

# The numerical evaluation of safety valve size in the pipelines of cryogenic installations

**Z M Malecha**

Department of Cryogenic, Aeronautic and Process Engineering , Wrocław University of technology, Poland, ul. Wyb. Wyspińskiego 37, 50-370 Wrocław, Poland

E-mail: [ziemowit.malecha@pwr.edu.pl](mailto:ziemowit.malecha@pwr.edu.pl)

**Abstract.** The flow of cold helium in pipes is a fundamental issue of any cryogenic installation. Pipelines for helium transportation can reach lengths of hundreds of meters. The proper selection of size for individual pipelines and safety valves is a crucial part in the consideration of costs for the entire installation and its safe operation. The size of the safety valve must be properly designed in order to avoid a dangerous pressure build-up during normal operation, as well as in the case of emergency. The most commonly occurring dangerous situation is an undesired heat flux in the helium as a result of a broken insulation. In this case, the heat flux can be very intense and the build-up of the pressure in the pipe can be very rapid. In the present work, numerical calculations were used to evaluate the build-up of pressure and temperature in the pipe, in the case of a sudden and intense heat flux. The main goal of the applied numerical procedure was to evaluate the proper sizes of the safety valves in order to avoid a rise in pressure above the safety limit. The proposed numerical model and calculations were based on OpenFOAM, an open source CFD toolbox.

## 1. Introduction

The proper selection of safety equipment is a very important part of the design study of any Cryogenic Distribution System (CDS). Safety valves are some of the most common safety devices located on the pipelines. They must be properly chosen in order to counteract any undesired increase pressure inside the pipes. The most common cause of a sudden increase in pressure is often due to intense heat flux as a consequence of broken insulation. It is also important to estimate the proper sizes of the individual pipes of the CDS in order to avoid overestimation and an unnecessary increase in cost. To accomplish both of these tasks, it is necessary to predict the dynamics of the pressure increase for each individual pipeline, and for the given heat flux.

The evaluation of the sizes of the pipelines may be based on zero dimensional analysis, but such models are very limited and tend to result in overestimation. Moreover, this type of analysis may be insufficient for the proper calculation of the size of the safety valves. On the other hand, a full three-dimensional (3D) CFD analysis would be prohibitively long because each individual pipe is hundreds of meters long, thus requiring a large number of various calculations. The wrong estimation of the safety valves can lead to very serious consequences [1].

A 2D numerical analysis based on the proper mathematical model seems to provide the ideal approach to this problem. 2D calculations are orders of magnitude faster than their 3D originals. The main challenge is in the proper transformation of the 3D original geometry to its 2D numerical model, and the adequate interpretation of the results.



In the works [2, 3], it was shown that the sufficient mathematical model, in order to calculate the dynamics of gases in cryogenic temperatures, could at least include the compressible Navier-Stokes equations with ideal gas and additive mixing approximations. Both works dealt with the propagation of the cryogenic gas (argon and helium) in the tunnel. Additionally, the work [3] compared the numerical results with the experimental data.

## 2. Numerical model of the pipeline

### 2.1. Transformation of the 3D geometry to its 2D representation.

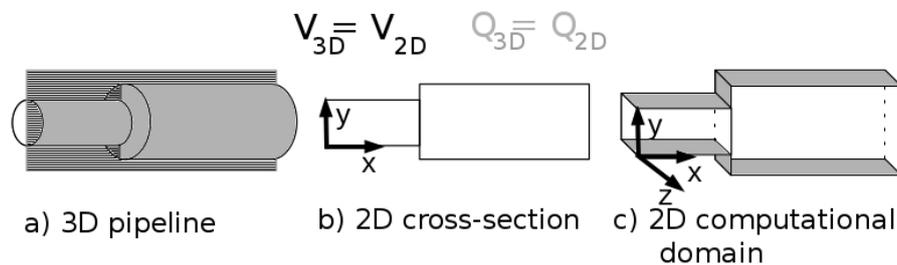
Due to the extensive length of the pipelines of the typical cryogenic installation, and in order to avoid prohibitively time consuming calculations, a 2 dimensional (2D) simplification was adopted, figure 1. It was assumed that the flow was invariant in width direction,  $(\cdot)/\partial z = 0$ .

