COMPLOT TEST SECTION OUTLET CFD OPTIMIZATION (PRE - TEST AND DIMENSIONING)

[1] MARIA MANUELA PROFIR, [1] VINCENT MOREAU and [2] GRAHAM KENNEDY

[1] CRS4, Centre for Advanced Studies, Research and Development in Sardinia, Italy
[2] SCK•CEN, Studiecentrum voor Kernenergie, Mol, Belgium
e-mail: manuela@crs4.it

Abstract

In the framework of the FP7 MAXSIMA European project, the COMPLOT (COMPonent LOop Testing) LBE experimental facility is employed for thermal-hydraulic experiments aimed to test and qualify, among other components, a buoyancy driven safety/control rods (SR/CR) system, as key components for the safe operation of the MYRRHA reactor. This paper focuses mainly on a simplified CFD representation of the SR test section outlet in order to optimise it for the testing program. Parametric cases, associated with different positions of the SR assembly have been set up and analysed. A quasi-static analysis has been performed for each case, accounting for the LBE volume displaced by the insertion of the SR bundle, by introducing appropriately positioned additional mass sources. Velocity and pressure fields, as well as pressure drop magnitudes and mass flow rates through relevant guide tube hole outlets have been calculated and compared. The CFD analysis proved that the outer boundary of the test section does not impact the expected performance of the SR (rapid transient downward insertion). Preliminary simulations reproducing the timely repositioning of the SR/CR in COMPLOT using procedures of automatic volume mesh regeneration, consistently with the rod imposed displacement, are illustrated.

Keywords: *MYRRHA, COMPLOT, CFD, parametric study*

Introduction

The COMPLOT (COMPonent LOop Testing) is an experimental facility under development at SCK•CEN [1] that will be employed to support the MYRRHA [2] reactor design to characterise the hydrodynamic behaviour of full-scale reactor components in a flowing LBE environment such as Fuel Assembly (FA), Spallation Target (ST), Control Rod (CR), Safety Rod (SR). COMPLOT will be an isothermal loop, operating within a temperature range of 200 °C – 400 °C with upwards LBE flow. It will be used to test the components pressure drop characteristics, the flow induced vibrations and the dynamics of the moving parts in LBE, such as control and safety rods.

This work focuses specifically on the test section outlet to be used for the SR, in the framework of FP7 MAXSIMA European project [3]. In the MYRRHA reactor design, the SR component consists of a long guide tube in which the LBE enters from the bottom at the core inlet and flows upward past/through the SR internals. Above the core, the LBE exits the guide tube through a series of outlet holes during steady state normal operation, but more importantly during the rapid transient downward insertion of the SR.

To ensure that the COMPLOT test section outlet is representative for the MYRRHA conditions, it needs to be optimised to ensure that its performances are not influenced by any test section design features. SCK•CEN have conceptualised a test section design where the lower part of the guide tube is inserted into a hexagonal "pipe" representing the core region. 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. It is the diameter of this annulus that must be optimised, to reduce space, weight, and cost limitations for the testing program.

The aim of the CFD (Computational Fluid Dynamics) analysis presented here is to optimise the test section outer diameter such that it does not impact the performance of the SR. The concern is that if the test section outlet diameter (D) is too small it is likely that the flows from the guide tube holes will strongly impinge on the test section wall with some feedback effects.

Model settings. Boundary conditions

The LBE fluid properties and the mass flow rate at the inlet of the guide tube are given in Table 1 below:

Table 1 Thermophysical properties of molten LBE

Parameter	Values
LBE Temperature	200°C
Density	10470kg/m³
Dynamic viscosity	2.432E-03Pa.s
Mass flow rate in inlet	8 kg/s

While during steady state normal operation, the LBE flows into the bottom of the SR guide tube at 8 kg/s, the downward insertion of the SR implies that additional mass source should be considered. We calculated the additional mass sources by assuming a linear insertion speed V of the component and used it boundary conditions.

