FLOODSTAND

FP7-RTD- 218532

Integrated Flooding Control and Standard for Stability and Crises Management

FLOODSTAND-deliverable:

Results of the computational study on the pressure losses in openings, air pipes and effects of ventilation

Author	Marek Kraskowski
Organisation	CTO S.A.
Revision	1.02
Deliverable No.	D2.4b

Date	18.01.2011

Document identification sheet

FLOODSTAND	**Integrated Flooding Control and Standard for Stability and Crises Management**	FP7-RTD- 218532

Title: Results of the computational study on the pressure losses in openings, air pipes and effects of ventilation	**Other report identifications:** CTO S.A. RH No. RH - 2010/T - 047E
Investigating partners: CTO, STX, NAPA **Author: Marek Kraskowski** **Reviewed by**: **Paweł Dymarski**	CTO reference: 5.167.01.221
☐ **Outline** ☐ **Draft** ☒ **Final** **Version number:** 1.02 **Revision date:** 18.01.2011 **Next version due:** **Number of pages: 31**	☒ **A deliverable** ☐ **Part of a deliverable** ☐ **Cover document for a part of a deliverable** ☐ **Deliverable cover document** ☐ **Other** **Deliverable number**: D2.4b **Work Package**: WP2 **Deliverable due at month**: 17
Accessibility: ☒ **Public** ☐ **Restricted** ☐ **Confidential (consortium only)** ☐ **Internal (accessibility defined for the final version)**	**Available from**: CTO **Distributed to**: **Disclosees when restricted:** **Comments:** k-friction & k-sum corr. on p. 31

Abstract: The report contains the results of CFD analysis of the compressible air flow through air pipes. The goal of the analyses is to evaluate the pressure loss coefficients for air flows and to check if the simplified approach elaborated by IMO for the water pipes can be also applicable for air pipes.

Acknowledgements: The research leading to these results has received funding from the European Union's Seventh Framework Programme (FP7/2007-2013) under grant agreement n° 218532. The financial support is gratefully appreciated.

EXECUTIVE SUMMARY (public)

The report contains the results of CFD analyses of the air flow through the air pipes, being the elements of the venting system of the ship compartments. The goal of the analysis is to evaluate the pressure loss coefficients for air flows and to check if the simplified approach, elaborated by IMO for the water pipes, can be also applicable for air pipes.

The presented analyses apply to the following situation: flooding of the ship's double bottom causes air compression in the compartments located far from the damage region, and the effect of air cushion appears. The air discharge through the air pipes of the compartment venting system influences the flooding rate.

The computational models are reduced to the air pipes only, with prescribed overpressure at the inlet and atmospheric pressure at the outlet. Such model allows for evaluation of the pressure loss coefficient as a function of overpressure for particular air pipes.

Two types of air pipes were considered: an air pipe with free outlet and air pipe with air cap on the outlet (the air cap closes the pipe outlet in case of water on deck). The presented results include:

- Visualization of the pressure and velocity distribution in the airpipes;
- Values of air mass flow rate for given overpressures;
- Derived quantities: speed reduction factor and pressure loss coefficient for given overpressures.

The CFD results (pressure loss coefficient) for the airpipe with free outlet are compared with the results of simplified calculation based on the IMO resolution No. MSC.245(83). This comparison shows that the simplified approach yields considerably higher values of pressure loss coefficient than CFD computations.

CENTRUM TECHNIKI OKRĘTOWEJ S.A.
SHIP DESIGN AND RESEARCH CENTRE S.A.
CTO S.A.

RESEARCH AND DEVELOPMENT DEPARTMENT
SHIP HYDROMECHANICS DIVISION

Address: 65, Szczecińska St.
80-392 Gdańsk
Poland

Phone: (48 58) 556 11 25
Fax: (48 58) 553 16 43
E-mail: modbas@cto.gda.pl

TECHNICAL REPORT

N⁰ RH-2010/T-047E

RESULTS OF THE COMPUTATIONAL STUDY ON THE PRESSURE LOSSES IN OPENINGS, AIR PIPES AND EFFECTS OF VENTILATION

Page 1
Number of pages 33
Copy N⁰

Prepared by: Marek Kraskowski, M.Sc.

Head of Ship Hydromechanics Division: Wojciech Górski, M.Sc.

Approved by: Paweł Dymarski, Ph.D.

Project Manager: Mateusz Weryk, M.Sc.

Gdańsk, August 2010

GENERAL INFORMATION

Customer: European Commission, Research Directorate – General

Acronym: FLOODSTAND

Project Title: Integrated flooding control and standard for stability and crises management

Project no.: FP7-RTD- 218532

CTO reference: 5.167.01.221

Work Package no 2: Flooding progression modelling

Task no 2.4: Computational studies & RANSE CFD

Subtask no 2.4.2: Effects of air compression

Subject of report
The report contains the results of CFD analysis of the compressible air flow through air pipes. The goal of the analyses is to evaluate the pressure loss coefficients for air flows and to check if the simplified approach elaborated by IMO for the water pipes can be also applicable for air pipes.

Keywords:

CFD, air pipes, venting, flooding…………………………………….

Distribution of the Report:

Consortium: ……. copies
CTO S.A.: …. copies

CONTENTS

1 EXECUTIVE SUMMARY 8
2 NUMERICAL MODEL USED FOR THE ANALYSES 9
3 COMPUTATIONAL CASES 10
3.1 Case 1 10
3.2 Case 2 15
4 RESULTS 20
4.1 Convergence criteria 20
4.2 Flow visualization 22
4.3 Quantitative results 28
5 CONCLUSIONS 33

1 EXECUTIVE SUMMARY

The report contains the results of CFD analyses of the air flow through the air pipes, being the elements of the venting system of the ship compartments. The goal of the analysis is to evaluate the pressure loss coefficients for air flows and to check if the simplified approach, elaborated by IMO for the water pipes, can be also applicable for air pipes.

The presented analyses apply to the following situation: flooding of the ship's double bottom causes air compression in the compartments located far from the damage region, and the effect of air cushion appears. The air discharge through the air pipes of the compartment venting system influences the flooding rate.

The computational models are reduced to the air pipes only, with prescribed overpressure at the inlet and atmospheric pressure at the outlet. Such model allows for evaluation of the pressure loss coefficient as a function of overpressure for particular air pipes.

Two types of air pipes were considered: an air pipe with free outlet and air pipe with air cap on the outlet (the air cap closes the pipe outlet in case of water on deck). The presented results include:

- Visualization of the pressure and velocity distribution in the airpipes;
- Values of air mass flow rate for given overpressures;
- Derived quantities: speed reduction factor and pressure loss coefficient for given overpressures.

The CFD results (pressure loss coefficient) for the airpipe with free outlet are compared with the results of simplified calculation based on the IMO resolution No. MSC.245(83). This comparison shows that the simplified approach yields considerably higher values of pressure loss coefficient than CFD computations.

2 NUMERICAL MODEL USED FOR THE ANALYSES

The situation reproduced by the numerical model is presented in Figure 1: flooding of the ship's double bottom through the damaged shell plating generates the air cushion in the compartments located farther from the damage region.

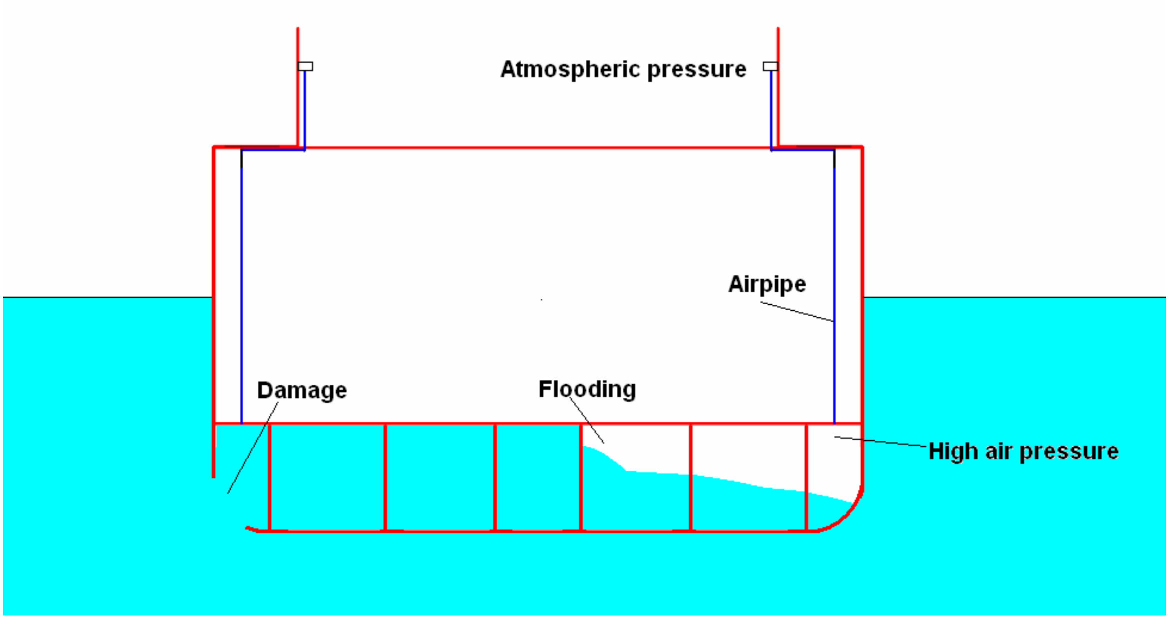


Figure 1 Situation reproduced by the CFD model

The CFD model is reduced to the air pipe and small cylindrical regions surrounding the inlet and outlet of the air pipe. The prescribed pressure at the boundaries of these cylindrical regions was applied as a boundary condition (Figure 2).

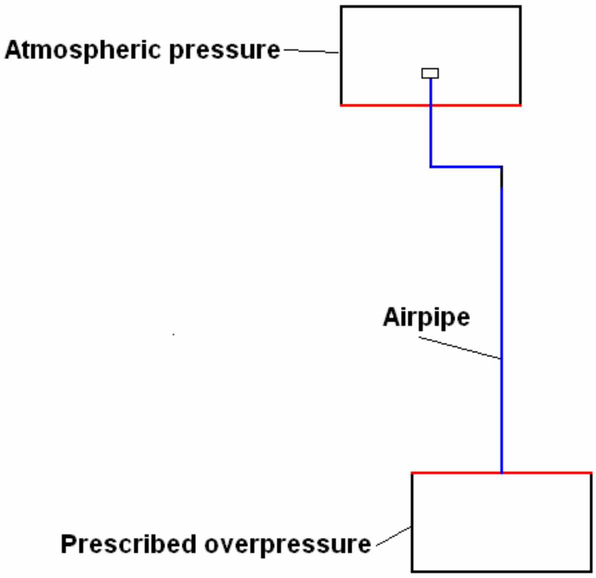


Figure 2 Numerical model

The following assumptions were done for the computations:

- The air compressibility was taken into account; an ideal gas equation was used;
- Gravity was taken into account;
- The reference pressure was set to 101325 Pa, reference temperature – to 300K;

3 COMPUTATIONAL CASES

Two computational cases were considered: air pipe with free outlet (Case 1) and air pipe with air cap at the outlet (Case 2). The documentation of the geometry of air pipes was provided by STX Europe. The CAD models were created with Rhinoceros.

3.1 Case 1

The geometry of the air pipe analyzed in Case 1 is presented in Figure 3.

