WAHA Code
Development of Single Phase Code

I. Tiselj, G. Černe, I. Parzer, A. Horvat

Ljubljana, Junij 2002

Institute "Jožef Stefan", Ljubljana, Slovenia
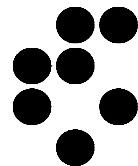
Table of contents

1 Introduction

WAHA single phase code is designed for simulations of a single phase 1D transients with pressure pulses and shocks expanding through a piping system. The code uses second-order accurate numerical scheme and is being used for testing the phenomenology of single phase pressure pulses and modelling of different elements of the piping system for further development of the WAHA two-phase code.

2 Nomenclature

A	section cross-section	***Subscripts / Superscripts***	
c	speed of sound	e	elastic pipe
C	Jacobian matrix	*pipe*	property of piping system
d	pipe wall thickness		
D	pipe diameter	***Greek letters***	
e	specific internal energy	ρ	density
E	Young's modulus	φ	vector of conservative variables
L	transformation matrix	ψ	vector of primitive variables
M	transformation matrix	ξ	vector of characteristic variables
p	pressure	Λ	eigenvalue matrix
t	time	ϕ	limiter function
v	velocity	Θ	slope detector
w	specific total energy		
x	spatial coordinate		

3 Installation

The WAHA program is being developed on Unix platform. Program is executable on Sun Ultra workstation and furthermore it is tested also on SGI Origin computer. There are two non-standard Sun calls that must be deleted before the installation is attempted on other computers and operating systems. In the `"main.f"` file the two lines with calls to `"ieee_handler"` and `"nonstandard_arithmetic"` should be commented.

WAHA code contains the following files:

- FORTRAN code files: `main.f, input_output.f, three_var.f,`
- `fluid_prop.f, models.f, sth.f`
- system files: `compile,var.h,struct.h,dim.h,sth2l.dat,`
- input file examples: `abr_ch1.dat, abr_ch2.dat, bm1.dat, tee.dat,`
- `circle.dat`

Files `"sth.f"` and `"sth2l.dat"` contain water properties from RELAP5 [9] and are thus not accessible to users without RELAP5 code. This dependency on RELAP5 will be removed when the new subroutines for water properties will be prepared.

All the files have to be put on the same directory. A subdirectory named "`results`" has to be created before running the code. The executable code named "`waha_exe`" is produced by the command:

```
compile waha_exe
```

The transient is run by the command

```
waha_exe input.dat output
```

where "`input.dat`" is the input file name and "`output`" is the name of the output file. During the calculation, the code produces output files in the subdirectory "`results`" named "`output.nnn`". where"`nnn`" is a sequence number of the output file, which is produced every time interval specified in the input file. If the name of the output file is omitted, the default name is "`output.nnn`". The output file consists of values of all the variables for the whole piping system at specific time moment. The form of data are suitable for presentation with the `GnuPlot` program. The input file name cannot be omitted.

4   Single phase model

4.1   Basic single phase flow equations

The one-dimensional single phase fluid flow can be is described with following conservative equations for density, momentum and specific total energy:

$$\partial_t \rho + \partial_x (\rho v) = 0 \tag{1}$$

$$\partial_t (\rho v) + \partial_x (\rho v^2 + p) = 0 \tag{2}$$

$$\partial_t (\rho w) + \partial_x (v(\rho w + p)) = 0 \tag{3}$$

where $w = e + v^2/2$ is a specific internal and kinetic energy together. This system of equations (1-3) can be also written in matrix notation:

$$\partial_t \begin{bmatrix} \rho \\ \rho v \\ \rho w \end{bmatrix} + \partial_x \begin{bmatrix} \rho v \\ \rho v^2 + p \\ v(\rho w + p) \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix} \tag{4}$$

If the piping system has variable cross-sections the system of equations (1-3) changes to

$$\partial_t (\rho A) + \partial_x (\rho v A) = 0 \tag{5}$$

$$\partial_t (\rho v A) + \partial_x ((\rho v^2 + p)A) = p\partial_x A \tag{6}$$

$$\partial_t (\rho w A) + \partial_x (v(\rho w + p)A) = -p\partial_t A \tag{7}$$

and in matrix notation (Toro [7]):

$$\partial_t \begin{bmatrix} \rho A \\ \rho v A \\ \rho w A \end{bmatrix} + \partial_x \begin{bmatrix} \rho v A \\ (\rho v^2 + p)A \\ v(\rho w + p)A \end{bmatrix} = \begin{bmatrix} 0 \\ p\partial_x A \\ -p\partial_t A \end{bmatrix} \tag{8}$$

4.2 Tube elasticity

In the present version of the developing numerical code the diameter of a piping system can change due to elasticity of tubes:

$$A(x,t) = A_0 + A_e(p(x,t)) \tag{9}$$

Smooth cross-section variations are not incorporated in the code yet. The pulse change the pipe diameter in accordance with relation (10) found in Martin and Wiggert [4].

$$\frac{dA}{A} = \frac{D}{d}\frac{dp}{E} \tag{10}$$

The elasticity of tubes also influences the speed of sound. In a system with rigid piping system, the expression for speed of sound (12) can be derived from the total derivative of specific internal energy $e(p,\rho)$:

$$de = \partial_p e\big|_\rho \, dp + \partial_\rho e\big|_p \, d\rho \tag{11}$$

with help of continuity (1), momentum (2) and energy (3) equations.

$$c^2 = \frac{p}{\rho^2}\partial_e p\big|_\rho - \partial_\rho p\big|_e \tag{12}$$

Following the same procedure using equations (11) and (5-7) the speed of sound in system with elastic pipes changes to :

$$\frac{1}{c^2_{pipe}} = \frac{1}{c^2} + \frac{\rho D}{Ed} \tag{13}$$

4.3 Jacobian matrix

Taking into account the modified speed of sound c_{pipe} from (6), the system of transport equation (8) in non-conservative form changes to:

$$\partial_t \begin{bmatrix} \rho \\ v \\ p \end{bmatrix} + \begin{bmatrix} v & \rho c_{pipe}^2/c^2 & 0 \\ 0 & v & 1/\rho \\ 0 & \rho c_{pipe}^2 & v \end{bmatrix} \partial_x \begin{bmatrix} \rho \\ v \\ p \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix} \tag{14}$$

or in short notation

$$\partial_t \vec{\psi} + \underline{C}\, \partial_x \vec{\psi} = 0 \tag{15}$$

The relation between primitive and conservative varibles can be written in matrix form as:

$$\begin{bmatrix} A & 0 & \rho A\, D/Ed \\ vA & \rho A & \rho vA\, D/Ed \\ wA + \rho\alpha A & \rho vA & \rho wA\, D/Ed + \rho\beta A \end{bmatrix} \delta \begin{bmatrix} \rho \\ v \\ p \end{bmatrix} = \delta \begin{bmatrix} \rho A \\ \rho vA \\ \rho wA \end{bmatrix} \tag{16}$$

where $\alpha = \partial_p e\big|_\rho dp$ and $\beta = \partial_\rho e\big|_p d\rho$. In short notation (16) can be written also as:

$$\underline{M}\, \delta\vec{\psi} = \delta\vec{\phi} \tag{17}$$

Applying transformation matrix M, the system of transport equations (8) in conservative form changes to:

$$\partial_t \vec{\phi} + \underline{C}^* \partial_x \vec{\phi} = 0 \tag{18}$$

where $\underline{C}^* = \underline{M}\,\underline{C}\,\underline{M}^{-1}$.

4.4 Characteristic form of transport equations

In order to determine efficient and stable numerical scheme, eigenvalues of the Jacobian matrix $\underline{C}$ (or $\underline{C}^*$) have to be known. The eigenvalues represent the propagation velocities of characteristic variables ξ_i, which are invariants in space and time. If the system of equations (14) is transformed into the form with characteristic variables, the Jacobian matrix $\underline{C}$ (or $\underline{C}^*$) is reduced to matrix with system eigenvalues $\underline{\Lambda}$ (25)

The transformation between primitive and characteristic variables can be written as:

$$\delta\vec{\psi} = \underline{L}\,\delta\vec{\xi} \qquad (19)$$

where $\underline{L}$:

$$\underline{L} = \begin{bmatrix} 1 & \dfrac{1}{c^2} & \dfrac{1}{c^2} \\ 0 & -\dfrac{1}{\rho c_{pipe}} & \dfrac{1}{\rho c_{pipe}} \\ 0 & 1 & 1 \end{bmatrix} \qquad (20)$$

and consequently $\underline{L}^{-1}$:

$$\underline{L}^{-1} = \begin{bmatrix} 1 & 0 & -\dfrac{1}{c^2} \\ 0 & -\dfrac{\rho c_{pipe}}{2} & \dfrac{1}{2} \\ 0 & \dfrac{\rho c_{pipe}}{2} & \dfrac{1}{2} \end{bmatrix} \qquad (21)$$

The columns of matrix $\underline{L}$ are the right eigenvectors and rows of matrix $\underline{L}^{-1}$ are the left eigenvectors of matrix $\underline{C}$. Applying the transformation (19) to (14) :

$$\underline{L}\partial_t\vec{\xi} + \underline{C}\,\underline{L}\partial_x\vec{\xi} = 0 \qquad (22)$$

and further

$$\partial_t\vec{\xi} + \left(\underline{L}^{-1}\underline{C}\,\underline{L}\right)\partial_x\vec{\xi} = 0 \qquad (23)$$

the system of equation (14) is translated to its characteristic form

$$\partial_t\vec{\xi} + \underline{\Lambda}\partial_x\vec{\xi} = 0 \qquad (24)$$

where $\underline{\Lambda} = \underline{L}^{-1}\underline{C}\,\underline{L}$. The result of matrix $\underline{C}$ diagonalization is the matrix $\underline{\Lambda}$ which consist only from system eigenvalues on its diagonal:

$$\underline{\Lambda} = \begin{bmatrix} v & 0 & 0 \\ 0 & -c_{pipe}+v & 0 \\ 0 & 0 & c_{pipe}+v \end{bmatrix} \qquad (25)$$

4.5 Numerical scheme

The second order accurate TVD scheme, developed and successfuly practiced by Tiselj & Petelin [2,3] was taken as a foundation for equations discretization. The method uses limiters to control numerical diffusion of the upwind scheme and numerical dispersion introduced by the Lax-Wendroff mathod.

