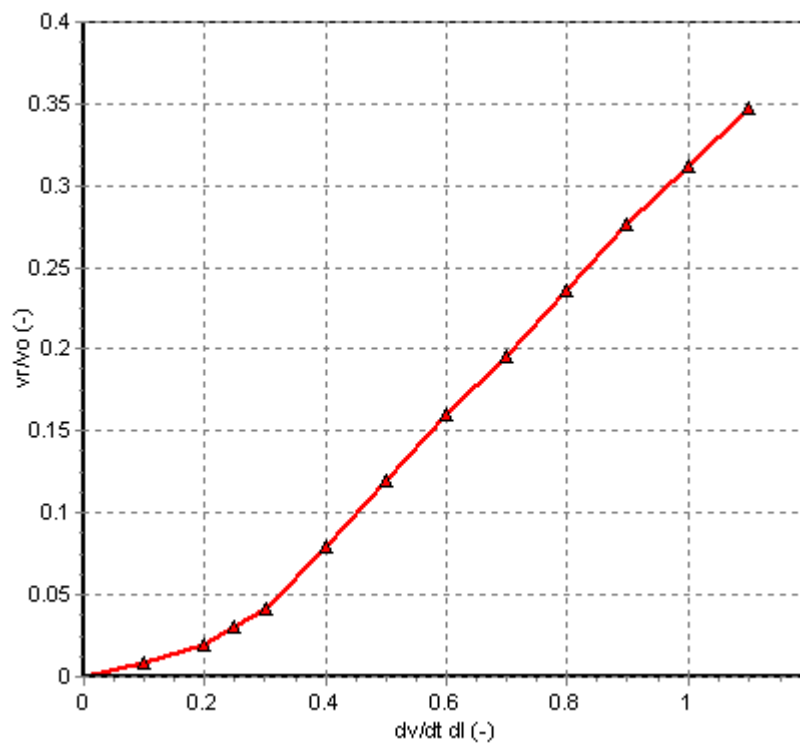
None

4.9.5.4 Component messages

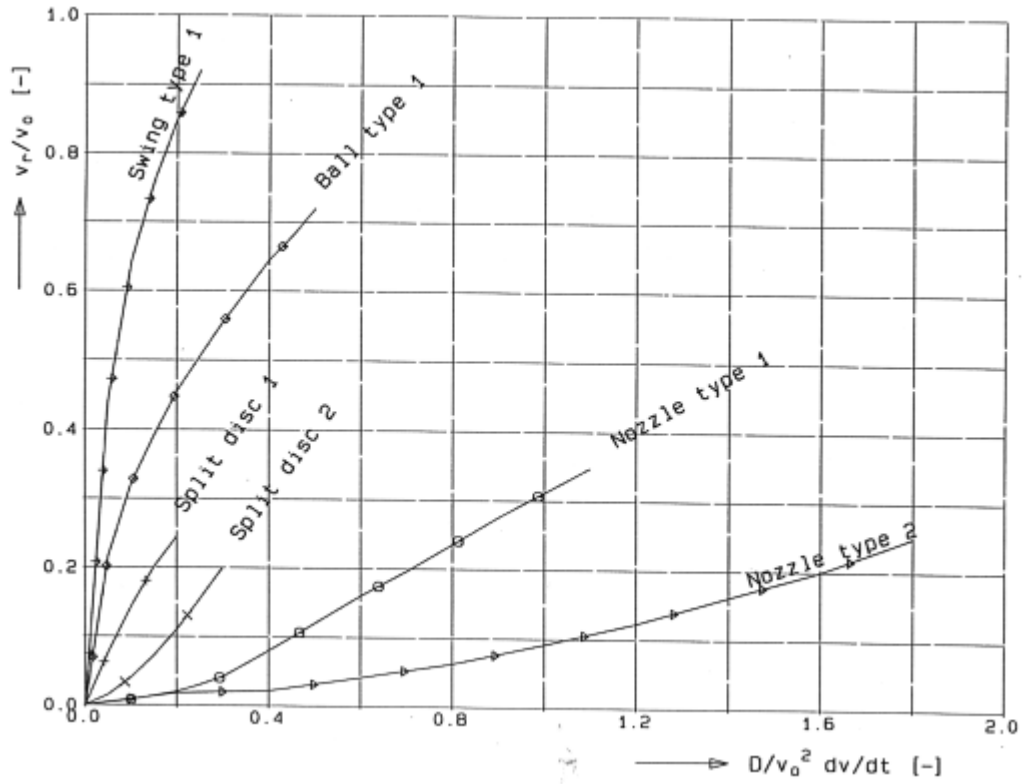
Message	Type	Explanation
starts in closed phase	Info	
starts in open phase	Info	
closes	Info	
opens	Info	

4.9.5.5 Example

Type	Checkvalve (undamped, dimless)
Comment: horizontal position; spring loaded	
Model name	Nozzle type
Inner diameter (mm)	800.0
Loss coefficient (-)	4.500
Delta_p for reopen (kPa)	10.000
Critical velocity V_0 (m/s)	2.50



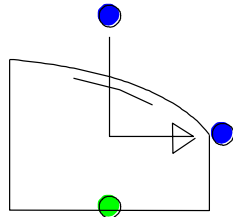
Some typical dimensionless dynamic characteristics are shown in the picture below. These characteristics are for reference only.



Consult your check valve manufacturer for the real dynamic characteristics of his product.

4.10 Collector

4.10.1 Collector (class)



Short open channel with constant slope and lateral inflow

Fall type

type label	description	active
Collector	Short open channel with constant slope and lateral inflow	no

Note: In transient mode, this component computes a steady-state profile for each time-step.

4.10.1.1 Mathematical model

The collector model is used for (increasing) spatially-varied flow problems. It is used to compute the upstream and downstream flow depths and velocities in a channel with constant slope (-8° to $+8^\circ$, or -14% to $+14\%$). Figure 1 presents the definition sketch of the collector model. The collector is closed at the upstream end and the lateral inflow is distributed uniformly along the length of the collector. The hydraulic control from which the upstream flow depth is computed can be inside the collector or at the downstream end. If the control is at the downstream end then it is assumed that the flow is either critical or subcritical. The collector does not account for the occurrence of a hydraulic jump (which might occur when the hydraulic control is between the upstream and downstream boundaries and the downstream boundary has a subcritical flow depth). Furthermore, the flow out of the collector always occurs at the downstream end and never laterally.

The geometry of the collector cross-section can be specified by the user using a depth-width table or simply by selecting a rectangular or half-round shape. Furthermore, the channel cannot overflow when the calculated flow depth exceeds the last entered depth-width table entry, since the model uses the last depth-width entry for such situations. Finally the model does not support dry channel situations (zero flow depth).

The depth profile along the length of the collector is curved for the following reasons:

- The flow velocity and head loss increases due to the lateral inflow.
- The lateral inflow introduces large local losses near the upstream end of the collector.
- The lateral inflow has no impulse in the streamwise direction.

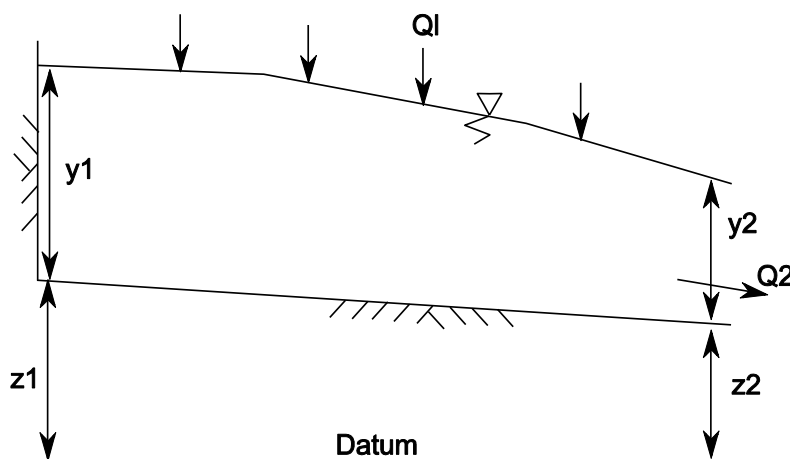


Figure 1: Definition sketch of collector

Flow profile computation

The flow profile (and ultimately the upstream flow depth) is computed from

$$\frac{dy}{dx} = \frac{S_0 - S_f - \frac{2\alpha Qq^*}{gA^2}}{1 - \frac{\alpha Q^2}{gA^2 D}} \quad (1)$$

where

S_0 = bed slope [-]

S_f = friction slope [-]

α = energy coefficient [-]

Q = discharge [m³/s]

q^* = lateral discharge per unit length [m³/s]

g = gravity acceleration [m/s²]

A = cross-sectional area [m²]

D = hydraulic depth [m].

equation (1) is solved using the direct-step method. The computation starts at a known downstream depth. A default depth is assumed for the following upstream point and its slope is computed from equation (1). If the slopes of the upstream and downstream points do not differ by more than 1%, then the computation proceeds to the next upstream point. If the two slopes differ by more than 1% then the default depth of the upstream point is adjusted. Its slope is recomputed and again compared to the slope of the downstream point.

Hydraulic control

In order to use equation (1) a known downstream flow depth is required. The computation proceeds upstream from this known depth, assuming subcritical flow. The known flow depth is determined as follows:

The algorithm first checks to see whether a hydraulic control occurs between the upstream and downstream boundaries of the collector. The location and flow depth of this hydraulic control is determined from a singular point analysis [1,2]. If there is such a hydraulic control then. If no hydraulic control occurred between the upstream and downstream boundaries, then the computation starts at the downstream boundary. At the downstream boundary the flow depth is the maximum of: the critical depth (free overfall), the uniform flow depth (if specified by the user) or the head of the downstream hydraulic node.

The critical flow depth y_c is determined from

$$y_c = \frac{Q_2^2}{A_2^2(y_c) \cdot g} \quad Fr = \frac{Q_2}{A_2 \sqrt{g \cdot y_c}} \quad (2)$$

The uniform flow depth y_n is determined from

$$Q = A(y_n) \cdot C \sqrt{R_h(y_n) \cdot S_b} \quad (3)$$

where

y_c = critical flow depth [m]

y_n = uniform flow depth [m]

Q = discharge [m³/s]

g = gravity acceleration [m/s²]

A = cross-sectional area at the downstream end conditions [m²]

Fr = Froud number [-]

C = Chezy friction factor [m^{0.5}/s]

R_h =hydraulic radius [m]

S_b = bed slope [-]

Equations (2) and (3) are solved iteratively. The Chezy friction factor is computed from:

$$C = \sqrt{\frac{8g}{f}} \quad (4)$$

where

f = Colebrook-White friction factor.

This second friction factor is computed from:

$$\frac{1}{\sqrt{f}} = -2 \log \left(\frac{k/D_h}{3.7} + \frac{2.51}{Re \sqrt{f}} \right) \quad (5)$$

with

$$Re = \frac{v \cdot D_h}{\nu} = \frac{v \cdot 4R_h}{\nu} \quad (6)$$

where

k =absolute roughness of the channel [m],

D_h = hydraulic diameter [m]

Re =Reynolds number [-]

v = flow velocity [m/s]

ν =kinematic viscosity [m²/s]

They are computed at the collector midpoint (L/2).

4.10.2 Collector

4.10.2.1 Hydraulic specifications

Description	Input	Unit	range	default	remarks
Length	Real	[m]	(0 ; 1000)		
Wall roughness	Real	[mm]	(0 ; 1000)		
Cross section	rectangular / semi-circular / table			rectangular	
Width	Real	[m]	(0 ; 10)		If cross-section is specified as rectangular or semi-circular
Depth-width table	table				If cross-section is specified as table
Upstream bottom offset	Real	[m]	[-140 ; 140]		w.r.t. downstream H-node
Downstream condition	Uniform/ Free overfall				

See also "Mathematical model" on page 245.

Remarks

The upstream and downstream bed levels are equal to the sum of their respective bottom offsets and the downstream H-node geometric height. The slope of the collector bed must be less than 14 % due to constraints of the mathematical model.

The component upstream of the collector should specify a discharge and not a head (a BOUNDQ or WEIR component, for example). The discharge through the collector must be greater than zero (i.e. the collector cannot be drained).

As with the CHANNEL component, the collector does not allow for transitions between sub- and supercritical flow. The component will issue a warning if it detects a hydraulic jump or a transition from subcritical to supercritical flow.

4.10.2.2 Component specific output

Description	Unit	remarks
Velocity downstream	[m/s]	
Depth upstream	[m]	
Depth downstream	[m]	
Normal depth downstream	[m]	
Critical depth downstream	[m]	
Froude number downstream	[-]	

4.10.2.3 H-actions

None

4.10.2.4 Component messages

Message

Info: Slope horizontal/adverse; normal depth infinite

Error: Lateral inflow may not be zero.

Info: Critical flow between upstream and downstream boundaries

Info: Downstream Froude ≤ 1 : Tranquil flow

Info: Downstream Froude > 1 : Shooting flow

Terminating Error: Slope $> 14\%$

explanation

Bottom offset values, result in a slope which is either horizontal or adverse

No discharge is being supplied by component upstream of collector

Hydraulic control occurs between upstream and downstream boundaries

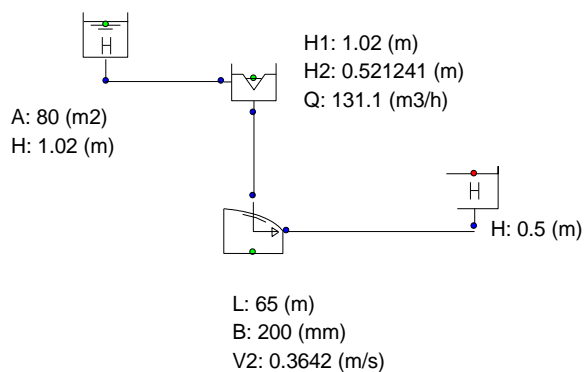
Flow at downstream end is subcritical. The subcritical flow depth at the downstream boundary is used for the profile computation

Flow at downstream end is critical. The critical flow depth at the downstream boundary is used for the profile computation

Slope exceeds 14%. Check elevations of the connected H-nodes.

4.10.2.5 Example

The following example demonstrates how to model a settling tank for sewage treatment works.



The V-notch weir discharges into the collector. This represents the settling tank overflowing into the surrounding channel.

Literature

- V.T. Chow, Open channel hydraulics, Int. ed., McGraw-Hill, Singapore, 1973.
- H. Chanson, 'The hydraulics of open channel flow: an introduction', Elsevier, Amsterdam, 2004

4.11 Conduit



Fall type

type label	description	active
Free surface flow conduit	Pipe model for free surface (atmospherically) flow and pressurised flow without waterhammer calculation	No

The free surface conduit is capable of simulating the behaviour of free surface phenomena. The application of this component is useful for simulation of slow filling and draining procedures.

The free surface conduit is capable of handling circular cross sections or arbitrary cross sections defined by a table.

4.11.1 Free surface flow conduit

4.11.1.1 Mathematical model

The dynamic behaviour of a free surface pipe is described by the continuity and momentum equation.

The mathematical model is described in chapter "Free surface flow conduit" on page 184.

A summary of the relevant aspects for the user with respect to the input are repeated in this section.

It is assumed that the gas (air) above the free surface in this pipe or channel can enter or leave the pipe freely without causing pressure surges.

The applicability of the continuity and momentum equations for free-surface flow (Saint-Venant equations) is limited to slopes of 1:7 (8 degrees or 14%).

In addition to the maximum slope, the slope change in two consecutive free surface elements is limited. The maximum slope change should be less than 0.14. Formally:

The element length of free surface pipes is calculated from the time step. However in Engineering mode the time step is not present. Therefore the user should specify explicitly a maximum element length, which is used in Engineering mode. It is recommended to set this property to $200 \cdot D$ (diameter) or less. In Transient mode the element length is calculated from the time step and it is checked whether the calculated element length is less than the maximum element length. It is recommended to set the maximum element length to $200 \cdot D$ or less, in order to obtain sufficient accuracy.

4.11.1.2 Geometry

Each CONDUIT must have a length input and height location. This so-called profile can be defined in several ways:

1. Scalar value for length
2. Length-height profile
3. Isometric layout specified with absolute XYZ co-ordinates
4. Isometric layout specified with differential XYX co-ordinates.

Each type of profile input is translated to a longitudinal profile (X-H profile). In case of a scalar value the height of the beginning and the end of the conduit is derived directly from the adjacent H-nodes. The input height is the height of the bottom (!) of the conduit, as indicated in figure below. Note that the pressurised pipe uses the centre line for elevation reference.

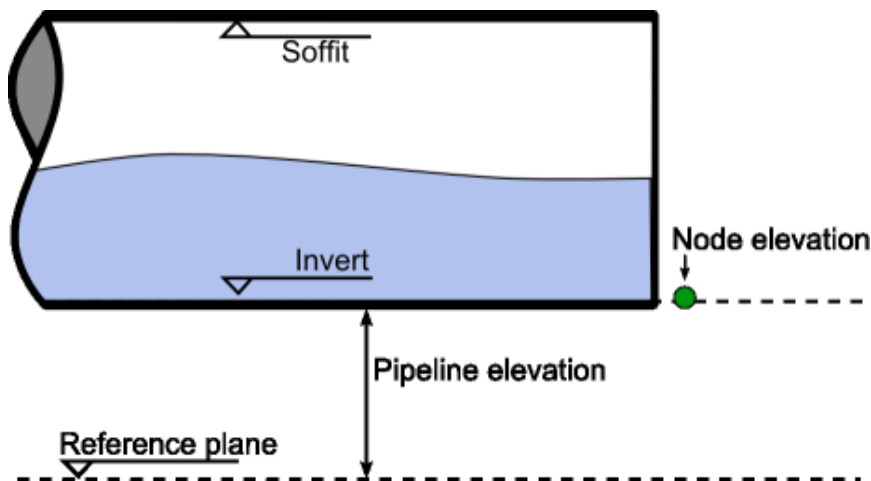


Figure 4.2: Definition of the height of the Conduit and the node elevation.

The CONDUIT use the same geometry profile type as used in the PIPE, For details see the PIPE component.

4.11.1.3 Hydraulic specifications

description	Input	unit	default	remarks
Cross section	Table Circle	[-]	Circle	
Height-width table				if cross section=Table
Inner diameter	real	[m]		if Cross section=Circle
Initial flow condition	Qinit > 0 Qinit = 0		Qinit>0	if Cross section=Rectangle
Drained	Complete Offtakes		Complete	if Initial flow condition = Qinit=0 See additional remark
Offtake locations	Table			If Drained =offtakes See additional remark
Friction model	D-W k D-W f	[-]	D-W-f	Darcy Weisbach friction model with wall roughness or friction factor input
Wall roughness	real	[m]		If friction model = D-W k

description	Input	unit	default	remarks
Friction factor	real	[-]		If friction model = D-W f
Dynamic friction	none Quasi-steady		Quasi-steady	Only If friction model = D-W k
Max. element length	real	[m]		
Geometry input	Length l-h xyz xyz diff	[-]		
Length	real	[m]		if Geometry input = Length
Profile	table			if Geometry input = L-h, xyz or xyz diff

Remarks Initial flow condition / drained

Contrary to the “initial flow condition: $Q_{init} > 0$ ” the “initial flow condition: $Q_{init} = 0$ ” ipe is intended to compute steady states with zero flow and an arbitrary filling status. With zero flow there may exist pools of liquid around the deep points in the pipe profile. In how far the pipe will be drained is specified by the user input item “drained”. This property can either be “complete” (pipe will be completely empty) or “offtakes” in which case offtake locations where the pipe will be drained are specified. The offtake locations are specified by means of a 3-column table in which the first two columns correspond to the profile table of the pipe (lines may be skipped, however). Offtake locations will be active when there is a positive non-zero entry in the third column at the station required. They are inactive when there is either a zero or an empty field in the third column. The offtake level will be the bottom elevation of the pipe (i.e. profile elevation minus pipe height: diameter or highest table entry).

For instance a 4 m diameter conduit with profile:

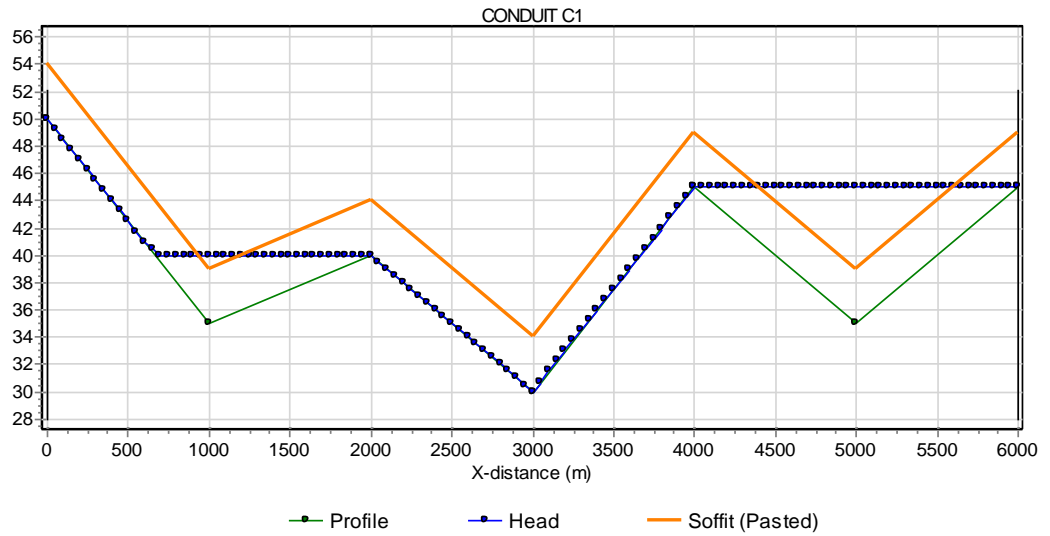
X	Z
0	50
1000	35
2000	40
3000	30
4000	45
5000	35
6000	45

and offtake table:

X	Z	offtake
0	50	0
3000	30	1

6000 | 45 | 0

will have the following solution:



Note: the soffit line is not standard output but pasted from a spreadsheet (elevation soffit = profile points + diameter)

4.11.1.4 Component specific output

Output	Description
Froude [-]	Froude number
Depth [m]	fluid depth related to bottom
Level [m]	fluid level related to horizontal reference plane

4.11.1.5 H-actions

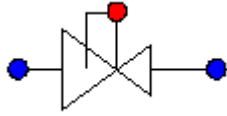
None

4.11.1.6 Example

None

4.12 Control valve

4.12.1 FCV



Flow control valve

Fall type

type label	description	active
Flow control valve	Flow control valve (FCV).	yes

4.12.1.1 Mathematical model

This valve controls the flow rate by adjusting the valve position. The basic pressure loss equation for a Newtonian fluid and without cavitation is:

$$\Delta H = \xi(\theta) \frac{v^2}{2g} \quad \text{in which} \quad H = z + \frac{p}{\rho g}$$

in which:

ΔH	=	pressure head difference across the valve	[m]
ξ	=	valve position dependent loss coefficient	[-]
θ	=	valve position	[-]
g	=	gravitational acceleration	[m/s ²]
v	=	velocity	[m/s]
z	=	elevation	[m]
p	=	pressure	[N/m ²]
ρ	=	density of the fluid	[kg/m ³]

Note that z_1 (upstream) and z_2 (downstream) are not necessarily the same, so the sign of the pressure difference may be even opposite to the head difference with the same flow.

Instead of the loss characteristic, the discharge coefficients K_v or C_v may also be used:

$$K_v = \frac{Q}{\sqrt{\Delta p}}$$

with:

K_v	=	discharge coefficient	[m ³ /h/ $\sqrt{\text{bar}}$]
Q	=	flow	[m ³ /h]
Δp	=	net pressure difference across the valve	[bar]

In words: the discharge coefficient K_v denotes the water flow rate in m³/h through a valve at a pressure difference of 1 bar.

Apart from K_v , C_v is also defined as a discharge coefficient for American units.

$$C_v = \frac{Q}{\sqrt{\Delta p}}$$

with

C_v	=	discharge coefficient	[USGPM/ $\sqrt{\text{psi}}$]
Q	=	flow	[US gallons/min]
Δp	=	Net pressure difference across the valve	[psi]

The relation between ξ , K_v and C_v is as follows:

$$\xi = 1.6 \cdot 10^9 \frac{1}{K_v^2} \cdot D^4$$

$$K_v = 0.865 C_v$$

A valve is characterised by $\xi = f(\xi)$ or $K_v = f(\xi)$ or $C_v = f(\xi)$.

ξ denotes the dimensionless valve opening. ξ ranges from 0 to 1 (in SI-units, or 0 to 100% in percentage annotation).

$\xi = 0$, valve is closed (0% open)

$\xi = 1$, valve is completely open (100% open).

The different discharge characteristics are always translated to the following equation:

$$\Delta H = a \xi |Q_1|$$

in which:

ΔH	=	$H_1 - H_2$ in positive flow direction	[m]
H_1	=	upstream head	[m]
H_2	=	downstream head	[m]
a	=	$1 / (2 g A r^2)$	[s ² /m ⁵]
ξ	=	loss coefficient	[-]

Q_1	=	discharge upstream	[m ³ /s]
A_f	=	discharge area valve	[m ²]

The discharge characteristic may be defined by one of Deltares' standard characteristics (See Hydraulic specifications) or by a user-defined discharge characteristic. If the valve position does not coincide with a tabulated position, interpolation must be performed to obtain the discharge coefficient for intermediate valve positions. The standard characteristics and the user-defined ξ characteristic are interpolated logarithmically according to the following equation:

$$\theta(z) = z \cdot \theta_1 + (1 - z) \cdot \theta_2$$

$$\xi(\theta(z)) = \xi_1^z \cdot \xi_2^{1-z}$$

The user-defined K_v and C_v characteristics are interpolated such that K_v or C_v values are interpolated linearly:

$$\frac{1}{\sqrt{\xi(\theta(z))}} = \frac{z}{\sqrt{\xi_1}} + \frac{1-z}{\sqrt{\xi_2}}$$

in which:

θ_1, θ_2	=	tabulated valve positions	[-]
ξ	=	fraction, which defines intermediate valve position ($0 < z < 1$)	[-]
$\theta(\xi)$	=	intermediate valve position	[-]
ξ_1, ξ_2	=	loss coefficient at θ_1, θ_2	[-]
$\xi(\theta(z))$	=	interpolated loss coefficient	[-]

If the valve is fully closed the governing equation is:

$$Q_1 = 0$$

Controlling actions

This valve controls the flow rate by adjusting the valve position. The valve position is adjusted with a speed proportional to the error (measured value minus set value: error = $Q_1 - Q_{set}$). The speed is truncated to the full stroke closing speed if the absolute value of the error is greater than the proportionality range.

$$\dot{\theta}_{\text{valve}} = \frac{Q_1 - Q_{\text{set}}}{Q_{\text{cutoff}} T_c} \quad \wedge \quad \text{abs}(\dot{\theta}_{\text{valve}}) \leq \frac{1}{T_c}$$

in which:

- $\dot{\theta}_{\text{valve}}$ = (angular) velocity of valve [s⁻¹]
- Q1 = discharge [m³/s]
- Qset = set discharge [m³/s]
- Qcutoff = proportionality range [m³/s]
- Tc = full stroke closing time [s]



For the initial situation either the valve position or the flow rate can be prescribed. In case the initial position is prescribed, the computed flow rate will generally differ from the set value. The set value will then be attained during the unsteady state computation. In case the initial flow rate is chosen to be the set value, the valve position will be determined on basis of the hydraulic solution. This is the preferred method if a balanced start of the unsteady state computation is required.

The maximum and minimum valve position may be different from 1 (100 % open) and 0 (fully closed) respectively by defining the valve characteristic table only between the desired lower and the upper limits. For example if the characteristic is specified as:


position θ	loss coef. ξ
0.3	5600
0.4	1600
0.5	900
0.6	500
0.7	300


The valve position will remain within 0.3 and 0.7. The initial position must also be within these limits or an error message will result. This is not applicable for the standard valve characteristics.

The controlling action is defined in the table below. The normal action is with a positive head difference and flow (hence: $H_1 > H_2$). This is shown in the second column of the table. For a negative head difference the valve will go to the minimum valve position (normally fully closed). since the (always positive) set value can not be reached. This is shown in the third column.

Q ctrl (FCV)	$H_1 > H_2$	$H_1 < H_2$
$Q < Q_{\text{set}}$		\downarrow
$Q > Q_{\text{set}}$		\downarrow

In which:

 : open further until set value reached;

 : close further until set value reached;

⊥ : go to the minimum valve position (normally fully closed) at maximum speed;

Cavitation

Cavitation depends on the pressure conditions around the valve. Usually these pressure conditions are defined by a pressure relation. Several different definitions are use in the industrial standards.

In WANDA the factor X_f is used, according the German VDMA standard.

$$X_f = \frac{\Delta p}{p_1 - p_v}$$

in which:

X_f	=	pressure ratio	[-]
Δp	=	net pressure difference across the valve	[N/m ²]
p_1	=	absolute pressure upstream of the valve	[N/m ²]
p_v	=	vapour pressure of the fluid	[N/m ²]

The pressure ratio depends of the valve opening: $X_f = f(\theta)$

The pressure ratio X_f is only calculated for positive flow; for negative flow $X_f = 0$.

If the cavitation characteristic is specified, the program calculates the pressure ratio in the system and warns the user if it exceeds the available value as defined in the characteristic.

Note:

In other standards (ISA, BS, IEC) the pressure ratio X_T is used:

$$X_T = \frac{p_1 - p_2}{p_1} = \frac{\Delta p}{p_1}$$

In some standards (e.g. ISA) the Thoma number σ is used. $\sigma = \frac{p_1 - p_v}{\Delta p}$

in which:

σ	=	pressure ratio	[-]
Δp	=	net pressure difference across the valve	[N/m ²]

P_1	=	absolute pressure upstream of the valve	[N/m ²]
p_v	=	vapour pressure of the fluid	[N/m ²]

The relationship between X_f and σ is: $\sigma = \frac{1}{X_f}$

Another definition for the Thoma number is based on the downstream pressure p_2 :

$$\sigma_2 = \frac{p_2 - p_v}{\Delta p}$$

where $\sigma = 1 + \sigma_2$ and $\sigma_2 = \frac{1}{X_f} - 1$

Remarks

None

4.12.1.2 Hydraulic specifications

Description	input	Unit	range	default	remarks
Characteristic type	Standard/ Kv/ Cv/ Xi				
Standard type	Buttrfly/ Ball/ Gate/ Gate_sqr			Buttrfly	if char.type = Standard
Kv characteristic	Table				if char.type = Kv
Cv characteristic	Table				if char.type = Cv
Xi characteristic	Table				if char.type = Xi
Inner diameter	Real	[m]	(0-1000]		
Initial setpoint	Real	[m ³ /s]	(0-10]		does not have to be the same as the initial flow
Initial setting	Position / Discharge				
Initial position (open)	Real	[-]	[0-1]		0 = closed 1 = open If init_setting = Position

Initial discharge	Real	[m ³ /s]	(0, 10]	If init_setting = Discharge
Full stroke closing time	Real	[s]	(0-10 ⁷)	
Proportionality range	Real	[m ³ /s]		Related to error see mathematical model
Check cavitation	Yes/No			
Cavitation table	Table			If check cavitation = Yes

4.12.1.3 Component specific output

Valve position [-]

Pressure ratio X_f system [-]

4.12.1.4 H-actions

The setpoint of the valve can be changed over time by means of the action table.

Example action table:

Time [s]	Discharge [m ³ /s]
0	1.0
10	1.0
20	2.0

Note:

The setpoint may be also changed by a control signal line; the control signal line overrules the action table.

4.12.1.5 Component messages

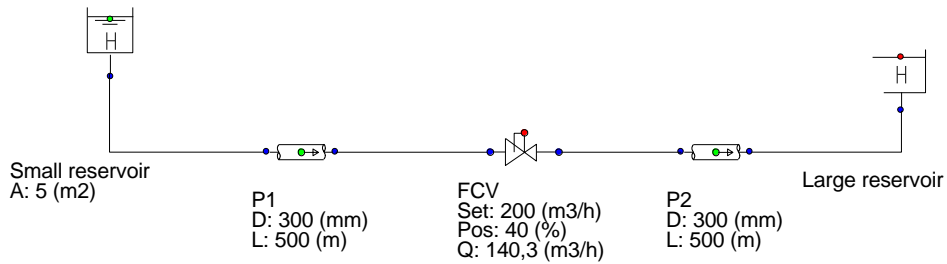
Message	Type	Explanation
Valve resistance in max. open position too large to obtain prescribed state or Valve resistance in min. open position too small to obtain prescribed state	Info	The specified discharge cannot be obtained. The required loss coefficient is outside of the bounds of the valve characteristic. Either increase valve table or adjust settings.
Initial setting not physically realistic Discharge opposite to pressure drop	Info	The specified discharge cannot be obtained. Change initial discharge value or change setting to position
Inconsistent valve position between	Info	If difference is less than 1% value of action

steady state results and action table		table will be used otherwise user had to change input
Start controlling	Info	The valve (re)opens and starts balancing
Closes	Info	Valve fully closed; position is 0
Reaching maximum opening	Info	Valve runs out of control; capacity in maximum open position to small too obtain required upstream pressure
Reaching minimum opening	Info	Valve runs out of control; capacity in minimum open position to large too obtain required upstream pressure

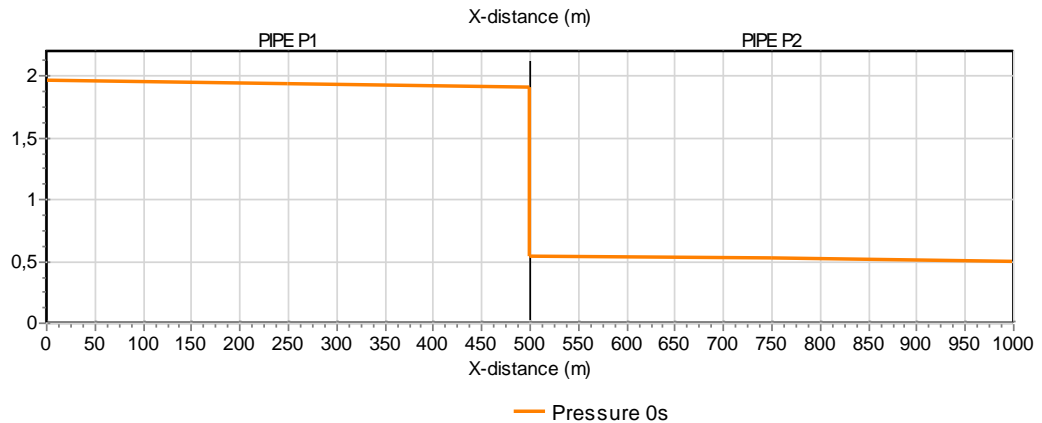
4.12.1.6 Example

A flow control valve is located in the middle of a 1000 m long pipeline.

The initial position of the valve is set to 40 % open. The steady state discharge is 140 m³/h bar. The required discharge is 200 m³/h.



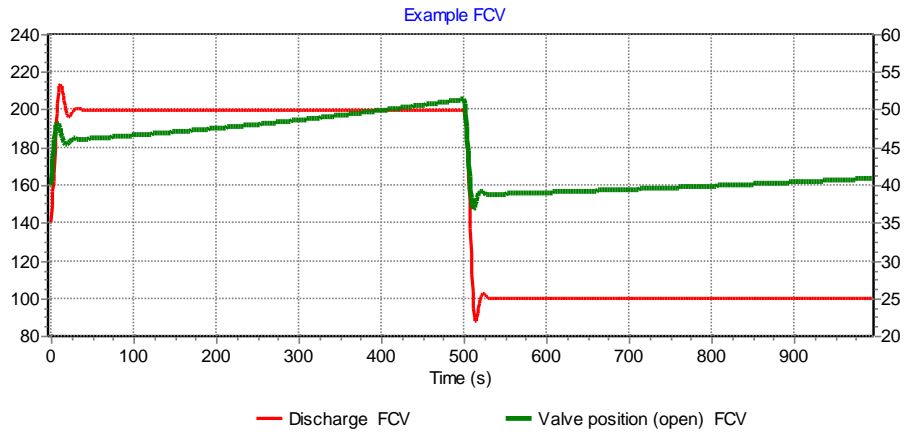
Example FCV



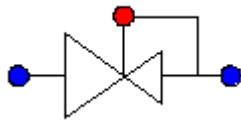
In the unsteady state the FCV starts controlling. Due to the decreasing level in the small reservoir the PdCV adjusts the position continuously to maintain the set discharge (200 m³/h).

At t= 500 sec, the setpoint has been changed to 100 m³/h(via the action table).

Next figure shows the results.



4.12.2 PdCV



Downstream pressure control valve

Fall type

type label	description	active
Downstream pressure control valve	Also known as pressure reducing valve (PRV).	yes

4.12.2.1 Mathematical model

This valve controls the pressure downstream by adjusting the valve position. The basic pressure loss equation for a Newtonian fluid and without cavitation is:

$$\Delta H = \xi(\theta) \frac{v^2}{2g} \quad \text{in which} \quad H = z + \frac{P}{\rho g}$$

in which:

ΔH = pressure head difference across the valve [m]

ξ	=	valve position dependent loss coefficient	[-]
θ	=	valve position	[-]
g	=	gravitational acceleration	[m/s ²]
v	=	velocity	[m/s]
z	=	elevation	[m]
p	=	pressure	[N/m ²]
ρ	=	density of the fluid	[kg/m ³]

Note that z_1 (upstream) and z_2 (downstream) are not necessarily the same so the sign of the pressure difference may be even opposite to the head difference with the same flow.

Instead of the loss characteristic, the discharge coefficients K_v or C_v may also be used:

$$K_v = \frac{Q}{\sqrt{\Delta p}}$$

with:

K_v	=	discharge coefficient	[m ³ /h/ $\sqrt{\text{bar}}$]
Q	=	flow	[m ³ /h]
Δp	=	net pressure difference across the valve	[bar]

In words: the discharge coefficient K_v denotes the water flow rate in m³/h through a valve at a pressure difference of 1 bar.

Besides K_v , C_v is also defined as a discharge coefficient for American units.

$$C_v = \frac{Q}{\sqrt{\Delta p}}$$

with

C_v	=	discharge coefficient	[USGPM/ $\sqrt{\text{psi}}$]
Q	=	flow	[US gallons/min]
Δp	=	Net pressure difference across the valve	[psi]

The relation between ξ , K_v and C_v is as follows:

$$\xi = 1.6 \cdot 10^9 \frac{1}{K_v^2} \cdot D^4$$

$$K_v = 0.865 C_v$$

A valve is characterised by $\xi = f(\theta)$ or $K_v = f(\theta)$ or $C_v = f(\theta)$.

θ denotes the dimensionless valve opening. θ ranges from 0 to 1 (in SI-units, or 0 to 100% in percentage annotation).

- $\theta = 0$, valve is closed (0% open)
- $\theta = 1$, valve is completely open (100% open)

The different discharge characteristics are always translated to the following equation:

$$\Delta H = a \xi Q_1 |Q_1|$$

in which:

ΔH	=	$H_1 - H_2$ in positive flow direction	[m]
H_1	=	upstream head	[m]
H_2	=	downstream head	[m]
a	=	$1 / (2 g A_f^2)$	[s ² /m ⁵]
ξ	=	loss coefficient	[-]
Q_1	=	discharge upstream	[m ³ /s]
A_f	=	discharge area valve	[m ²]

The discharge characteristic may be defined by one of Deltares' standard characteristics (See Hydraulic specifications) or by a user-defined discharge characteristic. If the valve position does not coincide with a tabulated position, interpolation must be performed to obtain the discharge coefficient for intermediate valve positions. The standard characteristics and the user-defined ξ characteristic are interpolated logarithmically according to the following equation:

$$\theta(z) = z \cdot \theta_1 + (1 - z) \cdot \theta_2$$

$$\xi(\theta(z)) = \xi_1^z \cdot \xi_2^{1-z}$$

The user-defined K_v and C_v characteristics are interpolated such that K_v or C_v values are interpolated linearly:

$$\frac{1}{\sqrt{\xi(\theta(z))}} = \frac{z}{\sqrt{\xi_1}} + \frac{1-z}{\sqrt{\xi_2}}$$

in which:

θ_1, θ_2	=	tabulated valve positions	[-]
ξ	=	fraction, which defines intermediate valve position ($0 < z < 1$)	[-]
$\theta(\xi)$	=	intermediate valve position	[-]
ξ_1, ξ_2	=	loss coefficient at θ_1, θ_2	[-]
$\xi(\theta(z))$	=	interpolated loss coefficient	[-]

If the valve is fully closed the governing equation is:

$$Q_1 = 0$$

Controlling actions

This valve controls the pressure downstream by adjusting the valve position. The valve position is adjusted with a speed proportional to the error (measured value minus set value: error = $p_2 - p_{set}$). The speed is truncated to the full stroke closing speed if the absolute value of the error is greater than the proportionality range.

$$\dot{\theta}_{\text{valve}} = \frac{p_2 - p_{\text{set}}}{p_{\text{cutoff}} T_c} \quad \wedge \quad \text{abs}(\dot{\theta}_{\text{valve}}) \leq \frac{1}{T_c}$$

in which:

$\dot{\theta}_{\text{valve}}$	=	(angular) velocity of valve	[s-1]
p2	=	downstream pressure	[N/m2]
pset	=	set pressure	[N/m2]
pcutoff	=	proportionality range	[N/m2]
Tc	=	full stroke closing time	[s]

For the initial situation either the valve position or the set value can be chosen. In case the initial position is prescribed, the computed pressure will generally differ from the set value. The set value will then be attained during the unsteady state computation. In case the initial setting is chosen to be the set value, the valve position will be determined on basis of the hydraulic solution. This is the preferred method if a balanced start of the unsteady state computation is required.

The maximum and minimum valve position may be different from 1 and 0 respectively by defining the valve characteristic table only between the desired lower and the upper limits. For example if the characteristic is specified as:

position θ	loss coef. ξ
0.3	5600
0.4	1600
0.5	900
0.6	500
0.7	300

The valve position will remain between 0.3 and 0.7. The initial position must also be within this range or an error message will result. This is not applicable for the standard valve characteristics.

The controlling action is defined in the table below. The normal action is with a positive head difference and flow (hence: $H_1 > H_2$). This is shown in the second column of the table. H_{set} is the set pressure head at the downstream node. In the third column the action is shown if the set value can not be reached but the error is minimised. This will lead to fully opening.

For a negative head difference (that means negative flow) the valve will go to the minimum valve position (normally fully closed). This is shown in the fourth column

p2 ctrl (PRV)	H1 > H2		H1 < H2
	$H_1 > H_{set}$	$H_1 < H_{set}$	
$p_2 < p_{set}$			↓
$p_2 > p_{set}$			↓

In which:

: open further until set value reached;

: open fully, set value will not be reached;

: close further until set value reached;

: go to the minimum valve position (normally fully closed) at maximum speed;

: not applicable

The valve will close if the valve position becomes zero.

The valve will start controlling if there is a positive pressure head difference ($H_1 > H_2$) and the upstream pressure is above the setpressure ($p_1 > p_{set}$).

Cavitation

Cavitation depends on the pressure conditions around the valve. Usually these pressure conditions are defined by a pressure relation. Several different definitions are use in the industrial standards.

In WANDA the factor X_f is used, according the German VDMA standard.

$$X_f = \frac{\Delta p}{p_1 - p_v}$$

in which:

X_f	=	pressure ratio	[-]
Δp	=	net pressure difference across the valve	[N/m ²]
p_1	=	absolute pressure upstream of the valve	[N/m ²]
p_v	=	vapour pressure of the fluid	[N/m ²]

The pressure ratio depends of the valve opening: $X_f = f(\theta)$

The pressure ratio X_f is only calculated for positive flow; for negative flow $X_f = 0$.

If the cavitation characteristic is specified, the program calculates the pressure ratio in the system and warns the user if it exceeds the available value as defined in the characteristic.

Note:

In other standards (ISA, BS, IEC) the pressure ratio X_T is used:

$$X_T = \frac{P_1 - P_2}{P_1} = \frac{\Delta p}{P_1}$$

In some standards (e.g. ISA) the Thoma number σ is used. $\sigma = \frac{P_1 - P_v}{\Delta p}$

in which:

σ	=	pressure ratio	[-]
Δp	=	net pressure difference across the valve	[N/m ²]
P_1	=	absolute pressure upstream of the valve	[N/m ²]
p_v	=	vapour pressure of the fluid	[N/m ²]

The relationship between X_f and σ is: $\sigma = \frac{1}{X_f}$

Another definition for the Thoma number is based on the downstream pressure p_2 :

$$\sigma_2 = \frac{P_2 - P_v}{\Delta p}$$

where $\sigma = 1 + \sigma_2$ and $\sigma_2 = \frac{1}{X_f} - 1$

4.12.2.2 Hydraulic specifications

Description	input	Unit	range	default	remarks
Characteristic type	Standard/ Kv/ Cv/ Xi				
Standard type	Buttrfly/ Ball/ Gate/ Gate_sqr			Buttrfly	if char.type = Standard
Kv characteristic	Table				if char.type = Kv
Cv characteristic	Table				if char.type

				= Cv
Xi characteristic	Table			if char.type = Xi
Inner diameter	Real	[m]	(0-1000)	
Initial setpoint	Real	[N/m ²]	[-10 ⁵ – 10 ⁷]	does not have to be the same as the initial pressure
Initial setting	Position / P_downstr			
Initial position (open)	Real	[-]	[0-1]	0 = closed 1 = open If init_setting = Position
Initial downstream pressure	Real	[N/m ²]	[-10 ⁵ – 10 ⁷]	If init_set = P_downstr
Full stroke closing time	Real	[s]	(0-10 ⁷)	
Proportionality range	Real	[N/m ²]		Related to error see mathematical model
Check cavitation	Yes/No			
Cavitation table	Table			If check cavitation = Yes

4.12.2.3 Component specific output

Valve position [-]

Pressure ratio X_f system [-]

4.12.2.4 H-actions

The setpoint of the valve can be changed over time by means of the action table.

Example action table:

Time [s]	Pressure [N/m ²]
0	7·10 ⁵
10	7·10 ⁵
20	5·10 ⁵

Note:

The setpoint may be also changed by a control signal line; the control signal line overrules the action table.

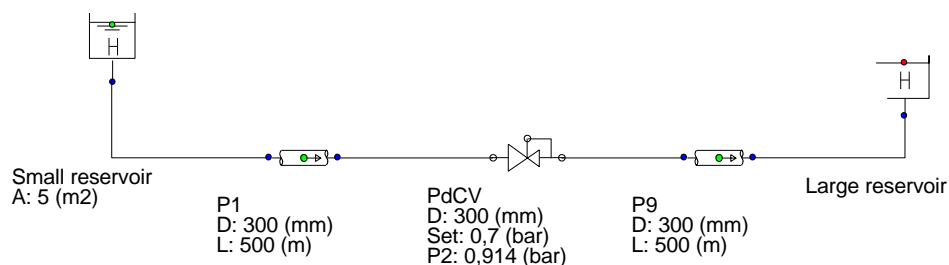
4.12.2.5 Component messages

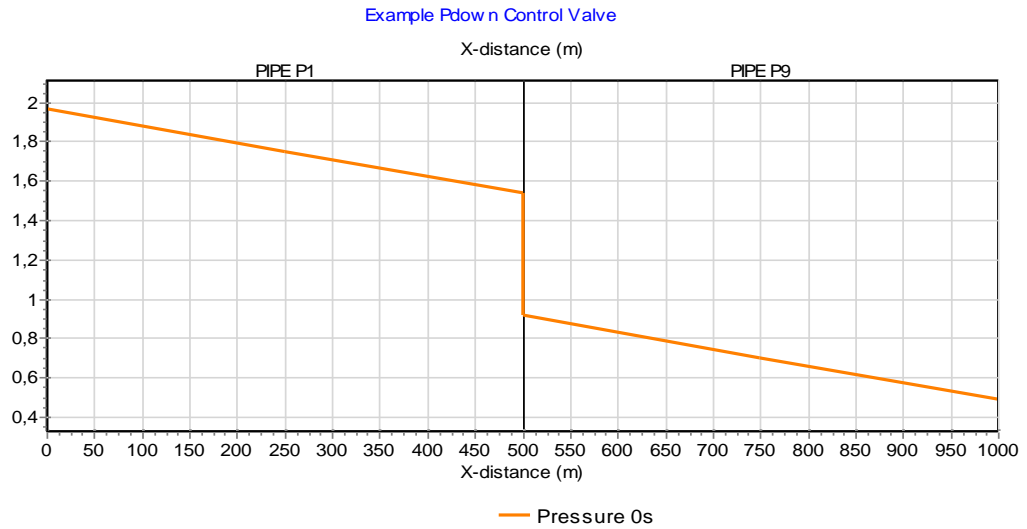
Message	Type	Explanation
Valve resistance in max. open position too large to obtain prescribed state or Valve resistance in min. open position too small to obtain prescribed state	Info	The specified downstream pressure cannot be obtained. The required loss coefficient is outside of the bounds of the valve characteristic. Either increase valve table or adjust settings.
Initial setting not physically realistic Discharge opposite to pressure drop	Info	The specified downstream pressure cannot be obtained. Change initial downstream pressure value or change setting to position
Inconsistent valve position between steady state results and action table	Info	If difference is less than 1% value of action table will be used otherwise user had to change input
Start controlling	Info	The valve (re)opens and starts balancing
Closes	Info	Valve fully closed; position is 0
Reaching maximum opening	Info	Valve runs out of control; capacity in maximum open position to small too obtain required upstream pressure
Reaching minimum opening	Info	Valve runs out of control; capacity in minimum open position to large too obtain required upstream pressure

4.12.2.6 Example

A pressure control valve is located in the middle of a 1000 m long pipeline.

The initial position of the valve is set to 80 % open. The steady state downstream pressure is 0.914 bar. The required pressure is 0.7 bar.

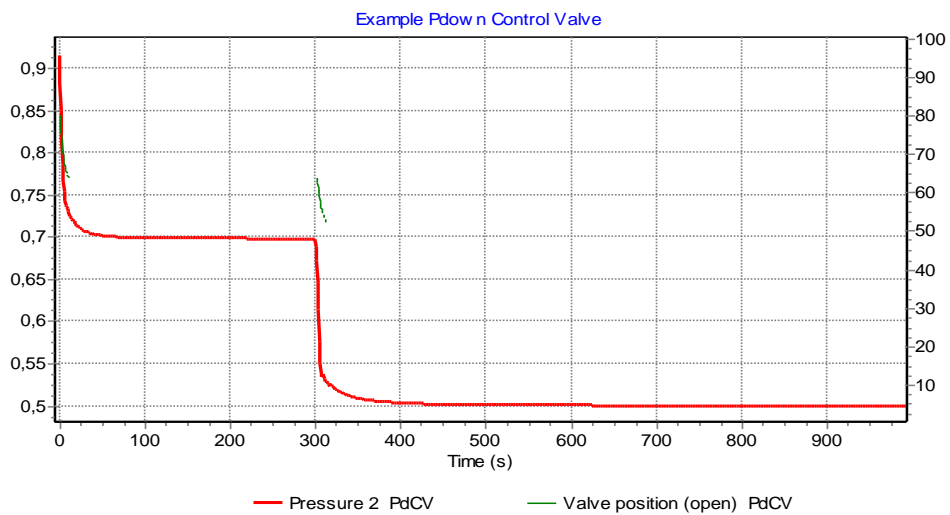




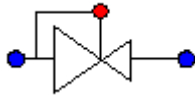
In the unsteady state the PdCV starts controlling. Due to the decreasing pressure in the small reservoir the PdCV adjusts the position continuously to maintain the set pressure (0.7 bar).

At T= 300 sec, the setpressure has been changed to 0.5 bar (via the action table).

Next figure shows the results.



4.12.3 PuCV



Upstream pressure control valve

Fall type

type label	description	active
Upstream pressure control valve	Also known as pressure sustaining valve (PSV).	yes

4.12.3.1 Mathematical model

This valve controls the pressure upstream by adjusting the valve position. The basic pressure loss equation for a Newtonian fluid and without cavitation is:

$$\Delta H = \xi(\theta) \frac{v^2}{2g} \quad \text{in which} \quad H = z + \frac{p}{\rho g}$$

in which:

ΔH	=	pressure head difference across the valve	[m]
ξ	=	valve position dependent loss coefficient	[-]
θ	=	valve position	[-]
g	=	gravitational acceleration	[m/s ²]
v	=	velocity	[m/s]
z	=	elevation	[m]
p	=	pressure	[N/m ²]
ρ	=	density of the fluid	[kg/m ³]

Note that z_1 (upstream) and z_2 (downstream) are not necessarily the same so the sign of the pressure difference may be even opposite to the head difference with the same flow.

Instead of the loss characteristic, the discharge coefficients K_v or C_v may also be used:

$$K_v = \frac{Q}{\sqrt{\Delta p}}$$

with:

K_v	=	discharge coefficient	[m ³ /h/√bar]
Q	=	flow	[m ³ /h]
Δp	=	net pressure difference across the valve	[bar]

In words: the discharge coefficient K_v denotes the water flow rate in m³/h through a valve at a pressure difference of 1 bar.

Besides K_v , C_v is also defined as a discharge coefficient for American units.

$$C_v = \frac{Q}{\sqrt{\Delta p}}$$

with

C_v	=	discharge coefficient	[USGPM/√psi]
Q	=	flow	[US gallons/min]
Δp	=	Net pressure difference across the valve	[psi]

The relation between ξ , K_v and C_v is as follows:

$$\xi = 1.6 \cdot 10^9 \frac{1}{K_v^2} \cdot D^4$$

$$K_v = 0.865 C_v$$

A valve is characterised by $\xi = f(\theta)$ or $K_v = f(\theta)$ or $C_v = f(\theta)$.

θ denotes the dimensionless valve opening. θ ranges from 0 to 1 (in SI-units, or 0 to 100% in percentage annotation).

= 0, valve is closed (0% open)

= 1, valve is completely open (100% open)

The different discharge characteristics are always translated to the following equation:

$$\Delta H = a \xi Q_1 | Q_1 |$$

in which:

ΔH	=	H1 - H2 in positive flow direction	[m]
$H1$	=	upstream head	[m]
$H2$	=	downstream head	[m]
a	=	$1 / (2 g A f^2)$	[s ² /m ⁵]

ξ	=	loss coefficient	[-]
Q1	=	discharge upstream	[m3/s]
Af	=	discharge area valve	[m2]

The discharge characteristic may be defined by one of Deltares' standard characteristics (See Hydraulic specifications) or by a user-defined discharge characteristic. If the valve position does not coincide with a tabulated position, interpolation must be performed to obtain the discharge coefficient for intermediate valve positions. The standard characteristics and the user-defined ξ characteristic are interpolated logarithmically according to the following equation:

$$\theta(z) = z \cdot \theta_1 + (1 - z) \cdot \theta_2$$

$$\xi(\theta(z)) = \xi_1^z \cdot \xi_2^{1-z}$$

The user-defined K_v and C_v characteristics are interpolated such that K_v or C_v values are interpolated linearly:

$$\frac{1}{\sqrt{\xi(\theta(z))}} = \frac{z}{\sqrt{\xi_1}} + \frac{1-z}{\sqrt{\xi_2}}$$

in which:

θ_1, θ_2	=	tabulated valve positions	[-]
ξ	=	fraction, which defines intermediate valve position ($0 < z < 1$)	[-]
$\theta(\xi)$	=	intermediate valve position	[-]
ξ_1, ξ_2	=	loss coefficient at θ_1, θ_2	[-]
$\xi(\theta(z))$	=	interpolated loss coefficient	[-]

If the valve is fully closed the governing equation is:

$$Q_1 = 0$$

Controlling actions

This valve controls the pressure upstream by adjusting the valve position. The valve position is adjusted with a speed proportional to the error (measured value minus set value: error = $p_1 - p_{set}$). The speed is truncated to the full stroke closing speed if the absolute value of the error is greater than the proportionality range.

$$\dot{\theta}_{valve} = \frac{p_1 - p_{set}}{p_{range} T_c} \quad \wedge \quad \text{abs}(\dot{\theta}_{valve}) \leq \frac{1}{T_c}$$

in which:

$\dot{\theta}_{\text{valve}}$	=	(angular) velocity of valve	[s ⁻¹]
p_1	=	upstream pressure	[N/m ²]
p_{set}	=	set pressure	[N/m ²]
p_{range}	=	proportionality range	[N/m ²]
T_c	=	full stroke closing time	[s]

For the initial situation either the valve position or the set value can be chosen. In case the initial position is prescribed, the computed pressure will generally differ from the set value. The set value will then be attained during the unsteady state computation. In case the initial setting is chosen to be the set value, the valve position will be determined on basis of the hydraulic solution. This is the preferred method if a balanced start of the unsteady state computation is required.

The maximum and minimum valve position may be different from 1 (100 % open) and 0 (fully closed) respectively by defining the valve characteristic table only between the desired lower and the upper limits. For example if the characteristic is specified as:

position θ	loss coef. ξ
0.3	5600
0.4	1600
0.5	900
0.6	500
0.7	300

The valve position will remain between 0.3 and 0.7. The initial position must also be within this range or an error message will result. This is not applicable for the standard valve characteristics.

The controlling action in open phase is defined in the table below. The normal action is with a positive head difference and flow (hence: $H_1 > H_2$). This is shown in the second column of the table. H_{set} is the set pressure head at the upstream node. In the third column the action is shown if the set value can not be reached but the error is minimised. This will lead to fully opening.

For a negative head difference (that means negative flow) the valve will go to the minimum valve position (normally fully closed). This is shown in the fourth column

p1 ctrl (PSV)	H1 > H2		H1 < H2
	H2 < Hset	H2 > Hset	
p1 > pset			⌞
p1 < pset			⌞

In which:

: open further until set value reached;

: open fully, set value will not be reached;

: close further until set value reached;

⌞ : go to the minimum valve position (normally fully closed) at maximum speed;

: not applicable

The valve will close if the valve position becomes zero.

The valve will start controlling if there is a positive pressure head difference ($H_1 > H_2$) and the upstream pressure is above the setpressure ($p_1 > p_{set}$).

Cavitation

Cavitation depends on the pressure conditions around the valve. Usually these pressure conditions are defined by a pressure relation. Several different definitions are use in the industrial standards.

In WANDA the factor X_f is used, according the German VDMA standard.

$$X_f = \frac{\Delta p}{p_1 - p_v}$$

in which:

- X_f = pressure ratio [-]
- Δp = net pressure difference across the valve [N/m²]
- p_1 = absolute pressure upstream of the valve [N/m²]
- p_v = vapour pressure of the fluid [N/m²]

The pressure ratio depends of the valve opening: $X_f = f(\theta)$

The pressure ratio X_f is only calculated for positive flow; for negative flow $X_f = 0$.

If the cavitation characteristic is specified, the program calculates the pressure ratio in the system and warns the user if it exceeds the available value as defined in the characteristic.

Note:

In other standards (ISA, BS, IEC) the pressure ratio X_T is used:

$$X_T = \frac{p_1 - p_2}{p_1} = \frac{\Delta p}{p_1}$$

In some standards (e.g. ISA) the Thoma number σ is used. $\sigma = \frac{p_1 - p_v}{\Delta p}$

in which:

σ	=	pressure ratio	[-]
Δp	=	net pressure difference across the valve	[N/m ²]
P_1	=	absolute pressure upstream of the valve	[N/m ²]
p_v	=	vapour pressure of the fluid	[N/m ²]

The relationship between X_f and σ is: $\sigma = \frac{1}{X_f}$

Another definition for the Thoma number is based on the downstream pressure p_2 :

$$\sigma_2 = \frac{p_2 - p_v}{\Delta p}$$

where $\sigma = 1 + \sigma_2$ and $\sigma_2 = \frac{1}{X_f} - 1$

4.12.3.2 Hydraulic specifications

Description	input	Unit	range	default	remarks
Characteristic type	Standard/ Kv/ Cv/ Xi				
Standard type	Buttrfly/ Ball/ Gate/ Gate_sqr			Buttrfly	if char.type = Standard See Valve for standard char.
Kv characteristic	Table				if char.type = Kv

Cv characteristic	Table			if char.type = Cv
Xi characteristic	Table			if char.type = Xi
Inner diameter	Real	[m]	(0-1000]	
Initial setpoint	Real	[N/m ²]	[-10 ⁵ – 10 ⁷]	does not have to be the same as the initial pressure
Initial setting	Position / P_upstream			
Initial position (open)	Real	[-]	[0-1]	0 = closed 1 = open If init_setting = Position
Initial upstream pressure	Real	[N/m ²]	[-10 ⁵ – 10 ⁷]	If init_setting = P_upstream
Full stroke closing time	Real	[s]	(0-10 ⁷]	
Proportionality range	Real	[N/m ²]		Related to error see mathematical model
Check cavitation	Yes/No			
Cavitation table	Table			If check cavitation = Yes

4.12.3.3 Component specific output

Valve position [-]

Pressure ratio X_f system [-]

4.12.3.4 H-actions

The setpoint of the valve can be changed over time by means of the action table.

Example action table:

Time [s]	Pressure [N/m ²]
0	$7 \cdot 10^5$
10	$7 \cdot 10^5$
20	$5 \cdot 10^5$

Note:

The setpoint may be also changed by a control signal line; the control signal line overrules the action table.

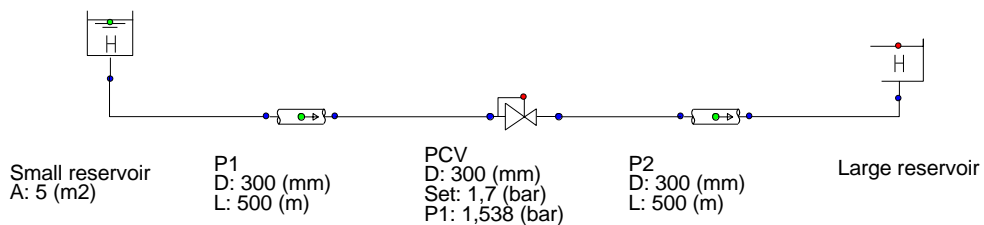
4.12.3.5 Component messages

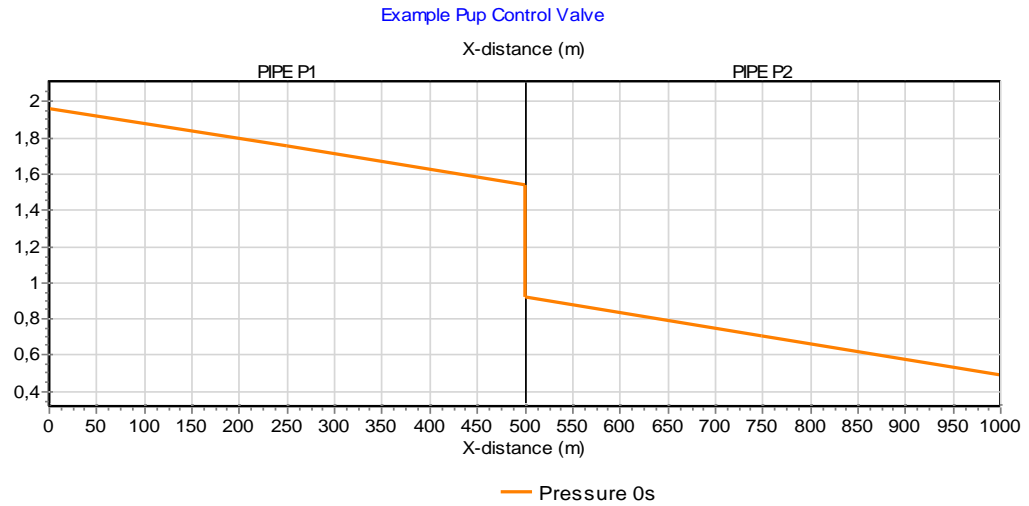
Message	Type	Explanation
Valve resistance in max. open position too large to obtain prescribed state or Valve resistance in min. open position too small to obtain prescribed state	Info	The specified upstream pressure cannot be obtained. The required loss coefficient is outside of the bounds of the valve characteristic. Either increase valve table or adjust settings.
Initial setting not physically realistic Discharge opposite to pressure drop	Info	The specified upstream pressure cannot be obtained. Change initial upstream pressure value or change setting to position
Inconsistent valve position between steady state results and action table	Info	If difference is less than 1% value of action table will be used otherwise user had to change input
Start controlling	Info	The valve (re)opens and starts balancing
Closes	Info	Valve fully closed; position is 0
Reaching maximum opening	Info	Valve runs out of control; capacity in maximum open position to small too obtain required upstream pressure
Reaching minimum opening	Info	Valve runs out of control; capacity in minimum open position to large too obtain required upstream pressure

4.12.3.6 Example

A pressure control valve is located in the middle of a 1000 m long pipeline.

The initial position of the valve is set to 80 % open. The steady state upstream pressure is 1.54 bar. The required pressure is 1.7 bar.

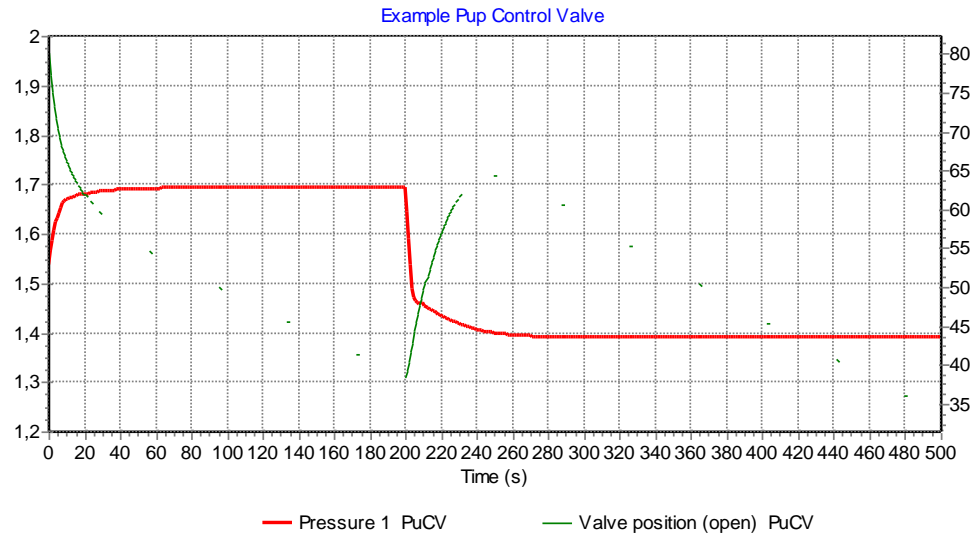




In the unsteady state the PuCV starts controlling. Due to the decreasing pressure in the small reservoir the PuCV adjusts the position continuously to maintain the set pressure (1.7 bar).

At T= 200 sec, the setpressure has been changed to 1.4 bar (via the action table).

Next figure shows the results.



4.13 Damper, Surge vessel

4.13.1 Damper, Surge vessel



Damper, Surge vessel (vertical bladdered)

Supplier type

Type label	description	active
Damper, Surge vessel	Vertically positioned, prismatic (constant area), bladder or piston type damper	No

4.13.1.1 Mathematical model

The damper or surge absorber is a vessel which allows for some dynamic (temporary) liquid storage upon exceedance of a certain (filling) pressure by compressing an enclosed gas volume. The gas is separated from the liquid by a bladder or a piston, to prevent it from being dissolved in the liquid. The damper described herein is a vertical, prismatic (e.g. a standing cylinder) vessel, with constant storage area.

If during the computation the system pressure exceeds the initial filling pressure, the behaviour of the vertical damper (or accumulator) is governed by two equations. The first equation describes the behaviour of the enclosed gas:

$$PV^k = C \quad (1)$$

in which:

P	= absolute gas pressure	[N/m ²]
V	= gas volume	[m ³]
k	= Laplace coefficient	[-]
C	= constant	[Nm]

The Laplace coefficient (also known as polytropic exponent) depends on the thermodynamic behaviour of the gas. Isothermal expansion is described by $k = 1$. Adiabatic expansion is described by $k = 1.4$ (for air or nitrogen). In the adiabatic case the polytropic exponent is the same as the isentropic exponent (κ), also known as the

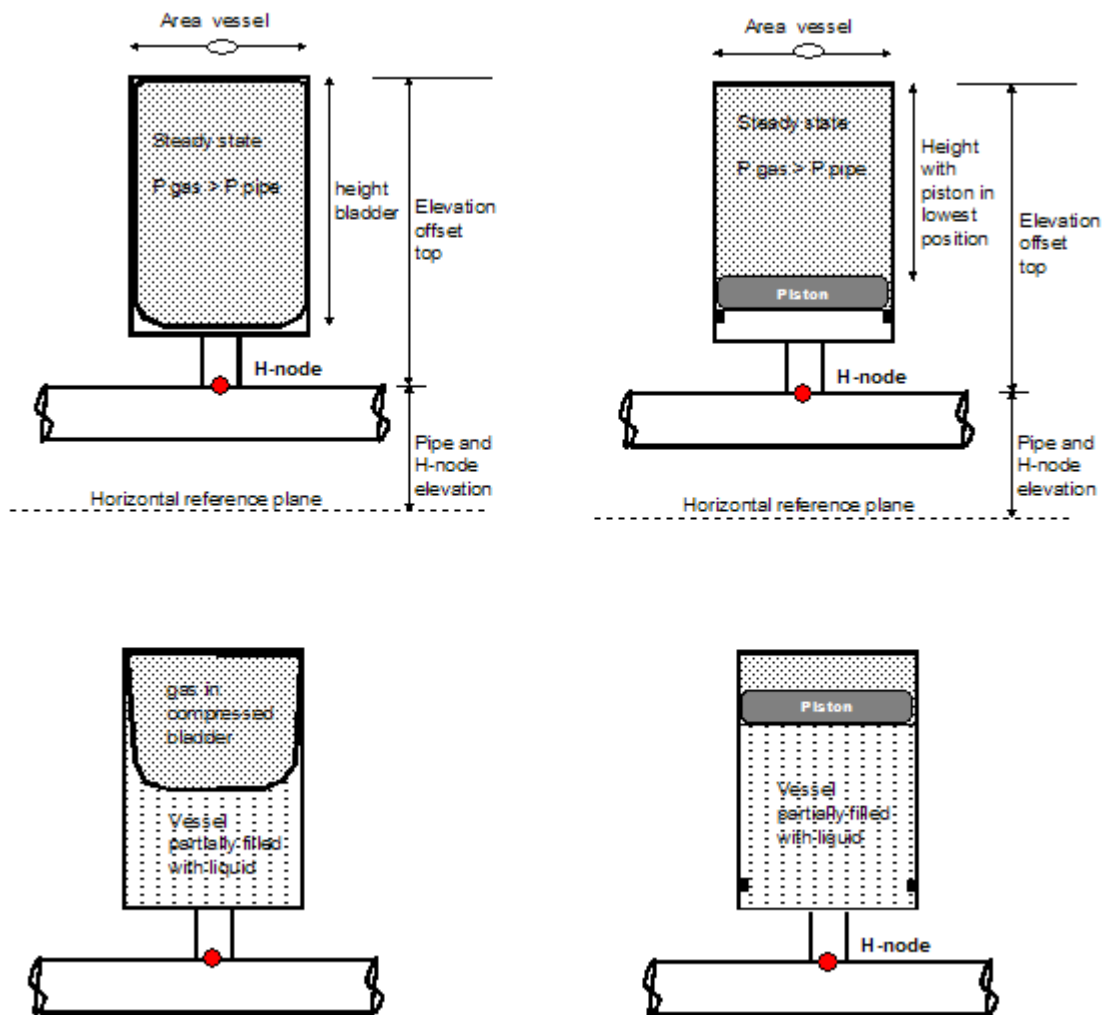
ratio of specific heats c_p/c_v . The gas is separated from the liquid by a bladder or a piston. The constant C is based on the filling pressure and the air volume of the damper with the bladder or piston fully down.

The second equation governs the amount of supplying discharge Q :

$$Q = -A_{av} \frac{dx_f}{dt} \tag{2}$$

in which A_{av} denotes the storage surface and x_f the (effective) liquid level in the vessel with respect to the horizontal reference plane. The minus sign indicates that the vessel supplies fluid to the system when the fluid level is decreasing in time (thus a positive flow if supply to the system and a negative flow if supply to the vessel) in steady state and if the system pressure is below the initial filling pressure, the air vessel does not supply liquid. Hence in this state the governing equation is:

$$Q = 0 \tag{3}$$



4.13.1.2 Hydraulic specifications

description	Input	unit	range	default	remarks
height	real	[m]			See remarks
area	real	[m ²]	(0-100]		See remarks
elevation offset top	real	[m]			See remarks
filling pressure	real	[Pa]			Relative to atmospheric pressure
Laplace coefficient	real	[-]	[1-1.4]		

Remarks

The initial gas volume: $V_{\text{gas}} = \text{height} \times \text{area}$

The “elevation offset top” is introduced to use the same input parameters for differently located dampers. The real elevation of the top of the vessel is determined by:

Elevation top = elevation H-node + elevation offset top

Piston weight and friction and/or bladder tension are neglected

Inlet/outlet losses are not included in this damper model, but they can be easily taken into account by applying the additional component RESIST

4.13.1.3 Component specific output

Gas pressure [N/m².a] (Note: absolute pressure)

Gas volume [m³]

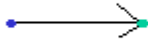
Fluid level [m]

Note

The default output parameters “Pressure” and “Pressure head” represent the P and H in the H-node of the Damper.

4.14 Infinite Pipe

4.14.1 INFPIP (class)



Infinite pipe

Supplier type

type label	description	active
Infinite Pipe (H-bound)	Infinite pipe with initial pressure head	No
Infinite Pipe (Q-bound)	Infinite pipe with initial discharge	No

4.14.1.1 Mathematical model

The infinite pipe model is used in case of detailed analyses of e.g. pumping stations. In such cases the internal piping and check valves are modelled in detail leading to a small time step. As a consequence the long transportation line departing from the station would get far too many elements for the computation to be feasible anymore. As long as the simulation time is shorter than $2L/c$ no reflection other than line packing will be present. With the infinite pipe component the user can apply a reflection-free pipe boundary condition. The initial hydraulic gradient in the infinite pipe is determined by:

$$\frac{\Delta H}{\Delta x} = \frac{\lambda}{D} \frac{v^2}{2g} \quad (1)$$

in which:

ΔH	=	head difference.	[m]
Δx	=	pipe element length	[m]
λ	=	Darcy Weisbach friction factor.	[-]
D	=	internal pipe diameter	[m]
v	=	flow velocity	[m/s]
g	=	gravitational acceleration	[m/s ²]

The length L is determined by the water hammer wave velocity c and the computational time step Δt with $L = c \Delta t$. For further explanation of the input parameters see "Definitions" on page 147. The initial flow or head results from the steady state balance.

Each time step the equation specified by the infinite pipe is the standard C- water hammer equation (see Conceptual Model):

$$(H - H_i) - R(Q - Q_i) + S Q_i |Q_i| = 0 \quad (2)$$

in which:

H	=	unknown head at node	[m]
Q	=	unknown discharge into pipe	[m ³ /s]
H _i	=	head in pipe at previous time step	[m]
Q _i	=	discharge in pipe at previous time step	[m ³ /s]
R	=	Inertance	[s/m ²]
S	=	Resistance	[s ² /m ⁵]

The values of H_i and Q_i are computed together with two extra internal points in the pipe. At each internal point the C⁺ and C⁻ water hammer equations are solved, except the last point, where only the C⁻ equation is available and the second equation is constituted by time extrapolation.

Tip:

INFPIP may be used for flows from the system into the infinite pipe ($Q > 0$) or for flows from the infinite pipe into the system ($Q < 0$).

4.14.2 Infinit Pipe (H-bound)

4.14.2.1 Hydraulic specifications

Description	input	unit	range	default	remarks
inner diameter	real	[m]	(0-5)		
friction factor	real	[-]	(0-1)		
wall thickness	real	[m]	(0-0.50)		only in transient mode
Young's modulus	real	[N/m ²]	(0-10 ¹⁴)		only in transient mode
initial head	real	[m]			

See also "Mathematical model" on page 284 and "PIPE" on page 331.

4.14.2.2 Component specific output

Delivery rate [m³/s]

4.14.2.3 H-actions

None

4.14.2.4 Component messages

None

4.14.3 Infinit Pipe (Q-bound)

4.14.3.1 Hydraulic specifications

Description	input	unit	range	default	remarks
inner diameter	real	[m]	(0-5)		
friction factor	real	[-]	(0-1)		
wall thickness	real	[m]	(0-0.50)		only in transient mode
Young's modulus	real	[N/m ²]	(0-10 ¹⁴)		only in transient mode
initial discharge	real	[m ³ /s]			

See also "Mathematical model" on page 284

4.14.3.2 Component specific output

Delivery rate [m³/s]

4.14.3.3 H-actions

None

4.14.3.4 Component messages

None

4.15 T-JUNCTION



type label	description	active
T-straight	Three-node junction with the combined flow in the straight leg. Both straight legs have the same diameter	no
T-branch	Three-node junction with the combined flow in the branch. Both straight legs have the same diameter	no

Both components can be used for all kind of flow regimes in a 90 degree T-junction. The difference between the two T-junctions has to do with the positive flow definition. The arrows in the symbol specify the positive direction of the flow (and the velocity).

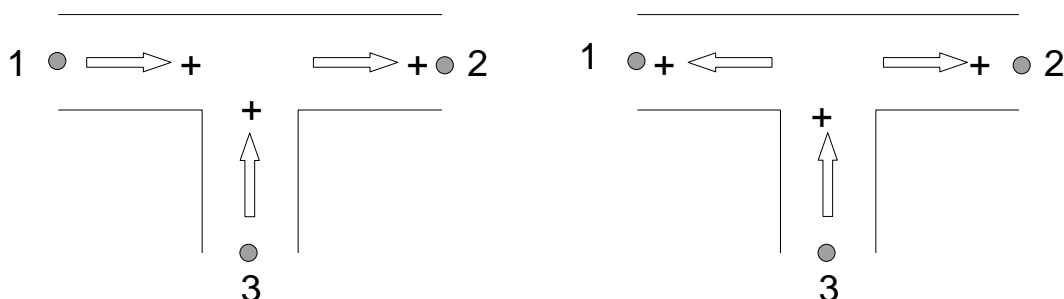
The two different symbols allow the user to express his design (normal) flow scenario with positive flows. If he expects in the steady state the combined (total) flow in the straight part he may prefer the left symbol. A combined flow, coming into or out of the branch, gives preference to the right symbol.

4.15.1 Mathematical model

4.15.1.1 Positive flow definition

The definition for the positive direction of flow (and velocity) is important to understand the results in the property window. Each T-junction type has his own definition.

For the **T-straight** component the positive flow definition is defined in the left figure below. For the **T-branch** component (right figure) the positive flow definition for the first connection point is opposite to that of the T-straight component; for the two other connection points the positive flow definitions are the same as for the T-straight component.



4.15.1.2 Flow regimes

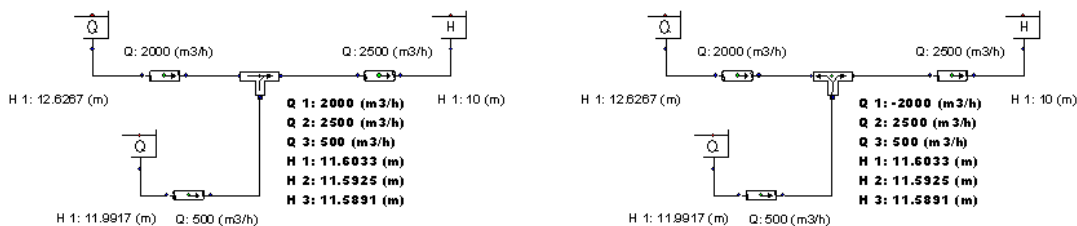
In the table below all different flow regimes which can occur are listed and explained with a scheme with corresponding actual flow arrows. It is called straight flow if the total flow (in some literature also called: combined flow) is in one of the straight legs, branch flow means that the total flow is in the branch.

Table 1: definition of flow regimes

Combining straight flow	<p>Positive straight flow</p>	<p>Negative straight flow</p>
Dividing straight flow	<p>Positive straight flow</p>	<p>Negative straight flow</p>
Combining branch flow		
Dividing branch flow		

The sign of the discharge and velocity output (results) depends of the T-junction type chosen, but is independent of the flow regime definition. The example below will illustrate this.

The main flow is from West to East with a supply from the South. That means we are dealing with a **combin~~ing~~** straight flow with the **combin~~ed~~** flow in leg 2. The solution of the calculation for the T-straight model (positive Q) as well as the T-branch model (negative Q) is shown in the figures below.



4.15.1.3 Equations

The head loss over a 3-node component depends on the distribution of the discharges and the area of the connected legs. The T-junction supports two different ways for retrieving the resistance coefficient ξ (ζ).

1. the head loss functions according to Idelchik's Handbook [ref. 1]
2. user specified table with ξ depending on discharge ratio. These table values belong to a fixed area ratio.

Please note that the loss coefficients according to the handbooks apply to junctions with a certain minimum pipe length interval in between multiple junctions.

The head loss in 3-node components is a function of the combined flow in the *combined* leg, which is either one leg (1 or 2) of the straight part or the branch (leg 3). Including the continuity equation (no production or loss of mass in the component) the general set of the three equations of the T-junction is:

Mass continuity, example T-straight	$f_1 = Q_1 - Q_2 + Q_3 = 0$
ΔH T-junction straight, combined flow in leg 2	
Head loss between straight legs	$f_2 = H_1 - H_2 - \xi_{12} \frac{Q_2 Q_2 }{2gA_2^2}$
Head loss between branch and straight leg	$f_3 = H_3 - H_2 - \xi_{32} \frac{Q_2 Q_2 }{2gA_2^2}$
ΔH T-junction straight, combined flow in leg 1	
Head loss between straight legs	$f_2 = H_1 - H_2 - \xi_{12} \frac{Q_1 Q_1 }{2gA_1^2}$
Head loss between straight leg and branch	$f_3 = H_1 - H_3 - \xi_{13} \frac{Q_1 Q_1 }{2gA_1^2}$
ΔH T-junction branch, combined flow in branch	
Head loss between branch and leg 1	$f_2 = H_3 - H_1 - \xi_{31} \frac{Q_3 Q_3 }{2gA_3^2}$
Head loss between branch and leg 2	$f_3 = H_3 - H_2 - \xi_{32} \frac{Q_3 Q_3 }{2gA_3^2}$

Where:

Q_i = total discharge in leg i	[m ³ /s]
H_i = energy head in connect point i	[m]
A_i = pipe area leg i (leg with total flow)	[m ²]
ξ_{ij} = loss coefficient between point i and j	[-]

The subscripts (1), (2) and (3) correspond to the different legs. The subscript x at the discharge Q_x refers always to the leg in which the combined (total) flow Q occurs.

For positive straight dividing flow and negative straight combining flow, Q_1 will be taken into account. For positive straight combining flow and negative straight dividing flow, Q_2 will be taken into account.

4.15.1.4 Resistance coefficients based on formulas

The resistance coefficients based of formulas are taken from the Idelchik Handbook with the following restriction to the section areas:

$$A_1 + A_3 > A_2$$

$$A_1 = A_2$$

For all different flow regimes the formulas used are described, and a table shows the ξ -values for various area and discharge ratios. Note that all indices used in the formulas are based on positive flow.

Straight flow combining

For the straight leg, the resistance coefficient is calculated according to:

$$\xi_{12} \approx 1.55 \frac{Q_3}{Q_2} - \left(\frac{Q_3}{Q_2} \right)^2$$

Table 23 shows the ξ_{12} values according to the formula above for various values of the discharge ratio.

Table 2: ξ_{12} values for combining flow

		A ₃ /A ₂
		Valid for all ranges
Q ₃ /Q ₂	0,0	0,00
	0,1	0,14
	0,2	0,27
	0,3	0,38
	0,4	0,46
	0,5	0,52
	0,6	0,57
	0,7	0,59
	0,8	0,60
	0,9	0,58
	1,0	0,55

For the branch, the resistance coefficient ξ_{32} is calculated according to:

$$\xi_{32} = A \left[1 + \left(\frac{Q_3}{Q_2} \frac{A_2}{A_3} \right)^2 - 2 \left(1 - \frac{Q_3}{Q_2} \right)^2 \right]$$

The coefficient A depends on the area and discharge ratios again. Table 3 shows the values.

Table 3: Values of the variable A .

A_3/A_2	≤ 0.35	> 0.35	
Q_3/Q_2	$0 - 1.0$	≤ 0.4	> 0.4
A	1.0	$0.9 \left(1 - \frac{Q_3}{Q_2} \right)$	0.55

Table 4 shows the ξ_{32} values according to the formulas above for various values of the discharge and area ratios.

Table 4: ξ_{32} values for combining flow.

		A_3/A_2									
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0
Q_3/Q_2	0,0	-1,00	-1,00	-1,00	-0,90	-0,90	-0,90	-0,90	-0,90	-0,90	-0,90
	0,1	0,38	-0,37	-0,51	-0,45	-0,47	-0,48	-0,49	-0,49	-0,49	-0,49
	0,2	3,72	0,72	0,16	-0,02	-0,09	-0,12	-0,14	-0,16	-0,17	-0,17
	0,3	9,02	2,27	1,02	0,37	0,24	0,17	0,13	0,10	0,08	0,07
	0,4	16,28	4,28	2,06	0,69	0,50	0,39	0,33	0,29	0,26	0,24
	0,5	25,50	6,75	3,28	1,13	0,83	0,66	0,56	0,49	0,44	0,41
	0,6	36,68	9,68	4,68	1,61	1,17	0,92	0,78	0,68	0,62	0,57
	0,7	49,82	13,07	6,26	2,14	1,53	1,20	1,00	0,87	0,78	0,72
	0,8	64,92	16,92	8,03	2,71	1,91	1,48	1,22	1,06	0,94	0,86
	0,9	81,98	21,23	9,98	3,32	2,32	1,78	1,45	1,24	1,09	0,98
1,0	101,00	26,00	12,11	3,99	2,75	2,08	1,67	1,41	1,23	1,10	

Straight flow dividing

For the straight leg the resistance coefficient is given using the following formula:

$$\xi_{12} = \tau_{12} \frac{Q_3}{Q_1}$$

where τ_{12} depends on the area and discharge ratios as given in table 5:

Table 5: Values of τ_{12}

A_3/A_1	≤ 0.4	0.4	
Q_3/Q_1	0-1.0	≤ 0.5	>0.5
τ_{12}	$0.4 (Q_3/Q_1)$	$0.2(2Q_3/Q_1 - 1)$	$0.3(2Q_3/Q_1 - 1)$

Table 6 shows the ξ_{12} values according to the formula above for various values of discharge ratios.

Table 6: ξ_{12} values for dividing flow

		A_3/A_1										
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0	
Q_3/Q_1	0,0	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00
	0,1	0,00	0,00	0,00	0,00	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02
	0,2	0,02	0,02	0,02	0,02	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02
	0,3	0,04	0,04	0,04	0,04	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02
	0,4	0,06	0,06	0,06	0,06	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02
	0,5	0,10	0,10	0,10	0,10	0,00	0,00	0,00	0,00	0,00	0,00	0,00
	0,6	0,14	0,14	0,14	0,14	0,04	0,04	0,04	0,04	0,04	0,04	0,04
	0,7	0,20	0,20	0,20	0,20	0,08	0,08	0,08	0,08	0,08	0,08	0,08
	0,8	0,26	0,26	0,26	0,26	0,14	0,14	0,14	0,14	0,14	0,14	0,14
	0,9	0,32	0,32	0,32	0,32	0,22	0,22	0,22	0,22	0,22	0,22	0,22
1,0	0,40	0,40	0,40	0,40	0,30	0,30	0,30	0,30	0,30	0,30	0,30	

The resistance coefficient ξ_{13} for the branch is calculated using the following formula:

For $A_3/A_1 \leq 2/3$:

$$\xi_{13} = A' \left[1 + \left(\frac{Q_3 A_1}{Q_1 A_3} \right)^2 \right]$$

For $A_3/A_1 = 1$ (up to velocity ratio to $v_3/v_1 \approx 2.0$):

$$\xi_{13} = A' \left[1 + 0.3 \left(\frac{Q_3 A_1}{Q_1 A_3} \right)^2 \right]$$

The values of A' are given in table 7.

Table 7: A' values

A ₃ /A ₁	≤ 0.35		> 0.35	
Q ₃ /Q ₁	≤ 0.4	> 0.4	≤ 0.6	> 0.6
A'	1.1 – 0.7 Q ₃ /Q ₁	0.85	1.0 – 0.65 Q ₃ /Q ₁	0.6

Table 8 shows the ζ₁₃ values according to the formulas above for various values of discharge and area ratio.

Table 8: ξ₁₃ values for dividing flow

		A ₃ /A ₁									
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1
Q ₃ /Q ₁	0	1.10	1.10	1.10	1.00	1.00	1.00	1.00	1.00	1.00	1.00
	0,1	1.34	1.11	1.06	0.95	0.95	0.94	0.94	0.94	0.94	0.94
	0,2	2.11	1.25	1.09	0.94	0.91	0.90	0.89	0.89	0.88	0.88
	0,3	3.29	1.49	1.16	0.94	0.89	0.87	0.85	0.84	0.83	0.83
	0,4	4.76	1.80	1.26	0.96	0.88	0.84	0.81	0.80	0.78	0.78
	0,5	7.23	2.44	1.56	0.99	0.88	0.82	0.78	0.75	0.74	0.73
	0,6	10.03	3.15	1.87	1.02	0.87	0.79	0.74	0.71	0.69	0.68
	0,7	13.35	3.97	2.24	1.15	0.95	0.85	0.78	0.74	0.71	0.69
	0,8	17.17	4.93	2.66	1.32	1.06	0.92	0.84	0.78	0.74	0.72
	0,9	21.51	6.01	3.14	1.51	1.18	1.00	0.90	0.83	0.78	0.75
1	26.35	7.23	3.68	1.73	1.32	1.10	0.97	0.88	0.82	0.78	

Branch flow combining

WANDA calculates the head loss coefficients for each leg (ζ₃₁ and ζ₃₂) from the following formula:

$$\zeta_{31} = A \left\{ 1 + \left(\frac{A_3}{A_1} \right)^2 + 3 \left(\frac{A_3}{A_1} \right)^2 \left[\left(\frac{Q_1}{Q_3} \right)^2 - \left(\frac{Q_1}{Q_3} \right) \right] \right\}$$

In which A is the same factor as used in the T-junction straight combining (table 3).

Table 9 shows the ζ₃₁ values according to the formulas above for various values of discharge and area ratio (identical for ζ₃₂ in leg 2).

Table 9: ξ_{31} and ξ_{32} values for combining branch flow

		A ₁ /A ₃									
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0
Q ₁ /Q ₃	0,0	101,00	26,00	12,11	6,52	4,50	3,40	2,74	2,31	2,01	1,80
	0,1	74,00	19,25	9,11	4,51	3,18	2,45	2,02	1,73	1,54	1,40
	0,2	53,00	14,00	6,78	3,06	2,22	1,76	1,48	1,30	1,18	1,09
	0,3	38,00	10,25	5,11	2,09	1,56	1,28	1,11	0,99	0,92	0,86
	0,4	29,00	8,00	4,11	1,48	1,14	0,96	0,85	0,78	0,73	0,69
	0,5	26,00	7,25	3,78	1,41	1,10	0,93	0,83	0,76	0,72	0,69
	0,6	29,00	8,00	4,11	1,51	1,17	0,98	0,86	0,79	0,74	0,70
	0,7	38,00	10,25	5,11	1,82	1,36	1,12	0,97	0,87	0,80	0,75
	0,8	53,00	14,00	6,78	2,34	1,69	1,34	1,13	1,00	0,90	0,84
	0,9	74,00	19,25	9,11	3,06	2,16	1,67	1,37	1,18	1,05	0,95
1,0	101,00	26,00	12,11	3,99	2,75	2,08	1,67	1,41	1,23	1,10	

Branch flow dividing

WANDA calculates the head loss coefficients for each leg (ξ_{31} and ξ_{32}) from the following formula:

$$\xi_{31} = 1 + k \left(\frac{Q_1 A_3}{Q_3 A_1} \right)^2$$

In which k is 0.3 valid for welded tees.

Table 10 shows the ξ_{31} values according to the formulas above for various values of discharge and area ratio (identical for ξ_{32} in leg 2).

Table 10: ξ_{31} and ξ_{32} values for dividing branch flow

		A ₁ /A ₃									
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0
Q ₁ /Q ₃	0,0	1,00	1,00	1,00	1,00	1,00	1,00	1,00	1,00	1,00	1,00
	0,1	1,30	1,08	1,03	1,02	1,01	1,01	1,01	1,00	1,00	1,00
	0,2	2,20	1,30	1,13	1,08	1,05	1,03	1,02	1,02	1,01	1,01
	0,3	3,70	1,68	1,30	1,17	1,11	1,08	1,06	1,04	1,03	1,03
	0,4	5,80	2,20	1,53	1,30	1,19	1,13	1,10	1,08	1,06	1,05
	0,5	8,50	2,88	1,83	1,47	1,30	1,21	1,15	1,12	1,09	1,08
	0,6	11,80	3,70	2,20	1,68	1,43	1,30	1,22	1,17	1,13	1,11
	0,7	15,70	4,68	2,63	1,92	1,59	1,41	1,30	1,23	1,18	1,15
	0,8	20,20	5,80	3,13	2,20	1,77	1,53	1,39	1,30	1,24	1,19
	0,9	25,30	7,08	3,70	2,52	1,97	1,67	1,50	1,38	1,30	1,24
1,0	31,00	8,50	4,33	2,88	2,20	1,83	1,61	1,47	1,37	1,30	

4.15.2 T-Junction properties

The input properties of both T-junction are not exactly the same. The differences are located in the sequence of the Xi tables and the description labels for input tables and output properties. The input and output properties for both types are specified below.

4.15.2.1 Properties T-junction straight

Input properties T-junction straight

Description	input	unit	range	default	Remarks
Diameter straight leg	Real	[mm]			
Diameter branch leg	Real	[mm]			
Xi method	Formula Tables			Formula	
Xi tables valid for	Combining Dividing All				Only when Xi method = Tables
Xi straight flow combining, leg straight	Table (ξ_{12})				if Xi tables = Combining or All
Xi straight flow combining, leg branch	Table (ξ_{32})				if Xi tables = Combining or All
Xi straight flow dividing, leg straight	Table (ξ_{12})				if Xi tables = Dividing or All
Xi straight flow dividing, leg branch	Table (ξ_{13})				if Xi tables = Dividing or All
Xi branch flow combining, leg 1	Table (ξ_{13})				if Xi tables = All
Xi branch flow combining, leg 2	Table (ξ_{23})				if Xi tables = All
Xi branch flow dividing, leg 1	Table (ξ_{31})				if Xi tables = All
Xi branch flow dividing, leg 2	Table (ξ_{32})				if Xi tables = All

With the tables for the Xi-values it is possible to choose user-defined loss coefficients, for instance for a certain kind of T-junction with rounded angles. The user has to specify a set of *loss coefficient values* related to the *discharge ratio* between the side and combined branch. If the user does not specify both combining and dividing Xi-tables, WANDA uses the formulas according to the Idelchik handbook for the unspecified flow regimes.

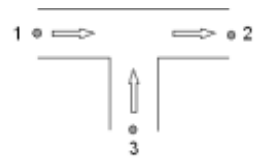
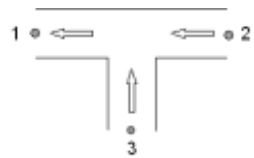
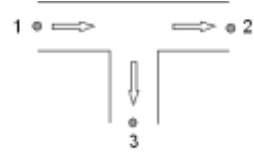
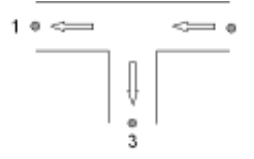
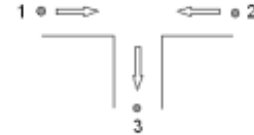
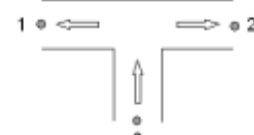
Example of Xi table; straight flow combining, leg branch (ξ_{32})
valid for Area ratio = 0.5

Discharge ratio [-]	ξ_i [-]
0.0	0.90
0.2	0.12
0.4	0.39
0.6	0.92
0.8	1.48
1.0	2.08

Component specific output T-junction straight

Loss coefficient straight [-]	The ξ_{12} for positive flow and ξ_{21} for negative flow In case of branch flow regime this is the ξ_{31} coefficient
Loss coefficient branch [-]	The ξ_{13} for positive flow and ξ_{31} for negative flow In case of branch flow regime this is the ξ_{32} coefficient
Head loss straight [m]	The ΔH_{12} for positive flow and ΔH_{21} for negative flow In case of branch flow regime this is the ΔH_{13} for combining and ΔH_{31} for dividing flow
Head loss branch [m]	The ΔH_{13} for positive flow and ΔH_{31} for negative flow In case of branch flow regime this is the ΔH_{23} for combining and ΔH_{32} for dividing flow

Component messages T-junction straight

Message	Type	Explanation
Combining	Info	Positive straight flow 
		Negative straight flow 
Dividing	Info	Positive straight flow 
		Negative straight flow 
Branch combining	Info	
Branch dividing	info	

With the tables for the ξ -values it is possible to choose user-defined loss coefficients, for instance for a certain kind of T-junction with rounded angles. The user has to specify a set of *loss coefficients values* related to the *discharge ratio* between the side and combined branch. If the user does not specify both combining and dividing ξ -tables, WANDA uses the formulas according to the Idelchik handbook for the unspecified flow regimes.

NOTE: The numbering of the branches in the literature is not always consistent with the WANDA definition as given under ‘Mathematical Model’ above. The Miller Handbook for instance indicates the side branch with 1, the straight branch with 2 and the combined one as 3.

Furthermore please note that there is a discontinuity when the flow in one of the legs goes through zero. This might lead to unstable behaviour.

4.15.2.2 Properties T-junction branch

Input properties T-junction branch

Description	input	unit	range	default	Remarks
Diameter straight leg	Real	[mm]			
Diameter branch leg	Real	[mm]			
Xi method	Formula Tables			Formula	
Xi tables valid for	Combining Dividing All				Only when Xi method = Tables
Xi branch flow combining, leg 1	Table (ξ_{13})				if Xi tables = Combining or All
Xi branch flow combining, leg 2	Table (ξ_{23})				if Xi tables = Combining or All
Xi branch flow dividing, leg 1	Table (ξ_{31})				if Xi tables = Dividing or All
Xi branch flow dividing, leg 2	Table (ξ_{32})				if Xi tables = Dividing or All
Xi straight flow combining, leg straight	Table (ξ_{12})				if Xi tables = All
Xi straight flow combining, leg branch	Table (ξ_{32})				if Xi tables = All
Xi straight flow dividing, leg straight	Table (ξ_{12})				if Xi tables = All
Xi straight flow dividing, leg branch	Table (ξ_{13})				if Xi tables = All

Component specific output T-junction branch

Loss coefficient 1 [-]	The ξ_{31} for dividing flow and ξ_{13} for combining flow In case of straight flow regime this is the ξ_{12} or ξ_{21} coefficient
Loss coefficient 2 [-]	The ξ_{32} for dividing flow and ξ_{23} for combining flow In case of straight flow regime this is the ξ_{32} or ξ_{23} coefficient
Head loss 1 [m]	The ΔH_{31} for dividing flow and ΔH_{13} for combining flow In case of straight flow regime this is the ΔH_{12} or ΔH_{21}
Head loss 2 [m]	The ΔH_{32} for dividing flow and ΔH_{23} for combining flow In case of straight flow regime this is the ΔH_{32} or ΔH_{23}

Component messages T-junction branch

Message	Type	Explanation	
Branch combining	Info		
Branch dividing	info		
Combining straight	Info	Positive straight flow	Negative straight flow
Dividing straight	Info	Positive straight flow	Negative straight flow

With the tables for the ξ -values it is possible to choose user-defined loss coefficients, for instance for a certain kind of T-junction with rounded angles. The user has to specify a set of *loss coefficients values* related to the *discharge ratio* between the side and combined branch. If the user does not specify both combining and dividing ξ -tables, WANDA uses the formulas according to the Idelchik handbook for the unspecified flow regimes.

NOTE: The numbering of the branches in the literature is not always consistent with the WANDA definition as given under ‘Mathematical Model’ above. The Miller Handbook for instance indicates the side branch with 1, the straight branch with 2 and the combined one as 3.

Furthermore please note that there is a discontinuity when the flow in one of the legs goes through zero. This might lead to unstable behaviour.

4.16 Y-JUNCTION



type label	description	active
Y-junction combining	Three-node junction with a non right angle branched leg and a combining flow. Both straight legs have the same diameter	no
Y-junction dividing	Three-node junction with a non right angle branched leg and a dividing flow. Both straight legs have the same diameter	no

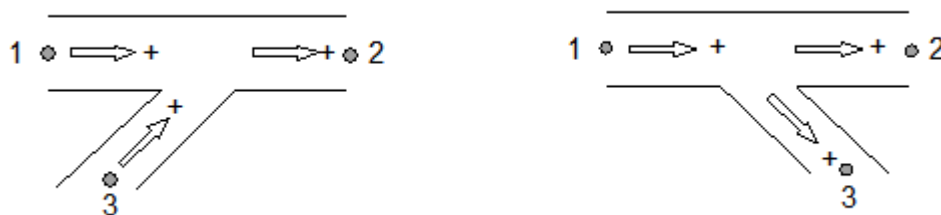
Both components can be used for all kind of flow regimes in a Y-junction with several angles. The supported angles for the formula based types are 30, 45 and 60 degrees. The difference between the two Y-junctions has to do with the flow regime in the branched legs, combining either dividing. The arrows in the symbol specify the positive direction of the flow (and the velocity).

4.16.1 Mathematical model

4.16.1.1 Positive flow definition

The definition for the positive direction of flow (and velocity) is important to understand the results in the property window. Each Y-junction type has his own definition.

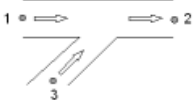
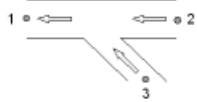
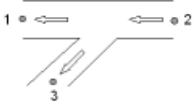
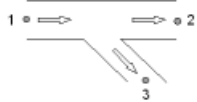
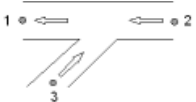
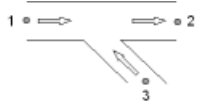
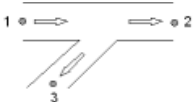
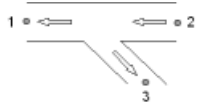
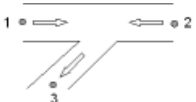
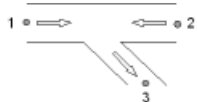
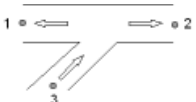
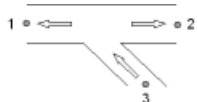
For the **Y-junction combining** component the positive flow definition is defined in the left figure below. For the **Y-junction dividing** component (right figure) the positive flow definition for the branch leg is opposite to that of the Y-junction combining. The positive flow definitions in the straight legs are the same for both types.



4.16.1.2 Flow regimes

In the table below all different flow regimes which can occur are listed and explained with a scheme with corresponding actual flow arrows. It is called straight flow if the total flow (in some literature also called: combined flow) is in one of the straight legs, branch flow means that the total flow is in the branch.

Table 1: definition of flow regimes for both types

	Y-junction combining type	Y-junction dividing type
1 Combining straight flow	 <p>Positive straight flow</p>	 <p>Negative straight flow</p>
2 Dividing straight flow	 <p>Negative straight flow</p>	 <p>Positive straight flow</p>
3 Combining straight flow	 <p>Negative straight flow</p>	 <p>Positive straight flow</p>
4 Dividing straight flow	 <p>Positive straight flow</p>	 <p>Negative straight flow</p>
5 Combining branch flow	 <p>Negative branch flow</p>	 <p>Positive branch flow</p>
6 Dividing branch flow	 <p>Positive branch flow</p>	 <p>Negative branch flow</p>

4.16.1.3 Equations

The head loss over a 3-node component depends on the distribution of the discharges and the area of the connected legs. The Y-junction supports two different ways for retrieving the resistance coefficient ξ (ζ).

1. the head loss functions according to Idelchik's Handbook [ref. 1] for angles of 30, 45 and 60 degree

2. user specified table with ξ_i depending on discharge ratio. These table values belong to a fixed area ratio.

Please note that the loss coefficients according to the handbooks apply to junctions with a certain minimum pipe length interval in between multiple junctions.