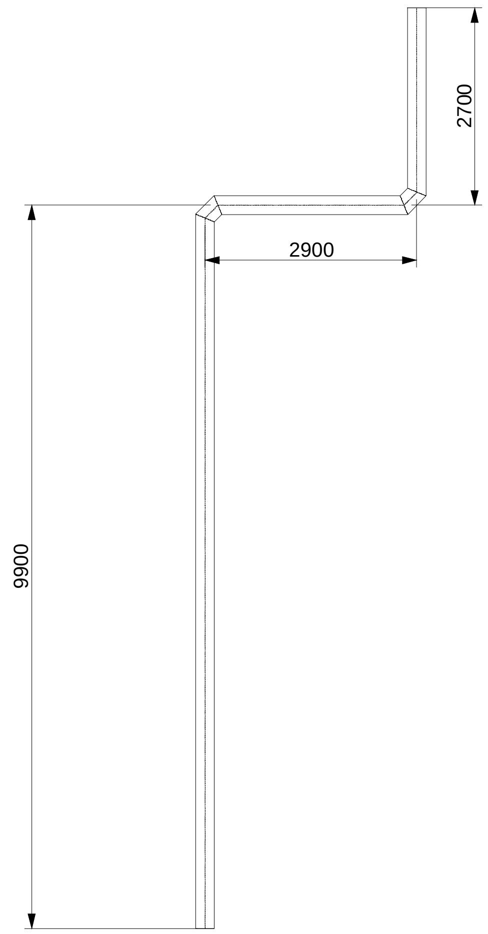


Figure 3 Case 1 – geometry of the air pipe

The air pipe includes two double mitre bends, presented in detail in Figure 4.

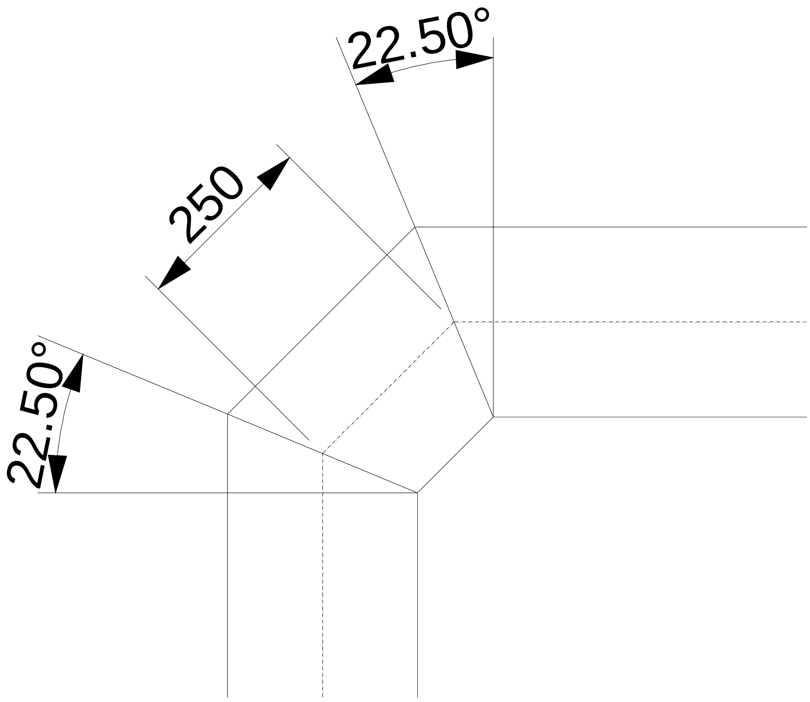


Figure 4 Double mitre bend of the air pipe in Case 1

The cross – section of the pipe is presented in Figure 5. Thickness of the pipe wall is important for correct modeling of the inlet and outlet.

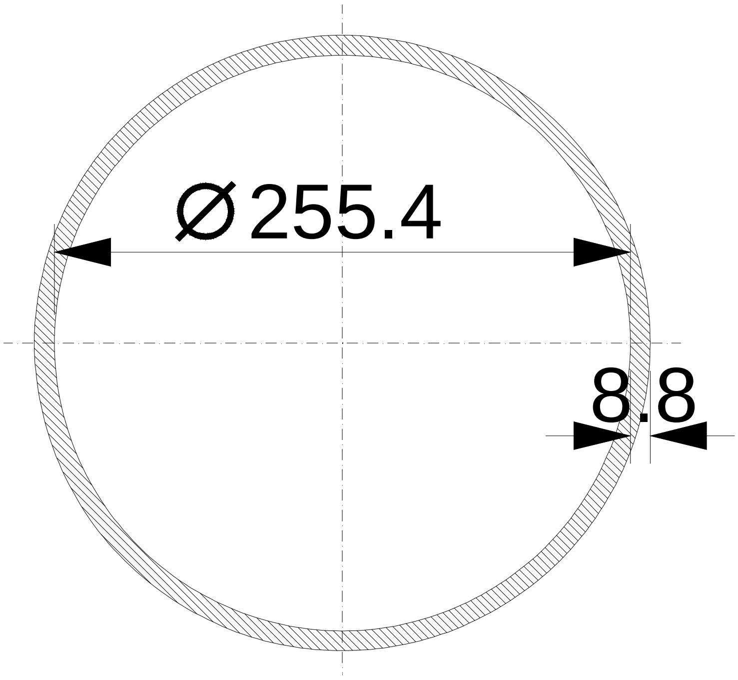


Figure 5 Cross-section of the air pipe in Case 1

Dimensions of the cylindrical zones surrounding the inlet and outlet of the pipe (see Figure 2), necessary for the CFD model, are presented in Figures 6 and 7.

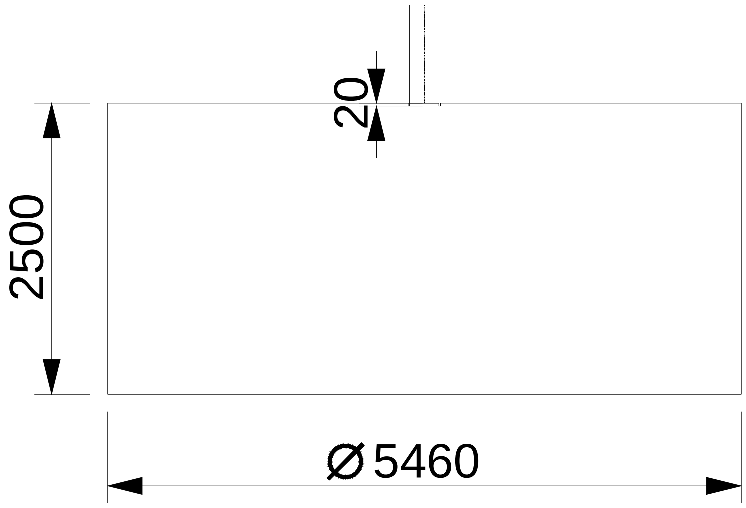


Figure 6 Cylindrical zone surrounding the air pipe inlet – Case 1

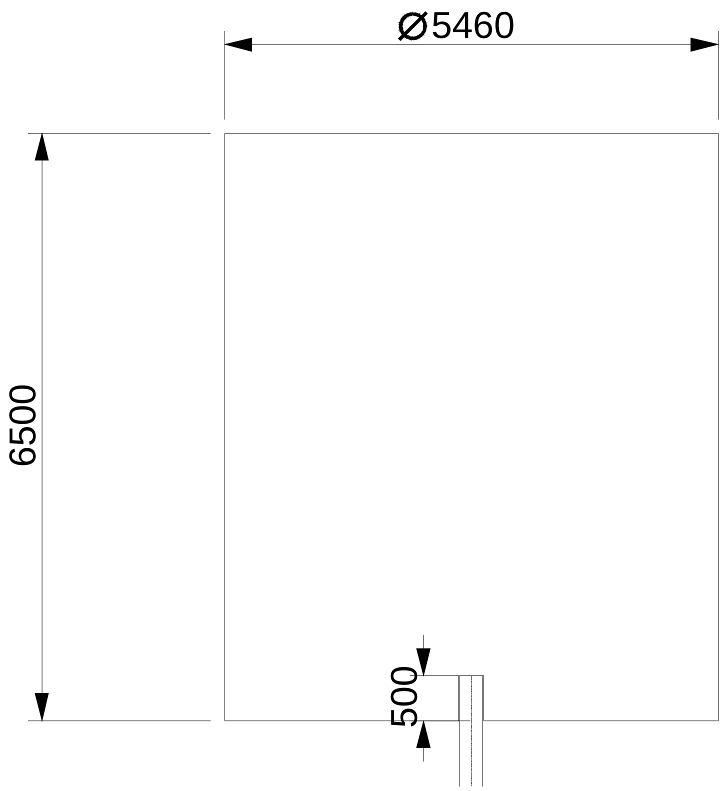


Figure 7 Cylindrical zone surrounding the air pipe outlet – Case 1

The features of the numerical mesh used for the CFD computations are listed below:

- Type of mesh: hexahedral, unstructured;
- Approximate number of cells: 800 000 (Case 1);
- Mesh generator: STAR CCM+.

Only half of the flow domain was taken into account in the computations due to symmetry. Visualization of the numerical mesh for Case 1 is presented in Figures 8-11. The following boundary conditions were applied:

– Inlet zone and outlet zone (orange surfaces in Fig.8): constant pressure;
– Symmetry plane (blue surface): zero values of normal velocity component and zero values of normal component of all gradients;
– Decks and pipe surface (gray surfaces): zero values of normal and tangential velocity components; the "wall function" was applied for the boundary layer.