In order to calculated limiters the equations have to discretized in its characteristic form:

$$\frac{\vec{\xi}_j^{n+1} - \vec{\xi}_j^n}{\Delta t} + \left(\underline{\Lambda}^{++}\right)_{j-1/2}^n \frac{\vec{\xi}_j^n - \vec{\xi}_{j-1}^n}{\Delta x} + \left(\underline{\Lambda}^{--}\right)_{j+1/2}^n \frac{\vec{\xi}_{j+1}^n - \vec{\xi}_j^n}{\Delta x} = 0 \tag{26}$$

Using a form of Steger-Warming decomposition [6], the elements of the diagonal matrices $\underline{\Lambda}^{++}$ and $\underline{\Lambda}^{--}$ are

$$\lambda_k^{++} = \max\left(0, \lambda_k\right) + \frac{\phi_k \lambda_k}{2}\left(\lambda_k \frac{\Delta t}{\Delta x} - \frac{\lambda_k}{|\lambda_k|}\right) \tag{27}$$

and

$$\lambda_k^{--} = \min\left(0, \lambda_k\right) - \frac{\phi_k \lambda_k}{2}\left(\lambda_k \frac{\Delta t}{\Delta x} - \frac{\lambda_k}{|\lambda_k|}\right) \tag{28}$$

where Δt is a timestep and Δx a spatial discretization distance and $k=1,3$. The ϕ_k is a limiter of the second order scheme, which is calculated as

$$\phi_k = \max\left(0, \min\left(\Theta_k, 1\right)\right) \qquad \text{for MINMOD limiter} \tag{29}$$

$$\phi_k = \left(|\Theta_k| + \Theta_k\right) / \left(1 + |\Theta_k|\right) \qquad \text{for van Leer limiter} \tag{30}$$

$$\phi_k = \max\left(0, \min\left(2\Theta_k, 1\right), \min\left(\Theta_k, 2\right)\right) \quad \text{SUPERBEE limiter} \tag{31}$$

The limiter ϕ_k is calculated of each cell boundary. In order to detect a slope of simulated variable ξ_i the following expression was used:

$$\Theta_k = \frac{\xi_{k,j+1-m} - \xi_{k,j-m}}{\xi_{k,j+1} - \xi_{k,j}}, \quad m = \frac{\lambda_{k,j+1/2}}{|\lambda_{k,j+1/2}|}, \quad k = 1..3 \tag{32}$$

As the calculations were carried out with primitive variables (14), the system of equations (26) has to be translated back into the form with primitive variables:

$$\vec{\Psi}_j^{n+1} = \vec{\Psi}_j^n - \frac{\Delta t}{\Delta x}\left(\underline{C}_{j-1/2}^{++}\left(\vec{\Psi}_j^n - \vec{\Psi}_{j-1}^n\right) + \underline{C}_{j+1/2}^{--}\left(\vec{\Psi}_{j+1}^n - \vec{\Psi}_j^n\right)\right) \tag{33}$$

where $\underline{C}_{j-1/2}^{++} = \underline{L}_{j-1/2}\underline{\Lambda}_{j-1/2}^{++}\underline{L}_{j-1/2}^{-1}$ and $\underline{C}_{j+1/2}^{--} = \underline{L}_{j+1/2}\underline{\Lambda}_{j+1/2}^{--}\underline{L}_{j+1/2}^{-1}$.

4.6 Numerical algorithm

The numerical algorithm for WAHA code is schematically presented in Fig. 1.
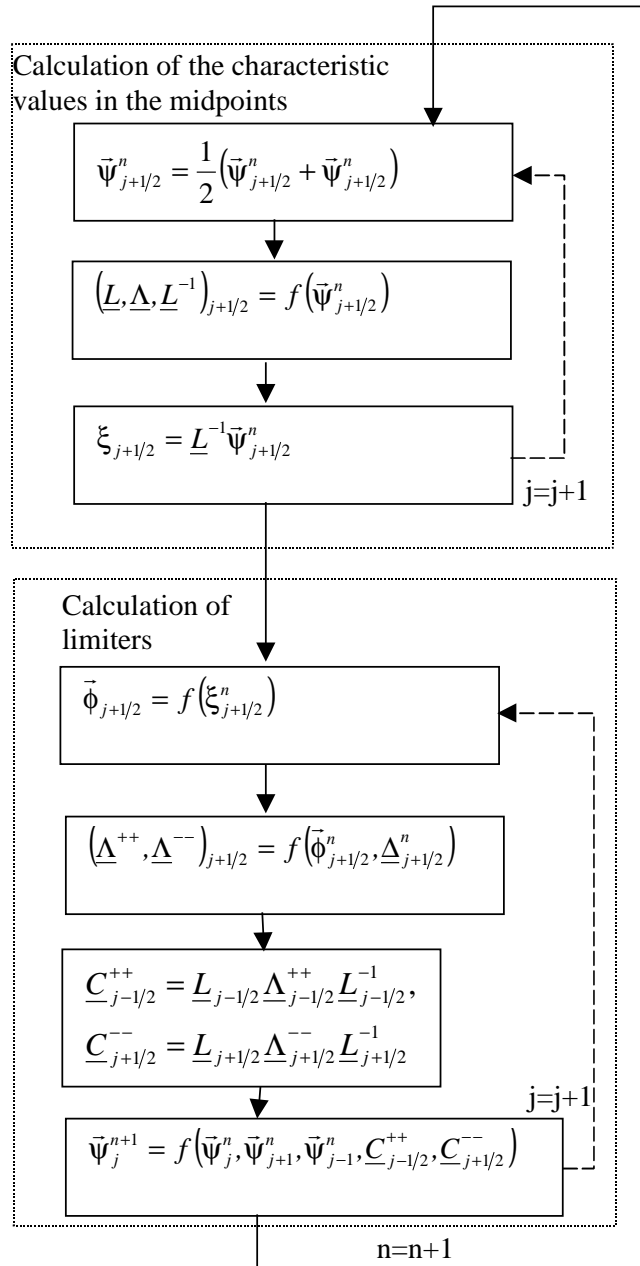

Figure 1: Numerical scheme of the WAHA code.

5 WAHA code data-structures for piping system

The WAHA code operates with two objects: **pipe** and **single volume**.

5.1 Pipe

Pipe is a basic element of the piping system. It is described by the geometric data like length, cross section at the beginning and end, wall thickness and wall elasticity as shown in Fig 2.
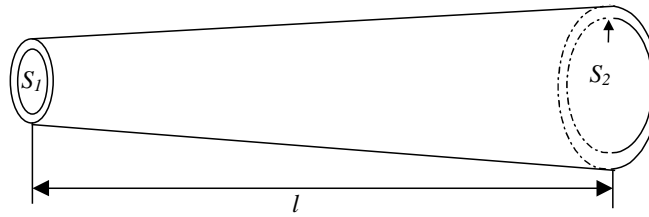

Figure 2: Geometrical description of the pipe.

5.1.1 Description of fluid state in the pipe

A single pipe is discretized in several segments i.e. volumes. The minimum number of a single pipe volumes is 1, while the maximum number of volumes is not limited. Only the maximum numbers of all the volumes in piping system is limited by the parameter NODS. Also the maximum number of pipe segments is limited with the parameter NPIPE. Both parameters are prescribed in the file "dim.h".

The number of variables necessary for the description of the fluid state is defined by parameter VAR in the file "var.h". In the WAHA single phase code, where the fluid state is described by density, velocity and pressure, the value of parameter VAR = 3.

The variables are defined in the volume centres. While the pipe is divided into *n* volumes, the length of a single volume is equal to $\Delta x = l/n$. The discretization scheme is shown in the Fig. 3.
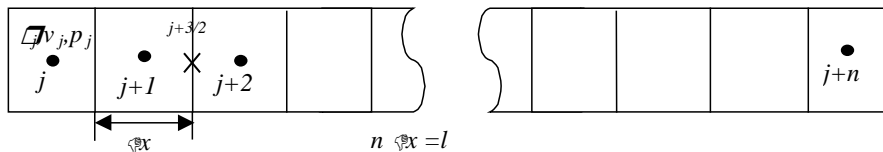

Figure 3: Discretization scheme.