Sketch a) from figure 1 shows an exemplary 3D geometry of the exemplary pipeline, composed of two pipes with a different diameter. Sketch b) shows a 2D longitudinal cross-section in the middle of the 3D domain. Sketch c) shows a numerical domain which was used in the current 2D calculations. The numerical domain from sketch c) is a 3D extrusion of the 2D cross-section from sketch b). It is clearly visible that the geometries from sketches a) and c) are not identical. In order for flow and thermal conditions in the numerical model to be as similar as possible to the real (original) conditions, the following measures were preserved:

- The volume of the original pipeline and its numerical model are equal,  $V_{3D} = V_{2D}$ .
- The total heat delivered through the walls is the same for the original pipeline and the numerical model,  $Q_{3D} = Q_{2D}$ , where:  $Q_{3D} = q_{3D}A_{3D}$  and  $Q_{2D} = q_{2D}A_{2D}$ . Notice that  $A_{3D} \neq A_{2D}$ , where  $A_{3D}$  and  $A_{2D}$  correspond to the area of the wall of the original pipeline and its numerical model, respectively.
- The cross-section of the safety valves is the same for the original pipeline and its numerical model,  $A_{Ve3D} = A_{Ve2D}$  (typically located on each end of the pipeline, not shown on the figure).

The first condition ensures that the same amount of He is in the original pipeline and in its numerical model. The second condition ensures that the same amount of heat is delivered to the He.

The change of the original circular geometry to the corresponding 2D channel, changed the value of a hydraulic diameter by  $\sim 10\%$ . Nevertheless, the resulting pressure drop difference can be considered as small and of second importance if compared to the pressure gradients developed in the considered flow.



**Figure 1.** Transformation of the 3D model of the original pipeline to its numerical geometry. Notice that the numerical geometry has 3 dimensions (length, height and width), but the numerical model is 2D because it is width invariant,  $(\cdot)/\partial z = 0$ . The heat flux is delivered to the He through the shaded walls. Notice that the area of the walls of the 3D model of the original pipeline is not equal to the area of the walls of the numerical model,  $A_{3D} \neq A_{2D}$ .

## 2.2. Mathematical model and numerical implementation

Numerical calculations were made using the *sonicFOAM* solver implemented in OpenFOAM (Open Source Field Operation and Manipulation) CFD toolbox [4]. OpenFOAM has been used effectively in diverse and challenging applications [3, 5]. In the work [5], the behaviour of a gas mixture, subjected to very high temperatures and high frequency sound waves, was modelled and then compared with analytical solutions. In the work [3], the emergency ejection of helium in cryogenic temperatures into the tunnel was considered, and then additionally compared with the experiment. Both works proved that OpenFOAM is a trustworthy and reliable CFD software.

For the present application, finite volume discretization was employed in conjunction with the PISO (Pressure Implicit with Splitting of Operators) algorithm for compressible flows [6, 7]. Like the widely-used SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm, PISO schemes belong to the family of pressure correction methods. However, the PISO method requires less computational effort since the conservation of mass is satisfied within the predictor–corrector steps.

The *sonicFOAM* solves for a transient, trans-sonic/supersonic, laminar or turbulent flow of a compressible gas. This choice was dictated by the fact that, in the case of the considered flow, high speeds are expected. The sudden opening of an safety valve, especially, can cause the creation of a shock wave. *sonicFOAM* uses numerical schemes that can capture these features while avoiding spurious oscillations.

The velocity field,  $\mathbf{u} = (u, v, w)$  was calculated by solving the compressible Navier-Stokes equations [8]:

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot (\mu \nabla \mathbf{u}) \quad (1)$$

which, along with the continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (2)$$

constitute a closed system. The density was calculated using the ideal gas equation and the Sutherland approximation was used to calculate the viscosity [4].

The temperature of the He was calculated using the energy transport equation:

$$\frac{\partial \rho e}{\partial t} + \nabla \cdot (\rho \mathbf{u} e) = \nabla \cdot \left( \frac{k}{C_v} \right) \nabla e + p \nabla \cdot \mathbf{u} \quad (3)$$

where  $k$  is heat conductivity and  $e = C_v T$ .