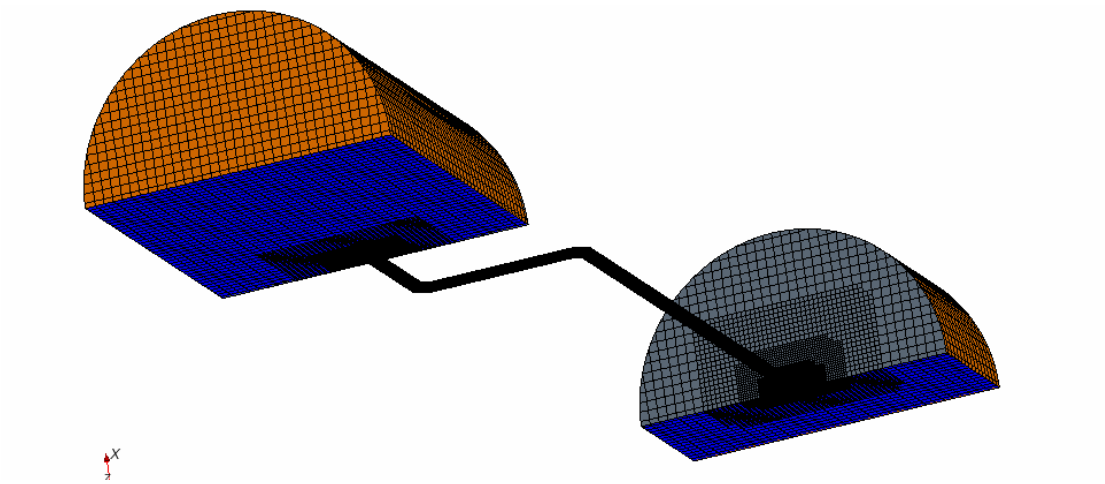


Figure 8 Computational mesh for Case 1 – overall view

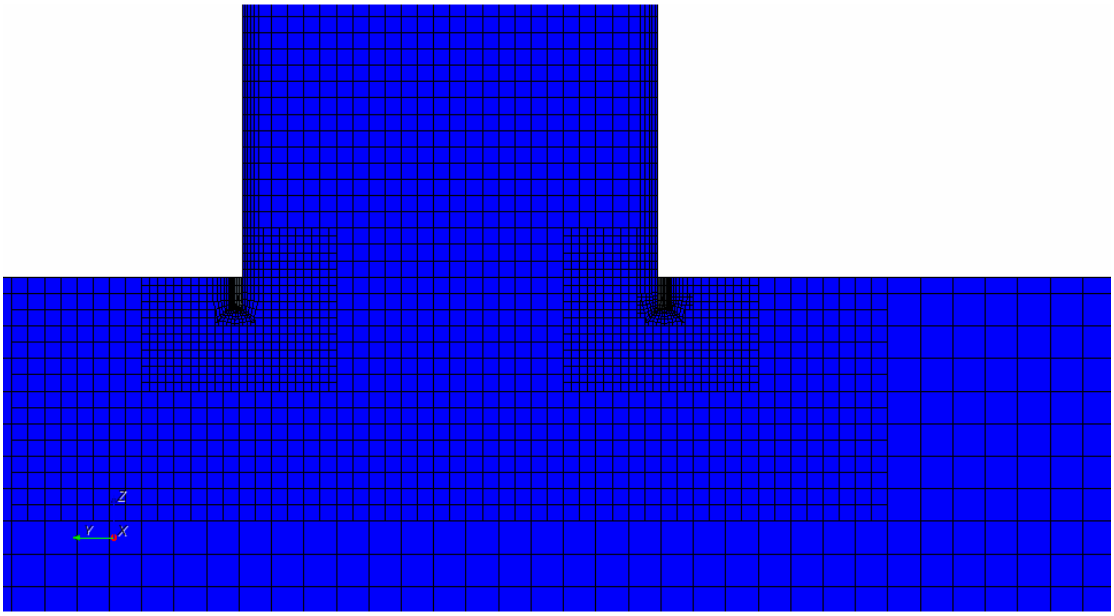


Figure 9 Computational mesh for Case 1 – mesh density at the pipe inlet region

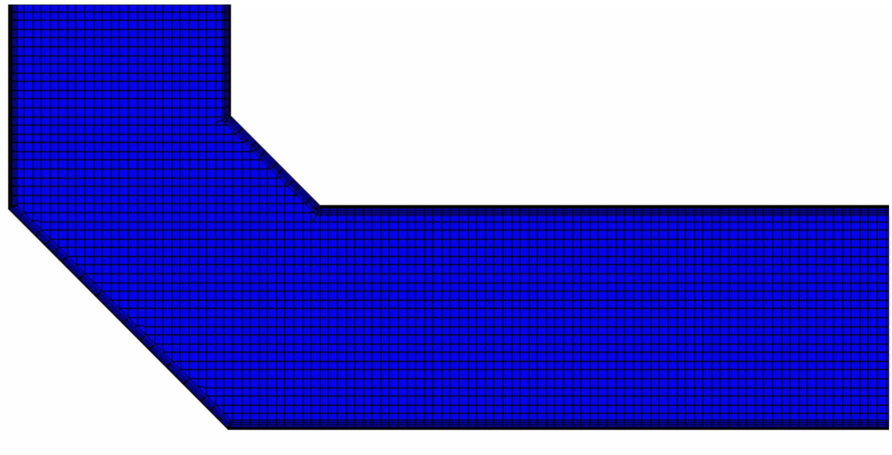


Figure 10 Computational mesh for Case 1 – mesh density at the bend

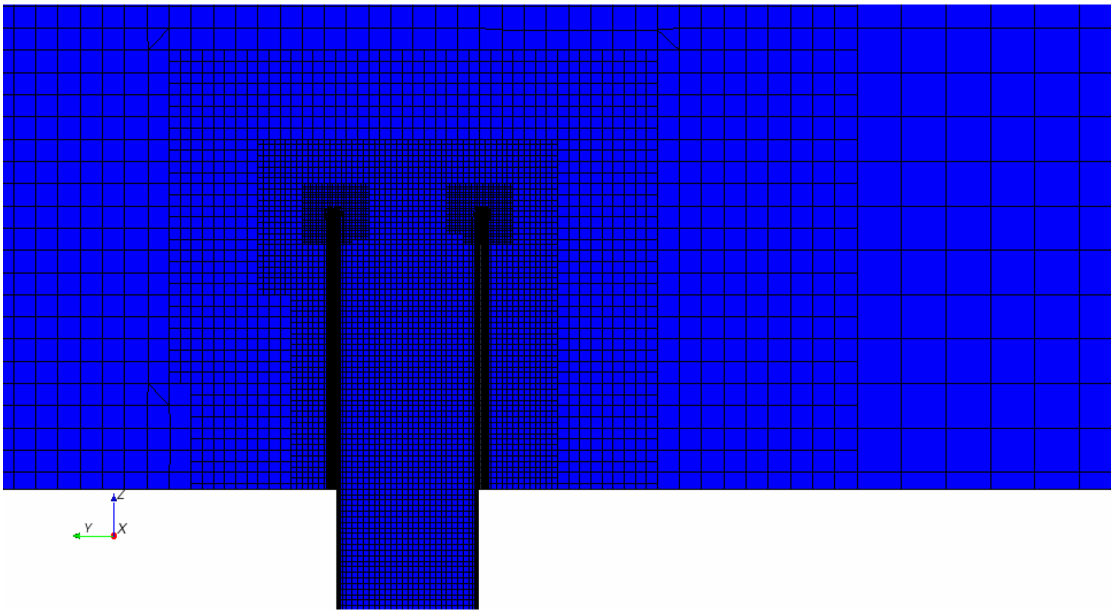


Figure 11 Computational mesh for Case 1 – mesh density at the pipe outlet region

The STAR CCM+ solver was applied. The solver settings for Case 1 were as follows:

- Flow solver type: segregated;
- Turbulence model: k-epsilon;
- The flow was solved as an unsteady one;
- Time step: dt=0.01s

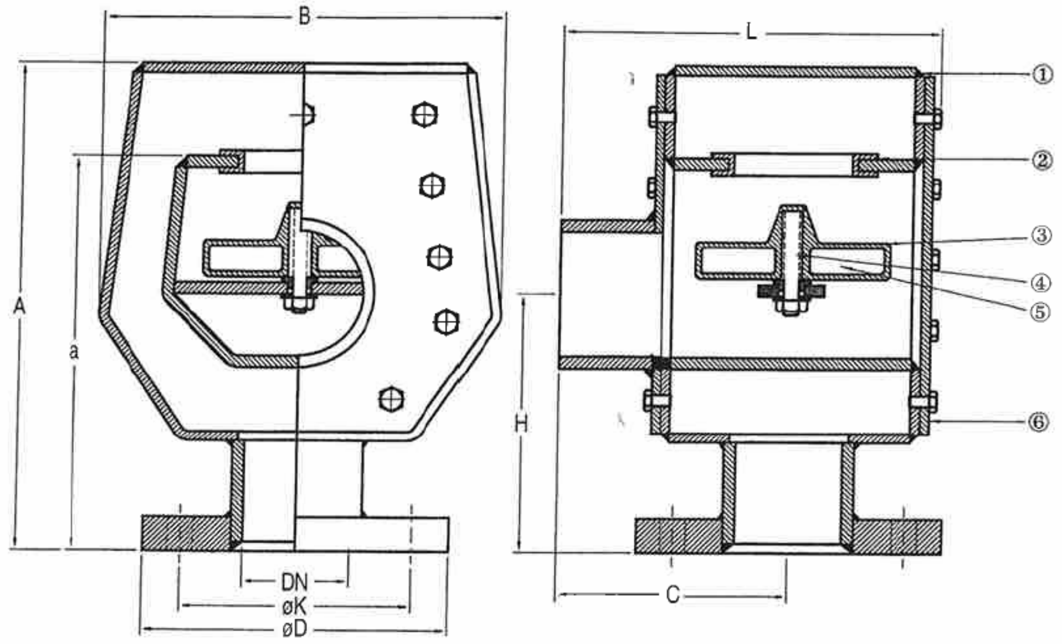
Three values of overpressure at the inlet Δp were considered: 1 kPa, 10 kPa and 20 kPa. The reasons for using these values are as follows:

- 20 kPa was considered to represent the maximum overpressure for a large tank with small air pipes;
- 1 kPa is approximately the minimum overpressure that has a noticeable effect for damage stability calculations;
- 10 kPa is roughly an average of these two values.

3.2 Case 2

In Case 2, the pipe of the same diameter as in Case 1 was used (DN250). The pipe was straight and 6800mm long, with the air cap mounted on the outlet. Additional drawings of the pipe itself are not necessary; the details of the air cap are presented in Figure 12 (a drawing was taken from the manufacturer's brochure).

The air cap used in the analysis is a product of John Gjerde A.S., type AERO 1.2, size DN250. The hat-shaped disc in the centre of the air cap is a float which closes the hole above it if the water enters the air cap.



DN	Inch	A	a	B	L	øK	øD	C	H
250	10	660	530	590	465	350	395	290	258

Figure 12 Air cap details

Similarly as in Case 1, the cylindrical zones were added at the pipe inlet and the air cap outlet. The entire computational domain is presented in figure 13. The dimensions of the inlet and outlet zones are presented in Figures 14-15.

