EUROPEAN ORGANIZATION FOR NUCLEAR RESEARCH
European Laboratory for Particle Physics

Large Hadron Collider Project

LHC Project Report 859

EXPERIMENTAL VALIDATION OF THE LHC HELIUM RELIEF SYSTEM FLOW MODELING

M. Chorowski[1], J. Fydrych[1], G. Riddone[2]

Abstract

In case of simultaneous resistive transitions in a whole sector of magnets in the Large Hadron Collider, the helium would be vented from the cold masses to a dedicated recovery system. During the discharge the cold helium will eventually enter a pipe at room temperature. During the first period of the flow the helium will be heated intensely due to the pipe heat capacity. To study the changes of the helium thermodynamic and flow parameters we have simulated numerically the most critical flow cases. To verify and validate numerical results, a dedicated laboratory test rig representing the helium relief system has been designed and commissioned. Both numerical and experimental results allow us to determine the distributions of the helium parameters along the pipes as well as mechanical strains and stresses.

1 Wroclaw University of Technology, Wroclaw, Poland
2 CERN, Accelerator Technology Department, Geneva, Switzerland

Presented at the CEC-ICMC'05 Conference
29 August-2 September 2005, Keystone, Colorado, USA

CERN
CH - 1211 Geneva 23
Switzerland

Geneva, 11/01/2006

EXPERIMENTAL VALIDATION OF THE LHC HELIUM RELIEF SYSTEM FLOW MODELING

M. Chorowski[1], J. Fydrych[1], G. Riddone[2]

[1]Wroclaw University of Technology,
Wroclaw, Poland

[2]Accelerator Technology Department, CERN,
1211 Geneva 23, Switzerland

ABSTRACT

In case of simultaneous resistive transitions in a whole sector of magnets in the Large Hadron Collider, the helium would be vented from the cold masses to a dedicated recovery system. During the discharge the cold helium will eventually enter a pipe at room temperature. During the first period of the flow the helium will be heated intensely due to the pipe heat capacity. To study the changes of the helium thermodynamic and flow parameters we have simulated numerically the most critical flow cases. To verify and validate numerical results, a dedicated laboratory test rig representing the helium relief system has been designed and commissioned. Both numerical and experimental results allow us to determine the distributions of the helium parameters along the pipes as well as mechanical strains and stresses.

KEYWORDS: cryogen relieve system, thermal flow modeling,
PACS: 47.11.+j, 44.15.+a

INTRODUCTION

The Large Hadron Collider accelerator, presently under construction at CERN, will make extensive use of superconducting magnets located in an underground tunnel about 27 km in length and divided into eight sectors, each containing about 7300 kg of helium. In case of simultaneous resistive transitions of all the magnets in one sector, the magnetic energy of about 1.5 GJ will be dissipated in the coil and partly transferred to the helium. The helium will be vented from the cold masses to the cold recovery header D in the cryogenic distribution line QRL, and further it will be partially relieved via the pressure valve PV and quench line QL to two buffer volumes, each composed of four 250-m^3

medium-pressure tanks located at ground level at both extremities of the sector (FIGURE 1). There also exists the possibility of discharging the helium from header D via a parallel path composed of safety valve SV or two bursting disks BD and the vent line VL, which directly opens to the environment. In both cases the helium leaving the header D enters a pipe at room temperature. During the first period of the flow, helium will be heated intensely due to the pipe heat capacity. To study the changes of the helium thermodynamic and flow parameters we have simulated numerically the most critical flow cases [1].

The numerical models of helium flows from header D through the QL and VL lines have been developed using the ANSYS 7.1 Code [2] and the Computational Fluid Dynamic module FLOTRAN CFD. In the model the lines QL and VL were straightened hydraulically and all the flow obstructions were replaced by equivalent lengths of straight pipes. The medium pressure tanks were replaced with one tank of equivalent volume capacity.

