

SCALING ANALYSIS FOR THE EUROPEAN HEAVY LIQUID METAL SCALED POOL FACILITY ESCAPE

K. Van Tichelen¹, Matthias Vanderhaegen^{1*}, S. Jajarayu², S. Keijers¹ and F. Roelofs²

¹ SCK•CEN, Mol, Belgium

² NRG, Petten, Netherlands

Abstract

The understanding of the thermal-hydraulic phenomena occurring in the reactor pool is a critical issue in the design of the MYRRHA system, the European Technology Pilot Plant for the Lead Fast Reactor. The E-SCAPE facility is a thermal-hydraulic scale model of the MYRRHA reactor using lead-bismuth as coolant. Non-dimensional analysis and CFD simulations are used to determine the scaling factors and transposition of the experimental results to the real scale situation. The joint efforts of SCK•CEN and NRG within the European project THINS in applying CFD to fix the design parameters of E-SCAPE is reported.

Introduction

Europe, through the Sustainable Nuclear Energy Technology Platform, has defined its strategy and priorities for fast neutron reactors that are the most likely to meet Europe's energy needs in the long term, in terms of security of supply, safety, sustainability and economic competitiveness. In its Concept Paper of the European Sustainable Nuclear Industrial Initiative, MYRRHA is identified as the European Technology Pilot Plant for the Lead Fast Reactor [1]. Figure 1 gives a view of the MYRRHA system [2]. MYRRHA is a pool-type reactor with liquid lead-bismuth eutectic (LBE) as primary coolant. The diaphragm (1) separates the cold lower plenum from the hot upper plenum. Primary heat exchangers (2), primary pumps (3), core containing fuel assemblies (4), spallation target (5), interim fuel storage (6) and fuel handling machines (7) are all submerged in the reactor pool.

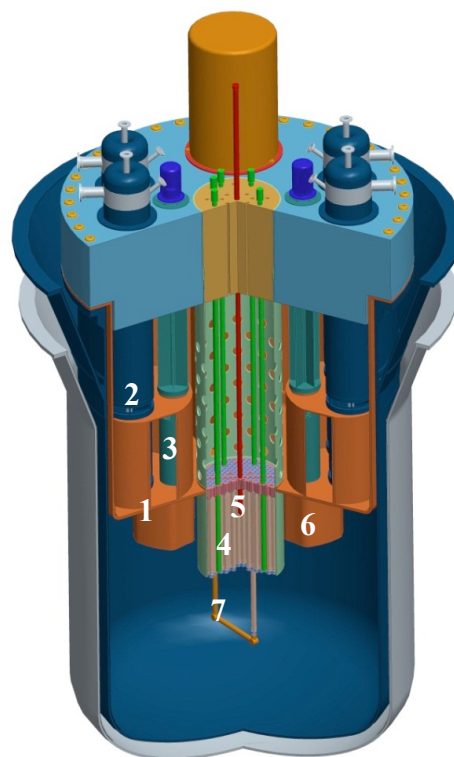


Figure 1 The MYRRHA system

The proper understanding of the thermal-hydraulic phenomena occurring in the upper and lower plena of the reactor pool is an important issue in the design of the MYRRHA system and lead-

* Currently: CEA, Cadarache, France

In [3] and [4], the thermal-hydraulic challenges in the plena of a pool-type liquid metal cooled reactor are identified:

- Also important to investigate are the flow distribution e.g. between different heat exchangers, the primary pump jet behaviour, residence times of fluid particles and the velocity and temperature fields in non-symmetrical flow conditions e.g. with pump or heat-exchanger failure as assessed in preliminary analyses for a simplified lower plenum [5].

