NUMERICAL ANALYSIS OF MYRRHA CONTROL ROD SYSTEM DYNAMICS

M. Profir[1], V. Moreau[1] and G. Kennedy[2]
[1]CRS4, Sardinia, Italy
[2]SCK•CEN, Mol, Belgium
gkennedy@sckcen.be, manuela@crs4.it, moreau@crs4.it

ABSTRACT

This paper focuses on the CFD simulation of the MYRRHA Control Rod (CR) transient behaviour in a flowing LBE environment during a SCRAM emergency insertion. During normal operation the control rods are inserted in the LBE, below the active core. The high density of the liquid metal coolant allows buoyancy to be the driving force for the emergency insertion of the control rods during SCRAM, which must be performed in less than 1s. In the numerical model, the original geometry of the fluid path and the absorber pins bundle is modelled with high fidelity for the steady state simulations, while some simplifications are made for the moving component in the transient simulations.

The adopted moving mesh techniques initially use automatic volume mesh adaptation with re-meshing of the underlying geometry and evolve naturally to the use of the more versatile overlapping grids. The overset mesh method applies well to complex flow paths, once the ability of handling small or zero gaps is acquired. The method is applied to reproduce the CR displacement as in the COMPLOT experiment, with several preliminary simplifications and gradually increased geometrical accuracy. Issues like the non-conservation of the mass flow and related oscillations in the calculated pressure and force fields are addressed. At first, the displacement is imposed, according to SCK•CEN theoretical calculations. Then it should be inferred by Newton's law from the force balance on the CR. In this paper, the imposed displacement and its numerical handling are illustrated. The numerical methodology and results will in the future be validated against the COMPLOT experimental tests and will be used further to support the MYRRHA design.

KEYWORDS
MYRRHA, liquid lead, safety rod systems, dynamic mesh

1. INTRODUCTION

MYRRHA (Multi-purpose hYbrid Research Reactor for High-tech Applications) is a flexible fast-spectrum research reactor under design at SCK•CEN, the Belgian Nuclear Research Centre. MYRRHA is a pool-type reactor with Lead Bismuth Eutectic (LBE) as primary coolant. Conceived as an accelerator driven system prototype, it is able to operate in sub-critical mode. Operating in critical mode, MYRRHA is identified as the European Technology Pilot Plant for the Lead Cooled Fast Reactor which is one of the Generation IV reactor concepts.

The safety and control rod system is a key component for the safe and reliable operation of any nuclear reactor. The MYRRHA control rod system comprises an absorber bundle within a guide tube filled with LBE. During normal operation the control rods are inserted in the LBE, below the active core. The high density of the liquid metal coolant allows buoyancy to be the driving force for the insertion of the control rods during a SCRAM event, i.e. an emergency reactor shutdown. The MYRRHA design requires that in the event of SCRAM, the insertion of the control and safety rods should take less than 1 s. The operation of this control rod system within liquid metal is rather different from standard systems. Therefore, to support the MYRRHA reactor design, the dynamics of a full-scale mock-up of the MYRRHA control rod system are characterized and qualified in the COMPonent LOop Testing (COMPLOT) LBE experimental facility at SCK•CEN. These experimental tests in COMPLOT will allow the hydrodynamics to be tested over a wide range of operating conditions, taking into account the actual transient acceleration and inertial effects of the rod bundle and liquid metal coolant, and thereby provide valuable data for the qualification of the control rod system.

In the framework of the European FP7 MAXSIMA project [1], CRS4 is dedicated to the particular task of simulating the MYRRHA control rod system with CFD, in the COMPLOT experimental facility. The CFD simulations provide pre-test simulation support to the experimental test section design [2], [3], [4], but more importantly, the numerical methodology and post-test simulation results will be validated against the COMPLOT experimental results. The validated CFD methodology will subsequently be used to simulate the MYRRHA control rod system and therefore serve as input to the MYRRHA safety analyses.

This paper focuses on the CFD simulation of the MYRRHA Control Rod (CR) transient behaviour in the COMPLOT LBE experimental facility, during a SCRAM emergency insertion. The geometrical modelling and meshing setup as well as the numerical methodologies are explained, together with the interpretation of some important results.

2. OVERVIEW OF THE CONTROL ROD TEST SECTION

The COMPLOT LBE facility is a closed-loop facility, designed to characterise the hydraulic and hydrodynamic behaviour of various full-scale MYRRHA components in a flowing LBE environment, representative of the MYRRHA conditions. The vertical test section is representative of a single MYRRHA core position or in-pile section (IPS) at full height, with the LBE flowing upwards as in MYRRHA. COMPLOT is an isothermal loop, operating within the temperature range of 200°C–400°C.

In the MYRRHA reactor design, the CR components essentially consist of long guide tubes which are secured on the top of the reactor cover. The guide tubes pass through the Above Core Structure (ACS) region and also penetrate into the core region. The LBE enters the bottom of each guide tube at the core inlet and flows upward within the guide tube and through the CR bundle and internals. The 19-pin absorber bundle is roller guided within the guide tube and is mechanically linked with a central rod that links to a hollow tube which continues through the reactor cover to the reactor exterior. The flow path inside the guide tube is formed by a damper, a bypass and a labyrinth flowing out the LBE through the 12 outlet holes, as shown in Figure 1. Above the core, in the so-called ACS, the LBE exits the guide tube via a series of outlet holes into a larger annular tube representing the MYRRHA upper plenum.
Since COMPLOT is an experimental piping loop, space, weight, and financial limitations are an issue. To ensure that the COMPLOT test section outlet is representative of the MYRRHA conditions, the test section outlet needed to be optimized to ensure that the component performances are not influenced by any piping design features. SCK•CEN conceptualized a test section outlet: in the lower region of the test section, the design considers a cylindrical region enclosed in the hexagonal region of the active core where the guide tube is inserted. Above the core region, the test section expands into a larger annular flow area which receives the LBE flowing out from the guide tube outlet holes. This annulus represents the MYRRHA ACS. Its diameter has been optimized, to reduce space, weight, and cost limitations for the testing program, by means of a parameterization study performed with CFD simulations, described in the report [5]. The annular flow area in the case of the CR is rather complex and is illustrated in detail in 2.1.

