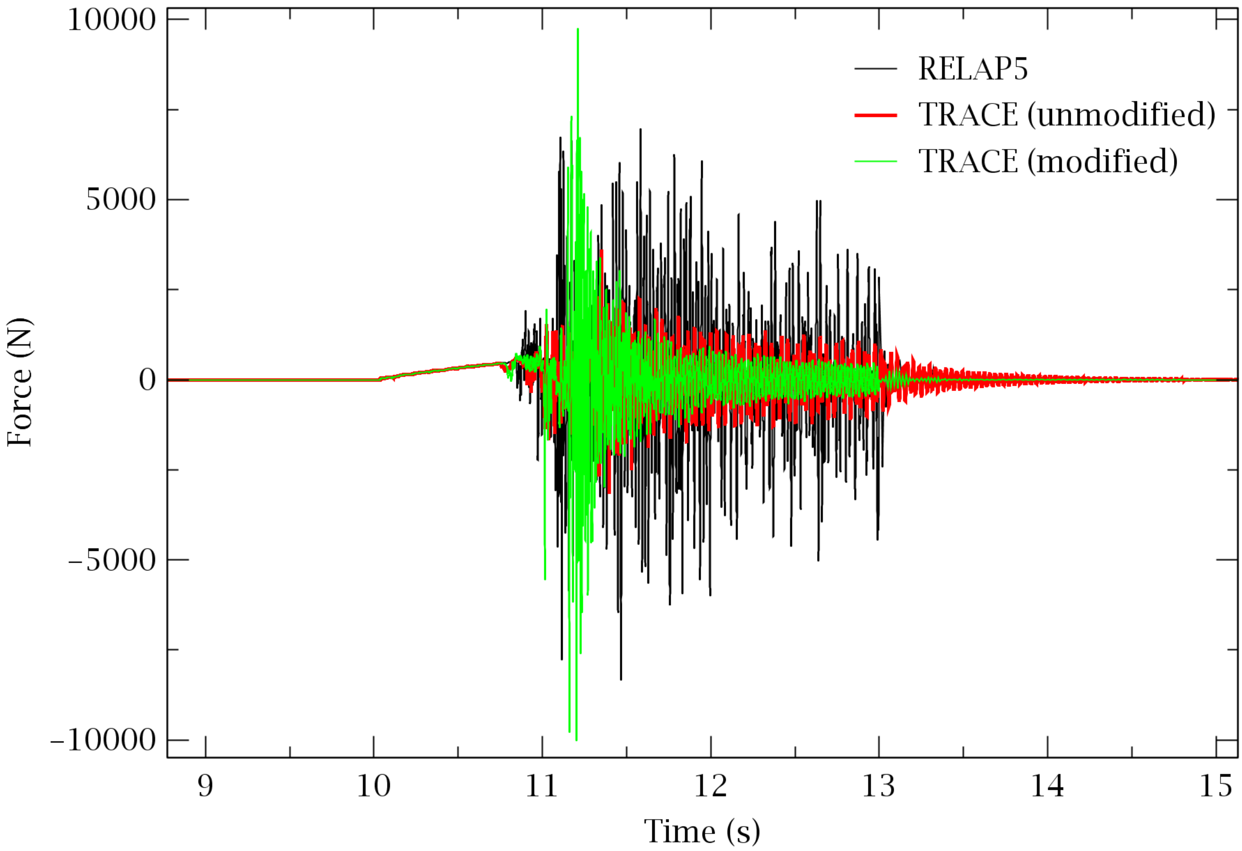
CHALMERS
UNIVERSITY OF TECHNOLOGY



Analysis of Fast Pressure Transients using RELAP5 and TRACE

Master's thesis at Ringhals AB within the Nuclear Engineering program

Joakim Holmström
Anton Lundin

Division of Nuclear Engineering
Department of Applied Physics
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2014

Report no: CTH-NT-296
ISSN: 1653-4662

Report no: CTH-NT-296 Darwin-ID: 2279313
ISSN: 1653-4662

MASTER’S THESIS

Analysis of Fast Pressure Transients using RELAP5 and TRACE

Joakim Holmström

Anton Lundin

Supervisor:

Anna Nyström, Ringhals AB

Examiner:

Anders Nordlund, Chalmers

Division of Nuclear Engineering
Department of Applied Physics
CHALMERS UNIVERSITY OF TECHNOLOGY
SE-412 96 Göteborg
Sweden
Telephone: + 46 (0)31-772 1000

Gothenburg, Sweden 2014

The image on the front page illustrates the calculated dynamic loads on a pipe segment in the residual heat removal system of Ringhals unit 4 following a pump stop transient.

Abstract

Simulations of water hammer transients are important e.g. regarding structural analysis of piping systems in a nuclear power plant. This project aims to validate TRACE with respect to analysis of such transients. Validation is performed against RELAP5 and also, when available, against experimental data. RELAP5 is currently used for such analyses at Ringhals nuclear power plant, and is approved by the authorities for this purpose. Since this system code is about to be phased out in a couple of years, it is of interest to determine whether TRACE may be a suitable successor.

Five main cases are studied, including e.g steam collapse in horizontal and vertical pipes, closing of inertial swing check valves and other valves, as well as some pump modelling. The parameters of interest for this project are mainly process-related ones such as pressures, void fractions and mass flows. In some cases forces on the piping systems are considered. All simulations are carried out in both codes, and the results from TRACE are compared mostly to the corresponding ones from RELAP5. Efforts are then made in TRACE to improve the results to be more similar to RELAP5, e.g. by modifying the interfacial heat transfer coefficients.

It is concluded that TRACE in general works as well as RELAP5 for this type of analysis. There are some differences between the codes though, when it comes to certain modelling techniques and correlations. Further, a number of bugs and other problems exist in the software which need to be fixed. Also, both codes are rather poor at handling steam collapse. This may be due to a too small predicted interfacial area at the moments just before the last steam is about to collapse. In case the interfacial heat transfer is increased in TRACE, the modelling of steam collapse is slightly better in this code than in RELAP5. However, further improvements are required in order to fully capture cavitation water hammers. Finally, it is noted that neither RELAP5 nor TRACE is originally intended for use in analysing this kind of fast pressure transients.

Keywords: TRACE, RELAP5, SNAP, water hammer, vapourous cavitation, condensation, steam collapse, system codes, validation, modelling, interfacial heat transfer, inertial swing check valve

Acknowledgements

We would like to give a special thanks to Anna Nyström, Mikael Fjällborg, Daniel Edebro, Claes Joheman, Maria Jordebo, Anders Nordlund and József Bánáti for your kind support and patience with our many questions during the course of this project. Thanks also to everyone else at RTPS, RTPP, Ringhals and Chalmers who has helped us to carry out and complete the project. We hope our results and findings will be helpful in your continued use of TRACE, and we look forward to seeing what new uses it may have at Ringhals in the future.

J. Holmström, A. Lundin, 2014-06-04

Nomenclature

a	Local speed of sound (m/s)
$\tilde{a}, \tilde{b}$	Generalised force terms included in the water packing detection logic
a_{gf}	Interfacial area per unit volume ($\mathrm{m}^2/\mathrm{m}^3$)
A_c	Cross sectional area (m^2) at the vena contracta
A_{flapper}	Flapper disc area (m^2)
A_j	Cross sectional area (m^2) of cell j
$A_{j+1/2}$	Cross sectional area (m^2) at cell boundary $j + \frac{1}{2}$
A_{pipe}	Cross sectional area (m^2) of a certain pipe
A_{seat}	Valve seat area (m^2)
A_{valve}	Valve opening area (m^2)
$\tilde{c}$	Large constant used in the water packing detection logic
$C_{p\mathrm{f}}$	Specific heat of liquid (J/K)
d_{b}	Average bubble diameter (m)
d_{max}	Maximum bubble diameter (m)
D'	Certain length constant (m) for a bubbly flow
D_H	Hydraulic diameter (m)
f_w	Fanning friction factor
F	Dynamic force or load (N)
F_1, F_2, F_3, F_4, F_5	Different smoothing functions
g	Standard gravity ($= 9.81\ \mathrm{m/s}^2$)
h_{fg}	Latent heat of evaporation (J/kg)
h_{ip}	Interfacial heat transfer coefficient ($\mathrm{W}/(\mathrm{m}^2{\cdot}\mathrm{K})$) for phase p
h_{p}	Specific enthalpy (J/kg) for phase p
Δh	Difference in altitude (m) between two adjacent cells
H	Rated pump head rise (m)

H_{ip}	Volumetric interfacial heat transfer factor (W/(m^3·K)) for phase p
I_{tot}	Moment of inertia (kg·m^2) of the flapper disc
k	'Strength' of the damping torque
k_{p}	Thermal conductivity (W/(m·K)) for phase p
K	Form loss factor
K_{RELAP5}	Form loss factor calculated by RELAP5
K_{TRACE}	$= \frac{\varepsilon_T^2}{\varepsilon} K_{\mathrm{RELAP5}}$
L	Length (m) of a certain pipe segment
m	Mass (kg)
$\dot{m}$	Mass flow (kg/s)
M_{damp}	Time-controlled damping torque (Nm)
M_{mg}	Torque (Nm) due to weight
$M_{\Delta P}$	Torque (Nm) due to pressure difference
M_{str}	Torque (Nm) due to flow velocity
M_{tot}	Total torque (Nm) acting on the flapper disc
N	Number of cells in a certain component
Nu	Nusselt number
P	Pressure (Pa)
P_{v}	Partial vapour pressure (Pa)
$\Delta P_{\mathrm{irrecoverable}}$	Irrecoverable pressure loss (Pa)
$\Delta P_{\mathrm{Joukowsky}}$	Joukowsky pressure head rise (Pa)
Pr_{f}	Liquid Prandtl number
q_{ip}	Heat transfer (W/m^3) for phase p
Re	Reynolds number
Re_{b}	Bubble Reynolds number
t	Time (s)
t_0	Point in time (s) used to indicate a valve closure
t_{damp}	Point in time (s) when the time-controlled damping is deactivated
$\Delta t_{\mathrm{Courant}}$	Material Courant limit (s)
$\Delta t_{\mathrm{critical}}$	Critical Courant Limit (s)
$\Delta t_{\mathrm{critical\ sampling}}$	Critical sampling interval (s)
T	Pipeline period (s)

T_{p}	Temperature (K) for phase p
T_{sv}	Saturation temperature (K) corresponding to the partial vapour pressure
ΔT_{sf}	$= T_{\mathrm{sv}} - T_{\mathrm{f}}$
v	Absolute flow velocity (m/s)
v_0	Steady-state flow velocity (m/s)
v_{fg}	Relative velocity (m/s) between the phases
v_{rel}	Relative flow velocity (m/s) used in the model of an inertial swing check valve
$v_{\mathrm{rel,b}}$	Bubble relative velocity (m/s)
$v_{\mathrm{rel},\infty}$	Relative terminal velocity (m/s) in an infinite medium
v_{reverse}	Backflow (m/s) through a check valve
v_{term}	Bubble terminal velocity (m/s)
$\dfrac{\Delta v}{\Delta t}$	Discrete approximation of the flow acceleration (m/s^2)
We	Weber number
x_{cg}	Distance (m) from the flapper disc attachment to the centre of gravity
x_{fc}	Distance (m) from the attachment of the flapper disc to its centre
$\Delta x,\ \Delta x_j$	Node length (m) for cell j
X_{n}	noncondensable quality

Greek

α_{bub}	Bubble void fraction
α, α_{g}	Void fraction
$\delta\alpha$, $\Delta\alpha_{cut}$, α_{lev}	Limits used in the conditional logic for the level tracking scheme
Γ	Total mass exchange rate (kg/(m^3·s))
Γ_{ip}	Interfacial mass exchange rate (kg/(m^3·s)) for phase p in the bulk fluid
Γ_{w}	Interfacial mass exchange rate (kg/(m^3·s)) between the phases near a wall
ϵ	Surface roughness (m)
ε	$= \dfrac{A_{j+1}}{A_j}$
ε_c	$= \dfrac{A_c}{A_{j+1/2}} = 0.62 + 0.38\varepsilon_T^3$ (correlation for the vena contracta)

ε_T	$= \dfrac{A_{j+1/2}}{A_j}$
θ	Current angle (rad) of the flapper disc
$\theta_{\max}$	Maximum opening angle (rad) of an inertial swing check valve
θ_s	Angle (rad) of the flapper disc to the vertical line at closed valve position
λ_T	Darcy friction factor ($= 4f_w$)
μ_{f}	Dynamic viscosity (Pa·s) of the liquid
ρ	Density (kg/m^3)
$\Delta\rho$	Density difference (kg/m^3) between the liquid and the gas phase
σ	Surface tension (N/m)
ω	Angular velocity (rad/s)
$\dot{\omega}$	Angular acceleration (rad/s^2)

Subscripts

c	Vena contracta
f	Liquid phase
g	Gas or vapour phase
j	Cell index j
$j + \frac{1}{2}$	Right boundary index of cell j
p	Phase index, i.e. liquid (f) or gas (vapour) (g)

Superscripts

n	Current time step index
SCG	Subcooled gas
SCL	Subcooled liquid
SHG	Superheated gas
SHL	Superheated liquid

Table of Contents

Chapter 1

Introduction

A nuclear power plant (NPP) is a large technical system where many scientific disciplines are represented, ranging from fluid mechanics and electrical engineering to neutron physics and water chemistry (the interested reader is referred to available literature on these subjects, e.g. [1, 2, 3]). Many of the sub-systems of a nuclear power plant are designed to increase the electricity production or to improve safety. Many of these subsystems are hydraulic systems with the purpose of transporting water to different parts of the plant, mainly to be used for cooling of the reactor core or other components.

In order to maintain the safety of a nuclear power plant it is among other things important to preserve the structural integrity of piping systems, since without such systems it is not possible to transport water around the plant. One part of the structural verification is the analysis of water hammers, which is the basis for the topic of this project.

1.1 Background

The Swedish Radiation Safety Authority (Strålsäkerhetsmyndigheten, SSM) requires that all Swedish NPPs must perform safety analyses of events, e.g. water hammers, that may challenge the structural integrity of the plant [4]. Today, the system code RELAP5, developed by the US Nuclear Regulatory Commission (NRC), is approved by SSM for use in such analyses. However, within a few years RELAP5 will be phased out [5], and alternative analysis tools must be found. One such tool is TRACE, another system code developed by NRC.

It is of interest for Ringhals nuclear power plant[1] to investigate whether TRACE is a suitable successor to RELAP5 regarding the analysis of fast pressure transients, in order to be able to perform these analyses also in the future. Therefore, this project was initiated in order to validate TRACE, mainly with respect to RELAP5.

The analysis of this kind of fast pressure transients is not only a requirement from the authorities. By accurately predicting the loads on piping systems, e.g. support structures can be designed in an optimal way, which can reduce the costs of such structures.

[1] Ringhals nuclear power plant is where this project has been carried out. It is located in the municipality of Varberg on the West Coast of Sweden.

If calculations lead to very conservative designs, i.e. design with very large safety margins, large amounts of money may need to be spent, without actually improving safety. Hence, it is important to make the calculations as accurate as possible, while still maintaining a degree of conservatism. The focus on steam collapse in this project originates from the fact that both RELAP5 and TRACE are known to handle this phenomenon rather poorly [6, 7, 8]. Also, another limitation is the fact that both codes are one-dimensional. Obviously, real 3D phenomena cannot be fully captured because of this approximation[2].

1.2 Purpose

The purpose of this project is to validate the system code TRACE with respect to fast pressure transients, such as water hammers and steam collapse, and to evaluate whether it is a suitable code for analysing such transients. Validation is performed against RELAP5 simulations, as well as against experimental data where such is available. Possible adjustments or modifications required in TRACE, for obtaining the same or similar results as in RELAP5, are also identified.

1.3 Method

The project is carried out by performing computer simulations with the two system codes RELAP5 and TRACE. Simulation results are compared between the codes and with experimental data. Studied parameters include e.g. pressure, mass flow, void fraction and forces. Models are created in both RELAP5 and TRACE. Also, RELAP5 models are converted to TRACE. Finally a few existing RELAP5 models (not created within this project) are converted to TRACE.

1.4 Outline

The outline of this report is as follows. First, some relevant theory is presented in Chapter 2 in order to get a better understanding of the different cases covered in Chapters 4-8. The purpose of the theory chapter is not to give a complete description about the mathematics and physics behind the investigated phenomena or about the system codes, but to give the reader a broad overview of these topics. In Chapter 3 the general methodology and motivation for the different cases are described, as well as some standard settings utilised for most of the simulations. In Chapter 9 the most important results and findings are summarised and discussed, and the final conclusion of this project is presented in Chapter 10. This last concluding chapter is kept short; all important details are instead gathered in the previous chapter.

[2]TRACE, however, does have a 3D `VESSEL` component [9] for modelling e.g. the Reactor Pressure Vessel (RPV). This is not considered in the project though.

Chapter 2

Theory

This chapter summarises some of the theory and background information relevant for this project. First, a section about the nature of water hammers and steam collapse is presented. Then, a general description about RELAP5 and TRACE is given, followed by a rather extensive section about some of the correlations encountered in the project. The last section is about the inertial swing check valve later used in Chapters 6 and 8.

Please note that the nomenclature used in this and the following chapters is not always the same as the ones used in the RELAP5 or TRACE manuals. The reason for this is simply to have a unified nomenclature throughout this report.

2.1 Water Hammers and Steam Collapse

Significant pressure surges (or water hammers) may arise in piping systems following a rapid event, e.g. a sudden valve closure, a large pipe break or a pump trip [6, 10], at which the momentum of the fluid is abruptly changed. A pressure wave is then initiated and will propagate through the system at the same order of magnitude as the speed of sound, i.e. approximately 1440 m/s in case of water [10]. Often, a characteristic 'bang' is heard that usually sounds like a hammer, hence the name water hammer. A simple example of such an event from everyday life is the abrupt closing of a water tap, which may be heard far away as the pressure wave transmits through the water pipes.

Furthermore, vapourous cavitation can appear in cases when the pressure falls below saturation conditions. Typically, this may occur at the discharge side after a pump trip or in the downstream pipe after a valve closure [11]. Similarly, vapourous cavitation may also be present at the suction side or in the upstream pipe, respectively, following a reflected relief wave. Later on, these steam formations may collapse violently as the propagation wave returns and again increases the pressure above the saturation level. The steam collapse often results in a large pressure pulse that may even exceed the initial head rise caused by the rapid event [12].

Large water hammers can, moreover, severely influence the mechanical integrity of the pipe, its support and of internal components, leading to permanent deformation or rupture. Also, in case of e.g. a rapid closing of a turbine trip valve in the main steam line of a BWR, the water hammers may even cause a power increase in the reactor core [13]. The reason for this is the pressure wave, following the valve closure, that will propagate backwards into the Reactor Pressure Vessel (RPV), causing the steam to collapse, increasing the moderator density and thus the reactivity of the core.

2.1.1 Vapourous Cavitation and Steam Collapse Following the Closing of a Valve

Because the closing of a valve is very central in this project (see e.g. Chapter 4, 6 and 7), this section is dedicated to the cause of the pressure waves and the vapourous cavitation following a valve closure in a horizontal pipe. Two cases are considered. First, the wave propagation and cavitation in the upstream pipe are described when the valve is located downstream between two reservoirs. Second, the corresponding phenomena in the downstream pipe are presented briefly [11]. It should be emphasised that in both cases it is assumed that the initial conditions are such that cavitation in the vicinity of the valve occurs.

In the first case (upstream pipe), consider the general setup shown in Fig. 2.1 in which a steady-state flow velocity is established in a pipe connected to a large water reservoir in each end. Downstream there is a valve located that is about to close instantaneously at the time $t = t_0$, causing the water to accumulate near the closed valve. A high pressure is built up and a pressure wave is sent backwards through the pipe towards the upstream reservoir. The maximum magnitude of this initial pressure pulse is determined by the Joukowsky equation [10]:

$$\Delta P_{\text{Joukowsky}} = \rho \cdot a \cdot |\Delta v| \text{ (Pa)} \tag{2.1}$$

where ρ is the fluid density (kg/m^3), a is the speed of sound (m/s) in the fluid and $|\Delta v| = |v_{\text{final}} - v_{\text{initial}}|$ is the change in velocity (m/s) due to the valve closure. Fig. 2.2 illustrates conceptually the pressure head rise at the time t_0 (point A-B) of the instantaneous valve closure. The time it takes for the pressure wave to return to the closed valve, after being reflected in the reservoir, is determined by the pipeline period T defined as:

$$T = \frac{2L}{a} \text{ (s)} \tag{2.2}$$

where L is the length (m) of the pipe between the reservoir and the valve. The reflection at the time $t = \frac{1}{2}T$ in the reservoir results in a relief wave travelling towards the valve. Meanwhile, the water is accumulating upstream the valve and a high pressure is built up, in turn causing the flow to reverse. When the relief wave reaches the valve at the time $t = t_0 + T$ the pressure falls to the vapour pressure (point C-D). When this occurs, vapourous cavitation forms in the vicinity of the valve. When the wave once again has been reflected in the reservoir it will travel against the valve with a positive sign, increasing the pressure on its way. At some point in time ($\frac{5}{2}T < t - t_0 < 3T$) the pressure gets high enough for the steam near the valve to collapse, which gives rise to a second pressure pulse (point E-F) in addition to the wave originating from the valve closure [12].

A superposition of these two waves may then appear as a short duration pulse when the original wave arrives at the valve for the third time (point G-H). The length and magnitude of this pulse, as well as the time of existence for the vapour[1], depend e.g. on the initial steady-state conditions and of the length of the pipe. The following pressure drop (point I-J) occurs when the reflected relief wave from the steam collapse reaches the valve. A new round of steam formation then follows when also the original relief wave arrives at the valve (point K-L). These cavities will later on collapse (M-N) and the cycle is repeated.

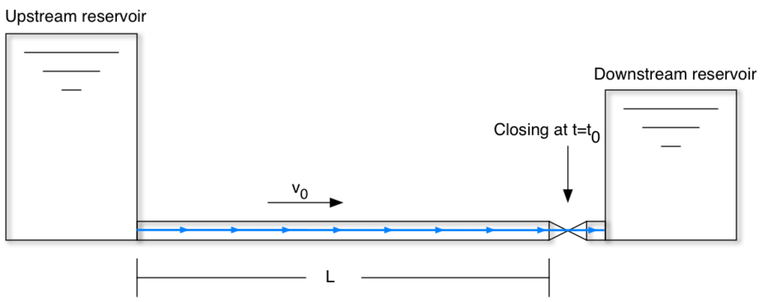


Figure 2.1: Setup and initial conditions for demonstrating vapourous cavitation in the upstream pipe.

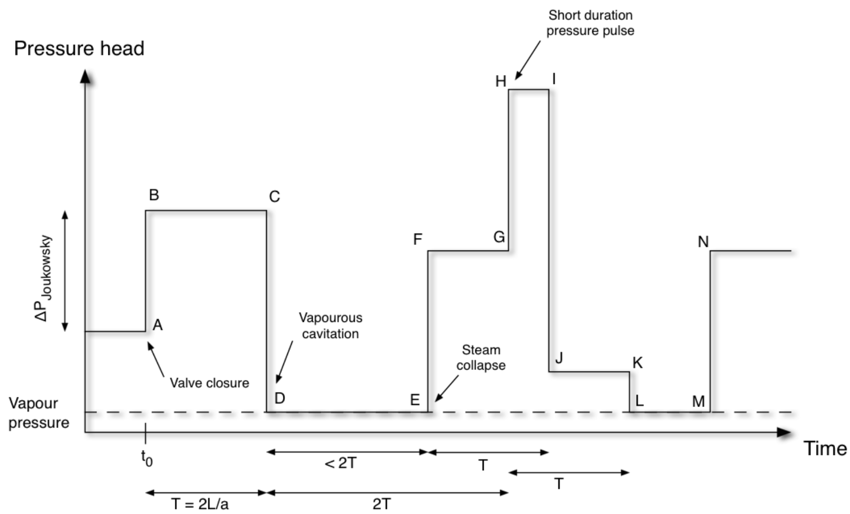


Figure 2.2: Conceptual illustration of the pressure head rise just upstream the valve following the sudden valve closure (see Fig. 2.1). Important moments are highlighted by A-N and are described further in Section 2.1.1 above.

In the second case (downstream pipe), consider Fig. 2.3 in which a certain steady-state flow velocity is established. When the valve is closed instantaneously the water column downstream still has some momentum and will continue to flow, leaving a low-pressure

[1]The interested reader is recommended to look at the graphs in [12], which illustrate how different initial flow velocities affect the time of existence for the vapourous cavities and the magnitude of the short duration pulse. For instance, it is shown that a higher initial flow velocity increases the time before the cavities collapse.

zone near the valve. If the initial conditions are such (as assumed) that the pressure falls to the saturation level, a vapourous cavity is formed [11]. After a while, the water column will come to rest and start to reverse its direction due to the pressure gradient across the downstream reservoir and the pipe. The closing of the valve also gives rise to an initial pressure relief wave that propagates downstream towards the reservoir, in which it is reflected. The pressure wave then returns to the valve, and either reflects in the steam cavity or makes it collapse depending on the state of the water column and the size of the cavity. When the steam collapse occurs, a second pressure pulse is created in addition to the first one. A repeated sequence of steam formation and collapse is then initiated [11].

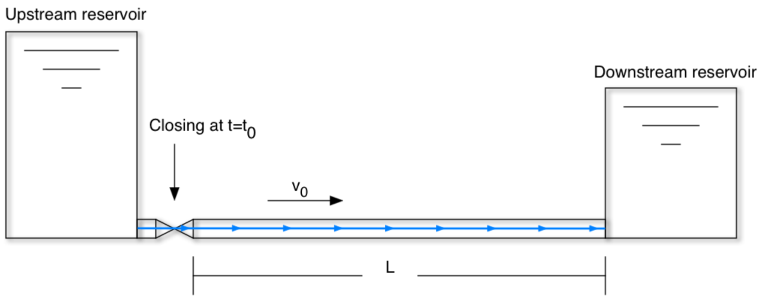


Figure 2.3: Setup and initial conditions for demonstrating vapourous cavitation in the downstream pipe.

In both cases presented above, the cycles of formation and collapse of vapourous cavities sooner or later die out due to frictional effects, Fluid-Structure Interaction (FSI) and other losses [10].

2.1.2 Dynamic Pipe Loads

As mentioned above, the rapid change of momentum due to valve closures, pump trips, etc. may cause severe loads on the pipes. It is therefore important to be able to estimate these loads on the piping system. In 1D codes like RELAP5 and TRACE, this can be achieved by e.g. calculating the dynamic force F (N) on a certain pipe segment in the direction of the flow according to:

$$F = -\frac{\mathrm{d}}{\mathrm{d}t}\int_0^L \dot{m}\cdot \mathrm{d}x \approx -\frac{\mathrm{d}}{\mathrm{d}t}\sum_j \dot{m}_j \cdot \Delta x_j \tag{2.3}$$

where L is the length (m) of the pipe segment considered and $\dot{m}_i$ is the mass flow rate (kg/s) in the pipe cell j with a node length (m) of Δx_j (cf. Fig. 2.5 in Section 2.2.2 below). If the pipe is bent, like e.g. the one in Fig. 2.4, the force is usually calculated separately for the respective straight pipe segments. This is because the loads in the pipe segment after a bend may challenge the structural integrity of e.g. the pipe segment before the bend, due to a force component acting perpendicular to the pipe. For instance, the load on the pipe segment L_4 (see Fig. 2.4) may be relevant for the integrity of segment L_3 and L_5.

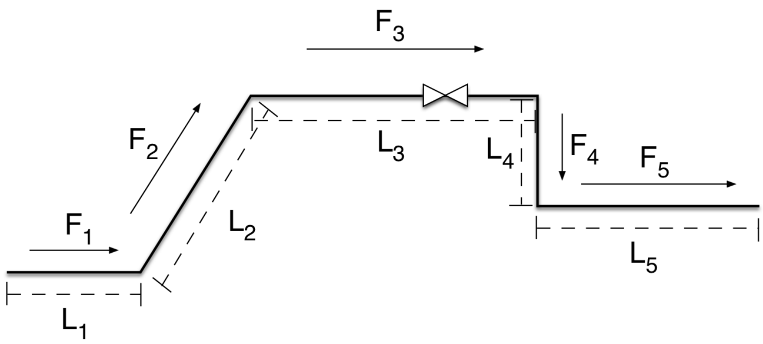


Figure 2.4: A certain pipe divided into a number of straight pipe segments (L_1, L_2, L_3, L_4 and L_5). Usually, the force (F) is calculated on each of these segments using Eq. 2.3.

2.2 Computer Codes

A number of computer codes are available for modelling thermal hydraulic systems. One such code is RELAP5 (Reactor Excursion and Leak Analysis Program), which is a 1D code developed by the Idaho National Laboratory for the US Nuclear Regulatory Commission (NRC).

The first code in the RELAP series was released in the 1960s [14]. RELAP5 was the first code in the series to utilise a nonhomogeneous non-equilibrium model, and the first version was released in the 1980s. The code version used in this project is mainly RELAP5/MOD3.3Patch04. In some cases[2] RELAP5/MOD3.3Patch03 is used. There are no major differences between the two versions though [15]. RELAP5 is currently approved by SSM for use in analyses of fast pressure transients such as water hammer events.

Within a few years the RELAP5 code will be phased out [5], i.e. no further updates will be released. NRCs successor to RELAP5 is TRACE (TRAC/RELAP Advanced Computational Engine), which, as the name suggests, is based on RELAP5 as well as the two other NRC codes TRAC-P and TRAC-B[3]. TRACE was previously known as TRAC-M and development of this code began in 1997 [16]. TRACE, like RELAP5, is using a nonhomogeneous, non-equilibrium model. However, unlike RELAP5, TRACE also has some 3D modelling capabilities[4]. The version of the TRACE code used in this project is mainly TRACE V5.830. In some cases TRACE V5.0 is used. It should be noted that TRACE V5.830 is a BETA version, and that it may be subject to change before the actual release.

The general solution strategy of these two codes is briefly presented in the following sections, together with a brief description of the components used in this project. Also, a short introduction to SNAP (Symbolic Nuclear Analysis Package) is given. This is a graphical user interface that can be used to create models in both RELAP5 and TRACE.

[2]For sensitivity analyses in Chapter 4.

[3]The 'P' denotes PWR and the 'B' denotes BWR.

[4]By the utilisation of a special VESSEL component, which, as mentioned before, is not considered in this project.

2.2.1 Field Equations

The basic approach used by both RELAP5 and TRACE for solving the two-phase field is the utilisation of the conservation equations for mass, momentum and energy, resulting in three PDE:s for each phase to be solved[5]. In the most general form, the conservation of a certain quantity in a control volume (or cell) is determined by the rate of accumulation (time variation), transport by convection and diffusion (spatial variation), and possible source or dissipation terms [17]. In case of mass conservation, the only source term is the mass exchange rate Γ between the phases. The conservation of momentum further includes e.g. the momentum change due to a pressure difference, gravity and frictional losses. The source terms in the energy equation are, in addition to the ones mentioned for momentum, due to interfacial (see Section 2.3.2), wall and direct heat transfer. An example of the latter one is the presence of radioactive decay in the fluid. For the exact formulation of the field equations, and how these are implemented in the respective code, the interested reader is referred to the theory manuals for RELAP5 [14] and TRACE [18].

A number of closure models, either thermodynamic state relations or correlations, are required in order to make the equations solvable. A few of these correlations are presented in Section 2.3.

To be able to solve the partial differential conservation equations they first have to be averaged in both space and time. The next two sections describe briefly how this is done in the codes.

2.2.2 Nodalisation

RELAP5 and TRACE use the same 1D nodalisation scheme with a staggered mesh, with the purpose of dividing a component into a certain number of control volumes. This is conceptually illustrated in Fig. 2.5, with the indexes $j-1$, j and $j+1$ representing the cell centers of three adjacent cells and $j-\frac{3}{2}$, $j-\frac{1}{2}$, $j+\frac{1}{2}$ and $j+\frac{3}{2}$ their respective boundary (junction or edge). Scalar quantities (pressures, temperatures, energies, void fractions, etc.) are defined at cell centers while vector quantities (velocities, mass flows, etc.) are defined on the cell boundaries [14, 18].

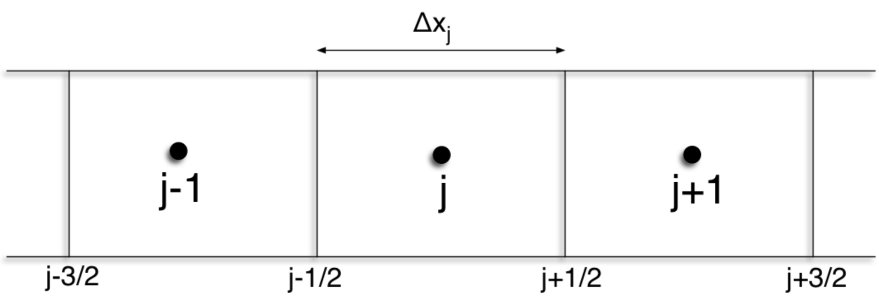


Figure 2.5: Conceptual 1D nodalisation scheme used by RELAP5 and TRACE.

[5] If also e.g. noncondensable gases are present, additional conservation equations are solved for these.

2.2.3 Solution Methods

The differential conservation equations briefly described in Section 2.2.1 are spatially integrated over the control volumes in order to obtain cell-averaged properties. Further, the time derivatives are approximated by their finite difference analogies, which can be evaluated either explicitly, implicitly or by a combination of these two. How this time advancement numerical scheme is implemented in the respective code is summarised below.

2.2.3.1 RELAP5

The default semi-implicit numerical scheme in RELAP5 is, as the name suggests, partially implicit in time [14]. The overall methodology is to use the implicit scheme only when small time constants are expected in order to reduce the computational time. The time step control is based on a rather complex algorithm in which one of the requirements to be fulfilled, for an explicit condition, is the material Courant limit defined as

$$\Delta t_{\text{Courant}} \leq \frac{\Delta x}{v} \text{ (s)} \tag{2.4}$$

where Δx is the cell length (m) and v is the flow velocity (m/s) in the current cell. Thus, a finer nodalisation scheme or a higher velocity results in smaller time steps to be used. The Courant limit is evaluated[6] (before the hydrodynamic calculations) for all cells in the model. The next smallest of these values is then chosen as the current time step size, thus partly violating the Courant limit [14].

There is also a choice of using the so-called nearly-implicit scheme, which may be suitable for slow transients where large time steps are adequate. In order to obtain a stable solution this calculation would have to be fully implicit, which in turn would require a very large number of equations to be solved. A solution to this calculation problem is to use the 'fractional step' method utilised by the nearly-implicit scheme, which reduces the number of equations. The time step size is chosen to be 20 times that of the semi-implicit scheme [14].

2.2.3.2 TRACE

In TRACE, two different schemes are available: the default Stability Enhancing Two-Step (SETS) method and a semi-implicit scheme similar to the one used in RELAP5. The advantage of using the SETS method is that the Courant limit can be eliminated, by using a 'stabiliser step', but with the cost of a higher numerical diffusion compared to the semi-implicit method [18]. Moreover, according to [18], the numerical diffusion reaches a minimum when the time step size equals the material Courant limit defined by Eq. (2.4).

[6] The equation used for calculating the Courant limit is slightly modified by taking the phasic velocities into account (see Eq. (8.1-9) in [14]).

It can also be mentioned that in TRACE V5.830 (BETA), there is a choice of utilising different second-order spatial difference schemes by setting a certain value of the `NAMELIST` variable `SPACEDIF` [9]. However, these are regarded as experimental and can only be used in combination with the semi-implicit method.

2.2.3.3 Recommended Time Step Size

Since the transients investigated in this report usually involve pressure waves propagating with the speed of sound a, the material Courant limit may not be sufficient in order to fully capture the details of these events. Therefore, a different critical time step size $\Delta t_{\text{critical}}$ is preferably used as the maximum allowed time step in those cases. This critical time step is also known as the critical Courant limit and may be defined as:

$$\Delta t_{\text{critical}} = \frac{\Delta x}{a} \text{ (s)}. \tag{2.5}$$

Furthermore, it has been shown that a time step size of $0.1 \cdot \Delta t_{\text{critical}}$ is adequate in fast pressure transients [19]. It is also important to choose the sampling frequency carefully, since important information may be lost if the sampling frequency is too low. On the other hand, if the sampling frequency is too high, the size of the output files may be very large. According to [19] the sampling frequency should not be lower than the critical Courant limit, i.e.

$$\Delta t_{\text{critical sampling}} \leq \frac{\Delta x}{a} \text{ (s)}. \tag{2.6}$$

2.2.4 Flow Regimes

The properties and appearance of a two-phase flow is widely dependent on the type of flow regime, which in turn is determined by e.g. the void fraction, flow velocity, temperature of the fluid and dimensions of the pipe. Since most of the correlations used for solving the two-phase field are flow regime-dependent, it is important for the codes to know what flow regime to choose in a certain situation. The main flow regimes that RELAP5 and TRACE can choose from are bubbly, slug, plug, annular-mist and horizontally/vertically stratified flow. These apply for pre-CHF (Critical Heat Flux) flows. For post-CHF, when the liquid no longer can stay in contact with the hot surface, the flow regimes are 'inverted'. For further details, see [14, 18].

2.2.5 Steam Tables

Both codes utilise built-in steam tables in order to determine the steam and water properties at a certain thermodynamic state.

The default steam table in RELAP5/MOD3.3Patch03 and Patch04 is based on the International Association for the Properties of Water and Steam (IAPWS) formulation from 1995 [14]. The user has the alternative choice of using the old 'H2OOLD' steam table.

The default steam table in TRACE V5.830 (BETA) is based on the IAPWS formulation from 1997, which is basically the same as the one used in RELAP5. The user has the alternative choice to use a curve-fit formulation inherited from TRAC-PF1 [18]. It should be mentioned that the latter option is default in TRACE V5.0 and that the IAPWS type can be chosen by the user.

2.2.6 Component Models

When creating models in RELAP5 and TRACE, a number of pre-defined components are available. A selection of these components relevant for this project are described briefly below. Many other components are available but these are outside the scope of this project, and the interested reader is referred to the manuals of the respective codes [20, 21] for more information regarding this.

2.2.6.1 Junctions

A junction (or edge) is used to connect two computational volumes. Junctions are very similar in RELAP5 and TRACE. The exception is the `SINGLE-JUNCTION` component, which is required in RELAP5 to connect different components. This component is not required in TRACE (although it is possible to use it). Instead, connecting volumes share their end junctions.

For each junction it is possible to specify a loss coefficient (K-factor)[7]. K-factors are described in more detail in Section 2.3.1.2.

2.2.6.1.1 RELAP5

For junctions the `jefvcahs`-flags are available. A full description of what each flag does is available in Volume II of the RELAP5 manual [20]. The only flags which have been modified in this project is the `a`-flag, which affects the abrupt area model, described in detail in Section 2.3.1.2, and the `c`-flag, which affects the choked flow model. The options are `a` = [0] (no abrupt area), `a` = [1] (full abrupt area) and `a` = [2] (partial abrupt area, meaning that the user has to specify the losses), and `c` = [0] (choked flow) and `c` = [1] (no choked flow).