5.1.2 Piping structure connections

Pipes with different cross sections can be connected together. In one point maximum three pipes (T - connection) can be joined. The Fig. 4 shows an example of a piping system.
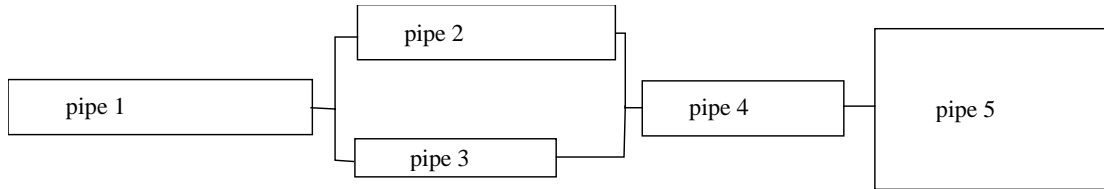

Figure 4: Scheme of the piping system.

The fluid state in all the volumes is stored in the 2D array named "`state(VAR,NODS)`". The first index is the variable name (density, velocity or pressure), whereas the second index is a volume location in the piping system.

A pipe is an individual element of the modelling. The pipe characteristics are described in file "`struct.h`". The information structure consists from the following elements:

`l`	pipe length
`s1`	cross section at the pipe beginning
`s2`	cross section at the pipe end (present version: `s1=s2`)
`elast`	elasticity of the pipe
`thick`	thickness of the pipe wall
Δx	length of the pipe volume
`i1`	position of the volume at the pipe beginning
`i2`	position of the volume at the pipe end
`bound1`	Id number of the piping element connected to the pipe beginning
`bound2`	Id number of the piping element connected to the pipe end
`bound1a`	Id number of the second piping element connected to the pipe beginning in the case of T - connection
`bound2a`	Id number of the second piping element connected to the pipe end in the case of T - connection
`b1`	orientation of the beginning connection
`b2`	orientation of the end connection
`b1a`	orientation of the connection of the second piping element at the pipe beginning in the case of T - connection
`b2a`	orientation of the connection of the second piping element at the pipe end in the case of T - connection

The fluid data for a particular pipe are located on the same place in the array. On the other hand the data for two pipes connected together are not necessary located one next to other in the array. The scheme of the array "`state(VAR,NODS)`" is presented in the Fig. 5.
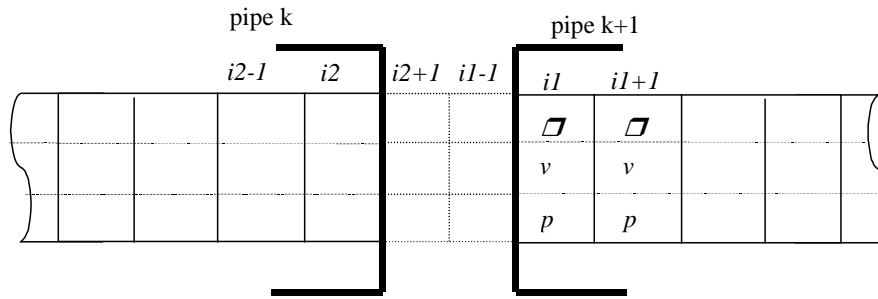
Figure 5: The structure of the array "`state(VAR,NODS)`".

The pipe characteristics are stored in the array "`pipe(NPIPE)`". The fluid data for each pipe occupy the array "`state`" from index *i1* to including *i2* (*i2-i1=n* is the number of volumes in the pipe). The "`bound1`" is the Id number of the pipe, which is connected to the pipe *k* beginning, and its value does not need to be *k-1*. Similar is valid for the "`bound1a`". The element "`b1`" is logical and is equal "`true`", if the another pipe end is connected to the pipe beginning. Otherwise, if two beginnings or two ends are connected, its value is "false". The "`bound2`" is the Id number of the pipe, which is connected to the pipe *k* end, and its value is not necessary *k+1*. Similar is valid for the "`bound2a`".Values "`b2, b1a, b2a`" have similar definition as "`b1`".

When pipe variables are located in computer memory array, two additional cells are used for each pipe end: *i*1-1 and *i*2+1. These two cells are used for the calculation of the boundary conditions (see chapter 6). In the case, when two pipes with equal cross sections are connected together (the pipe *k* end is connected to the pipe *l* beginning), the fluid state in the cell *pipe(k).i*2+1 is equal to the *pipe(l).i*1 and vice versa. The fluid state in the cell *pipe(l).i*1-1 is equal to *pipe(k).i*2. Otherwise the state in the boundary cells is calculated according to the abrupt area change model (see chapter 6.4) or prescribed with boundary conditions.

Since only one boundary cell is defined, the second order accuracy is currently not achieved on the boundaries. In the case, when the second order limiter ϕ_k needs two boundary cells, first order discretization is used.

5.2 Single volume

Single volume element is used for the boundary condition of constant pressure (e.g large water tank). It has no physical volume and in memory it occupies only one cell (Fig. 6). The single volumes are positioned after the last pipe in the piping system.
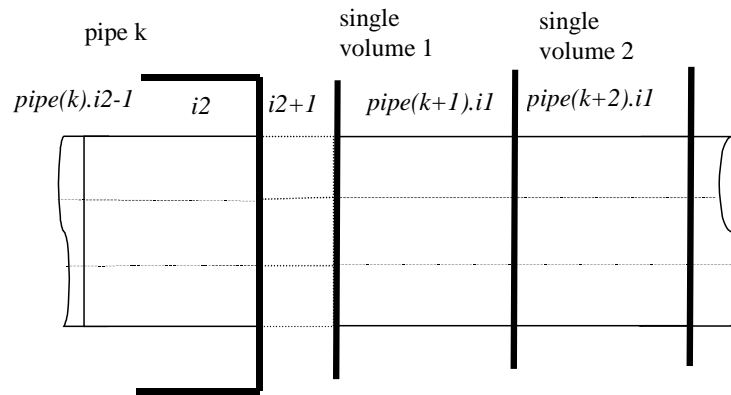
Figure 6: The single volume structure in the array `"state(VAR,NODS)"`

6 Special models applied in WAHA code

WAHA single phase program can calculate the flow of ideal gas or single phase liquid water. All models in the code are based on ideal properties of fluid. At that time no viscous or any other friction are included. Neither gravity or momentum changes due to the curved pipes are taken into account. The modelled piping is strictly one dimensional in horizontal direction.

The WAHA single phase code uses special models for the basic boundary conditions (closed pipe, open pipe), water tank, abrupt area change and T-connection.

6.1 Closed end boundary condition

The closed end boundary prescribes zero velocity on the pipe edge $v_{x=bound} = 0$. Since velocity is not defined on the pipe edge, but in the volume centre, such condition is achieved when fluid in the boundary cell has the opposite velocity as in the cell inside the pipe (fig 7):

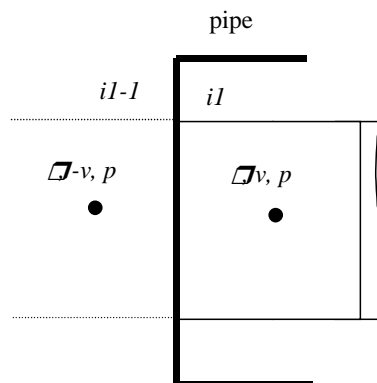$$\rho_{i1-1} = \rho_{i1}, \quad v_{i1-1} = -v_{i1}, \quad p_{i1-1} = p_{i1} \tag{34}$$

Figure 7: Closed end boundary condition.

6.2 Open end boundary condition

The open end boundary prescribes free outflow from the pipe that is achieved with the zero gradient of fluid flow variables:

$$\rho_{i1-1} = \rho_{i1}, \quad v_{i1-1} = v_{i1}, \quad p_{i1-1} = p_{i1} \tag{35}$$

Although the numerical stability was obtained with the boundary condition (35), their physical background might be questionable in the case of critical flow.

6.3 Water tank

The water tank is modelled as a infinite large vessel of fluid that can be treated as a constant pressure boundary condition. The single volume element is used to prescribe pressure and density of the water tank liquid, while the velocity is used in the same way as at the open end boundary condition (Fig. 8):

$$\rho_{i1-1} = \rho_{\text{water tank}}, \quad v_{i1-1} = v_{i1}, \quad p_{i1-1} = p_{\text{water tank}} \tag{36}$$

Although the numerical stability was obtained with the boundary condition (36), the physical background might not be completely correct for high velocities.
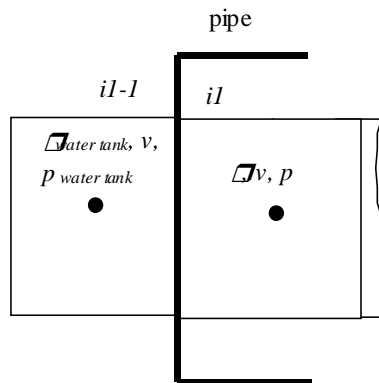

Figure 8: Water tank boundary condition on the left side of the pipe.