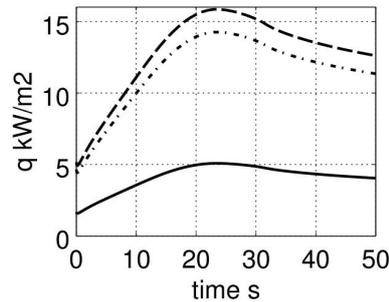
## 3. Exemplary results and discussion

The introduced numerical model was used to calculate the increase of pressure in two typical pipelines, as a consequence of broken insulation. The goal of the analysis was to investigate whether the sizes of the used safety vales had been appropriately chosen.

The sizes and working parameters of the two pipelines considered below were taken from a design study of a large cryogenics installation. Figure 2 shows the predicted heat flux for the considered installations in the case of broken insulation, and the recalculated heat flux for the corresponding 2D numerical geometries.

### 3.1. Pipeline with one change of diameter

The first considered pipeline was made up of two pipes with different diameters. The first section had length  $L_1 = 355$  m and diameter  $d_1 = 72.1$  mm, while the second section had  $L_2 = 55$  m and  $d_2 = 38.4$  mm. Additionally, one safety valve was located at each end of the pipeline. The



**Figure 2.** The predicted heat flux for the considered installations in the case of broken insulation (solid line), and the recalculated heat flux for the corresponding 2D numerical geometries (dashed line – pipeline considered in §3.1, dash-dotted line – pipeline considered in §3.2).

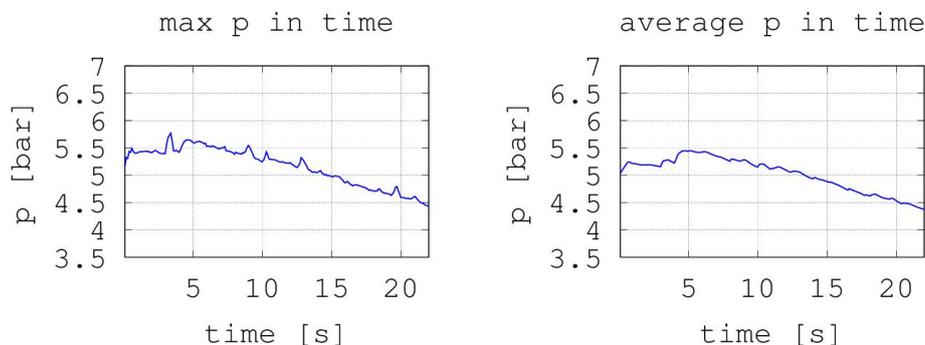
above sizes of the original pipeline were recalculated according to the principles listed in section 2.1.

The nominal pressure of the pipeline was designed to be 4 bar a. The maximum pressure allowed in the pipeline was 6 bar a. The safety valves were chosen to open if the pressure reached 5 bar a. The goal of the numerical analysis was to calculate the minimally required cross-section of the safety valves.

It was assumed that at the beginning, the pipeline had uniform pressure  $p_{IC} = 5$  bar a, with two open safety valves and with heat flux being delivered according to the characteristics showed on the figure 2.

The performed numerical analysis showed that the minimal diameter of the safety valves should be  $d_{v1} = 60.3$  mm and  $d_{v2} = 32.1$  mm, respectively (these are recalculated values for the original 3D geometry). The analysis took into consideration influence of the jet contraction, which can reduce the useful diameter of the safety valve by 30%. Note that the sizes were additionally restricted by the diameters of the individual pipes.

Figure 3 shows the pressure build-up in time for the pipeline equipped with safety valves with minimally required diameters. The left plot shows the maximum pressure in time,  $|p|_{max}$ , while the right plot shows the average pressure in time,  $p_{av} = \frac{1}{A} \int_A p dA$ . It can be seen that for this configuration, the maximum pressure may rise close to 6 bar a, but never goes above it.