2.2.6.1.2 TRACE

The junction options for TRACE include the possibility to modify the losses across the junction via the `NFF`-variable ('Friction factor correlation option'). This is done e.g. to activate the abrupt area model. The options are `NFF` = [0] (only user-supplied K-factor[8]), `NFF` = [1] (K-factor and friction), `NFF` = [−1] (K-factor and friction and abrupt area change) and `NFF` = [−100] (K-factor and abrupt area).

[7] If desired, it is possible to specify different loss coefficients for forward and reverse flow.

[8] The K-factor (form loss factor) is an irreversible loss coefficient. It is described in more detail in Section 2.3.1.2.

There is also the possibility to use a choked flow model.

2.2.6.2 Pipes

The most commonly used component (in almost any application) is the `PIPE` component. It consists of a number of volumes (computational cells) connected with junctions, which are described in Section 2.2.6.1.

2.2.6.2.1 RELAP5

Fig. 2.5 shows the nodalisation scheme of a general pipe. It should be noted that the first junction is located after the first cell in RELAP5, and that the last junction is located before the last cell; i.e. in a pipe with N cells there are $N - 1$ junctions. The reason for this is that the `SINGLE-JUNCTION` component is used to connect two `PIPE` components (or a `PIPE` component with any other component).

Pipes are bent between cells in RELAP5 (i.e. in junctions). E.g a 90°-bend will consist of one vertical and one horizontal cell. The junction between will be either vertical or horizontal.

The available flags for the volumes in the `PIPE` component (or in almost any component) are the `tlpvbfe`-flags. The flags of interest in this project are the `l`-, `p`- and `e`-flags. The `l`-flag determines whether the level tracking scheme is used, with the options `l` = [0] (level tracking off) and `l` = [1] (level tracking on). The `p`-flag determines whether the water packing mitigation scheme should be used. The options are `p` = [0] (water packing mitigation scheme is on) and `p` = [1] (water packing mitigation scheme is off). The `e`-flag determines whether a non-equilibrium or an equilibrium model should be used. In the first case the two phases (liquid and vapour) are allowed to have different temperatures, whereas in the latter case the two phases have the same temperature. The latter case is considered to correspond to having an infinitely fast interfacial heat transfer rate (see Section 2.3.2). The options are `e` = [0] (non-equilibrium) and `e` = [1] (equilibrium).

2.2.6.2.2 TRACE

In TRACE the first junction is located before the first cell, and the last junction is located after the last cell; i.e. in a `PIPE` with N cells there are $N + 1$ junctions. This means that the `SINGLE-JUNCTION` component is not required in TRACE. Instead, the junctions at the ends of a `PIPE` are shared with any connecting component.

Pipes are by default bent in cell centers in TRACE. E.g. a 90°-bend will consist of only one cell, with the first half being vertical and the second half being horizontal. It is possible to change this using the `NAMELIST` variable `IELV`, which specifies how gravity terms should be entered (the changes are applied to all pipes). More information about this can be found in the TRACE User's manual [21]. Setting `IELV` = [2] means that pipes are bent between cells (i.e. in the same way as in RELAP5).

2.2.6.3 Valves

Valves are used to control the flow rate through a pipe by changing the flow area. This is exactly how they are modelled in RELAP5 and TRACE – as a junction with a variable flow area.

2.2.6.3.1 RELAP5

The `VALVE` component in RELAP5 is a special case of a `SINGLE-JUNCTION` component, for which it is possible to control the flow area. It is possible to model several types of valves using the built-in models in RELAP5, including inertial swing check valves, motor-controlled valves and relief valves. The reader is referred to Volume V of the RELAP5 manual [22] for details on the different valve types. The most flexible valve type is the servo valve, where the valve stem position can be controlled by any control variable. The most common control variable for this purpose is time, but it is possible to use any variable in the model[9].

The same flags as for a junction are used for a `VALVE` component.

2.2.6.3.2 TRACE

The `VALVE` component in TRACE is a special case of a `PIPE` component rather than of a `SINGLE-JUNCTION` component. It is however possible to use valves consisting of only a junction. The typical valve consists of two cells with the actual valve junction (i.e. the junction with variable flow area) located between them (i.e. at the second junction). It is possible to use any number of cells, except for one single cell, and to put the actual valve junction anywhere in the component.

The flow area of the `VALVE` component can e.g. be controlled by a table or by a control variable. It can also be set as constant (in order to model an orifice). Finally, it is possible to change the way that valve losses are modelled using the variable `INTLOSSOFF` (instead of the also available `NFF` variable). The options are `INTLOSSOFF` = [0] (losses calculated by TRACE according to the theory in Section 2.3.1.2) and `INTLOSSOFF` = [1] (losses calculated from a user-supplied form loss table). Details about the different available valve types can be found in Volume I and II of the TRACE manual [9, 21].

2.2.6.4 Pumps

A pump is used to add energy to the fluid in terms of increased pressure. In both RELAP5 and TRACE, this is modelled as a source term in the motion equations. The behaviour of the pump is largely determined by the homologous curves. These are normalised curves (using rated parameters of the pump) which describe the pump head and torque in different flow conditions (normal pump, energy dissipation, turbine and reverse pump) for single-phase flow. There are also similar curves for a degraded pump, i.e. a pump

[9]The inertial swing check valve discussed in Section 6 uses a servo valve with its valve stem position controlled by a large number of control variables.

which works in two-phase flow. A full explanation is outside the scope of this project, and the interested reader is referred to the existing literature on the subject.

The `PUMP` component in RELAP5 and TRACE can be supplied with homologous curves as well as rated values, dimensions, etc. Not all homologous curves need to be supplied, but if the codes during simulation enter a condition for which no curve is given, the simulation will result in an error.

2.2.6.4.1 RELAP5

The `PUMP` component in RELAP5 consists of a single cell in which the source term is applied. It should be noted that connections to other components do not require `SINGLE-JUNCTION`s since these are included in the `PUMP` model. Also, it should be noted that the degraded homologous curves are specified as the difference between the single-phase curves and the two-phase curves.

2.2.6.4.2 TRACE

The `PUMP` component in TRACE can consist of any number of cells except one. The source term is applied at the second junction. The degraded homologous curves, unlike for the RELAP5 `PUMP` component, are specifed directly, rather than as the difference between single-phase and two-phase curves.

2.2.6.5 Boundary Conditions

Boundary conditions are components that are mostly used to simulate large tanks or pipe breaks. The boundary conditions can either be constant in time or vary according to a pre-defined table. There are mainly two types of boundary conditions – flow conditions and pressure conditions. It is also possible to regard dead ends as a type of boundary condition. Throughout this project, pressure boundary conditions are most common, together with a number of dead ends.

2.2.6.5.1 RELAP5

The pressure boundary condition in RELAP5 is called a `TIME-DEPENDENT VOLUME` (`TMDPVOL`). The name is a little misleading since it is also possible to use variables other than time to control this component. There are several types of a `TMDPVOL`; the most common is one where pressure and temperature are specified. They can be set to vary according to a function specified in a table using the chosen variable. This corresponds to setting the `t`-flag to `t` = 3. Sometimes noncondensable gases are used; then one sets `t` = 4. There are other options for the `t`-flag; these are described in the Input Manual for RELAP5 (Appendix A of Volume II) [20].

The dimensions of the `TMDPVOL` do not affect the behaviour of the boundary condition. The specified thermodynamic properties are applied at the junction connecting to the next volume in the model.

Flow boundary conditions are specified using a `TIME-DEPENDENT JUNCTION` (`TMDPJUN`) as the junction connecting to the next component from the `TMDPVOL`. For the `TMDPJUN` it is possible to specify either the flow velocity or the mass flow through the junction.

2.2.6.5.2 TRACE

In TRACE pressure boundary conditions are implemented using the `BREAK` component. It is very similar to the `TMDPVOL` in RELAP5, but with one major difference; the dimensions of the `BREAK` component affect how the pressure boundary condition is applied. This is because the pressure applied at the junction connecting to the next volume is the average of the pressure of the volume and the `BREAK` component, respectively, weighted with the lengths of the two components.

Because of this, there are two recommendations for the dimensions of the `BREAK` component; either it is specified as having a large volume (10^6 m^3) and a short length (10^{-6} m), or it is specified as having the same dimensions as the connecting cell. The former corresponds to the user-specified pressure being equal to the dynamic pressure, whereas the latter corresponds to the user-specified pressure being equal to the static pressure. When the boundary condition is used to model a large pressure source or sink (e.g. a large tank) the former is normally recommended. More details about this can be found in Volume II of the TRACE manual [21].

The flow boundary conditions in TRACE are implemented using the `FILL` component. It works in a similar way as the `BREAK` component, except that it is required to input flow velocity or mass flow as well.

2.2.7 Control Systems and Trips

Both codes calculate and save a number of quantities during a simulation, e.g. pressures in all cells and velocities at all edges. However, sometimes another type of parameter may be of interest, such as the flow acceleration which, of course, is the time derivative of the flow velocity. This is where the control systems come into play. The control systems primarily consist of signal variables and control blocks. The former ones are used to measure a certain parameter, which may be of a number of different categories. In this project, the signal variables that are considered are typically general signals (e.g. current time step), volumetric signals (e.g. pressure in cell j) and junction or edge signals (e.g. total mass flow at cell boundary $j + \frac{1}{2}$). These signals can then be manipulated by the usage of control blocks, which typically represent a certain mathematical operation (e.g. summation, division or differentiation), in order to calculate the specific quantity of interest. These quantities may be important to study by themselves, or they may be used for controlling other components in the model. For instance, the flow area of a valve may be controlled by time or valve stem position, which is done using signal variables and possible control blocks.

Another important concept is trips, which often are used in combination with signal variables and control blocks. Trips can only have two values, ON (=true or 1) or OFF (=false or 0), depending on the input conditions and the setpoint values defined for the trip. A typical usage of a trip is to determine whether a valve is opened or closed. This information may then e.g. be used in a control system to make the valve change its position.

The implementation of control systems and trips is very similar in RELAP5 and TRACE. The difference is mainly the maximum number of control components that can be used in a model and the restrictions in how these components are numbered. The following two sections describe some of these differences. For a more comprehensive description, the reader is referred to the respective manual [9, 20, 21].

2.2.7.1 RELAP5

In RELAP5, two different types of 'cards' are available for the implementation of control systems, but only one of these can be used within a model. Either the card `205CCCNN` or `205CCCCN` is chosen by the user. The former card allows a control component numbering in the range 001-999, compared to 1-9999 in the latter case. Thus, ten times as many control blocks can be used by choosing `205CCCCN`. It can be mentioned that signal variables are not given a component number in RELAP5, but are instead associated by the signal type and at which location the signal is measured.

There are three different card series available for defining the numbering and how many trips that are allowed in a model, but only one card series can be used at a time. The cards that are used during this project are `401-799`, which allow a maximum of 199 variable trips (`401-599`) and 199 logical trips (`601-799`) to be used simultaneously. Note that if this trip card series is used in combination with the control system card `205CCCNN`, the total number of control blocks and trips are limited to 999, with the trips restricted to be in their respective range. However, the other two card series (see [20]) allow a maximum of 1000 or 10000 trips of each type (variable and logical), respectively.

2.2.7.2 TRACE

In TRACE, all types of control components (signal variables, control blocks and trips) are assigned a unique component number. The numbering is, however, much less restricted compared to RELAP5. Signal variables are in TRACE identified by the ID number $1 \leq \texttt{IDSV} \leq 99000$, control blocks by $-99000 \leq \texttt{IDCB} \leq -1$ and trips by $1 \leq |\texttt{IDTP}| \leq 9999$ [9].

2.2.8 SNAP

The Symbolic Nuclear Analysis Package (SNAP) is a graphical user interface which can be used to create models in both RELAP5 and TRACE, as well as a number of other codes. It can also be used to analyse the results of simulations, using the Animation-plugin.

For this project, there are two other important properties of SNAP; the ability to import an input file for RELAP5, and the ability to convert a RELAP5 model to a TRACE model. It is also possible to import input files for TRACE, as well as to export RELAP5 and TRACE models to ASCII format.

Further, it is possible to define restart[10] cases within SNAP. Calculations in SNAP are handled using Job Streams, which consist of a number of calculation steps (e.g. a base case and several restart cases). The results of one step is submitted to the next step. It is also possible to read results from a file.

2.3 Special Process Models and Correlations

This section includes a few of the correlations that have been considered to a greater extent in the project. These are correlations for pressure losses, interfacial heat transfer, level tracking and water packing.

2.3.1 Pressure Losses

Pressure losses in flow through pipes occur mainly due to friction and changes in the flow geometry. The latter is referred to as form loss. The following sections describe how these two phenomena are modelled in RELAP5 and TRACE.

2.3.1.1 Friction Losses

Pressure losses due to friction can be determined using different correlations. The ones used in RELAP5 and TRACE are briefly presented below. It should be noted that the friction factors are used in somewhat different ways in the two codes, which means that they are not explicitly comparable. The interested reader is referred to the manuals of the respective codes [14, 18] for more details on friction losses.

2.3.1.1.1 RELAP5

RELAP5 makes use of the Darcy friction factor, λ_T, which is calculated using different correlations depending on the Reynolds number. For turbulent flow (Re$>$ 3000) the Zigrang-Sylvester approximation [23] to the Colebrook-White correlation [24] is used. It is given by

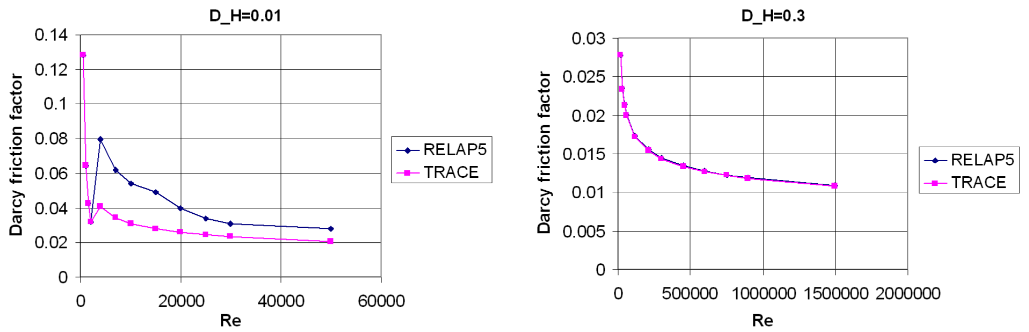
$$\frac{1}{\sqrt{\lambda_T}} = -2\log_{10}\left(\frac{\epsilon}{3.7D_H} + \frac{2.51}{\mathrm{Re}}\left(1.14 - 2\log_{10}\left(\frac{\epsilon}{D_H} + \frac{21.25}{\mathrm{Re}^{0.9}}\right)\right)\right), \tag{2.7}$$

where ϵ is the surface roughness and D_H is the hydraulic diameter.

2.3.1.1.2 TRACE

TRACE makes use of the Fanning friction factor f_w, which is equal to the Darcy friction factor divided by four, i.e. $\lambda_T = 4f_w$. The Churchill formula [25] is used to determine

[10] A restart case is a re-run of a simulation where typically only minor changes have been made to the model.



(a) $D_H = 0.01$ m.

(b) $D_H = 0.3$ m.

Figure 2.6: Plot showing how the friction factors vary with Reynolds number. The Fanning friction factor as calculated by Eq. (2.8) for TRACE has been multiplied by four in order to get comparable results with the Darcy friction factor calculated by Eq. (2.7) for RELAP5, since $\lambda_T = 4f_w$. Note that the difference becomes smaller for a pipe with larger hydraulic diameter.

the Fanning friction factor in TRACE. It is given by

$$f_w = 2\left(\left(\frac{8}{\mathrm{Re}}\right) + \frac{1}{(a+b)^{3/2})}\right)^{1/12}, \tag{2.8}$$

with

$$a = \left(2.457\ln\left(\frac{1}{\left(\frac{7}{\mathrm{Re}}\right)^{0.9} + 0.27\left(\frac{\epsilon}{D_H}\right)}\right)\right)^{16} \tag{2.9}$$

and

$$b = \left(\frac{3.753 \cdot 10^4}{\mathrm{Re}}\right)^{16}. \tag{2.10}$$

Fig. 2.6 shows how the friction factors vary with Reynolds number (the Fanning friction factor has been multiplied by four in order to get comparable results).

2.3.1.2 Form Losses

As mentioned above, pressure losses occur in flows due to e.g. friction. But there are also losses related to changes in flow area. These can be separated into recoverable and irrecoverable losses. The former occurs where there is a smooth change in flow area and, as the name suggests, the loss can be recovered as the flow area changes back to its original value. The latter, however, occurs e.g. where there is a sudden – abrupt – change in flow area. This type of loss is not possible to recover.

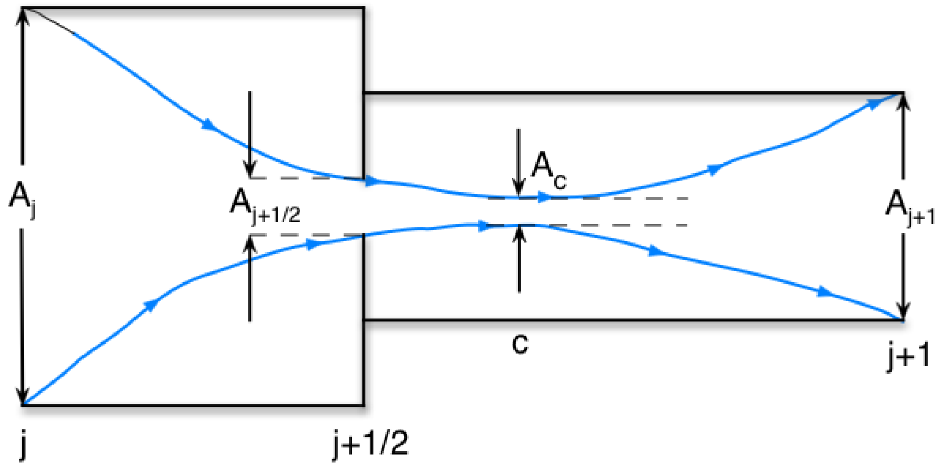


Figure 2.7: A general geometry for an abrupt area change. Note that indexes j and $j+1$ correspond to cell centres, whereas index $j+1/2$ corresponds to a cell edge.

In both RELAP5 and TRACE there is a possibility to enter a form loss coefficient for an edge. In the case of an abrupt area change, it is also possible to let the codes calculate this coefficient. It should be noted though, that although the two codes are based on the same model, there are some slight differences, which will be described below. First, a brief description of form losses is given. This is valid for both codes.

Generally, the pressure drop due to form loss between two cells can be expressed by the Bernoulli equation:

$$\Delta P_{j\to j+1} = P_j - P_{j+1} = \rho\left(\frac{v_{j+1}^2}{2} - \frac{v_j^2}{2}\right), \tag{2.11}$$

where the indices refer to Fig. 2.7, P is the pressure, ρ is the density and v is the velocity of the fluid. If there is an abrupt area change between the cells or any other irrecoverable loss, an additional term must be included in Eq. (2.11). This term is usually expressed as

$$\Delta P_{\text{irrecoverable}} = \rho K \frac{v^2}{2}, \tag{2.12}$$

using the form loss factor K. Eq. (2.11) then becomes

$$\Delta P_{j\to j+1} = P_j - P_{j+1} = \rho\left(\frac{v_{j+1}^2}{2} - \frac{v_j^2}{2}\right) + \rho K \frac{v^2}{2}, \tag{2.13}$$

where the velocity v is taken at different locations in RELAP5 and TRACE, as described below. This expression can be used not only for abrupt area changes, but also for flow through pipe bends and tees. In the case of an abrupt area change the form loss factor K is modelled somewhat differently in RELAP5 and TRACE, as will be described below. These differences can lead to difficulties e.g. when modelling valves.

It should be noted that in addition to the form loss factors calculated by RELAP5 and TRACE, respectively, for an abrupt area change it is possible for the user to specify an additional constant form loss coefficient, which is added to the calculated one in order to get the total form loss. Such a user-specified form loss coefficient can also be specified in a junction where no abrupt area change occurs.

2.3.1.2.1 RELAP5

In RELAP5 the velocity v in Eq. (2.13) is taken as the velocity downstream the abrupt area change, i.e.

$$v = v_{j+1}. \tag{2.14}$$

The approach in modelling the form loss factor K in RELAP5 is more general than in TRACE. It should be noted that RELAP5 takes into account the formation of a vena contracta, i.e. that the flow is contracted more than the actual area reduction, as indicated at point c in Fig. 2.7. The following relationship is used to determine the form loss factor for an abrupt area change:

$$K = \left(1 - \frac{\varepsilon}{\varepsilon_c \varepsilon_T}\right)^2, \tag{2.15}$$

where

$$\varepsilon = \frac{A_{j+1}}{A_j}, \tag{2.16}$$

$$\varepsilon_T = \frac{A_{j+1/2}}{A_j} \tag{2.17}$$

and

$$\varepsilon_c = \frac{A_c}{A_{j+1/2}} = 0.62 + 0.38\varepsilon_T^3. \tag{2.18}$$

The second equality in Eq. (2.18) comes from a correlation for the contraction in the vena contracta. The interested reader is referred to Volume I and IV of the RELAP5 manual [14, 26] for details.

Further, the model is modified slightly for the case of two-phase flow, taking interphase drag into account. In principle it is the same model, though. The reader is again referred to Volume I and IV of the RELAP5 manual [14, 26] for details regarding this.

2.3.1.2.2 TRACE

In TRACE the velocity v in Eq. (2.13) is taken as the velocity at the location of the abrupt area change, i.e.

$$v = v_{j+1/2}. \tag{2.19}$$

TRACE also uses different equations to model the form loss factor depending on if the abrupt area change consists of a contraction, an expansion or an orifice. If there is a sudden contraction, Eq. (2.20) is used:

$$K = \left(0.5 - 0.7\varepsilon + 0.2\varepsilon^2\right)\left(\frac{\varepsilon_T}{\varepsilon}\right)^2. \tag{2.20}$$

If there is a sudden expansion, Eq. (2.21) is used:

$$K = \left(1 - \frac{1}{\varepsilon}\right)^2 \varepsilon_T^2. \tag{2.21}$$

In the case of an orifice TRACE is not able to automatically calculate the form loss factor. Instead it must be specified manually. In this case Eq. (2.22) should be used to manually determine K (according to the TRACE Theory manual [18]):

$$K = \left(1 + 0.707\sqrt{1-\varepsilon_T} - \varepsilon_T\right)^2. \tag{2.22}$$

A special case of an orifice is a valve, which in principle works as an orifice with an adjustable flow area. The form loss coefficient for a valve is automatically calculated by TRACE according to Eq. (2.23):

$$K = \frac{1-\varepsilon_T}{2} + (1-\varepsilon_T)^2. \tag{2.23}$$

Instead of Eq. (2.23) it is possible to input a table relating the form loss coefficient to the flow area fraction (by turning off the internal loss model by setting `INTLOSSOFF` = [1]). Finally, it should be noted that when the additive user-specified form loss coefficient is input as 0.0, TRACE automatically changes it to 0.03 for most valve types [21].

Since there are differences in how the two codes calculate the form loss factor for abrupt area changes, slight differences are expected in the simulation results. Also, the fact that K is multiplied by v_{j+1} in RELAP5 and $v_{j+1/2}$ in TRACE can introduce differences in the results. If one is interested in having the exact same losses in TRACE as in RELAP5 for an abrupt area change, the following expression can be used to convert the RELAP5 form loss to TRACE, taking into account the fact that K multiplies different velocities in the two codes:

$$K_{TRACE} = \frac{\varepsilon_T^2}{\varepsilon} K_{RELAP5} = \left(\frac{1}{\varepsilon_c}\left(1 - \frac{\varepsilon_T \varepsilon_c}{\varepsilon}\right)\right)^2, \tag{2.24}$$

where ε, ε_T and ε_c are given by Eqs. (2.16)-(2.18), K_{TRACE} is the user-specified form loss factor in TRACE and K_{RELAP5} is the form loss factor as calculated by RELAP5 according to Eq. (2.15). It should be noted that Eq. (2.24) is not necessary to use for form losses related to e.g. bends, since it reduces to $K_{TRACE} = K_{RELAP5}$ in this case. For valves, Eq. (2.24) can be used as input into the table relating form loss to flow area fraction, which is used when `INTLOSSOFF` = [1]. Fig. 2.8 shows the difference between the form loss factors for a valve as calculated by Eqs. (2.23) and (2.24), i.e. using the internal form loss model of TRACE and the converted form loss model from RELAP5, respectively. For simplicity it is assumed that the upstream flow area is equal to the downstream flow area.

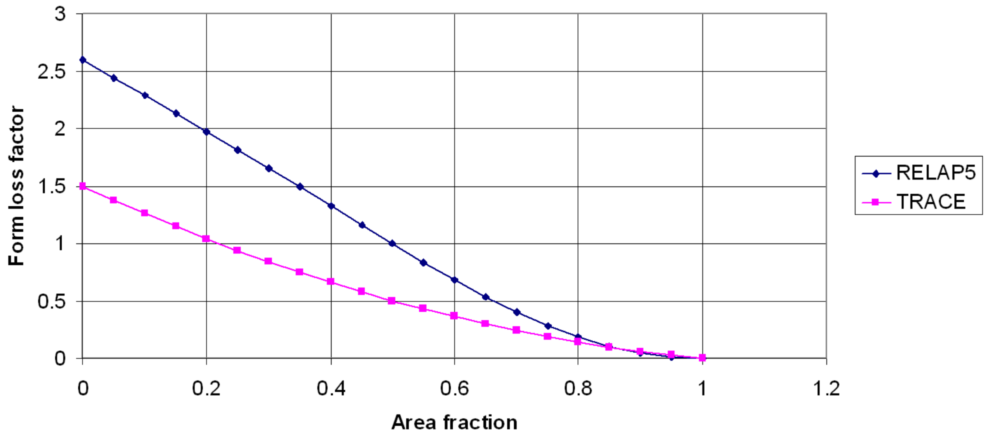


Figure 2.8: Difference between the form loss factors for a valve as calculated by Eqs. (2.23) and (2.24), i.e. using the internal form loss model of TRACE and the converted form loss model from RELAP5, respectively. For simplicity it is assumed that the upstream flow area is equal to the downstream flow area. It can be seen that the difference is smaller for a valve which is almost fully open.

2.3.2 Interfacial Heat Transfer (for Bubbly Flows)

The rate at which condensation and vapourisation occur for a two-phase flow is determined by the heat transfer between the phases in the bulk fluid and by possible heat transfer between each phase and a heated wall[11]. In both RELAP5 and TRACE this is generally expressed in terms of a total mass exchange rate Γ (kg/(m^3·s)), made up of two parts:

$$\Gamma = \Gamma_{\mathrm{ip}} + \Gamma_{\mathrm{w}} \tag{2.25}$$

where Γ_{ip} is the interfacial mass exchange rate for phase p in the bulk fluid and Γ_{w} is the mass exchange rate between the phases near the wall. Due to continuity reasons, the total mass transfer rate from the liquid to the gas phase (i.e. vapourisation, $\Gamma > 0$) equals the negative mass transfer rate in the opposite direction (i.e. condensation, $\Gamma < 0$). Only bulk interfacial heat transfer is considered in this section, for details regarding wall

[11] One may also account for a possible direct heating source, e.g. radioactive decay in the fluid.

heat transfer the reader is referred to the RELAP5 [14, 26] and TRACE [18] manuals. Neglecting Γ_{w}, the bulk interfacial mass exchange term in Eq. (2.25) is modelled using a thermal energy 'jump relation':

$$\Gamma_{\mathrm{ig}} = -\Gamma_{\mathrm{if}} = \frac{-(q_{\mathrm{ig}} + q_{\mathrm{if}})}{h_{\mathrm{g}} - h_{\mathrm{f}}} \tag{2.26}$$

where q_{ig}, q_{if} is the heat transfer per unit volume (W/m^3) and h_{g}, h_{f} is the specific enthalpy (J/kg) for the respective phase. Fig. 2.9 illustrates the concept, and directions, of mass and energy exchange between the phases through the interface.

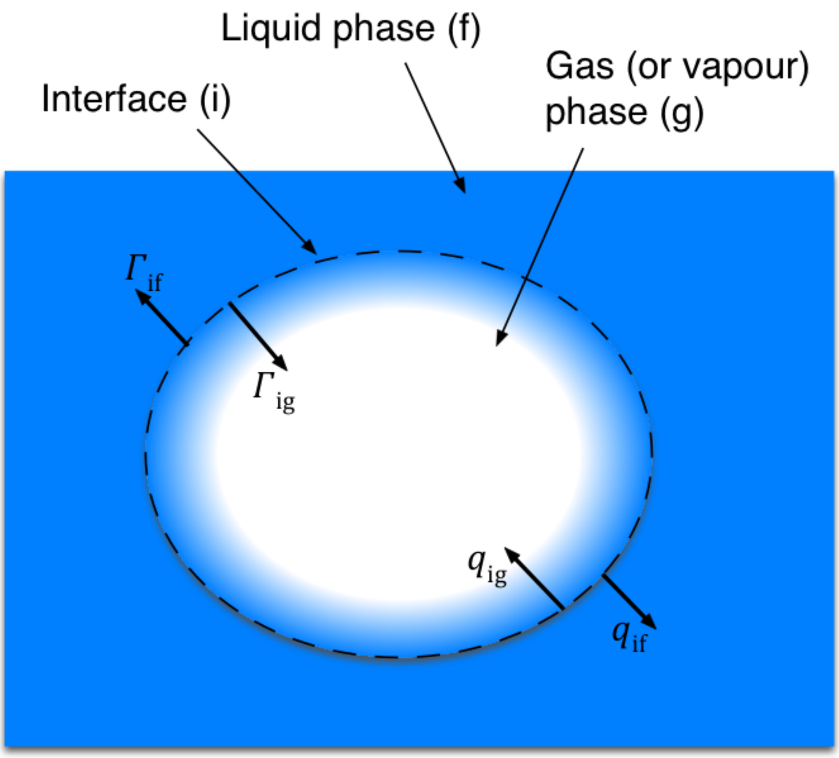


Figure 2.9: Concept of mass (Γ) and energy (q) exchange rates between the liquid (f) and gas (or vapour) phase (g). $\Gamma_{\mathrm{ig}} = -\Gamma_{\mathrm{if}}$ due to continuity.

The heat transfer quantities in Eq. (2.26) are included as source terms in the field equations and are modelled according to

$$q_{\mathrm{ig}} = H_{\mathrm{ig}}(T_{\mathrm{sv}} - T_{\mathrm{g}}) \tag{2.27}$$

for the vapour side[12] and

$$q_{\mathrm{if}} = H_{\mathrm{if}}(T_{\mathrm{sv}} - T_{\mathrm{f}}) \tag{2.28}$$

[12] TRACE, however, also includes a factor $\frac{P_{\mathrm{v}}}{P}$ for the vapour side equation.

for the liquid side, where T_{sv} is the saturation temperature corresponding to the partial vapour pressure. H_{ip} (i.e. H_{ig} or H_{if}) is a volumetric interfacial heat transfer factor (W/(m$^3\cdot$ K)) and is modelled, in a very general manner, as

$$H_{\mathrm{ip}} = \frac{k_{\mathrm{p}}}{L}\mathrm{Nu}\ a_{\mathrm{gf}} = h_{\mathrm{ip}} a_{\mathrm{gf}} \tag{2.29}$$

where k_{p} is the thermal conductivity (W/(m·K)) for phase p, L is a characteristic length (m), Nu is the dimensionless Nusselt number, a_{gf} is the interfacial area per unit volume (m^2/m^3) and h_{ip} is the interfacial heat transfer coefficient (W/(m^2·K)) for phase p. Finally, both h_{ip} and a_{gf} are modelled utilising a certain correlation depending on the flow regime chosen by the code. Because bubbly flow is one of the most common flow regimes in a non-heated piping system, the correlations for this flow type are presented briefly and compared in the following sections. Further, possible ways of adjusting the interfacial heat transfer rate in the two codes are mentioned. For a complete description of the correlations used for another certain regime, the reader is referred to the RELAP5 [26] and TRACE [18] manuals.

2.3.2.1 RELAP5

In both RELAP5 and TRACE, the interface between the two phases is kept constant at the saturation temperature T_{sv}. However, both the gas and liquid phases are allowed to be either subcooled ($T_{\mathrm{p}} < T_{\mathrm{sv}}$) or superheated ($T_{\mathrm{p}} > T_{\mathrm{sv}}$) and may thus contribute to the heat transfer both to and from the interface. All the heat transferred to the interface from either superheated liquid (SHL) or superheated gas (SHG) contributes to vapourisation and has a positive mass exchange rate Γ_{ig}. Similarly, all the heat transferred from the interface to subcooled liquid (SCL) or subcooled gas (SCG) contributes to condensation and has a negative mass exchange rate $-\Gamma_{\mathrm{ig}}$. The net condensation or vapourisation rate is then determined by the sum of these positive and negative contributions.

The volumetric heat transfer factor H_{ip} is in RELAP5 modelled for each fluid condition (SHL, SHG, SCL, and SCG) and flow regime. In the case of **bubbly flow**, the following correlations are used for superheated liquid:

$$H_{\mathrm{if}}^{\mathrm{SHL}} = \left\{ \max \begin{bmatrix} -\dfrac{k_{\mathrm{f}}}{d_{\mathrm{b}}}\dfrac{12}{\pi}\Delta T_{\mathrm{sf}}\dfrac{\rho_{\mathrm{f}}C_{p\mathrm{f}}}{\rho_{\mathrm{g}}h_{\mathrm{fg}}} & \text{(Plesset-Zwick)} \\ \dfrac{k_{\mathrm{f}}}{d_{\mathrm{b}}}(2.0 + 0.74\mathrm{Re}_{\mathrm{b}}^{0.5}) & \text{(modified Lee-Ryley)} \end{bmatrix} + 0.4|v_{\mathrm{f}}|\rho_{\mathrm{f}}C_{p\mathrm{f}}F_1 \right\} (a_{\mathrm{gf}}F_2F_3) \tag{2.30}$$

or $H_{\mathrm{if}}^{\mathrm{SHL}} = 0.0$ if $\alpha_{\mathrm{g}} = 0$ and $\Delta T_{\mathrm{sf}} \geq 0$, where

$$\Delta T_{\mathrm{sf}} = T_{\mathrm{sv}} - T_{\mathrm{f}}$$

$$h_{\mathrm{fg}} = \text{latent heat of evaporation}$$

$$C_{p\mathrm{f}} = \text{specific heat of liquid}$$

$$\mathrm{Re_b} = \frac{(1-\alpha_{\mathrm{bub}})\rho_\mathrm{f} v_{\mathrm{fg}} d_\mathrm{b}}{\mu_\mathrm{f}} = \frac{\mathrm{We}\ \sigma(1-\alpha_{\mathrm{bub}})}{\mu_\mathrm{f}(v_{\mathrm{fg}}^2)^{1/2}}$$

$$\mathrm{We} = \text{Weber number}$$

$$\sigma = \text{surface tension}$$

$$\mathrm{We}\ \sigma = \max(\mathrm{We}\ \sigma, 10^{-10})$$

$$d_\mathrm{b} = \text{average bubble diameter } (= \frac{1}{2} d_{\mathrm{max}}) = \frac{\mathrm{We}\ \sigma}{\rho_\mathrm{f}\ v_{\mathrm{fg}}^2}, \mathrm{We} = 5$$

$$a_{\mathrm{gf}} = \frac{3.6\alpha_{\mathrm{bub}}}{d_\mathrm{b}}$$

$$\alpha_{\mathrm{bub}} = \max(\alpha_\mathrm{g}, 10^{-5})$$

$$v_{\mathrm{fg}} = \begin{cases} v_\mathrm{g} - v_\mathrm{f} & \alpha_\mathrm{g} \geq 10^{-5} \\ (v_\mathrm{g} - v_\mathrm{f})\alpha_\mathrm{g} \cdot 10^5 & \alpha_\mathrm{g} < 10^{-5} \end{cases}$$

$$v_{\mathrm{fg}}^2 = \max\left[v_{\mathrm{fg}}^2, \frac{\mathrm{We}\ \sigma}{\rho_\mathrm{f}\ \min(D'\alpha_{\mathrm{bub}}^{1/3}, D_H)}\right]$$

$$D_H = \text{hydraulic diameter}$$

$$D' = 0.005 \text{ m for bubbly flow}$$

$$F_1 = \frac{\min(0.001, \alpha_{\mathrm{bub}})}{\alpha_{\mathrm{bub}}}$$

$$F_2 = \frac{\min(0.25, \alpha_{\mathrm{bub}})}{\alpha_{\mathrm{bub}}}$$

$$F_3 = \begin{cases} 1 & \Delta T_{\mathrm{sf}} \leq -1 \\ \max\Big[0.0, \big[F_4(1+\Delta T_{\mathrm{sf}}) - \Delta T_{\mathrm{sf}}\big]\Big] & -1 < \Delta T_{\mathrm{sf}} < 0 \\ \max(0.0, F_4) & \Delta T_{\mathrm{sf}} \geq 0 \end{cases}$$

$$F_4 = \min\Big[10^{-5}, \alpha_{\mathrm{g}}(1 - X_{\mathrm{n}})\Big] \cdot 10^5$$

$$X_{\mathrm{n}} = \text{noncondensable quality}$$

The Plesset-Zwick [27] correlation in Eq. (2.30) is based on an analytical equation for the growth rate of the bubble radius. This approach has several assumptions, e.g. that the bubble remains spherical as it grows and that translation velocity of the bubble is neglected. The second correlation by Lee and Ryley [28] is used for taking a higher relative bubble velocity into account. The Nusselt number is in this case modelled as

$$\mathrm{Nu} = 2.0 + 0.74\mathrm{Re}_{\mathrm{b}}^{1/2}\mathrm{Pr}_{\mathrm{f}}^{1/3} \tag{2.31}$$

where it is assumed that the Prandtl number $\mathrm{Pr}_{\mathrm{f}} = \dfrac{C_{p\mathrm{f}}\mu_{\mathrm{f}}}{k_{\mathrm{f}}} \approx 1$ and is therefore neglected. However, some doubts are raised in [26] about the validity of this correlation due to, for instance, the deformation of the spherical bubbles in a bubbly flow. Also, the pressures at which the correlation has been tested are far below the typical operating pressures in a nuclear power plant. The second term in Eq. (2.30) is added to represent the effect of increasing nucleation at small void fractions, while the functions F_1, F_2, F_3 and F_4 are different smoothing functions.