The head loss in 3-node components is a function of the combined flow in the *combined* leg, which is either one leg (1 or 2) of the straight part or the branch (leg 3). Including the continuity equation (no production or loss of mass in the component) the general set of the three equations of the Y-junction is:

Mass continuity, example Y-combining	$f_1 = Q_1 - Q_2 + Q_3 = 0$
ΔH Y-junction combining, combined flow in leg 2	
Head loss between straight legs	$f_2 = H_1 - H_2 - \xi_{12} \frac{Q_2 Q_2 }{2gA_2^2}$
Head loss between branch and straight leg	$f_3 = H_3 - H_2 - \xi_{32} \frac{Q_2 Q_2 }{2gA_2^2}$
ΔH Y-junction dividing, combined flow in leg 1	
Head loss between straight legs	$f_2 = H_1 - H_2 - \xi_{12} \frac{Q_1 Q_1 }{2gA_1^2}$
Head loss between straight leg and branch	$f_3 = H_1 - H_3 - \xi_{13} \frac{Q_1 Q_1 }{2gA_1^2}$
ΔH Y-junction, combined flow in branch	
Head loss between branch and leg 1	$f_2 = H_3 - H_1 - \xi_{31} \frac{Q_3 Q_3 }{2gA_3^2}$
Head loss between branch and leg 2	$f_3 = H_3 - H_2 - \xi_{32} \frac{Q_3 Q_3 }{2gA_3^2}$

Where:	Q_i = total discharge in leg i	[m ³ /s]
	H_i = energy head in connect point i	[m]
	A_i = pipe area leg i (leg with total flow)	[m ²]
	ξ_{ij} = loss coefficient between point i and j	[-]

The subscripts (1), (2) and (3) correspond to the different legs. The subscript x at the discharge Q_x refers always to the leg in which the combined (total) flow Q occurs.

For positive straight dividing flow and negative straight combining flow, Q_1 will be taken into account. For positive straight combining flow and negative straight dividing flow, Q_2 will be taken into account.

4.16.1.4 Resistance coefficients based on formulas

The resistance coefficients based of formulas are taken from the Idelchik Handbook with the following restriction to the section areas:

$$A_1 + A_3 > A_2$$

$$A_1 = A_2$$

The formulas are only valid for the normal flow regimes 1 and 2, as explained in table 1. For each formula the ξ -values for various area and discharge ratios are collected in a table. Note that all indices used in the formulas are based on positive flow.

For the other flow regimes where one of the flow direction is around the acute angle there are no formulas available. For those situations a fixed value of $\xi = 2,0$ will be used.

Straight flow combining

For the **straight** leg, the resistance coefficient ξ_{12} is calculated according to:

$$\xi_{12} \approx 1 - \left(1 - \frac{Q_3}{Q_2}\right)^2 - C_\alpha \frac{A_2}{A_3} \left(\frac{Q_3}{Q_2}\right)^2$$

C_α is a coefficient, which depends on the angle α . In WANDA three fixed α .values are included:

- $\alpha = 30^\circ$: $C_\alpha = 1,74$;
- $\alpha = 45^\circ$: $C_\alpha = 1,41$;
- $\alpha = 60^\circ$: $C_\alpha = 1,00$.

Table 2a until 2c shows the ξ_{12} values according the formula above for various ratios of discharge and areas for the supported angles, respectively 30, 45 and 60 degrees.

Table 2a: Values of ξ_{12} for $\alpha=30^\circ$ straight flow combining

		A_3/A_2									
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0
Q_3/Q_2	0,0	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00
	0,1	0,02	0,10	0,13	0,15	0,16	0,16	0,17	0,17	0,17	0,17
	0,2	-0,34	0,01	0,13	0,19	0,22	0,24	0,26	0,27	0,28	0,29
	0,3	-1,06	-0,27	-0,01	0,12	0,20	0,25	0,29	0,31	0,34	0,35
	0,4	-2,14	-0,75	-0,29	-0,06	0,08	0,18	0,24	0,29	0,33	0,36
	0,5	-3,60	-1,42	-0,70	-0,34	-0,12	0,03	0,13	0,21	0,27	0,31
	0,6	-5,42	-2,29	-1,25	-0,73	-0,41	-0,20	-0,05	0,06	0,14	0,21
	0,7	-7,62	-3,35	-1,93	-1,22	-0,80	-0,51	-0,31	-0,16	-0,04	0,06
	0,8	-10,18	-4,61	-2,75	-1,82	-1,27	-0,90	-0,63	-0,43	-0,28	-0,15
	0,9	-13,10	-6,06	-3,71	-2,53	-1,83	-1,36	-1,02	-0,77	-0,58	-0,42
	1,0	-16,40	-7,70	-4,80	-3,35	-2,48	-1,90	-1,49	-1,17	-0,93	-0,74

Table 2b: Values of ξ_{12} for $\alpha=45^\circ$ straight flow combining

		A_3/A_2									
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0
Q_3/Q_2	0,0	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00
	0,1	0,05	0,12	0,14	0,15	0,16	0,17	0,17	0,17	0,17	0,18
	0,2	-0,20	0,08	0,17	0,22	0,25	0,27	0,28	0,29	0,30	0,30
	0,3	-0,76	-0,12	0,09	0,19	0,26	0,30	0,33	0,35	0,37	0,38
	0,4	-1,62	-0,49	-0,11	0,08	0,19	0,26	0,32	0,36	0,39	0,41
	0,5	-2,77	-1,01	-0,42	-0,13	0,05	0,16	0,25	0,31	0,36	0,40
	0,6	-4,24	-1,70	-0,85	-0,43	-0,18	-0,01	0,11	0,21	0,28	0,33
	0,7	-6,00	-2,54	-1,39	-0,82	-0,47	-0,24	-0,08	0,05	0,14	0,22
	0,8	-8,06	-3,55	-2,05	-1,30	-0,84	-0,54	-0,33	-0,17	-0,04	0,06
	0,9	-10,43	-4,72	-2,82	-1,87	-1,29	-0,91	-0,64	-0,44	-0,28	-0,15
	1,0	-13,10	-6,05	-3,70	-2,52	-1,82	-1,35	-1,01	-0,76	-0,57	-0,41

Table 2c: Values of ξ_{12} for $\alpha=60^\circ$ straight flow combining

		A_3/A_2									
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0
Q_3/Q_2	0,0	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00
	0,1	0,09	0,14	0,16	0,17	0,17	0,17	0,18	0,18	0,18	0,18
	0,2	-0,04	0,16	0,23	0,26	0,28	0,29	0,30	0,31	0,32	0,32
	0,3	-0,39	0,06	0,21	0,28	0,33	0,36	0,38	0,40	0,41	0,42
	0,4	-0,96	-0,16	0,11	0,24	0,32	0,37	0,41	0,44	0,46	0,48
	0,5	-1,75	-0,50	-0,08	0,12	0,25	0,33	0,39	0,44	0,47	0,50
	0,6	-2,76	-0,96	-0,36	-0,06	0,12	0,24	0,33	0,39	0,44	0,48
	0,7	-3,99	-1,54	-0,72	-0,31	-0,07	0,09	0,21	0,30	0,37	0,42
	0,8	-5,44	-2,24	-1,17	-0,64	-0,32	-0,11	0,05	0,16	0,25	0,32
	0,9	-7,11	-3,06	-1,71	-1,03	-0,63	-0,36	-0,17	-0,02	0,09	0,18
	1,0	-9,00	-4,00	-2,33	-1,50	-1,00	-0,67	-0,43	-0,25	-0,11	0,00

For the **branch leg**, the resistance coefficient is calculated according to:

$$\xi_{32} = A \left[1 + \left(\frac{Q_3}{Q_2} \frac{A_2}{A_3} \right)^2 - 2 \left(1 - \frac{Q_3}{Q_2} \right)^2 - C_\alpha \frac{A_2}{A_3} \left(\frac{Q_3}{Q_2} \right)^2 \right]$$

The coefficient A depends on the area and discharge ratios again. Table 3 shows the values:

Table 3: Values of the variable A .

A_3/A_2	$\leq 0,35$	$> 0,35$
Q_3/Q_2	$0,0 - 1,0$	$\leq 0,4$ $> 0,4$
A	1,0	$0,9 \left(1 - \frac{Q_3}{Q_2} \right)$ 0,55

C_α is a coefficient, which depends on the angle α . In WANDA three fixed α .values are included:

- $\alpha = 30^\circ$: $C_\alpha = 1,74$;
- $\alpha = 45^\circ$: $C_\alpha = 1,41$;
- $\alpha = 60^\circ$: $C_\alpha = 1,00$.

Table 4a until 4c shows the ξ_{32} values according the formula above for various ratios of discharge and areas for the supported angles, respectively 30, 45 and 60 degrees.

Table 4a: Values of ξ_{32} for $\alpha=30^\circ$ branch flow combining

		A_3/A_2									
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0
Q_3/Q_2	0,0	-1,00	-1,00	-1,00	-0,90	-0,90	-0,90	-0,90	-0,90	-0,90	-0,90
	0,1	0,21	-0,46	-0,57	-0,49	-0,50	-0,50	-0,51	-0,51	-0,51	-0,51
	0,2	3,02	0,37	-0,07	-0,15	-0,19	-0,21	-0,21	-0,22	-0,22	-0,22
	0,3	7,45	1,49	0,50	0,12	0,04	0,01	-0,01	-0,02	-0,03	-0,03
	0,4	13,50	2,89	1,13	0,32	0,20	0,14	0,11	0,10	0,09	0,09
	0,5	21,15	4,57	1,83	0,54	0,35	0,26	0,21	0,19	0,18	0,17
	0,6	30,42	6,55	2,59	0,75	0,48	0,35	0,29	0,25	0,24	0,23
	0,7	41,29	8,81	3,42	0,96	0,59	0,42	0,33	0,29	0,26	0,25
	0,8	53,78	11,35	4,32	1,17	0,69	0,46	0,35	0,29	0,26	0,25
	0,9	67,89	14,18	5,28	1,39	0,77	0,48	0,34	0,27	0,23	0,21
1,0	83,60	17,30	6,31	1,60	0,84	0,48	0,31	0,21	0,17	0,14	

Table 4b: Values of ξ_{32} for $\alpha=45^\circ$ branch flow combining

		A_3/A_2									
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0
Q_3/Q_2	0,0	-1,00	-1,00	-1,00	-0,90	-0,90	-0,90	-0,90	-0,90	-0,90	-0,90
	0,1	0,24	-0,44	-0,56	-0,48	-0,49	-0,50	-0,50	-0,50	-0,50	-0,51
	0,2	3,16	0,44	-0,02	-0,12	-0,17	-0,19	-0,20	-0,21	-0,21	-0,21
	0,3	7,75	1,64	0,60	0,17	0,08	0,04	0,01	0,00	-0,01	-0,01
	0,4	14,02	3,15	1,31	0,39	0,25	0,19	0,15	0,13	0,12	0,12
	0,5	21,98	4,99	2,10	0,65	0,44	0,33	0,28	0,25	0,23	0,22
	0,6	31,60	7,14	2,99	0,91	0,61	0,46	0,38	0,33	0,31	0,29
	0,7	42,91	9,62	3,96	1,19	0,77	0,57	0,46	0,40	0,36	0,34
	0,8	55,90	12,41	5,02	1,47	0,92	0,66	0,52	0,44	0,39	0,36
	0,9	70,56	15,52	6,17	1,75	1,06	0,73	0,55	0,45	0,39	0,36
1,0	86,90	18,95	7,41	2,05	1,20	0,79	0,56	0,44	0,37	0,32	

Table 4c: Values of ξ_{32} for $\alpha=60^\circ$ branch flow combining

		A_3/A_2									
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0
Q_3/Q_2	0	-1,00	-1,00	-1,00	-0,90	-0,90	-0,90	-0,90	-0,90	-0,90	-0,90
	0,1	0,28	-0,42	-0,54	-0,47	-0,49	-0,49	-0,50	-0,50	-0,50	-0,50
	0,2	3,32	0,52	0,03	-0,09	-0,14	-0,17	-0,18	-0,19	-0,20	-0,20
	0,3	8,12	1,82	0,72	0,23	0,13	0,08	0,05	0,03	0,02	0,01
	0,4	14,68	3,48	1,52	0,48	0,32	0,25	0,20	0,18	0,16	0,15
	0,5	23,00	5,50	2,44	0,79	0,55	0,43	0,36	0,32	0,29	0,28
	0,6	33,08	7,88	3,48	1,12	0,77	0,59	0,50	0,44	0,40	0,37
	0,7	44,92	10,62	4,63	1,46	0,99	0,75	0,62	0,54	0,48	0,45
	0,8	58,52	13,72	5,90	1,83	1,21	0,90	0,72	0,62	0,55	0,51
	0,9	73,88	17,18	7,28	2,21	1,43	1,03	0,81	0,68	0,59	0,54
1,0	91,00	21,00	8,78	2,61	1,65	1,16	0,89	0,72	0,62	0,55	

Straight flow dividing

For the straight leg the resistance coefficient is given using the following formula:

$$\xi_{12} = \tau_{12} \frac{Q_3}{Q_1}$$

where τ_{12} depends on the area and discharge ratios as given in table 5:

Table 5: Values of τ_{12}

A_3/A_1	≤ 0.4	0.4	
Q_3/Q_1	0-1.0	≤ 0.5	>0.5
τ_{12}	$0.4 (Q_3/Q_1)$	$0.2(2Q_3/Q_1 - 1)$	$0.3(2Q_3/Q_1 - 1)$

Table 6 shows the ξ_{12} values according to the formula above for various values of discharge ratios.

Table 6: ξ_{12} values for dividing flow

		A_3/A_1										
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0	
Q_3/Q_1	0,0	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00
	0,1	0,00	0,00	0,00	0,00	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02
	0,2	0,02	0,02	0,02	0,02	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02
	0,3	0,04	0,04	0,04	0,04	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02
	0,4	0,06	0,06	0,06	0,06	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02	-0,02
	0,5	0,10	0,10	0,10	0,10	0,00	0,00	0,00	0,00	0,00	0,00	0,00
	0,6	0,14	0,14	0,14	0,14	0,04	0,04	0,04	0,04	0,04	0,04	0,04
	0,7	0,20	0,20	0,20	0,20	0,08	0,08	0,08	0,08	0,08	0,08	0,08
	0,8	0,26	0,26	0,26	0,26	0,14	0,14	0,14	0,14	0,14	0,14	0,14
	0,9	0,32	0,32	0,32	0,32	0,22	0,22	0,22	0,22	0,22	0,22	0,22
1,0	0,40	0,40	0,40	0,40	0,30	0,30	0,30	0,30	0,30	0,30	0,30	

The resistance coefficient ξ_{13} for the branch is calculated using the following formula:

$$\xi_{13} = A' \left[1 + \left(\frac{Q_3 A_1}{Q_1 A_3} \right)^2 - 2 \left(\frac{Q_3 A_1}{Q_1 A_3} \right) \cos \alpha \right]$$

The values of A' are given in table 7.

Table 7: A' values

A_3/A_1	≤ 0.35		> 0.35	
Q_3/Q_1	≤ 0.4	> 0.4	≤ 0.6	> 0.6
A'	$1.1 - 0.7 Q_3/Q_1$	0.85	$1.0 - 0.65 Q_3/Q_1$	0.6

Table 8a until 8c shows the ξ_{13} values according to the formulas above for various values of discharge and area ratio for the supported angles of 30, 45 and 60 degrees.

Table 8a: Values of ξ_{13} for $\alpha=30^\circ$ branch flow dividing

		A ₃ /A ₁									
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0
Q ₃ /Q ₁	0,0	1,10	1,10	1,10	1,00	1,00	1,00	1,00	1,00	1,00	1,00
	0,1	0,28	0,40	0,55	0,59	0,65	0,69	0,72	0,75	0,77	0,78
	0,2	1,47	0,26	0,28	0,33	0,41	0,46	0,51	0,55	0,58	0,60
	0,3	4,28	0,58	0,24	0,21	0,26	0,31	0,36	0,40	0,43	0,46
	0,4	8,26	1,26	0,38	0,20	0,19	0,21	0,25	0,28	0,32	0,35
	0,5	14,74	2,48	0,76	0,27	0,18	0,17	0,18	0,21	0,23	0,26
	0,6	22,62	4,08	1,31	0,40	0,22	0,16	0,15	0,16	0,18	0,20
	0,7	32,19	6,11	2,04	0,62	0,32	0,20	0,16	0,15	0,15	0,17
	0,8	43,47	8,56	2,97	0,92	0,47	0,28	0,20	0,16	0,15	0,15
	0,9	56,45	11,44	4,08	1,30	0,67	0,39	0,26	0,19	0,16	0,15
1,0	71,13	14,74	5,39	1,75	0,92	0,53	0,34	0,24	0,19	0,16	

Table 8b: Values of ξ_{13} for $\alpha=45^\circ$ branch flow dividing

		A ₃ /A ₁									
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0
Q ₃ /Q ₁	0,0	1,10	1,10	1,10	1,00	1,00	1,00	1,00	1,00	1,00	1,00
	0,1	0,60	0,56	0,66	0,66	0,71	0,74	0,77	0,78	0,80	0,81
	0,2	2,08	0,56	0,48	0,47	0,52	0,56	0,59	0,62	0,64	0,66
	0,3	5,12	1,00	0,52	0,40	0,41	0,44	0,46	0,49	0,51	0,54
	0,4	9,30	1,78	0,73	0,43	0,38	0,37	0,38	0,40	0,42	0,44
	0,5	16,09	3,16	1,21	0,54	0,40	0,35	0,34	0,34	0,35	0,37
	0,6	24,24	4,89	1,85	0,69	0,45	0,36	0,32	0,31	0,31	0,31
	0,7	34,09	7,06	2,67	0,95	0,59	0,43	0,35	0,32	0,30	0,30
	0,8	45,63	9,64	3,69	1,30	0,78	0,54	0,41	0,35	0,32	0,31
	0,9	58,88	12,65	4,89	1,73	1,02	0,68	0,50	0,40	0,35	0,32
1,0	73,83	16,09	6,29	2,23	1,30	0,85	0,61	0,48	0,40	0,35	

Table 8c: Values of ξ_{13} for $\alpha=60^\circ$ branch flow dividing

		A ₃ /A ₁									
		0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0
Q ₃ /Q ₁	0,0	1,10	1,10	1,10	1,00	1,00	1,00	1,00	1,00	1,00	1,00
	0,1	1,03	0,77	0,80	0,76	0,79	0,81	0,82	0,83	0,84	0,85
	0,2	2,88	0,96	0,75	0,65	0,66	0,68	0,69	0,71	0,72	0,73
	0,3	6,23	1,56	0,89	0,65	0,61	0,60	0,61	0,62	0,63	0,64
	0,4	10,66	2,46	1,18	0,74	0,62	0,58	0,56	0,56	0,56	0,56
	0,5	17,85	4,04	1,79	0,89	0,68	0,58	0,54	0,52	0,51	0,51
	0,6	26,35	5,95	2,55	1,07	0,76	0,61	0,54	0,50	0,47	0,46
	0,7	36,55	8,29	3,49	1,39	0,94	0,72	0,60	0,53	0,50	0,47
	0,8	48,45	11,05	4,63	1,80	1,18	0,87	0,70	0,60	0,54	0,50

	0,9	62,05	14,24	5,95	2,29	1,46	1,05	0,82	0,68	0,60	0,55
	1,0	77,35	17,85	7,46	2,85	1,80	1,27	0,97	0,79	0,67	0,60

4.16.2 Y-Junction properties

The input properties of both Y-junction are exactly the same. The input and output properties for both types are specified below.

4.16.2.1 Properties Y-junction (combining and dividing type)

Input properties T-junction straight

Description	input	unit	range	default	Remarks
Diameter straight leg	Real	[mm]			
Diameter branch leg	Real	[mm]			
Xi method	Formula Tables			Formula	
Xi tables valid for	Combining Dividing Both				Only when Xi method = Tables
Xi combining straight	Table (ξ_{12})				if Xi tables = Combining or Both
Xi combining branch	Table (ξ_{32})				if Xi tables = Combining or Both
Xi dividing straight	Table (ξ_{12})				if Xi tables = Dividing or Both
Xi dividing branch	Table (ξ_{13})				if Xi tables = Dividing or Both

With the tables for the Xi-values it is possible to choose user-defined loss coefficients, for instance for a certain kind of Y-junction with rounded angles. The user has to specify a set of *loss coefficient values* related to the *discharge ratio* between the branch leg and the straight combined flow leg. If the user does not specify both combining and dividing Xi-tables, WANDA uses the formulas according to the Idelchik handbook for the unspecified flow regimes.

Example of Xi table; straight flow combining, branch leg (ξ_{32})

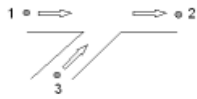
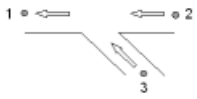
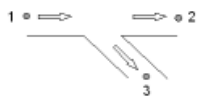
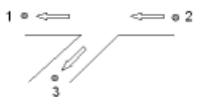
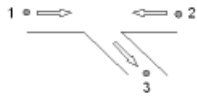
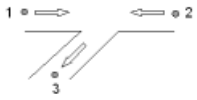
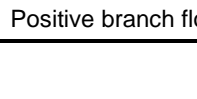
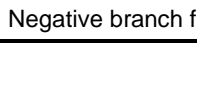
valid for Area ratio = 0.5

Discharge ratio [-]	ξ_i [-]
0.0	0.90
0.2	0.12
0.4	0.39
0.6	0.92
0.8	1.48
1.0	2.08

Component specific output Y-junction straight

Loss coefficient straight [-]	The ξ_{12} for positive flow and ξ_{21} for negative flow In case of branch flow regime this is the ξ_{31} coefficient
Loss coefficient branch [-]	The ξ_{13} for positive flow and ξ_{31} for negative flow In case of branch flow regime this is the ξ_{32} coefficient
Head loss straight [m]	The ΔH_{12} for positive flow and ΔH_{21} for negative flow In case of branch flow regime this is the ΔH_{13} for combining and ΔH_{31} for dividing flow
Head loss branch [m]	The ΔH_{13} for positive flow and ΔH_{31} for negative flow In case of branch flow regime this is the ΔH_{23} for combining and ΔH_{32} for dividing flow

Component messages Y-junction straight

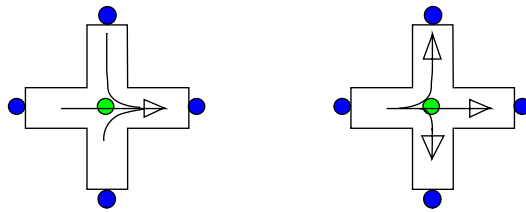
Message	Type	Explanation
Combining	Info	Positive straight flow 
		Negative straight flow 
Dividing	Info	Positive straight flow 
		Negative straight flow 
Branch combining	Info	Positive branch flow 
		Negative branch flow 
Branch dividing	Info	Positive branch flow 
		Negative branch flow 



With the tables for the ξ -values it is possible to choose user-defined loss coefficients, for instance for a certain kind of T-junction with rounded angles. The user has to specify a set of *loss coefficients values* related to the *discharge ratio* between the side and combined branch. If the user does not specify both combining and dividing ξ -tables, WANDA uses the formulas according to the Idelchik handbook for the unspecified flow regimes.

NOTE: The numbering of the branches in the literature is not always consistent with the WANDA definition as given under 'Mathematical Model' above. The Miller Handbook for instance indicates the side branch with 1, the straight branch with 2 and the combined one as 3.

4.17 X-JUNCTION



type label	description	active
X-junction (merging)	Four-node junction with rectangular legs and a merging flow. Both straight legs have the same diameter	no
X-junction (dividing)	Four-node junction with rectangular legs and a dividing flow. Both straight legs have the same diameter	no

Both components supports only a “3-flow” merging and dividing flow in the positive and negative main flow direction (from connect node 1 to 2 or vice versa).

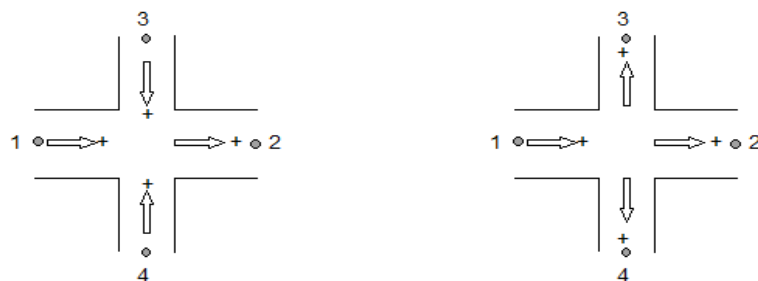
The merging type supports the dividing flow regime and the dividing type supports the merging flow regime

4.17.1 Mathematical model

4.17.1.1 Positive flow definition

The definition for the positive direction of flow (and velocity) is important to understand the results in the property window. Each X-junction type has his own definition. The arrows in the symbol specify the positive direction of the flow (and the velocity).

For the **X-junction (merging)** component the positive flow definition is defined in the left figure below. For the **X-junction (dividing)** component (right figure) the positive flow definition for the branch leg is opposite to that of the Y-junction combining. The positive flow definitions in the main legs (1 to 2) are the same for both types.

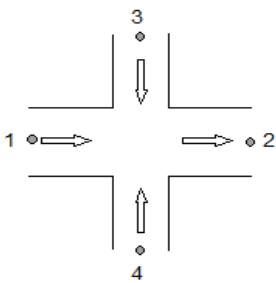
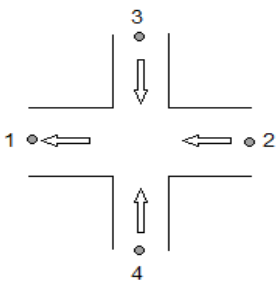
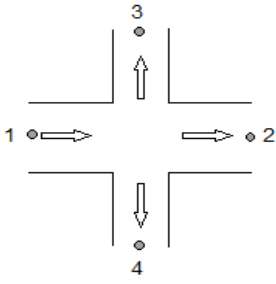
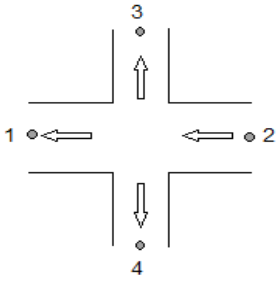


4.17.1.2 Flow regimes

In the table below all supported flow regimes are listed and explained with a scheme with corresponding actual flow arrows.

For all other not supported flow regimes the local losses in the X-junction will be ignored, that means the component is frictionless.

Table 1: definition of supported flow regimes for both X-junction types

<p>1 Combining flow</p>	 <p>Positive flow</p>	 <p>Negative flow</p>
<p>2 Dividing flow</p>	 <p>Positive flow</p>	 <p>Negative flow</p>

4.17.1.3 Equations

The head loss over a 4-node component depends on the distribution of the discharges and the area of the connected legs. The X-junction supports only the resistance coefficient ξ according the head loss functions of Idelchik’s Handbook

Please note that the loss coefficients according to the handbooks apply to junctions with a certain minimum pipe length interval in between multiple junctions.

The head loss in 4-node components is a function of the combined flow in the *combined* leg, which is either one leg (1 or 2) of the straight part. Including the continuity equation (no production or loss of mass in the component) the general set of the four equations of the X-junction is:

X-junction merging	
Mass continuity	$f_1 = Q_1 - Q_2 + Q_3 + Q_4 = 0$
ΔH X-junction merging, combined flow in leg 2	
Head loss between main straight legs	$f_2 = H_1 - H_2 - \xi_{12} \frac{Q_2 Q_2 }{2gA_2^2}$

Head loss between branch leg 3 and straight leg 2	$f_3 = H_3 - H_2 - \xi_{32} \frac{Q_2 Q_2 }{2gA_2^2}$
Head loss between branch leg 4 and straight leg 2	$f_4 = H_4 - H_2 - \xi_{42} \frac{Q_2 Q_2 }{2gA_2^2}$
X-junction dividing	
Mass continuity	$f_1 = Q_1 - Q_2 - Q_3 - Q_4 = 0$
ΔH X-junction dividing, combined flow in leg 1	
Head loss between main straight legs	$f_2 = H_1 - H_2 - \xi_{12} \frac{Q_1 Q_1 }{2gA_1^2}$
Head loss between branch leg 3 and straight leg 2	$f_3 = H_1 - H_3 - \xi_{13} \frac{Q_1 Q_1 }{2gA_1^2}$
Head loss between branch leg 4 and straight leg 2	$f_4 = H_1 - H_4 - \xi_{14} \frac{Q_1 Q_1 }{2gA_1^2}$

Where:	Q_i = total discharge in leg i	[m ³ /s]
	H_i = energy head in connect point i	[m]
	A_i = pipe area leg i (leg with total flow)	[m ²]
	ξ_{ij} = loss coefficient between point i and j	[-]

The subscripts (1), (2), (3) and (4) correspond to the different legs. The subscript x at the discharge Q_x refers always to the leg in which the combined (total) flow Q occurs.

For positive dividing flow and negative merging flow, Q_1 will be taken into account. For positive merging flow and negative dividing flow, Q_2 will be taken into account.

4.17.1.4 Resistance coefficients based on formulas

The resistance coefficients based of formulas are taken from the Idelchik Handbook with the following restriction to the section areas:

$$A_1 > A_3$$

$$A_1 = A_2$$

$$A_3 = A_4$$

The formulas are only valid for the supported flow regimes as explained in table 1. For each formula the ξ -values for various area and discharge ratios are collected in a table. Note that all indices used in the formulas are based on positive flow.

For the other flow regimes the resistance coefficient can not be retrieved and a $\xi=0$ will be applied .

X-junction merging

For the **straight** leg, the resistance coefficient ξ_{12} is calculated according to:

$$\xi_{12} = 1.2 + \left(\frac{Q_1}{Q_2}\right)^2 - \left(\frac{Q_1}{Q_2}\right)^2 \frac{1 + Q_1/Q_2}{(0.75 + 0.25 Q_1/Q_2)^2}$$

Table 2 shows the ξ_{12} values according the formula above for several discharge ratios

Table 2: ξ_{st} values for the combining case.

Q ₁ /Q ₂	0,0	0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1
ξ_{12}	1,20	1,19	1,17	1,12	1,05	0,96	0,85	0,72	0,56	0,39	0,20

For the both **branch** legs, the resistance coefficient is calculated according to:

$$\xi_i = 1 + \left(\frac{Q_i}{Q_2} \frac{A_2}{A_i}\right)^2 - 8 \left(\frac{Q_i}{Q_2}\right)^2 \frac{[Q_2/Q_i - (1 + Q_j/Q_i)]^2}{4 - (1 + Q_j/Q_i) Q_i/Q_2}$$

The subscript *i* is for the interested branch leg, while *j* is for the other branch leg.

Table 3 shows some ξ_{i2} values according the formula above for several discharge ratios and area ratios

Table 3: ξ_{i2} values for the X-junction merging.

Q _i /Q ₂	Q _i /Q ₂						
	0,00	0,1	0,2	0,3	0,4	0,5	0,6
	A_i/A₂= 0.2						
0,5	-0,999	-0,25	0,94	2,57	4,62	7,10	9,97
1	-0,999	-0,097	1,200	2,874	4,900	7,250	-
2	-0,999	0,191	1,624	3,224	-	-	-
	A_i/A₂= 0.4						
0,5	-0,999	-0,439	0,191	0,881	1,624	2,409	3,224
1	-0,999	-0,285	0,450	1,186	1,900	2,563	-
2	-0,999	0,003	0,874	1,537	-	-	-
	A_i/A₂= 0.6						
0,5	-0,999	-0,474	0,051	0,566	1,064	1,535	1,966
1	-0,999	-0,320	0,310	0,872	1,341	1,689	-
2	-0,999	-0,032	0,734	1,222	-	-	-
	A_i/A₂= 0.8						
0,5	-0,999	-0,486	0,003	0,459	0,874	1,237	1,537
1	-0,999	-0,332	0,263	0,764	1,150	1,391	-
2	-0,999	-0,044	0,686	1,115	-	-	-
	A_i/A₂= 1						
0,5	-0,999	-0,491	-0,019	0,408	0,784	1,096	1,334
1	-0,999	-0,337	0,240	0,714	1,060	1,250	-
2	-0,999	-0,049	0,664	1,064	-	-	-

X-junction dividing

For the straight passage the resistance coefficient is:

$$\xi_{12} = \tau_1 \frac{Q_{br}}{Q_1}$$

where Q_{br} is the mean discharge in the branches (i.e. $0.5 \cdot (Q_3 + Q_4)$) and τ_1 depends on the area and discharge ratios as given in table 5:

Table 5: Values of τ_1 .

A_i/A_1	≤ 0.4	> 0.4	
Q_{br}/Q_1	0 - 1.0	≤ 0.5	> 0.5
τ_1	$0.4 (Q_{br}/Q_1)$	$0.2(2Q_{br}/Q_1 - 1)$	$0.3(2Q_{br}/Q_1 - 1)$

For the branch legs there are two cases, depending on area ratio:

1. For $A_i/A_1 \leq 2/3$

$$\xi_{1i} = A' \left[1 + \left(\frac{Q_i A_1}{Q_1 A_i} \right)^2 \right]$$

2. For $A_i/A_1 = 1$ (up to $v_i/v_1 \approx 2.0$)

$$\xi_{1i} = A' \left[1 + 0.3 \left(\frac{Q_i A_1}{Q_1 A_i} \right)^2 \right]$$

where the values of A' are taken from table 6.

Table 6: Values for A' .

A_i/A_1	$\leq 0,35$		$> 0,35$	
Q_i/Q_1	$\leq 0,4$	$> 0,4$	$\leq 0,6$	$> 0,6$
A'	$1,1 - 0,7 \frac{Q_i}{Q_1}$	0.85	$1,0 - 0,65 \frac{Q_i}{Q_1}$	0.6

As an example some values for the resistance coefficients ξ_{12} and ξ_{1i} are given in tabel 7 and 8, depending on the section area and discharge ratios.

Table 7: ξ_{12} values for diverging case.

		ξ_{12}		
		$A_i/A_2 \leq 0.4$	$A_1/A_2 > 0.4$	
Q_2/Q_1	0	0,000	0,000	-
	0,1	0,004	-0,016	-
	0,2	0,016	-0,024	-
	0,3	0,036	-0,024	-
	0,4	0,064	-0,016	-
	0,5	0,100	0,000	-
	0,6	0,144	-	0,036
	0,7	0,196	-	0,084
	0,8	0,256	-	0,144
	0,9	0,324	-	0,216
	1	0,400	-	0,3

Table 8: ξ_s values for diverging case.

v_i/v_1	$A_i/A_1 \leq 2/3$	$A_i/A_1 = 1$
0,0	1,10	1,00
0,1	1,10	1,00
0,2	1,10	0,88
0,4	1,17	0,77
0,6	1,32	0,68
0,8	1,52	0,71
1,0	1,78	0,78
1,2	2,07	0,85
1,4	2,51	0,95
1,6	3,02	1,06
2,0	4,25	1,32
2,6	6,59	-
3,0	8,50	-
4,0	14,45	-
5,0	22,10	-
6,0	31,45	-
8,0	55,25	-
10,0	85,85	-

4.17.2 X-Junction properties

The input and output properties of both X-junction are exactly the same.

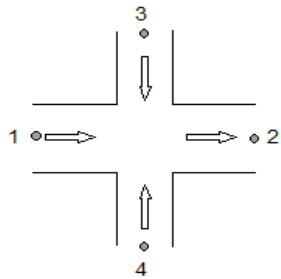
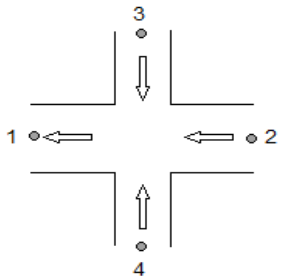
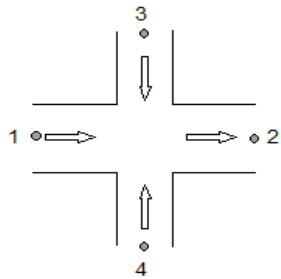
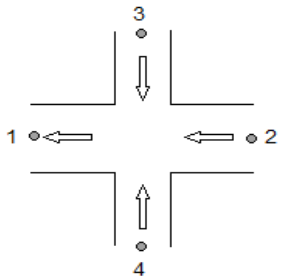
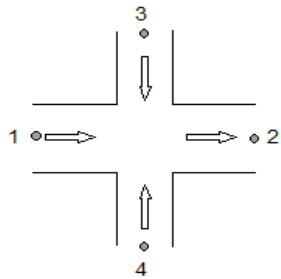
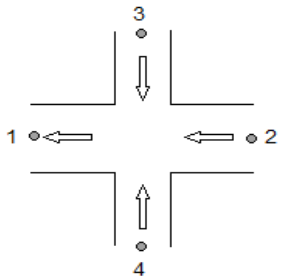
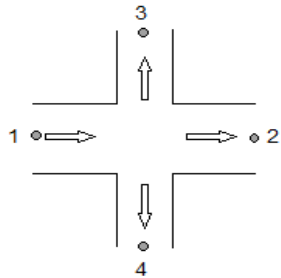
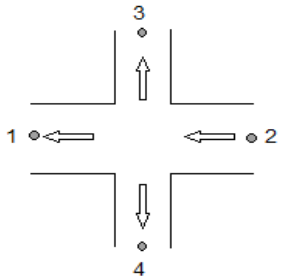
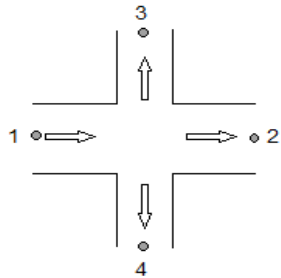
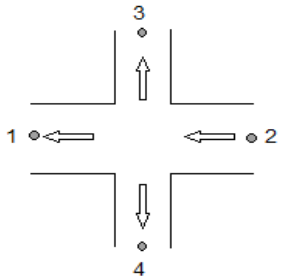
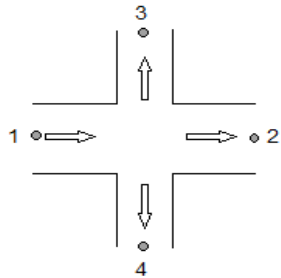
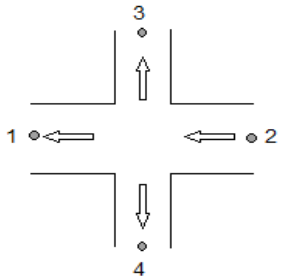
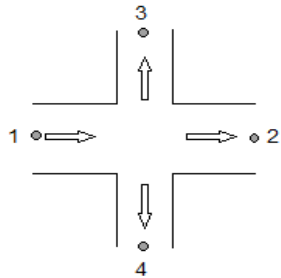
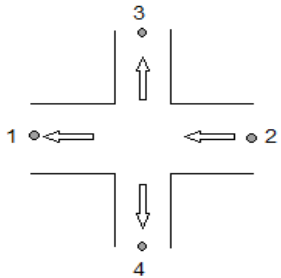
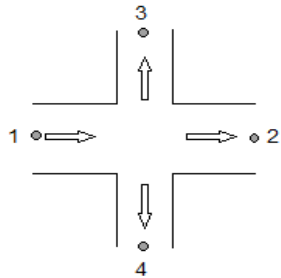
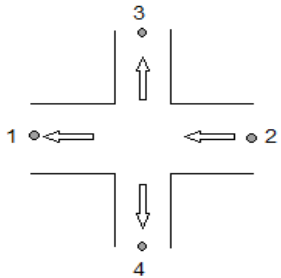
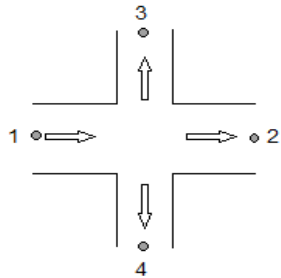
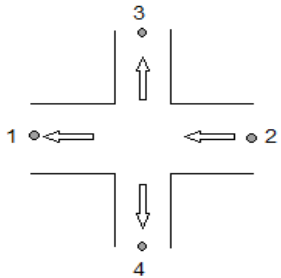
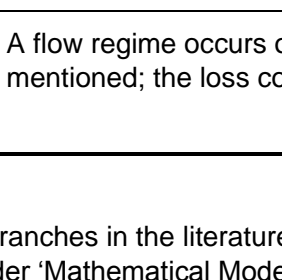
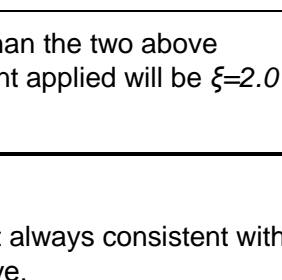
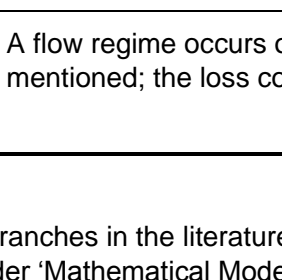
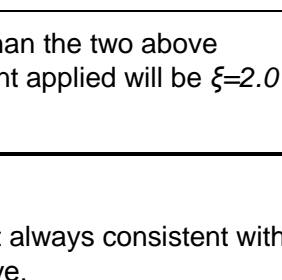
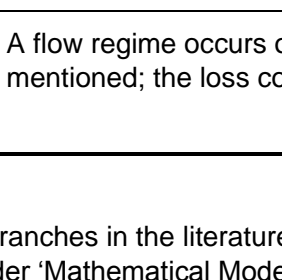
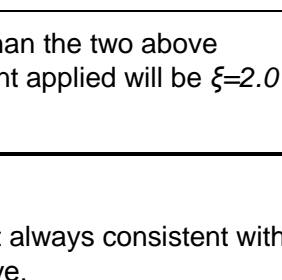
4.17.2.1 Properties X-junction (merging and dividing type)

Description	input	unit	range	default	Remarks
Diameter straight leg	Real	[mm]			D leg 1 = D leg 2
Diameter branch leg	Real	[mm]			D leg 3 = D leg 4

Component specific output X-junction (merging and dividing)

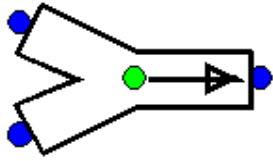
Loss coefficient straight [-]	The ξ_{12} for positive flow and ξ_{21} for negative flow
Loss coefficient branch 1 [-]	The ξ_{13} for positive flow and ξ_{31} for negative flow
Loss coefficient branch 2 [-]	The ξ_{14} for positive flow and ξ_{41} for negative flow
Head loss straight [m]	The ΔH_{12} for positive flow and ΔH_{21} for negative flow
Head loss branch 1 [m]	The ΔH_{13} for positive flow and ΔH_{31} for negative flow
Head loss branch 2 [m]	The ΔH_{14} for positive flow and ΔH_{41} for negative flow

Component messages Y-junction (combining and dividing)

Message	Type	Explanation				
Merging	Info	<table border="0"> <tr> <td style="text-align: center;">Positive straight flow</td> <td style="text-align: center;">Negative straight flow</td> </tr> <tr> <td style="text-align: center;">  </td> <td style="text-align: center;">  </td> </tr> </table>	Positive straight flow	Negative straight flow		
		Positive straight flow	Negative straight flow			
						
<table border="0"> <tr> <td style="text-align: center;">Positive straight flow</td> <td style="text-align: center;">Negative straight flow</td> </tr> <tr> <td style="text-align: center;">  </td> <td style="text-align: center;">  </td> </tr> </table>	Positive straight flow	Negative straight flow				
Positive straight flow	Negative straight flow					
						
Dividing	Info	<table border="0"> <tr> <td style="text-align: center;">Positive straight flow</td> <td style="text-align: center;">Negative straight flow</td> </tr> <tr> <td style="text-align: center;">  </td> <td style="text-align: center;">  </td> </tr> </table>	Positive straight flow	Negative straight flow		
		Positive straight flow	Negative straight flow			
						
<table border="0"> <tr> <td style="text-align: center;">Positive straight flow</td> <td style="text-align: center;">Negative straight flow</td> </tr> <tr> <td style="text-align: center;">  </td> <td style="text-align: center;">  </td> </tr> </table>	Positive straight flow	Negative straight flow				
Positive straight flow	Negative straight flow					
						
Not supported flow regime	Info	A flow regime occurs other than the two above mentioned; the loss coefficient applied will be $\xi=2.0$				

NOTE: The numbering of the branches in the literature is not always consistent with the WANDA definition as given under 'Mathematical Model' above.

4.18 Merger



type label	description	active
2-merger	Three-node junction which merges 2 separate branch flows into a combined flow. Cross section can be circular or rectangular.	no

The merger differs from the Y-junctions because there is no straight main flow. The area of the three legs may be different.

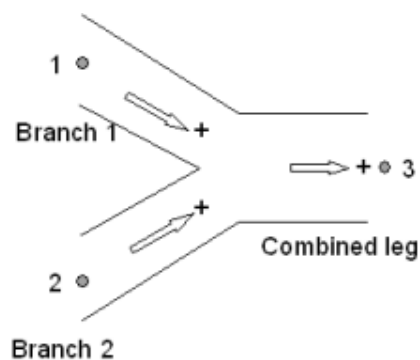
Instead of negative flow the Merger component acts as a Splitter component.

4.18.1 Mathematical model

4.18.1.1 Positive flow definition

The definition for the positive direction of flow (and velocity) together with the location of the connect point numbers is important to understand the results in the property window. Notice that the connect point numbering of the Merger differs from that of the Splitter.

For the Merger-**junction** the positive flow definition and connect point numbers are defined in the figure below.

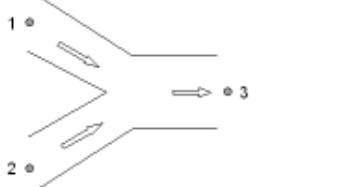
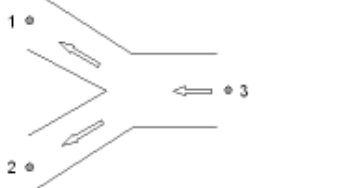
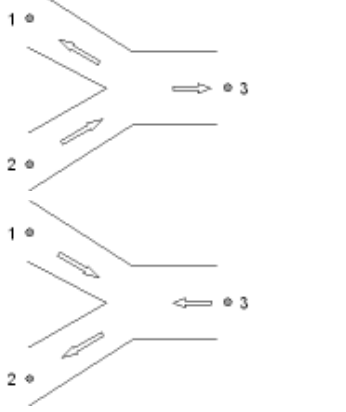


4.18.1.2 Flow regimes

The Merger supports only the positive merging flow and for negative flow the splitting variant. The other flow regimes are not supported and for these circumstances the local losses are ignored, that means the Merger becomes frictionless.

In the table below the different flow regimes are explained with a scheme with corresponding actual flow arrows.

Table 1: definition of flow regimes for both types

<p>1 Merging flow</p>	 <p>Positive flow</p>
<p>2 Splitting flow</p>	 <p>Negative flow</p>
<p>Not supported flow regimes</p>	

4.18.1.3 Equations

The head loss over a 3-node component depends on the distribution of the discharges and the area of the connected legs. The Merger supports only the user specified resistance coefficient ξ (ξ) depending on discharge ratio's for a fixed area ratio.

The head loss in 3-node components is a function of the combined flow in the combined leg. Including the continuity equation (no production or loss of mass in the component) the general set of the three equations of the Y-junction is:

Mass continuity	$f_1 = Q_1 + Q_2 - Q_3 = 0$
Head loss between branch 1 and combined leg	$f_2 = H_1 - H_3 - \xi_{12} \frac{Q_3 Q_3 }{2gA_3^2}$
Head loss between branch 2 and combined leg	$f_3 = H_2 - H_3 - \xi_{23} \frac{Q_3 Q_3 }{2gA_3^2}$

Where:

Q_i = total discharge in leg i	[m ³ /s]
H_i = energy head in connect point i	[m]
A_i = pipe area leg i (leg with total flow)	[m ²]
ξ_{ij} = loss coefficient between point i and j	[-]

The subscripts (1), (2) and (3) correspond to the different legs.

For negative splitting flow the formulas of the Splitter component will be used with Q_1 taken into account instead of Q_3 .

For the not supported flow regimes the head loss equation becomes $H_1 - H_3 = 0$ and $H_2 - H_3 = 0$

4.18.2 Merger properties

The input and output properties for both types are specified below.

Input properties

Description	input	unit	range	default	Remarks
Cross section	Circular Rectangular			Circular	
Diameter combined leg	Real	[mm]			If Cross section = Circular
Diameter merge branch 1	Real	[mm]			If Cross section = Circular
Diameter merge branch 2	Real	[mm]			If Cross section = Circular

Width combined leg	Real	[mm]	If Cross section = Rectangular
Height combined leg	Real	[mm]	If Cross section = Rectangular
Width merge branch 1	Real	[mm]	If Cross section = Rectangular
Height merge branch 1	Real	[mm]	If Cross section = Rectangular
Width merge branch 2	Real	[mm]	If Cross section = Rectangular
Height merge branch 2	Real	[mm]	If Cross section = Rectangular
Xi tables valid for	Merging Both		
Xi merging branch 1	Table (ξ_{13})		if Xi tables = Splitting or Both
Xi merging branch 2	Table (ξ_{23})		if Xi tables = Splitting or Both
Xi splitting branch 1	Table (ξ_{31})		if Xi tables = Both
Xi splitting branch 2	Table (ξ_{32})		if Xi tables = Both

Example of Xi table; Merging branch 1 (ξ_{13})

valid for Area ratio = 0.5

Discharge ratio [-]	Xi [-]
0.0	0.90
0.2	0.12
0.4	0.39
0.6	0.92
0.8	1.48
1.0	2.08

Component specific output properties

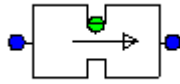
Loss coefficient branch 1 [-]	The ξ_{13} for positive flow and ξ_{31} for negative flow
Loss coefficient branch 2 [-]	The ξ_{23} for positive flow and ξ_{32} for negative flow
Head loss branch 1 [m]	The ΔH_{13} for positive flow and ΔH_{31} for negative flow
Head loss branch 2 [m]	The ΔH_{23} for positive flow and ΔH_{32} for negative flow

Component messages

Message	Type	Explanation
Merging flow	Info	
Splitting flow	Info	
Not supported flow regime	Info	

4.19 ORIFICE

4.19.1 ORIFICE (class)



Orifice

Fall type

type label	Description	active
Orifice	Orifice plate complying with ISO 5167-2:2003	No

4.19.1.1 Mathematical model

The resistance in a weakly compressible liquid of an orifice plate that complies with the international standard ISO 5167-2:2003 is described by the following equations:

$$Q = \frac{C}{\sqrt{1-\beta^4}} \frac{\pi}{4} d^2 \sqrt{\frac{2 \cdot \Delta p}{\rho}} = \frac{C}{\sqrt{1-\beta^4}} \frac{\pi}{4} \beta^2 D^2 \sqrt{2g \cdot (H_1 - H_{vc})} \quad (1)$$

$$H_1 - H_2 = \frac{\sqrt{1-\beta^4(1-C^2)} - C\beta^2}{\sqrt{1-\beta^4(1-C^2)} + C\beta^2} (H_1 - H_{vc}) \quad (2)$$

in which:

H_1	=	Upstream head	[m]
H_2	=	Downstream head	[m]
H_{vc}	=	Head in the vena contracta	[m]
P_{vc}	=	Pressure in the vena contracta	[Pa]
z_i	=	Elevation of connecting up- (1) and downstream (2) pipe	[m]
Q	=	discharge through orifice	[m ³ /s]
C	=	Discharge coefficient	[-]
d	=	Throat diameter	[m]
D	=	Pipe diameter	[m]
β	=	Diameter ratio d / D	[-]

Equation (2) represents the pressure recovery by expansion of the jet downstream of the orifice plate. Pressure recovery is completed approximately 6D downstream of the orifice plate.

Equations (1) and (2) can be rewritten to a general head loss equation:

$$H_1 - H_2 = \xi \frac{v|v|}{2g} = \xi \frac{Q|Q|}{2gA^2} \quad (3)$$

$$\xi = \left(\frac{\sqrt{1 - \beta^4(1 - C^2)}}{C \cdot \beta^2} - 1 \right)^2 \quad (4)$$

Where ξ , denoted as K in ISO 5167, is the dimensionless head loss coefficient.

ISO 5167-2:2003 proposes the Reader-Harris/Gallagher (RHG) equation to model the discharge coefficient, C, as a function of the location of the pressure tapings. The RHG-equation is:

$$C = 0.5961 + 0.0261\beta^2 - 0.216\beta^8 + 5.21 \cdot 10^{-4} \left(\frac{10^6 \beta}{\text{Re}_D} \right)^{0.7} + \quad (5)$$

$$(0.0188 + 0.0063A) \beta^{3.5} \left(\frac{10^6}{\text{Re}_D} \right)^{0.3} +$$

$$(0.043 + 0.08e^{-10L_1} - 0.123e^{-7L_1})(1 - 0.11A) \frac{\beta^4}{1 - \beta^4} -$$

$$0.031(M_2 - 0.8M_2^{1.1}) \beta^{1.3}$$

Where $D < 71.12$ mm (2.8 in.), the following term shall be added to equation (5):

$$+0.011(0.75 - \beta) \left(2.8 - \frac{D}{0.0254} \right)$$

In these equations

Re_D	=	Reynolds number calculated with respect to D	[-]
L_1	=	quotient of distance of upstream tapping to the orifice and the pipe diameter; l_1 / D	[-]
L_2	=	quotient of distance of downstream tapping to the orifice and the pipe diameter; l_2 / D	[-]
M_2	=	$\frac{2L_2}{1 - \beta}$	[-]
A	=	$\left(\frac{19000\beta}{\text{Re}_D} \right)^{0.8}$	[-]

The model assumes that the vena contracta occurs $D/2$ downstream of the orifice plate, such that the RHG equation for D and $D/2$ tapings is applicable; hence $L_1 = 1$ and $L_2 = 0.47$.

The location of the vena contracta determines whether local cavitation and choking flow will occur. If local cavitation occurs the above equations are not valid anymore. WANDA

will generate a warning. The velocity in the vena contracta is approximated, based on Bernoulli, by:

$$\left. \begin{aligned} P_{vc} &= (H_{vc} - z_2) \rho g \\ v_{vc} &= \sqrt{2g(H_1 - H_{vc}) + v_D^2} \end{aligned} \right\} , Q > 0 \quad (6)$$

$$\left. \begin{aligned} P_{vc} &= (H_{vc} - z_1) \rho g \\ v_{vc} &= \sqrt{2g(H_2 - H_{vc}) + v_D^2} \end{aligned} \right\} , Q \leq 0 \quad (7)$$

If the flow reverses, the same loss coefficient is applied, which is physically correct if the orifice plate is not bevelled. If the orifice plate is bevelled, the numerical result for negative flow rates are unreliable.

Cavitation depends on the pressure conditions around the orifice. Usually these pressure conditions are defined by a pressure ratio, for which various specific numbers are used.

In WANDA the factor X_f is used, according the German VDMA standard.

$$X_f = \frac{\Delta p}{P_1 - P_v}$$

in which:

X_f	=	pressure ratio	[-]
Δp	=	net pressure difference across the valve	[N/m ²]
p_1	=	absolute pressure upstream of the valve	[N/m ²]
p_v	=	vapour pressure of the fluid	[N/m ²]

The pressure ratio X_f is only calculated for positive flow; for negative flow $X_f = 0$.

4.19.2 Orifice plate

4.19.2.1 Hydraulic specifications

Description	input	unit	range	default	remarks
Pipe diameter	real	[m]	(0-5]		
Initial setting	throat d. / discharge			throat d.	if the user specifies the initial discharge, then the required throat diameter is calculated
Throat diameter	real	[m]	(0-4]		if initial setting = throat d.
Initial discharge	real	[m ³ /s]	(0-100]		if initial setting = discharge

See also "Mathematical model" on page 326.

4.19.2.2 Component specific output

Pressure vena contracta	[Pa]
Head vena contracta	[m]
Velocity vena contracta	[m/s]
Loss coefficient (ξ)	[-]
Pressure ratio X_f	[-]
Throat diameter	[m]
Beta (d/D)	[-]

4.19.2.3 H-actions

None

4.19.2.4 Component messages

Message	Type	Explanation
Throat diameter is out of scope of standard	Warning	Standard requires $d \geq 12.5$ mm. Same equations are applied but uncertainty of standard is not guaranteed.
Pipe diameter is smaller than standard scope	Warning	Standard requires $D \geq 50$ mm. Same equations are applied but uncertainty of standard is not guaranteed.
Pipe diameter is greater than standard scope	Warning	Standard requires $D \leq 1000$ mm. Same equations are applied but uncertainty of standard is not guaranteed.
Diameter ratio is smaller than standard scope	Warning	Standard requires $\beta \geq 0.1$. Same equations are applied but uncertainty of standard is not guaranteed.
Diameter ratio is greater than standard scope	Warning	Standard requires $\beta \leq 0.75$. Same equations are applied but uncertainty of standard is not guaranteed.
Reynolds number drops out of scope	Warning	The actual Reynolds number has dropped below the minimum Re-number in the standard. The applicable Re-number is fixed to the minimum Re-number. The uncertainty may increase.
Reynolds number returns into scope	Info	The actual Reynolds number returns into the scope of the standard.
Degassing may occur	Warning	This message is displayed if the pressure in the vena contracta drops to vapour pressure + 0.5 bar
Vapour pressure reached. Choking flow	Warning	Vapour pressure reached in the vena contracta. Choking flow is not supported at this stage. Results become unreliable.
Reverse flow	Warning	If orifice is bevelled, then results are unreliable.

4.20 PIPE



Fall type

type label	description	active
Pipe	Pressurised pipe	No

As from the WANDA 4.0 release there is one pipe-model which includes all former pipe models. Although there are still two different symbols, the functionality is exactly the same.

4.20.1 Pressurised pipe

4.20.1.1 Mathematical model

The PIPE can be considered the primary component in WANDA. The PIPE is used to simulate the actual pressure wave propagation during transient simulations.

In steady state (Engineering Mode) $dQ/dt = 0$ and the PIPE is considered as a fall type, modelled only by the following equation:

$$H_1 - H_2 = \frac{fLQ^2}{2gDA^2} \quad (1)$$

in which:

$H_1 - H_2$	=	head loss from H-node 1 to H-node 2	[m]
Q	=	discharge through the pipe	[m ³ /s]
f	=	friction factor	[-]
D	=	inner diameter	[m]
A	=	cross sectional area	[m ²]

WANDA calculates the friction factor iteratively using the wall roughness or it can be directly specified by the user. Several friction models are included in WANDA, see "Friction models" on page 166 for details. Please note that the friction factor is only recalculated during the calculation if the Dynamic Friction is set to 'Quasi-Steady'. With other friction models the friction factor is kept constant. For more details see "Wall roughness k-pipes" on page 168.

In Transient Mode the pipe model depends on the input property "Calculation mode". If it is set to "waterhammer", the PIPE is handled as a waterhammer pipe and the full waterhammer equations are solved. The wave propagation speed is calculated based on pipe and fluid properties. The user has also the option to specify the wave

propagation speed according to his own needs. The waterhammer equations are given below, details about the wave propagation and the equations involved are discussed in Theory on waterhammer pipes in Chapter "General definitions" on page 149.

Momentum:

$$\frac{L}{A_f} \frac{\partial Q}{\partial t} + g \frac{\partial H}{\partial s} + \frac{f Q|Q|}{2D A_f^2} = 0 \quad (2)$$

Continuity:

$$\frac{L}{c_f^2} \frac{\partial H}{\partial t} + \frac{1}{g A_f} \frac{\partial Q}{\partial s} = 0 \quad (3)$$

If the "calculation mode" is set to "rigid column", the PIPE is considered as a pipe with infinite propagation speed. In a relatively short pipe the propagation speed can be regarded as infinite. Relatively short means smaller than the element length, ds , in which the compressible pipes are divided, such that the fluid velocity may be assumed constant ($dv/ds = 0$). In other words, the fluid in the pipe is regarded as incompressible. Hence, only the momentum equation is solved:

$$H_1 - H_2 = \frac{L}{gA} \frac{dQ}{dt} + \frac{fL}{8RgA^2} Q|Q| \quad (4)$$

in which:

H_1	=	upstream head	[m]
H_2	=	downstream head	[m]
Q	=	discharge	[m ³ /s]
L	=	pipe length	[m]
D	=	Inner pipe diameter	[m]
A	=	cross sectional area	[m ²]
R	=	hydraulic radius (A / P)	[m]
f	=	friction factor	[-]

4.20.2 Pressurised pipe Properties

4.20.2.1 Cross section

The cross section of the pipe can be circular or rectangular. For rectangular cross sections, it is possible to define the size of the fillets (filling of the corners, see figure 1) and their structural contribution. A larger fillet and/or more structural contribution will increase the stiffness of the cross section, resulting in higher wave speed. A structural contribution of 0% means that the fillets have no effect on the wave speed, whereas 100% gives full influence according to the theory as explained in section "Wave propagation speed" on page 173.

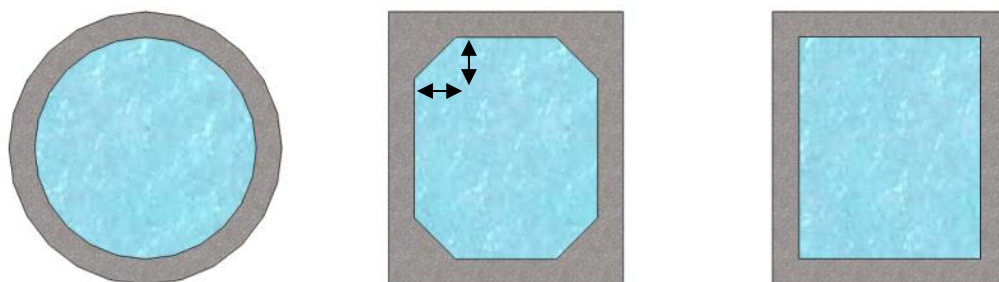


Figure 1: Circular and rectangular cross section (with and without fillets).

4.20.2.2 Additional losses

If several local losses (like bends, Tee's etc) exists in a pipe section, you may model them separately (pipe-resist-pipe-resist) or combine them in the pipe model. Combining them can be done in 2 different ways: using the ξ -annotation (single value or table) or the equivalent length annotation. For example:

for an elbow ξ_{elbow} is about 0.2

or

for an elbow L_{eq} is about 15 D

WANDA calculates the total resistance (friction and local losses) in the steady state and derives a combined friction factor that will be used during the unsteady state calculation. In the tables of the local losses the diameter of the local resistance may differ from the pipe diameter, but the combined friction factor will be related to the pipe diameter only. If the diameter of local loss differs from the main pipe, it can be specified in the column "Diameter". If this column remains blank, the pipe diameter will be used.

	local losses	Loss coef Xi (-)	Number	Diameter (mm)	Comment
1	Elbow 90 degr; R=1.5D	0.2000			
2	Elbow 45 degr; R=1.5D	0.1200			
3	Ball valve 100% open	0.2500			
4	Diffusor 1:4	0.1000			
5	Tee; flow through run	0.3000			
6	Tee; flow through branch	0.9000			

4.20.2.3 Geometry

Each PIPE must have a length input and height location. This so-called profile can be defined in several ways:

1. Scalar value for length
2. Length-height profile
3. Isometric layout specified with absolute XYZ co-ordinates
4. Isometric layout specified with differential XYZ co-ordinates.

Each type of profile input is translated to a longitudinal profile (X-H profile). In case of a scalar value the height of the beginning and the end of the pipe is derived directly from the adjacent H-nodes. The input height is the height of the centreline of the pipe, as indicated in figure 2 below.

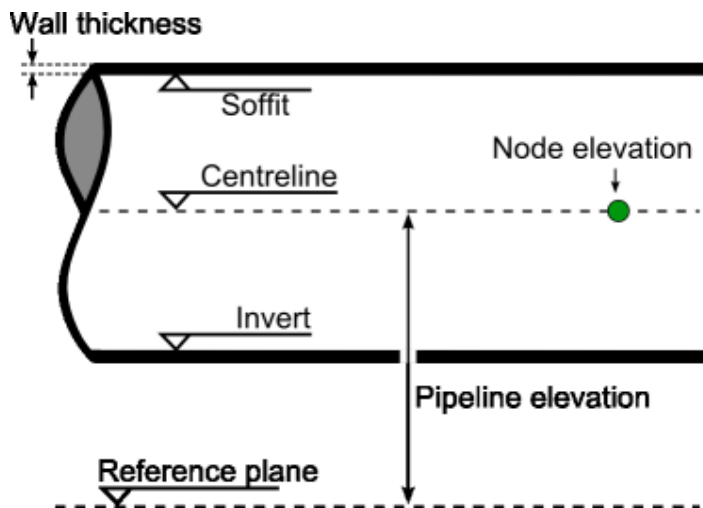


Figure 2: Definition of the height of the pipe and the node elevation.

WANDA compares the input of the H-node height with the beginning and end of the Pipe. If there is a difference of more than half the pipe diameter, a warning is displayed in the property field "H-node height check". Otherwise, "OK" is displayed.

Below an example of differential XYZ input based on the given isometric is given.

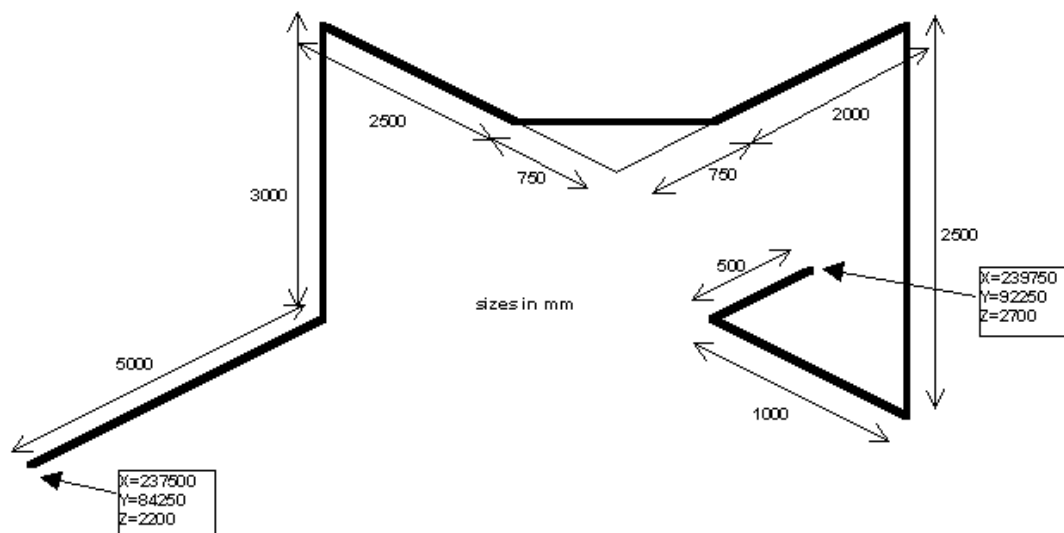


Figure 3: PIPE isometric with differential input.

	x-distance (m)	height (m)	X-abs (m)	Y-abs (m)	Z-abs (m)	X-diff (m)	Y-diff (m)	Z-diff (m)
1	0.0000	2.20000	237.5000	84.25000	2.200000	237.5000	84.25000	2.200000
2	5.0000	2.20000	237.5000	89.25000	2.200000	0.000000	5.000000	0.000000
3	5.0000	5.20000	237.5000	89.25000	5.200000	0.000000	0.000000	3.000000
4	7.5000	5.20000	240.0000	89.25000	5.200000	2.500000	0.000000	0.000000
5	8.5607	5.20000	240.7500	90.00000	5.200000	0.7500000	0.7500000	0.000000
6	10.561	5.20000	240.7500	92.00000	5.200000	0.000000	2.000000	0.000000
7	10.561	2.70000	240.7500	92.00000	2.700000	0.000000	0.000000	-2.500000
8	11.561	2.70000	239.7500	92.00000	2.700000	-1.000000	0.000000	0.000000
9	12.061	2.70000	239.7500	92.50000	2.700000	0.000000	0.5000000	0.000000

Please note the first row starts with the absolute XYZ coordinates followed by the differential coordinates.

The input is translated to real XYZ values and subsequently to a x-H profile. This x-H profile is shown in the location charts.

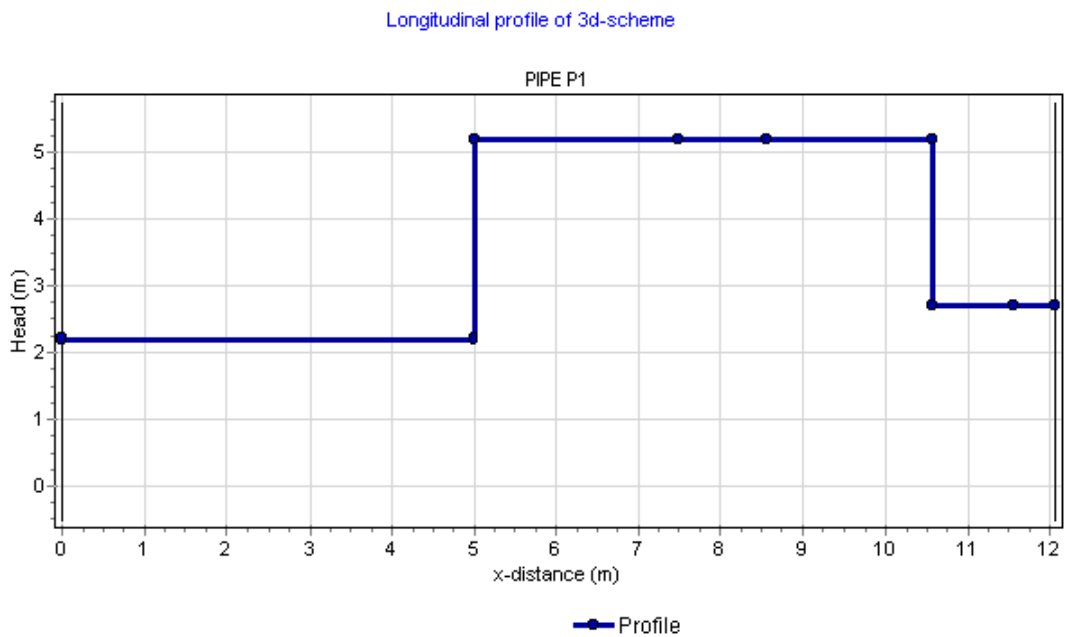


Figure 4: result of XYZ diff specified profile translated to x-H profile

It is not necessary that the pipe profile start with zero. Any positive value is possible. If you split a pipe into two parts (e.g. a VENT must be connected to the pipe) you only have to copy the pipe and delete the unnecessary rows.

	x-distance (m)	height (m)
1	31090	132.910
2	31670	131.510
3	32270	131.010
4	32850	129.810
5	33400	127.210

The location chart (route) starts with the value specified in the first row of the table.

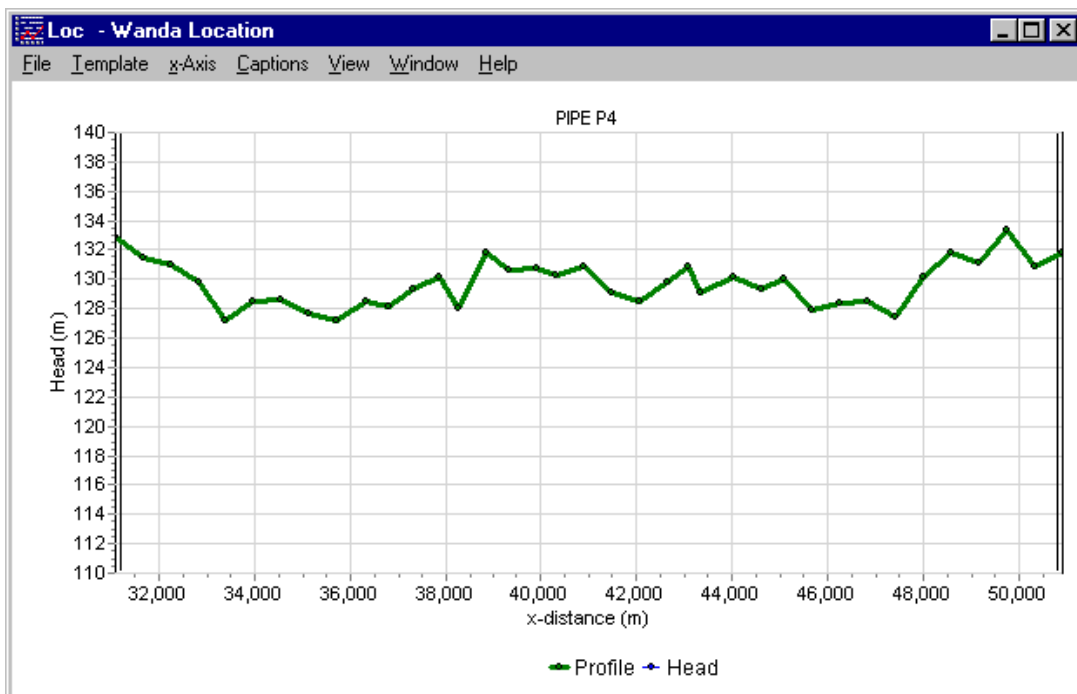


Figure 5: Pipe profile starting with a non-zero begin location

4.20.2.4 Hydraulic specifications

description	input	unit	default	remarks
Cross section	Circle Rectangle	[-]	Circle	
Inner diameter	real	[m]		if Cross section=Circle
Inner width	real	[m]		if Cross section=Rectangle
Inner height	real	[m]		if Cross section=Rectangle
Fillet size	real	[m]	0	if Cross section=Rectangle
Fillet structural contribution	real	[-]	0,67	if Cross section=Rectangle
Calculation mode	Waterhammer Rigid column		Waterhammer	Only in transient mode

description	input	unit	default	remarks
Wave speed mode	Physical specified		Physical	if Calculation mode = Waterhammer
Wall thickness	real	[m]		if Wave speed mode = physical
Young's modulus	real	[N/m ²]		if Wave speed mode = physical
Specified wave speed	real	[m/s]		if Wave speed mode = specified
Friction model	D-W k D-W f H-W C-M	[]	D-W k	See "Friction models" on page 166
Wall roughness	real	[mm]		if Friction model = D-W k
Dynamic friction	Quasi-steady none			Only in transient mode and if Friction model = D- W k
Friction factor	real	[-]		if Friction model = D-W f
C coefficient	real	[-]		if Friction model = H-W
n coefficient	real	[-]		if Friction model = C-M
Additional losses	None Xi Xi-pos Xi-neg Xi table L_eq table	[-]	None	
Local losses coeff	real	[-]		if Additional losses = Xi
Local losses pos. dir.	real	[-]		if Additional losses = Xi-pos Xi-neg
Local losses neg. dir	real	[-]		if Additional losses = Xi-pos Xi-neg
Local losses Xi	table			if Additional losses = Xi table
Local losses L_eq	table			if Additional losses = L_eq table
Geometry input	Length l-h xyz xyz diff	[-]		
Length	real	[m]		if Geometry input = Length
Profile	table			if Geometry input = L-h, xyz or xyz diff
Upper limit pressure	real	[N/m ²]		Only visible if checked in "Mode&options" window
Lower limit pressure	real	[N/m ²]		Only visible if checked in "Mode&options" window

description	input	unit	default	remarks
Location	real	[m]		If empty, chart button creates a location serie If filled in, chart button creates a time serie for nearest internal node

See also help on using the "Property Window" on page 43.

Remarks

The tables with local losses (Xi and L_eq tables) are derived from the tables as specified in the "Initial values" window (see menu/model). A local copy of this table is made and may be modified for this particular pipe. If you want to use other global settings for this case, first edit the "Initial values" tables before adding PIPE's to the diagram.

The location charts of "Pressure" and "Head" shows the specified limits only if they are filled in. It is possible to specify these values after the simulation. If these fields are not visible, first check the "Upper/Lower limits in pipes" fields in the Mode & option window. You may change these values without re-calculating the case.

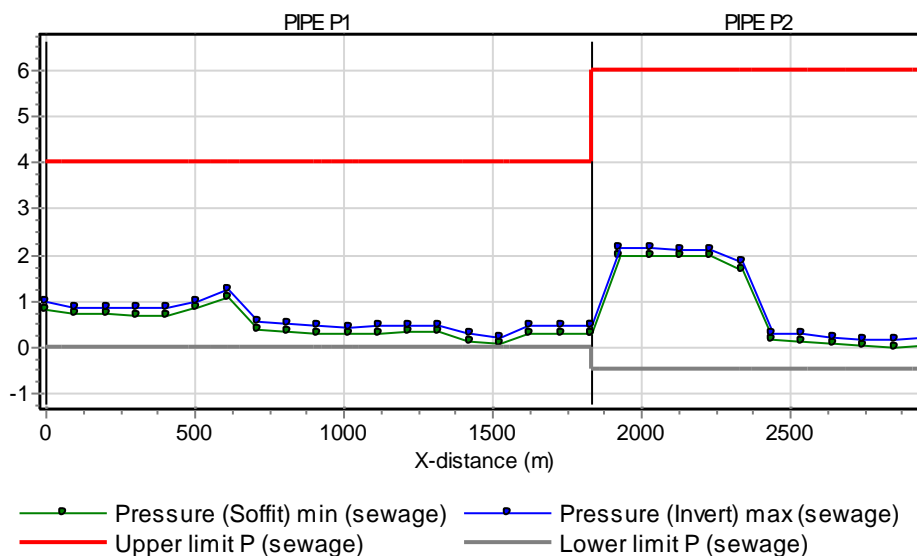


Figure 6: Pressure location chart with additional pressure rating series

Note that the reference for the pressure output (centre line / soffit (top) / invert (bottom)) is defined in the Mode & options window.

The main output properties like "Discharge" and "Pressure" contains vector based output (output values in each pipe node or element). That means that the chart button automatically creates a location chart. If you want a time chart in a particular pipe node you have to specify the location in the "Location" field.

To have direct access to the first and last node of the pipe (corresponding with the left and right H-node) you may extend the output property list with these points by checking the field "Connect point output in pipes".

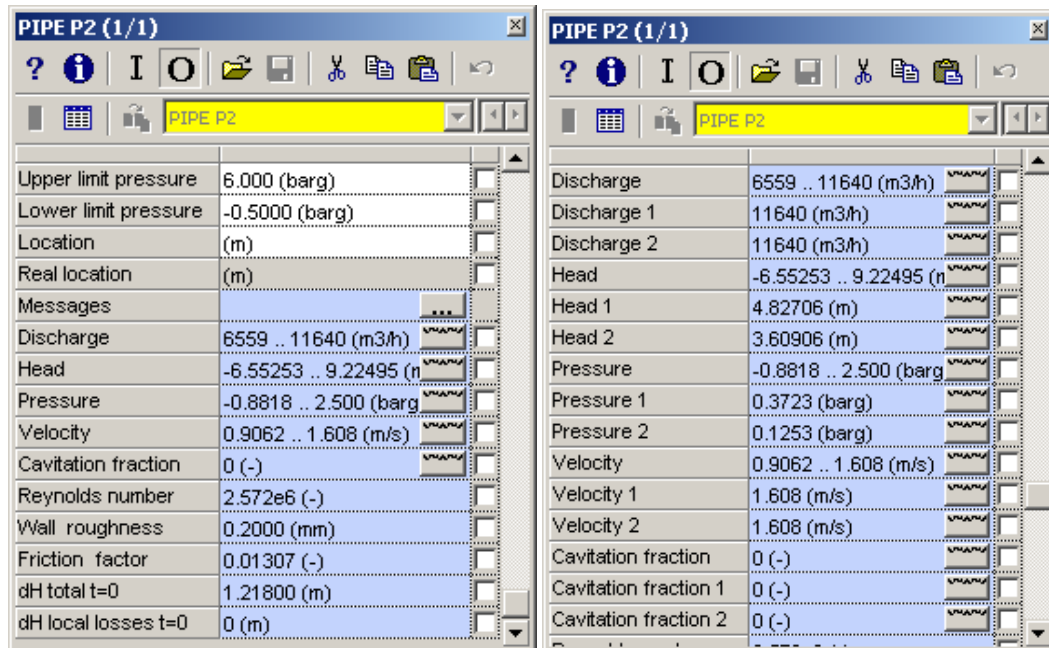


Figure 7: Two appearances of output properties window depending on Mode & Options setting "Connect point output in pipes".

Pressing the chart button in these additional fields automatically generates a time series chart.

4.20.2.5 Component specific output

Output	Description
Pipe length [m]	Total pipe length based on entered profile
Wave speed [m/s]	Wave propagation speed based on fluid properties, pipe properties and time step
Pipe element count [-]	Amount of elements of same length in which pipe is divided.
Adapted wave speed [m/s]	Wave speed is adapted in such way that a integer number of elements (minimal 1) fits in the total length
Deviation adapted [%]	Ratio between wave speed and adapted wave speed. Must be less than 0.25 for a valid deviation
Cavitation fraction [-]	Ratio between cavitation volume and element volume Must be less than 0.20 for valid range of cavitation algorithm
H-node height check	Input validation between H-node height with corresponding pipe node elevation. OK means the difference is less than 0,5 D, otherwise "Warning" is displayed.
Wall roughness	The wall roughness based on the friction factor and including local losses
dH total t= 0	Total head losses across the pipe at steady state
dH local losses t=0	Head loss due to local losses at steady state

4.20.2.6 H-actions

None

4.20.2.7 Example

None

4.21 Fast filling pipe

4.21.1 PIPE FFP (class)



Fall type

type label	description	active
Pipe	Fast Filling Pipe. Pipe model which can be used to model the fast filling of an empty pipeline. The water front moves vertically through the pipeline (perpendicular to the axis).	No

Fast Filling Pipe

The fast filling pipe can be used to simulate the fast filling of a single pipeline. Filling of branched systems is not possible with this component. In case of fast filling, the water front moves vertically through the pipe (perpendicular to the pipe axis). Filling can be modelled from either left or right side but filling from both sides simultaneously is not possible. The user does not have to specify the filling direction.

4.21.1.1 Mathematical model

The mathematical model for the moving water front is based on ref [1]. The behaviour of the fast filling pipe is divided into four phases:

1. Initialisation phase
2. Waiting to start with filling phase
3. Filling phase
4. Completely filled phase

The criterion for “fast filling” is derived from ref. [3]. The user needs to check the flow number. The flow number should be above 0.9 to make sure that all air is transported and does not remain trapped at local high points.

Initialisation phase

This phase is used in the steady state to initialize an empty or partly filled pipeline at the beginning of the transient simulations. The head in the pipeline is initialized at the elevation initial air pressure specified by the user of the pipeline.

Waiting to start with filling phase

The pipe is waiting to be filled from one of the connection points. As soon as the head at one of the connection points increases above the elevation of the pipe at the connection point, the pipe goes to the filling phase, in which the pipe is filled from the connection point where the head is raised. If the head is raised on both sides simultaneously, a warning is given and the calculations are continued with filling from connection point 1.

Filling phase

During the filling phase, the pipeline is filled from the side at which the head is increased first. The filling flow rate is determined from (see ref [2]):

$$\frac{1}{A} \frac{dQ}{dt} = \frac{g}{L_f} \left(H_{1,2} - H_a - f \frac{L_f}{D} \frac{Q|Q|}{2gA^2} - (1 + \xi_{entr}) \frac{Q|Q|}{2gA^2} \right) \quad (1)$$

In which:

A	=	Area of the pipeline	[m ²]
Q	=	Flow rate	[m ³ /s]
dt	=	Time step	[s]
g	=	Gravitational acceleration	[m/s ²]
L _f	=	Length of the fluid column	[m]
H _{1,2}	=	Head at side from which the pipe is filling	[m]
H _a	=	Air pressure head at the end of the liquid column	[m]
f	=	friction factor	[-]
D	=	Diameter pipe	[m]
ξ _{entry}	=	Entry loss	[-]

The air pressure head is calculated from:

$$\frac{dH_a}{dt} = \lambda \frac{H_a}{L_a} \left(Q - \left(\frac{P_{atm}}{P_{air}} \right)^{\frac{1}{\lambda}} Q_{air} \right) \quad (2)$$

In which:

λ	=	Laplace coefficient	[-]
L _a	=	Length of air column	[m]
Q _{air}	=	Air flow rate at atmospheric pressure	[Nm ³ /s]

The air discharge is calculated based on the air pressure, in the same way as for air valves.

Equation 1 is based on the rigid column model. During the calculations, the length of the fluid column is followed. The head is set to the height of the pipe for locations which are not filled yet. For the remainder (elements that are filled) the head decreases linearly to the initialized head. When the length of the liquid column becomes identical with the length of the pipeline, the pipeline is completely filled and phase 4 is started.

This transition results in strong transients, which may depend upon the time step used. It is therefore recommended to do a sensitivity analysis on the time step to ensure physical correct results are obtained.

Completely filled phase

When the pipeline is completely filled, the calculation method is changed to water hammer. Based on the boundary condition, e.g. closed valve or reservoir, the resulting head and flow rate are calculated.

4.21.2 Fast filling pipe properties

4.21.2.1 Hydraulic specifications

description	input	unit	default	remarks
Cross section	Circle	[-]	Circle	
Inner diameter	real	[m]		if Cross section=Circle
Calculation mode	Waterhammer		Waterhammer	Only in transient mode
Wave speed mode	Physical specified		Physical	if Calculation mode = Waterhammer
Wall thickness	real	[m]		if Wave speed mode = physical
Young's modulus	real	[N/m ²]		if Wave speed mode = physical
Specified wave speed	real	[m/s]		if Wave speed mode = specified
Wall roughness	real	[mm]		if Friction model = D-W k
Dynamic friction	Quasi-steady none			Only in transient mode and if Friction model = D-W k
Geometry input	Length l-h xyz xyz diff	[-]		
Length	real	[m]		if Geometry input = Length
Profile	table			if Geometry input = L-h, xyz or xyz diff
Upper limit pressure	real	[N/m ²]		Only visible if checked in "Mode&options" window
Lower limit pressure	real	[N/m ²]		Only visible if checked in "Mode&options" window
Location	real	[m]		If empty, chart button creates a location serie If filled in, chart button creates a time serie for nearest internal node
Discharge coefficient air	Real			

description	input	unit	default	remarks
Discharge area air	Real			
Entrance loss coefficient flow	Real			
Laplace coefficient	Real	[-]		
Initial air pressure	Real	[bar.a]	1.014	
Initial length liquid column	Real	[m]	0	

Remarks

Please note that the fast filling pipe is a dedicated component and can only be used to simulate the filling of a single pipeline. Furthermore, it is recommended for stability to use a small time step, to ensure the transition from filling to water hammer phase is taken place smoothly.

4.21.2.2 Component specific output

Output	Description
Pipe length [m]	Total pipe length based on entered profile
Wave speed [m/s]	Wave propagation speed based on fluid properties, pipe properties and time step
Pipe element count [-]	Amount of elements of same length in which pipe is divided.
Adapted wave speed [m/s]	Wave speed is adapted in such way that an integer number of elements (minimal 1) fits in the total length
Deviation adapted [%]	Ratio between wave speed and adapted wave speed. Must be less than 0.25 for a valid deviation
Cavitation fraction [-]	Ratio between cavitation volume and element volume Must be less than 0.20 for valid range of cavitation algorithm
H-node height check	Input validation between H-node height with corresponding pipe node elevation. OK means the difference is less than 0.5 D, otherwise "Warning" is displayed.
Wall roughness	The wall roughness based on the friction factor and including local losses
dH total t= 0	Total head losses across the pipe at steady state
dH local losses t=0	Head loss due to local losses at steady state
Flow number	Flow number, can be used to determine if it is a fast or slow filling.
Air pressure	Pressure of the air
Length liquid column	The length of the liquid column
Air flow rate	Flow rate of air out (+) or in (-)

Message	Type	Explanation
Filling starts from connection point 1	info	

Message	Type	Explanation
Filling starts from connection point 2	info	
Head increases on both sides, filling starts from connection point 1	warning	
Pipe completely filled	info	

References

- [1] Zhou, F., Hicks, F.E., Steffler, P.M., 2002, *Transient Flow in a Rapidly Filling Horizontal Pipe Containing Trapped Air*, J. Hydraul. Eng., 128 (6), 625-634.
- [2] van 't Westende, J.M.C., Pothof, I.W.M., Heinsbroek A.G.T.J., Tukker M.J., Guijt W. Incident analysis of a fuel loading line, 14th International Conference on Multiphase Production Technology, Cannes, France: 17th – 19th June 2009
- [3] Pothof, I.W.M., Co-currunt air-water flow in downward sloping pipes, 2011, ISBN: 978-90-89577-018-5

4.22 Pressure relief valve

4.22.1 PRV (class)



Pressure relief valve, safety valve

Fall type

Type label	description	active
PRV (Ideal)	This PRV monitors the system pressure and opens upon exceedance of a set pressure. Opening time, friction, choking flow and valve dynamics are not taken into account.	No
PRV (simplified char.)	The opening pressure, as well as the full lift pressure and discharge, must be specified. (accumulation is taken into account).	No

4.22.1.1 Mathematical model

The PRV is always closed in the steady state. In steady state and closed state the governing equation of the PRV component is very simple:

$$Q_1 = 0 \tag{1}$$

in which:

$$Q_1 = \text{upstream discharge} \quad [m^3/s]$$

The PRV has been implemented as a Fall type (up- and downstream head and flow) to allow a downstream pressure that is different from atmospheric pressure. Only the PRV (simpl. char.) is influenced by the downstream pressure head.

If the system pressure (measured at the upstream H-node: H_1) exceeds the set pressure (gauge pressure) the PRV opens, maintains the set pressure, P_{set} , and the governing equation of the ideal PRV becomes:

$$H_1 = \frac{P_{set}}{\rho_f g} + h \tag{2}$$

in which:

H1	=	upstream pressure head	[m]
Pset	=	set pressure (gauge)	[N/m ²]
P_f	=	fluid density	[kg/m ³]
g	=	gravitational acceleration	[m/s ²]
h	=	elevation of PRV	[m]

The ideal PRV is not influenced by the downstream head, because it maintains the set pressure ideally.

In practice, however the PRV will not act ideally. The non-ideal behaviour can be modelled with a simplified characteristic of the PRV. After exceeding the set pressure the valve body will lift and gradually open upon increasing pressure and discharge. If the internal pressure keeps rising, the valve body will eventually hit a stop and the full lift condition is entered. From then on, the PRV will act as a resistance with constant loss coefficient. The simplified characteristic PRV is influenced by the downstream head so that the user may specify a boundary condition that is different from atmospheric pressure. One could think of a pressurised tank at the downstream side of the PRV.

In the partial lift phase ($0 < Q_1 < Q_{full}$) the governing equations are:

$$\Delta P = \Delta P_{set} + (\Delta P_{full} - \Delta P_{set}) \frac{Q_1}{Q_{full}} \quad (3)$$

$$Q_1 = Q_2$$

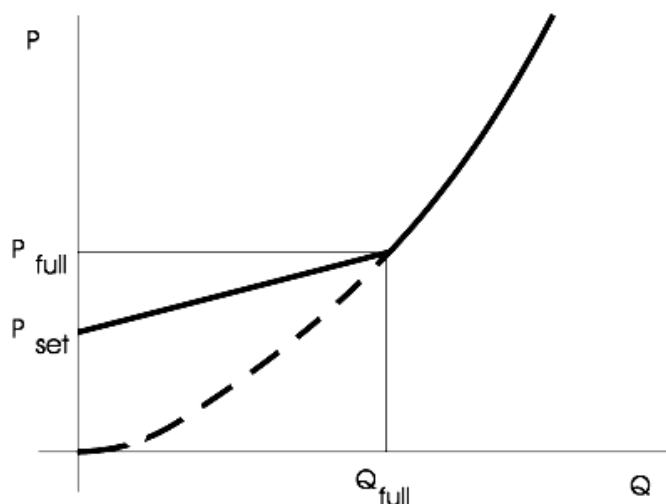
in which:

ΔP	=	pressure difference over PRV: $\rho g (H_1 - H_2)$	[N/m ²]
ΔP_{set}	=	set pressure difference	[N/m ²]
ΔP_{full}	=	full lift pressure difference	[N/m ²]
Qfull	=	full lift discharge	[m ³ /s]

In the full lift phase ($Q_1 = Q_{full}$) the governing equations are:

$$\Delta P = \frac{\Delta P_{full}}{Q_{full}^2} Q_1^2 \quad (4)$$

The Q P relation is shown in the figure below:



The PRV closes again if $Q_1 < 0$.

4.22.2 PRV (simplified char.)

4.22.2.1 Hydraulic specifications

Description	input	unit	range	default	remarks
set pressure (ΔP)	real	[N/m ²]	(0-10 ⁸)		
full lift pressure (ΔP)	real	[N/m ²]	(0-10 ⁸)		
full lift discharge	real	[m]	(0-10 ⁸)		

See also "Mathematical model" on page 346.

Tip:

The downstream head of the PRV should be set to the elevation level of the PRV if you want to model an atmospheric pressure at the downstream side.

4.22.2.2 Component specific output

None

4.22.2.3 H-actions

None

4.22.2.4 Component messages

Message	Type	Explanation
closes	Info	
partially open	Info	
fully open	Info	

4.22.3 PRV (Ideal)

4.22.3.1 Hydraulic specifications

Description	Input	unit	range	default	remarks
set pressure (P upstr.)	Real	[N/m ²]	(0-10 ⁸)		
Elevation	Real	[m]			

See also "Mathematical model" on page 346.

Tip:

The downstream head of the PRV should be set to the elevation level of the PRV if you want to model an atmospheric pressure at the downstream side.

4.22.3.2 Component specific output

None

4.22.3.3 H-actions

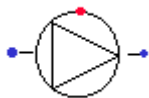
None

4.22.3.4 Component messages

Message	Message	explanation
Closes	Info	
Opens	Info	
Not valid ($H_2 > H_1$), downstream capacity exceeded	Warning	H_2 does not play a role in the current mathematical model of the ideal PRV. If however H_2 becomes greater than H_1 the PRV "drowns" and effectively the computation loses physical relevance. This is a useful aid to design the downstream capacity.

4.23 Pump

4.23.1 PUMP (class)



Pump

Fall type

type label	description	active
Pump	Complete pump model with various data and drives. Positive and negative speeds	yes

4.23.1.1 Mathematical model

In the most general case a pump can operate in four QH-quadrants for $N \geq 0$ and in four QH-quadrants for $N < 0$. In WANDA, two pump models are distinguished: the pump model limited to the $N \geq 0$ case (section 4.23.1.2) and the complete pump model for all N (< 0 , $= 0$ and > 0), see section 4.23.1.3.

Basis equations

The running pump is characterised by the following functions in the first quadrant for $N > 0$:

$$H = f(Q), N = N_r \quad (1)$$

$$E = f(Q), N = N_r \quad (2)$$

in which:

H	=	pump head	[m]
Q	=	pump discharge	[m ³ /s]
E	=	pump efficiency	[-]
N	=	pump speed	[rpm]
N _r	=	rated pump speed (at which the functions f and g are specified)	[rpm]

To describe a tripping pump the torque function must be known as well:

$$T = f(Q), N = N_r \quad (3)$$

To derive the functions (1), (2) and (3) for other pump speeds N_i than the rated pump speed N_r the affinity rules are used:

$$\begin{aligned} \frac{H(N_i)}{H(N_r)} &= \left(\frac{N_i}{N_r}\right)^2 \\ \frac{Q(N_i)}{Q(N_r)} &= \frac{N_i}{N_r} \\ \frac{T(N_i)}{T(N_r)} &= \left(\frac{N_i}{N_r}\right)^2 \end{aligned} \quad (4)$$

The affinity rules – also known as the homologous relations – are based on the assumption that the flow field and the accompanying pressure field are independent of the size of the pump as long as the shape (proportions) remain the same, and that also the efficiency does not change with the size of the pump.

To simulate a tripping pump Newton's second law is used in terms of rotating mass:

$$T_d - T = I_p \frac{dN}{dt} \frac{2\pi}{60} \quad (5)$$

in which T_d is the driving torque and T is the torque exerted by the fluid on the pump impeller. I_p denotes the polar mass moment of inertia [kgm²] of rotating mass (=motor + impeller + fluid). Note: sometimes the value GD^2 (GeeDeeSquared) is used which is four times the I_p

In normal running operation $T_d = T$ and the pump will run at a steady speed. Due to e.g. power failure, $T_d = 0$, and therefore during pump trip the decaying pump speed can be calculated from:

$$dN = -\frac{60}{2\pi} \frac{T}{I_p} dt \quad (6)$$

In the procedure this differential equation is solved using Euler's method. The torque for N_i is calculated using the affinity rules (4). In the case of a starting pump, T_d is determined by the speed-torque relation (in tabular form) of the driving motor-gearbox combination.

4.23.1.2 The H-model for $N \geq 0$

The functions (1) and (2) are usually unknown in algebraic form. A pump manufacturer normally provides f and g in graphics and then only in the first quadrant for $N = N_r$. For the procedure these graphics are specified as tables. Hence at the start of the simulation only the Q-H table and the Q-E table are known for N_r . From both tables the torque table can be derived using:

$$T = \frac{\rho_f g Q H}{E \omega_r} \quad (7)$$

in which:

ρ_f	= fluid density	[kg/m ³]
g	= grav. acceleration	[m/s ²]
E	= efficiency	[-]
ω_r	= rated pump speed	[rad/s]

However, to be able to simulate pump trip the functions H and T have to be known in the second and fourth quadrant as well. To get the values outside the table range the tables are extended to the 'left hand' and 'right hand' side with so-called extrapolation parabolas. These parabolas have the following general forms:

Q-H at the right of the table range:

$$H / H_r = A_H(Q / Q_r)^2 + B_H(Q / Q_r) + C_H$$

Q-H at the left of the table range:

$$H / H_r = D_H(Q / Q_r)^2 + E_H(Q / Q_r) + F_H$$

Q-T at the right of the table range:

$$T / T_r = A_T(Q / Q_r)^2 + B_T(Q / Q_r) + C_T \quad (8)$$

Q-T at the left of the table range:

$$T / T_r = D_T(Q / Q_r)^2 + E_T(Q / Q_r) + F_T$$

As we can see, the parabolas are given in dimensionless form. The H , T and Q are made dimensionless with respect to H_r , T_r and Q_r i.e. the values in the maximum efficiency point (at the rated pump speed N_r). The extrapolation coefficients are derived in a quite specific way. The A and D coefficients are calculated from empirical data. These coefficients are given in the tables 1 through 4 inclusive below as a function of the pump specific speed N_s .

Table 1: AH coefficients and derivatives versus the pump specific speed N_s

NS	AH	dAH/dNS
10	-0.446	-0.004100
20	-0.487	-0.005075
50	-0.649	-0.007263
100	-1.068	-0.009730
150	-1.622	-0.008070
200	-1.875	0.004690
250	-1.153	0.014440

Table 2: DH coefficients and derivatives versus the pump specific speed NS

NS	DH	dDH/dNS
10	0.566	0.005100
20	0.617	0.006400
50	0.822	0.009650
100	1.389	0.013480
150	2.170	0.007370
200	2.126	-0.009100
250	1.260	-0.017320

Table 3: AT coefficients and derivatives versus the pump specific speed NS

NS	AT	dAT/dNS
10	-0.332	-0.003800
20	-0.370	-0.004675
50	-0.519	-0.007588
100	-0.977	-0.010660
150	-1.585	-0.006080
200	-1.585	0.008050
250	-0.780	0.016100

Table 4: DT coefficients and derivatives versus the pump specific speed NS

NS	DT	dDT/dNS
10	0.704	0.005800
20	0.762	0.007375
50	0.999	0.010050
100	1.566	0.011270
150	2.126	0.002140
200	1.780	-0.013450
250	0.781	-0.019980

The pump specific speed for a particular pump is defined as:

$$N_s = \frac{N_r(Q_r)^{0.5}}{H_r^{0.75}} \quad (9)$$

This quantity characterises the pump. Low values (<50) indicate a centrifugal (or radial) type of pump and high values (>200) indicate an axial type of pump. Suppose a given pump results in a certain N_{spec} . Usually this value is not equal to one of the given table values N_S and interpolation is required. This is done using the derivatives (which are also given in the tables) in the following way:

$$A_H(N_{\text{spec}}) = A_H(N_S) + (N_{\text{spec}} - N_S)dA_H/dN_S \quad (10)$$

The same method is applied for D_H , A_T and D_T . When these empirical values are known, the remaining coefficients are determined as follows:

- B_H and C_H are calculated using the last two Q-H table points.
- E_H and F_H are calculated using the first two Q-H table points.
- E_T and F_T are calculated using the first two Q-T table points.

To obtain the B_T and C_T coefficients the method is different:

A. First, the intersection point of

$$H/H_r = A_H(Q/Q_r)^2 + B_H(Q/Q_r) + C_H \quad (11)$$

and

$$H/H_r = \phi(Q/Q_r)^2 \quad (T=0 \text{ line})$$

is determined. The $T=0$ line describes the relation between H and Q when the pump is maintaining a speed that results exactly in a hydraulic torque $T = 0$. The coefficient ϕ is an empirical value: $\phi = -0.3$. It is assumed to be a constant independent of the pump specific speed (although in reality it may vary between about -0.1 and -0.4). The intersection results in the discharge Q_{T0} i.e. the discharge for which $T = 0$.

B. B_T and C_T are now determined from the last table point of the Q-T table and the point $Q = Q_{T0}$, $T = 0$.

In dimensionless form the various tables and extrapolation parabolas are given as an example in figure 1. In this figure $Q_{T0} = 3.52$ (dimensionless!).

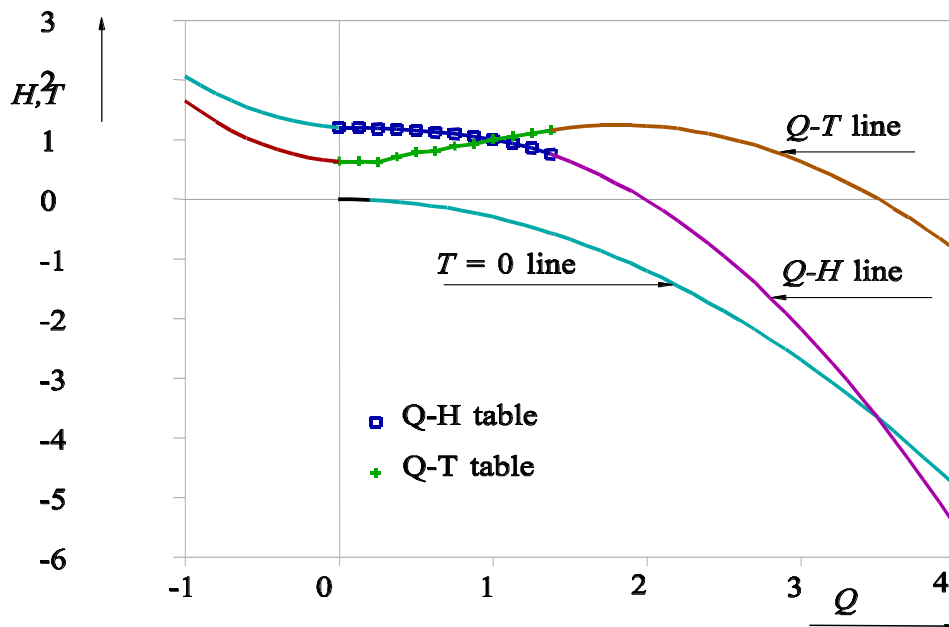


Figure 1: Dimensionless curves of pump for $N>0$

4.23.1.3 The complete pump model

In order to obtain the pump function for positive and negative flow as well as positive and negative pump speed, heads and torque's for all combinations of Q and N have to be measured and plotted as isolines in an N - Q graph. This has been carried out for a number of pumps in literature. These pumps are of varying type (radial flow, mixed flow, axial flow) denoted by specific speeds of 25, 147 and 261 (SI units) respectively (see Wylie, *Fluid Transients in Systems*, 1993).

It has proven convenient to use dimensionless quantities for head, torque, discharge and speed, by dividing the quantities by their respective values at rated (r) conditions.

$$h = \frac{H}{H_r} ; \quad \beta = \frac{T}{T_r} ; \quad q = \frac{Q}{Q_r} ; \quad \alpha = \frac{N}{N_r} \tag{12}$$

Expressed in these dimensionless quantities the affinity laws read:

$$\frac{h}{\alpha^2} \text{ vs. } \frac{q}{\alpha} \quad \frac{\beta}{\alpha^2} \text{ vs. } \frac{q}{\alpha} \quad \frac{h}{q^2} \text{ vs. } \frac{\alpha}{q} \quad \frac{\beta}{q^2} \text{ vs. } \frac{\alpha}{q} \tag{13}$$

From a computational viewpoint, these relations are difficult to handle (dividing by zero when pump speed passes zero, and changing signs during transients). Marchand and Suter have overcome this difficulty by using:

$$WH = \frac{h}{\alpha^2 + q^2} \text{ vs. } \tan^{-1} \frac{q}{\alpha} \quad \text{and} \quad WB = \frac{\beta}{\alpha^2 + q^2} \text{ vs. } \tan^{-1} \frac{q}{\alpha} \tag{14}$$

For complete pump data in all zones of operation, a polar diagram of $\tan^{-1} \frac{q}{\alpha}$ vs. WH and $\tan^{-1} \frac{q}{\alpha}$ vs. WB can be plotted as closed curves. This is usually done in the form of a rectangular plot (see figure 2) in which the abscissa $x = 2\pi$ ($= \tan^{-1} \frac{q}{\alpha}$). The addition of π is intended to have only positive values for x (ranging from 0 to 2).

