Experimental and Numerical Investigation of Cryogenic Two-Phase Flows and Application to Liquid Rocket Propulsion Systems

S. Soller[*], K. Claramunt[**], L. Peveroni[***], N. Pouvesle[#], S. Schmidt[##], J. Steelant[###]

[**] Ariane Group, D-82084 Taufkirchen
[**] Numeca, B-1170 Brussels
[***] Von-Karman Institute for Fluid Dynamics, B-1640 Rhode-St-Genèse
[#] Energie-Technologie GmbH, D-85649 Brunnthal
[##] Technische Universität München, D-85747 Garching
[###] ESTEC, NL-2210 AZ Noordwijk

ABSTRACT

Within a Belgian-German cooperation, research institutes, academia and industry have performed experiments to improve the modelling capabilities for two phase flow effects occurring in liquid rocket engines: cavitation and transient two-phase flow in cooling circuits. Experiments were performed at Energie Technology (D) and VKI (B), modelling activities were performed by ArianeGroup (D), Numeca (B) and TUM. (D) The test data were used to benchmark different engineering and CFD tools.

KEY WORDS

Cryogenic Two-Phase Flow, Cavitation, Chill-Down, Boiling, Liquid Rocket Engine, Hydrogen

SYMBOLS & ABBREVIATIONS

CHT	Conjugate Heat Transfer
ESA	European Space Agency
ESPSS	European Space Propulsion System Simulation
ESTEC	European Space Technology Centre
HEM	Homogeneous Equilibrium Model
LES	Large Eddy Simulation
LH2	Liquid Hydrogen
LN2	Liquid Nitrogen
TVD	Total Variation Diminishing
TUM	Technische Universität München
VKI	Von-Karman-Institute

1 INTRODUCTION

Within a Belgian-German GSTP project, partners from industry and academia have investigated transient two-phase flow phenomena in liquid rocket engines.

In liquid rocket engine thrust chambers, several two-phase flow phenomena can be observed during the transient start-up operation. The experiments and numerical simulations provide a better understanding and a more accurate modelling of the processes dominating the chill-down behaviour of the thrust chamber and its off-design cooling performance.

1.1 Two-Phase Flow in Liquid Rocket Engines

Before ignition, the cryogenic fuel is actually used to chill down the turbomachinery, the valves and the structure of the engine. During this phase, the fuel passes the coolant circuit, continuously cooling down the combustion chamber wall. Entering the coolant channels in liquid state, the fuel is heated up and starts to boil during the passage of the coolant channels. Low-frequency mass flow oscillations with phases of choked gaseous flow are characteristic of this phase. In order to minimise the propellant mass, which is used to condition the engine before start-up and which is effectively lost for propulsion of the launcher, an accurate prediction of the mass flow rate is required.

From tests with the HM-7BB upper stage engine another aspect to be taken into account is known: a too long chill-down duration eventually ended in pronounced temporally combustion instabilities during the ignition and ramp-up transient of the motor, potentially risking flow blockage and pump stall of the engine.

In addition to the transient boiling effects during chill-down, cavitation effects need to be taken into account in the coolant circuit of the engine. For example, HM-7B uses a small amount of mass flow which is tapped off from the main coolant circuit via dedicated orifices to cool the lower part of the nozzle extension. The propellant is dumped at the nozzle exit via miniature nozzles. During engine ignition and start-up, cavitation effects taking place at both orifices (the inlet and the outlet nozzles) limit the mass flow rate being available to cool the wall structure of the nozzle extension. In order to verify that the orifice configuration provides a sufficient high mass flow rate of fuel for cooling the nozzle not only during steady state operation but also during the start-up transient, an accurate modelling of the varying discharge coefficient of these orifices and the phase state of the fuel is required.

1.2 Scope of Activities / Project Setup

The research activities presented in this publication were performed within an ESA GSTP project (Contract No. 4000111616/14/NL/PA). Partners from academia and industry teamed up to contribute with their individual strengths to the project.

AiraneGroup GmbH designs, builds and operates liquid rocket engine thrust chambers like the HM7, Vulcain 2 and the Vinci. To predict the transient behaviour of these engines, different codes are available at ArianeGroup. For transient engine system simulations, the in-house code Smart and the European Space Propulsion System Simulation toolbox ESPSS are in use. Dedicated two-phase simulations are performed with THESEUS, a software package which was derived from a code developed by the German Gesellschaft für Reaktorsicherheit (GRS) [1] and which provides a set of equations to describe the heat transfer and phase change phenomena using a one-dimensional homogeneous equilibrium model. The Institute for Aerodynamics and Fluidmechanics of TUM provides expertise in 3-dimensional and time-resolved modelling of two-phase flow phenomena with its CATUM code. Brussels-based Numeca has developed the multi-physics software package Fine™/Open, providing additional three-dimensional computing capability to the project. The experiments were performed at the Von-Karman-Institute (VKI) for Fluid Dynamics in Rhode-Saint-Genèse and at Energie-Technolgie GmbH (ET) In Brunnthal. While the experiments at VKI were performed using liquid nitrogen as substitute fluid, ET operates facilities to perform the tests with liquid hydrogen.

2 ORIFICE FLOW WITH CAVITATION

Figure 1 shows a detail of HM7 coolant circuit, which was investigated in more detail during the project.

In the coolant inlet manifold, a number of small orifices is used to tap off a certain amount of liquid hydrogen to cool the tube-wall nozzle extension downstream. The coolant mass flow is dumped to the ambient at the end of the coolant passage. In the engine's operating cycle, the amount of coolant flowing to the nozzle extension varies due to a change in the pressure cascade in the system. The orifice geometry and the actual discharge coefficient are additional parameters defining the amount of mass flow, which is available to cool the nozzle extension.

During transient operation the orifice inlet pressure of the coolant circuit is below the critical pressure of hydrogen (p_{crit} = 12,9 bar and T_{crit} = 33 K), and local cavitation may occur in the orifice.