For subcooled liquid, the corresponding heat transfer factor is modelled as

$$H_{\mathrm{if}}^{\mathrm{SCL}} = \frac{F_3 F_5 h_{\mathrm{fg}} \rho_{\mathrm{g}} \rho_{\mathrm{f}} \alpha_{\mathrm{bub}}}{\rho_{\mathrm{f}} - \rho_{\mathrm{g}}} \text{ (modified Unal and Lahey)} \tag{2.32}$$

where $\rho_{\mathrm{f}} - \rho_{\mathrm{g}} = \max(\rho_{\mathrm{f}} - \rho_{\mathrm{g}}, 10^{-7})$, F_5 is a smoothing function while F_3 and α_{bub} are the same as for bubbly SHL. The interfacial area a_{gf} is in Eq. (2.32) rewritten in terms of other physical quantities in a similar was as in Eq. (2.34) below. For more details regarding this correlation the reader is referred to [26] or e.g. the original publication by Unal [29].

The heat transfer factors for superheated and subcooled gas are simply expressed as

$$H_{\mathrm{ig}}^{\mathrm{SHG}} = H_{\mathrm{ig}}^{\mathrm{SCG}} = h_{\mathrm{ig}} F_6 F_7 a_{\mathrm{gf}} \tag{2.33}$$

where $h_{\mathrm{ig}} = 10^4$ W/(m^2·K) and F_6 and F_7 are different smoothing functions (see [26]). The constant value for h_{ig} is used for quickly bringing the gas temperature to the saturation temperature.

For all fluid conditions in a bubbly flow, the interfacial area is modelled as

$$a_{\mathrm{gf}} = \frac{3.6\alpha_{\mathrm{bub}}}{d_{\mathrm{b}}} = 0.72\frac{\alpha_{\mathrm{g}}\rho_{\mathrm{f}}(v_{\mathrm{g}} - v_{\mathrm{f}})^2}{\sigma}, \tag{2.34}$$

which is based on a probability distribution for the bubble diameter and where it is assumed that the average bubble diameter $d_{\mathrm{b}} = \frac{1}{2}d_{\max}$.

Basically, the only way to adjust or modify the interfacial heat transfer rate in RELAP5 is to turn on the equilibrium model (e-flag) in appropriate control volumes. In this way, the two phases are assumed to be in thermal equilibrium (i.e. having the same temperature), which in turn results in an instantaneous phase transition in both directions (i.e. condensation and vapourisation).

2.3.2.2 TRACE

In contrast to the bubbly flow correlations in RELAP5, the heat transfer for the liquid side is the same for both vapourisation ($H_{\mathrm{if}}^{\mathrm{SHL}}$) and condensation ($H_{\mathrm{if}}^{\mathrm{SCL}}$) and can be expressed as

$$\left.\begin{matrix} H_{\mathrm{if}}^{\mathrm{SHL}} \\ \\ H_{\mathrm{if}}^{\mathrm{SCL}} \end{matrix}\right\} = \frac{k_{\mathrm{f}}}{d_{\mathrm{b}}}\mathrm{Nu}\; a_{\mathrm{gf}} = h_{\mathrm{if}}a_{\mathrm{gf}} \tag{2.35}$$

where the interfacial area depends on the void fraction and the bubble diameter according to

$$a_{\mathrm{gf}} = \frac{6\alpha_{\mathrm{g}}}{d_{\mathrm{b}}} \tag{2.36}$$

which seems to be larger than the corresponding area in RELAP5, see Eq. (2.34). However, a different model is used for the bubble diameter (d_{b}). In TRACE, it is calculated from

$$d_{\mathrm{b}} = 2\sqrt{\frac{\sigma}{g\Delta\rho}} \tag{2.37}$$

where σ is the surface tension (N/m) and $\Delta\rho = \rho_{\mathrm{f}} - \rho_{\mathrm{g}}$ (kg/m^3). Further, the diameter is restricted to be in the range 10^{-4}(m) $\leq d_{\mathrm{b}} \leq 0.9D_H$ (i.e a rather vague restriction).

The Nusselt number (Nu) in Eq. (2.35) is based on the following correlation by Ranz and Marshall [30]:

$$\mathrm{Nu} = 2.0 + 0.6\mathrm{Re}_{\mathrm{b}}^{1/2}\mathrm{Pr}_{\mathrm{f}}^{1/3}, \tag{2.38}$$

which is very similar to Eq. (2.31) for RELAP5 apart from a different factor (0.6 vs. 0.74) and the fact that the Prandtl number is not neglected in this case. However, there are also some differences in the modelling of the Reynolds number for the dispersed bubble. In TRACE, this dimensionless number is calculated by

$$\mathrm{Re}_{\mathrm{b}} = \frac{\rho_{\mathrm{f}} v_{\mathrm{rel,b}} d_{\mathrm{b}}}{\mu_{\mathrm{f}}} \tag{2.39}$$

where $v_{\mathrm{rel,b}}$ is the bubble relative velocity which is not (necessarily) the same as the phasic relative velocity $v_{\mathrm{vg}} = |v_{\mathrm{g}} - v_{\mathrm{f}}|$. According to [18], this is explained by the fact that the vapour phase is assumed to consist of bubbles of different size, resulting in $v_{\mathrm{rel,b}} \ll |v_{\mathrm{g}} - v_{\mathrm{f}}|$. A better estimate of the bubble relative velocity is then given by

$$v_{\mathrm{rel,b}} = \mathrm{Min}\Big[|v_{\mathrm{g}} - v_{\mathrm{f}}|, v_{\mathrm{term}}\Big] \tag{2.40}$$

where the bubble terminal velocity (v_{term}) is calculated according to

$$v_{\mathrm{term}} = v_{\mathrm{rel},\infty}(1 - \alpha_{\mathrm{g}})^{1.39} \tag{2.41}$$

In this equation[13], the relative terminal velocity in an infinite medium is determined by

$$v_{\mathrm{rel},\infty} = \sqrt{2}\left(\frac{\sigma \cdot g \cdot \Delta\rho}{\rho_{\mathrm{f}}^2}\right)^{1/4} \tag{2.42}$$

The heat transfer for the vapour side is, similarly to RELAP5, modelled as a constant factor that is large enough for bringing the conditions not too far from thermal equilibrium. This is simply coded as:

$$H_{\mathrm{ig}}^{\mathrm{SHG}} = H_{\mathrm{ig}}^{\mathrm{SCG}} = h_{\mathrm{ig}} a_{\mathrm{gf}} = 10^3 \cdot a_{\mathrm{gf}} \tag{2.43}$$

According to [18], the interfacial heat transfer coefficient for the vapour is usually not important for the heat and mass exchange in bubbly flows. It is noticed that this constant value (10^3) is only one tenth of the value implemented in RELAP5 (10^4). However, no smoothing functions are applied for this correlation in TRACE (cf. Eq. (2.33)).

[13] An error occurs in TRACE V5.0 where the exponent is implemented as 2.39 instead of 1.39 (see p.207 in the TRACE Theory Manual [18]). It is unclear whether this has been corrected in version V5.830.

In TRACE V5.830 (BETA) it is possible to modify the phase transition rate for the entire model by a multiplicative factor[14] in front of the calculated interfacial heat transfer coefficient h_{ip}. This modification can be done individually for both the gas and the liquid side, as well as for the different flow regimes. However, this feature is not available in the V5.0 release, and it is unclear whether it will be available in the next actual release of the TRACE code.

2.3.3 Level Tracking

Both RELAP5 and TRACE use mean void fraction as the variable to keep track of the distribution of steam and liquid within a computational cell. This means that small cell sizes are required in order to resolve sudden changes in the amount of void, e.g. as in the case of a liquid level, where the void fraction changes from zero to one over an infinitely thin interface. It is however desirable to keep the cells as large as possible in order to minimise computational cost. Therefore, the method of level tracking has been developed, which determines whether a liquid level exists within a volume, and, if so, modifies the field equations within that volume in order to more realistically model the behaviour of the fluid. Also, the computational volume is divided into two sub volumes – one with mainly liquid and one with mainly vapour – which have different flow topologies, including void fractions. Finally, the volumes of the two sub volumes must be known as well as the velocity of the liquid level [31]. It should be noted that level tracking can only be used in vertical components.

Fig. 2.10 shows a schematic of three cells, of which one contains a liquid level. The notation in the figure will be used throughout this section, with α denoting void fraction.

[14] Also known as 'sensitivity coefficients' in SNAP.

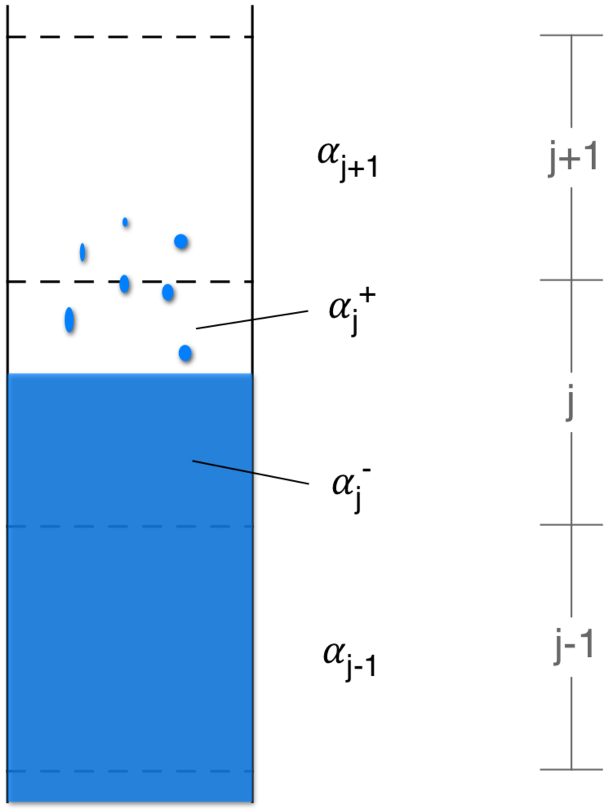


Figure 2.10: Schematic of a vertical pipe containing a liquid level.

In order for the level tracking method to be used in the calculations (assuming that the user has activated the corresponding flag), several conditions must be fulfilled. In TRACE one of the following conditions must be fulfilled:

$$(\alpha_j - \alpha_{j-1} > \delta\alpha) \wedge (\alpha_{j+1} - \alpha_j > \Delta\alpha_{cut}) \wedge (\alpha_{j+1} > \alpha_{lev}) \tag{2.44}$$

or

$$(\alpha_j - \alpha_{j-1} > \Delta\alpha_{cut}) \wedge (\alpha_{j+1} - \alpha_j > \delta\alpha) \wedge (\alpha_{j+1} > \alpha_{lev}), \tag{2.45}$$

with $\delta\alpha = 0.005$, $\Delta\alpha_{cut} = 0.2$ and $\alpha_{lev} = 0.7$.

In RELAP5 the conditions (for a normal void profile, i.e. where the vapour is above the liquid) are either:

$$(\alpha_{j+1} - \alpha_j > \Delta\alpha_{cut}) \wedge (\alpha_{j+1} > \alpha_{lev}) \tag{2.46}$$

or

$$(\alpha_j - \alpha_{j-1} > \Delta\alpha_{cut}) \wedge (\alpha_{j+1} > \alpha_{lev}), \tag{2.47}$$

with $\Delta\alpha_{cut} = 0.2$ and $\alpha_{lev} = 0.7$, i.e. the same values as for TRACE. It should be noted that $\delta\alpha$ is not included in the conditions used in RELAP5 though. Further, neither the TRACE manual [9, 18, 21] nor the RELAP5 manual [14, 15, 20, 22, 26] give any information of how e.g. dead ends are treated in the level tracking model, and what values are assigned to variables in cell $j + 1$ in such a case.

If the conditions in Eqs. (2.44)-(2.45) or (2.46)-(2.47) are fulfilled, respectively, modifications are made to the field equations. The details are left out and the reader is referred to the manuals of the two respective codes [14, 18]. Further, variables are created which describe the position and velocity of the liquid level as well as the void fractions above and below the level. These are used in the modifications of the field equations.

For a detailed description of level tracking in thermal-hydraulic codes, the reader is referred to [31].

2.3.4 Water Packing

In an Eulerian finite-difference scheme it is possible to encounter some unphysical pressure spikes when water (or any liquid) is entering a cell that already is almost completely water-filled. The cause of these pressure spikes is the large change in compressibility between pure water and a two-phase mixture with a small void fraction, together with the approximations made in the discrete momentum equations. The higher compressibililty of the two-phase mixture means that a small pressure change is enough to achieve a significant change in volume, or vice versa. Thus, a high mass influx (into the cell) is allowed without a large pressure increase. At the time step when the cell is about to be completely filled with liquid ('water packed'), the compressibility is rapidly decreased, and a large pressure spike is likely to be calculated in the next time step [14, 18].

To overcome this numerical issue, a water packing mitigation scheme is used (by default, but it can be turned off) in both RELAP5 and TRACE. A detection logic[15] is utilised for determining when water packing is about to occur. This logic is similar, but not identical, for the two codes. The modifications to the momentum equations, when water packing has been detected, are also slightly different but the basic idea is the same. Consider a standard motion equation for the liquid velocity at a cell edge $j + \frac{1}{2}$ expressed as

$$v_{j+1/2}^{n+1} = v_{j+1/2}^{n} + \tilde{a} + \tilde{b}(P_j^{n+1} - P_{j+1}^{n+1}) \tag{2.48}$$

where the superscripts n and $n + 1$ represent the current and the next time step, respectively. If the water packing detection logic returns true for the cell j, the equation is modified according to

$$v_{j+1/2}^{n+1} = v_{j+1/2}^{n} + \tilde{a} + \tilde{b}(\tilde{c}P_j^{n+1} - P_{j+1}^{n+1}) \tag{2.49}$$

where the factor $\tilde{c}$ is chosen to be large enough for a small pressure change to give rise to a high liquid flow velocity out of the cell. In this way, the 'packing' of water, and thus the large pressure spikes, are avoided.

[15] With 'detection logic' when talking about water packing, it is meant the conditions tested by the codes to see whether water packing is likely to occur or not.

Some of the differences in the detection logic between RELAP5 and TRACE are presented below.

2.3.4.1 RELAP5

A number of requirements are tested if water packing is present in the cell j. First of all, the pressure increase during one time step must fulfill:

$$P_j^{n+1} \geq P_j^n + 0.0023P_j^n \tag{2.50}$$

Also, the following must be fulfilled: void fraction $\alpha_g \leq 0.12$, liquid temperature $T_{\mathrm{f}} \leq T_{\mathrm{sv}}$, vertically stratified flow regime and the volume above must be 'highly voided'. The last two requirements mean that water packing is not considered at all in a horizontal component (also confirmed in the Developmental Assessment of the RELAP5 manual [15]).

2.3.4.2 TRACE

The upper limit for the void fraction in cell j is lower in TRACE ($\alpha_g \leq 0.08$), but the requirement for the liquid temperature ($T_{\mathrm{f}} \leq T_{\mathrm{sv}}$) is the same. Also, the void fraction in adjacent cells must be larger than 0.1, which is more precisly defined than the, in RELAP5, correspondingly 'highly voided'. However, nothing is told in the TRACE Theory manual [18] about whether only vertical flows are considered.

2.4 Inertial Swing Check Valve

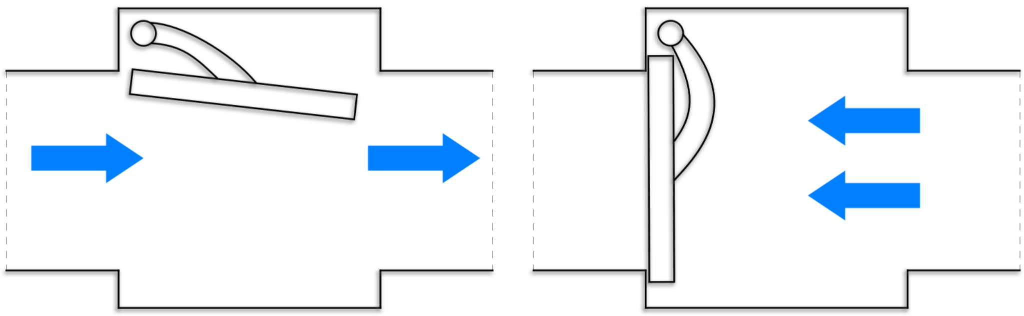
Later on (in Chapter 6 and 8), a model of an inertial swing check valve, created at Ringhals, is utilised and investigated rather extensively. The methodology and physical correlations used to build this model are therefore presented in this section. First, however, is a short introduction to check valves in general.

The aim of using check valves is to prevent backflow following for instance a pump stop or a pipe break. An ideal check valve will close instantaneously at the moment when the flow velocity is zero before starting to reverse. In reality, though, some backflow will always occur, resulting in a sudden slam when the valve flapper (or disc) hits the seat, cf. Fig. 2.11. This sudden closure can in turn give rise to large water hammers with a maximum magnitude given by the Joukowsky equation [10] as:

$$\Delta P_{\mathrm{Joukowsky}} = \rho \cdot a \cdot v_{\mathrm{reverse}} \quad \mathrm{(Pa)}. \tag{2.51}$$

In order to avoid these water hammers, the valve should preferably close instantly (before a backflow is established) or in a very slow manner (if a backflow already is present) [32]. Since the backflow itself may cause mechanical damage to e.g. a pump, it is desirable to have the smallest possible backflow through the system.

A number of different basic check valve designs exist, but one of the most common is the (inertial) swing check valve illustrated in Fig. 2.11. Simple models of such a valve are available in both RELAP5 [14] and TRACE [21] and are essentially the same model. However, they are known to be limited in their application, mainly because the torque on the flapper is calculated using only the pressure difference across the valve and the weight of the flapper[16]. These limitations mean e.g. that the flapper is allowed to accelerate even though its velocity is greater than the velocity of the fluid, and that a vertically installed valve component cannot be modelled [33]. Comparisons with experimental data have been carried out in an earlier Master's thesis [34], from which the conclusion was that the built-in models in both RELAP5 and TRACE have too fast closure rates and thus give non-conservative results. This is because a fast closure rate leads to less backflow, which means that the force required to stop the flow is smaller because of its lower momentum. Furthermore, in the User's Guidelines (Volume V) of the RELAP5 manual [22], the built-in inertial swing check valve model is recommended not to be used due to lack of user experience.



(a) Opened position.

(b) Closed position.

Figure 2.11: Basic design and functionality of a swing check valve.

Therefore, a better RELAP5 model of such a valve has been created at Ringhals using control variables, as presented in [33]. The main advantage with the usage of control variables is that specific properties needed for describing the valve motion can be added. This model is based on, and has been validated against, a version originally implemented in the DRAKO code [33]. The main difference compared to the built-in models in RELAP5 and TRACE is that the torque due to the relative velocity between the fluid and the flapper disc is taken into account. In this model, the total torque acting on the flapper is expressed as

$$M_{\text{tot}} = \sum M = M_{\text{mg}} + M_{\text{str}} + M_{\Delta P} \; (+M_{\text{damping}}) \tag{2.52}$$

where M_{mg} is the torque due to the flapper weight, M_{str} is the torque due to the flow velocity and $M_{\Delta P}$ is the torque due to the pressure difference across the flapper disc. M_{damping} is a time-controlled damping term used in the code for achieving a faster steady-state.

[16] Although the torque due to friction is also taken into account in the built-in models in both RELAP5 and TRACE, this contribution is only based on a user-specified additional pressure difference required for the valve to open [14] [21].

The weight term is modelled according to

$$M_{\mathrm{mg}} = -m \cdot g \cdot \sin(\theta_s + \theta) \cdot \left(1 - \frac{\rho}{\rho_{\mathrm{flapper}}}\right) \cdot x_{\mathrm{cg}} \tag{2.53}$$

where m is the flapper mass (kg), $\rho_{\mathrm{flapper}} = 7800$ kg/m^3 is the density of the flapper material (steel)[17], θ_s is the flapper angle (rad) to the vertical line at closed valve position, θ is the flapper angle (rad) at the current valve position and x_{cg} is the distance (m) from the flapper attachment to the centre of gravity (see Fig. 2.12).

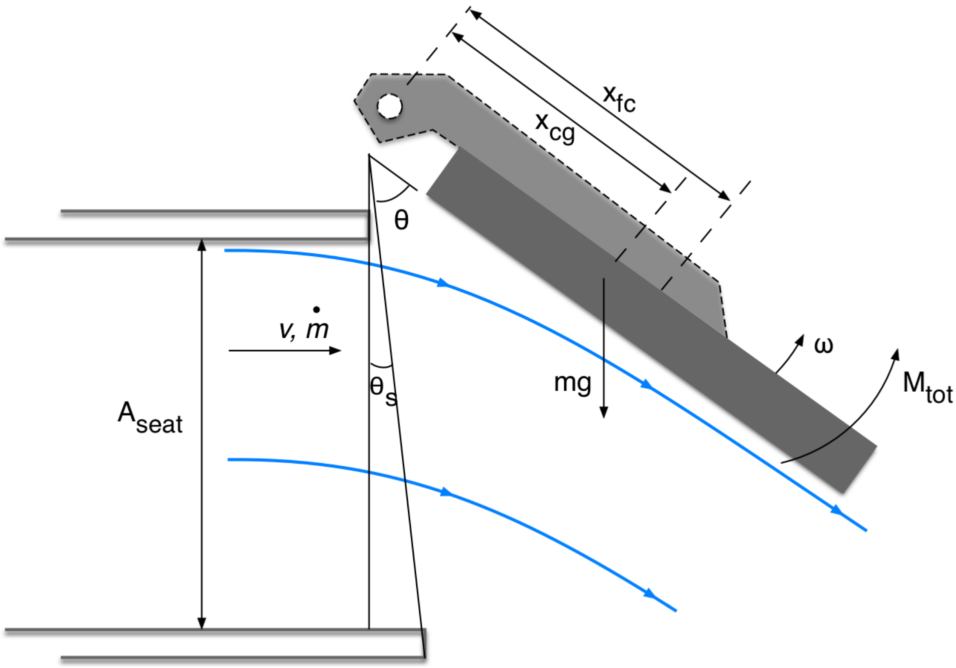


Figure 2.12: Parameters used for calculating the torque on the flapper disc due to weight, flow velocity and pressure difference.

The torque due to the flow velocity is represented by

$$M_{\mathrm{str}} = v_{\mathrm{rel}} \cdot |v_{\mathrm{rel}}| \cdot \rho \cdot A_{\mathrm{seat}} \cdot \cos^2(\theta_s + \theta) \cdot x_{\mathrm{fc}} \tag{2.54}$$

where x_{fc} is the distance (m) from the flapper attachment to the centre of the disc and A_{seat} is the valve seat area (m^2). v_{rel} is the relative velocity (m/s) between the fluid and the flapper in the direction of the flow and is defined as

$$v_{\mathrm{rel}} = v - \omega \cdot x_{\mathrm{fc}} \cdot \cos(\theta_s + \theta) \tag{2.55}$$

where ω is the angular velocity (rad/s) of the flapper disc.

[17]This parameter is used for taking the buoyancy effect on the flapper into account.

The torque due to the pressure difference is calculated by

$$M_{\Delta P} = \left(P_j - P_{j+1} - \rho \cdot g \cdot \Delta h - \rho \cdot \Delta x \cdot \frac{\Delta v}{\Delta t}\right) \cdot A_{\text{flapper}} \cdot x_{\text{fc}} \tag{2.56}$$

where P_j is the pressure (Pa) in the cell just upstream the valve, P_{j+1} is the corresponding downstream pressure (Pa), Δh is the difference in altitude (m) between two cells, Δx is the node length, $\frac{\Delta v}{\Delta t}$ is the discrete approximation of the flow acceleration (m/s^2) and A_{flapper} is the flapper disc area (m^2). It can be noticed that Eq. (2.56) takes into account a possible vertical installation of the valve. None of these two last terms in Eq. (2.56) is included in the built-in models used by RELAP5 and TRACE.

The time-controlled damping torque is modelled as

$$M_{\text{damping}} = \begin{cases} -k \cdot \omega, & t < t_{\text{damp}} \\ 0, & t \geq t_{\text{damp}} \end{cases} \tag{2.57}$$

where the constant k is the 'strength' of the damping torque and where t_{damp} usually is chosen as the starting time of the transient.

Once the total torque is determined, the angular acceleration $\dot{\omega}$ (rad/s^2) of the flapper disc is calculated by

$$\dot{\omega} = \frac{M_{\text{tot}}}{I_{\text{tot}}} \tag{2.58}$$

where I_{tot} is the moment of inertia (kg·m^2) of the flapper. The angular velocity ω is then calculated by the time integration of the angular acceleration, and the flapper angle θ is in turn the time integration of the angular velocity.

This model of an inertial swing check valve is universal and can be utilised for a specific valve by adjusting a number of parameters. The data for the valve type ('Wheatley 50 mm') that was used in [33] and in this report is listed in Table 2.1.

Table 2.1: Parameters used in the model of an inertial swing check valve. The data represents a 'Weatley 50 mm' valve type and is further described in [33].

Description	**Symbol**	**Value**
Flapper disc area	A_{flapper}	0.0044 m^2
Total flapper wight (disc + arm)	m	0.739 kg
Moment of inertia	I_{tot}	0.0018 kg
Distance, attachment → centre of gravity	x_{cg}	0.044 m
Distance, attachment → flapper disc centre	x_{fc}	0.05 m
Maximum opening angle	θ_{max}	70°
Angle from vertical line to closed valve	θ_s	14.8°
Pipe area (connected to the valve)	A_{pipe}	0.00216 m^2
Seat area	A_{seat}	0.00216 m^2

The valve characteristics, in terms of normalised flow area as a function of valve stem position, is calculated for a specific valve using an Excel-sheet included in [33]. The interested reader is referred to this document for further instructions on how this is made.

An important finding that is discussed in Chapter 6 is how the absolute flow velocity v is calculated. In the model, it is defined and implemented as the flow velocity *into* the valve (see Fig. 2.12) and is calculated according to

$$v = \frac{\dot{m}}{\rho \cdot A_{\text{seat}}} \tag{2.59}$$

where $\dot{m}$ is the mass flow (kg/s) through the valve measured by the code. However, the effects of instead measuring the flow velocity *through* the valve itself is investigated in Chapter 6. The mathematical difference compared to Eq. (2.59) is that A_{seat} is exchanged to A_{valve}, i.e. the actual valve flow area used by the codes.

Chapter 3

Methodology

A number of simulations have been carried out in this project in order to validate TRACE as a tool for modelling fast pressure transients in nuclear power plant piping systems. These simulations span a wide range of applications typical for load calculations[1], including e.g. sudden valve closure, steam collapse and pump modelling. Five main cases are considered and they are presented separately in Chapters 4 to 8. The general methodology in each of these cases is as follows. A model geometry is either built up from scratch in RELAP5 (using SNAP version 2.2.6) or imported from an existing input (.i) file. The RELAP5 model is then simulated and the results are graphically illustrated in AptPlot (version 6.5.1). The parameters that are studied in components of interest are typically mass flows, pressures, dynamic loads, void fractions and possible valve closure characteristics. The purpose is then to make the same kind of simulations with a corresponding TRACE model, obtained by converting the RELAP5 model using the built-in conversion tool in SNAP. In some cases, the TRACE model is also built up from scratch for comparison with the converted model. Experimental data is available for one of the cases (see Chapter 4), allowing for both codes to be compared to real measurements. Otherwise, the RELAP5 results are assumed to be the 'benchmark' which the TRACE results should be validated against.

In all simulations, some basic settings are used as standard and are presented and motivated in the following sections.

First of all, `BREAK` components in the TRACE models are set to having a small length (10^{-6} m) and a large volume (10^{6} m^{3}), giving a large connecting area (10^{12} m^{2}). This is one of the two recommended `BREAK` dimensions according to the TRACE Modeling Guidelines [21] (see also Section 2.2.6.5) and should preferably be used whenever the boundary condition represents a large pressure source or sink (which is most often the case in the models considered).

The typical cell length that is used for most cases is in the range 0.1-0.2 m, which is a rather common choice (for this type of analyses) that is well-balanced between

[1]By load calculation, it is meant what in Swedish is called 'belastningsunderlag', which is created for the purpose of calculating loads on piping systems following a certain transient. A more correct English translation has not been found, but perhaps a better term is water hammer simulation. These two terms should therefore be regarded as synonyms throughout this report. The calculation of the loads is described in Section 2.1.2.

computational time and accuracy. In order to fulfill the corresponding critical Courant limit (see Eq. 2.5) an appropriate maximum time step size of 10^{-5} s is used during the most important parts of a transient. However, for steady-state calculations a larger maximum time step size (e.g. $10^{-3} - 10^{-2}$ s) is usually sufficient. The standard sampling interval is set to 10^{-4} s during transients and $10^{-2} - 10^{-1}$ s for steady-state calculations. A smaller sampling interval than 10^{-4} s can give very large result files and usually does not provide any further relevant information.

The default code versions which are used in the simulations are RELAP5/MOD3.3Patch04 and TRACE V5.830 (BETA), which currently (May 2014) are the latest available. The use of TRACE V5.830 is further motivated by the ability to modify a number of sensitivity parameters such as the interfacial heat transfer coefficients (see Section 2.3.2.2). The default steam table used for all simulations is based on the IAPWS formulation and is available in both RELAP5 and TRACE. The equilibrium flag in RELAP5 is turned off by default, and the interfacial heat transfer is left untouched in TRACE. Any deviations from these standard settings are mentioned in the cases concerned.

Irrecoverable form losses at valves and orifices (see Section 2.3.1.2 for more details on this) are managed in RELAP5 by applying the 'full abrupt area' model, which is widely used whenever the exact losses are unknown. In a few cases, these form losses are implemented using a constant loss coefficient which has earlier been calculated by the use of [35]. The corresponding losses in TRACE are in most cases calculated by using the option '[0] Constant FRIC' and optimising an additional additive form loss (K-factor) so that the same steady-state condition (e.g. mass flow or valve position) as in the RELAP5 model is established. In a few cases where a large number of abrupt area changes are present in the model, making the optimisation against the RELAP5 model unmanageable, one of the following two options is used. First, the (after conversion) default option '[-1] Flow Factor + FRIC + Abrupt' is tried. This option is then compared with the results obtained by instead implementing a form loss value (or table in case of a valve) manually by turning off the internal loss model (`INTLOSSOFF`) and specifying the losses according to Eq. (2.24). The option that gives results most similar to RELAP5 is then the one that is used.

Much attention is dedicated throughout the report on ways to adjust the condensation rate calculated by the codes. In RELAP5 this is modified by activating the equilibrium flag, either locally in appropriate cells or globally for all cells in the model, making both condensation and vapourisation basically instantaneous. In TRACE, the corresponding adjustment is made by multiplying the interface-to-liquid heat transfer coefficients h_{if} with a suitable factor.

In the remainder of this chapter, the aims and methodologies of the different cases are presented briefly.

The first case in Chapter 4 is about water hammers taking place after a sudden valve closure. As described in Section 2.1, these pressure peaks are likely to appear in (and may cause damage to) piping systems in nuclear power plants. The experimental setup consists of a horizontal pipe connected upstream to a large water reservoir at a high pressure. A sudden closing of a valve downstream causes the water column in the pipe to quickly decelerate and a pressure wave to propagate backwards through the system, then reflecting back and forth against the reservoir and the closed valve. This will in turn cause a repeated series of vapourisation and steam collapse as described in Section 2.1.1. The main purpose of this experiment is to compare the water hammers, calculated by RELAP5 and TRACE just upstream the valve, with experimental data that is accessible e.g. in [36]. Since this is the only considered case where real measurements are available, a number of sensitivity analyses are carried out in order to test the robustness of the models and the uncertainty of the results.

The case in Chapter 5 is all about the study and comparison of modelling steam collapse in a vertical pipe. The upper part of the pipe is closed (i.e. a dead end) and the lower part is connected to a water reservoir keeping such a pressure that a vapour pocket is established at the top. The pressure is then quickly increased at the bottom (representing a simple simulation of a pump start) making the steam collapse, which in turn results in severe water hammers. This kind of phenomenon may occur in a system containing e.g. a pump and a closed valve placed in a vertical pipe segment. The aim of this experiment is to investigate possible differences between RELAP5 and TRACE regarding the collapse of the vapour pocket and the following water hammers.

The modelling of the inertial swing check valve presented in Section 2.4 is carried out in Chapter 6. This RELAP5 model is developed at Ringhals and is regularly used for water hammer simulations. It is therefore of interest whether it can be used also for modelling in TRACE. In this case, an existing input file of the valve model is imported to SNAP, put into place in a certain geometrical setup and then converted to a TRACE model. The closing behaviour is studied carefully and compared with that of the original RELAP5 model. It should be mentioned that both RELAP5 and TRACE provide a built-in model of an inertial swing check valve. However, these are known to be inadequate and are not considered in this report. A study of these built-in models have earlier been carried out in [34].

In Chapter 7, an existing RELAP5 load calculation [37] is imported to SNAP and converted to a TRACE model. This chapter is very important because it somewhat demonstrates whether TRACE is useful and reliable for this kind of work. The load calculation considered is about opening or closing of a safety valve in the residual heat removal system (system 321) at Ringhals 3 and contains most of the concepts investigated in Chapter 4 and 5. First, the system response of opening a safety valve is studied in Section 7.1, where hot water suddenly drains a piping system initially partly vapour-filled. In Section 7.2, the same safety valve is instead closed, leading to a pressure depressurisation and vapourisation downstream the valve.

Last but not least, in Chapter 8 an existing RELAP5 load calculation [38] is imported to SNAP and converted to a TRACE model (i.e. similar procedure as in Chapter 7). In this water hammer simulation a part of the residual heat removal system (system 321) at Ringhals 4 is modelled, and the simulated transient represents a pump stop. The model further contains several independent instances of the inertial swing check valve presented in Section 2.4 and studied in Chapter 6. The purpose of this chapter is twofold; firstly, it is of interest how the conversion of a pump component is handled and, secondly, if the gained experience of the inertial swing check valve is applicable also when it is utilised in a more complex system.

Chapter 4

Steam Collapse in a Horizontal Pipe

This chapter is about water hammers in a horizontal pipe originating from a sudden valve closure, which, as described in earlier sections, are common events in piping systems. The setup in this chapter is kept simple to be able to study the wave propagation in the pipe. The main focus is on how RELAP5 and TRACE model steam collapse and how this affects the pressure peaks just upstream the valve. The results from the two codes are compared to experimental data obtained from [36] and [39], in which the same experiment has been carried out in RELAP5. The geometry has been chosen in order to correspond to the geometry in [36] and [39].

4.1 Method

The experimental setup is shown in Fig. 4.1 and consists of a 36 m horizontal pipe connected upstream to a large water tank (with a constant pressure and temperature of 1 MPa and 436 K, respectively) and downstream to atmospheric conditions. The steady-state velocity in the pipe is 0.4 m/s, and at the end of the pipe there is a valve which will be closed instantaneously. The experimental data that is available is the pressure measured just upstream the valve as represented by the figure. This is also what the results from RELAP5 and TRACE are compared with.

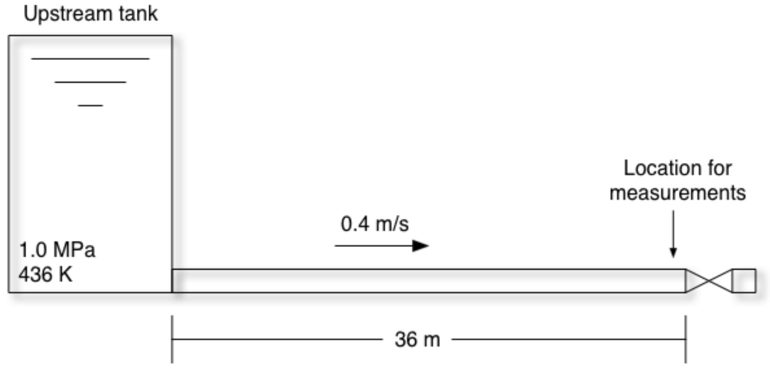


Figure 4.1: Experimental setup for the case of studying steam collapse in a horizontal pipe following a sudden closing of a valve. Downstream the valve there are atmospheric conditions.

4.1.1 Modelling in RELAP5

A representation of the RELAP5 model is illustrated in Fig. 4.2. The pipe consists of four segments, each with 72 cells with a node length of 0.125 m and a cross-sectional area of 0.1 m^2, giving the same cell dimensions as in [36]. The upstream tank is modelled using a `TMDPVOL` with the given pressure and temperature conditions. A `TMDPVOL` is also used downstream the valve with a constant pressure of 0.1 MPa and temperature of 300 K, representing atmospheric conditions. The valve flow area is set to $1.44 \cdot 10^{-3}$ m^2 and the pipe wall roughness is set to $2.5 \cdot 10^{-5}$ m, giving the steady-state initial velocity of 0.4 m/s. The valve closure time is set to 1 ms. The 'full abrupt area' model is used for the valve. The component data is summarised in Table 4.1.

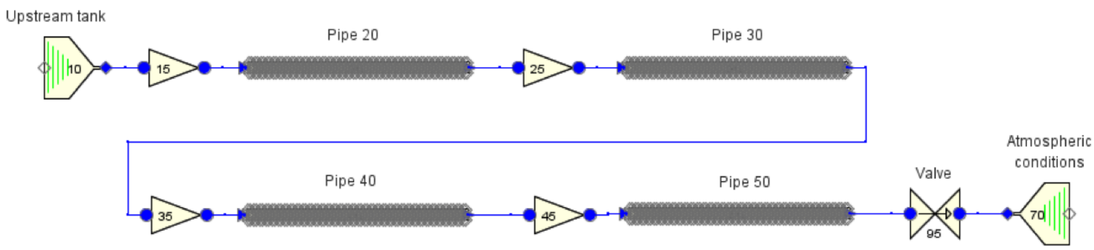


Figure 4.2: RELAP5 model of the setup shown in Fig. 4.1.

Table 4.1: Model-related data used for the RELAP5 model of steam collapse in a horizontal pipe.

Component type	**Component number**	**Description**
`TMDPVOL` (Upstream tank)	010	`TIME-DEPENDENT VOLUME` with constant pressure (1.0 MPa) and constant temperature (436 K). Component dimensions: volume = 10^6 m^3, length = 10^{-6} m, area = 10^{12} m^2.
Junctions	015, 025, 035, 045	Flow area = 0.1 m^2.
Pipe	020, 030, 040, 050	Four `PIPE` segments, each with 72 cells with a node length of 0.125 m and cross-sectional area of 0.1 m^2. Wall roughness = $2.5 \cdot 10^{-5}$ m. Steady-state flow velocity = 0.4 m/s.
Valve	095	Servo valve with initial flow area of $1.44 \cdot 10^{-3}$ m^2. 'full abrupt area' model is used. Closure time of 1 ms.
`TMDPVOL` (Atmospheric conditions)	070	`TIME-DEPENDENT VOLUME` with constant pressure (0.1 MPa) and constant temperature (300 K). Component dimensions: volume = 10^6 m^3, length = 10^{-6} m, area = 10^{12} m^2.

The experiment is simulated for 5 s in order to reach steady-state. The valve is then closed in a linear manner between 5.000 s and 5.001 s. The following transient is simulated for another 0.25 s for which experimental data is available. The pressure is studied in the last cell of `PIPE` 050 just upstream the valve. Two RELAP5 simulations are performed; one using the default non-equilibrium model and one when the equilibrium flag (`e` = [1]) is activated in all pipe cells.