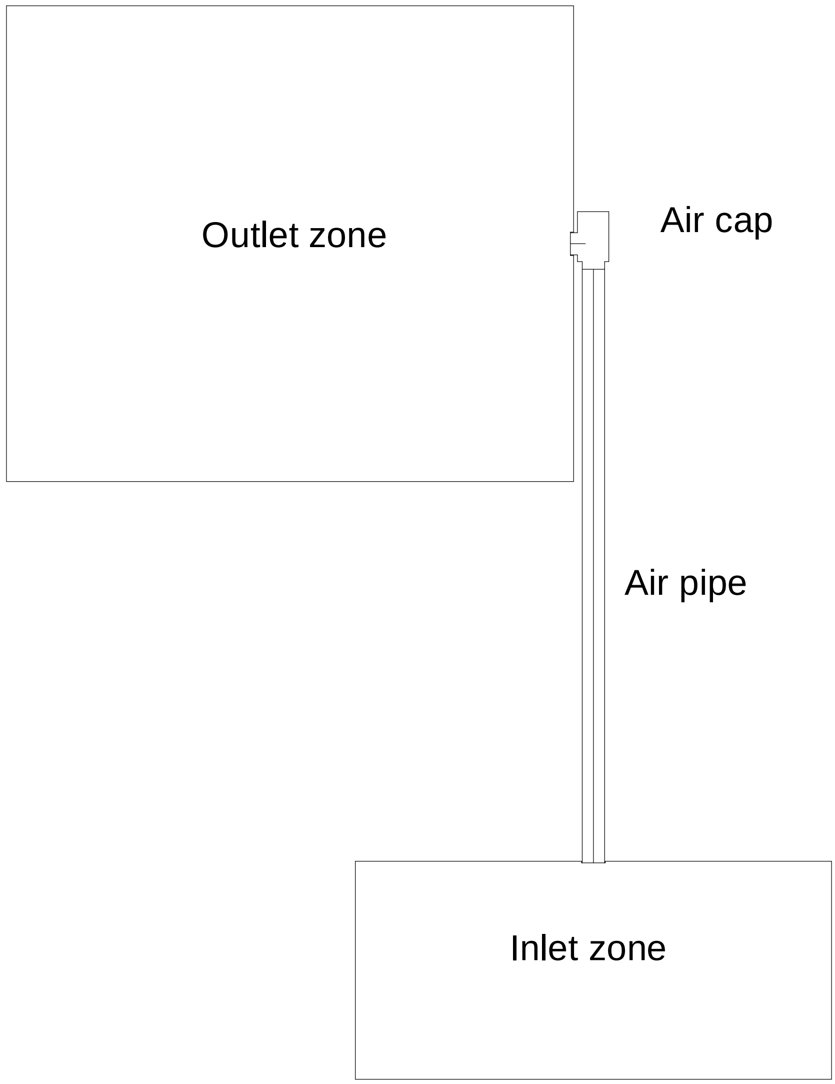


Figure 13 Computational domain – Case 2

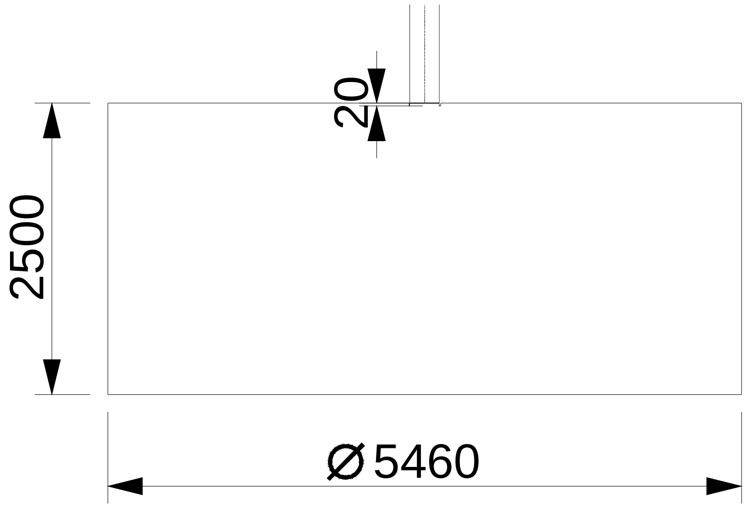


Figure 14 Cylindrical zone surrounding the air pipe inlet – Case 2

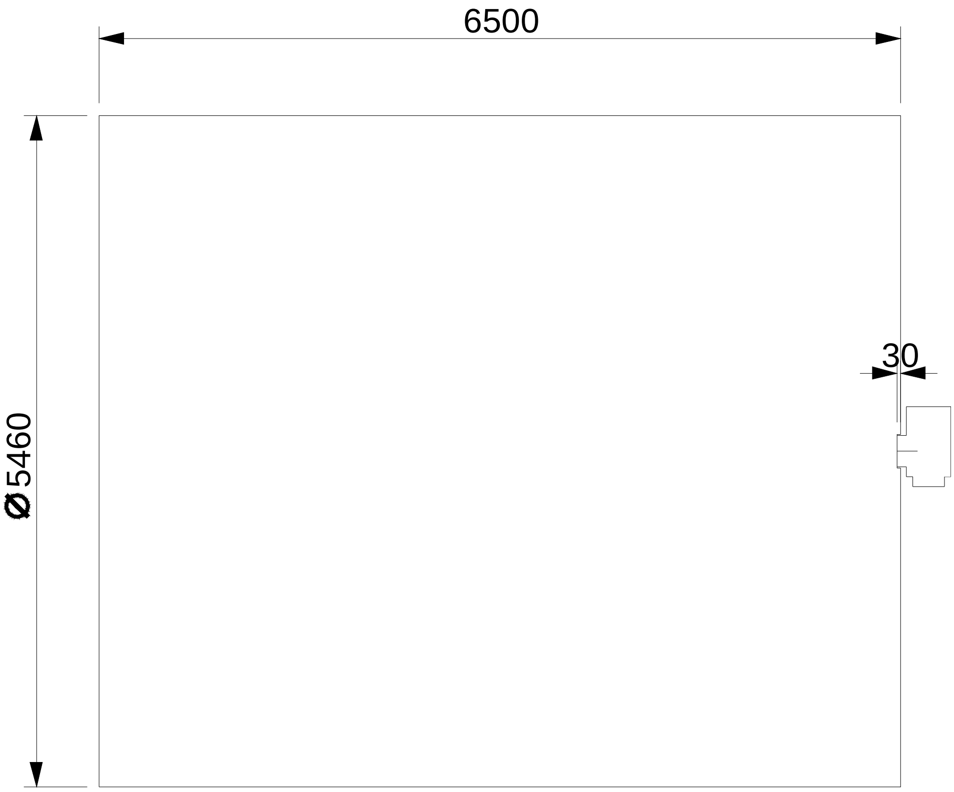


Figure 15 Cylindrical zone surrounding the air pipe outlet – Case 2

The numerical mesh used for Case 2 was similar to that used for Case 1, the approximate number of cells was 1 400 000. Visualization of the numerical mesh for Case 2 is presented in Figures 16-19.