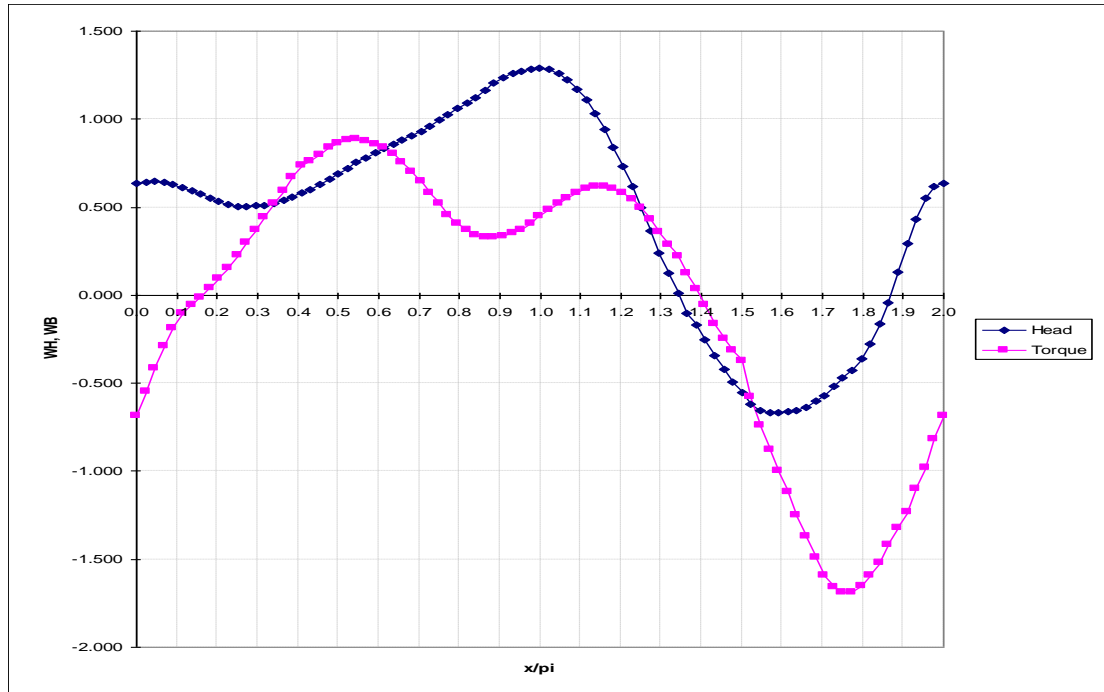


Figure 2: Dimensionless Suter curves of radial pump ($N_s=25$, SI units)

These curves can for any α be transformed into head and torque as functions of discharge and made dimensional by substitution of actual values in (12) and (14). An example transformed from figure 2 is given in figure 3.

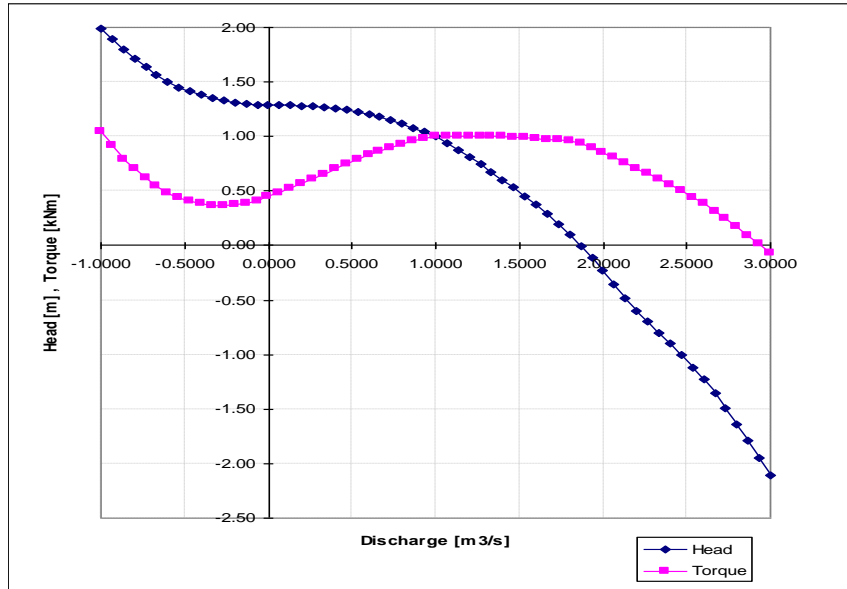


Figure 3: Dimensionless curves of pump for $N > 0$

This procedure works equally well for $\alpha < 0$, $\alpha = 0$ and $\alpha > 0$. The user input consists of the two tables $WH(x)$ and $WB(x)$ among other quantities. If only a QHE-table is available the user must prepare the WH/WB tables by manual transformation and fill in the empty parts by interpolating from known curves for comparable specific speed pumps. The results of such interpolations must however be viewed with scepticism.

4.23.1.4 Determination of the new pump speed

The calculation of the pump speed at $t_{\text{new}} (= t_{\text{old}} + \Delta t)$ is quite specific. Equation (6) is integrated using Euler's method and sub-timesteps Δt^* :

$$N_{\text{new}} = N_{\text{old}} + \frac{60}{2\pi} \frac{(T_d - T)_{(N_{\text{old}})}}{I_p} \Delta t \tag{15}$$

in which:

- N_{new} = pump speed at $t = t_{\text{new}}$
- N_{old} = pump speed at $t = t_{\text{old}}$
- T_d = driving torque at $t = t_{\text{old}}$, hence for $N = N_{\text{old}}$
- T = hydraulic torque at $t = t_{\text{old}}$, hence for $N = N_{\text{old}}$

For a tripping pump $T_d = 0$. To obtain sufficient accuracy the difference between N_{new} and N_{old} may not be too large for each Euler integration sub-timestep. This difference is denoted as $DN = N_{\text{new}} - N_{\text{old}}$. DN must have a value as given in figure 4. This value is denoted as DN_w ("wanted" DN).

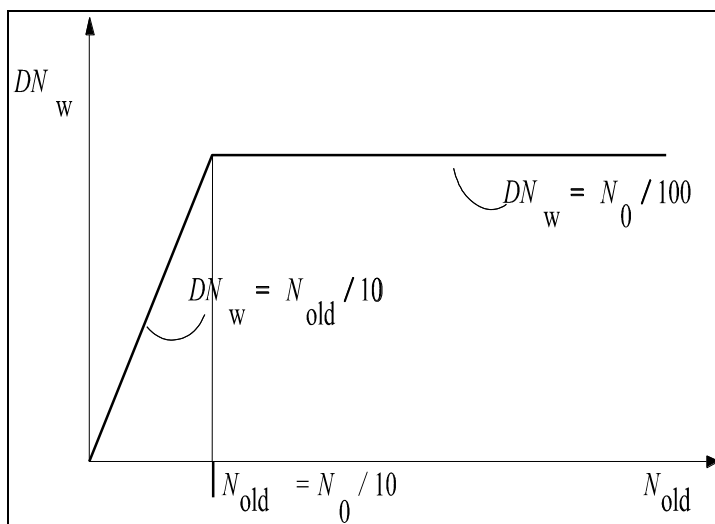


Figure 4: DN_w as a function of N

As long as $N_{old} > N_0/10$, DN_w is fixed to $DN = N_0/100$. At a certain instant the pump speed drops below the value $N_0/10$. From that moment on DN_w is varying according to $DN = N_{old}/10$. Based on this arbitrarily chosen DN_w a new (sub) time step Δt is calculated in the following way:

- N_{new} is calculated using Δt (first estimate)
- The difference DN is calculated
- Depending on N_{old} , DN_w is calculated (see figure 4)
- The number of (sub) time steps within Δt is calculated: $N_{t-steps} = DN/DN_w$
- The new time step $\Delta t^* = \Delta t/N_{t-steps}$ is calculated.

Once this is done the N_{new} is calculated using Δt^* (provided that $N_{t-steps} > 1$) until $t = t_{old} + \Delta t$.

4.23.1.5 Motor drive

For the motor driven pump the following considerations are to be taken into account. The pump speed is computed using equation (16) of the mathematical model description in the beginning of this section.

$$T_d - T = I_p \frac{dN}{dt} \frac{2\pi}{60} \tag{16}$$

Here, the driving torque T_d is determined by the motor properties. In general, when the motor is powered on, T_d is a function of the pump speed and the frequency of the speed control. In case the power is off, T_d is of course zero and we then have a situation equivalent to the tripping pump.

Typical speed-torque curves for electric motors and diesel engines are plotted below.

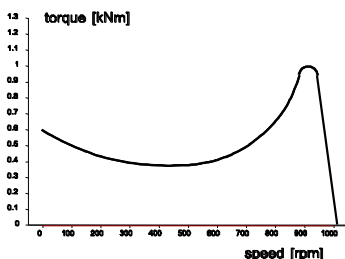


Fig. 5: Electric motor

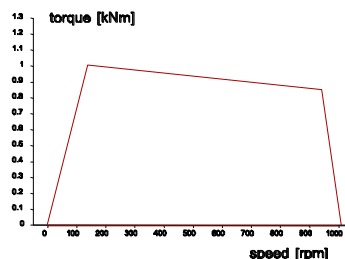


Fig. 6: Diesel engine

Both curves are characterised by a sharp drop (cut-off) around the operating speed. The result of this is that with load increase the speed does not drop much. This is denoted as elastic behaviour. If the maximum torque just before the drop is sufficiently large the steady state value of the pump speed is well defined. In steady state WANDA will compute two equilibria: 1) pump (QH) curve versus system resistance (QH relation) and 2) hydraulic torque versus motor torque. Both are varying at each iteration step and to reach convergence a dedicated iteration strategy has been employed.

For electric motors the pump speed at which the torque goes through zero is denoted by the term “synchronous speed”. This speed is in general equal to 3000 RPM (the frequency of the AC power) divided by the number of coils, which is half the number of poles. In practice many two-coil (four pole) and three-coil (six pole) motors are used with their respective synchronous speeds of 1500 RPM and 1000 RPM. An electric motor can only generate torque if the shaft RPM exhibits a certain “slip” relative to the synchronous speed. The nominal power (and torque) will be delivered at somewhere between 1 and 3 % slip, this is specified by the manufacturer. The peak torque is about 3 times as large as the nominal torque. The start torque is about 2.2 times as large as the nominal torque and the sag torque (the relative minimum half way) is about 2 times the nominal torque.

To be able to “brake” or “force down” the pump by means of a temporarily negative torque, the speed torque table has to be mirrored about the synchronous point. This can easily be done by hand by adding the difference of the synchronous speed and the speed value to the synchronous speed and reversing the torque value of each speed-torque co-ordinate pair. An example is given in figure 7 below:

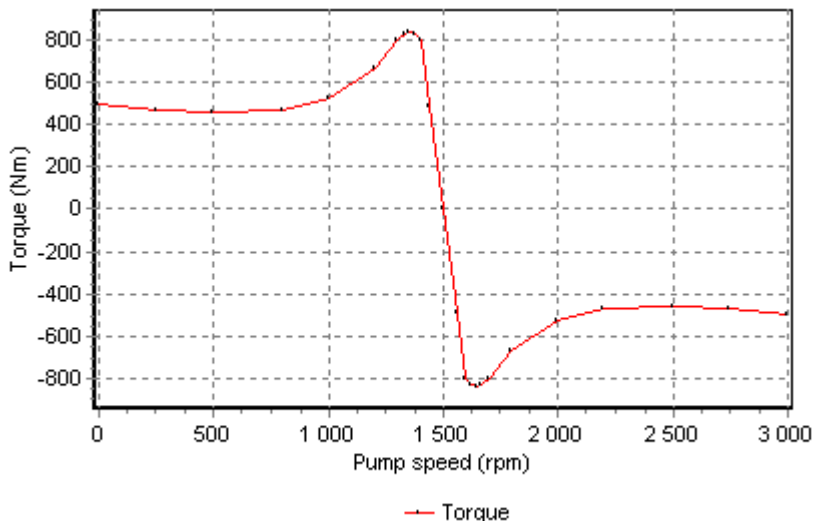


Figure 7: Complete speed-torque curve for electric motor

The nominal power for the speed torque curve in figure 7 can be computed by multiplying the nominal torque (which could be 220 Nm) and the speed (which is in that case 155 rad/s). The resulting nominal power is 34.1 kW. However in this case the motor can also run at 150.8 rad/s (1440 RPM) delivering a torque of 490.7 Nm (this is the penultimate co-ordinate just before the synchronous point), leading to a power of 74 kW. The motor slip is in that case 60 RPM. Whether this will lead to thermal problems depends on the design of the motor and this mode of operation has to be confirmed by the manufacturer.

The speed values in the curve are offset by the relative frequency, thus moving the curve in horizontal direction, thereby moving also the synchronous point to the left or right (decreasing or increasing of RPM). Theoretically the relative frequency can even become zero but then the motor torque will be zero throughout. In order to “force” the pump down in RPM the speed-torque curve must also contain negative torque values for speeds greater than the synchronous speed.

In unsteady state the relative frequency should not be changed faster than the pump RPM can follow, otherwise the operating point (intersection between impeller speed-torque and motor speed-torque curves) will drop in the start-up zone.

4.23.2 Pump properties

4.23.2.1 Hydraulic specifications

Description	input	unit	Range	default	remarks
Characteristic type	QHE / Suter			QHE	
QHE table	table				if Char. type = QHE
Suter table	table				if Char. type = Suter
Rated speed	real	[rad/s]	(0-10 ¹⁰)		
Q in best point of eff.					if Char. type = Suter
H in best point of eff.					if Char. type = Suter
T in best point of eff.					if Char. type = Suter
Initial setting	Speed / H_upstream / H_downstream / Discharge / Frequency			Speed	
Initial speed		[rad/s]			if Initial setting = Speed
Initial suction head		[m]			if Initial setting = H_upstream
Initial pressure head		[m]			if Initial setting = H_downstream
Initial discharge		[m ³ /s]			if Initial setting = Discharge
Initial relative frequency		[-]			if Initial setting = Frequency
Speed-Torque char.of motor	table				if Initial setting = Frequency
Polar moment of inertia	real	[kgm ²]	(0-2000)		only in transient mode
Drive type	Running/ Trip / Speed / Motorfreq / OnOff			Run- ning	
Trip time		[s]			if Drive type = Trip
Action table speed	time vs. speed				if Drive type = Speed
Minimum speed for tripping		[rad/s]			if Drive type = Speed
Action table motorfreq	time vs. rel. motorfreq.				if Drive type = Motorfreq
Minimum rel.freq. for trip.		[-]			if Drive type = Motorfreq
Level On		[m]			if Drive type =

				OnOff
Level Off		[m]		if Drive type = OnOff
Start time (0 to max)		[s]		if Drive type = OnOff
Stop time (max to 0)		[s]		if Drive type = OnOff
Required speed	real	[rad/s]	(0-10 ¹⁰)	only visible in system characteristics
Minimum speed	real	[rad/s]	(0-10 ¹⁰)	only visible in pump energy
Maximum speed	Real	[rad/s]	(0-10 ¹⁰)	only visible in pump energy
Maximum power	Real	[W]	(0-10 ¹⁰)	only visible in pump energy

Because the Suter curves are dimensionless the discharge (Q), head (H) and torque (T) in the best efficiency point must also be supplied if the characteristic type is "Suter".

In case of the QHE characteristic the Q and H at maximum efficiency will be found in the table. Hence the user must take care for one maximum value within the E range.

Used table types

QHE table (Q to run from 0.0 upwards, H to decrease monotonously, E to be zero at either H=0.0 or Q=0.0)

Q [m3/s]	H [m]	E [-]
Real	Real	Real
....
....
real	real	real

Suter table (x/π to run from 0.0 to 2.0)

x/π [-]	WH [-]	WB [-]
Real	Real	Real
....
....
real	real	real

Three Suter curves according to Wylie and 14 curves according to Thorley have been included as property templates (see ...Wanda3\Property templates\Examples\Suter_pump)

Speed torque table in case of motor driven pump (see examples)

N [rad/s]	T [Nm]
Real	Real
....
....
real	real

Important:

- Minimum required rows is 5, maximum is 100.
- The heads in the Q-H table must be monotonously decreasing.
- The Q-E table must have a maximum value for the efficiency E.
- Efficiency must be zero where either Q=0 or H=0.
- It is recommended to start QHE tables with $Q = 0$ [m³/s].
- The first and last row values of WH and WB should respectively have the same values (for $x/\pi=0$ and $x/\pi=2$), thus forming closed curves.

Steady state settings

To determine the steady state condition of the pump the choice property “Initial setting” has to be set accordingly. The user has five choices:

Speed	The operational speed is specified by the user. QH curve will be applied in computation and H1, H2 and Q are computed depending on system
H_upstream	The head upstream is specified by the user. H1 = C is applied in computation and H2 and Q are computed. Afterwards, the pump speed will be computed, such that the combination H2-H1 and Q is on the QH curve.
H_downstream	The head downstream is specified by the user. H2 = C is applied in computation and H2 and Q are computed. Afterwards, the pump speed will be computed, such that the combination H2-H1 and Q is on the QH curve.
Discharge	The discharge is specified by the user. Q = C is applied in computation and H2 and Q are computed. Afterwards, the pump speed will be computed, such that the combination H2-H1 and Q is on the QH curve.
Frequency	The relative motor frequency is specified by the user. This is the actual frequency (e.g. 40 Hz) divided by the nominal frequency (e.g. 50 Hz) yielding the value of 0.8. With this relative frequency the speed-torque curve of the motor is adapted (shifted horizontally) and the steady state (H1, H2 and Q) will be solved such that motor torque and hydraulic (impeller) torque intersect. The point of intersection determines the actual pump speed.

Unsteady state settings

For the computation of the pump during unsteady state there are five possibilities to choose from in the property "Drive type"

Running	The pump speed and the QH curve will be kept constant, throughout the complete simulation time. Value for "polar moment of inertia" is not used
Trip	The pump speed and QH curve will be maintained until the user specified "trip time". Thereafter the motor torque will be zero and the pump will trip according to equation 15.
Speed	The pump speed will be determined by applying the time-speed table (which is specified by the user). Polar moment of inertia is not taken into account during use of the table If the value in the field "min speed for tripping" has been undershoot, the speed table is overruled and the pump acts as a tripping pump, see remark 3
Motorfreq	The pump speed will be computed by applying the transformed speed-torque curve of the motor on basis of the time table for the relative frequency (which is specified by the user)
OnOff	Fix speed pump is switched on if upstream head $H_1 > \text{"Level On"}$; speed increases in "Start time" seconds from 0 to "Rated speed" Pump is switched off if upstream head $H_1 < \text{"Level Off"}$; speed decreases in "Stop time" seconds from rated speed to 0 Polar moment of inertia is not taken into account This pump type can be used for simple pump operation control; use Control module for advanced operational control

Remarks

- It is most logical to apply drive type "Speed" in combination with initial setting "Speed" and drive type "Motorfreq" in combination with initial setting "Frequency" in order to have a smooth transition between the steady state and unsteady state computations. However, different combinations are in principle possible. The user must take care that the pump speeds match between steady and unsteady. Differences will however be solved dynamically: the unsteady state solution process aims at achieving a new balance.
- All "Drive type" types may be controlled by a control loop (C-components).
- To trigger the "trip" action on a pump with drivetype = speed at certain point in time T, the table value of the speed before T must be greater than the value of the input property "Minimum speed for tripping" and the table value after T must be equal to zero.

4.23.2.2 Component messages

Message	Type	Explanation
Q-H curve not monotonously Decreasing	Warning	To prevent instability, the derivative of $H(Q)$ must be negative.
Pump efficiency must be zero if either $Q=0$ or $H=0$ in input	Error	To have a finite torque, this is by definition.

Message	Type	Explanation
and non-zero elsewhere		
Pump specific speed =.....	Info	Small number (<40) denotes centrifugal pump, large number (>100) denotes axial pump.
Pump efficiency less than 0 or greater than 1	Error	
Motor overloaded Fluid torque = [kNm] Max. motor torque = [kNm]	Error	The intersection between hydraulic and motor torque is at the left of the maximum motor torque.
Convergence between fluid and motor torque less accurate than 0.1 % Fluid torque = [kNm] Motor torque = [kNm]	Warning	This will lead to a further convergence process in TRANSIENT, therefore we are not yet in a steady state.
Initial pump speed = [rpm]	Info	.
Initial-running status contradictive with control based running status	Warning	For example, according to the initial status the suction controlled pump is running, whereas the measured suction level is below the switch off level.
Unbalance exceeding rpm sensitivity fluid torque = Nm motor torque = Nm	Warning	Within the accuracy of pump speed computation, it was not possible to obtain balance between hydraulic and motor torques. The pump speed is correct however and the torque unbalance will settle during the following time steps (dynamics).
Starts	Info	.
Initial value of action table does not coincide with initial value of pump speed.	Warning	A possibly unintended "jump" in the pump speed will occur upon starting the unsteady state computation.
Motor overloaded	Warning	The hydraulic torque has become larger than the maximum (peak) motor torque. The motor is not longer capable of holding the normal pump speed. This is called a powered trip.
Trips	Info	.
Inconsistent pump speed between steady state results and control system; steady state value will be used	Warning	Message given if discrepancy between steady state speed and control value is less than 1 %
Inconsistent pump speed between steady state results and control system; Modify input	Error	Message given if discrepancy between steady state speed and control value is more than 1 %

4.23.2.3 Component specific output

Pump speed [rad/s]

Fluid torque [Nm]

Motor torque [Nm]

On/Off [-]

Efficiency [-]

Power [W].

4.23.2.4 H-actions

How to activate the pump is dependent of the drive type. If the type is “running” there will be no action applicable. The pump just keeps running and the fixed QH-curve will apply throughout the steady state computation. The speed may be overruled by a control connection (signal line).

If the drive type is “trip” the input value “trip time” will determine the time at which the driving torque becomes zero and the pump speed will solely be determined by the hydraulic torque. The speed may be overruled by a control connection (signal line)

If the drive type is “speed” the applicable action table will be of the time-speed type. In case the initial control is also “speed” the user interface will copy the initial speed into the table. In other cases (init_H1, init_H2, init_Q and motor), the initial pump speed will be calculated by the computational core (STEADY) and it is the user’s responsibility to have the time-speed table starting with the same pump speed. A warning will be issued however in case of a discrepancy. The action table may be overruled by a control connection (signal line).

If the drive type is “MotorFreq” the action table will be of the type “relative frequency” which is in effect a dimensionless table. The percentage will be used to scale the motor speed-torque curve relative to the y-axis (speeds will be multiplied with the scaling factor). The action table may be overruled by a control connection (signal line).

If the drive type is “Level On/Off” the pump acts as a start/stop pump if the upstream head over/undershoot the On/Off values specified. The speed increase/decrease is defined by the ramp parameters.

4.24 Pump Energy

4.24.1 Pump Energy components

A special set H-components are used for the module PumpEnergy. These components are used in combination with the PUMP component. The PumpEnergy components are ignored in the steady and transient calculation.

Overview available components

type	description
PUMPSCEN	Pump operation scenario
Q_PATTRN	Discharge pattern as function of time
Q_FREQ	Discharge frequency in a certain period
SYSCHAR	System characteristic

4.24.1.1 Mathematical model

The PumpEnergy module calculates the total power (kW) of a multi-pumpstation (two or more pumps in parallel arrangement) for a specified operation scenario. If the time is specified the total energy (kWh) is calculated as well. For one or more discharge ranges, the user specify which pumps are running. The running pumps can be speed controlled or fixed speed. If they are speed controlled, the program calculates the speed in such way that the required head (as defined by the given system characteristic) is satisfied.

The pump curves of pumps in parallel placement are summarised. Intermediate points are determined by linear interpolation.

For a certain discharge the point of intersection of the summarised pump curve with the specified system characteristic is computed. In case of variable speed pumps, the affinity rules are used:

$$\frac{H(N_i)}{H(N_r)} = \left(\frac{N_i}{N_r}\right)^2$$

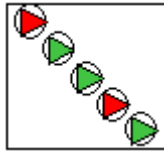
$$\frac{Q(N_i)}{Q(N_r)} = \frac{N_i}{N_r}$$

All variable speed pumps are controlled with the same ratio according:

$$N = N_{\min} + Ratio(N_{\max} - N_{\min})$$

Subsequently the duty point of each individual pump is determined, that means the discharge and head, but also the efficiency and power and, if the duration is known, the energy.

4.24.2 PUMPSCEN (Pump Scenario)



4.24.2.1 Input specifications

description	input	unit	range	default	remarks
Scenario	table				See below

The scenario table contains all non-disused pumps available in the diagram.

Q from (m3/h)	Q to (m3/h)	PUMP 1 (rpm)	PUMP 2 (rpm)	PUMP 3 (rpm)
0	1200	0		
1200	2400			0
2400	4000		1500	0

For each discharge range (phase) the operating pumps are defined:








- value = 0 (zero) means a variable speed pump
- value > 0 means the actual fixed speed
- empty field means pump is not operating

4.24.2.2 Output specifications

If the pump energy can be computed (depends of Q_PATTRN or Q_FREQ component), the energy consumption for each phase is mentioned in the scenario table. A separate output property is used for the total energy amount.

Q from (m3/h)	Q to (m3/h)	PUMP 1 (rpm)	PUMP 2 (rpm)	PUMP 3 (rpm)	Energy (MWh)	Energy ratio (%)

The output properties are:

Messages	Error	
Discharge	(m3/h)	
Pump head	(m)	
Speed	(rpm)	
Efficiency	(%)	
power	(kW)	
Energy	(MWh)	

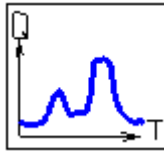
The message property contains messages with respect to all components use for the PumpEnergy calculation;

Next output properties can be visualised in a chart as function of discharge or time (last one only in combination with Q_PATTRN)

- Discharge
- Head
- Speed
- Efficiency
- Power

In the example the coherence of the Pump Energy components are explained.

4.24.3 Q_PATTRN (discharge pattern)



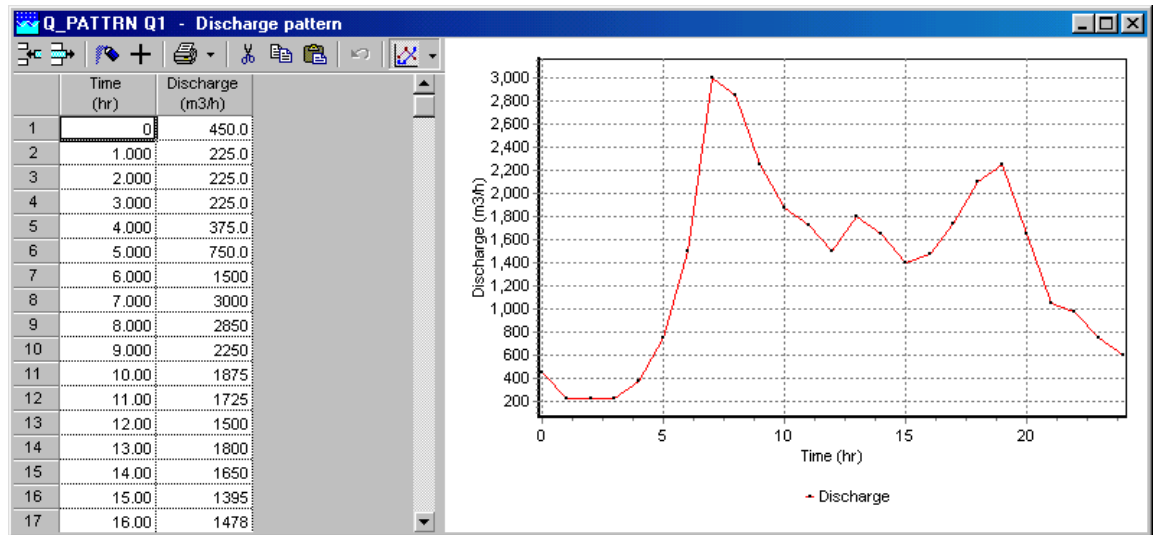
4.24.3.1 Input properties

description	input	unit	range	default	remarks
Discharge pattern	table				See below

The discharge pattern table is a 2-columns table

Time	Discharge
(s)	(m3/s)

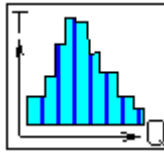
Example:



4.24.3.2 Output properties

None

4.24.4 Q_FREQ (discharge frequency)



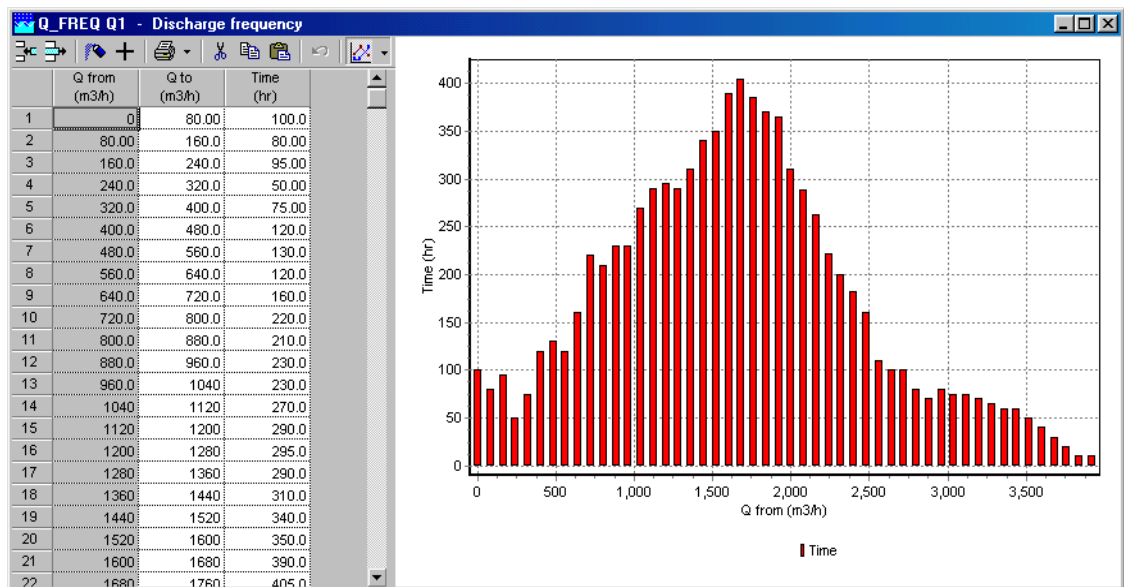
4.24.4.1 Input properties

description	input	unit	range	default	remarks
Discharge frequency	table				See below

The discharge pattern table is a 3-columns table; column one and two are used to specify the range, only the second column can be edit; the first column is automatically filled, starting with zero.

Q from	Q to	Time
(m3/s)	(m3/s)	(s)

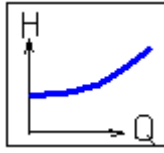
Example:



4.24.4.2 Output properties

None

4.24.5 SYSCHAR (System characteristic)



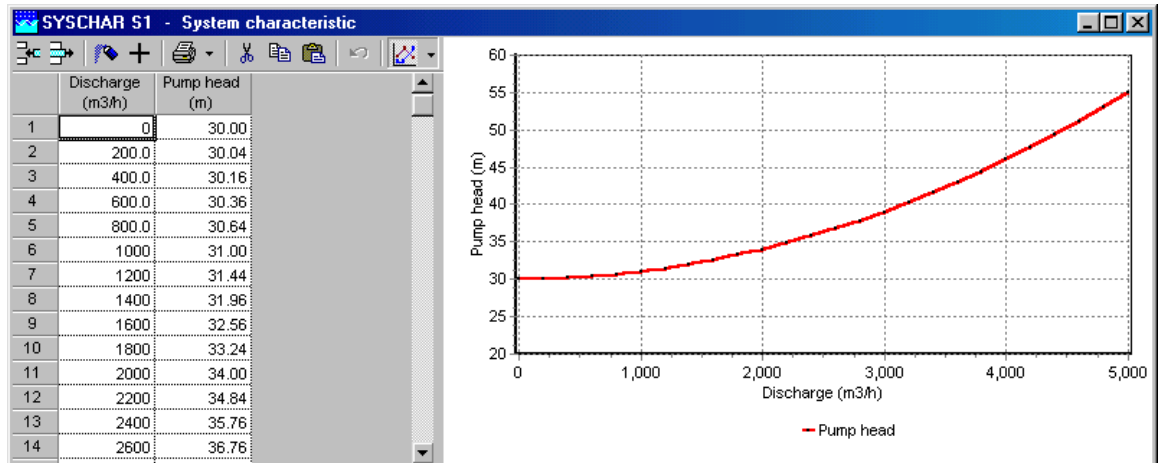
4.24.5.1 Input properties

description	input	unit	range	default	remarks
System characteristic	table				See below

The system characteristic is a 2-columns table;

Discharge (m ³ /s)	Pump head (m)

Example:



The Pump Head is defined as the sum of the static head and dynamic head:

Note: this characteristic can be derived from the System Characteristic module.

See also "System characteristic" on page 111.

Create the wanted system characteristic, show the chart and copy the chart data to the clipboard.

Next, copy the chart data into the SYSCHAR table.

4.24.5.2 Output properties

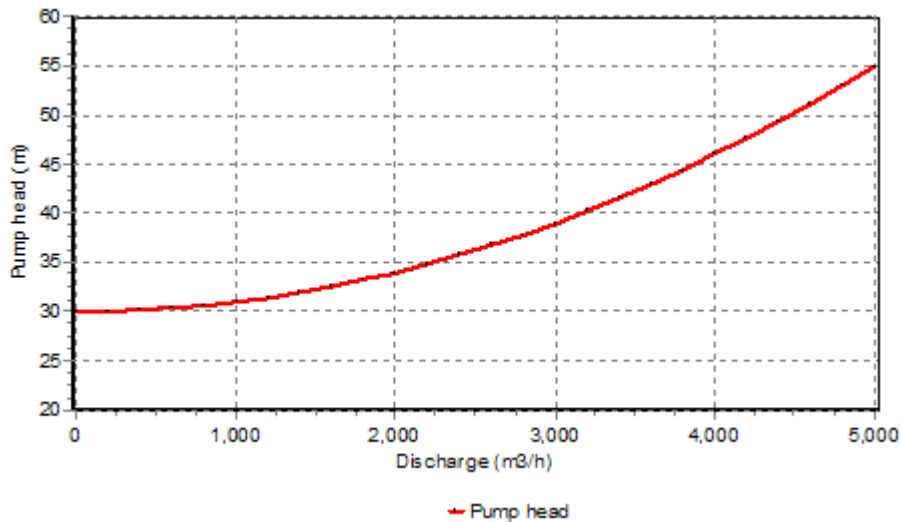
None

4.24.5.3 Example

A pumpstation with three different pumps supplies to a drinking water network. The maximum delivery rate is 4000 m³/h. From history reports the annual delivery ranges are known (see chart given by Q_FREQ)

Note: this example is included in the example directory

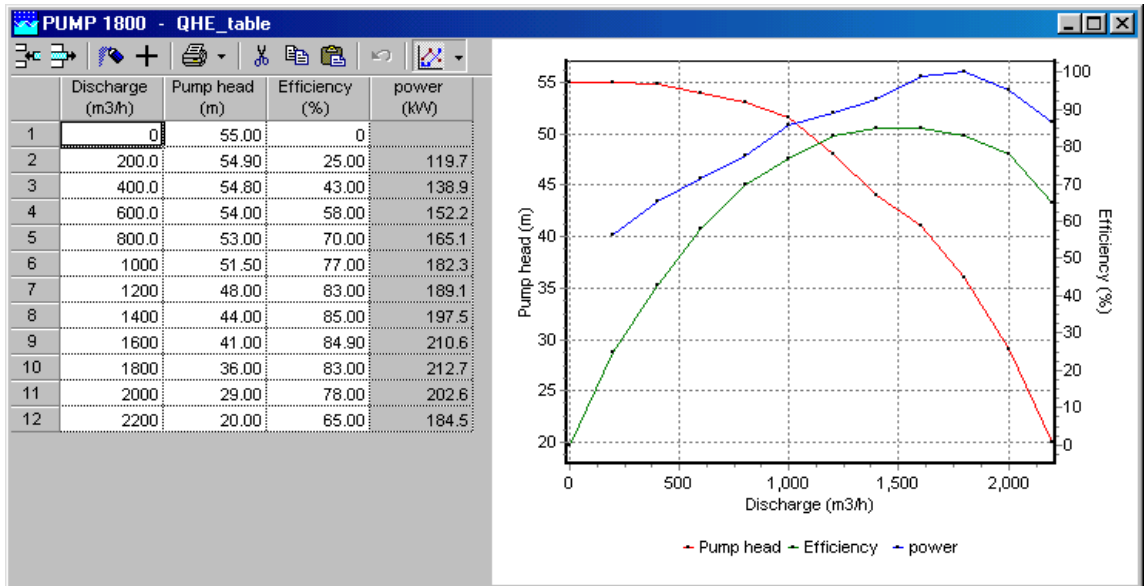
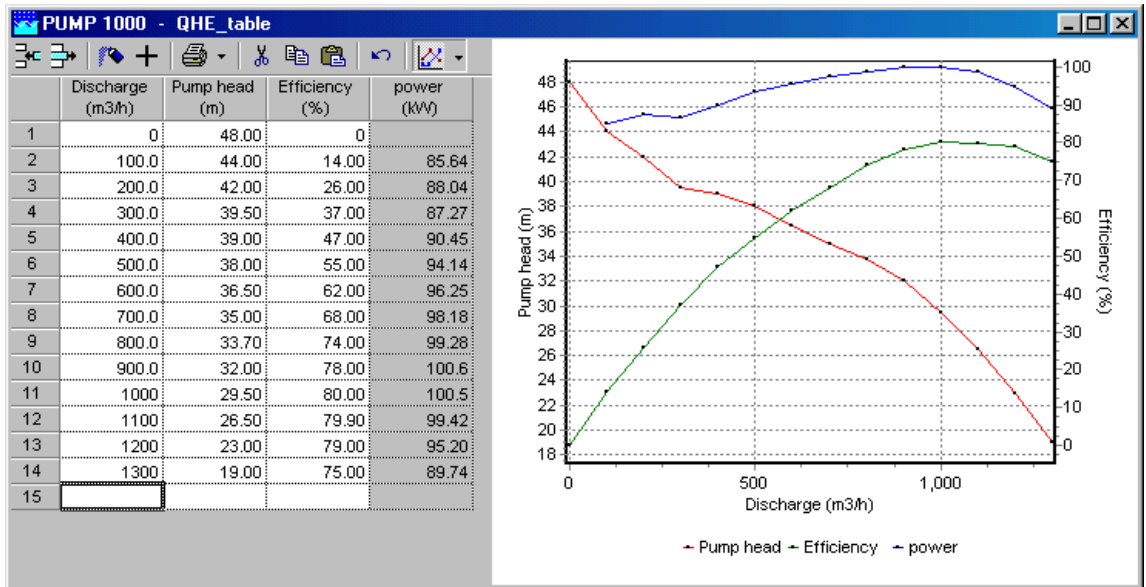
The desired pressure in the network (pressure side pump) is defined by the system characteristic:



Pumpdata:

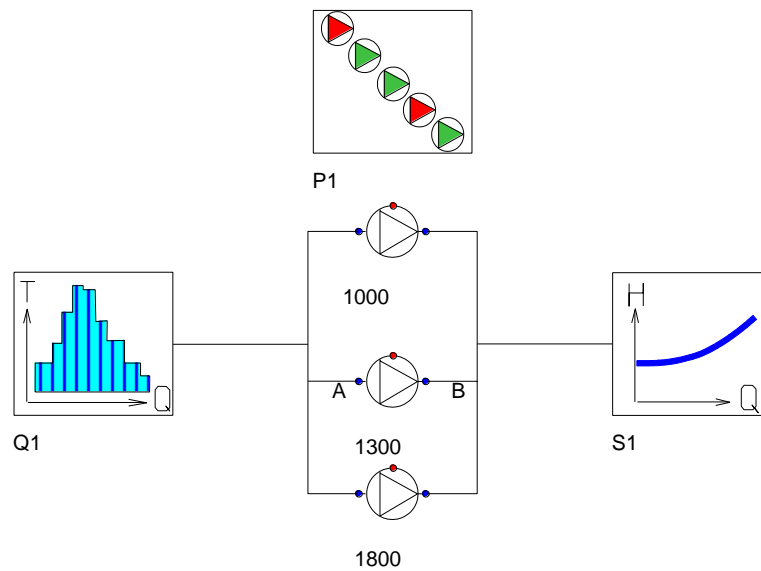
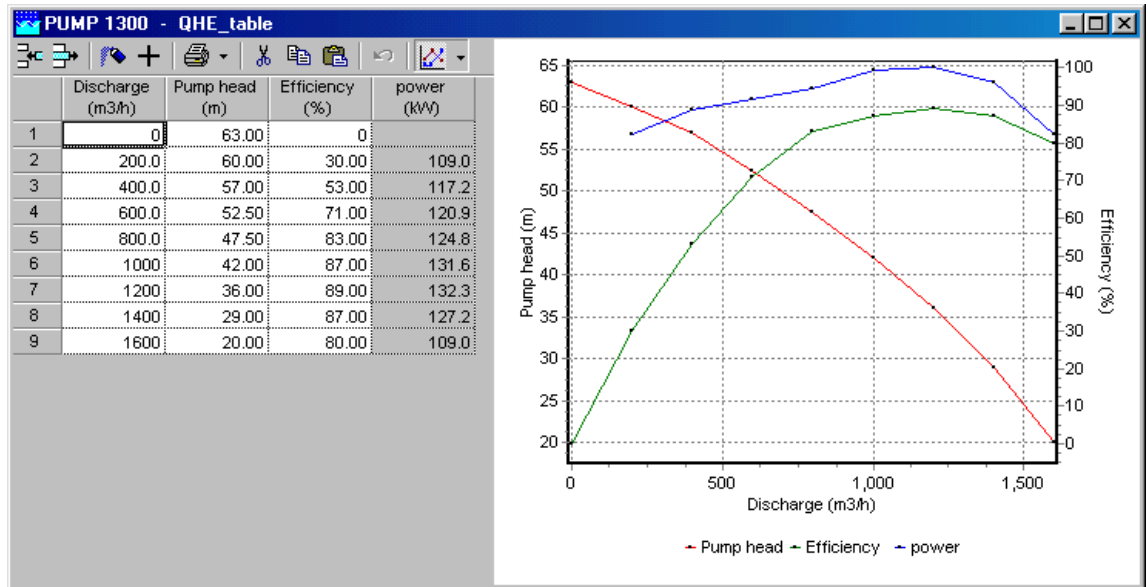
Pump id	design capacity [m3/h]	Rated speed curve [rpm]	min speed [rpm]	max speed [rpm]	max power [kW]
1000	1000	1200	1000	1500	150
1300	1300	1200	1000	1500	200
1800	1800	1200	1000	1500	300

The pump curves are given below:



Question: Optimise the pump scenario.

Step 1: add components into diagram:



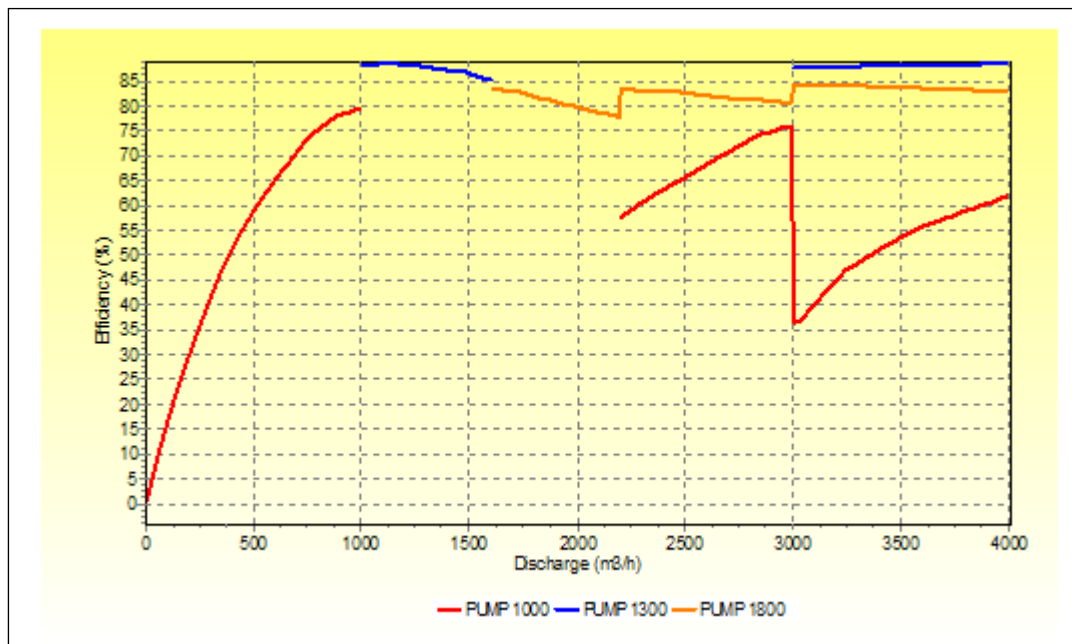
Step 2: input of pump data and system characteristic

Step 3: choose a scenario

	Q from (m3/h)	Q to (m3/h)	PUMP 1000 (rpm)	PUMP 1300 (rpm)	PUMP 1800 (rpm)	Energy (MWh)	Energy ratio (%)
1	0	1000	0			133.6	7.69
2	1000	1600		0		297.7	17.1
3	1600	2200			0	560.8	32.3
4	2200	3000	0		0	448.9	25.8
5	3000	4000	0	0	0	296.3	17.1

The total energy consumption (annual base according specified frequency) is 1737 MWh

Step 4: analyse the scenario output (especially efficient)

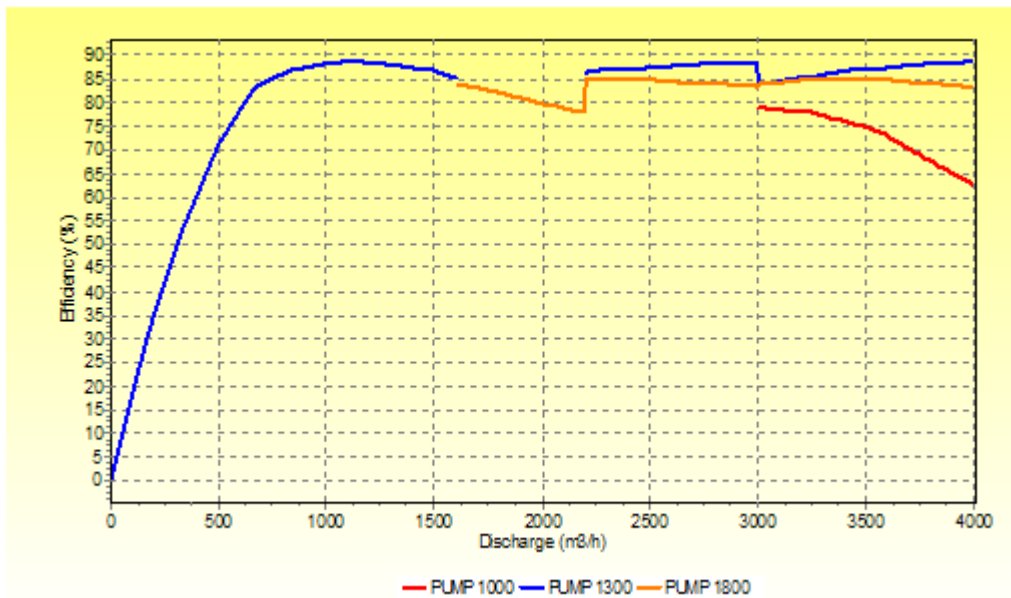


Step 5: adapt the scenario

Pump 1300 has a better efficiency, use this pump also in the first stage and fourth stage instead of Pump 1000. The last stage, use Pump 1000 at a fixed speed of 1350 rpm.

	Q from (m3/h)	Q to (m3/h)	PUMP 1000 (rpm)	PUMP 1300 (rpm)	PUMP 1800 (rpm)	Energy (MWh)	Energy ratio (%)
1	0	1000		0		128.4	7.67
2	1000	1600		0		297.7	17.8
3	1600	2200			0	560.8	33.5
4	2200	3000		0	0	405.6	24.2
5	3000	4000	1350	0	0	280.9	16.8

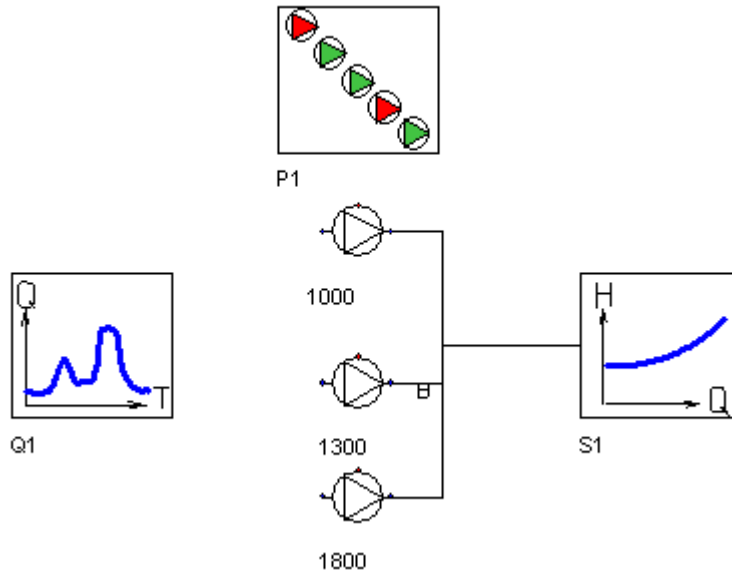
The total energy consumption is now reduced to 1673 MWh; that means a reduction of 4 %.



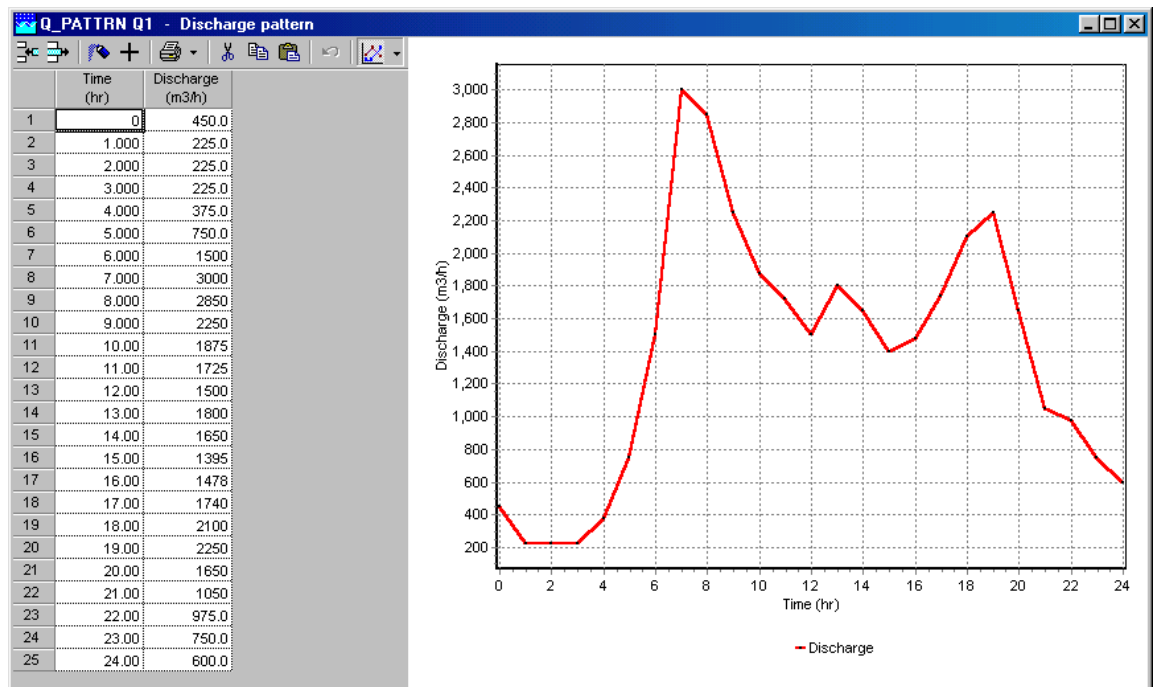
Step 6: analyse a 24 hours period

Replace the Q_FREQ by the Q_PATTRN and specify the discharge pattern

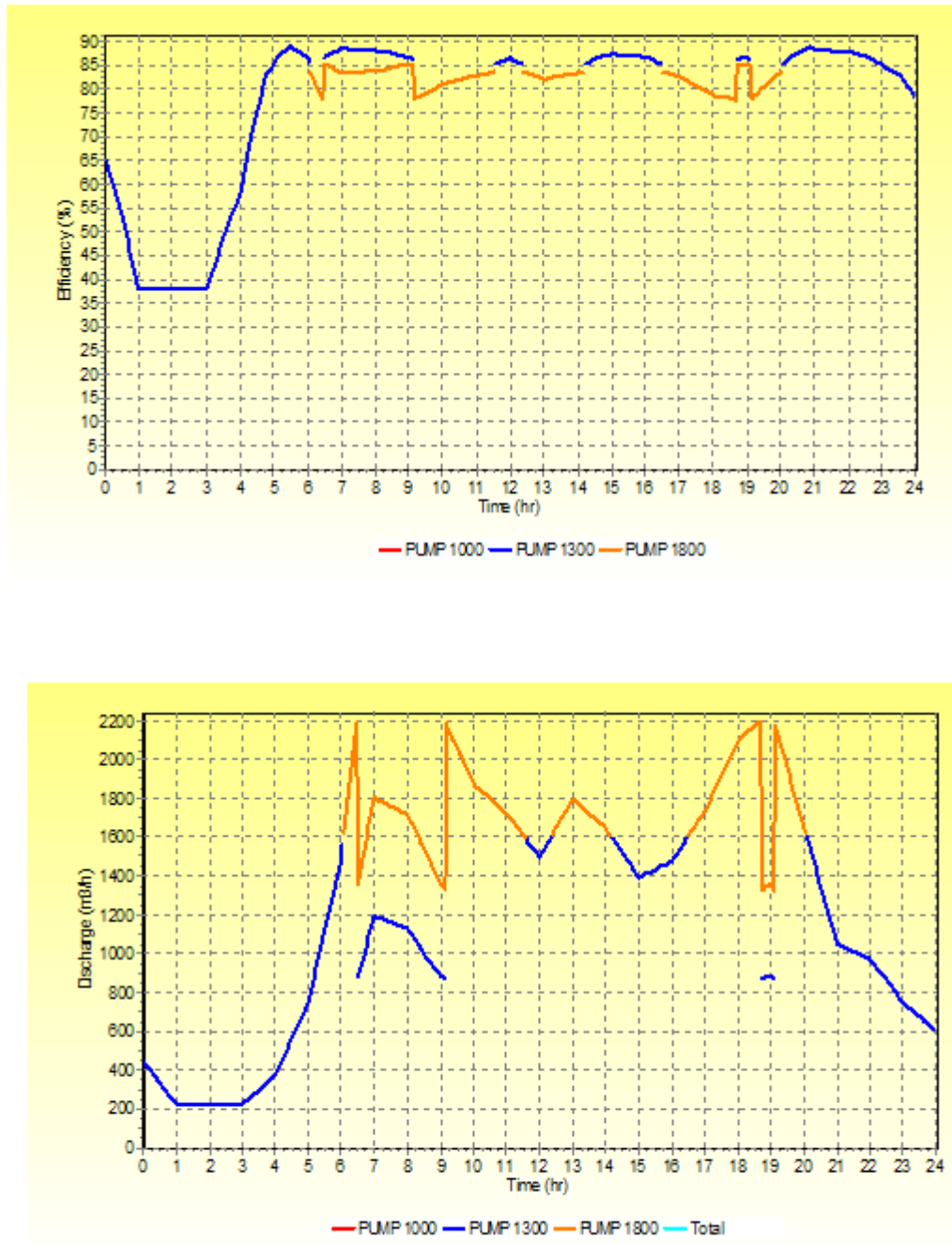
Note: it is not necessary to connect the Q_PATTRN to the pumps



Step 7: Specify data for Q_PATTRN



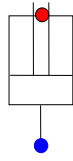
Step 8: analyse the results



Advise: use for the low discharges (0 – 400 m³/h) a pump with lower capacity.

4.25 PUMPSS

4.25.1 PUMPSS (class)



Single cylinder, single-acting pump with mechanically driven piston

Supplier type

type label	description	active
Pump Simplex, single-acting	A single cylinder, with a mechanically driven single-acting piston..	Yes

A single cylinder reciprocating pump model can be created by connecting the cylinder to the node shared by two check valves in series See Figure 1.

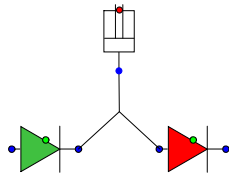


Figure 1: Simplex, single-acting reciprocating pump model

A multi-cylinder pump model can be created by placing multiple instances of these cylinders and check valves in series

4.25.1.1 Mathematical model

Figure 2 shows the geometry of the one cylinder piston pump and its motion.

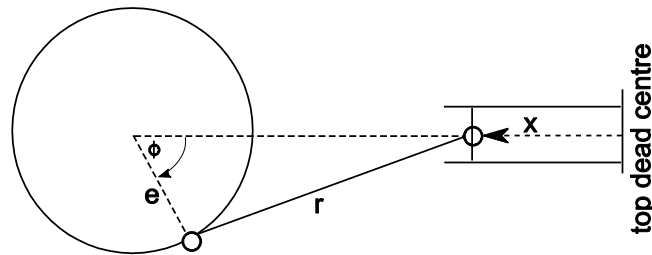


Figure 2: Geometry of piston motion

The crank angle is defined as::

$$\phi = \omega t + c \quad (1)$$

where ϕ is the crank angle relative to the top dead centre (radians), ω is the constant pump speed (radians/sec), t is time and c is the crankshaft offset angle (radians). For a variable pump speed equation (1) can be rewritten as:

$$\phi^n = \phi^{n-1} + \omega^n \Delta t \quad (2)$$

where n denotes the values of the present time-step, $n-1$ denotes the values of the previous time-step, Δt is the time-step length and x is the piston position relative to the top dead centre, e is the crank radius and r is the connecting rod length.

$$x = e(1 - \cos\phi) + r \left(1 - \sqrt{1 - \left(\frac{e}{r} \sin\phi \right)^2} \right) \quad (3)$$

The piston position may also be normalized [-1,+1] as:

$$x_{norm} = \frac{x - e}{e} \quad (4)$$

The cylinder volume V and piston flow rate Q_p (derived from the piston position) is given by:

$$V = V_{TDC} + A_p \left[e(1 - \cos\phi) + r \left(1 - \sqrt{1 - \left(\frac{e}{r} \sin\phi \right)^2} \right) \right] \quad (5)$$

$$Q_p = -\omega A_p e \sin\phi \left[1 + \frac{e \cos\phi}{r \sqrt{1 - \left(\frac{e}{r} \sin\phi\right)^2}} \right] \quad (6)$$

where V_{TDC} is the cylinder volume at top dead centre and A_p is the piston area.

In the steady state there is no piston movement, hence no discharge is entering or leaving the cylinder. This can be written as a residual :

$$Q = 0 \quad (7)$$

During motion the following continuity equation applies:

$$Q = Q_p - \frac{dP}{dt} \frac{V}{K} \quad (8)$$

where Q is the discharge out of the cylinder, Q_p is the volumetric discharge of the piston, P is the pressure inside the cylinder and K is the bulk modulus of the fluid inside the cylinder. Dividing equation by 8 by fluid density and gravity acceleration and integrating fully implicitly in time gives:

$$F^n = H^n - H^{n-1} + (Q^n - Q_p^n) \frac{K \Delta t}{\rho g V^n} = 0 \quad (9)$$

Where H is the pressure head within the cylinder.

The piston pump model does not account for cavitation occurring within the pump cylinder.

4.25.2 PUMPSS (Piston Pump)

4.25.2.1 Hydraulic specifications

Description	input	unit	Range	default	remarks
Initial motor speed	Real	rad/s			
Initial phase	Real	radians	0 - 2π	0	
Cylinder volume	Real	m ³			volume at top dead centre
Piston area	Real	m ²			
Crank radius	Real	m			
Connecting rod length	Real	m			

4.25.2.2 Component specific output

Motor speed [rad/s]

Piston position (normalized) [-]

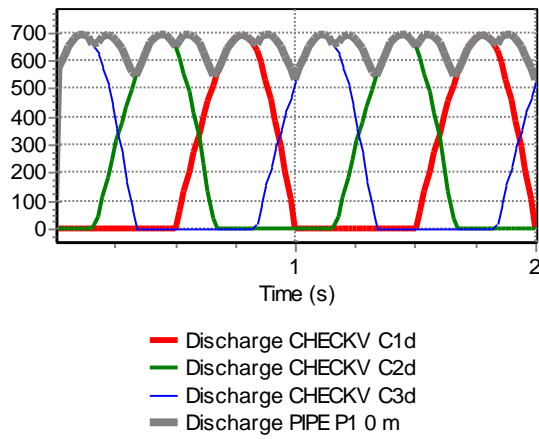
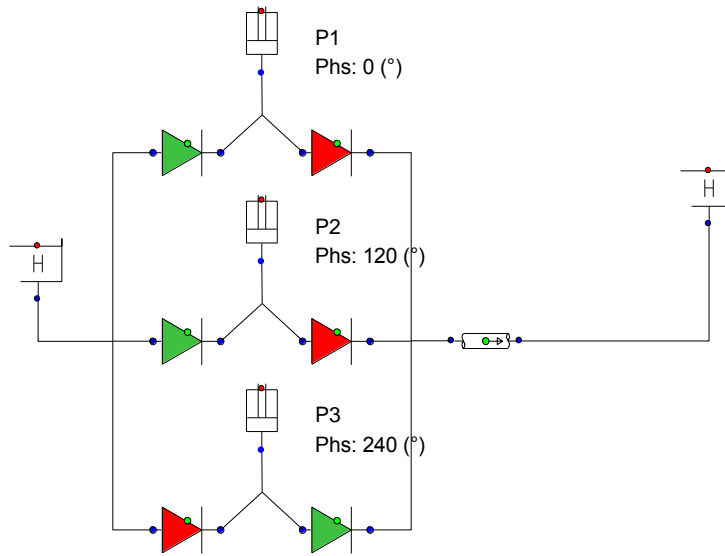
Crank shaft angle [radians]

4.25.2.3 H-actions

Example:

To simulate a duplex, triplex or any other multicylinder pump, multiple instances of the PUMPSS component together with both checkvalves need to be placed parallel, with each cylinder having a unique phase angle.

The example below shows a triplex piston pump, with initial phase angle of respectively 0°, 120° and 240°.

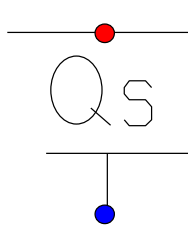


4.25.2.4 Component messages

Message	Type	Explanation
Time-step should be smaller than reciprocating pump period	Error	Time-step is too large

4.26 QSCHAR

4.26.1 QSCHAR (Class)



Varying discharge boundary condition used to determine the system characteristic

Supplier type

type label	description	active
QSCHAR	varying discharge to determine the system characteristic	yes

4.26.1.1 Mathematical model

This hydraulic component is used to determine the system characteristic of (a part of) the hydraulic system. The system characteristic is a non-linear function $H=f(Q)$ at a certain location.

For simple systems this curve can be derived easily while the coefficient C in $H = C Q^2$ is constant. For branched systems where other head or flow dependant devices (like pumps) influence the head this coefficient C is not constant. The curve can be determined exactly by WANDA using a repeating calculation of steady.

Each calculation step the QSCHAR acts as a BOUNDQ component with a constant value. This value varies from 0 m³/s until Q_{max} m³/s in 25 equidistant steps

This H-component is especially used in the System characteristic module.

See help on the Model Menu "System characteristic chart" on page 109.

Running the modules Steady and Unsteady the QSCHAR exactly behaves like the BOUNDQ component.

For more information see "BOUNDQ (class)" on page 224.

4.26.1.2 Hydraulic model aspects

Generally the QSCHAR represents a (set of) pumps. If you want to use this component in a model where the pump station is already modelled, it is not necessary to delete these pumps, check valves and H-nodes. Just toggle the “disuse” property of the pumps from “No” to “Yes”. Now the pumps are ignored in the calculation

Be careful using the QSCHAR component together with TAPS or other discharge prescribed components. Calculating the 25 Q-steps the values of the other discharge prescribed components are kept constant. These values are not scaled.

A model with only discharge prescribed components and no Head boundary condition at all will lead to a numerical error (singular matrix)

4.26.2 QSCHAR (BoundQ for system characteristic)

4.26.2.1 Hydraulic specifications

Description	input	unit	range	default	remarks
maximum discharge	real	[m ³ /s]			
suction level	real	m			
Marker Qmin	real	[m ³ /s]			
Marker Qmax	real	[m ³ /s]			

See also "Mathematical model" on page 385.

4.26.2.2 Component specific output

None

4.26.2.3 H-actions

In principal the QSCHAR is used for the system characteristic calculation. This is a repeating steady calculation. In transient mode the QSCHAR behaves like a BOUNDQ.

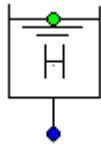
See “BOUNDQ (class)” on page 224.

4.26.2.4 Component messages

Pump head (m)

4.27 Reservoir

4.27.1 RSVOIR (class)



Pressure head boundary condition

Supplier type

type label	Description	active
RSVOIR (reservoir)	Constant or time-dependent head	No

4.27.1.1 Mathematical model

A RSVOIR prescribes the head in a certain point of the system.

$$H = f(t) \tag{1}$$

The head (level) is constant during the steady state calculation. During the transient calculation the head varies in time in accordance with the net inflow into the reservoir.

The RSVOIR is a reasonable model, if the storage area is relatively small (compared with the net in- or outflow during the transient calculation). If the level variation during the transient calculation is negligible, the component "BOUNDH (reservoir)" on page 223 should be used instead.

To achieve the specified head the RSVOIR has to supply or consume fluid to or from the system. The discharge will be determined by the continuity equation for the H-node to which the RSVOIR is connected. If a RSVOIR is located between two pipes it actually decouples the system: the pipes are behaving independently of each other and waterhammer waves will reflect completely (reflection factor -1).

4.27.2 RSVOIR (reservoir)

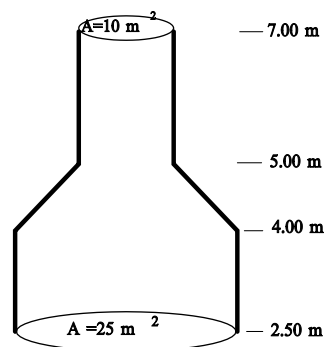
4.27.2.1 Hydraulic specifications

description	input	unit	range	default	remarks
Area type	Constant /Variable	[m]			
Area	Real	[m ²]			If Area type = constant
Height-area table	Table				If Area type = variable
Head at t = 0	Real	[m]			

See also "Mathematical model" on page 387.

Note: While the area of the RSVOIR is limited, the volume is not. When specifying a height-area table, the values at the lowest and highest point are assumed to extend to respectively infinite depth and height.

Example of table



height [m]	area [m ²]
2.50	25.0
4.00	25.0
5.00	10.0
7.00	10.0

4.27.2.2 *Component specific output*

None

4.27.2.3 *H-actions*

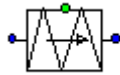
None

4.27.2.4 *Component messages*

None

4.28 Resistance

4.28.1 RESIST (class)



Local resistance or loss

Fall type

type label	Description	active
Resist quadr. xi	Dimensionless -value from loss equation	No
Resist quadr. C	-value from loss equation	No
Resist linear C	-value from loss equation $\Delta H = CQ$	No
Resist quadr. Initial Q	Initial is specified, from loss equation is computed.	No
Resist 2-way quadr.xi	Dimensionless -value from loss equation -value depends on flow direction	No
Resistance polynomial	Resistance according user-defined polynomial	No

4.28.1.1 Mathematical model

A resistance is modelled by the following equations:

$$F1: H_1 - H_2 = a\xi Q_1 |Q_1| \quad (1)$$

in which:

H_1	=	Upstream head	[m]
H_2	=	Downstream head	[m]
a	=	$1/(2g A_r^2)$	[s ² /m ⁵]
ξ	=	loss coefficient	[-]
Q_1	=	discharge through RESIST	[m ³ /s]
A_r	=	discharge area resistance	[m ²]

The discharge area does not have to be equal to the area of the adjacent pipes.

The area is calculated from the resistance diameter D_r according to

$$A_r = \pi D_r^2 / 4 \quad (2)$$

Tip:

The RESIST can be used to model all kinds of resistant components without a relevant length, e.g.:

- A number of bends
- Entrance, exit losses
- Filters
- Heat exchangers, etc.

4.28.2 Resist quadr. xi

4.28.2.1 *Hydraulic specifications*

Description	input	unit	range	default	remarks
inner diameter	real	[m]	(0-5]		
loss coefficient	real	[-]	[0-100]		

See also "Mathematical model" on page 390.

4.28.2.2 *Component specific output*

None

4.28.2.3 *H-actions*

None

4.28.2.4 *Component messages*

None

4.28.3 Resist quadr. C

4.28.3.1 Hydraulic specifications

description	input	Unit	range	default	remarks
C-value ($\Delta H=CQ^2$)	real	[s ² /m ⁵]	[0-100]		

See also "Mathematical model" on page 390.

4.28.3.2 Component specific output

None

4.28.3.3 H-actions

None

4.28.3.4 Component messages

None

4.28.4 Resist linear C

4.28.4.1 Hydraulic specifications

description	input	unit	range	default	remarks
linear C-value ($\Delta H=CQ$)	real	[s/m ²]	[0-100]		

See also "Mathematical model" on page 390.

4.28.4.2 Component specific output

None

4.28.4.3 H-actions

None

4.28.4.4 *Component messages*

None

4.28.5 Resist quadr. Initial Q

4.28.5.1 Hydraulic specifications

description	input	unit	range	default	remarks
initial discharge	real	[m ³ /s]	(0-10]		

See also "Mathematical model" on page 390.

4.28.5.2 Component specific output

None

4.28.5.3 H-actions

None

4.28.5.4 Component messages

Message	Type	Explanation
C-value (resistance) = [s ² /m ⁵]	Info	

4.28.6 Resist 2-way quadr.xi

4.28.6.1 Hydraulic specifications

Description	input	unit	range	default	remarks
inner diameter pos flow	real	[m]	(0-5]		
inner diameter neg flow	real	[m]	(0-5]		
loss coeff. pos flow (xi+)	real	[-]	[0-100]		
loss coeff. new flow (xi-)	real	[-]	[0-100]		

See also "RESIST" on page 390

4.28.6.2 Component specific output

None

4.28.6.3 H-actions

None

4.28.6.4 Component messages

None

4.28.7 Resistance polynomial

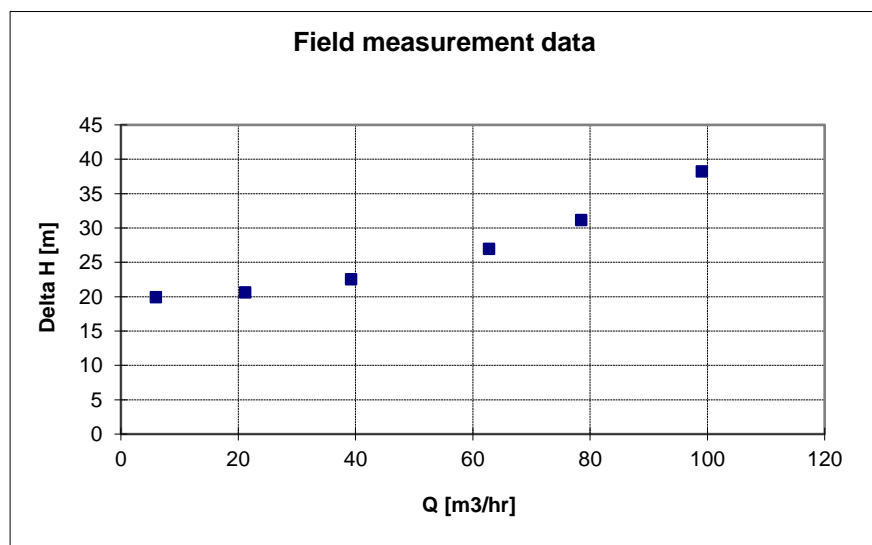


Fall type

type label	description	active
Resistance polynomial	Resistance described by second order polynomial	no

4.28.7.1 Mathematical model

The resistance of a particular hydraulic object is mostly a non-linear function. If field measurements are carried out, the pressure head difference and related discharge can be shown in a chart, see figure below.



The function of this resistance can be described by a second order polynomial:

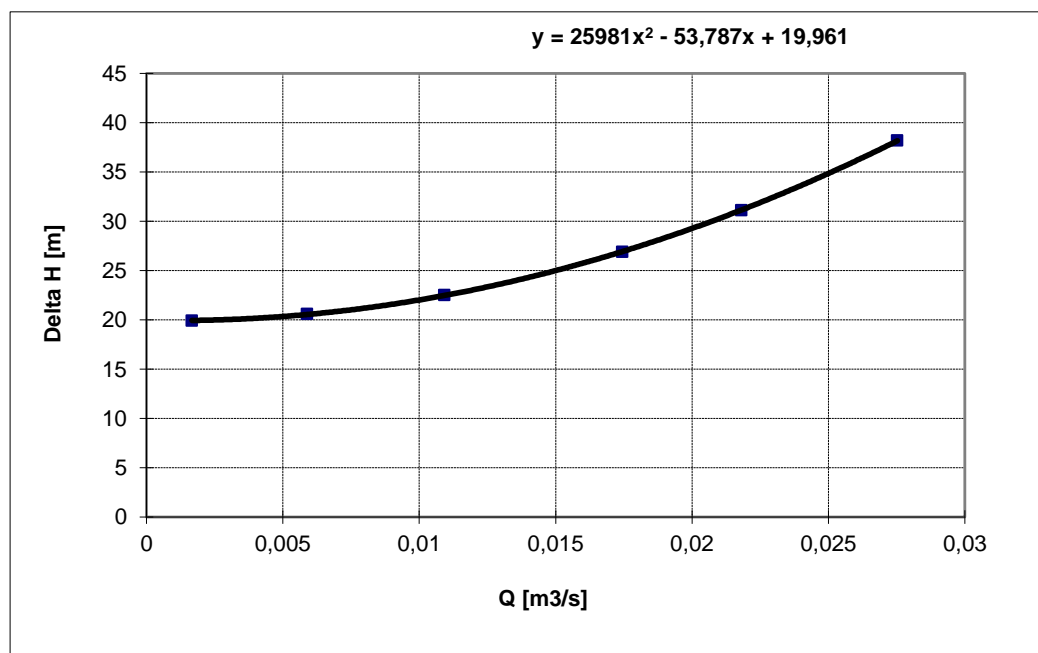
- $\Delta H = a + bQ + cQ|Q|$ (1)

in which:

- ΔH = pressure drop $H_1 - H_2$ [m]
- Q = Discharge through resist [m^3/s]
- a, b, c = coefficients [-]
- H_1 = upstream pressure head [m]
- H_2 = downstream pressure head [m]

The coefficients a , b , and c can be derived e.g. using EXCEL (insert trendline). The coefficient a is the static pressure head.

In the figure below, the derived function together with the measured points are drawn. Note that the coefficients are based on SI-units.



Remark:

The polynomial is valid for positive and negative flow.

Note that the coefficient “A” is a constant head loss and is only valid for a positive flow direction. That means that with a negative flow, the headloss contribution “A” (defined as $H_1 - H_2$) is the same as for positive flow.

4.28.7.2 Hydraulic specifications

Description	input	unit	range	default	remarks
Coefficient a in $\Delta H = a + bQ + cQ Q $	real	[-]	$(-10^8 - 10^8)$		
Coefficient b in $\Delta H = a + bQ + cQ Q $	real	[-]	$(-10^8 - 10^8)$		
Coefficient c in $\Delta H = a + bQ + cQ Q $	real	[-]	$(-10^8 - 10^8)$		

4.28.7.3 Component specific output

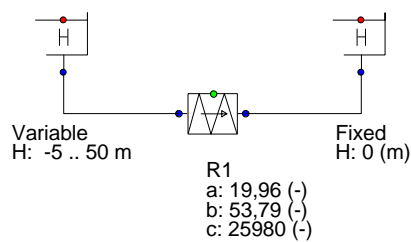
None

4.28.7.4 H-actions

None

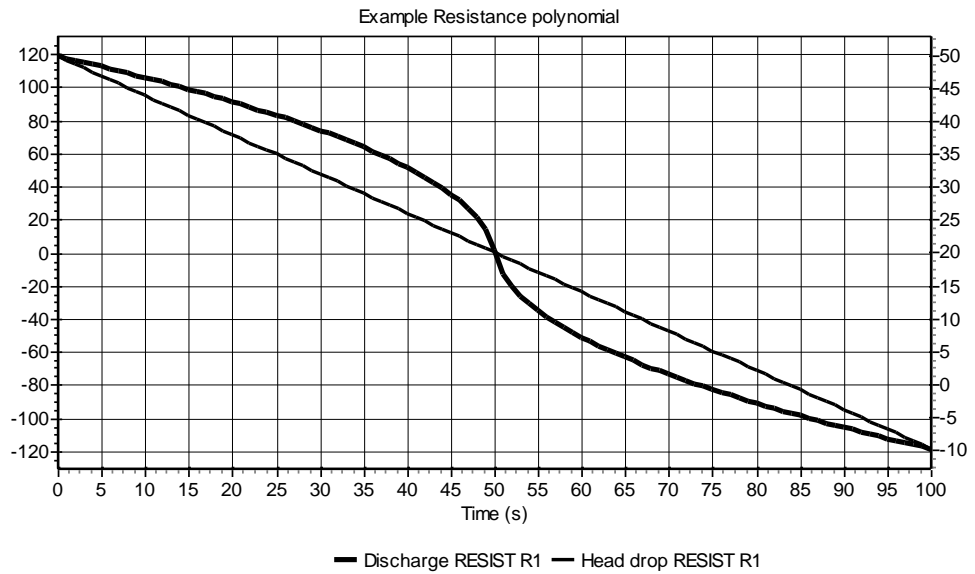
4.28.7.5 Component messages

None

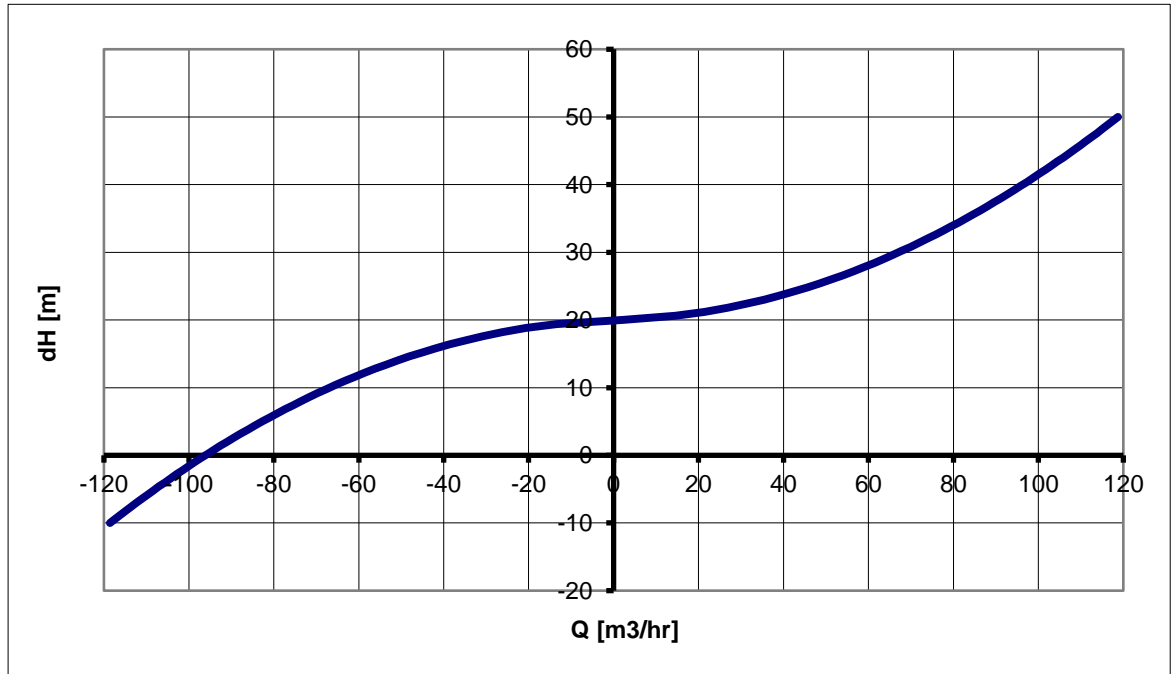
4.28.7.6 Example

The level in the upstream reservoir varies from 50 m to -5 m; the level in the downstream reservoir remains constant at $H = 0$ m.

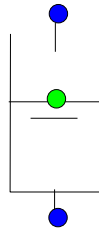
Next figure shows the discharge and level as functions of time.



The next chart shows the pressure head ΔH related to the discharge Q . The WANDA results are exactly the same as the theoretical polynomial.



4.29 Shaft



Fall type supplier

type label	description	active
Shaft	Shaft with inflow.	No

The shaft models a storage element with inflows at two elevations. The edge level of the inflow at the top is an input parameter of the shaft. The other inflow is at the bottom of the shaft and corresponds to the node elevation, which should therefore be lower than the elevation of the edge level. The shaft level equals the downstream head.

A shaft is typically applied to model steep pipe sections that may drain or fill. However the pipe inertia and friction are neglected in this model. During transients the shaft behaves like a surge tower with height-dependent storage area.

4.29.1 Mathematical model

Based on the inflow over the edge, different conditions for the component can be derived:

1. submerged (level in the shaft, H_2 , above edge level, Z_{edge})
2. partially filled (level in the shaft below edge level, inflow over edge)
3. drained at top (level in the shaft below edge level, no inflow over edge)

The basic equations governing the hydraulics of the shaft depend on the condition:

Governing equations

For all conditions:	$Q_1 - Q_2 = A \frac{dH_2}{dt}$	(1)
---------------------	---------------------------------	-----

Furthermore, the following equations in the different states

Submerged:	$H_1 = H_2 > z_{edge}$	(2)
partially filled:	$H_1 = z_{edge} ; H_2 < z_{edge}$	(3)
drained at top:	$Q_1 = 0 , H_1, H_2 < z_{edge}$	(4)

Head H1 is connected to the node with the inflow edge, head H2 is connected to the bottom of the shaft. z_{edge} is the elevation of the edge level relative to the reference plane of the WANDA model. The shaft is drained at top when the head in the node H1, is lower than the edge level elevation of the shaft.

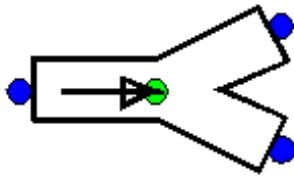
4.29.2 Hydraulic specifications

Description	input	unit	range	default	remarks
height-area table	table				height in metres relative to the reference plane
Initial state	toggle	[-]	partially filled; submerged; Drained top; $Q_2=0$	partially filled	see below

The initial state defines how the shaft is operated during steady state calculations, as indicated in the table below.

Initial state	Description
partially filled	Initial shaft level is lower top edge level. Upstream head equals the top edge or is higher, such that inflow via the edge takes place. The shaft level is determined by the downstream system such that the net inflow during the steady state equals 0.
submerged	The shaft is pressurised if the shaft level exceeds the top edge. Upstream and downstream head are equal. The net inflow in the steady state is 0.
Drained top	There is no flow through the shaft. The upstream head is less than or equal to the top edge. The shaft level is determined by the downstream system.

4.30 Splitter



type label	description	active
2-splitter	Three-node junction which splits up a combined flow into 2 separate branch flows. Cross section can be circular or rectangular.	no

The splitter differs from the Y-junctions because there is no straight main flow. The area of the three legs may be different.

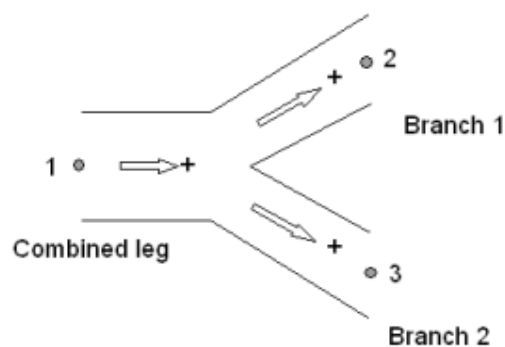
Instead of negative flow the Splitter component acts as a Merger component.

4.30.1 Mathematical model

4.30.1.1 Positive flow definition

The definition for the positive direction of flow (and velocity) together with the location of the connect point numbers is important to understand the results in the property window. Notice that the connect point numbering of the Splitter differs from that of the Merger.

For the Splitter-**junction** the positive flow definition and connect point numbers are defined in the figure below.

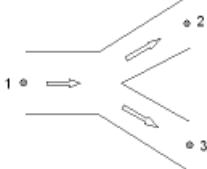
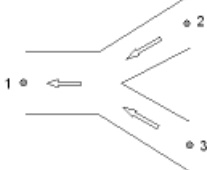
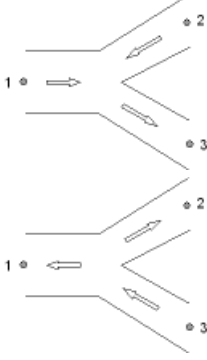


4.30.1.2 Flow regimes

The Splitter supports only the positive dividing flow and for negative flow the merging variant. The other flow regimes are not supported and for these circumstances the local losses are ignored, that means the splitter becomes frictionless.

In the table below the different flow regimes are explained with a scheme with corresponding actual flow arrows.

Table 1: definition of flow regimes for both types

<p>1 Splitting flow</p>	 <p>Positive flow</p>
<p>2 Merging flow</p>	 <p>Negative flow</p>
<p>Not supported flow regimes</p>	

4.30.1.3 Equations

The head loss over a 3-node component depends on the distribution of the discharges and the area of the connected legs. The Splitter supports only the user specified resistance coefficient X_i (ξ) depending on discharge ratio's for a fixed area ratio.

The head loss in 3-node components is a function of the combined flow in leg 1. Including the continuity equation (no production or loss of mass in the component) the general set of the three equations of the Y-junction is:

Mass continuity	$f_1 = Q_1 - Q_2 - Q_3 = 0$
-----------------	-----------------------------

Head loss between combined leg and branch 1	$f_2 = H_1 - H_2 - \xi_{12} \frac{Q_1 Q_1 }{2gA_1^2}$
Head loss between combined leg and branch 2	$f_3 = H_1 - H_3 - \xi_{13} \frac{Q_1 Q_1 }{2gA_1^2}$

Where:

Q_i = total discharge in leg i	[m ³ /s]
H_i = energy head in connect point i	[m]
A_i = pipe area leg i (leg with total flow)	[m ²]
ξ_{ij} = loss coefficient between point i and j	[-]

The subscripts (1), (2) and (3) correspond to the different legs.

For negative merging flow the formulas of the Merger component will be used with Q_1 taken into account instead of Q_3 .

For the not supported flow regimes the head loss equation becomes $H_1 - H_2 = 0$ and $H_1 - H_3 = 0$

4.30.2 Splitter properties

The input and output properties for both types are specified below.

Input properties

Description	input	unit	range	default	Remarks
Cross section	Circular Rectangular			Circular	
Diameter combined leg	Real	[mm]			If Cross section = Circular
Diameter split branch 1	Real	[mm]			If Cross section = Circular
Diameter split branch 2	Real	[mm]			If Cross section = Circular
Width combined leg	Real	[mm]			If Cross section = Rectangular
Height combined leg	Real	[mm]			If Cross section = Rectangular
Width split branch 1	Real	[mm]			If Cross section = Rectangular
Height split branch 1	Real	[mm]			If Cross section = Rectangular
Width split branch 2	Real	[mm]			If Cross section = Rectangular
Height split branch 2	Real	[mm]			If Cross section = Rectangular
Xi tables valid for	Splitting				

Both		
Xi splitting branch 1	Table (ξ_{12})	if Xi tables = Splitting or Both
Xi splitting branch 2	Table (ξ_{13})	if Xi tables = Splitting or Both
Xi merging branch 1	Table (ξ_{21})	if Xi tables = Both
Xi merging branch 2	Table (ξ_{31})	if Xi tables = Both

Example of Xi table; splitting branch 1 (ξ_{12})

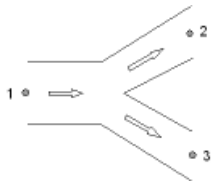
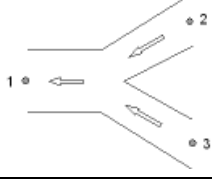
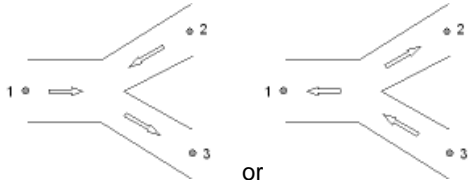
valid for Area ratio = 0.5

Discharge ratio [-]	Xi [-]
0.0	0.90
0.2	0.12
0.4	0.39
0.6	0.92
0.8	1.48
1.0	2.08

Component specific output properties

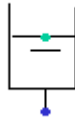
Loss coefficient branch 1 [-]	The ξ_{12} for positive flow and ξ_{21} for negative flow
Loss coefficient branch 2 [-]	The ξ_{13} for positive flow and ξ_{31} for negative flow
Head loss branch 1 [m]	The ΔH_{12} for positive flow and ΔH_{21} for negative flow
Head loss branch 2 [m]	The ΔH_{13} for positive flow and ΔH_{31} for negative flow

Component messages

Message	Type	Explanation
Splitting flow	Info	
Merging flow	Info	
Not supported flow regime - no resistance used	Info	

4.31 Surge tower, surge shaft

4.31.1 SURGTW (class)



Surge tower, surge shaft

Supplier type

type label	Description	active
Surgetower constant area	Storage area is independent of liquid height (pressure head)	No
Surgetower height dep. area	Storage area is a function of liquid height (pressure head)	No

4.31.1.1 Mathematical model

A surge tower or shaft or open tank (atmospheric pressure above fluid level) with constant area supplies fluid to the system according to the differential equation:

$$Q = -A_{tw} dH / dt \quad (1)$$

in which:

Q	=	Discharge from the tower to the system	[m ³ /s]
A _{tw}	=	Storage surface of the tower	[m ²]
H	=	fluid level in the tower	[m]

In case the area of the surge tower is a function of height the following differential equation has to be applied:

$$Q = -dV_f / dt \quad (2)$$

in which:

V _f	=	Volume of the fluid in the surge tower	[m ³]
----------------	---	--	-------------------

In steady state the surge tower does not supply fluid. In that case the equation is simply:

$$Q = 0 \quad (3)$$

4.31.2 Surgetower constant area

4.31.2.1 *Hydraulic specifications*

description	Input	unit	range	default	remarks
area	Real	[m ²]	(0-500]		

See also "Mathematical model" on page 406.

4.31.2.2 *Component specific output*

Fluid level [m]

4.31.2.3 *H-actions*

None

4.31.2.4 *Component messages*

None

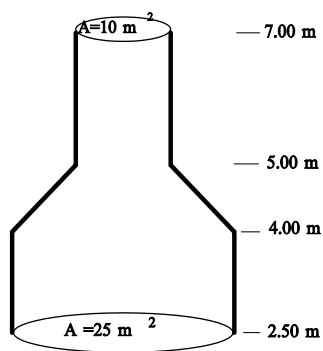
4.31.3 Surgetower height dep. area

4.31.3.1 Hydraulic specifications

description	input	unit	Range	default
height-area table				

See also "Mathematical model" on page 406.

Example of table



height [m]	area [m ²]
2.50	25.0
4.00	25.0
5.00	10.0
7.00	10.0

4.31.3.2 Component specific output

Fluid level [m]

4.31.3.3 H-actions

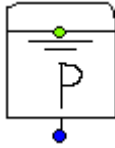
None

4.31.3.4 Component messages

None

4.32 TANK

4.32.1 TANK (class)



Pressure boundary condition

Supplier type

type label	description	active
TANK (constant pressure)	Pressurised tank with constant gas pressure (e.g. floating roof tank)	No

4.32.1.1 Mathematical model

A TANK prescribes the pressure in a certain point of the system.

$$H_l = \frac{P_{set}}{\rho_f g} + h \quad (1)$$

The gas pressure remains constant during the steady state and transient calculation. During the transient calculation the fluid level varies in time in accordance with net inflow into the tank. The TANK is a reasonable model, if the gas volume is relatively large (compared with the net in- or outflow during the transient calculation). If the gas volume is of the same order of magnitude as the net inflow, the component "Air Vessel" (see page 199) should be used instead.

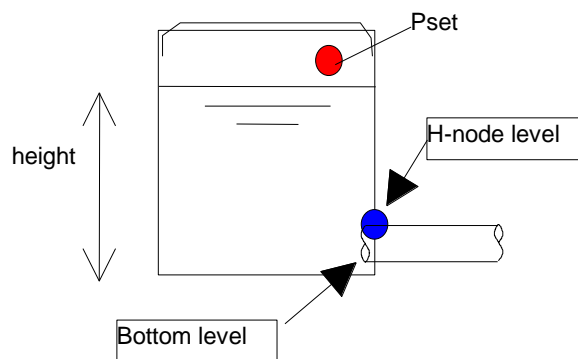
The discharge is determined by the continuity equation for the H-node to which the TANK is connected. If a TANK is located between two pipes it actually decouples the system: the pipes are behaving independently of each other and waterhammer waves will reflect completely (reflection factor -1).

4.32.2 TANK (constant pressure)

4.32.2.1 Hydraulic specifications

description	input	unit	range	default	Remarks
Set pressure at t = 0	real	[Pa]			
Area	real	[m ²]			
Level bottom	real	[m]			
Height fluid column	real	[m]			

See also "Mathematical model" on page 409.



4.32.2.2 Component specific output

None

4.32.2.3 H-actions

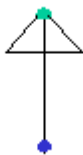
None

4.32.2.4 Component messages

None

4.33 Tap point / distribution area

4.33.1 TAP (class)



Tap point or distribution area

Supplier type

type label	description	active
Tap return flow	If system head drops below downstream head, TAP supplies back to system.	No
Tap non-return flow	If system head drops below downstream head, TAP does not supply back to system.	No
Tap non-return flow with dP for reopen	If system pressure drops below downstream pressure, TAP does not supply back to system. Supply to system (re)starts when a minimum required pressure is exceeded.	No

4.33.1.1 Mathematical model

In steady state the governing equation of this component is very simple:

$$Q = -Q_D \tag{1}$$

in which:

- Q = system discharge to the TAP [m³/s]
- Q_D = delivery rate [m³/s]

The negative sign appears because of the convention that a flow out of the system is defined negative. After obtaining the steady state solution the head loss coefficient is computed with:

$$C = (H - H_D) / Q^2 \tag{2}$$

in which:

- C = head loss coefficient [s²/m⁵]
- H = system pressure head [m]
- H_D = downstream pressure head [m]

The downstream head is either given by the user or for the Tap non-return flow 2 based on the downstream pressure specified by the user.

This head loss coefficient is kept constant throughout the unsteady state computation.

The governing equation then is:

$$H - H_D = C Q |Q| \tag{3}$$

For the **return** type of the TAP component equation (3) is valid for positive as well as negative values of Q. For the **non-return** type equation (3) is only valid for positive values of Q. In case Q becomes negative the component changes from H-phase and closes like a check valve. The governing equation the simply is:

$$Q = 0 \tag{4}$$

The non-return tap opens again if $H > H_D$. For the Tap non-return flow 2 an additional pressure drop can be set which is required to open the tap.

4.33.2 Tap return flow

4.33.2.1 Hydraulic specifications

Description	Input	unit	range	default	remarks
downstream head	Real	[Pa]			
initial delivery rate	Real	[m³/s]	(0-5]		

See also "Mathematical model" on page 411.

4.33.2.2 Component specific output

None

4.33.2.3 H-actions

None

4.33.2.4 Component messages

Message	Type	Explanation
System pressure head (..... m) is lower than delivery head (..... m)	Error	Initially the tap discharge must be out of the system. Since the computed head loss coefficient may not be negative the system pressure head must be higher than the delivery (downstream) head.

See also "Mathematical model" on page 411.

4.33.3 Tap non-return flow

4.33.3.1 Hydraulic specifications

description	input	unit	range	default	remarks
downstream head	Real	[m]			
initial delivery rate	Real	[m ³ /s]	(0-5]		

See also "TAP" on page 411.

4.33.3.2 Component specific output

None

4.33.3.3 H-actions

None

4.33.3.4 Component messages

Message	Type	Explanation
System pressure head (..... m) is lower than delivery head (..... m)	Error	Initially the tap discharge must be out of the system. Since the computed head loss coefficient may not be negative the system pressure head must be higher than the delivery (downstream) head.
opens	Info	
closes	Info	

4.33.4 Tap non-return flow with DP reopen

4.33.4.1 Hydraulic specifications

description	input	unit	range	default	remarks
downstream pressure	Real	[Pa]			
initial delivery rate	Real	[m ³ /s]	(0-5]		
Delta P for reopen	Real	[Pa]			

See also "TAP" on page 411.

4.33.4.2 Component specific output

None

4.33.4.3 H-actions

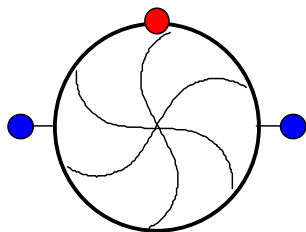
None

4.33.4.4 Component messages

Message	Type	Explanation
System pressure (..... barg) is lower than delivery pressure (.....barg)	Error	Initially the tap discharge must be out of the system. Since the computed pressure loss coefficient may not be negative the system pressure must be higher than the delivery (downstream) pressure.
opens	Info	
closes	Info	

4.34 Turbine

4.34.1 Turbine (class)



Turbine with adjustable vanes

Fall type

type label	description	active
Turbine	A turbine whose speed is controlled by its vane position, absorbed power and upstream/downstream hydraulic conditions.	yes

The turbine is an active Fall-type component. It is controlled by its vane position, the power absorbed by the electricity grid and the upstream/downstream hydraulic conditions.

Notes:

1. The absorbed power must be less than the maximum available hydraulic power ($= \rho g Q H \eta$) at all times.
2. A hydraulic system based on BoundH boundary conditions are recommended. Convergence problems could be encountered when a BoundQ boundary condition is used (multiple solutions could exist).

4.34.1.1 Mathematical model

The basic equations governing the flow in the turbine are:

Continuity

$$Q_1 = Q_2 \quad (1)$$

Head balance

$$H_1 - H_2 = H(Q, N, Y) \quad (2)$$

Torque-speed

$$T(Q, N, Y)N - P_{abs} = I_p N \frac{dN}{dt} \quad (3)$$

Where

Symbol	Description	SI-units
Q_1	upstream discharge	[m ³ /s]
Q_2	downstream discharge	[m ³ /s]
H_1	upstream head	[m]
H_2	downstream head	[m]
$H(Q, N, Y)$	Head loss across the turbine	[m]
$T(Q, N, Y)$	Torque generated by the turbine	[Nm]
N	Turbine speed	[rad/s]
Y	Turbine vane position	[rad]
TN	Power generated by the turbine	[W]
P_{abs}	Power absorbed by the power grid	[W]
I_p	Turbine polar moment of inertia	[kg.m ²]
t	time	[s]
ρ	Actual fluid density (entered in fluid specifications window)	[kg/m ³]
ρ_o	Reference fluid density (typically 1000 for water of 4 °C)	[kg/m ³]

4.34.1.2 Turbine data curves

In order to solve equations 1, 2 and 3 the turbine headloss $H(Q, N, Y)$ and torque $T(Q, N, Y)$ need to be determined. These values are calculated from the data provided by the manufacturer, using the following relationships:

$$Q(H, N, Y) = Q_{11}(N_{11})D^2\sqrt{H} \quad (4)$$

$$T(H, N, Y) = T_{11}(N_{11})D^3H \rho/\rho_o \quad (5)$$

With

$$N_{11} = \frac{ND}{\sqrt{H}} \quad (6)$$

Where $Q_{11}(N_{11})$ and $T_{11}(N_{11})$ are the unit value curves provided by the turbine manufacturer. D is the reference diameter. Usually the $Q_{11}(N_{11})$ and $T_{11}(N_{11})$ unit value curves do not allow for accurate interpolation of an arbitrary vane position. Improved interpolation is achieved by transforming the unit value curves into $Q_{11}(N_{11} y/y_{max})$ and $T_{11}(N_{11} y/y_{max})$ curves. The Wanda turbine component computes these transformed unit value curves from two .trb input files, which contain a set of interpolation curves. (Note: y_{max} is actually a reference value, hence the dimensionless y/y_{max} may be > 1).

Figure 1.1 presents an example set of transformed unit value curves. For a fixed head drop, vane position, diameter and speed, the discharge and torque can be found. For example:

speed: $N = 600$ rpm
 reference diameter: $D = 1590$ mm
 head drop: $H = 350$ m
 relative vane position: $y/y_{max} = 1$ (100%)

yields: $N_{11} = \frac{ND}{\sqrt{H}} = 51 \text{ rpm} \cdot \text{m}^{1/2}$ and then from Figure 1.1: $Q_{11} = 0.224 \text{ m}^3/\text{s}$ and $T_{11} = 370 \text{ N/m}^3$ and with equations 4 and 5: $Q = 10.6 \text{ m}^3/\text{s}$ and $T = 521 \text{ kNm}$.

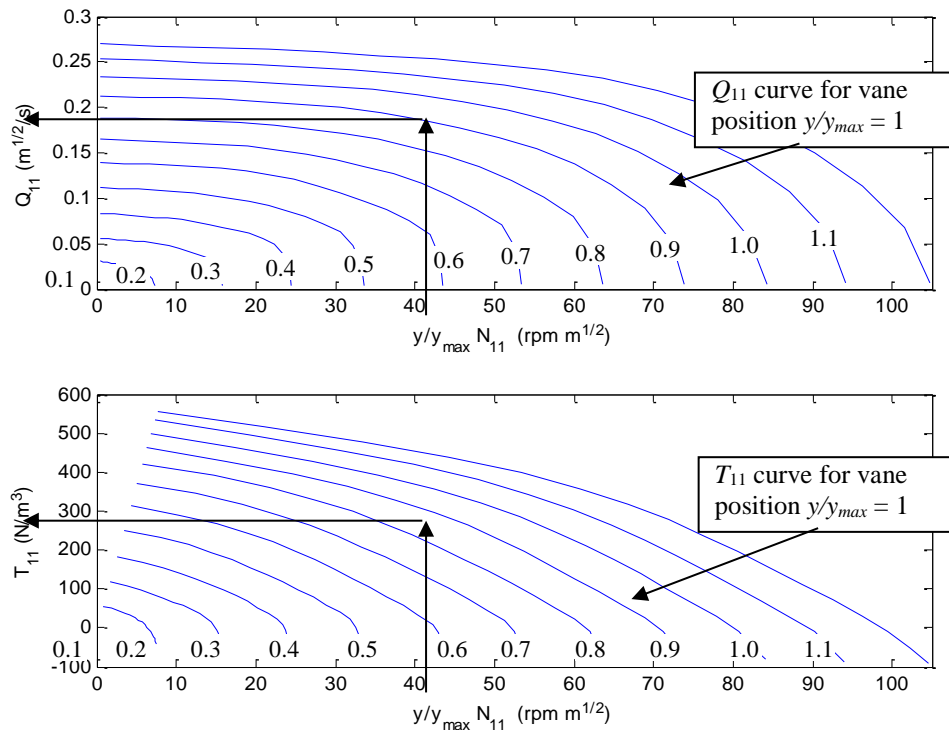


Figure 1.1: Example turbine unit value curves

References

- [1] E.B. Wylie, V.L. Streeter, "Fluid Transients", McGraw-Hill, New York, 1978.
- [2] A.P. Boldy, N. Walmsley "Representation of the characteristics of reversible pump turbines for use in waterhammer simulations", Paper G1, Fourth Int. conference on Pressure Surges, University of Bath, England, September 21-23, 1983
- [3] C.S. Martin, "Transformation of pump-turbine characteristics for hydraulic transient analysis", Proceedings, Eleventh Symposium of the section on Hydraulic Machinery, Equipment and Cavitation, IAHR, Amsterdam, 1982, Paper 30

4.34.2 Turbine model

4.34.2.1 Hydraulic specifications

Description	Input	SI-unit	Default	Remarks
Turbine file nr	Real	[-]	1	Reference number for the turbine input data (.trb) files: CASE_###T11.trb and CASE_###Q11.trb. See below.
Reference diameter	Real	[m]		Turbine diameter related to the unit value curves in the input files.
Polar moment of inertia	Real	[kg.m ²]		Moment of inertia.
Rated head	Real	[m]		Rated (or nominal) head, used for start vector in solution process.
Prescribed relative vane position	Table			Relative vane position y/y_{\max} as function of time.
Initial power absorbed	Real	[W]		Power absorbed by the electricity grid at $t=0$ s.
Initial speed	Real	[rad/s]		Speed at $t=0$. A realistic value should be used (see below).
Action table	Real	[W]		The absorbed power can be changed using this time-power action table.