In determining the boundary conditions we must take into account that the SR is formed by two separate components, namely the absorber pins bundle below and the tungsten ballast above. The absorber bundle consists in 12 pins separated by a pushing rod and a shaft attached to the pushing rod and long all way through the guide tube. During the downward insertion of the safety rod the ballast is free to move on the shaft, with a delayed response relative to the balance of the weight, drag and buoyancy of the tungsten ballast in the LBE fluid. Hence the additional mass source due to bundle/ballast insertion will come from different sources. In addition, the downward movement of the bundle/ballast assembly will increase the fluid volume at the top of the guide tube, likely creating a suction effect and pulling LBE into that space. Considering these complex possible boundary conditions, our approach considers the most extreme (up and down) and some intermediate Safety Rod positions, each one with the corresponding boundary conditions.

For each SR position, the test section outer tube of dimension "D" was varied between two extremes in order to understand the relative effect of the outer annulus on the LBE flows through the guide tube holes, taking as minimum and maximum outer tube external dimensions 6" and 12" respectively. The study was implemented in the commercial CFD code STAR-CCM+ [3]. The geometrical model was constructed entirely with the embedded 3D-CAD modeller, paying particular attention to the order in which the operations have been performed such that a complex parametric study could be easily realized. The SR bundle, the guide tube and the annular pipe have been constructed separately in different CAD models and successively assembled through import/export operations.

The parametric study was possible due to the variation of a minimal number of properly chosen dimensions imposed as design parameters. The three design parameters were: the *annular pipe Outer Diameter* and *Inner Diameter* (the choice is motivated by the fact that the annular pipe was constructed by revolving the sketch profile which gives the pipe's thickness) and the *quote position of the bundle* (up, intermediate, down), implemented as a translation vector.

Thanks to the symmetry with respect to a vertical axis, only a half of the computational domain has been considered for the analysis, reducing in this way the computational costs associated to the simulations. The polyhedral mesh model was employed, for a total number of cells varying in the range of 2 - 3 millions. Volumetric controls have been applied for the mesh refinement necessary in the zone of the absorber pins. The geometric models of some of the considered cases are illustrated in Figure 1 with a detail view on the mesh.

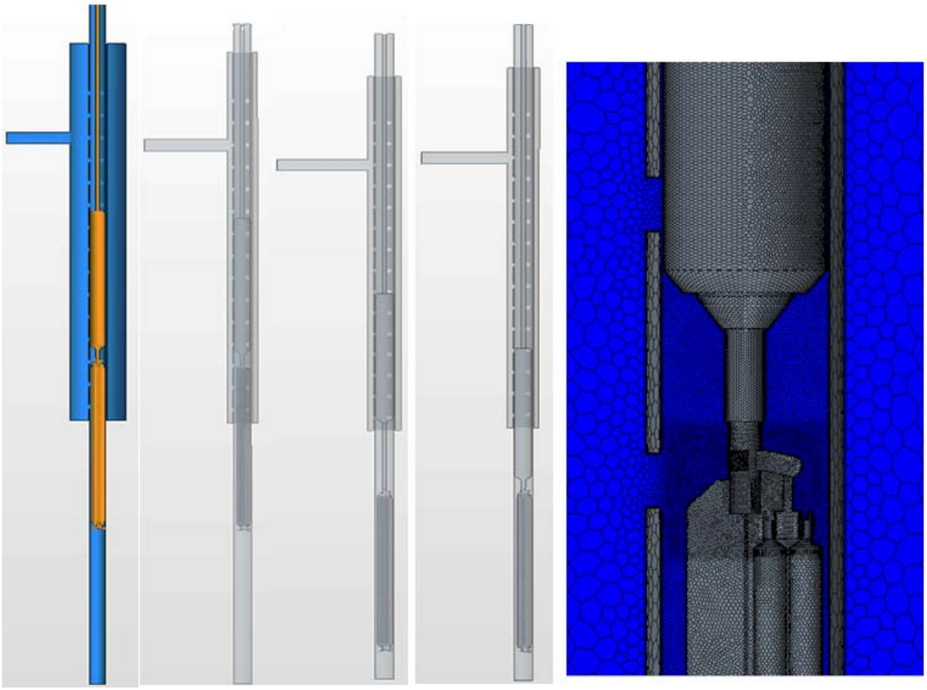


Fig. 1 Geometry and mesh in the case of the dimension D = 12"/6", with the bundle in up/intermediate/down position

In order to determine the additional mass sources originating from the insertion of the SR, we calculated the resulting mass flow rates relative to the cross-sectional area of each part of the assembly and to the reference velocity, V, and we obtained the following quantities:

i) 4.4 kg/s from the insertion of the shaft;

ii) 73.1 kg/s from the displacement of the ballast moving freely on the shaft;

iii) 39.5 kg/s from the displacement of the absorber pins bundle.