2.1. Computational Domain. Geometry and Mesh

The simulations were performed with the commercial CFD code STAR-CCM+, starting with version 8 and reaching the latest released version 9.06 [6]. Globally, the original geometry is modelled with high accuracy, keeping a reasonable balance between computational power and physical behaviour. The fluid domain is modelled in full detail, while some simplifications are made for the moving component.
The CFD geometry considers only half of the domain, thanks to its planar symmetry**;** it started with the construction of the CR hosting section, namely the guide tube and annular tube. Then attention focused on the moving component and on the complex upper flow path. Since the shell of the control rod test section is rather complex, the reproduction of the movement proved to be complicated. For this reason, the attention was initially paid to the simulation of the control rod system in the stationary case. A CFD model was constructed, with reasonable simplifications, but as realistic as possible. The characteristic mesh sizes are referred to the inlet of the guide tube which has a diameter of 10 cm; the discretization accounts for about 20 cells along this diameter. A prism layer with the thickness of 1 mm is generated since a turbulent flow is treated in the simulation. The mesh was refined at boundaries level and volumetric controls were employed in order to better capture some details (rollers, narrow gaps), resulting in about 3 million volume cells, Figure 1 (right). On the left, some geometrical details are shown; the different colours indicate the distinguished inputs of the boundaries, split for an optimal boundary level refinement.

Figure 1: Geometry damper, bypass, labyrinth, piston (left). Volume mesh detail labyrinth (right)

2.2. Steady State Calculations

Steady state simulations have been performed. The flow is turbulent. The K-Epsilon Turbulence model is applied. A two-step approach was adopted for the boundary conditions: i) imposing the mass flow rate of 18.5 kg/s at the inlet; ii) imposing the stagnation pressure of 2.7 bar calculated in the previous step as Stagnation Inlet. The nominal expected pressure drop over the control rod bundle in MYRRHA is 2.5 bar. This value is fundamental while the mass flow rate is a derived quantity.

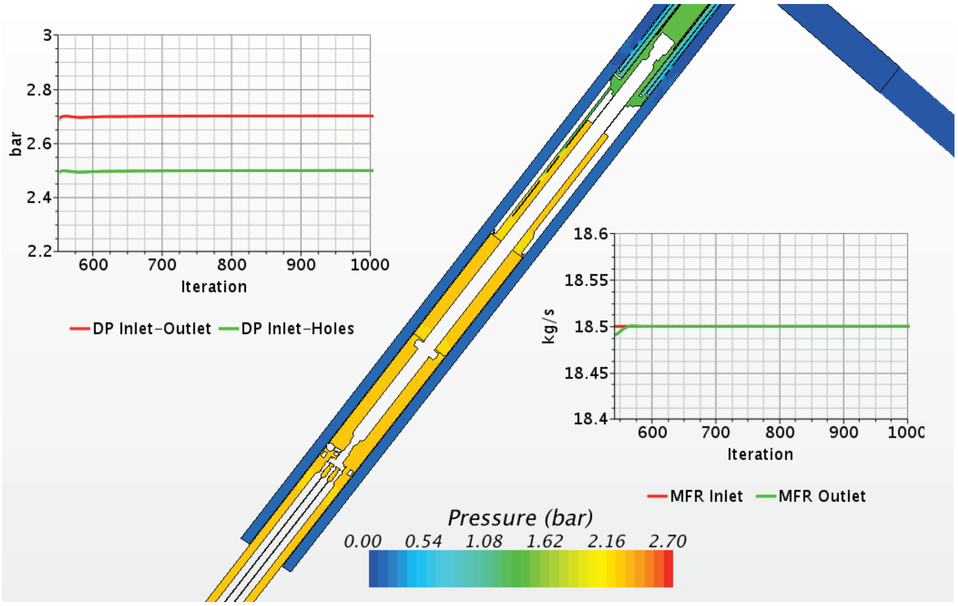

Figure 2: Steady state solution

3. MOTION WITH MESH MORPHING METHODOLOGY

Given the complexity of both the geometrical model and the physics, the work was split into two distinct steps. The preliminary stage considers half of the movement of the absorber pin bundle while we became familiar with the geometry. We developed moving mesh techniques and implemented strategies for the conservation of a good quality mesh. Particular attention is paid to the solution independence with respect to the re-meshing operations. Suitable boundary conditions, compatible with the imposed displacement, are determined.

3.1. Movement Settings. Strategy of Re-meshing

The first method used to reproduce the movement of the control rod involved morphing motion and re-meshing of the volume mesh. The computational domain is divided in four regions: one enclosing the moving component, two regions upward and downward, appropriately coupled with the moving part region through interfaces and the fourth one containing the rest of the domain. This approach was developed prior to this project, on quite simple geometries and the intention was to extend it to a more complex geometry such as the CR model. The presence of the damper, as a restriction of the guide tube, forbids the use of a simple sliding mesh for the moving component.

The movement couples a rigid body motion (translation) of the absorber pin bundle region with a morphing motion of the up and down regions, letting stationary the rest of the domain. The morphing produces deformations of the mesh (compression and stretching of the volume cells). In order to prevent the degeneration of the mesh, a re-meshing strategy is developed and applied only on the morphed parts, avoiding the pin bundle re-meshing, the most costly in terms of computational power. The strategy of re-meshing consists of coupling the physical displacement performed by the control rod with the updated underlying geometry. The effective displacement performed by the rod is retrieved by means of a report measuring its position and imposed as a translation vector of the corresponding geometrical body at CAD level. At the same time, the quality of the mesh is controlled. When the quality of the moving mesh becomes smaller, by a given amount, than the quality of the initial static mesh, then a re-meshing is performed. The automatic implementation of this strategy is coupled with the simulation through a Java

script on a time step basis. When the mesh quality criterion is no longer satisfied, the geometry at CAD level is updated and the surface and volume mesh of the morphed regions is regenerated
However, the test section which hosts the control rod is not just a guide tube with holes but it has complex flow paths such as a damper with slots, a bypass and a labyrinth guiding the LBE flow out through the outlet holes. The moving control rod consists of a 19 pin bundle linked to a central rod that links further to a hollow tube. Due to the presence of these components, the foreseen strategy of automated re-meshing coupled with geometry updating cannot be applied to the whole model. For this reason only a part of the movement is reproduced, restricting the morphing region only to the cylindrical part of the guide tube.