NOTE: Turbine file format

The 'interpolation curves' in the turbine data files can be used to determine the values of $Q_{11}(N_{11} y/y_{\max})$ and $T_{11}(N_{11} y/y_{\max})$ or to reconstruct the unit value curves for an arbitrary vane position. The data files contain the number of vane positions and an array of vane positions in the header and the number of interpolation curves and the actual interpolation curves (x and y values) in the body. Rows starting with '*' are ignored.

The file name consists of the CASENAME, the number specified in the input field, Q_{11} or T_{11} and the .trb extension. The interpolation curves are sorted from lower

$N_{11} y/y_{\max}$ values at the top of the file to higher values at the bottom.

The interpolation curves define the points for each vane position. Therefore, the number of columns for each curve is the same as the number of vane positions. Each curve has a row for $N_{11} y/y_{\max}$ [rpm.m^{1/2}] and a row for Q_{11} [m³/s] or T_{11} [N/m³] corresponding to the actual vane positions. Please note that the units of N_{11} in the turbine files are defined as [rpm.m^{1/2}], which is not conform the SI-unit standard.

```

1  * Francis turbine (T11 versus N11*y/ymax)
2  *
3  *
4  * The first row gives the vane positions: Number of vane positions = number of columns.
5  *   In this file we have 11 vane positions hence we have 11 columns.
6  * # VANE POSITIONS
7  11
8  * List of vane positions (y/ymax fractions)
9  0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1.0 1.1
10 *
11 *
12 * # INTERPOLATION CURVES
13 17
14 * Each interpolation curve has two rows; the columns correspond to the vane positions above
15 * row 1: N11*y/ymax values
16 * row 2: T11 values
17 *
18 * Interpolation curve 1:
19 8.1382344E-01 1.6410418E+00 2.5290259E+00 3.4465626E+00 4.2869270E+00 5.0758145E+00 5.7422397E+00
20 5.6154618E+01 1.1724455E+02 1.8282203E+02 2.5058196E+02 3.1264273E+02 3.7090193E+02 4.2011732E+02
21 * Interpolation curve 2
22 1.8086890E+00 3.8293637E+00 5.7864977E+00 7.9790587E+00 1.0351830E+01 1.2483895E+01 1.4767228E+01
23 5.0161878E+01 1.0728041E+02 1.6857407E+02 2.3185225E+02 2.8992266E+02 3.4446175E+02 3.9048861E+02
24 * Interpolation curve 3
25 2.7514762E+00 5.4989515E+00 8.3473620E+00 1.1498953E+01 1.5104397E+01 1.8180247E+01 2.1755054E+01
26 4.4160669E+01 9.7212141E+01 1.5418070E+02 2.1289943E+02 2.6688356E+02 3.1759231E+02 3.6030446E+02
27 * Interpolation curve 4
28 3.5531236E+00 6.9232355E+00 1.0485109E+01 1.4426752E+01 1.8905556E+01 2.2808252E+01 2.7477029E+01
29 3.3333333E+01 3.3333333E+01 3.3333333E+01 3.3333333E+01 3.3333333E+01 3.3333333E+01 3.3333333E+01

```

Figure: Example of a part of the input file "Turbine_001T11.trb"

NOTE: Initial speed

An estimate of the initial speed should be given such the generated and absorbed power are in equilibrium at $t=0$ s. The equilibrium initial speed can be found through trial-and-error calculation where hydraulic conditions, vane position and absorbed power are kept constant during the simulation. By plotting the turbine speed in time series, the equilibrium speed can be identified. However, multiple solutions exist. The user has to check whether the Generated Power equals the Absorbed Power. The right equilibrium speed must result in the turbine generated power being equal to the absorbed power at $t=0$.

4.34.2.2 Component specific output

Description	SI-unit	Remarks
Turbine speed	[rad/s]	Rotational speed of the turbine
Relative vane position	[-]	Relative position (y/y_{max}) of the vanes
Power generated	[W]	Power generated by the turbine
Power absorbed	[W]	Power absorbed by generator / electricity grid
Torque generated	[Nm]	Torque generated by the turbine
Torque absorbed	[Nm]	Torque absorbed by the generator
N11	[rpm.m ^{1/2}]	Transformed turbine speed (in rpm Instead of rad/s)
Q11	[m ^{1/2} /s]	Transformed turbine discharge
T11	[N/m ³]	Transformed turbine torque

4.34.2.3 H-actions

The turbine component differs from other components regarding H-actions. Although for most components the H-action controlled by the control signal line is the same as what can be controlled through an action table, the turbine has two different H-actions:

- Action table: changes the absorbed power
- Control signal line: changes the vane position

4.34.2.4 Component messages

Message	Explanation
Warning: Could not solve Torque-speed equation: Inappropriate Q11/T11 turbine data, too large absorbed power or too small polar moment?	A solution could not be found for the torque-speed equation at the start of the time-step. This might be caused by a too small polar moment, incorrect Q11/T11 turbine data or that the absorbed power is larger than $\rho g Q H \eta$.
Warning: Could not solve head for given discharge and speed.	Unable to compute a head from the Q11/T11 turbine files (.trb), based on the present discharge and speed.
Warning: N11*y/y _{max} below turbine data range	Computed N11*y/y _{max} value is less than the minimum value in the Q11/T11 turbine files (.trb)
Warning: N11*y/y _{max} above turbine data range	Computed N11*y/y _{max} value is larger than the maximum value in the Q11/T11 turbine files (.trb)
Error: Input file not found	The turbine data files: CASENAME_###T11.trb and/or CASENAME_###Q11.trb could not be found.
Error: Input file already in use by other RDFILE	The turbine data file is already in use by another turbine component.
Error: Error with input file	The data in the turbine input file contains an error
Error: Error in turbine file	The data in the turbine input file contains an error
Error: decrease NVANES	Number of vanes used in turbine data file is greater than the maximum number of vanes allowed in WANDA (20).
Error: decrease NCURVES	Number of 'interpolation curves' used in the turbine data file is greater than the maximum number of

	'interpolation curves' allowed in WANDA (20).
Error: Inconsistent initial vane position at $t=0s$ and input of control system	The initial control system value and the initial vane position specified by the user are different.
Error: Control system gives negative vane position	Only positive vane positions are allowed.

4.34.3 Examples

4.34.3.1 Model description

The following example is meant to explain the function of the turbine component. The turbine is placed between two reservoirs, as shown in Figure . As this example is an academical case, the pipeline systems at both sides of the turbine are neglected. However, in practical applications the influence of the pipeline should be considered (see example 2).

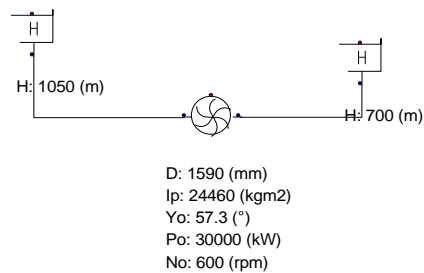


Figure 1.3: Wanda scheme.

The turbine interpolation curves are saved in the example files Turbine_001T11.trb and Turbine_001Q11.trb. Figure presents a visual representation of these curves. In addition, the unit curves have been reconstructed from the interpolation curves and have also been added to the figure.

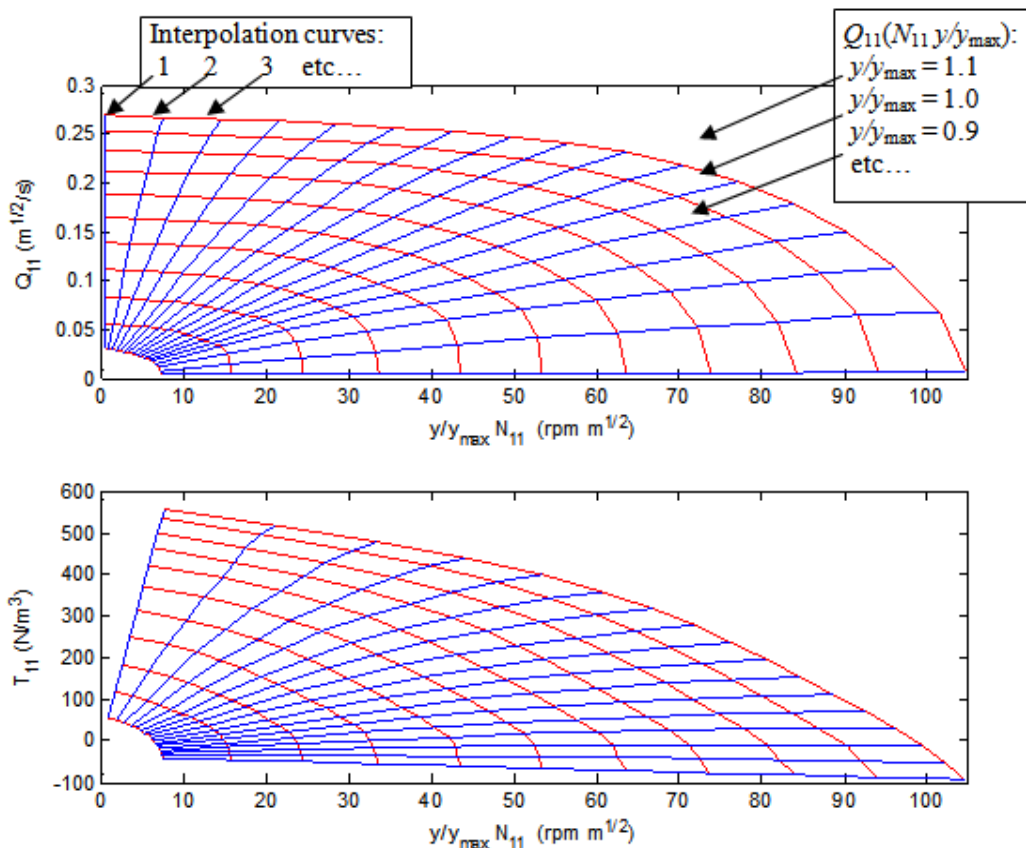


Figure 1.4: Turbine unit curves with interpolation curves (see example directory for the input files “Turbine_001T11.trb” and “Turbine_001Q11.trb”)

4.34.3.2 Finding the equilibrium speed for a specified P_{abs}

The following procedure can be followed to find the speed that places the generated power in equilibrium with the absorbed power: An initial speed equal to the rated speed is input (for this example 600 rpm) and a transient calculation performed without varying the hydraulic conditions, vane position or absorbed power. Figure shows that the absorbed power and generated power are not in equilibrium with an initial speed of 600 rpm. For the next 35s the turbine speeds up in order to balance the absorbed and generated power. After 35s the turbine has reached an equilibrium with a speed of approximately 751.6 rpm. This equilibrium speed can be used in subsequent transient calculations, where the turbine is initially in equilibrium.

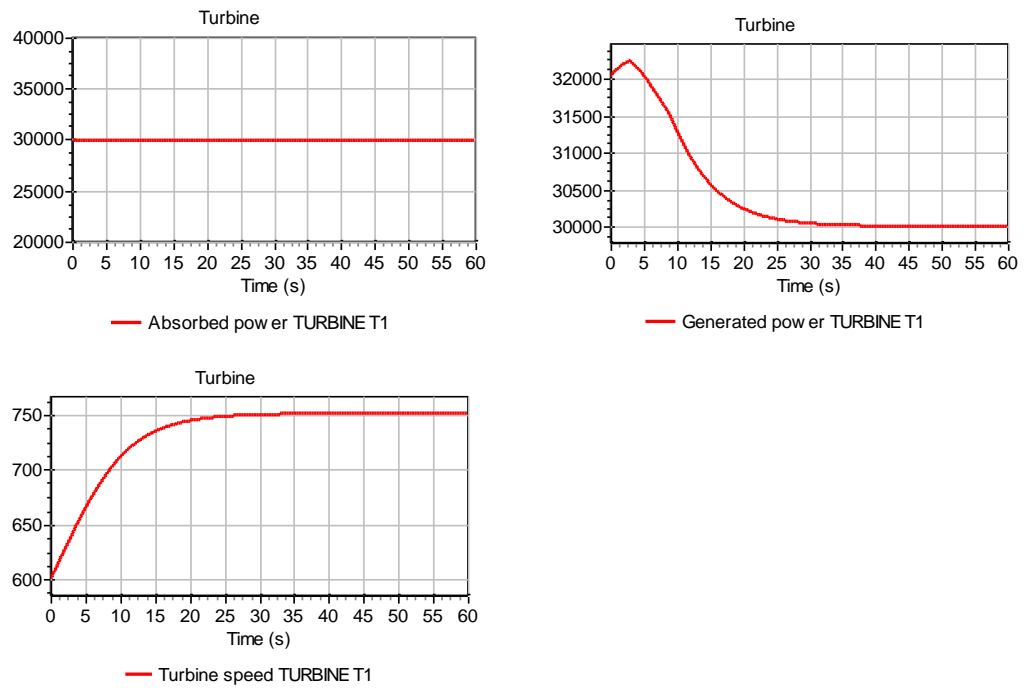
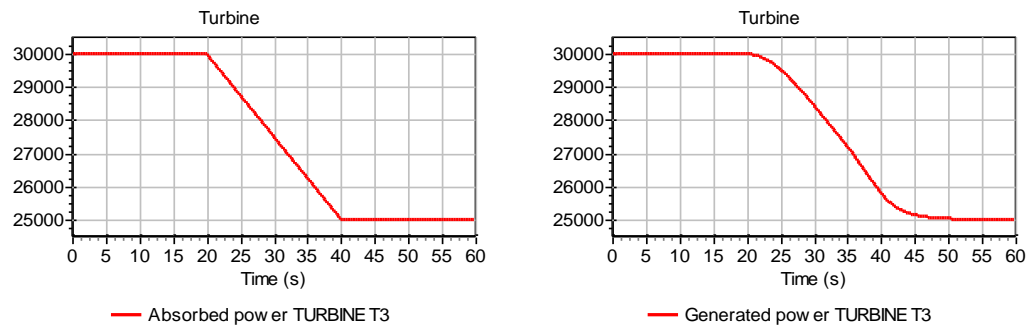


Figure 1.5. Turbine speed increases as power is not in equilibrium at $t=0s$.

4.34.3.3 Varying the absorbed power P_{abs}

At $t=0s$ there is an equilibrium between generated and absorbed power. The absorbed power is lowered from 30000 at $t=20s$ to 25000 kW at $t=40s$ (using the action table of the turbine). Due to the inertia of the turbine a disequilibrium will develop. The turbine will speed up in order to lower the generated power and to restore the equilibrium. Figure shows the increase of the turbine speed at constant vane position.



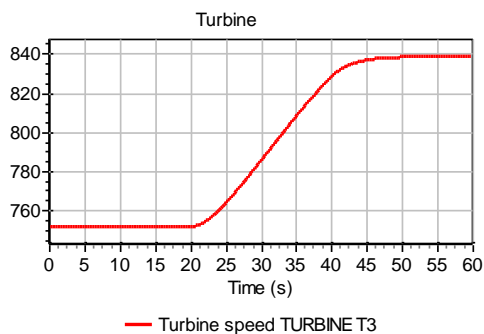


Figure 1.6. Decreasing power results in increasing turbine speed.

4.34.3.4 Varying the vane position

By changing the vane position, it is possible to control the speed of the turbine. A control signal line is connected to the turbine as shown in Figure . It is also possible to vary the vane position with the table property. If both the control table and the property table are present, the control table will prevail. The signal decreases the vane position from 100% of the maximum angle at t=5s to 95% at t=25s. For this example the absorbed power is kept constant ($P_{abs} = 30\ 000\ kW$).

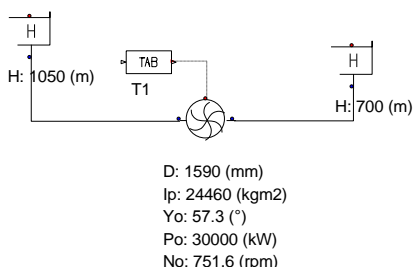


Figure 1.7. Wanda scheme and connection of signal line.

The system is in equilibrium at t=0s. The discharge and generated power will decrease due to the decreasing vane position, as can be expected from the turbine unit curves. Figure 1.8 shows the generated power and the vane position. After t=25s the vane position is kept constant.

Due to the imbalance between generated and absorbed power the speed of the turbine changes. Because the generated power is lower than the absorbed power, the speed of the turbine is decreasing according to equation 3. At t=60s the turbine is in equilibrium again as the speed is constant and the generated power equals the absorbed power.

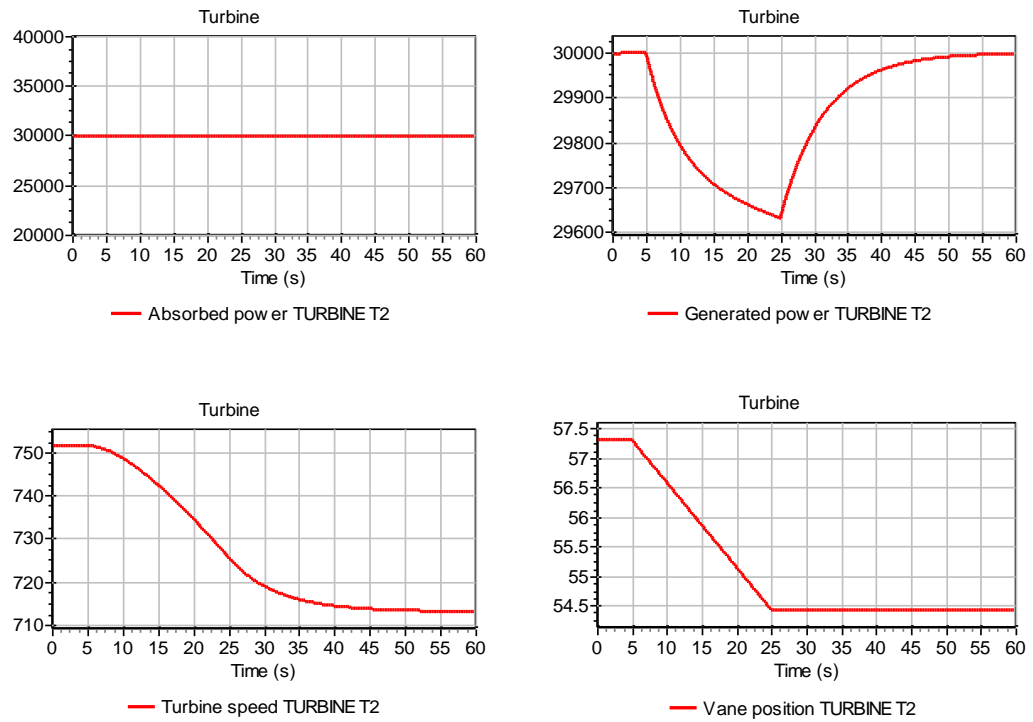


Figure 1.8. Changes in generated power due to reduced vane position.

4.34.3.5 More realistic example

To show the influence of the pipeline system on the turbine a more realistic case is included as well. Here the absorbed power is initially in equilibrium at 27970 kW. At t=10s the absorbed power drops to 22000 kW at t=30s. The vane position is reduced to prevent too high speed values. Since the change of vane position is a reaction to the drop in absorbed power, the vane position changes with a delay of 5 seconds from 100% (at t=15s) to 76% (at t=35s). Figure 1.9 shows the scheme for this example including the pipelines.

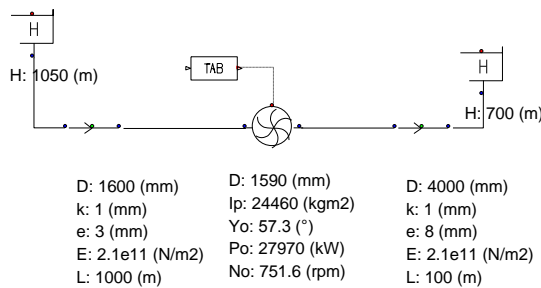


Figure 1.9 Wanda scheme

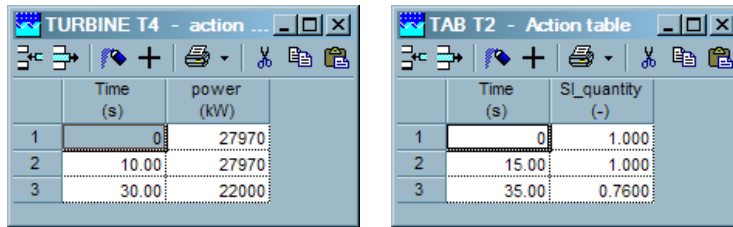


Figure 1.10: Input via action table (left) and control-table (right)

Figure 1.11 shows the change in power, speed and vane position. Due to the inertia there is a 'overshoot': the generated power drops below the equilibrium value when the absorbed power is kept constant again (at t=30s). At t=50 the system is in equilibrium again.

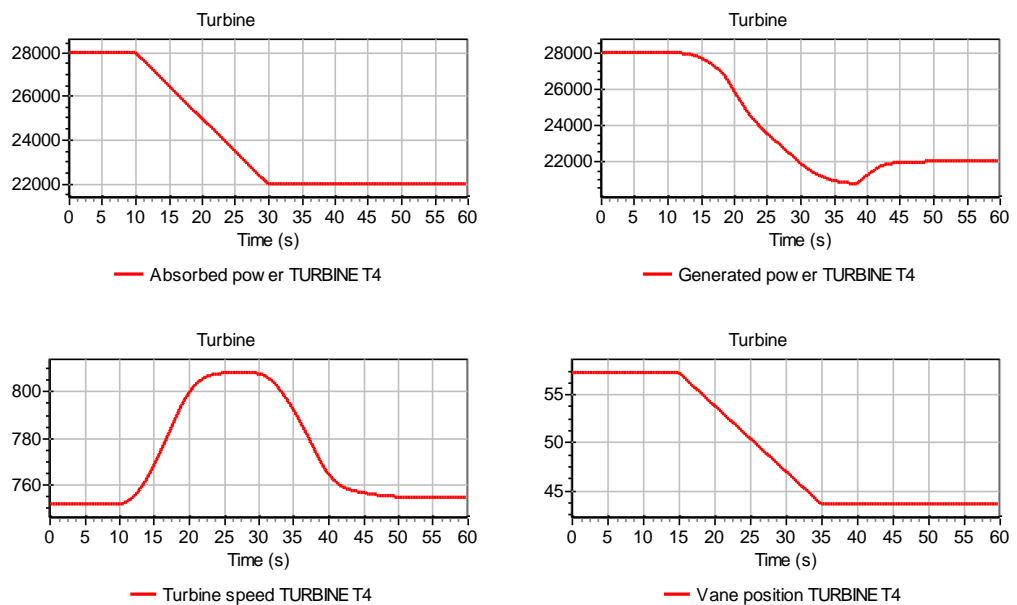
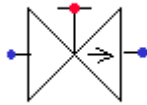


Figure 1.11: Power, speed and vane position

It is of course also possible to build a control system with a sensor to measure the pump speed and feeding this into a PID controller which feeds the vane position back into the turbine.

4.35 VALVE

4.35.1 Valve (class)



Control valve, block valve

Fall type

type label	description	active
Valve	Control or block valve with choice out of four predefined DELTARES standard head loss characteristics or user specified characteristics; several initial settings can be used for flow or pressure balancing	Yes

4.35.1.1 Mathematical model

The discharge characteristic indicates the relation between the flow Q through the valve and the pressure Δp across the valve as a function of the valve position. This relation is expressed in a discharge coefficient K_v or C_v and a loss coefficient ξ .

The coefficients are derived from the general equation for a Newtonian flow through a restriction in a pipeline under cavitation free circumstances:

$$\Delta p = \xi \frac{\rho v^2}{2}$$

in which:

Δp	=	net pressure difference across the valve	[N/m ²]
ξ	=	loss coefficient of the valve	[-]
ρ	=	density of the fluid	[kg/m ³]
v	=	velocity	[m/s]

or

$$\Delta H = \xi \frac{v^2}{2g}$$

in which:

ΔH	=	pressure head difference across the valve	[m]
ξ	=	loss coefficient of the valve	[-]
v	=	velocity	[m/s]
g	=	gravitational acceleration	[m/s ²]

In practice the discharge coefficients K_v is used.

$$K_v = \frac{Q}{\sqrt{\Delta p}}$$

with:

K_v	=	discharge coefficient	[m ³ /h/ $\sqrt{\text{bar}}$]
Q	=	flow	[m ³ /h]
Δp	=	net pressure difference across the valve	[bar]

In words: the discharge coefficient K_v denotes the flow in m³/h which flows through a valve at a pressure difference of 1 bar.

Apart from K_v , C_v is also defined as a discharge coefficients for American units.

$$C_v = \frac{Q}{\sqrt{\Delta p}}$$

with

C_v	=	discharge coefficient	[USGPM/ $\sqrt{\text{psi}}$]
Q	=	flow	[US gallons/min]
Δp	=	net pressure difference across the valve	[psi]

The relation between ξ , K_v and C_v is as follows:

$$\xi = 1.6 \cdot 10^9 \frac{1}{K_v^2} \cdot D^4$$

$$K_v = 0.865 C_v$$

A valve is characterised by $\xi = f(\theta)$ or $K_v = f(\theta)$ or $C_v = f(\theta)$.

θ denotes the dimensionless valve opening. θ ranges from 0 to 1 (in SI-units, or 0 = 100 % in percentage annotation)

- $\theta = 0$, valve is closed.
- $\theta = 1$, valve is completely open.

The different discharge characteristic are always translated to the following equation:

$$\Delta H = a \xi Q_1 | Q_1 |$$

in which:

ΔH	=	$H_1 - H_2$ in positive flow direction	[m]
H_1	=	upstream head	[m]
H_2	=	downstream head	[m]
a	=	$1 / (2 g A_f^2)$	[s ² /m ⁵]
ξ	=	loss coefficient	[-]
Q_1	=	discharge upstream	[m ³ /s]
A_f	=	discharge area valve	[m ²]

The discharge characteristic may be defined by one of Deltares' standard characteristics (See Hydraulic specifications) or by a user-defined discharge characteristic. If the valve position does not coincide with a tabulated position, interpolation must be performed to obtain the discharge coefficient for intermediate valve positions. The standard characteristics and the user-defined ξ characteristic are interpolated logarithmically according to the following equation:

$$\theta(z) = z \cdot \theta_1 + (1 - z) \cdot \theta_2$$

$$\xi(\theta(z)) = \xi_1^z \cdot \xi_2^{1-z}$$

The user-defined K_v and C_v characteristics are interpolated such that K_v or C_v values are interpolated linearly:

$$\frac{1}{\sqrt{\xi(\theta(z))}} = \frac{z}{\sqrt{\xi_1}} + \frac{1-z}{\sqrt{\xi_2}}$$

in which:

θ_1, θ_2	=	tabulated valve positions	[-]
z	=	fraction, which defines intermediate valve position ($0 < z < 1$)	[-]
$\theta(z)$	=	intermediate valve position	[-]
ξ	=	loss coefficient at θ_1, θ_2	[-]
$\xi(\theta(z))$	=	interpolated loss coefficient	[-]

If the valve is fully closed the governing equations is:

$$Q_1 = 0$$

Cavitation

Cavitation depends on the pressure conditions around the valve. Usually these pressure conditions are defined by a pressure relation. Several different definitions are use in the industrial standards.

In WANDA the factor X_f is used, according the German VDMA standard.

$$X_f = \frac{\Delta p}{p_1 - p_v}$$

in which:

X_f	=	pressure ratio	[-]
Δp	=	net pressure difference across the valve	[N/m ²]
p_1	=	absolute pressure upstream of the valve	[N/m ²]
p_v	=	vapour pressure of the fluid	[N/m ²]

The pressure ratio depends of the valve opening: $X_f = f(\theta)$

The pressure ratio X_f is only calculated for positive flow; for negative flow $X_f = 0$.

If the cavitation characteristic is specified, the program calculates the pressure ratio in the system and warns the user if it exceeds the available value as defined in the characteristic.

Note:

In other standards (ISA, BS, IEC) the pressure ratio X_T is used:

$$X_T = \frac{p_1 - p_2}{p_1} = \frac{\Delta p}{p_1}$$

In some standards (e.g. ISA) the Thoma number σ is used. $\sigma = \frac{p_1 - p_v}{\Delta p}$

in which:

σ	=	pressure ratio	[-]
Δp	=	net pressure difference across the valve	[N/m ²]
P_1	=	absolute pressure upstream of the valve	[N/m ²]
p_v	=	vapour pressure of the fluid	[N/m ²]

The relationship between X_f and σ is: $\sigma = \frac{1}{X_f}$

Another definition for the Thoma number is based on the downstream pressure p_2 :

$$\sigma_2 = \frac{p_2 - p_v}{\Delta p}$$

where $\sigma = 1 + \sigma_2$ and $\sigma_2 = \frac{1}{X_f} - 1$

4.35.2 Valve properties

4.35.2.1 Hydraulic specifications

Description	input	Unit	range	default	remarks
Characteristic type	Standard/ Kv/ Cv/ Xi				
Standard type	Buttrfly/ Ball/ Gate/ Gate_sqr			Buttrfly	if char.type = Standard
Kv characteristic	Table				if char.type = Kv
Cv characteristic	Table				if char.type = Cv
Xi characteristic	Table				if char.type = Xi
Inner diameter	Real	[m]	(0-1000]		
Initial setting	Position/ H_upstream H_downstr Discharge P_upstream P_downstr				
Initial position (open)	Real	[-]	[0-1]		0 = closed 1= open If init_set = Position
Initial upstream head	Real	[m]	[-1000 – 1000]		If init_set = H_upstream
Initial downstream head	Real	[m]	[-1000 – 1000]		If init_set = H_downstr
Initial discharge	Real	[m ³ /s]	(0, 10)		If init_set = Discharge
Initial upstream pressure	Real	[N/m ²]	[-10 ⁵ – 10 ⁷]		If init_set = P_upstream
Initial downstream pressure	Real	[N/m ²]	[-10 ⁵ – 10 ⁷]		If init_set = P_downstr
Check cavitation	Yes/No				
Cavitation table	Table				If check cavitation= Yes

Deltares standard characteristics

Butterfly valve

θ	ξ
0.000	1.0E+10
0.010	10000000
0.025	1700000
0.050	140000
0.075	23000
0.100	6000
0.125	2400
0.150	1150
0.200	440
0.250	195
0.300	97.5
0.400	31.0
0.500	13.8
0.600	5.80
0.700	2.40
0.800	1.00
0.900	.420
1.000	0.150

Ball valve

θ	ξ
0.000	1.0E+10
0.015	900000
0.025	350000
0.050	40000
0.075	9500
0.100	2750
0.150	650
0.200	270
0.300	79.5
0.400	30.0
0.500	13.8
0.600	6.1
0.700	2.7
0.800	1.03
0.900	0.14
1.000	0.01

Gate valve

θ	ξ
0.000	1.0E+10
0.0025	270000
0.025	2850
0.050	625
0.075	270
0.100	140
0.150	58
0.200	31
0.300	11.5
0.400	5.35
0.500	2.55
0.600	1.27
0.700	0.67
0.800	0.355
0.900	0.188
1.000	0.100

Square gate valve

θ	ξ
0.000	1.0E+10
0.0025	249000
0.050	850
0.075	370
0.100	195
0.150	82
0.200	45
0.300	17.8
0.400	8.2
0.500	4.0
0.600	2.1
0.700	0.95
0.800	0.39
0.900	0.09
1.000	0.001

4.35.2.2 Component specific output

Valve position (open) [-]

Pressure ratio X_f system [-]

4.35.2.3 H-actions

A valve can be opened or closed. To do that the valve must be activated. How the valve opens or closes is arranged via a θ -time relation in tabular form (menu 'actions' in MODEL).

An example:

**Input of table valve type
VALVE ACTION**

Time (s)	Position (-)
0	1.
0.5	0.
10.0	0.
12.0	1.
14.0	0.5
18.0	0.5

Note: the position unit depends on the setting made in menu Units

The valve closes linearly in 0.5 s. It remains closed until 10 s. Then the valve opens again in 2 s and starts to close directly until $\theta = 0.5$ at 14.0 s. From then on the valve remains in that position.

4.35.2.4 Component messages

Message	Type	Explanation
starts in open phase	Info	
Opens	Info	
starts in closed phase	Info	
Closes	Info	
Xf valve exceeded, valve may be choking	Info	Only if "Check Cavitation" = "Yes" Current pressure ratio exceeds the table value

4.36 Vent, Air valve

4.36.1 VENT (class)



Vent (air inlet and/or outlet)

Supplier type

type label	description	active
VENT (discharge coeff.)	Vent with the possibility of air in/outlet, air inlet only or air outlet only Capacity specified with discharge coefficients	No
VENT (inflow/outflow char.)	Vent with the possibility of air in/outlet, air inlet only or air outlet only Capacity specified by inflow/outflow characteristics	No

4.36.1.1 Mathematical model

The vent or air valve is a component through which air can enter into or be expelled from the hydraulic system. This is usually achieved by a floating-ball valve mechanism (figures 1, 2 and 3). The purpose of the vent is to prevent cavitation or intolerable under pressures.

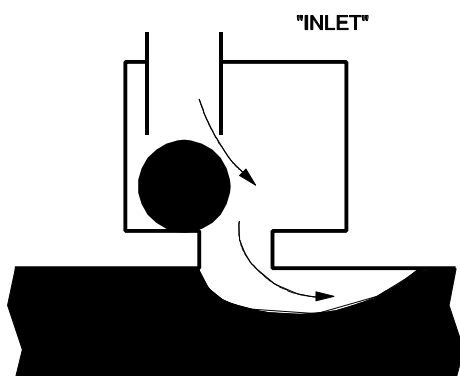


Figure 1: In/outlet vent in air inlet status

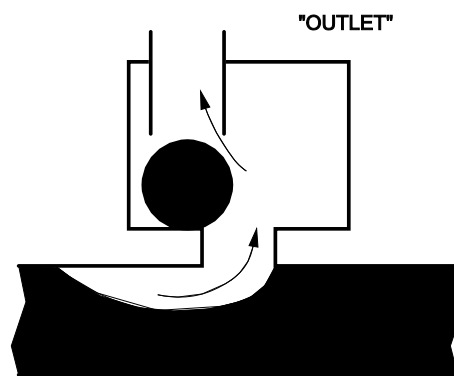


Figure 2: In/outlet vent in air outlet status

If the internal pressure in the system drops below the elevation of the vent, the floating-ball valve opens and air can enter the system (figure 1). The air entering the system will expand due to the internal pressure being lower than atmospheric pressure. If the air remains in the vicinity of the vent (e.g. in case the vent is located at a "high" point in the system) it will be expelled through the same vent if the internal pressure rises again above the elevation of the vent (figure 2). In this case all entered air will be expelled through the same vent. Meanwhile the air will also be compressed due to the internal pressure being higher than atmospheric. The vent closes at the instant the amount of

expelled air is equal to the amount of entered air (figure 3). This type of vent is denoted by "in/outlet".

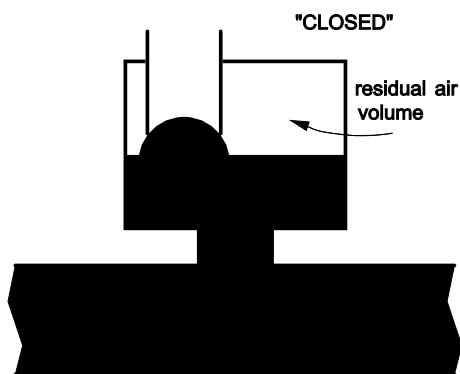


Figure 3: In/outlet vent in closed status

If, however, the fluid flow causes the air to be moved away from the vent, the air will not be expelled through the vent. As soon as the system pressure rises again above the vent elevation the vent closes. This type of vent is denoted by "inlet". The vent types "in/outlet" and "inlet" are in fact not different physically, but only circumstantially.

A third option is the vent that only allows air to be expelled from the system (figure 4). This is achieved by adding an "air check valve" to the vent. This valve is closed if the internal system pressure is below vent elevation (figure 5). A "sane" computation has to start with a positive initial air volume. This type of vent is denoted by "outlet".

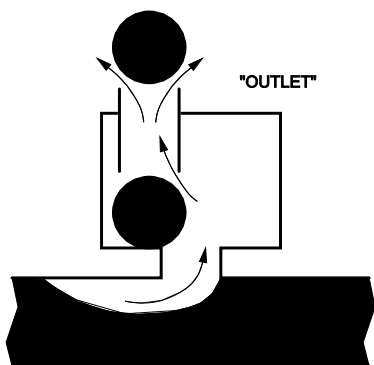


Figure 4: Outlet vent in air outlet status

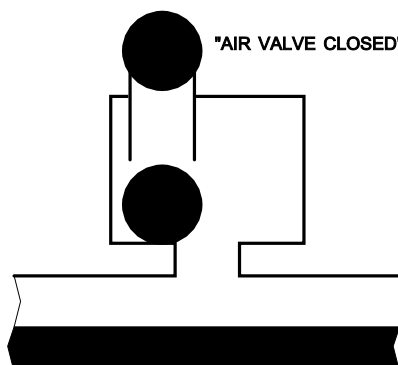


Figure 5: Outlet vent in closed air outlet status

The air flow is compressible, with the consequence that due to local occurrence of supersonic velocities and shock waves, choking flow may arise. The air inlet capacity is therefore truncated to a certain level after which further reducing the internal pressure does not increase the air flow anymore. The expansion and compression of the air may be isothermal, adiabatic or polytropic. Since WANDA is not a two-phase flow computer code it is not capable of describing the entrance and transportation of air in the pipeline as such. This means that the vent model is merely a boundary condition describing a pressure-discharge relation. In reality the amount of liquid within the system decreases when air enters the system. In the discharge supplied by the vent to the system the compressibility of the air is taken into account. The vent component supplies liquid to the system and therefore an error in the momentum balance (inertia forces) will be introduced. This error is small however if the amount of air is small compared to the pipeline volume. The continuity balance is not violated.

Changing water levels due to air entrance or expansion/compression can be taken into account by specifying a level-area table. This table represents the storage area in the components in the neighbourhood of the VENT against the fluid level drop. As the entered volume of air increases, the fluid level drops according to this level-area table. If the fluid-level is taken into account (user-specified), then air enters the system more difficult and is expelled more easily, compared to the situation without fluid-level effect.

For the description of the mathematical model four states are defined:

- closed floating ball valve,
- air inlet,
- air outlet,
- manual outlet.

In case of "closed-in" air (states 1 and 4) the component behaves like an air vessel with adiabatic expansion and compression:

$$PV^k = C \quad (1)$$

in which:

P	=	absolute air pressure on fluid level	[N/m ²]
V	=	air volume	[m ³]
k	=	Laplace coefficient (ratio of specific heats)	[-]
C	=	Constant	[Nm]

The second equation governs the amount of supplying discharge Q:

$$Q = dV / dt \quad (2)$$

In the "open" states 2 and 3 the change of air volume in time is dependent of two phenomena. Firstly the compression/expansion of the air, secondly the amount of air leaving/entering the system. The former is handled in the same way as with states 1 and 4. The latter is determined by either formula (3) to (6) or by the air valve characteristic, depending on which vent component has been selected. Both options are described here.

Vent (capacity defined by coefficients)

1. Subsonic air flow in

$$Q_{\text{air}} = C_{\text{in}} A_{\text{in}} \sqrt{7 R T_0} \sqrt{\left(\frac{P}{P_0}\right)^{1.4286} - \left(\frac{P}{P_0}\right)^{1.714}} ; P_0 > P > 0.53 P_0 \quad (3)$$

in which:

A_{in}	=	inlet area	[m ²]
C_{in}	=	inlet discharge coefficient	[-]
P	=	abs. internal pressure on fluid level	[Pa]
P_0	=	atmospheric pressure	[Pa]
R	=	gas constant	[J/kg·K]
T_0	=	ambient air temperature	[K]
Q_{air}	=	air flow (positive if into system: supplier!)	[m ³ /s]

2. Critical flow in

$$Q_{\text{air}} = C_{\text{in}} A_{\text{in}} \sqrt{7 R T_0} \cdot 0.259 ; P < 0.53 P_0 \quad (4)$$

3. Subsonic air flow out

$$Q_{\text{air}} = -C_{\text{out}} A_{\text{out}} \sqrt{7 R T_0} \left(\frac{P}{P_0}\right)^{\frac{k+1}{2k}} \sqrt{\left(\frac{P_0}{P}\right)^{1.4286} - \left(\frac{P_0}{P}\right)^{1.714}} ; \frac{P_0}{0.53} > P > P_0 \quad (5)$$

in which:

A_{out}	=	outlet area	[m ²]
C_{out}	=	outlet discharge coefficient	[-]
k	=	Laplace coefficient (ratio of specific heats)	[-]

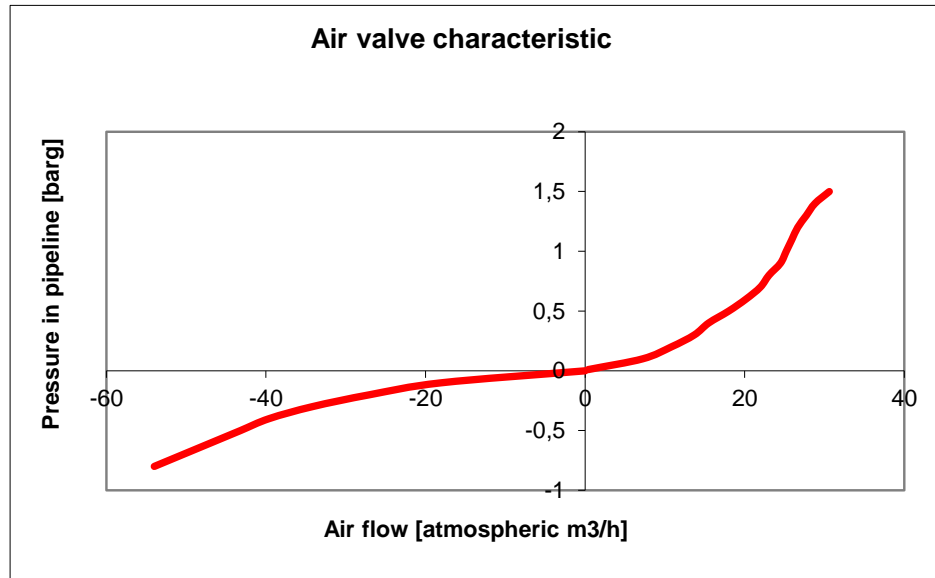
4. Critical flow out

$$Q_{\text{air}} = -C_{\text{out}} A_{\text{out}} \sqrt{7 R T_0} \left(\frac{P}{P_0}\right)^{\frac{k+1}{2k}} \cdot 0.259 ; \frac{P_0}{0.53} < P \quad (6)$$

Vent (capacity defined by characteristic)

The capacity of the vent can also be defined by the air valve characteristic (see example characteristic). In this case, you can define both the characteristic for inflow and outflow, which ever is applicable. The characteristic should be supplied as pressure against discharge.

Note: Often the characteristic supplied by the manufacturer will have the axes reversed from this definition. See also the example characteristic in this manual.



For both types of vents, the air volume is initialised by:

$$V = V_{\text{init}} + V_{\text{res}} \quad (7)$$

in which:

V_{init}	=	Initial air volume to be expelled	[atmospheric m ³]
V_{res}	=	Residual air volume	[m ³]

If, according to equation (2), the air volume V becomes $\leq V_{\text{res}}$ the vent closes. If the system pressure head H drops below the vent elevation h the vent enters the inlet state. If the system pressure rises again above the vent elevation the vent enters the outlet state.

Following table may be helpful to decide which combination of VENT type and initial state must be chosen.

Scenario	VENT type	Initial state
upstream pump trip or valve closure with automatic inlet/outlet VENTs without initial air volumes.	In/outlet	closed
final stage of a slow filling procedure with inlet/outlet VENTs. 'Slow' means 'slowly relative to the system's characteristic time'. See remarks below.	In/outlet	outlet
upstream pump trip or valve closure with automatic inlet VENTs without initial air volumes.	Inlet	closed
final stage of a slow filling procedure with outlet VENTs. 'Slow' means 'slowly relative to the system's characteristic time'. See remarks below.	Outlet	outlet
start-up scenario's or downstream valve closure with automatic outlet VENTs and initial air volumes; steady state pressure must be sub-atmospheric.	Outlet	manual
fast filling procedures or venting procedure with manually operated outlet VENTs. If the steady state pressure is super-atmospheric, then the VENTs open the first transient timestep. See remarks below.	Outlet	manual

Remarks

The initial state 'Inlet' should only be used if no other combination seems appropriate.

The difference between the initial states *outlet* and *manual* is the steady state computation. The *manual* initial state of vent type *outlet* computes a steady state flow of 0 m³/s through the VENT. During the transient calculation the VENT starts to expel air from the system (if steady pressure is super-atmospheric) causing pressure waves in the system.

This *manual* initial state can be used to simulate a manual vent, which is closed in steady state and starts with air outlet at $t = 0$ s. The difference with the *closed* initial state, is the presence of an initial air volume in the system, which cannot be specified in the *closed* initial state. The *outlet* initial state is used to simulate the outlet of an air volume in the system. It computes a hydrodynamic balance between the VENT outflow and the pipe flows in steady state.

This balance is a realistic simulation, only if the air volume in the real system is large enough, i.e. if the outflow of this air volume requires a period, which is greater than 5 times the characteristic time of the system. The characteristic time of the system is 2 times the characteristic length divided by the average wave propagation speed. By taking a small residual air volume the simulation starts at the moment the large air volume has almost entirely left the system through the vent.

If you want to investigate the difference between both initial states, it is recommended to build a simple model and compare the transient results of both initial states (the initial air volume should be set sufficiently high and the level effect should be set to NO).

4.36.2 VENT (discharge coeff.)

4.36.2.1 Hydraulic specifications

description	Input	unit	range	default	remarks
elevation offset	Real	[m]			See remark
Laplace coefficient	Real	[-]	[1-1.4]		
ambient air temperature	Real	[°C]			
inlet discharge coefficient	Real	[-]	(0-1)		See remark
outlet discharge coefficient	Real	[-]	(0-1)		See remark
inlet discharge area	Real	[m ²]	≥0		
outlet discharge area	Real	[m ²]	≥0		
initial air volume	Real	[m ³]	[0-10]		atmospheric m ³
residual air volume	Real	[m ³]	[0-1]		
type	In/outlet Inlet Outlet			In/outlet	See remark
initial state	Closed Inlet Outlet Manual			Closed	
level effect	YES NO			YES	
level-area table	table				See remark

See also "Mathematical model" on page 434.

Remark

The point of reference for the elevation offset of the vent is the geometric height of the H-node connecting the vent to the rest of the system.

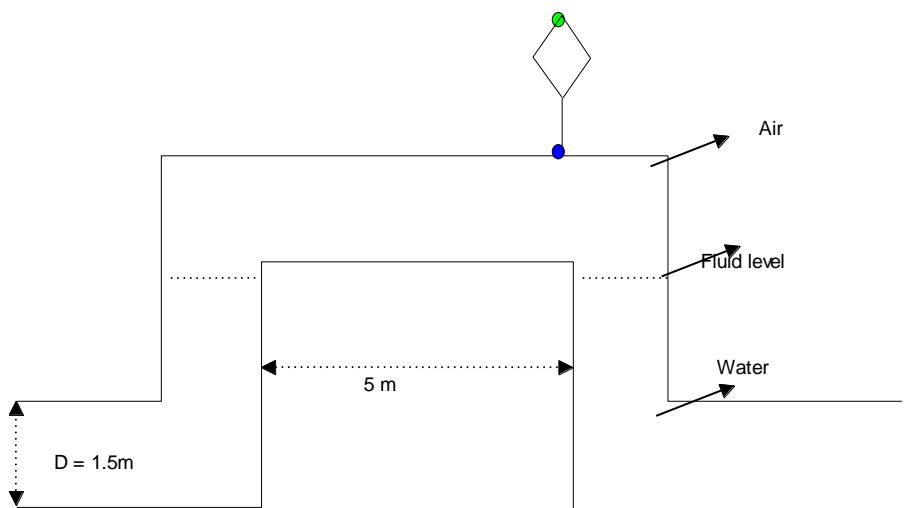
Please be aware that for deriving the discharge coefficients C_{in} and C_{out} from a K_V -value (m³/h if = 1 bar) the manufacturer most probably specifies the capacity in **atmospheric** cubic metres per second. In general, the discharge coefficients describe the amount of contraction of the air flow through the orifices and will probably be in the range of 0.5 to 1.0.

The in/out type should be chosen, if the air enters and leaves the system at the same location. The inlet type should be chosen, if the air enters the system at the VENT location and is transported with the flow. The outlet type should be chosen, if the air leaves the system at the VENT location (high point) and can not enter the system. The outlet can be used in particular for the simulation of de-aeration procedures with manually operated VENT type, in combination with the *manual* initial state.

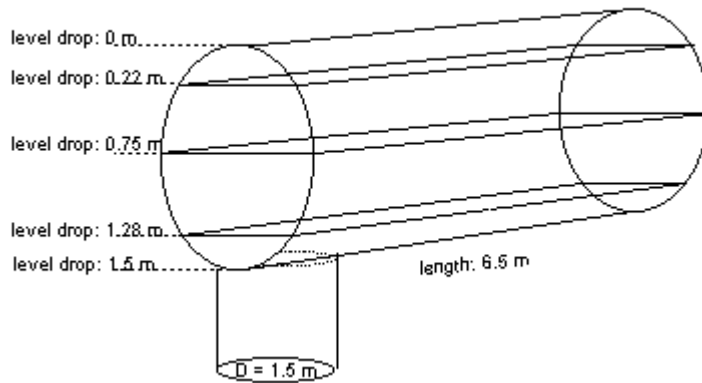
This data input template is based on a "sharing" concept. That is, the type can be changed without the need to re-enter the other data items. The consequence of this is that not all data are required for each type. For instance the outlet loss coefficient is not required in case the type is "inlet". However, for the program to be able to determine that the input data are complete, all fields must be filled by the user.

The level-area table has the elevation offset of the vent as point of reference. At positive distances below this point of reference the area of (part of) the horizontal cross-section of the pipeline is given. This area determines where in the pipelines the air volume will form if air enters the system through the vent. It does not necessarily have to be equal to the cross-section of the pipe. For example, if the flow pushes the air volume to a corner of the pipeline the area at some levels will be smaller than the cross-section of the pipeline. The user must make an educated guess on where the air volume will form in the system. An estimation of the cross-section of an oblique pipeline is made by the volume of the pipeline times the height of the pipeline. The area at the lowest level-drop that is given will be used for the rest of the pipeline. If an abrupt change in area occurs, the two different areas should be ascribed to two levels with only a small distance between them.

The level-area table models the storage area of the pipes and components in the neighbourhood of the VENT with level effect. The storage area of all components should be modelled in which air may collect during air inlet. The following examples may be helpful on specifying the level-area table.

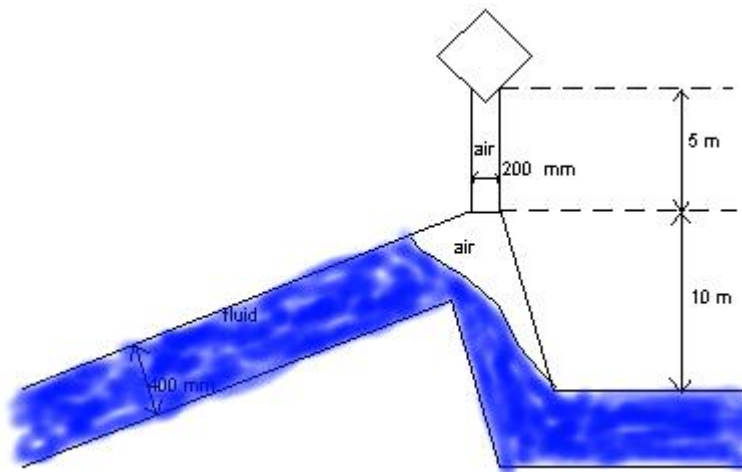


The figure above shows a schematised pipe bridge with a length of 5 m. All pipe diameters are 1.5 m. The elevation offset is 0 in this example. The first column of the level-area table is the level drop relative to the offset elevation and must start with level drop 0 (area may be greater than 0). The second column represents the storage area at a fluid level, which is “level drop” m below the vent elevation. The storage area of the horizontal circular pipe is discretised at 3 intermediate levels with an average length of 6.5 m. The storage area of the two vertical pipes increases as the level drops from 0 to 1.5 m. Consequently following table is entered for the situation above.



Example of level drop table:

Level drop [m]	area [m ²]
0.0	0.0
0.22	7.31
0.75	9.75
1.28	7.31
1.5	1.76



The figure above shows a schematised sawtooth pipeline with a height of 10 m. The offset of the vent is 5 m. The diameter of the connecting pipeline is 0.2 m, all other pipelines have a diameters 5.4 m. At a level drop of 5 m, the area changes gradually from 0.03 m² to 1.76 m². Because of the flow, the air volume will be pushed in the corner and downwards in the short pipeline. All this information is used in the level-area table. Below the level drop of 0.4 m the area remains 0.355 m² for 10 m, until the direction of the main pipeline changes to horizontal. This provides with a large enough volume for the air that enters the system.

Example of level drop table

Level drop [m]	area [m ²]
0.0	0.03
5.0	0.03
5.1	1.76
5.3	0.4
5.4	0.355

Important Notice: In case of vents with large capacity it is well possible to have a very sensitive computation in which small pressure differences cause large air flows and volumetric effects. Although the program is optimised to find the most accurate solution, numerical oscillations may still occur (inertia is not taken into account). In that case it is up to the user, to adjust (reduce) the time step and the convergence criteria to reduce the oscillations.

See also "Mathematical model" on page 434.

4.36.2.2 Component specific output

Air volume [m³]

Air flow [m³/s]

Air pressure (absolute) [N/m².abs]

Air temperature [°C]

4.36.2.3 H-actions

None

4.36.2.4 Component messages

Message	Type	Explanation
Positive initial air volume not allowed in combination with "initial state = closed"	Error	
Contradiction between type and initial state	Error	
"Initial state = outlet" not possible with zero initial air volume	Error	
Starts in closed phase	Info	
Absolute pressure below vapour pressure	Error	This would lead to an infinite air volume. This error can only occur if the VENT is connected to a BOUNDH or other <i>H</i> pre-scribing component.

Message	Type	Explanation
Closes	Info	The floating ball valve closes.
Starts in air inlet phase	Info	
Air inlet	Info	System pressure has dropped below the vent elevation.
Starts in air outlet phase	Info	
Air outlet	Info	System pressure has risen above the vent elevation and not all air has yet been expelled.
Starts with closed air valve	Info	
Air valve closes	Info	There may still be air in the system which will be expelled if the pressure rises again above the vent elevation.

4.36.2.5 Example

A certain type of vent is specified by the manufacturer with the following parameters:

DN: 80/100

Inlet:

A:	3850 mm ²
Δp :	0.1 bar
T ₀ :	20 °C = 293 K
Q _{air} :	0.4 m ³ /s

Outlet:

A:	50 mm ²
Δp :	0.1 bar
T:	20 °C = 293 K
Q _{air} :	0.005 m ³ /s

Residual air volume: 0.02 m³.

The pressure differences are in both cases (inlet and outlet) in the subsonic range (0.9 and 1/0.909 respectively). For the determination of C_{in} we apply equation (3) with the gas constant of air: $R = 287 \text{ J/kg}\cdot\text{K}$. This results in a value of 0.85 for the inlet coefficient. The outlet coefficient C_{out} is determined with equation 3. If we assume isothermal behaviour ($k = 1$) the value of the outlet coefficient will be 0.77.

4.36.3 VENT (inflow/outflow char.)

4.36.3.1 Hydraulic specifications

description	Input	unit	range	default	remarks
elevation offset	Real	[m]			See remark
Laplace coefficient	Real	[-]	[1-1.4]		
ambient air temperature	Real	[°C]			
initial air volume	Real	[m ³]	[0-10]		atmospheric m ³
residual air volume	Real	[m ³]	[0-1]		
type	In/outlet Inlet Outlet			In/outlet	See remark
initial state	Closed Inlet Outlet Manual			Closed	
level effect	YES NO			YES	
level-area table	table				See remark
Air inflow characteristic	table				See remark
Air outflow characteristic	table				See remark
Normal air temperature char.	Real	[°C]			See remark

See also "Mathematical model" on page 434.

Remark

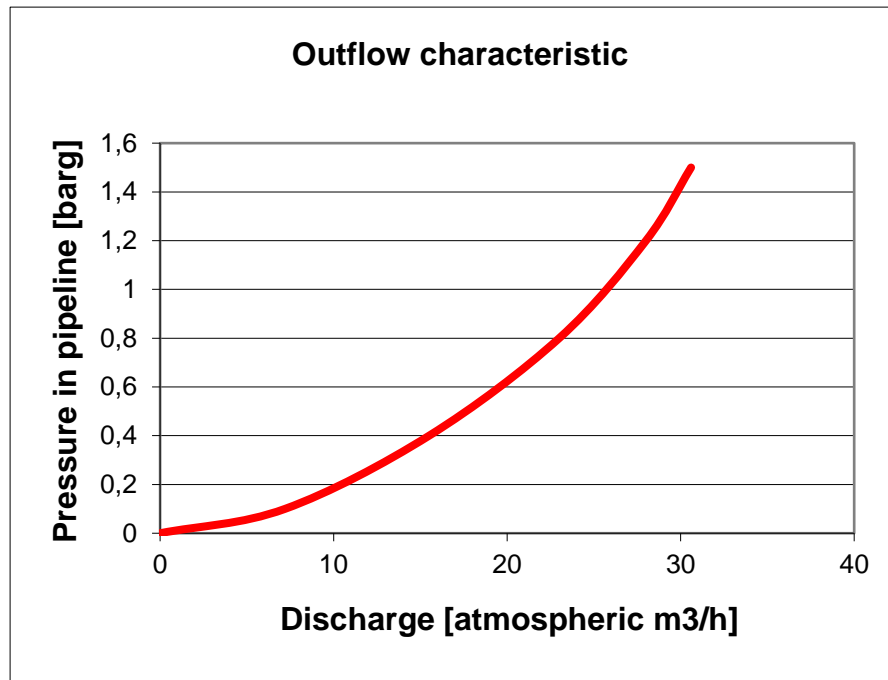
For remarks with respect to elevation offset, type and level effect, see the remarks for "VENT (discharge coeff.)" on page 440.

The table for air inflow or outflow is consists of the pressure difference between the pressure in the pipeline at the location of the valve and the ambient pressure. This corresponds to the pressure in the pipeline in [barg], if the ambient pressure is equal to atmospheric pressure. The table should start at 0 and should be monotonously increasing. The discharge should be supplied in atmospheric m³/h. For both characteristic, the values are given as positive values (see examples). When the pressure drop falls out of table, the corresponding air flow will be linearly extrapolated. Please note that this does not include choking flow or other phenomena. It is highly recommended to extend the characteristic if the warning "table out of range" is given.

The characteristic for in/outflow is sometimes measured at a temperature different from the actual ambient temperature. Therefore, the temperature used to define the characteristic has to be specified as well. If this unknown, you can use the same temperature as the ambient temperature.

Example of outflow table

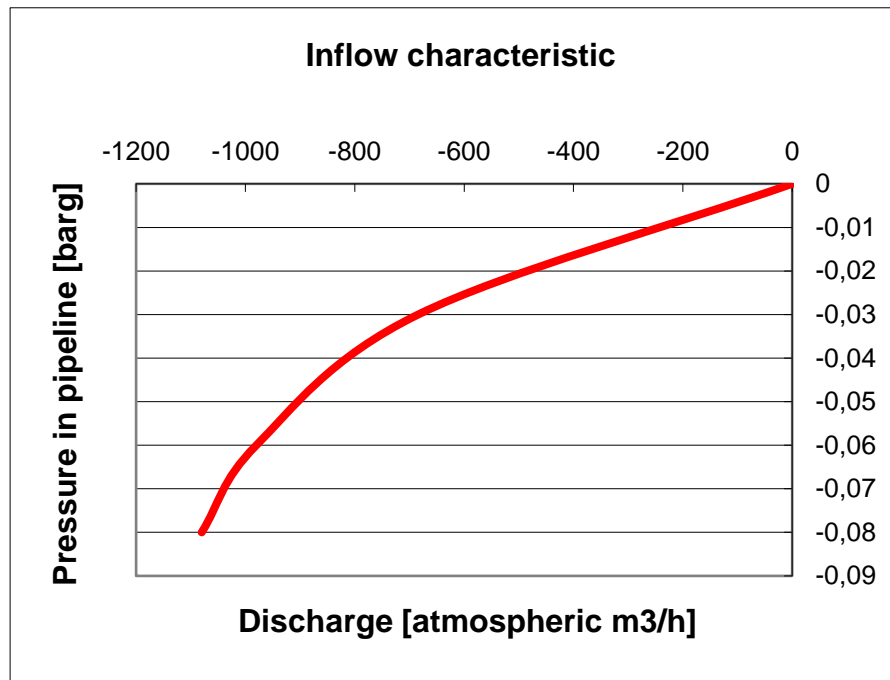
Below you see an example of a outflow characteristic as supplied by the manufacturer, together with the corresponding input table for Wanda.



Pressure difference [bar]	Air flow [m ³ /h]
0	0
0.1	7.2
0.4	15.5
0.8	23
1.2	28
1.5	30.6

Example of inflow table

Below you see an example of a inflow characteristic as supplied by the manufacturer, together with the corresponding input table for Wanda. Please note that the pressure difference and discharge are entered as positive values.



Pressure difference [bar]	Air flow [m ³ /h]
0	0
0.01	360
0.02	540
0.03	684
0.04	792
0.05	864
0.06	936
0.07	-1008
0.08	1080

4.36.3.2 Component specific output

Air volume [m³]

Air flow [m³/s]

Air pressure (absolute) [N/m².abs]

Air temperature [°C]

4.36.3.3 H-actions

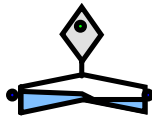
None

4.36.3.4 Component messages

Message	Type	Explanation
Positive initial air volume not allowed in combination with "initial state = closed"	Error	
Contradiction between type and initial state	Error	
"Initial state = outlet" not possible with zero initial air volume	Error	
Starts in closed phase	Info	
Absolute pressure below vapour pressure	Error	This would lead to an infinite air volume. This error can only occur if the VENT is connected to a BOUNDH or other <i>H</i> prescribing component.
Closes	Info	The floating ball valve closes.
Starts in air inlet phase	Info	
Air inlet	Info	System pressure has dropped below the vent elevation.
Starts in air outlet phase	Info	
Air outlet	Info	System pressure has risen above the vent elevation and not all air has yet been expelled.
Starts with closed air valve	Info	
Air valve closes	Info	There may still be air in the system which will be expelled if the pressure rises again above the vent elevation.

4.37 Vent, Air valve with filling pipe section

4.37.1 VENT2LEG (class)



Fall type

type label	description	active
vent2leg	Vent with automatic in/outflow. Capacity specified with discharge coefficients. The level effect is automatically taken into account based on the profile of the pipelines connected to the vent	No

4.37.1.1 Mathematical model

The vent or air valve is a component through which air can enter into or be expelled from the hydraulic system. This is usually achieved by a floating-ball valve mechanism (figures 1, 2 and 3). The purpose of the vent is to prevent cavitation or intolerable underpressures.

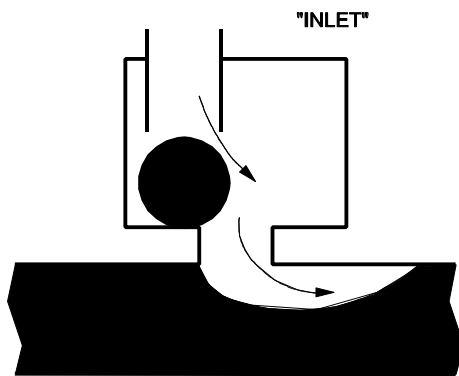


Figure 1: In/outlet vent in air inlet status

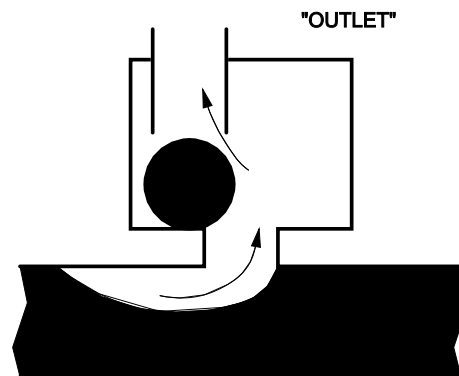


Figure 2: In/outlet vent in air outlet status

If the internal pressure head in the system drops below the elevation of the vent, the floating-ball valve opens and air can enter the system (Figure 1). The air entering the system will expand due to the internal pressure being lower than atmospheric pressure. If the air remains in the vicinity of the vent (e.g. in case the vent is located at a "high" point in the system) it will be expelled through the same vent if the internal pressure rises again above the elevation of the vent (Figure 2). Meanwhile the air will also be compressed due to the internal pressure being higher than atmospheric. The vent closes at the instant the amount of air is smaller than the residual air volume (Figure 3).

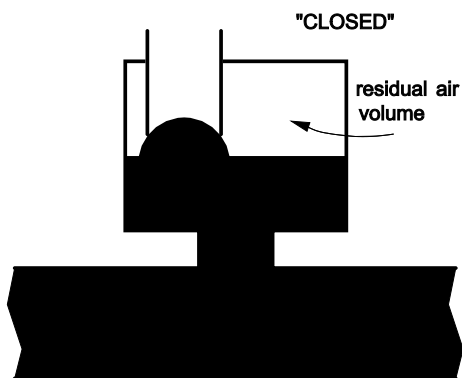


Figure 3: In/outlet vent in closed status

The air flow is compressible, with the consequence that due to local occurrence of supersonic velocities and shock waves, choking flow may arise. The air inlet capacity is therefore truncated to a certain level. If the internal pressure reduces below this level, the air flow does not increase anymore. The expansion and compression of the air may be isothermal, adiabatic or polytrophic. Since Wanda is not a two-phase flow computer code, it is not capable of describing the entrainment and transportation of air in the pipeline as such. The calculations are performed under the assumption that air is not transported into the pipeline. A warning is included when this assumption is no longer valid. This check is based upon the value of the flow number. For more information see Ref. [1] and Ref. [2].

The vent model is merely a boundary condition describing a pressure-discharge relation. In reality, the amount of liquid within the system decreases when air enters the system. The compressibility of the air is taken into account in the discharge supplied by the vent to the system. The vent component supplies liquid to the system and therefore an error in the momentum balance (inertia forces) will be introduced. This error is small, if the amount of air is small compared to the pipeline volume. The continuity balance is not violated.

Changing water levels due to air entrainment or expansion/compression is automatically taken into account based on the profile of the connecting pipes. As the volume of air increases, the fluid level drops accordingly.

For the description of the mathematical model two states are defined:

- Closed air valve,
- Open air valve

In case of "closed-in" air (state 1) the component behaves like an air vessel:

$$P_{air} V_{air}^k = C \tag{1}$$

in which:

- | | | | |
|-----------|---|---|-------------------|
| P_{air} | = | absolute air pressure on fluid level | [Pa] |
| V_{air} | = | air volume | [m ³] |
| k | = | Laplace coefficient (ratio of specific heats) | [-] |
| C | = | Constant | [J] |

The second equation is the continuity equation, which states that the change of volume in air is equal to the in- and outflow of the vent to both connecting nodes.

$$Q_1 - Q_2 + \left(\frac{P_{atm}}{P_{air}} \right)^{\frac{1}{k}} Q_{air} = 0 \quad (2)$$

in which:

Q_1	=	Discharge at connection point 1	[m ³ /s]
Q_2	=	Discharge at connection point 2	[m ³ /s]
P_{atm}	=	Atmospheric pressure	[Pa]
Q_{air}	=	Air flow rate at atmospheric pressure and temperature	[Nm ³ /s]

During the closed state the change of volume of air is due to compression and expansion of the air.

The third equation is an equation equalling the air pressure on both sides of the vent.

The air pressure is given by:

$$P = P_{atm} + \rho g (H_i - w_i) \quad (3)$$

in which:

P	=	Absolute air pressure on fluid level	[Pa]
ρ	=	Density of the fluid	[kg/m ³]
g	=	Gravitational acceleration	[m/s ²]
H	=	Head at i th connection point	[m]
w	=	Fluid level at i th connection point	[m]

Since the air pressure on both sides of the vent is equal, this equation becomes:

$$H_1 - H_2 = w_1 - w_2 \quad (4)$$

In the open state (2) the change of air volume in time is dependent of two phenomena:

1. The compression/expansion of the air.
2. The amount of air leaving/entering the system.

The former is handled in the same way as with state 1 (closed). The latter is determined by formula (5) to (8). Based on the air pressure the direction of the air flow is automatically determined.

Vent capacity (defined by coefficients)

The following formulae are based on Ref. [3].

1. Subsonic air flow in

$$Q_{air} = C_{in} A_{in} \sqrt{7RT_0} \sqrt{\left(\frac{P}{P_{atm}}\right)^{1.4286} - \left(\frac{P}{P_{atm}}\right)^{1.714}} ; P_{atm} > P > 0.53P_{atm} \quad (5)$$

in which:

A_{in}	=	inlet area	[m ²]
C_{in}	=	inlet discharge coefficient	[-]
R	=	gas constant	[J/kg·K]
T_0	=	ambient air temperature	[K]

2. Critical flow in

$$Q_{air} = C_{in} A_{in} \sqrt{7RT_0} 0.259; P < 0.53P_{atm} \quad (6)$$

3. Subsonic air flow out

$$Q_{air} = -C_{out} A_{out} \sqrt{7RT_0} \left(\frac{P}{P_{atm}}\right)^{\frac{k+1}{2k}} \sqrt{\left(\frac{P_{atm}}{P}\right)^{1.4286} - \left(\frac{P_{atm}}{P}\right)^{1.714}} ; \frac{P_{atm}}{0.53} > P > P_{atm} \quad (7)$$

in which:

A_{out}	=	outlet area	[m ²]
C_{out}	=	outlet discharge coefficient	[-]

4. Critical flow out

$$Q_{air} = -C_{out} A_{out} \sqrt{7RT_0} \left(\frac{P}{P_{atm}}\right)^{\frac{k+1}{2k}} 0.259; \frac{P_{atm}}{0.53} < P \quad (8)$$

Level effect

The level effect is automatically included based upon the pipe profile of pipes directly connecting to the 2-leg air valve. It is therefore only possible to connect one pipe to each side of the air valve and no other components. A connection pipe between the air valve and the pipeline can be included by specifying its diameter and height, see Figure 4. For the level effect an engineering approach is used to calculate the volume in bends. Since the most important result is the expelling of the last air, this approximation is acceptable. The dynamic behaviour of the water levels in both legs depends on the actual water levels relative to the bottom elevation. If we denote the local pipe bottom elevation with b , then the following states are feasible:

w_1	w_2	Description	Applicable equations
> b	> b	both levels are identical.	$w_1 = w_2$

			$Q_1 - Q_2 = A_1(w_1) \frac{dw_1}{dt} + A_2(w_2) \frac{dw_2}{dt}$
= b	< b	leg 1 fills leg 2, if $Q_1 > 0$	$w_1 = b$ $Q_1 - Q_2 = A_2(w_2) \frac{dw_2}{dt}$
< b	= b	leg 2 fills leg 1, if $Q_2 < 0$	$w_2 = b$ $Q_1 - Q_2 = A_1(w_1) \frac{dw_1}{dt}$
< b	< b	system is fully decoupled	$Q_1 = A_1(w_1) \frac{dw_1}{dt}$ $Q_2 = A_2(w_2) \frac{dw_2}{dt}$

The fact that the water level remains equal to the local bottom elevation implies that the overflow is idealised.

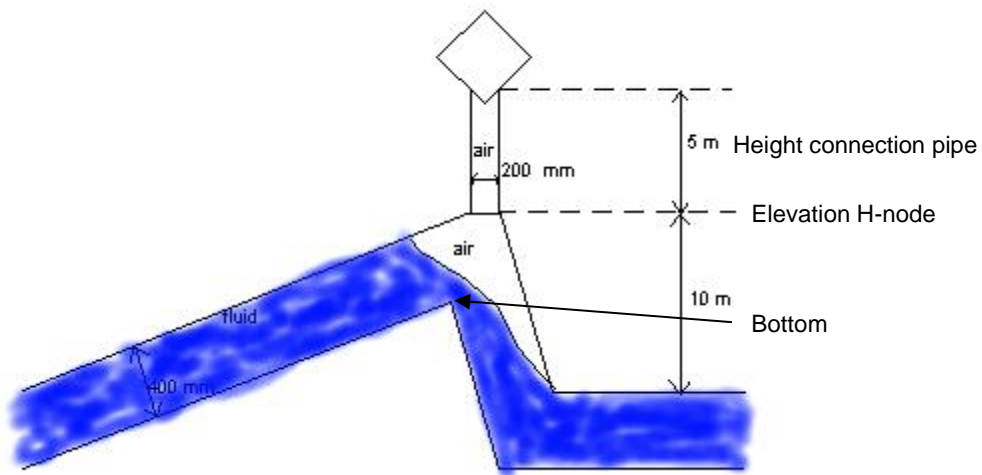


Figure 4: Example of 2-leg air valve with standpipe

Air transport

Wanda is a single phase simulation tool, thus air transport through pipelines is not included. The air valve model can be used for all processes (such as slow filling) where air is expelled or taken in at the same air valve where air is not transported through the pipeline. Air is partially transported when the flow number is greater than 0.6: If the flow number is greater than 0.9 all air is transported. The flow number is given by:

$$Fr = \frac{v}{\sqrt{gD}} \quad (9)$$

in which:

Fr	=	Flow number	[-]
v	=	Flow velocity	[m/s]
D	=	Pipe diameter	[m]

A warning is given when one of these criteria is exceeded. Note that simulation results are invalid if the flow number exceeds 0.6. For more background see Ref. [1] and Ref. [2].

4.37.2 VENT2LEG properties

4.37.2.1 Hydraulic specifications

description	Input	unit	range	default	remarks
Height connection pipe	Real	[m]			
Diameter connection pipe	Real	[m]			
Laplace coefficient	Real	[-]	[1-1.4]		
Ambient air temperature	Real	[°C]			
Inlet discharge coefficient	Real	[-]	[0-1]		See remark
Outlet discharge coefficient	Real	[-]	[0-1]		See remark
Inlet discharge area	Real	[m ²]	≥0		
Outlet discharge area	Real	[m ²]	≥0		
Delta P for open	Real	[Pa]			
Residual air volume	Real	[m ³]	[0-1]		
Initial state	Closed Open			Closed	
Initial upstream fluid level	Real	[m]			
Initial downstream fluid level	Real	[m]			

See also "Mathematical model" on page 449.

Remark

Please be aware that for deriving the discharge coefficients C_{in} and C_{out} from a K_V -value (m³/h if = 1 bar), the manufacturer most probably specifies the capacity in **atmospheric** cubic metres per second. In general, the discharge coefficients describe the amount of contraction of the air flow through the orifices and will probably be in the range of 0.5 to 1.0.

To simulate an air valve, which only lets air into the system the outlet discharge coefficient can be set to 0, in the same way an outlet air valve can be modelled by setting the inlet discharge coefficient to zero.

Important Notice: In case of vents with large capacity it is well possible to have a very sensitive computation in which small pressure differences cause large air flows and volumetric effects. Although the program is optimised to find the most accurate solution, numerical oscillations may still occur (inertia is not taken into account). In that case it is up to the user to adjust (reduce) the time step and the convergence criteria to reduce the oscillations.

See also "Mathematical model" on page 449.

4.37.2.2 Component specific output

Air volume [m³]

Air flow [m³/s]

Air pressure (absolute) [N/m².abs]

Fluid level 1 [m]

Fluid level 2 [m]

Flow number []

4.37.2.3 H-actions

None

4.37.2.4 Component messages

Message	Type	Explanation
Not exactly one pipe connected to connection point i	Error	
Initial upstream fluid level above air valve top	Error	
Initial downstream fluid level above air valve top	Error	
Initial upstream fluid level above pipe bottom high point and initial downstream fluid level below pipe bottom high point	Error	
Initial downstream fluid level below pipe bottom high point and initial upstream fluid level above pipe bottom high point	Error	
Starts in open phase	Error	
Air valve opens	Info	

Message	Type	Explanation
Air valve closes	Info	
Air is transported	Warning	
All air is transported	Warning	

References

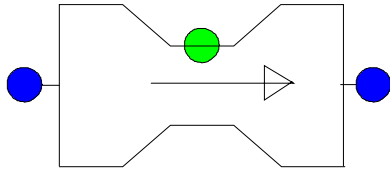
[1] Pothof, I.W.M., Co-currunt air-water flow in downward sloping pipes, 2011, ISBN: 978-90-89577-018-5

[2] Tukker, M.J., Kooi, C, Pothof, I. (2013): Hydraulic design and management of wastewater transport systems, CAPWAT Manual, published by IOS Press BV, Netherlands, <http://capwat.deltares.nl>

[3] Streeter, V.L., Wylie, E.B., 1978: Fluid Transients. McGraw-Hill, New York.

4.38 Venturi

4.38.1 Venturi (class)



Venturi for discharge measurement in open channel

Fall type

type label	description	active
Venturi_open	Venturi for discharge measurement in an open channel, according to norm ISO 1438-1975	no

4.38.1.1 Mathematical model

The VENTURI can be used for a discharge measurement in an open channel by measuring the resistance in the throat of the venturi. The equations which are used to model the venturi are taken from the ISO 1438-1975 norm.

A venturi is described by:

$$Q = \left(\frac{2}{3}\right)^{\frac{3}{2}} C_v C_e \sqrt{g} b h_v^{\frac{3}{2}} \quad \text{or} \quad h_v = \left(\frac{Q}{\left(\frac{2}{3}\right)^{\frac{3}{2}} C_v C_e \sqrt{g} b} \right)^{\frac{2}{3}} \quad (1)$$

in which the constants C_e and C_v are given by:

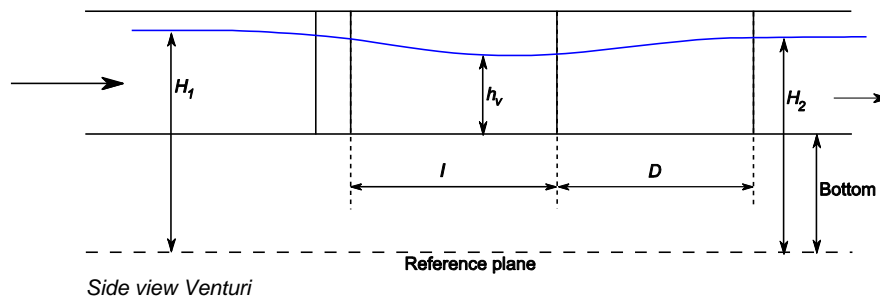
$$C_e = \left(\frac{b}{b + 0,004l} \right)^{1,5} \left(\frac{h_1 - 0,003l}{h_1} \right)^{1,5} \quad (2)$$

$$C_v = 0.3587 \left(\frac{b}{B}\right)^3 - 0.0362 \left(\frac{b}{B}\right)^2 + 0.0651 \frac{b}{B} + 0.9951 \quad (3)$$

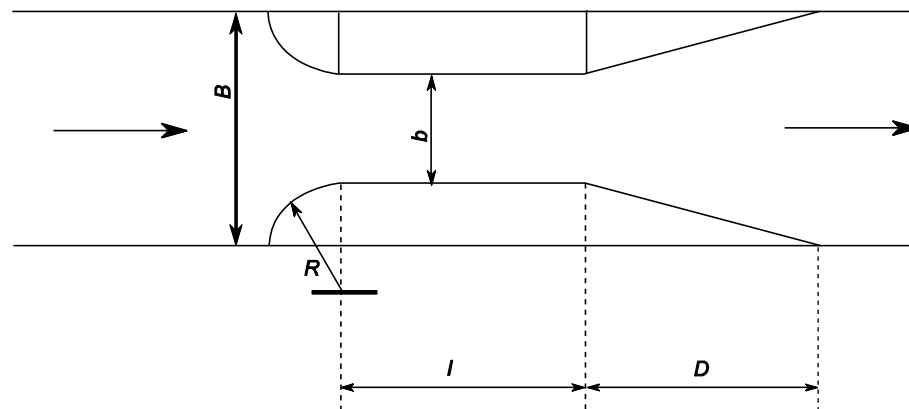
with:

Symbol	Unit	Description
Q	[m ³ /s]	discharge
C _v	[-]	Jameson's velocity coefficient
C _e	[-]	Horner's discharge coefficient
g	[m ² /s]	Gravitational acceleration
b	[m]	Throat width venturi
h _v	[m]	Fluid level in the venturi throat relative to the bottom of the channel
h ₁	[m]	Upstream water level
L	[m]	Throat length
B	[m]	Width of upstream channel

NB: This component can not be used for venturi's without a flat bottom. The use of the venturi component is limited to venturi's in which the throat is formed by a narrowing of the channel.



Side view Venturi



top view Venturi

4.38.2 Venturi_open

4.38.2.1 Hydraulic specifications

Description	Input	Unit	Range	default	Remarks
Length venturi throat	Real	[-]	(0, 1]		
Width venturi	Real	[-]	(0, 1]		
Width upstream channel	Real	[m]	(0; 100]		
Height bottom	Real	[m]	(-100; 10000]		Relative to reference point

See also: "Mathematical model" on page 457.

4.38.2.2 Component specific output

Description	Unit	Remarks
Water depth upstream	[m]	
C _v coefficient Jameson	[-]	
C _e coefficient Horner	[-]	

4.38.2.3 H-actions

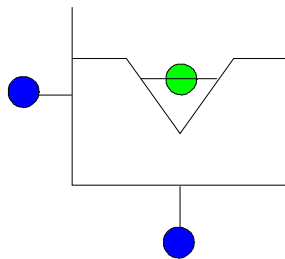
None

4.38.2.4 Component messages

Message	explanation
Error: width ratio outside Cv range	Ratio width throat/channel is outside the range given in the ISO norm [0.1-0.7]
Warning: downstream level above upstream level	Downstream head is larger than upstream head ($H_2 > H_1$), discharge in opposite direction
Warning: Downstream level above 60 % upstream level	Depth is larger than 60% of upstream water level

4.39 Vee notch Weir

4.39.1 V-weir (class)



Thin plate weir with one or more triangular shaped notches

Fall type

type label	Description	active
Vee notch Weir	Thin plate weir with one or more Vee shaped notches, according to ISO 1438/1 1980(E)	no

4.39.1.1 Mathematical model

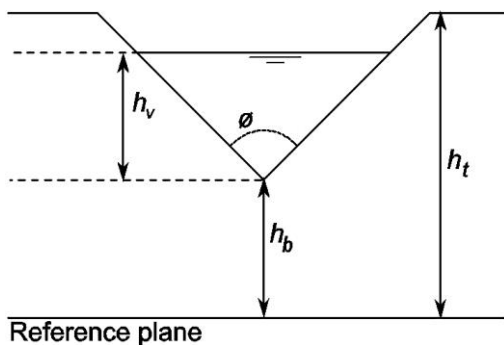
The Vee notch weir is a short, sharp weir with one or more Vee shaped notches. The model can be used for a Vee shaped measuring weir or a Vee shaped weir of a tank.

A precondition is that the upstream fluid level is always in the V-shape and the weir discharge is independent of the downstream fluid level. So, the downstream fluid level should always be below the bottom of the Vee shaped weir.

The weir discharge per Vee shape is determined by:

$$Q = C_D \cdot \frac{8}{15} \cdot \sqrt{2g} \cdot \tan\left(\frac{\theta}{2}\right) \cdot h_v^{\frac{5}{2}} \quad (1)$$

$$F_1 : Q_{tot} = N_v \cdot Q \quad (2)$$



with:

Symbol	Unit	description
Q	[m ³ /s]	Discharge per V shape
C _D	[-]	Discharge coefficient, depended on the shape of the weir.
g	[m ² /s]	Gravitational acceleration
θ	[°]	total V angle
h _v	[m]	Upstream fluid level, relative to bottom of V
h _b	[m]	Bottom level of V shape relative to reference plane
h _t	[m]	Top level of V shape relative to reference plane
N _v	[-]	Number of V's
Q _{tot}	[m ³ /s]	Total weir discharge

The level of the top (h_t) and bottom (h_b) of the Vee notch are given relative to the reference plane.

If the upstream fluid level is above the top of the V shape (h₁>h_t), water flows over the whole edge. There is no critical flow anymore when the downstream fluid level is above the bottom of the V shape (h₂>h_b). In both cases the programme gives an error because you are outside the range of the discharge equation.

The weir is dry when the upstream fluid level drops below the bottom of the V (h₁<h_b). In this case:

$$F_1 : Q = 0 \tag{3}$$

The storage upstream of the weir is no part of the Vee notch weir component and should be modelled with another component (SURGTW).

4.39.2 Vee notch Weir

4.39.2.1 Hydraulic specifications

Description	Input	Unit	Range	default	Remarks
bottom V-shape	Real	[m]	(-100 ; 10000]		Downstream fluid level should be below bottom of V- shape
top level V-shape	Real	[m]	(-100; 10000]		Upstream fluid level should be below top of V-shape
Discharge coefficient	Real	[-]	(0, 1]	0,578	This is a mean value from the standard
V-shape angle	Real	[°]	(20; 100]	60	See standard
Number of V's	Integer	[-]	(1, 1000]	1	
Initial status			vritical / dry	critical	

See also "Mathematical model" on page 460.

4.39.2.2 Component specific output

None

4.39.2.3 H-action

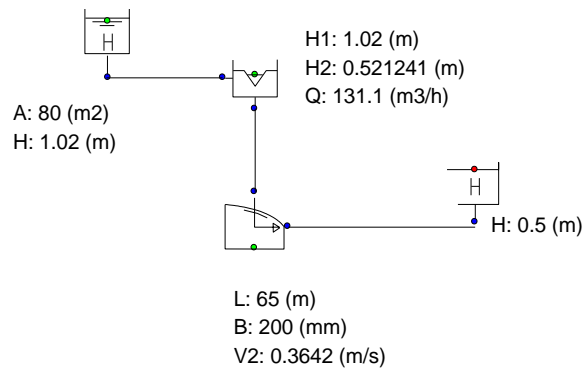
None

4.39.2.4 Component messages

Message	explanation
Starts in critical state	Informative: Upstream fluid level is above the bottom of the V-shape in steady state. $Q > 0$.
Starts in closed state.	Informative: Upstream fluid level is below the bottom of the V-shape in steady state. $Q = 0$.
Closes	Informative, weir becomes dry.
Enters critical state	Informative: Upstream fluid level becomes larger than bottom of V-shape. $Q > 0$
Error: upstream level above top V-notch. Calculation is no longer valid	Upstream level becomes larger than top of V-shaped notch. The calculated discharge is not correct anymore.
Error: Return flow not allowed	Downstream fluid level is larger than upstream fluid level; discharge in opposite direction is not allowed.

4.39.2.5 Example

The following example demonstrates how to model a settling tank for sewage treatment works.



4.40 Weir

4.40.1 WEIR (class)



Weir

Fall type

type label	description	active
WEIR (two way)	Broad crested weir, capable of simulating discharge in positive direction and negative direction.	No

4.40.1.1 Mathematical model

The component WEIR is often used in sewage water systems. The WEIR can have a discharge in positive or negative direction or can be closed. If the upstream head, H_1 , is larger than the downstream head, H_2 , then the WEIR will have a discharge in positive direction. The component can be modelled by the following equations if the flow over the edge is critical (see figure 1):

$$Q = \left(\frac{2}{3}\right)^{1.5} \sqrt{g} B C (H_1 - H_e)^{1.5} \quad (1)$$

$$F_1: \quad \approx 1,7 B C (H_1 - H_e)^{1.5}$$

in which:

Q_1	=	Discharge through WEIR	[m ³ /s]
C	=	Discharge coefficient	[-]
B	=	width of the threshold	[m]
H_1	=	head upstream	[m]
H_e	=	edge level	[m]

The first equation is only valid if the downstream head, H_2 , is lower than the flush criterion head and the upstream head is greater than the edge level:

$$H_2 - H_e \leq \frac{2}{3} (H_1 - H_e) \quad (2)$$

$$H_1 \geq H_e \quad (3)$$

and

$$H_1 \geq H_2 \tag{4}$$

in which H_2 is the head downstream of the weir [m].

If expression (2) is violated, a warning is issued. In this case the WEIR behaves like a flush or submerged WEIR (see figure 2) and equation (1) is no longer valid. If the weir is submerged and , the flow is no longer influenced by the weir. This situation should therefore be avoided. For a submerged weir the following equations are describing the behaviour:

$$F_1: Q = BC(H_2 - H_e)\sqrt{2g(H_1 - H_2)} \tag{5}$$

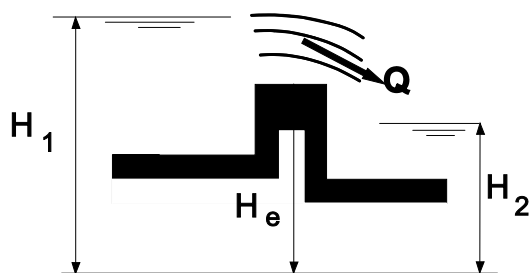


Figure 1: Critical flow over weir

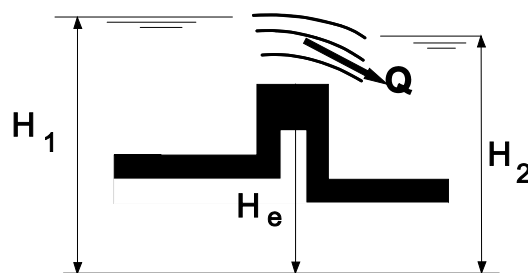


Figure 2: Flush flow over weir

If expression (4) is violated, i.e. the downstream head is larger than the upstream head. The WEIR can cope with this situation and will have a discharge in negative direction. This component can be modelled by the following equations if the negative flow over the edge is critical.

$$F_1: Q = -1.7 \cdot BC(H_2 - H_e)^{3/2} \tag{6}$$

This flow is critical if the upstream head is lower than the flush criterion head and the downstream head is greater than the edge level:

$$H_1 - H_e \leq \frac{2}{3}(H_2 - H_e) \tag{7}$$

$$H_2 \geq H_e \tag{8}$$

and

$$H_2 \geq H_1 \tag{9}$$

If expression (7) is violated, a message is issued. In this case the WEIR behaves like a flush or submerged WEIR. If the weir is submerged, the flow is no longer influenced by the weir. For a submerged weir with negative flow equation (6) is no longer valid. In that case the following equations are describing the behaviour:

$$F_1: Q = BC(H_1 - H_e)\sqrt{2g(H_2 - H_1)} \tag{10}$$

If expression (3) and (8) is violated the WEIR runs dry ('closes'), thus if $H_1 < H_e$ then

$$F_1 : Q_1 = 0 \quad (11)$$

Or verbatim: there is no discharge over the WEIR.

In steady state the WEIR can be 'open' ($H_1 > H_e$ or $H_2 > H_e$) or 'closed' ($H_1 \leq H_e$ or $H_2 \leq H_e$).

Whether the WEIR is open or closed depends on the hydraulic state of the total system. In steady state the user must initialise the initial state of the WEIR (critical, submerged or closed). The program does not calculate the initial state automatically.

4.40.1.2 Discharge coefficient

Broad crested weirs

According to the literature the coefficient C in equations 1 and 2 is the product of two separate coefficients: The approach velocity coefficient C_v which corrects for the velocity upstream of the weir, and the Discharge coefficient C_d which corrects for the energy losses that occur at the weir:

$$C = C_v \cdot C_d \quad (12)$$

The discharge coefficient must be obtained through calibration of the weir. If the discharge coefficient is unknown, an estimate can be obtained with the following equations.

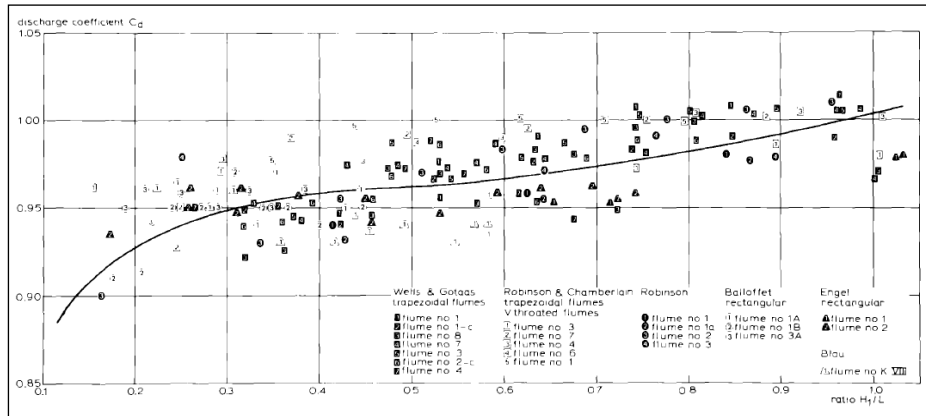
The approach velocity coefficient is given by:

$$C_v = \left(\frac{H_1}{h_1} \right)^{1.5} = \left(1 + \frac{v_1^2}{2gh_1} \right)^{1.5} \quad (13)$$

The water depth and velocity of the approach flow are unknown to Wanda, which means that the calculation of the approach velocity coefficient must be done by the user.

The discharge coefficient C_d corrects for the assumptions made for equation 1.

The figure below shows the relation between the ratio H_1/L (the head upstream of the weir and the length of the weir) and the discharge coefficient.



Short crested weirs

Short or sharp crested weirs are used for flowrate measurements in open channels. ISO 1438/1-1980 presents the following equation:

$$Q = C \cdot \frac{2}{3} \sqrt{2g} \cdot b \cdot \left(h_1 + \frac{v_1^2}{2g} \right)^{3/2} = C \cdot \frac{2}{3} \sqrt{2g} \cdot b \cdot H_1^{3/2} \tag{14}$$

With:

- Q flow over the weir [m³/s]
- C Discharge coefficient [-]
- b width of the weir [m]
- H₁ Head upstream of the weir [m]
- g gravitational acceleration [m/s²]

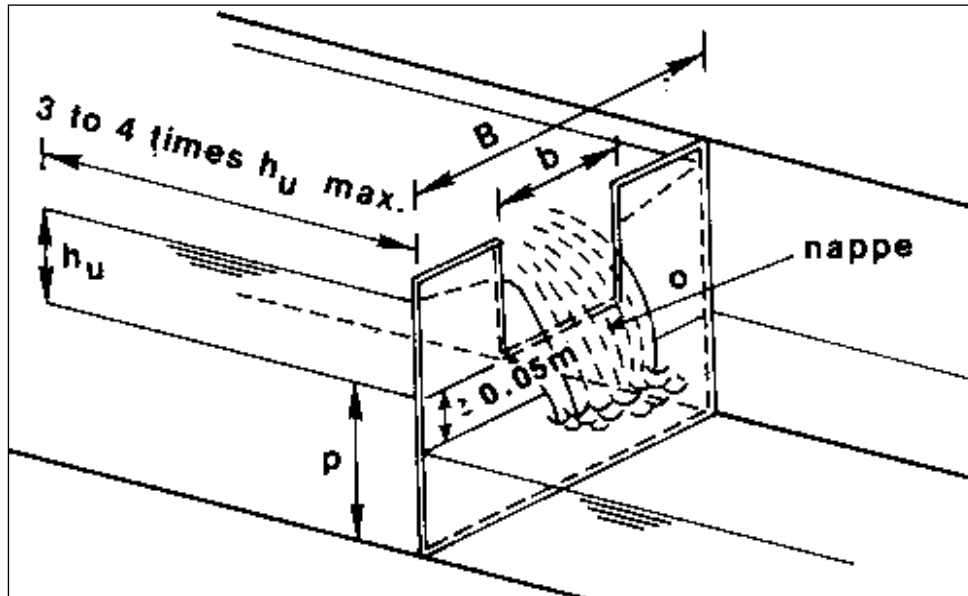
According to ISO, the discharge coefficient is given by the following equation:

$$C = 0,627 + 0,018 \left(\frac{H_1}{p} \right)^{3/2} \tag{15}$$

with:

H ₁	head upstream of the weir	[m]
p	height of the weir	[m]

The following figure illustrates the different parameters



The ISO equation for the discharge over the weir is not the same as the equation that Wanda uses, but this can be compensated with the discharge coefficient. To model the ISO-compliant weir in Wanda we need to use the following equation for the discharge coefficient C:

$$C_{Wanda} = 1,732 \cdot \left(0,627 + 0,018 \left(\frac{H_1}{p} \right)^{\frac{3}{2}} \right) \quad (16)$$

Wanda cannot solve this equation because the water depth upstream of the weir (and thus p) is unknown.

References

- Bos, M.G. "The Use of Long-Throated Flumes to Measure flows in Irrigation and Drainage canals" *Agricultural Water Management*, 1 (1977) p111-126.
- Boiten, W. "Flow measurement structures" *Flow measurement and Instrumentation*, 13 (2002) pag. 203-207
- International Standards Organization (ISO), "Water flow measurement in open channels using weirs and venturi flumes" *International Standard ISO 1438/1-1980*, 1980.

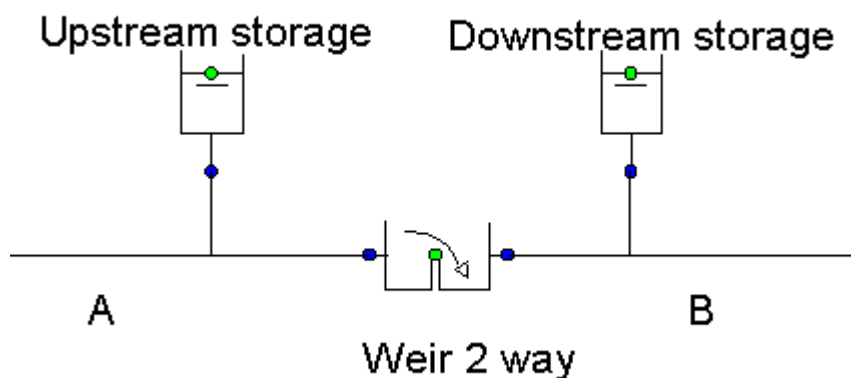
4.40.2 WEIR (two way)

4.40.2.1 Hydraulic specifications

description	Input	Unit	range	default	remarks
edge level	Real	[m]			
width	Real	[m]	(0-1000]		
discharge coefficient	real	[-]	(0-2]		
discharge coeff. (Q<0)	real	[-]	(0-2]		
initial state	critical / submerged/ closed / crit. Q<0/ subm. Q<0			critical	Q<0

It is preferable to take an initial state of the WEIR in the steady state with a discharge in positive direction or closed.

The WEIR does not model the storage of the weir. To model both the up- and downstream storage of the WEIR, three discharges should be known. The discharge entering and leaving the WEIR are necessary to determine how much fluid stays in the storage of the WEIR. The discharge over the edge of the WEIR is necessary to determine how this fluid is divided over the up- and downstream storage. Only two of these variables can be known to Wanda, therefore it is impossible to model the storage both up- and downstream of the WEIR. It is left to the user to, if necessary, model the storage using a surge tower up- and downstream of the WEIR.



See also "WEIR" on page 464

4.40.2.2 *Component specific output*

None

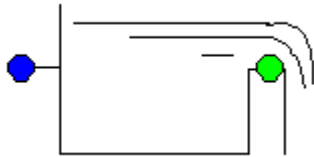
4.40.2.3 *H-actions*

None

4.40.2.4 *Component messages*

Message	Type	Explanation
starts in critical state	Info	
starts in submerged state	Info	
starts in closed state	Info	
enters critical state	Info	
enters submerged state	Info	
closes	Info	
enters critical state, neg. flow	Info	
enters submerged state, neg. flow	Info	

4.40.3 WEIR_S (class)



Weir

Supplier type

type label	description	active
Outlet WEIR (incl. storage)	Broad-crested weir, only capable of simulating discharge in positive direction. Storage upstream of the weir is modelled.	No

4.40.3.1 Mathematical model

The component WEIR_S is often used in sewage water systems. The WEIR_S can have a discharge in positive direction or be closed. If the upstream head, H_1 , is larger than the edge of the WEIR_S, H_e , then the WEIR_S will have a discharge in positive direction. The component can be modelled by the following equations if the flow over the edge is critical (see figure 1):

$$F_1: Q = 1.7 \cdot BC(H_1 - H_e)^{3/2} \tag{1}$$

in which:

- Q = Discharge through WEIR [m³/s]
- C = Discharge coefficient [m^{1/2}/s]
- B = width of the threshold [m]
- H_1 = head upstream [m]
- H_e = edge level [m]

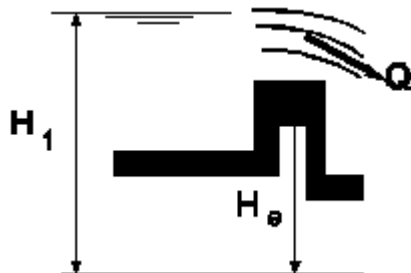


Figure 1: Critical flow over weir

(For more information about the discharge coefficient see "Discharge coefficient" on page 466)

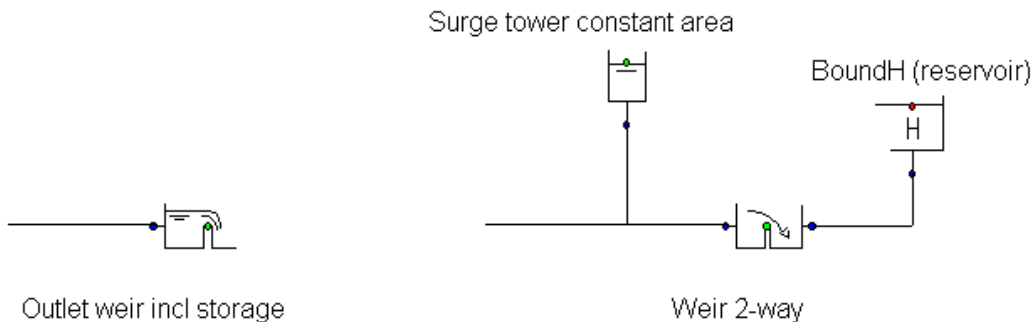
4.40.4 Outlet WEIR (incl. storage)

4.40.4.1 Hydraulic specifications

description	Input	Unit	range	default	remarks
edge level	Real	[m]			
width	Real	[m]	(0-1000]		
discharge coefficient	real	[-]	(0-2]		
Area upstream storage		[m ²]	(0.001-500]		
initial state	critical/ dry			critical	

The WEIR_S (with storage) is a one way weir with critical flow only. The storage upstream of the weir is modelled. This weir can only be used as boundary condition in a system.

For outlet structure the WEIR_S can be used Instead of the combination Surge tower – Weir (2-way) – Boundh;



See also "WEIR_S (class)" on page 471.

4.40.4.2 Component specific output

Fluid level [m]

Discharge about weir [m³/s]

4.40.4.3 H-actions

None

4.40.4.4 Component messages

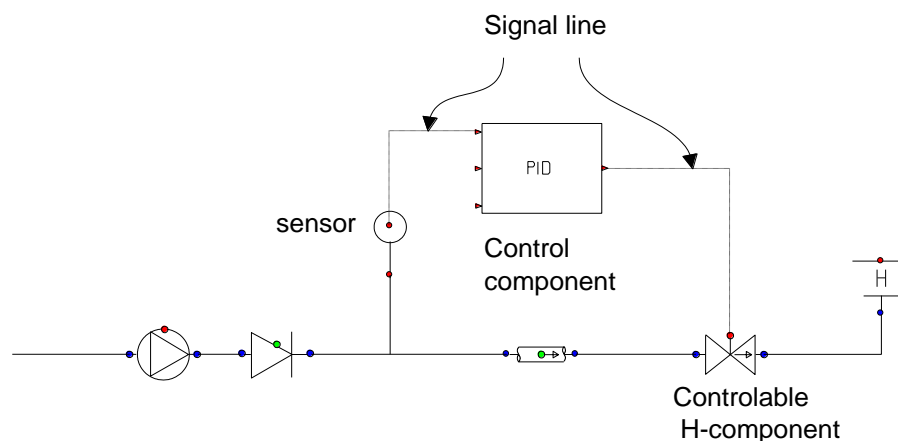
Message	Type	Explanation
starts in critical state	Info	
starts in dry state	Info	
enters critical state	Info	
Enter dry state	Info	

5. WANDA Control

5.1 Control Components

WANDA CONTROL is a further development of the WANDA hydraulic analysis tool. It enables the user to simulate a wide variety of control systems added to the hydraulic system. This is performed by superimposing a control network over the hydraulic network. The control network can respond autonomously to events and values occurring elsewhere in the system. The added value of WANDA CONTROL lies in the far more realistic simulation of the dynamic behaviour of the controlled system than with conventional control simulation packages. Another added value lies in the possibility of having scenarios simulated in a far more automated way than WANDA without CONTROL.