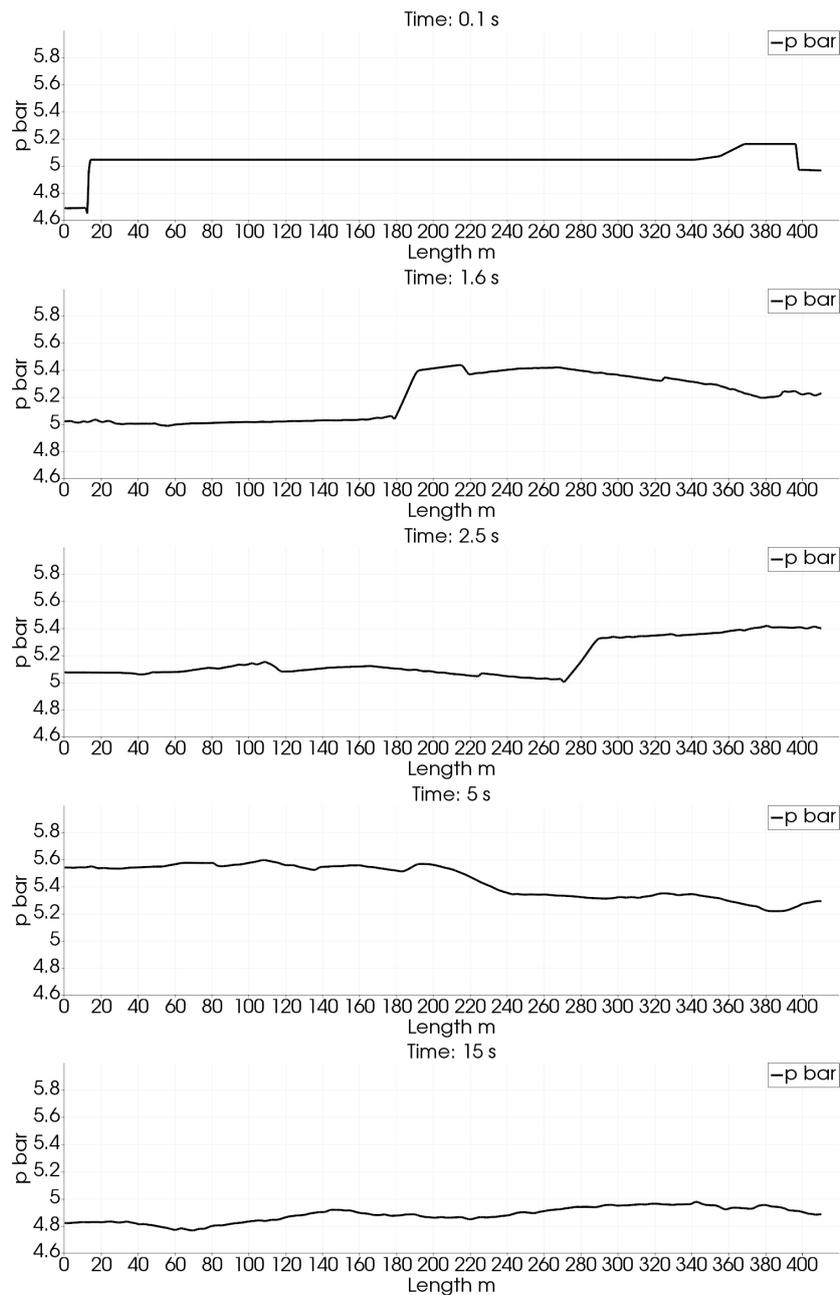


**Figure 3.** The change of pressure in time for the first considered pipeline equipped with safety valves of minimally required diameters.

Figure 4 shows the time sequence of the pressure distribution along the pipeline,  $p(x, y = 0)$ ,