4.1.2 Modelling in TRACE

A representation of the TRACE model is illustrated in Fig. 4.3. Similar to the RELAP5 model, the pipe consists of four segments, each with 72 cells with a node length of 0.125 m and a cross-sectional area of 0.1 m^2. The upstream tank is modelled using a `BREAK` component with the given pressure and temperature conditions. A `BREAK` is also used for the downstream atmospheric conditions. The flow area of the valve is set to $1.44 \cdot 10^{-3}$ m^2 and the wall roughness is $2.5 \cdot 10^{-5}$ m. In contrast to the RELAP5 model, the abrupt area change model is not used for the valve. Instead, the option '[0] Constant FRIC' is used, together with an additional additive constant K-factor (=1.1), in order to obtain a steady flow of 0.4 m/s through the pipe. Another choice would be to implement the same loss coefficient as calculated by the 'full abrupt area' model in RELAP5 according to the methodology in Eq. (2.24). The valve closure time is 1 ms. The component data is summarised in Table 4.2.

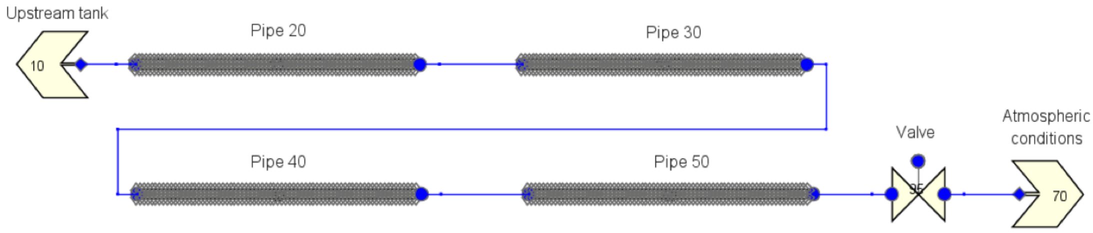


Figure 4.3: TRACE model of the setup shown in Fig. 4.1.

Table 4.2: Model-related data used for the TRACE model of steam collapse in a horizontal pipe.

Component type	**Component number**	**Description**
`BREAK` (Upstream tank)	10	`BREAK` with constant pressure (1.0 MPa) and constant temperature (436 K). Component dimensions: volume = 10^6 m^3, length = 10^{-6} m, area = 10^{12} m^2.
Pipe	20, 30, 40, 50	Four `PIPE` segments, each with 72 cells with a node length of 0.125 m and cross-sectional area of 0.1 m^2. Wall roughness = $2.5 \cdot 10^{-5}$ m. Steady-state flow velocity = 0.4 m/s.
Valve	95	Valve type: 'Flow area fraction table'. Initial flow area of $1.44 \cdot 10^{-3}$ m^2. '[0] Constant FRIC' + K-factor=1.1. Closure time of 1 ms.
`BREAK` (Atmospheric conditions)	070	`BREAK` with constant pressure (0.1 MPa) and constant temperature (300 K). Component dimensions: volume = 10^6 m^3, length = 10^{-6} m, area = 10^{12} m^2.

The experiment is simulated for 5 s in order to reach steady-state. The valve is then closed in a linear manner between 5.000 s and 5.001 s. The following transient is simulated for another 0.25 s for which experimental data is available. The pressure is studied in the last cell of `PIPE` 50 just upstream the valve. Two TRACE simulations are performed; one for the default case (unmodified interfacial heat transfer coefficient h_{if}) and one by setting $h_{\text{if}} \times 10^6$. The latter simulation is to make the interfacial heat transfer (and thus the condensation and vapourisation rates) basically as fast as in the equilibrium case in RELAP5.

4.1.3 Converted Model

The equilibrium RELAP5 model is converted to a TRACE model using the built-in conversion tool in SNAP. Two simulation cases are considered: one 'as is' (without any parameter modifications) and one where necessary parameters are changed to the ones used in the original TRACE model described in the previous section.

4.1.4 Sensitivity Analysis

A number of simulations are performed in order to test the robustness of the models and the uncertainty of the results. The equilibrium case in RELAP5 and the case with $h_{\text{if}} \times 10^6$ in TRACE are used as standard for all simulations (as will be clear in Section 4.2). The standard maximum time step is 10^{-5} s and the sampling interval is 10^{-4} s. RELAP5/MOD3.3Patch04 and TRACE V5.830 (BETA) are the standard code versions.

First, a sensitivity analysis is carried out for different interfacial heat transfer coefficients (h_{if}) in TRACE in order to find a suitable value for matching the results from RELAP5 and from the measurements. Simulations are performed for the default case (unmodified h_{if}) and for $h_{\text{if}} \times 5$, $h_{\text{if}} \times 50$ and $h_{\text{if}} \times 10^6$.

Then, the following sensitivity analyses are carried out for both RELAP5 and TRACE:

- The influence of different cell sizes is investigated. The node lengths that are used are 0.0625 m, 0.125 m (original case) and 0.250 m.
- The original case with a maximum time step of 10^{-5} s is compared with maximum time steps of 10^{-4} s and 10^{-6} s. The sampling interval is still 10^{-4} s in all cases.
- Possible effects of switching off the water packing model are studied.
- The original linear valve closure is compared with a more realistic S-curve characteristic with a total closure time of 1 ms.
- RELAP5/MOD3.3Patch03 and Patch04 are compared. The same for TRACE V5.0 and V5.830 (BETA).
- A sensitivity analysis is performed in RELAP5 using the default IAPWS steam table and the user-specified 'H2OOLD'. A similar analysis is carried out using the default IAPWS steam table in TRACE V5.830 and the old table inherited from TRAC-PF1.

4.2 Results and Discussion

The results are presented in three main sections below. First, the overall pressure behavior calculated by RELAP5 and TRACE is compared with the corresponding experimental data. Second, the results using the converted model are presented briefly. The results from the different sensitivity analyses are presented in the last section.

4.2.1 Overall Pressure Behaviour Upstream the Valve

The pressure behaviour just upstream the valve, following the instantaneous valve closure, is shown in Fig. 4.4 for the default case where the non-equilibrium model is utilised in RELAP5 and the interfacial heat transfer coefficient h_{if} is unmodified in TRACE. It can be observed that the results from the two codes are almost identical up to about 5.05 s (0.05 s after the valve closure), where the pressure wave has been reflected in the upstream tank and returned to the valve with a negative sign (cf. Section 2.1.1). At this point, RELAP5 shows a small pressure undershoot which is not apparent in either TRACE or the experimental data. The amplitude of the first pressure peak (between 5.00 and 5.05 s) is overestimated in both codes compared to the measurements. This has also been seen in other simulations using the same experimental data (see e.g. [36], [39] and [40]). This may be due to a slightly higher steady-state velocity in the simulations compared to the experiment. However, reducing this velocity to about 0.36 m/s, so that the first pressure peak coincides with the measurements, results in the second pressure peak (between 5.1 s and 5.2 s) deviating even more compared to the experimental pressure, both in time and shape. This second pressure peak (first caused by the steam collapse and later by a superposition with the original pressure wave from the valve closure) differs rather much in comparison with the experimental data. While RELAP5 shows a similar shape and maximum amplitude compared to the measurements, TRACE has a completely different look. It has been found that both RELAP5 and TRACE have a too slow condensation rate, which underestimates the pressure pulse and gives a dampened behaviour. This has also been confirmed in [6] and [7], where the slow condensation rates predicted by both codes are considered as an issue and that further development of interfacial heat transfer modelling is required. The largely distorted shape of the pressure peak in TRACE is assumed to be due to the condensation rate being even slower in TRACE than in RELAP5. This is illustrated in Figs. 4.8 and 4.9 below, where it is discussed further.

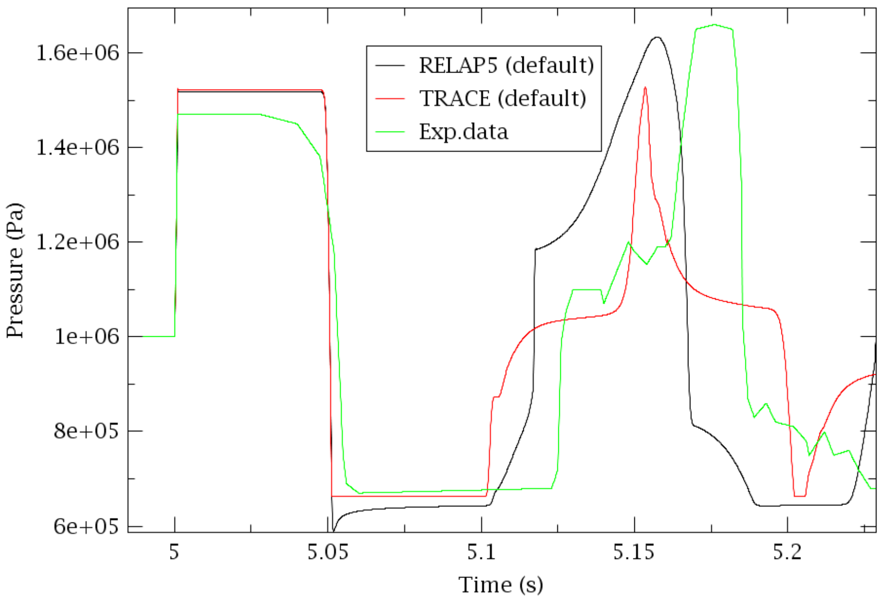


Figure 4.4: Predicted and measured pressure upstream the valve. The transient starts at 5.0 s where the valve is closed instantaneously. The default configurations are the non-equilibrium model (RELAP5) and unmodified interfacial heat transfer (TRACE), respectively. The difference in shape is obvious, both between the codes and compared to experimental data. It has been found that this is due to a too slow condensation rate as calculated by the codes.

Fig. 4.5 shows the case when the interfacial heat transfer rate has been modified, in RELAP5 by using the equilibrium model and in TRACE by setting a multiplicative factor of 10^6 in front of h_{if}. Both codes then give very similar results also for the second pressure peak, and are also closer to the shape of the measurements. However, the maximum amplitude is instead overestimated by roughly 1.4 bar. It should be mentioned though that the equilibrium case is not realistic either since the condensation rate now is (more or less) instantaneous for both codes. In order to obtain more conservative results, the equilibrium model in RELAP5 and a corresponding large factor for h_{if} in TRACE should preferably be used in this type of water hammer experiment. This has earlier been concluded and recommended for RELAP5 [36, 40] and for TRACE [6, 7].

It can also be noticed that the second pressure peak occurs earlier in the simulations compared to the real case, which most likely is due to the elasticity of the pipe that is not taken into account in either RELAP5 or TRACE. This lack of FSI (fluid-structure interaction) is also another shortage of the codes mentioned in e.g. [7]. No solution approach has been found to overcome this time offset.

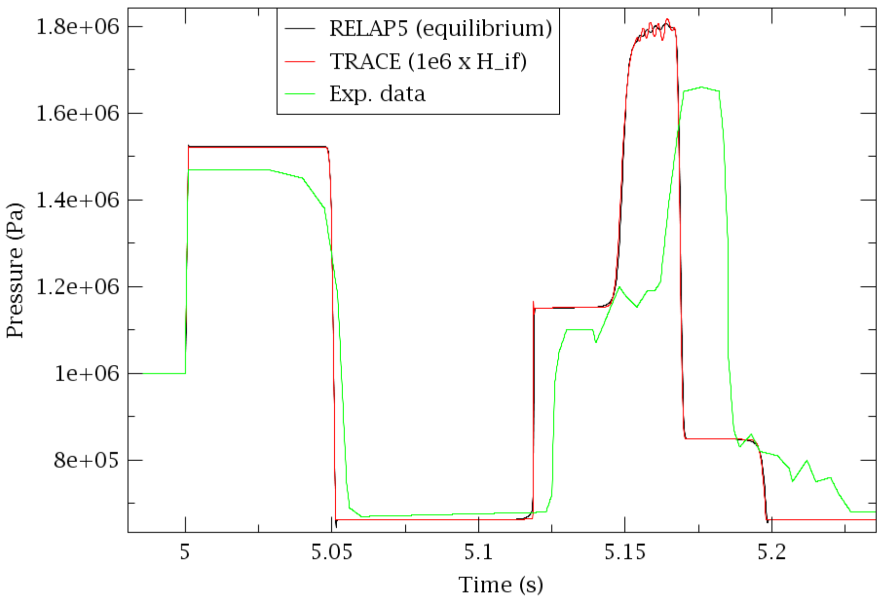


Figure 4.5: Predicted and measured pressure upstream the valve. The RELAP5 equilibrium flag is activated in all pipe cells. The interfacial heat transfer rate is modified in TRACE by setting $h_{\text{if}} \times 10^6$. The codes give very similar results, and are also closer to the experimental results than in Fig. 4.4.

4.2.2 Converted Model

Fig. 4.6 shows the results using the converted TRACE model compared with the 'built-from-scratch' RELAP5 and TRACE models presented in the previous section. It can be seen that the converted model is identical to the TRACE model – when a few parameters have been modified. The changes that are made in the converted model are the following (where the parameter values are changed to the ones used in the original TRACE model):

- The dimensions of the `BREAK` components
- The 'Friction Factor Correlation Option' (`NFF`) for the valve
- 'Liquid to interface bubbly-slug heat transfer coefficient' (h_{if})

By not adjusting these parameters, the results will be similar to the TRACE default case shown in Fig. 4.4, but with a noticeably higher first (and slightly lower second) pressure peak. This is due to a higher steady-state flow velocity (0.44 m/s compared to 0.40 m/s) obtained when the friction option for the valve is not adjusted. The fact that the losses across the valve is different in TRACE compared to RELAP5 is explained by how the abrupt area change is modelled in the two codes (see Section 2.3.1.2).

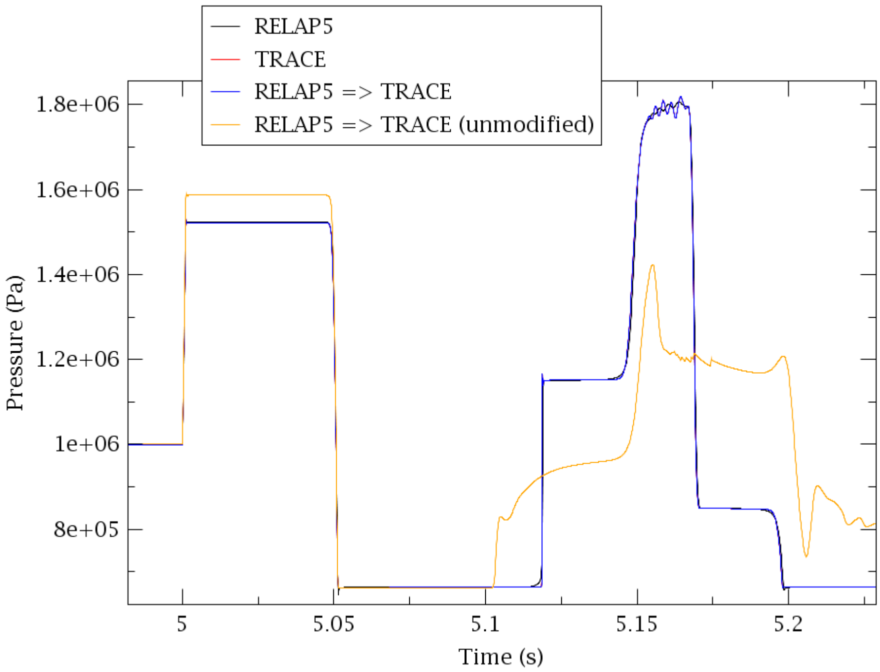


Figure 4.6: Pressure calculated using the converted model. The pressure is identical to the original TRACE model when a few parameters have been changed. The unmodified model is 'as is', i.e. no parameters have been changed after the conversion.

4.2.3 Sensitivity Analysis

The results from the different sensitivity analyses are presented in the sections below, starting with the interfacial heat transfer.

4.2.3.1 Interfacial Heat Transfer Coefficient (h_{if}) in TRACE

Fig. 4.7 shows how the second pressure peak is affected by the interfacial heat transfer coefficient. The default case gives a pressure behaviour that deviates quite dramatically from both the other cases and from the experimental data, which is due to a too slow condensation rate (as mentioned in Section 4.2.1 above). The effect of the (in reality) rapid steam collapse is therefore not fully captured in TRACE, resulting in a dampened behaviour and an underestimated amplitude. However, the amplitude and sharpness of the peak increase with increasing heat transfer coefficient, but a factor larger than about 50× shows only little further improvements.

This phenomenon is further illustrated in Fig. 4.8 where the vapour generation rate (kg/(m^3· s)) is shown for the different cases (positive values represent vapourisation and negative values condensation). Like in Fig. 4.7 the results are similar up to about 5.1 s (during the first vapourisation phase) but starts to differ when condensation starts to occur. The default case clearly has a relatively slow and prolonged condensation phase, while the $h_{\mathrm{if}} \times 50$ and $h_{\mathrm{if}} \times 10^6$ cases exhibit much steeper gradients.

For comparison, Fig. 4.8(b) shows the vapour generation rate for the equilibrium and non-equilibrium cases in RELAP5 together with the $h_{\mathrm{if}} \times 10^6$ case in TRACE. It can be seen that both condensation and vapourisation are very slow when the default non-equilibrium model is used, and that RELAP5 and TRACE can be quite similar when their respective heat transfer rate is modified.

A final remark is how much steam that is produced during the vapourisation phases. Fig. 4.9(a) illustrates that the void fraction is especially small in the RELAP5 default case. Even though the void fraction is greater for the TRACE default case, the condensation rate is (at least) as slow as in RELAP5. The condensation rates of the modified cases (equilibrium and $h_{\mathrm{if}} \times 10^6$, respectively) are much faster. However, the most interesting part is Fig. 4.9(b), which clearly shows that a small remaining vapour pocket is left in the TRACE default case. Thus, no complete steam collapse occurs, which further explains the dampened behaviour of the pressure peak in Fig. 4.4.

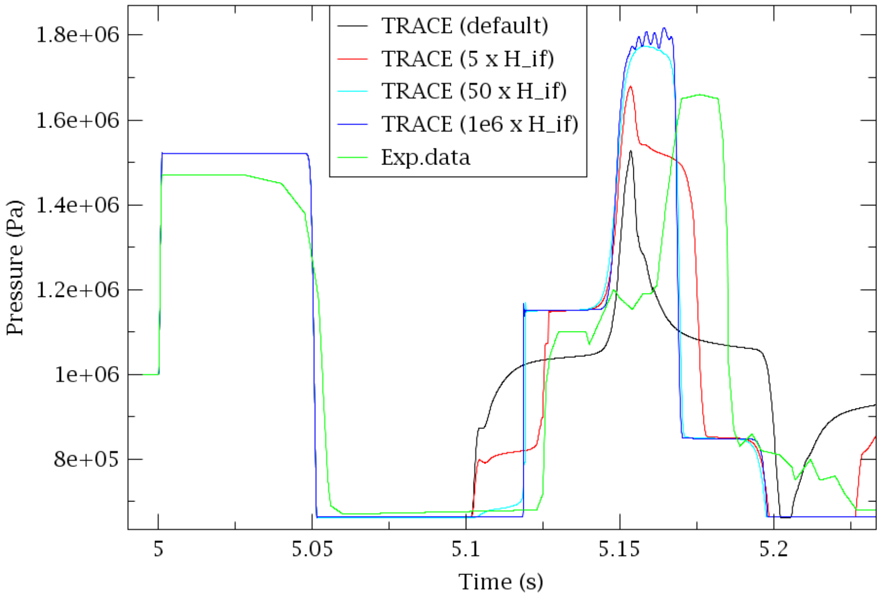
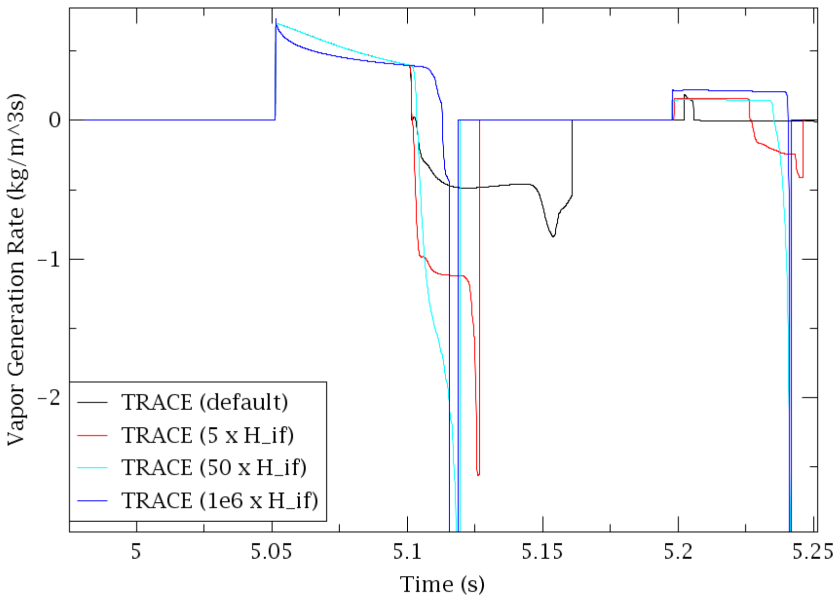
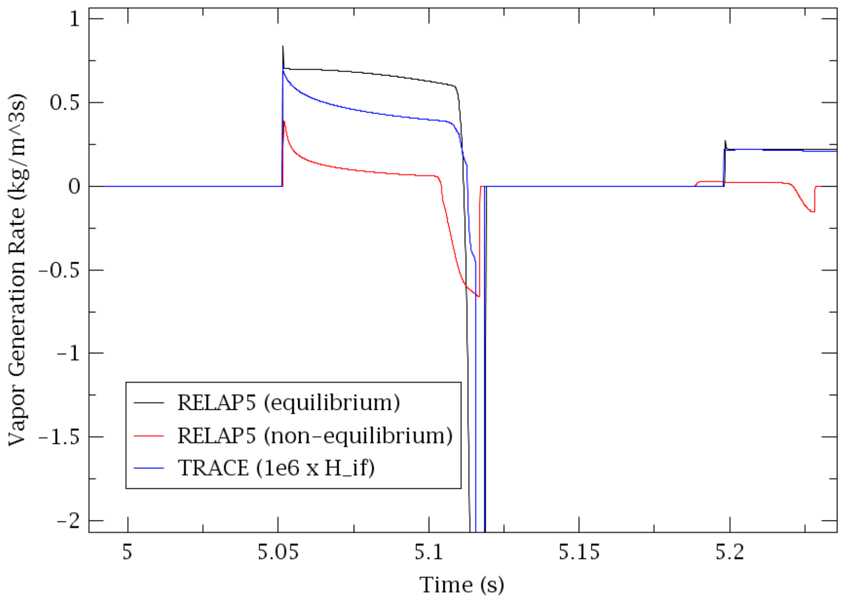


Figure 4.7: The effect of different interfacial heat transfer coefficients in TRACE. The shape of the curve becomes more similar to experimental data as the heat transfer coefficient is increased. Also, the pressure peak is overpredicted, which is more conservative.

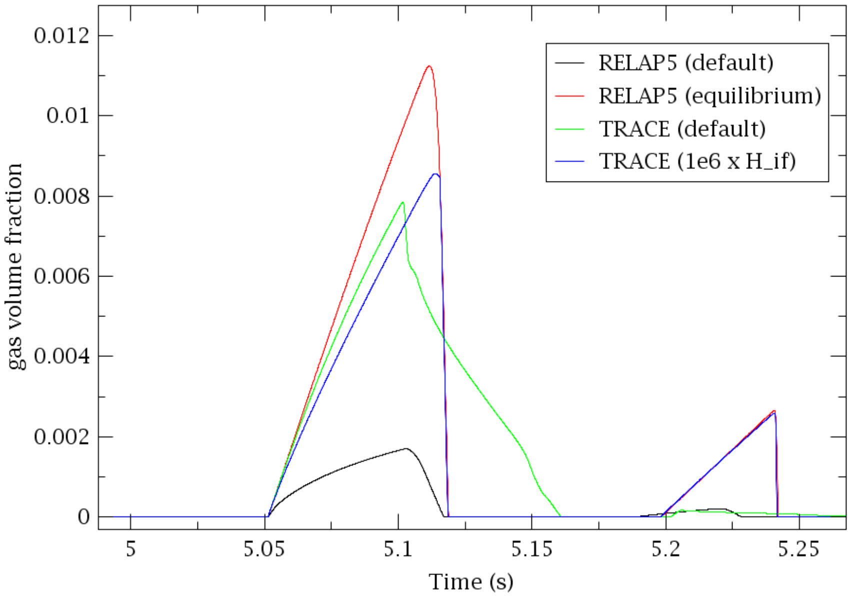


(a)

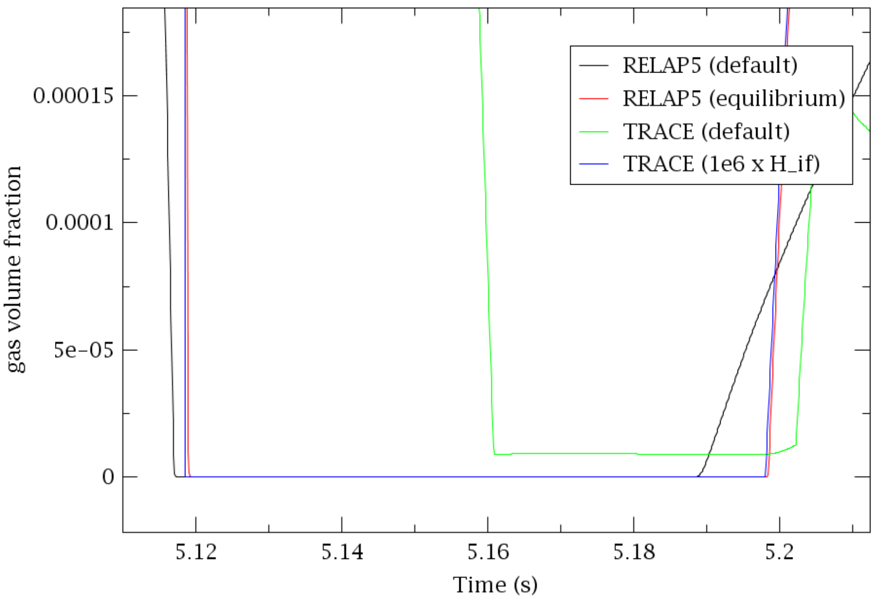


(b)

Figure 4.8: Rate of vapourisation (positive values) and condensation (negative values) in TRACE and RELAP5. Increasing the heat transfer coefficient results in faster condensation (larger negative vapour generation rate) in TRACE. The same is observed when the equilibrium flag is activated in RELAP5.



(a)

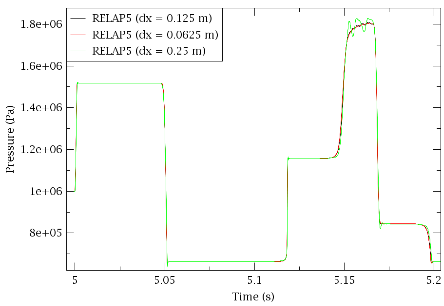


(b)

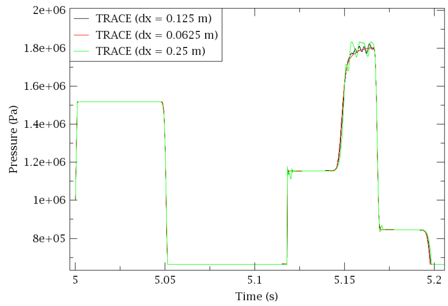
Figure 4.9: Void fraction between RELAP5 (default and equilibrium) and TRACE (default and $h_{\text{if}} \times 10^6$). Observe in (a) how much steam that is produced in the different cases and how fast it condenses. Part (b) shows a magnification of the void fraction after the first steam collapse. Notice that a small vapour pocket is still left in the TRACE default case.

4.2.3.2 Cell Size

The influence of different cell sizes is shown in Fig. 4.10(a) for RELAP5 and Fig. 4.10(b) for TRACE. The overall difference is minor, but it is clear that a smaller cell size gives a smoother pressure behaviour with less unphysical oscillations. The choice of 0.125 m as the standard node length is satisfactory, but 0.0625 m should preferably be used when calculation effort is not a limiting factor. However, reducing the cell size also means that the time step may need to be reduced correspondingly in order to fulfill the critical Courant limit.



(a) RELAP5.

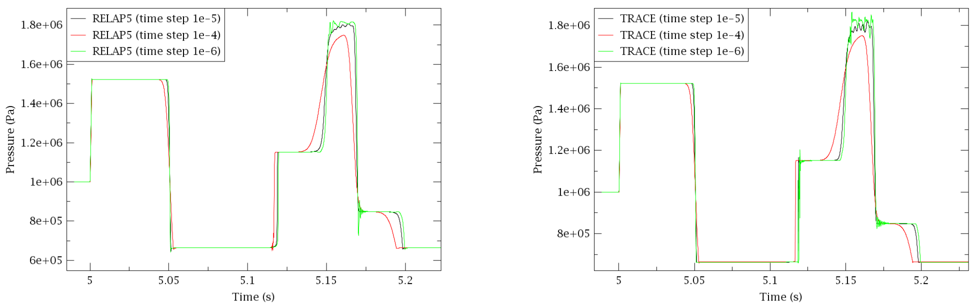


(b) TRACE.

Figure 4.10: The effect of different cell lengths (0.125 m is default). There are no major differences between the different cell sizes, although the peak becomes more 'wavy' for larger cell sizes.

4.2.3.3 Time Step

Figs. 4.11 show the effect of different maximum time step sizes. The overall results are similar in both codes, i.e. that a large time step captures less details and gives a more dampened behaviour compared to a smaller time step. However, a smaller time step also shows some numerical oscillations. The default maximum time step of 10^{-5} s seems acceptable in this case.

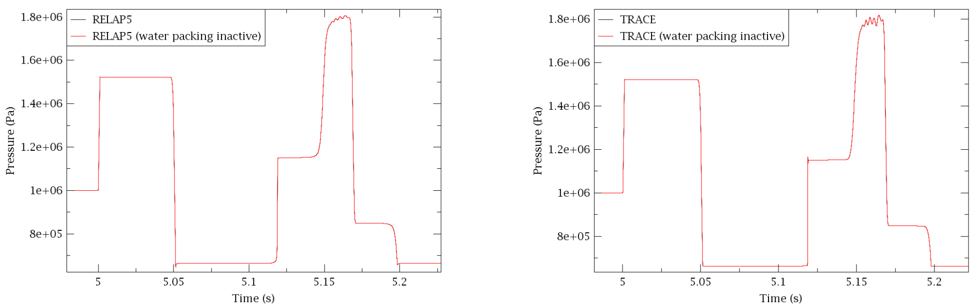


(a) RELAP5. (b) TRACE.

Figure 4.11: The effect of different maximum time steps (10^{-5} s is default). A larger time step results in slightly smaller amplitudes, whereas a smaller time step results in some numerical noise.

4.2.3.4 Water Packing

Fig. 4.12 show that switching off the water packing model has no effect in this case for neither RELAP5 or TRACE. This should not be surprising, at least not in RELAP5, since the water packing mitigation scheme is not activated in this code for horizontal flows (see Section 2.3.4). Some degree of horizontal stratification is present in both codes close to the valve, further indicating that water packing is not likely to occur.

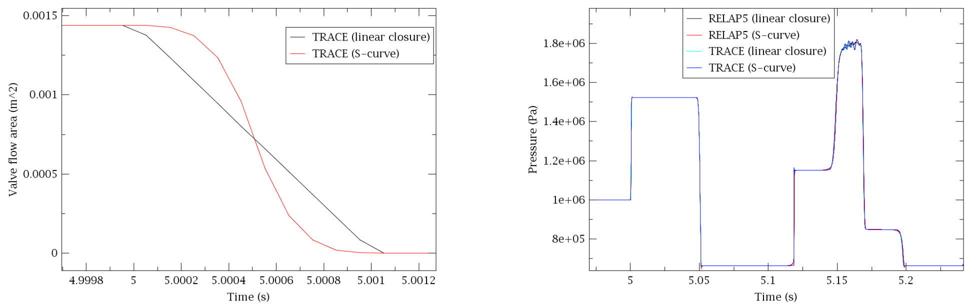


(a) RELAP5. (b) TRACE.

Figure 4.12: The effect of switching off the water packing model. The results are identical.

4.2.3.5 Valve Closure Characteristic

Fig. 4.13(a) illustrates the shape of the more realistic S-curve compared with the standard linear closure characteristic and Fig. 4.13(b) shows the corresponding pressure for both RELAP5 and TRACE. It is clearly seen that the difference is negligible, which is expected for such a short closure time (1 ms).



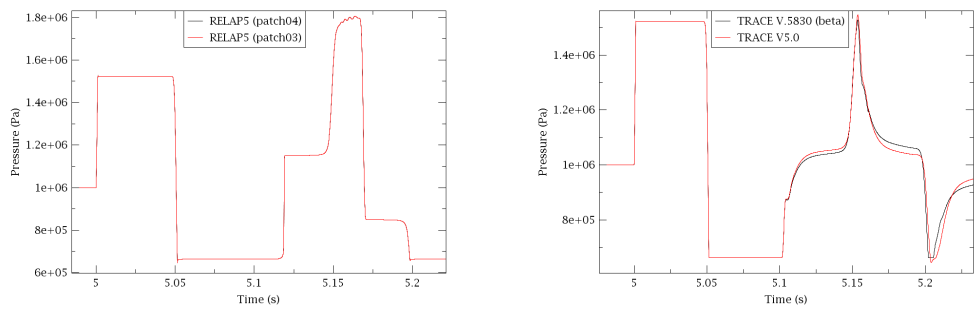
(a) (b)

Figure 4.13: The effect of different valve closure characteristic. The results are very similar within each code.

4.2.3.6 Code Version

Fig. 4.14(a) shows the difference between the two different patches of RELAP5/MOD3.3 (Patch04 and Patch03). They are found to be identical in this experiment.

Fig. 4.14(b) shows the difference between the V5.830 (BETA) and V5.0 (Patch 3) versions of TRACE. It has to be mentioned that the two versions had to be compared using an unmodified interfacial heat transfer coefficient (default case) since V5.0 does not support changing this parameter. The same IAPWS steam table is used for both cases. The underlying reason for the small difference between the two versions is unknown. However, there are some deviations in the vapour generation that may explain the slightly different behavior. The recommendation is to use the V5.830 (BETA) version to be able to adjust the interfacial heat transfer rate.



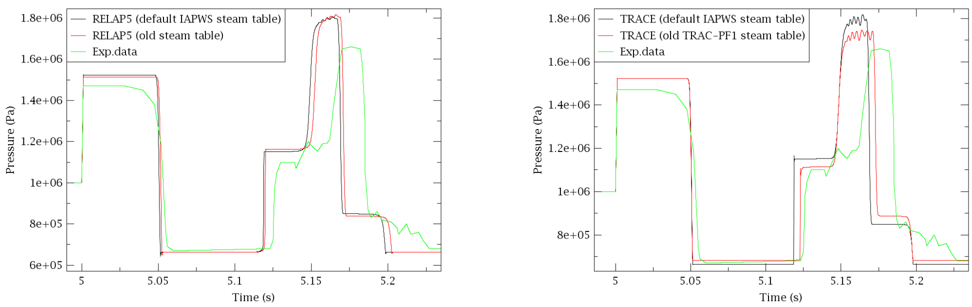
(a) RELAP5. (b) TRACE.

Figure 4.14: The effect of different code versions. No difference is observed in RELAP5. In TRACE the result is slightly different.

4.2.3.7 Steam Table

Fig. 4.15(a) shows the RELAP5 results using the default IAPWS steam table and the old 'H2OOLD' steam table[1]. There are some minor differences between the two tables, although not significant. The recommendation is simply to use the default table.

The corresponding results using TRACE are shown in Fig 4.15(b). In this case, there is a noteworthy difference between the default and the old TRAC-PF1 steam table, where the old one gives results more similar to the experimental data. Both the timing of the steam collapse (at about 5.125 s) and the pressure peak are slightly closer to the measurements. Thus, the old steam table may be recommended to use, at least in this type of experiment. It should also be mentioned that the default steam table in TRACE V5.0 is the old TRAC-PF1 steam table. At this point, it is uncertain which steam table will become default in the next actual release of TRACE.



(a) RELAP5. (b) TRACE.

Figure 4.15: The effect of different steam tables. Some differences are noticeable, especially in TRACE where the second pressure peak is slightly offset in time and having a smaller amplitude compared to the default steam table. It may be motivated, at least in this type of transient, to use the old TRAC-PF1 steam table. The differences in RELAP5 are rather minor though.

[1]There seems to be a bug in SNAP when switching to 'H2OOLD' in RELAP5. In order to make the simulations work one has to rename the steam table file, located in the same folder as the executable RELAP5 file, from 'tpfh2o' to 'tpf2ho'.

Chapter 5

Steam Collapse in a Vertical Pipe

This chapter is about water hammers in a vertical pipe. The origin of such water hammers could e.g. be the start of a pump together with a stuck closed valve in a vertical pipe segment. No experimental data is available, which means that the codes are only compared to each other. The main focus is, as in Chapter 4, to study how the two codes handle steam collapse. The geometry is slightly more complicated in this case though, since a vertical pipe is involved. The dimensions of the pipe have been chosen in order to simulate some sort of 'typical' case, which could arise in a nuclear power plant. Further, the length of the pipe should preferably be as long as possible in order to have a reasonable time gap between the water hammers caused by sonic pressure waves propagating up and down. However, the pipe should preferably not be unrealistically tall either. Thus, a length of 20 m seems like a good compromise.

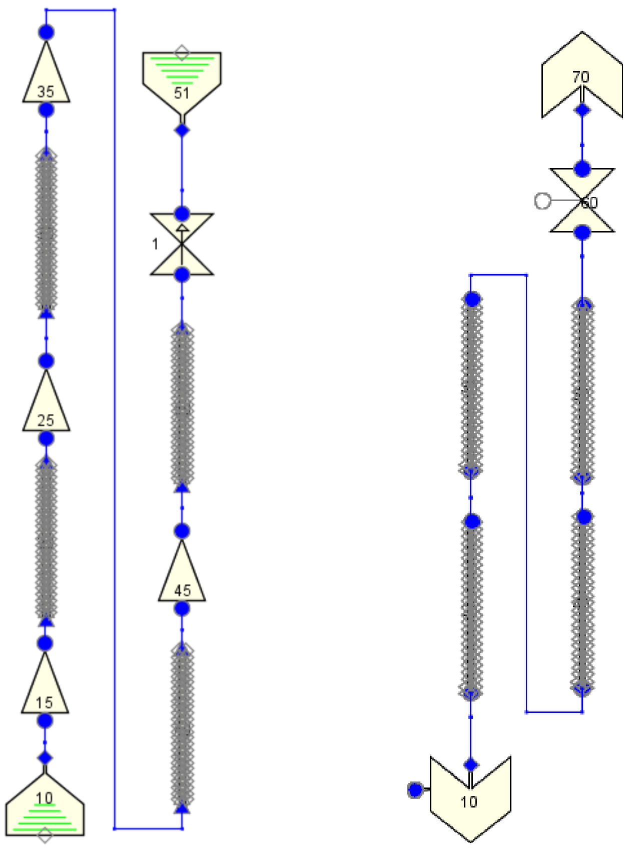
The effects of using the level tracking method are investigated. Also, the time of pressure increase in the bottom of the pipe is studied.