The finite element method has been applied to solve the set of the three-dimensional equations of mass, momentum, energy and turbulence transports (k-ε model). Helium mass flows and heat transfers were considered simultaneously. For the numerical simulations the FLUID-141 element with axis-symmetry option was chosen, and the flows were modelled as transient (unsteady), turbulent, thermal and compressible. For helium property calculations, the FLOTRAN gas model was applied, and the gas model checked with the helium property values obtained from HePak version 3.4 [3] with the maximum deviation less than 2 %.

The results of the numerical models allow determination of the helium pressure, temperature, density and velocity distributions along the lines as well as the evolution of mass-flow rate. FIGURE 2 presents the helium pressure and temperature distribution along QL after PV opening. The calculated results include some uncertainty because of the complex geometry of the relief system, the very large aspect ratio (length and diameter) of the pipe, and transient as well as turbulent character of the flows. To validate the mathematical model an experimental verification was carried out.

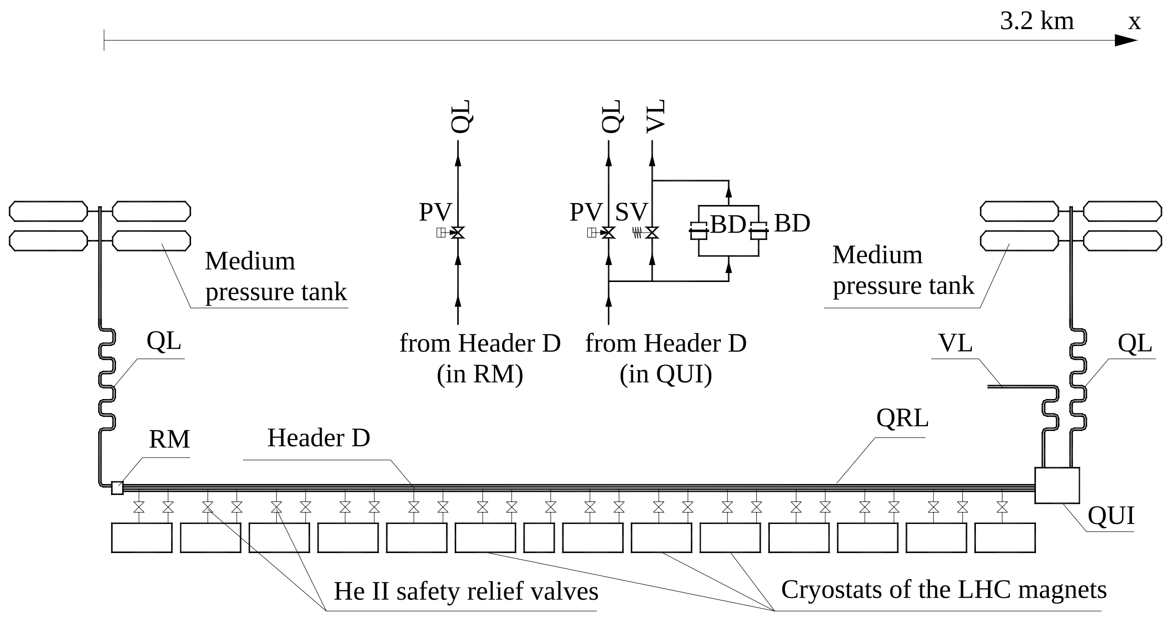


FIGURE 1. Helium relief system scheme.