The control system is schematised as a network of sensors, signal lines and control components ("blocks").



These terms will be defined more exactly in the next section.

There are basically two types of control systems:

- Conditional actions
- Continuous control

The former generates an event, like start of a pump, closure of a valve, etceteras. The latter generates a value for the controlled variable such as valve position or pump speed.

5.2 Overview Control components

In the table below an overview is given of all control components available in WANDA.

Numerical output signal generators	
SENSOR	Measured signal from hydraulic system
CONST	Generates constant output value
TAB	Generates output value by interpolation in a time table mono-directional
TAB2	Generates output value by interpolation in a time table bi-directional
FUN	Generates output value by evaluating a time function
WAVE	Generates a periodic, time dependent signal.
TIME	Outputs the model time at every time-step
LtoN	Transfer logical to numeric
MSP	Multi stage pattern
Conditional actions	
COND	Condition (comparison)
COND2	Condition (comparison) with additional enable input port
2COND	Two Conditions
LCS	Level Control Switch
LCS2	Level Control Switch with additional enable input port
SELECT	Select one or other input channel depending status third input channel
Continuous controllers	
PID	Proportional, Integrating, Differentiating controller (Continuous)
PID_N	Proportional, Integrating, Differentiating controller (Continuous), normalized
PIDSS	Proportional, Integrating, controller with start- stop
PIDSS_N	Proportional, Integrating, controller with start- stop; normalized
Logical operators	
AND	AND (all)
OR	OR (at least one)
XOR	Exclusive OR (either the one or the other)
NOT	Negation
Numerical operators	
SUM	Addition
MUL	Multiplication
SUB	Subtraction
DIV	Division
SFUN	Transformation of an input signal according to a standard function
STAB	Transformation of an input signal according to a table
MIN	The smallest of the input signals
MAX	The largest of the input signals
MOVAVG	Moving average of the input signal
ABS	Absolute value of input signal
SIGN	Sign of input signal
BIAS	Addition by a constant value
GAIN	Multiplication by a constant value
RAMP	Maximum change per second
TRUNC	Truncation of input signal
INTGRT	Integrator
MOD	Modulo
Time delay components (travel time of signals)	
DELAY	Delay for numerical (measured) signals
LDELAY	Delay for logical signals
Logical signal generators	
LCON	Generates constant logical output value

LTAB	Generates logical output value by interpolation in a time table
Various	
RDFILE	Read input from file
WRFILE	Write output to file

5.3 Modeling aspects

5.3.1 Measured signals

The signals that may be measured are dependent of the location to which the sensor is attached. In general, pressure type quantities are defined in H-nodes, whereas flow type quantities are defined within (entry and exit) of components. Furthermore, it is possible to measure certain component specific quantities such as pump speed, pump status (running, off), valve position, etceteras.

An overview of the measurable quantities is given in the table below:

Hydraulic object	Measured variables
H-node	H, P
Pipe	Q, v (only at begin or end of Pipe)
Fall type	Q, DH, DP, component specific output
Supplier type	Q, component specific output

in which:

H	=	pressure head	[m]
P	=	pressure	[Pa]
Q	=	discharge	[m ³ /s]
v	=	velocity	[m/s]
DH	=	pressure head difference	[m]
DP	=	pressure difference	[Pa]

The component specific quantities are the same as the component specific output described at the H-component definitions

5.3.2 Controlled hydraulic components

In the table below the controllable H-components are listed, together with their manipulated variable.

H-component	property remark	Type of input signals	Manipulated variable
BOUNDH	-	numerical	Pressure head [m]
BOUNDQ	-	numerical	Discharge [m ³ /s]
FCV	-	numerical	setpoint
PdCV	-	numerical	setpoint
PuCV	-	numerical	setpoint
PUMP	Drive type =running	numerical	Pump speed [rad/s]
	Drive type =trip	Both	Run (T), Trip (F) ^{*)} ; Pump speed [rad/s]
	Drive type = speed	numerical	Pump speed [rad/s]
	Drive type = motorfreq	numerical	Relative frequency [-]
PUMPSS		numerical	Motor speed [rad/s]
VALVE		both	Valve position [-] ^{**)}

(*) One way only

(**)The valve can be connected to both a numerical signal line and a logical signal line. The logical signal line will enable/disable the control of the valve depending on its value: TRUE/FALSE. If the control is disabled the valve will maintain its last control value.

If both an action table and a control table are connected/defined, the control table will prevail.

Hence in steady state the hydraulic system as well as the control system must be in balance, a message is given if the initial hydraulic steady state value of the controllable H-component differs to much from the control value.

5.3.3 Data flow Unsteady

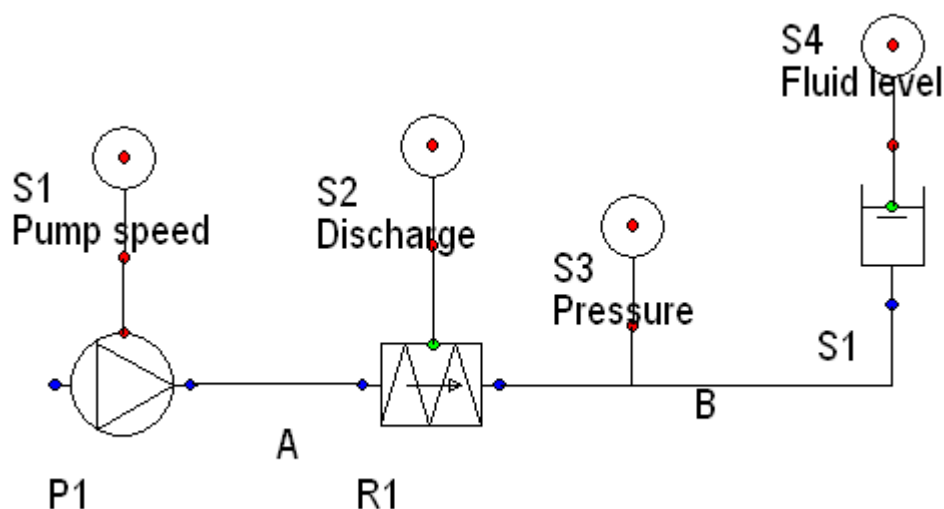
For every time step the control system is evaluated after the hydraulic solution has been obtained. The consequence of this is that the output signals of the control system are synchronous with the hydraulic solution. At the beginning of the next time step all hydraulic components which are connected to actuated signal lines will process the signals. This implies that the actuation has a delay of one time step. This is an explicit approach in contrary of an implicit approach in which an integrated solution of hydraulic and control variables would be obtained (at the cost of substantially more modelling of the control components physics).

5.3.4 Control Diagram aspects

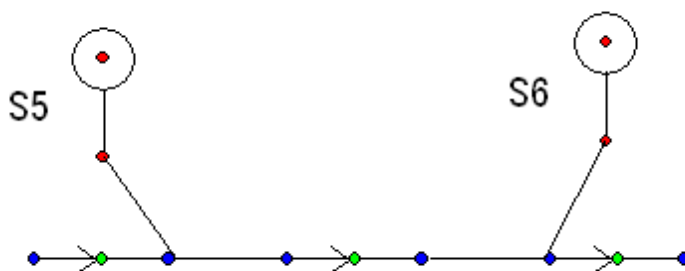
While entering the control system the user is bound by the following rules:

- Combined control relations can be constructed using:
 - arithmetic operator blocks (for numeric values): '+', '-', '*', and '/'.
 - logical operator blocks (for logical values): 'AND', 'OR', 'NOT' and 'XOR'.
- The commutative operators '+', '*', 'AND' and 'OR' have one input channel to which many signal lines may be connected. The non-commutative operators '-' and '/' have two input channels, the first for the left part (a) of the expression, the second for the right part (b) of the expressions a-b and a/b. The 'NOT' operator is unary, that is it has only one input signal. The 'XOR' operator has one input channel at which exactly two logical signal lines must be connected.
- Sensors can be connected to the hydraulic diagram in several ways:

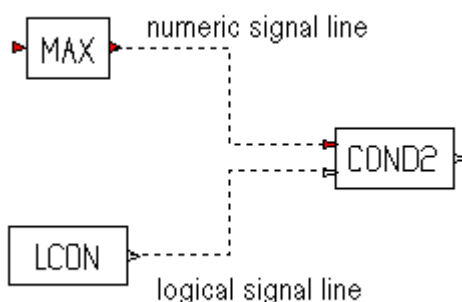
Examples are given in the figure below. Measured are: pump speed (sensor S1), discharge through resistance (sensor S2), pressure in H-node (sensor S3) and level of surge tower (sensor S4).



- For pipes only the flow values Q and v at the first and last pipe-nodes can be measured. The sensor must be placed at the hydraulic connect points (the blue ones) of the pipe. Examples of sensor placement are given in the figure below:



- C-components can be placed freely in the diagram. While the box contains the class annotation the shape can not be flipped (only rotated)
- The maximum numbers of signal lines allowed to be connected to the input and output channels are defined in the particular class description: the C-component descriptions. In general, the number of connections to output channels is unlimited.
- Signal lines can only connect output and input channels of the same type (logical to logical and numeric to numeric).



- The unit dimension of the value measured by the sensor is always in SI-units. Changing the unit mode has no affect on the Control scheme.
- When a logical input channel is not connected to a signal line the default value taken by the C-component procedure will be TRUE. A numeric input channel must always be connected to at least one signal line.
- The measured quantity of a SENSOR is initially undefined, which is indicated by a "?".
- When the unit dimensions of two signal lines connected to the same input channel do not match, a warning is issued, but the connection is accepted. This is due to the fact that the user yet has to specify some of the measured quantities. At the final check of the control system the unit dimension must be of the same type.
- When a C-component is deleted, all connected signal lines are removed as well. Deletion of a signal line has no consequences. Changing of the type of an H-component, will not result in the deletion of the connected signal line. It is possible that the connected C-component or signal line become invalid due to this changing if the definition of the former one doesn't match the definition of the latter one.

5.3.5 Control Output properties

After completion of the computation time history plots of the output signals of all C-components can be made. The unit dimension will always be "dimensionless": [-]. It is the user's responsibility to select signals for multi-line charts such that the ranges of the selected signals are of the same order of magnitude.

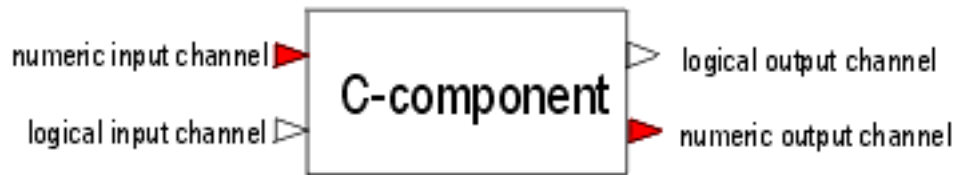
A very interesting opportunity is the possibility to use C-components and signal lines to combine hydraulic output and generate derived signals. For instance one can make a time history plot of the sum of two discharges or the difference between two pressures.

It is therefore well conceivable that the user enters a control system in which not all output channels are connected to signal lines.

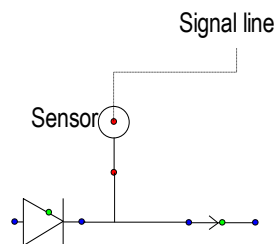
5.4 Definitions

The control extension of WANDA involves some terms and expressions, which are explained here in a hierarchical fashion. After the definitions some general remarks concerning the control diagram are made.

Control components	Blocks that can be placed anywhere in the scheme. They generate one or more output signals, based on zero or more input signals (by conversion). To enable signals to be transferred the control blocks are provided with input and output channels.
Input channel	Hot spot of control component via which one or more input signals can enter. It is denoted by a triangle pointing towards a control component. The signal may be numeric or logical, denoted by respectively a filled (solid) or an empty (hollow) triangle.
Output channel	Hot spot of control component via which one output signal can exit (through multiple signal lines). It is denoted by a triangle pointing away from a control component. The signal may be numeric or logical, denoted by respectively a filled (solid) or an empty (hollow) triangle.



Sensor	Component that "measures" a quantity in a hydraulic component or hydraulic node. The measured quantity can be further transferred as a signal to a input channel of a control component denoted by respectively a filled or an empty triangle.
---------------	--



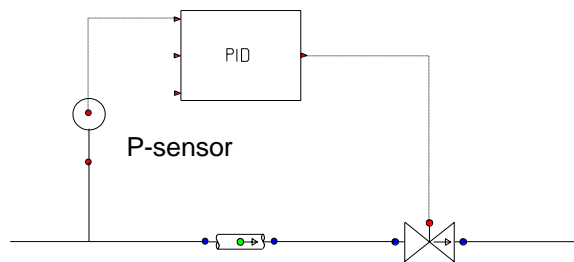
Signal generator	Component that generates a signal autonomously. May be a constant, a time table or a time function.
Conditional action	Generates an event if a certain condition becomes true.
Controller	Continuously controls/steers a system based on the difference (error) between a set value and a measured value.
Operator	
Arithmetic	Operator that converts and combines numbers, like '+', '-', '*' and '/'.
Logic	Operator that converts and combines logical values (TRUE/FALSE), like 'AND', 'OR', 'NOT' and 'XOR'.
Signal line	Line connecting an output channel of a control component to an input channel of another control component. It may also come from a sensor or go to a controlled hydraulic component. Signal lines can convey numeric data as well as logical data. They are shown in the schematics as dashed lines.
Sensor line	Line connecting a hydraulic component to a sensor component. Sensor lines only convey numeric data. They are shown in the schematics as solid lines.
Controlled hydraulic component	Hydraulic component (pump, valve) to be actuated by the control system, by means of a change of the controlled variable (pump speed, pump on/off, valve position).

5.4.1 Control diagram remarks

The elements that can be applied in the control scheme are:

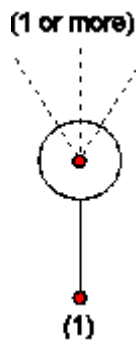
sensor	This element is graphically represented by a circle connected to the end of a solid line. This line is to be connected to the hydraulic component/node to be measured and conveys the measured signal. This is actually the input channel of the sensor. The output channel is situated at the middle of the circle and is the starting point of one or more signal lines.
signal line	This element is represented by a dashed line
control component	This element is represented by a box with small triangles attached. The triangles pointing towards the box are input channels, those pointing away from the box are output channels. The filled (opaque) triangles denote channels conveying numerical signals, the empty (transparent) triangles denote channels conveying logical signals. The class of the control component is denoted by a text within the box (COND: condition, LCS: level control switch, PID: PID controller, etceteras).

In the figure below an example system is shown containing the above mentioned control elements. The system consists of a valve controlling the upstream pressure. The actual upstream pressure is measured by a sensor. From here a signal line conveys the pressure value to the - numerical - input channel of the PID control component. This component converts the input value to a numerical output value (valve position), which is conveyed to the valve by a second signal line. The fact that the valve is controllable is denoted by a red dot at top of the symbol



5.5 SENSOR

Symbol



Purpose

Measures a quantity from the hydraulic system and sends the values via signal lines to other components.

Procedure

After obtaining the hydraulic solution at each time step, the measured quantity is sent to the output channel.

Parameters

Parameter	input	unit	range	default	remarks
Quantity					

Remarks

The output channel is located at the centre of the sensor circle.

Sensors can be used to measure hydraulic quantities (eg. head, discharge, velocity) or component specific quantities (eg. valve position, pump speed)

Example

Figure 3 presents three sensors measuring different quantities along a pipeline:

- The left sensor is connected to a hydraulic node and is set to measure pressure head.
- The middle sensor is connected to the downstream end of a pipe and is set to measure discharge (the sensor could also have been connected to the upstream end).
- The right sensor is connected to a valve and is set to measure the head loss across the valve.

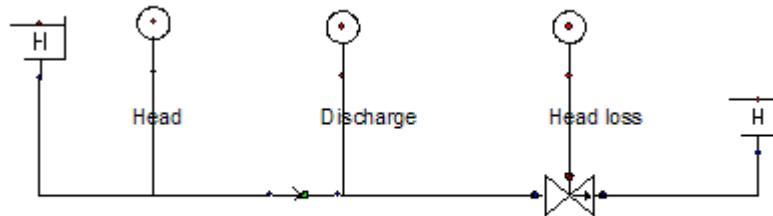
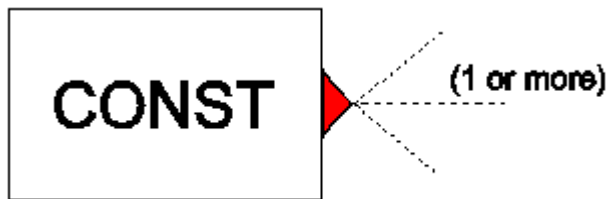


Figure 3: Three sensors measuring hydraulic quantities at different positions along a pipeline

5.6 CONST

Symbol



Purpose

Generates a constant value that can be used as input for Hydraulic components or Control components.

Procedure

This component generates a constant value, which is always (steady state, unsteady state) available at the output channel.

Parameters

Parameter	input	unit	range	default	remarks
Value		real			

Remarks

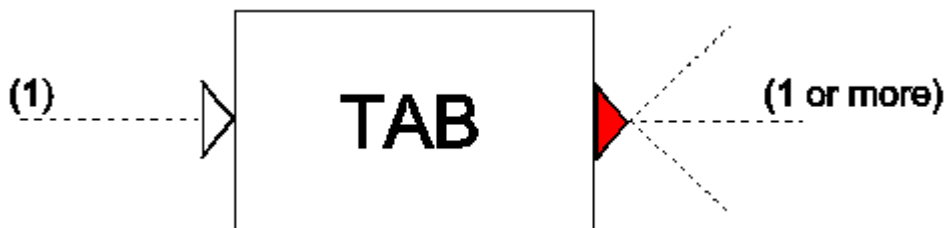
None.

Example

None.

5.7 TAB

Symbol



Purpose

Generates a time dependent numerical value which can be used by other components. Mono-directional.

Procedure

The TAB component is similar to a standard hydraulic action table, with only one difference. The time in the 'action table' is relative to the instant at which the input signal becomes TRUE (=1). When the input channel becomes FALSE (=0), the output remains constant until the input signal becomes TRUE again, after which the table will resume from its present value. Hence, the table is mono-directional.

Parameters

Parameter	input	unit	range	default	remarks
Table type	Valvepos/ pumtpsp/ boundhth/ boundqtq/ CtrlActn			CtrlActn	This type is not used anymore; type will be ignored
Action table	table				

Remarks

If no logical signal line is connected to the input channel a value of TRUE is assumed. In such a case the TAB component is similar to a standard hydraulic action table. Because the control system has a delay of one time step, the point in time of TAB is corrected with -dt to act like the hydraulic action table. As a consequence of this shift the TAB interpolates in steady state between the 1st row (t=0) and 2nd row (t=T1).

Example

Figure 1a presents the input 'action table' for the TAB component. Figure 1b shows the output of TAB if no conditional control is connected to the second input channel.

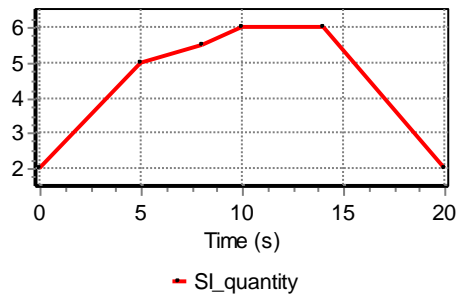
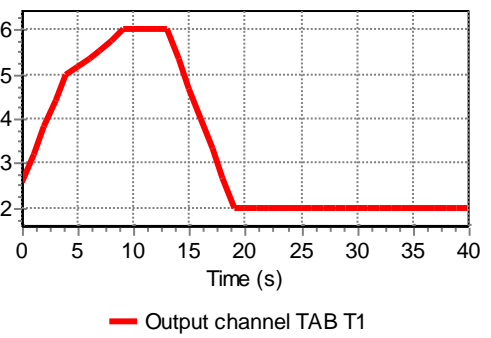
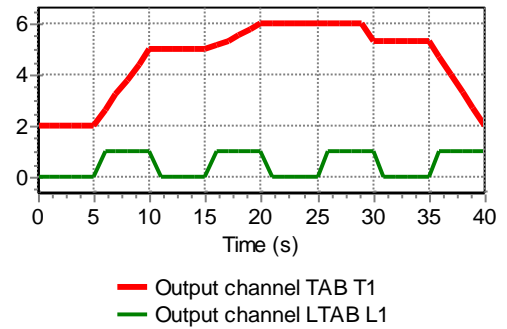


Fig 1a: input TAB



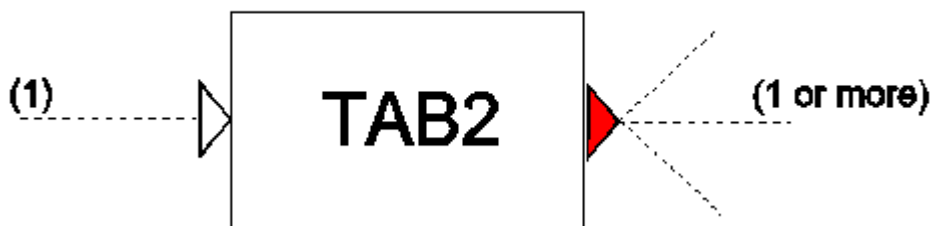
output TAB (be aware of -dt shift)

Figure 2 shows the output of TAB (thick red line) together with the logical input signal (thin green line), specified by a LTAB component.



5.8 TAB2

Symbol



Purpose

Generates a time dependent numerical value which can be used by other components.

Procedure

The TAB2 component can be used to operate a hydraulic component (e.g.: start/stop of a pump, open/close a valve). The time in the 'action table' is relative to the instant at which the input signal becomes TRUE(=1). This holds for the TRUE control 'action table'. If the input signal becomes FALSE (=0), the second 'action table' will be used to generate output values. The TAB2 component takes care that the transition from TRUE to FALSE is continuous (the current output value will be the starting point in the second table). The two 'action tables' allows the TAB2 component to be bi-directional. The start value of the first table must correspond with the end value of the second table and vice versa. The tables should be monotoniously decreasing and increasing or vice versa.

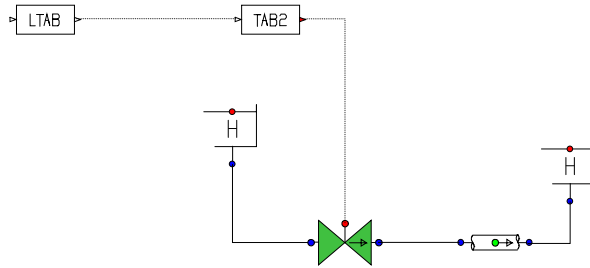
Parameters

Parameter	input	unit	range	default	remarks
Table type	Valvepos/ pumtpsp/ boundhth/ boundqtq/ CtrlActn			CtrlAcnt	This type is not used anymore; type will be ignored
Action table (true)	table				start value = end value of Action table (false);
Action table (false)	table				start value = end value of Action table (true)

Remarks

If no logical signal line is connected to the input channel a value of TRUE is assumed. In such a case the TAB2 component is similar to a standard action table.

Example



The table below presents the TRUE and FALSE 'action tables' for the TAB2 component with respectively the non-linear opening and closing control of a valve. Figure 2a presents its output and input signals. The green line is the logical input signal, while the red line is the numerical output signal.

True table used for VALVE opening	
Time (s)	SI_quantity (-)
0.0	0
8.0	0.2
10.0	1.0

False table used for VALVE closing	
Time (s)	SI_quantity (-)
0.0	1
2.0	0.2
10.0	0.0

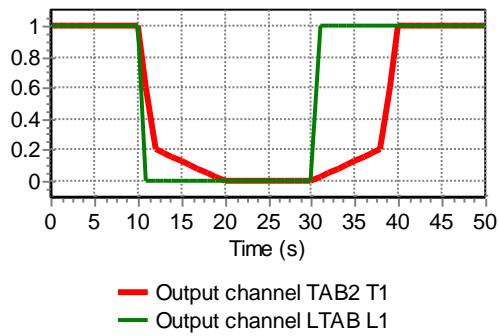


Fig 2a: Control output

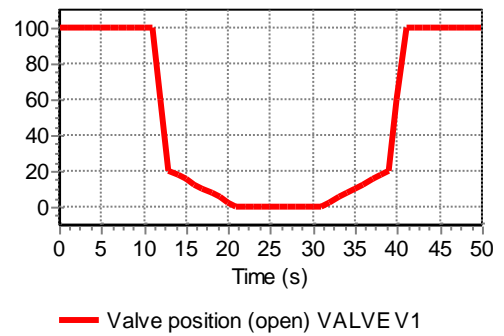


Fig 2b: Valve position

5.9 FUN

Symbol



Purpose

To create a time-dependent signal.

Procedure

If the input signal is FALSE, the output signal is zero. If the input signal is TRUE, one of the following signals are output:

Sine
$$y = A \sin(B(t - t_{start}) - C)$$

Parabola
$$y = A + (B + C(t - t_{start}))(t - t_{start})$$

Exponential
$$y = A e^{B(t - t_{start}) - C}$$

where t_{start} is the time that the logical input signal became TRUE.

Parameters

Parameter	input	unit	range	default	remarks
Function	Sine/ Polynome/ Exp				
A	real				
B	real				
C	real				

Remarks

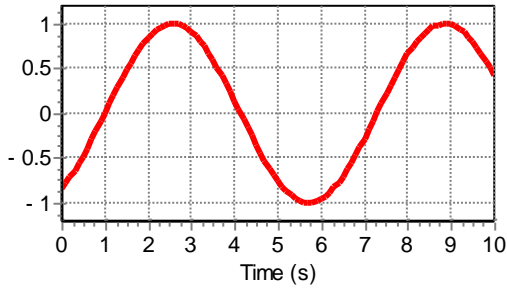
If no input signal is connected to the input channel a value of TRUE is assumed

Examples



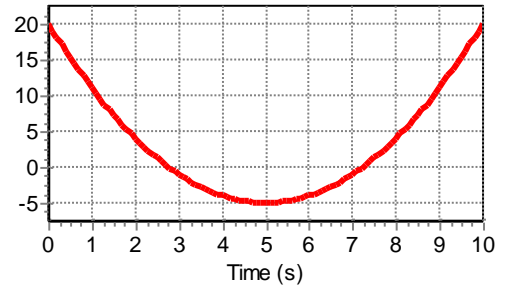
L1

Sine



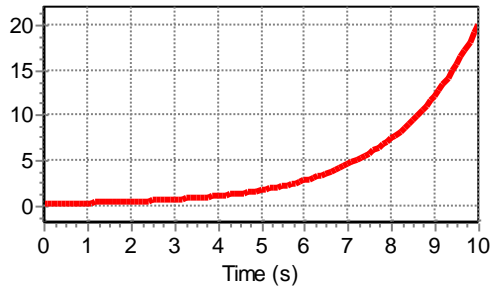
— Output channel FUN Sine

Sine (input: A=1, B= 1, C=1)



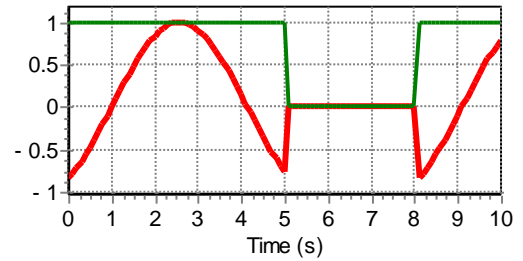
— Output channel FUN Polynome

Polynome (input: A=20, B=-10, C=1)



— Output channel FUN Exponential

Exponential (input: A=1, B=0.5, C=2)

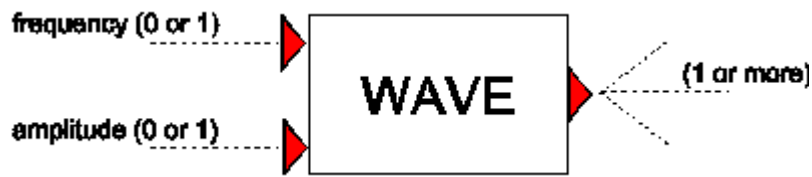


— Output channel FUN Sine
— Output channel LTAB L1

Sine function (input: A=1, B= 1, C=1) controlled by a logical input signal

5.10 WAVE

Symbol



Purpose

Generates a periodic, time dependent signal.

Procedure

The WAVE component can produce Sine, Square and Sawtooth signals:

The signal is defined by an amplitude, period ($=1/\text{frequency}$) and bias as explained in figure 1.

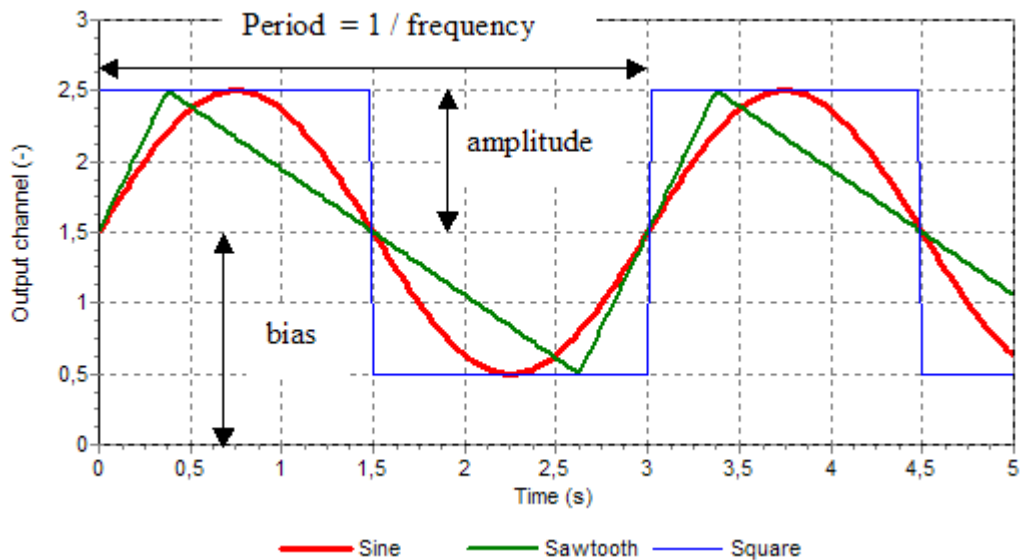


Figure 1: the three different wave types and input definitions

The start of the wave is defined by the so-called initial phase fraction. This phase is a fraction of a period. In figure 2 the wave signals of sine, sawtooth and square have an initial phase fraction of respectively 0.25, 0.5 and 0.75.

For the Sine signal next equation is used:

$$y = a \cdot \sin(2\pi(f \cdot b \cdot t - c)) + d$$

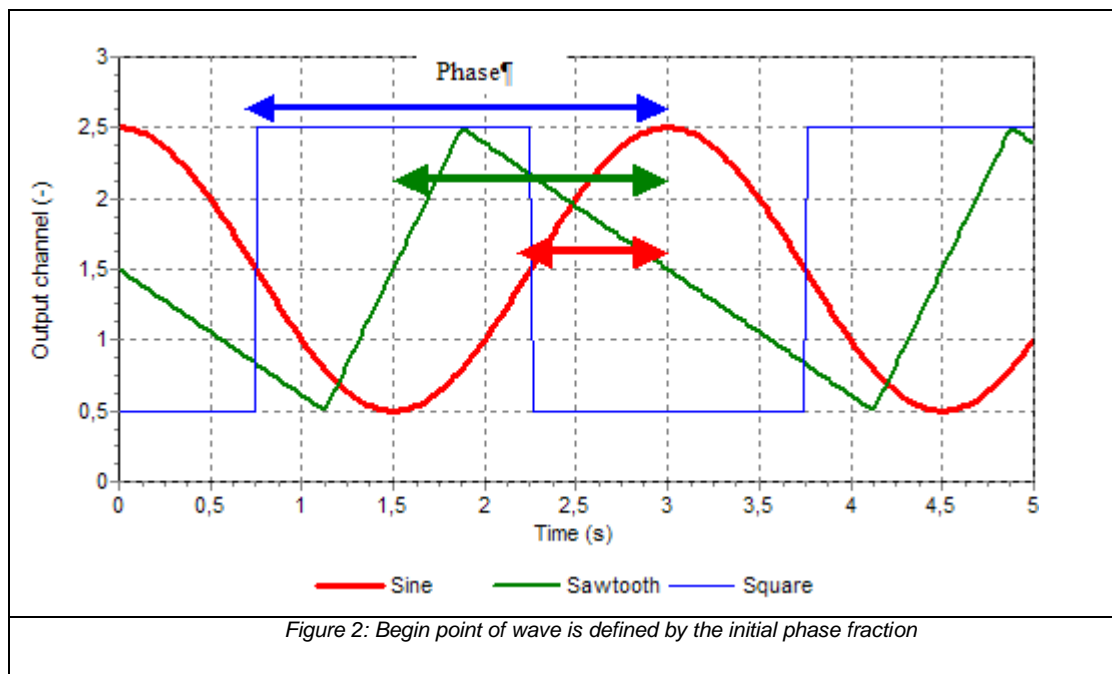
in which:

- a = amplitude (-)
- b = frequency multiplier (-)
- f = frequency (Hz)
- t = time (s)
- c = initial phase fraction (-)
- d = bias (-)

The frequency and amplitude are input parameters but they can be overruled by the signal lines connected to the input channels.

Using the frequency input channel the ratio between the specified frequency and amplitude parameters can be maintained (Freq dependent amplitude = True) during the simulation.

To prevent start-up phenomena the amplitude can be increased from 0 to its full value within a specified start-up time.



Parameters

Parameter	input	unit	range	default	remarks
Waveform	Sine/ Square/ Sawtooth				
Frequency dependent amplitude	Boolean		YES/ NO		Ignored if a signal is connected to the amplitude channel
Initial frequency	real	Hz			
Initial amplitude	real	[-]			
Initial phase fraction	real	[-]	[0,1]	0	defines begin of wave; part of period shifted to left
Rising phase fraction	real	[-]	[0,1]		If type = Sawtooth: Fraction of sawtooth period for rising part
Frequency multiplier	real	[-]	[1e-8, 1e8]	1	
Wave bias	real			0	
Start-up time	real	[s]	[0, 1e8]	0	

Messages

Message	Type	Remarks
Timestep too large for Period (1/freq)	Warning	Required time step less than 1/10 of period
Period (1/freq) must be at least 2 * dt	Error	Frequency and time step do not match; use smaller time step

Remarks

An application for the frequency multiplier are pump induced vibrations. These vibrations depends on the number of blades but the main frequency is defined by the pump speed which is connected to the frequency channel. The frequency multiplier is the number of blades.

Example

The input parameters of the three waves signals of Figure 1 are:

Initial frequency	0.33 (Hz)
Initial amplitude	1 (-)
Initial phase fraction	0.0 (-)
Frequency multiplier	1 (-)
Wave bias	1.5 (-)
Start-up time	0 (s)

In figures 3 – 7 some combinations of two sine waves are shown. The input parameters of the basic sine wave are listed below:

- Initial frequency 1 (Hz)
- Initial amplitude 1 (-)
- Initial phase fraction 0.0 (-)
- Frequency multiplier 1 (-)

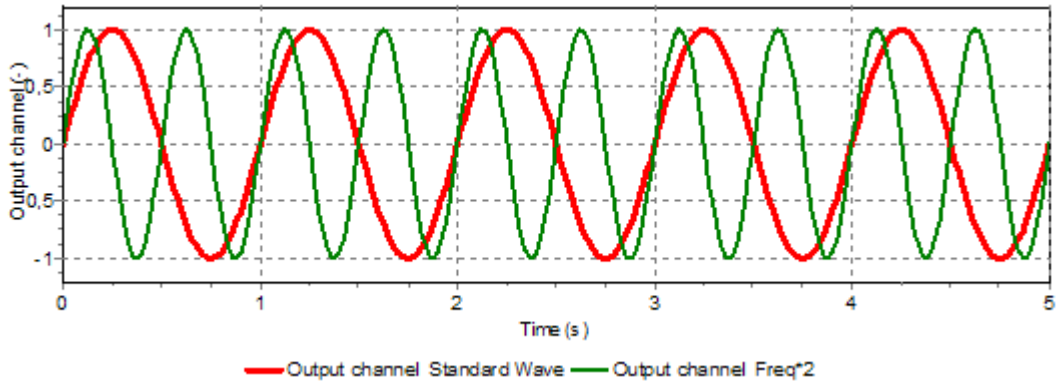


Fig 3: Different initial frequencies (1 and 2 Hz)

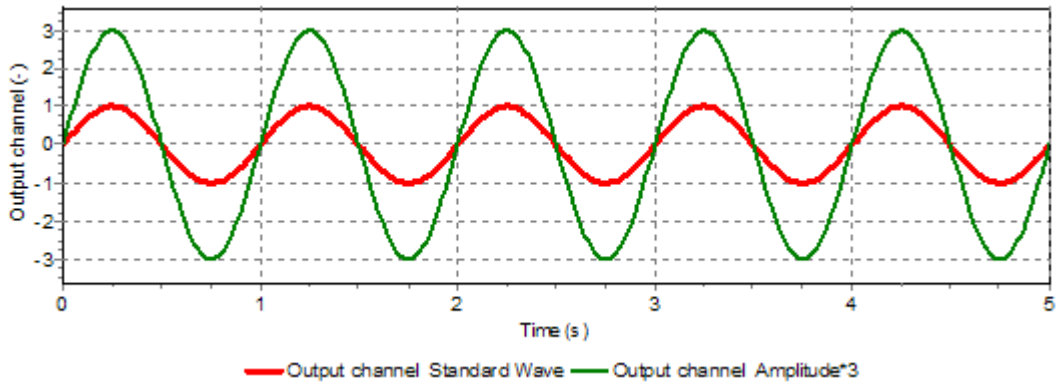


Fig 4: Different initial amplitudes (1 and 3)

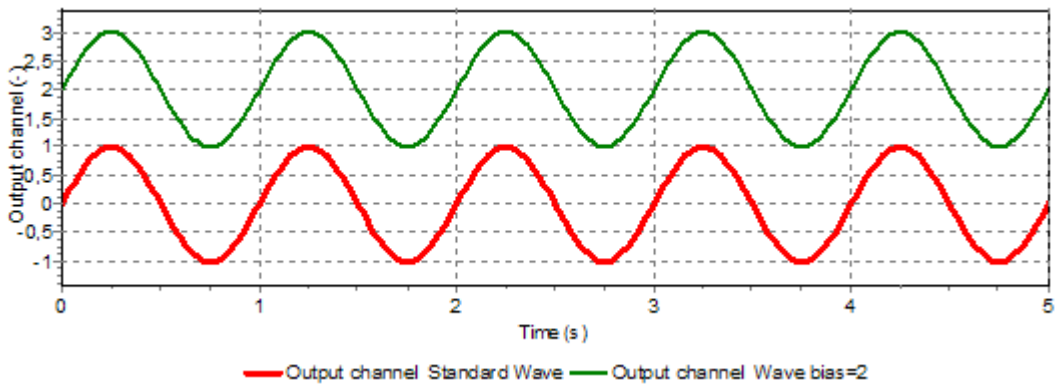


Fig 5: Different wave bias (0.0 and 2.0)

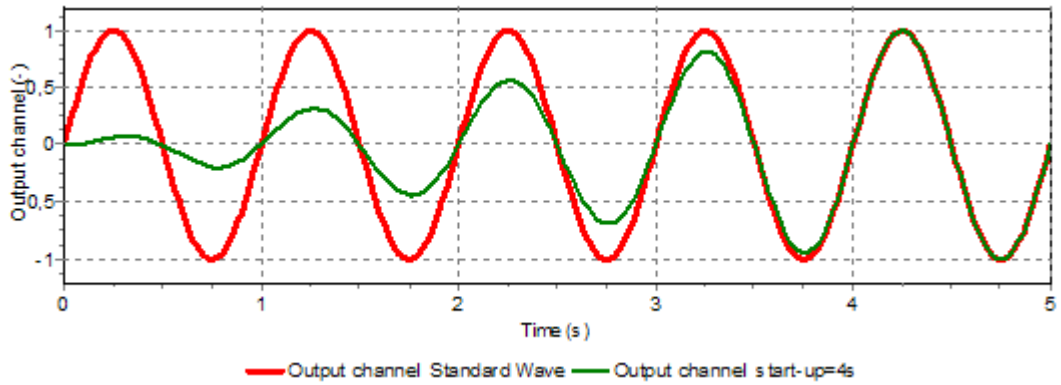


Fig 6: Sine waves with and without start-up times (0.0 and 4.0)

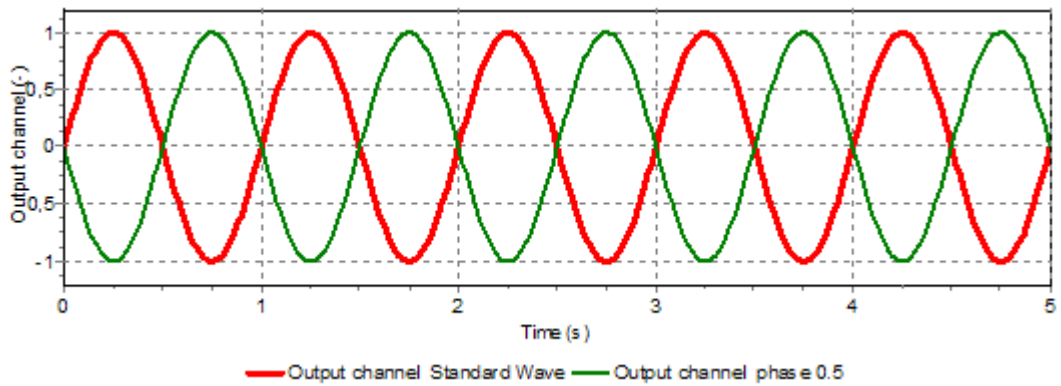
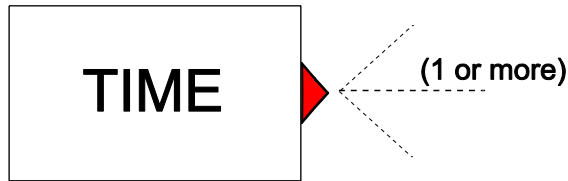


Fig 7: Different initial phases (0.0 and 0.5)

5.11 TIME

Symbol



Purpose

Outputs the model time at every time-step.

Procedure

None

Parameters

None

Remarks

None

Example

Figure 1 presents an example where the TIME component is used to switch a pump off between 5:00 and 21:00 and on between 21:00 and 5:00 every day. Figure 2 (a & b) shows the output of the MOD and OR component for a simulation duration of 1 week.

Component TAB2 is used to translate logical output to a pumpspeed including start/stop time in 5 s. Figure 3 shows the pump speed.

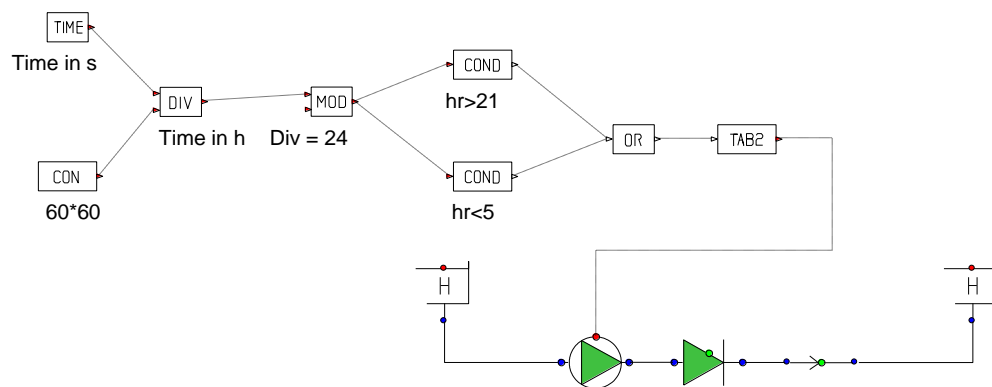


Fig 1: control scheme to operate a pump each day between 21 h and 5 hr

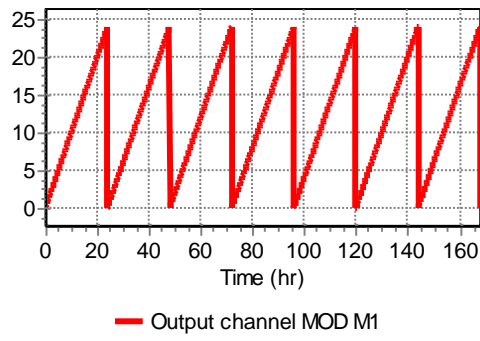


Fig 2a: Output of Modulo function

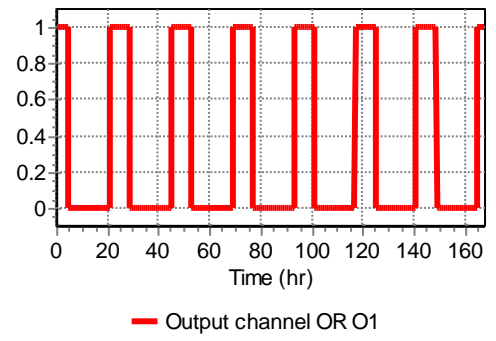


Fig 2b On/Off signal for Pump

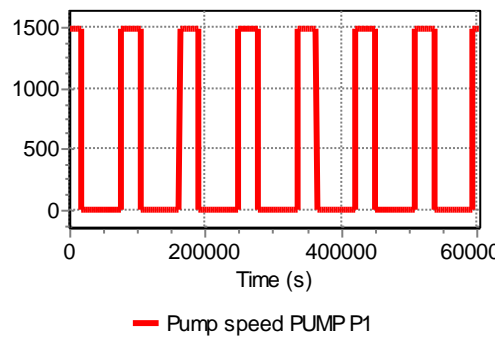


Fig 3: Pump speed

5.12 LtoN

Symbol



Purpose

Outputs a constant value or table value, depending on whether the input is TRUE or FALSE.

Mathematical description

None

Parameters

Parameter	input	unit	range	default	remarks
Output when TRUE	scalar/table			scalar	
Output value TRUE	real	[-]	[-1e8,1e8]		if scalar
Action table (TRUE)	table				if table
Output when FALSE	scalar/table			scalar	
Output value FALSE	real	[-]	[-1e8,1e8]		if scalar
Action table (FALSE)	table				if table

Remarks

The time in the action table is relative to the instant at which the input signal becomes True(1) or False(0)

In contrast with the TAB2 component there is no continuous transition from TRUE to FALSE and vice versa. A table will not have any relation with the other table or constant scalar.

Example

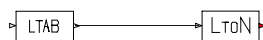


Fig 1: Control scheme

Input of LtoN component:

Output when TRUE: Table

0	0
5.000	10.00
10.00	30.00
15.00	100.0

Output when FALSE: Scalar = 50

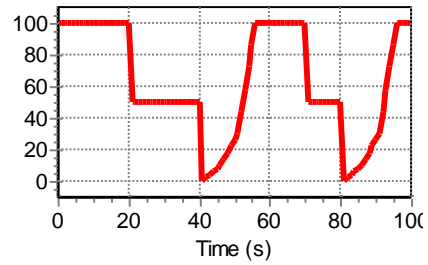
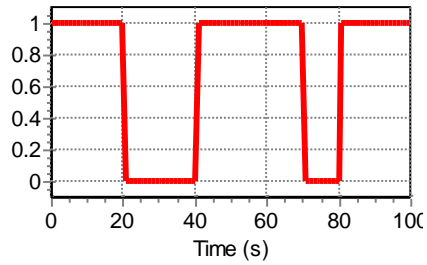


Fig 2: Time dependent logical input signal and numeric output signal

5.13 MSP

Symbol



Purpose

The MSP (Multi Stage Pattern) can be used to simulate closure or opening pattern of a valve or pump start/stop with multiple stages including freeze periods.

Procedure

MSP has two input signals and one output signal. The first input signal is a real input signal, which represents for example a time series of pump speeds or valve position. The second input signal is a logical signal. It represents for example the trigger condition for a shut-down.

When the logical input signal is false, the output signal is the same as the real input signal. When the trigger signal becomes true, the output signal is fixed to the actual value of the real input. As long as the logical input is true, the change of the output signal follows the MSP (according to the freeze periods, ramp values). When during this time the logical input signal becomes false again, the output signal switches instantaneously back to the actual real input signal.

The MSP control is described by a maximum of 5 stages, each with its own freeze period, ramp speed and end point. Every stage can start with a freeze period in which the output signal is kept constant at the current value (Note that the freeze period can be set to zero). For all stages except stage 1, this value is the end point of the previous stage. When the freeze time has passed, the output value is changed with the specified ramp towards the end point of that stage. When the end point of the last stage is reached, the output is kept constant.

Parameters

Parameter	input	unit	range	default	remarks
Number of stages	1/2/3/4/5	[-]		1	
Freeze period 1	Real	[s]	0-1e5	0 s	
Ramp 1	Real	[1/s]	0-1e5		
End point 1	Real	[-]	0-1e5		
Freeze period 2	Real	[s]	0-1e5		
Ramp 2	Real	[1/s]	0-1e5		
End point 2	Real	[-]	0-1e5		
Freeze period 3	Real	[s]	0-1e5		
Ramp 3	Real	[1/s]	0-1e5		
End point 3	Real	[-]	0-1e5		
Freeze period 4	Real	[s]	0-1e5		

Ramp 4	Real	[1/s]	0-1e5
End point 4	Real	[-]	0-1e5
Freeze period 5	Real	[s]	0-1e5
Ramp 5	Real	[1/s]	0-1e5
End point 5	Real	[-]	0-1e5

Remarks

The different end points should be either monotonously increasing or decreasing.

If the end points are monotonously increasing the following applies:

- If the real input signal is below the end point of stage 1, the MSP control starts with the freeze period.
- If the real input signal is equal to or above the end point of stage 1, the control pattern starts immediately (no freeze period).
- If the real input signal is equal to the end point of any stage (at freeze level), the freeze period of this stage is not taken into account and the MSP starts immediately.

If the end points are monotonously decreasing the following applies:

- If the real input signal is above the end point of stage 1, the MSP control starts with the freeze period.
- If the real input signal is equal to or below the end point of stage 1, the control pattern starts immediately (no freeze period).
- If the real input signal is equal to the end point of any stage (at freeze level), the freeze period of this stage is not taken into account and the MSP starts immediately.

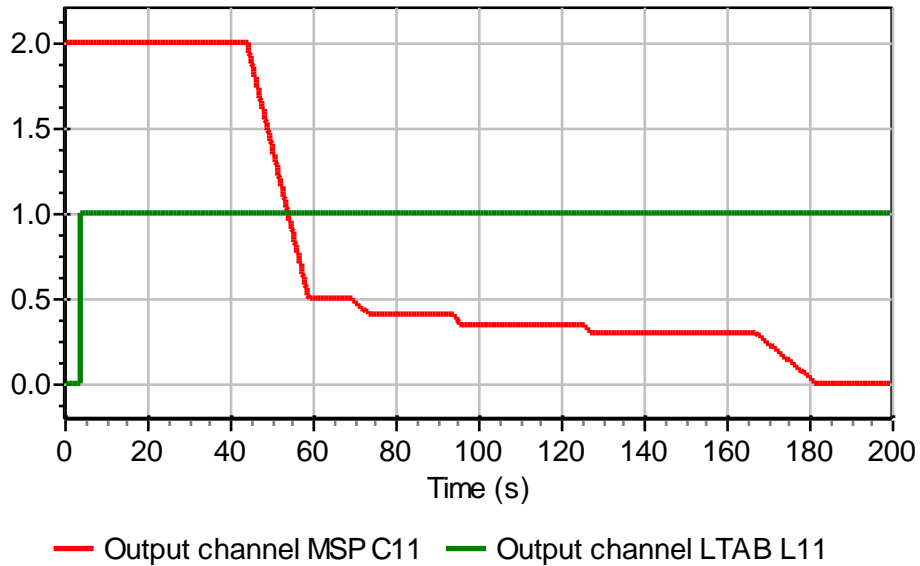
Example 1

The screenshot shows the configuration window for a Pattern named 'C11'. The configuration includes various parameters such as Name, Comment, Keywords, User name, Modified date, Type, Sequence number, Disuse, Number of stages, and five stages of Freeze period, Ramp, and End point. The 'Ramp 3' value is highlighted as 0.03000 (-). The 'Output channel' is set to 1.000 (-). To the right, a schematic diagram shows an MSP block with three input lines labeled CON, C1, and LTAB, and one output line labeled L1.

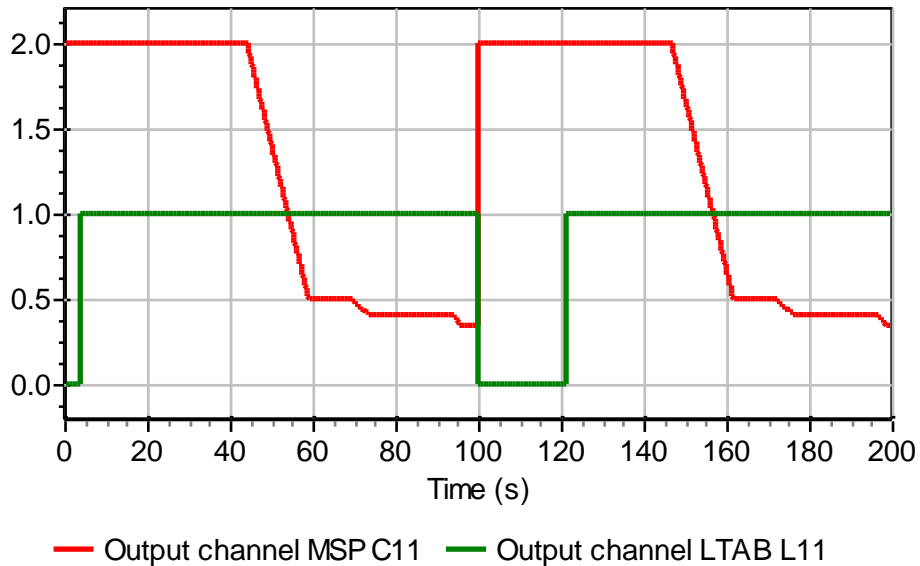
Name	C11
Comment	
Keywords	
User name	zwan
Modified	2012 Nov 2 9:12
Type	Pattern
Sequence number	0
Disuse	No
Number of stages	5
Freeze period 1	40.00 (s)
Ramp 1	0.1000 (-)
End point 1	0.5000 (-)
Freeze period 2	10.00 (s)
Ramp 2	0.02000 (-)
End point 2	0.4000 (-)
Freeze period 3	20.00 (s)
Ramp 3	0.03000 (-)
End point 3	0.3500 (-)
Freeze period 4	30.00 (s)
Ramp 4	0.04000 (-)
End point 4	0.3000 (-)
Freeze period 5	40.00 (s)
Ramp 5	0.02000 (-)
End point 5	0 (-)
Messages	
Output channel	1.000 (-)

Based on the input and scheme in the above figure several sub-examples have been made with different inputs for a constant (CON) and LTAB.

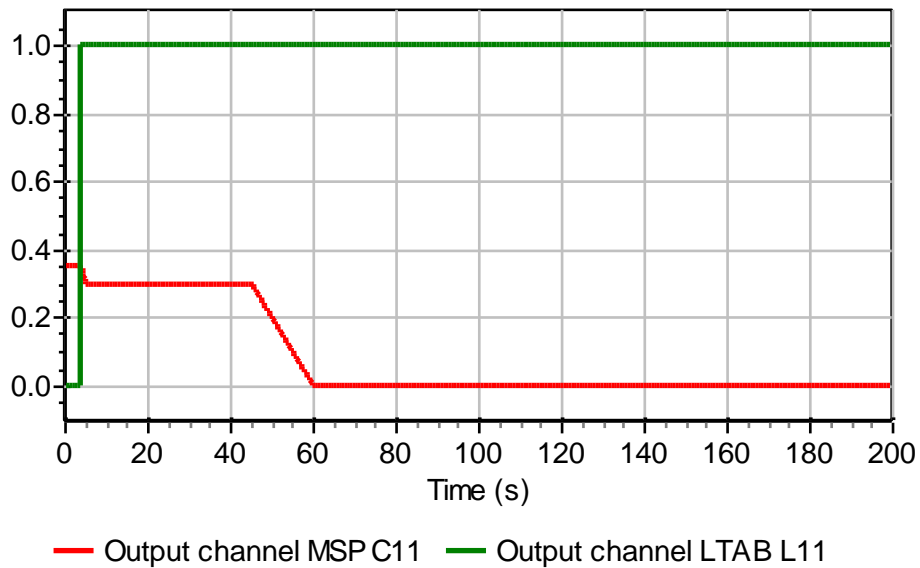
Constant value is assumed to be 2, the logical signal LTAB becomes true at $t = 4$ s. Because the real value at the time $t = 4$ sec is 2.0 and therefore larger than the value of End point 1 (0.5), the MSP starts with the first freeze period (40 seconds). After 44 seconds, the output signal of MSP is ramped down according to the given pattern (from Ramp 1 until End point 5).



The above example is modified slightly. The real input value is still a constant value of 2, LTAB becomes true at $t = 4$ s but at $t = 100$ s it becomes false. At $t = 100$ s, the output value gets again the value of the real input (here $CON = 2$, for a time series this would be the actual value at $t = 100$ s). At $t = 120$ s, LTAB becomes true again. As $CON = 2$ is larger than the value for End point 1 (0.5), output of MSP is then again according to the prescribed control including an initial freeze period of 40 seconds..

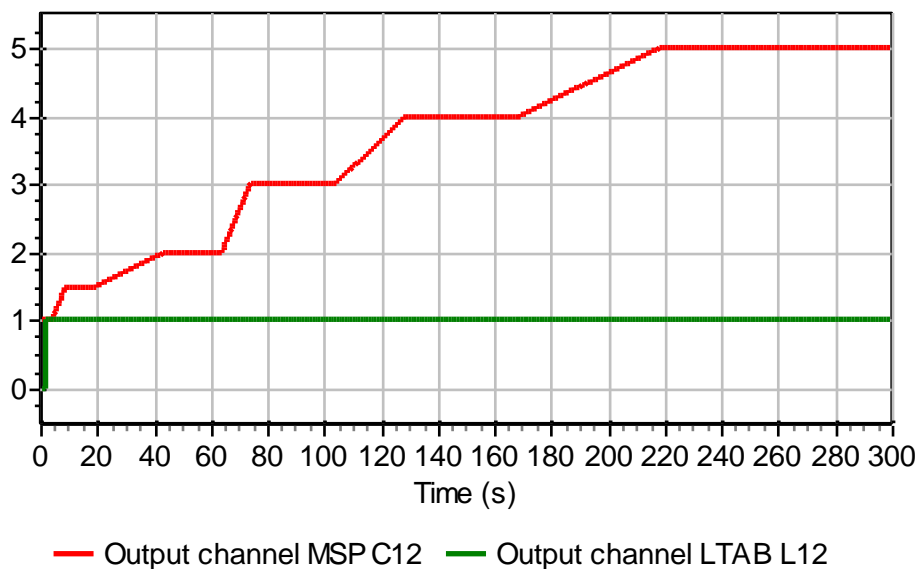


In the third sub-example, CON is set to 0.35. LTAB becomes true after 4 s. The MSP control identifies the correct position in the pattern (example; between End Point 3 and End Point 4). The MSP starts in the middle of Ramp 4 and continues with Freeze 5 until End point 5.



Example 2

By changing the inputs of the MSP the signal can also increase in time towards the value of the last end point.



5.14 COND

Symbol



Purpose

Compares the value from the input channel with a set value.

Procedure

The (measured) value at the input channel is compared with the set value. The type of comparison is chosen by the 'Logical operator'.

Parameters

Parameter	input	unit	range	default	remarks
Logical operator	LT (less than)/ GT (greater than)/ LE (less equal)/ GE (greater equal)/ EQ (equal)/ NE (not equal)				
Set value	real				
Reset time (true)	real	[s]		0	
Reset time (false)	real	[s]		0	

Remarks

The condition must hold for a period longer than the 'reset time' before the output will switch its original value. If the condition does not hold for a period longer than the 'reset time' then the output will keep its original value and the timing period will be restarted.

Examples

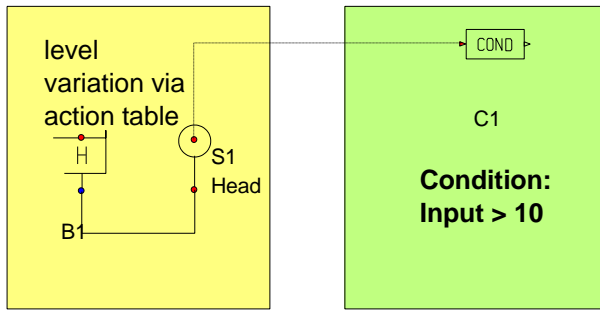


Fig 1: Control scheme

Figures 2a and 2b show the output when both 'reset times' are not used (0 seconds)

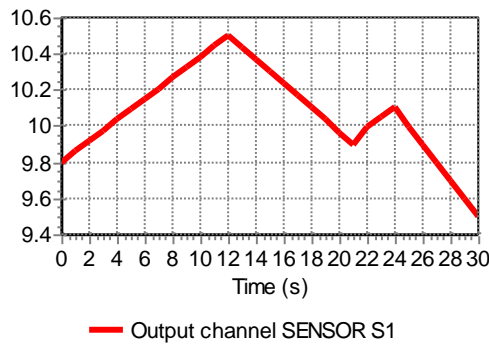


Fig 2a: variation of Head measured by Sensor

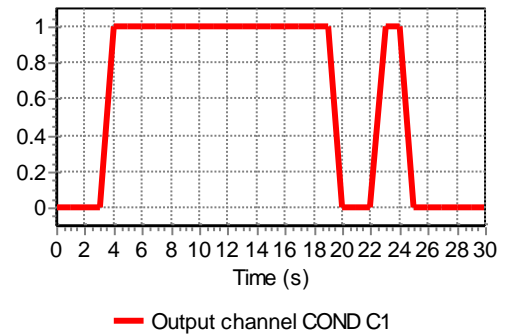


Fig 2b Output of COND

Figures shows the output when 'reset time (true)' is set to 5 seconds and 'reset time (false)' is set to 2 seconds.

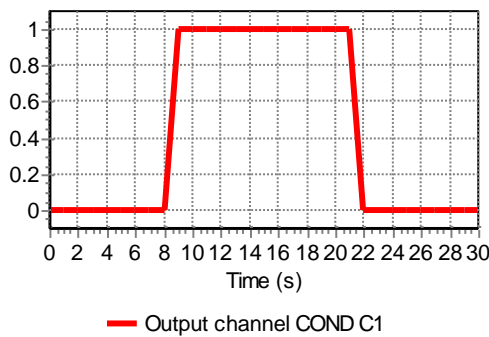


Fig 3 Output of COND including reset times (true =5 s, false = 2s)

5.15 COND2

Symbol



Purpose

Compares the value from the first input channel with a set value. Furthermore, the comparison can be enabled/disabled depending on the logical condition of the second input channel.

Procedure

The (measured) value at the first input channel is compared with the set value. The type of comparison is chosen by the 'Logical operator'. If the second input channel is TRUE, then the value of the condition is output. If the second output channel is FALSE, then the output is FALSE. Table 1 presents the possible scenarios for different input values.

Table 1: Possible scenarios

Scenario	condition	2nd input	Output
1	TRUE	TRUE	TRUE
2	FALSE	TRUE	FALSE
3	TRUE	FALSE	FALSE
4	FALSE	FALSE	FALSE

Parameters

Parameter	input	unit	range	default	remarks
Logical operator	LT (less than)/ GT (greater than)/ LE (less equal)/ GE (greater equal)/ EQ (equal)/ NE (not equal)				
Set value	real				
Reset time (true)	real	[s]		0	
Reset time (false)	real	[s]		0	

Remarks

If no logical signal line is attached to the second input channel it will default to TRUE and hence the COND2 will act identical to COND.

Example

The example of COND is extended with the logical table LTAB to enable/disable the second (logical) input channel of COND2.

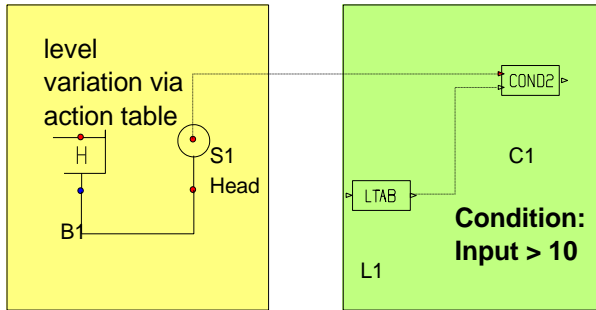


Fig 1: Control scheme

The input of LTAB is:

Time (s)	On(1) Off(0)
0	0
8.000	1.000
14.00	0
16.00	1.000

Figures 2a and 2b show the output when both 'reset times' are not used (0 seconds).

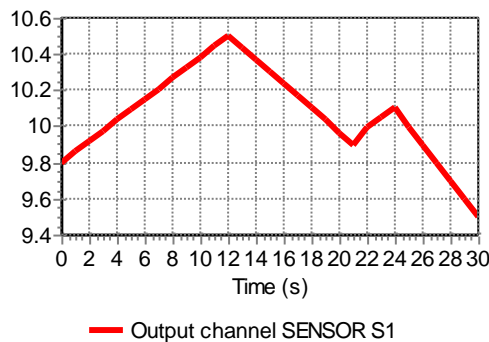


Fig 2a: variation of Head measured by Sensor

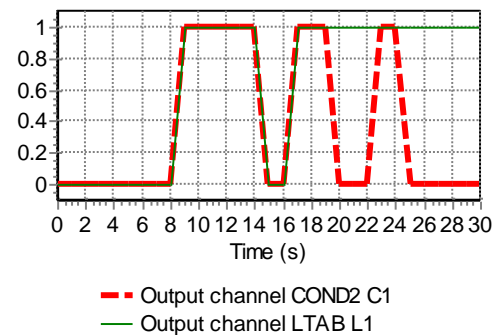


Fig 2b Output of COND and LTAB

5.16 2COND

Symbol



Purpose

Compares the results of two conditions (2COND).

Procedure

Each input channel value is compared with its corresponding set value. The type of comparison is chosen by the 'Logical operator'. Condition 1 is evaluated with the first (upper) input channel value. Condition 2 is evaluated with the second (lower) input channel value. If condition 1 becomes true, the output value is set to true; if condition 2 becomes true, the output value becomes false. Table 1 presents the output values for different condition combinations.

Table 1: Output values for different scenarios

Scenario	condition 1	condition 2	Output
1	TRUE	TRUE	FALSE
2	TRUE	FALSE	TRUE
3	FALSE	TRUE	FALSE
4	FALSE	FALSE	TRUE

Parameters

Parameter	input	unit	range	default	remarks
Logical operator C1	LT (less than)/ GT (greater than)/ LE (less equal)/ GE (greater equal)/ EQ (equal)/ NE (not equal)				
set value C1	real				
reset time C1	real	[s]		0	
Logical operator C2	LT (less than)/ GT (greater than)/ LE (less equal)/ GE (greater equal)/ EQ (equal)/ NE (not equal)				

set value C2	real		
reset time C2	real	[s]	0
Initial status	TRUE/ FALSE		

Remarks

Both conditions must hold for periods longer than their corresponding 'reset time' before the output will switch its original value. If both conditions do not hold for periods longer than their 'reset time' then the output will keep its original value and the timing periods will be restarted.

Examples

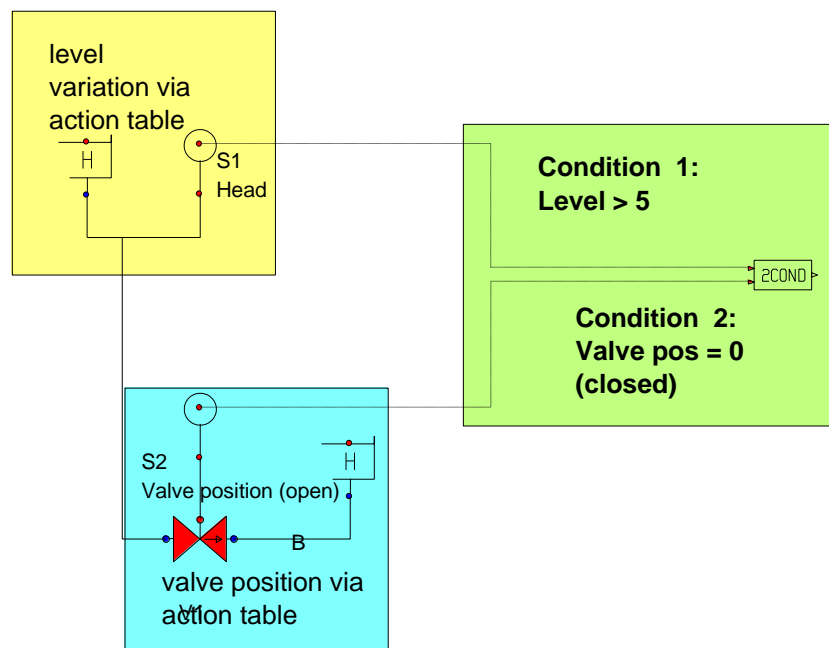


Fig 1: Control scheme

The input signals of 2COND are gathered from a prescribed Head and valve position and shown in figure 2a and 2b.

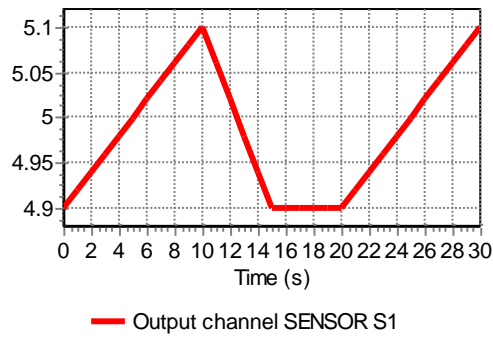


Fig 2a: prescribed head

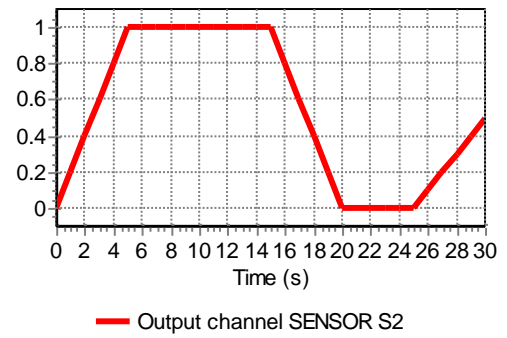


Fig 2b: prescribed valve position

The initial state of 2COND is set to false, because the level (condition 1) is below 5 m. The behaviour of 2COND with both reset times 0 s, is shown in figure 3a.

Introducing a reset time of 6 s for both condition gives a rather different behaviour shown in figure 3b.

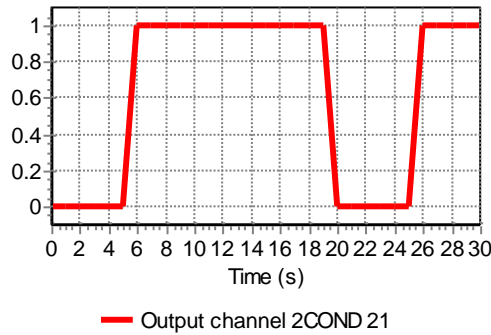


Fig 3a: Output 2COND with both reset times 0 s

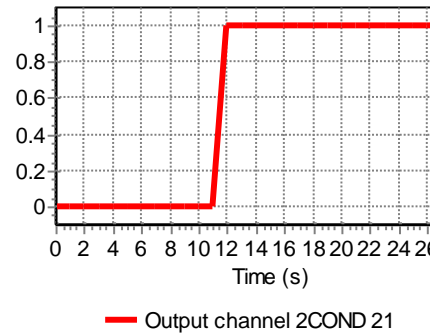


Fig 3b: Output 2COND with both reset times 6 s

5.17 LCS

Symbol



Purpose

The Level Control Switch (LCS) can be used to switch on at one level and to switch off at another level. In fact it is a combination of two COND components. For example:

Pump sump (On-value > Off-value): When the level in the collection reservoir exceeds a certain value the pump is switched on. When the level drops below a certain value the pump is switched off.

Pressure system (Off-value > On-value): When the pressure in the system falls below a certain value the pump is switched on, to re-pressurise the system. When the pressure exceeds a certain value the pump is switched off.

Procedure

The sense of the operation depends on which of the two values are greater.

- On-value > Off-value:
 - Output will become TRUE if the input signal exceeds the On-value.
 - Output will become FALSE if the input signal drops below the Off-value.
- On-value < Off-value
 - Output will become TRUE if the input signal drops below the On-value.
 - Output will become FALSE if the input signal exceeds the Off-value.

Parameters

Parameter	input	unit	range	default	remarks
On value	real				
Off value	real				
Reset time (on)	real	[s]			
Reset time (off)	real	[s]			
Initial status	ON/ OFF				

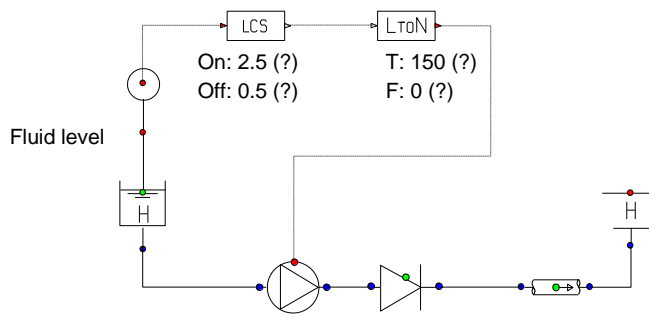
Remarks

Both On/Off-conditions must hold for periods longer than their corresponding 'reset time' before the output will switch to TRUE or FALSE. If the conditions do not hold for periods longer than their 'reset time' then the output will not switch.

Example

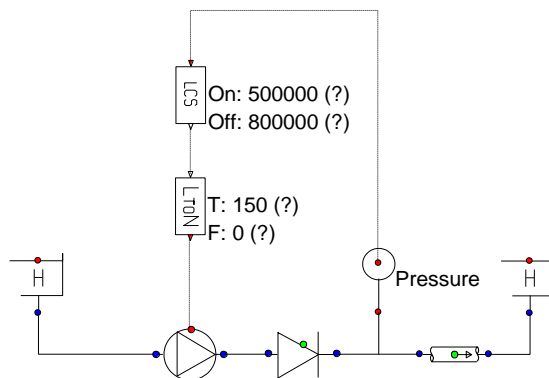
Pump sump: On-level > Off-level.

If the level in the collection reservoir has risen to a certain level, it is emptied by the pump to a certain lower level



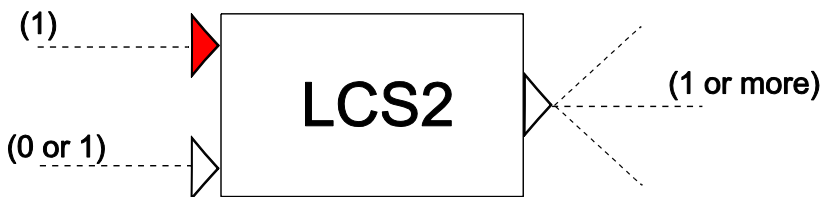
Pressure system: Off-Level > On-level

If the pressure in the system falls below a certain level the pump starts to re-pressure the system. After reaching a certain high level the pump stops.



5.18 LCS2

Symbol



Purpose

The Level Control Switch (LCS2) can be used to switch on at one level and to switch off at another. For example:

Pump sump (On-value > Off-value): When the level in the collection reservoir exceeds a certain value the pump is switched on. When the level drops below a certain value the pump is switched off.

Pressure system (Off-value > On-value): When the pressure in the system falls below a certain value the pump is switched on, to re-pressurise the system. When the pressure exceeds a certain value the pump is switched off.

Furthermore, the LCS2 component can be enabled/disabled depending on the logical input signal.

Procedure

The sense of the operation depends on which of the two values are greater.

- On-value > Off-value:
 - Output will become TRUE if the input signal exceeds the On-value.
 - Output will become FALSE if the input signal drops below the Off-value.
- On-value < Off-value
 - Output will become TRUE if the input signal drops below the On-value.
 - Output will become FALSE if the input signal exceeds the Off-value.

The above procedure will only be executed when the logical input signal is TRUE (enabled). When the logical input signal is FALSE the output signal is set to FALSE and the internal reset times are reset.

Parameters

Parameter	input	unit	range	default	remarks
On value	real				
Off value	real				
Reset time (on)	real	[s]			
Reset time (off)	real	[s]			
Initial status	ON/ OFF				

Remarks

Both On/Off-conditions must hold for periods longer than their corresponding 'reset time' before the output will switch to TRUE or FALSE. If the conditions do not hold for periods longer than their 'reset time' then the output will not switch.

If no logical signal line is attached to the logical input channel, the default value (TRUE) is used. Hence the LCS2 component will behave in the same way an LCS.

Example

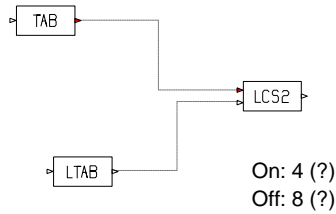


Fig 1: Example of control scheme

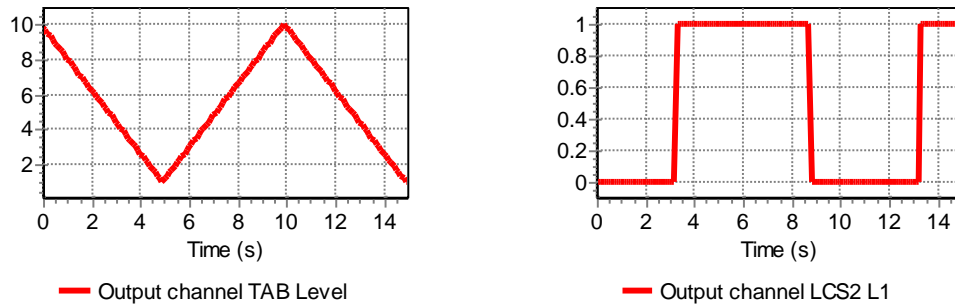


Fig 2: Behaviour of LCS2 without LTAB connected

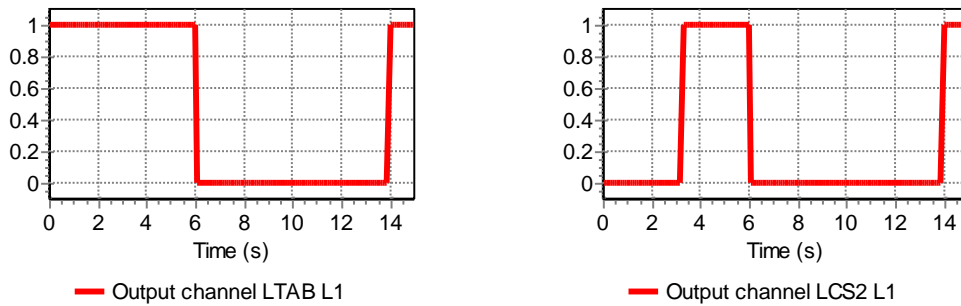


Fig 3: Behaviour of LCS2 with LTAB connected

See also "LCS" on page 516.

5.19 SELECT

Symbol



Purpose

Outputs one of two possible signals, depending on whether the switch value is TRUE or FALSE.

Procedure

Switching between input signals A and B is determined by whether the logical input signal is TRUE or FALSE. Signal A is output when the logical input is TRUE. Signal B is output when the logical input is FALSE.

Parameters

None

Remarks

None

Example

Figure 1 shows the SELECT component switching between two numerical input signals: a sine and a constant. When the logical input is TRUE, the sine is output. When the logical input is FALSE, the constant is output.

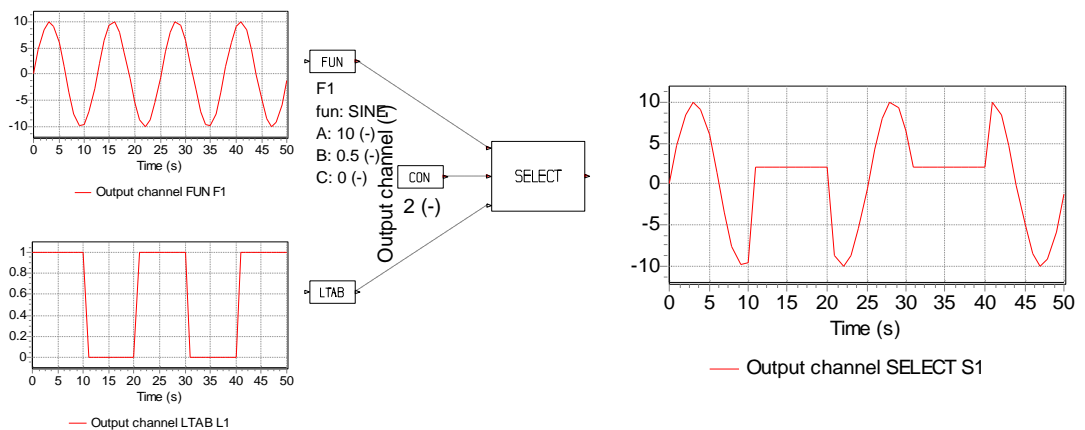
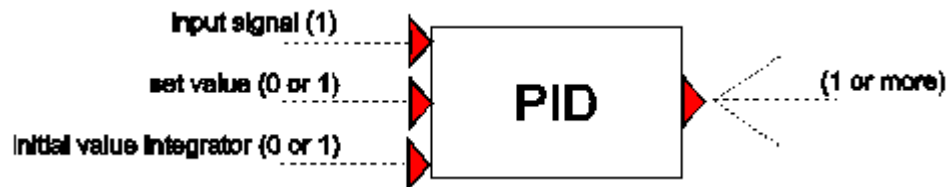


Fig 1: Control scheme with SELECT

5.20 PID

Symbol



Purpose

To achieve a certain set value by adjusting a controlled variable of a hydraulic component, based on the difference between the input value and the set value.

Procedure

The Proportional, Integrate, Differentiate (PID) component computes for each sample period Δt the error ε between an input signal $x[n]$ and a set value x_{set} :

$$\varepsilon[n] = x[n] - x_{set}$$

Where n denotes the present sample. The output signal is computed by Proportioning:

$$y_p[n] = c_p \varepsilon[n]$$

Integrating:

$$y_i[n] = y_i[n-1] + \frac{c_p}{c_i} \varepsilon[n] \Delta t$$

or Differentiating the error:

$$y_d[n] = c_p c_d \frac{\varepsilon[n] - \varepsilon[n-1]}{\Delta t}$$

where c_p is the “gain” (whose units are determined by the units of the output signal), c_i is the “integration time” (time units), c_d is the “differentiation time” (time units), Δt is the sample time-step and $[n-1]$ denotes the previous sample. The above equations can be combined to give an output signal, which is either:

Proportional (P):

$$y = y_p + c_0$$

where c_0 is an offset value (Bias).

Proportional and Integrated (PI):

$$y = y_p + y_i$$

or Proportional, Integrated and Differentiated (PID):

$$y = y_p + y_i + y_d$$

Parameters

Parameter	input	unit	range	default	remarks
Initial setpoint	real				Can be overruled by 2 nd input channel
Accuracy of recorded value	real				See remarks
Sample time interval	real	[s]			See remarks
Type of control	P/ PI/ PID				
Gain	real				
Integration time constant	real	[s]			
Differentiation time constant	real	[s]			
Offset value (bias)	real				only if 'Type of Control' = P
Initial value of integrator	real				only if 'Type of Control' = PI or PID Can be overruled by 3 rd input channel
Lower bound	real				
Upper bound	real				
Ramp value CTRL_UP	real				max. positive change per second
Ramp value CTRL_DOWN	real				max. negative change per second

Remarks

The Sample time interval is used to decrease the resolution of the input signal. A PLC-controller can have a different sample time than the simulation time.

If the difference between the measured (recorded) value and setpoint is smaller than the accuracy, the error is set to zero.

The PID component allows an 'upper bound' and a 'lower bound' to be imposed on the output signal.

When the output value "hits" the upper or lower bound the control system can not follow the error anymore and has no possibility to feed back to the system. If this situation remains long enough the integrator fills with a large value, which inhibits timely reaction if the error value changes direction (sign). To prevent this, a simple anti-reset windup provision is implemented. This provision suspends the accumulating of the error during the time that the output value violates the lower or upper bound.

Furthermore, a maximum allowable 'ramp value' can also be specified to control the rate of change of the output.

The integrating part can be initialised with a non-zero value. This can be used to adjust the PID controller to the hydraulic steady state (see examples).

In the steady state, the error should be equal to zero. This ensures that the entire system is in a true steady state.

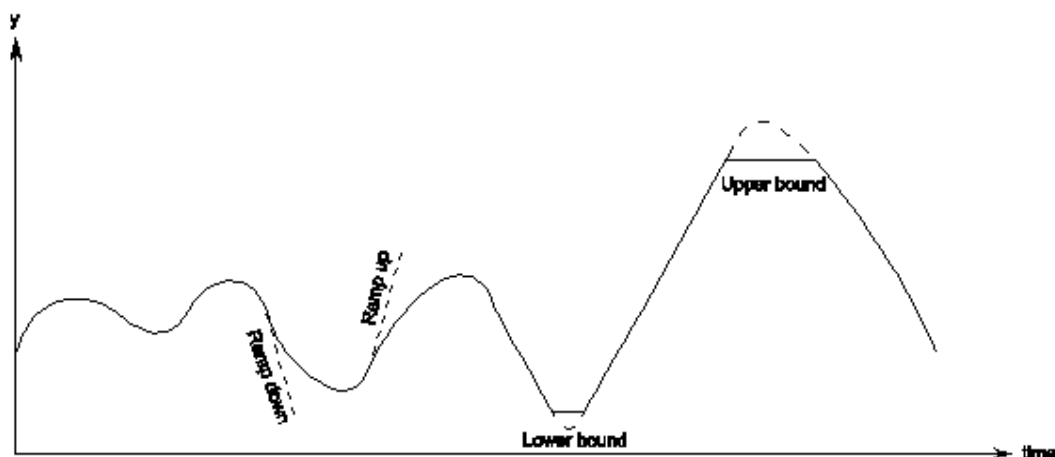


Figure 4: PID operation

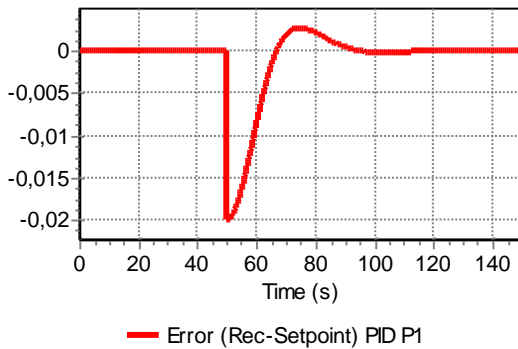
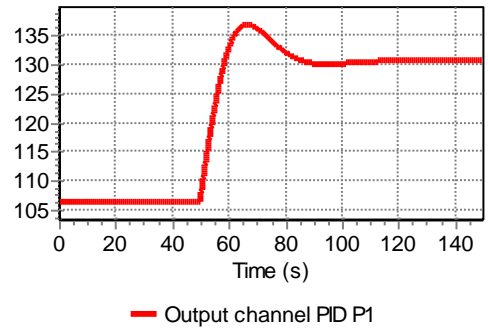
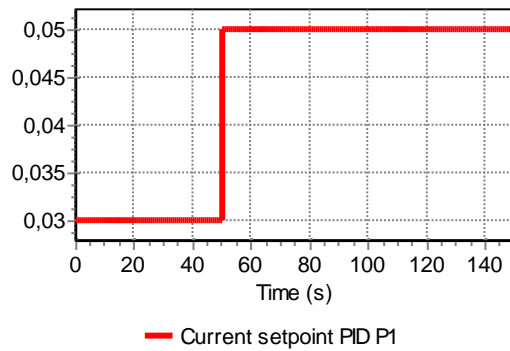
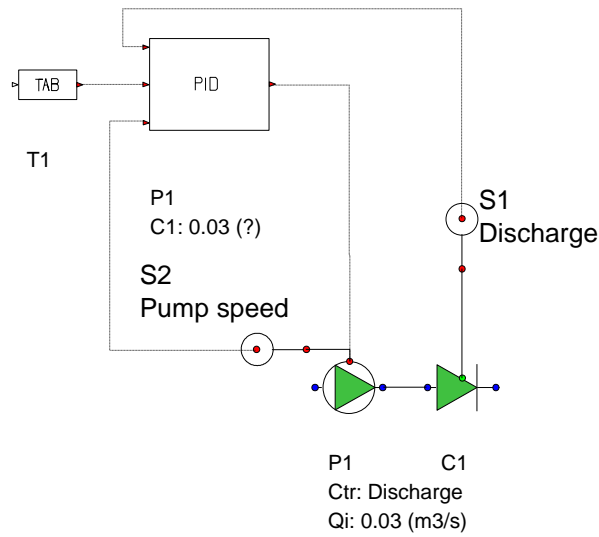
It is the user’s responsibility to choose, and so that the output signal can be used by the controlled hydraulic component. For instance, if a valve is the controlled hydraulic component then the PID output signal must have a lower bound of 0 and an upper bound of 1.

Component Messages

Message	Explanation
Integrator must be less than upper bound	Input validation
Integrator must be greater than lower bound	Input validation
Offset must be less than upper bound	Input validation
Offset must be greater than lower bound	Input validation
Discrepancy with setpoint value and measured value	
ERROR: Discrepancy with setpoint value more than 25 %	The control system and hydraulic system must be in equilibrium for the steady state, see remarks
Target below lowerbound, anti-reset windup provision activated	Informative
Target above lowerbound, anti-reset windup prov. de-activated	Informative
Target above upperbound, anti-reset windup provision activated	Informative
Target below upperbound, anti-reset windup prov. de-activated	Informative

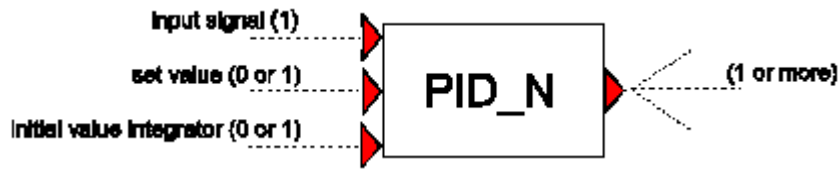
Example

The following example shows the use of the PID controller:



5.21 PID_N

Symbol



Purpose

To achieve a certain set value by adjusting a controlled variable of a hydraulic component, based on the difference between the input value and the set value. The PID_N component is similar to the PID component, except that the error can be normalized, as well as the output.

Procedure

The Proportional, Integrate, Differentiate Normalized (PID_N) component computes for each sample period Δt the error ε between an input signal $x[n]$ and a set value x_{set} :

$$\varepsilon[n] = x[n] - x_{set}$$

where n denotes the present sample. This error is then normalized by the input lower bound and upper bound values:

$$\varepsilon' = \frac{\varepsilon}{x_{upper} - x_{lower}}$$

The unscaled output signal is computed by

Proportioning:

$$y'_p[n] = c_p \varepsilon'[n]$$

Integrating:

$$y'_i[n] = y'_i[n-1] + \frac{c_p}{c_i} \varepsilon'[n] \Delta t$$

or Differentiating the normalized error:

$$y'_d[n] = c_p c_d \frac{\varepsilon'[n] - \varepsilon'[n-1]}{\Delta t}$$

where c_p is the “gain”, c_i is the “integration time constant”, c_d is the “differentiation time constant”, Δt is the sample time-step and $[n-1]$ denotes the previous sample. The equations above can be combined to give an unscaled output signal, which is either:

Proportional (P):

$$y' = y'_p + c_0$$

Proportional and Integrated (PI):

$$y' = y'_p + y'_i$$

or Proportional, Integrated and Differentiated (PID):

$$y' = y'_p + y'_i + y'_d$$

where c_0 is an offset value. Finally, the scaled output signal is obtained using the output lower bound and upper bound values:

$$y = y' (y_{upper} - y_{lower}) + y_{lower}$$

Parameters

Parameter	input	unit	range	default	remarks
Initial setpoint	real				Can be overruled by 2 nd input channel
Lower bound (input)	real				
Upper bound (input)	real				
Accuracy of recorded value	real				See remarks
Sample time interval	real	[s]			See remarks
Type of control	P/ PI/ PID				
Gain	real				
Integration time constant	real	[s]			
Differentiation time constant	real	[s]			
Offset value (bias)	real				only if "Type of Control" = P
Initial value of integrator	real				only if "Type of Control" = PI or PID Can be overruled by 3 rd input channel
Lower bound	real				
Upper bound	real				
Ramp value CTRL_UP	real				max. positive change per second
Ramp value CTRL_DOWN	real				max. negative change per second

Remarks

The Sample time interval is used to decrease the resolution of the input signal. A PLC-controller can have a different sample time than the simulation time.

If the difference between the measured (recorded) value and setpoint is smaller than the accuracy, the error is set to zero.

The PID component allows an 'upper bound' and a 'lower bound' to be imposed on the output signal.

When the output value "hits" the upper or lower bound the control system can not follow the error anymore and has no possibility to feed back to the system. If this situation remains long enough the integrator fills with a large value, which inhibits timely reaction if the error value changes direction (sign). To prevent this, a simple anti-reset windup provision is implemented. This provision suspends the accumulating of the error during the time that the output value violates the lower or upper bound.

Furthermore, a maximum allowable 'ramp value' can also be specified to control the rate of change of the output.

The integrating part can be initialised with a non-zero value. This can be used to adjust the PID controller to the hydraulic steady state (see examples).

In the steady state, the error should be equal to zero. This ensures that the entire system is in a true steady state.

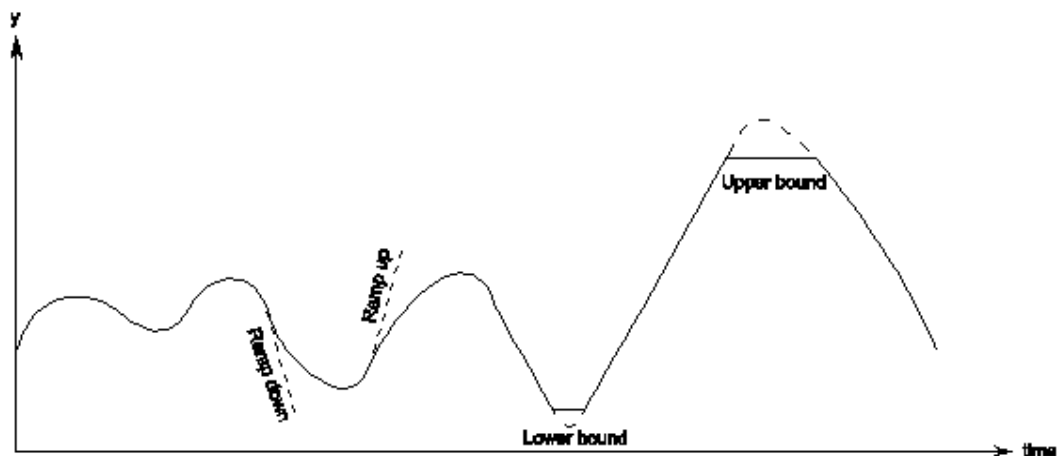


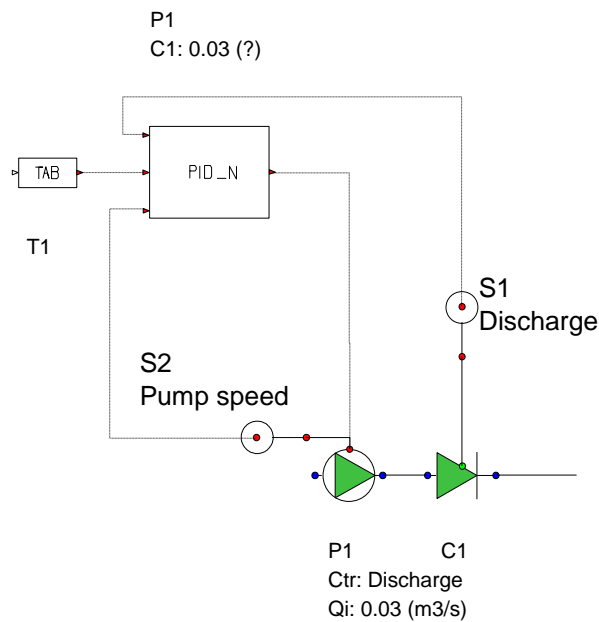
Figure 5: PID operation

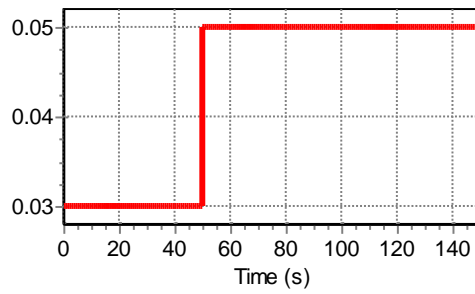
It is the user's responsibility to choose, and so that the output signal can be used by the controlled hydraulic component. For instance, if a valve is the controlled hydraulic component then the PID output signal must have a lower bound of 0 and an upper bound of 1.

Component Messages

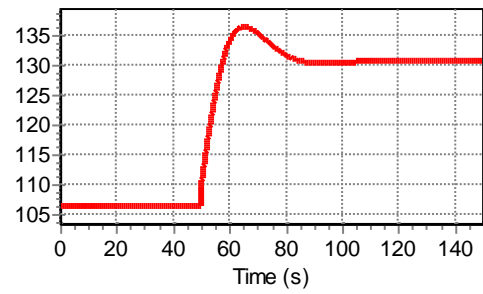
Message	Explanation
Integrator must be less than upper bound	Input validation
Integrator must be greater than lower bound	Input validation
Offset must be less than upper bound	Input validation
Offset must be greater than lower bound	Input validation
Discrepancy with setpoint value and measured value	
ERROR: Discrepancy with setpoint value more than 25 %	The control system and hydraulic system must be in equilibrium for the steady state, see remarks
Target below lowerbound, anti-reset windup provision activated	Informative
Target above lowerbound, anti-reset windup prov. de-activated	Informative
Target above upperbound, anti-reset windup provision activated	Informative
Target below upperbound, anti-reset windup prov. de-activated	Informative

Example

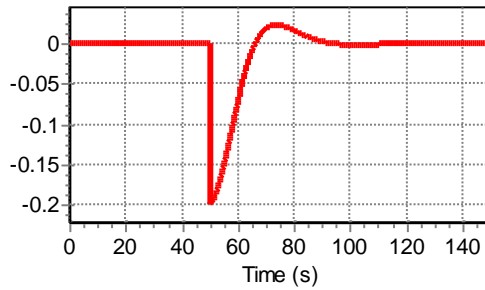




— Current setpoint PIDN P1



— Output channel PIDN P1



— Error (Rec-Setpoint) PIDN P1

5.22 PIDSS

Symbol



Purpose

To achieve a certain set value by adjusting a controlled variable of a hydraulic component, based on the difference between the input value and the set value. This component includes an ON/OFF switch and is mainly used for pump sumps in sewage systems.

Procedure

The Proportional, Integrate, Differentiate Start/Stop (PIDSS) component requires a minimum of two input signals: The first 'input signal' is used to compute the output signal. The second 'switch value' signal is used to determine whether to switch the PIDSS component ON or OFF. The PIDSS switches ON when the "switch value" signal is higher than the "input value to switch ON" property and switch OFF when the "switch value" signal is lower than the "input value to switch OFF" property.

The PIDSS component has four states: OFF, STARTING, ON and STOPPING (See figure 1). Assuming the initial state is OFF, at the moment the input 'switch value' signal exceeds the 'switch ON' level, the state will change to STARTING. The output signal will then change linearly in time (bound by the 'ramp value') from the 'output value when OFF' to the 'output start value'. The state then changes to ON.

During the first part of the ON state the output remains equal to the 'output start value' for a duration specified by the 'stabilize time'. The second part of the ON state is the CONTROLLING part of the PIDSS component. If the input 'switch value' drops below the 'switch OFF' level the state will change to STOPPING. In this state the output value will decrease linearly from the 'lower bound value' to the 'output value when OFF', bound by the 'ramp value'. Thereafter the state will revert to OFF.

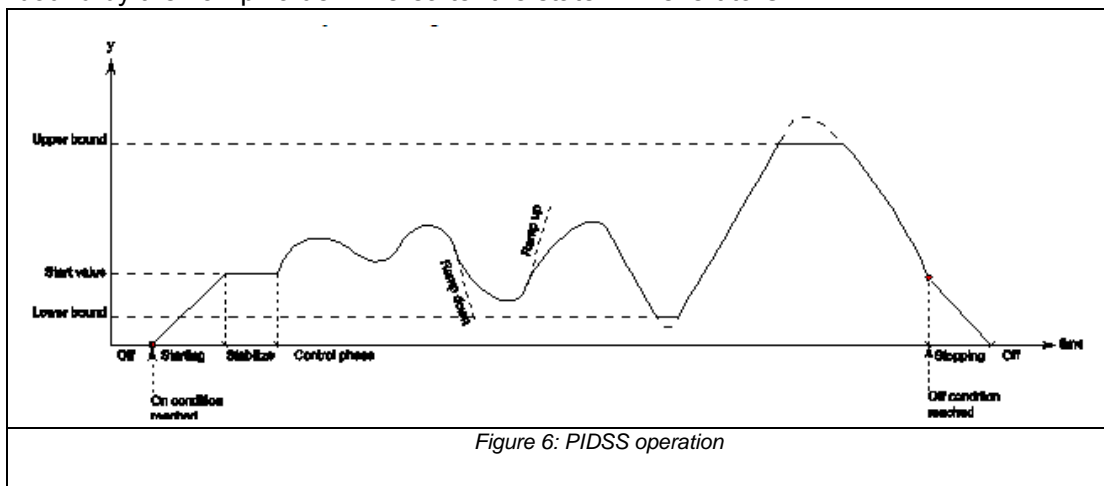


Figure 6: PIDSS operation

The PIDSS component computes for each sample period Δt the error ε between the input signal $x[n]$ and the setpoint x_{set} . The error is given by:

$$\varepsilon[n] = x[n] - x_{set}$$

Where n denotes the present sample. The output signal is computed by proportioning, integrating or differentiating the error:

The proportional error is given by:

$$y_p[n] = c_p \varepsilon[n]$$

The integrated error is given by:

$$y_i[n] = y_i[n-1] + \frac{c_p}{c_i} \varepsilon[n] \Delta t$$

The differentiated error is given by:

$$y_d[n] = c_p c_d \frac{\varepsilon[n] - \varepsilon[n-1]}{\Delta t}$$

where c_p is the “gain” (whose units are determined by the units of the output signal), c_i is the “integration time” (time units), c_d is the “differentiation time” (time units), Δt is the sample time-step and $[n-1]$ denotes the previous sample. The equations above can be combined to give an output, which is either:

Proportional (P):

$$y = y_p + c_0$$

Proportional and Integrated (PI):

$$y = y_p + y_i$$

or Proportional, Integrated and Differentiated (PID):

$$y = y_p + y_i + y_d$$

where c_0 is an offset value.

Parameters

Parameter	input	unit	range	default	remarks
input value to switch ON	real				
input value to switch OFF	real				
Reset time (on)	real	[s]			
Reset time (off)	real	[s]			
Initial status	ON/ OFF				
Initial setpoint	real				Can be overruled by 3 rd input channel
Accuracy of recorded value	real				See remarks
Sample time interval	real	[s]			See remarks
Type of control	P/ PI/ PID				
Offset value (bias)	real				only if 'Type of Control' = P
Gain	real				
Integration time constant	real	[s]			
Initial value of integrator	real				only if 'Type of Control' = PI or PID Can be overruled by 4 th input channel
Start value (output)	real				
Use stabilize time		YES/ NO			
Stabilization time	real				Only if "Use stabilize time" = YES
Lower bound (output)	real				
Upper bound (output)	real				
Ramp (max. change/sec)	constant/ variable				
Ramp value	real				Only if Ramp = constant
Ramp value START	real				Only if Ramp = variable
Ramp value STOP	real				Only if Ramp = variable
Ramp value CTRL_UP	real				Only if Ramp = variable
Ramp value CTRL_DOWN	real				Only if Ramp = variable
Output value when OFF	real				

Remarks

The Sample time interval is used to decrease the resolution of the input signal. A PLC-controller can have a different sample time than the simulation time.