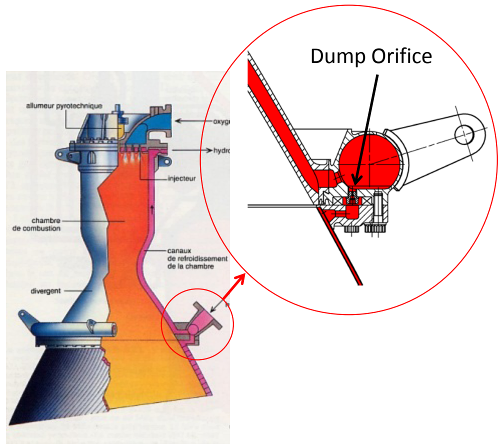


Figure 1: Coolant inlet manifold with dump orifice

This may reduce the coolant mass flow rate in the dump cooled nozzle extension. During steady state operation, the pressure downstream of the orifice is below the critical pressure of hydrogen. Two-phase-flow effects may occur due to the massive expansion of the fluid passing the orifice.

The experiments performed within the project aimed at providing a comprehensive data base to accurately predict the discharge coefficient of the dump orifice during operation.

2.1 Experimental Setup

The tests were performed at ET's facilities in Brunnthal. Figure 2 and Figure 3 illustrate the experimental setup.

The orifice is installed in a tube, which is submerged into a vessel filled with liquid hydrogen. Due to the pressure difference, hydrogen flows upward through the orifice into a settling chamber, from where it is dumped to the ambient. The settling chamber is a cylinder with an inner diameter of 58 mm and a length of 200 mm. Two orifice diameters were tested, 0.5 mm and 0.8 mm respectively. The orifices feature an inlet radius of 0.4 mm.

A hydrogen-operated pressurisation system is used to adjust the feed pressure up a level of 50 bar. Due to the contact with the pressurant gas, which is stored at ambient temperature, the temperature in the liquid hydrogen entering the test section varies approximately between 25 K and 40 K during the test. This allows assessing the effect of the fuel inlet temperature on the pressure drop behaviour of the system. Pressure and temperature are measured in the tube upstream of the orifice and in the settling chamber after the orifice. For the temperature measurement, PT 1000 resistance temperature sensors were used. The mass flow is measured at the exit of the test section. All sensors were sampled at a rate of 100 Hz

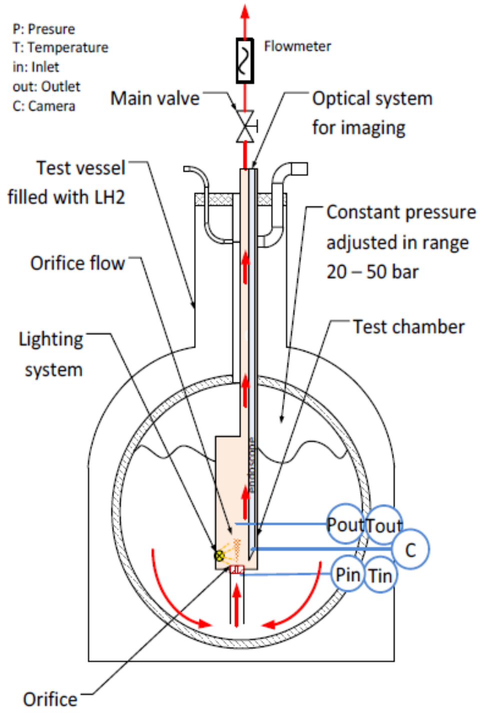


Figure 2: Experimental setup for orifice tests

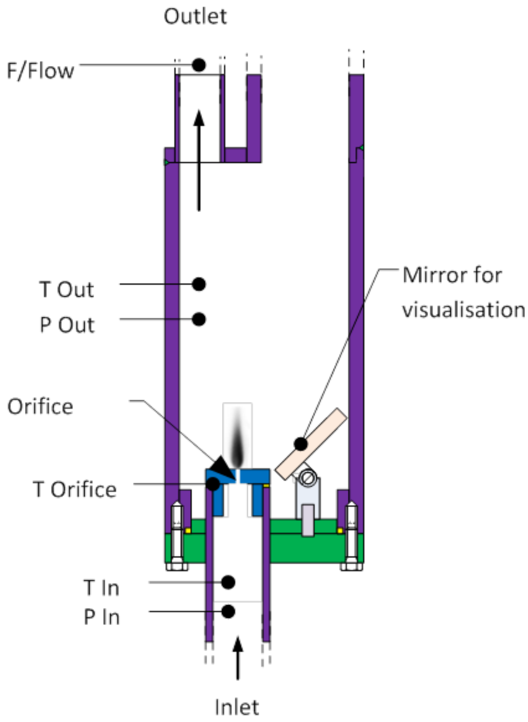


Figure 3: Installation of orifice in experimental setup

Table 1: Operating conditions orifice test

Parameter	**Range**	**Unit**	**Accuracy**
$\dot{m}_{H_2}$	1 - 40	kg/h	± 0,5
$T_{H2,in}$	25 - 35	K	± 1
$T_{H2,out}$	25 - 35	K	± 1
$p_{H2,in}$	25 - 50	bar	± 0.5
$p_{H2,out}$	1 - 10	bar	± 0.5

In addition to the standard measurements of pressure, temperature and mass flow, the experimental setup is equipped with a LED lighting system and an optical system. This was used for backlight imaging of the spray cone during the experiment. Imaging was done with a Imaging Development Systems UI-5240CP-M-GL, providing a resolution of 1280 x 1024 pixels with a frame rate up to 50 Hz. The exposure time can be set in a range from 0,009 ms to 2000 ms. An adjustable valve in the exhaust duct was used to set the backpressure downstream of the orifice to be characterised.