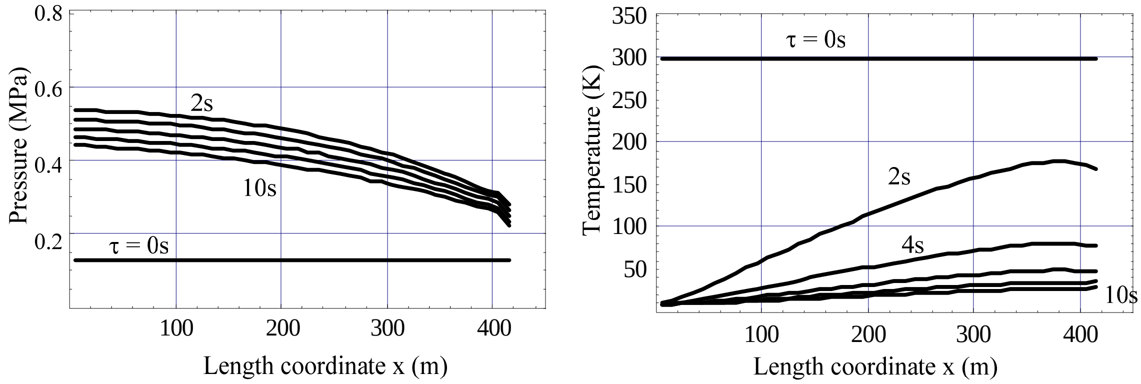


FIGURE 2. Pressure and temperature distribution along QL after the opening of PV.

METHODOLOGY OF THE VALIDATION

The verification of the mathematical modeling was done under laboratory conditions. A direct validation method implies the use of cold helium vapour, thus leading to some technical complications, such as manufacturing of a helium cryogenic vessel to withstand very high pressure, and assuring similar heat transfer condition. Therefore, we decided to validate the mathematical model by building a dedicated test-rig representing the helium relief system and by using cold nitrogen vapour. For our study we applied the following four-step verification methodology:

Step 1: experimental investigation of cold nitrogen vapor flow through a small scale VL or QL. The chosen scale was 1 to 15 to have the same length to diameter ratio as for the LHC,

Step 2: numerical calculation of the cold nitrogen vapor flow through the small scale model (the same numerical model used for the helium flow analysis),

Step 3: comparison of the numerical and experimental results,

Step 4: result analysis and conclusion.

For the helium flow in the LHC discharge system and for the nitrogen flow in the test rig density-wave oscillations are not expected because the fluids are quite far from the two-phase and supercritical states. Thermo-acoustic oscillations will also not occur due to the large pipe diameter (in case of the LHC discharge system), and too small temperature ratio between the ends of the pipe (in case of the nitrogen flow in the test rig) [4].

EXPERIMENTAL TEST RIG

The main part of the experimental test rig is a simplified model representing of the helium relief system (see FIGURE 3), scaled 1 to 15. The model consists of a quick-acting valve, a long copper pipe and a pressure tank. The pipe length and inner diameter are 25 m and 10 mm, respectively, and its ratio is similar to that in the real relief system. The pipe outlet is connected to a 180-dm^3 medium-pressure tank or it may be open directly to the atmosphere, simulating the flows through QL (to medium-pressure tanks) and through VL (to the atmosphere). The basic source of cryogen vapour for the test rig is a standard liquid-nitrogen pressure vessel equipped with an inner evaporator and a pressure control system. The test rig can also be supplied with other cryogens.