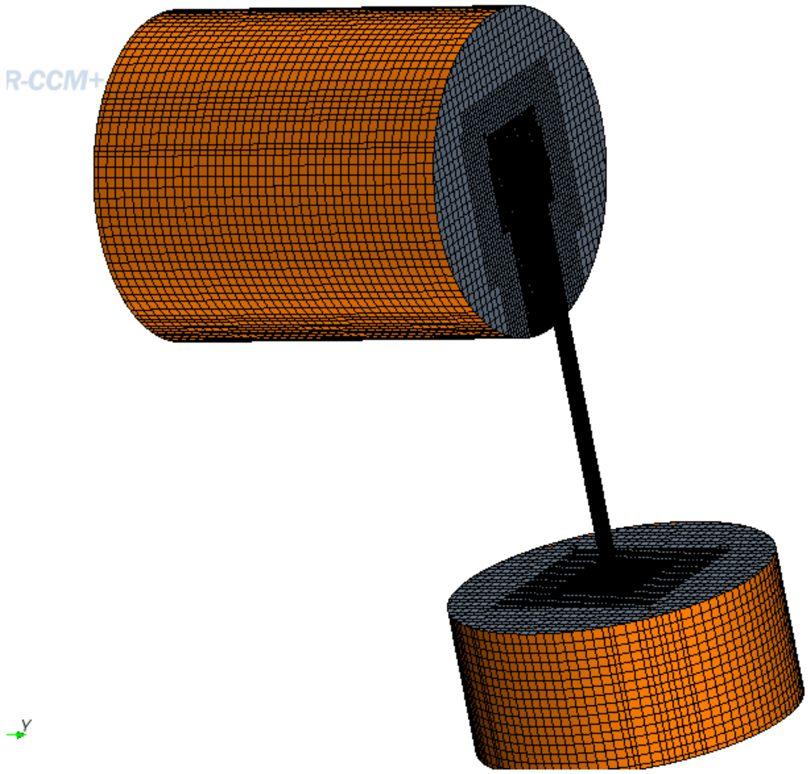


Figure 16 Computational mesh for Case 2 – overall view

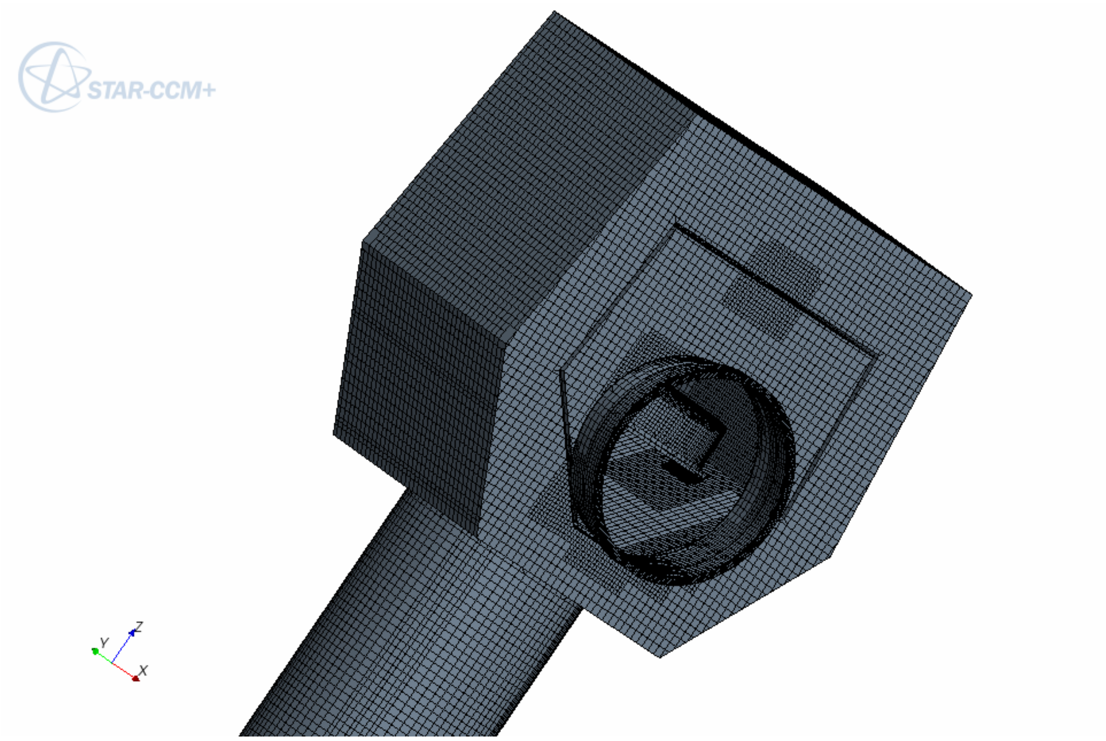


Figure 17 Computational mesh for Case 2 - surface mesh on the air cap

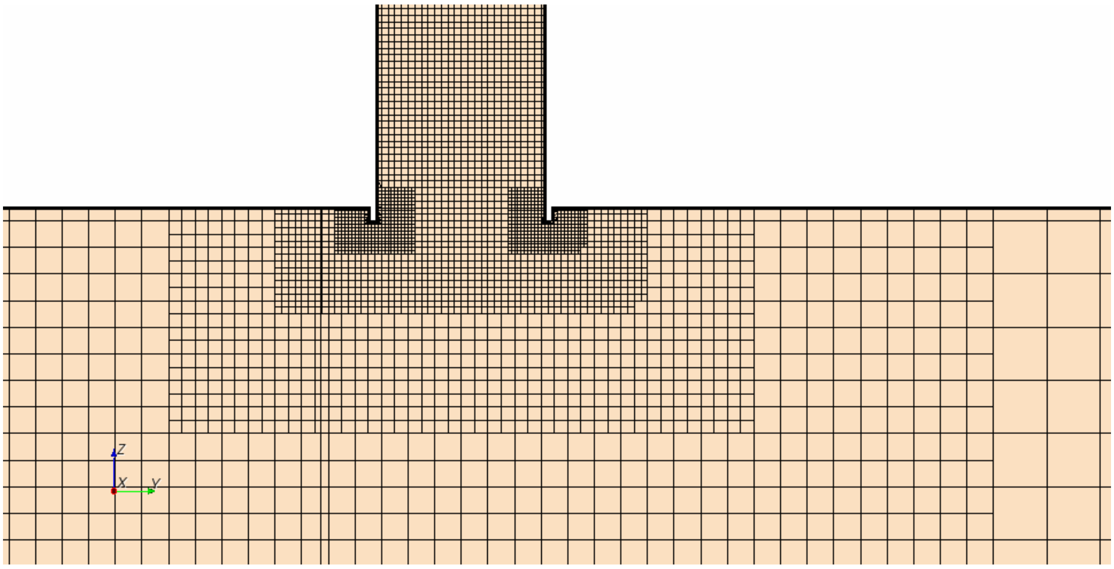


Figure 18 Computational mesh for Case 2 – mesh density at the pipe inlet region

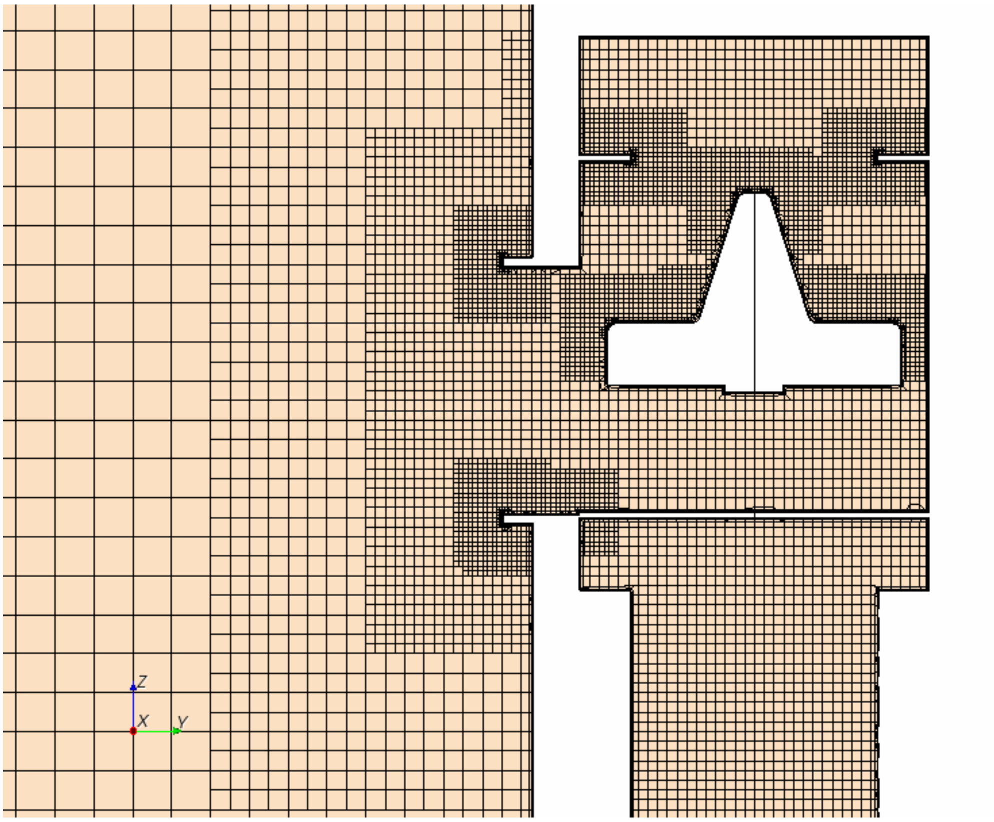


Figure 19 Computational mesh for Case 2 – mesh density at the air cap outlet region

The solver settings for Case 2 were similar as for Case 1 (see page 13), only the time step was reduced to 0.005s for higher overpressure at the inlet zone.

Three values of overpressure Δp at the inlet zone were considered: 1 kPa, 10 kPa and 20 kPa (the same values as for Case 1).

4 RESULTS

The results are presented for both cases together to enable the comparison between them. Presentation of the results include:

- Flow visualization (pressure and velocity)
- Quantitative results (mass flow rate, speed reduction factor, pressure loss coefficient).

Convergence of the computations is also presented.

4.1 Convergence criteria

For each case, the flow was considered converged when the value of mass flow rate in the pipe became constant and decrease of residuals was satisfactory. Sample time histories of residuals and mass flow rates are presented in Figures 20-23.