The CR movement was reproduced, imposing a translation motion to the region containing the bundle. The same velocity was set as a boundary condition in the regions undergoing the morphing motion, namely, assigning a Grid Velocity to the interfaces between the morphed regions and the bundle region. The other boundaries of the morphed regions (portions of the guide tube) are Floating boundaries, letting the control points on their surfaces free to follow the motion of the interfaces. In order to verify the independence of the solution with respect to the re-meshing operations, reports of pressure and mass flows were prepared; ideally, the plots show no perturbations, or at least no important effects of the re-meshing on the flow.

Two preliminary setups of the simulation were implemented: i) a steady-state simulation with a specified mass flow rate at the inlet and a pressure outlet; ii) a steady-state simulation with a stagnation pressure at the inlet and a pressure outlet with the calculated pressure drop from the first setting, resulting in almost exactly the same inlet mass flow rate as in setup i). In the case of the simulation with the stationary bundle, both approaches proved to be equivalent, as expected. In the case of the transient simulation, with the movement actuated, the boundary conditions had to be properly chosen, compatible with the imposed displacement of the bundle.

3.2. Transient with Inlet Mass Flow Rate (MFR)

After the stabilization of the flow in steady state condition, we continued to run the same simulation switching to the transient setting, keeping the bundle stationary for a sufficient duration to stabilize the flow. After that, the bundle was allowed to move, considering a constant acceleration $a = 15\ m/s^2$, as an estimation of the buoyancy effect. Several re-meshing operations were applied during the movement, but the re-meshing of the region containing the pin bundle was avoided, which alone contains about 2.5 M cells. The displacement was performed in about 0.2 s, until the seal attached to the rod reached the entrance to the damper.

During the movement, the simulation revealed a strong drop in pressure, independent of the time step, indicating a probable but not foreseen physical origin. We reached to the following considerations:

- The consequence of specifying both a mass flow rate at the inlet and imposing acceleration to the bundle is an inversion of the acceleration of the flow displaced by the moving bundle which has to fill in the empty space left at the bottom of the bundle.
- The integral, along the guide tube's portion involved in the motion, of the variations of velocity (both of the inlet flow and of the bundle) is compensated by the variation of the integral of the pressure gradient along the same portion of the guide tube.

Hence, we can conclude that the pressure inversion is perfectly physical. It is induced by the combined effect of the prescribed mass flow rate in inlet and of the prescribed bundle acceleration.

3.3. Transient with Inlet pressure

The MFR boundary condition does not seem compatible with the imposed displacement. A different approach was thus adopted: i) initially, the unsteady simulation with the stationary bundle was performed with a specified MFR and the pressure drop between inlet and outlet was calculated; ii) afterwards, this pressure drop was imposed at the inlet (Stagnation Inlet boundary condition) and the bundle was allowed to move with the specified acceleration.

During the motion, the report measuring the pressure drop across the control rod bundle registered a decrease of about 0.5 bars in the first iteration and continued to decrease to about 1 bar during the movement. The inlet MFR was also monitored and a continuous increase from 19 kg/s at the beginning up to 52 kg/s at the end of the simulation was observed. An illustration of the simulation at the end of the first half of the movement is given in Figure 3.

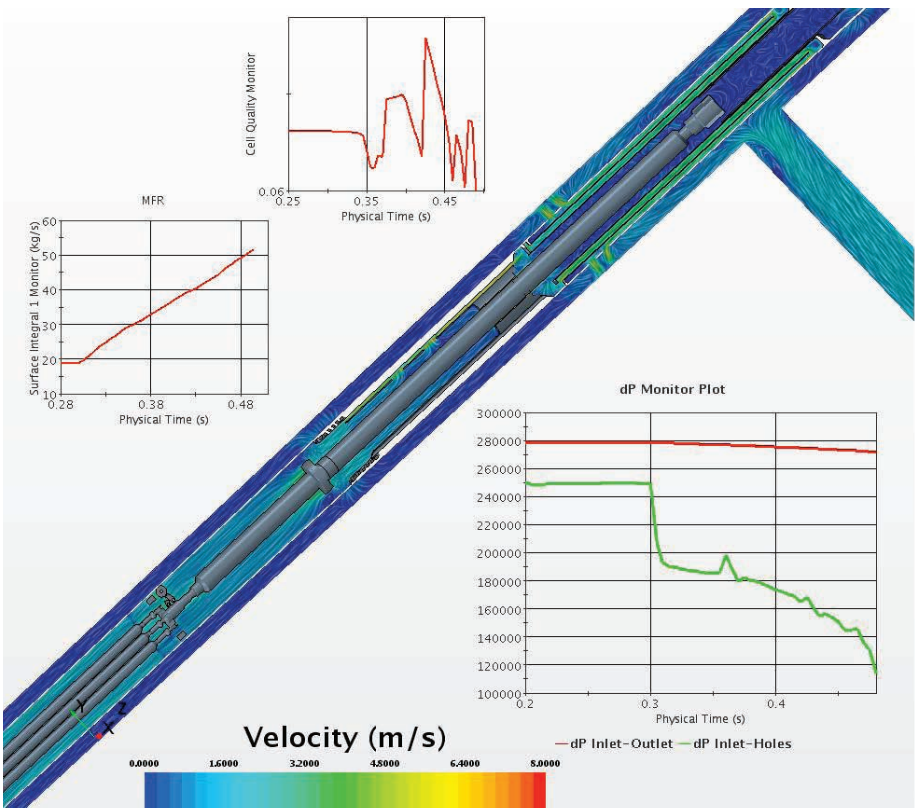

Figure 3: End of the first part of the movement, performed with morphing and re-meshing

3.4. Effects of the Re-meshing on the Physics

The jumps in the pressure field due to the re-meshing operations, visible on the Figure 3, required the strategy of re-meshing to be improved, with the following actions: i) adopt a better criteria for deciding when to re-mesh; ii) adopt a decreasing time step when the bundle reaches the entrance in the damper; iii) refine the mesh in the morphed regions. Re-meshing was also performed upon global change of the morphed region volumes by more than 50% or upon their incremental change by more than 10%.