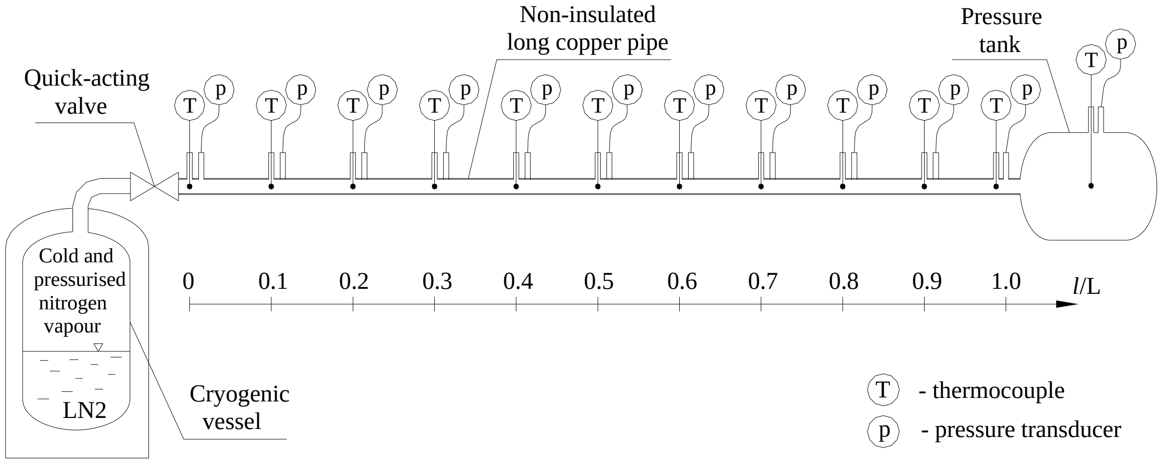


FIGURE 3. Test rig schematic diagram.

To reduce the dimensions of the test rig, the pipe was formed into a coil with a diameter almost 70 times bigger than the pipe inner diameter (see FIGURE 4). Because of the high value of the diameter to length ratio, the influence of the pipe curvature on the flow can be neglected. The pipe and the tank are equipped with temperature and pressure sensors as shown in FIGURE 3. This instrumentation enables measurement of the temperature and pressure evolutions and profiles.

A digital camcorder (see FIGURE 4) records the phenomena which occur on the outer surface of the pipe coil during the cryogen flow. The obtained sets of film frames can be used for the visualisation of the process of moisture condensation and solidification.

EXPERIMENTAL AND NUMERICAL RESULTS

During the experimental investigation the pressure and temperature values were measured every second in each selected cross section. The initial pressure and temperature of the vapour were equal to 1.0 MPa and 130 K, and the outlet of the pipe was open to atmosphere (see FIGURE 5).

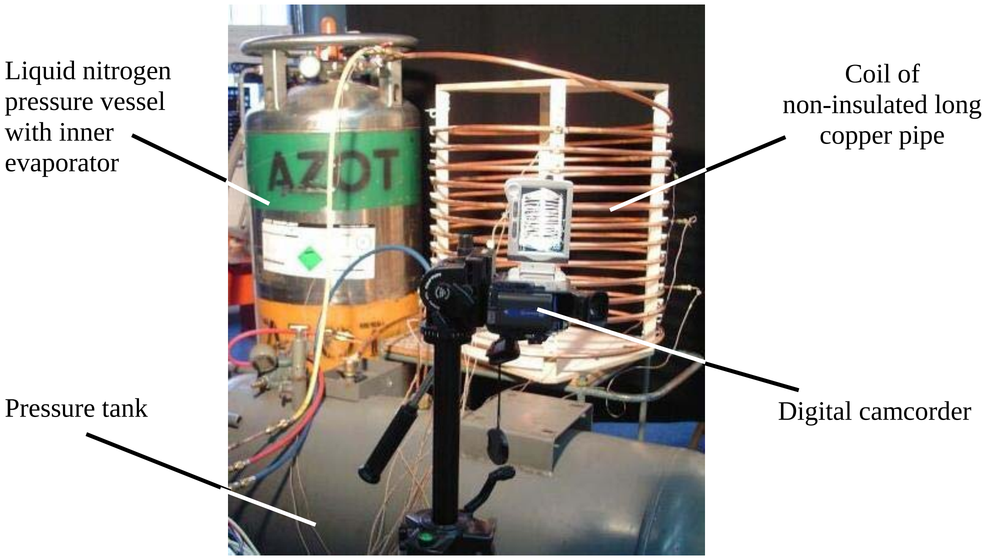


FIGURE 4. View of the test rig.