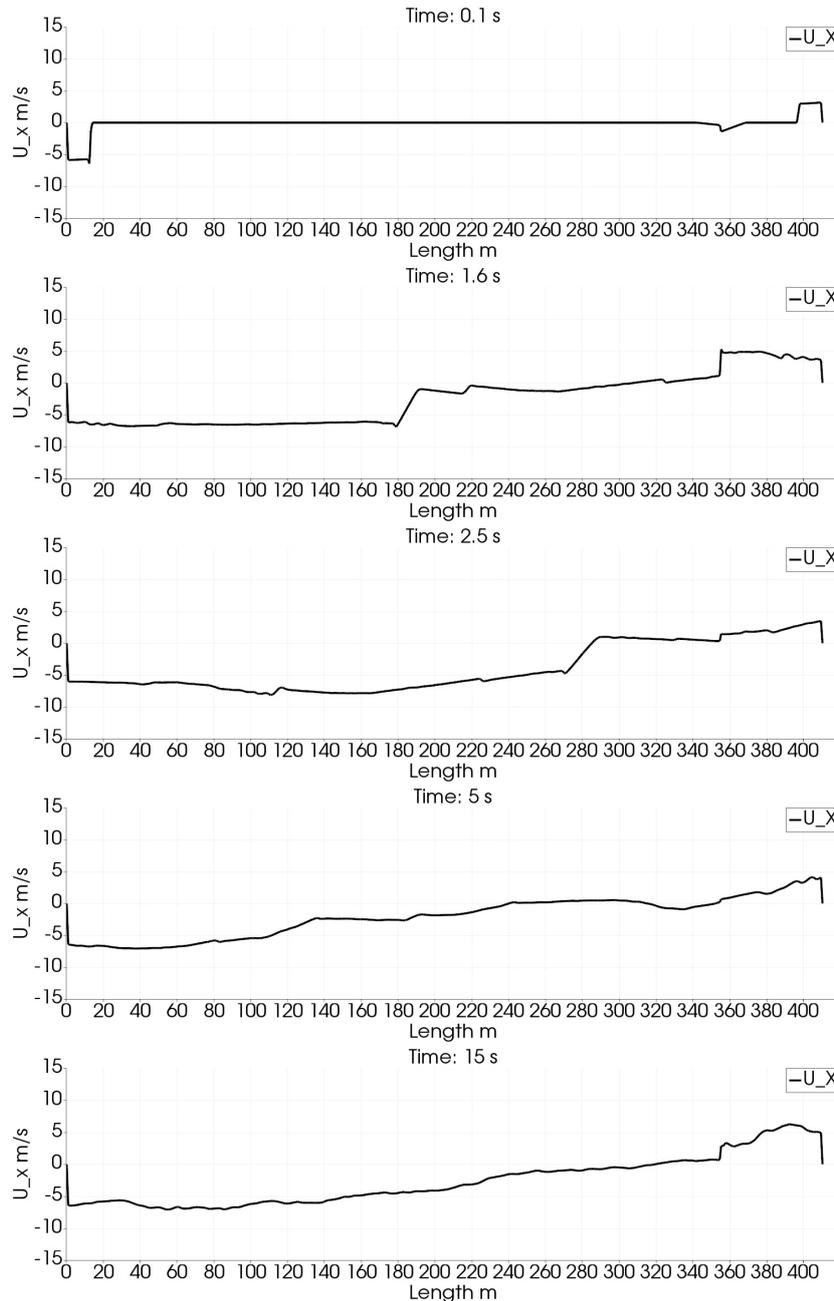
for times  $t = (0.1, 1.6, 2.5, 5, 15)$  s. The pressure distribution along the pipeline taken from a different location (e.g., closer to the lower or upper wall) would be nearly the same, because the pipeline was "long and thin", which causes the pressure to be nearly constant in the vertical direction.

It can be seen that the sudden opening of the safety vales caused the shock waves to be created at both ends of the pipeline. The waves travelled along the pipeline, collided, and then travelled backwards. After 5 s, the waves started to flatten and after 15 s, the pressure went below 5 bar a.



**Figure 4.** Change of the pressure along the pipeline,  $p(x, y = 0)$ , for  $t = (0.1, 1.6, 2.5, 5, 15)$  s. The consecutive plots show the travelling shock waves caused by the sudden opening of the safety valves located at each end of the pipeline.