If the difference between the measured (recorded) value and setpoint is smaller than the accuracy, the error is set to zero.

The PID component allows an 'upper bound' and a 'lower bound' to be imposed on the output signal.

When the output value "hits" the upper or lower bound the control system can not follow the error anymore and has no possibility to feed back to the system. If this situation remains long enough the integrator fills with a large value, which inhibits timely reaction if the error value changes direction (sign). To prevent this, a simple anti-reset windup provision is implemented. This provision suspends the accumulating of the error during the time that the output value violates the lower or upper bound.

Furthermore, a maximum allowable 'ramp value' can also be specified to control the rate of change of the output.

The integrating part can be initialised with a non-zero value. This can be used to adjust the PID controller to the hydraulic steady state (see examples).

In the steady state, the error should be equal to zero. This ensures that the entire system is in a true steady state.

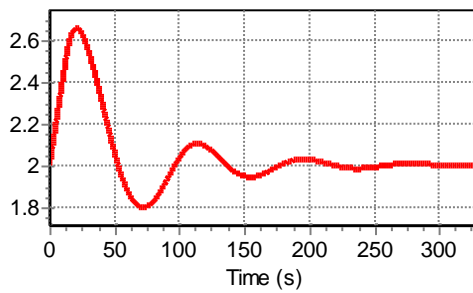
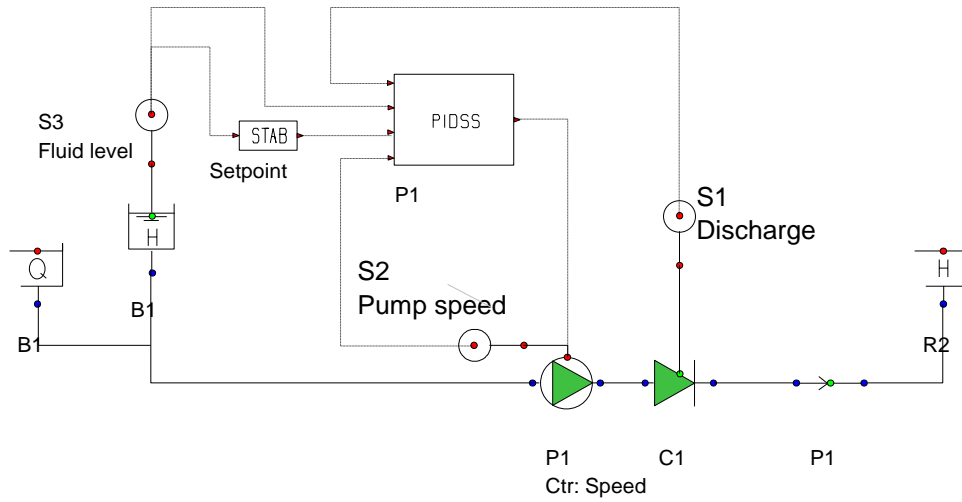
It is the user's responsibility to choose, and so that the output signal can be used by the controlled hydraulic component. For instance, if a valve is the controlled hydraulic component then the PID output signal must have a lower bound of 0 and an upper bound of 1.

Component Messages

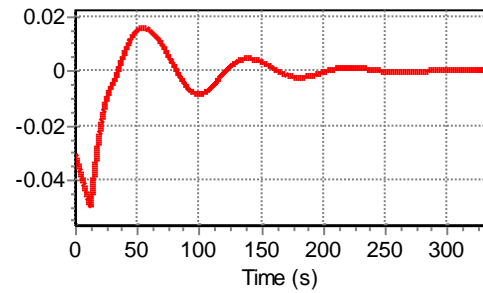
Message	Explanation
Integrator must be less than upper bound	Input validation
Integrator must be greater than lower bound	Input validation
Offset must be less than upper bound	Input validation
Offset must be greater than lower bound	Input validation
Discrepancy between initial state and on/off value	Switch value must be between (input) lower and upper bounds
ERROR: Discrepancy with setpoint value more than 25 %	The control system and hydraulic system must be in equilibrium for the steady state, see remarks
Discrepancy with setpoint value and measured value	
Target below lowerbound, anti-reset windup provision activated	Informative
Target above lowerbound, anti-reset windup prov. de-activated	Informative
Target above upperbound, anti-reset windup provision activated	Informative
Target below upperbound, anti-reset windup prov. de-activated	Informative
change to Starting phase	Informative
change to Stabalize phase	Informative
change to Control phase	Informative
change to Off phase	Informative

Example

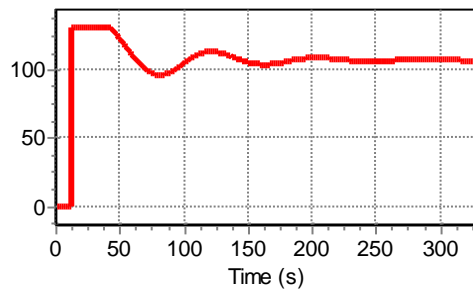
PID start/stop component controlling discharge with a fluid-level dependent setpoint.



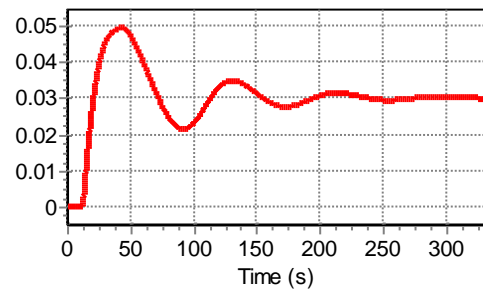
— Fluid level RSVOIR B1



— Error (Rec-Setpoint) PIDSS P1



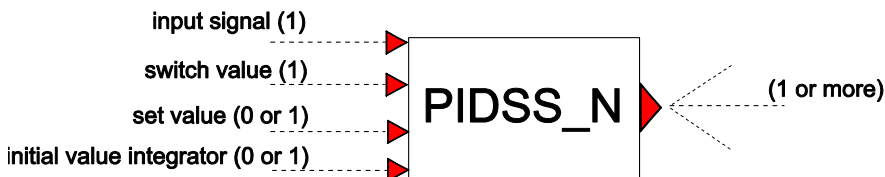
— Pump speed PUMP P1



— Discharge PUMP P1

5.23 PIDSS_N

Symbol



Purpose

To achieve a certain set value by adjusting a controlled variable of a hydraulic component, based on the difference between the input value and the set value. The PIDSS_N component includes an ON/OFF switch and is therefore similar to the PIDSS component, except that the error and output can be normalized. The component is mainly used for pump sumps in sewage systems.

Procedure

The Proportional, Integrate, Differentiate Start/Stop (PIDSS_N) component requires a minimum of two input signals: The first 'input signal' is used to compute the output signal. The second 'switch value' signal is used to determine whether to switch the PIDSS_N component ON or OFF.

The PIDSS_N component has four states: OFF, STARTING, ON and STOPPING (See Figure 4). Assuming the initial state is OFF, at the moment the input 'switch value' signal exceeds the 'switch ON' level, the state will change to STARTING. The output signal will then change linearly in time (bound by the 'ramp value') from the 'output value when OFF' to the 'output start value'. The state then changes to ON.

During the first part of the ON state the output remains equal to the 'output start value' for a duration specified by the 'stabilize time'. The second part of the ON state is the CONTROLLING part of the PIDSS_N component. If the input 'switch value' drops below the 'switch OFF' level the state will change to STOPPING. In this state the output value will decrease linearly from the 'lower bound value' to the 'output value when OFF', bound by the 'ramp value'. Thereafter the state will revert to OFF.

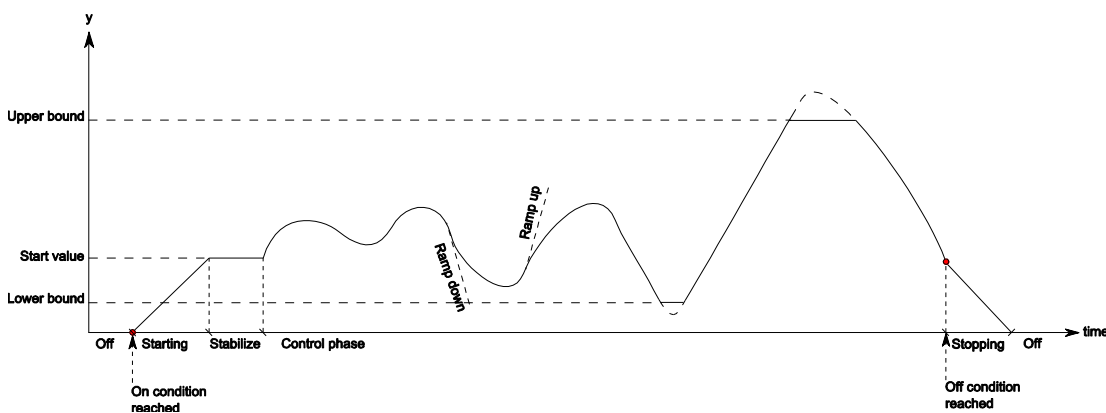


Figure 7: PIDSS_N operation

The Proportional, Integrate, Differentiate Normalized (PIDSS_N) component computes for each sample period Δt the error ϵ between an input signal $x[n]$ and a set value x_{set} .

$$\varepsilon[n] = x[n] - x_{set}$$

and upper bound values:

$$\varepsilon' = \frac{\varepsilon}{x_{upper} - x_{lower}}$$

The unscaled output signal is computed by

Proportioning:

$$y'_p[n] = c_p \varepsilon'[n]$$

Integrating:

$$y'_i[n] = y'_i[n-1] + \frac{c_p}{c_i} \varepsilon'[n] \Delta t$$

or Differentiating the normalized error:

$$y'_d[n] = c_p c_d \frac{\varepsilon'[n] - \varepsilon'[n-1]}{\Delta t}$$

where c_p is the “gain” (whose units are determined by the units of the output signal), c_i is the “integration time” (time units), c_d is the “differentiation time” (time units), Δt is the sample time-step and $[n-1]$ denotes the previous sample. The equations above can be combined to give an unscaled output signal, which is either:

Proportional (P):

$$y' = y'_p + c_0$$

Proportional and Integrated (PI):

$$y' = y'_p + y'_i$$

or Proportional, Integrated and Differentiated (PID):

$$y' = y'_p + y'_i + y'_d$$

where c_0 is an offset value. Finally, the scaled output signal is obtained using the output lower bound and upper bound values:

$$y = y' (y_{upper} - y_{lower}) + y_{lower}$$

Parameters

Parameter	input	unit	range	default	remarks
input value to switch ON	real				
input value to switch OFF	real				
Reset time (on)	real	[s]			
Reset time (off)	real	[s]			
Initial status	ON/ OFF				
Initial setpoint	real				Can be overruled by 3 rd input channel
Lower Bound (input)	real				
Upper Bound (input)	real				
Accuracy of recorded value	real				See remarks
Sample time interval	real	[s]			See remarks
Type of control	P/ PI/ PID				
Offset value (bias)	real				only if 'Type of Control' = P
Gain	real				
Integration time constant	real	[s]			
Initial value of integrator	real				only if 'Type of Control' = PI or PID Can be overruled by 4 th input channel
Start value (output)	real				
Use stabilize time		YES/ NO			
Stabilization time	real				Only if "Use stabilize time" = YES
Lower bound (output)	real				
Upper bound (output)	real				
Ramp (max. change/sec)	constant/ variable				
Ramp value	real				Only if Ramp = constant
Ramp value START	real				Only if Ramp = variable
Ramp value STOP	real				Only if Ramp = variable
Ramp value CTRL_UP	real				Only if Ramp = variable
Ramp value CTRL_DOWN	real				Only if Ramp = variable
Output value when OFF	real				

Remarks

The Sample time interval is used to decrease the resolution of the input signal. A PLC-controller can have a different sample time than the simulation time.

If the difference between the measured (recorded) value and setpoint is smaller than the accuracy, the error is set to zero.

The PID component allows an 'upper bound' and a 'lower bound' to be imposed on the output signal.

When the output value "hits" the upper or lower bound the control system can not follow the error anymore and has no possibility to feed back to the system. If this situation remains long enough the integrator fills with a large value, which inhibits timely reaction if the error value changes direction (sign). To prevent this, a simple anti-reset windup provision is implemented. This provision suspends the accumulating of the error during the time that the output value violates the lower or upper bound.

Furthermore, a maximum allowable 'ramp value' can also be specified to control the rate of change of the output.

The integrating part can be initialised with a non-zero value. This can be used to adjust the PID controller to the hydraulic steady state (see examples).

In the steady state, the error should be equal to zero. This ensures that the entire system is in a true steady state.

It is the user's responsibility to choose, and so that the output signal can be used by the controlled hydraulic component. For instance, if a valve is the controlled hydraulic component then the PID output signal must have a lower bound of 0 and an upper bound of 1. q

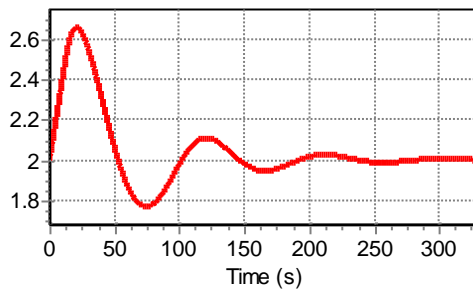
Component Messages

Message	Explanation
Integrator must be less than upper bound	Input validation
Integrator must be greater than lower bound	Input validation
Offset must be less than upper bound	Input validation
Offset must be greater than lower bound	Input validation
Discrepancy between initial state and on/off value	Switch value must be between (input) lower and upper bounds
ERROR: Discrepancy with setpoint value more than 25 %	The control system and hydraulic system must be in equilibrium for the steady state, see remarks
Discrepancy with setpoint value and measured value	
Target below lowerbound, anti-reset windup provision activated	Informative
Target above lowerbound, anti-reset windup prov. de-activated	Informative
Target above upperbound, anti-reset windup provision activated	Informative
Target below upperbound, anti-reset windup prov. de-activated	Informative

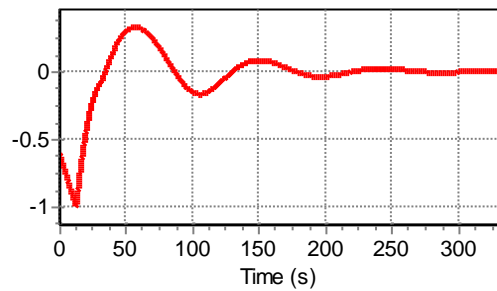
change to Starting phase	Informative
change to Stabalize phase	Informative
change to Control phase	Informative
change to Off phase	Informative

Example

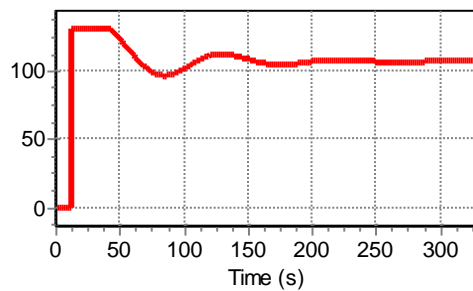
PID start/stop normalized component controlling discharge with a fluid-level dependent setpoint.



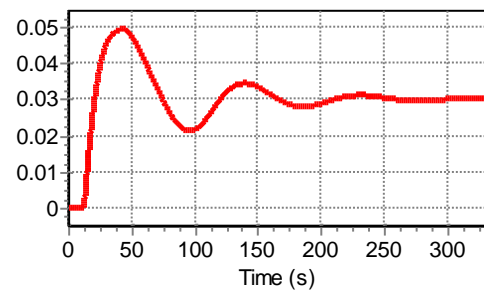
— Fluid level RSVOIR B1



— Error (Rec-Setpoint) PIDSSN P1



— Pump speed PUMP P1



— Discharge PUMP P1

5.24 AND

Symbol



Purpose

To perform a logical AND operation.

Mathematical description

The output becomes TRUE (=1) if all logical input signals are TRUE. If at least one input signal is FALSE, the output will become FALSE (=0).

Parameters

None.

Remarks

None.

Examples

Table 1 presents the possible output values that can result from two input signals. Table 2 presents the possible output values that can result from three input signals.

Table 1: Possible output values for two input signals

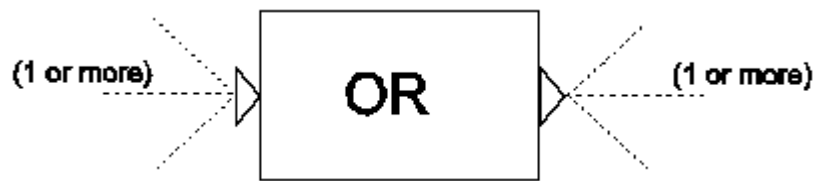
Scenario	input 1	input 2	Output
1	TRUE	TRUE	TRUE
2	TRUE	FALSE	FALSE
3	FALSE	TRUE	FALSE
4	FALSE	FALSE	FALSE

Table 2: Possible output values for three input signals

Scenario	input 1	input 2	input 3	Output
1	TRUE	TRUE	TRUE	TRUE
2	TRUE	TRUE	FALSE	FALSE
3	TRUE	FALSE	TRUE	FALSE
4	FALSE	TRUE	TRUE	FALSE
5	TRUE	FALSE	FALSE	FALSE
6	FALSE	TRUE	FALSE	FALSE
7	FALSE	FALSE	TRUE	FALSE
8	FALSE	FALSE	FALSE	FALSE

5.25 OR

Symbol



Purpose

To perform a logical OR operation.

Procedure

The output becomes TRUE (=1) if at least one input signal is TRUE. Only when all input signals are FALSE does the output become FALSE (=0).

Parameters

None.

Remarks

None.

Examples

Table 1 presents the possible output values that can result from two input signals. Table 2 presents the possible output values that can result from three input signals.

Table 1: Possible output values for two input signals

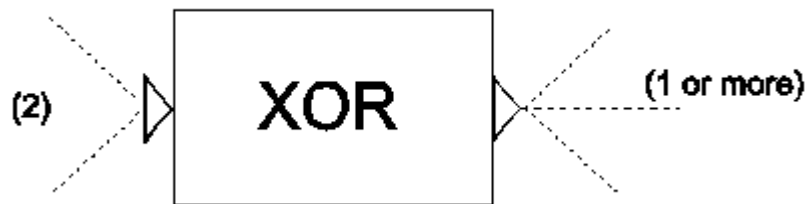
Scenario	input 1	input 2	Output
1	TRUE	TRUE	TRUE
2	TRUE	FALSE	TRUE
3	FALSE	TRUE	TRUE
4	FALSE	FALSE	FALSE

Table 2: Possible output values for three input signals

Scenario	input 1	input 2	input 3	Output
1	TRUE	TRUE	TRUE	TRUE
2	TRUE	TRUE	FALSE	TRUE
3	TRUE	FALSE	TRUE	TRUE
4	TRUE	FALSE	FALSE	TRUE
5	FALSE	TRUE	TRUE	TRUE
6	FALSE	TRUE	FALSE	TRUE
7	FALSE	FALSE	TRUE	TRUE
8	FALSE	FALSE	FALSE	FALSE

5.26 XOR

Symbol



Purpose

To perform a logical XOR (exclusive OR) operation.

Procedure

The output becomes TRUE (=1) if one and only one of the two input signals are TRUE.

Parameters

None.

Remarks

The output value shows whether the input values are mutually exclusive.

Example

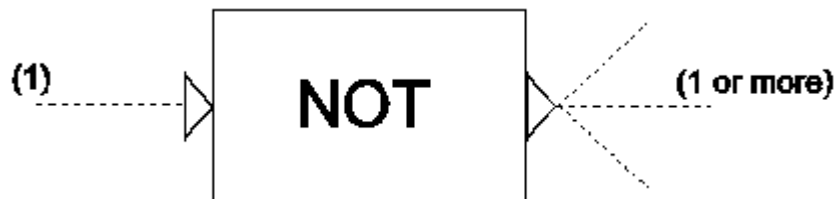
Table 1 presents the possible output values for the XOR component.

Table 1: Possible output values

Scenario	input 1	input 2	Output
1	TRUE	TRUE	FALSE
2	TRUE	FALSE	TRUE
3	FALSE	TRUE	TRUE
4	FALSE	FALSE	FALSE

5.27 NOT

Symbol



Purpose

Inverts the logical input signal.

Procedure

Changes a TRUE input signal to FALSE and a FALSE input signal to TRUE

Parameters

None.

Remarks

None.

Example

None.

5.28 SUM

Symbol



Purpose

To add the values of signal lines.

Procedure

The output value is equal to the sum of all input signals. Summation is commutative, so the order of the input signals is unimportant.

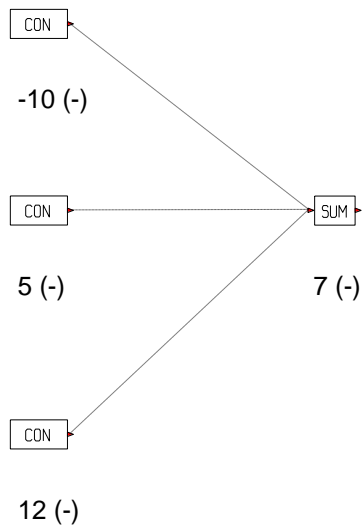
Parameters

None.

Remarks

None.

Example



5.29 MUL

Symbol



Purpose

Multiplication of input signals.

Procedure

The output value is determined by the product of all the input signals. Multiplication is commutative, so the order of the input signals is insignificant.

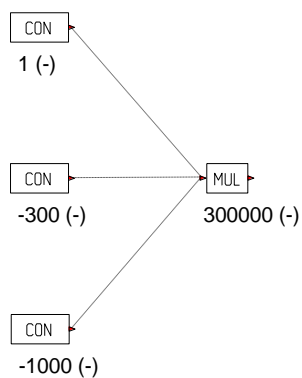
Parameters

None.

Remarks

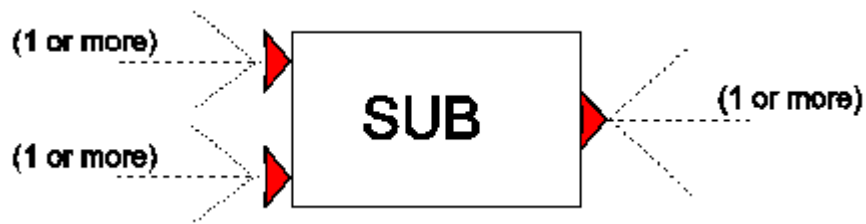
None.

Example



5.30 SUB

Symbol



Purpose

To subtract the values from one or more signals from each other.

Procedure

The output value is equal to the sum of all the signals of the upper channel minus the sum of all the signals of the lower channel.

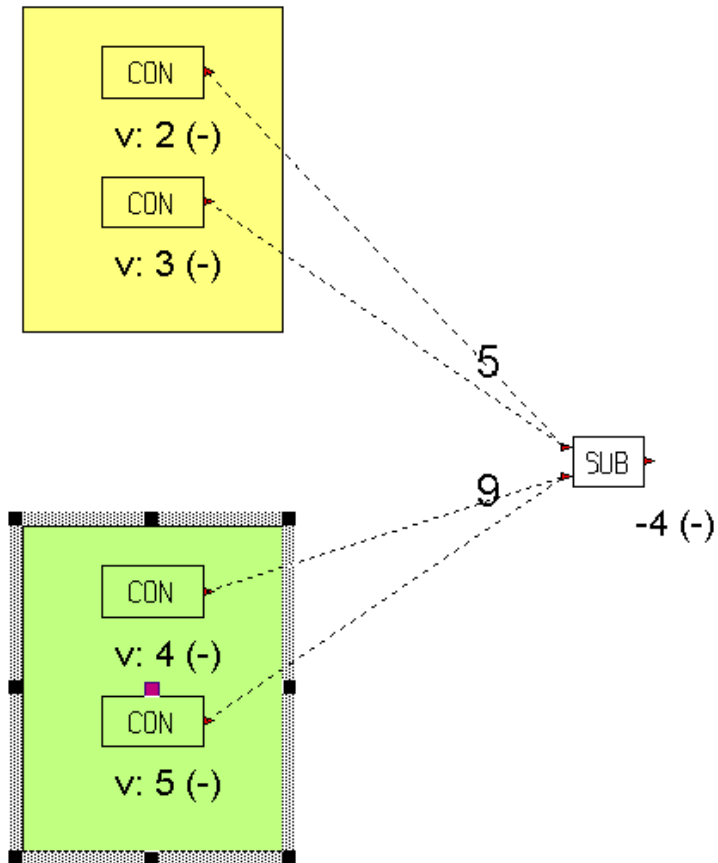
Parameters

None.

Remarks

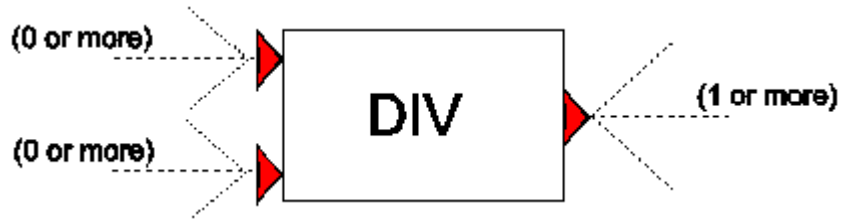
In case of empty input channels (no signal lines connected) a value of zero is assumed.

Example



5.31 DIV

Symbol



Purpose

Division of input signals.

Procedure

The output value is determined by the product of all the signals of the upper input channel divided by the product of all the signals of the lower input channel.

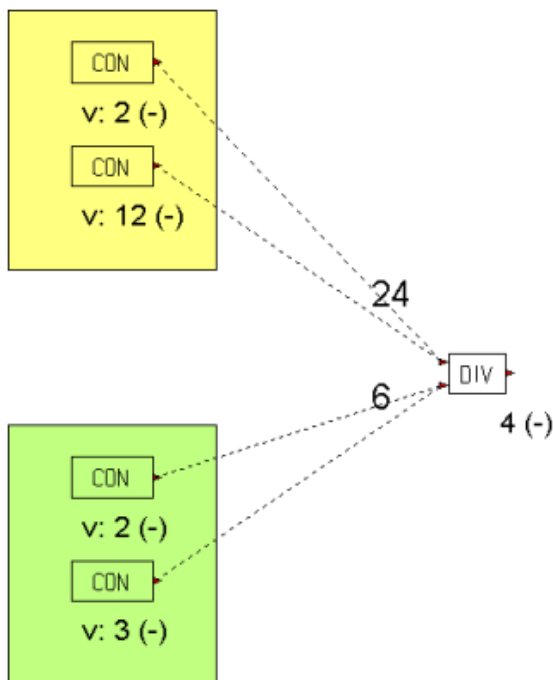
Parameters

None.

Remarks

In case of empty input channel 1 a value of 0 (zero) is assumed. In case of empty input channel 2 a value of 1 (one) is assumed.

Examples



5.32 SFUN

Symbol



Purpose

To transform an input signal . The transformation function can be either a sine-, polynome-, exponential- or power-function.

Procedure

One of the following functions is used to transform the input signal :

Sine	$y = A \cdot \sin(Bx - C)$
Polynome	$y = A + (B + Cx)x$
Exponential	$y = Ae^{Bx-C}$
Power	$y = \text{sign}(Ax + B) \times \text{abs}(Ax + B)^C$

Parameters

Parameter	input	unit	range	default	remarks
Function	Sine/ Polynome/ exp/ power				
A	real				
B	real				
C	real				

Remarks

The SFUN component is different from the FUN component, because the time itself is not used within the function. It will be obvious that if the input x is time-dependent, the output varies in time also.

Example

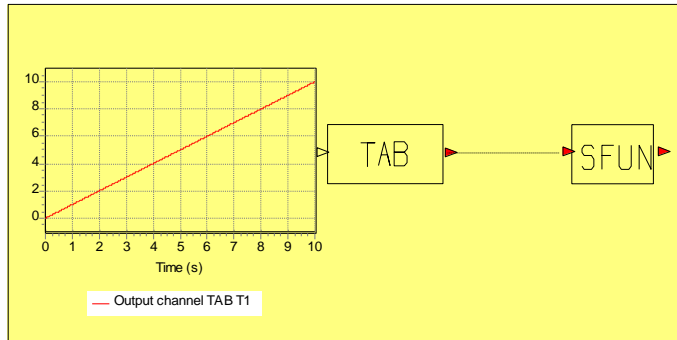


Fig 1: Control Scheme

Figures 2a – 2d present output of different type of functions, where the input signal does vary with time (increasing from 0 – 10, see fig 1) and hence the output signals are also time-dependent.

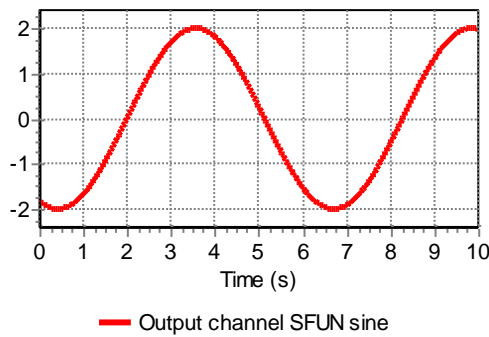


Figure 2a: Sine signal (A= 2, B=1, C=2)

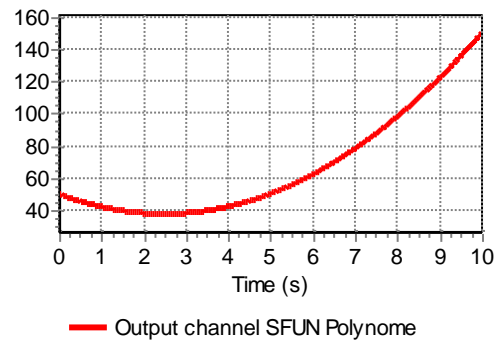


Figure 2b: Sine signal (A= 50, B=-10, C=2)

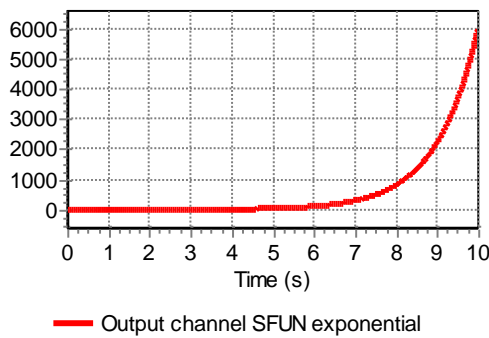


Figure 2c: Exponential signal (A= 2, B=1, C=2)

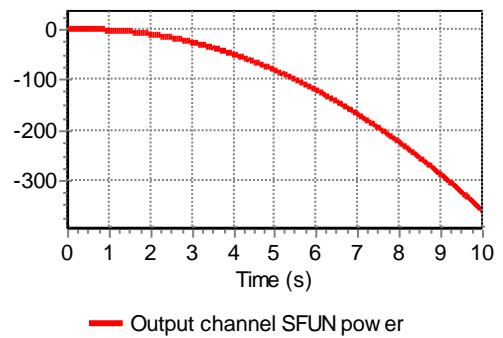


Figure 2d: Poweer signal (A= -20, B=1, C=2)

5.33 STAB

Symbol



Purpose

To transform an input signal . The transformation is done according to a table.

Procedure

Interpolation of the 'function table' is done if the input signal falls within the 'function table' limits. No extrapolation is done if the input signal falls outside the limits, instead the nearest value in the 'function table' is output.

Parameters

Parameter	input	unit	range	default	remarks
Function table	table				

Remarks

Both columns in the 'function table' must be in SI-units.

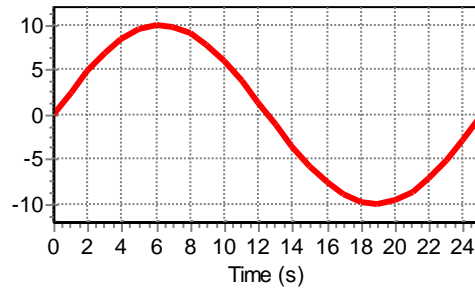
Example

Table 1 presents a STAB 'function table'. Figure 1a presents the input signal (sine) generated by FUN component and figure 1b presents the output signals for the STAB component.

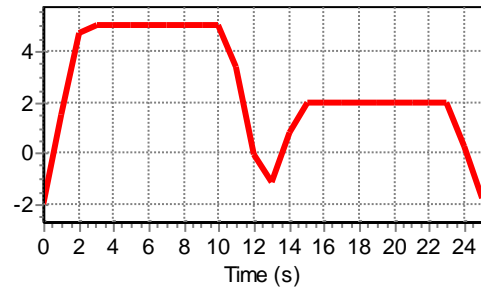
It can be seen that from 2 s to 10 s the input signal is above the 'function table' limits and a value of 5 is output. From 15 s to 23 s the input signal is outside the limits and a value of 2 is output. In the other time intervals the input signal is within limits and interpolation is used.

Table 1: A function table

Input	Output
-5.0	2.0
0.0	-2.0
5.0	5.0



— Output channel FUN F1



— Output channel STAB s1

5.34 MIN

Symbol



Purpose

Selects the minimum value amongst one or more input signals.

Purpose

At each time step the lowest value of all the input signals is output.

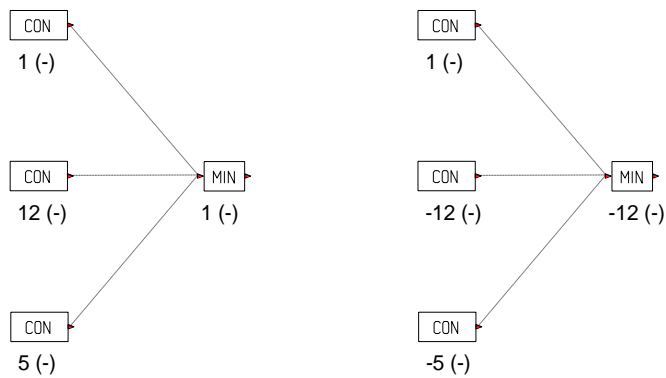
Parameters

None.

Remarks

None.

Example



5.35 MAX

Symbol



Purpose

Selects the maximum value amongst one or more input signals.

Procedure

At each time step the highest value of all the input signals is output.

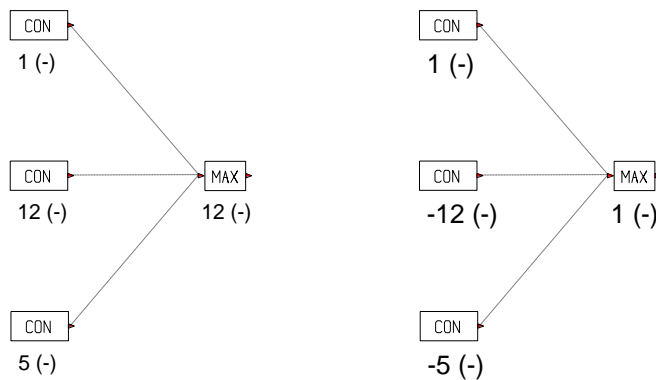
Parameters

None.

Remarks

None.

Example



5.36 MOVAVG

Symbol



Purpose

Computes the moving average of the input signal. The supported moving averages include:

- Simple Moving Average (SMA)
- Cumulative Moving Average (CMA)
- Weighted Moving Average (WMA)
- Exponential Moving Average (EMA)

Procedure

The procedure used to calculate the average varies depending on the type of averaging specified:

Moving Average Method	Formula
Simple Moving Average	$SMA_t = \frac{x_t + x_{t-1} + \dots + x_{t-(n-1)}}{n}$
Cumulative Moving Average	$CMA_t = \frac{x_1 + \dots + x_t}{t}$
Weighted Moving Average	$WMA_t = \frac{nx_t + (n-1)x_{t-1} + \dots + x_{t-(n-1)}}{n + (n-1) + \dots + 1}$
Exponential Moving Average	$EMA_t = \alpha x_t + (1 - \alpha) EMA_{t-1}$

where x_t is the signal value at time step t , and n is the window size to be used for averaging. Note that the user specifies the window size in seconds, such that the averaging is not heavily influenced by the choice of time step size.

For the SMA and WMA methods a queue of size n will be used to store past input values. This queue is filled with the steady state value, so the user does not have to specify these values themselves.

To calculate the moving average for successive values, it is possible to make computation of the SMA, CMA and WMA faster by rewriting the formulas to use the average value calculated at the previous time step. This results in the following expressions, which have been used for the implementation in Wanda:

Moving Average Method	Implementation in Wanda
Simple Moving Average	$SMA_{t+1} = SMA_t - \frac{x_{t-n}}{n} + \frac{x_{t+1}}{n}$
Cumulative Moving Average	$CMA_{t+1} = \frac{t \cdot CMA_t + x_{t+1}}{t+1}$
Weighted Moving Average	$O = \frac{n(n+1)}{2}$ $A_t = x_t + \dots + x_{t-n+1}$ $B_t = n \cdot x_t + (n-1)x_{t-1} + \dots + x_{t-n+1}$ $WMA_t = \frac{B_t}{O}$ $A_{t+1} = A_t + x_{t+1} - x_{t-n+1}$ $B_{t+1} = B_t + n \cdot x_{t+1} - A_t$ $WMA_{t+1} = \frac{B_{t+1}}{O}$
Exponential Moving Average	Unchanged

Calculating the average for the current time step by using the values of the previous time step can result in cumulative errors, which is inherent to the usage of floating point numbers. By using a double precision accumulator it is possible to reduce these errors, while keeping the speed-up by using these recursive expressions.

Parameters

Parameter	input	unit	range	default	remarks
Window size	real	[s]	[Δt, 2000* Δt]	Δt	Only for Simple and Weighted Moving Averages.
Weight factor	real	-	[0..1]	1	Only for Exponential Moving Average.

Remarks

The user specified window size is limited to a maximum of 2000 times the time step due to memory constraints. Because the window size *n* and the input values *x_t* are discrete values, the queue size (i.e. effective window size) shall be rounded to the nearest integer.

Examples

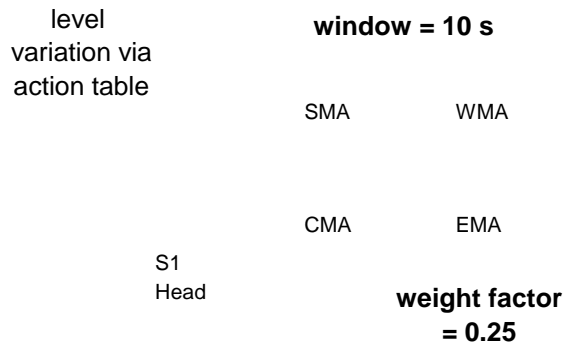


Fig 1: Control scheme

Figures 2a through 2d show the input and output of the various moving average methods.

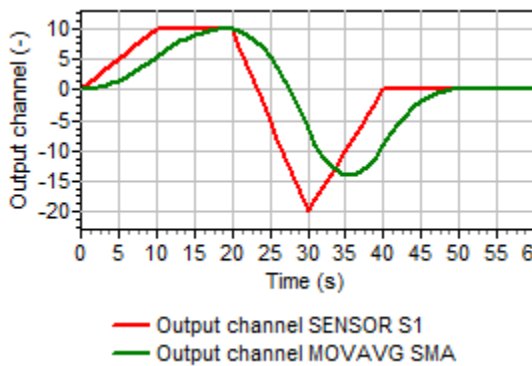


Fig 2a: Input and output of a simple moving average with a window of 10 seconds.

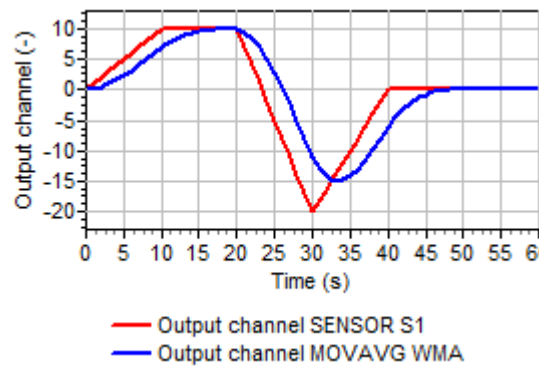


Fig 2b Input and output of a weighted moving average with a window of 10 seconds.

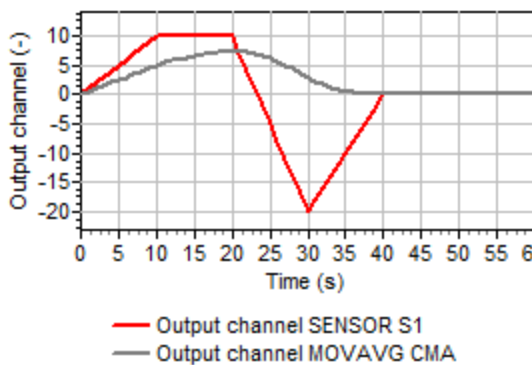


Fig 2c: Input and output of a cumulative moving average.

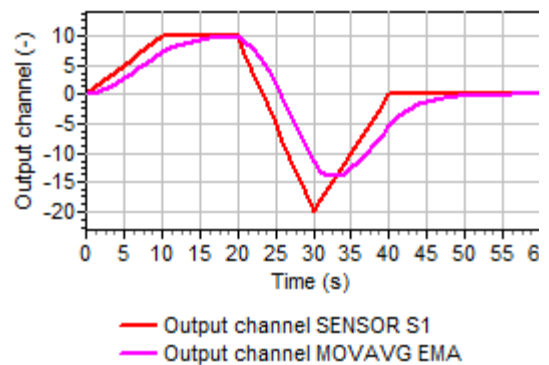


Fig 2d: Input and output of an exponential moving average with a weight factor of 0.25.

5.37 ABS

Symbol



Purpose

To transform a signal into its absolute value.

Procedure

At each time step the absolute value of the input signal is transferred to the output channel.

Parameters

None.

Remarks

None.

Examples

$$abs(-5.2) = 5.2$$

$$abs(5.2) = 5.2$$

5.38 SIGN

Purpose

Symbol



Purpose

Returns the sign of an input value.

Procedure

At each time step the sign of the input signal is output according to the following scheme:

- If $x > 0$ then output = +1.
- If $x = 0$ then output = 0.
- If $x < 0$ then output = -1.

Parameters

None.

Remarks

None.

Example

None.

5.39 BIAS

Symbol



Purpose

To add a constant value (offset, shift) to a signal.

Procedure

The input signal is added to the constant value and the result is transferred to the output channel.

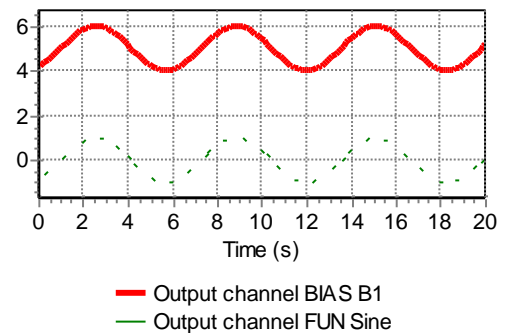
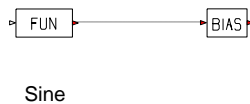
Parameters

Parameter	input	unit	range	default	remarks
Add value	real				

Remarks

None.

Example



The red thick sine wave from the BIAS component is obtained from the thin green sine wave from the FUN component by adding a bias of '+5'.

5.40 GAIN

Symbol



Purpose

To multiply (amplify) a signal.

Procedure

The output signal is equal to the input signal multiplied by the 'Multiply value'.

Parameters

Parameter	input	unit	range	default	remarks
Multiply value	real				

Remarks

None.

Example

Figure 1 shows a normal sine signal (the FUN component) being amplified by the GAIN component.

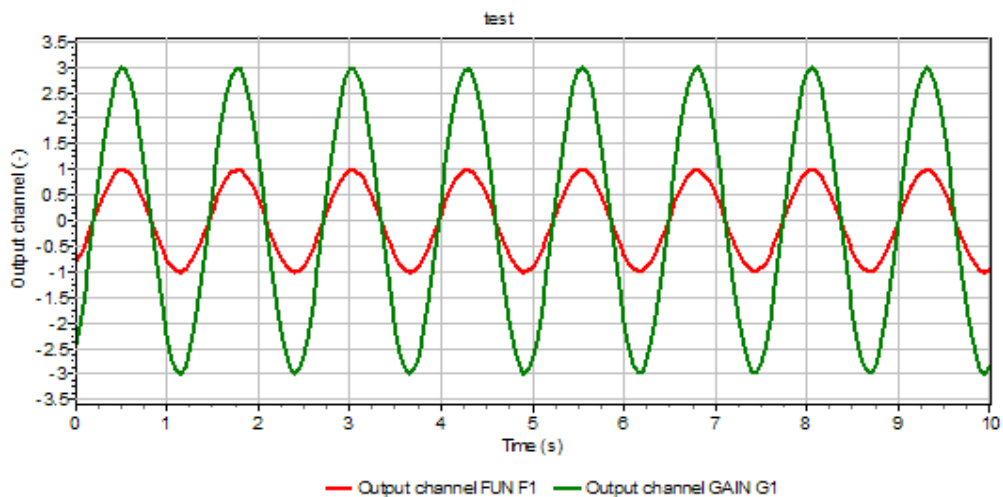


Figure 1: A sine signal

5.41 RAMP

Symbol



Purpose

To limit the rate of change of the input signal (value per second). This component can for instance be used to limit the rate of opening or closing a valve

Procedure

The input signal is limited to a maximum rate of change. Different rates are allowed for increases and decreases in the input signal.

Parameters

Parameter	input	unit	range	default	remarks
Ramp increase	real				
Ramp decrease	real				

Remarks

None.

Example

Figure 1 presents the input and output signals for a ramp component. The thin green line represents the input signal, while the thick red line represents the output signal. The input signal is a block function. The used 'ramp increase' parameter is 1, that means that the maximum change per second is 1 and it takes 10 seconds to reach the required input value. The used 'Ramp decrease' parameter is 2; so it takes 5 seconds to change from 10 to 0.

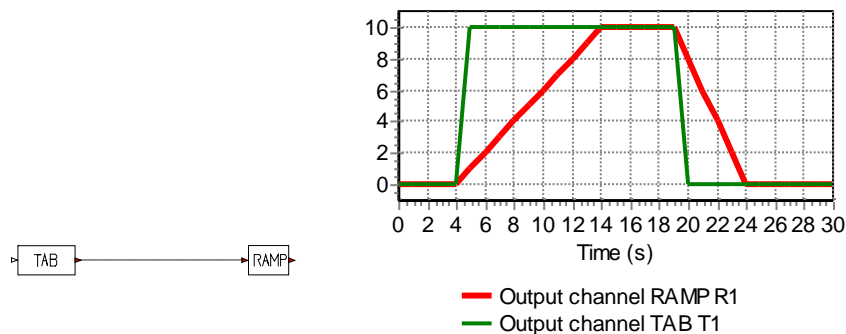


Figure 1: Input and output signals for RAMP (incr = 1, decr = 2)

5.42 TRUNC

Symbol



Purpose

Truncates an input signal if it exceeds lower or upper bound values.

Procedure

If the values of an input signal exceed either the lower or upper bound values, then those input values are replaced with the corresponding bound values.

Parameters

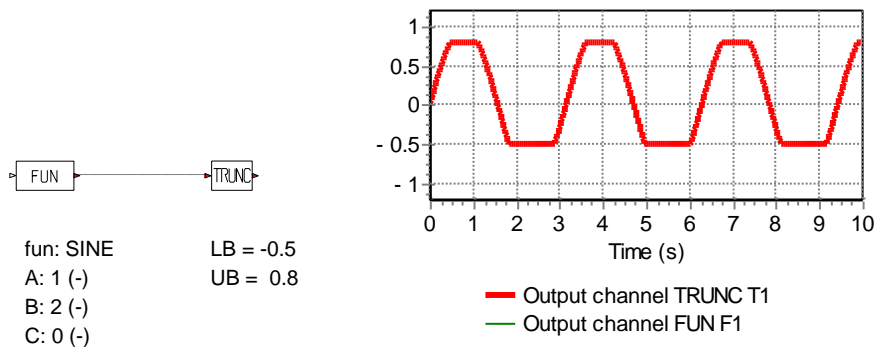
Parameter	input	unit	range	default	remarks
Lower bound	real				
Upper bound	real				

Remarks

None.

Example

Below figure presents an input sine signal, which has been truncated by a lower bound value of -0.5 and an upper bound value of +0.80



5.43 INTGRT

Symbol



Purpose

Integrates an input signal with respect to time.

Procedure

The INTGR module integrates an input signal with respect to time, as follows:

$$y = \int x(t)dt + c$$

Parameters

Parameter	input	unit	range	default	remarks
Initial value c	real				

Remarks

None

Example

Figure 1 presents a simple case where the INTGRT component is used to compute the amount of water delivered by two pumping stations over a period of one hour.

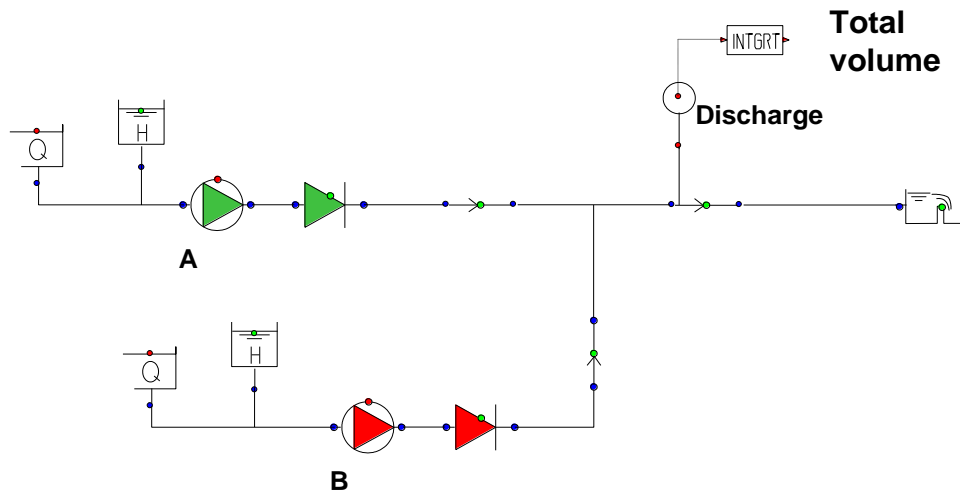
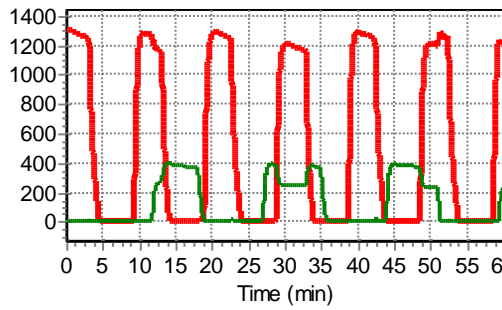
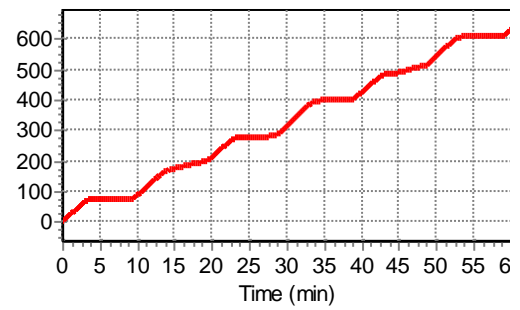


Figure 1: Scheme with a bulk water meter



— Discharge PUMP A — Discharge PUMP B

Figure 2: discharge of both pumping stations



— Output channel INTGRT I1

Figure 3: Computed flow volume

5.44 MOD

Symbol



Purpose

Computes the modulo of a number for a given divisor.

Procedure

The MOD component returns the remainder when the input value is divided by a specified divisor:

$$y = x - \text{floor}(x/c) \cdot c$$

where x is the input signal, c is the divisor, and $\text{floor}(x/c)$ is the greatest integer less than or equal x .

Parameters

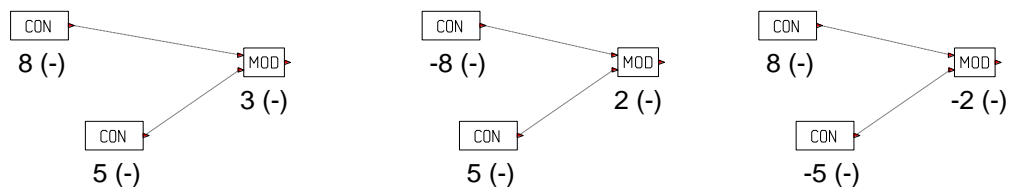
Parameter	input	unit	range	default	remarks
divisor	real				must be non-zero

Remarks

If a signal line is connected to the second input channel, the value of that signal line is used as divisor. Otherwise, the value specified in the component properties is used.

The result has the same sign as the divisor (See examples).

Example



5.45 DELAY

Symbol



Purpose

Delay (or lag) a numeric signal for a specific time.

Procedure

A memory queue is formed into which at one end the current value of the input signal is fed and at the other end the value at 'delay time' is output. Initially the queue (pipeline) is filled with the steady state value, so the queue does not have to be "loaded" during the first delay time period.

Parameters

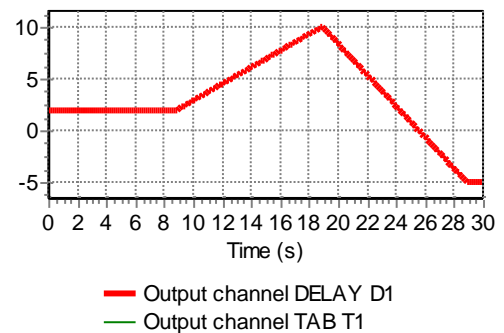
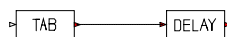
Parameter	input	unit	range	default	remarks
Delay time	real	[s]			Due to memory constraints this must be less than or equal to 2000 times the time step.

Remarks

None.

Example

The figure below shows two signals: The thin green red line is the input signal, while the thick red line is the output signal. The output signal has been delayed by 4 seconds.



5.46 LDELAY

Symbol



Purpose

Delay (or lag) a logical signal for a specific time.

Procedure

A memory queue is formed into which at one end the current value of the input signal is fed and at the other end the value at 'delay time' is output. Initially the queue (pipeline) is filled with the steady state value, so the queue does not have to be "loaded" during the first delay time period.

Parameters

Parameter	input	unit	range	default	remarks
Delay time	real	[s]			Due to memory constraints this must be less than or equal to 2000 times the time step.

Remarks

None.

Example

Figure 1 shows two signals: The thin green line is the input signal, while the thick red line is the output signal. The output signal has been delayed by two seconds.

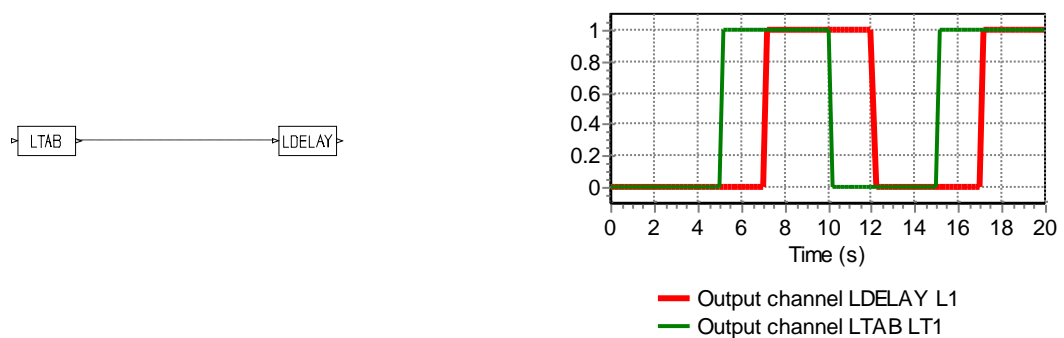
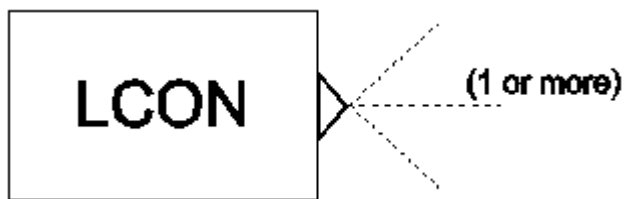


Figure 1: An input signal and a delayed output signal

5.47 LCON

Symbol



Purpose

Generates a logical constant value (TRUE or FALSE), which can be combined with other logical C-components.

Procedure

This component generates a constant logical value (TRUE=1, FALSE=0), which is always available (steady state and unsteady state) at the output channel.

Parameters

Parameter	input	unit	range	default	remarks
Logical value	TRUE/ FALSE				

Remarks

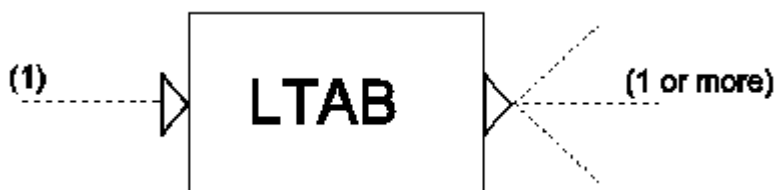
None.

Example

None.

5.48 LTAB

Symbol



Purpose

Generates a time dependent logical value (TRUE or FALSE) which can be used by other control components.

Procedure

This component is equivalent to the ON_OFF action table as used in some hydraulic components, although there is one difference. The time axis is not absolute anymore but relative to the instant at which the input signal becomes TRUE. When the input channel becomes FALSE, the output will be FALSE and the table is reset.

Parameters

Parameter	input	unit	range	default	remarks
On(1)/Off(0) table	table				

Remarks

If no logical signal line is connected to the input channel a value of TRUE is assumed and the effect is the same as with the action tables.

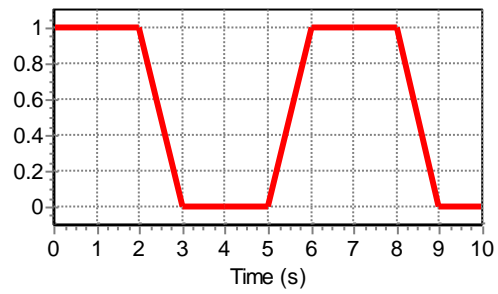
The table is a block function; the table value will be used after (!) the specified point in time (see example).

Example

The input table of LTAB is:

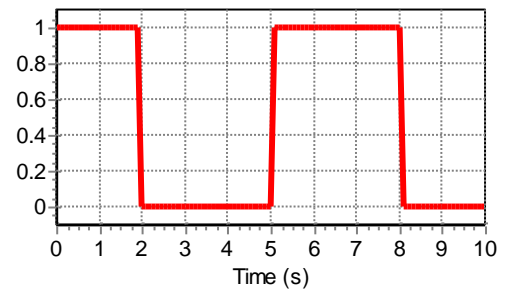
Time (s)	On(1) Off(0) (-)
0.0	1.0
2.0	0.0
5.0	1.0
8.0	0.0

Next two figures show the output channel of LTAB with different used time steps



— Output channel LTAB L1

Fig 1a: output with $dt = 1.0$ s

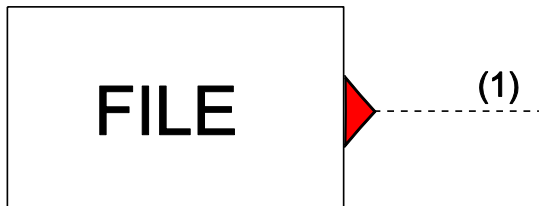


— Output channel LTAB L1

Fig 1b: output with $dt = 0.1$ s

5.49 RDFILE

Symbol



Purpose

Read values from a time dependent file.

Mathematical description

None

Parameters

Parameter	input	unit	range	default	remarks
CASE_I###.csv	integer	[-]	[0,999]		Number which gets appended to the casename
Function type	Continue/Step				
Time unit	ms/s/ min/hr				unit used for time column

Remarks

The file must be located in the same directory as the Wanda case files. The name of the file must have the following format: `<casename>_I###.csv`. Where `<casename>` is the full name of the Wanda casefiles (without extension) and `###` is the number entered in the component properties.

The ascii-input file contains 2 columns, separated by a comma, tab or space. The first column contains the time in the unit system as specified with the third input parameter. The second column contains the function value. The unit system of this function value is not known by the next H- or C-component and it will be handled as SI-unit. The time have to be started with 0.

With the function type "Continue" intermediate points are interpolated. If the simulation time exceeds the last time row, the last function value will be maintained.

With the function type "Step" the function value will remain constant until the next point in the file.

The file may contain a header with comments. These comment records have to be started with a '#' or '*'.

Note: The last record of the file must be finished with a carriage return.

Example

* example of input file

```
* time in ms, function value is pressure (bar) at location X
*
* note: output of RDFILE is bar; this value must be multiplied by 100.000 to get
* pressures in the SI-unit (N/m2)

0, 2.10
100, 2.80
# comment within the table is allowed
200, 3.25
300, 2.60
400, 2.80
500, 2.60
```

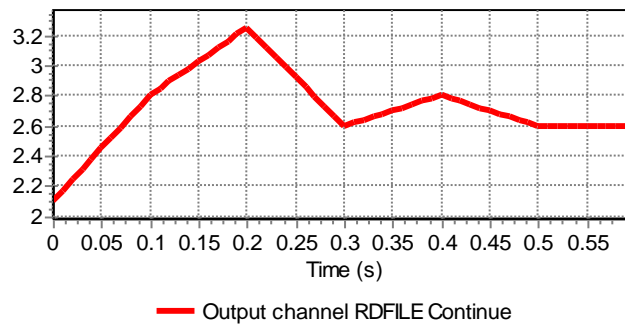


Fig 1: Result with Type = Continue

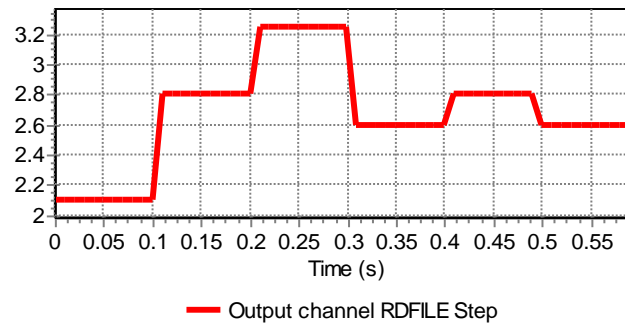
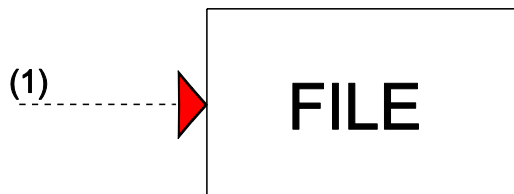


Fig 2: Result with Type = Step

5.50 WRFILE

Symbol



Purpose

Write values to a file.

Mathematical description

None

Parameters

Parameter	input	unit	range	default	remarks
CASE_O###.csv	integer	[-]	[0,999]		Number which gets appended to the casename
Time unit	ms/ s/ min/ hr				
Output incr	integer	[-]			Output written every 'output incr' time-steps to the file

Remarks

The WRFILE component can be used to write results to a file, which can be read into a WANDA model or spreadsheet program at a later stage. The comment field of the WRFILE component will be used as a header in the output file.

The output file will be placed in the same directory as the Wanda case files. The name of the file will have the following format: <casename>_O###.csv. Where <casename> is the full name of the Wanda casefiles (without extension) and ### is the number entered in the component properties.

Example

Figure 1 presents an example where the WRFILE component is used. The product of two different signals (a sine and a polynome) is written to a case related file with output number 48. The simulation time step is 1 s.

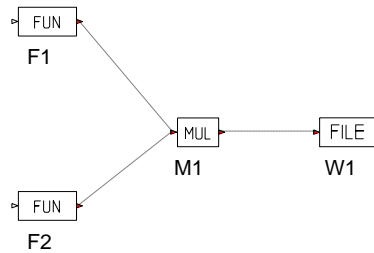


Fig 1: Control scheme

The input of the WRFILE component is:

- Name: W1
- Comment: product sine*poly
- CASE_O###.csv: 48
- Time unit: s
- Output incr: 5

The WRFILE component writes the result of the multiplication of the two input signals to a file named "MyCase_O048.csv". This file can be opened with any text editor or spreadsheet.

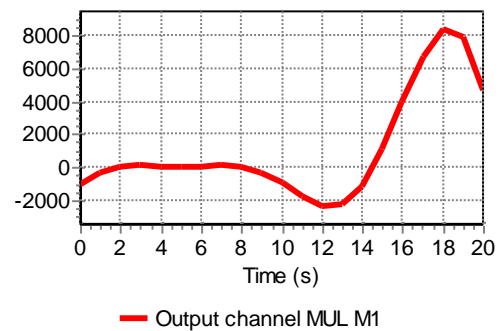
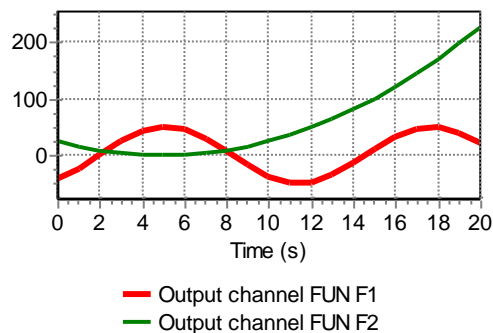


Fig 2: Input signals and result of multiplication

The created file "MyCase_048.csv" contains two columns: The first column specifies the time, while the second column specifies the values given to the WRFILE component. The complete contents of the file "MyCase_048.csv" is shown below:

# Time [s]	product sine*poly
0.000000E+00	0.000000E+00
5.000000	42.07355
10.00000	-280.6266
15.00000	-1131.633
20.00000	7825.173

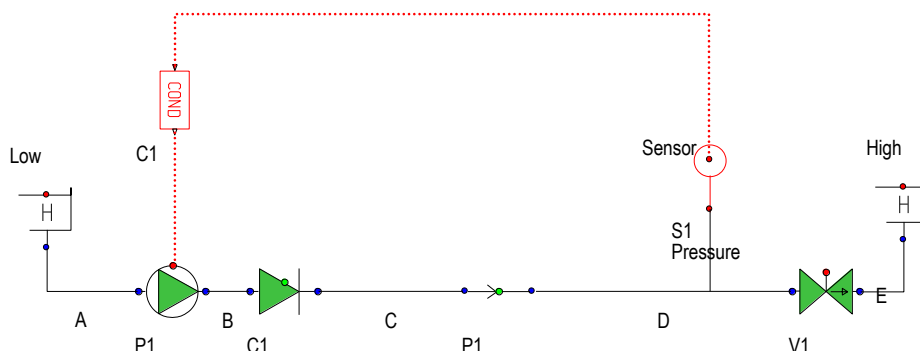
5.51 Examples

In this chapter a few illustrative examples will be discussed to demonstrate some of the possibilities of WANDA CONTROL. The examples are part of the WANDA set-up (see subdirectory Wanda\Examples).

5.51.1 Tripping pump

In the figure below a very simple system is depicted. A pump drives a transport line, which is terminated by a shut off valve. When the shut off valve starts to close the pressure at H-node D will rise, as well as the pressure in pipe P1. The pressure is measured by a sensor and transmitted to the C-component COND(itional action).

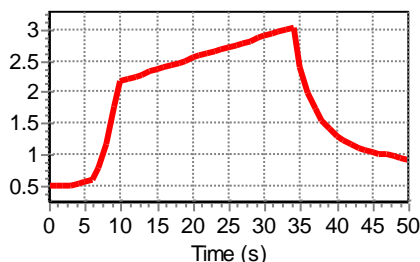
The pump must be turned off if the pressure in H-node D exceeds the set pressure of 2 barg during 2 seconds.



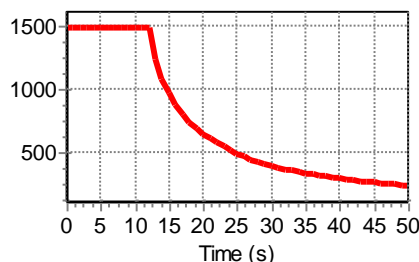
The parameters of COND are as follows:

- logical operator: LT
- set value: 20000 [-]
- reset time (true) (Pump = on): 0 [s]
- reset time (false) (Pump = off): 2 [s].

As soon as the measured pressure has risen above 2 barg for at least 2 seconds (reset time), the output will become FALSE. This logical signal is fed into the pump of drive type TRIP. For this type of pump a control signal becoming FALSE is interpreted as the signal to start the tripping. The result is thus that, when the pressure at H-node D goes through the 200000 value (Pressure in N/m² corresponding with 2 barg) at T=10 seconds, the pump will trip at T=12 seconds.

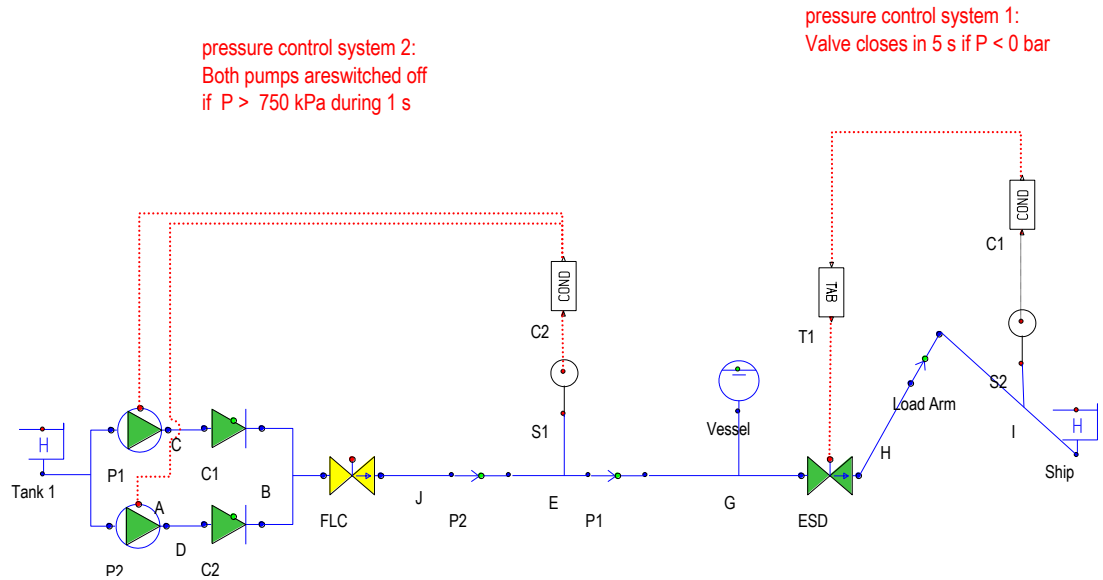


— Pressure H-node D (Trip_pmp)

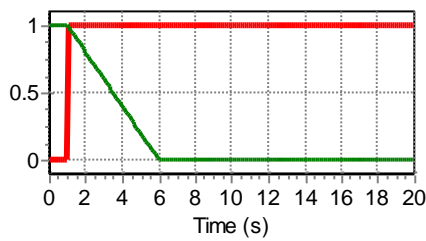


— Pump speed PUMP P1 (Trip_pmp)

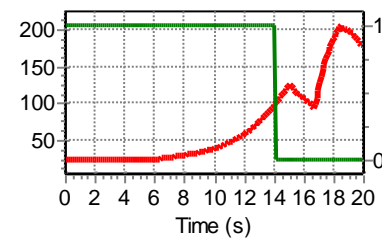
5.51.2 Loading Arm



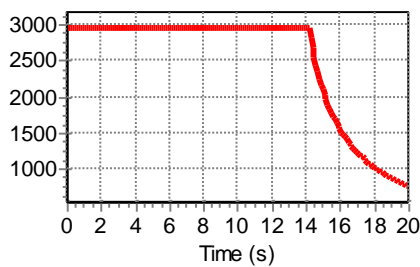
The figure above shows a loading line from tank 1 to ship B. If the back pressure drops below a certain set level, e.g. in case the ship breaks loose, the conditional action closes valve V1. A pressure wave is then propagated through pipe P1, which is monitored by a sensor just downstream of the pumping station. If the set pressure is exceeded the pumps are switched off.



— Output channel COND C1 (Load01)
 — Output channel TAB T1 (Load01)



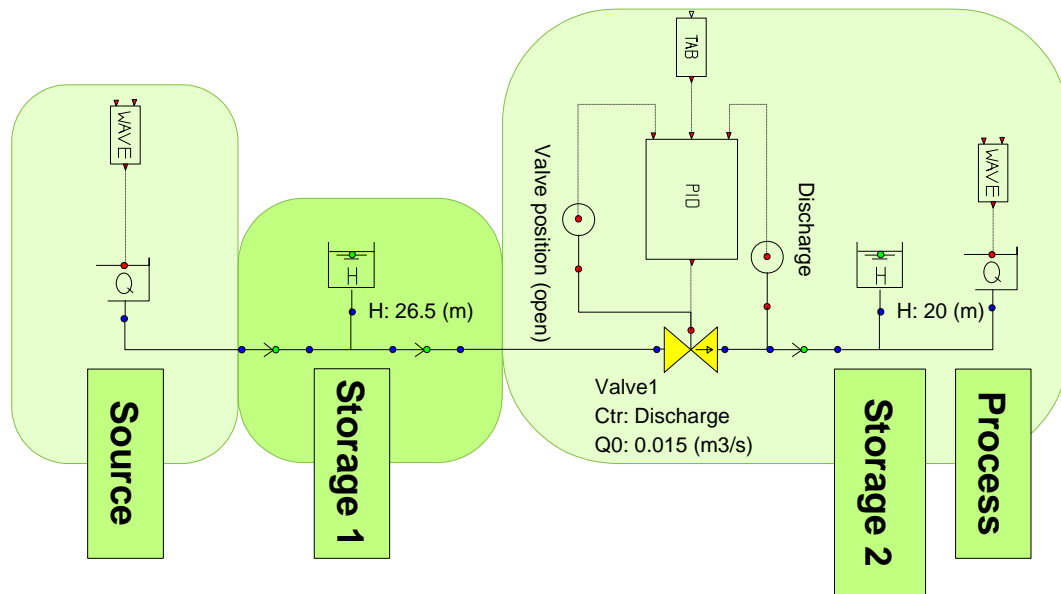
— Head H-node E (Load01)
 — Output channel COND C2 (Load01)



— Pump speed PUMP P1 (Load01)

5.51.3 Flow control valve

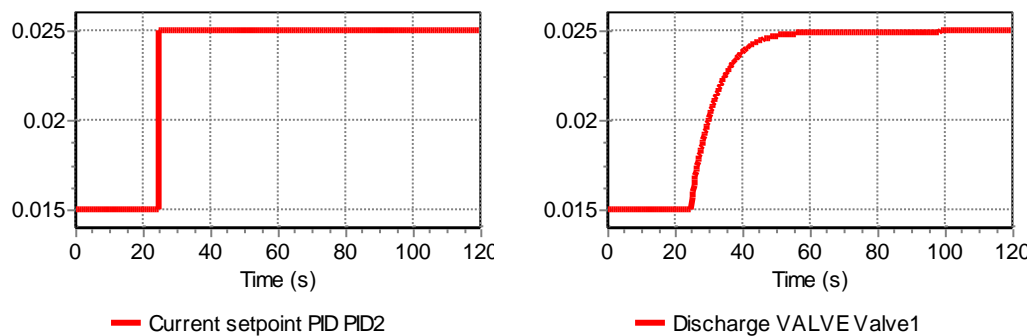
To demonstrate the operation of a PID-controller a simple flow control system is shown in the figure below.

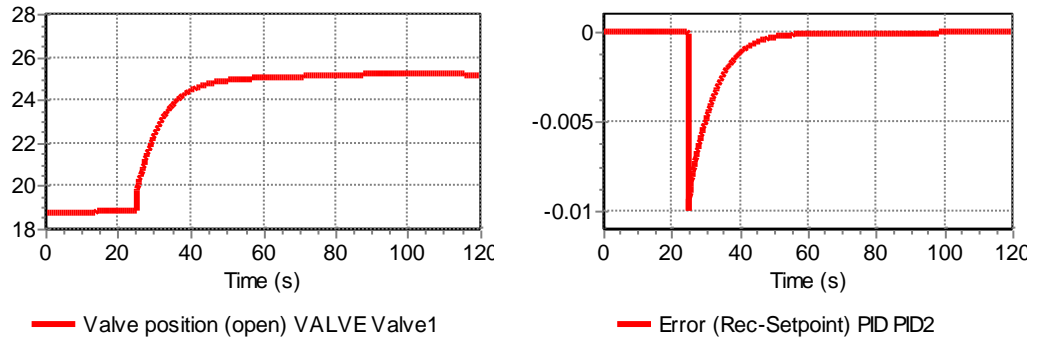


In steady state the control scheme must be in balance too. The setpoint of the flow control valve is 15 l/s (because the SI-unit of discharge is m³/s, the specified property value will be 0,015) To achieve this, the initial setting property of the valve is set to 'Discharge', with an initial discharge equal to the setpoint of the PID controller. As a result of the hydraulic calculation, the valve position will be calculated. The valve position is measured with a sensor and fed back into the PID controller as the initial value of the integrator, ensuring that the control system is in balance with the hydraulic system.

At the begin and end of the system a WAVE component is used to introduce some flow disturbances.

At t = 25 s the setpoint is changed from 0.015 m³/s to 0.025 m³/s using the TAB-component. The PID controller adjusts the valve position to reach the new setpoint.

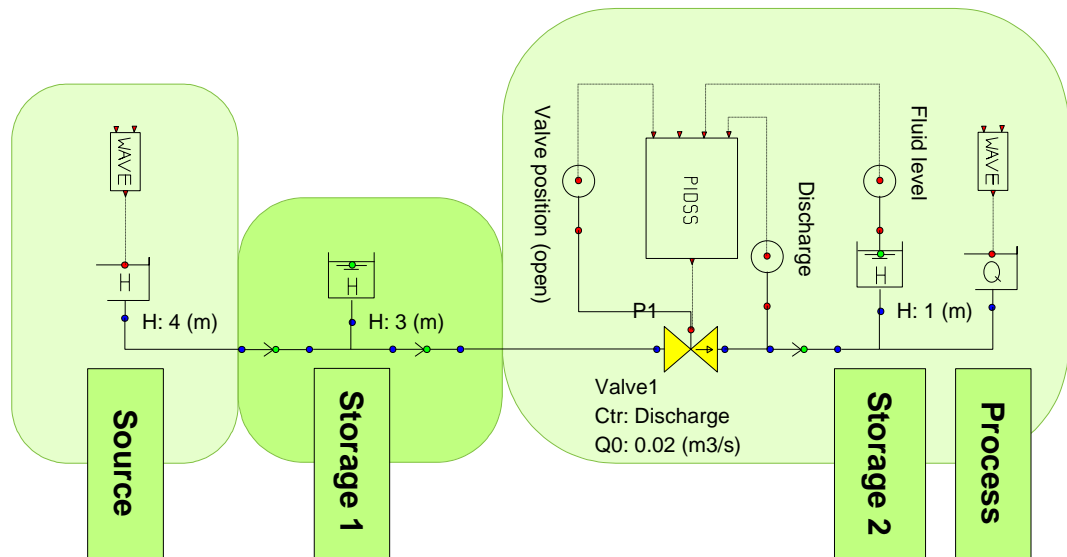




5.51.4 Flow control valve with start/stop conditions

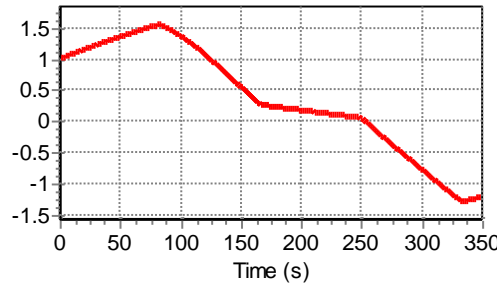
To demonstrate the operation of a PIDSS-controller the flow control system similar to the previous example is extended with an open/close condition.

In the scheme below the flow control valve fills storage tank 2. If the tank is full enough the valve must be closed and if the tank becomes empty the valve must be opened. During operation a required flow must be achieved.

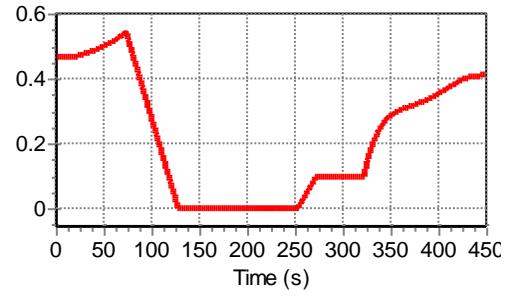


The PIDSS controller has an extra input signal, which is used to switch the controller on and off at a certain value.

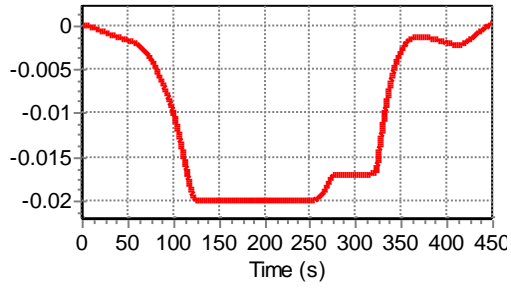
The discharge setpoint is 0,02 m³/s. The controller switches on and off based on the fluid level in the second storage tank. The ON-level is 0 m, the OFF level is 1,5 m.



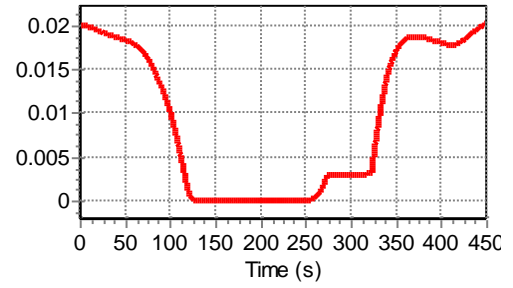
— Fluid level RSVOIR Storage #2



— Valve position (open) VALVE Valve1



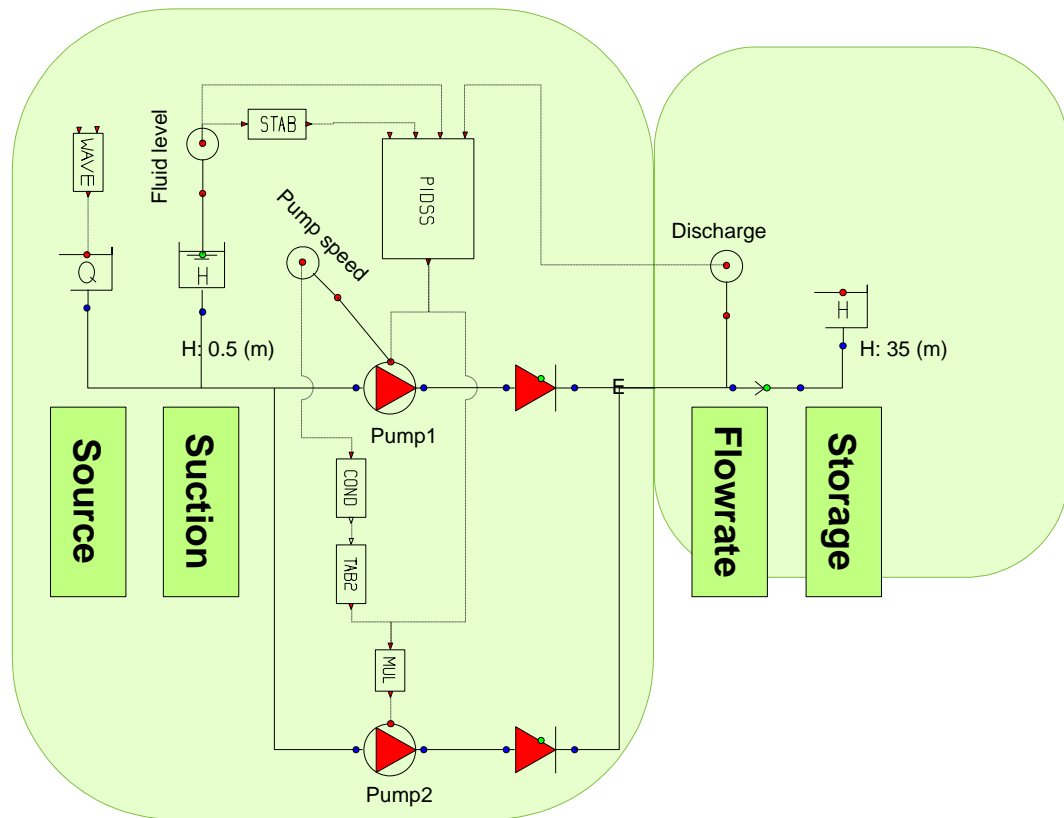
— Error (Rec-Setpoint) PIDSS P1



— Discharge VALVE Valve1

5.51.5 Control of a pumping station

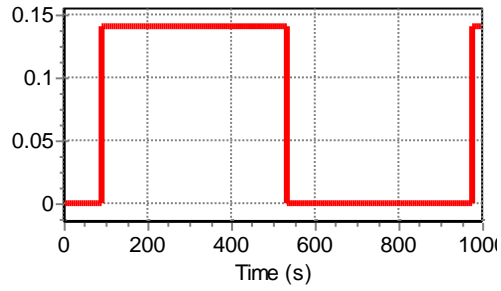
In the figure below a characteristic sewage water transport system is shown. The pumping station consists of two pumps. The system has a discharge controller with a sump-level dependant set-point.



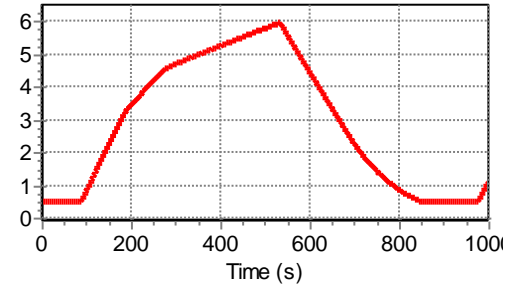
If the capacity of the first pump is not enough to maintain the required setpoint, the second pump becomes in operation.

Pump 1 starts if the level in the sump becomes above the IN-level (3 m) and stops if the level becomes below the OUT-level (0,5 m)

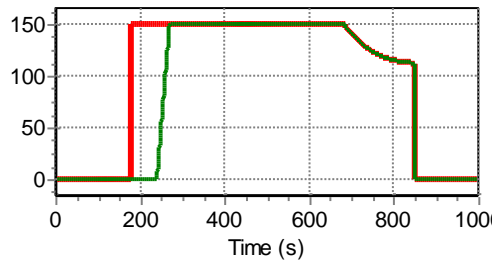
The pumpspeed of pump 1 is measured using a sensor and fed into the COND block. The second pump only starts if the first pump is at it's maximum speed for longer than 60 s. It runs for at least 3 minutes before shutting down again. This is done via de TAB2 block, which outputs either 1 or 0. The multiplication with the output signal of the PID controller causes the second pump to run



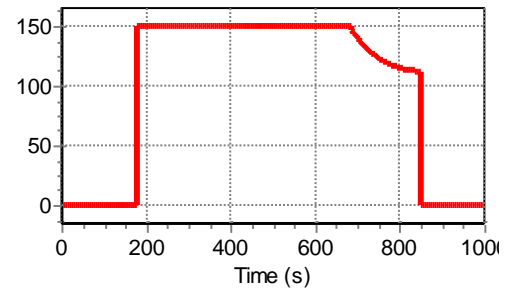
— Discharge BOUNDQ Source



— Fluid level RSVOIR Suction



— Pump speed PUMP Pump1
— Pump speed PUMP Pump2



— Output channel PIDSS PIDSS 1

6. WANDA Heat

6.1 HEAT domain

This part of the Help documentation contains all relevant information of the HEAT domain. First the core quantities which the in the heat domain are used, are described. An important difference with the Liquid domain are the temperature dependant properties of the fluid. All available component and node types for the HEAT domain are described. Each class description of the Heat-component contains the symbol, the list of types and the mathematical model. Each type description consists of the input list (with descriptions, dimensions and ranges), the component specific output list, the actions list and a list of error messages and explanations.

6.1.1 Core quantities

In the heat domain the following core quantities are used:

- Total pressure p_t [N/m²]
- mass flow rate \dot{m} [kg/s]
- temperature T [K]

With:

$$p_t = \rho gh + \frac{\rho v^2}{2} \quad (1)$$

$$\dot{m} = \rho Av \quad (2)$$

In which:

p	=	the manometric (or piezometric) pressure	[N/m ²]
\dot{m}	=	mass flow	[kg/s]
ρ	=	density	[kg/m ³]
g	=	gravitational acceleration	[m/s ²]
h	=	pressure head	[m]
v	=	velocity	[m/s]
A	=	cross section area	[m ²]

The pressure p , discharge Q and (energy) head H (the core quantities as used in the liquid domain) are calculated as post processing according:

$$p = p_t + \frac{\rho v^2}{2} \quad (3)$$

$$Q = v \cdot A \quad (4)$$

$$H = z + \frac{p}{\rho g} + \frac{v^2}{2g} \quad (5)$$

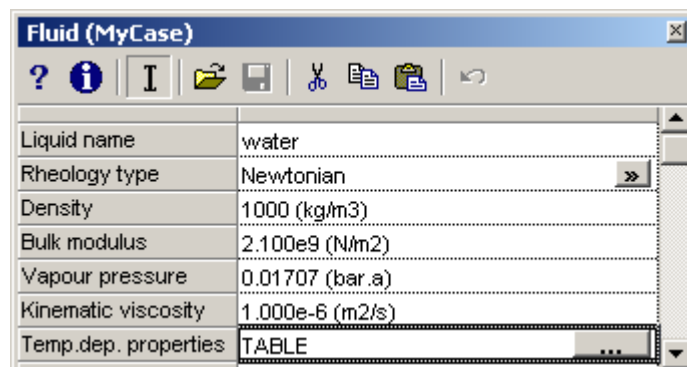
In which:

p_t	=	total pressure	[N/m ²]
Q	=	discharge (volume flow)	[m ³ /s]
H	=	energy head	[m]
z	=	elevation	[m]

The temperature T in the user interface is displayed in degree Celsius as standard and only possible unit. This has been chosen due to limitations in the unit system. Controls using temperature should also use degree Celsius. The calculation core however calculates everything in Kelvin.

6.1.2 Fluid properties

In Wanda Heat the properties of the fluid are depending on the temperature and therefore a table with “**Temperature dependent fluid properties**” has to be filled before starting any simulation. This table can be found in the “**Fluid window**” under the “**Model**” menu.



Fluid window

The following properties are depending on the actual temperature:

- Density [kg/m³]
- Kinematic viscosity [m²/s]
- Vapour pressure [bar.a]
- Specific heat [J/kg/K]

The above properties of the fluid have to be specified within the computational range (the range of temperatures Wanda encounters during the computation). An error will result if Wanda computes a temperature anywhere outside this range at any time ('Temperature outside table range').

It should be noted that Wanda linearly interpolates between the temperatures at which the values are specified in the table. Too little rows in this table (especially where the fluid properties are not or less linear varying with temperature) therefore might result in an erroneous fluid definition.

	Temperature (°C)	Density (kg/m3)	Kin.viscosity (m2/s)	Vapour pressure (bar.a)	Specific heat (J/kg.K)
1	0	999.80	1.7920e-6	0.0061100	4217.0
2	1.0000	999.90	1.7310e-6	0.0065700	4213.0
3	2.0000	999.90	1.6740e-6	0.0070500	4210.0
4	3.0000	1000.0	1.6200e-6	0.0075700	4207.0
5	4.0000	1000.0	1.5690e-6	0.0081300	4205.0
6	5.0000	1000.0	1.5200e-6	0.0087200	4202.0
7	6.0000	1000.0	1.4730e-6	0.0093500	4200.0
8	7.0000	1000.0	1.4290e-6	0.010010	4198.0
9	8.0000	999.90	1.3860e-6	0.010720	4196.0

Example of fluid table

The fluid window contains also the property fields "Density", "Vapour pressure" and "Kinematic viscosity". These are ignored by WANDA Heat but only used in WANDA Liquid.

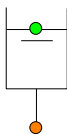
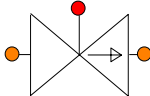

By default the fluid property table for water is loaded (stored in program directory Wanda4\Property Templates\Examples\Fluids). The user might build a database of fluid properties to be loaded / saved in the fluid window. Fluid property template files can be loaded into the fluid properties window by clicking on the 'open property template file' button or pressing 'Ctrl + O' in the fluid window. In a similar way the user might save (key: 'Ctrl + S') his fluid property templates.

6.1.3 Overview of Heat Components

Currently, the following Heat components as listed below are available in WANDA. Your license specifies if you are authorised to use them.

Type	Symbol	Short description
Heat air vessel		Vertically positioned, prismatic (constant area), non-vented air vessel
Heat boundary: Heat BoundPT		Pressure - temperature boundary condition
Heat boundary: Heat BoundMT		Mass flow - temperature boundary condition

Type	Symbol	Short description
Heat boundary: Heat BoundT(PM)		Temperature – Pressure boundary condition or Temperature - Mass flow boundary condition
Check valve / Non-return valve		Ideal check valve; closes if $Q < 0$, opens if $p_1 - p_2 > \Delta p_{preopen}$
Heat exchanger: Heat supply QHeat in		Supply heat to or extract heat from the fluid by directly specifying a heat input.
Heat exchanger: Heat supply Tdown		Supply heat to or extract heat from the fluid by specifying a temperature at downstream Heat node.
Heat exchanger: Heat supply QHeat in limit on T		Supply heat to or extract heat from the fluid by directly specifying a heat input, with user specified limits for the minimum and maximum temperatures
Heat exchanger: Heat exchanger		Exchange heat with surroundings due to temperature difference between fluid and ambient temperature.
Heat exchanger: Heat demand		Heat demand for district heating systems with different input parameters for heating and warm (tap) water.
Pipe		Heat pipe including friction model, friction generated heat and heat transfer to or from the surroundings.
Pump		Complete pump model with various data and drives; positive and negative speed.
Heat pump		Heat pump: transports heat from low to high temperature via an internal liquid

Type	Symbol	Short description
Surge tower, surge shaft		Surge tower with storage area independent of liquid level.
Valve		Control or block valve with choice out of four predefined Deltares standard head loss characteristics or user characteristics (Cv Kv, Xi).
Converters		Converter to enable the connection of liquid component to heat components

6.1.4 Component properties

All input and output properties are displayed in the property window (selection property window: Shift-F11). The property window can be divided in 5 parts:

Part of property window	Properties
General input data	Name, comment, keywords etc
Component specific input data	Type dependent
Action table	Type dependent for active types only (actions)
General output data	Messages, default HEAT output variables
Component specific output data	Type dependent (including pressure loss, temperature loss, mass)

The default HEAT output variables (for each connect point) are: discharge, energy head, pressure, velocity, density, temperature and mass flow rate.

The visibility of some properties can be managed using the checkboxes in the upper part of the Mode & Options window in the “Model” menu.

The properties are type dependent. They are described for each component type separately. Some details about the action table are described below.

6.1.5 Hydraulic and thermal action tables

Hydraulic or thermal transients in a pipeline system are caused by changes in the flow conditions or thermal boundary conditions. These changes are initiated by manoeuvring Heat components in the system, called actions. Not all components in a pipeline system can be activated. A component is categorised as either an active or a passive component, depending on its manoeuvrability.

An action is described using a control parameter specified as a function of time. The control parameter depends on the component type. In WANDA, a control parameter function is specified through an action table in the properties of the component or by connecting a control system, see chapter 5: “Control Components”. The control system overrules the component specific action table.

6.2 Heat nodes

A Heat node connects two or more Heat components which each other. In the diagram this Heat node is represented by a connection line, connected to the hydraulic connect point of a Heat component (orange dot). In case more than two connect points have to be connected the Heat node consists of a master and one or more slave connection lines, thus forming a tree. For more information about the connection line, see "Connecting components" on page 34.

Heat node type

Type label	Description
Heat node	Default Heat node for connecting components in WANDA Heat
Heat node init pT	Heat node defining initial pressure and temperature
Heat node init p	Heat node defining initial pressure
Heat node init T	Heat node defining initial temperature
Heat node with demand	Heat node with base-demand and pattern (for use in drinking water networks)
Heat node conditional init pT; heat node with conditional initial pressure and temperature	Heat node, which only defines the temperature and pressure when it is not defined

6.2.1 Mathematical model

Temperature definition for Heat node

The temperature in the Heat node is calculated based on an energy balance. The energy balance takes all energy flows connected to the Heat node into account. For n components the equation is as follows:

$$\sum_{i=1}^n c_{p_i} T_i \dot{m}_i = 0$$

In which:

c_p	=	specific heat of the fluid	[J/kg/K]
T	=	temperature	[K]
\dot{m}	=	mass flow	[kg/s]

The temperature of the Heat node itself is defined by setting the temperature of the outflow (flow away from the Heat node) equal to the temperature of the Heat node itself. This means that the temperature in the Heat node is modeled as completely mixed.

Cavitation in Heat node

If the pressure drops below the vapour pressure, cavitation occur. Cavitation in a Heat node is only taken into account if the Heat node is connected to a PIPE.

Default output properties

Each type of Heat node has a few standard output properties. These are:

- Pressure [N/m²]
- Temperature [°C]
- Total Pressure [N/m²]
- Void fraction (cavitation) [-]

6.2.2 Heat node

This Heat node is the standard hydraulic node for WANDA Heat, by default generated after connecting two Heat components.

The output can be calculated if the Heat node is part of a fully connected (“open”) hydraulic network with one or more boundary conditions for the pressure and one or more boundary conditions for the temperature (use of BOUNDPT, BOUNDMT or BOUNDT(PM)). “Open” means that each Heat component has a certain relation between pressure and mass flow and that there is at least one component prescribing the pressure (e.g. no fully closed valves).

Input properties

Description	Input	SI-units	Remarks
Elevation	real	[m]	

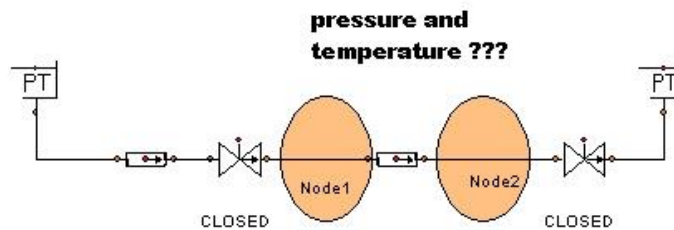
Messages

Message	Message	Explanation
Cavitation in steady state not allowed	error	The lowest pressure in the Heat node is less than the vapour pressure
Pressure < P _{vapour} ; Cavitation not supported for node without PIPE connections	warning	In this Heat node there is no PIPE component connected. The cavitation model needs at least 1 PIPE to compute the vapour volume. This warning is only given the first time the pressure condition is true; the rest of the simulation is physically invalid.
Cavitates	informative	Heat node starts cavitating; the lowest pressure drops below the vapour pressure
Cavitation collapses	informative	Void fraction becomes 0
Change Type to Heat node with initial pressure	error	Heat node is part of an isolated part of the network in which no information is available for the pressure; the user must define the pressure in this part of the model.
Change Type to Heat node with initial temperature	error	Heat node is either part of an isolated part of the network in which no information is available for the temperature or there is no flow in the connected components; the user must define the temperature in this part of the model.

6.2.3 Heat node init pT

If the Heat node is part of a “closed” or ”isolated” hydraulic network, there is no connection with a boundary condition like a BOUNDPT, then the actual pressure and temperature in the steady state computation cannot be determined.

This is explained in the figure below: the pressure and temperature in the nodes Node1 and Node2 cannot be calculated.



It is therefore necessary to define the pressure and temperature for this area in one of the Heat nodes. It is up to the user which Heat node the pressure will be prescribed (changed into the type: Heat node init pT). The type of the node can be changed via the property window (drop down list in “Type” field).

Note: the initial pressure and temperature are only applied during steady state. If during transient an “isolated network” occurs the pressure and temperature of the previous time step will be used.

Note: By using the node with initial pressure, the mass flow at the node is no longer set to zero. This can thus result in a non-zero mass flow from the node. A warning is issued when this occurs.

Input properties

Description	Input	SI-units	Remarks
Elevation	real	[m]	
Pressure t=0	real	[N/m ²]	
Temperature t=0	real	[°C]	

Messages

Message	Message	Explanation
Cavitation in steady state not allowed	error	The lowest pressure in the Heat node is less than the vapour pressure
Pressure < Pvapour; Cavitation not supported for Heat node without PIPE connections	warning	In this Heat-node there is no PIPE component connected. The cavitation model needs at least 1 PIPE to compute the vapour volume. This warning is only given the first time the pressure condition is true; the rest of the simulation is physically invalid.
Cavitates	informative	Heat node starts cavitating; the lowest pressure drops below the vapour pressure
Cavitation collapses	informative	Void fraction becomes 0
Mass continuity not observed	Warning	There is a mass flow from the node

6.2.4 Heat node init p

As with the “Heat node init pT” above, the “Heat node init p” can be used in an isolated part of the model to initiate the pressure in the H-node. When using this H-node, there should be another component or H-node which defines the temperature.

The initial pressure is only applied during steady state. If during transient an “isolated network” occurs the pressure and temperature of the previous time step will be used.

Note: By using the node with initial pressure, the mass flow at the node is no longer set to zero. This can thus result in a non-zero mass flow from the node. A warning is issued when this occurs.

Input properties

Description	Input	SI-units	Remarks
Elevation	real	[m]	
Pressure t=0	real	[N/m ²]	

Messages

Message	Message	Explanation
Cavitation in steady state not allowed	error	The lowest pressure in the Heat node is less than the vapour pressure
Pressure < Pvapour; Cavitation not supported for H-node without PIPE connections	warning	In this Heat node there is no PIPE component connected. The cavitation model needs at least 1 PIPE to compute the vapour volume. This warning is only given the first time the pressure condition is true; the rest of the simulation is physically invalid.
Cavitates	informative	Heat node starts cavitating; the lowest pressure drops below the vapour pressure
Cavitation collapses	informative	Void fraction becomes 0
Mass continuity not observed	Warning	There is a mass flow from the node
Change Type to Heat node with initial temperature	error	Heat node is either part of an isolated part of the network in which no information is available for the temperature or there is no flow in the connected components; the user must define the temperature in this part of the model.

6.2.5 Heat node init T

As with the “Heat node init pT” above, the “Heat node init T” can be used in an isolated part of the model to initiate the temperature in the Heat node. When using this Heat node, there should be another component or Heat node which defines the pressure.

The initial temperature is only applied during steady state and only if there are no connected components which already specify the temperature. In transient the value of the former time step is used.

Input properties

Description	Input	SI-units	Remarks
Elevation	real	[m]	
Temperature t = 0 [s]	real	[°C]	

Messages

Message	Message	Explanation
Cavitation in steady state not allowed	error	The lowest pressure in the H-node is less than the vapour pressure
Pressure < Pvapour; Cavitation not supported for H-node without PIPE connections	warning	In this H-node there is no PIPE component connected. The cavitation model needs at least 1 PIPE to compute the vapour volume. This warning is only given the first time the pressure condition is true; the rest of the simulation is physically invalid.
Cavitates	informative	H-node starts cavitating; the lowest pressure drops below the vapour pressure
Cavitation collapses	informative	Void fraction becomes 0
Change Type to Heat node with initial pressure	error	Heat node is part of an isolated part of the network in which no information is available for the pressure; the user must define the pressure in this part of the model.

6.2.6 Heat node with demand

This Heat node will subtract a demand volume flow from the network. This Heat node is very useful for modelling drinking water distribution networks over a longer period of time and the application is comparable to the EPANET approach.

With this heat node the volumetric base demand can be specified as the discharge to end-users connected to this node. The pattern can be used to vary the actual (instantaneous) discharge over time (e.g. high peaks during morning and evening but low at night). The pattern will be repeated if the simulation time is larger than the table period.

Input properties

Description	Input	SI-units	Remarks
Elevation	real	[m]	
Base demand	real	[m ³ /s]	
Pattern	None Table		
Pattern table	table		If Pattern = Table Multiplication factor applied to base pattern

Specific output properties

Node specific output for this Heat node is:

- Demand [m³/s]
- Volume delivered [m³]

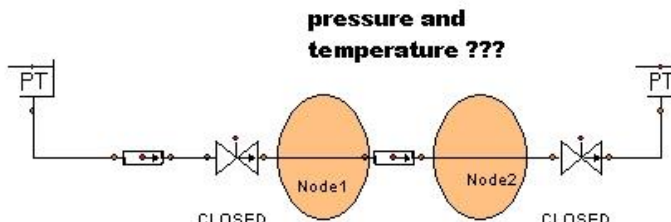
Messages

Message	Message	Explanation
Cavitation not allowed	Error	Cavitation is not possible for the Heat node with demand.

6.2.7 Heat node conditional init pT

If the Heat node is part of a “closed” or ”isolated” hydraulic network, there is no connection with a boundary condition like a BOUNDPT, then the actual pressure and temperature in the steady state computation cannot be determined.

This is explained in the figure below: the pressure and temperature in the nodes Node1 and Node2 cannot be calculated.