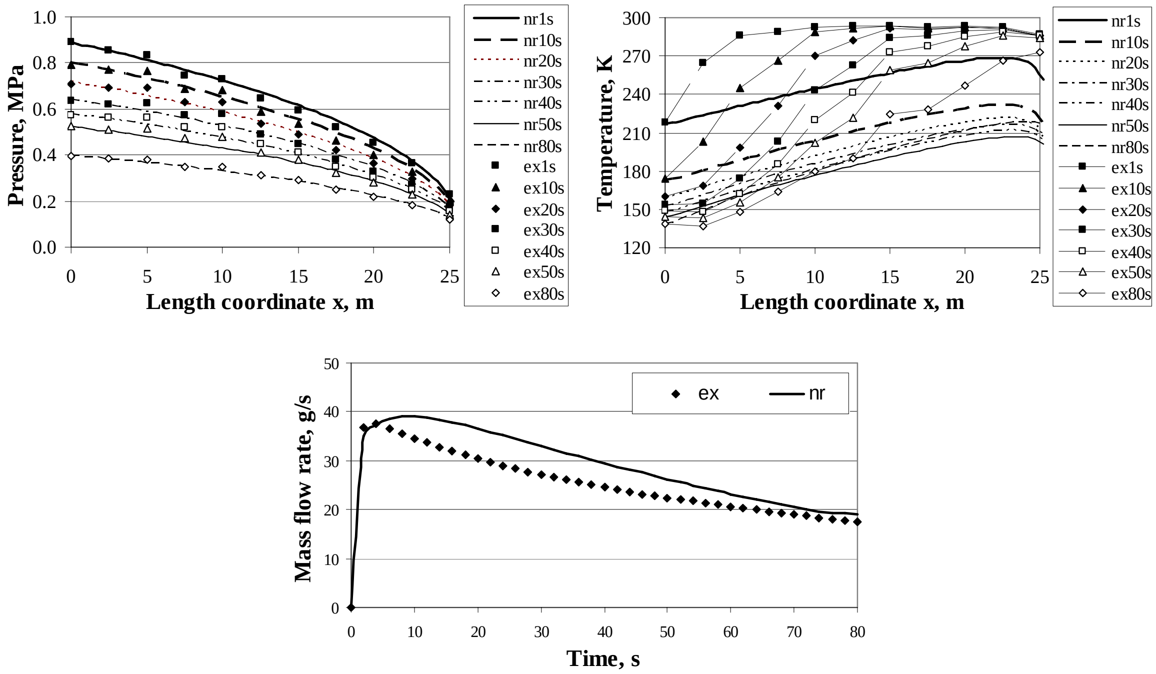


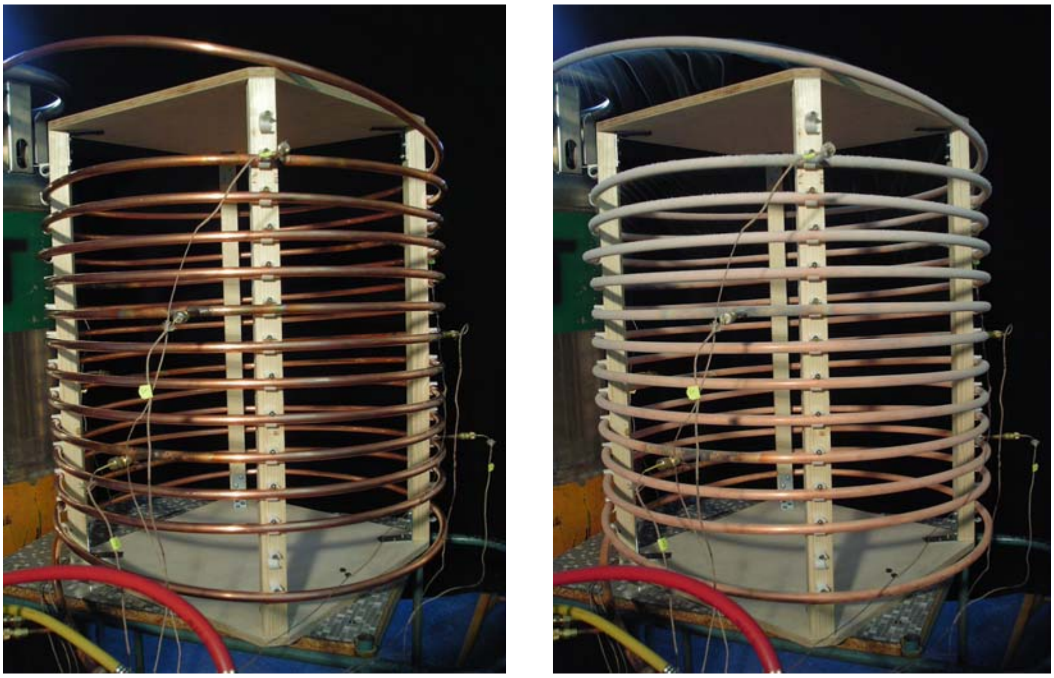
FIGURE 5. Pressure and temperature distributions along the small-scale VL and the mass flow rate evolution; comparison of numerical (nr) to experimental results (ex).