6.4 Abrupt area change

When two pipes with different cross sections are connected, a special model has to be applied for the calculation of the boundary state at each end of the pipe. The applied model calculates the boundary states as described by Moody [8] and is shown in Fig. 9.
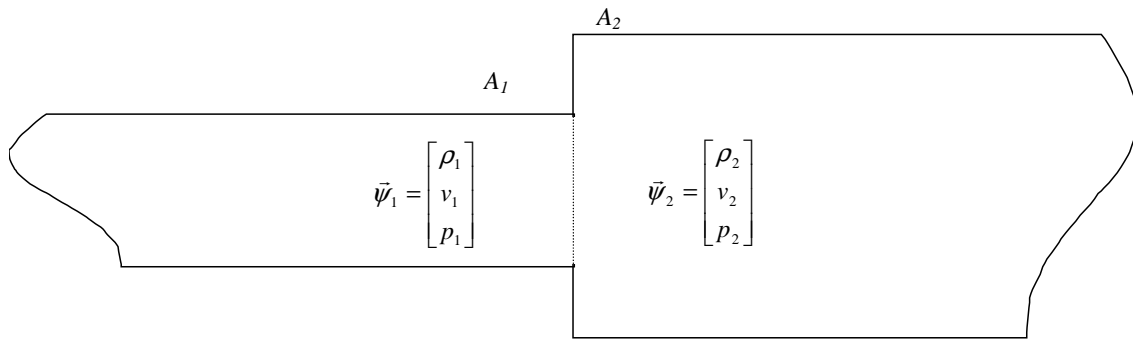
Figure 9: Abrupt area change: geometrical arrangement.

At time $t = n$ the fluid in last cell of the left pipe in Fig. 9 is at the state ψ_1, whereas the fluid in the first cell of the right pipe at state ψ_2. In the left pipe the characteristic variables ξ_1 with positive eigenvalues are known. Also, the characteristic variables ξ_2 with negative eigenvalues are known in the right pipe (39).

$$p_1 + \rho_1 c v_1 = p_1' + \rho_1 c v_1' \tag{37}$$

$$p_2 - \rho_2 c v_2 = p_2' - \rho_2 c v_2' \tag{38}$$

$$\rho_1 - \frac{p_1}{c^2} = \rho_1' - \frac{p_1'}{c^2} \tag{39}$$
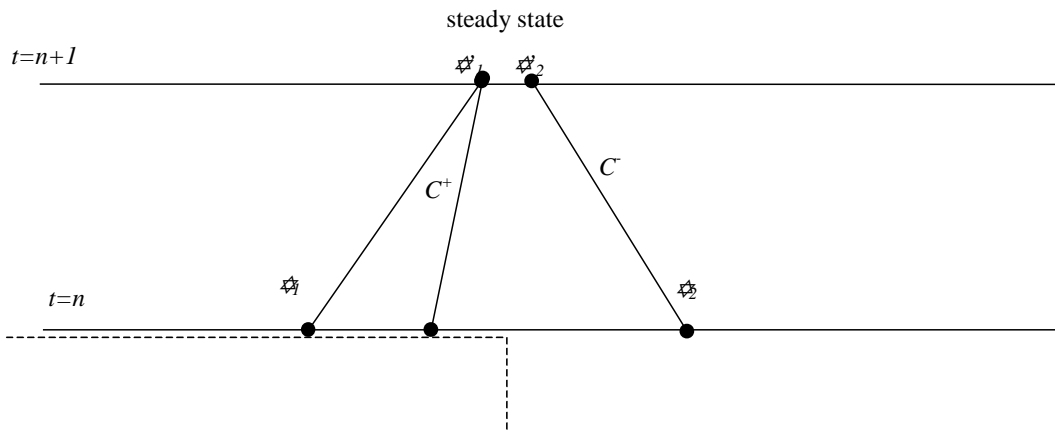
Figure 10: Abrupt area change: numerical arrangement.

The model assumes that stationary state is formed immediately at the connection and therefor the states ψ_1' and ψ_2' at time $t = n+1$ are connected by the following mass, momentum and energy conservation equations:

$$\rho_1' v_1' A_1 = \rho_2' v_2' A_2 \tag{40}$$

$$(v_2' - v_1')(v_2' + v_1') = (p_1' - p_2')\left(\frac{1}{\rho_2'} + \frac{1}{\rho_1'}\right) \tag{41}$$

$$\frac{p_1'}{\rho_1'} + \frac{1}{2}v_1'^2 + e_1' = \frac{p_2'}{\rho_2'} + \frac{1}{2}v_2'^2 + e_2' \qquad (42)$$

The unknown values ρ_1', ρ_2', p_1', p_2', v_1', v_2' are calculated numerically from the system of equation (37-42). The system of six nonlinear equations does not always give a real solution. In the case where a numerical algorithm cannot find the solution, the following simplification of eqs. (41 and 42) is used

$$\rho_1' = \rho_2' \qquad (43)$$

$$p_1' = p_2' \qquad (44)$$

Especially, when the fluid is nearly incompressible more accurate simplification can be applied changing (44) with

$$\frac{p_1'}{\rho_1'} + \frac{1}{2}v_1'^2 = \frac{p_2'}{\rho_2'} + \frac{1}{2}v_2'^2 \qquad (45)$$

The control switch No. 3 controls the model of the abrupt area change calculation:

`sw(3)=1 or less`   the equations (40), (43) and (44) are used
`sw(3)=2`   the equations (40), (43) and (45) are used
`sw(3)=3 or more`   the equations (40), (41) and (42) are used

6.5   T - connection

A T-connection model is applied when three pipes are connected in a single point (Fig 11).
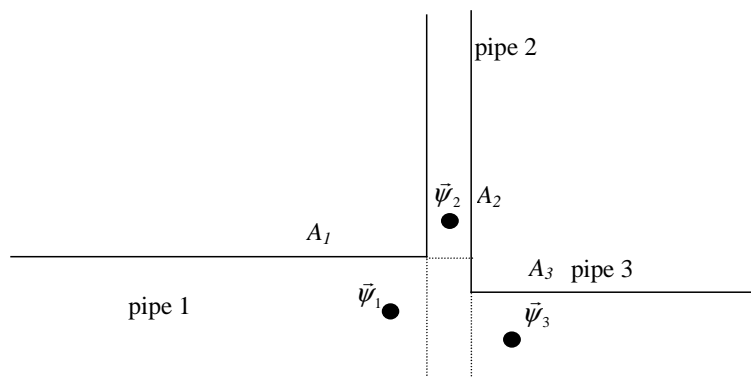

Figure 11: T-connection geometrical arrangement

In the T-connection model the main pipe is first defined. The fluid that flows through the main pipe splits to two streams through the connected pipes.

The main pipe is defined as a pipe with the largest mass flow:

$$\phi_{main} = \max\left(\phi_1, \phi_2, \phi_3\right) \tag{46}$$

where ϕ_i is the mass flow in the connected pipe. The state in the other two connected pipes is averaged. In the case when the main pipe is pipe has index $i = 1$, the averaged state is

$$\rho_{av} = \frac{\rho_2 A_2 + \rho_3 A_3}{A_2 + A_3} \tag{47}$$

$$v_{av} = \frac{\rho_2 v_2 A_2 + \rho_3 v_3 A_3}{\rho_{av}\left(A_2 + A_3\right)} \tag{48}$$

$$p_{av} = \frac{p_2 A_2 + p_3 A_3}{A_2 + A_3} \tag{49}$$

The boundary value of the pipe is then calculated in the same was as in the abrupt area change model in Fig. 9 with $\vec{\psi}_1 \leftarrow \vec{\psi}_{main}$, $\vec{\psi}_2 \leftarrow \vec{\psi}_{av}$ and $A_2 \leftarrow (A_2 + A_3)$.

7 Input/output

7.1   Input file

The input data of WAHA single phase code is an ASCII file that summarises the geometrical and physical characteristics of the piping structures as well as time characteristic of the calculation algorithm. The basic element of the input file and the calculation procedure is a card.

7.1.1 Time step card

Time step cards include time parameters and other related parameters for the calculation. It consists from six words:

W1   time at the beginning of calculation
W2   time at the end of calculation
W3   time sequence for major outputs
W4   time sequence for minor outputs
W5   time step determination. It is written in units of Courant number (for value larger than 1.0 the calculation becomes unstable, recommended value 0.9)
W6   type of fluid in piping system (integer): 1- liquid water, otherwise – ideal gas
W7   accuracy of the calculation (integer): 2 – second order, otherwise – first order
W8   model for an abrupt area changes and T-connections: 1 – mass conservation, 2 – mass and momentum conservation, 3 – mass, momentum and energy conservation (see section 6.4)

7.1.2 Minor output card

Minor output card is not necessary. When it exists, it must begin with two dummy lines, where the first one begins with "*".

W1   Id number of the pipe

W2   Id of the volume in the pipe

W3   the variable for which the time history is followed: 1 – density, 2 – velocity, 3 - pressure

7.1.3 Pipe card

Pipe cards includes physical and geometrical characteristics of the piping elements. It consists of two cards.

The first card has six words
W1   Id number of the piping element (integer). W1>0

W2   Id number of the adjacent pipe connected to the beginning of the pipe being specified. If W2=0 the pipe beginning is a closed end, if W2=-1, the pipe beginning is open

W3   orientation of the beginning connection: W3=1 - the beginning of the pipe is connected to another pipe's beginning, W3=99 – the beginning of the pipe is connected to the another pipe's end.

W4   Id number of the another pipe connected to the end of the pipe being specified. If W4=0 the pipe end is closed end, if W4=-1, the pipe's end is open.