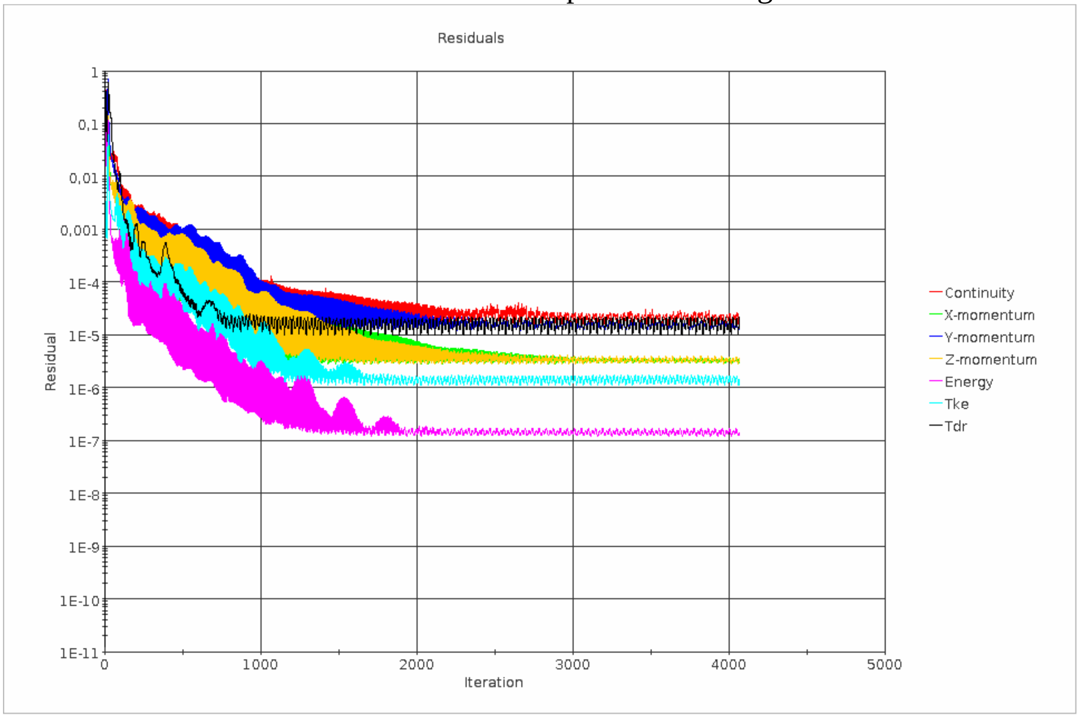


Figure 20 Residuals – Case 1, overpressure 10 kPa

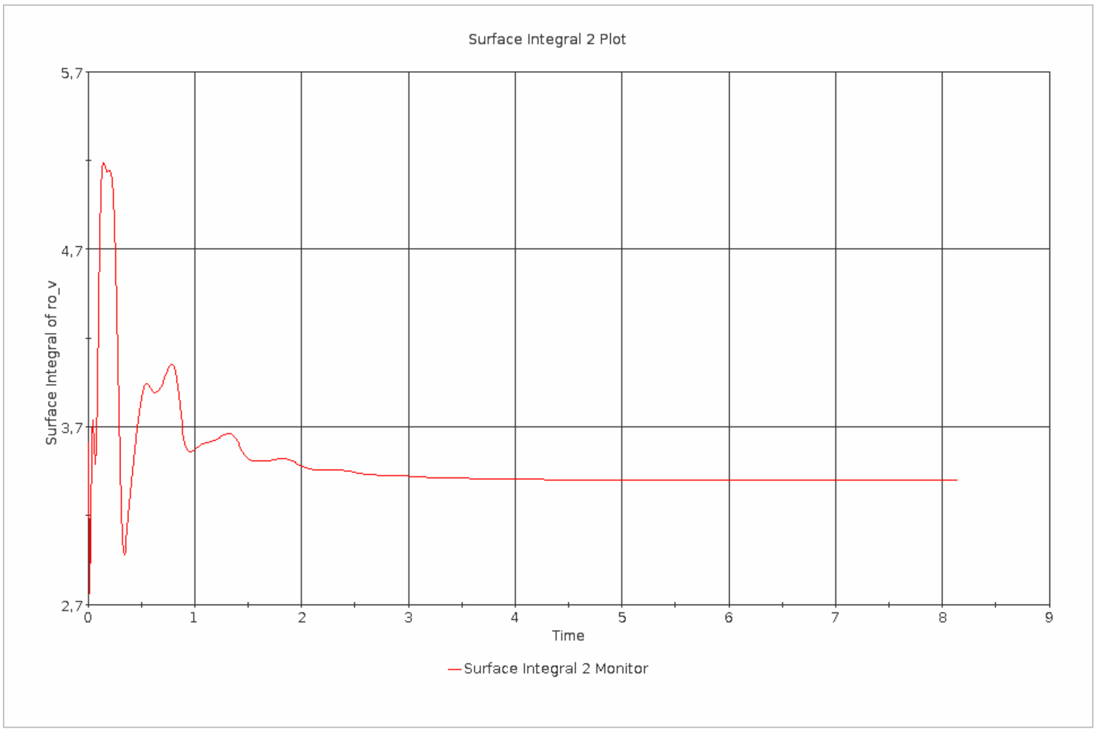


Figure 21 Mass flow rate - Case 1, overpressure 10 kPa

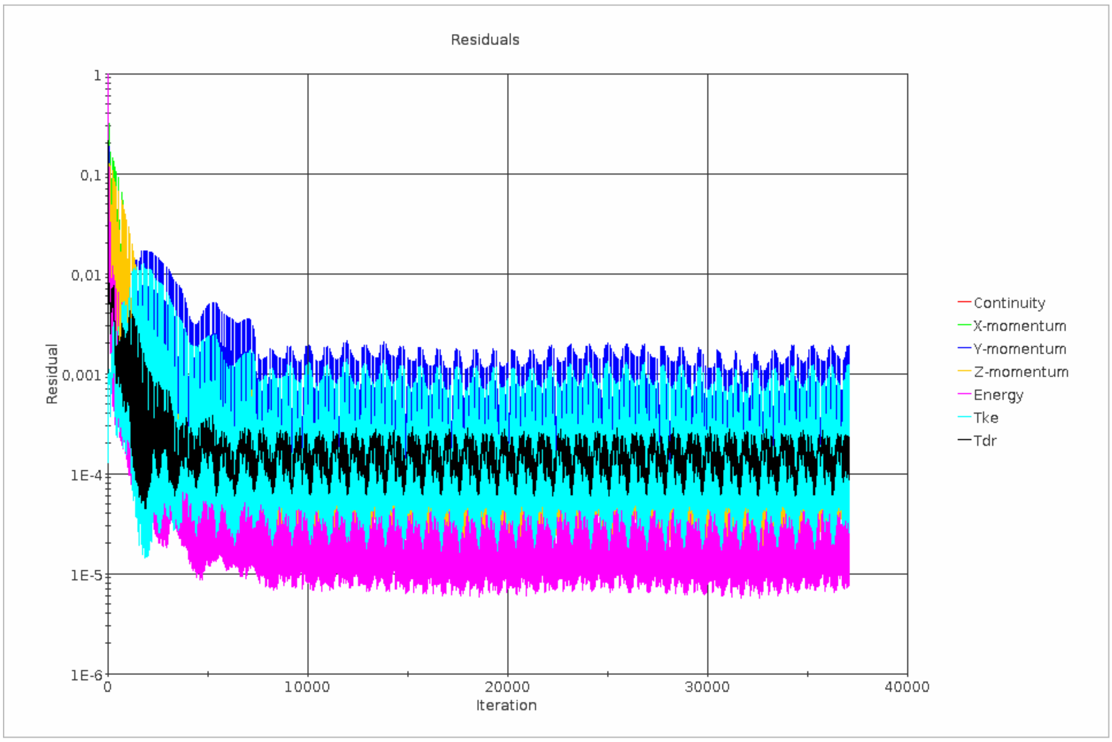


Figure 22 Residuals – Case 2, overpressure 20 kPa

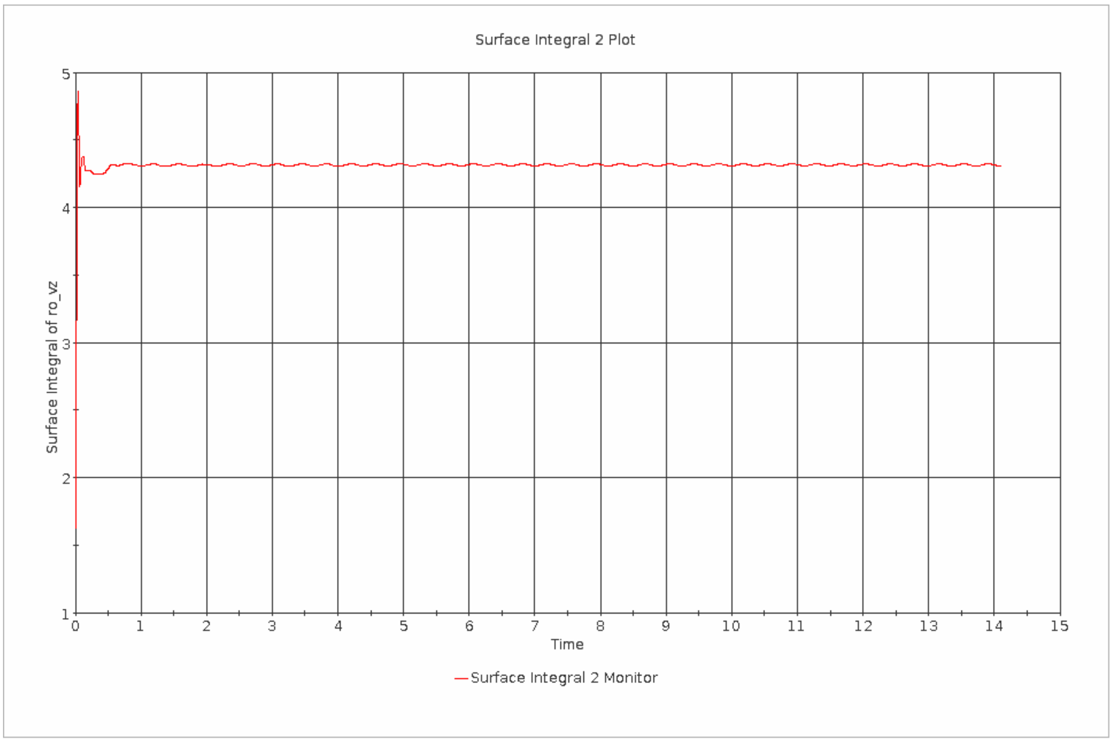


Figure 23 Mass flow rate - Case 2, overpressure 20 kPa

4.2 Flow visualization

The pressure and velocity magnitude are presented in non-dimensional form, so as to enable the comparison of the flow at different values of overpressure at the inlet.

The pressure was normalized by Δp (overpressure at the inlet zone boundary) and the velocity magnitude was normalized by $\sqrt{\frac{2\Delta p}{\rho_0}}$ - a theoretical maximum speed in the air pipe at overpressure Δp, without any losses (ρ_0 is the reference density, $\rho_0 = 1.177$ kg/m^3.

Note: Calculation of the theoretical maximum speed is based on the assumption that the flow character at the pipe outlet is quite different from the flow at the inlet:

- At the outlet - an air jet is forming, so the flow is similar to that in the pipe. The pressure is approximately constant around the outlet and inside the pipe close to the outlet;
- At the inlet - the velocity can be assumed to be zero at some distance from the pipe inlet.

The following points are then used for the calculation of theoretical maximum speed basing on the Bernoulli equation:

- A point located well below the pipe inlet, where $p_1 = \Delta p$ and $v_1 = 0$;
- A point inside the pipe, very close to the outlet, where $p_2 = 0$ and v_2 is the theoretical maximum speed to be computed.

<u>Normalized pressure – pipe inlet region</u>

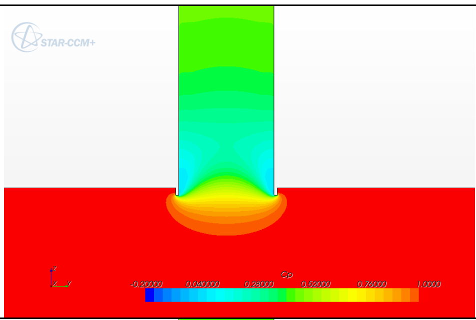
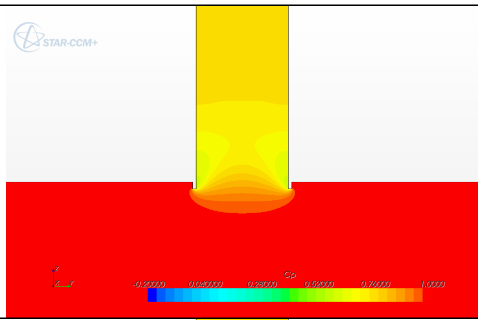
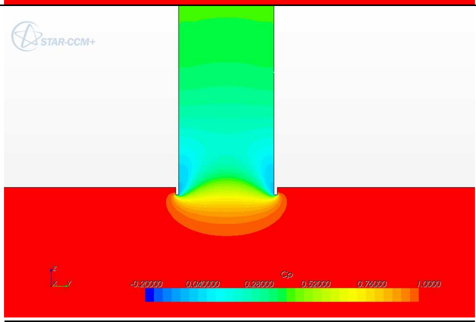
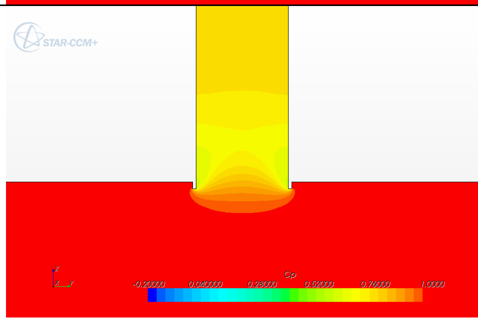
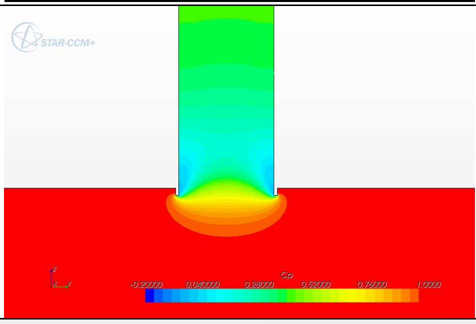
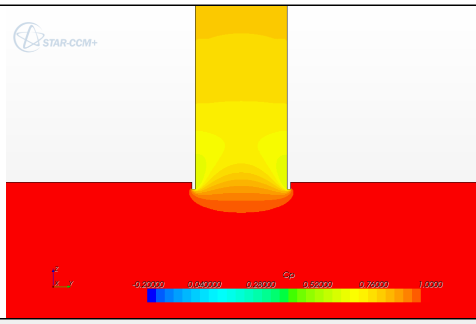
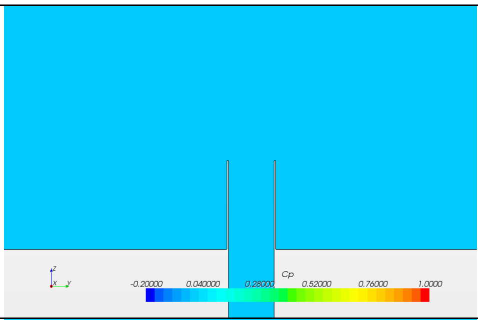
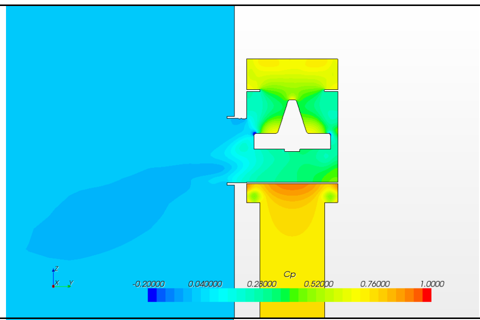
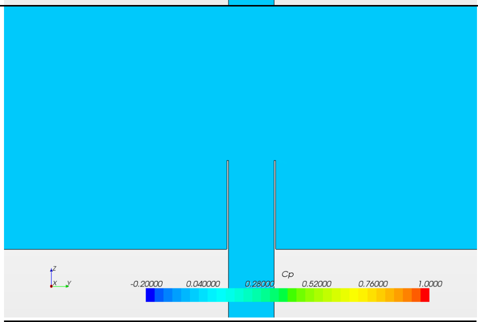
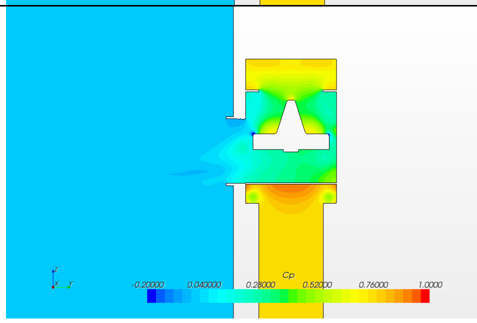
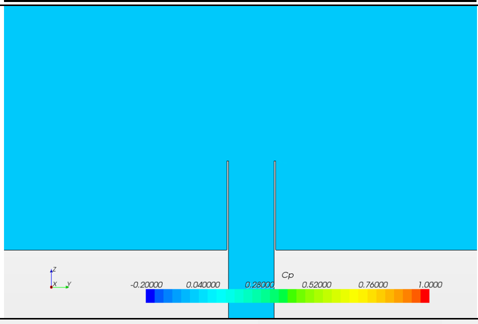
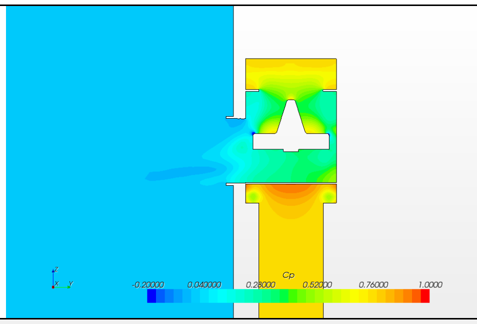
Δ*p*	Case 1 (free outlet)	Case 2 (air cap)
1 kPa		
10 kPa		
20 kPa		