2.2 Modelling Approach

At ArianeGroup, a simplified 0-dimensional model to predict the pressure drop behaviour of cavitating flows is being used. The model is based on the following assumptions:

- Isentropic expansion from total condition p_{in} ,T_{in} to p_{out}
- Real gas properties for para-hydrogen
- Energy balance is used to evaluate the velocity
- Pipe friction is considered as total pressure loss
- Expansion is limited to choked condition if $p_{out} < p_{crit}$ (p_{crit} has to be found iteratively)
- If p_{out} is lower than p_{sat} the Homogeneous Equilibrium Model (HEM) two-phase model is used to evaluate:
    - Vapor fraction x
    - p_{crit}
    - Mixture velocity w_m
    - Mixture sound speed c_m

At TUM, software for simulation of cavitating flows has been developed and applied since more than 10 years. One output is the CFD code "CATUM" (Cavitation Technische Universität München), which was applied within this project. The components of this code are shortly summarized in the following list:

- 3-D finite volume method operating on body-fitted hexahedral block-structured grids (grid generator ICEM)
- 4-stage 2nd order low-storage Runge-Kutta time integration
- Low Mach number consistent Riemann solver with 2nd order reconstruction (total variation diminishing/TVD limiters)
- Conservative form of fully compressible governing equations (here: Euler equations)

- Tabulated thermodynamics (here: parahydrogen), thermodynamic equilibrium assumption, tables generated with REFPROP, 3000² resolution
- Single fluid formulation (just one set of governing equations, no reconstruction of phase boundaries)

Within the project, Numeca used its Fine™/Open CFD solver with equations of state for para-hydrogen in the range from subcritical to supercritical conditions. The modelling approach applied is characterized by the following elements:

- Navier-Stokes (preconditioning for low-Mach)
- Turbulence model: High-Reynolds k-ε model with wall functions
- Thermo-tables for parahydrogen - LH2 (equilibrium), based on NIST/Refprop. The equation of state (EOS) used is a Helmholtz equation [1]
- Hakimi preconditioning adjusted for phase transitions [4]
- Heat conduction solved in the solid domains and heat flux transferred between fluid and solid domains by means of a conjugate heat transfer (CHT) connection.

Figure 4 illustrates the CFD grid used by Numeca. The computational domain covers the inflow section and the settling chamber downstream of the orifice.

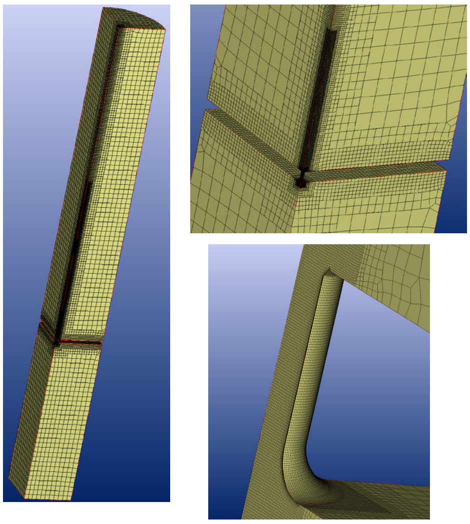


Figure 4: CFD mesh used with Fine™/Open

As the simulations of TUM indicated that the inlet radius visible in Figure 4 may have a significant impact on the cavitation behaviour of the flow in the orifice, this radius was measured with a microscope on manufacturing samples.

2.3 Results & Discussion

The tests varied the orifice diameter (0.5 mm and 0.8 mm), the inlet pressure (25 bar and 50 bar) and the hydrogen inlet temperature (varying from 25 K to 40 K).

Figure 5 shows still images taken during a test with an inlet pressure of p_{In} = 50 bar and an outlet pressure of p_{Out} = 1.4 bar. The obvious change in the jet topology correlates with the increase in vapour mass fraction predicted by the engineering model in Figure 6. A more detailed analysis was performed by TUM by simulating the effect of a change in hydrogen temperature from 25 K to 30 K. *Figure 7* shows the expected vapour fraction in the orifice and in the jet exiting the orifice. The effect of the increased hydrogen temperature on the cavitation zone within the orifice passage itself and on the jet plume downstream of the orifice is clearly visible.

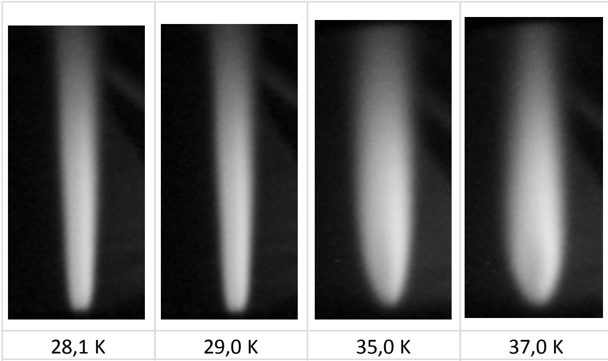


Figure 5: Influence of temperature on jet topology

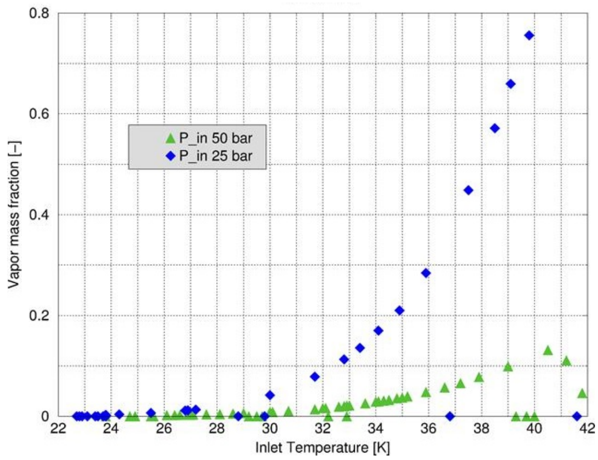


Figure 6: Vapour mass fraction as function of temperature for different inlet pressure levels