5.1 Method

In this case a vertical pipe of length 20 m with a diameter of 0.2 m, as can be seen in Figs. 5.1(a) and 5.1(b), is studied. The upper end of the pipe consists of a closed valve (using the `VALVE` component[1]), and the lower end consists of a pressure boundary condition. The fluid temperature is set to 293 K and the pipe is initially filled with liquid water up to 19.64 m, whereas the uppermost part of the pipe contains vapour. This is achieved by setting the pressure boundary condition to an initial value that would result in saturation conditions at a height of 19.64 m. At 293 K this corresponds to 1.94615 bar. A sensitivity analysis with respect to the node length is performed (using

[1] It should be mentioned that the dead end as well can be represented by just the pipe end itself. The reason for choosing a closed valve is to have a more realistic setup, even though the results should be the same. However, it can be shown (in both RELAP5 and TRACE) that the results, using a closed valve versus only the pipe end, are not identical but differ slightly. The small differences appear mostly in the numerical dampening of the water hammers, but also slightly in the pressure amplitudes. The reason for this is unclear though.

pressure as the benchmarking parameter), resulting in the pipe being divided into 100 nodes of length 0.2 m, which are evenly distributed across four PIPE components. The reason for dividing the pipe into several components is that RELAP5 has an upper limit of 99 cells per component. Other tested node lengths are 0.1 m and 0.4 m.



(a) RELAP5. (b) TRACE.

Figure 5.1: Schematics of the vertical pipe. The valves at the top of the pipe in the respective model is closed at all times.

The boundary condition at the bottom is modelled as a TMDPVOL component in RELAP5 and as a BREAK component in TRACE. The geometry of the BREAK component is set according to the guidelines in Volume II of the TRACE manual [21] which are described in Section 2.2.6.5. The BREAK component is set to have a length of 10^{-6} m and a volume of 10^6 m^3. This way of specifying the geometry of the BREAK component is in agreement with the guidelines. The data is summarised in Table 5.1.

Table 5.1: Data used in the simulations of steam collapse in a vertical pipe.

Pipe length	20 m
Pipe diameter	0.2 m
Node length	0.2 m
Temperature	293 K
Dead end	Closed valve
Initial pressure at lower boundary	1.94615 bar
Initial height of water level	19.64 m
`BREAK` length (TRACE)	10^{-6} m
`BREAK` volume (TRACE)	10^{6} m
Pressure increase time	1, 100, 500 or 2000 ms

First an initialisation is performed, with the purpose of reaching steady-state conditions in all cells of the system – i.e. to make sure that all velocities are zero and that there are no significant changes to other parameters such as pressure or void fraction in any cell.

At time $t = 1$ s (assuming steady-state is achieved at time $t = 0$ s) the pressure in the bottom of the pipe is increased to 13.42385 bar[2] over a number of different time periods (with the standard case used for most simulations being 100 ms), which results in a collapse of the steam in the upper part of the pipe. This in turn gives rise to a large pressure wave in the pipe. Such a situation could e.g. occur in a system where a pump is suddenly started, without downstream valves opening. However, the simulated system is constructed in an as simple manner as possible, in order to minimise complexity due to pressure wave reflections etc. Therefore, the model is constructed as described above.

The most interesting parameters to study are pressures, void fractions, mass flows etc. Since no experimental data is available for this case, the main objective is to find settings in TRACE which yield similar results as RELAP5. In some simulations it is also considered whether it is possible to get more realistic results from TRACE. However, 'more realistic' in this context is based on the (limited) experience of the authors of this report. Generally, a conservative approach is used, i.e. simulations giving higher pressure peaks are preferred over those giving lower pressure peaks.

In RELAP5 simulations are performed with the equilibrium flag turned on and off, respectively. In TRACE different multiplicative factors for the interface-to-liquid heat transfer coefficient h_{if}, which is described in Section 2.3.2, are tested. In both codes simulations are also performed with the level tracking flag turned on and off, respectively.

All simulations (in both codes) are performed using the default IAPWS steam table.

[2] The reason for this particular pressure increase (13.42385 − 1.94615 = 11.4777 bar) is that this should correspond to the rated pressure head of a typical power plant centrifugal pump. After some consideration, the choice was to use the data from the pump studied in [34], which is based on one of the pumps (323P4) in the emergency core cooling system (ECCS) of unit 1 and 2 at Forsmark nuclear power plant. The rated head rise (H) of this pump is 117 m, which, converted to pressure, gives $P = \rho g H = 1000 \cdot 9.81 \cdot 117 = 11.4777$ bar.

Finally, the model built-in RELAP5 is converted to TRACE and compared to the models built in RELAP5 and TRACE, respectively.

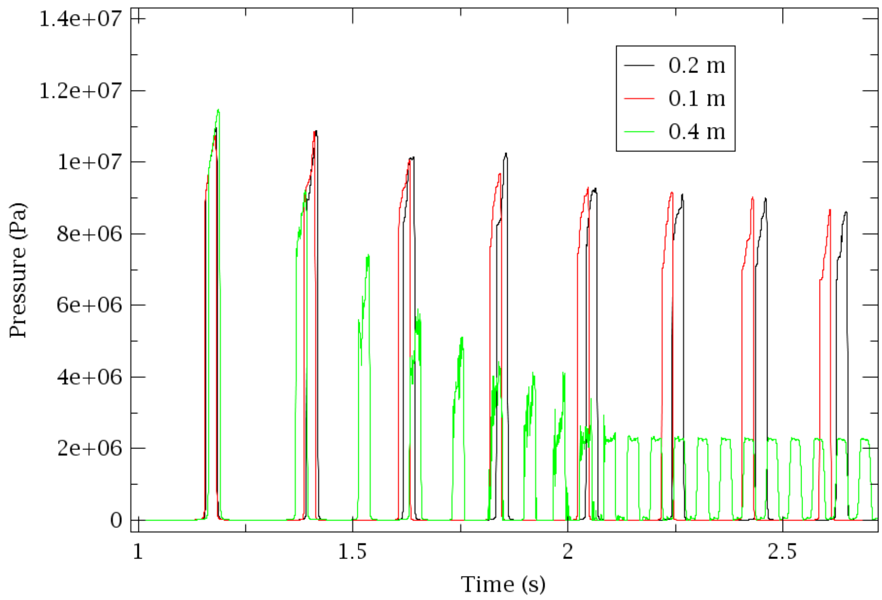
5.2 Results and Discussion

First it should be mentioned that there are difficulties in reaching steady-state in the case of the vertical pipe. Since the thermodynamic properties around the liquid level are very close to saturation conditions, very long simulation times (in the order of 10^5 s) are required in order to reduce the small oscillations in void fraction in the uppermost cells to negligible values. TRACE in particular required very long simulation times in order to completely remove small oscillations in some parameters. However, this steady-state simulation only needs to be run once for each code, since all steady-state parameters then can be saved and imported for all other simulations.

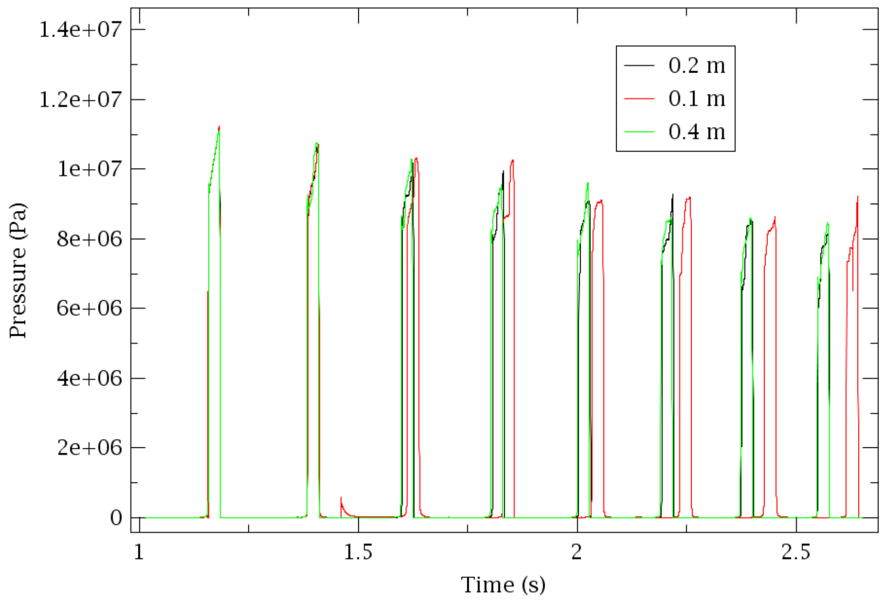
It can also be mentioned that the converted model performs basically the same as the TRACE model built from scratch in SNAP.

5.2.1 Sensitivity Analysis

Once steady-state is been reached, the sensitivity analysis with respect to the node length is performed. Nodes having lengths of half and double the initial node length were tested, respectively. As can be seen in Figs. 5.2(a) and 5.2(b), there is very little difference between a node length of 0.2 m and 0.1 m (except for a slight shift in frequency), whereas the simulation using a node length of 0.4 m differs rather significantly in the RELAP5 case. Thus, a node length of 0.2 m is used for all simulations, in order to reduce the computational time as compared to a node length of 0.1 m. For simplicity, the same node length is used for both TRACE and RELAP5, although Fig. 5.2(b) suggests that larger nodes could be used in the TRACE simulations, in order to reduce computational time further.



(a) RELAP5.



(b) TRACE.

Figure 5.2: Results of the sensitivity analysis with respect to node length for the vertical pipe. The difference is small between 0.1 m and 0.2 m for both codes. Thus, a node length of 0.2 m is chosen as standard.

5.2.2 Effects of Modifying the Interfacial Heat Transfer

From the results of the steam collapse in a horizontal pipe (described in Chapter 4) it is assumed that changing the equilibrium flag in RELAP5 and modifying the interfacial heat transfer coefficients in TRACE will affect the simulation results. Hence, such a study is performed and described in the following sections.

5.2.2.1 RELAP5

In Chapter 4 it is found that the equilibrium model should be used in RELAP5 when dealing with steam collapse, since the determination of the condensation rate in the non-equilibrium case seems to be too slow. However, this turns out to be a major problem in the vertical pipe. When the equilibrium flag is turned on it is not possible to run the simulations past the first steam collapse. The (possible) reasons for this is described below.

As the pressure is increased at the bottom of the pipe, it is expected that the pressure wave will propagate up through the pipe, leading to pressure increase in all cells (with a small delay due to the limited speed of wave propagation). However, the pressure in the final cell (just below the closed valve) starts to decrease, which is unphysical. Since the pressure is already very low in this cell, it soon drops below the limit which can be handled by the steam table, which results in the crashing of the simulation. Several attempts have been made to solve this problem, e.g. by changing the steam table or code version, or by slightly changing initial conditions. All such attempts have proven unsuccessful though.

Therefore, the results from the RELAP5 simulations are not expected to be very accurate compared to experimental data (which is not available in this case, as mentioned before).

5.2.2.2 TRACE

The results of Chapter 4 show that the interface-to-liquid heat transfer coefficient h_{if} should be modified by a large multiplicative factor in TRACE. It can be seen in the figures of Chapter 4 that there is very little difference between using a multiplicative factor of 50 and 10^6. Thus, it is assumed here that 50 can be considered large enough. Fig. 5.3 shows how different factors affect the results for a pressure increase time of 1 ms.

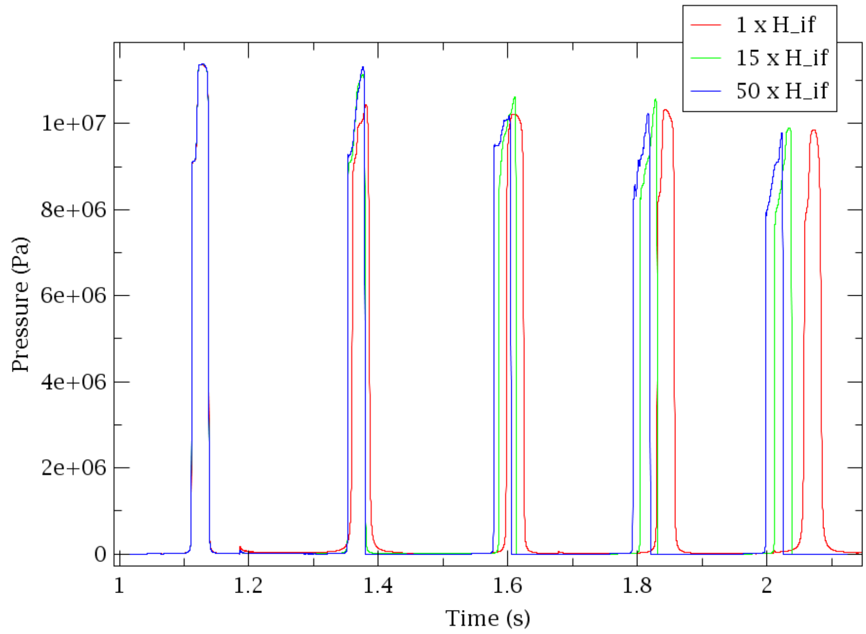


Figure 5.3: Results for different interface-to-liquid heat transfer coefficients in TRACE. The first two peaks are very similar for the factors 15 and 50. The unmodified case is not very different either, although a slightly larger time shift is observed.

It can be seen that the effects of changing the heat transfer coefficient are rather small. However, when the heat transfer is faster the pressure peaks become slightly higher, which means that this result is more conservative.

This conservatism is not always valid, though. Fig. 5.4, which is the same simulation as Fig. 5.3 but simulated for a longer time, shows that the dampening of the pressure peaks is slower in the case of no modification to the heat transfer coefficient. In general, however, only the first pressure peaks are of interest, since these are the ones with the highest amplitude. Thus, from the perspective of this project, it is of little importance that there is a difference in the peaks occurring later, as long as they remain smaller than the initial ones. Also, as discussed in e.g. [7], errors are introduced in the simulations which means that the results are questionable after the first few pressure peaks.

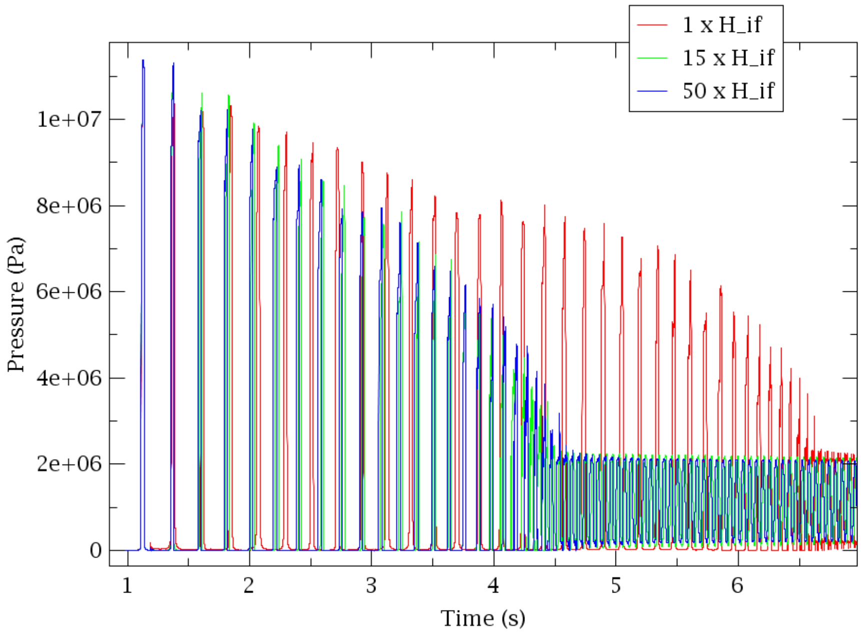
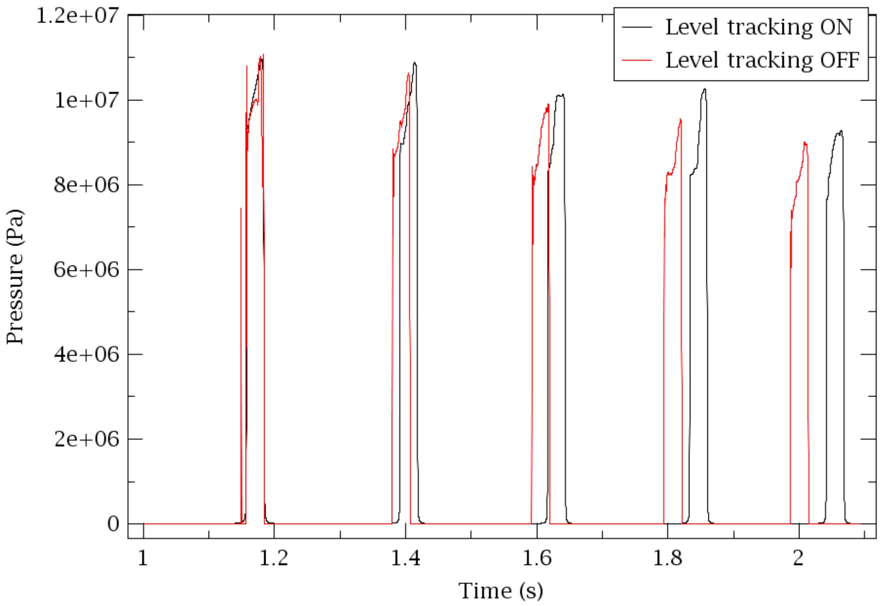


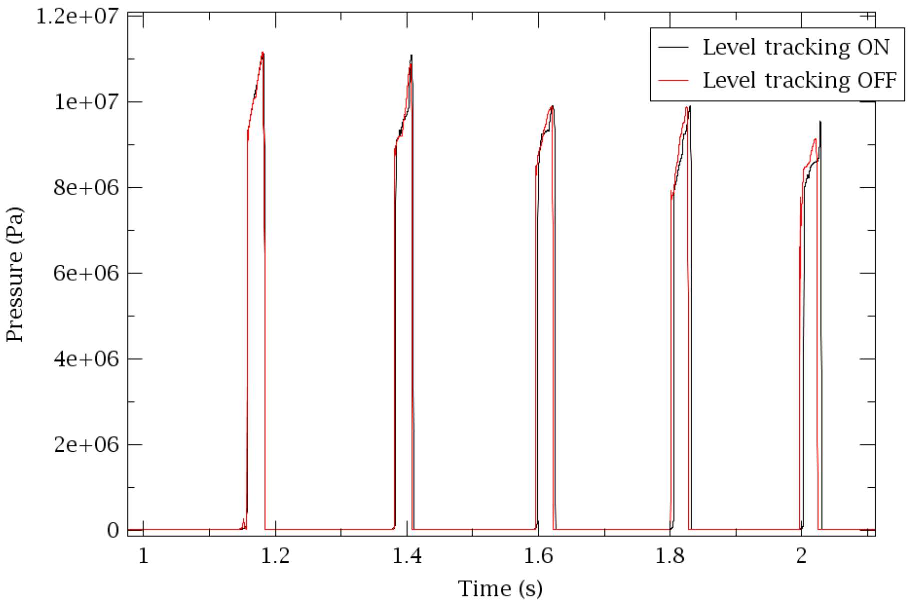
Figure 5.4: Results for different interface-to-liquid heat transfer coefficients in TRACE, simulated for a longer time. It is observed that the steam collapses occur for a longer time in the unmodified case (i.e. the peaks die out later).

5.2.3 Effects of Level Tracking

The second thing to be studied is the effects of using the level tracking method, which is described in Section 2.3.3. Figs. 5.5(a) and 5.5(b) show how the amplitude of the pressure peaks is affected by the use of the level tracking method for a pressure increase time of 100 ms. As can be seen, the use of level tracking results in slightly more conservative results, in particular for RELAP5. Also, some numerical irregularities are removed when the method is used.



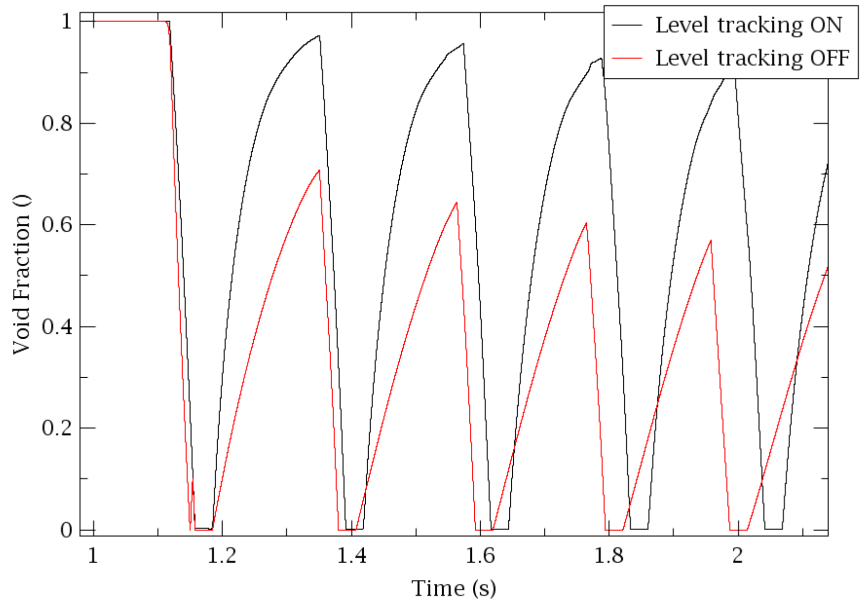
(a) RELAP5.



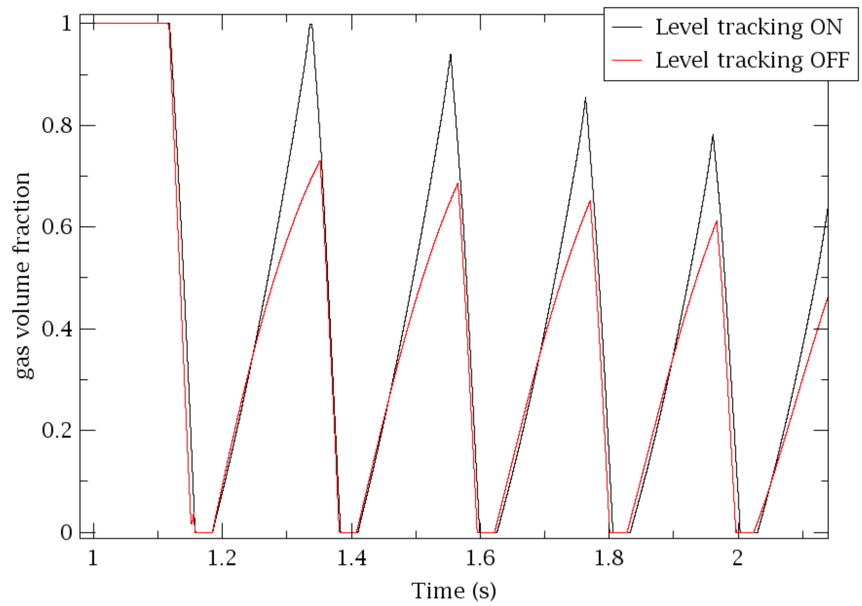
(b) TRACE.

Figure 5.5: Effects on the pressure peaks of using level tracking. A small difference is observed in both codes. The difference is slightly larger for RELAP5.

If void fraction is considered (rather than pressure) the differences between using the level tracking method and not using it is more obvious. This can be seen in Fig. 5.6, where the disabling of the level tracking method results in less amount of void. However, the pressure amplitude is more important when it comes to structural integrity due to water hammers, as has been mentioned before.



(a) RELAP5.



(b) TRACE.

Figure 5.6: Effects on the void fraction of using level tracking. More void is produced when level tracking is activated.

Although the level tracking method does have an impact on the results, it does not work as intended at all times. This is discussed further below.

5.2.3.1 RELAP5

When using the level tracking method together with default heat transfer in RELAP5 (i.e. non-equilibrium model), it is observed that vapourisation takes place in several cells at the same time. This can be seen in Fig. 5.7.

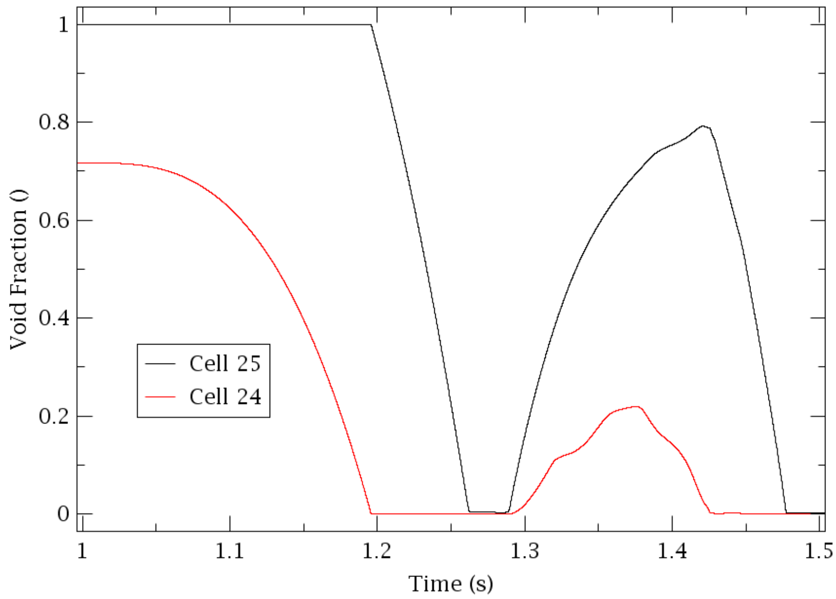


Figure 5.7: Void fraction in the two uppermost cells using level tracking in RELAP5. Cell 25 is above cell 24. As the void fraction increases again, it does so in both cells at the same time.

There is no suitable variable in RELAP5 which describes the position of the liquid level. It is possible (and recommended in the RELAP5 manual [20]) to use the variable `vollev` as a measure of the liquid level. This does not work very well in this case, though, as can be seen in Fig. 5.8, since the level is not tracked at all in the uppermost cell. Note that the variable in Fig. 5.8 is the `vollev`-variable, which can only determine the position of the liquid level within a cell. As can be seen, no useful data can be extracted from this variable, since the level only varies discretely between full and empty for the uppermost cell, and is just slightly more accurate in the cell below.

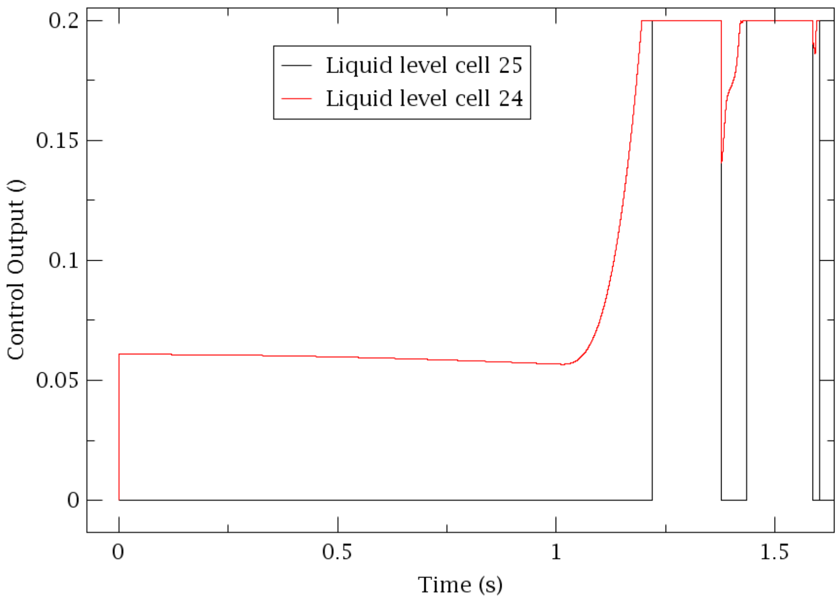


Figure 5.8: Liquid level with the `vollev`-variable using level tracking in RELAP5. The vertical axis refers to the height of the liquid level within the cell. Cell 25 is above cell 24. The discontinuities indicate that the variable is not suitable for this purpose.

5.2.3.2 TRACE

Fig. 5.9 shows the void fraction for the two uppermost cells in the pipe (i.e. cells 24 and 25, with cell 25 being the upper one) for the cases with default heat transfer coefficient and with a multiplicative factor of 50. In the former case the void fraction behaviour is as expected, i.e. the cells are drained/filled with void one by one (which is what will happen if there is a liquid level moving in the pipe). For the condensation part (i.e. when the void fraction decreases) the same result is obtained as the heat transfer coefficient is increased. There is however a large difference in how the vapour is produced once the pressure decreases again (just before $t = 1.3$ s). Instead of vapourisation first occurring in the uppermost cell and then in the second, both cells start to vapourise at the same time, which may be unphysical. As a matter of fact, vapourisation occurs in most of the cells in the pipe at the time, although at much slower rates in cells further from the upper end. Only in the 13 cells at the bottom of the pipe (corresponding to 2.6 m) there is no vapourisation at all.

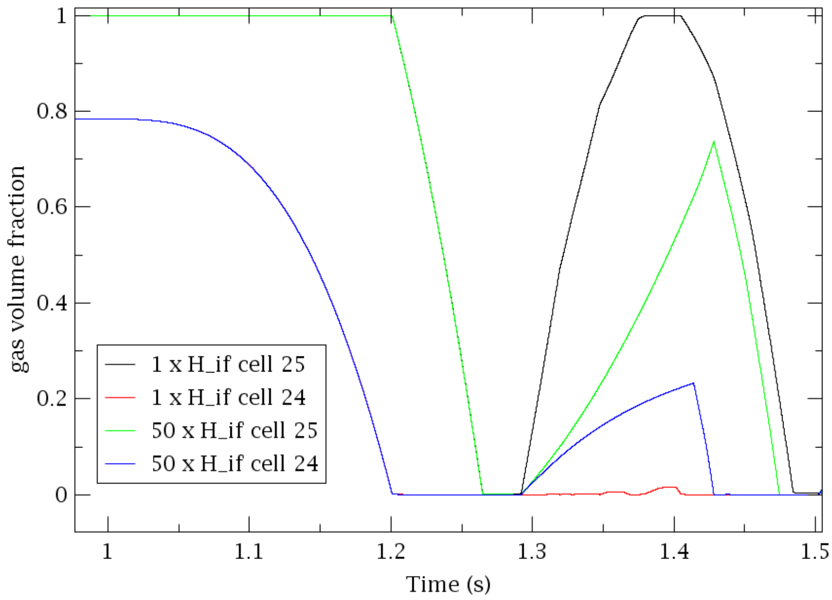


Figure 5.9: Void fraction in the two uppermost cells using level tracking in TRACE. Cell 25 is above cell 24. The same behaviour as in RELAP5 is observed (with void production in both cells at the same time.

Fig. 5.10 shows the liquid level in the upper quarter of the pipe (as calculated by TRACE). It should be pointed out that the liquid level never actually reaches 5.0 m (i.e. the top of the pipe) in any of the cases, even though the void fraction is zero. In the case where a modified heat transfer coefficient is used, the level does get closer to 5.0 m though, as can be seen in Fig. 5.11. Fig. 5.11 also shows that the level tracking method cannot handle the case where the liquid level reaches a closed valve (or a dead end). This in itself is not necessarily a problem, though. Fig. 5.10 shows that the liquid level drops to zero for a short time around $t = 1.2$ s in both cases. This happens when the level moves from cell 24 to cell 25. However, as can be seen in Fig. 5.10 at $t \approx 1.32$ s and onwards (i.e. as the level tracking method starts to work again), the behaviour of the liquid level in the case of modified heat transfer is very different of that in the case of standard heat transfer. It seems that TRACE is not able to accurately model the vapourisation when an increased heat transfer coefficient is used.

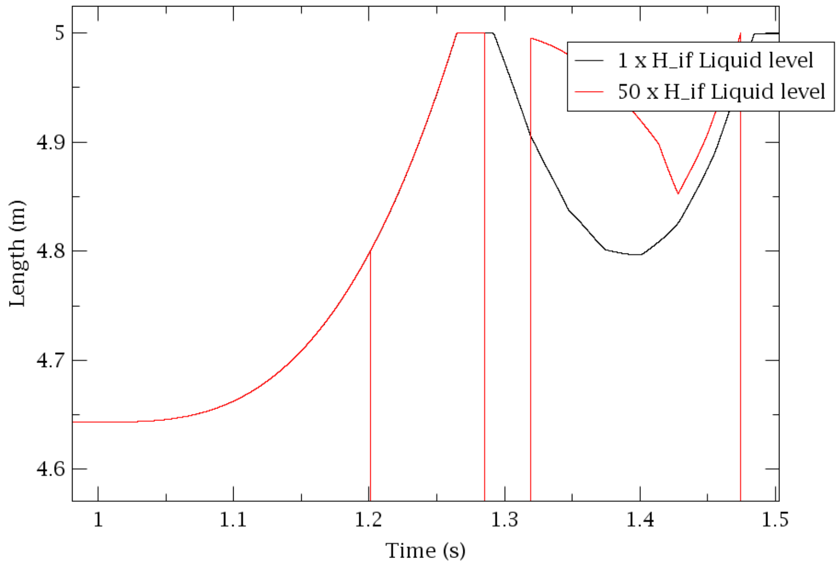


Figure 5.10: Liquid level in the upper quarter of the pipe using level tracking in TRACE. A strange behaviour is observed for the case with an increased heat transfer coefficient.

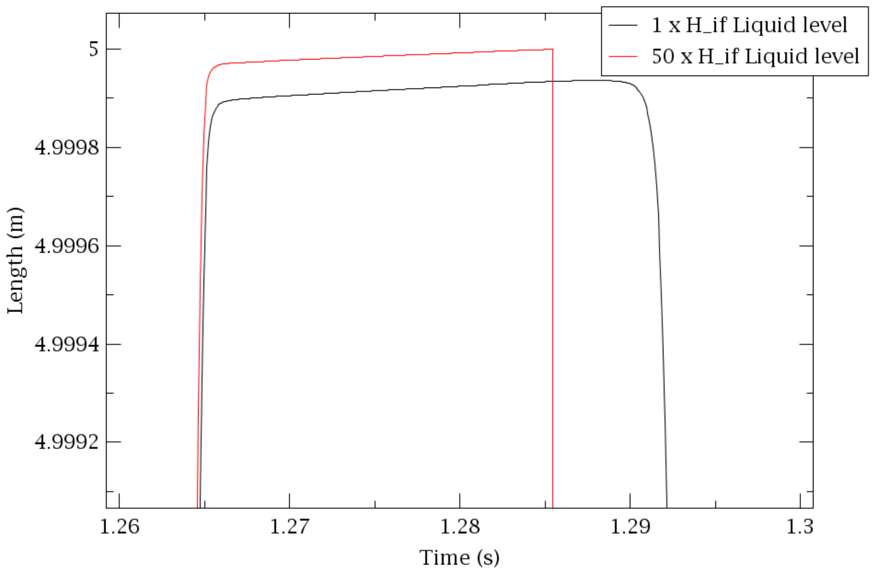


Figure 5.11: Zoom-in of the liquid level in the upper quarter of the pipe using level tracking in TRACE. The level never reaches the top of the pipe before the level tracking scheme crashes.

It should be noted that the level tracking method leads to strange behaviour of the liquid level and void fraction also in the case of standard heat transfer. However, this happens only after several seconds, as can be seen in Fig. 5.12.

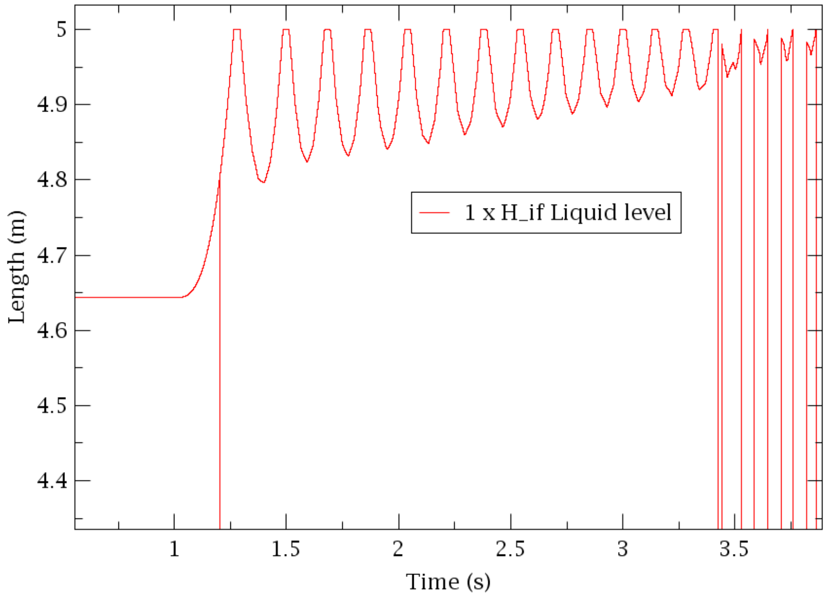


Figure 5.12: Liquid level with standard heat transfer coefficient using level tracking in TRACE. The strange behaviour is observed at a later time than for the case with increased heat transfer.

Because of the behaviour described above, the level tracking method in both TRACE and RELAP5 should be used with caution for this kind of fast transients.

5.2.4 Different Pressure Increase Times

As has been noted, the results in the preceding sections have been obtained using different pressure increase times. Several different cases are investigated; 1 ms, 100 ms, 500 ms and 2000 ms. The behaviour of the first few pressure peaks is very similar for all the cases. However, when looking at slightly longer timescales some differences appear, in particular for the 100 ms case. From Fig. 5.4 above, which shows the behaviour of the pressure peaks for the 1 ms case, it is observed that when increasing the interfacial heat transfer coefficient, the dampening of the pressure peaks is faster (although the dampening is rather constant for large heat transfer coefficients). The same observation can be made in the 500 ms and 2000 ms case, which are shown in Figs. 5.13 and 5.14, respectively.