The obtained pressure and temperature evolutions at the inlet of the pipe were applied as transient boundary conditions in the corresponding numerical model. The model geometry was composed of a straight long pipe of diameter and length equal to 10 mm and 25 m, respectively. The flow area was divided into 850 elements. On each node the initial pressure and temperature were 0.1 MPa and 293 K. The bulk temperature was equal to 293 K and the convective heat-transfer coefficient (film coefficient) was assumed equal to 15 $W/m^2 \cdot K$.

The model was solved using the ANSYS 7.1 FLOTRAN CFD Code as transient (unsteady), turbulent, thermal and compressible flow of cold nitrogen vapours. The obtained numerical results for different times are presented in FIGURE 5 and compared to the experimental results. The comparison shows a good agreement between the numerical and experimental pressure distributions. The shape of the pressure curves is not linear and reflects the tendency of the measured values.

The temperature distributions are not in such good agreement as the pressure distributions. The measured temperature profiles rise quicker than the corresponding numerical profiles. In the first second of the discharge the nitrogen temperature reaches 293 K close to the inlet of the pipe, whilst the numerical profile is much more flat and increases only to 270 K near the pipe outlet. Note that the numerical and experimental temperature values approach each other for the following profiles. The existing difference indicates that the heat-transfer mechanism is more complicated than assumed in the numerical model. The model took into account only natural convection on the outer surface of the pipe, omitted the process of moisture condensation and solidification on the pipe during the cool-down of the pipe (see FIGURE 6), and omitted entry-length effects.

The numerical mass-flow rate evolution is slightly above the experimental results. It may signify that the flow resistance in the numerical analysis is lower than in the experiment. If the cryogen temperature is higher, the velocity will be higher as well, and the flow rate will be lower. This confirms that the heat inleaks to the flowing cryogen have to be simulated more precisely.



t = 0 s t = 80 s

FIGURE 6. Frost formation on the small-scale pipe during the cryogen vapor discharge.

Three main phases of heat flow to cold nitrogen vapour inside the non-insulated pipe can be qualitatively identified (see FIGURE 7). Phase I occurs at the beginning when cold cryogen vapour flows into the pipe. In this phase the vapour removes heat from the pipe wall, which depends on forced convection coefficient α_i and pipe wall material thermal conductivity λ_w. Phase II begins when the pipe is frost-free and the pipe wall is partly cooled down. The natural convection coefficient α_o starts then to be as important as α_i and λ_w. The time of this phase depends strongly on the ambient air humidity. If the humidity in the air increases, the time decreases. When the frost layer appears on the pipe then phase III begins. During this phase the frost thermal conductivity λ_f takes part in the heat flow mechanism. The heat exchange from ambient air meets the highest resistance and the heat flux is much lower than during the phases I and II.