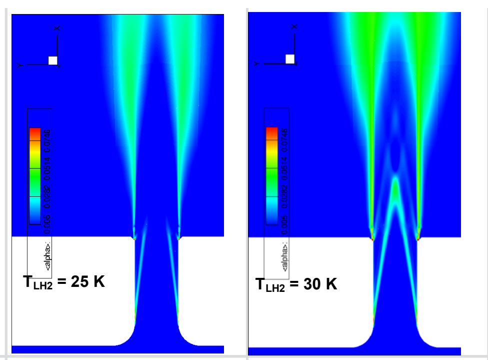


Figure 7: CATUM simulation of vapour fraction (pin = 50 bar, pout = 5 bar)

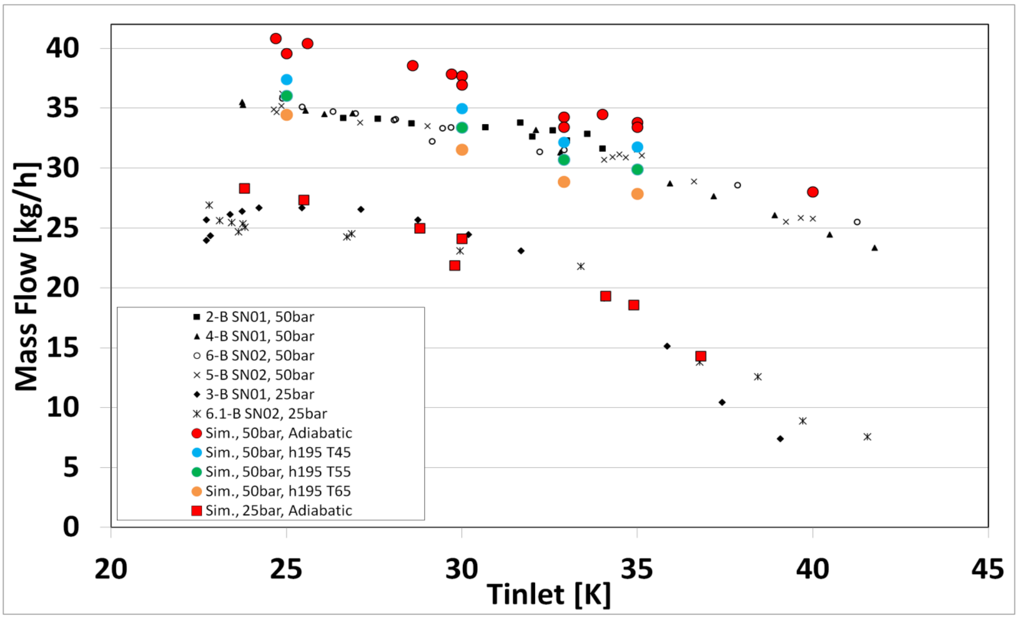


Figure 8: Comparison of Numeca's Fine™/Open simulation results with experimental data

The diagram in Figure 8 compares the simulation results obtained with the Fine™/Open model with the data recorded during the experiments. On the ordinate axis, the measured mass flow rate is plotted against the fluid inlet temperature on the abscissa. The experimental data (black symbols) comprise tests with 50 bar inlet pressure as well as with 25 bar inlet pressure, resulting in two distinct groups of data in the diagram.

For the test case with 25 bar inlet pressure, the simulation results (square red symbols) match exactly the decreasing mass flow rate with increasing inlet temperature of the hydrogen. For the test case with 50 bar inlet pressure (circle red symbols) however, there is a distinct difference for the simulation results from the experimental test data for inlet temperatures below 32 K. The same difference between simulation and test was observed when comparing the experimental data to the simulation results of TUM and ArianeGroup, indicating that some additional effect of the experimental setup influences the test data.

To better understand potential influences not taken into account in the CFD models, Numeca performed additional simulations to consider conjugate heat transfer (CHT) phenomena in the inlet tube and the test chamber itself. As indicated in Figure 2, the liquid level in the vessel reduces continuously during the test. Thus, the test chamber and the feeding tube are exposed to hydrogen gas at a significant higher temperature. A series of simulations was performed, assuming different gas temperatures and a reduced liquid level as illustrated in Figure 9. Blue, green and orange circle symbols in Figure 8 show simulation results for assumed gas temperature of 45, 55 and 65 K, respectively. Figure 10 exemplarily shows how heat transfer phenomena may affect the temperature of the liquid hydrogen being ejected from the orifice. With these simulations it could be shown that such heat transfer phenomena are the most likely root cause of the differences observed between experimental data and simulation results.

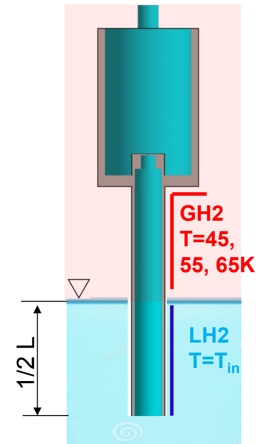


Figure 9: Schematic of reduced liquid level during test

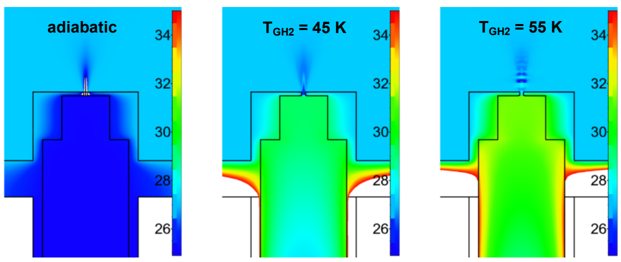


Figure 10: Effect of heat transfer on fluid temperature

3 COOLING CHANNEL CHILL-DOWN

During the chill-down-process of liquid propellant engines, a small mass flow rate of fuel is fed to the coolant circuit of the thrust chamber to reduce the structural temperature to a level, which allows mastering the mass flow rate through this passage during the start-up transient.