W5   orientation of the pipe end connection: W5=1 – the end of the pipe is connected to the another pipe's beginning, W5 = 99 – the end of the pipe is connected to another pipe's end.

W6   number of the nodalization volumes of the pipe. If W1>0 the element is a pipe, if W1=0 the piping element is a single volume used for constant pressure boundary.

The second card has eight words
W1   length of the pipe (m)

W2   elasticity of the pipe (N/m^2) – if zero, pipe elasticity is not considered

W3   width of the pipe (m)

W4   cross section of the pipe beginning (m^2)

W5   cross section of the pipe end (m^2)

W6   density of the fluid in the pipe (kg/m^3)

W7   velocity of the fluid in the pipe (m/s)

W8   pressure of the fluid in the pipe (Pa)

7.1.4 Examples of the input cards

Shock in the abrupt area change

```
shock tube
--------time constants-------------------------------
beg    end         print    diff    fluid   order   abr_model
0     2.0e-2       2e-4     0.99     0       2       3
------------pipes   -----------------------------
No   from    beg/end  to  beg/end  nods
1     0         00    2      01     100
length  elsast  thick  b_area    e_area    dens    veloc   press
 1.0-0   0.0    0.0  1.0e-2   1.0e-2   1.92e-0   0.0   2.0e5
---------------------------------------------------
No   from    beg/end  to  beg/end  nods
2     1         99    3      01     100
length  elsast  thick  b_area    e_area    dens    veloc   press
 1.0-0   0.0    0.0  1.0e-2   1.0e-2   1.92e-0   0.0   2.0e5
---------------------------------------------------
No   from    beg/end  to  beg/end  nods
3     2         99    4      01     100
length   elast thick b_area   e_area     dens     veloc   press
1.0e-0    0.0   0.0 1.0e-2    1.0e-2   1.347e-0   0.0    1.0e5
---------------------------------------------------
No   from    beg/end  to  beg/end  nods
4     3         99    5      01     100
length   elast thick b_area   e_area     dens     veloc   press
1.0e-0    0.0   0.0 1.0e-2    1.0e-2   1.347e-0   0.0    1.0e5
---------------------------------------------------
No   from    beg/end  to  beg/end  nods
5     4         99    0      00     100
length   elast thick b_area   e_area     dens     veloc   press
1.0e-0    0.0   0.0 1.0e-2    1.0e-2   1.347e-0   0.0    1.0e5
******************************************************
```

Water hammer at the tee connection

```
bm1 test
--------time constants-----------------------------
beg    end      maj_out  min_out  diff   fluid  order  abr_model
0     3.5e-2   3.0e-4   3.0e-5   0.83    1      2         1
*-------minor print out---------------------------
pipe    volume   variable
1        90        3
3         2        2
2        10        3
-------------pipes  -------------------------------
No    from    beg/end  to  beg/end    nods
1     4        00     2      00        100
length   elast thick  b_area    e_area   dens      veloc   press
10.0-0    0.0    0.0   1.0e-1   1.0e-1   1.001918e+3   1.0   8.0e6
-------------------------------------------------------
No    from    beg/end  to  beg/end  nods
2     1        00     0      00       50
length elast thick  b_area    e_area   dens      veloc   press
5.0-0  0.0    0.0    1.0e-1   1.0e-1   1.001918e+3  0.7692   8.0e6
-------------------------------------------------------
No    from    beg/end  to  beg/end  nods
3     1        00     5      00       10
length elast thick  b_area    e_area   dens      veloc   press
1.0-0    0.0    0.0   3.0e-2    3.0e-2  1.001918e+3  0.7692   8.0e6
-------------------------------------------------------
No    from    beg/end  to  beg/end  nods
4     0        0      1      00       0
length   elast   thick  b_area  e_area    dens      veloc   press
0.0e-0    0.0    0.0    2.0e-1  2.0e-1   1.001918e+3   0.0    8.0e6
-------------------------------------------------------
No    from    beg/end  to  beg/end  nods
5     3        0      0      00       0
length   elast   thick  b_area  e_area    dens      veloc   press
0.0e-0    0.0    0.0    2.0e-1  2.0e-1   1.001918e+3   0.0    8.0e6
*************************************************************
```

7.2 Output file

7.2.1 Major output file

The major output file is written periodically as it is specified in the input file. Every time a new major output file "output_filename.xxx" is written, where the "xxx" marks the Id number of an output file. The file is in ASCII format, where the fluid state of the whole piping system is written in 5 columns:

1 –   additive x positions of the point in the pipe (additive means that the new pipe begins at position, where the previous ends),
2 –   density,
3 –   velocity,
4 –   pressure,
5 –   speed of sound.

The format of the output file is suited for presenting the results with the Gnuplot program, where a direct presentation of the time sequence in the piping system is also possible.

7.2.2 Minor output file

The minor output file is named "output_filename.out" and includes the time history of variables which are specified in the input file. It is possible to specify up to 10 variables. The file is updated periodically as it is also specified in the input file. The file is in ASCII format and has a several columns. The first column is time, whereas in the other columns are values of specified variables.

8 Simulation results

8.1 Shock tube with the abrupt area change

In this case five pipes with two different cross sections are connected as shown in Fig. 11. Each pipe is 1m long and divided in 100 volumes.
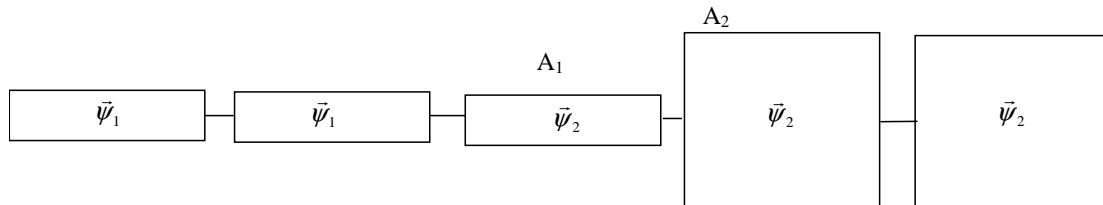

Figure 12: Shock tube with an abrupt area change.

The state in the left two pipes is ρ_1=1.92 kg/m^3, v_1=0 kg/m^3, p_1=2bar, whereas in the right three pipes ρ_2=1.347kg/m^3, v_2=0 m/s, p_2=1bar is prescribed. Due to the pressure difference

between the 2nd and 3rd pipe two pressure pulses starts to expand upwards and downwards the piping system. At the abrupt area change with the cross section ratio $A_1/A_2=1\text{cm}^2/5\text{cm}^2$ part of it is reflected and the rest is transmitted further.

Figs. 13-15 show the pressure, velocity and density in the piping system at the time moment t=0.005sec, calculated with 2 different numerical accuracy (second-order accuracy, first-order accuracy) and with RELAP5 computer code [9]. Both calculations were done with the most accurate abrupt area change model.
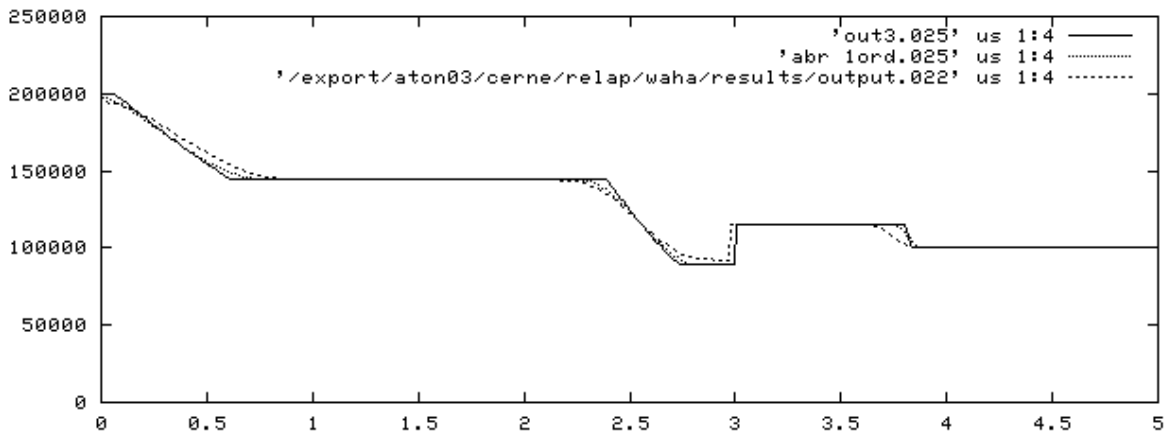
Figure 13: Reflection of the pressure pulse on the abrupt area change with different numerical accuracies; pressure distribution.
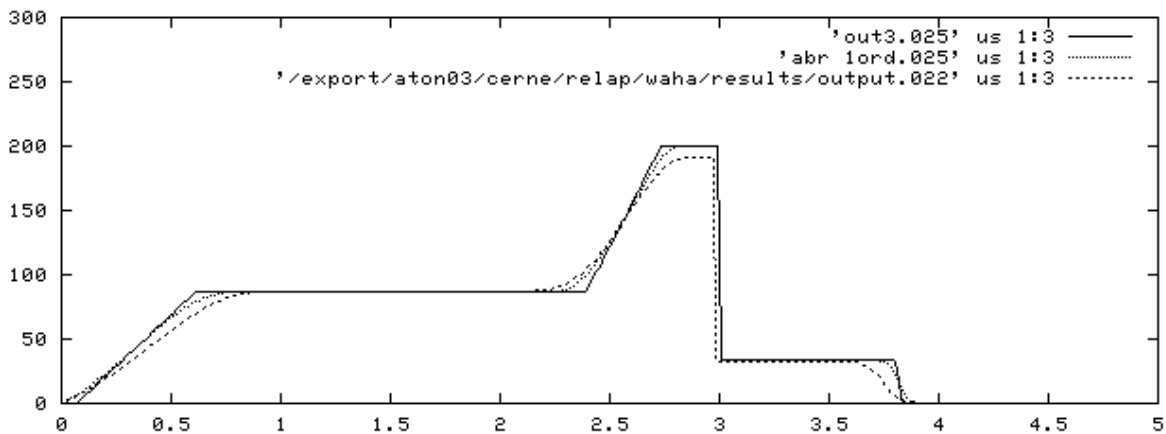