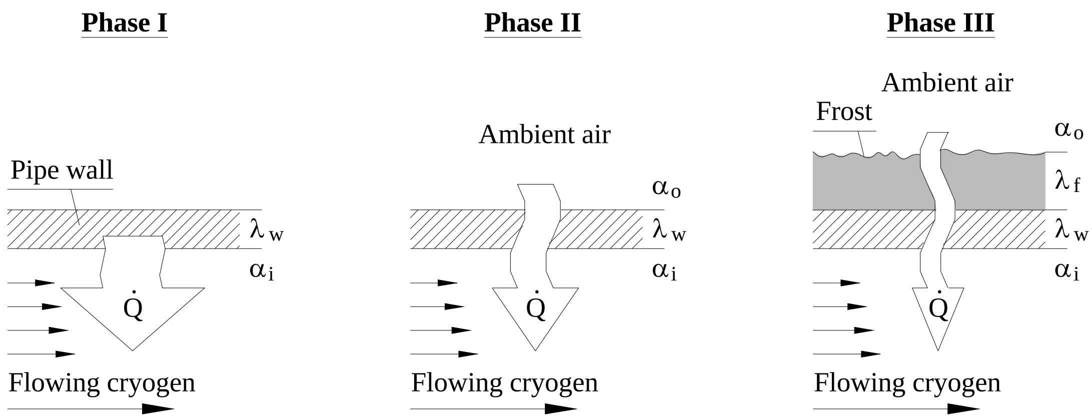


FIGURE 7. Phases of heat flow to cryogen vapour flowing in a non-insulated pipe.

Because the heat flow mechanisms depends on many factors (α_i, λ_w, λ_f, α_o, pipe wall and frost layer thickness, air humidity, etc.), which are function of time and current local temperature, a very exact simulation of the transient thermal boundary condition is extremely time consuming and not applicable in the analyzed of thermal-flow issue. Therefore some approximations of the transient thermal boundary conditions have been taken into account for the improvement of the numerical model.

At the beginning of phase I the most important factor is α_i, which can reach values of about 1500 – 2000 W/m^2·K [5]. On the other hand, in phase III the coefficient α_o has the greatest influence on the heat flux. In the mathematical model the simplified thermal boundary condition can be a function of time in such a way that at the beginning of the process the film coefficient is equal to $\alpha_i \approx 1700$ W/m^2·K and then successively reduces to $\alpha_o \approx 15$ W/m^2·K.

The numerical results are shown in FIGURE 8. In this simulation the film coefficient was steadily decreasing from α_i to α_o, with the rate that after 80 s from the beginning of the process it was equal to 500 W/m^2·K. The improvement of the numerical temperature profiles matching to experimental temperature values is remarkable. The profiles are not perfectly reflecting the measured values but the matching is much better than shown in FIGURE 5. In the first second of the simulation the fitting of numerical results to real temperature values is perfect. This confirms the assumption that in the first phase of heat inflow the forced convection coefficient α_i is the most important factor.

The comparison of pressure profiles in FIGURES 5 and 8 show that the pressure distributions are not significantly affected by the film coefficient changes. In the analyzed flows the pressure profiles are not considerably sensitive to thermal boundary conditions. They depend on the difference between inlet and outlet pressures, the pipe aspect ratio and friction coefficient.

The mass-flow rate evolution shown in FIGURE 8 better fits the experimental results except for the first few seconds of the flow. This confirms again that the model with decreasing film coefficient better represents the analyzed flows.