Being able to accurately predict the transient two-phase flow behaviour is essential to assess the behaviour of the thrust chamber during this transient process. The two-phase-flow caused by the transient boiling of liquid in the coolant circuit can cause pressure and mass flow oscillations, which influences the structural chill-down. Two-phase processes during LOX-dome priming can affect the operation of the injector. As a consequence, this may result in low-frequency oscillations (chugging) during the ignition transient.

The experimental setup shall be designed to provide benchmarking data for transient modelling tools.

3.1 Experimental Setup

The experimental setup to investigate the coolant channel chill-down was designed to be operated with liquid nitrogen LN2 as well as with liquid hydrogen LH2. The LN2 tests were performed at the Cryogenic Chilldown Experimental Facility (CHIEF) of VKI at Rhode-Saint-Genèse. Detailed results have been published by Peveroni et al.[9]. The LH2 tests are currently being performed at ET in Brunnthal.

Table 2 compares the operating conditions of the application case with the characteristics of the two test facilities and shows how these affect the relevant similarity parameters like the Weber, the Reynolds, the Prandtl and Jakob number. The design of the experiment and the seletced operating conditions allow comparing the results from the different test facilities directly.

In both test facilities, the cooling channel setup was installed in an evacuated chamber to minimise the convective heat transfer from the ambient to the channel. Figure 11 shows this vacuum chamber and the cooling channel as used at VKI. The cooling channel section is made of copper. Two quartz window inserts for optical access allow for monitoring the void fraction during the experiment.

In both facilities, the pressure drop along the test section is measured via pressure transducers at the inlet and the outlet of the channel. The evolution of the coolant channel wall temperature is measured at various axial positions.

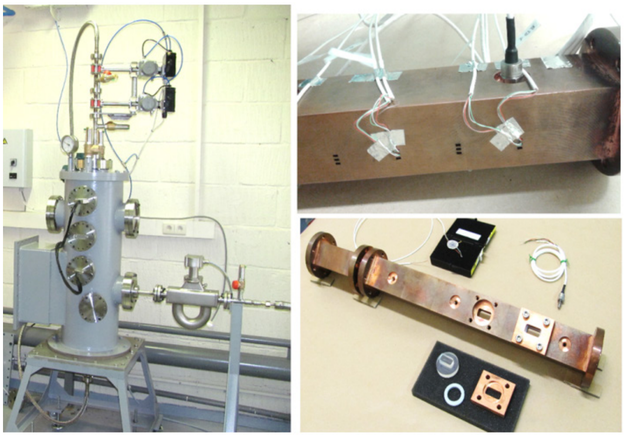


Figure 11: Cryogenic Chilldown Experimental Facility and cooling channel experiment used at VKI

Table 2: Operating conditions and similarity parameters chill down test [7]

Facility	Working fluid	Test section size [mm]	Represent. conditions [bar], [K]	Mass flow rate [Kg/s]	Weber Number [-]	Reynolds Number [-]	Boiling Efficiency [%]	Prandtl Number [-]	Jakob Number [-]
Real Engine	LH_2	4.65 x 1.75	Range: [1.5 3.2] bar [22 25] K Selected value: 2.65 bar 23 K	0.00154	Range: [800 1160] Selected value: **1009**	Range: [4.05 5.1] 10^4 Selected value: **4.4 10^4**	Range: [0.12 0.14] Selected value: **0.13**	**1.2**	if $T_w = 109$ K **2.3**
CHIEF	LN_2	9.3 x 3.5	Range: [2 6] bar [80 96] K Selected value: **5 bar** **94 K**	Range: [0.025 0.037] Selected value: **0.027**	Range: [800 1160] Selected value: **1009**	Range: [2.2 6] 10^4 Selected value: **4.6 10^4**	Range: [0.09 0.26] Selected value: **0.16**	Range: [1.75 2.13] Selected value: **1.8**	$T_w = 293$ K Range: [2.1 2.8] Selected value: **2.3**
				Range: [0.017 0.027] Selected value: **0.019**	Range: [360 530] Selected value: **462**	Range: [1.5 4] 10^4 Selected value: **3.4 10^4**	Range: [0.13 0.4] Selected value: **0.24**		
ET Technology	LH_2	10 x 3.8	Range: [1.5 3.2] bar [22 25] K Selected value: 2.65 bar 23 K	0.0033	Range: [360 530] Selected value: **462**	Range: [4.06 5.06] 10^4 Selected value: **4.38 10^4**	Range: [0.25 0.32] Selected value: **0.29**	**1.2**	if $T_w = 109$ K **2.3**

3.2 Modelling Approach

ArianeGroup uses a one-dimensional engineering model to predict the transient behaviour of propulsion systems subjected to phase-change phenomenal called THESEUS (Thermal Equation Solver for Engines Used in Spacecraft). This code was developed for using hydrogen as coolant with support of the German Gesellschaft für Reaktorsicherheit, GRS, based on its ATHLET code [5]. In the past, it was applied to the cooling circuit of the HM-7B rocket thrust chamber [6]. The code uses a basic 4-equation system to describe the conversation of liquid and vapour mass as well as the mixture momentum and the mixture energy. The code uses a lumped parameter method to solve mass and energy equations using the finite volume technique. The fluid flow equations are integrated spatially for each control volume. To close the resulting set of ordinary differential equations for mass, momentum and energy, the following set of equations is used:

- Wolfert model for the mass and energy transfer between the phases
- Drift flux model for the relative velocity of gas and liquid
- Modified Martinelli/Nelson model for two-phase pressure drop
- A set of various correlations describing the wall heat transfer coefficient

In recent years, the THESEUS code was coupled to the finite element program CalculiX [2] in order to be able to describe more accurately the chill-down process of liquid propulsion components like manifolds or injector domes. For the experiments at VKI, the THESEUS property data base was expanded to also cover nitrogen. Additionally, storable propellants have been added to the data base in the past.