2. Scaling approach

2.1 Non-dimensional analysis

- Group 1: Richardson $Ri \equiv \frac{g\beta\Delta TL}{U^2} = \frac{\text{Buoyancy}}{\text{Inertia}}$

	Euler	$Eu \equiv \left(\frac{fL}{D} + K \right)_i = \frac{\text{Friction}}{\text{Inertia}}$
	Heat source number	$H.S. \equiv \frac{q'''L}{\rho c_p U \Delta T} = \frac{\text{Heat source}}{\text{Axial energy change}}$
• Group 2:	Stanton number	$St \equiv \frac{h}{\rho c_p U} = \frac{\text{Heat transfer at wall}}{\text{Convection}}$
• Group 3:	Reynolds	$Re \equiv \frac{\rho UL}{\mu} = \frac{\text{Inertia}}{\text{Viscosity}}$
	Péclet number	$Pe \equiv \frac{\rho c_p UL}{k} = \frac{\text{Convection}}{\text{Conduction}}$

The Stanton number used in this text is derived from Eguchi's boundary condition number using the Péclet number. The term Heat source number is taken from Ishii's reference paper on the three-level scaling approach [8]. The numbers in group 1 are from source terms in the momentum and energy equations and they have direct impact on the solutions U, p and T. If they are relevant in the case at hand, priority must be given to these numbers when selecting similitude conditions. The number in group 2 is due to boundary conditions and is negligible if heat flux through fluid boundary is negligibly small in both experiment and actual plant. Thermal insulation and large wall thicknesses make this a valid assumption in this work. The numbers in group 3 are coefficients of second derivatives in the momentum and energy equations. In most cases, they have rather local and less significant effect on the solutions, partly because their effect is limited to areas where the value of the second derivatives of U and T are large enough and mainly because of molecular diffusion is often smeared out by overwhelming turbulent diffusion. In this work, one assumes the latter is valid if Reynolds numbers remain above 10000 and Péclet numbers above 100.

When flow patterns near free surfaces are of interest and/or flow is driven by gravity, the Froude number should be preserved:

$$Fr \equiv \frac{U^2}{gL} = \frac{\text{Inertia}}{\text{Gravity}}$$

The flow in the lower plenum is mainly dictated by the two pump jets. The flow in the core exit region is also a multi-jet region. Peterson [9] states that for forced jets, if the aspect ratio between jet penetration length and inlet diameter is preserved, similar mixing and entrainment can be expected with full scale and reduced scale jets. If, for buoyant plumes, also the Richardson number is preserved, the transition height from jet to plume behaviour scales properly.

In principle, a scaling strategy can be derived which preserves all non-dimensional quantities above while using the prototypic fluid. Reynolds, Péclet, Richardson and Heat source number analogy are preserved if:

$$U \sim \frac{1}{L} \quad \Delta T \sim \frac{1}{L^3} \quad q''' \sim \frac{1}{L^5}$$

Since pressure loss coefficients are a combinations of friction and form loss terms, the Euler number is easily fulfilled by adapting form loss coefficients to compensate for changes in friction loss terms (e.g. by using orifices). The Stanton number defines the heat transfer coefficient at the boundaries.

The latter is generally determined by the Reynolds and Péclet number. Preserving all non-dimensional quantities and using the prototypic fluid, leads to values for temperature changes and power densities that cannot be realized in practice. A selection of relevant non-dimensional quantities is thus necessary for each flow phenomenon investigated.

2.2 The E-SCAPE case

In the first stage, E-SCAPE focuses on high-velocity forced convection flow. In this case, the Euler and heat source number are the driving non-dimensional numbers. The preservation of the Euler number is fulfilled if pressure loss coefficients in the black boxes are preserved. Again, this requirement is easily fulfilled by adapting form loss coefficients to compensate for changes in friction loss terms (e.g. by using orifices). The heat source number provides a linear relation between the power density in a black-box and the temperature difference between in- and outlet.