Refining the mesh in the morphed regions and having the same mesh size at the interface provided a smoother mapping of the previously calculated solution on the newly recreated mesh, according to the nearest neighbour interpolation scheme. Applying the actions illustrated above, a smoother motion was obtained, with almost no influence of the re-meshing on the physics, as shown in Figure 4.

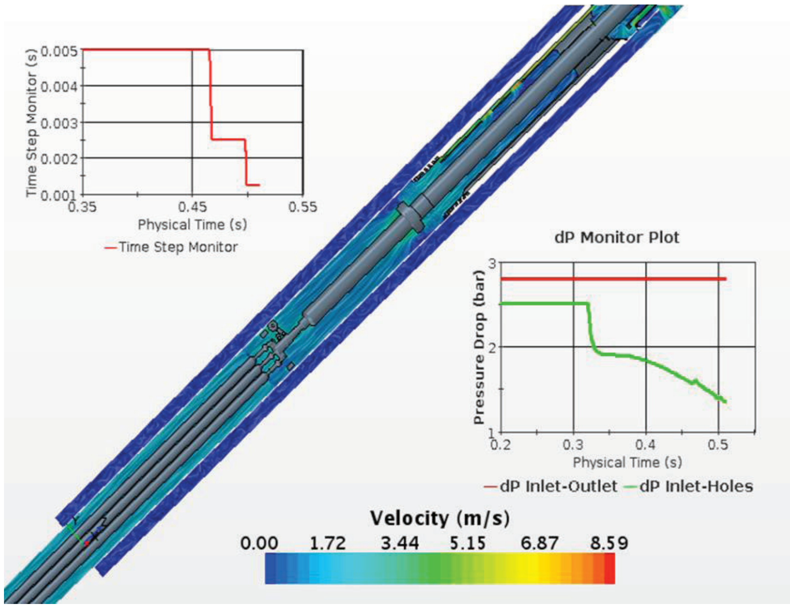

Figure 4: Motion with almost no influence of the re-meshing on the pressure field

4. MOTION WITH OVERSET MESH METHODOLOGY

The second step of the work would have been to reproduce the second part of the CR movement in a much stiffer configuration, facing the limitation of the re-meshing technique. However, in the meantime, the overset mesh methodology implemented in STAR-CCM+ reached sufficient maturity, allowing the entire CR displacement to be simulated with no need to re-mesh. The background domain and the volume enclosing the moving component are separate overlapping regions, each with its own mesh. A volume interface is created in order to couple the solutions on the two overlapping grids. The method is explained on a simplified model of the test section and is applied to reproduce the full scale CR movement.

4.1. Simplified Control Rod Motion with Overset Mesh Method

While simulating the motion of the CR system with morphing and re-meshing strategies, it was understood that reproducing the motion into the upper flow path would become impossible using the same technique because the CR displacement would create almost instantaneous high mesh distortion and continuous re-meshing. Hence attention was focused on an alternative moving mesh methodology, more exactly on the **Overset Mesh Method**, recently implemented in STAR-CCM+. As described in the CD-adapco's "Spotlight on Overset Mesh Version902", overset meshes, also known as "Chimera" or overlapping meshes, are used to discretize a computational domain with several different meshes that overlap each other. Using overset meshes does not require any mesh modification after generating the initial mesh, thus offering greater flexibility over the standard meshing techniques.

The stationary parts, guide tube, outer annular tube, labyrinth, damper, etc. form the **Background region.** The moving component is enclosed in a box, the **Overset region,** meaning that the moving component surface is not obtained with subtraction from the stationary background but from this box**.** The boundaries of the moving object are **Walls**, while the boundaries forming the box will have **Overset Mesh** type boundary condition. Two mesh continua must be employed, one for each region, and an **Overset Mesh interface** between the two regions is created.

The volume cells involved in the exchange of information between the two regions are classified as follows. The initialization of the interface creates a layer of **acceptor cells** in the background mesh, attached to the overset boundary in the overset region. The **active cells** are the ones where the discretized governing equations are solved. The **inactive cells** are the ones where no equation is solved, however, these cells become active when the overset region is moving. The **acceptor cells** separate active and inactive cells in the background region; they are used to couple solutions on the two overlapping grids, by receiving interpolated information from the other mesh **donor** cells. There are two necessary conditions for a successful initialization and simulation of the motion: i) 1-2 layers of cells between the moving body boundary and the box boundary; ii) the same size of the mesh in both regions in the overlapping zone.

The two volume meshes (for the background and overlapping regions) were generated, for a simplified moving component, resulting in about 3 M cells. Circular ring volumetric controls with customized mesh size were created in the overlapping zones, in the narrow space above the damper, and at the outlet holes. With these mesh refinements, the error in the mass conservation, specific when using the overset mesh method, was limited to below 1%. The foreseen pressure drop of 2.5 bar across the bundle corresponds to an inlet MFR=22 kg/s.

5. CONTROL ROD MOVEMENT WITH ACCURATE GEOMETRY

Once full control of the simulation of the movement was acquired, correctly coupled with the physics, attention was focused on the treatment of the narrow gaps. The objective was to design more and more accurate geometry, especially regarding the dimensions of the seal. Ideally, the seal attached on the piston matches the damper perfectly when the seal enters the damper, forcing the LBE flow through the slots. Due to the current limitations of the overset mesh methodology (to be eliminated in the future versions), this situation cannot be reproduced. Thus, an acceptable compromise between the mesh density and the geometrical accuracy had to be found, with compensation for the reduced accuracy by modelling an increased viscosity.

5.1. Approach the narrow gaps in the flow path

The dimensions of the seal were made so as to have a gap of 2.5 mm with the wall of the damper, as visible in Figure 5 (left). The most suitable mesh surface size is 1.5 mm in the overlapping zones, for a volume mesh representation of about 9 M cells, Figure 5 (right).