We obtained different additional mass sources, negative or positive, depending on the position of the various components of the SR assembly, which we located as follows: A) at the Top of the Ballast; B) Between ballast and bundle; C) at the Bottom of the Bundle merged with the Inlet, as shown in Figure 2. The computation of the mass sources together with their locations is described in Table 2. When is not at rest, the ballast is assumed to move with V/2 and will therefore be mid-way through the stroke, displacing 36.5 kg/s.

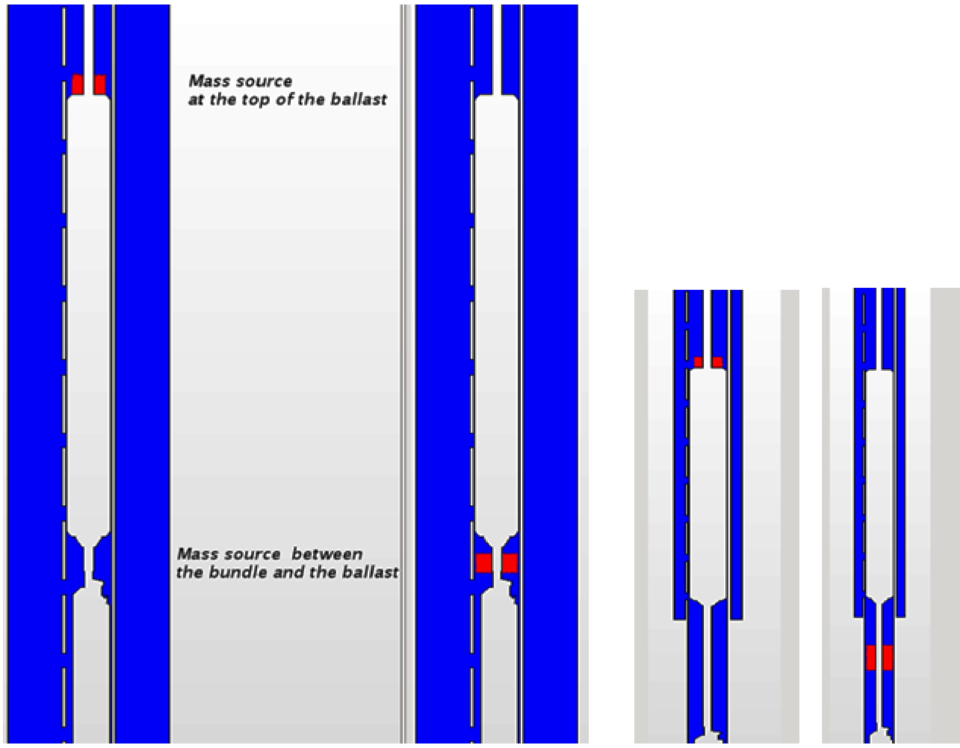


Fig. 2 Mass sources location for Bundle Up/Ballast Up (left) and Bundle Down/Ballast Intermediate (right)

Table 2 Additional mass flow rates and mass sources location

Description/Additional mass flow rates [kg/s]	**Source Top of Ballast**	**Source Between bundle and ballast**			**Source Bottom of Bundle**	
	Ballast	Shaft	Ballast	Bundle	Bundle	Inlet
Ballast up at rest Bundle up/down (at V)	0	+4.4	0	-39.5	+39.5	+8
Total	0	-35.1			+47.5	
Ballast intermediate (V/2) Bundle down (at rest)	- 73.1/2	+4.4	+73.1/2	-39.5	+39.5	+8
Total	-36.55	+1.45			+47.5	
Ballast intermediate (V/2) Bundle down (at rest)	-73.1/2	0	+73.1/2	0	0	+8
Total	-36.55	+36.55			+8	

On the basis of the resulting mass flow rates calculated above, we implemented the mass, the momentum and the turbulence sources. The implementation of the mass sources in STAR-CCM+ is realized by means of a series of field functions that are meant to do the following:

• define volumes properly localized as volume integrals of the characteristic function;

• define negative/positive mass sources by dividing the calculated additional mass flow rates to the corresponding volumes;

• define the total mass source by summing the effective mass sources corresponding to the localized volumes.