Figure 24 Flow visualization – normalized pressure at the pipe inlet region

Normalized pressure – pipe outlet region

Δp	Case 1 (free outlet)	Case 2 (air cap)
1 kPa		
10 kPa		
20 kPa		

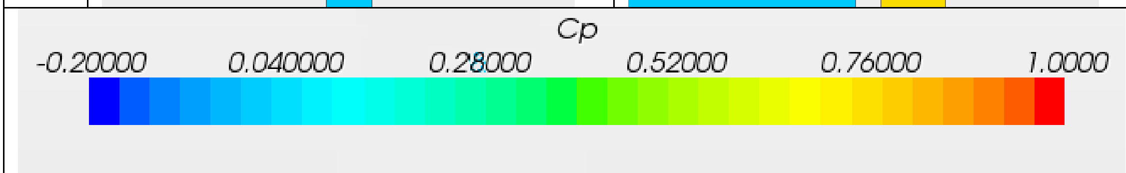


Figure 25 Flow visualization – normalized pressure at the pipe outlet region

<u>Normalized velocity magnitude – pipe inlet region</u>

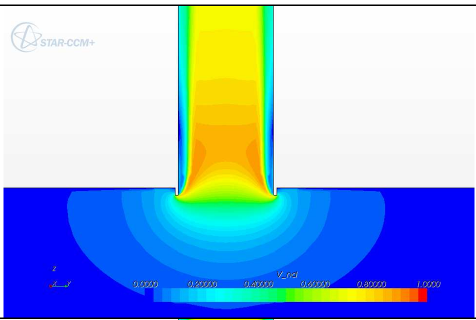
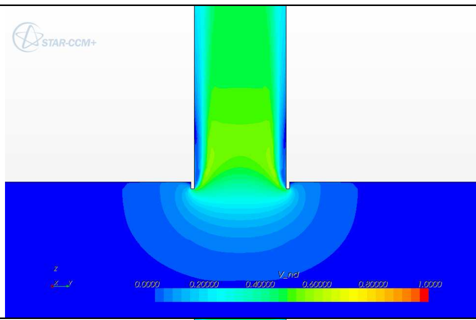
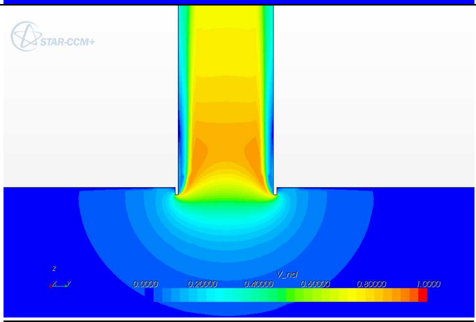
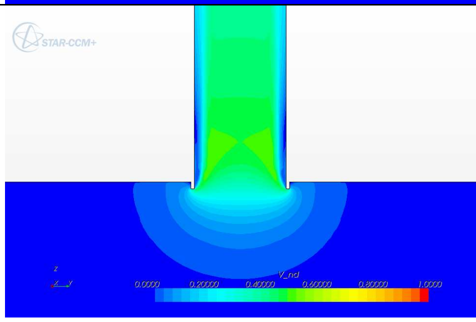
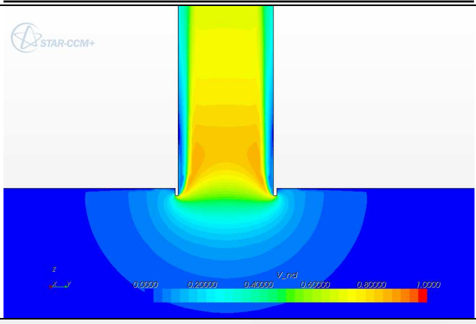
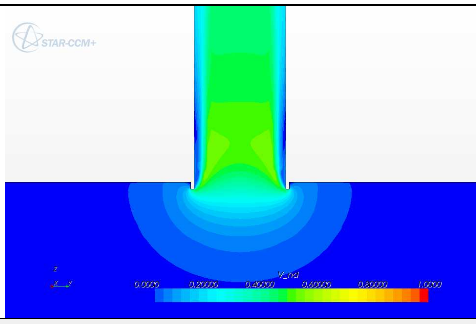
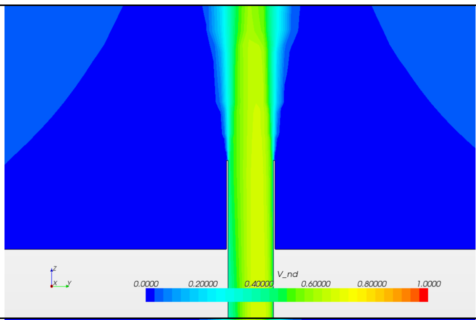
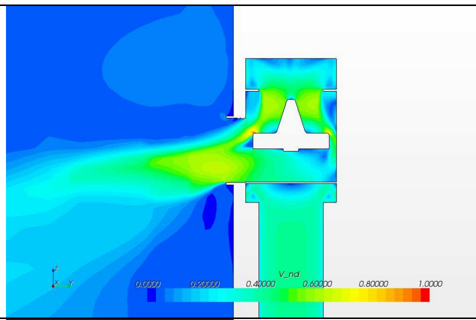
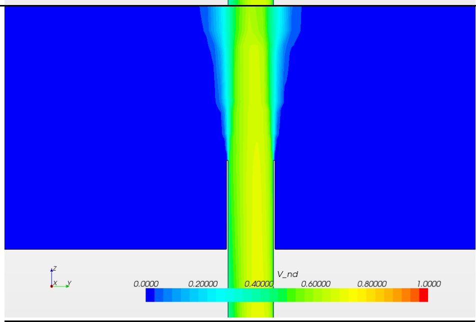
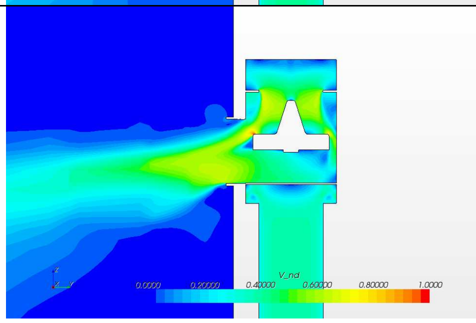
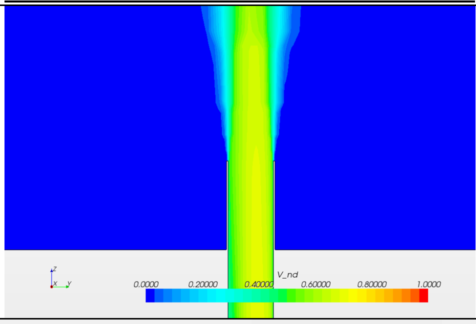
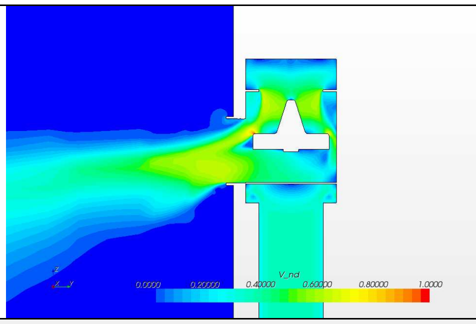
Δ*p*	Case 1 (free outlet)	Case 2 (air cap)
1 kPa		
10 kPa		
20 kPa		

Figure 26 Flow visualization – normalized velocity magnitude at the pipe inlet region

rmalized velocity magnitude – pipe outlet region

Δp	Case 1 (free outlet)	Case 2 (air cap)
1 kPa		
10 kPa		
20 kPa		

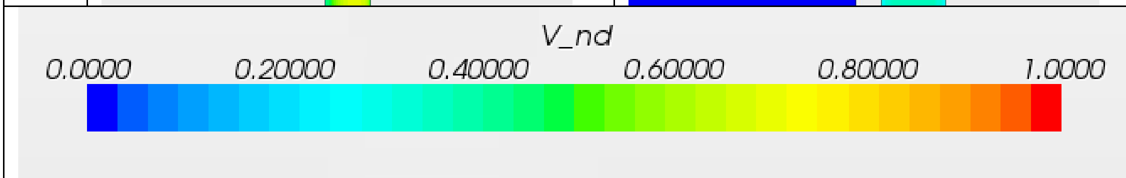


Figure 27 Flow visualization – normalized velocity magnitude at the pipe outlet region

Figure 28 shows the normalized velocity magnitude contour map in the symmetry plane of the pipe for Case1, in the double mitre bends region (overpressure 10 kPa). For other values of overpressure, the qualitative character of the flow is similar.

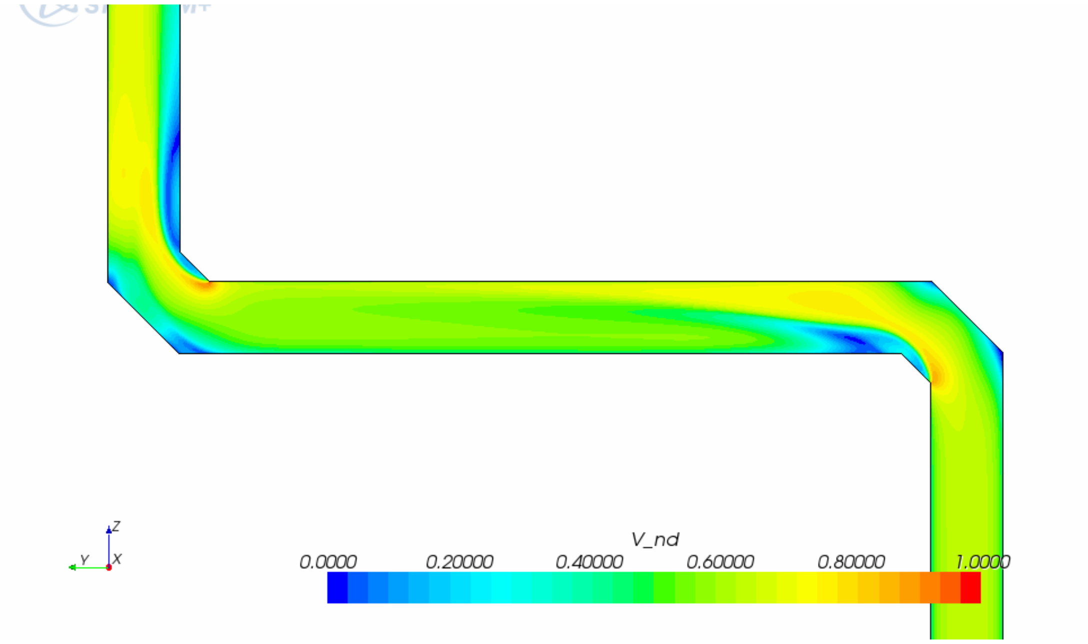


Figure 28 Normalized velocity magnitude in the double mitre bends region (Case 1, overpressure 10 kPa)

4.3 Quantitative results

The quantity resulting directly from the CFD computations is the air mass flow rate in the pipe. It was computed as an integral of the normal velocity component and local density value over the pipe cross-section:

$$\dot{m} = \int_A \vec{v}_n \cdot \rho \cdot dA$$

($\vec{v}_n$ is the velocity component normal to the pipe cross-section).

For better accuracy, the mass flow rate was computed in two sections of the pipe (close to the inlet and close to the outlet) and averaged. Although the mass flow rate is theoretically constant in the pipe, some difference occurs due to numerical errors; the difference between the computed mass flow rates at two ends of the pipe was well below 0.5% for each case.

Sample procedure of computing the speed reduction factor and pressure loss coefficient is presented below, for Case 1 and overpressure 10 kPa.

The speed reduction factor F_{Air} is defined as:

$$F_{Air} = \frac{\dot{m}_{Real}}{\dot{m}_{Theor}} = \frac{\rho \cdot Q_{Real}}{\rho_0 \cdot Q_{Theor}}$$

where $\dot{m}_{Real}$ - actual mass flow rate, $\dot{m}_{Theor}$ - theoretical mass flow rate (according to Bernoulli's equation), Q_{Real} – actual discharge, Q_{Theor} - theoretical discharge, ρ – density in the section where Q_{Real} is computed, ρ_0 – density at reference pressure and reference temperature.