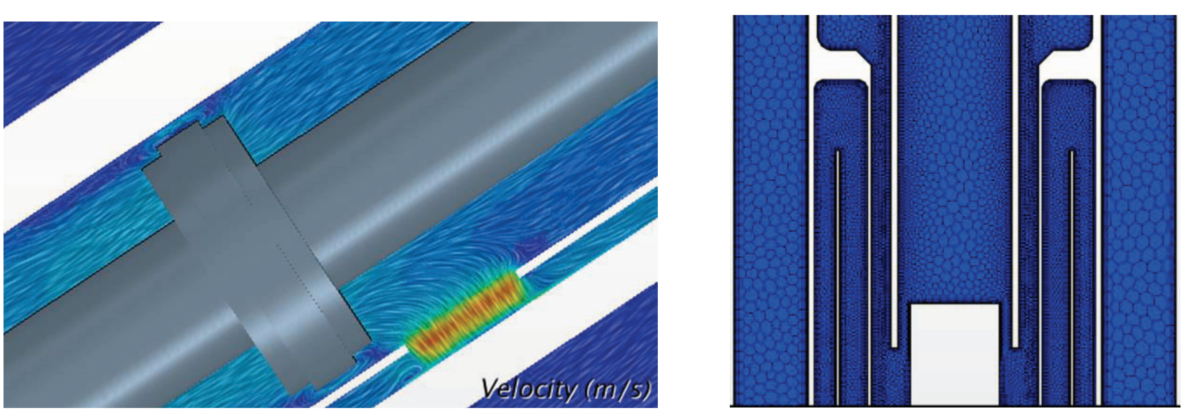

Figure 5: Left, seal almost conform, 2.5 mm of gap to the damper. Right, volume mesh 9 M Cells

The convergence in steady and unsteady state before motion started is essentially reached, with a relatively small mass error, less than 0.1 kg/s. The pressure drop at the outlet holes was calculated to be 2.58 bar for an inlet MFR=23 kg/s (Figure 6). A more accurate steady state calculation, performed by scaling the inlet mass flow rate value to 22.6 kg/s, provided 2.5 bar of pressure drop across the bundle (measured at the outlet guide tube holes) and 2.72 bar of overall pressure drop.

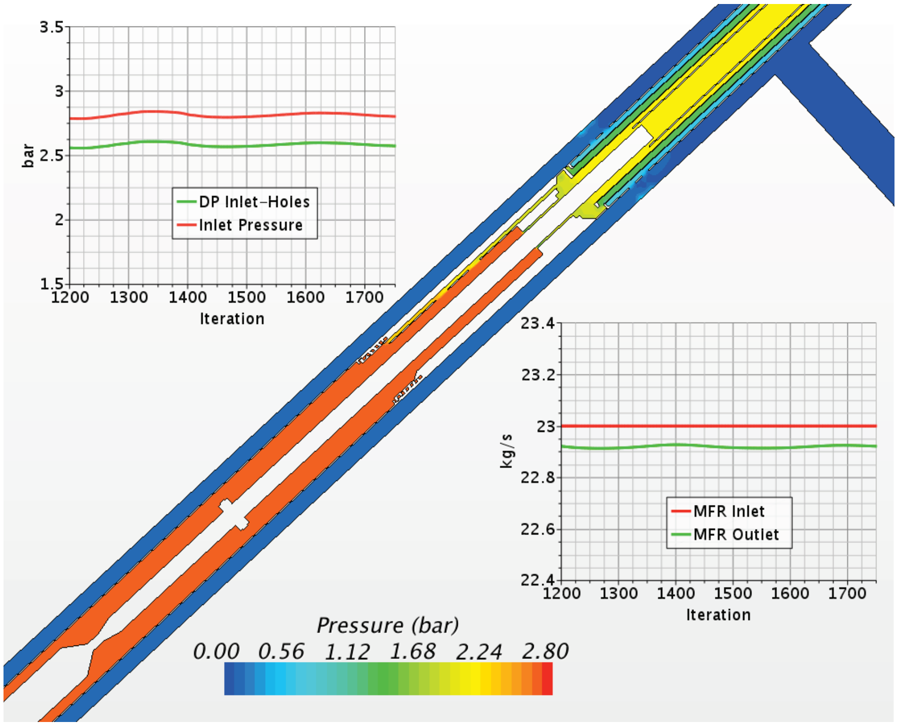

Figure 6: Steady state results in model with simplified bundle and overlapping grids

The motion was simulated with a fixed inlet stagnation pressure for the LBE fluid and an imposed translation velocity for the CR; the displacement was estimated by SCKCEN in theoretical calculations, based on the transient accelerations given to the bundle and is showed in the plots in Figure 7 below. The curves of the drag force, pressure and mass flows are shown in Figure 8.

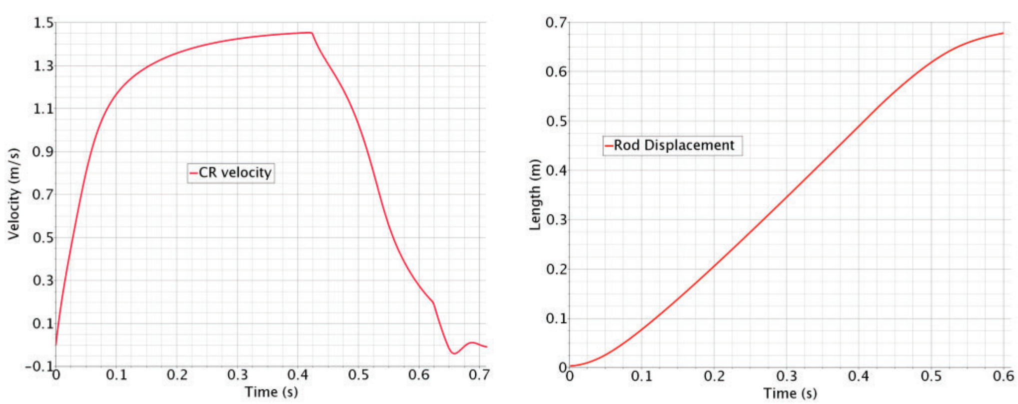

Figure 7: Imposed displacement, provided by integrated transient accellerations

.