Numerical results.Velocity, pressure, mass flow profiles
For all the cases, we retrieved the contour plots of the velocity and pressure fields on the symmetry plane section of the domain and the velocities at the relevant guide tube hole outlets (the three lower layers). We calculated the mass flow rates through these relevant outlet holes, as surface integrals of the radial component of the velocity localized at each layer of holes and compared them by varying the "D" dimension.

We illustrate here the results of some of the cases, due to space limitations, while all the considered cases are described in [5]. In particular, Figure 3 shows the radial velocity plots on the three lower layers of holes for D = 12"/10"/8"/6" in the case of the bundle down - ballast at rest, while in Table 3 we report the mass flow rates through holes and the pressure drops.

The comparison clearly shows that the development of the LBE flow through the outlets is quite similar for all the geometries considered.

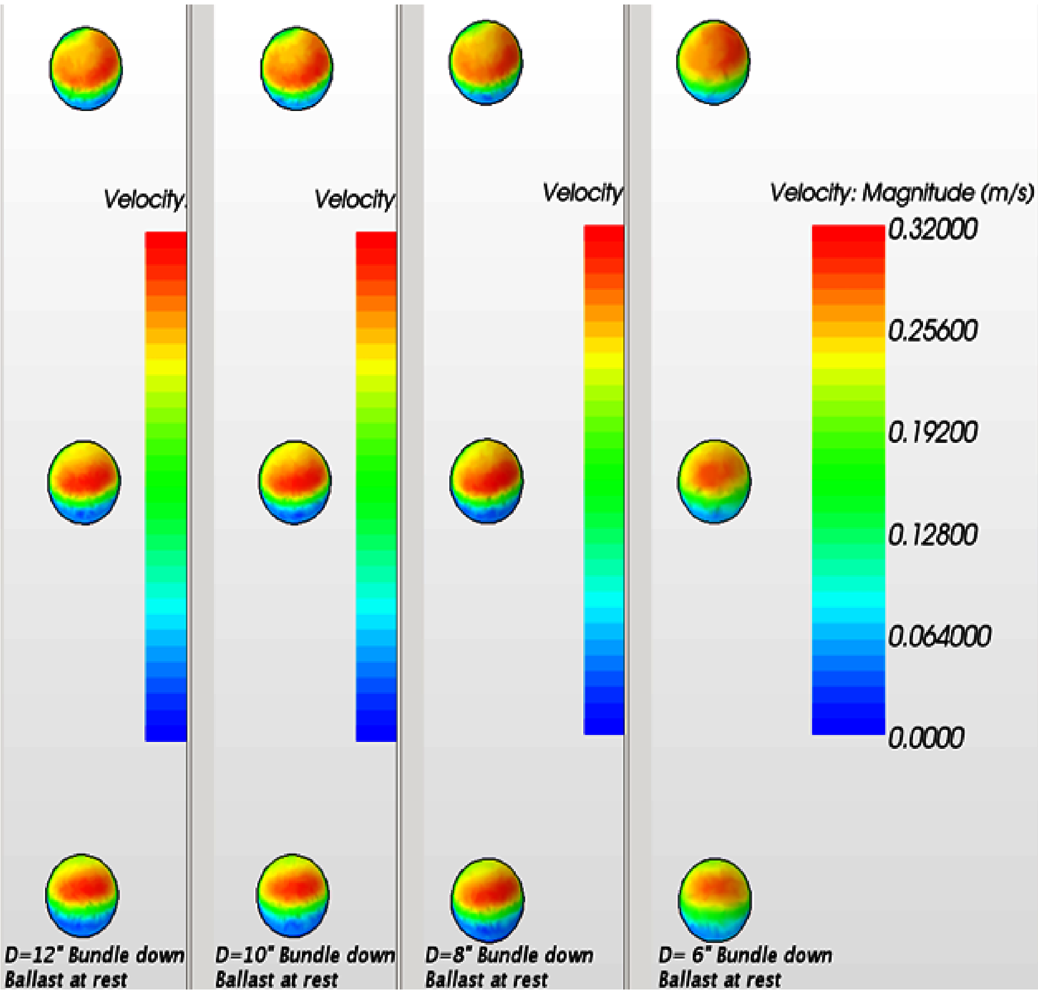


Fig. 3 Velocity magnitudes at the three lower holes for
D = 12”/10”/8”/6”, Bundle down - Ballast up at rest