Figure 12 shows the Theseus model of the CHIEF setup. The CalculiX tool has only been applied to the copper coolant channel structure.

In parallel to the well-established THESEUS model, VKI started updating the current version of the European Space Propulsion System Simulation toolkit (ESPSS) to simulate transient two-phase flow phenomena with phase change [7], [8].

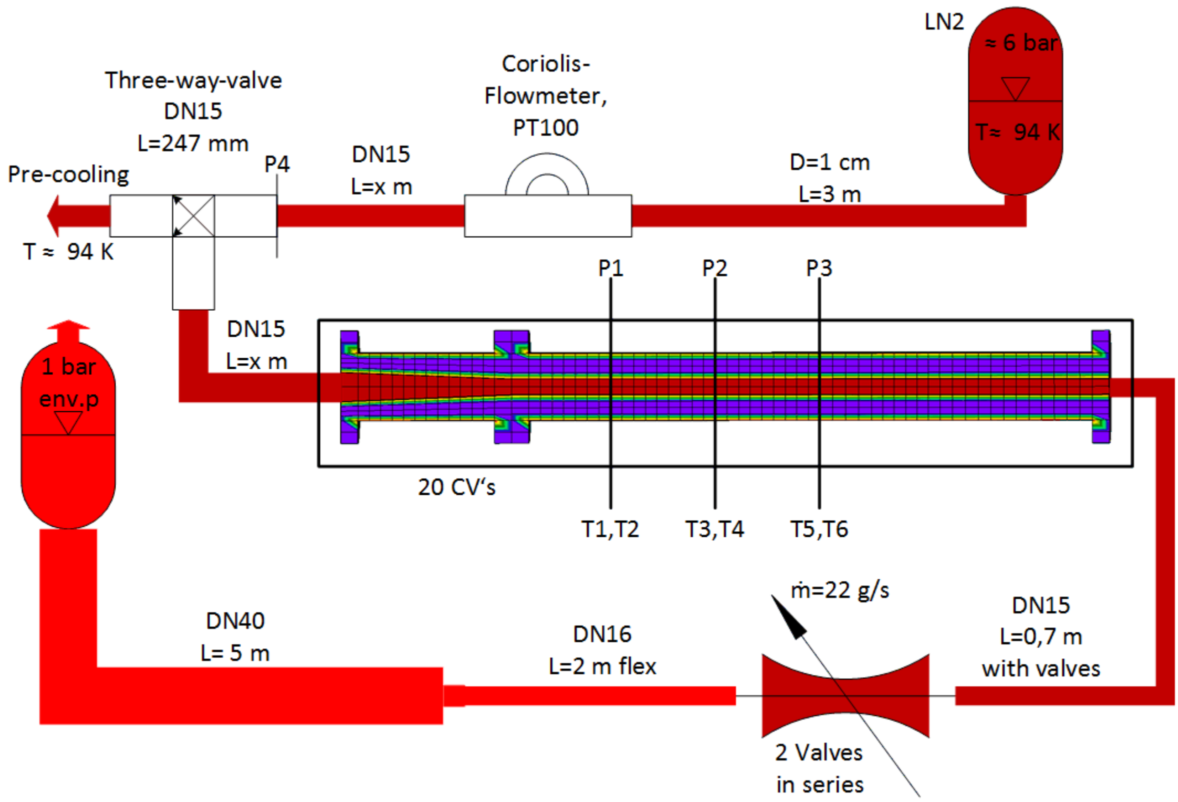
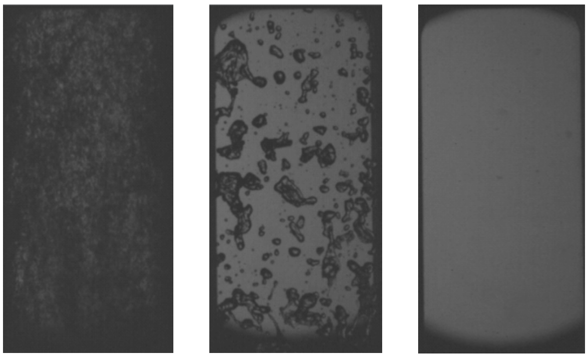


Figure 12: THESEUS model of the CHIEF setup

3.3 Results and Discussion

The experimental data recorded during the tests at VKI are discussed in detail in [9] and [10]. During the chilldown experiment, all two-phase flow regimes can be observed at the quartz windows. Figure 13 shows a typical image of mist flow, disperse droplet flow and gas pocket observed during the first phase of the experiment.



(a) (b) (c)

Figure 13: Flow observed during the inception phase: mist flow (a), disperse droplet flow (b), gas pocket (c).

The experimental conditions, which were used in the experiments, are given in Table 3. Figure 14 to Figure 16 show results obtained at CHIEF with mass flow rates of 10 g/s (black), 20 g/s (green) and 30 g/s (violet), respectively. The diagrams show (from top to bottom) the evolution of the pressure, the mass flow rate and the temperature at the end of the test section.

Table 3: Test conditions chill down test

Test ID	**# 1**	**# 2**	**# 3**
$\dot{m}_{LN_2}$ [kg/s]	0.01	0.02	0.03
$T_{LN2,in}$ [K]	94	94	94
$p_{LN2,in}$ [bar]	5	5	5

Typical characteristic pressure oscillation signatures were observed in the first phase of the chill-down. Once the structure is cooled down sufficiently, these oscillations vanish. With increasing mass flow rate, the chill-down duration decreases from 700 s in test # 1 to 400 s in test # 3. The pressure signature consists of a very low frequency oscillation 0.05 Hz and some higher superimposed oscillations with about 1.75 Hz and 3 Hz. All major frequencies are much lower than the acoustic eigenmodes of the channel (> 400 Hz) and are therefore of Helmholtz-type, i.e. have no spatial distribution (bulk oscillation).