Figure 14: Reflection of the pressure pulse on the abrupt area change with different numerical accuracies; velocity distribution.
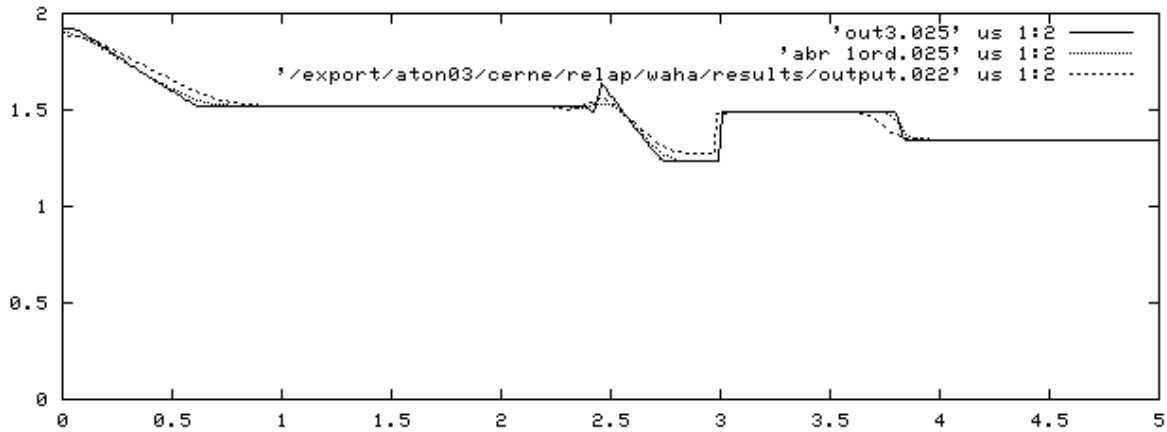
Figure 15: Reflection of the pressure pulse on the abrupt area change with different numerical accuracies; density distribution.

Figs. 16-18 show performance of three different abrupt area change models (out3 - the most accurate, out1 - the least accurate). Pressure, velocity and density distributions are presented at *t*=0.005sec.

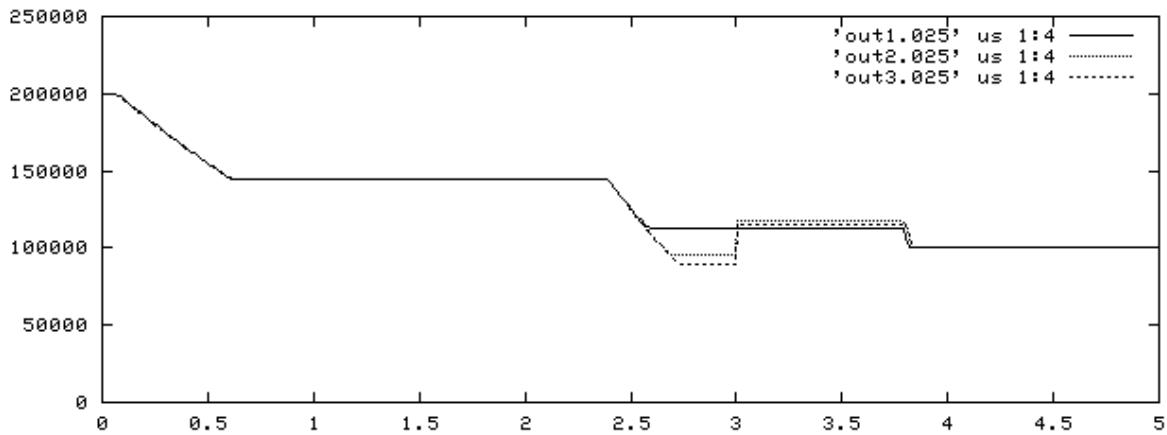**Figure 16:** Reflection of the pressure pulse on an abrupt area change calculated with different models; pressure distribution.
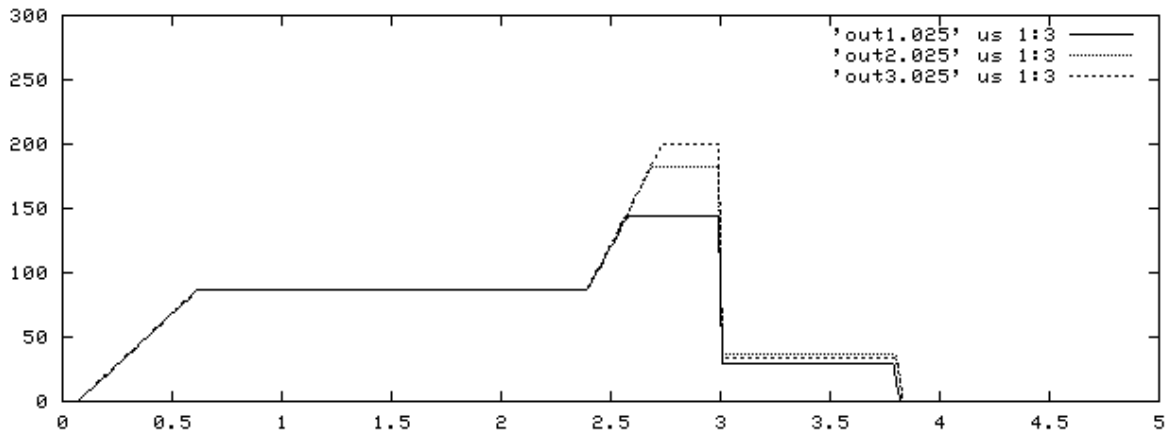
Figure 17: Reflection of the pressure pulse on an abrupt area change calculated with different models; velocity distribution.
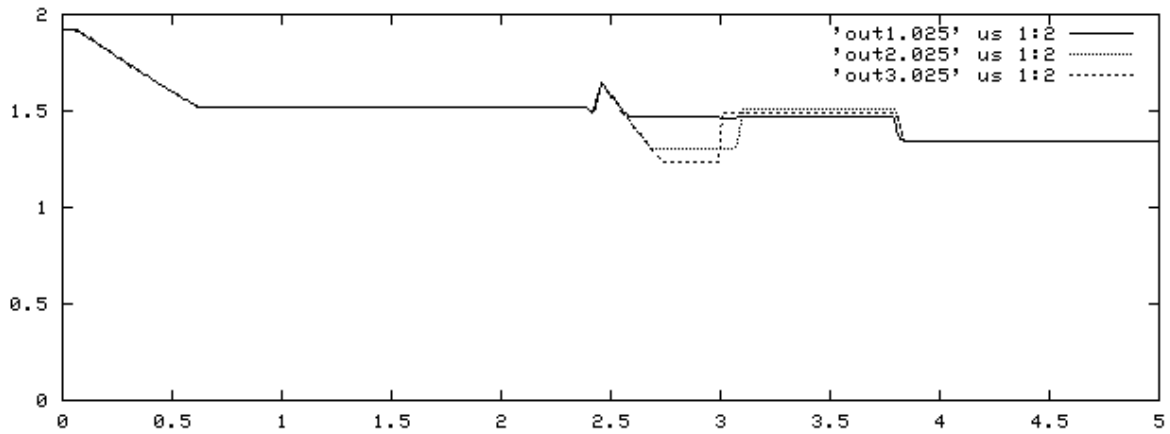

Figure 18: Reflection of the pressure pulse on an abrupt area change calculated with different models; density distribution.

8.2 Pressure pulse through the tee connection

This case shows behaviour of the pressure pulse through the T-connection (see input file in section 7.1.4). Three different pipes are connected in the same point as shown in Fig. 19.
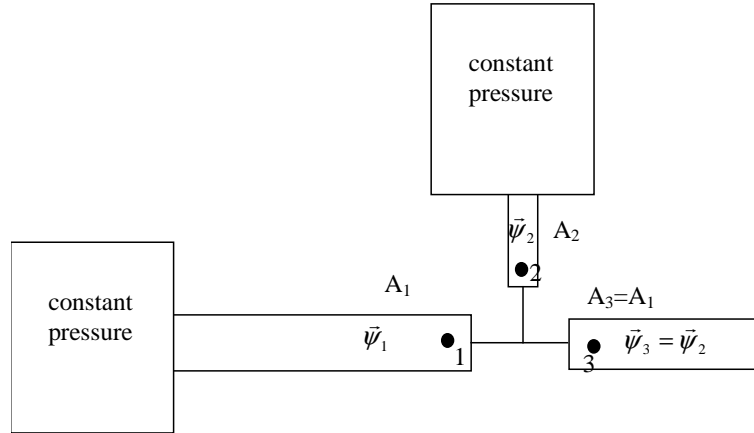

Figure 19: Model of T-connection.