Table 3 Mass flow rate and pressure drop for
Bundle down–Ballast up at rest

Holes position	**Mass flow rate through holes** [kg/s]			
	D = 12"	D = 10"	D = 8"	D = 6"
1st layer	1.46	1.43	1.47	1.45
2nd layer	1.80	1.78	1.80	1.75
3rd layer	2.07	2.12	2.08	2.11
	Pressure drop Inlet-Outlet [kPa]			
	17.60	17.64	17.58	17.20

We also measured the pressure drops across the bundle, as the difference of pressure between the bottom of the rods bundle and the head of the shaft, and across the ballast, as the difference of pressure between the head of the shaft and the top of the ballast.

In Table 4 and Table 5 are illustrated some numerical results by comparing the two extreme dimensions for the case considering the bundle just after kick-off: Bundle Up (at V) and Ballast Up (at V/2), respectively.

Table 4 Mass flow rates through holes
Bundle up (at V), Ballast Up (at V/2)

"D" Dimension	**Mass flow rate** [kg/s]		
	1st layer	2nd layer	3rd layer
D=12"	5.13	6.43	8.58
D= 6"	5.30	6.53	8.33

Table 5 Pressure drop
Bundle up (at V), Ballast Up (at V/2)

"D" Dimension	**Pressure drop** [kPa]		
	Inlet – Outlet	Across bundle	Across ballast
D=12"	30.8	21.2	13.0
D= 6"	30.3	21.0	14.0

The contour plots of the velocity field on the symmetry plane section of the domain and the velocities at the relevant guide tube hole outlets are illustrated in Figure 4. The comparison clearly shows that the diameter of the outer pipe doesn’t influence the LBE flow.