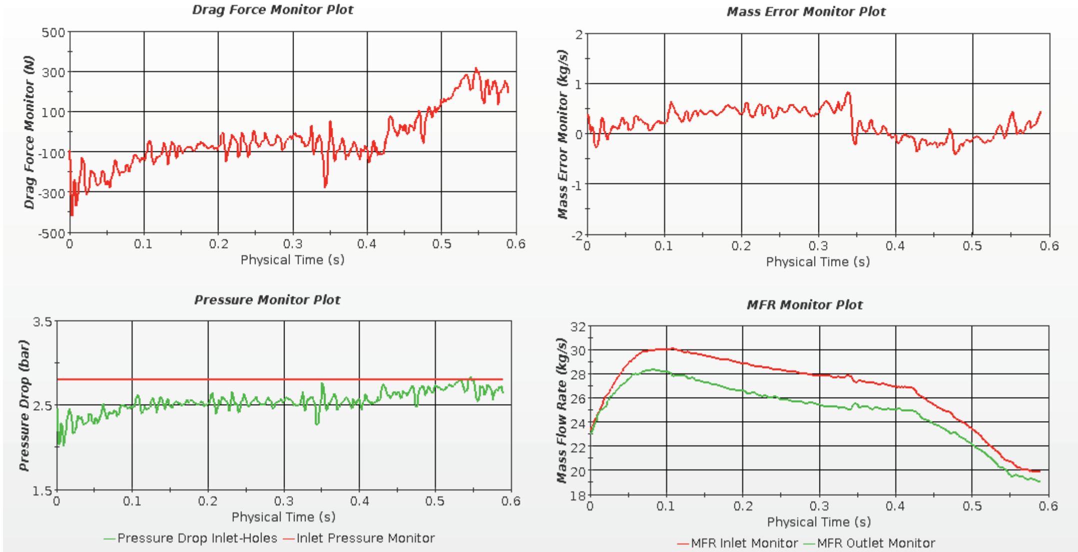

Figure 8: Results of the transient simulation with seal almost conform for 0.6 s of motion

5.2. Towards the geometrical accuracy

The gap between the seal and the damper is blocked by increasing the viscosity of the LBE around the seal by four orders of magnitude. The increased viscosity is defined as follows. First, we create the characteristic function χ of the volume where the viscosity has to be modified, taking into account the CR current position. Then, the increased dynamic viscosity of the LBE is $\mu_1=\mu_0+ \mu_0 10^4\chi$.

The fluid is also blocked in the channel at the top of the damper (Figure 9, left). In this channel, the real gap is 1.25 mm but a 2.5 mm gap is kept in order to be able to reproduce the overset movement.

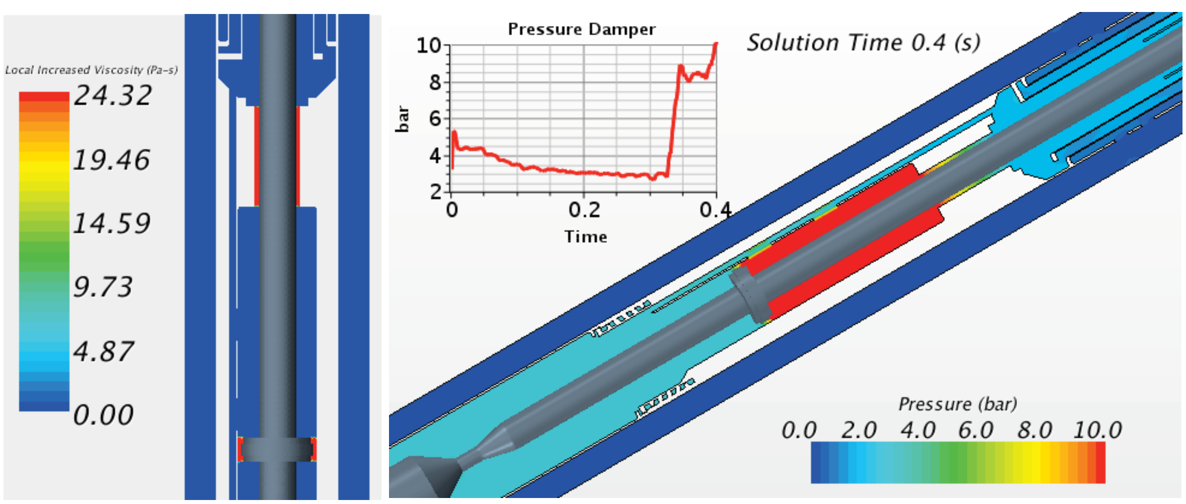

Figure 9: Left, volumes with four orders of magnitude increased viscosity. Right, overpressure due to the seal "sliding" on the damper, compressing the LBE

During the steady state simulation the viscosity alteration induces a relevant change in the flow pattern as almost no flow can pass through the annular channel at the top of the damper. To keep the pressure drop of about 2.5 bar, a MFR = 19.5 kg/s was needed.

The pressure was monitored at a point situated in the upper part of the damper, above the slots. The plot, Figure 9 right, shows a consistent increase of the pressure, initiated at the moment of the entrance in the damper (at time 0.33 s). The LBE flow reaches high velocities, more than 10 m/s at the damper slots (Figure 10). The behaviour of the flow, in terms of pressures and mass flow rates, was monitored during the control rod insertion and is illustrated in the Figure 11.

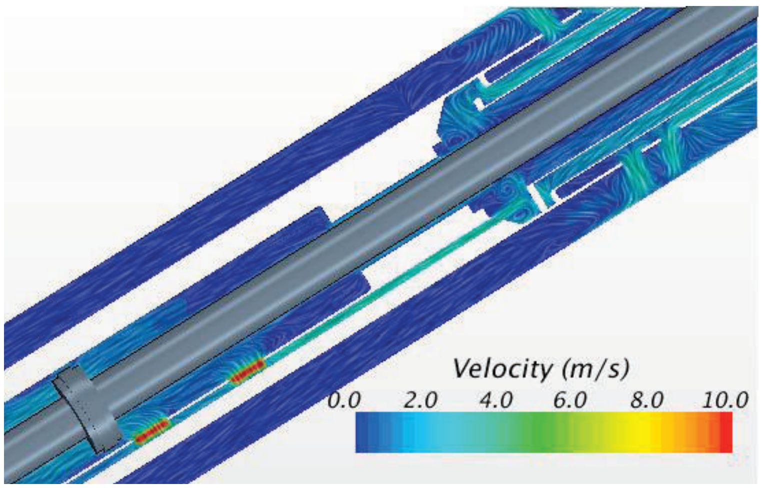

Figure 10: High velocity reached by the LBE flow at the damper slots

.