During the chill-down-phase, which is characterised by the aforementioned oscillations, a continuous increase in mass flow rate and a corresponding continuous decrease in wall temperature, the void fraction in the channel decreases until the flow is fully liquid.

The simple 1-D transient two-phase model THESEUS can recalculate the pressure oscillation signature, as can be seen in the left graph in Figure 17 for the low mass flow Test#1. For the tests with higher mass flows the low frequency oscillation vanishes in the simulations. This can be seen in Figure 17, right.

Model analysis has shown that the origin of the low frequency oscillation is an interaction of structural heat transfer to the fluid, fluid condition, and amount of heat transfer, which is linked to the fluid condition itself or wall superheating. The higher frequency oscillating condition is generally a two-phase condition, driven by fluctuating vaporization/condensation phenomena and even flow reversal. This condition is interrupted by reaching a temporarily stable fluid condition (only one phase), i.e. liquid or supercritical. This temporarily stable heat flux leads to a pressure and vapor fraction increase until the structure to fluid heat transfer mechanism changes again. In the current test condition, the change occurs from super-critical heat transfer to film boiling heat transfer.

To further investigate this effect and provide additional data for validating the ESPSS and THESEUS models, the tests which are currently being performed at ET with liquid hydrogen will also use a second cooling channel setup in which the thermal inertia of the setup has been reduced as far as possible by reducing the wall thickness without compromising access for the different measurement instruments.