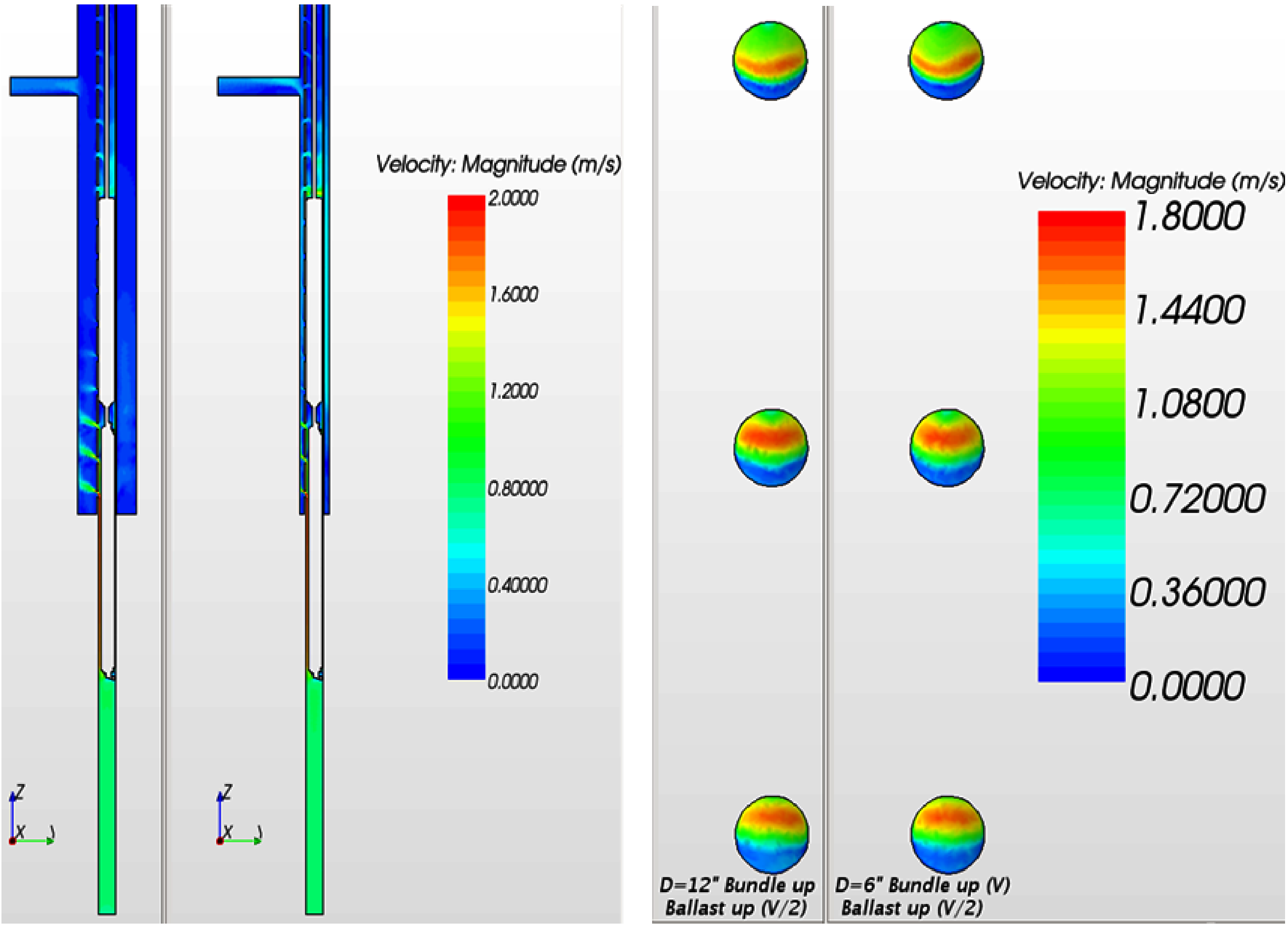


Fig. 4 Velocity plots through sectioned domain (left) and through the outlet holes (right) for D = 12"/6", Bundle Up (at V), Ballast Up (at V/2)

Further work. Preliminary transient simulations
The further work to be performed in COMPLOT is to simulate the timely repositioning of the SR/CR assembly, in a flowing LBE environment. For the moment, in pre-test phase, we considered a simplified computational model of the CR component since the effective one is quite large and difficult to manage in the framework of CFD transient simulations involving moving meshes.

The main set up for reproducing the insertion of the rod consists of dividing the computational domain into regions: one containing the moving part, two regions up and down, appropriately coupled with the moving part through interfaces and the rest of the domain.

Our strategy for reproducing the movement consists of combining an imposed translation motion of the absorber pins bundle with a morphing motion of the up and down regions, leaving the rest of the domain stationary.

Since the morphing produces deformations of the mesh (compressions and extensions), we are able to apply re-meshing strategies that we have previously developed [5], in a selective manner, only on the affected parts. This represents an important achievement since we can prevent from re-meshing the pin bundle assembly, which is the most costly in terms of computational power (in COMPLOT we expect to employ about 10 M Cells).

The strategy of re-meshing consists of coupling the effective displacement performed by the rod with the update of the underlying geometry, by imposing the displacement as translation vector at CAD level, and keeping the quality of the mesh under control by means of appropriate metrics.

In the three pictures in Figure 5 the mesh of the up and down regions is shown deformed (left) and regenerated (right) before the successive time steps of the translation motion.

The aim is to numerically reproduce the control/safety rod system in COMPLOT with imposed displacement history (taking into account acceleration and inertial effects), to measure the numerical forces applied to the control rod and to compare them with the experimental tests.

We reproduced the mechanical movement with the strategy described above, imposed a velocity to the bundle with a given acceleration, measured the drag force and verified its independence of the mesh and of the re-meshing operations (illustrative animations have been included in [6]).

Indeed, the plot of the drag force shows that only a small perturbation, without important effects on the flow, has verified when the mesh had to be regenerated.