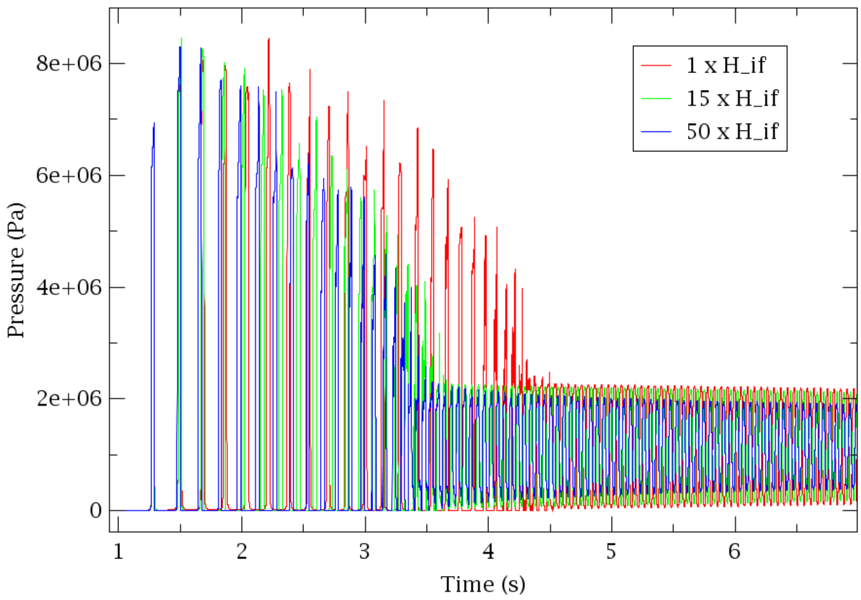


Figure 5.13: Results for different interface-to-liquid heat transfer coefficients in TRACE, simulated for a longer time. The pressure increase time is 500 ms.

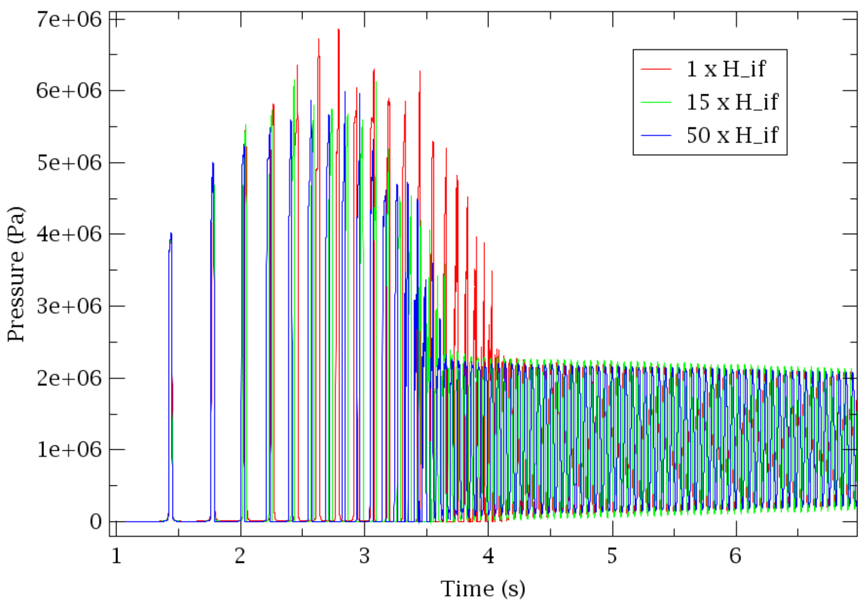


Figure 5.14: Results for different interface-to-liquid heat transfer coefficients in TRACE, simulated for a longer time. The pressure increase time is 2000 ms.

However, in the 100 ms case shown in Fig. 5.15, this behaviour cannot be seen. Instead, the dampening of the pressure peaks seems to be independent of the heat transfer coefficient in this case. The reason for this remains unclear, but it could be due to some unfortunate combination of variable values in this particular case. However, it was decided not to focus on this too much, because of the great uncertainties present in the simulation results after such (relatively speaking) long simulation times (as observed by [7]).

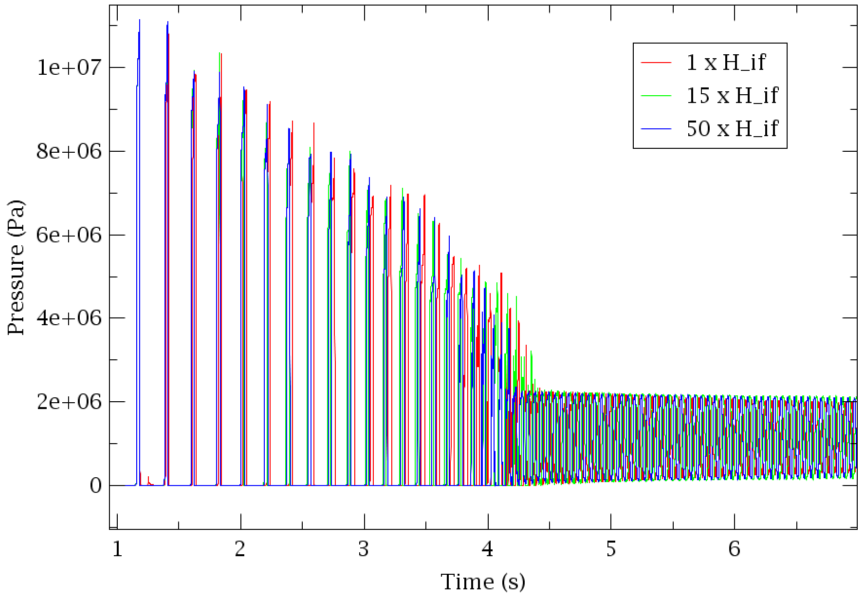


Figure 5.15: Results for different liquid to interface heat transfer coefficients in TRACE, simulated for a longer time. The pressure increase time is 100 ms. Notice that the case with default heat transfer is similar to those with increased heat transfer, which has not been the case when other pressure increase times have been used. The reason for this remains unclear.

Chapter 6

Closing of an Inertial Swing Check Valve

The purpose of this chapter is to convert the RELAP5 model of an inertial swing check valve presented in Section 2.4 to a TRACE model and compare the results, especially in terms of the closure characteristics and possible chattering behaviour. Also, the possibility of installing the valve vertically is studied briefly.

The geometry of the valve and of the surrounding system is chosen to be the same as in [33] for the sake of verification that no errors are introduced during the import of the RELAP5 model.

6.1 Closure Characteristics

The effects of closing the inertial swing check valve is investigated in this section, where the flow is established, and then reversed, by the usage of a `TIME-DEPENDENT JUNCTION`.

6.1.1 Method

First, the RELAP5 input file of the valve model is imported to SNAP and put into place in a setup shown in Fig. 6.1 (the same setup used for validation against the DRAKO model in [33]). This arrangement consists of two pipes of length 1 m and 9.999 m, respectively, between which the valve is located. Both the two pipes and the fully open valve has a flow area of $2.16 \cdot 10^{-3}$ m^2 ($\varnothing \approx 2$”). The node length is 0.1 m for the shorter pipe and 0.101 m for the longer one[1]. The wall roughness is set to zero and the 'full abrupt area' model is used for the valve. A `TIME-DEPENDENT VOLUME` is connected to each end. The volume of these are set arbitrarily to 1 m^3.

[1]The reason for not just using a 10 m pipe, with a node size of 0.1 m, is the upper limit (99) for the number of cells in a RELAP5 pipe component.

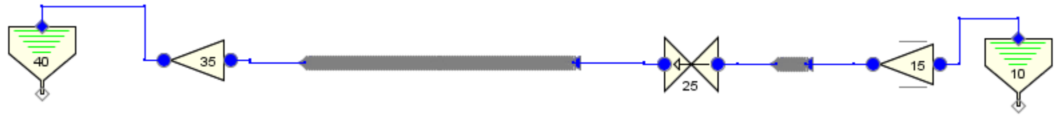


Figure 6.1: Model setup (in RELAP5) for studying the inertial swing check valve. A `TIME-DEPENDENT JUNCTION` (component number 15) is used in both codes for controlling the flow through the valve.

A pressure of 10 bar and a temperature of 293 K is set both upstream and downstream the valve. A constant flow velocity of 3 m/s is established in the pipes using a `TIME-DEPENDENT JUNCTION` located upstream according to Fig. 6.1. At steady-state conditions this results in the inertial swing check valve being partly open and having a throughput velocity of approximately 3.5 m/s (or a mass flow of 6.5 kg/s). At time $t = 10$ s (when steady-state has been established) the flow velocity is reduced at a rate of 20 m/s^2, leading to closure of the valve. The time and characteristics of the closure are then studied. A set of control variables are implemented in order to calculate the loads in the downstream pipe according to Eq. (2.3).

The imported model is converted to TRACE using the SNAP built-in conversion tool. Possible adjustments and modifications are made in order to make the model work properly (these are described in Section 6.1.2 below). Finally the behaviour of the inertial swing check valve in the TRACE model is compared to that of the valve in the RELAP5 model.

6.1.2 Results and Discussion

Before the converted model can be run in TRACE, some discovered bugs generated by the SNAP conversion tool have to be fixed. Most severe is the fact that the numerator and denominator in the division operator are exchanged. This has to be corrected manually in all division control blocks. It can easily be verified, by just converting a single control block separately, that this bug seems to be consistent in most, but not all, cases. A few division operators are actually converted correctly during the conversion of the RELAP5 model with three identical valves connected in series (see Section 6.3.2 below). However, one should keep in mind to check all division operators after the conversion.

Another bug-related feature is that all traces of the valve characteristic table disappear in the conversion and it has to be re-added in the TRACE `VALVE` component. Furthermore, the cell in the upstream pipe, in which the pressure before the valve is measured, is mysteriously changed from the last to the next last cell of the pipe. This also needs to be corrected in order to obtain the right pressure loss over the valve.

Much effort has also been put into finding the reason why TRACE (even after the bugs mentioned above have been fixed) exhibits a noticeably faster valve closure rate compared to that of the RELAP5 model[2]. Since a faster closure rate for a check valve results in less backflow, which in turn leads to less severe loads on the system, the

[2]In order to exclude further conversion bugs, the complete model of the inertial swing check valve has also been built up from scratch in TRACE, but the problem still remained.

TRACE results may be non-conservative and underestimate the impact of the valve closure. However, it is finally found that the way the absolute flow velocity through the valve is calculated (or measured) makes a significant difference in how fast the valve is closing. In the original RELAP5 model, this velocity is defined as the absolute velocity *into* the valve (i.e. not *through* the valve opening itself) and is calculated from the mass flow rate and the constant valve seat area according to Eq. (2.59) in Section 2.4. By instead measuring (using a signal variable) the flow velocity through the actual valve, the valve closure rate in TRACE becomes much closer to the one in RELAP5. Oddly, the overall effects of making the same adjustment to the RELAP5 model turn out to be very minor. One may speculate that is due to the fact that different velocities (see Section 2.3.1.2) are used in the equations for the abrupt area form losses. Further investigations are recommended though in order to find the underlying reason why it makes such a difference in TRACE and not in RELAP5.

In the rest of this section, the results from three different model versions are presented in Figs. 6.2 - 6.6. The benchmark version to be compared with is the original RELAP5 model. In the unmodified TRACE version, the flow velocity is unchanged (i.e. it is calculated according to Eq. (2.59)), whereas in the modified TRACE version the velocity is adjusted as described above. Also, the friction option '[0] Constant FRIC' is used, together with an additional form loss ($K = 0.009$), for the modified case in order to obtain the same steady-state mass flow and flapper disc position. The (after conversion) default friction option '[-1] Flow Factor + FRIC + Abrupt' is used in the unmodified version, which in this case gives satisfactory steady-state results. It should be mentioned, though, that the implementation of a form loss table based on Eq. (2.24) is also tried out for the two different TRACE versions. This results in a small deviation from the steady-state flapper position of the RELAP5 model but also slightly slower closure rates compared to the simulations with the respective friction option specified above. Thus, these results are somewhat more conservative and may motivate the usage of such a form loss table in future use of the inertial swing check valve model in TRACE.

Fig. 6.2 shows the flapper angle of the inertial swing check valve after the flow reversal initiated at 10 s. The RELAP5 model has a total closure time of approximately 180 ms, while the unmodified TRACE version is fully closed already after about 160 ms. The modified TRACE version follows the original RELAP5 case nicely.

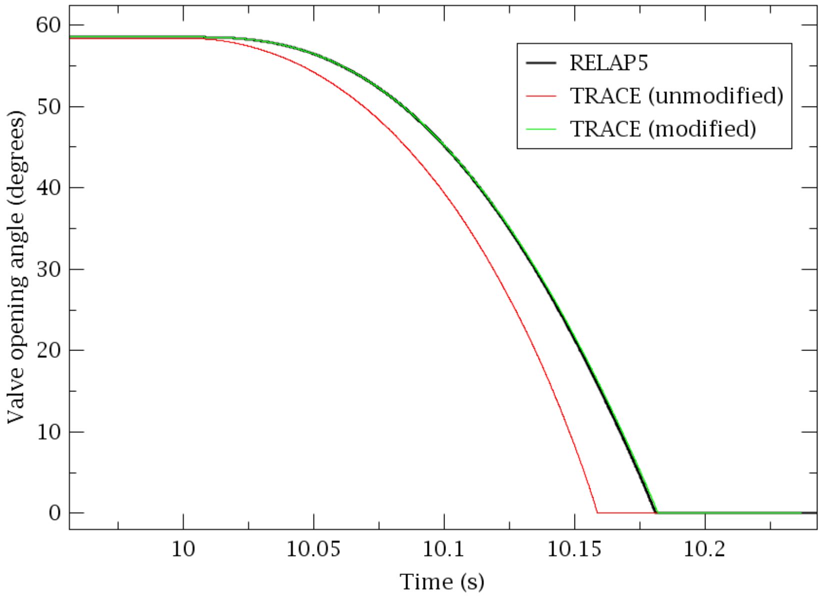


Figure 6.2: Valve opening angle. The flow starts to decelerate at 10 s. The valve closes faster in the unmodified TRACE case.

The pressure just downstream the valve is illustrated in Fig. 6.3. It is easily observed that the unmodified TRACE version results in much smaller pressure peaks compared to the RELAP5 and modified TRACE versions, which are very similar. This is explained by the smaller backflow in the unmodified case (see Fig. 6.4) at the precise moment when the valve is about to close.

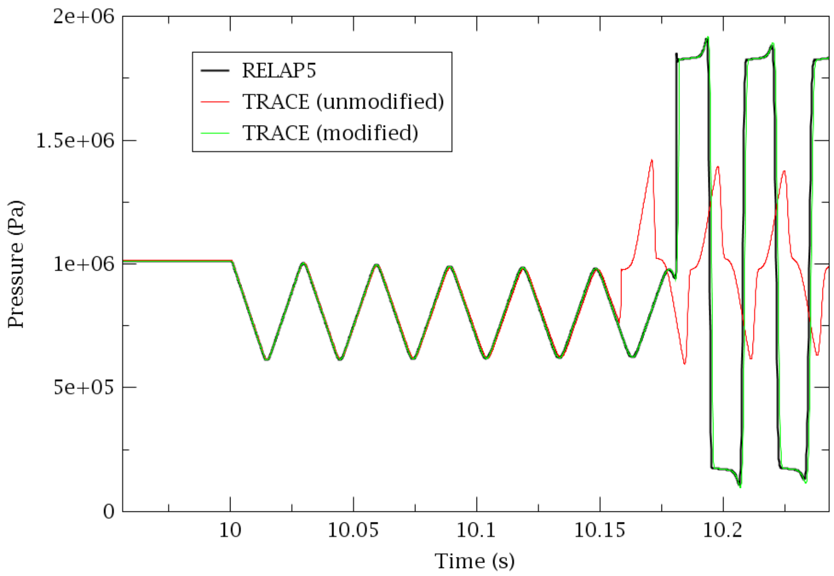


Figure 6.3: Pressure just downstream the valve. The pressure peaks are much lower in the unmodified case.

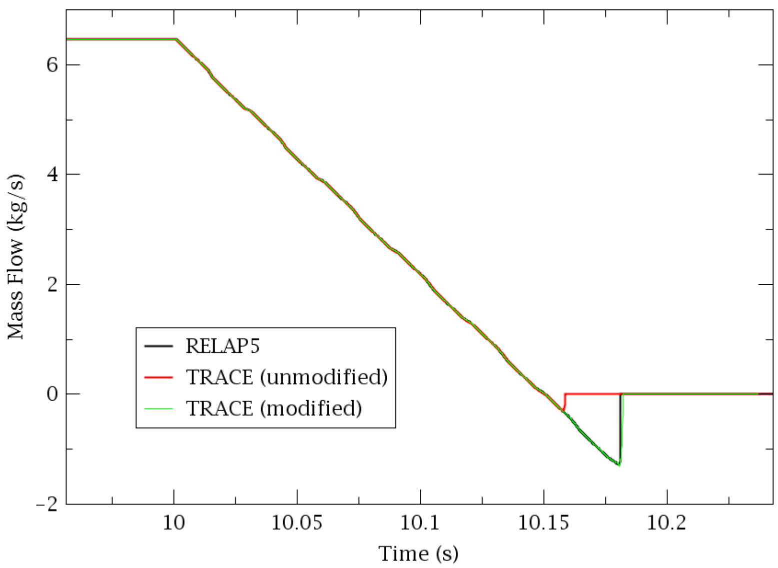


Figure 6.4: Mass flow rate through the valve. In the unmodified case the valve closes faster, which means that there is less time for a large backflow to develop. This leads to the lower pressure peaks seen in Fig. 6.3.

The only investigated variable that differs considerably for the modified TRACE version compared to RELAP5 is the angular velocity of the flapper disc during the last few milliseconds before fully closed, as can be seen in Fig. 6.5. Unsurprisingly, the unmodified version differs even more compared to both of these. While RELAP5 has a maximum absolute angular velocity of about 20 rad/s, the modified TRACE version peaks at 15 rad/s. This can most likely be explained by the different ways the velocities are calculated (through or into the valve) when the valve is about to close and the valve area goes to zero. However, this difference seems to influence neither the system pressure nor the calculated loads and may thus be ignored. The modified TRACE version is closer to the original DRAKO-model on which [33] is based, since the undershoot just before closure is smaller.

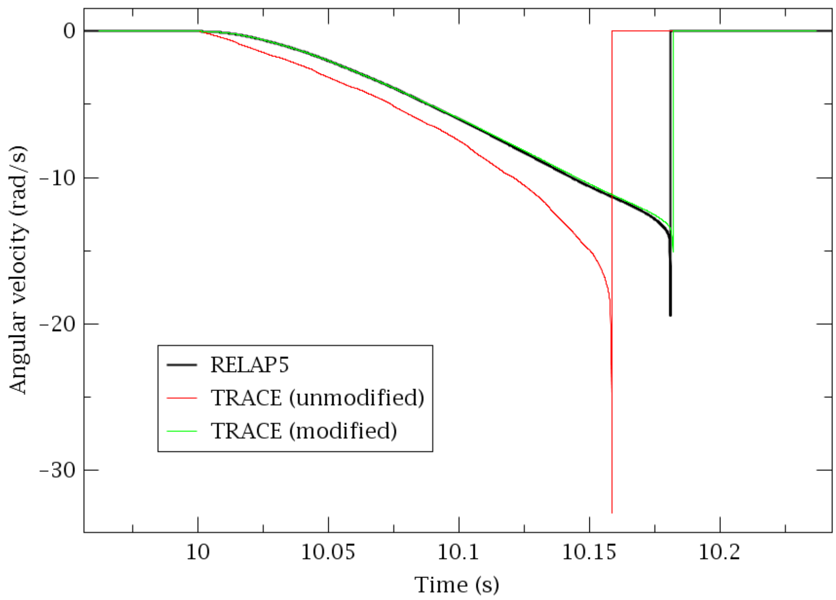


Figure 6.5: Flapper angular velocity. The modified TRACE case is closer to the original DRAKO-model on which [33] is based, since the undershoot just before closure is smaller.

Fig. 6.6 shows the loads on the downstream pipe, calculated according to Eq. (2.3), due to the decelerated water column. Even in this case, the RELAP5 and the modified TRACE versions are very similar. However, a small inexplicable difference can be observed in the middle of each peak where the two codes seem to calculate a minor additional spike, but in opposite directions. It is uncertain if either of these has any physical explanation or if they are completely numerical.

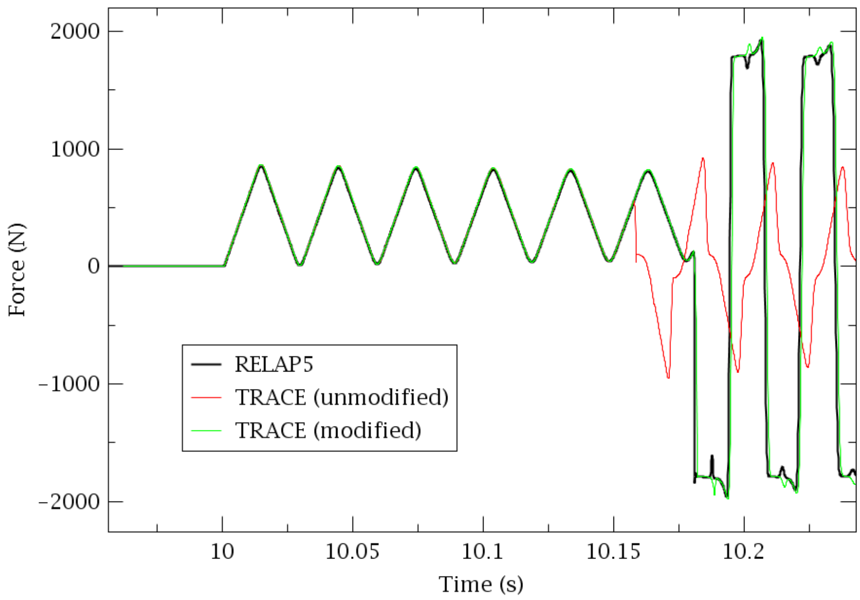


Figure 6.6: Calculated loads on the downstream pipe. Again, a large difference is observed for the unmodified TRACE version. Also, a small difference can be noticed between the modified TRACE version and RELAP5 in the middle of each peak.

To conclude, the existing RELAP5 model of an inertial swing check valve can be successfully converted and implemented in TRACE. However, some important modifications have to be applied in order to make it work properly. For instance, a number of bugs have been discovered in SNAP, but they are fairly easy to fix once they are known. But, even though the results using the modified TRACE version are in most cases fully comparable with the RELAP5 model, the modification of the flow velocity may need to be further investigated.

6.2 Chattering of a Single Valve

It is known from earlier experience (see e.g. [38]) with the RELAP5 model that it may exhibit a chattering behavior in some situations. This is a common occurrence that is likely to appear especially for slow depressurisation rates [21], which can cause the pressure difference across the valve to fluctuate and in turn causing the valve control trips to be switched on and off alternately. Therefore, it is of interest to investigate whether this behaviour is present also in the converted model in TRACE. In order to induce this phenomenon, the flow boundary condition used in the previous section is replaced with suitable pressure boundary conditions for establishing a steady-state flow. A small additional test is also carried out where the valve is installed in a vertical position.

6.2.1 Method

The experimental setup is the same as shown in Fig. 6.1, except that the `TIME-DEPENDENT JUNCTION` is replaced by a `SINGLE-JUNCTION`. Two different cases (A and B) are considered, representing a small and a large pressure drop, respectively (see Table 6.1). In Case A, the downstream pressure is set to 10 bar and the upstream pressure initially to 10.1 bar, giving a steady-state mass flow of approximately 5 kg/s through the system. The upstream pressure is then reduced to 9.9 bar during 0.2 s. The temperature is constant at 293 K both upstream and downstream. In Case B, the downstream pressure is set to 10 bar at a temperature of 333 K. The corresponding upstream conditions are initially set to 11 bar and 333 K, resulting in an almost fully open valve and a steady-state mass flow of approximately 19 kg/s. The pressure and temperature are then reduced during 0.2 s to 1 bar and 293 K, respectively. Possible chattering following the valve closure is then studied for the two cases. A sensitivity analysis is also performed in which the influence of a larger maximum time step size (10^{-4} s instead of 10^{-5} s) is investigated. The same simulations are then performed using a converted TRACE model.

Table 6.1: Parameters used for the model setup of the inertial swing check valve in order induce a chattering behaviour. The flow through the system is established by using suitable pressure boundary conditions, i.e. no flow boundary condition is utilised. Two different cases (A and B) are considered. The arrows for the upstream pressures and temperatures indicate a change from the left value to the right in a linear manner during 0.2 s.

Parameter	**Case A**	**Case B**
Upstream pressure (bar)	10.1 → 9.9	11 → 1
Upstream temperature (K)	293 → 293	333 → 293
Downstream pressure (bar)	10	10
Downstream temperature (K)	293	333

A final experiment is carried out by, more or less, turning the setup shown in Fig. 6.1 by 90°. A steady-state upward flow is established by keeping a high pressure (11.5 bar) in the bottom of the pipe and a lower pressure (10 bar) in the top of the pipe. The pressure at the bottom is then reduced to 9.9 bar, causing (together with gravity) the flow to reverse and the valve to close. The model equations are slightly modified for taking the vertical geometry into account. First, the angle θ_s is increased by 90° in Eq. (2.53). Second, the difference in altitude in Eq. (2.56) is set to $\Delta h = 0.1$ m. This is done in both RELAP5 and TRACE. The results are studied and compared briefly.

6.2.2 Results and Discussion

In order to obtain the same steady-state valve opening angle as in RELAP5, the form loss option '[0] Constant FRIC' is used together with an additional K-factor. In Case A, the additional K-factor is 0.02 for both the unmodified and modified TRACE versions. In Case B, the corresponding K-factor is 0.06 for the unmodified TRACE version and 0.0 for the modified ditto. It should also be mentioned that smaller time steps (e.g. 10^{-3} s) are apparently necessary for the modified TRACE version to converge to a steady-state.

Figs. 6.7 and 6.10 clearly illustrate that the model of the inertial swing check valve is sensitive to chattering when the flow is controlled using only pressure boundary conditions.

In the case of the small pressure drop (Case A) the RELAP5 model exhibits some pronounced chattering that do not seem to die out (see Fig. 6.7). However, none of the TRACE versions show any significant chattering. The chattering in RELAP5 also results in greater forces on the pipe compared to TRACE, as seen in Fig. 6.8. The time between two of the 'opening peaks' in RELAP5 is usually (but not always) 0.0133 s (see Fig. 6.9), corresponding to a sonic pressure wave travelling the distance 1440 m/s $\times$ 0.0133 s $=$ 19.152 m $\approx$ 20 m, i.e. up and down through the 10 m pipe.

The sensitivity analysis shows that the chattering tendency in RELAP5 disappears completely when a maximum time step size of 10^{-4} s is used instead of the default 10^{-5} s, most likely since the larger time step cannot capture this fast behaviour. However, it may instead be an indication of that the chattering in this low pressure case (Case A) is numerical. If so, the TRACE results may then be more realistic. However, it can also mean that e.g. the numerical damping in TRACE requires an even shorter time step to be used in order to exhibit the same amount of chattering. Therefore, both the unmodified and modified TRACE models have been tested using 10^{-6} s as the maximum time step size, but no (more) chattering appear. Also, a simulation has been performed using the semi-implicit numerical scheme, since this should (theoretically according to Section 2.2.3) have less numerical diffusion compared to the SETS method. No difference is seen though. To summarise, further investigation is required in order to determine whether the chattering is numerical or physical, and why TRACE is less sensitive to this behaviour.

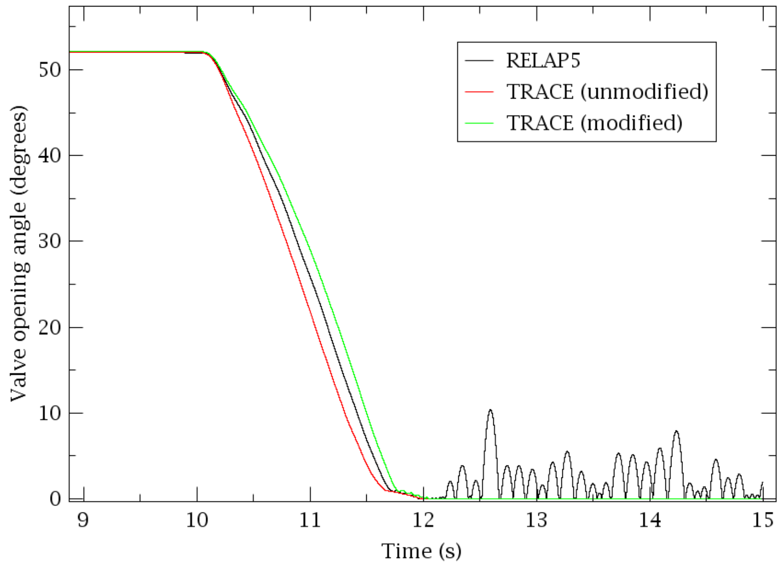


Figure 6.7: Case A: Chattering behaviour for the inertial swing check valve model when a small pressure drop is applied (10.1 bar → 9.9 bar). The TRACE simulations show that the valve has less tendency to chatter in this code. This may indicate that the chattering in RELAP5 is just a numerical phenomenon.

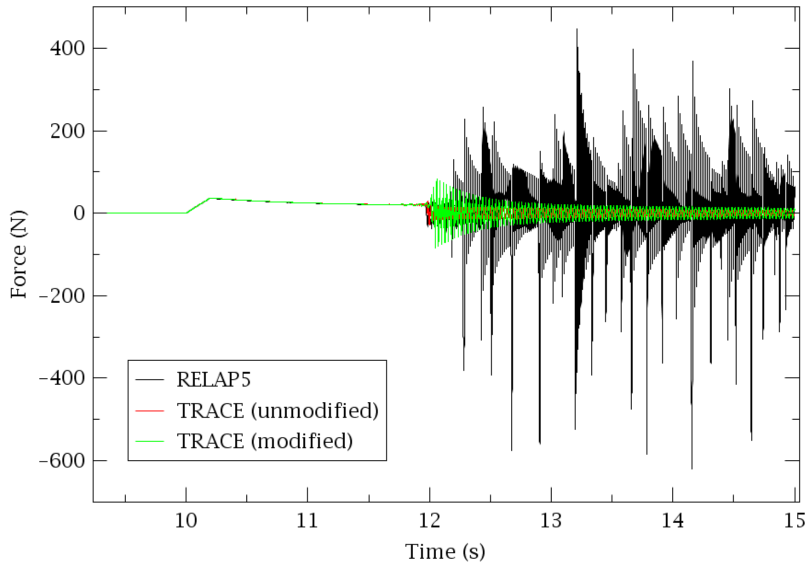


Figure 6.8: Case A: Calculated loads due to chattering. The forces arise when the valve closes, which happens several times for RELAP5.

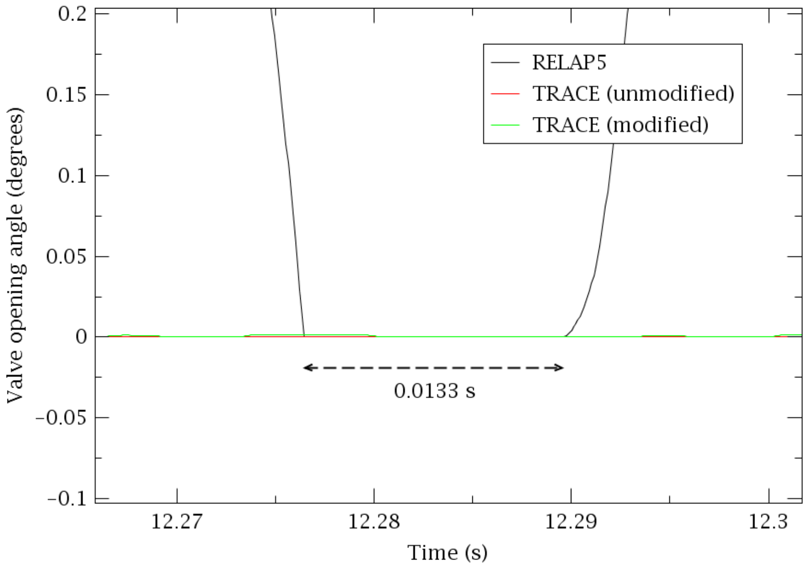


Figure 6.9: Case A: Time between two of the peaks shown in Fig. 6.7. This time shows for how long the valve stays fully closed and corresponds to the time it takes for a pressure wave to travel up and down the longer 10 m pipe.

In the case of the large pressure drop (Case B), the chattering behaviour is striking where the valve repeatedly changes between being completely closed and partly or fully open, at least in RELAP5 and the modified TRACE version (see Fig. 6.10). The explanation for this strong chattering is the frequent steam collapse in the short pipe upstream the valve, initiated by the Joukowsky water hammer following the valve closure. Each time such steam collapse occurs, the pressure upstream the valve increases rapidly, causing the valve to open. Fig. 6.11 shows that the valve opening in RELAP5 coincides beautifully with the pressure spikes in the first upstream cell. The corresponding graph for the modified TRACE version is shown in Fig. 6.12. The correlation is not as evident in this case, though. Furthermore, the pressure spikes are smaller in TRACE, which likely is due to the fact that the very last vapour pocket seldom collapses in TRACE. Thus, the following pressure spikes are less severe. This is consistent with the conclusion from Chapter 4 that the condensation rate of the last remaining steam is too slow, both compared to RELAP5 and to experimental results. Finally, the reason why the unmodified TRACE version soon dies out is probably the slightly faster closure rate resulting in less backflow, and thus less severe water hammers.

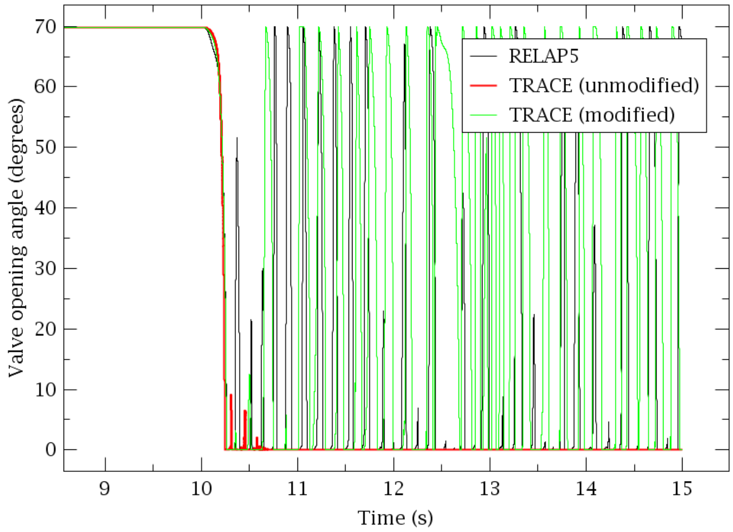


Figure 6.10: Case B: Chattering behaviour for the inertial swing check valve model when a large pressure drop is applied (11 bar → 1 bar). It is noticed that also the valve in the modified TRACE version chatters a lot in this case. This may indicate that the chattering (in this case) is physical and not only a numerical phenomenon.

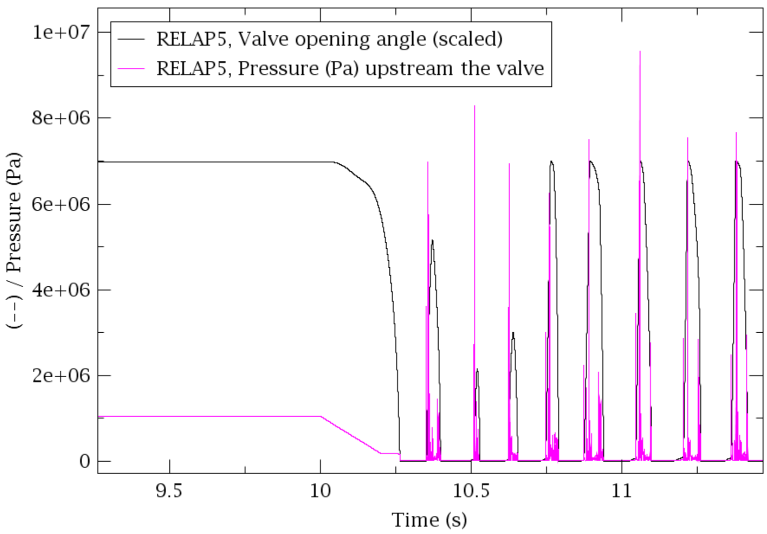


Figure 6.11: Case B: The temporal correlation between the valve opening in RELAP5 and the pressure spikes caused by steam collapses upstream the valve. For clarity, the valve opening angle is scaled to the same order of magnitude as the pressure. It can be observed that the valve chattering is triggered by the rapid increase in upstream pressure, causing the valve to open.

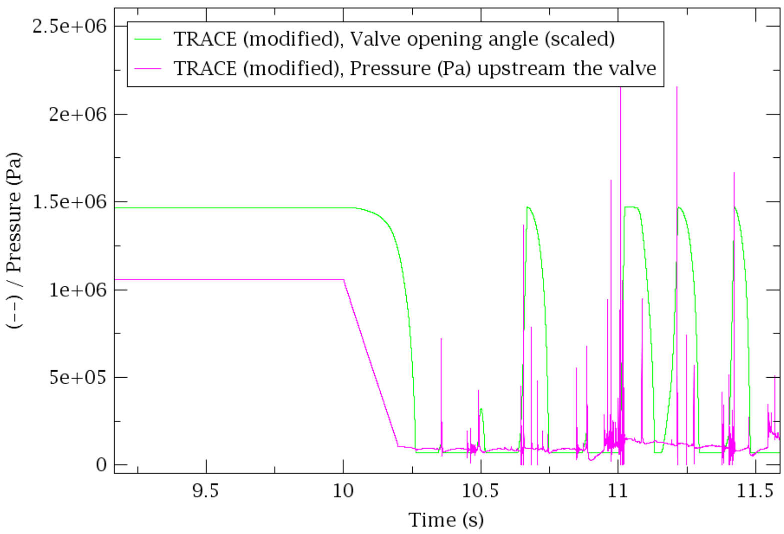


Figure 6.12: Case B: The temporal correlation between the valve opening in the modified TRACE version and the pressure spikes caused by steam collapses upstream the valve. For clarity, the valve opening angle is scaled to the same order of magnitude as the pressure. It can be observed that the valve chattering is (often, but not always) triggered by the rapid increase in upstream pressure, causing the valve to open. The pressure amplitudes are smaller than in RELAP5 (see Fig. 6.11), probably due to a slower condensation rate.

The experiment with a vertically installed valve turns out fine and the results seem reasonable. A few different pressure boundary conditions are tested until a moderate steady-state mass flow is established. Since no flow boundary condition is used, some degree of chattering is visible for all three model versions. However, like in the previous results, the unmodified TRACE version is less sensitive to chattering due to a smaller backflow. Overall, the valve model seems to work also for a vertical geometry, which the built-in models in RELAP5 and TRACE are not designed for.