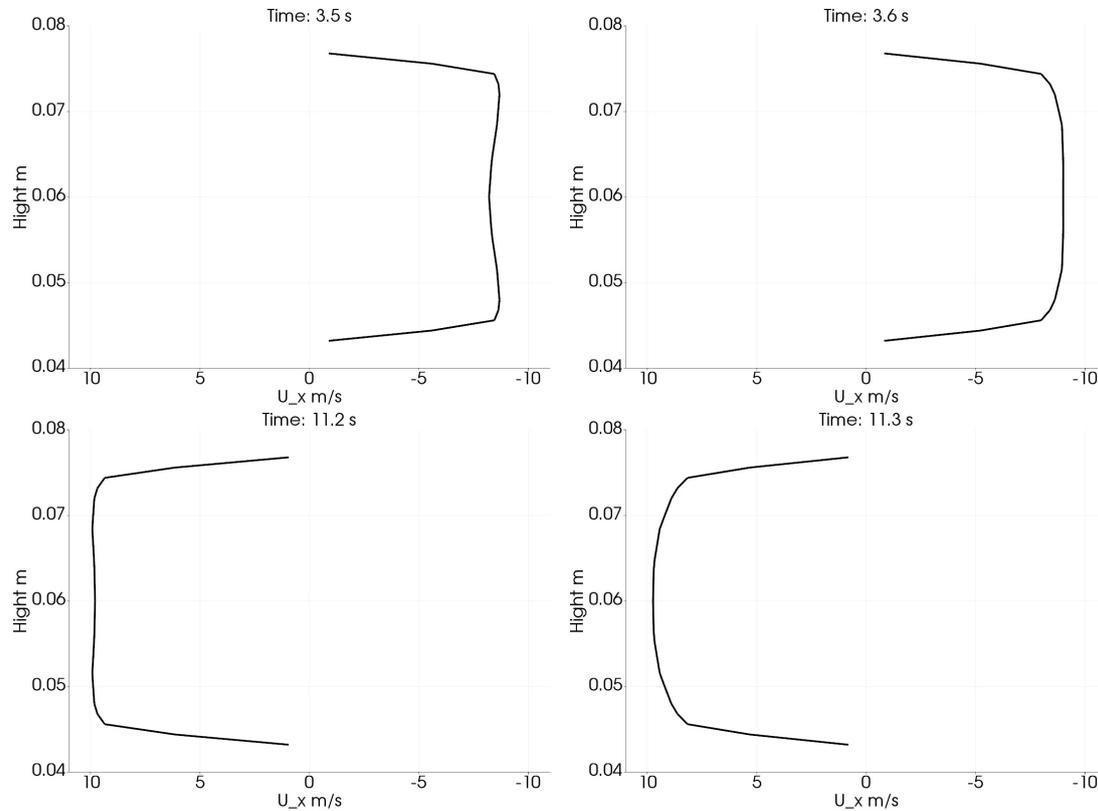
Figure 5 shows the time sequence of the  $x$  component of the velocity along the pipeline,  $u(x, y = 0)$ , for times  $t = (0.1, 1.6, 2.5, 5, 15)$  s. It can be seen that the shock waves travel against the flow. The shock wave created at the left (right) end of the pipeline, moves to the right (left), but the flow is directed to the left (right), note the sign of  $u$  and compare with the figure 4.



**Figure 5.** Change of the  $x$  component of velocity vector along the pipeline,  $u(x, y = 0)$ , for  $t = (0.1, 1.6, 2.5, 5, 15)$ . Note the negative (positive) value of the velocity indicates that the flow is directed to the left (right). The shock wave is travelling opposite to the direction of the flow, compare with the plots shown in figure 4

Figure 6 shows the time sequence of the instantaneous velocity profile across the pipeline, at

$x = 30$  m,  $u(x = 30, y)$ , for  $t = 3.5, 3.6, 11.2, 11.3$  s. It can be seen that the maximum velocity established inside the larger pipe of the considered pipeline is  $\approx 10$  m/s.



**Figure 6.** Change of the  $x$  component of velocity vector across the pipeline, at  $x = 30$  m,  $u(x = 30, y)$ , for  $t = (3.5, 3.6, 11.2, 11.3)$  s. The consecutive plots show the instantaneous velocity profile established in the pipe. Note that the maximum velocity is  $\approx 10$  m/s.