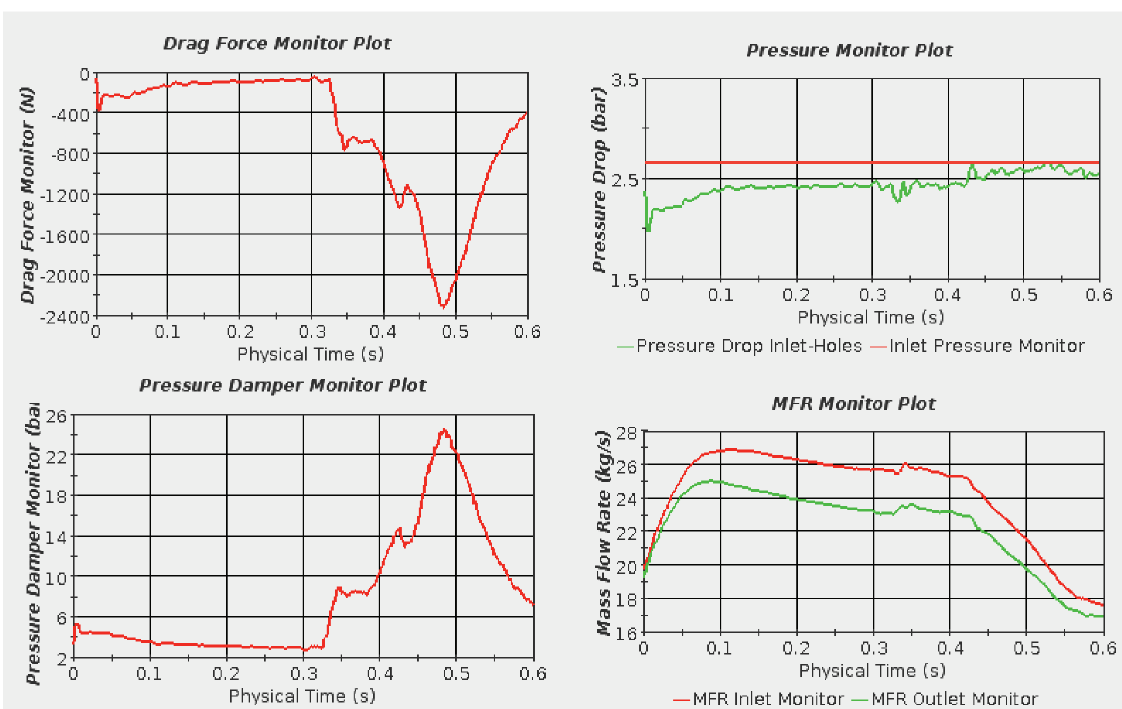

Figure 11: Plots at the end of the insertion in the case with increased viscosity

5.2.1. Interpretation of the drag curve

The drag force curve shown in Figure 11 was obtained by measuring the upward force exerted by the fluid on the moving assembly and is positive when the CR is at rest. For better visualization and interpretation, a new drag curve representing the downward force exerted by the fluid, was adopted. We also scaled the curve of the pressure measured in the upper part of the damper cylinder, by the area of the section represented by the seal. The two scaled curves are represented in Figure 12, top left; they are essentially shifted by about 200N. The pressure monitoring thus captures all the variations of the drag.

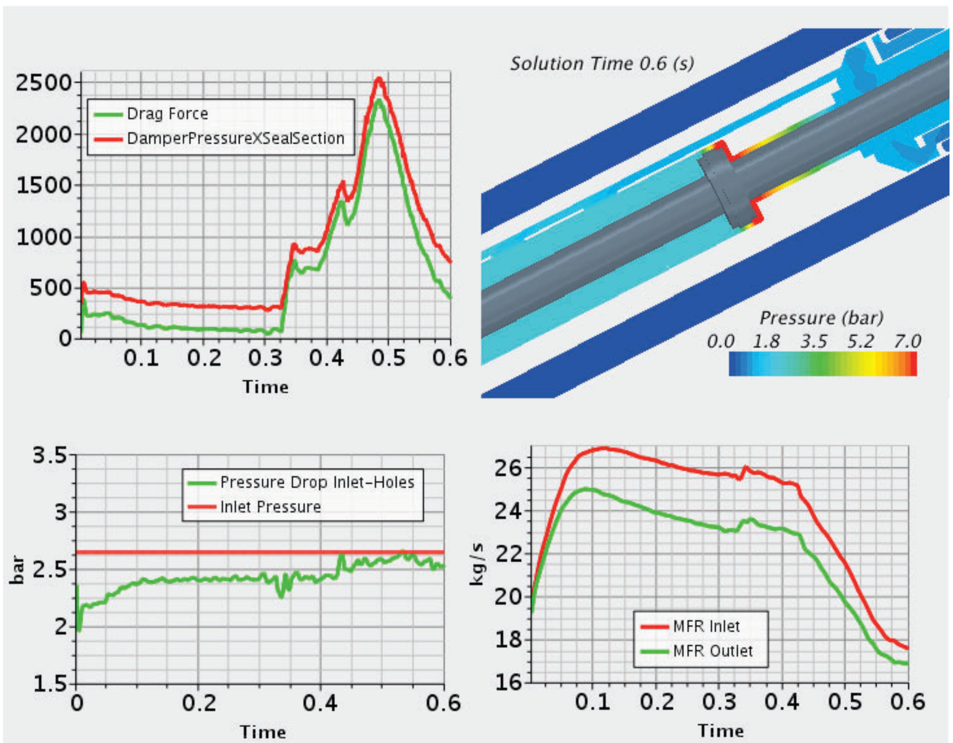

Figure 12: Drag force and pressure in the damper in full correspondence

The pressure drop plots in Figure 13 (left) show the comparison between the most accurate geometry of the seal that could be afforded, and the same geometry adjusted with the viscosity. The plots essentially indicate that an increased viscosity has a "calming" effect on the physics.