6.3 Chattering of Multiple Valves

A separate test is also performed where three identical inertial swing check valves are connected in series[3]. The purpose is to compare the results using the two different codes, but also to investigate the convenience in RELAP5 and TRACE of implementing (i.e. duplicating) another valve component into the model when a large set of control variables are involved for each of these components. The motivation is that each valve must have a unique set of control blocks, trips, etc. and that possible restrictions in the

[3]The reason for choosing three valves, and not e.g. two or seven, is simply a balance between having a *more* complex experimental setup compared to a single valve (which is the purpose of this section) and having a *too* complex model (since every additional valve requires its own set of control variables). Thus, three seems like a nice number. Further, it is in reality very unlikely that more than a few check valves share the same flow path.

codes, regarding control system numbering, may be problematic during the duplication process.

6.3.1 Method

The setup of this test is shown in Fig. 6.13, in which the downstream pipe is split into three parts of approximately 8 m, 1 m and 1 m. The two smaller pipe segments are then connected between the valves according to the arrangement in the figure. Thus, the total length of the pipes is kept the same. The same boundary conditions as in Case A and B (described above) are applied. A general study of the behavior of the valves is performed, and the results are compared to the corresponding cases with just one single valve.

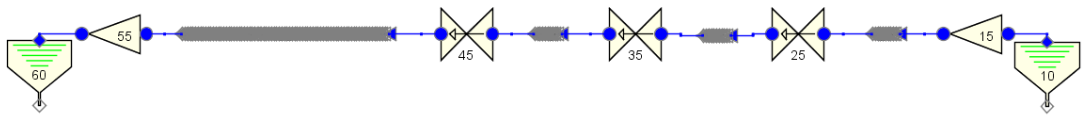


Figure 6.13: Model setup of three identical inertial swing check valves connected in series.

6.3.2 Results and Discussion

The results from the test of connecting three identical inertial swing check valves in series are similar to the case with just a single valve. Using the same pressure boundary conditions as in Case A above, all three valves close basically simultaneously and with only a minor degree of chattering. This is true for all models, both RELAP5 and the two TRACE versions. The additional K-factor added in TRACE (in order to obtain the same steady-state mass flow and opening angles as in the RELAP5 model) are 0.08 for the unmodified version and 0.01 for the modified version, respectively. A probable reason why chattering is not, in this case, apparent in the RELAP5 model is the slightly smaller mass flow through the system, which in turn is due to the form losses introduced by the two extra valves. In Case B, the closure of the three valves is no longer synchronised. Otherwise, the results are comparable to the ones with just a single valve, i.e. the RELAP5 and the modified TRACE models exhibit a rather chaotic chattering behaviour while the chattering for the unmodified TRACE version quickly dies out. The explanation for the latter is probably, once again, the smaller backflow resulting from the faster closure rate.

The purpose of this experiment with three identical valves is also to investigate the convenience of implementing another component, with a large set of control variables, into the model. Because the RELAP5 model of the inertial swing check valve uses the control variable card type `205CCCNN`, the available control component numbers are restricted to be in the range 001-999. Also, the standard trip data card is utilised, which means that variable trips must have component numbers in the range 401-599 and logical trips in the range 601-799 (see Section 2.2.7). This limitation makes things more difficult when all the control blocks and trips associated with the RELAP5 model are duplicated twice (each valve must have its own set of control variables) since the numbering has to be offset using a clever methodology, in order to simplify the handling of the control system. A lot of time has to be dedicated for e.g. renumbering the trips so they all have unique ID numbers in the permitted range. It can be mentioned that attempts have been made in order to change the control variable card type `205CCCNN` to `205CCCCN` and thus allowing a numbering range of 1-9999. Since this feature does not exist (or at least has not been found) in SNAP[4], this is performed by opening the RELAP5 input file in a text editor and manually change the data card type. However, when the model then is re-imported to SNAP, the connections between the control blocks are unfortunately lost.

In TRACE, the implementation is much more convenient and systematic since all control blocks, signal variables, trips, etc. can easily be offset by, let's say, 2000 without any issues. This means that all variables corresponding to the first valve can be grouped together on numbers `1XXX`, all variables corresponding to the second valve on numbers `2XXX`, etc. The only restrictions in TRACE are the limits described in Section 2.2.7.2, which are far less restricted in comparison with RELAP5. This fact may actually be a distinct practical advantage when modelling complex control systems.

[4]When building a RELAP5 model from scratch in SNAP, the (better) control variable card type `205CCCCN` is chosen as default.

Chapter 7

Ringhals 3 - 321WH/SV1

This chapter describes a water hammer simulation performed at Ringhals nuclear power plant. A RELAP5 model has previously been created at Ringhals for investigation of the effects of water hammers on the structural integrity of the residual heat removal system (RHRS) of Ringhals 3 (system 321), a case known as 30321WH/SV1[1] which is described in [37]. The model was used to study how the use of RHRS as pressure relief for the reactor coolant system using two safety valves could lead to forces on the piping system, and to estimate the magnitudes of these forces.

Fig. 7.1 shows a schematic representation of the studied system[2]. In this chapter two cases have been studied – opening and closing of the safety valve A, respectively. The two cases are described in the following sections starting with the opening of the valve.

This case is studied because it is a fairly simple model used in a water hammer simulation at Ringhals. The purpose is to investigate how TRACE performs in a 'real' case.

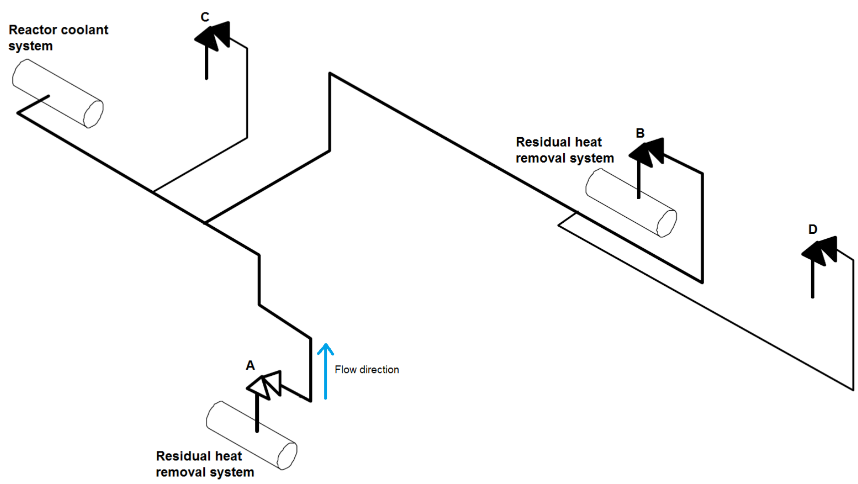


Figure 7.1: Schematic representation of the model in 30321WH/SV1. Note the location of valve A in the lower part of the figure.

[1]WH/SV = Water Hammer Safety Valve.

[2]For information about the actual plant-specific component names used at Ringhals the reader is referred to [37].

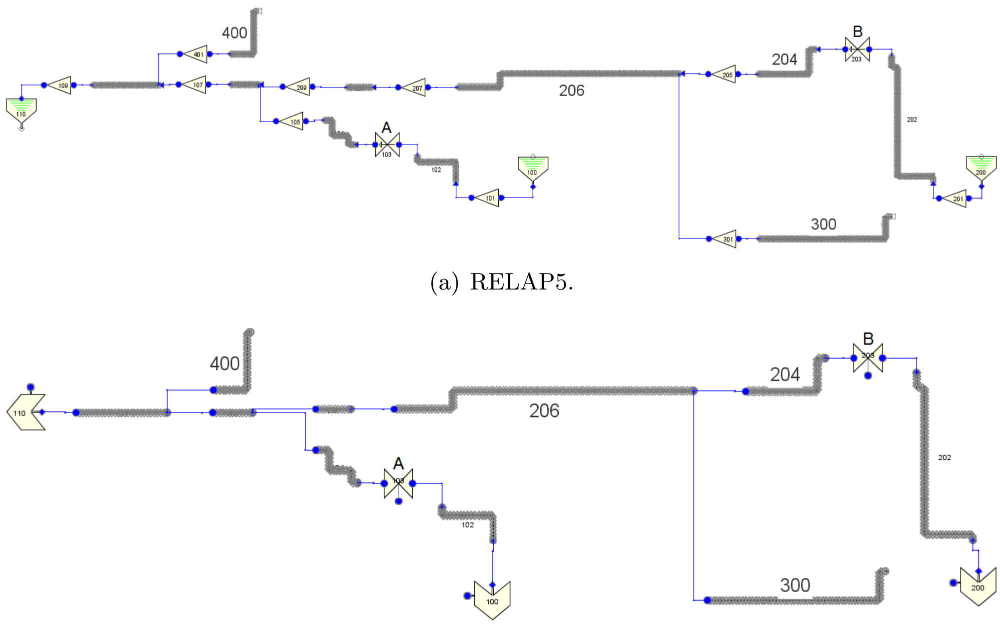
7.1 Case 1a – Opening of Safety Valve

The opening of the safety valve is described in the following section, starting with a description of the method used. Then, the results are presented together with a discussion.

7.1.1 Method

The methodology for the case of opening of the valve has been as follows. First, the RELAP5 model, which is available as an input (.i) file, is imported to SNAP. Second, the imported model is simulated and compared to the results presented in [37], in order to make sure that the import function does not introduce any errors. Third, the imported model is converted to TRACE. Fourth, the converted model is simulated and compared to the RELAP5 model. Fifth, modifications are made to the TRACE model in order to improve the results.

The RELAP5 and TRACE models, as represented in SNAP, are shown in Fig. 7.2. As can be seen, only A and B are modelled as valves, while the other two valves (C and D) are assumed to be closed during the transient and are therefore modelled as dead ends. Also the valve B is closed during this particular case. It should also be mentioned that some pipes actually have their direction into or out of the paper. However, the graphical interface of SNAP can only show two-dimensional representations of the models.



(a) RELAP5.

(b) TRACE.

Figure 7.2: Models used in 30321WH/SV1.

The initial conditions for the simulations are summarised in Table 7.1. All valves except A have been kept closed during the transient, since this is the configuration which is present when the system is used (for this purpose) in the real plant. From previous experience the length and volume of the BREAK components are set to 10^{-6} m and 10^6 m^3, respectively. At time $t = 5$ s the valve A is opened (in 0.111 s), and the consecutive transient is studied. Parameters which are studied are mainly pressure and void fraction downstream A, as well as mass flow through the valve.

Since no experimental data is available for this case, the TRACE results have only been compared to the results from the RELAP5 simulations.

Table 7.1: Data used in the simulations of Case 1a.

Fluid conditions upstream valve A	Liquid water (60°C, 35.1 bar)
Fluid conditions downstream valve A	Nitrogen gas (25°C, 1 bar)
Node length	Varying, around 0.2 m
Wall roughness (all pipes)	$4.5 \cdot 10^{-5}$ m

7.1.2 Results and Discussion

The pressure downstream valve A is shown in Figs. 7.3 and 7.4. The RELAP5 result is more or less identical to that given in [37], indicating that the import function works as intended. The TRACE result however, shows some unexpected behaviour, with large pressure spikes occuring at regular intervals. This can be seen even better in Fig. 7.5 which has the same scale as Fig. 7.3. A closer investigation showed that these spikes could be related to the filling of a computational cell with liquid water – i.e. the phenomenon known as water packing, which is described in Section 2.3.4.

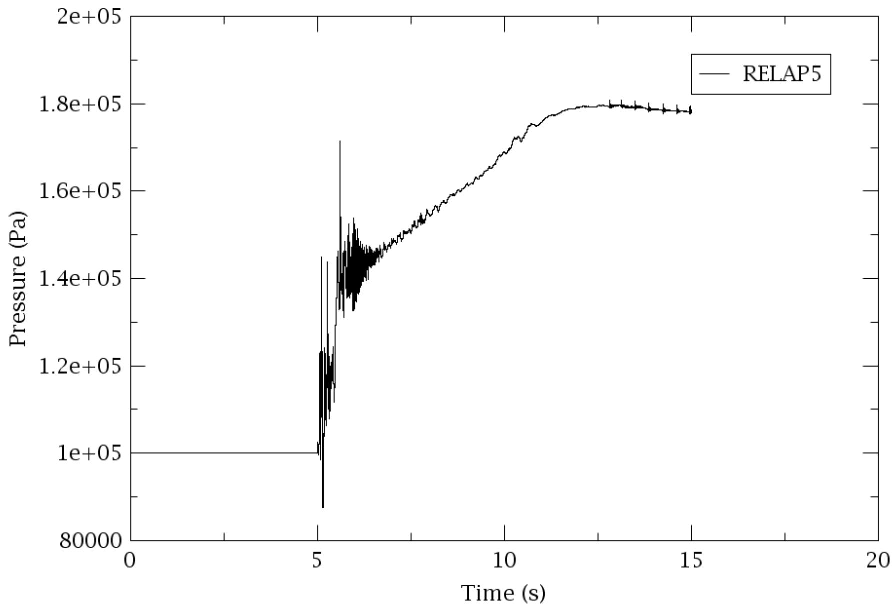


Figure 7.3: Pressure just downstream valve A in RELAP5. Some noise is observed in the first seconds after valve opening.

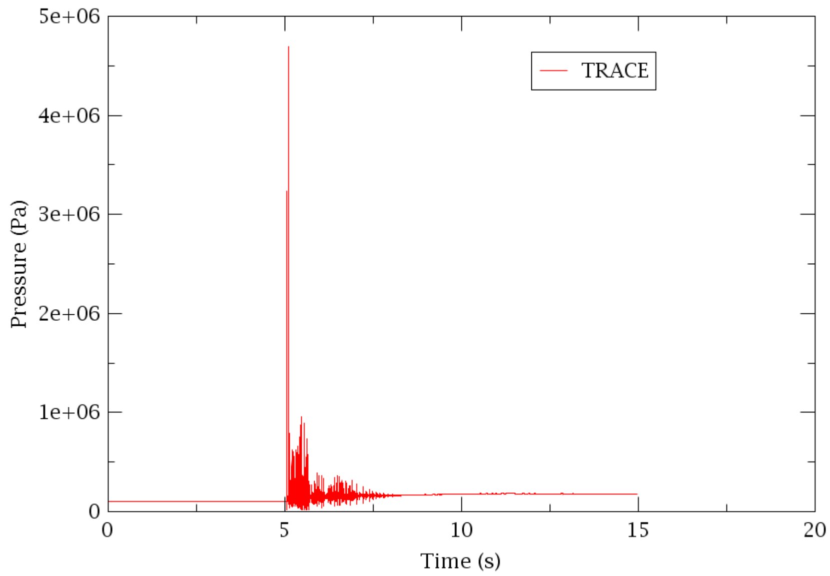


Figure 7.4: Pressure just downstream valve A in TRACE. The noise is much larger compared to the RELAP5 case.

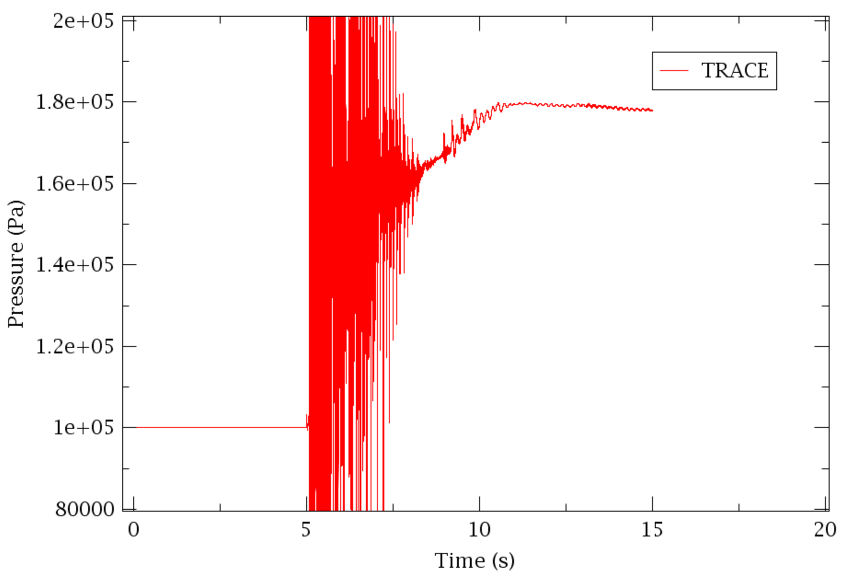


Figure 7.5: Pressure just downstream valve A in TRACE. Zoomed in to the same scale as Fig. 7.3. No useful information can be extracted during the three seconds after valve opening due to the large amount of noise.

Figs. 7.6 and 7.7 show the void fraction in the first four cells downstream valve A just after the valve has started to open. The main difference between the two codes is that TRACE fills one cell at a time with liquid water (indicating plug flow), whereas RELAP5 fills several cells simultaneously (indicating horizontally stratified flow). The horizontally stratified flow seems to be more realistic, since a liquid entering a gas-filled pipe should fall to the bottom of the pipe because of the differences in density between the gas and the liquid. As can be seen this is not the case in TRACE though. It is assumed that this is the reason for the water packing spikes that are present in Fig. 7.4.

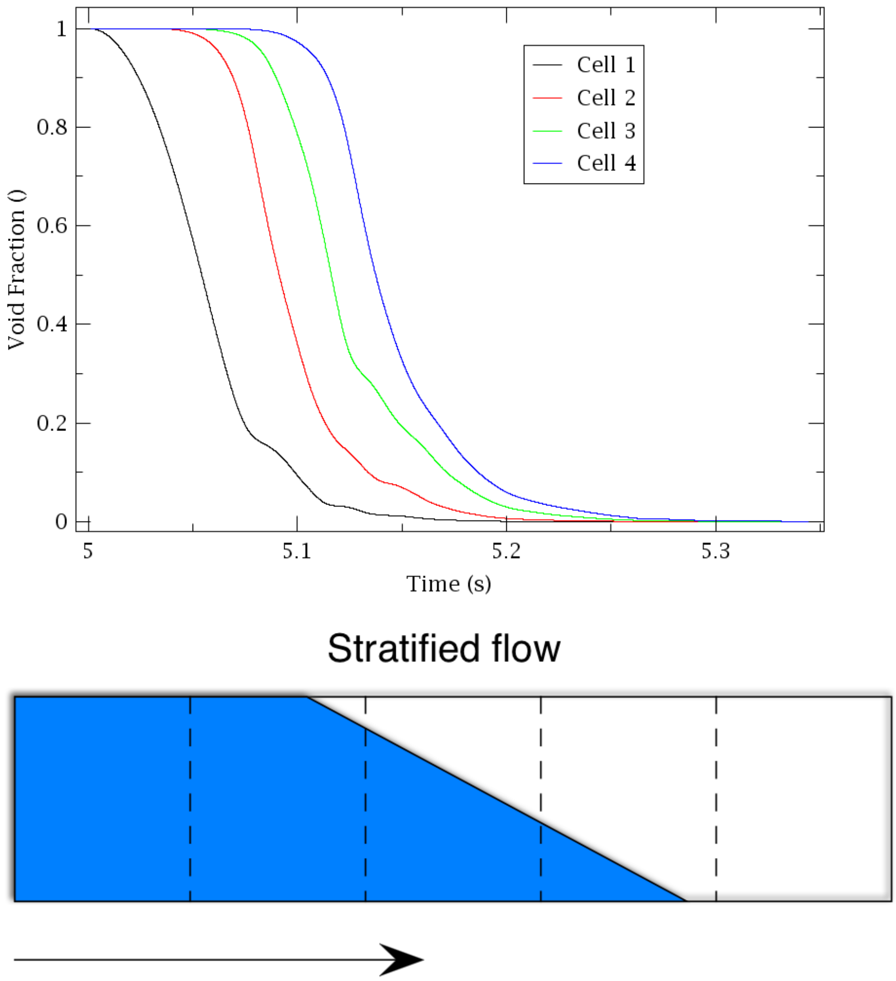
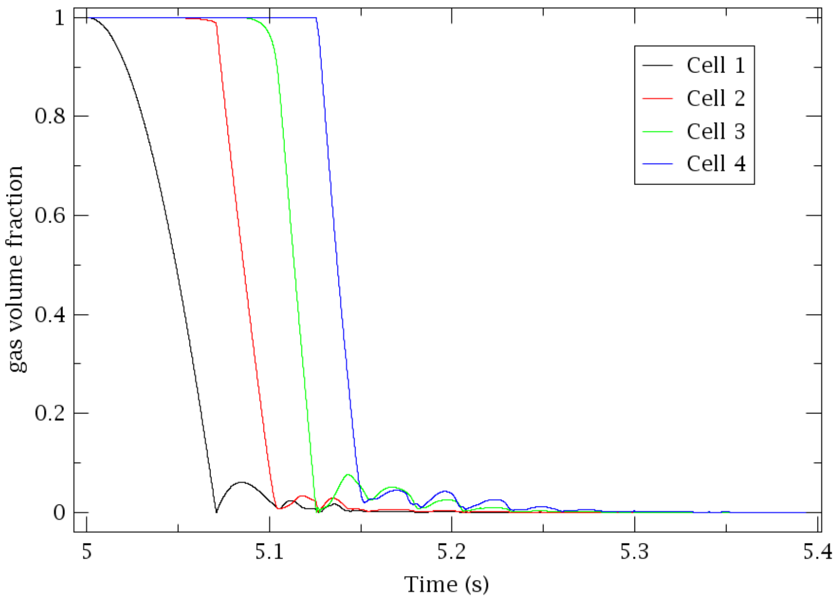


Figure 7.6: Void fraction in the first four cells downstream valve A in RELAP5. Cell 1 is closest to the valve. Several cells are filled with liquid water simultaneously, indicating a horizontally stratified flow.



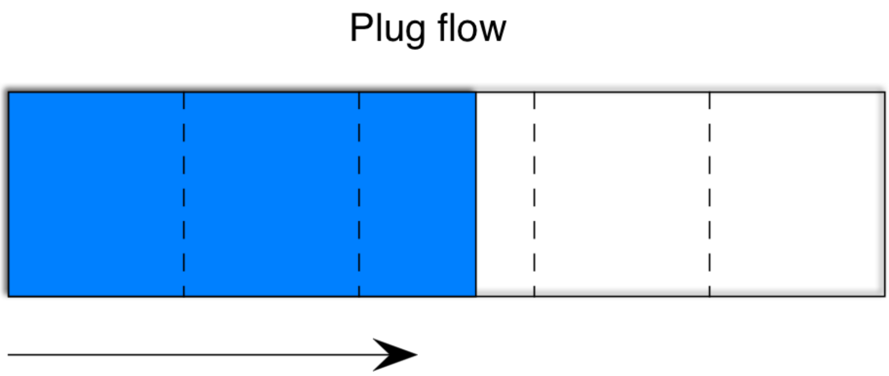


Figure 7.7: Void fraction in the first four cells downstream valve A in TRACE. Cell 1 is closest to the valve. Only one cell is filled with liquid water at a time, indicating plug flow. Also, cells which have been completely filled with liquid are suddenly drained of part of their liquid.

Both TRACE and RELAP5 contain a water packing mitigation scheme though, with the purpose of reducing pressure spikes originating from water packing. For some reason (which still remains unknown) the water packing mitigation scheme is not active in TRACE, and hence the spikes are not reduced. One explanation could be that water packing in RELAP5 is only applicable in vertical pipes, probably because it is not required in horizontal pipes because of the horizontally stratified flow. No information about this is given in the TRACE manual, but assuming that the same is true for TRACE as for RELAP5, it could explain why the water packing mitigation scheme is not active. Still, the problem is probably not the water packing scheme itself, but rather the inaccuracy of using plug flow. The presence of plug flow is what leads to the packing of the water, which in turn leads to (numerical) pressure spikes.

It should be noted that the horizontally stratified flow weighting flag in TRACE (which can have a value between 0 and 1, where 0 means no horizontal stratification and 1 means that the cell is horizontally stratified) does have a value greater than 0 (roughly 0.3 in the first cell, and slightly lower in cells further downstream, as can be seen in Fig. 7.8). Thus, the flow should be in the transition between plug and stratified. However, the results in Fig. 7.7 indicate that horizontal stratification is not at all what is actually obtained in the calculations.

Finally, it should be noted that some pressure spikes can be seen in the RELAP5 result as well. These are much smaller than in the TRACE result, though, and the overall behaviour (especially between t = 5 s and t = 8 s) is significantly more clear. It is assumed that this is because of the horizontally stratified flow.

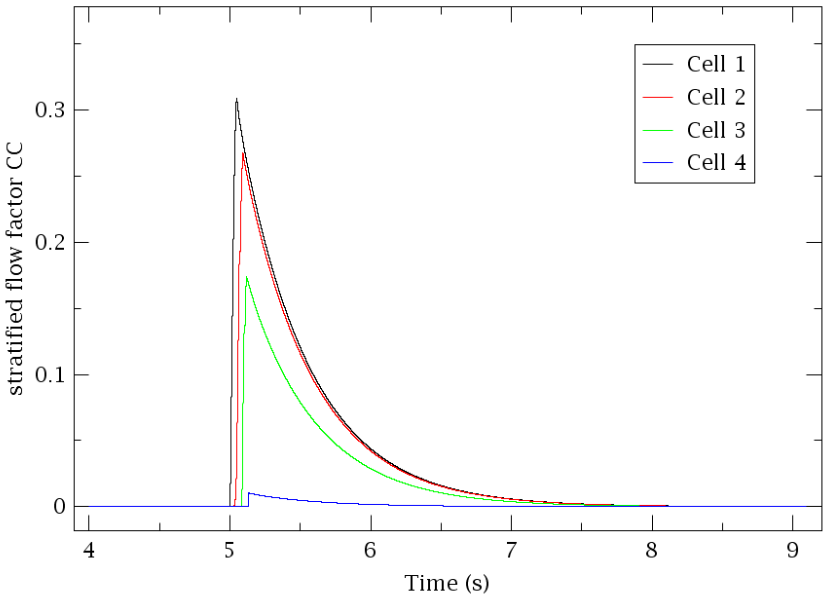


Figure 7.8: Horizontally stratified flow factor in the first four cells downstream valve A. Cell 1 is closest to the valve. It should be noted that cell 4 is the first vertical cell in this pipe component. It can be seen that the cells are in some transition between horizontally stratified and plug flow. No traces of the horizontally stratified flow can be seen in Fig. 7.7 though.

Several attempts have been made to improve the TRACE simulation. The first is to substitute the model of valve A (which in the converted model consists only of a junction) to a more appropriate TRACE valve model where one or more computational cells is added on each side of the junction. Although an improvement could be seen when running the simulation at longer time steps (10^{-3} s), no such improvement was visible as the time step was reduced to the required 10^{-5} s.

Another attempt to improve the results is to use one of the second-order spatial integration schemes available in TRACE 2.2.3.2. However, although better results were again obtained for long time steps, no improvements were seen for short time steps.

A similar attempt is made with the temporal scheme, where the standard SETS scheme is substituted for the semi-implicit scheme. This do not affect the solution in any observable way.

In order to reduce the complexity of the system, a simplified model has also been constructed, which consists only of two horizontal pipes with a valve between. The downstream valve is filled with nitrogen gas. Several different cases are then studied, including different pressures and temperatures, substituting the nitrogen for steam, and changing the pipe diameter (in the original model, the pipes upstream and downstream the valve have different diameters). Still, no improvement is observed.

It has been concluded that the underlying reason for the strange behaviour of the TRACE simulations (i.e. the extremely high pressure spikes) is probably the fact that liquid water gets mixed with nitrogen gas (or steam). This has not been the case in previous simulations, and it is assumed that TRACE simply does not handle this type of flow in a good way. Hence the plug flow observed from Fig. 7.7 emerges, and eventually leads to water packing spikes.

7.2 Case 3a – Closing of a Safety Valve

The closing of the safety valve is described in the following section, starting with a description of the method used. Then, the results are presented together with a discussion.

7.2.1 Method

The methodology used in this case is very similar to that which is used for Case 1a (see Section 7.1), since the same model is used. The difference is that valve A starts in its open position. After a time $t = 5$ s the valve is closed and the following transient is studied. What will happen is similar to the explanation of vapourous cavitation in the downstream pipe described in Section 2.1.1.

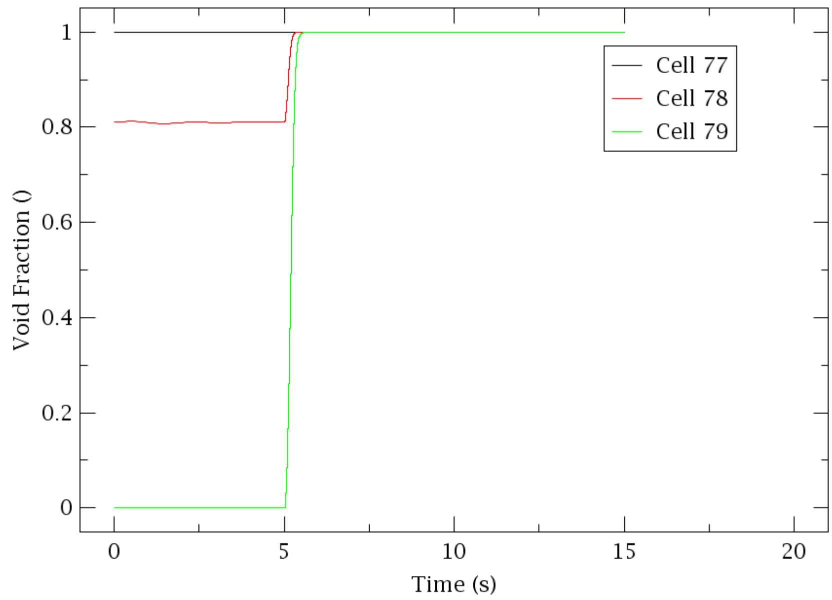
Before the transient can be started, steady-state must be reached in the piping system. Initially, the part of the system downstream valve A is filled with nitrogen gas (as in the previous case). Since the valve is open, the downstream part of the system starts filling with water. After sufficiently long time a steady state is been reached. The time at which the valve closes is measured relative to this steady-state.

It should be noted that at steady-state, parts of the system are still filled with nitrogen gas. These are located in the `PIPE` components labelled 204, 206, 300 and 400 (see Fig. 7.2), which contain regions above the level of the inflowing liquid water, and which end with dead ends, resulting in nitrogen gas being trapped here.

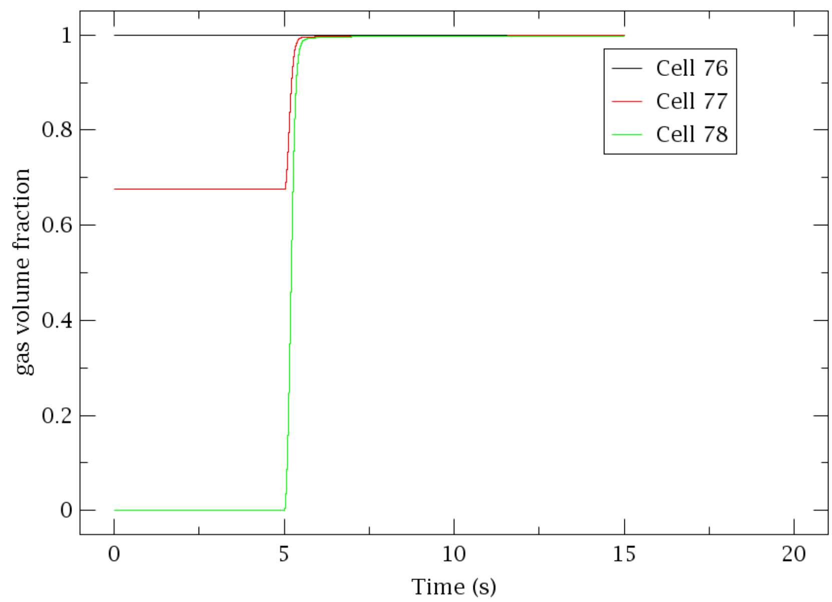
As has been noted in previous sections, steam collapse in general is rather difficult to model using system codes. Using the results of these previous sections the interfacial heat transfer coefficients in TRACE have been increased by different factors. The results have been compared to both standard RELAP5 simulations (using default settings) as well as RELAP5 simulations were the equilibrium flag has been turned on in the `PIPE` component just downstream valve A. Studied parameters include, as in previous cases, pressure and void fraction downstream the valve, and mass flow through the valve. Also, liquid levels in pipes 204, 206, 300 and 400 have been monitored (at steady-state). The effects of using the level tracking method have been studied.

7.2.2 Results and Discussion

Due to the thermodynamic conditions in the simulation, two liquid levels are present in the system at steady-state – one in pipe 206 and one in pipe 400. Above the levels, all cells are filled with nitrogen gas. It has been found that the effects of using level tracking are rather negligible, and activating the method did not have any major impact on the results. Fig. 7.9 shows that the location of the liquid level in pipe 206 varies slightly between RELAP5 and TRACE (in steady-state, i.e. $t < 5$ s). However, the results are considered to be similar enough.



(a) RELAP5.



(b) TRACE.

Figure 7.9: Void fraction in selected cells of pipe 206. Steady-state conditions apply for $t < 5$ s. All plotted cells are in the vertical location of pipe 206 and increased cell number means decreased elevation. In RELAP5 the liquid level is present in cell 78. In TRACE the liquid level is present in cell 77, i.e. slightly above the RELAP5 case. The difference is not very large though, since the cell length is only 0.2 m.

The pressure downstream valve A as calculated by RELAP5 is plotted in Fig. 7.10 for the standard case and the case where the equilibrium flag has been turned on. Three things are noticed. First, the amplitude of the first peak is greatly increased in the case with the equilibrium flag turned on. Second, the dampening of the pressure peaks is faster in this case. Third, the frequency of the pressure peaks is lower when using non-equilibrium. Since the geometry of the system is rather complex, it is difficult to get an intuitive feeling for which of the two cases is more correct. However, as is known from previous chapters, the RELAP5 non-equilibrium model underpredicts the amplitude of the pressure peaks, whereas the equilibrium model overpredicts them. Hence, it is expected that experimental measurements should give results somewhere between these two cases. Unfortunately, no such data has been available. Regarding the dampening of the pressure peaks, it would seem reasonable that the equilibrium case gives more correct results. However, the frequency for the peaks is virtually impossible to estimate except for very simple geometries (without branching pipes). Therefore it is not possible to say whether the equilibrium case is better or worse than the non-equilibrium case regarding this. An educated guess would be that the true result is somewhere between these two cases.

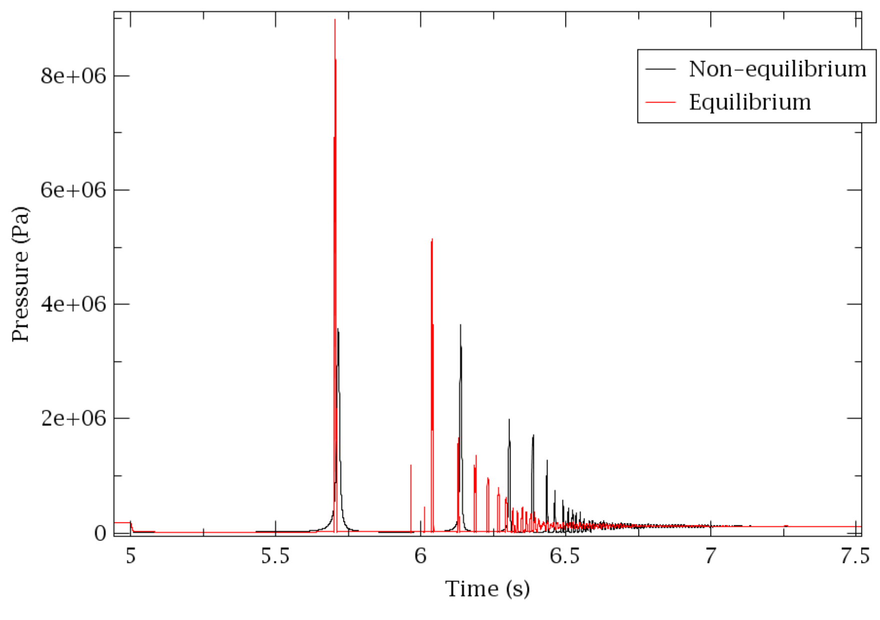


Figure 7.10: Pressure downstream valve A as calculated by RELAP5 using the non-equilibrium and equilibrium models, respectively. The peak values are much larger for the equilibrium case. Also, the peaks die out faster in this case.

Fig. 7.11 shows the pressure downstream valve A as calculated by TRACE for three different factors for the interfacial heat transfer coefficient. It can be noticed that for all three cases the magnitude of the first pressure peak is between the non-equilibrium and equilibrium cases calculated by RELAP5. This indicates that the TRACE simulations may be more accurate. Also, the dampening of the peaks is faster in all TRACE cases.

Finally the difference in frequency is much smaller between the different TRACE cases compared to between the two RELAP5 cases. Increasing the heat transfer by an even greater factor only increases the amplitude (mainly of the first peak) and frequency slightly.

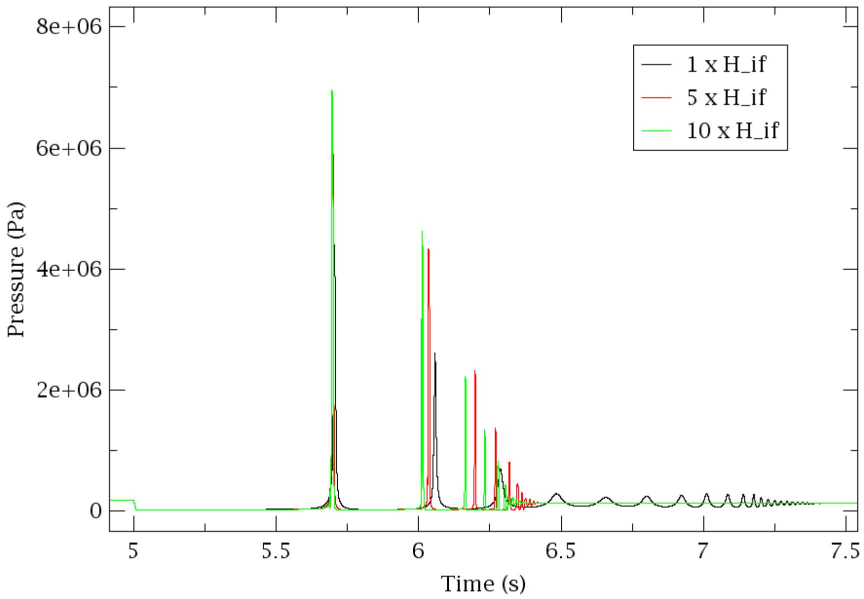


Figure 7.11: Pressure downstream valve A as calculated by TRACE using three different factors for the interfacial heat transfer coefficient. The differences between the cases are less distinct, although a slight increase of the peak amplitude is observed for an increased heat transfer coefficient.

One thing to notice is the fact that the first pressure peak comes a short time earlier in TRACE compared to RELAP5. Also, the amplitude of the second peak for the two cases with increased heat transfer corresponds rather well to the RELAP5 equilibrium case. It should be mentioned that the behaviour occurring after the first two or three peaks should not be considered as being entirely correct, as is discussed in [7]. This is mainly because the codes do not consider fluid-structure interaction (meaning that the peaks are not broadened in the way that they are in reality), and because the solution at this time is very sensitive to modelling details.