The left pipe is 10m long and its left end is connected to a large water tank modelled as a constant pressure boundary condition. The upper pipe is 1m and the right pipe is 5m long. The left and the right pipe have a same cross-section $A=0.1$ m^2, while the upper pipe has a cross-section $A_2=0.03$ m^2. The following stationary state is applied initially: $\rho_1=\rho_2=\rho_3=1001.9$kg/m^3, $v_1=1$m/s, $v_2=v_3=0.769$m/s and $p_1=p_2=p_3=80$bar. At time $t=0$ the right end of the right pipe is closed and the pressure pulse expands upward the piping system.

Figs. 20-22 shows the time history of the pressure and velocity in the points indicated in the fig. 19.
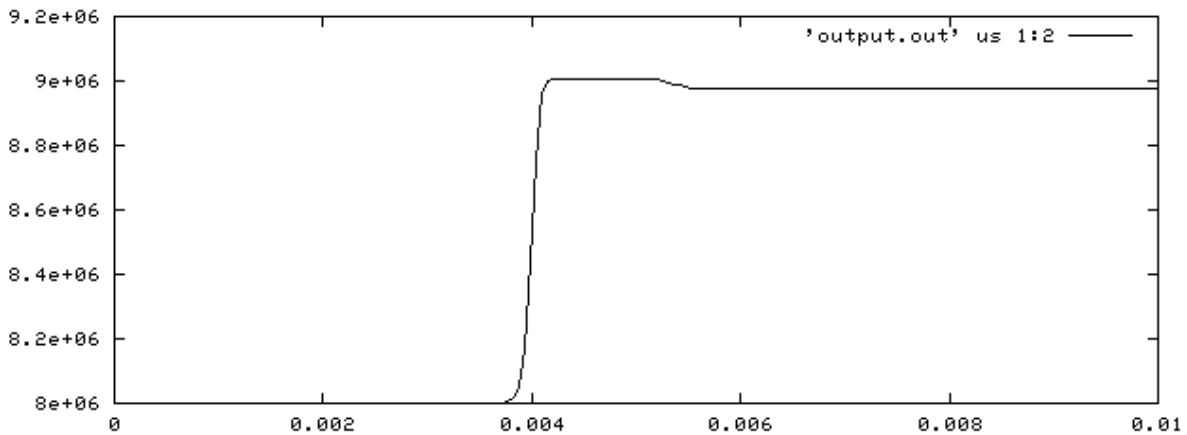

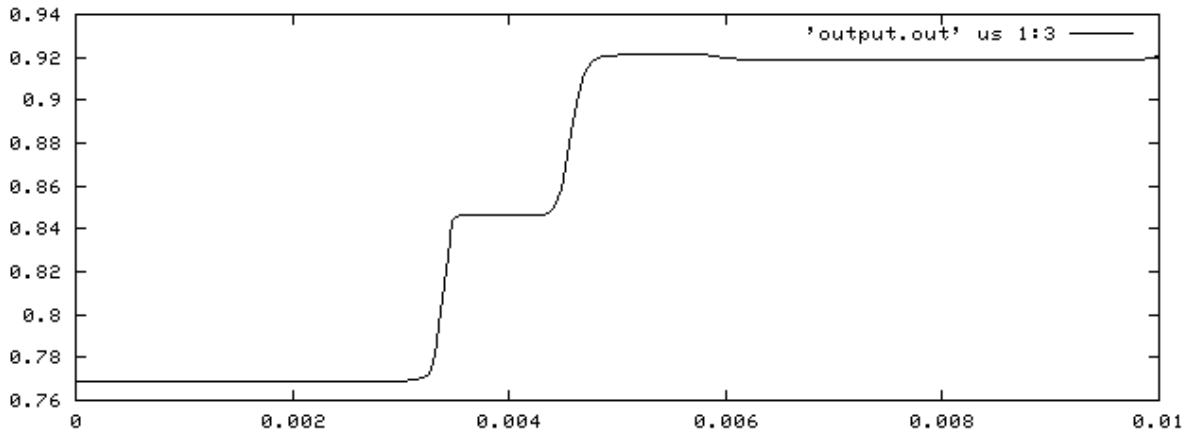**Figure 20:** Pressure versus time in the first pipe of T-connection.

Figure 21: Velocity versus time in the second pipe of T-connection.
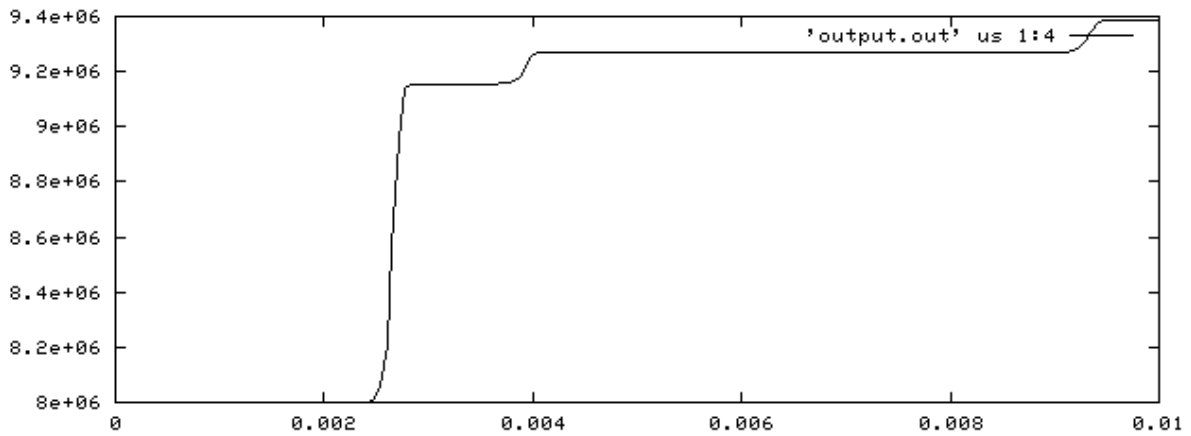

Figure 22: Pressure versus time in the third pipe of T-connection.

9 Description of files and subroutines

9.1 Files of the WAHA single phase code

dim.h   — "include file" with parameters for the memory size allocation:

NPIPE=100 – maksimum number of pipes,
NODS=10000 - maximum number of volumes,
TIME=6 – number of time parameters of the calculation, which describe:

t_par(1) – beginning of the transient
t_par(2) – end of the transient
t_par(3) – major print output time interval
t_par(4) – ratio of the time step to the Courant number
t_par(5) – time step
t_par(6) – minor print output time interval

SWNO=2 – number of switch parameters describing special characteristics of the calculation:

sw(1) – name of the fluid (water or ideal gas)
sw(2) – accuracy order of the calculation (first order or second order)
sw(3) – type of the abrupt area change model

fluid_prop.f   -FORTRAN code file, which includes subroutines for the calculation of the fluid properties: sound_speed int_energy, and subroutines for the water properties calculation (suf, scf, surft, spf, srf, scf, surgt, spg, srg, sug, scg, stanjef, stanjeg, stsat)

input_output.f   -FORTRAN code file, which includes subroutines for reading the input file and writing the output results. The subroutines are: initial_state, wrt_state, wrt_time, get_ekst, axial_force

main.f   -FORTRAN code, which beside the main program includes also subroutines for space integration, second order numerical scheme and mathematical operations between matrices and vectors. These subroutines are: t_integr7, connect_left, connect_right, lambda, limiter, bbb, averaging, mat_vec_mult

models.f   -FORTRAN code file, which includes subroutines for the abrupt area change model. These subroutines are: area_change, approx_ststate, calc_shock, approx_refl, funv6, funcv3 and numerical recipes subroutines (mnewt, lubksb, ludcmp, usrfun, fdjac)

sth.f   -FORTRAN code file from RELAP5 code, which includes subroutines for the calculation of the water properties

`sth2l.dat`	- RELAP5 steam tables
`struct.h`	- "`include file`" for the definition of the structure used to describe the geometrical characteristics of the piping elements
`three_var.f`	-It is a FORTRAN code file, which includes subroutines for the calculation of the eigenvectors and eigenvalues, which depend on the model. In this case the three variable model is used, which describes the fluid flow with density, velocity and pressure. This file also includes subroutines for some basic boundary conditions. The subroutines are:: `eigen`, `char_val`, `conservatrion`, `closed_end`, `bound_cond`, `tee_branch`
`var.h`	- "`include file`" for the determination of the number of variables (`VAR=3`)
`waha`	- command file for compiling of the FORTRAN code into the executable file

9.2 Other files not directly connected with the WAHA code

`film_press`	gnuplot file for presenting the time history of the pressure
`film_dens`	gnuplot file for presenting the time history of the density
`film_veloc`	gnuplot file for presenting the time history of the velocity
`plot, plot2, plot2`	gnuplot files for making graphs in *.pbm or *.ps format suitable for windows

9.3 Subroutines of the FORTRAN code file

The location of subroutines and calling dependence is shown in fig. 23.