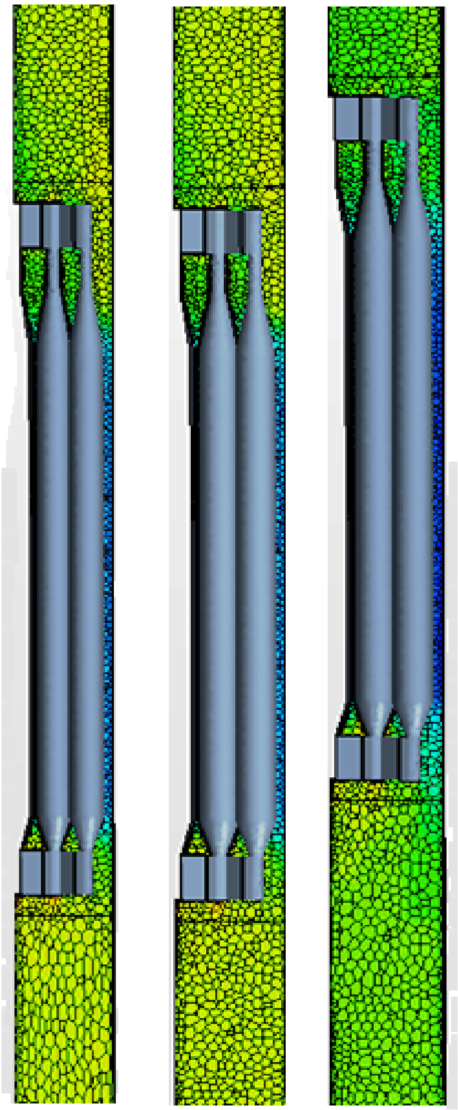
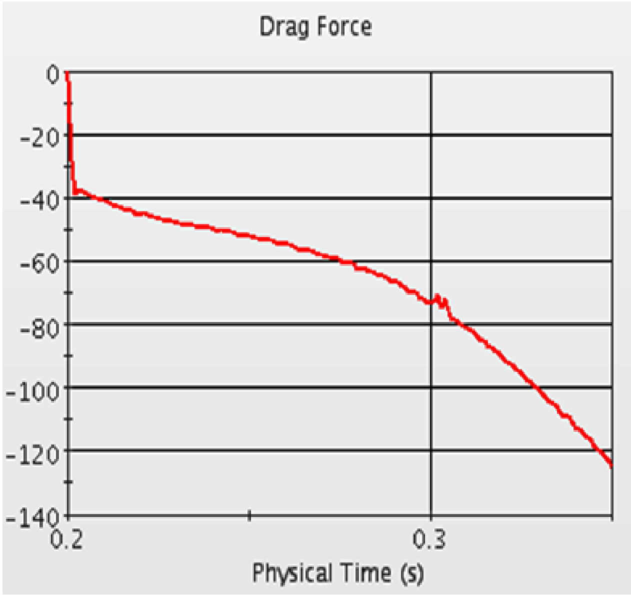


Fig. 5 Re-meshing procedure



Conclusions

A parametric study aimed at the dimensioning of a test section was set up in a CFD quasi – static analysis with additional mass sources imposed as boundary conditions. The analysis proved that the diameter of the outer annular pipe which receives the LBE flow did not influence the flow induced by the displacement of the moving component, thus helping the designer to decide for the optimization of the facility in the testing program.

Preliminary transient simulations have been performed on a simplified CR model in order to test the independence of the solution of the re-meshing operations and to prepare the conditions for full-scale reproduction of moving parts in a LBE flow.

References

[1] MYRRHA Team (2012) "MYRRHA Technical Description Rev. 1.4", http://search.sckcen.be

[2] MAXSIMA official website http://maxsima.sckcen.be/

[3] STAR-CCM+, Version 7.02, (2012), http://www.cd-adapco.com/

[4] Kennedy, G., Moreau, V., Profir, M.M. (2012) "CFD Optimization of the COMPLOT Test Section Outlet", CRS4 - SCK•CEN Technical Report.

[5] Profir, M. M. (2012) "Automated moving mesh techniques and re-meshing strategies in CFD applications using morphing and rigid motions", CRS4 Internal Report

[6] Profir, M.M. (2013) "CFD simulation of the control/safety rod system", MAXSIMA 1st semi-annual review meeting, May 6-8, 2013, Polaris, Sardinia