The actual discharge is taken from the computations, the theoretical discharge is computed basing on the following formulae:

$Q = v \cdot A$ (definition of the discharge)

where v – axial velocity component in the pipe, A – cross-section of the pipe;

$\Delta p = \frac{1}{2} \rho_0 v^2$ (based on the Bernoulli's equation)

where Δp – overpressure (difference between the pressure in the vented compartment and atmospheric pressure)
Consequently:

$$\Delta p = \frac{1}{2}\rho_0\left(\frac{Q_{Theor}}{A}\right)^2$$

$$Q_{Theor} = A\sqrt{\frac{2\Delta p}{\rho_0}}$$

Substituting the density value ρ_0:

$$\rho_0 = 1.177\,{}^{kg}\!/\!{}_{m^3}$$

one obtains:

$$Q_{Theor} = 6.679\,{}^{m^3}\!/\!{}_{s}$$

and $\rho_0 \cdot Q_{Theor} = 7.859\,{}^{kg}\!/\!{}_{s}$

The computed value of $\rho \cdot Q_{Real}$ is:

$$\rho \cdot Q_{Real} = 4.741\,{}^{kg}\!/\!{}_{s}$$

The speed reduction factor F_{Air} equals:

$$F_{Air} = 0.603$$

The pressure loss coefficient k and the speed reduction factor are related to each other by the following formula:

$$F_{Air} = \frac{1}{\sqrt{k}}$$

$$k = \frac{1}{F_{Air}^2}$$

$$k = 2.748$$

For Case 2 (pipe with air cap), the speed reduction factor and pressure loss coefficient were also extracted for the air cap itself, basing on the pressure values at the air cap inlet and outlet.

The quantitative results for all cases are summarized in tables 1 – 3.

Table 1 Quantitative results – Case 1 (air pipe with free outlet)

Overpressure [kPa]	**1.0**	**10**	**20**
Actual mass flow rate [kg/s]	1.427	4.741	6.793
Theoretical mass flow rate [kg/s]	2.485	7.859	11.115
Speed reduction factor F_{Air} [-]	0.574	0.603	0.611
Pressure loss coefficient k [-]	3.034	2.748	2.677
Maximum velocity [m/s]	37.8	124.7	176.5

Table 2 Quantitative results – Case 2 (air pipe with air cap)

Overpressure [kPa]	**1.0**	**10**	**20**
Actual mass flow rate [kg/s]	0.919	2.992	4.320
Theoretical mass flow rate [kg/s]	2.485	7.859	11.115
Speed reduction factor F_{Air} [-]	0.370	0.381	0.389
Pressure loss coefficient k [-]	7.321	6.901	6.619
Maximum velocity [m/s]	42.1	136.4	192.0

In Table 3, the speed reduction factors and pressure loss coefficients for the air cap itself are given. These values were also extracted from the computations named Case 2, only the overpressure value was taken from the air cap inlet.

Table 3 Quantitative results for the air cap (extracted from Case 2)

Overpressure [kPa]	**0.74**	**7.52**	**15.25**
Actual mass flow rate [kg/s]	0.919	2.992	4.320
Theoretical mass flow rate [kg/s]	2.132	6.817	9.705
Speed reduction factor F_{Air} [-]	0.431	0.439	0.445
Pressure loss coefficient k [-]	5.385	5.192	5.047

Figure 29 shows the speed reduction factor F_{Air} as a function of overpressure, for both analyzed cases and the air cap.

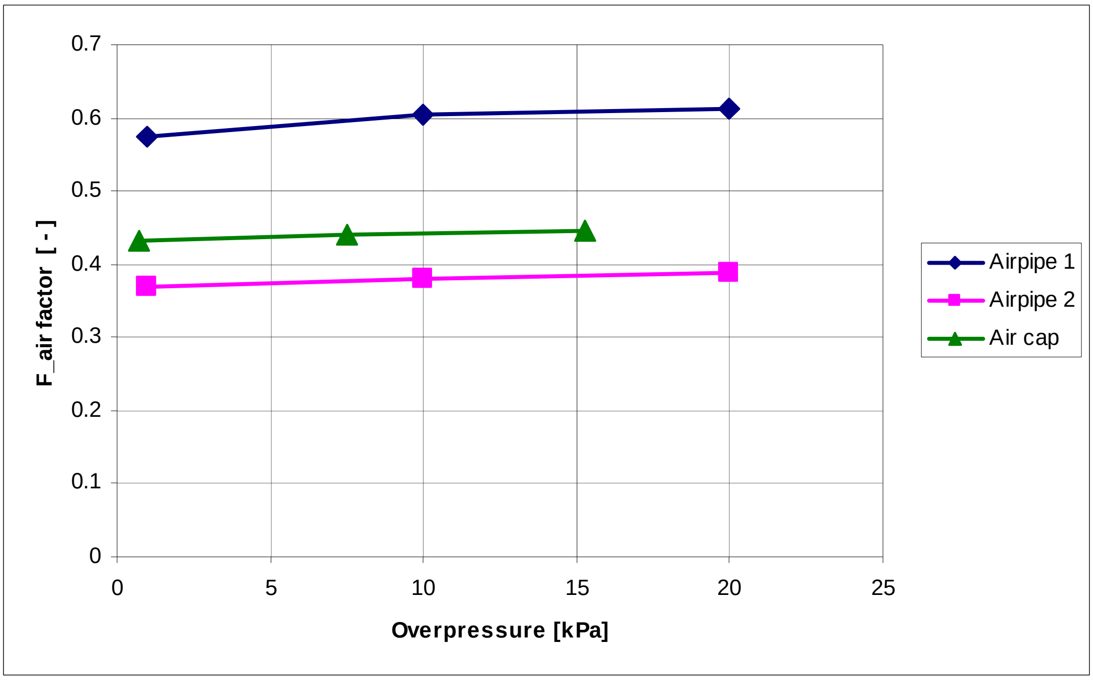


Figure 29 Speed reduction factor F_{Air} as a function of overpressure

The simplified approach, based on IMO resolution **MSC.245(83),** consists in calculating the pressure loss coefficient k by adding the k values for particular elements of the pipe system (inlet, outlet, valves, bends etc.). These values are listed in the resolution (Figure 30).

A sample calculation of the k coefficient for the pipe analyzed in Case 1 is presented here:

Inlet	k = 0.43
Outlet	k = 1.00
no valve	k = 0.00
2 x 90 deg double mitre bend	k = 2 x 0.44
Pipe friction	k = 0.02xL/D = 1.21 (corrected.)
Sum	**k = 3.52 (corrected.)**
Result of CFD	k = 2.748

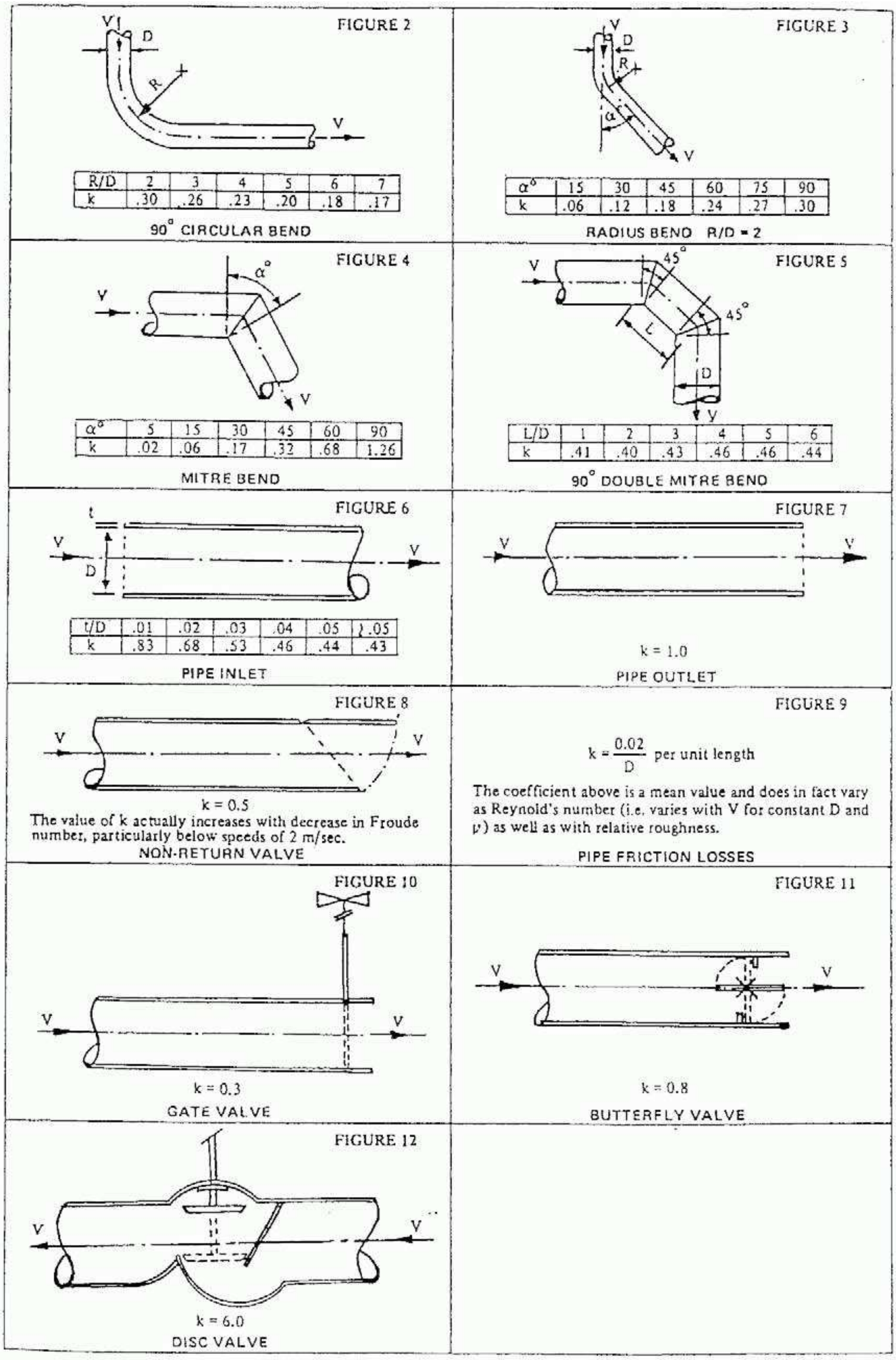


Figure 30 List of “k” values for particular elements of the pipe system

5 CONCLUSIONS

The analysis presented in this report can be summarized as follows:

- Speed reduction factors and pressure loss coefficients were computed for two types of venting air pipes and three values of overpressure at the inlet (in the vented compartment);
- The variation of pressure loss coefficients and speed reduction factors is small within the considered range of overpressure, so they can be assumed constant for simplified computations;
- The maximum velocity for highest overpressure reaches the value of app. 190m/s, which corresponds to Mach number 0.56. Using the compressible gas model is advisable for such values;
- Comparison of the CFD results with the simplified approach based on IMO resolution **MSC.245(83)** shows that the simplified approach yields notably higher values of pressure loss coefficient than CFD computations. This can be due to the fact that the IMO resolution is elaborated for water flows, for which the Reynolds number is usually smaller, which means higher pressure losses in general.