Chapter 8

Ringhals 4 - 321WH/PT2

In this chapter, an existing water hammer simulation [38] about different pump transients in the Residual Heat Removal System (RHRS) of Ringhals 4 (system 321) is considered. One of these transients, called WH/PT2[1], is studied in this chapter and represents a pump stop during Low Head Safety Injection (LHSI). The main purpose of this system is to mitigate the effects of a LOCA by pumping water from the borated water storage tank (RWST) into the reactor pressure vessel through the cold legs.

This case is made up of a slightly more complicated model than the one in Chapter 7. The purpose is to validate TRACE for another 'real' case, this time containing a number of different components (a large number of pipes, several orifices and inertial swing check valves as well as a pump).

8.1 Method

The part of the system that is modelled for the pump stop transient is shown in Fig. 8.1[2]. The model includes a `PUMP` component (for which homologous head and torque curves are given), pipes and valves. The heat exchanger shown in Fig. 8.1 is modelled as `PIPE` components. Nine check valves are included in the model, each modelled in the same way as described in Section 2.4. Thus, a significant number of control and signal variables are involved in the model. The rest of the valves seen in the figure are passive and are represented as `SINGLE-JUNCTION`s with a certain form loss. The same applies for pipe bends. For the details regarding how this model is implemented in RELAP5, and what boundary and initial conditions that are used, the reader is referred to [38].

[1] WH/PT = Water Hammer Pump Transient.

[2] The reason why an image of the RELAP5 or TRACE model is not used in this case is the rather messy arrangement of components when the input file has been imported to SNAP.

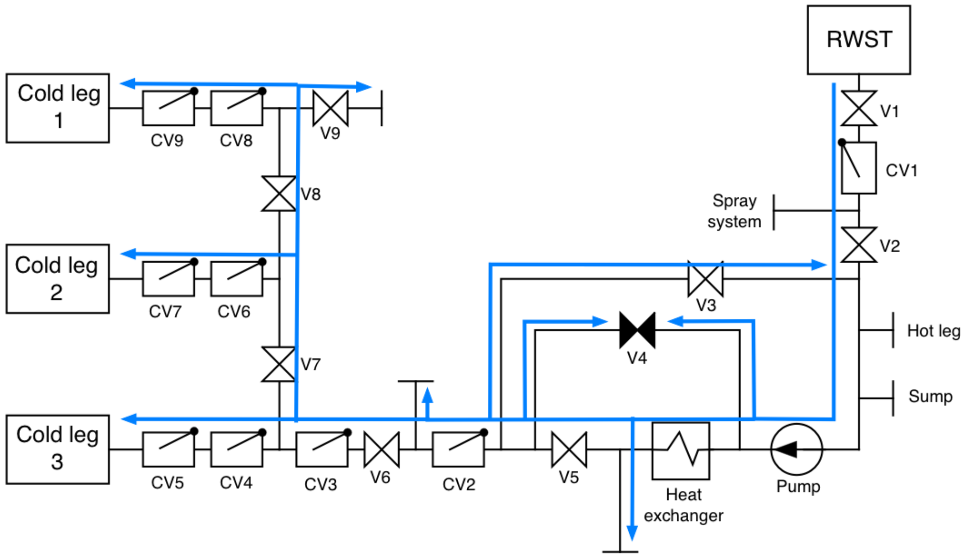


Figure 8.1: Simplified model of the WH/PT2 pump stop transient. The pump is initially in full operation, pumping water from the RWST into the three cold legs. The bypass line and the pipeline from the hot leg are closed during the transient. Possible flow paths are illustrated by blue lines. The nine check valves (CV1-CV9) are each modelled according to Section 2.4 while the other valves (V1-V9) are passive and modelled (in RELAP5) as `SINGLE-JUNCTION`s having a certain form loss. In TRACE, these passive valves are represented by the common edge between two adjacent pipes. For the plant-specific component names, the reader is referred to [38].

The methodology, to begin with, is to import and simulate the existing RELAP5 input file of the WH/PT2 pump stop model. In order to reach a steady-state where the pump is in full operation, the simulation is divided into three parts by making use of restart cases, as was performed in the original water hammer simulation. First, a transient simulation is performed during 50-200 s. Second, a steady-state simulation is started at 200 s and stops whenever the convergence criteria are met (or latest at 1000 s). Third, a transient simulation is performed during 0-25 s where the pump speed is reduced exponentially starting from $t = 10$ s. The parameters that are studied after the simulation are especially the ones related to the pump, e.g. the mass flow and pump head. Further, the calculated dynamic forces downstream the pump are investigated, as well as the behaviour of the check valves.

The RELAP5 model is then converted to a TRACE model using (as always) the SNAP conversion tool. The experience from the earlier chapters is taken into account for fixing possible bugs in SNAP or making other modifications in order to make the model work properly. Especially for the findings from Chapter 6 it is of great interest to see whether they apply also when several inertial swing check valves are put into a more complex model. Therefore, two different versions of the TRACE model is used depending on what velocity (through or into the valve) is used in the calculations. As in Section 6.1.2, these are called the unmodified and modified TRACE version, respectively. The two model versions are then simulated, using the same type of restart cases as in the RELAP5 simulation.

8.2 Results and Discussion

The first thing to mention is regarding the restart cases present in this model. When converting the RELAP5 model to TRACE in SNAP, the restart cases are lost, and have to be redefined. Although not a major issue, it is still a drawback which is important to be aware of.

Another problem with the conversion is related to the results obtained previously with bent pipes. When loops are present in the system it is possible that the conversion will result in two connecting pipes which have different altitudes. It is possible to simply ignore this (by not performing the 'Loop check' in TRACE[3]) and run the simulation anyway, but one should be aware that this may cause problems. In the present model a height difference of 12 cm is found by the 'Loop check'. This has been ignored in the analysis of the results, although it could be part of the reason for the differences in the results.

Further, it should be noted that the modified TRACE case, i.e. where the velocity *through* the respective inertial swing check valve is used instead of the velocity *into* the valve, requires a rather large damping torque (see Eq. (2.57) of Section 2.4) to be included in order to reach steady-state. This damping torque is later set to zero when the actual transient starts.

Because the model includes an extensive number of area changes (both smooth and abrupt) in valves and junctions, it is practically unmanageable to deal with these individually in order to achieve the same pressure losses everywhere as in the RELAP5 model. Therefore, only the form losses for the nine check valves are considered (and only for the modified TRACE version), while the losses for the passive valves and other edges are set to the respective default option (usually '[1] Flow Factor + FRIC' with or without an additional K-factor). For the check valves, three different approaches are tested: using the default option '[-1] Flow Factor + FRIC + Abrupt', using '[0] Constant FRIC', and using an implemented form loss table based on Eq. (2.24). It turns out that the overall results are very similar, but all three tests exhibit a slightly too small mass flow through the system and, often, combined with a larger valve opening angle. This indicates that the remaining losses elsewhere in the model are too large in TRACE, which, at least for abrupt area changes, is consistent with Eq. (2.24).

The conversion of the pump component seems to work very well though, which is the most important result obtained from this case. This is probably because the pump component works very similar in RELAP5 and TRACE. The only major difference is where the momentum source is applied. In RELAP5 the source term is applied in the pump computational cell, whereas in TRACE it is applied in the edge between two computational cells. This has not been found to affect the simulation results noticably. Pump parameters such as head, angular speed and torque are converted without any problems, as well as homologous curves. It should be mentioned that no degraded pump curves were supplied for this model though. If such curves are present, problems arise because of the differences in inputs required for RELAP5 and TRACE, as described in Section 2.2.6.4[4].

[3]This is a model validation tool with the purpose of detecting errors in elevation within loops.

[4]Converting degraded pump curves has not been tested explicitly in this project, but has earlier been

Fig. 8.2 shows the mass flow through the pump as calculated by RELAP5 and TRACE (both unmodified and modified), respectively. It can be seen that all three simulations have the same general behaviour. However, the unmodified TRACE case is much smoother than the RELAP5 case and the modified TRACE case. This means that there is less acceleration of the water in the system, which in turn leads to lower forces acting on the pipes. Fig. 8.3 shows an example of a force, in `PIPE` component 345, which is the pipe downstream valve CV2 (see Fig. 8.1). The force in this particular pipe segment was chosen because it is one of the forces with the highest amplitude (in the RELAP5 simulations). As can be seen, the amplitude is much lower for the unmodified TRACE case than for the other two cases.

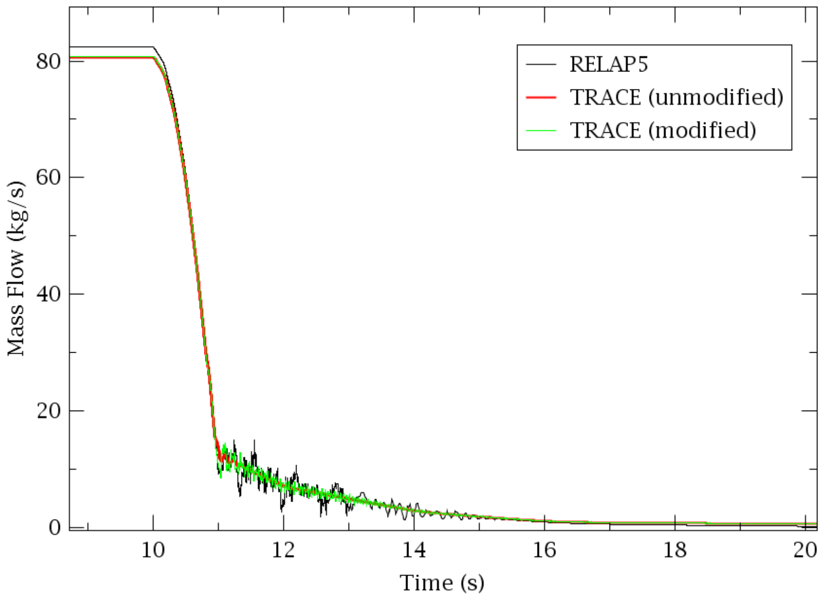


Figure 8.2: Mass flow through the pump in the WH/PT2 transient, in which the pump speed is reduced exponentially starting at $t = 10$ s. The slightly different steady-state mass flows are likely due to different form losses at some of the area changes included in the model. The general behaviour is very similar though.

confirmed by Mikael Fjällborg, RTPP.

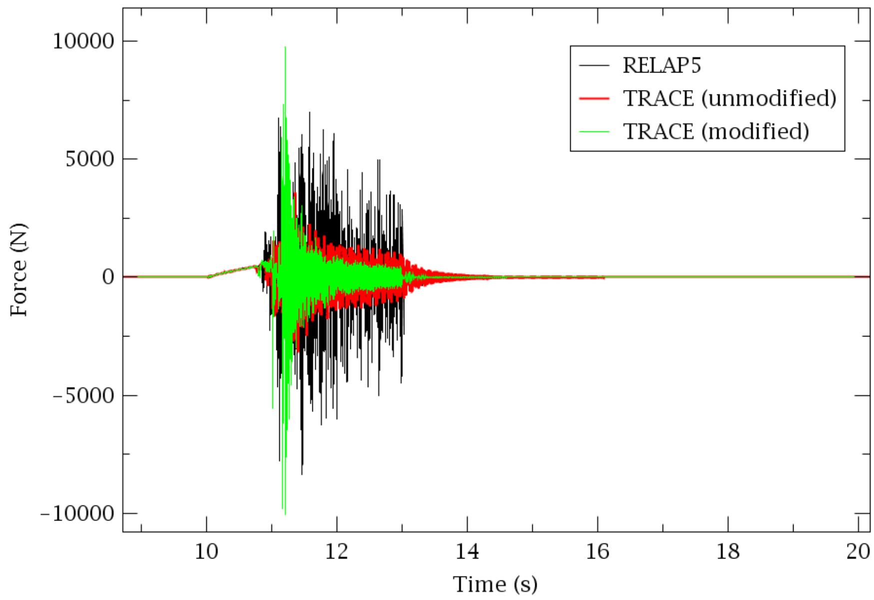


Figure 8.3: Dynamic force on pipe 345, which is the pipe downstream valve CV2 shown in Fig. 8.1. Observe how smoothly the TRACE simulations die out, because of less chattering for the inertial swing check valves. The drastic 'dead' of RELAP5 at 13 s is due to the increase of the maximum time step to 10^{-4} s.

Furthermore, although the maximum force amplitude calculated by the modified TRACE model corresponds rather well with the result from the RELAP5 simulation, the behaviour over time is much less similar. The force amplitude in RELAP5 remains rather constant over the following seconds, whereas in TRACE it is dampened over time. A sensitivity analysis shows that the forces calculated by the two codes are highly dependent on the maximum time step size chosen by the user. In the simulation, this time step size is changed from the standard 10^{-5} s to 10^{-4} s at $t = 13$ s in order to save computational time during the rest of the simulation (as was done originally in [38][5].). This corresponds to the time when the force in RELAP5 abruptly disappears. By changing to a larger time step size a little bit earlier (e.g. at $t = 12$ s), the abrupt end then takes place at this point. By doing the same in TRACE, the effect is simply that the smooth 'roll-out' gets slightly quicker. Overall, the forces over time are smaller when a larger maximum time step size is used due to a larger numerical damping. In order to not underestimate the loads, a smaller time step (e.g. 10^{-5} s) should preferably be utilised during the majority of the transient. It should also be noted that the maximum (absolute) force amplitude is higher in TRACE than in RELAP5 for the same time step, i.e. the TRACE result is more conservative.

The differences between the simulations originate from the phenomenon which gives rise to the forces – the closing of the inertial swing check valves. Every time an inertial swing

[5] In [38], the time step size is changed from 10^{-5} s to 10^{-3} s at $t = 13$ s

check valve closes a large force arises, since the mass flow more or less instantaneously is reduced to zero. The main difference between the simulations is actually the behaviour of these inertial swing check valves. For instance, Fig. 8.4 shows the opening angle of valve CV3 during the transient. As can be seen, the valve behaves very differently in the three cases. In RELAP5 the valve repeatedly opens and closes, with rather large opening angles. Almost no such behaviour can be seen in the unmodified TRACE case (there is a small opening of the valve at $t = 11.3$ s, which is not visible in the figure though). Because of this, the amplitude of the force becomes much smaller. In the modified TRACE case, the valve opens and closes a few times during the first half second after the initial closure. The opening angles are much smaller than in the RELAP5 case though. Also, after this first half second, the valve is more or less constantly closed[6]. The same overall appearance applies also for the other eight inertial swing check valves. By increasing the maximum time step size in RELAP5 to 10^{-4} s the chattering tendency, which may be numerical as speculated in Section 6.2.2, is suppressed and the forces become smaller over time and more like TRACE. Thus, even though no experimental data is available for this case, it does seem that TRACE may be more realistic than RELAP5 due to the probable numerical chattering of the intertial swing check valves.

It should further be noted that the opening angles are slightly different during steady-state as well (the difference is around 4° between RELAP5 and the modified TRACE version in the case of valve CV3). The fact that the deviation is larger for the modified TRACE version compared to the unmodified ditto is probably explained by different K-factors required after the modification of the valve model (also noted in Chapter 6). Different losses elsewhere in the model may also contribute to the deviation from RELAP5. Since no experimental data is available it is not possible to say which code is more correct.

To conclude; the conversion of the pump works very well, and the conversion of the inertial swing check valves works acceptably well. In order to not underestimate the loads (compared to RELAP5) the check valves should be modified in the same way as in Chapter 6. The big issue in this case is how to handle the pressure losses at the large number of area changes appearing in the model.

[6]In both TRACE simulations small oscillations in the opening angle can be seen (in the order of less than 10^{-3} degrees). These small openings and closures result in oscillations of the force in pipe 345 in the order of less than 1 mN, and are therefore considered negligible.

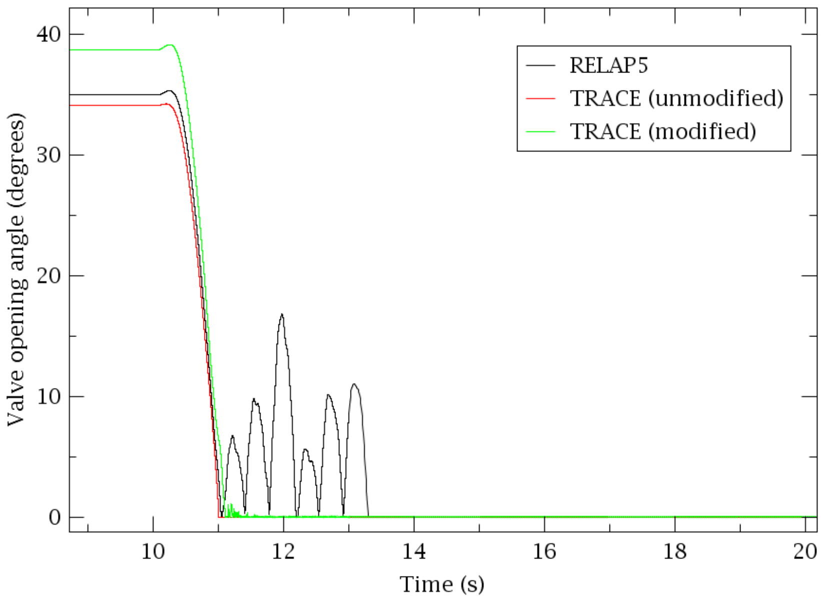


Figure 8.4: Opening angle of valve CV3 during the transient. Note the different chattering tendency of the valve in the different simulations. As observed in Chapter 6, the inertial swing check valve tends to chatter less in TRACE than in RELAP5. Also, the deviation in steady-state opening angles (between the two TRACE versions) is probably due to different pressure losses across the valve when the model has been modified, as noted in Chapter 6.

Chapter 9

General Results and Important Findings

A number of important discoveries made during the project are summarised and discussed in the sections below. These include both differences between the two codes as well as bugs found, mainly in the conversion tool.

9.1 Condensation Rate

As was clearly demonstrated in Chapter 4, both RELAP5 and TRACE seem to have a too slow condensation rate when a pressure wave causes vaporous cavities to collapse. The reason why this is of such an importance to model correctly is the underpredicted water hammers resulting from the slow steam collapse. Using default options, none of the two codes was able to fully capture the measured short duration pulse. Especially TRACE exhibited a rather disperse pressure peak, which was found to be caused by a small remaining vapour pocket that did not collapse. Because the compressibility of a two-phase flow (even for very small void fractions) is much greater than that of a pure liquid, the dampened behaviour seems understandable.

The same conclusion about the condensation rate has been drawn in earlier analyses of using RELAP5 and/or TRACE for fast pressure transients, thus it is a known area of concern. Some of these analyses are [6], [7] and [8]. An e-mail conversation has been carried out during this project with two of the authors[1] of [6] and [7] in order to discuss the underlying reason for the slow steam collapse and possible ways of improving the results. There are likely two things that seem plausible to be the culprit: either the modelling of the interfacial heat transfer coefficient h_{if} or the interfacial area a_{gf} (or both). As described in Section 2.3.2, these coefficients strongly influence the heat and mass exchange rates between the phases, and thus the condensation and vapourisation rates. The authors' speculation was that a_{gf} calculated by the codes may be too small during the collapse of a steam bubble, at least in case of a stratified flow regime. The motivation was that, in reality, local turbulence makes the surface very wavy, which

[1] Annalisa Manera, University of Michigan and Werner Barten, Swiss Federal Nuclear Safety Inspectorate ENSI.

would give a larger total interfacial area compared to a flat surface. According to the TRACE Theory Manual, this 'stratified wavy' flow regime *is* taken into account when calculating h_{if}, but not a_{gf} as it seems (see p. 219-221 in [18]). However, when studying the results of the experiment in Chapter 4, the flow regime flagged by TRACE was only stratified to a minor degree. Therefore, the corresponding interfacial heat transfer model for stratified flow was probably not utilised in this case[2]. Further, according to their reasoning, the interfacial area would then be reduced when the water column hits the steam bubble and comes at rest, reducing the amount of turbulence and waviness of the surface. A proposed improvement was therefore to, in some way, calculate the time-dependence of a_{gf} in order to account for the occasionally larger interfacial area. Independently of the underlying reason and possible ways of improvements, the main problem is perhaps the basic fact that both RELAP5 and TRACE are 1D system codes and that real 3D phenomena thus are greatly simplified, as speculated in [7].

What they did to improve the results was to modify the source code by making subroutines in order to increase the heat transfer factor H_{if}, which is similar to the approach used in this project. The investigation in especially Chapter 4 showed that the condensation rate could be increased significantly even without modifying the source code. In RELAP5, this was done by activating the equilibrium flag in those cells where steam formation was expected. However, this approach may be too extreme because the phase transition then becomes more or less instantaneous. There is no 'middle way', which resembles a more realistic condensation rate. But the results, when compared to measurements, were still more credible with the equilibrium flag activated. In TRACE, the corresponding methodology was somewhat more sophisticated because the heat transfer coefficients for each flow regime and phase could be modified by a multiplicative factor[3]. However, the drawback of this is that the entire model is affected, and not only a certain part of interest. It should be mentioned, though, that the results in Chapter 4 were almost identical to the RELAP5 equilibrium case when a very large factor (e.g. 10^6) was used in front of h_{if}. In order to not underestimate the water hammers, these settings are the recommended ones when steam collapse is present in the model. Still, further research and development is recommended for TRACE to be able to fully capture these cavitation water hammers.

9.2 Pressure Losses

It has been noted throughout the project that even though the same parameters have been supplied to a RELAP5 model and a TRACE model, the results have been slightly different, also for very simple cases. The following sections are aimed at explaining where the differences come from. For a more detailed review, see Section 2.3.1.

[2]It might have been weighted by a small factor though.

[3]Once again, this feature is not available in the current commercial release of TRACE (V5.0), and it is uncertain whether it will be included in the next actual release. Until then, it is recommended to use the V5.830 (BETA) version.

9.2.1 Friction Loss

As described in Section 2.3.1.1, the friction loss models in RELAP5 and TRACE are slightly different. As can be seen in Fig 2.6 the difference is largest for pipes with small hydraulic diameter, i.e. small flow area. It has been proposed [41] that the friction loss model in TRACE is erroneous and that it should be modified. In the current version of TRACE this has not been done though.

For comparative analysis it is possible to turn off the friction models in both RELAP5 and TRACE, but this is not recommended for ordinary analyses.

9.2.2 Form Loss

It has been found previously in this project that there are differences in the way that pressure losses are modelled. The main difference lies in the velocity which multipies the K-factor. In RELAP5 it is the downstream velocity which is used, wheras in TRACE it is the velocity in the actual edge considered which is used. More about this can be found in Section 2.3.1.2.

There are also differences in the abrupt area change models of the two codes, especially when it comes to valves. It is important to be aware of this when e.g. converting RELAP5 models to TRACE, since the exact same results will not be achieved. This does not mean that the TRACE results are wrong, but rather that the two codes use slightly different modelling approaches regarding form losses, which can affect mass flows and pressures slightly. In water hammer simulations these are important parameters which can greatly affect what loads that will be calculated to act on the system. It is therefore important to make sure that reasonable values are achieved for these parameters.

Further it should be noted that the difference in the modelling of K-factors does not affect pressure losses in pipe bends, since the velocity is constant if no area change occur. It is 'only' in valves, orifices, abrupt contractions and abrupt expansions that different simulation results are expected from the two codes, because of the modelling of form loss factors.

9.3 Boundary Conditions

The implementation of boundary conditions is rather straightforward in RELAP5, since the geometry of the `TMDPVOL`-component is irrelevant for how the code interprets the supplied pressure. In TRACE however, it is important to define the geometry of the `BREAK`-component according to the guidelines in Volume II of the TRACE manual [21] in order to achieve the expected results. These guidelines are briefly summarised in Section 2.2.6.5. For a full description of the guidelines as well as a full explanation of the underlying reasons, the reader is referred to Volume II of the TRACE manual [21].

It has been found that for the models studied in this project, the `BREAK` component should be defined as follows: the length of the component should be set to a very small value, and the volume should be set to a very large value (10^{-6} m and 10^{6} m^3 for the length and volume, respectively, have been used in this project).

9.4 Conversion Problems

During the course of this project a number of models of different complexity have been converted from RELAP5 to TRACE. Several problems have been found with the conversion tool, which are presented hereafter. Some have been mentioned earlier wheareas some are mentioned for the first time in this section.

First, division operators are not converted correctly. There seems to be a bug in the SNAP conversion tool which switches the denominator for the numerator and vice versa. This can obviously lead to large errors in control systems. It is expected that this bug will be fixed in upcoming versions of SNAP, but until then it is an important issue to be aware of.

Second, it was found during the study of the inertial swing check valve (see Chapter 6) that signal variables are not always converted correctly. The control system for the inertial swing check valve used in Chapter 6 involves the pressure difference upstream and downstram the valve. Therefore, two signal variables are present which measure the pressure in the cell immediately upstream and downstream the valve, respectively. For some reason, the signal variable related to the upstream position instead measured the pressure *two* cells upstream after conversion. Although the effects on the behaviour of the inertial swing check valve are probably minor, it could lead to problems in certain cases. Especially since the root cause for this error remains unknown, which means that it is possible for errors with larger impact on simulation results to arise during conversion of signal variables.

Third, degraded homologous curves for the `PUMP` component are converted incorrectly. This is a bug that probably arises because of the different inputs required for these curves in RELAP5 and TRACE, as described in Section 2.2.6.4.

Fourth, if restart cases are present in the SNAP model, they are not converted to the TRACE model. Restart cases must then be defined again in the converted model.

Fifth, the mixture level tracking scheme is not converted. If such a scheme is present at any location in the RELAP5 model it must be re-activated in the converted TRACE model.

Sixth, valve characteristics tables are, in some cases, lost during conversion and have to be re-entered in the TRACE model.

Seventh, conversion of models containing both vertical and horizontal pipes can lead to problems with the altitude of certain pipes. Since RELAP5 and TRACE bend pipes at different locations (at junctions in RELAP5 and at cell centres in TRACE), errors can obviously arise during conversion. It is important to make sure that all bends occur at the correct positions in pipes in order to accurately simulate a system. If loops are present in the model, the erroneous conversion of bent pipes can lead to inconsistency at loop closures, i.e. two connecting pipes having different altitudes. This will cause an error in the 'Loop check' validation test. It is still possible to run the model (by e.g. turning off the 'Loop check') but one should be aware that incorrect simulation results may be achieved. Instead, the incorrect pipes should be located and manually modified in order to correctly model the physical system. If required, it is however possible to instruct TRACE to bend pipes in the same way as RELAP5 does, by setting the `NAMELIST` variable IELV to alternative [2]. Still, modifications must be made in all bends to make sure that they occur at the right locations. It should be noted that this can be a tedious work if the model is large.

It should be noted that, by converting to TRACE patch 1, pipes are automatically bent in the same way as in RELAP5 (i.e. at junctions). This is because the NAMELIST variable IELV is set to option [2] by default when performing this conversion. However, there may still be errors related to altitude in the converted model, and all pipe bends must then be revised manually. If the RELAP5 way of bending pipes is desired, IELV = [2] can be set also in the TRACE patch 3 converted model. A manual revision is required in this case as well.

9.5 Valve Modelling

Valves work similarly in RELAP5 and TRACE although the valve geometry is different, as described in Section 2.2.6.3 (in RELAP5 a valve is made up of a single junction, whereas in TRACE it is made up of several cells). A RELAP5 valve will be converted to a valve with 0 cells in TRACE, which is more or less identical to the original RELAP5 valve.

The differences regarding form loss factors described in Section 2.3.1.2 and 9.2.2 need to be considered when modelling valves. Which of the two codes that uses a more correct approach is outside the scope of this project to determine, but it is important to be aware that there are differences.

9.5.1 Inertial Swing Check Valve

The RELAP5 model of an inertial swing check valve developed at Ringhals turned out to work satisfactory when converted to TRACE, at least after a few modifications. First, the following conversion bugs (earlier mentioned in Section 9.4) had to be fixed; the division operators, the lost valve characteristics and a few signal variables that measure the pressure in wrong cells. After fixing these bugs, the TRACE model can be simulated with fairly decent results. However, the closing time is slightly shorter than in RELAP5, which may give non-conservative results. One further modification is necessary in order to achieve the same closure characteristic, namely to measure the flow velocity through the actual valve instead of the velocity into the valve (see Section 2.4). It remains a mystery though why this modification is required, and further investigation is recommended for finding the underlying reason. One may speculate that the difference has something to do with the different velocity ($v_{j+1/2}$ or v_{j+1}) that is used in the equation for the pressure loss (see Section 2.3.1.2).

It can also be mentioned that it may be more difficult to reach steady-state when the velocity through the valve is measured. In some cases, it is enough to decrease the maximum time step during the steady-state calculation, while in other cases a larger damping term (see Eq. (2.57)) is needed.

Finally, the pronounced chattering behaviour seen in RELAP5 at small time steps is not visible in TRACE to the same extent. This may be due to a higher numerical damping in TRACE for the same time step. However, it can also indicate that the chattering seen in RELAP5 is numerical and that TRACE, in such a case, is more realistic. It would be interesting to compare the simulated chattering with measurements of a corresponding real swing check valve in order to find out which of the two codes that is closest to reality.

9.6 Pump Modelling

Pumps work in very similar ways in RELAP5 and TRACE. The main difference is where the source term is applied. In RELAP5 it is applied in the single cell of the `PUMP` component, whereas in TRACE it is applied in the junction between the first and second cell. Also, a `pump` can have any number of cells except 1 in TRACE, whereas it can have only 1 cell in RELAP5. During conversion this is solved by simply splitting the single RELAP5 cell into two cells of half the size of the RELAP5 cell.

One should be aware of the erroneous conversion of the degraded homologous curves for the `PUMP` component, as mentioned above (see Section 9.4).

More about the `PUMP` component can be found in Section 2.2.6.4 or in the manuals of the respective codes [14, 18]. A last remark is how simplified the 1D pump models actually are when compared to a real 3D pump, or any other turbomachine [42].

Chapter 10

Conclusion

The aim of this project is to validate TRACE, against RELAP5 but also against measurements, as a tool for modelling fast pressure transients in nuclear power plant piping systems. During the course of the project, a number of simulations – spanning over a wide range of possible sequences of events – have been performed in both of these codes, and the results have been studied and compared. A good view of the potential and limitations of TRACE for this purpose has been obtained, which are presented briefly in this concluding chapter.

In general, the TRACE code is a suitable successor to RELAP5 regarding modelling of fast presssure transients. Although both codes have rather poor capabilities to model steam collapse and cavitation water hammers, the conclusions from this project are that TRACE performs more or less as well as RELAP5. The most important issue at the moment is the development of the conversion tool in SNAP in order to reduce the number of bugs in the software (which are described in Section 9.4).

Over a longer perspective it is of interest to develop TRACE to better handle this type of fast transients. Special effort should be put into the modelling of steam collapse. It should be remembered that neither TRACE nor RELAP5 are originally intended for simulation of this type of transients, though[1]. In fact, fast pressure transients such as water hammer events may be on the border, or even beyond the border, of their applicability.

[1] An attempt has been made to create a code which captures the phenomena of water hammers better than todays system codes, in a project called WAHALoads [43].

Bibliography

[1] Nordlund, A. (2012) *Introduction to Nuclear Reactors.* Chalmers University of Technology, Göteborg.

[2] Demazière, C. (2012) *Physics of Nuclear Reactors.* Chalmers University of Technology, Göteborg.

[3] Choppin, G., et al. (2002) *Radiochemistry and Nuclear Chemistry.* Third edition. Oxford: Butterworth-Heinemann.

[4] SSMFS 2008:1. *Strålsäkerhetsmyndighetens föreskrifter och allmänna råd om säkerhet i kärntekniska anläggningar.* Stockholm, Strålsäkerhetsmyndigheten

[5] US Nuclear Regulatory Commission. (2013) Computer Codes *U.S.NRC* `http://www.nrc.gov/about-nrc/regulatory/research/safetycodes.html` (2014-05-07)

[6] Barten, W. et al. (2008) Analysis of the Capability of System Codes to Model Cavitation Water Hammers: Simulation of UMSICHT Water Hammer Experiments with TRACE and RELAP5. *Nuclear Engineering and Design*, vol. 238, pp. 1129–1145.

[7] Barten, W. et al. (2011) Analysis of the UMSICHT Water Hammer Benchmark Experiment 329 using TRACE and RELAP5. *Multiphase Science and Technology*, vol. 23, pp. 1-27.

[8] Barna, I.F., et al. (2010) Experimental and theoretical study of steam condensation induced water hammer phenomena. *Nuclear Engineering and Design*, vol. 240, pp. 46-150.

[9] Nuclear Regulatory Commission et al. (2013) *TRACE V5.830 User's manual Volume 1: Input specification.* Washington: USNRC.

[10] Thorley, A.R.D. (2004) *Fluid transients in pipeline systems: a guide to the control and suppression of fluid transients in liquids in closed conduits.* London: Professional Engineering Publishing.

[11] Shu, J.J. (2003) Modelling vaporous cavitation on fluid transients. *International Journal of Pressure Vessels and Piping*, vol. 80, pp. 187-195.

[12] Simpson, A.R., Wylie, E.B. (1991) Large Water-Hammer Pressures for Column Separation in Pipelines. *Journal of Hydraulic Engineering*, vol. 117, pp. 1310-1316.

[13] Barten, W., et al. (2006) RETRAN-3D analysis of the base case and the four extreme cases of the OECD/NRC Peach Bottom 2 Turbine Trip benchmark. *Annals of Nuclear Energy*, vol. 33, pp. 99–118.

[14] Information System Laboratories Inc. (2010) *RELAP5/MOD3.3Patch04 Code manual, Volume I: Code structure, system models, and solution methods.* Rockville, Idaho Falls: USNRC.

[15] Information System Laboratories Inc. (2010) *RELAP5/MOD3.3Patch04 Code manual, Volume III: Developmental assessment problems.* Rockville, Idaho Falls: USNRC.

[16] Los Alamos National Laboratory. (2001) *TRAC-M/FORTRAN90 (Version 3.0) User's Manual* Los Alamos: USNRC.

[17] Andersson, B. et al. (2013) *Computational Fluid Dynamics for Engineers.* Ninth Edition. Cambridge: Cambridge Univesity Press.

[18] Nuclear Regulatory Commission et al. (2013) *TRACE V5.830 Theory Manual.* Washington: USNRC.

[19] Björndahl, O., et al. (2007) *Rekommendationer för analys av spänningsrespons i rörsystem utsatta för termohydrauliska transienter.* SKI Rapport 2008:13. ISSN: 1104-1374.

[20] Information System Laboratories Inc. (2010) *RELAP5/MOD3.3Patch04 Code manual, Volume II: User's guide and input requirements.* Rockville, Idaho Falls: USNRC.

[21] Nuclear Regulatory Commission et al. (2013) *TRACE V5.830 User's manual Volume 2: Modeling guidelines.* Washington: USNRC.

[22] Information System Laboratories Inc. (2010) *RELAP5/MOD3.3Patch04 Code manual, Volume V: User's guidelines.* Rockville, Idaho Falls: USNRC.

[23] Zigrang, D.J., Sylvester, N.D. (1985) A Review of Explicit Friction Factor Equations. *Journal of Energy Resources Technology*, vol. 107, pp. 280-283.

[24] Colebrook, C.F. (1939) Turbulent Flow in Pipes with Particular Reference to the Transition Region Between Smooth and Rough Pipe Laws. *Journal of Institution Civil Engineers*, vol. 11. pp. 133-156.

[25] Churchill, S.W. (1977) Friction Factor Equation Spans All Fluid-Flow Regimes. *Chemical Engineering*, vol. 84, pp 91-92.

[26] Information System Laboratories Inc. (2010) *RELAP5/MOD3.3Patch04 Code manual, Volume IV: Models and correlations.* Rockville, Idaho Falls: USNRC.

[27] Plesset, M.S., Zwick, S.A. (1954) The Growth of Vapor Bubbles in Superheated Liquids. *Journal of Applied Physics*, vol. 25, pp.493-500.

[28] Lee, K., Ryley, D.J. (1968) The Evaporation of Water Droplets in Superheated Steam. *Journal of Heat Transfer*, vol. 90, pp. 445-451.

[29] Unal, H.C. (1976) Maximum Bubble Diameter, Maximum Bubble-Growth Time and Bubble Growth Rate During the Subcooled Nucleate Flow Boiling of Water

up to 17.7 MN/m^2. *International Journal of Heat and Mass Transfer*, vol. 19, pp. 643-649.

[30] Ranz, W.E., Marshall, W.R. (1952) Evaporation from drops. *Chemical Engineering Progress*, vol. 47, pp. 141-146.

[31] Aktas, B. (2003) *Level tracking in thermal-hydraulic simulations of nuclear reactors.* Ph.D. Thesis, The Pennsylvania State University, State College.

[32] Ballun, J.V. (2007) A methodology for predicting check valve slam. *American Water Works Association. Journal*, vol. 99, pp. 60-65.

[33] Joheman, C. (2010) *Beräkningsmodell RELAP5 Klaffbackventil.* Internal Ringhals document, Darwin-ID: 2029239/2.0.

[34] Björklund, K. (2010) *Comparison between RELAP5 and TRACE for modelling different loads on pipe systems during transient conditions.* M.Sc. thesis at Uppsala University, Uppsala.

[35] CRANE Co. (1983) *Flow of Fluids Through Valves, Fittings, and Pipe. Metric Edition - SI units.* Technical Paper No. 410M. United Kingdom. (Also available as an internal Ringhals document, Darwin-ID: 2005527).

[36] Andersson, C. (2008) *FoU - Simulering av ångkollaps med DRAKO och RELAP5.* Internal Ringhals document, Darwin-ID: 1949221/2.0.

[37] Edebro, D. (2013) *Ringhals 3 - Belastningsunderlag 321WH/SV1, 321RF/SV1 - Säkerhetsventillaster.* Internal Ringhals document, Darwin-ID: 2207942/2.0.

[38] Nyström, A. (2012) *Ringhals 4 - Belastningunderlag 321WH/PT1 och 321WH/PT2.* Internal Ringhals document, Darwin-ID: 2144596/3.0.

[39] Tiselj, I., Černe, G. (2000) Some Comments on the Behavior of the RELAP5 Numerical Scheme av Very Small Time Steps. *Nuclear Science and Engineering*, vol. 134, pp. 306-311.

[40] Andersson, P., Sandebert, C. (2011) *Investigation of solution methods suitable for modelling steam collapse and pressure peaks using RELAP5.* M.Sc. thesis at the Department of Nuclear Engineering, Chalmers University of Technology, Göteborg.

[41] Krotiuk, W.J. (2011) *Recommendation for Edge Friction Pressure Drop Improvements.* Available on internal bug-report page of USNRC.

[42] Dixon, S.L., Hall, C.A. (2010) *Fluid Mechanics and Thermodynamics of Turbomachinery.* Sixth Edition. Amsterdam: Elsevier.

[43] Giot, M. et al. (2004) *Two-phase flow water hammer transients and induced loads on materials and structures of nuclear power plants.* WAHALoads project