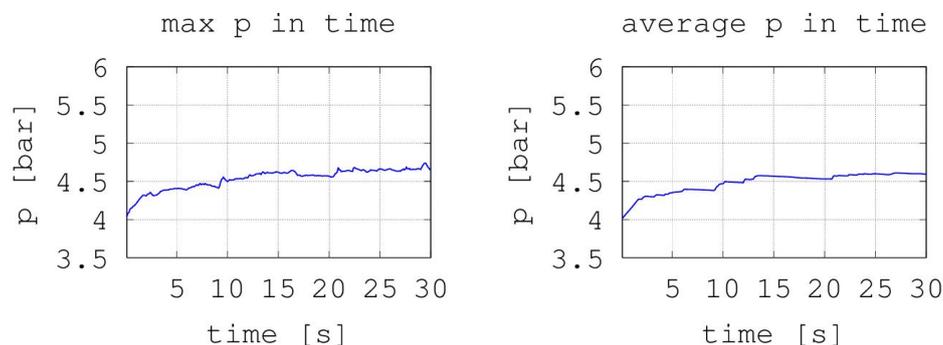
### 3.2. Pipeline with two changes of diameter

The second considered pipeline consisted of three pipes with different diameters:  $L_1 = 80$  m,  $L_2 = 255$  m,  $L_3 = 55$  m and  $d_1 = 267$  mm,  $d_2 = 214$  mm, and  $d_3 = 135$  mm, respectively. In the case of this pipeline, the diameters of the safety valves were restricted to  $d_v = 14$  mm and they were located at each end of the pipeline. In the case of emergency, both valves were designed to be open after the pressure reached 4 bar a.

This case differs from the previous one by way of much larger diameters for individual pipes, hence, the increase of cold He in the system, and also in the relatively small diameter of the safety valves, as compared to the general sizes of the pipes.

Similar to the previous case, it was assumed that at the beginning, the pipeline had uniform pressure  $p_{IC} = 4$  bar a, two open safety valves, and that the heat flux was delivered according to the characteristics from figure 2.

Figure 7 shows the pressure build-up in time. It can be seen that the safety valves ensured that the pressure never rose above 4.75 bar a. However, opposite to the previous case, the pressure remained high for a longer time (wide plateau in the figure). After 42 s, the maximum pressure dropped below 4.5 bar a (not show in the figure).



**Figure 7.** The change of pressure in time for the second considered pipeline equipped with the  $d_v = 14$  mm diameter safety valves.

#### 4. Conclusions

This work has presented a generic approach for the evaluation of the sizes of the pipelines and safety valves of a large cryogenic installation. The main virtue was found in the usage of the 2D numerical model, which was much faster when compared to the 3D model, and much more accurate and informative when compared to the zero- or one-dimensional model.

The transformation of the original 3D geometry into simplified numerical geometry was demonstrated, in order to solve the problem using the appropriate 2D mathematical model.

The proposed transformation kept the geometrical and flow similarities and ensured the preservation of the characteristic numbers (especially: Reynolds number, Peclet number, and Grashof number). For the sake of the presented analysis, a compressible Navier-Stokes model with ideal gas law was used.

The proposed numerical approach can be seen as a tool to help with the design process of any cryogenic installation. Its main benefits are: fast calculation time, geometrical flexibility, with the possibility to use more complex mathematical models rather than oversimplified zero-dimensional models. More importantly, it can reduce additional costs which result from the overestimation of the sizes of the pipelines for cryogenic installations.

#### Acknowledgments

The work has been supported by statutory funds from the Polish Ministry for Science and Higher Education, Statutory Founding 2014/2015, S40104/K0901.

#### References

- [1] Chorowski M, Fydrych J, Modlinski Z, Polinski J, Tavian L and Wach J 2011 *Proceedings of the ICEC 23-ICMC 2010, Oficyna Wydawnicza Politechniki Wroclawskiej* 879–884
- [2] Chorowski M, Malecha Z M and Polinski J 2015 *IOP Conference Series: Materials Science and Engineering* **101** 012160
- [3] Malecha Z, Chorowski M and Polinski J 2013 *Cryogenics* **57** 181
- [4] OpenFOAM 2009 *The open source CFD toolbox user guide* (Free Software Foundation)
- [5] Chini G, Malecha Z and Dreeben T 2014 *J. Fluid Mech* **744** 327
- [6] Ferziger J and Peric M 1999 *Computational Methods for Fluid Dynamics* (Germany: Springer)
- [7] Issa R, Gosman A and Watkins A 1986 *J. Comput. Phys.* **62** 66
- [8] Batchelor G 1967 *Introduction to fluid dynamics* (Cambridge: Cambridge University Press)