5.2.2. Numerical validation and comparison of the results

For the validation of the simulation, the case with increased viscosity was run while dividing the time step by five: **μ_1 (N=4), TS=0.0002**s. The results previously obtained are confirmed (very similar drag force curve). The main difference is a much longer computation time and more frequent oscillations in the pressure drop curve, as Figure 13 (right) shows. This confirms that there is no need to run with such a small time step until we have a complete control of all the physics involved and it is sufficient to run the simulation with TS=0.001s and 20 inner iterations.

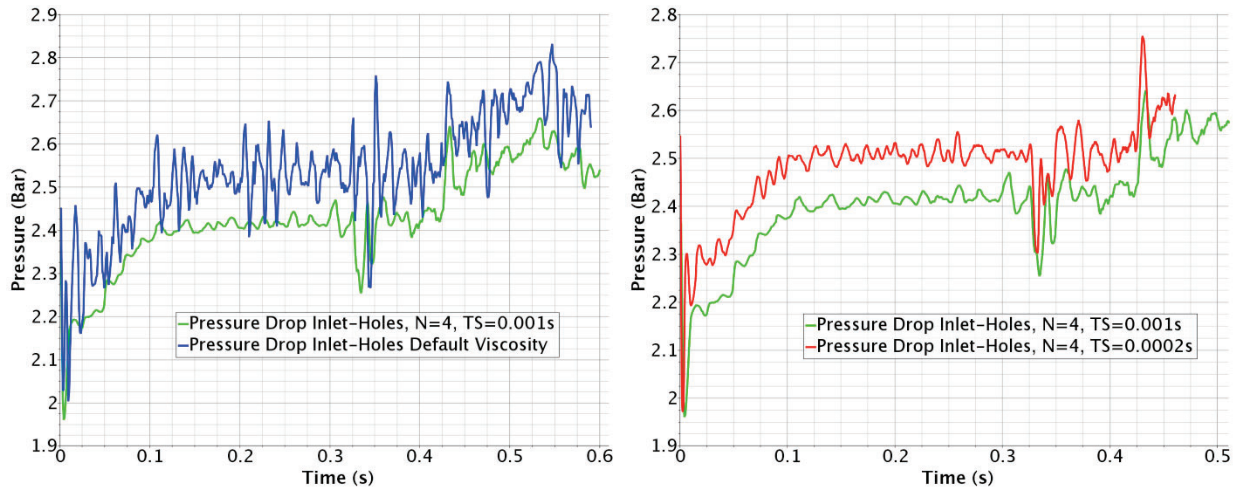

Figure 13: Pressure drop LBE default/increased viscosity (left), different time steps (right)

In Table I, an overview of the steady–state solutions in the last three cases, considered the most representative ones, is given. Notice that the shift of 0.1 bar in the pressure drop curves in Figure 13 (right) is also due to the higher pressure (2.72 bar) given as boundary condition.

Table I: Comparative results for default/increased viscosity and different time steps

LBE Viscosity	MFR (kg/s)		Pressure drop (bar)		Drag Force (N)
	Inlet	Outlet	Inlet-Outlet	Inlet-Holes	
μ_0, TS=0.001 s	22.6	22.7	2.72	2.5	75
μ_1 (N=4), TS=0.001 s	19.5	19.3	2.68	2.5	100
μ_1 (N=4), TS=0.0002 s	19.7	19.55	2.72	2.56	97

6. CONCLUSIONS

In the context of the MAXSIMA project, CFD analyses of the control rod dynamics have been performed with a numerical model reproducing a test section of the COMPLOT experimental facility.

In order to reproduce the movement of the control rod assembly, mesh morphing techniques and automatic optimized re-meshing strategies have been developed and employed, coupling the CFD code with Java scripts. These techniques have been applied in the first step for the reproduction of half of the movement. For the simulation of the second part of the movement, the approach had to be changed, given the complexity of the flow path. The recently developed methodology of overlapping grids in STAR-CCM+ was applied first on a simplified model, followed thereafter by gradually increasing the geometrical complexity.

The aim was to test this new moving mesh methodology on a rather complex geometry, which revealed a set of complications. Grid discretization issues provided mass loss in steady state and variations in the mass during the transient, with corresponding pressure drop variations. The imposed displacement adopted in this test phase of the method, provided by 1-D theoretical calculations, in terms of integrated transient accelerations, was not totally compliant with the geometry. We identified these two major aspects as the necessary conditions for a reliable interpretation of the results. To this end, a good control on the overset mesh methodology in a complex flow path configuration has been acquired. Issues like the non-conservation of the mass flow and the oscillations in the fields have been investigated and partially resolved in a still on-going process, optimizing the grid discretization in the overlapping zones. The continuous evolution of the software versions also contributed to the improvement of the simulations.

A method to implement the narrow gaps in the flow path was established, which is quite difficult to treat in the context of the overset mesh methodology. Recall that the control rod insertion must be stopped in its last phase by a piston entering a damper, ideally with a perfect seal. In this sense, an acceptable compromise between the mesh density and the geometrical accuracy was found, bringing the dimensions of the piston much closer to the real ones, for a volume mesh of about 9 million cells.

In order to correctly model the near zero leakage in the damper, the LBE viscosity was modified locally, by increasing it artificially in the remaining gaps. In this manner, a correct drag force curve was obtained, representative for the numerical model of the CR considered in the simulations described in this paper.

The drag force obtained with the imposed displacement is not fully consistent with the physics. We thus expect the real movement to show some velocity and acceleration differences; the confirmation will be given by a two–way coupling, determining the displacement from the force balance on the CR.

REFERENCES

1. Official website http://maxsima.sckcen.be/
2. M. Profir, V. Moreau, G. Kennedy, “COMPLOT Test Section Outlet CFD Optimization (pre-test and dimensioning)”, *Journal of Nuclear Research and Development*, vol. NO. 6, pages 19-24.
3. Moreau V., Kennedy G., “Report on the CFD Optimization of the COMPLOT Fuel Assembly Test Section Inlet CR/SR configuration”, CRS4 internal report
4. Profir M., Moreau V., Kennedy G., “Report on the CFD Optimization of the COMPLOT Fuel Assembly Test Section Inlet”, CRS4 internal report
5. Profir M., Moreau V., Kennedy G., “Report on the CFD Optimization of the Safety Rods Bundle Outlet test section”, CRS4 internal report
6. Official website http://www.cd-adapco.com/