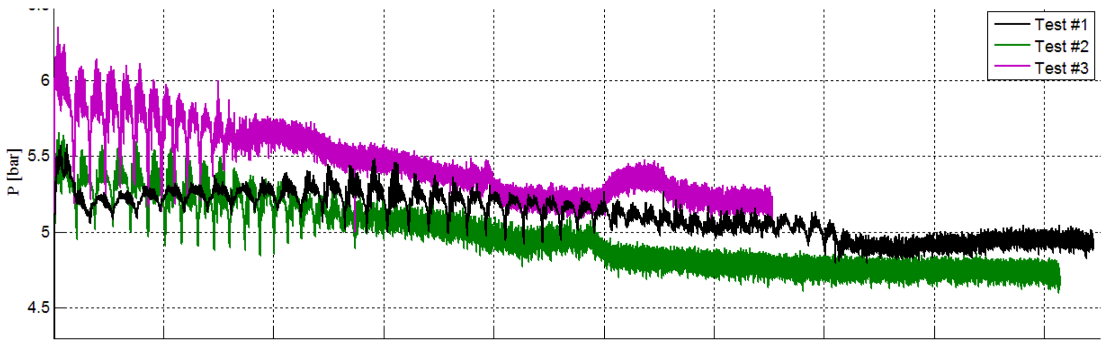


Figure 14: Coolant channel pressure recorded during chill-down tests at CHIEF

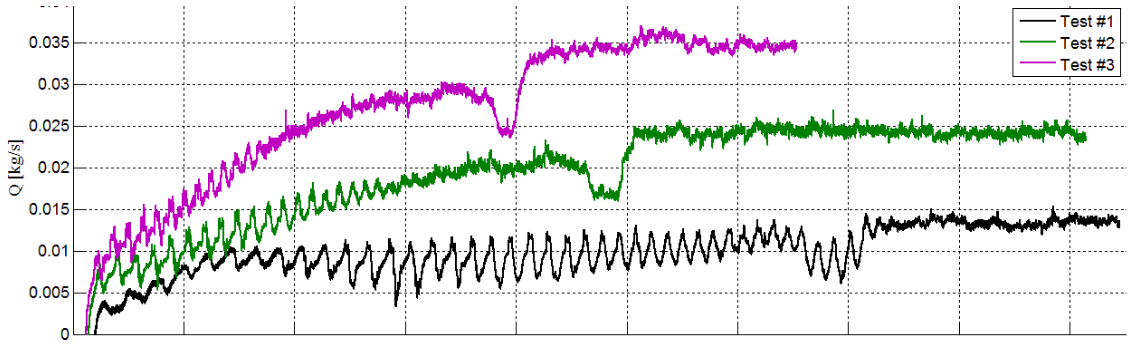


Figure 15: LN2 mass flow rate recorded during chill-down tests at CHIEF

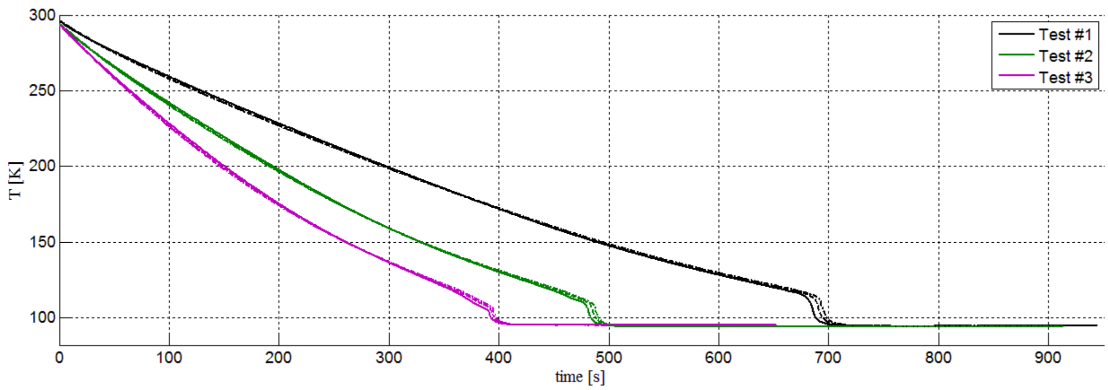


Figure 16: Coolant fluid temperature recorded during chill-down tests at CHIEF

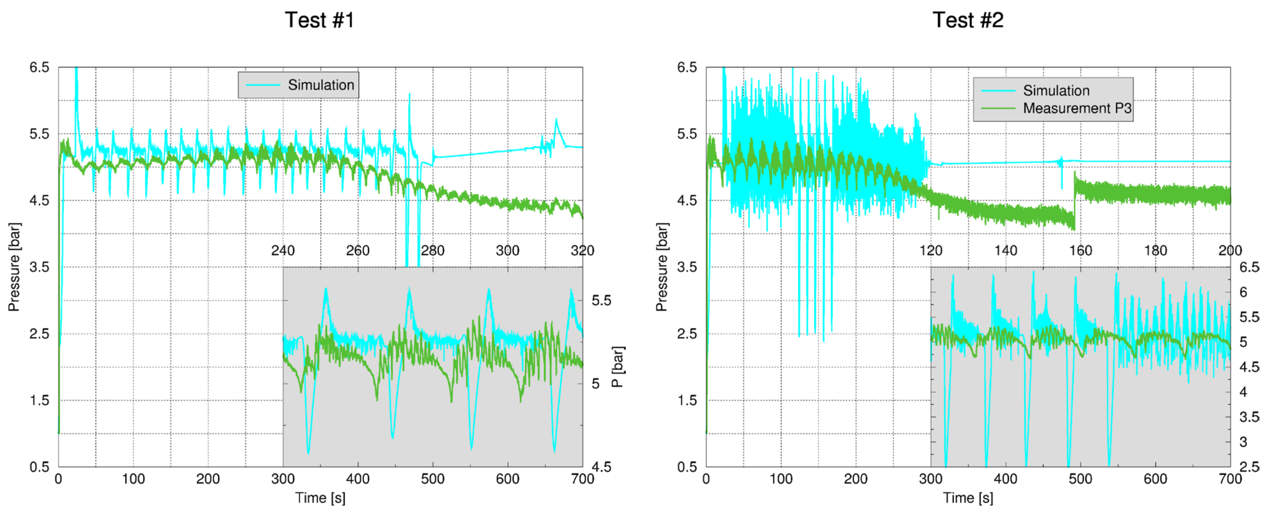


Figure 17: Comparison of test data from test # 1 (left) and test # 2 (right) with THESEUS simulation

4 Summary

Within a Belgian-German cooperation, partners from industry and academia performed investigations on transient two-phase flow phenomena which can occur in the operation of cryogenic liquid rocket engines.

Investigations on the flow of liquid hydrogen through a orifice representative of the orifices controlling the coolant mass flow rate to a dump cooled nozzle extension confirmed provided data to benchmark engineering tools against more accurate CFD tools. A comparison of the different simulation results with the experimental data showed that in nominal conditions, all models can predict the pressure drop and discharge coefficient of the orifice with high accuracy. However, the test data show that heat transfer effects may have a significant influence on the pressure drop behaviour and should be considered in the application in a rocket engine.

To more accurately simulate the chilldown-process of a liquid rocket engine thrust chamber before ignition, experiments with liquid nitrogen and liquid hydrogen were performed at two different test facilities. The tests with nitrogen provided a first data base to validate the 1-dimensional software tools THESEUS and ESPSS. Based on the experiments, the heat transfer correlations used in the tools can be checked with respect to their applicability to correctly simulate pressure oscillations caused by phase change phenomena in cooling circuits or propellant feed systems.

5 Acknowledgements

The research activities presented in this publication were performed within an ESA GSTP project (Contract No. 4000111616/14/NL/PA). The authors would like to thank ESA-ESTEC for supporting and coordinating the activities. The support of Belspo and DLR, which allowed bringing together Belgian and German experts from academia and industry to jointly perform this research activity, is highly appreciated.

6 References

[1] Gesellschaft für Reaktorsicherheit, *ATHLET Mod 3.0 cycle A, Program Overview*, Gesellschaft für Reaktorsicherheit mbH, 2012

[2] Dhont, G. and Wittig, K., CALCULIX: A Free Software Three-Dimensional Structural Finite Element Program, www.calculix.de

[3] Leachman, J.W., Jacobsen, R.T, Penoncello, S.G., Lemmon, E.W, *Fundamental Equations of State for Parahydrogen, Normal Hydrogen, and Orthohydrogen,* J. Phys. Chem. Ref. Data, 38(3):721-748, 2009

[4] Hakimi N, *Preconditioning methods for time dependent Navier-Stokes equation. Application to Environmental and Low Speed Flows,* PhD. Thesis, Vrije Universiteit Brussel, 1997

[5] Papdimitriou, P. and Skorek, T, One-dimensional thermohydraulic code THESEUS and its application to chilldown process simulation in two-phase hydrogern flows, Cryogenics 1992 Vol 32, No4, pp. 362-370

[6] Preclik, D., Two-Phase Flow in the Cooling Circuit of a Cryogenic Rocket Engine, AIAA-92-3731, AIAA/SAE/ASME/ASEE Joint Propulsion Conference & Exhibit, July 6-8, 1992, Nashville, TN

[7] Pinna F., Leonardi M., Nasuti F., Moral J. F. and Steelant J., *'Implementation of a Two-Phase Pipe components inside the ESPSS Library,* Space Propulsion 2018, Seville, Spain, 14-18 May 2018, SP2018-170.

[8] Leonardi M., Nasuti F., Di Matteo F., Steelant J., Moral Moral F. and Vila Vacas J., '*Improving Combustion Chamber and Pipe Components of the European Space Propulsion System Simulation (ESPSS) library with AUSM scheme*', Space Propulsion 2016, Rome, Italy, 2-6 May 2016, SP2016-3125133.

[9] Peveroni, L. et al., *Experimental investigation of Liquid Nitrogen Chilldown Two Phase Flow* Space Propulsion Conference 2016, Rome, SP2016_3124754

[10] Peveroni et al.: Liquid Nitrogen Chilldown Void Fraction Measurement and Feasibility of Extension to Liquid Methane Testing, Space Propulsion Conference 2018, Sevilla