`approx_refl` `(models.f)`	- calculates the transmitted and reflected states at an abrupt area change with the assumption of constant pressure and density across the pipe connection
`area_change` `(models.f)`	- calculates the transmitted and reflected states at an abrupt area change with the specific model, which are used as boundary conditions at the pipe connections
`averaging (main.f)`	- interpolation of state in the middle of two neighbouring mesh cells (used for the second order calculation)
`axial_force` `(input_output.f)`	- calculation of the axial force on the pipe – used for the BM1 test
`bbb (main.f)`	- multiplies three matrices: first output is a matrix with space derivatives of positive eigenvalues: $C^+=L\Lambda^+L^{-1}$ and the second output with negative eigenvalues $C^-=L\Lambda^-L^{-1}$
`bound_cond`	– boundary condition for the constant pressure – pressure and

(three_var.f)	density are determined with the single volume element, velocity gradient is zero
char_val (three_var.f)	- calculates the vector of eigenvalues
closed_end (three_var.f	- returns the boundary condition for a closed pipe
connect_left (main.f)	- calculates the left boundary condition of the pipe
connect_right (main.f)	- calculates the right boundary condition of the pipe
conservation (three_var.f)	– calculates the integral energy and mass of the piping system – it is used only for writing the screen output
eigen (three_var.f)	- calculates the matrix of eigenvectors, inverse matrix of eigenvectors and eigenvalues
fdjac (models.f)	- calculates the jacobian matrix for the derivatives of the equations used for the calculation of the abrupt area change model
funcv5 (models.f)	- calculates validity of the guessed state at the abrupt area change. If the the values of the conservative and characteristic equations are zero, the guessed state is correct. Mass and momentum equation are in conservative form, whereas the energy equation is non-conservative form
funcv6 (models.f)	- calculates validity of the guessed state at an abrupt area change. If the value of the conservative and characteristic equations are zero, the guessed state is correct. Mass, momentum and energy equations are in conservative form
get_ekst (input_output.f)	– the integer value transforms into the string for the purpose of indexing of the output file name (it is used for the file name extension)
initial_state (input_output.f)	– reads the input file specified in the parameter of the executable command, constructs the index array and the fluid state array
int_energy (fluid_prop.f)	– calculates the internal energy from the given state
lambda (main.f)	- calculates the vector of positive and a vector of negative eigenvalues according to the flux corrected transport eqs.
limiter (main.f)	- calculates the limiters for second order accuracy
lubksb (models.f)	- calculates the inverse of the Jacobian matrix needed in the abrupt area change model
ludcmp (models.f)	- calculates the inverse of the Jacobian matrix needed in the abrupt area change model
main (main.f)	- contains the main time integration loop, calls for minor and major output procedures
mat_vec_mult (main.f)	- multiplies matrix with vector
mnewt (models.f)	- calculates the value of variables, which would give the zero value of system of equations given by funcv6 or funcv5

|	
`pipe_elast` `(fluid_prop.f)`	- calculates the change of the sound speed due to the pipe elasticity
`scf (fluid_prop.f)`	- water property subroutine
`sound_speed` `(fluid_prop.f)`	- calculates the sound speed of the fluid at the given point
`stanjef` `(fluid_prop.f)`	- water property subroutine
`sth2x0 (sth.f)`	- water property subroutine from RELAP5
`sth2x6 (sth.f)`	- water property subroutine from RELAP5
`sth2xi (sth.f)`	- water property subroutine from RELAP5
`suf (fluid_prop.f)`	- water property subroutine
`t_integr (main.f)`	- calculates the fluid state in the next time step for the whole piping system
`tee_branch` `(three_var.f)`	– calculates the boundary values of the pipe connected at the T- branch
`usrfun (models.f)`	- calculates the values of the eqs. for the abrupt area change model
`wrt_state` `(input_output.f)`	– writes the total mass and energy during the calculation on the screen and in the major output files
`wrt_time` `(input_output.f)`	– writes the minor output file "`output_name.out`"
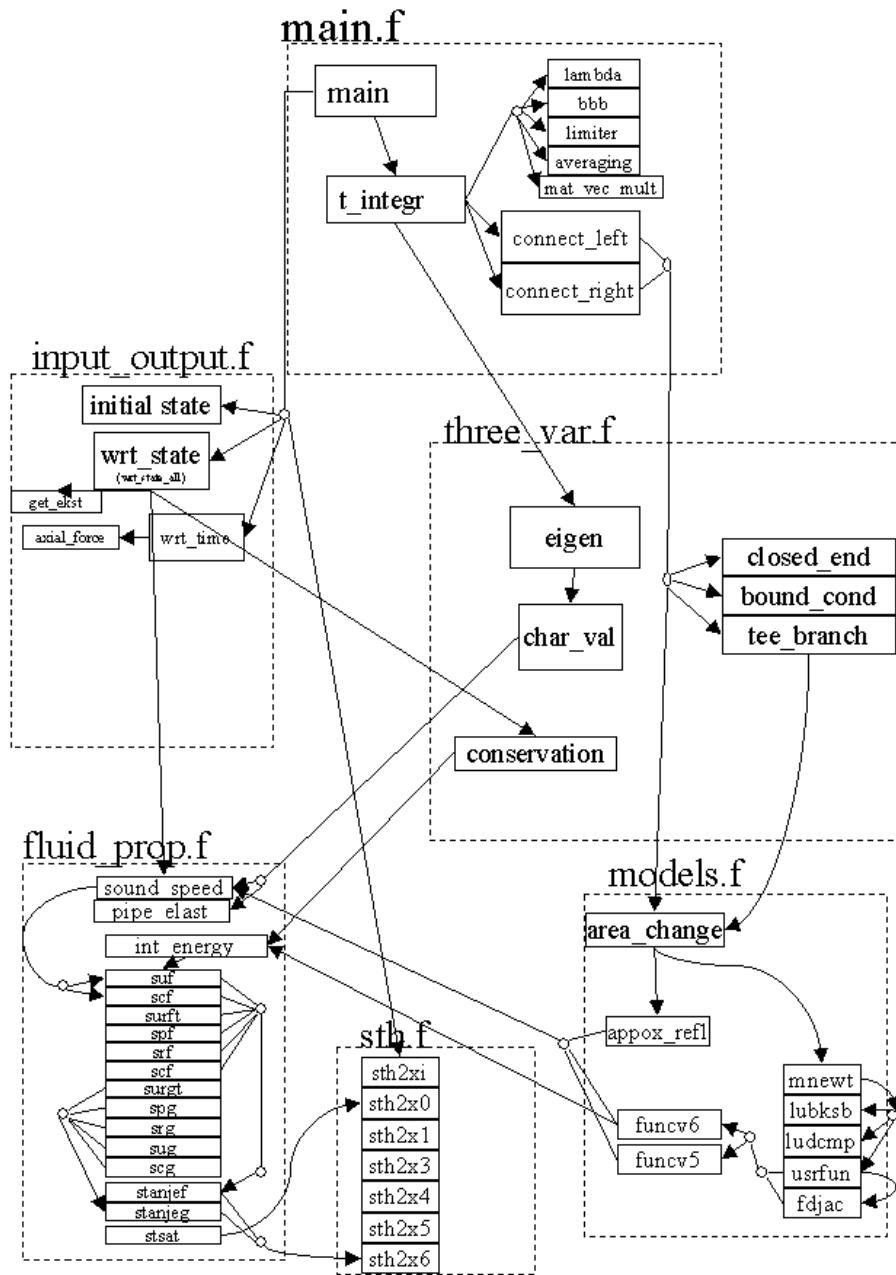
Figure 23: Call tree of subroutines in WAHA code

10 References

[1]   WAHALoads contract FIKS-CT-2000-0016, Annex I, Description of work: *Two-Phase Flow Water Hammer Transients and Induced Loads on Materials and Structures of Nuclear Power Plants*, September 2000.

[2]   I. Tiselj, S. Petelin, *Modelling of Two-Phase Flow with Second-Order Accurate Scheme*, J. Comp. Phycs, 136 (2), 1997, pp. 503-521.

[3]   Tiselj, I., Petelin. S., *First and Second-Order Accurate Schemes for Two-Fluid Models*, J. Fluids Eng. - ASME 120 (2), 1998, pp. 363-368.

[4] Martin, C.S., Wiggert, D., 1996, *Waterhammer and Fluid Structure Interaction in Piping System*, ASME Professional Development Programs, April 4.-5., 1996, p. I-8.

[5] I. Tiselj, G. Černe, I. Parzer, A. Horvat, "Numerical method of WAHA code", WAHALoads project, Deliverable D62 version Oct.2001, 2001.

[6] C. Hirsch, "Numerical computation of internal and external flow", Vol. 1-2, J.ohn Wiley & Sons, 1988.

[7] Toro, E.E., 1999, *Riemann Solvers and Numerical Methods for Fluid Dynamics*, 2nd Ed., Springer Verlag, , p.27.

[8] Moody, F. J., 1990, *Introduction of Unsteady Thermofluid Mechanics*, JohnWiley & Sons, 1990, Toronto.

[9]   RELAP5/MOD3   code   manual,   NUREG/CR-5535,   1995   (available   on http://www.nrc.gov/RES/RELAP5.