It is therefore necessary to define the pressure and temperature for this area in one of the Heat nodes. It is up to the user which Heat node the pressure will be prescribed. The type of the node can be changed via the property window (drop down list in “Type” field). However when using for example the parameter script it can be useful that the node only describes the pressure and temperature when they are part of an isolated system. For this purpose the conditional init PT node can be used. This node only pre-describes its pressure or temperature when it is part of an isolated system.

Note: the initial pressure and temperature are only applied during steady state. If during transient an “isolated network” occurs the pressure and temperature of the previous time step will be used.

Note: By using the node with initial pressure, the mass flow at the node is no longer set to zero. This can thus result in a non-zero mass flow from the node. A warning is issued when this occurs.

Input properties

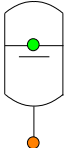
Description	Input	SI-units	Remarks
Elevation	real	[m]	
Pressure t=0	real	[N/m ²]	
Temperature t=0	real	[°C]	

Messages

Message	Message	Explanation
Cavitation in steady state not allowed	error	The lowest pressure in the Heat node is less than the vapour pressure
Pressure < P _{vapour} ; Cavitation not supported for Heat node without PIPE connections	warning	In this Heat-node there is no PIPE component connected. The cavitation model needs at least 1 PIPE to compute the vapour volume. This warning is only given the first time the pressure condition is true; the rest of the simulation is physically invalid.
Cavitates	informative	Heat node starts cavitating; the lowest pressure drops below the vapour pressure
Cavitation collapses	informative	Void fraction becomes 0
Mass continuity not observed	Warning	There is a mass flow from the node

6.3 Heat air vessel

Supplier type

Type label	Symbol	Description	Active
Heat Airvessel		Vertically positioned, prismatic (constant area), non-vented, isolated air vessel	No

6.3.1 Mathematical model

A vertical air vessel or air chamber without air inlet can be modelled by four equations. The first equation describes the behaviour of the enclosed air:

$$PV^k = C \quad (1)$$

in which:

P	=	absolute air pressure	[N/m ²]
V	=	air volume	[m ³]
k	=	Laplace coefficient	[-]
C	=	constant	[Nm] if k = 1

The Laplace coefficient depends on the thermodynamic behaviour of the air. Isothermal expansion is described by $k = 1$. Adiabatic expansion is described by $k = 1.4$.

The second equation governs the discharge Q (defined positive, when the flow is from the air vessel into the system) :

$$Q = -A_{av} \frac{dx_f}{dt} \quad (2)$$

in which A_{av} denotes the storage surface and x_f the fluid level in the air chamber with respect to the global co-ordinate system. The minus sign indicates that the chamber supplies fluid to the system when the fluid level is decreasing in time.

In steady state the air vessel does not supply fluid. Hence in steady state the governing equation is:

$$Q = 0 \quad (3)$$

The last equation (4) defines the temperature of the water in the air vessel. This air vessel model assumes that the vessel is thermally isolated, i.e. no heat is transferred to or from the outside world. The water in the air vessel is assumed to be mixed instantaneously (ideally mixed tank). Using an energy balance for the air vessel, the change of temperature can be expressed as a function of the energy inflow:

$$\begin{aligned} \dot{m} < 0 &\Rightarrow \frac{dV \rho c_p T}{dt} = \dot{m} c_p T_{in} \\ \dot{m} \geq 0 &\Rightarrow \frac{dT}{dt} = 0 \end{aligned} \quad (4)$$

During outflow the temperature in the air vessel does not change, the outflow temperature is taken equal to the temperature in the air vessel.

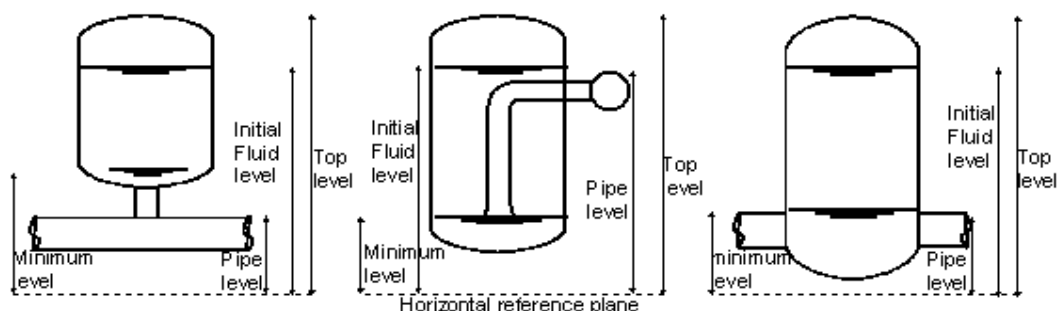
6.3.2 Airvessel properties

6.3.2.1 Hydraulic specifications

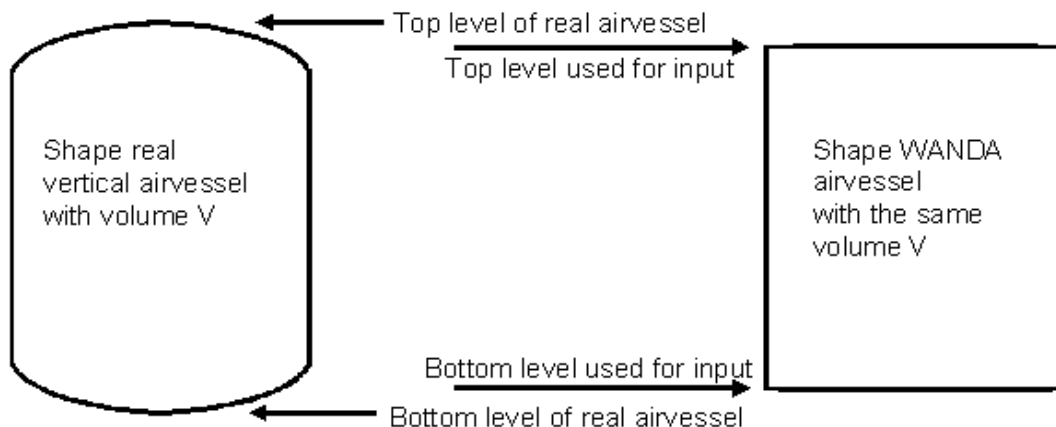
Description	Input	SI-units	Remarks
Bottom level	real	[m]	See remarks
Top level	real	[m]	See remarks
Air quantity by	Fluid Level Air volume Constant C		See remarks
Initial fluid level	real	[m]	If "air quantity by" = Fluid Level
Initial air volume	real	[m ³]	If "air quantity by" = Air volume
Initial C in P*V = C	real	[J]	If "air quantity by" = Constant C
Chamber area	real	[m ²]	
Laplace coefficient	real	[-]	
Temperature at t = 0 [s]	real	[°C]	

Remarks

The air volume and the fluid level in the airvessel are determined by either the initial fluid level, the initial air volume or the initial constant C. One quantity has to be given by the user, the other two quantities are calculated by Wanda.



An airvessel in WANDA is modelled with a straight instead of a rounded top and bottom. The volume of the real airvessel and the WANDA airvessel should be the same. The top and bottom level should therefore be taken respectively lower and higher than the level of the real airvessel. How much lower and higher depends on the shape of the real airvessel.



Meaning of Pressure in property window Airvessel:

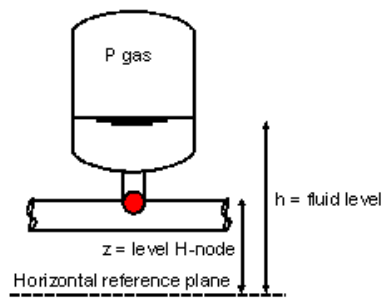
Head = Head in node of airvessel

Pressure P (in node) = $\rho \cdot g \cdot (H-z)$ (relative to atmosphere)

z = height specified by particular node

Air pressure (absolute) = $\rho \cdot g \cdot (H-h) + P_{atm}$

h = fluid level



6.3.2.2 Component specific output

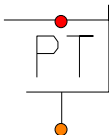
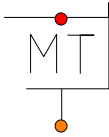
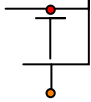
- Fluid level [m]
- Air pressure (absolute) [N/m².abs]
- Air volume [m³]
- Tank temperature [°C]

6.3.2.3 Component messages

Message	Type	Explanation
Top level below bottom level	Error	This is in input error by the user
Initial fluid level not in between top and bottom level of air chamber	Error	This is in input error by the user
Specified air volume larger than vessel volume	Error	This is in input error by the user
Empty airchamber	Warning	Computation can continue, since storage area is constant independent of the height. The results will not be physically correct, but may help the design of the air vessel.
Airchamber refilled	Info	There has been a time with completely emptied air chamber, above.

6.4 Heat boundary

Supplier type

Type label	Symbol	Description	Active
Heat BoundPT		Defines pressure and temperature (constant or time varying)	Yes
Heat BoundMT		Defines mass flow and temperature (constant or time varying)	Yes
Heat BoundT(PM)		Defines temperature and pressure or mass flow (constant or time varying)	Yes

Without boundary conditions applied at the extremities of the pipeline system, the model is undefined and no (valid) solution can be found. In a well defined model enough core variables (like pressure, temperature and mass flow) must be defined to find a solution for the entire system.

Only one variable can be activated via the control lines (or action table). The type name of the component indicates which variable can be controlled by the action table or control signal line and which variable can be controlled by a secondary time-table: the name consists of three parts:

Bound + controlled variable + time-table variable.

6.4.1 Mathematical model

Each boundary condition in WANDA Heat prescribes a combination of mass flow or pressure and temperature. The variables are prescribed using an (action) table or control signal line. Each boundary condition therefore contains one of the two following functions:

$$\begin{aligned} \dot{m} &= f(t) \\ p &= f(t) \end{aligned} \tag{1}$$

Next to this the following equation is used:

$$T = f(t) \tag{2}$$

Sign convention:

- positive flow is into the system
- negative flow is out of the system.

Reservoirs with a positive flow also pre-describe their temperature for the system.

6.4.2 Heat BoundPT

Pressure-temperature boundary

The Heat BoundPT prescribes the pressure and temperature in a certain point of the system: the pressure can be changed with an action table or control signal line, the temperature can be changed with a secondary time-table of the component.

6.4.2.1 Hydraulic specifications

Description	Input	SI-units	Remarks
Pressure at t = 0 [s]	real	[N/m ²]	
Temperature	Constant time varying		
Constant temperature	real	[°C]	Only if Temperature = constant
Temperature time table	table		Only if Temperature = time varying

6.4.2.2 Actions

In unsteady state both the pressure and the temperature can vary in time. The time variation of the pressure can be defined via a specified table or can be set through control components. An example action table is given below:

BoundPT Action table

Time [s]	Pressure [kPa]
0.0	10.0
0.99	10.0
1.0	20.0
1.99	20.0
2.00	0.0
2.99	0.0
3.00	20.0
10.00	25.0

In this example, the pressure remains at the steady state value until 0.99 s. At t = 1.0 s the pressure suddenly rises to 20.0 kPa and keeps that value until 1.99 s. At t = 2.00 s the pressure falls stepwise to 0 kPa, etc.

The temperature variation must be specified via a user defined table. Linear interpolation will be performed between two values in the table. No extrapolation will be done; the last function value will be held if the actual time is outside the last table value.

6.4.3 Heat BoundMT

Mass flow-temperature boundary

The Heat BoundMT prescribes the mass flow and temperature in a certain point of the system: the mass flow can be changed with an action table or control signal line, the temperature can be changed with a secondary time table of the component.

Positive mass flows represent flow into the system. Negative flow represents flow from the system

6.4.3.1 Hydraulic specifications

Description	Input	SI-units	Remarks
Mass flow at t= 0 [s]	real	[kg/s]	
Temperature	Constant Time varying		
Constant temperature	real	[°C]	If Temperature = Constant
Temperature time table	table		If Temperature = time varying

6.4.3.2 Actions

In unsteady state the mass flow can vary in time. The time variation must be defined in the action table or through a control signal line. An example is given below:

BoundMT Action table

Time [s]	Mass flow [kg/s]
0.0	0.0
1.00	200.0
10.00	20.0

In the example the mass flow is zero at steady state. It rises linearly in 1 s to 200 kg/s. After 1 s the mass flow decreases linearly in 9 s to 20 kg/s.

The temperature variation must be specified via a user defined table. Linear interpolation will be performed between two values in the table. No extrapolation will be done; the last function value will be held if the actual time is outside the last table value.

6.4.4 Heat BoundT(PM)

Temperature (mass flow/pressure) boundary condition

The Heat BoundT(PM) prescribes the temperature and either the pressure or mass flow in a certain point of the system: Temperature can be changed with action table or control signal line pressure or mass flow can be changed with a secondary time table of the component.

Positive mass flows represent flow into the system, negative mass flows represent flow from the system.

6.4.4.1 Hydraulic specifications

Description	Input	SI-units	Remarks
Temperature at t = 0 [s]	real	[°C]	
Prescribed	pressure mass flow		
Pressure	constant time varying		Only if Prescribed = pressure
Constant pressure	real	[N/m ²]	If Pressure = constant
Pressure time table	table		If Pressure = time varying
Mass flow	constant time varying		Only if Prescribed = mass flow
Constant mass flow	real	[kg/s]	If Mass flow = constant
Mass flow time table	table		If Mass flow = time varying

6.4.4.2 Actions

In unsteady state the temperature can vary in time. The time variation must be defined in the action table or through a Control signal line. An example table is given below:

BOUNT(MP) Action table

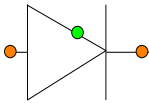
Time [s]	Temperature [°C]
0.0	0.0
1.00	20.0
10.00	15.0

In the example the temperature is zero at steady state. It rises linearly in 1 s to 20 °C. After 1 s the flow decreases linearly in 9 s to 15 °C.

The pressure or mass flow variation must be specified via a user defined table. Linear interpolation will be performed between two values in the table. No extrapolation will be done; the last function value will be held if the actual time is outside the last table value.

6.5 Check valve / Non-return valve

Fall type

Type label	Symbol	Description	Active
Heat Check valve		Ideal check valve; closes if $Q < 0$, opens if $p_1 > p_2 + \Delta p_{\text{reopen}}$	No

6.5.1 Mathematical model

Ideal check valve

An open check valve is modelled as a resistance by the following equations:

$$\text{Pressure difference: } \Delta p = p_1 - p_2 = \frac{\xi}{2\rho} \frac{\dot{m}|\dot{m}|}{A^2} + g\rho(y_2 - y_1) \quad (1)$$

$$\text{Continuity: } \dot{m}_1 = \dot{m}_2 \quad (2)$$

$$\text{Temperature difference: } \dot{m}c_{p2}T_2 = \dot{m}c_{p1}T_1 + fr \cdot Q_{gen} \quad (3)$$

In which:

p_1 / p_2	=	pressure at connection point	[Pa]
ρ	=	density at upstream connection point	[kg/m ³]
y_1 / y_2	=	Heat node (elevation) heights	[m]
ξ	=	valve loss coefficient	[-]
\dot{m}	=	mass flow	[kg/s]
A	=	valve inlet area	[m ²]
c_1 / c_2	=	specific heat of the fluid	[J/kg/K]
fr	=	fraction of generated heat that is transferred to the fluid	[-]

The discharge area of the check valve may differ from the cross sectional area of the adjacent pipes. The heat generated in the check valve (due to friction) is partially absorbed by the fluid. The extend to which this generated heat is absorbed by the fluid can be set with the 'fraction of generated heat that is transferred to the fluid'. The rest of the heat is lost. The generated heat is given by:

$$Q_{gen} = \xi \frac{\dot{m}^3}{2A^2\rho^2}$$

Next, the criteria for opening and closing must be defined.

Closure criteria

An ideal check valve closes when:

$$Q_1 < 0 \quad (4)$$

Opening criteria

A closed valve (re)opens when the pressure difference across the valve exceeds a specified value (the default is 0):

$$p_1 - p_2 > \Delta p_{reopen} \quad (5)$$

6.5.2 Heat Check valve properties

The check valve is modelled as an **ideal** check valve, which closes without reverse flow. However, in practice a check valve does not behave ideally, but closes after flow reversal. The closure is accompanied with a certain amount of reverse flow and pressure surges, depending on the valve type.

Alternatively, the closure behaviour of check valves may be modelled with the component **VALVE**. The valve position is then prescribed as function of (closure) time. In the case of damped check valves the valve may be closed in two stages: A stage of passive and a stage of active damping.

6.5.2.1 Hydraulic specifications

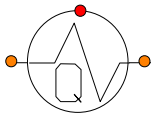
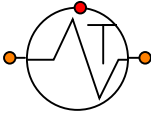
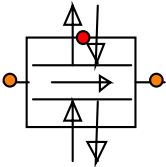
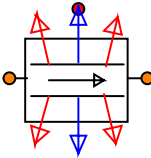

Description	Input	SI-units	remarks
Inner diameter	real	[m]	
Loss coefficient	real	[-]	
Delta_p for reopen	real	[N/m ²]	
Initial state	Open Closed		Default = open
Fraction gen. heat to fluid	real	[-]	Default = 0

6.5.2.2 Component messages

Message	Type	Explanation
Starts in closed phase	Info	
Starts in open phase	Info	
Closes	Info	
Opens	Info	

6.6 Heat exchanger

Fall type

Type label	Symbol	Description	Active
Heat supply QHeat in		Supply heat to or extract heat from the fluid by directly specifying a heat input	Yes
Heat supply Tdown		Supply heat or extract heat to the fluid by specifying a temperature at downstream Heat node	Yes
Heat supply with limits		Supply heat to or extract heat from the fluid by directly specifying a heat input, with user specified limits for the minimum and maximum temperatures	Yes
Heat exchanger		Exchange heat with surroundings due to temperature difference between fluid and ambient temperature.	Yes
Heat demand		Heat demand for district heating systems with different time scales for heating and warm (tap) water.	Yes
Gas boiler		Supply heat to the fluid by gas combustion	Yes
Solar collector		Heat supply via a solar collector	Yes

6.6.1 Heat supply QHeat in

The heat supply is used to supply a constant heat flux to the system. The user sets the amount of heat supplied (+) or extracted (-). Wanda calculates how much the fluid heats or cools down as a result. Next to this, the hydraulic losses are also included via a loss coefficient. This hydraulic loss can be converted to heat by the fraction-generated heat to fluid.

6.6.1.1 Mathematical model

The head loss (definition based on liquid domain) is given by:

$$\Delta H = CQ^2$$

In which:

ΔH	=	the head loss	[m]
C	=	the loss coefficient	[s ² /m ⁵]
Q	=	the flow rate	[m ³ /s]

This can be rewritten in terms of a pressure drop and mass flow as follows:

$$\Delta p = \rho g \left(C \frac{\dot{m}^2}{\rho^2} - \Delta z \right)$$

In which

Δp	=	the pressure drop	[N/m ²]
ρ	=	the density at the upstream side	[kg/m ³]
g	=	the gravitational acceleration	[m/s ²]
\dot{m}	=	mass flow	[kg/s]
Δz	=	the height difference between the two connection points	[m]

The temperature change is computed from:

$$\dot{m}(c_{p1}T_1 - c_{p2}T_2) + Q_{input} + fr \cdot Q_{gen} = 0$$

In which:

c_{p1} / c_{p2}	=	specific heat of the fluid	[J/kg/K]
T_1 / T_2	=	upstream / downstream Temperature	[K]
Q_{input}	=	heat supplied to the system	[W]
fr	=	fraction of generated heat that is transferred to the fluid	[-]
Q_{gen}	=	generated heat flux	[W]

The generate heat flux is given by:

$$Q_{gen} = Cg \frac{\dot{m}^3}{\rho^2}$$

The heat input is always supplied and therefore zero flow is not possible, since this would result in an infinite temperature. The heat supplied is calculated as:

$$Q_s = Q_{input} + fr \cdot Q_{gen}$$

6.6.1.2 Hydraulic specifications

Description	Input	SI-units	remarks
C-value (dH=CQ ²)	real	[s ² /m ⁵]	Q is fluid flow rate
Fraction gen. heat to fluid	real	[-]	Default = 0
Initial heat supply	real	[W]	Can be controlled by action table

6.6.1.3 Component specific output

- Heat supplied [W]
- Generated Heat flux [W]

6.6.1.4 Component messages

Message	Type	Explanation
Zero flow not allowed	Error	Heat input with zero flow not possible, because that would require the temperature to become infinite.

6.6.1.5 Actions

The action table (control connection) of this component controls the heat supply as function of time.

6.6.2 Heat supply Tdown

The heat supply Tdown is used to specify a fixed downstream temperature (constant or time-variable through action table). The amount of heat supplied (+) to or heat extracted (-) from the fluid is calculated by Wanda. The location of the downstream side is based on the actual flow direction (thus not based on the scheme definition).

6.6.2.1 Mathematical model

The head loss (definition based on liquid domain) is given by:

$$\Delta H = CQ^2$$

In which:

ΔH	=	the head loss	[m]
C	=	the loss coefficient	[s ² /m ⁵]
Q	=	the flow rate	[m ³ /s]

This can be rewritten in terms of a pressure drop and mass flow as follows:

$$\Delta p = \rho g \left(C \frac{\dot{m}^2}{\rho^2} - \Delta z \right)$$

In which

Δp	=	the pressure drop	[N/m ²]
ρ	=	the density at the upstream side	[kg/m ³]
g	=	the gravitational acceleration	[m/s ²]
\dot{m}	=	mass flow	[kg/s]
Δz	=	the height difference between the two connection points	[m]

The downstream temperature is set based on the user specified input. A change in flow direction will set the specified temperature at the opposite connection point.

The heat supplied is calculated from:

$$Q_s = \dot{m} c_{p1} T_1 - \dot{m} c_{p2} T_2$$

Q_s	=	Heat supplied	[W]
c_{p1} / c_{p2}	=	specific heat of the fluid	[J/kg/K]
T_1 / T_2	=	upstream / downstream Temperature	[K]

The generate heat flux is given by:

$$Q_{gen} = Cg \frac{\dot{m}^3}{\rho^2}$$

6.6.2.2 Hydraulic specifications

Description	Input	SI-units	remarks
C-value ($dH=CQ^2$)	real	[s ² /m ⁵]	Q is fluid flow rate
Fraction gen. heat to fluid	real	[-]	Default = 0
Initial downstr temp	real	[°C]	Can be controlled by action table

6.6.2.3 Component specific output

- Heat supplied [W]
- Generated Heat flux [W]

6.6.2.4 Component messages

None

6.6.2.5 Actions

The action table (control connection) of this component controls the downstream temperature as function of time.

6.6.3 Heat supply QHeat in limit on T

The heat supply is used to supply a constant heat flux to the system. The user sets the amount of heat supplied (+) or extracted (-). Wanda calculates how much the fluid heats or cools down as a result, the downstream temperature is limited by user specified boundaries. The hydraulic losses are included via a loss coefficient. This hydraulic loss can be converted to heat by the fraction-generated heat to fluid.

6.6.3.1 Mathematical model

The head loss (definition based on liquid domain) is given by:

$$\Delta H = CQ^2$$

In which:

ΔH	=	the head loss	[m]
C	=	the loss coefficient	[s ² /m ⁵]
Q	=	the flow rate	[m ³ /s]

This can be rewritten in terms of a pressure drop and mass flow as follows:

$$\Delta p = \rho g \left(C \frac{\dot{m}^2}{\rho^2} - \Delta z \right)$$

In which

Δp	=	the pressure drop	[N/m ²]
ρ	=	the density at the upstream side	[kg/m ³]
g	=	the gravitational acceleration	[m/s ²]
\dot{m}	=	mass flow	[kg/s]
Δz	=	the height difference between the two connection points	[m]

The temperature change is computed from:

$$\dot{m}(c_{p1}T_1 - c_{p2}T_2) + Q_{input} + fr \cdot Q_{gen} = 0$$

In which:

c_{p1} / c_{p2}	=	specific heat of the fluid	[J/kg/K]
T_1 / T_2	=	upstream / downstream Temperature	[K]
Q_{input}	=	heat supplied to the system	[W]
fr	=	fraction of generated heat that is transferred to the fluid	[-]

$$Q_{gen} = \text{generated heat flux} \quad [W]$$

If the downstream temperature comes below or above the user specified extremes the temperature is set to this value, the user is informed by a message. The actual used heat flux is given as result. The generate heat flux is given by:

$$Q_{gen} = Cg \frac{\dot{m}^3}{\rho^2}$$

The heat supplied is calculated as:

$$Q_s = Q_{input} + fr \cdot Q_{gen}$$

6.6.3.2 Hydraulic specifications

Description	Input	SI-units	remarks
C-value (dH=CQ^2)	real	[s2/m5]	Q is fluid flow rate
Fraction gen. heat to fluid	real	[-]	Default = 0
Minimum supply temperature	Real	[°C]	
Maximum supply temperature	Real	[°C]	
Initial heat supply	real	[W]	Can be controlled by action table

6.6.3.3 Component specific output

- Heat supplied [W]
- Generated Heat flux [W]

6.6.3.4 Component messages

Message	Type	Explanation
Temperature set to upper bound	Info	This message is given once when the heat temperature is set to the upper bound
Temperature set to lower bound	info	This message is given once when the heat temperature is set to the lower bound
Temperature within bounds	info	This message is given when the temperature is again within bounds

6.6.3.5 Actions

The action table (control connection) of this component controls the heat supply as function of time.

6.6.4 Heat exchanger

The heat exchanger component calculates in the transient state the heat transfer depending on the temperature difference between the ambient temperature and the fluid temperature using an integrated heat transfer coefficient. Four different initial states are possible:

- Heat exchanger
- DownT and C- coef
- DownT and Q supplied
- DeltaT and Q supplied.

These are detailed in the Mathematical model.

6.6.4.1 Mathematical model

The head loss is given by:

$$\Delta H = CQ^2$$

In which:

ΔH	=	the head loss	[m]
C	=	the loss coefficient	[s ² /m ⁵]
Q	=	the flow rate	[m ³ /s]

This can be rewritten in terms of a pressure drop and mass flow as follows:

$$\Delta p = \rho g \left(C \frac{\dot{m}^2}{\rho^2} - \Delta z \right)$$

In which

Δp	=	the pressure drop	[N/m ²]
ρ	=	the density at the upstream side	[kg/m ³]
g	=	the gravitational acceleration	[m/s ²]
\dot{m}	=	mass flow	[kg/s]
Δz	=	the height difference between the two connection points	[m]

The heat flow to (+) or from (-) the fluid is computed from:

$$Q_{input} = h(T_{\infty} - T_f)$$

In which:

h	=	the heat transfer coefficient	[W/K]
T_{∞}	=	is the temperature external temperature	[K]
T_f	=	is the temperature at the downstream connection point	[K]

The energy equation becomes then:

$$\dot{m}(c_{p1}T_1 - c_{p2}T_2) + h(T_{\infty} - T_f) + fr \cdot Q_{gen} = 0$$

In which:

c_{p1} / c_{p2}	=	specific heat of the fluid	[J/kg/K]
T_1 / T_2	=	upstream / downstream temperature	[K]
fr	=	fraction of generated heat that is transferred to the fluid	[-]
Q_{gen}	=	generated heat flux	[W]

The generate heat flux is given by:

$$Q_{gen} = Cg \frac{\dot{m}^3}{\rho^2}$$

The heat supplied is calculated as:

$$Q_s = Q_{input} + fr \cdot Q_{gen}$$

Several options are available for the steady state computation (initial state). This setting defines how the steady state is calculated. Based on this initial state a constant heat transfer coefficient and/or a loss coefficient are calculated which is used throughout the transient computation.

- Heat Exchanger
The initial state is calculated based on a heat transfer coefficient input by the user.
- DownT and C-coeff
Initial state is based on downstream temperature and a C-value. The downstream side is based on the initial flow direction and may differ from the scheme definition. The heat transfer coefficient is calculated as a result.
- DownT and Qhsup
Initial state is based on downstream temperature and a fixed heat supply. The downstream side is based on the initial flow direction and may differ from the scheme definition. The C-value and heat transfer coefficients are calculated as result of this initial steady state.
- DT and Qhsup
Initial state is based on a temperature difference (T1-T2) over the component and a fixed heat supply. The C-value and heat transfer coefficient are calculated as result of this initial steady state.

6.6.4.2 Hydraulic specifications

Description	Input	SI-units	remarks
Initial state	Heat exchange DownT and C-coef DownT and Qhsup DT and Qhsup		Only used for initial (steady state) calculation
C-value (dH=CQ ²)	real	[s ² /m ⁵]	Only if Initial state is: "Heat exchange" or "DownT and C-coef"
Fraction gen. heat to fluid	real	[-]	Default = 0
Heat transfer coefficient	real	[W/K]	Only if Initial state is "Heat exchange"
Initial heat supply	real	[kW]	Only if Initial state is

				“DownT and Qhsup” or “DT and Qhsup”
Initial downstr temp	real	[°C]		Only if Initial state is “DownT and C-coef” or “DownT and Qhsup”
Initial delta T	real	[°C]		Only if Initial state is “DT and Qhsup”
Initial ambient temperature	real	[°C]		Can be controlled by action table

6.6.4.3 Component specific output

- Heat supplied [W]
- Generated Heat flux [W]
- Ambient temperature [°C]

6.6.4.4 Component messages

Message	Type	Explanation
Unable to determine resistance and heat transfer coefficient: zero flow	Error	To calculate the C-value the flow needs to be non-zero
No heat transfer: outside temperature equals inside temperature	Error	The heat transfer coefficient cannot be determined when the ambient temperature equals the fluid temperature.
Unable to determine resistance: zero head difference	Error	To calculate the C-value the head difference needs to be non-zero
Negative hydraulic loss coefficient	Warning	Negative hydraulic loss adds energy to the system, which is probably not intended by the user.
Heat supply and delta T should have opposite signs	Error	If this is not the case the heat flow would be in a physical impossible direction

6.6.4.5 Actions

The action table (control connection) of this component controls the ambient temperature as function of time.

6.6.5 Heat demand

This component can be used to model the heat demand of a number of users in a district heating system. The heat demand consists of 2 parts: space heating and tap water heating.

6.6.5.1 Mathematical model

The head loss is given by:

$$\Delta H = CQ^2$$

In which:

ΔH	=	the head loss	[m]
C	=	the loss coefficient	[s ² /m ⁵]
Q	=	the flow rate	[m ³ /s]

This can be rewritten in terms of a pressure drop and mass flow as follows:

$$\Delta p = \rho g \left(C \frac{\dot{m}^2}{\rho^2} - \Delta z \right)$$

In which

Δp	=	the pressure drop	[N/m ²]
ρ	=	the density at the upstream side	[kg/m ³]
g	=	the gravitational acceleration	[m/s ²]
\dot{m}	=	mass flow	[kg/s]
Δz	=	the height difference between the two connection points	[m]

The temperature change is computed from:

$$\dot{m}(c_{p1}T_1 - c_{p2}T_2) - Q_{demand} + fr \cdot Q_{gen} = 0$$

In which:

c_{p1} / c_{p2}	=	specific heat of the fluid	[J/kg/K]
T_1 / T_2	=	upstream / downstream Temperature	[K]
Q_{demand}	=	heat demanded from the system	[W]
fr	=	fraction of generated heat that is transferred to the fluid	[-]
Q_{gen}	=	generated heat flux	[W]

The heat demand is calculated from:

$$Q_{demand} = Q_{heat} + \rho_f c_{pf} (T_{hot} - T_{cold}) Q_{hot}$$

In which:

Q_{heat}	=	heat demand as specified by the user	[W]
C_{pf}	=	average specific heat based on the temperature specified by the user	[J/kg/K]
ρ_f	=	average density of the tap water based on the temperature specified by the user	[kg/m ³]
T_{hot}	=	hot water temperature	[K]
T_{cold}	=	cold water temperature	[K]
Q_{hot}	=	hot water flow rate demand	[m ³ /s]

The generate heat flux is given by:

$$Q_{\text{gen}} = Cg \frac{\dot{m}^3}{\rho^2}$$

The total heat demanded is calculated as:

$$Q_d = Q_{\text{demand}} - fr \cdot Q_{\text{gen}}$$

Please note that this component always extracts the heat independent on the hot and cold water temperature. This also result in that zero flow is not possible.as this would result in an infinite temperature.

6.6.5.2 Hydraulic specifications

Description	Input	SI-units	Remarks
C-value (dH=CQ ²)	real	[-]	
Fraction gen. heat to fluid	real	[-]	Default = 0
Heat demand	Constant Time varying		
Constant heat demand	real	[W]	If Heat demand = Constant
Heat demand time table	table		If Heat demand = Time varying
Cold water temperature	real	[°C]	Input temperature of the secondary fluid (to be heated)
Hot water temperature	real	[°C]	Output temperature of the secondary fluid (after heating)
Hot water demand	real	[m ³ /s]	Controlled by action table

6.6.5.3 Component specific output

- Total heat demanded [W]
- Generated heat flux [W]

6.6.5.4 Component messages

Message	Type	Explanation
Zero flow not allowed	Error	Heat input with zero flow not possible, because that would require the temperature to become infinite.

6.6.5.5 Actions

The action table (control connection) of this component controls the hot water demand in [m³/s] as function of time.

6.6.6 Gas boiler

The heat supply is used to supply a constant heat flux to the system. The user sets the amount of heat supplied (+). Wanda calculates how much the fluid heats or cools down as a result. Next to this, the hydraulic losses are also included via a loss coefficient. This hydraulic loss can be converted to heat by the fraction-generated heat to fluid.

6.6.6.1 Mathematical model

The head loss is given by:

$$\Delta H = CQ^2$$

In which:

ΔH	=	the head loss	[m]
C	=	the loss coefficient	[s ² /m ⁵]
Q	=	the flow rate	[m ³ /s]

This can be rewritten in terms of a pressure drop and mass flow as follows:

$$\Delta P = \rho g \left(C \frac{\dot{m}^2}{\rho^2} - \Delta z \right)$$

In which

Δp	=	the pressure drop	[Pa]
ρ	=	the density at the upstream side	[kg/m ³]
g	=	the gravitational acceleration	[m/s ²]
\dot{m}	=	mass flow	[kg/s]
Δz	=	the height difference between the two connection points	[m]

The temperature change is computed from:

$$\dot{m}(c_{p1}T_1 - c_{p2}T_2) + Q_{input} + fr \cdot Q_{gen} = 0$$

In which:

\dot{m}	=	mass flow	[m ³ /s]
c_{p1} / c_{p2}	=	specific heat of the fluid	[J/kg K]
T_1 / T_2	=	upstream / downstream Temperature	[K]
Q_{input}	=	heat supplied to the system	[W]
fr	=	fraction of generated heat that is transferred to the fluid	[-]
Q_{gen}	=	generated heat flux	[W]

The generated heat flux is given by:

$$Q_{gen} = Cg \frac{\dot{m}^3}{\rho^2}$$

The heat input is always supplied and therefore zero flow is not possible, since this would result in an infinite temperature. The heat supply is calculated as:

$$Q_s = Q_{input} + fr \cdot Q_{gen}$$

The gross primary energy input by the fuel combustion is given by:

$$Q_{primary} = \frac{Q_{input}}{\eta}$$

η = efficiency of boiler [-]

The amount of fuel required is calculated as:

$$V_{fuel} = \frac{Q_{primary}}{c_{fuel} \rho_{fuel}} dt$$

In which:

V_{fuel}	=	Volume of fuel required	[m ³]
c_{fuel}	=	Heat of combustion	[J/kg]
ρ_{fuel}	=	Density of fuel	[kg/m ³]

6.6.6.2 Hydraulic specifications

Description	Input	SI-units	remarks
C-value (dH=CQ ²)	Real	[s ² /m ⁵]	
Fraction gen. heat to fluid	Real	[-]	Default = 0
Initial heat supply	Real	[W]	Can be activated by action table or control
Combustion heat fuel	Real	[J/kg]	
Efficiency boiler	Real	[-]	
Density fuel	Real	[kg/m ³]	

6.6.6.3 Component specific output

- Temperature change [°C]
- Heat supplied [W]
- Generated Heat flux [W]
- Primary energy [W]
- Discharge of fuel used [m³/s]

6.6.6.4 Component messages

Message	Type	Explanation
Zero flow not allowed	Error	Heat input with zero flow not possible, because that would require the temperature to become infinitive.

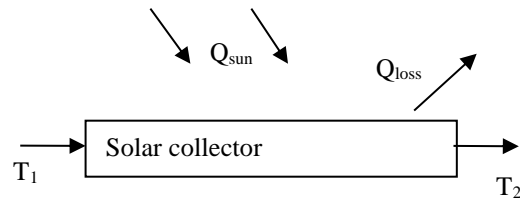
6.6.6.5 H-actions

The action table (control connection) of this component controls the heat supply as function of time.

6.6.7 Solar collector

Heat supply by solar radiation

6.6.7.1 Mathematical model



The solar collector collects energy from the sun that heats the fluid flowing through the solar collector. The hydraulic head loss in the solar collector is calculated from:

$$\Delta H = CQ^2$$

In which:

ΔH	=	the head loss	[m]
C	=	the loss coefficient	[s ² /m ⁵]
Q	=	the flow rate	[m ³ /s]

This can be rewritten in terms of a pressure drop and mass flow as follows:

$$\Delta P = \rho g \left(C \frac{\dot{m}^2}{\rho^2} - \Delta z \right)$$

In which:

Δp	=	the pressure drop	[bar]
ρ	=	the density at the upstream side	[kg/m ³]
g	=	the gravitational acceleration	[m/s ²]
\dot{m}	=	mass flow	[kg/s]
Δz	=	the height difference between the two connection points	[m]

The temperature is calculated from the following energy balance:

$$\dot{m} (c_{p1} T_1 - c_{p2} T_2) + A \phi_{sun} - A \phi_{hloss} + fr \cdot Q_{gen} = 0$$

In which:

c_{p1} / c_{p2}	=	specific heat of the fluid	[J/kg K]
T_1 / T_2	=	upstream / downstream temperature	[K]
A	=	Area of solar collector	[m ²]
ϕ_{sun}	=	Heat flux from the sun	[W/m ²]

ϕ_{hloss}	=	Heat loss flux to the surroundings	[W/m ²]
fr	=	fraction of generated heat that is transferred to the fluid	[-]
Q_{gen}	=	Generated heat flux	[W]

The generated heat flux is given by:

$$Q_{gen} = Cg \frac{\dot{m}^3}{\rho^2}$$

The heat supply is calculated as:

$$Q_s = A\phi_{sun} - A\phi_{hloss} + fr \cdot Q_{gen}$$

The heat loss flux is calculated from:

$$\phi_{hloss} = \alpha_1 \left(\frac{T_1 + T_2}{2} - T_{ambient} \right) + \alpha_2 \left(\frac{T_1 + T_2}{2} - T_{ambient} \right) \left| \frac{T_1 + T_2}{2} - T_{ambient} \right| + e\sigma \left(\frac{T_1 + T_2}{2} \right)^4$$

In which:

α_1	=	Heat loss coefficient 1	[W/m ² /K]
α_2	=	Heat loss coefficient 2	[W/m ² /K ²]
$T_{ambient}$	=	Ambient temperature	[K]
e	=	Emission coefficient	[-]
σ	=	Stefan Boltzmann constant	[W/m ² /K]

6.6.7.2 Hydraulic specifications

description	Input	unit	range	default	remarks
C-value (dH=CQ ²)	Real	[s ² /m ⁵]	0.001-200		
fraction generated heat to fluid	Real	[-]	0-1	0	
Heat loss coefficient 1	Real	[W/m ² /K]			
Heat loss coefficient 2	Real	[W/m ² /K ²]			
Area of the collector	Real	[m ²]			
Initial solar heat flux	Real	[W/m ²]			
Emission coefficient	Real	[-]	0-1	1	
Ambient temperature	Constant, time varying	[-]			

Constant Tamb	Real	[°C]	0-500		Only if Ambient temperature = constant
Tamb time table	Real (Table)	[°C]			Only if Ambient temperature = time varying see remark

Remark

The Tamb time table is a pattern table. When it reaches the last entry it restarts at the beginning. This enables to include a day variation for the temperature.

6.6.7.3 Component specific output

Qsupplied [W]

Qgenerated [W]

6.6.7.4 H-actions


The action table (control connection) of this component controls the solar heat flux as function of time

6.6.7.5 Component messages

Message	Type	Explanation

6.7 Surge tower, surge shaft

Supplier type

Type label	Symbol	Description	Active
Heat Surgetower		Thermally isolated surge tower. Storage area is independent of liquid height (pressure head)	No

6.7.1 Mathematical model

A surge tower or shaft or open tank (atmospheric pressure at fluid level) supplies fluid to the system according to the differential equation:

$$Q = -A_{tw} dH / dt \quad (1)$$

in which:

Q	=	Discharge from the tower to the system	[m ³ /s]
A _{tw}	=	Storage surface of the tower	[m ²]
H	=	fluid level in the tower	[m]

In steady state the surge tower does not supply fluid. In that case the equation is:

$$Q = 0 \quad (2)$$

The temperature in the surge tower is modelled as instantaneously mixed. Therefore, the temperature at the connecting node (Temperature 1) can change instantaneously when the flow will reverse from direction. To correctly calculate the tank temperature when fluid is supplied to the tank the bottom level also needs to be defined. No heat transfer between the fluid and the air outside is taken into account (isolated tank).

6.7.2 Surge tower properties

6.7.2.1 Hydraulic specifications

Description	Input	SI-units	Remarks
Bottom level	real	[m]	
Tank area	real	[m ²]	
Temperature at t=0s	real	[°C]	

6.7.2.2 Component specific output

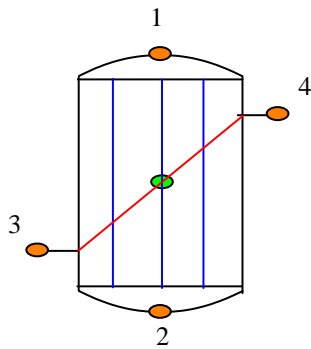
Fluid level [m]

Tank temperature [°C]

6.7.2.3 Component messages

Message	Type	Explanation
Initial fluid level below bottom of surge tower	Warning	The fluid level is below the bottom level, initially.
Surge tower empty	Warning	The fluid level is below the bottom level,

6.8 4-Way Heat exchanger



4-way heat exchanger

Fall type

type label	description	active
Heat exchanger	Heat exchanger with primary and secondary connection points. The fluid on both sides is assumed to be the same.	No

The four-way heat exchanger can be operated in co- and counter current flow. The heat transfer is determined by the temperature difference between the primary and secondary side and the heat transfer coefficient. The head loss is determined by a loss coefficient.

6.8.1 Mathematical model

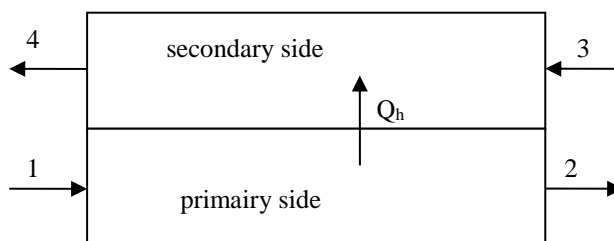


Figure 1 overview of positive flow directions and numbering of connection points

Figure 1 shows an overview of the positive flow direction and heat transfer direction.

The head loss on both the primary and secondary side is given by:

$$\Delta H_{p,s} = C_{h,p,s} Q_{p,s} |Q_{p,s}|$$

In which:

$\Delta H_{p,s}$	=	Head loss over heat exchanger on primary resp. secondary side	[m]
$C_{h,p,s}$	=	Hydraulic loss coefficient of primary res. Secondary side	[s ² /m ⁵]
$Q_{p,s}$	=	Flow rate on primary resp. secondary side	[m ³ /s]

This can be rewritten in terms of pressure and mass flow as follows (subscripts for primary and secondary side have been dropped for convenience):

$$\Delta P = \rho g \left(C \frac{\dot{m}^2}{\rho^2} - \Delta z \right)$$

In which

ΔP	=	Pressure loss over heat exchanger	[Pa]
g	=	gravitational acceleration	[m/s ²]
\dot{m}	=	Mass flow rate	[kg/s]
ρ	=	Density on the upstream side	[kg/m ³]
Δz	=	Height difference between connection points	[m]

For both sides also the following continuity equation apply:

$$\dot{m}_1 = \dot{m}_2$$

$$\dot{m}_3 = \dot{m}_4$$

The heat transfer can be modelled in four different ways, see ref 3:

ε-NTU method

P-NTU method

LMTD method (Log Mean Temperature Difference)

ψ-P method

See ref 3. We have used the ε-NTU method, since it is most practical in use for computer programs. The ε-NTU method uses the effectiveness (ε) of the heat exchangers. This is given by:

$$\dot{\epsilon} = \frac{\dot{Q}}{\dot{Q}_{max}} = \frac{C_{t,Cold} (T_{C,out} - T_{gen} - T_{C,in})}{C_{t,min} (T_{H,in} - T_{C,in})}$$

In which:

\dot{Q}	=	Heat transfer from primary to secondary side	[W]
\dot{Q}_{\max}	=	Maximum heat transfer from primary to secondary side	[W]
$C_{t,\text{cold}}$	=	Flow thermal capacity on the cold side	[J/K/s]
$C_{t,\text{min}}$	=	Minimum Flow thermal capacity from both the hot and cold side	[J/K/s]
$T_{C,\text{out}}$	=	Outflow temperature at the cold side	[K]
$T_{C,\text{in}}$	=	Inflow temperature at the cold side	[K]
$T_{H,\text{in}}$	=	Inflow temperature at the hot side	[K]
T_{gen}	=	Temperature increase generated by frictions	[K]

The cold side is the side with the lowest temperature, this can be either the primary or secondary side

The flow thermal capacity for both the hot and cold side is calculated from:

$$C_t = c_p \dot{m}$$

In which:

$$c_p = \text{Specific heat} \quad [\text{J/kg/K}]$$

The temperature increase generated by friction is calculated from:

$$T_{\text{gen}} = \frac{Q_{\text{gen}}}{c_p}$$

In which:

$$Q_{\text{gen}} = \text{Heat generated by friction} \quad [\text{W}]$$

The generated heat flux is calculated from:

$$Q_{\text{gen}} = C_h g \frac{\dot{m}^3}{\rho^2}$$

For different configurations of heat exchangers, formulas are available to calculate the effectiveness of the heat exchanger. We have implemented the following equation for the heat exchanger in co-current flow:

$$\dot{\phi} = \frac{1 - e^{-NTU*(1+RC)}}{1 + RC}$$

and for counter-current flow

$$\dot{\phi} = \frac{1 - e^{-NTU*(1-RC)}}{1 - RC e^{-NTU*(1-RC)}}$$

In which:

$$NTU = \text{Number of transfer units} \quad [-]$$

RC = Heat capacity ratio [-]

The number of transfer units is given by:

$$NTU = \frac{h}{C_{min}}$$

In which:

h = Overall heat exchange coefficient [W/K]

C_{min} = Minimum thermal capacity from the hot and cold side [W/K]

The heat capacity ratio is given by:

$$RC = \frac{C_{min}}{C_{max}}$$

C_{max} = Maximum thermal capacity from the hot and cold side [W/K]

To be able to calculate the temperatures on all connection points we need one more equation for the temperature which is an energy balance between the primary (1-2) and secondary (3-4) side This is given by:

$$\dot{m}_3 c_{p3} T_3 - \dot{m}_4 c_{p4} T_4 + fr \cdot Q_{gens} + \dot{m}_1 c_{p1} T_1 - \dot{m}_2 c_{p2} T_2 + fr \cdot Q_{genp} = 0$$

In which:

fr = Fraction generated heat to fluid [-]

6.8.2 4-Way heat exchanger

6.8.2.1 Hydraulic specifications

description	Input	unit	range	default	remarks
Loss coefficient primary side	Real	[s ² /m ⁵]	[0-200]		
Loss coefficient secondary side	Real	[s ² /m ⁵]	[0-200]		
Heat transfer coefficient	Real	[W/K]	[0-1e8]		
Correction factor	Real	[-]	[0-1]		
Fraction gen. heat to fluid	Real	[-]	[0-1]	0	

See also "Mathematical model" on page 632.

Remark

6.8.2.2 *Component specific output*

Heat transfer [W]

Generated heat flux [W]

6.8.2.3 *H-actions*

None

6.8.2.4 *Component messages*

Message	Type	Explanation
---------	------	-------------

References

1. Fundamentals of heat and mass transfer Frank Incorpera and David de Witt
Basic heat and mass transfer A.F. Mills
Heat exchanger design handbook, T. Kuppan

6.9 Heat resist

Resist

Fall type

type label	description	active
Heat resist	Resistance according user-defined polynomial	No

6.9.1 Mathematical model

The head loss is given by:

$$\Delta H = a + bQ + cQ^2$$

In which:

ΔH	=	Head loss over resist	[m]
a	=	Constant loss	[m]
b	=	Linear loss coefficient	[m ² /s]
c	=	Quadratic loss coefficient	[m ⁵ /s ²]
Q	=	Flow rate	[m ³ /s]

This can be rewritten in terms of pressure and mass flow rate by:

$$\Delta P = a\rho g + gb\dot{m} + gc\frac{\dot{m}^2}{\rho} - \Delta z\rho g$$

ΔP	=	Pressure loss over resist	[Pa]
ρ	=	Density at upstream connection point	[kg/m ³]
g	=	gravitational acceleration	[m/s ²]
\dot{m}	=	mass flow rate	[kg/s]
Δz	=	height difference over the resist	[m]

For the temperature the following energy balance applies:

$$c_{p1}\dot{m}T_1 - c_{p2}\dot{m}T_2 + fr \cdot Q_{gen} = 0$$

In which:

c_{pi}	=	Heat capacity of the fluid at the i th connection point	[W/kg/K]
T_i	=	Temperature at i th connection point	[kg/m ³]
fr	=	Fraction generate heat to the fluid	[-]
Q_{gen}	=	Generate heat by friction	[W]

The generate heat from friction is calculated from:

$$Q_{gen} = ag\dot{m} + bg \frac{\dot{m}^2}{\rho} + cg \frac{\dot{m}^3}{\rho^2}$$

Note that the coefficient “A” is a constant head loss and is only valid for a positive flow direction. That means that with a negative flow, the head loss contribution “A” (defined as H1-H2) is the same as for positive flow. This is useful for the modeling of osmotic membranes.

6.9.2 Heat Resist

6.9.2.1 Hydraulic specifications

description	Input	unit	range	default	remarks
a in a+bQ+CQ Q	Real	[m]			
b in a+bQ+CQ Q	Real	[s ² /m ²]			
c in a+bQ+CQ Q	Real	[s ⁴ /m ⁵]			
Frac. Gen. heat to fluid	Real	[-]	0-1	0	

See also "Mathematical model" on page 637.

6.9.2.2 Component specific output

Generated heat flux [W]

6.9.2.3 H-actions


None

6.9.2.4 Component messages

Message	Type	Explanation

References

6.10 Pipe

Type label	Symbol	Description	Active
Heat pipe		Pressurised pipe	Yes

6.10.1 Mathematical model

6.10.1.1 Fluid equations

The PIPE can be considered the primary component in WANDA to simulate the pressure wave propagation during transient simulations. In WANDA Heat, the PIPE is also used to simulate the advection and radial transfer of heat in the system.

In steady state $dQ/dt = 0$. The PIPE is considered as a fall type, in which the head loss is modelled by the following equations:

$$(p_2 - p_1) + \rho g (y_2 - y_1) + f \frac{1}{32\pi^2} \frac{L}{D^5} \frac{\dot{m}|\dot{m}|}{\rho} = 0 \quad (1)$$

$$\dot{m}_1 - \dot{m}_2 = 0$$

in which:

$p_1 - p_2$	=	pressure loss from H-node 1 to H-node 2	[N/m ²]
\dot{m}	=	mass flow through the pipe	[kg/s]
f	=	friction factor	[-]
D	=	inner diameter	[m]
$y_2 - y_1$	=	height difference between Heat node1 and Heat node 2	[m]
ρ	=	density	[kg/m ³]
g	=	gravitational acceleration	[m/s ²]
L	=	pipe length	[m]
D	=	inner diameter	[m]

WANDA calculates the friction factor, f , iteratively using the Darcy-Weisbach wall roughness, k . In Transient Mode the friction factor is recalculated each time step using a quasi-steady approach.

The wave propagation speed is calculated based on pipe and fluid properties (analogous to the Liquid pipe with “calculation mode” set to “waterhammer”). Details about the wave propagation and the equations involved are discussed in Theory on waterhammer pipes in Chapter 3: Theoretical and mathematical background. The waterhammer equations are given:.

Momentum:

$$\frac{1}{A_f} \frac{\partial Q}{\partial t} + g \frac{\partial H}{\partial s} + \frac{fQ|Q|}{2DA_f^2} = 0 \quad (2)$$

Continuity:

$$\frac{1}{c_f^2} \frac{\partial H}{\partial t} + \frac{1}{gA_f} \frac{\partial Q}{\partial s} = 0 \quad (3)$$

Note: only a circular cross section is supported in WANDA Heat.

6.10.1.2 Heat equations

The heat transfer in the pipe is included by dividing the pipe in elements, see figure 1. Per element the temperature is calculated based on the following energy balance:

$$\frac{\rho_i c_{pi} VT_i^{t+\Delta t} - \rho_i c_{pi} VT_i^t}{\Delta t} = (c_p \dot{m} T)_{i-1}^t - (c_p \dot{m} T)_i^t - Q_{loss} + fr \left(\frac{\xi}{L} + \frac{f}{D} \right) \frac{\dot{m}_i^2 |\dot{m}_i|}{2\rho_i^2 A^2} ds \quad (4)$$

In which:

ρ	=	Density	[kg/m ³]
c_p	=	Specific heat	[J/kg/K]
V	=	Volume of the observed element	[m ³]
T	=	Temperature	[K]
\dot{m}	=	Mass flow rate	[kg/s]
Q_{loss}	=	Heat loss	[W]
ξ	=	Local loss coefficient	[-]
L	=	Length of the entire pipe	[m]
f	=	Friction coefficient	[-]
A	=	Area of the pipe	[m ²]
fr	=	Fraction friction loss to heat	[-]
ds	=	Element length	[m]

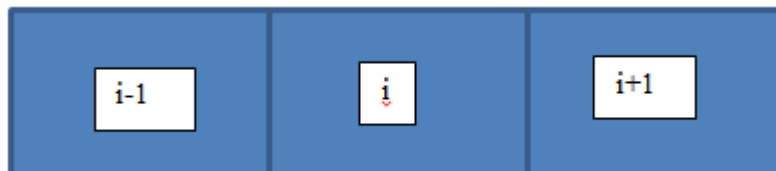


Figure 1, subdivision of pipe in elements for the temperature calculations

The division of the pipes in elements depends upon the type of pipe. For water hammer pipes the the water hammer grid is used. In a rigid column pipe a grid is created based on the advection velocity, which is defined by the user. The size of the grid is calculated from:

$$ds = \frac{L}{n} \quad (5)$$

n = number of internal grid cells [-]

The number of grid cells is calculated from:

$$n = \text{Integer} \left[\frac{L}{v_{adv} dt} \right] \quad (6)$$

v_{adv} = Maximum advection velocity [m/s]

dt = Time step [s]

The maximum advection velocity is set by the user and should be greater than the flow velocity in the pipe for numerical stability. By increasing the time step the number of grid cells can then be reduced. To ensure that steep temperature fronts are correctly conserved a flux limiter scheme is implemented. In Wanda the super bee limiter scheme is implemented see [4].

To increase the stability an additional robust check is built in when a pipe only contains one element. This can occur when there a model consists of long and short pipes. In this case it is checked whether the actual advection velocity (L/dt), which can be lower than the advection velocity specified by the user, is lower than the flow velocity. When this occurs, the entire volume of water in the pipe is refreshed during on time step. If there are no heat losses in this pipe the inflow temperature is equal to the outflow temperature. If there is a heat loss the outflow temperature is calculated with:

$$T_{out} = T_{in} - \frac{Q_{loss} dx}{v} \quad (7)$$

By using this equation only the heat loss is used for the flow which is in the pipe. If this is not done the heat loss would be too high.

The heat loss in equation 4 includes all the heat losses to the surrounding. The heat is transported from the fluid through the pipe wall into the surrounding subsoil. All these effects should be taken into account. In the following section these effects are discussed.

Heat transfer in fluid

The heat transport in the fluid is determined by the flow regime (laminar or turbulent).

For laminar flow the following applies [2] and [3]:

$$\begin{aligned} Nu &= 3.66 \quad \text{if } Gz > 0.1 \\ Nu &= 1.62Gz^{\frac{1}{3}} \quad \text{if } Gz < 0.05 \end{aligned} \quad (8)$$

In which

Nu = Nusselt number [-]

Gz = Graetz number [-]

The Graetz-number is given by:

$$Gz = \frac{a}{Dv} \quad (9)$$

in which

a	=	Thermal diffusion coefficient	[m ² /s]
D	=	Diameter of the pipe	[m]
v	=	Velocity of the fluid	[m/s]

The heat transfer coefficient can be calculated from:

$$h = \frac{Nu\lambda}{D} \quad (10)$$

In which:

h	=	Heat transfer coefficient	[-]
λ	=	Heat conductivity coefficient	[W/m/K]

For turbulent flow, the following formula can be used [5]:

$$Nu = 0.027Re^{0.8}Pr^{0.33} \text{ if } Re > 10^4; Pr \geq 0.7 \quad (11)$$

In which

Pr	=	Prandtl-number	[-]
----	---	----------------	-----

The Prandtl-number is given by:

$$Pr = \frac{\nu}{a} \quad (12)$$

In which

ν	=	Kinematic viscosity	[m ² /s]
---	---	---------------------	---------------------

Heat transfer through the pipe wall

For a general radial system, the heat transfer coefficient through a layer of material can be calculated from:

$$h = \frac{2\lambda}{D_2 \ln \frac{D_2}{D_1}} \quad (13)$$

In which:

D2	=	Outer diameter	[m]
D1	=	Inner diameter	[m]

Figure 2 shows the definitions of the parameters of equation 2. The total heat transfer coefficient of a pipeline element consisting of multiple material layers can be calculated with:

$$\frac{1}{h_{total}} = \sum \frac{1}{h_i} \quad (14)$$

Heat transfer through the subsoil

The subsoil can be included by using equation 13. For the outer diameter the distance of the undisturbed ground should be used. If this distance is in the same range as the depth of the pipe, the effect of the subsoil should be taken into account via the following equation [1]:

$$R_g = \frac{1}{2\pi\lambda_g} \ln \left[\frac{2H}{D_c} + \sqrt{\left(\frac{2H}{D_c}\right)^2 - 1} \right] \quad (15)$$

in which

R _g	=	Heat resistance of the subsoil	[mK/W]
λ _g	=	Thermal conductivity of the ground	[W/m/K]
H	=	Corrected depth of pipe centre	[m]
D _c	=	Equivalent diameter of casing	[m]

This can be transferred to a heat transfer coefficient via:

$$h = \frac{1}{\pi DR} \quad (16)$$

The corrected depth of the pipe can be calculated from:

$$H = H' + \frac{\lambda_g}{h_{gs}} \quad (17)$$

In which:

H'	=	Depth of the pipe centre	[m]
h _{gs}	=	Heat transfer coefficient at ground surface	[W/m ² /K]

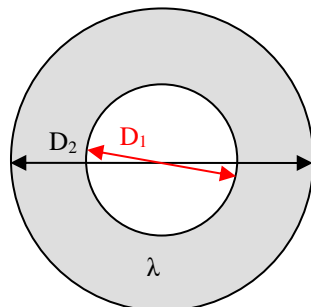


Figure 2. Definitions for formula 2 for the heat transfer coefficient.

Influence of adjacent pipe

The final effect which is of importance is the effect of adjacent pipes, see figure 3. This will influence the temperature in the ground around the pipes and therefore all effect need to be taken into account simultaneously. The heat loss for one pipe is given by [1]:

$$Q = L(U_1 - U_2)(T_1 - T_g) + LU_2(T_1 - T_2) \quad (18)$$

in which

U = Heat loss coefficient [W/m/K]

Subscript 1 is for the pipe for which the heat loss is calculated subscript 2 is for the adjacent pipe.

The heat loss coefficients are given by:

$$U_1 = \frac{R_{ground} + R_{iso}}{(R_{ground} + R_{iso})^2 - R_m^2} \quad (19)$$

$$U_2 = \frac{R_m}{(R_{ground} + R_{iso})^2 - R_m^2} \quad (20)$$

The heat resistance of the pipe and isolation can be calculated with help of formulas (8) to (14) and equation (16). The heat resistance of the ground can be calculated from:

$$R_g = \frac{1}{2\pi\lambda_g} \left[\ln \left(\frac{4H}{D_c} \right) \right] \quad (21)$$

β is given by:

$$\beta = \frac{\lambda_g}{\lambda_i} \ln \left(\frac{D_i}{D_p} \right) \quad (22)$$

In which:

λ_i = Thermal conductivity of the insulation [W/m/K]
 D_i = Diameter of insulation [m]
 D_p = External diameter of the pipe [m]

Finally the heat resistance due to the influence of a nearby pipe can be calculated from:

$$R_m = \frac{1}{4\pi\lambda_g} \ln \left(1 + \left(\frac{2H}{E} \right)^2 \right) \quad (23)$$

In which:

E = Distance between the centres of the pipes [m]

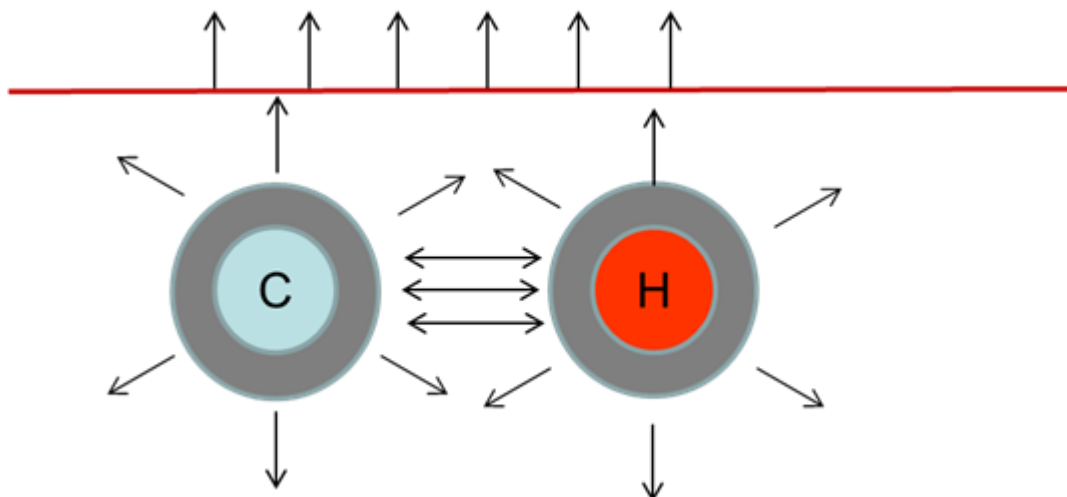


Figure.3 Heat loss from two adjacent pipe.

6.10.1.3 Additional losses

If several local losses (like bends, Tee's etc) exist in a pipe section, you can combine them in the pipe model (neglecting the exact position). Combining them can be done in 3 different ways, using either:

- the ξ -annotation (e.g. for one elbow ξ_{elbow} is about 0.2, hence for 5 elbows the combined ξ -value is 1.0)
- the equivalent length annotation (for an elbow L_{eq} is about 15 D)
- a percentage relative to the calculated loss in the pipe itself

6.10.1.4 Geometry

Each PIPE must have a length input and height location. This so-called profile can be defined in several ways:

- Scalar value for length (interpolating height from Heat nodes)
- Length-height profile
- Isometric layout specified with absolute XYZ co-ordinates
- Isometric layout specified with differential XYZ co-ordinates.

Each type of profile input is translated to a longitudinal profile (x-H profile). In case of a scalar value the height of the beginning and the end of the pipe is derived directly from the adjacent Heat nodes.

The input height is the height of the centreline of the pipe (see figure 4 below).

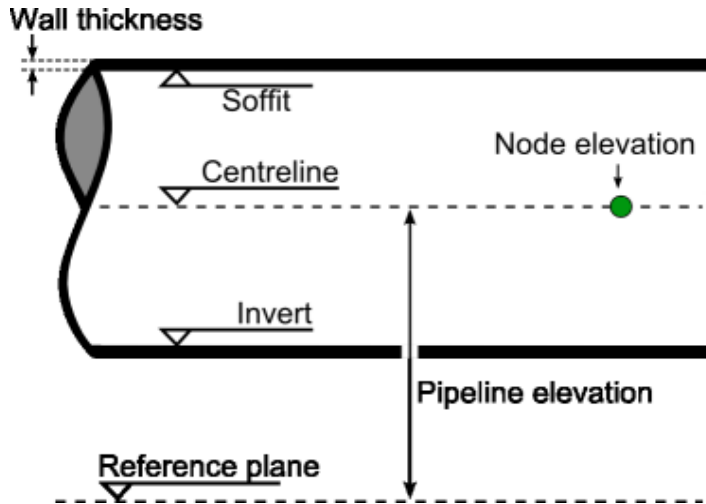


Figure 4: Definition of the height of the pipe and the node elevation.

WANDA compares the input of the beginning and end of the Pipe with the connecting Heat node heights. If there is a difference of more than half the pipe diameter, a warning is displayed in the property field "H-node height check". Otherwise, "OK" is displayed.

Below an example of differential XYZ input based on the given isometric is given.

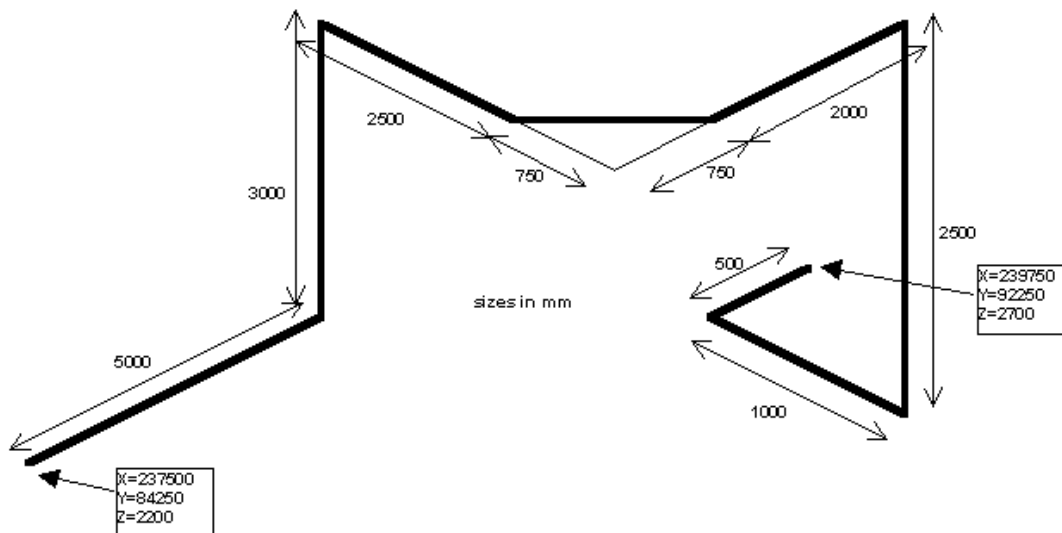


Figure 5: PIPE isometric with differential input.

	x-distance (m)	height (m)	X-abs (m)	Y-abs (m)	Z-abs (m)	X-diff (m)	Y-diff (m)	Z-diff (m)
1	0.0000	2.20000	237.5000	84.25000	2.200000	237.5000	84.25000	2.200000
2	5.0000	2.20000	237.5000	89.25000	2.200000	0.000000	5.000000	0.000000
3	5.0000	5.20000	237.5000	89.25000	5.200000	0.000000	0.000000	3.000000
4	7.5000	5.20000	240.0000	89.25000	5.200000	2.500000	0.000000	0.000000
5	8.5607	5.20000	240.7500	90.00000	5.200000	0.7500000	0.7500000	0.000000
6	10.561	5.20000	240.7500	92.00000	5.200000	0.000000	2.000000	0.000000
7	10.561	2.70000	240.7500	92.00000	2.700000	0.000000	0.000000	-2.500000
8	11.561	2.70000	239.7500	92.00000	2.700000	-1.000000	0.000000	0.000000
9	12.061	2.70000	239.7500	92.50000	2.700000	0.000000	0.5000000	0.000000

Please note the first row starts with the absolute XYZ coordinates followed by the differential coordinates.

The input is translated to real XYZ values and subsequently to a x-H profile. This x-H profile is shown in the location charts.

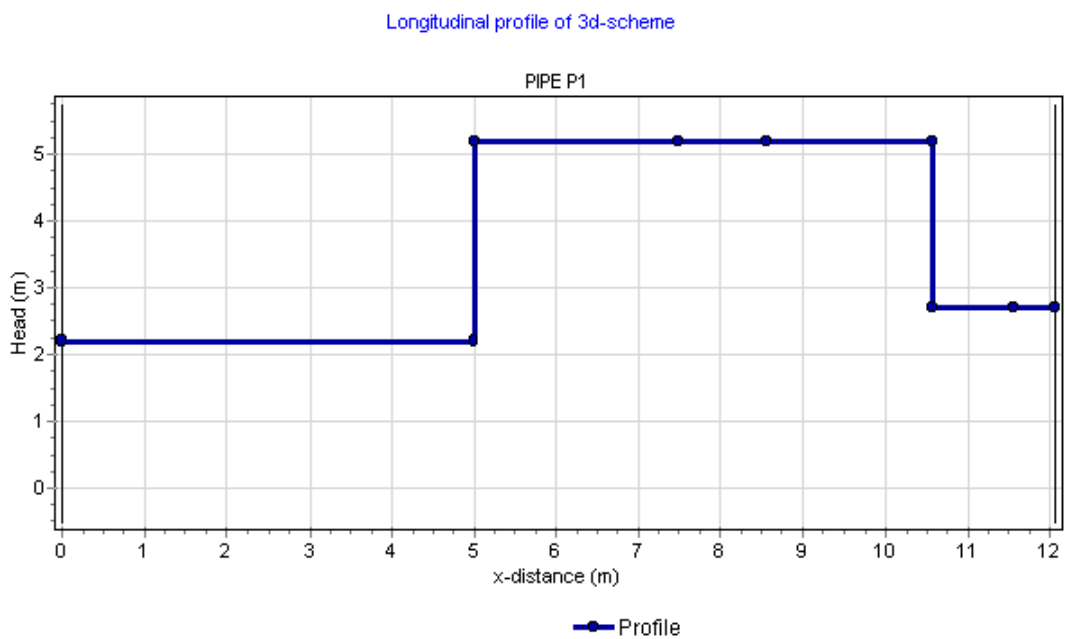


Figure 6: result of XYZ diff specified profile translated to x-H profile

It is not necessary that the pipe profile start with zero. Any positive value is possible. If you split a pipe into two parts, you only have to copy the pipe and delete the unnecessary rows.

	x-distance (m)	height (m)
1	31090	132.910
2	31670	131.510
3	32270	131.010
4	32850	129.810
5	33400	127.210

The location chart (route) starts with the value specified in the first row of the table.

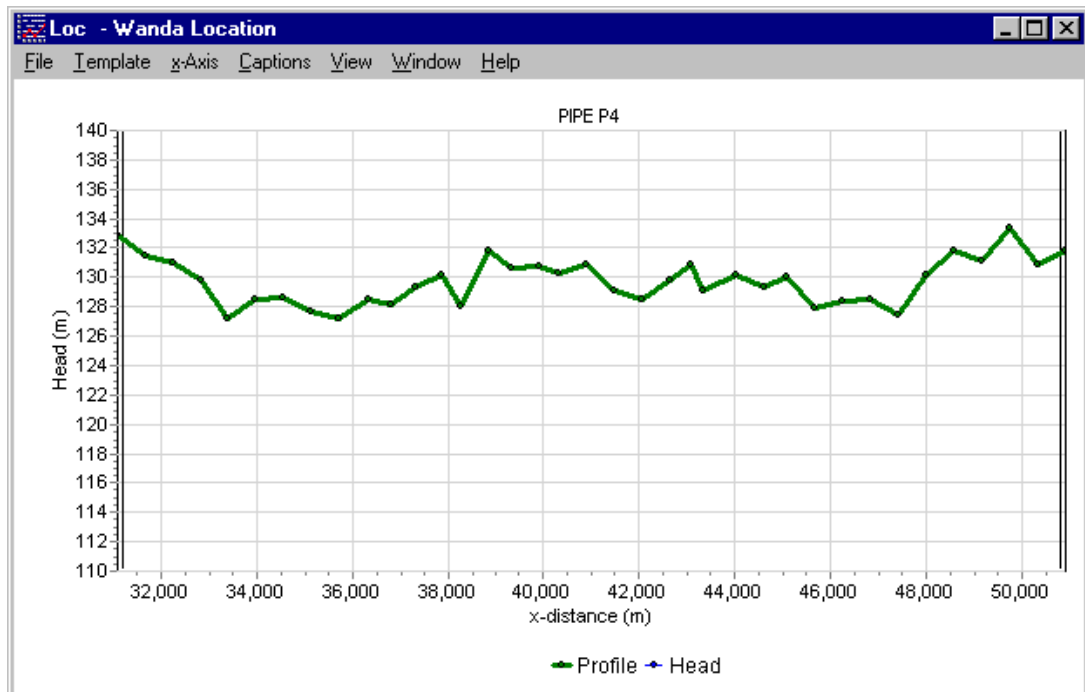


Figure 7: Pipe profile starting with a non-zero begin location

6.10.1.5 Reference

- [1] Boehm, B. On transient heat losses from buried district heating pipes, 2000, International journal of energy, 24; 1311-1334
- [2] Mills, A.F., *Basic heat and mass transfer*. second ed. 1999: Prentice Hall, Inc.
- [3] Incropera, F.P. and D.P. De Witt, *Fundamentals of heat and mass transfer*. 2 ed. 1985: John Wiley & Sons inc.
- [4] Dullemond C.P. and Wang A.A. , Lecture Numerical Fluid Dynamics, Course given at Heidelberg University, (<http://www.mpia.de/homes/dullemon/lectures/fluidynamics/>)
- [5] ISSO Publicatie 7, Grondleidingen voor warmte en koude transport, 2012, ISBN:978-90-5044-232-9
- [6] NEN-EN 13941+A1, Ontwerp en installatie voor geïsoleerde buissystemen voor stadsverwarming, 2010

6.10.2 Heat pipe

6.10.2.1 Hydraulic specifications

Input properties

Description	Input	SI-units	Remarks
Calculation mode	Waterhammer Rigid column		Only in transient mode, default = Waterhammer
Inner diameter	real	[m]	
Wall roughness	real	[mm]	
Wall thickness	real	[m]	Only in transient mode and with calculation mode Waterhammer
Young's modulus	real	[N/m ²]	Only in transient mode and with calculation mode Waterhammer
Advection velocity	real	[m/s]	Only in transient mode and with calculation mode Rigid column
Geometry input	Length l-h xyz xyz diff	[-]	Default = Length
Length	real	[m]	if Geometry input = Length
Profile	table		if Geometry input = L-h, xyz or xyz diff
Upper limit pressure	real	[N/m ²]	Only for post processing, not needed for calculation
Lower limit pressure	real	[N/m ²]	Only for post processing, not needed for calculation
Additional losses	None Xi Length Percentage	[-]	Default = None
Additional length	real	[m]	If Additional losses = Length
Local losses coeff	real	[-]	If Additional losses = Xi
Add. loss percentage	real	[%]	If Additional losses = Percentage
Fraction gen. heat to fluid	real	[%]	
Heat transf input	Value Table	[-]	
Heat transf coef	real	[W/m ² /K]	if Heat transf input = Value
Heat transf table	table		if Heat transf input = table, see remark
Heat trans. ground	No Yes	[-]	if Heat transf input = table
Ground coverage	real	[m]	If Heat trans. Ground = yes
Thermal conductivity of ground	real	[W/m/K]	If Heat trans. Ground = yes

Description	Input	SI-units	Remarks
Heat transfer coefficient at ground surface	real	[W/m ² /K]	If Heat trans. Ground = yes
Adjacent pipe	yes no		If Heat trans. Ground = yes
Pipe pair number	integer	[-]	If Adjacent pipe = yes see remark
Distance between centre of the pipes	real	[m]	if adjacent pipe = yes
Ambient temperature	real	[°C]	Controllable with control component
Location	real	[m]	If empty, chart button creates a location series If filled in, chart button creates a time series for nearest internal calculation node

Remarks

The location charts of "Pressure" and "Head" shows the specified limits only if they are filled in. It is possible to specify these values after the simulation. If these fields are not visible, first check the "Upper/Lower limits in pipes" fields in the Mode & option window. You may change these values without re-calculating the case. In the heat domain, the change in the maximum head criterion due to changing density is not taken into account.

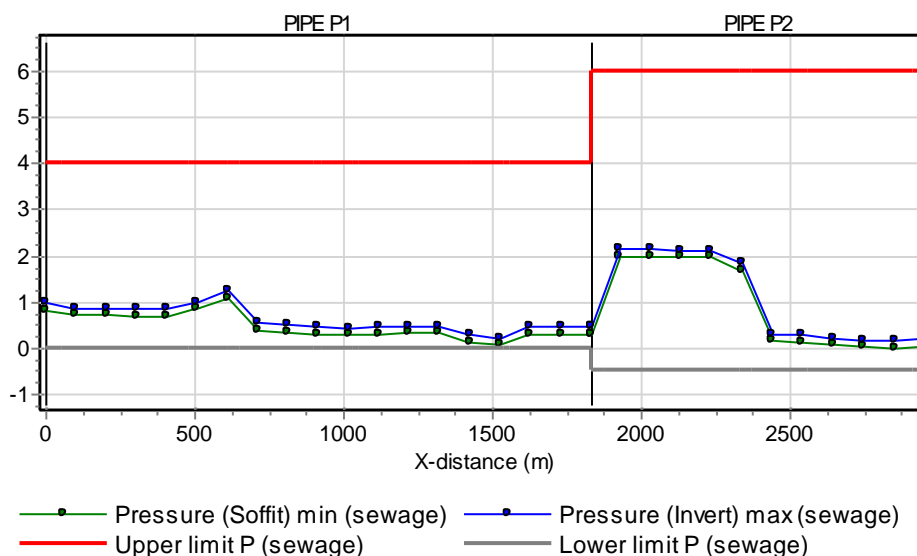


Figure 8: Pressure location chart with additional pressure rating series

Note that the reference for the pressure output (centre line / soffit (top) / invert (bottom)) is defined in the Mode & options window.

The main output properties like "Discharge" and "Pressure" contains vector based output (output values in each calculation node or element in the pipe). That means that the chart button automatically creates a location chart. If you want a time chart in a particular pipe node you have to specify the location in the "Location" field.

To have direct access to the first and last node of the pipe (corresponding with the left and right H-node) you may extend the output property list with these points by checking the field "Connect point output in pipes" in "Modes & Options" (ctrl-m). Pressing the chart button in these additional fields automatically generates a time series chart.

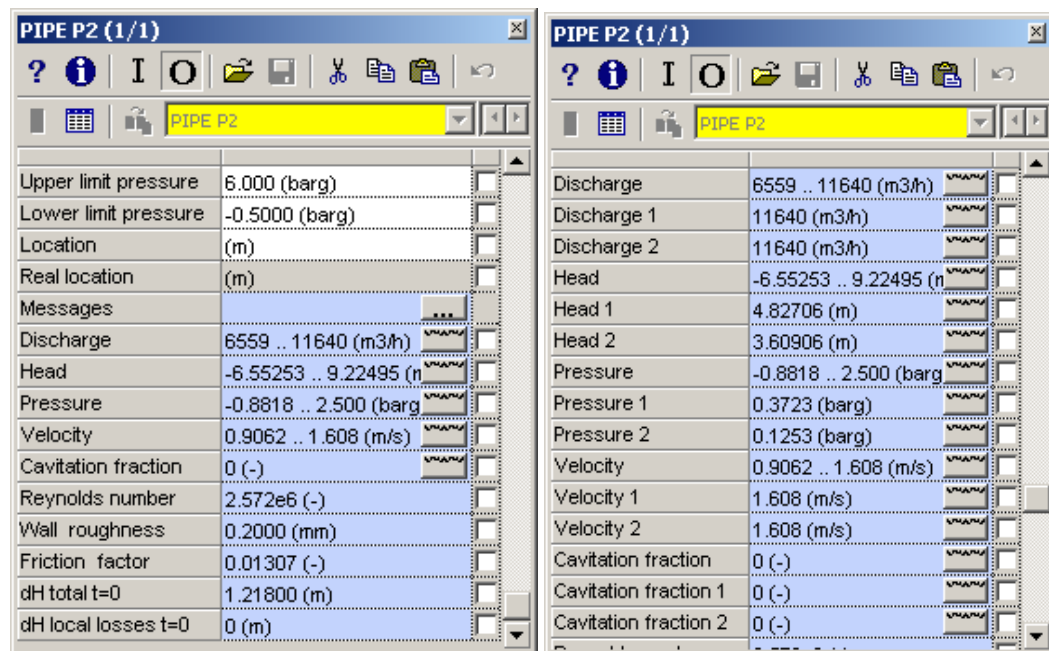


Figure 9: Two appearances of output properties window depending on Mode & Options setting "Connect point output in pipes".

Heat transfer coefficient

If the heat transfer coefficient table option is selected the user should fill in a table with the outer diameter of the material layer and the thermal conductivity coefficient. Equation 13 is used to calculate the heat transfer coefficient. For the inner diameter the outer diameter of the previous layer (previous row in the table is used).

Adjacent pipe property.

To include the effect of an adjacent pipe the user needs to specify pairs of pipes in the model via the pipe number input property. This value should be an unique number for each pair of pipes. If the same number is given for more than two pipes an error message is given.

The user should ensure that the number of elements in both pipes is the same. If this is not the case an error message is generated.

6.10.2.2 Component specific output

Output	Description
Wave speed [m/s]	Wave propagation speed based on fluid properties and pipe properties
Pipe element count [-]	Amount of elements of same length in which pipe is divided.
Adapted wave speed [m/s]	Wave speed is adapted in such way that a integer number of elements (minimal 1) fit in the total length
Deviation adapted [%]	Ratio between wave speed and adapted wave speed. Must be less than 25% for a valid computation

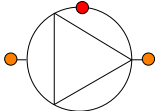
H-node height check	Input validation between H-node height with corresponding pipe node elevation. OK means the difference is less than 0,5 D, otherwise "Warning" is displayed.
Real location [m]	If a location is specified the nearest (internal) calculation node will be selected. The real location is the location of this calculation node.
Cavitation fraction [-]	Ratio between cavitation volume and element volume Must be less than 20% for valid range of cavitation algorithm
Generated heat flux [W]	The generated heat due to friction with the pipe wall
Heat flux out [W]	Heat radiated to the surroundings through the pipe wall
Total heat flux [W]	Nett heat flux including effect of 'fraction generated heat to fluid' parameter (heat actually absorbed by the fluid)
Ambient temperature [°C]	The friction factor is used in the calculation of the head loss. This factor is calculated iteratively based on the wall roughness.
Friction factor [-]	The friction factor is used in the calculation of the head loss. This factor is calculated iteratively based on the wall roughness.
Kinematic viscosity [m ² /s]	The kinematic viscosity.

6.10.2.3 Actions

The action table (control connection) of this component controls the ambient temperature as function of time.

6.11 Pump

Fall type

Type label	Symbol	Description	Active
incl. Heat		Complete pump model with various data and drives. Positive and negative speeds	yes

6.11.1 Mathematical model

A pump is not 100% efficient and therefore produces heat, a fraction of which (fr) is transferred to the fluid. The temperature increase of the fluid is modelled as a quasi-steady process, thereby neglecting temporary heat accumulation effects in the pump housing. The fluid temperature increase is modelled by:

$$\dot{m}c_2T_2 = \dot{m}c_1T_1 + fr(1-\eta)P \quad (1)$$

in which:

\dot{m}	=	mass flow at duty point	[kg/s]
η	=	pump efficiency at duty point	[-]
P	=	pump shaft power	[W]
T_i	=	temperature in heat node i	[K]
c_i	=	specific heat in heat node i	[J/kg/K]
fr	=	fraction of generated heat absorbed by fluid	[-]

The pump's efficiency is determined every time-step from the pump curve ($Q-H$ and $Q-\eta$). The pump data is supplied at a certain density. It is assumed that the pump curves remain valid over the entire range of temperatures and densities used in the calculation.

The remainder of this chapter is identical to the pump component described in the WANDA Liquid manual. The only difference is explained above and no additional information is included for the pump in WANDA Heat. Although the core variables have been changed (mass flow Instead of discharge, pressure Instead of head), the pump curves are still expressed in discharge, head and efficiency.

As the pump (QHE) curves as provided by the manufacturer are (normally) given in fluid head (H) and not in pressure, there is no need to 'translate' the curves to different densities if the actual fluid is different from the fluid used in the tests. The other (temperature dependent) fluid properties however most likely do have an effect on the pump performance (efficiency). This effect however is not proportional and therefore it is not included in the pump model. The validity of the pump curves is to be determined by the pump manufacturer.

6.11.1.1 Affinity rules

In the most general case a pump can operate in four QH-quadrants for $N \geq 0$ and in four QH-quadrants for $N < 0$. In WANDA, two pump models are distinguished:

5. Pump model limited to the $N \geq 0$ case, see section 6.11.1.2
6. Complete pump model for all $N (<0, =0 \text{ and } >0)$, see section 6.11.1.3

Basis equations

In the first quadrant for $N > 0$ a running pump is characterised by the following functions:

$$H = f(Q), N = N_r \quad (2)$$

$$\eta = g(Q), N = N_r \quad (3)$$

in which:

H	=	pump head	[m]
Q	=	pump discharge	[m ³ /s]
η	=	pump efficiency	[-]
N	=	pump speed	[rpm]
N_r	=	rated pump speed (at which the functions f and g are specified)	[rpm]

To describe a tripping pump the torque function must be known as well:

$$T = h(Q), N = N_r \quad (4)$$

To derive the functions (2), (3) and (4) for other pump speeds N_i than the rated pump speed N_r the affinity rules are used:

$$\frac{H(N_i)}{H(N_r)} = \left(\frac{N_i}{N_r}\right)^2 \quad (5)$$

$$\frac{Q(N_i)}{Q(N_r)} = \frac{N_i}{N_r}$$

$$\frac{T(N_i)}{T(N_r)} = \left(\frac{N_i}{N_r}\right)^2$$

The affinity rules – also known as the homologous relations – are based on the assumption that the flow field and the accompanying pressure field are independent of the size of the pump as long as the shape (proportions) remain the same, and that also the efficiency does not change with the size of the pump.

To simulate a tripping pump Newton's second law is used in terms of rotating mass:

$$T_d - T = I_p \frac{dN}{dt} \frac{2\pi}{60} \quad (6)$$

in which T_d is the driving torque and T is the torque exerted by the fluid on the pump impeller. I_p denotes the polar mass moment of inertia [kgm²] of rotating mass (=motor +

impeller + fluid). Note: sometimes the value GD^2 (GeeDeeSquared) is used which is four times the I_p

In normal running operation $T_d = T$ and the pump will run at a steady speed. Due to e.g. power failure, $T_d = 0$, and therefore during pump trip the decaying pump speed can be calculated from:

$$dN = -\frac{60}{2\pi} \frac{T}{I_p} dt \quad (7)$$

In the procedure this differential equation is solved using Euler's method. The torque for N_i is calculated using the affinity rules (5). In the case of a starting pump, T_d is determined by the speed-torque relation (in tabular form) of the driving motor-gearbox combination.

6.11.1.2 The pump model for $N \geq 0$ (QHE table)

The functions (2) and (3) are usually unknown in algebraic form. A pump manufacturer normally provides f and g in graphics and then only in the first quadrant for $N = N_r$. Often these graphics are specified as tables. Hence at the start of the simulation only the Q-H table and the Q-E table are known for N_r . From both tables the torque table can be derived using:

$$T = \frac{\rho_f g Q H}{E \omega_r} \quad (8)$$

in which:

ρ_f	= fluid density	[kg/m ³]
g	= grav. acceleration	[m/s ²]
E	= efficiency	[-]
ω_r	= rated pump speed	[rad/s]

However, to be able to simulate pump trip the functions H and T have to be known in the second and fourth quadrant as well. To get the values outside the table range the tables are extended to the 'left hand' and 'right hand' side with so-called extrapolation parabolas. These parabolas have the following general forms:

$$\text{Q-H, right of the table range:} \quad H / H_r = A_H(Q / Q_r)^2 + B_H(Q / Q_r) + C_H \quad (9)$$

$$\text{Q-H, left of the table range:} \quad H / H_r = D_H(Q / Q_r)^2 + E_H(Q / Q_r) + F_H$$

$$\text{Q-T, right of the table range:} \quad T / T_r = A_T(Q / Q_r)^2 + B_T(Q / Q_r) + C_T$$

$$\text{Q-T, left of the table range:} \quad T / T_r = D_T(Q / Q_r)^2 + E_T(Q / Q_r) + F_T$$

As we can see, the parabolas are given in dimensionless form. The H , T and Q are made dimensionless with respect to H_r , T_r and Q_r i.e. the values in the maximum efficiency point (at the rated pump speed N_r). The extrapolation coefficients are derived in a quite specific way. The A and D coefficients are calculated from empirical data.

These coefficients are given in the tables 1 through 4 inclusive below as a function of the pump specific speed N_S .

Table 1: A_H coefficients and derivatives versus the pump specific speed N_S

N_S	A_H	dA_H/dN_S
10	-0.446	-0.004100
20	-0.487	-0.005075
50	-0.649	-0.007263
100	-1.068	-0.009730
150	-1.622	-0.008070
200	-1.875	0.004690
250	-1.153	0.014440

Table 2: D_H coefficients and derivatives versus the pump specific speed N_S

N_S	D_H	dD_H/dN_S
10	0.566	0.005100
20	0.617	0.006400
50	0.822	0.009650
100	1.389	0.013480
150	2.170	0.007370
200	2.126	-0.009100
250	1.260	-0.017320

Table 3: A_T coefficients and derivatives versus the pump specific speed N_S

N_S	A_T	dA_T/dN_S
10	-0.332	-0.003800
20	-0.370	-0.004675
50	-0.519	-0.007588
100	-0.977	-0.010660
150	-1.585	-0.006080
200	-1.585	0.008050
250	-0.780	0.016100

Table 4: D_T coefficients and derivatives versus the pump specific speed N_S

N_S	D_T	dD_T/dN_S
10	0.704	0.005800
20	0.762	0.007375
50	0.999	0.010050
100	1.566	0.011270
150	2.126	0.002140
200	1.780	-0.013450
250	0.781	-0.019980

The pump specific speed for a particular pump is defined as:

$$N_S = \frac{N_r(Q_r)^{0.5}}{H_r^{0.75}} \quad (10)$$

This quantity characterises the pump. Low values (<50) indicate a centrifugal (or radial) type of pump and high values (>200) indicate an axial type of pump. Suppose a given pump results in a certain N_{spec} . Usually this value is not equal to one of the given table values N_S and interpolation is required. This is done using the derivatives (which are also given in the tables) in the following way:

$$A_H(N_{\text{spec}}) = A_H(N_S) + (N_{\text{spec}} - N_S)dA_H/dN_S \quad (11)$$

The same method is applied for D_H , A_T and D_T . When these empirical values are known, the remaining coefficients are determined as follows:

- B_H and C_H are calculated using the last two Q-H table points.
- E_H and F_H are calculated using the first two Q-H table points.
- E_T and F_T are calculated using the first two Q-T table points.

To obtain the B_T and C_T coefficients the method is different:

A. First, the intersection point is determined of:

$$H/H_r = A_H(Q/Q_r)^2 + B_H(Q/Q_r) + C_H \quad (12)$$

and

$$H/H_r = \phi(Q/Q_r)^2 \quad (T=0 \text{ line})$$

The $T=0$ line describes the relation between H and Q when the pump is maintaining a speed that results exactly in a hydraulic torque $T = 0$. The coefficient ϕ is an empirical value: $\phi = -0.3$. It is assumed to be a constant independent of the pump specific speed (although in reality it may vary between about -0.1 and -0.4). The intersection results in the discharge Q_{T0} i.e. the discharge for which $T = 0$.

B. B_T and C_T are now determined from the last table point of the Q-T table and the point $Q = Q_{T0}$, $T = 0$.

In dimensionless form the various tables and extrapolation parabolas are given as an example in figure 1. In this figure $Q_{T0} = 3.52$ (dimensionless!).

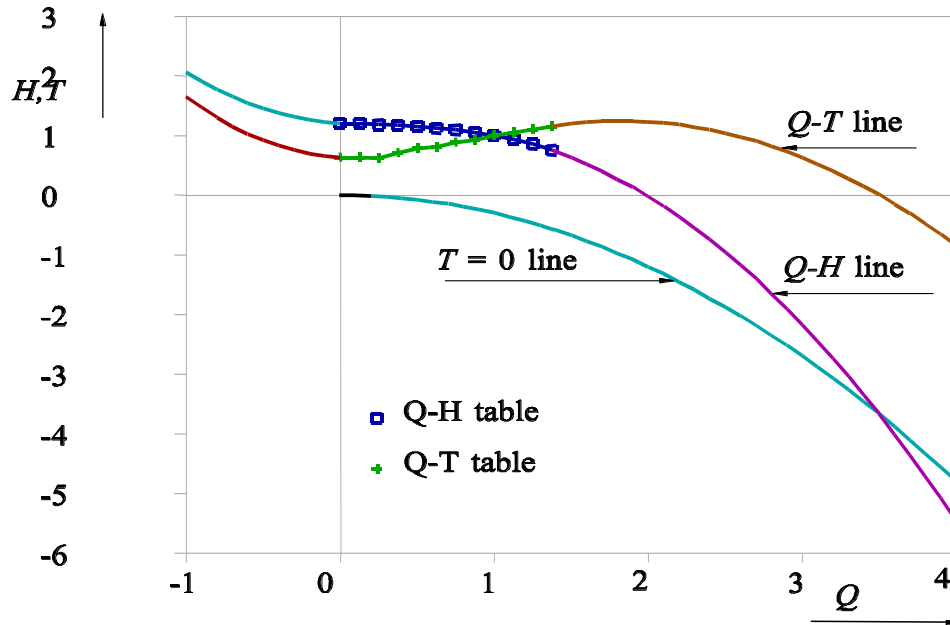


Figure 1: Dimensionless curves of pump for $N > 0$

6.11.1.3 The complete pump model (suter curve)

In order to obtain the pump function for positive and negative flow as well as positive and negative pump speed, heads and torque's for all combinations of Q and N have to be measured and plotted as isolines in an N - Q graph. This has been carried out for a number of pumps in literature. These pumps are of varying type (radial flow, mixed flow, axial flow) denoted by specific speeds of 25, 147 and 261 (SI units) respectively (see Wylie, *Fluid Transients in Systems*, 1993).

It has proven convenient to use dimensionless quantities for head, torque, discharge and speed, by dividing the quantities by their respective values at rated (r) conditions.

$$h = \frac{H}{H_r} ; \quad \beta = \frac{T}{T_r} ; \quad q = \frac{Q}{Q_r} ; \quad \alpha = \frac{N}{N_r} \tag{13}$$

Expressed in these dimensionless quantities the affinity laws read:

$$\frac{h}{\alpha^2} \text{ vs. } \frac{q}{\alpha} \quad \frac{\beta}{\alpha^2} \text{ vs. } \frac{q}{\alpha} \quad \frac{h}{q^2} \text{ vs. } \frac{\alpha}{q} \quad \frac{\beta}{q^2} \text{ vs. } \frac{\alpha}{q} \tag{14}$$

From a computational viewpoint, these relations are difficult to handle (dividing by zero when pump speed passes zero, and changing signs during transients). Marchand and Suter have overcome this difficulty by using:

$$WH = \frac{h}{\alpha^2 + q^2} \text{ vs. } \tan^{-1} \frac{q}{\alpha} \quad \text{and} \quad WB = \frac{\beta}{\alpha^2 + q^2} \text{ vs. } \tan^{-1} \frac{q}{\alpha} \tag{15}$$

For complete pump data in all zones of operation, a polar diagram of WH and WB can be plotted as closed curves. This can be done using a rectangular plot (see figure 2) with $\tan^{-1} \frac{q}{\alpha}$ on the x-axis.

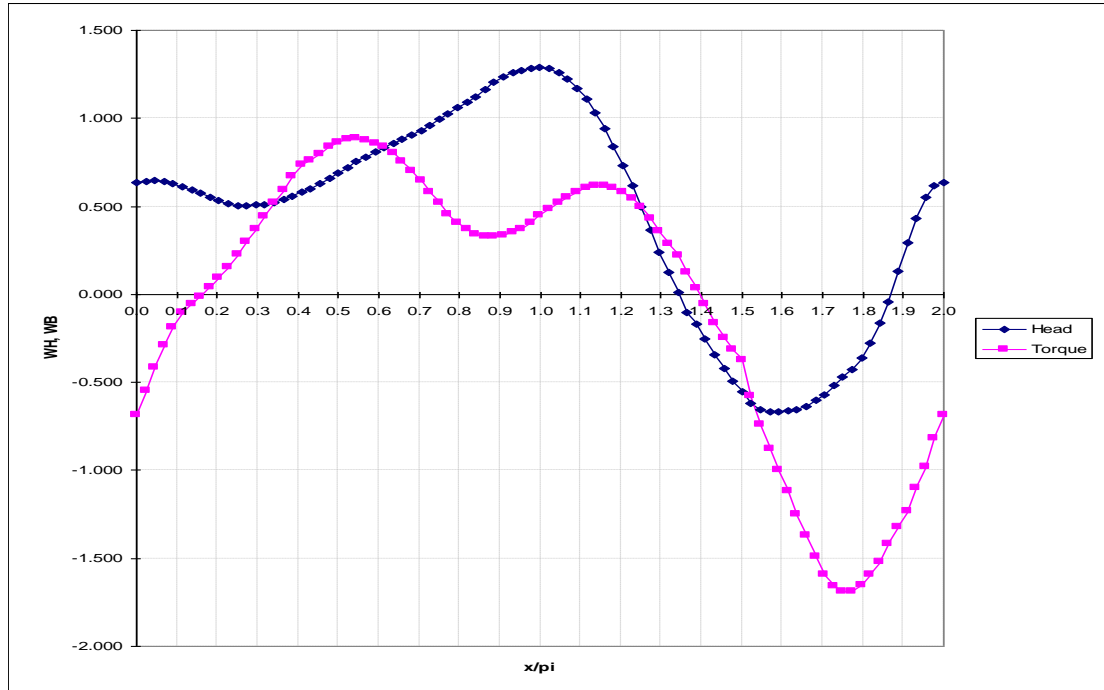


Figure 2: Dimensionless Suter curves of radial pump ($N_s=25$, SI units)

These curves can for any α be transformed into head and torque as functions of discharge and made dimensional by substitution of actual values in (13) and (15). An example transformed from figure 2 is given in figure 3.

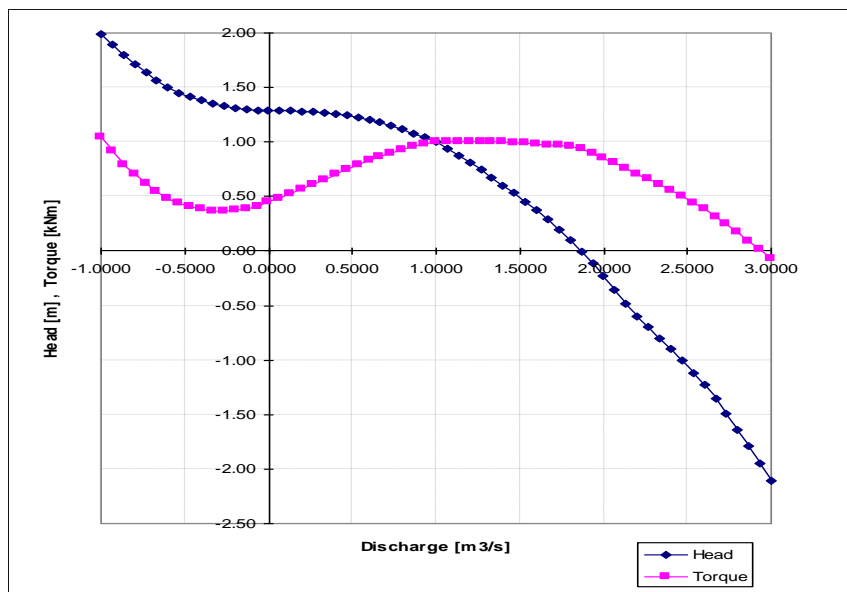


Figure 3: Dimensionless curves of pump for $N>0$

This procedure works equally well for $\alpha < 0$, $\alpha = 0$ and $\alpha > 0$. The user input consists of the two tables $WH(x)$ and $WB(x)$ among other quantities. If only a QHE-table is

available the user must prepare the WH/WB tables by manual transformation and fill in the empty parts by interpolating from known curves for comparable specific speed pumps. The results of such interpolations must however be viewed with scepticism.

6.11.1.4 Interpolation of the pump speed

The calculation of the pump speed at t_{new} ($= t_{old} + \Delta t$) is quite specific. Equation (7) is integrated using Euler's method and sub-timesteps Δt^* :

$$N_{new} = N_{old} + \frac{60}{2\pi} \frac{(T_d - T)_{(N_{old})}}{I_p} \Delta t \tag{16}$$

in which:

- N_{new} = pump speed at $t = t_{new}$
- N_{old} = pump speed at $t = t_{old}$
- T_d = driving torque at $t = t_{old}$, hence for $N = N_{old}$
- T = hydraulic torque at $t = t_{old}$, hence for $N = N_{old}$

For a tripping pump $T_d = 0$. To obtain sufficient accuracy the difference between N_{new} and N_{old} may not be too large for each Euler integration sub-timestep. This difference is denoted as $DN = N_{new} - N_{old}$. DN must have a value as given in figure 4. This value is denoted as DN_w ("wanted" DN).

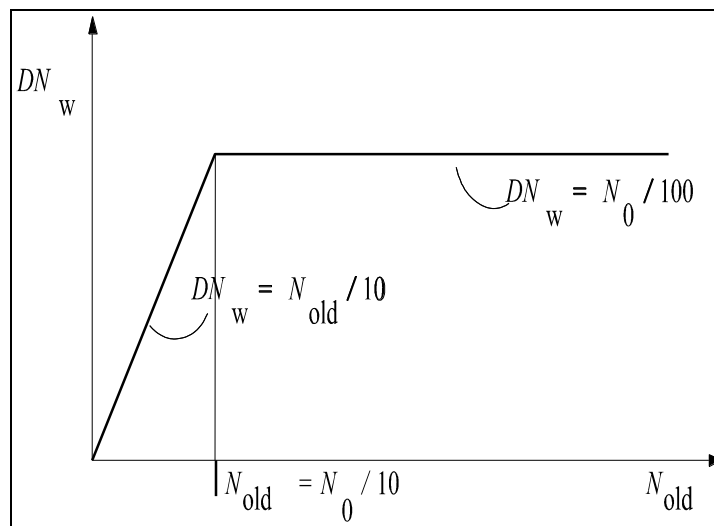


Figure 4: DN_w as a function of N

As long as $N_{old} > N_0/10$, DN_w is fixed to $DN = N_0/100$. At a certain instant the pump speed drops below the value $N_0/10$. From that moment on DN_w is varying according to $DN = N_{old}/10$. Based on this arbitrarily chosen DN_w a new (sub) time step Δt^* is calculated in the following way:

- N_{new} is calculated using Δt (first estimate)
- The difference DN is calculated
- Depending on N_{old} , DN_w is calculated (see figure 2)
- The number of (sub) time steps within Δt is calculated: $N_{t-steps} = DN/DN_w$
- The new time step $\Delta t^* = \Delta t/N_{t-steps}$ is calculated.

Once this is done the N_{new} is calculated using Δt^* (provided that $N_{t-steps} > 1$) until $t = t_{old} + \Delta t$.

6.11.1.5 Motor drive

For the motor driven pump the following considerations are to be taken into account. The pump speed is computed using equation (17) of the mathematical model description in the beginning of this section.

$$T_d - T = I_p \frac{dN}{dt} \frac{2\pi}{60} \quad (17)$$

Here, the driving torque T_d is determined by the motor properties. In general, when the motor is powered on, T_d is a function of the pump speed and the frequency of the speed control. In case the power is off, T_d is of course zero and we then have a situation equivalent to the tripping pump.

Typical speed-torque curves for electric motors and diesel engines are plotted below.