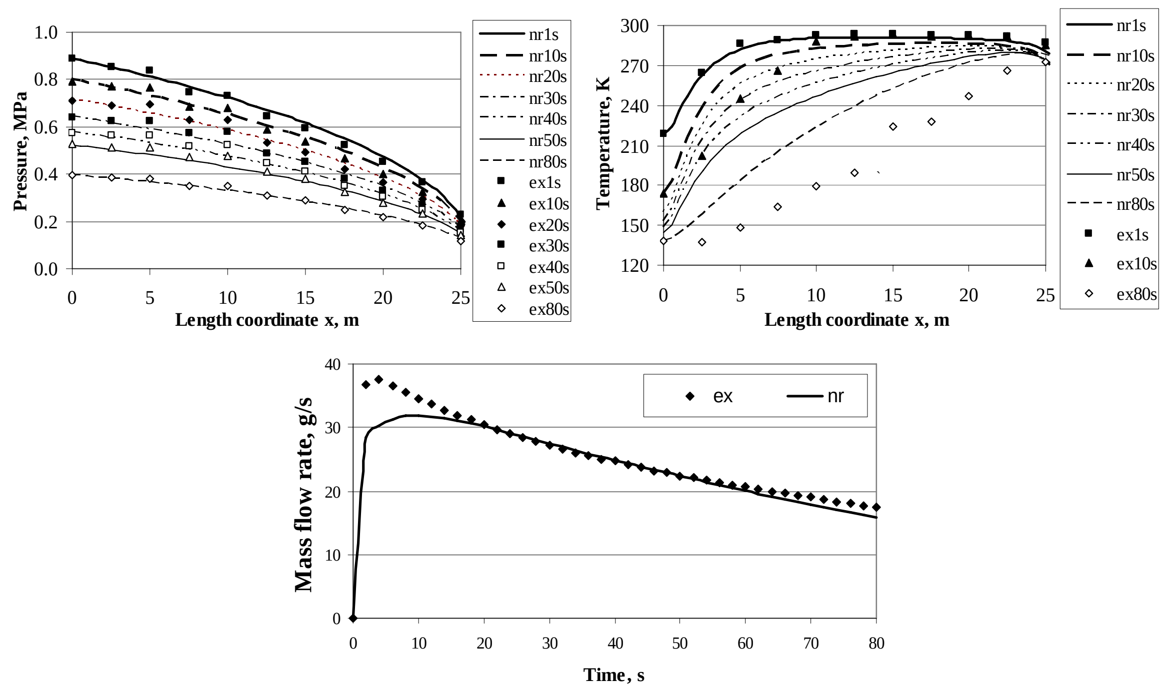


FIGURE 8. Example of pressure and temperature distributions for numerical model with decreasing film coefficient.

CONCLUSIONS

Simultaneous resistive transitions of the magnets in a whole sector of the LHC will cause cold helium to flow through a helium discharge system at room temperature. The thermo-hydraulic processes in the system have been numerically modeled to size the system and guarantee its safe operation. The model has been experimentally verified on a dedicated test rig. The experimental fluid was nitrogen since a direct validation method using cold helium vapour led to some technical complications. The verification showed that the numerical model gave correct pressure distributions, but the temperature profiles reflected much more severe thermal conditions. Therefore, the thermo-mechanical pipe strength calculations made on the basis of these numerical results can be considered as quite conservative.

To reproduce correctly the distribution of the temperature along the test pipe it was necessary to vary the film heat transfer coefficient during the calculations. This enabled one to correct the thermal boundary conditions in the model, which led to more accurate numerical results.

The proposed methodology can be used for the description of any cryogen flow in warm piping.

ACKNOWLEDGEMENTS

The work has been performed within the cooperation agreement K944 signed between CERN, Geneva and Wroclaw University of Technology, Poland.

REFERENCES

1. Chorowski, M., Fydrych, J., Riddone, G., "Analysis of Thermo-mechanical Pipe-strength for the LHC Helium Relief System and Corresponding Helium Flows Following a Resistive Transition of the Magnets," LHC Project Note 370, CERN, Geneva, 2005.
2. ANSYS Release 8.1, Code by ANSYS, Inc., Southpointe 275 Technology Drive, Canonsburg, PA 15317.
3. HEPAK Version 3.4, code by Cryodata, Inc., P.O. Box 173, Louiaville, CO 80027-0173.
4. Hands, B. A., *Cryogenic Engineering,* Academic Press, Inc, London, 1986.
5. Barron, R. F., *Cryogenic Heat Transfer*, Taylor & Francis, Philadelphia, PA, 1999.