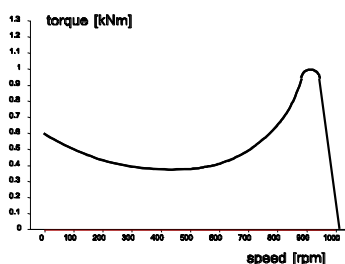


Fig. 5: Electric motor

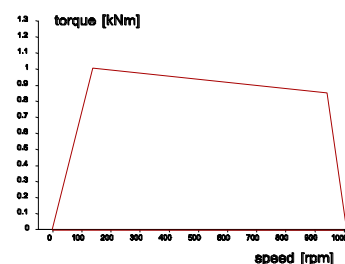


Fig. 6: Diesel engine

Both curves are characterised by a sharp drop (cut-off) around the operating speed. The result of this is that with load increase the speed does not drop much. This is denoted as elastic behaviour. If the maximum torque just before the drop is sufficiently large the steady state value of the pump speed is well defined. In steady state WANDA will compute two equilibria: 1) pump (QH) curve versus system resistance (QH relation) and 2) hydraulic torque versus motor torque. Both are varying at each iteration step and to reach convergence a dedicated iteration strategy has been employed.

For electric motors the pump speed at which the torque goes through zero is denoted by the term “synchronous speed”. This speed is in general equal to 3000 RPM (the

frequency of the AC power) divided by the number of coils, which is half the number of poles. In practice many two-coil (four pole) and three-coil (six pole) motors are used with their respective synchronous speeds of 1500 RPM and 1000 RPM. An electric motor can only generate torque if the shaft RPM exhibits a certain “slip” relative to the synchronous speed. The nominal power (and torque) will be delivered at somewhere between 1 and 3 % slip, this is specified by the manufacturer. The peak torque is about 3 times as large as the nominal torque. The start torque is about 2.2 times as large as the nominal torque and the sag torque (the relative minimum half way) is about 2 times the nominal torque.

To be able to “brake” or “force down” the pump by means of a temporarily negative torque, the speed torque table has to be mirrored about the synchronous point. This can easily be done by hand by adding the difference of the synchronous speed and the speed value to the synchronous speed and reversing the torque value of each speed-torque co-ordinate pair. An example is given in figure 7 below:

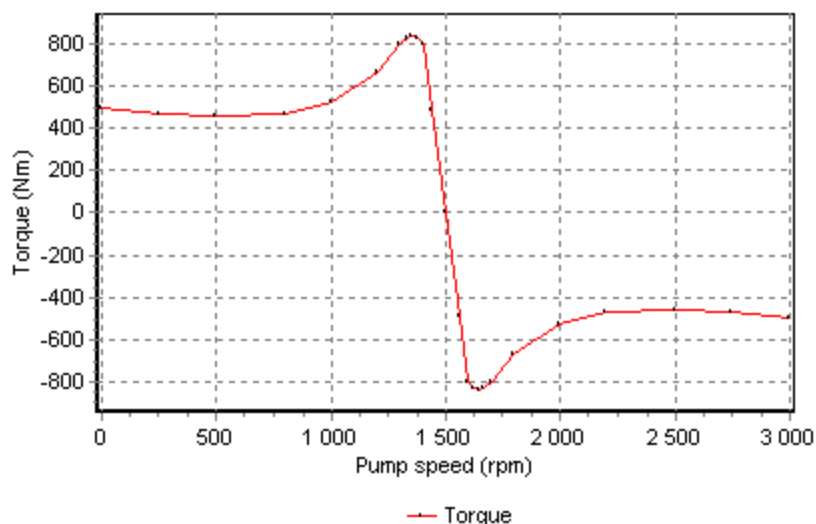


Figure 7: Complete speed-torque curve for electric motor

The nominal power for the speed torque curve in figure 7 can be computed by multiplying the nominal torque (which could be 220 Nm) and the speed (which is in that case 155 rad/s). The resulting nominal power is 34.1 kW. However in this case the motor can also run at 150.8 rad/s (1440 RPM) delivering a torque of 490.7 Nm (this is the penultimate co-ordinate just before the synchronous point), leading to a power of 74 kW. The motor slip is in that case 60 RPM. Whether this will lead to thermal problems depends on the design of the motor and this mode of operation has to be confirmed by the manufacturer.

The speed values in the curve are offset by the relative frequency, thus moving the curve in horizontal direction, thereby moving also the synchronous point to the left or right (decreasing or increasing of RPM). Theoretically the relative frequency can even become zero but then the motor torque will be zero throughout. In order to “force” the pump down in RPM the speed-torque curve must also contain negative torque values for speeds greater than the synchronous speed.

In unsteady state the relative frequency should not be changed faster than the pump RPM can follow, otherwise the operating point (intersection between impeller speed-torque and motor speed-torque curves) will drop in the start-up zone.

6.11.2 Pump properties

6.11.2.1 Hydraulic specifications

Description	Input	SI-units	Remarks
Characteristic type	QHE Suter		Default = QHE
QHE_table	table		if Char. type = QHE
Suter table	table	Dim. less	if Char. type = Suter
Rated speed	real	[rad/s]	
Q in best point of eff.	real	[m ³ /s]	if Char. type = Suter
H in best point of eff.	real	[m]	if Char. type = Suter
T in best point of eff.	real	[Nm]	if Char. type = Suter
Initial setting	Speed P_upstream P_downstream Mass flow Frequency		Default = Speed
Initial speed	real	[rad/s]	if Initial setting = Speed
Initial suction pressure	real	[N/m ²]	if Initial setting = P_upstream
Initial discharge pressure	real	[N/m ²]	if Initial setting = P_downstream
Initial mass flow	real	[kg/s]	if Initial setting = Mass flow
Initial relative frequency	real	[-]	if Initial setting = Frequency
Speed-Torque char.of motor	table		if Initial setting = Frequency
Polar moment of inertia	real	[kgm ²]	only in transient mode
Drive type	Running Trip Speed Motorfreq Level OnOff		Default = Running
Trip time	real	[s]	if Drive type = Trip
Action table speed	table		if Drive type = Speed
Minimum speed for tripping	real	[rad/s]	if Drive type = Speed
Action table motorfreq	table		if Drive type = Motorfreq
Minimum rel.freq. for trip.	real	[-]	if Drive type = Motorfreq
Level On	real	[m]	if Drive type = OnOff
Level Off	real	[m]	if Drive type = OnOff
Start time (0 to max)	real	[s]	if Drive type = OnOff
Stop time (max to 0)	real	[s]	if Drive type = OnOff
Fraction gen. heat to fluid	real	[-]	1 (100%)

Because the Suter curves are dimensionless the discharge (Q), head (H) and torque (T) in the best efficiency point must also be supplied if the characteristic type is "Suter".

In case of the QHE characteristic the Q and H at maximum efficiency will be found in the table. Hence the user must take care for one maximum value within the E range.

Used table types

QHE table (Q to run from 0.0 upwards, H to decrease monotonously, E to be zero at either H=0.0 or Q=0.0)

Q [m ³ /s]	H [m]	E [-]
Real	Real	Real
....
....
real	real	real

Suter table (x/π to run from 0.0 to 2.0)

x/π [-]	WH [-]	WB [-]
Real	Real	Real
....
....
real	real	real

Three Suter curves according to Wylie and 14 curves according to Thorley have been included as property templates (see [installation dir]\Property templates\Examples\Suter_pump)

Speed torque table in case of motor driven pump (see examples)

N [rad/s]	T [Nm]
Real	Real
....
....
real	real

Important:

- Minimum required rows is 5, maximum is 100.
- Efficiency must be specified dimensionless, not in percents: range 0 - 1 !
- The heads in the Q-H table must be monotonously decreasing.
- The Q-E table must have a maximum value for the efficiency E.
- Efficiency must be zero where either Q=0 or H=0.
- It is recommended to start QHE tables with Q = 0 [m³/s].
- The first and last row values of WH and WB should respectively have the same values (for $x/\pi=0$ and $x/\pi=2$), thus forming closed curves.

Steady state settings

To determine the steady state condition of the pump the choice property “Initial setting” has to be set accordingly. The user has five choices:

Speed	The operational speed is specified by the user. QH curve will be applied in computation and P1, P2 and Q are computed depending on system
P_upstream	The pressure upstream is specified by the user. P1 = C is applied in computation and P2 and Q are computed. Afterwards, the pump speed will be computed, such that the combination H2-H1 and Q is on the QH curve.
P_downstream	The pressure downstream is specified by the user. P2 = C is applied in computation and P2 and Q are computed. Afterwards, the pump speed will be computed, such that the combination H2-H1 and Q is on the QH curve.
Mass flow	The mass flow is specified by the user. $\dot{m} = C$ is applied in computation and P1 and P2 are computed. Afterwards, the pump speed will be computed, such that the combination H2-H1 and Q is on the QH curve.
Frequency	The relative motor frequency is specified by the user. This is the actual frequency (e.g. 40 Hz) divided by the nominal frequency (e.g. 50 Hz) yielding the value of 0.8. With this relative frequency the speed-torque curve of the motor is adapted (shifted horizontally) and the steady state (P1, P2 and Q) will be solved such that motor torque and hydraulic (impeller) torque intersect. The point of intersection determines the actual pump speed.

Unsteady state settings

For the computation of the pump during unsteady state there are five possibilities to choose from in the property “Drive type”

Running	The pump speed and the QH curve will be kept constant, throughout the complete simulation time. Value for “polar moment of inertia” is not used
Trip	The pump speed and QH curve will be maintained until the user specified “trip time”. Thereafter the motor torque will be zero and the pump will trip according to equation (16).
Speed	The pump speed will be determined by applying the time-speed table (which is specified by the user). Polar moment of inertia is not taken into account during use of the table. If the value for “min speed for tripping” has been undershoot, the speed table is overruled and the pump acts as a tripping pump, see remark 3.
Motorfreq	The pump speed will be computed by applying the transformed speed-torque curve of the motor on basis of the time table for the relative frequency (which is specified by the user)
Level OnOff	Fixed speed pump is switched on if upstream head H1 > “Level On”; speed increases in “Start time” seconds from 0 to “Rated speed” Pump is switched off if upstream head H1 < “Level Off”; speed decreases in “Stop time” seconds from rated speed to 0 Polar moment of inertia is not taken into account This pump type can be used for simple pump operation control; use a Control module for advanced operational control

Remark 1:

It is most logical to apply drive type “Speed” in combination with initial setting “Speed” and drive type “Motorfreq” in combination with initial setting “Frequency” in order to have a smooth transition between the steady state and unsteady state computations. However, different combinations are possible. The user must take care that the pump speeds match between steady and unsteady. Differences will however be solved dynamically: the unsteady state solution process aims at achieving a new balance.

Remark 2:

All "Drive type" types may be controlled by a control loop (C-components).

Remark 3:

To trigger the “trip” action at certain point in time T the minimum speed for tripping can be used. The table value of the speed just before T must be higher than this minimum speed and lower after T.

6.11.2.2 Component specific output

- Pump speed [rad/s]
- Fluid torque [Nm]
- Motor torque [Nm]
- On/Off [-]
- Efficiency [-]
- power [W]
- Energy [J]

6.11.2.3 Actions

How to activate the pump is dependent of the drive type:

Running

If the type is “running” there will be no action applicable. The pump just keeps running and the fixed QH-curve will apply throughout the computation. The speed may be overruled by a control connection (signal line).

Trip

If the drive type is “trip” the input value “trip time” will determine the time at which the driving torque becomes zero and the pump speed will solely be determined by the hydraulic torque. Up to the “trip time” the pump will be running at constant speed, although the speed may be overruled by a control connection (signal line).

Speed

If the drive type is “speed” the action table will be of the time-speed type. In case the initial control is also “speed” the user interface will copy the initial speed into the table. In other cases (init_P1, init_P2, init_Q and motor), the initial pump speed will be calculated by the computational core (STEADY) and it is the user’s responsibility to have the time-speed table starting with the same pump speed. A warning will be issued however in case of a discrepancy. The action table may be overruled by a control connection (signal line).

MotorFreq

If the drive type is “MotorFreq” the action table will be of the type “relative frequency” which is a dimensionless table. The percentage will be used to scale the motor speed-torque curve relative to the y-axis (the speed will be multiplied by the scaling factor) resulting in lower torque values. The action table may be overruled by a control connection (signal line).

Level On/Off

If the drive type is “Level On/Off” the pump acts as a start/stop pump if the upstream head over/undershoot the On/Off values specified. The speed increase/decrease is defined by the ramp parameters.

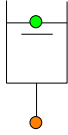
6.11.2.4 Component messages

Message	Type	Explanation
Pump specific speed =.....	Info	Small number (<40) denotes centrifugal pump, large number (>100) denotes axial pump.
Initial pump speed = [rpm]	Info	
Starts	Info	
Pump starting	Info	
Tripping	Info	
Pump stopping	Info	
Q-H curve not monotonously Decreasing	Warning	To prevent instability, the derivative of H(Q) must be negative.
Convergence between fluid and motor torque less accurate than 0.1 % Fluid torque = [kNm] Motor torque = [kNm]	Warning	This will lead to a further convergence process in TRANSIENT, therefore we are not yet in a steady state.
Initial-running status contradictive with control based running status	Warning	For example, according to the initial status the suction controlled pump is running, whereas the measured suction level is below the switch off level.
Unbalance exceeding rpm sensitivity fluid torque = Nm motor torque = Nm	Warning	Within the accuracy of pump speed computation, it was not possible to obtain balance between hydraulic and motor torques. The pump speed is correct however and the torque unbalance will settle during the following time steps (dynamics).
Initial value of action table does not coincide with initial value of pump speed.	Warning	A possibly unintended “jump” in the pump speed will occur upon starting the unsteady state computation.
Motor overloaded	Warning	The hydraulic torque has become larger than the maximum (peak) motor torque. The motor is not longer capable of holding the normal pump speed. This is called a powered trip.
Inconsistent pump speed between steady state results and control system; steady state value will be used	Warning	Message given if discrepancy between steady state speed and control value is less than 1 %
Pump efficiency must be zero if either Q=0 or H=0 in input and non-zero elsewhere	Error	To have a finite torque, this is by definition.

Message	Type	Explanation
Pump efficiency less than 0 or greater than 1	Error	
Motor overloaded Fluid torque = [kNm] Max. motor torque = [kNm]	Error	The intersection between hydraulic and motor torque is at the left of the maximum motor torque.
Inconsistent pump speed between steady state results and control system; Modify input	Error	Message given if discrepancy between steady state speed and control value is more than 1 %
Maximum torque of motor not occurring at pump speed higher than 0.1% of rated pump speed	Error	
Drive type=Motorfreq only in combination with Initial setting=Frequency	Error	
Can't compute pump speed	Error	
Pump table entries < 2	Error	
Operating point left of QH-table'	Error	
Polar mom. of inertia < 0	Error	

6.12 Surge tower, surge shaft

Supplier type

Type label	Symbol	Description	Active
Heat Surgetower		Thermally isolated surge tower. Storage area is independent of liquid height (pressure head)	No

6.12.1 Mathematical model

A surge tower or shaft or open tank (atmospheric pressure at fluid level) supplies fluid to the system according to the differential equation:

$$Q = -A_{tw} dH / dt \quad (1)$$

in which:

Q	=	Discharge from the tower to the system	[m ³ /s]
A _{tw}	=	Storage surface of the tower	[m ²]
H	=	fluid level in the tower	[m]

Insteady state the surge tower does not supply fluid. In that case the equation is:

$$Q = 0 \quad (2)$$

The temperature in the surge tower is modelled as instantaneously mixed. Therefore, the temperature at the connecting node (Temperature 1) can change instantaneously when the flow will reverse from direction. To correctly calculate the tank temperature when fluid is supplied to the tank the bottom level also needs to be defined. No heat transfer between the fluid and the air outside is taken into account (isolated tank).

6.12.2 Surge tower properties

6.12.2.1 Hydraulic specifications

Description	Input	SI-units	Remarks
Bottom level	real	[m]	
Tank area	real	[m ²]	
Temperature at t=0s	real	[°C]	

6.12.2.2 Component specific output

Fluid level [m]

Tank temperature [°C]

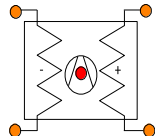
6.12.2.3 Component messages

Message	Type	Explanation
Initial fluid level below bottom of surge tower	Warning	The fluid level is below the bottom level, initially.
Surge tower empty	Warning	The fluid level is below the bottom level,

6.13 Heat pump

6.13.1 Heat pump (class)

Fall type

type label	Symbol	description	active
Heat pump		Heat pump: transports heat from low to high temperature via an internal liquid	Yes

6.13.1.1 Mathematical model

The heat pump is used to exchange heat between two systems (the primary and secondary side) in the opposite direction of the temperature gradient. Figure 1 shows an overview of a heat pump

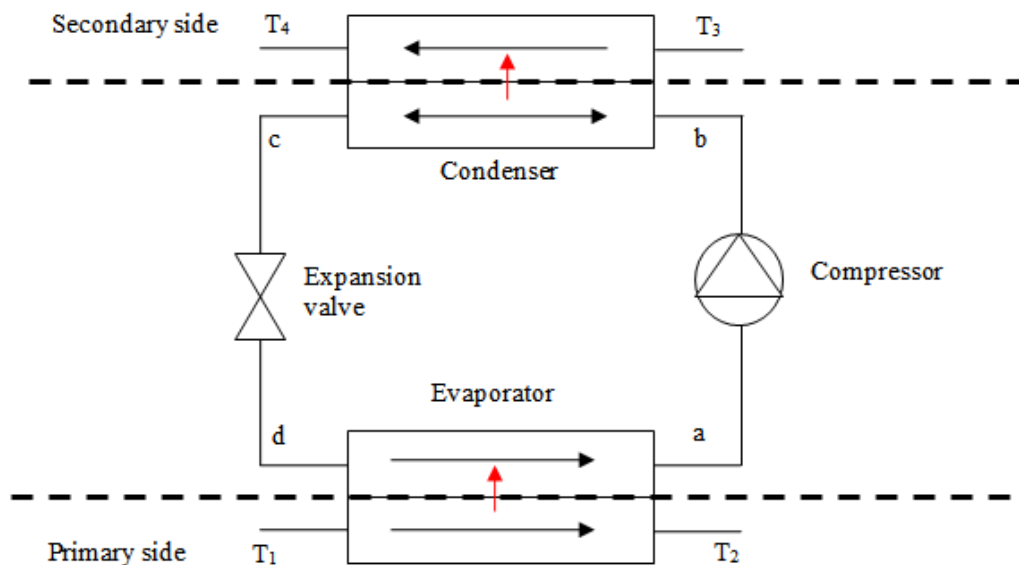


Figure 1 Overview of a heat pump system.

In the Wanda heat pump, the user connects the primary and secondary systems to the heat pump. The internal circuit of the heat pump is included in the heat pump model and does not need to be modelled by the user.

Hydraulic loss equations

The head loss on both the primary and secondary side is given by:

$$\Delta H = CQ^2 \tag{1}$$

In which:

- ΔH = Head loss over heat exchanger [m]
- C = Loss coefficient [s²/m⁵]

Q = Flow rate [m³/s]

This can be rewritten in terms of pressure and mass flow as:

$$\frac{P_i}{g\rho_i} - \frac{P_j}{g\rho_j} + z_i - z_j = gC \frac{\dot{m}^2}{\rho} \quad (2)$$

In which

P _i , P _j	=	Pressure at the two connection points	[Pa]
ρ _i , ρ _j	=	Density at the two connection points	[kg/m ³]
z _i , z _j	=	Height of the connection points	[m]
g	=	gravitational acceleration	[m/s ²]
\dot{m}	=	Mass flow rate	[kg/s]
ρ	=	Density on the upstream side	[kg/m ³]

For both sides also the following continuity equations apply:

$$\dot{m}_1 = \dot{m}_2 \quad (3)$$

$$\dot{m}_3 = \dot{m}_4 \quad (4)$$

Heat transfer

The heat pump transports the heat from primary to secondary side via an internal liquid circuit. This liquid is evaporated in the evaporator extracting heat from the primary side. A compressor raises the pressure of the vaporized liquid. Due to the compression also the temperature of the internal vapour is increased. The vapour condenses in the condenser releasing its heat at the secondary side. Since the temperature has increased the heat is released at a higher temperature and heat is thus transported from a low temperature side to a higher temperature side. Equations are derived for the different parts of the heat pump in the following sections.

Evaporator

The evaporator is a heat exchanger. The heat exchange process can be divided into two parts:

1. Evaporating section
2. Superheating section

In the evaporating section the coolant is evaporated, while the temperature of the internal liquid remains constant. When the coolant is evaporated completely the temperature of the coolant rises, this is called superheating. This superheating only occurs when the evaporator has enough heat exchange capacity.

6.13.2 Evaporating section

The amount of heat needed for evaporation is:

$$Q_{vap} = h_{vap} \dot{m}_{HP} \quad (5)$$

In which:

Qvap	=	Heat required for evaporation	[W]
h _{vap}	=	Evaporation heat of internal liquid	[J/kg]
\dot{m}_{HP}	=	Mass flow rate of the heat pump	[kg/s]

The temperature at the end of the evaporation section in the primary circuit is:

$$T_{endevap} = T_1 - \frac{Q_{vap}}{c_p \dot{m}_1} \quad (6)$$

In which

T _{endevap}	=	Temperature at the end of the evaporation section	[K]
c _p	=	Specific heat of the fluid on the primary side	[J/kg/K]
\dot{m}_1	=	Mass flow rate on primary side	[kg/s]

To determine if there is superheating, the part of the heat exchanger required to reach this temperature needs to be calculated. This can be done via the ϵ -NTU (effectiveness and number of transfer units of the heat exchanger). The effectiveness of the evaporation section of heat exchanger is given by:

$$\dot{\epsilon} = \frac{T_1 - T_{endevap}}{T_1 - T_d} = 1 - e^{-NTU} \quad (7)$$

in which:

$\dot{\epsilon}$	=	Effectiveness of the evaporation section of the heat exchanger	[-]
T _d	=	Temperature at the inflow of the evaporator (point d in figure 1)	[K]
NTU	=	Number of transfer units	[-]

The number of transfer unit can be determined from:

$$NTU = \frac{y_{evap} h}{m_1 c_p} \quad (8)$$

In which:

y _{evap}	=	Part of the heat exchanger required for the evaporation section	[-]
h	=	Heat transfer coefficient	[W/K]

Based on the above given equation, y can be calculated. If the result is greater than unity, the heat exchanger capacity is too small to completely evaporate the internal fluid and an error is given.

If y is one the total heat exchanger is used to evaporate the internal fluid. The temperature on the primary outlet is then equal to T_{endevap}, which is determined from equation 6.

If y is smaller than unity, superheating occurs, in which additional heat is exchanged. The temperature of the internal fluid rises, while the temperature at the primary side decreases.

The additional amount of heat exchanged can be calculated with the ϵ -NTU method. The ϵ -NTU method uses the effectiveness (ϵ) of the heat exchangers. This is given by:

$$\dot{\epsilon} = \frac{\dot{Q}}{\dot{Q}_{max}} = \frac{C_{Cold} (T_a - T_d)}{C_{min} (T_{endevap} - T_a)} \quad (9)$$

In which:

\dot{Q}	=	Heat transfer from primary to heat pump side in superheating section	[W]
\dot{Q}_{max}	=	Maximum heat transfer from primary to heat pump side in superheating section	[W]
C_{cold}	=	Flow thermal capacity on the heat pump side	[J/K/s]
C_{min}	=	Minimum flow thermal capacity from both sides	[J/K/s]
T_a	=	Outflow temperature at the heat pump side	[K]

The effectiveness is also given by:

$$\dot{\epsilon} = \frac{1 - e^{-NTU*(1-RC)}}{1 - RCe^{-NTU*(1-RC)}} \quad (10)$$

In which:

NTU	=	Number of transfer units	[-]
RC	=	Heat capacity ratio	[-]

The number of transfer units is given by:

$$NTU = \frac{y_{supheat} h}{C_{min}} \quad (11)$$

In which:

$y_{supheat}$	=	Part of the heat exchanger used for superheating	[-]
---------------	---	--	-----

The heat capacity ratio is given by:

$$RC = \frac{C_{min}}{C_{max}} \quad (12)$$

Furthermore it holds that:

$$y_{evap} + y_{supheat} = 1 \quad (13)$$

To calculate the temperatures at the outflow of the evaporator the energy balance between primary (1-2) and the heat pump side is needed. This is given by:

$$\dot{m}_{hp} c_{php} T_d - \dot{m}_{hp} c_{php} T_a + \dot{m}_1 c_{p1} T_{endevap} - \dot{m}_2 c_{p2} T_2 = 0 \quad (14)$$

The heat flux of the evaporator is given by:

$$Q_{HP} = \dot{m}_1 c_{p1} T_1 - \dot{m}_2 c_{p2} T_2 \quad (15)$$

The above set of equation can be used to calculate the heat transfer to the internal fluid at the evaporator side.

Compressor

The outflow temperature of the compressor, which is the inflow temperature of the condenser, is given by:

$$T_b = T_a R_c^{\frac{n-1}{n}} \quad (16)$$

In which:

Rc	=	The compression ratio = Pb/Pa	[-]
n	=	polytropic exponent = Cp/Cv	[-]
Pa,b	=	Absolute pressure at in- and outflow of compressor, location a resp. b	[Pa abs]
Ta,b	=	In- and outflow temperature of compressor, location a resp. in figure 1	[K]

The power used by the compressor can be calculated from:

$$Q_c = \frac{n}{n-1} \frac{m_{HP} P_a}{\rho_a} \left(R_c^{\frac{n-1}{n}} - 1 \right) \quad (17)$$

In which:

Qc	=	Net Power supplied by compressor at the given speed	[W]
----	---	---	-----

The pressure at the inflow (Pa) is the condensation pressure of the evaporator. With the pressure ratio specified by the user the pressure at the outlet can be determined. The outlet temperature is calculated with equation 16.

Condenser model

Based on the results of the compressor it is first checked if the saturation temperature is reached. The saturation temperature is determined on base of the pressure at the inlet of the condenser. If the temperature at the inlet is higher than the saturation temperature, the condenser starts with de-superheating to decrease the temperature towards the saturation temperature. If the inlet temperature is equal to the saturation temperature then the condenser starts with a condensing section. If the temperature is lower a warning is given since the coolant is already condensed in the compressor.

The heat transferred and the length of the de-superheating section can be calculated with the ε -Ntu method, see section on the evaporator. The heat transferred in this section is given by:

$$Q_{desup} = c_{pb} \dot{m}_{hp} (T_{sat} - T_b) \quad (18)$$

In which:

Qdesup	=	Heat exchange in the de-superheating section	[W]
c _{pb}	=	Coolant specific heat	[J/kg/K]
T _{sat}	=	Saturation temperature of the internal fluid	[K]

The temperature at the secondary side at the end of the de-superheating section is:

$$T_{3desup} = \frac{Q_{desup}}{c_{p3} \dot{m}_3} + T_3 \quad (19)$$

The efficiency of this section is:

$$\dot{\varepsilon} = \frac{T_3 - T_{3desup}}{T_3 - T_b} = \frac{1 - e^{-NTU*(1-RC)}}{1 - RCe^{-NTU*(1-RC)}} \quad (20)$$

this equation can be used in combination with equation 11 to determine the section of the evaporator required for de-superheating.

The de-superheating section is followed by the condensing section. The heat transfer and length of the section can be calculated in the same manner as for the evaporation section. The last part is the superheating section, which is also the same as for the evaporator. For the condenser it holds that:

$$y_{desuph} + y_{cond} + y_{suph} = 1 \quad (21)$$

In which:

y _{desuph}	=	length of de-superheating section	[-]
y _{cond}	=	length of condensing section	[-]
y _{suph}	=	length of superheating section	[-]

Finally there is a throttling valve in the heat pump circuit which decreases the pressure of the liquid towards the saturation pressure at the inlet of the evaporator.

For the calculations the following parameters need to be set:

1. Evaporator inlet temperature, this remains constant during the simulations
2. Pressure ratio of the condenser
3. Mass flow rate in the heat pump

Based on the mass flow rate and the pressure drop over the throttling valve a loss coefficient is calculated from:

$$C = \frac{(P_{compout} - P_{compin})}{\dot{m}_{mhp}^2} \quad (22)$$

The C-value remains constant during the simulations and is used to calculate a new pressure drop based on the actual mass flow. The actual mass flow is determined from equation 16. This assumes that the hydraulic losses over the heat exchangers are negligible compared to the head loss over the throttling valve.

6.13.2.1 Hydraulic specifications

description	Input	unit	range	default	remarks
C-value condenser	Real	[s ² /m ⁵]			
C-value evaporator	Real	[s ² /m ⁵]			
Heat transfer coefficient condenser	Real	[W/K]			
Heat transfer coefficient evaporator	Real	[W/K]			
Coolant	List				see remark
Temperature evaporator	Real	[°C]			
Initial pressure ratio	Real	[-]			
Mass flow rate heat pump	Real	[m ³ /s]			

Remarks:

The properties of the coolant inside the heat pump are determined with the Refprop library (<http://www.nist.gov/srd/nist23.cfm>). In Wanda the following selection of coolants is included:

- 1 Water
- 2 Ammonia
- 3 CO₂
- 4 Butane
- 5 Isobutane
- 6 R22
- 7 R134a
- 8 R407c
- 9 R410A

6.13.2.2 Component specific output

Generated heat flux [W]

Gross power supply [W]

Coefficient of Performance [-]

6.13.2.3 H-actions

None

6.13.2.4 Component messages

Message	Type	Explanation
Primary temperature in evaporator too low to enable heat exchange.	W	The calculated temperature at the primary side drops below the evaporation temperature and hence not all heat can be exchanged
Heat transfer coefficient too low to exchange all evaporation heat.	W	The heat transfer coefficient of the evaporator is too low to enable transfer all heat required for evaporation of the internal fluid.
Primary temperature in condenser too high to enable heat exchange	W	The temperature on the primary (outside) the heat pump is becoming higher than the condensing heat of the internal fluid and thus not all heat can be exchanged.

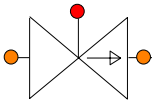
References

[1] Mills, A.F. Basic heat and mass transfer

[2] Moran, M.J. and Shapiro, H.N., Fundamentals of engineering thermodynamics

6.14 Valve

Fall type

Type label	Symbol	Description	Active
Heat Valve		Control or block valve with choice out of four predefined DELTARES standard head loss characteristics or user specified characteristics; several initial settings can be used for flow or pressure balancing	Yes

6.14.1 Mathematical model

6.14.1.1 Pressure loss coefficients

The discharge characteristic indicates the relation between the flow Q through the valve and the pressure loss Δp across the valve as a function of the valve position. This relation is expressed in a discharge coefficient K_v or C_v and a loss coefficient ξ .

The coefficients are derived from the general equation for a Newtonian flow through a restriction in a pipeline under cavitation free circumstances:

$$\Delta p = \xi \frac{\rho_1 v_1^2}{2}$$

in which:

Δp	=	net pressure difference across the valve	[N/m ²]
ξ	=	loss coefficient of the valve	[-]
ρ_1	=	upstream density of the fluid	[kg/m ³]
v_1	=	upstream velocity	[m/s]

In practice the discharge coefficients K_v is often used:

$$K_v = \frac{Q}{\sqrt{\Delta p}}$$

with:

K_v	=	discharge coefficient	[m ³ /h/ $\sqrt{\text{bar}}$]
Q	=	flow	[m ³ /h]
Δp	=	net pressure difference across the valve	[bar]

In words: the discharge coefficient K_v denotes the flow in m³/h which flows through a valve at a pressure difference of 1 bar.

Apart from K_v , C_v is also defined as a discharge coefficients for American units.

$$C_v = \frac{Q}{\sqrt{\Delta p}}$$

with

C_v	=	discharge coefficient	[USGM/ $\sqrt{\text{psi}}$]
Q	=	flow	[US gallons/min]
Δp	=	net pressure difference across the valve	[psi]

The relation between ξ , K_v and C_v is as follows:

$$\xi = 1.6 \cdot 10^9 \frac{1}{K_v^2} \cdot D^4$$

$$K_v = 0.865 C_v$$

A valve is characterised by $\xi = f(\theta)$ or $K_v = f(\theta)$ or $C_v = f(\theta)$.

θ denotes the dimensionless valve opening. θ ranges from 0 to 1 (in SI-units, or 0 = 100 % in percentage annotation)

- $\theta = 0$, valve is closed.
- $\theta = 1$, valve is completely open.

The different discharge characteristic are always translated to the following equation:

$$\Delta H = a \xi Q_1 |Q_1|$$

in which:

ΔH	=	$H_1 - H_2$ in positive flow direction	[m]
H_1	=	upstream head	[m]
H_2	=	downstream head	[m]
a	=	$1 / (2 g A_r^2)$	[s ² /m ⁵]
ξ	=	loss coefficient	[-]
Q_1	=	discharge upstream	[m ³ /s]

The discharge characteristic may be defined by one of Deltares' standard characteristics (See Hydraulic specifications) or by a user-defined discharge characteristic. If the valve position does not coincide with a tabulated position, interpolation must be performed to obtain the discharge coefficient for intermediate valve positions. The standard characteristics and the user-defined ξ characteristic are interpolated logarithmically according to the following equation:

$$\theta(z) = z \cdot \theta_1 + (1 - z) \cdot \theta_2$$

$$\xi(\theta(z)) = \xi_1^z \cdot \xi_2^{1-z}$$

The user-defined K_v and C_v characteristics are interpolated such that K_v or C_v values are interpolated linearly:

$$\frac{1}{\sqrt{\xi(\theta(z))}} = \frac{z}{\sqrt{\xi_1}} + \frac{1-z}{\sqrt{\xi_2}}$$

in which:

θ_1, θ_2	=	tabulated valve positions	[-]
z	=	fraction, which defines intermediate valve position ($0 < z < 1$)	[-]
$\theta(z)$	=	intermediate valve position	[-]
	=	loss coefficient at θ_1, θ_2	[-]
$\xi(\theta(z))$	=	interpolated loss coefficient	[-]

If the valve is closed the governing equations is:

$$Q_i = 0$$

To calculate the temperature the following equation is used:

$$\dot{m}c_{p1}T_1 - \dot{m}c_{p2}T_2 + fr \cdot Q_{gen} = 0$$

In which:

\dot{m}	=	mass flow rate	[kg/s]
c_{pi}	=	specific heat at connection point i	[J/kg/K]
T_i	=	temperature at connection point i	[K]
fr	=	fraction generated heat to fluid	[-]
Q_{gen}	=	generated heat by friction	[W]

The generated heat is given by:

$$Q_{gen} = \xi \frac{\dot{m}^3}{2\rho^2 A^2}$$

6.14.1.2 Cavitation

Cavitation depends on the pressure conditions around the valve. Usually these pressure conditions are defined by a pressure relation. Several different definitions are use in the industrial standards.

In WANDA the factor X_f is used, according the German VDMA standard.

$$X_f = \frac{\Delta p}{p_1 - p_v}$$

in which:

X_f	=	pressure ratio	[-]
Δp	=	net pressure difference across the valve	[N/m ²]
p_1	=	absolute pressure upstream of the valve	[N/m ²]
p_v	=	vapour pressure of the fluid	[N/m ²]

The pressure ratio depends of the valve opening: $X_f = f(\theta)$

The pressure ratio X_f is only calculated for positive flow; for negative flow $X_f = 0$.

If the cavitation characteristic is specified, the program calculates the pressure ratio in the system and warns the user if it exceeds the available value as defined in the characteristic.

Note:

In other standards (ISA, BS, IEC) the pressure ratio X_T is used:

$$X_T = \frac{p_1 - p_2}{p_1} = \frac{\Delta p}{p_1}$$

In some standards (e.g. ISA) the Thoma number σ is used. $\sigma = \frac{p_1 - p_v}{\Delta p}$

in which:

σ	=	pressure ratio	[-]
Δp	=	net pressure difference across the valve	[N/m ²]
P_1	=	absolute pressure upstream of the valve	[N/m ²]
p_v	=	vapour pressure of the fluid	[N/m ²]

The relationship between X_f and σ is: $\sigma = \frac{1}{X_f}$

Another definition for the Thoma number is based on the downstream pressure p_2 :

$$\sigma_2 = \frac{p_2 - p_v}{\Delta p}$$

where $\sigma = 1 + \sigma_2$ and $\sigma_2 = \frac{1}{X_f} - 1$

6.14.2 Valve properties

6.14.2.1 Hydraulic specifications

Description	input	SI-units	Remarks
Characteristic type	Standard Kv Cv Xi		
Standard type	Buttrfly Ball Gate Gate_sqr		if char.type = Standard, default = Buttrfly
Kv characteristic	table	[m ³ /h/√bar]	if char.type = Kv
Cv characteristic	table	[USGM/√psi]	if char.type = Cv
Xi characteristic	table	[-]	if char.type = Xi
Inner diameter	real	[m]	
Initial setting	Position P_upstream P_downstr Mass flow		
Initial position (open)	real	[-]	0 = closed 1= open If init_set = Position
Initial upstream pressure	real	[N/m ²]	If init_set = P_upstream
Initial downstream pressure	real	[N/m ²]	If init_set = P_downstr
Initial mass flow	real	[kg/s]	If init_set = Mass flow
Check cavitation	Yes/No		
Cavitation table	table		If check cavitation=Yes
Fraction gen. heat to fluid	real	[-]	Default = 1 (100%)

Deltares standard characteristics**Butterfly valve**

θ	ξ
0.000	1.0E+10
0.010	10000000
0.025	1700000
0.050	140000
0.075	23000
0.100	6000
0.125	2400
0.150	1150
0.200	440
0.250	195
0.300	97.5
0.400	31.0
0.500	13.8
0.600	5.80
0.700	2.40
0.800	1.00
0.900	.420
1.000	0.150

Ball valve

θ	ξ
0.000	1.0E+10
0.015	900000
0.025	350000
0.050	40000
0.075	9500
0.100	2750
0.150	650
0.200	270
0.300	79.5
0.400	30.0
0.500	13.8
0.600	6.1
0.700	2.7
0.800	1.03
0.900	0.14
1.000	0.01

Gate valve

θ	ξ
0.000	1.0E+10
0.0025	270000
0.025	2850
0.050	625
0.075	270
0.100	140
0.150	58
0.200	31
0.300	11.5
0.400	5.35
0.500	2.55
0.600	1.27
0.700	0.67
0.800	0.355
0.900	0.188
1.000	0.100

Square gate valve

θ	ξ
0.000	1.0E+10
0.0025	249000
0.050	850
0.075	370
0.100	195
0.150	82
0.200	45
0.300	17.8
0.400	8.2
0.500	4.0
0.600	2.1
0.700	0.95
0.800	0.39
0.900	0.09
1.000	0.001

6.14.2.2 Component specific output

- Valve position (open) [-]
- Pressure ratio X_f system [-]
- Generated Heat flux [W]

6.14.2.3 Actions

An activated valve can be opened or closed. How the valve opens or closes is arranged via a θ -time relation in tabular form.

An example:

Valve action table

Time (s)	Position (-)
0	1.
0.5	0.
10.0	0.
12.0	1.
14.0	0.5
18.0	0.5

Note: the position unit depends on the setting made in menu Units

The valve closes linearly in 0.5 s. It remains closed until 10 s. Then the valve opens again in 2 s and starts to close directly until $\theta = 0.5$ at 14.0 s. From then on the valve remains in that position.

6.14.2.4 Component messages

Message	Type	Explanation
starts in open phase	Info	
starts in closed phase	Info	
Opens	Info	
Closes	Info	
Initial valve position below minimum position; minimum table value used	Warning	
Initial valve position above maximum position; maximum table value used	Warning	
Valve position truncated to input range	Warning	
Indifferent valve position (zero discharge and zero dH) max. open position taken	Warning	
Valve characteristic table entries not in between [0-1]	Error	The input table is not correct
Valve resistance in max. open position too large to obtain prescribed state	Error	The desired flow rate is too large for this valve with the calculated pressure drop.
Valve resistance in min. open position too small to obtain prescribed state	Error	

Initial setting not physically realistic Discharge opposite to pressure drop	Error	Check your model or chose different settings.
--	-------	---

Cavitation in steady state not allowed'	Error	
---	-------	--

6.15 Converters



Converter

Fall type

type label	description	active
Converter liquid to heat	Converter to enable the connection of liquid component to heat components	Yes

6.15.1 Mathematical model

Liquid to Heat

The converter component converts the core quantities of one domain to the core quantities of another. This enables the coupling of different domains. No hydraulic losses are taken into account over the converter.

The basic equations for the liquid to heat converter are:

$$Q\rho = \dot{m}$$

$$\rho gH - \rho g z = P$$

in which:

Q	=	Flow rate of liquid connect point	[m ³ /s]
ρ	=	Density of fluid	[kg/m ³]
\dot{m}	=	Mass flow rate on heat connect point	[kg/s]
g	=	Gravitational acceleration	[m/s ²]
H	=	Energy head	[m]
z	=	Height of connect point	[m]
P	=	Total pressure	[barg]

When the flow direction is from the heat domain towards the liquid domain, the density is known (based on the fluid temperature at the connected heat node).

When the flow direction is from the liquid domain towards the heat domain, the temperature of the fluid can be specified in the component (either via a constant, via an action table, or via the control connection point). The temperature is used to determine the density from the fluid properties table.

6.15.2 Liquid to heat converter

6.15.2.1 Hydraulic specifications

Description	Input	unit	range	default	remarks
Inner diameter	Real	[m]			This is only used to calculate the velocity
Temperature at [t = 0 s]	Real	[°C]			

6.15.2.2 Component specific output

6.15.2.3 H-actions

In transient calculations the temperature can be change over time by using an action table or by control.

6.15.2.4 Component messages

Message	Type	Explanation

7. WANDA Locks

7.1 Locks domain

With Wanda-Locks, operation of locks can be modelled including salt transport. The salt is transported with the fluid in the hydraulic model (advective or barotropic transport) and an additional salt flux (baroclinic transport) can be modelled by using specific control components.

Salt intrusion through ship locks causes quality loss of fresh water. In the redesign of many locks measures are taken to reduce the intrusion of salt. To compare different measures, simulations are required. This simulation tool is created in such a way that different locks can be modelled and different measures can be included.

7.1.1 Core quantities

The lock domain uses the following core quantities:

- total pressure p_t [N/m²]
- mass flow rate \dot{m} [kg/s]
- Salt density S_ρ [kg/m³]

The manometric pressure p , discharge Q , (energy) head H , velocity v and salinity S are calculated (post processing) according to the following set of equations:

$$p = p_t - \frac{1}{2} \rho v^2 \quad (1)$$

$$Q = \frac{\dot{m}}{\rho} \quad (2)$$

$$H = z + \frac{p}{\rho g} + \frac{v^2}{2g} \quad (3)$$

$$v = \frac{Q}{A} \quad (4)$$

$$S = \frac{1000 S_\rho}{\rho} \quad (5)$$

In which:

p	=	manometric (or piezometric) pressure	[N/m ²]
ρ	=	density	[kg/m ³]
Q	=	discharge (volume flow)	[m ³ /s]
A	=	cross sectional area	[m ²]
g	=	gravitational acceleration	[m/s ²]

The salinity, S , in the user interface is displayed in practical salinity unit (psu) as standard unit.

The salinity conversion and density calculations can be based on the formulas of Eckard 1981 (default) or of Unesco 1981. The selection can be made in the physical constants window. The user can also specify the temperature of the water used in the conversion here.

The formulas used for Eckard are:

$$Z_1 = 5890 + 38T - 0.375T^2$$

$$Z_2 = 1779.5 + 11.25T - 0.0754T^2$$

$$Z_3 = 3.8 + 0.01T$$

$$\rho = \frac{1000(Z_1 + 3S)}{0.698(Z_1 + 3S) + Z_2 - Z_3S}$$

which are valid in the range

$$0 < T < 40$$

$$0 < S < 40$$

and in which

T	=	temperature	[°C]
ρ	=	density	[kg/m ³]
S	=	salinity	[psu]

For conversion between salinity and density based on Unesco 1981:

$$A = 8.24493 \cdot 10^{-1} - 4.0899 \cdot 10^{-3}T + 7.6438 \cdot 10^{-5}T^2 - 8.2467 \cdot 10^{-7}T^3 + 5.3875 \cdot 10^{-9}T^4$$

$$B = -5.72466 \cdot 10^{-3} + 1.02227 \cdot 10^{-4}T - 1.6546 \cdot 10^{-6}T^2$$

$$C = 4.8314 \cdot 10^{-4}$$

$$\rho_0 = 999.842594 + 6.793952 \cdot 10^{-2}T - 9.095290 \cdot 10^{-3}T^2 + 1.001685 \cdot 10^{-4}T^3 - 1.120083 \cdot 10^{-6}T^4 + 6.536332 \cdot 10^{-9}T^5$$

$$\rho = \rho_0 + AS + BS^{\frac{3}{2}} + CS^2$$

T	=	temperature	[°C]
ρ	=	density	[kg/m ³]
S	=	salinity	[ppt] (≈[psu])

7.1.2 Overview of lock components

Currently, the components listed below are available in Wanda-Locks.

Type	Symbol	Short description
Boundary		Boundary condition specifying head (pressure) and salinity
Pipe		Pipe including friction model.
Ideal pump		Idealised pump which delivers the requested flow rate independent of the pressure difference over the pump
Valve		Valve with choice out of four predefined Deltares standard head loss characteristics or user characteristics (Cv Kv, Xi). Can be used to model lock gates.
Lock chamber		Completely mixed lock chamber / surge tower with free surface area independent of water level. Salinity of the water can be changed by salt flux through the control connect point.
Converter		Converter to enable the connection of liquid component to heat components

In addition to the components mentioned above, there are two control component which are used to model the baroclinic transport.

Type	Symbol	Short description
Lock Exchange		Component calculating the baroclinic salt flux due to lock exchange from one lock chamber to another based on water level and salinity. See e.g. Shin et al., 2004

Type	Symbol	Short description
Lock Exchange v2		Like the "Lock Exchange" component, but capable of taking the flushing discharge through a lock head into account.
Lock Exchange v3		Like the "Lock Exchange v2" component, but capable of taking the ship's water displacement into account.

7.1.3 Component properties

All input and output properties are displayed in the property window (selection property window: Shift-F11). The property window can be divided in 5 parts:

Part of property window	Properties
General input data	Name, comment, keywords etc.
Component specific input data	Type dependent
Action table	Type dependent for active types only (actions)
General output data	Messages, default ZOUTLEK output variables
Component specific output data	Type dependent (including pressure loss, temperature loss, mass)

The visibility of the properties in the model schematic can be managed using the checkboxes in the upper part of the Mode & Options window in the "Model" menu.

Default output properties

The default output properties (for each connect point) of the lock components are:

- Discharge [m³/s]
- Head [m]
- Pressure [N/m²]
- Density [kg/m³]
- Salinity [kg/m³]
- Salinity psu [psu]
- Mass flow rate [kg/s]
- Salt flux [kg/s]
- Average salt flux [kg/s]
- Average water flow [kg/s]

Some properties are type dependent. They are described for each component type separately.

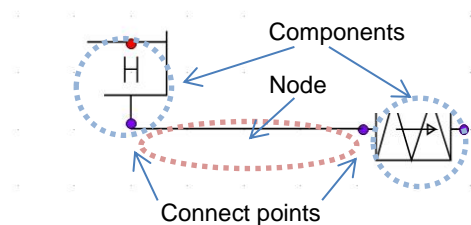
7.1.4 Action tables

Transients in the system are caused by changes in the flow conditions. These changes can be initiated by activate components in the system (e.g. valves). Not all components in a system are active.

An action is described using a control parameter specified as a function of time. The control parameter depends on the component type. In Wanda, a control parameter function can be specified through an action table in the properties of the component or by connecting a control system, see Chapter 5 of the Wanda manual: "Control Components". The control system overrules the component specific action table, although most components need the parameter value to be consistent with the control value at $t = 0$ s.

7.1.5 Lock nodes

A lock node connects two or more Lock components. In the diagram this Lock node is represented by a connection line, connected to the hydraulic connect point of a Locks component (purple dot, see figure below). In case more than two connect points have to be connected the Locks node consists of a master and one or more slave connection lines, thus forming a tree. For more information about connection lines, see "Connecting components" on page 32.



Zoutlek node type

Type label	Description
Locks node	Default Locks node for connecting Locks components in WANDA-Locks
Locks node init p	Locks node defining initial pressure.
Locks node init z	Locks node defining initial salinity.
Locks node init pz	Locks node defining initial pressure and salinity.

The latter node types, where an initial value is prescribed by the user, is only necessary when the node is not connected to either a lock chamber or boundary, and when there is no flow through said node. For example, a node between two fully closed valves will need a value for both the initial pressure and initial salinity.

7.1.6 Mathematical model

Pressure definition for Locks node

In the node the core variable p_t (total pressure) is calculated from a set of equations as explained in "Implementation" on page 147. The head H is derived from this head according Bernoulli (see "Bernoulli's theorem" on page 138).

The manometric pressure in the connect points of the adjacent components are derived from the pressure of the node. If the adjacent component has a diameter property (e.g: Valve, Pipe), the velocity v can be determined and thus, the velocity head can be taken into account, according to:

$$p = p_t - \frac{1}{2} \rho v^2$$

Because the diameters of the adjacent components may differ or the elevation of the centreline (in case of PIPE) may differ, the local pressure in the individual connect points may differ. The manometric pressure shown for the node is the lowest connect point pressure.

Salinity definition for Locks node

Salt transport in the Locks node is calculated based on a continuity equation for the salt mass. The balance takes all salt fluxes connected to the Locks node into account. For n components the equation is as follows:

$$\sum_{i=1}^n \frac{S_{\rho_i} \dot{m}_i}{\rho_i} = 0$$

In which:

S_{ρ}	=	Salinity	[kg/m ³]
\dot{m}	=	mass flow	[kg/s]
ρ_i	=	Density	[kg/m ³]

The salinity of the Locks node itself is defined by setting the salinity of the outflow (flow out of the node) equal to the salinity of the node itself. This means that the salinity in the Locks node is modeled as completely mixed.

Cavitation in Locks node

If the pressure drops below the vapour pressure, cavitation will occur. Cavitation in a Locks node is only taken into account if the Locks node is connected to a pipe.

Default output properties

Each type of Locks node has a few standard output properties. These are:

- Salt density [kg/m³]
- Salinity [psu]
- Total Pressure [N/m²]
- Void fraction (cavitation) [-]

7.2 Salinity boundary

Pressure and salinity boundary condition

Supplier type

type label	description	active
Sluice bound	Boundary prescribing head and salinity	Yes

Sign convention for suppliers is as follows:

- + into the system
- out of the system.

The salinity is prescribed only when fluid is supplied to the system. If the flow is directed out of the system, the salinity is taken from the salinity in the connected node.

7.2.1 ZL_BOUND

A ZL_BOUND prescribes the head and salinity in a certain point of the system when supplying fluid to the system. If the flow is directed towards this component the head is a function of pressure and salinity in the connected node.

The head specified is the total energy head and, because the velocity in the reservoir is zero, equal to the hydraulic head and equal to the level in the reservoir. Both the head and the salinity are constant during the steady state calculation. During the unsteady state calculation the head can vary in time according to the Action table or a connected Signal line and the salinity may vary according to a control table or an input file (CASE_LBD###.csv).

To achieve the specified head the ZL_BOUND has to supply or consume fluid to or from the system. The discharge will be determined by the continuity equation for the H-node the ZL_BOUND is connected to.

7.2.2 Mathematical model

As the core quantity for the Locks domain is the total pressure, the input value for the head and salinity is translated to a pressure and that pressure is used in the calculations:

$$P = \rho g (H - z_{node})$$

In which:

H	=	Input value for the head	[m]
z_{node}	=	Height of the connected node	[m]
ρ	=	Density (depends on salinity)	[kg/m ³]
g	=	Gravitational acceleration	[m ² /s]

7.2.2.1 Hydraulic specifications

description	Input	unit	range	default	remarks
Head at t = 0 [s]	Real	[m]			
Salinity	Select	[-]	Constant Varying		
Constant salinity	Real	[psu]	0 - 50	0	
Salinity time table	Table	[-]			
Salinity input file (CASE_LBD###.csv)	String	[-]			

7.2.2.2 Component specific output

description	unit	remarks
Head	[m]	The actual water level as prescribed by this component (as opposed to Head 1, which is the head computed from pressure and salinity in the connection point)
Salinity	[psu]	The actual salinity as prescribed by this component (as opposed to the Salinity 1, which is the salinity in the connection point)

7.2.2.3 H-actions

The hydraulic actions can be the following:

- Variable salinity through time table
- Variable head through action table or Control connection

7.2.2.4 Component messages

Message	Type	Explanation
Initial head from input template not the same as head at t=0.0s in action table; table value will be used.	Warning	
Inconsistent value at t=0 [s] between steady state results and control system; steady state value will be used.	Warning	The control value differs less than 1% with the steady state results.
Inconsistent value at t=0 [s] between steady state results and control system; modify input.	Error	The difference (see above) is more than 1%, which is not acceptable.

7.3 Lock chamber

Fall type

type label	description	active
LCKCMB2	Ideally mixed lock chamber with constant area. Head and Salinity are specified in steady state	No

The LCKCHM2 (Lock Chamber) component specifies the head (pressure) and salinity in the steady state.

During the transient calculation the head and salinity vary in time according to the net inflow into the lock chamber. Using the control connect point of the LCKCHM2 component an additional net (baroclinic) salt transport can be modelled resulting from the lock exchange driven by density currents (see the 'Lock Exchange' control component). The input value at this control point is a salt flux [kg/s].

There is no head difference or 'distance' between both connection points (1-d model of the lock chamber). There are two connection points to ensure that the inflow into the lock chamber is mixed with the water inside before it is flowing out at the other connection point. Any salt or salt water coming into the lock chamber is instantaneously mixed before flowing out at the other connection points.

Although ships sailing through the lock chamber will influence the exchange of water and salt, this is not taken into account.

7.3.1 Mathematical model

In steady state the pressure at a connection node is given by:

$$P_1 = \rho g (H - z_{node})$$

In which:

H	=	Input value for the head	[m]
z_{node}	=	Height of the connected node	[m]
ρ	=	Density (depends on salinity)	[kg/m ³]
g	=	Gravity acceleration	[m ² /s]

In both steady and transient the pressure on both sides is equal given the following equation:

$$P_1 = P_2$$

During transient the flow rate and water level in the lock are related as follows:

$$A \frac{dH}{dt} = Q_1 - Q_2$$

Rewriting this to the core quantities (mass flow, pressure and salinity):

$$\frac{A}{g} \left(\frac{P_{new}}{\rho_{new}} - \frac{P_{old}}{\rho_{old}} \right) = \frac{W_1}{\rho_1} \Delta t - \frac{W_2}{\rho_2} \Delta t$$

During inflow the density (ρ_1 or ρ_2 on connection point 1 or 2 respectively) is based on the salinity in the node. During outflow the density is based on the salinity in the lock chamber itself.

The control point of the LCKCMB2 is used to model baroclinic salt transport. The value in the control point is interpreted as mass of salt per unit time added to (+) or subtracted from (-) the lock chamber.

7.3.2 LCKCMB2

7.3.2.1 Hydraulic specifications

description	Input	unit	range	default	remarks
Head at t = 0 [s]	Real	[m]		-	
Salinity at t = 0 [s]	Real	[psu]	0-50	0	
Chamber area	Real	[m ²]	0.01-1000	-	
Bottom level	Real	[m]		-	

The head and salinity are specified at t = 0 s for steady state only.

The chamber area and bottom level are used to calculate the fluctuations in water level and salinity in time. Therefore, the value of the Chamber Area should represent the actual area of the free surface (used to calculate water level variations) and the value of the Bottom Level should be such that it represents the actual total volume of water present in the lock chamber divided by the value for Chamber Area.

7.3.2.2 (Component specific) output

description	Input	unit	remarks
Chamber head	Real	[m]	Fluid level <i>inside</i> the lock chamber
Chamber salinity	Real	[psu]	Salinity <i>inside</i> the lock chamber
Chamber volume	Real	[m ³]	Volume of water in the lock chamber

7.3.2.3 Component messages

Message	Type	Explanation
Volume below minimum value	Error	The water level has dropped below the bottom

Remarks

This component specifies the head (pressure) and salinity in the connected nodes. Therefore, to avoid singular matrices, this component should not be connected to another component prescribing the head (pressure) and/or salinity without some kind of resistance (valve / pipe / resist) in between.

7.4 Locks valve

VALVE

Fall type

type label	description	active
Sluice valve	Valve for the Locks domain (e.g. the ship lock gates)	Yes

7.4.1 Mathematical model

7.4.1.1 Pressure loss coefficients

The discharge characteristic indicates the relation between the flow Q through the valve and the pressure loss Δp across the valve as a function of the valve position. This relation is expressed in a discharge coefficient K_v or C_v and a loss coefficient ξ .

The coefficients are derived from the general equation for a Newtonian flow through a restriction in a pipeline under cavitation free circumstances:

$$\Delta p = \xi \frac{\rho_1 v_1^2}{2}$$

in which:

Δp	=	net pressure difference across the valve	[N/m ²]
ξ	=	loss coefficient of the valve	[-]
ρ_1	=	upstream density of the fluid	[kg/m ³]
v_1	=	upstream velocity	[m/s]

In practice the discharge coefficients K_v is often used:

$$K_v = \frac{Q}{\sqrt{\Delta p}}$$

with:

K_v	=	discharge coefficient	[m ³ /h/√bar]
Q	=	flow	[m ³ /h]
Δp	=	net pressure difference across the valve	[bar]

In words: the discharge coefficient K_v denotes the flow in m^3 which flows through a valve at a pressure difference of 1 bar.

Apart from K_v , C_v is also defined as a discharge coefficients for American units.

$$C_v = \frac{Q}{\sqrt{\Delta p}}$$

with

C_v	=	discharge coefficient	[USGM/ $\sqrt{\text{psi}}$]
Q	=	flow	[US gallons/min]
Δp	=	net pressure difference across the valve	[psi]

The relation between ξ , K_v and C_v is as follows:

$$\xi = 1.6 \cdot 10^9 \frac{1}{K_v^2} \cdot D^4$$

$$K_v = 0.865 C_v$$

A valve is characterised by $\xi = f(\theta)$ or $K_v = f(\theta)$ or $C_v = f(\theta)$.

θ denotes the dimensionless valve opening. θ ranges from 0 to 1 (in SI-units, or 0 = 100 % in percentage annotation)

- $\theta = 0$, valve is closed.
- $\theta = 1$, valve is completely open.

The different discharge characteristic are always translated to the following equation:

$$\Delta H = a \xi Q_1 |Q_1|$$

in which:

ΔH	=	$H_1 - H_2$ in positive flow direction	[m]
H_1	=	upstream head	[m]
H_2	=	downstream head	[m]
a	=	$1 / (2 g A r^2)$	[s ² /m ⁵]
ξ	=	loss coefficient	[-]
Q_1	=	discharge upstream	[m ³ /s]

The discharge characteristic may be defined by one of Deltares' standard characteristics for valves (See Hydraulic specifications) or by a user-defined discharge characteristic.

If the valve position does not coincide with a tabulated position, interpolation must be performed to obtain the discharge coefficient for intermediate valve positions. The standard characteristics and the user-defined ξ characteristic are interpolated logarithmically according to the following equation:

$$\theta(z) = z \cdot \theta_1 + (1 - z) \cdot \theta_2$$

$$\xi(\theta(z)) = \xi_1^z \cdot \xi_2^{1-z}$$

The user-defined K_v and C_v characteristics are interpolated such that K_v or C_v values are interpolated linearly:

$$\frac{1}{\sqrt{\xi(\theta(z))}} = \frac{z}{\sqrt{\xi_1}} + \frac{1-z}{\sqrt{\xi_2}}$$

in which:

θ_1, θ_2	=	tabulated valve positions	[-]
z	=	fraction, which defines intermediate valve position ($0 < z < 1$)	[-]
$\theta(z)$	=	intermediate valve position	[-]
ξ_1	=	loss coefficient at θ_1, θ_2	[-]
$\xi(\theta(z))$	=	interpolated loss coefficient	[-]

7.4.1.2 Coefficients based on gate dimensions

As the lock gates are known by width and (water) depth and valve characteristics typically are unknown, using standard valve characteristics is not preferred, but should be based on their appropriate dimensions. To this end, the hydraulic resistance of the lock gates and levelling sliders can be expressed in K_v values as follows:

$$K_v = \frac{Q}{\sqrt{\Delta p}} = \frac{\mu A \sqrt{2g\Delta h} \cdot 3600}{\sqrt{\Delta p}} = \mu A \sqrt{2g} \sqrt{10} \cdot 3600$$

in which:

μ	=	Contraction coefficient	[-]
A	=	Discharge opening area	[m ²]

7.4.2 Valve properties

7.4.2.1 Hydraulic specifications

Description	input	SI-units	Remarks
Characteristic type	Standard Kv Cv Xi		
Standard type	Buttrfly Ball Gate Gate_sqr		if char.type = Standard, default = Buttrfly
Kv characteristic	table	[m ³ /h/√bar]	if char.type = Kv
Cv characteristic	table	[USGM/√psi]	if char.type = Cv
Xi characteristic	table	[-]	if char.type = Xi
Inner diameter	real	[m]	
Initial position (open)	real	[-]	0 = closed 1 = open

Deltares standard characteristics

Butterfly valve

θ	ξ
0.000	1.0E+10
0.010	10000000
0.025	1700000
0.050	140000
0.075	23000
0.100	6000
0.125	2400
0.150	1150
0.200	440
0.250	195
0.300	97.5
0.400	31.0
0.500	13.8
0.600	5.80
0.700	2.40
0.800	1.00
0.900	.420
1.000	0.150

Ball valve

θ	ξ
0.000	1.0E+10
0.015	900000
0.025	350000
0.050	40000
0.075	9500
0.100	2750
0.150	650
0.200	270
0.300	79.5
0.400	30.0
0.500	13.8
0.600	6.1
0.700	2.7
0.800	1.03
0.900	0.14
1.000	0.01

Gate valve		Square gate valve	
θ	ξ	θ	ξ
0.000	1.0E+10	0.000	1.0E+10
0.0025	270000	0.0025	249000
0.025	2850	0.050	850
0.050	625	0.075	370
0.075	270	0.100	195
0.100	140	0.150	82
0.150	58	0.200	45
0.200	31	0.300	17.8
0.300	11.5	0.400	8.2
0.400	5.35	0.500	4.0
0.500	2.55	0.600	2.1
0.600	1.27	0.700	0.95
0.700	0.67	0.800	0.39
0.800	0.355	0.900	0.09
0.900	0.188	1.000	0.001
1.000	0.100		

7.4.2.2 Component specific output

- Valve position (open) [-]
- Pressure ratio X_f system [-]

7.4.2.3 Actions

A valve can be opened or closed by using an action table or control connection. How the valve opens or closes is arranged via a θ -time relation in tabular form.

Note: the position unit depends on the setting made in menu Units

An example:

Valve action table

Time (s)	Position (-)
0	1.
0.5	0.
10.0	0.
12.0	1.
14.0	0.5
18.0	0.5

The valve closes linearly in 0.5 s. It remains closed until 10 s. Then the valve opens again in 2 s and starts to close directly until $\theta = 0.5$ at 14.0 s. From then on the valve remains in that position.

7.4.2.4 (Component specific) output

description	Input	unit	Remarks
Valve position (open)	Real	[-]	

7.4.2.5 Component messages

Message	Type	Explanation
starts in open phase	Info	
starts in closed phase	Info	
Opens	Info	
Closes	Info	
Initial valve position below minimum position; minimum table value used	Warning	
Initial valve position above maximum position; maximum table value used	Warning	
Valve position truncated to input range	Warning	
Indifferent valve position (zero discharge and zero dH) max. open position taken	Warning	
Valve characteristic table entries not in between [0-1]	Error	The input table is not correct

7.5 Ideal pump

Fall type

Type label	Symbol	Description	Active
incl. salt transport		Simplified pump model that delivers a specified discharge.	yes

The IDPUMP component specifies the discharge in steady state and transient calculation. It is an idealistic representation of a pump that is always capable delivering the requested discharge.

7.5.1 Mathematical model

In both steady and transient mode the mass flow on both sides is equal given the following equation:

$$\dot{m} = \rho_0 Q_r$$

in which:

\dot{m}	=	mass flow rate	[kg/s]
ρ_0	=	density of fluid	[kg/m ³]
Q_r	=	requested flow rate	[m ³ /s]

Note that this mathematical model uses the density of the fluid defined in the fluid properties window, whereas other components of the Wanda-Locks domain use the density as defined by the Eckard or UNESCO conversion formulas.

Furthermore, the mass flow rate and salinity are conserved by this component:

$$\dot{m}_1 = \dot{m}_2$$

$$S_1 = S_2$$

7.5.2 IDPUMP

7.5.2.1 Hydraulic specifications

description	Input	unit	range	default	remarks
Initial discharge	Real	[m ³ /s]		-	

The initial discharge is at $t = 0$ s. This value is maintained throughout the transient simulation. The setpoint can be overridden by using the control point of the IDPUMP component, which sets the requested flow rate.

Remarks

This component specifies the mass flow on adjacent nodes. The user will therefore have to be careful not to cause physically unrealistic situations, e.g. where the pump would force mass flow through a closed valve. The latter sets the mass flow on adjacent nodes to zero. When two components set a different value on a node, it will result in a singular matrix.

7.6 Salt pipe

Type label	Symbol	Description	Active
Salt pipe		Pressurised pipe with salt transport	Yes

7.6.1 Mathematical model

7.6.1.1 Fluid equations

The PIPE can be considered the primary component in WANDA to simulate the pressure wave propagation during transient simulations. In Wanda-Locks, the PIPE is also used to simulate the advection and radial transfer of heat in the system.

In steady state $dQ/dt = 0$. The PIPE is considered as a fall type, in which the head loss is modelled by the following equations:

$$(p_2 - p_1) + \rho g (y_2 - y_1) + f \frac{1}{32\pi^2} \frac{L}{D^5} \frac{\dot{m} |\dot{m}|}{\rho} = 0 \quad (1)$$

$$\dot{m}_1 - \dot{m}_2 = 0$$

in which:

$p_1 - p_2$	=	pressure loss from H-node 1 to H-node 2	[N/m ²]
\dot{m}	=	mass flow through the pipe	[kg/s]
f	=	friction factor	[-]
D	=	inner diameter	[m]
$y_2 - y_1$	=	height difference between Heat node1 and Heat node 2	[m]
ρ	=	density	[kg/m ³]
g	=	gravitational acceleration	[m/s ²]
L	=	pipe length	[m]
D	=	inner diameter	[m]

WANDA calculates the friction factor, f , iteratively using the Darcy-Weisbach wall roughness, k . In Transient Mode the friction factor is recalculated each time step using a quasi-steady approach.

The wave propagation speed is calculated based on pipe and fluid properties (analogous to the Liquid pipe with “calculation mode” set to “waterhammer”). Details about the wave propagation and the equations involved are discussed in Theory on waterhammer pipes in Chapter 3: Theoretical and mathematical background. The waterhammer equations are given:

Momentum:

$$\frac{1}{A_f} \frac{\partial Q}{\partial t} + g \frac{\partial H}{\partial s} + \frac{fQ|Q|}{2DA_f^2} = 0 \quad (2)$$

Continuity:

$$\frac{1}{c_f^2} \frac{\partial H}{\partial t} + \frac{1}{gA_f} \frac{\partial Q}{\partial s} = 0 \quad (3)$$

Note: only a circular cross section is supported in WANDA Heat.

7.6.1.2 Salt equations

The salt transfer in the pipe is included by dividing the pipe in elements, see figure 1.

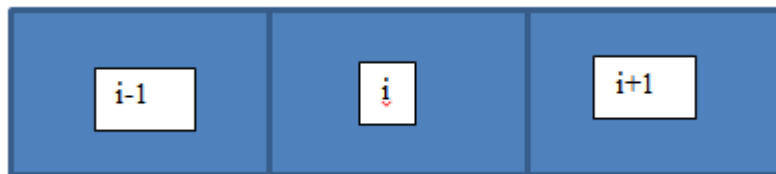


Figure 1, subdivision of pipe in elements for the temperature calculations

The division of the pipes in elements depends upon the type of pipe. For water hammer pipes the the water hammer grid is used. In a rigid column pipe a grid is created based on the advection velocity, which is defined by the user. The size of the grid is calculated from:

$$ds = \frac{L}{n} \quad (5)$$

In which

ds	=	distance between grid elements	[m]
n	=	number of internal grid cells	[-]
L	=	total length of the pipe	[m]

The number of grid cells n is calculated as follows:

$$n = \text{Integer} \left[\frac{L}{v_{adv} dt} \right] \quad (6)$$

v_{adv}	=	Maximum advection velocity	[m/s]
dt	=	Time step	[s]

The maximum advection velocity is set by the user and should be greater than the flow velocity in the pipe for numerical stability. By increasing the time step the number of grid cells can then be reduced. To ensure that steep temperature fronts are correctly conserved a flux limiter scheme is implemented. In Wanda the super bee limiter scheme is implemented see [1].

To increase the stability an additional robust check is built in when a pipe only contains one element. This can occur when there a model consists of long and short pipes. In this case it is checked whether the actual advection velocity (L/dt), which can be lower than the advection velocity specified by the user, is lower than the flow velocity. When his occurs, the entire volume of water in the pipe is refreshed during on time step.

7.6.1.3 Additional losses

If several local losses (like bends, Tee's etc) exist in a pipe section, you can combine them in the pipe model (neglecting the exact position). Combining them can be done in 3 different ways, using either:

- the ξ -annotation (e.g. for one elbow ξ_{elbow} is about 0.2, hence for 5 elbows the combined ξ -value is 1.0)
- the equivalent length annotation (for an elbow L_{eq} is about 15 D)
- a percentage relative to the calculated loss in the pipe itself

7.6.1.4 Geometry

Each PIPE must have a length input and height location. This so-called profile can be defined in several ways:

- Scalar value for length (interpolating height from Heat nodes)
- Length-height profile
- Isometric layout specified with absolute XYZ co-ordinates
- Isometric layout specified with differential XYZ co-ordinates.

Each type of profile input is translated to a longitudinal profile (x-H profile). In case of a scalar value the height of the beginning and the end of the pipe is derived directly from the adjacent Heat nodes.

The input height is the height of the centreline of the pipe (see figure 4 below).

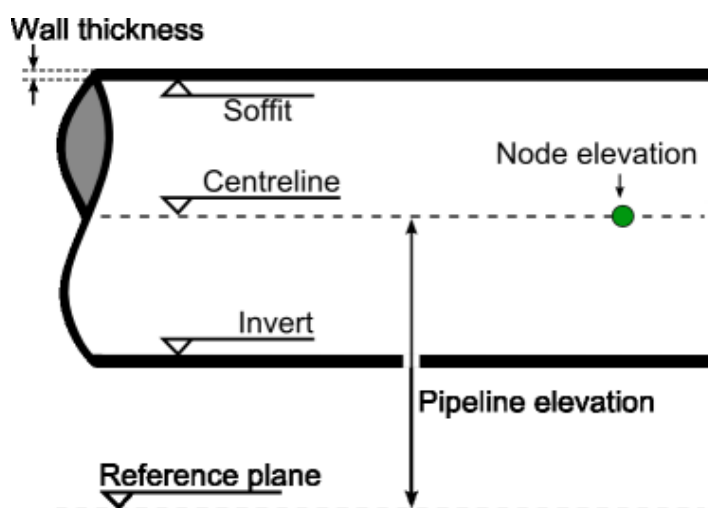


Figure 4: Definition of the height of the pipe and the node elevation.

WANDA compares the input of the beginning and end of the Pipe with the connecting Heat node heights. If there is a difference of more than half the pipe diameter, a

warning is displayed in the property field “H-node height check”. Otherwise, “OK” is displayed.

Below an example of differential XYZ input based on the given isometric is given.

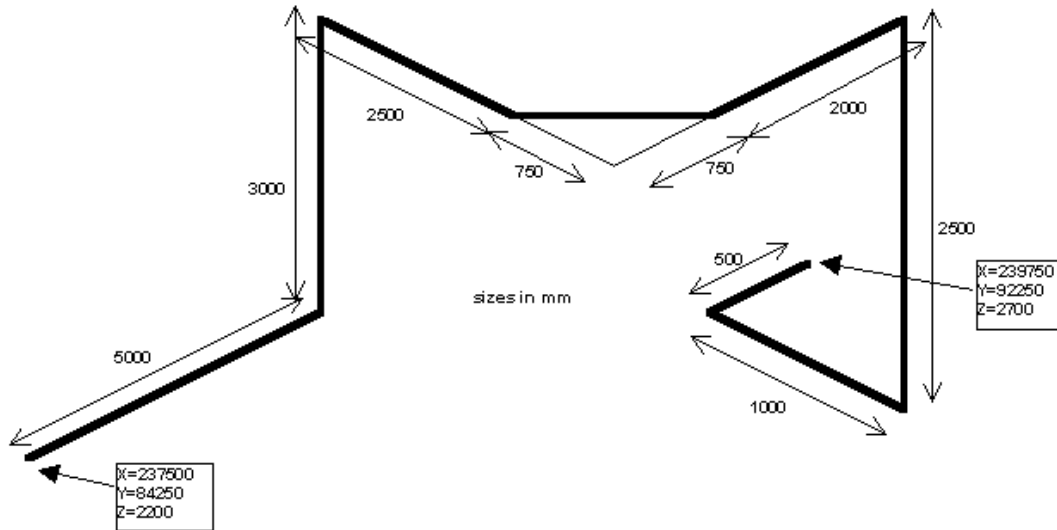


Figure 5: PIPE isometric with differential input.

	x-distance (m)	height (m)	X-abs (m)	Y-abs (m)	Z-abs (m)	X-diff (m)	Y-diff (m)	Z-diff (m)
1	0.0000	2.20000	237.5000	84.25000	2.200000	237.5000	84.25000	2.200000
2	5.0000	2.20000	237.5000	89.25000	2.200000	0.000000	5.000000	0.000000
3	5.0000	5.20000	237.5000	89.25000	5.200000	0.000000	0.000000	3.000000
4	7.5000	5.20000	240.0000	89.25000	5.200000	2.500000	0.000000	0.000000
5	8.5607	5.20000	240.7500	90.00000	5.200000	0.7500000	0.7500000	0.000000
6	10.561	5.20000	240.7500	92.00000	5.200000	0.000000	2.000000	0.000000
7	10.561	2.70000	240.7500	92.00000	2.700000	0.000000	0.000000	-2.500000
8	11.561	2.70000	239.7500	92.00000	2.700000	-1.000000	0.000000	0.000000
9	12.061	2.70000	239.7500	92.50000	2.700000	0.000000	0.5000000	0.000000

Please note the first row starts with the absolute XYZ coordinates followed by the differential coordinates.

The input is translated to real XYZ values and subsequently to a x-H profile. This x-H profile is shown in the location charts.

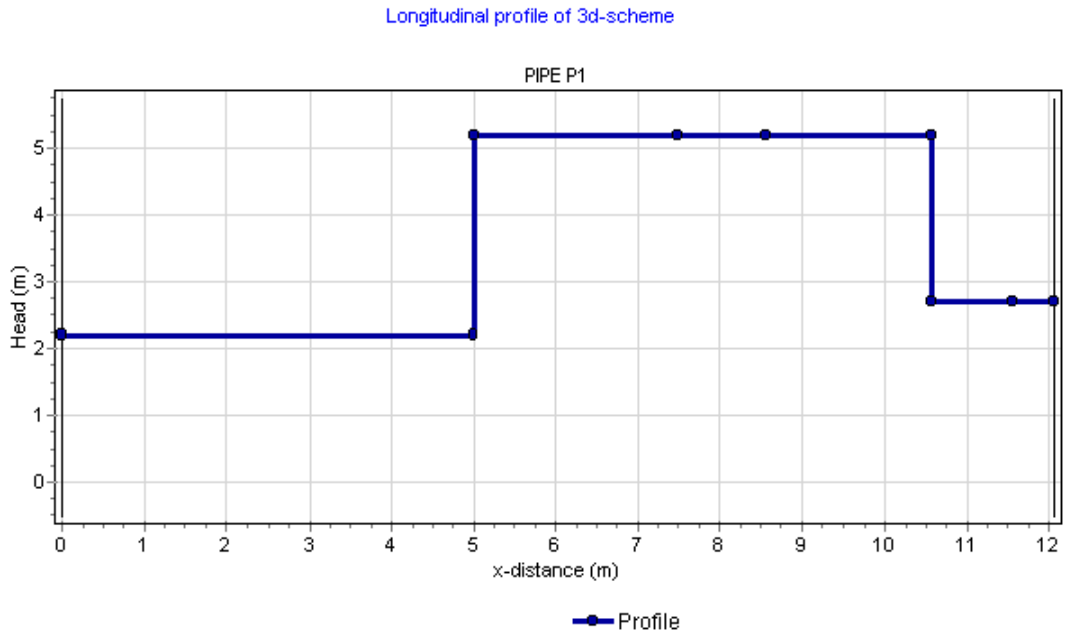


Figure 6: result of XYZ diff specified profile translated to x-H profile

It is not necessary that the pipe profile start with zero. Any positive value is possible. If you split a pipe into two parts, you only have to copy the pipe and delete the unnecessary rows.

	x-distance (m)	height (m)
1	31090	132.910
2	31670	131.510
3	32270	131.010
4	32850	129.810
5	33400	127.210

The location chart (route) starts with the value specified in the first row of the table.

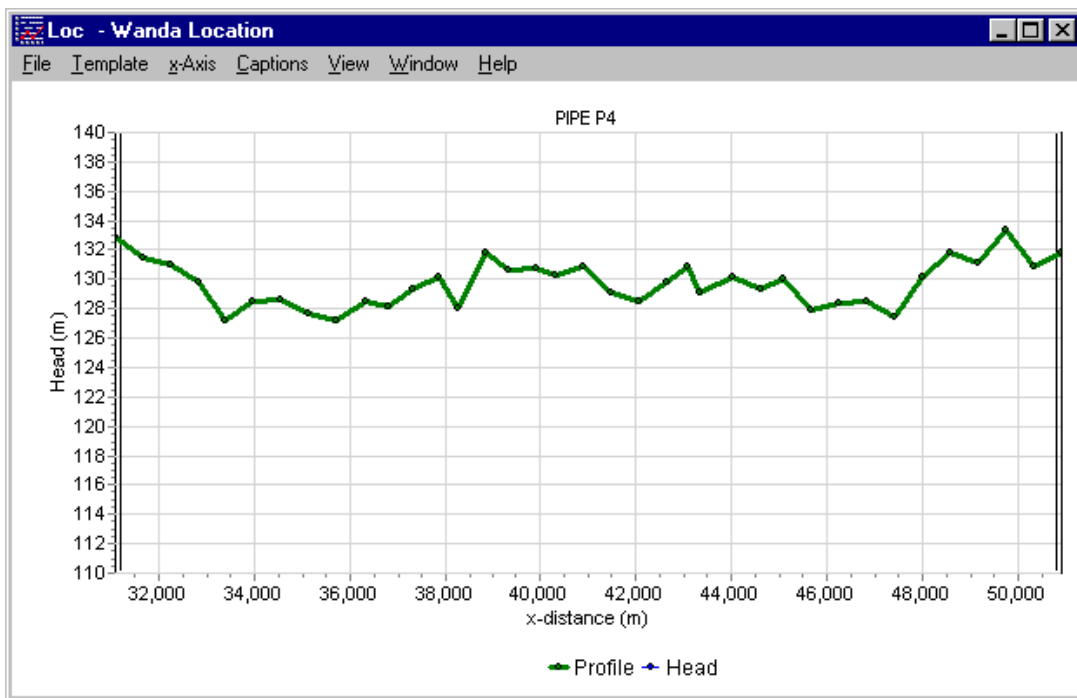


Figure 7: Pipe profile starting with a non-zero begin location

7.6.1.5 Reference

- [1] Dullemond C.P. and Wang A.A. , Lecture Numerical Fluid Dynamics, Course given at Heidelberg University, (<http://www.mpia.de/homes/dullemon/lectures/fluidynamics/>)

7.6.2 Salt pipe

7.6.2.1 Hydraulic specifications

Input properties

Description	Input	SI-units	Remarks
Calculation mode	Waterhammer Rigid column		Only in transient mode, default = Waterhammer
Inner diameter	real	[m]	
Wall roughness	real	[mm]	
Wall thickness	real	[m]	Only in transient mode and with calculation mode Waterhammer
Young's modulus	real	[N/m ²]	Only in transient mode and with calculation mode Waterhammer
Advection velocity	real	[m/s]	Only in transient mode and with calculation mode Rigid column
Geometry input	Length l-h xyz xyz diff	[-]	Default = Length

Description	Input	SI-units	Remarks
Length	real	[m]	if Geometry input = Length
Profile	table		if Geometry input = L-h, xyz or xyz diff
Upper limit pressure	real	[N/m ²]	Only for post processing, not needed for calculation
Lower limit pressure	real	[N/m ²]	Only for post processing, not needed for calculation
Additional losses	None Xi Length Percentage	[-]	Default = None
Additional length	real	[m]	If Additional losses = Length
Local losses coeff	real	[-]	If Additional losses = Xi
Add. loss percentage	real	[%]	If Additional losses = Percentage
Location	real	[m]	If empty, chart button creates a location series If filled in, chart button creates a time series for nearest internal calculation node

Remarks

The location charts of "Pressure" and "Head" shows the specified limits only if they are filled in. It is possible to specify these values after the simulation. If these fields are not visible, first check the "Upper/Lower limits in pipes" fields in the Mode & option window. You may change these values without re-calculating the case. In the heat domain, the change in the maximum head criterion due to changing density is not taken into account.

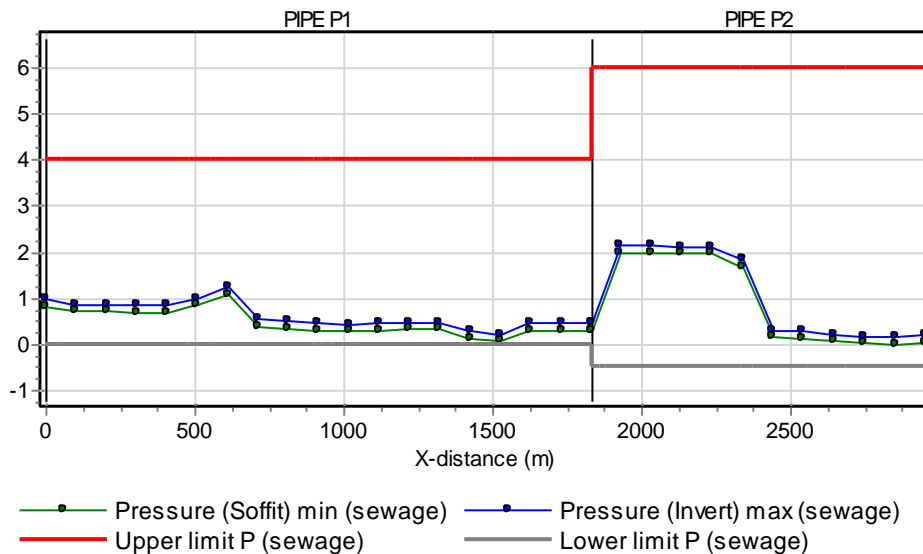


Figure 8: Pressure location chart with additional pressure rating series

Note that the reference for the pressure output (centre line / soffit (top) / invert (bottom)) is defined in the Mode & options window.

The main output properties like "Discharge" and "Pressure" contains vector based output (output values in each calculation node or element in the pipe). That means that the chart button automatically creates a location chart. If you want a time chart in a particular pipe node you have to specify the location in the "Location" field.

To have direct access to the first and last node of the pipe (corresponding with the left and right H-node) you may extend the output property list with these points by checking the field "Connect point output in pipes" in "Modes & Options" (ctrl-m). Pressing the chart button in these additional fields automatically generates a time series chart.

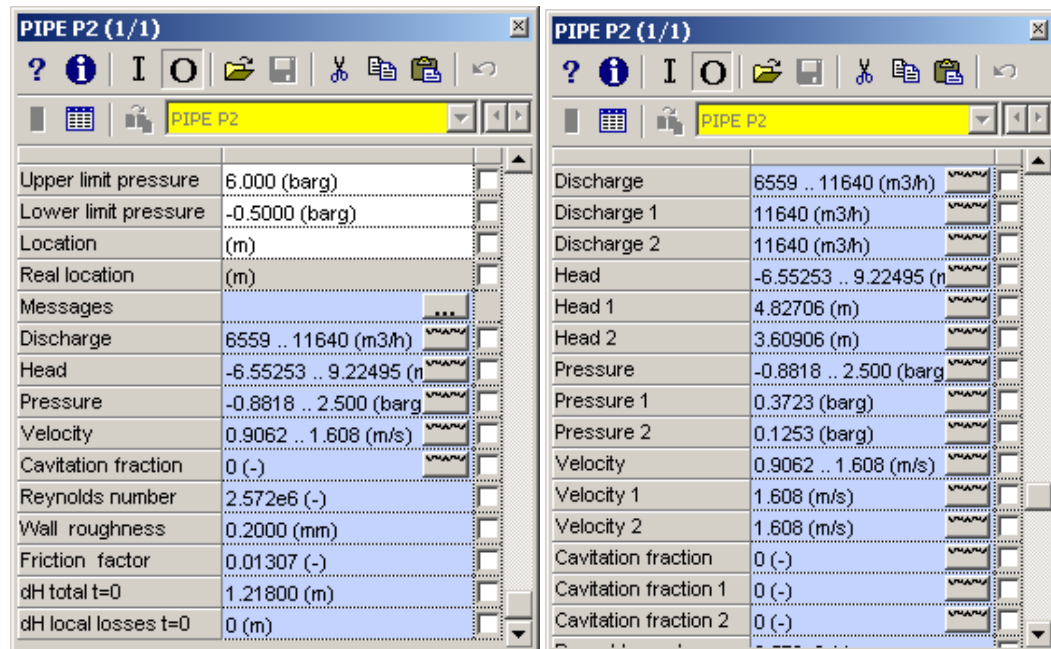


Figure 9: Two appearances of output properties window depending on Mode & Options setting "Connect point output in pipes".

7.6.2.2 Component specific output

Output	Description
Wave speed [m/s]	Wave propagation speed based on fluid properties and pipe properties
Pipe element count [-]	Amount of elements of same length in which pipe is divided.
Adapted wave speed [m/s]	Wave speed is adapted in such way that a integer number of elements (minimal 1) fit in the total length
Deviation adapted [%]	Ratio between wave speed and adapted wave speed. Must be less than 25% for a valid computation
H-node height check	Input validation between H-node height with corresponding pipe node elevation. OK means the difference is less than 0,5 D, otherwise "Warning" is displayed.
Real location [m]	If a location is specified the nearest (internal) calculation node will be selected. The real location is the location of this calculation node.

Cavitation fraction [-]	Ratio between cavitation volume and element volume Must be less than 20% for valid range of cavitation algorithm
Friction factor [-]	The friction factor is used in the calculation of the head loss. This factor is calculated iteratively based on the wall roughness.
Kinematic viscosity [m ² /s]	The kinematic viscosity.

7.7 Lock Exchange

Symbols

Purpose

The control component Lock Exchange is the Wanda implementation of the existing lock exchange model by Rob Uittenbogaard (ref. Deltares report 1201226-011-ZKS-0002, dated July 2nd, 2010 and literature on density currents and internal waves, e.g. Shin e.a., 2004). Comparison with measurements and other validation can be found in Deltares report 1209463-000-HYE-0002-v2, dated January 4th 2016. It can be used to model the salt intrusion by baroclinic transport at ship locks and the effectiveness of mitigating measures such as bubble screens.

The model is based on the internal (density) wave equations for a lock exchange problem, which defines the density current velocity as follows:

$$U = Fr \sqrt{g'H} \quad ; \quad g' = \frac{\rho_+ - \rho_-}{\frac{1}{2}(\rho_+ + \rho_-)} g$$

In which:

U	:	Velocity of density current
Fr	:	Froude number for internal flows
H	:	The water depth
g'	:	Reduced gravitation

Using theoretical values for the height over which the lock exchange takes place (h/H) and the internal Froude number (Fr), the initial density current is known (see e.g. Shin e.a., 2004). Multiplication by the actual opening width of the lock gate and salinity difference provides the resulting net salt mass flux:

$$\dot{M}_0 = \eta \cdot \frac{h}{H} \cdot Fr \cdot H \cdot \sqrt{g'H} \cdot \phi B \cdot \Delta S \quad [\text{kg}^3 / \text{s}]$$

In which:

η	:	Reduction factor based on the mitigation measures taken (1 = no reduction)
h	:	Height of density wave
ϕ	:	Opening percentage of the lock gate
B	:	Width of the gate
ΔS	:	Salt density difference over the gate at start of opening

In the equation for the salt flux above:

- The height of the opening, H , is the water level above the level height of the gate sill. The water level is taken as average of the heads at both sides of the gate opening.
- The gate opening width is the total width of the lock gate (B) multiplied by the actual position (ϕ , opening percentage) of the gate.
- The values for g' and ΔS are taken at the time when the lock gate starts to open.

Regarding the salt flux at the opening lock gate there are the following stages:

- State 0: Gate closed, no salt mass flux;
- State 1: Gate open, internal wave travelling towards the opposite (closed) gate with constant velocity. The salt mass flux is taking place with Froude number 0.5 and over half the water depth ($h/H = 0.5$). This state remains active until half of the effective volume of the lock chamber is exchanged.
- State 2: Gate open, internal wave is reflected on the closed gate and travels back towards the open gate. The internal wave is now taking place over a smaller depth than in state 1 ($h/H = 0.347$) and the internal Froude number is increased ($Fr = 0.527$). This state remains until 85% of the effective volume of the lock chamber is exchanged.
- State 3: Gate open, the internal wave has returned to the open gate when 85% of the lock chamber volume is exchanged. For the remaining time, the lock exchange follows the original equation ($h/H = 0.5$, $Fr = 0.5$), but the values of the parameters (g' and ΔS) are now calculated based no longer taken constant

Mitigating measures

Air (bubble) screen

The effective lock exchange can be reduced by use of an air (bubble) screen, resulting in a significantly longer time before the complete lock chamber volume has been exchanged. The optimal air flow rate corresponds to a reduction factor of 0.25 (Wanda will calculate the necessary air flow rate). If the user choses to specify the air flow rate manually, the corresponding reduction factor is calculated automatically. We take q_{int} as the quantity of intruding salt water per unit width of lock, and $q_{int,0}$ as intrusion in the situation without a bubble screen. The reduction factor and air flow rate are then related as follows (see Abraham & v.d. Burgh, 1962):

$$\eta = \frac{q_{int}}{q_{int,0}} = \frac{0.185}{\phi} + \phi^3 - 0.86\phi$$

Which is valid for

$$0.16 < \phi < 0.65$$

Furthermore,

$$\phi = \frac{C_1}{\sqrt{g'H}} \quad ; \quad C_1 = \frac{4}{3} \alpha \sqrt[3]{\frac{g q_a}{(H_{atm} + H_1)/H_{atm}}}$$

Where

- H_{atm} : the atmospheric pressure head (in m water column).
- H_1 : the water depth at the level of air injection.
- q_a : the specific air flow (per m width of the lock gate).
- α : typical values in the range [0.40, 0.47] (see Bulson, 1961).

The specific air flow per meter width of the lock gate is given in (Nm³/s), which is the flow (m³/s) at atmospheric pressure.

Based on the air flow calculated according to the equations given above, the necessary power for air compression can be calculated. In Wanda the needed power is calculated according to the expression for adiabatic compression with a Laplace coefficient of $\gamma = 1.4$:

$$P_{adiab} = \frac{\gamma}{\gamma - 1} p_{atm} Q_a \left\{ \left(\frac{p_1 + p_{loss}}{p_{atm}} \right)^{\frac{\gamma - 1}{\gamma}} - 1 \right\} \text{ [Watt]}$$

In which:

- P_{adiab} : The power used for the air screen
- γ : The Laplace coefficient
- p_{atm} : Atmospheric pressure
- Q_a : Air flow rate
- p_1 : Pressure at level air screen
- p_{loss} : Additional pressure required for orifice

Water screen

A fresh water jet can be used in addition to an air bubble screen to reduce the salt intrusion even further. At current, a water screen can only be applied in combination with an optimal air bubble screen, resulting in a combined reduction factor of 0.15. There is not much (theoretical) background, but measurements in the Stevin Locks show that a fresh water jet of 1.6 m³/s over a width of 14.0 m and depth of 4.7 m is most favourable. Therefore, in Wanda the fresh water discharge needed for the water screen is calculated according to:

$$Q_{fresh} = \frac{1.6}{14.0 * 4.7} W.H$$

In which:

- Q_{fresh} : Required fresh water flow rate
- W : Width of the lock gate
- H : Depth of the lock chamber

Besides helping the effectiveness of the air bubble screen, the fresh water flow resulting from the water screen is reducing the salinity as well by mixing with the water in the lock chamber. This effect should be included in the hydraulic model with for example the ideal pump. The top output channel of the lock exchange is the water flow and can thus be used to control the water screen flow.

Elevated sill

An elevated sill reduces the height over which the lock exchange takes place. This height is reduced by 80% of the sill height above the lock chamber floor. In other words: the effective gate opening height as used in Wanda is equal to the gate opening height increased with 20% of the sill height. The sill height is the height of the sill with respect to the highest bottom of the two chambers connected (char. bottom level) as depicted in figure 1. It is possible to have a sill with a variable height. The user specifies the water level above the sill and the minimum and maximum height of the sill. The salt exchange component automatically calculates the correct sill height and the salt flux based on this.

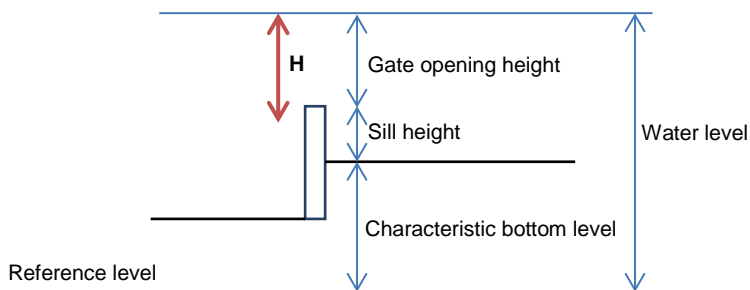


Figure 1 Different elevations and offsets used in the Slat exchange component

Flushing discharge (Lock exchange v2 and v3 only)

A flushing discharge through the lock head when the lock exchange process is running will influence the rate at which salt is exchanged. In Wanda-Locks we assume that the two processes can be superimposed for the first two phases of the lock exchange process, that is the density wave entering the lock chamber and the reflected wave leaving the lock chamber. This is illustrated below. In the last phase, where diffusion occurs, no compensation for a flushing discharge takes place.

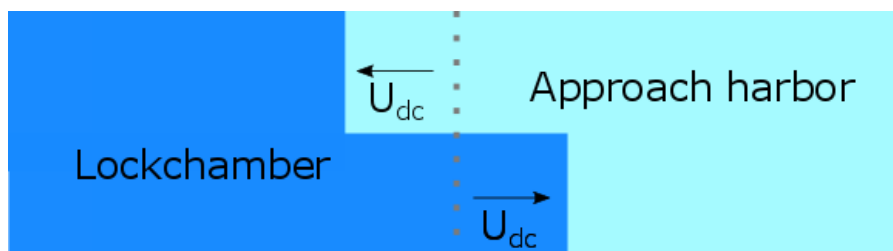


Figure 2 Velocity of density current without flushing discharge

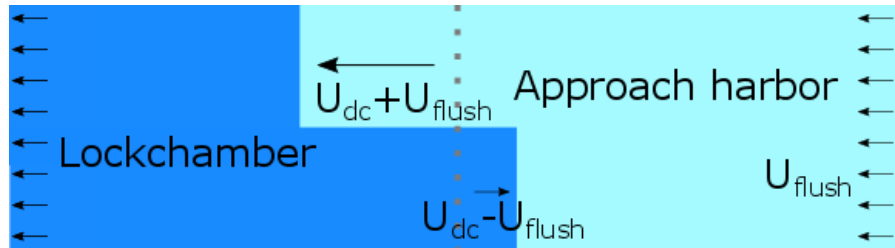


Figure 3 Velocity of density current superimposed with flushing discharge's equivalent velocity

The flushing discharge is averaged over the height of the lock head to get an equivalent velocity. This velocity is subtracted from the velocity of the density current, to get a new resulting velocity:

$$U_{dc} = U_{dc, noflush} - U_{flush}$$

When bubble screens are used, the following set of equations apply:

$$U_{dc} = \eta U_{dc, unprotected} - U_{flush}$$

For comparison with measurements of this approach (and others) to incorporating the flushing discharge, we refer to Deltares report 1220106-001-HYE-0002-v2, dated January 4th, 2016.

Note that, because the baroclinic and barotropic are completed separated in the current implementation of Wanda-Locks, the Lock Exchange component will output zero when the flushing velocity is equal to or larger than the density current's velocity. This is because the advective transport of transport (which does not take into account the density current), already takes care of all transport of water and the accompanying salinity.

Ship's displacement (Lock exchange v3 only)

When a ship enters the lock chamber, it effectively pushes water out of the lock chamber into the approach harbour. This effect is modelled in a simplified way in Wanda-Locks. Ship's leave the lock entirely when the door is barely open, just before the density current starts. The displacement of ships entering the lock chamber is accounted for when the doors close, just after the density current stops. Note that this process does not result in actual (advective) flows in Wanda-Locks, but only salt transport.

There are many questions about the exact influence of a ship and its impeller on the lock exchange process, but not enough is known to model these accurately. Therefore Wanda-Locks contains only (one of) the major components of the ship's displacement, described below. For further discussion about implementing the ship's displacement we refer to Deltares report 1210765-004-HYE-0001.

When apply the following set of equations for the ship's displacement, taking as an example a ship entering the lock from the left side:

$$V_{AH, left} = A_{AH, left} \cdot (H_{water, left} - H_{bottom, left head})$$

$$\Delta M = (S_{lock} - S_{AH, left}) \cdot V_{ship}$$

in which:

$V_{AH, left}$	=	volume of the left approach harbor	[m ³]
$H_{water, left}$	=	the water level in the left approach harbor	[m] (datum)
$H_{bottom, left head}$	=	the depth of the bottom of the left lock head	[m] (datum)
ΔM	=	the mass of salt that has to be transported	[kg]
S_{lock}	=	salinity of the lock chamber before the doors open	[kg/m ³]
$S_{AH, left}$	=	salinity of the left approach harbour before the doors open	[kg/m ³]
V_{ship}	=	the ship's displacement	[m ³]

The resulting salt flux that then has to be applied on this component's output channel can then be calculated as follows:

$$\dot{M}_{left} = \frac{\Delta M}{\Delta t}$$

$$\dot{M}_{right} = 0$$

In which:

\dot{M}_{left}	=	the mass flux to be applied to the left output channel for a single time step, i.e. the approach harbour in the example	[kg/s]
ΔM	=	the mass of salt that has to be transported	[kg]
Δt	=	the time step of the simulation	[s]
\dot{M}_{right}	=	the mass flux to be applied to the right output channel for a single time step, i.e. the lock chamber in the example	[kg/s]

Internally, Wanda-Locks also immediately updates the salinity of the left approach harbour and lock (unchanged in this example), to set the correct driving salinity difference for the density current which starts in the same time step.

Input parameters (Lock exchange v1 and v2)

The input signals to this component are defined as follows (numbering from left to right):

1. Salinity (left)
2. Head (left)
3. Valve position (optional, 100% open if not connected)
4. Head (right)
5. Salinity (right)

Parameter	input	unit	range	default	remarks
Gate width	Real	m		-	
Sill	Constant variable				Default = Constant
Constant sill height	Real	m		0	If Sill = Constant
Max. sill level	Real	m			If Sill = variable
Min. sill level	Real	m			If Sill = variable
Water level above sill	Real	m			If Sill = variable
Char. surface area	Real	m ²		-	
Char. bottom level	Real	m		-	
Air screen	No Auto Spec. reduc. fac Spec. air flow				
Location air screen	Left Sill Right				Default = Sill
Air flow	Real	m ³ /s			Only if Air screen = Spec air flow.
Reduction factor	Real				Only if Air screen = Spec reduc. fac
Water screen	Select	No Auto		No	Only if Air screen is auto
Start gate open	Real	-	0 – 1	0	Specifies when the salt flux is started.
Pressure loss air screen	Real	Bar			

The input parameter 'Start gate open' can be used to neglect small valve openings (The effective width is 0% at this opening percentage up to 100% of the gate width at 100% opening).

Input parameters (Lock exchange v3)

The input signals to this component are defined as follows (numbering from left to right):

6. Salinity (left)
7. Head (left)
8. Valve position (optional, 100% open if not connected)
9. Head (right)
10. Salinity (right)

Parameter	input	unit	range	default	remarks
Lock chamber	Left Right				Position of lock chamber relative to this lock exchange component

Gate width	Real	m	-		
Sill	Constant variable			Default = Constant	
Constant sill height	Real	m	0	If Sill = Constant	
Max. sill level	Real	m		If Sill = variable	
Min. sill level	Real	m		If Sill = variable	
Water level above sill	Real	m		If Sill = variable	
Surf. area lock chamber	Real	m ²	-		
Surf. area approach harbor	Real	m ²			
Char. bottom level	Real	m	-		
Air screen	No Auto Spec. reduc. fac Spec. air flow				
Location air screen	Left Sill Right			Default = Sill	
Air flow	Real	m ³ /s		Only if Air screen = Spec air flow.	
Reduction factor	Real			Only if Air screen = Spec reduc. fac	
Water screen	Select	No Auto	No	Only if Air screen is auto	
Start gate open	Real	-	0 – 1	0	Specifies when the salt flux is started.
Pressure loss air screen	Real	Bar			

The input parameter 'Start gate open' can be used to neglect small valve openings (The effective width is 0% at this opening percentage up to 100% of the gate width at 100% opening).

Output parameters

Parameter	unit	range	remarks
Output channel 1	-		Salt flux towards the left.
Output channel 2	-		Salt flux towards the right, equals minus channel 1
Air flow (calc)	Nm ³ /s		Only if Air screen = auto, this is the calculated optimum air flow rate in atmospheric m ³ /s.
Water flow	m ³ /s		Only if Water screen = auto, this is the calculated optimum water flow rate
Salt flux	kg/s		Instantaneous salt flux value
Salt load	kg/s		Daily averaged salt flux
Driving salinity difference	psu		The salinity difference used in the equations

Component messages

Message	Type	Explanation
Starting in state 0	Info	The gate is closed at $t = 0$
Starting in state 3	Info	The gate is open at $t = 0$
Switching to state 1	Info	The salt flux is started as the gate opening becomes larger than specified by the input parameter 'Start gate open'.
Switching to state 2	Info	Half of the effective lock chamber volume is exchanged, the internal wave is reflected on the closed gate.
Switching to state 3	Info	The internal wave has returned to the open gate, 85% of the effective lock chamber volume has been exchanged.
Switching to state 0	Info	The lock exchange is stopped, as the gate position becomes smaller than the input parameter 'Start gate open'.
No salinity difference, switching to state 3	Info	If there is initially no salinity difference, the exchange flow will follow the expression as given for state 3.
Level difference over gate greater than 0.15 m, while opening gate	Warning	Large level difference at opening of the gate should be avoided as this has implications for the validity of the lock exchange.
Reduction factor below application area	Warning	Reduction factor becomes smaller than 0.25, which is outside of the validated range
Reduction factor below zero	Error	Negative reduction factor will result in erroneous salt fluxes

Remarks

- Many physical processes are not captured in this lock exchange component, due to lack of validation data. Even the currently implemented methods have tested only on specific locks, which specific dimensions. It is to the user to verify their applicability to other locks of other dimensions.

7.8 Converters

Converter

Fall type

type label	description	active
Converter liquid to locks	Converter to enable the connection of liquid component to heat components	Yes

7.8.1 Mathematical model

Liquid to Lock

The converter component converts the core quantities of one domain to the core quantities of another. This enables the coupling of different domains. No hydraulic losses are taken into account over the converter.

The basic equations for the liquid to lock converter are:

$$Q\rho = \dot{m}$$

$$\rho gH - \rho g z = P$$

in which:

Q	=	Flow rate of liquid connect point	[m ³ /s]
ρ	=	Density of fluid	[kg/m ³]
\dot{m}	=	Mass flow rate on heat connect point	[kg/s]
g	=	Gravitational acceleration	[m/s ²]
H	=	Energy head	[m]
z	=	Height of connect point	[m]
P	=	Total pressure	[barg]

When the flow direction is from the Lock domain towards the liquid domain, the density is known (based on the salinity at the connected node).

When the flow direction is from the liquid domain towards the Lock domain, the salinity of the fluid can be specified in the component (either via a constant, via an action table, or via the control connection point). The specified salinity is used to determine the density from the fluid properties table.

7.8.2 Liquid to lock converter

7.8.2.1 Hydraulic specifications

description	Input	unit	range	default	remarks
Inner diameter	Real	[m]			This is only used to calculate the velocity
salinity at [t = 0 s]	Real	[°C]			

7.8.2.2 Component specific output

7.8.2.3 H-actions

In transient calculations the salinity can be change over time by using an action table or by control.

7.8.2.4 Component messages

Message	Type	Explanation