The relation between velocity and length scaling should be determined on the basis of other dimensionless numbers (taking into account the use of the prototypic fluid):

- If free surface appearance is considered important, the Froude number dictates that velocity scales as the square root of height. (In principle, a different scaling can be chosen for heights and horizontal lengths. This is not considered for E-SCAPE since the preservation of aspect ratio is important for jet behaviour.) This is true as long as Reynolds and Péclet numbers are high enough so that turbulent diffusion dominates.
- If this is not the case or if local turbulence characteristics are important (e.g. to assess erosion), velocity should scale as the inverse of length according to Reynolds.
- If time conservation is important (e.g. when chemical reactions are of interest) velocity should scale as length.
- When buoyancy is important, the Richardson number provides a relation between velocity, length and temperature difference. Power density (or equivalently temperature difference), length and velocity scale can no longer be chosen independently. If one specifies to preserve temperature differences, one can derive that – similar to the Froude scaling - velocity scales as the square root of height. Power density should scale as the inverse of the square root of height.

Reynolds scaling leads to very high velocities. From a technical point, however, it is not feasible to have flow velocities above 2 m/s in the experiment. It was therefore decided to consider an alternative "maximum velocity scaling" which preserves the velocities at the pump and core exits. In this case, the Reynolds number scales as the length.

A final criterion to be considered in the scaling analysis is cost. For this purpose, the maximal scaling factor for E-SCAPE is set to 1/5.

The scaling parameters for the four different scaling approaches that were investigated in the frame of E-SCAPE are listed in Table 1. Selecting one approach for a particular phenomenon investigated leads to compromises for other dimensionless numbers. Scaling factors are then to be chosen such that the general flow characteristics remain valid (e.g. high Reynolds for turbulent flow and high Péclet number for negligible axial conduction). Figure 2 shows the evolution of the Reynolds and the Péclet number at the pump exit in forced and natural convection as a function of the scale for the Froude/Richardson scaling approach. For scaling factors below 1/8 Reynolds and Péclet numbers become too low.

Technical reasons limit the core power and density in the scale model. No scaling strategy can therefore reproduce full power conditions. For the scaling factors considered for E-SCAPE, the 7% peak core decay heat can be simulated for Froude, Richardson and time scaling. For maximum velocity and Reynolds scaling, the scaled core power and density are too high.

Parameter	Froude/Richardson		Reynolds		Time		Max. velocity	
Length	l	1/5	l	1/5	l	1/5	l	1/5
Diameter	l	1/5	l	1/5	l	1/5	l	1/5
Velocity	$\sqrt{l}$	1/2.24	1/ l	5	l	1/5	1	1
Flow rate	$\sqrt{l^5}$	1/55.9	l	1/5	l^3	1/125	l^2	1/25
Δ Pressure	l	1/5	1/ l^2	25	l^2	1/25	1	1
Δ Temp.	1	1	1	1	1	1	1	1
Core power	$\sqrt{l^5}$	1/55.9	l	1/5	l^3	1/125	l^2	1/25
Power/vol.	1/ $\sqrt{l}$	2.24	1/ l^2	25	1	1	1/ l	5
Time	$\sqrt{l}$	1/2.24	l^2	1/25	1	1	l	1/5
Reynolds	$\sqrt{l^3}$	1/11.2	1	1	l^2	1/25	l	1/5
Péclet	$\sqrt{l^3}$	1/11.2	1	1	l^2	1/25	l	1/5
Richardson	1	1	l^3	1/125	1/ l	5	l	1/5
Froude	1	1	1/ l^3	125	l	1/5	1/ l	5
Euler	1	1	1	1	1	1	1	1
Heat source	1	1	1	1	1	1	1	1

Table 1 Scaling parameters for different scaling approaches and values for a scaling factor of 1/5

3. CFD analysis

To verify the validity of the scaling approaches CFD calculations have been performed by NRG using Fluent and SCK•CEN using CFX, both on the full-scale reactor and on different scale models. Since E-SCAPE focuses on high-velocity forced convection flow in the first stage, all calculations presented in this work were isothermal. For comparison reasons, if necessary, velocities and turbulent kinetic energies were non-dimensionalised with respect to the pump outlet values and lengths with respect to the pump outlet diameter.

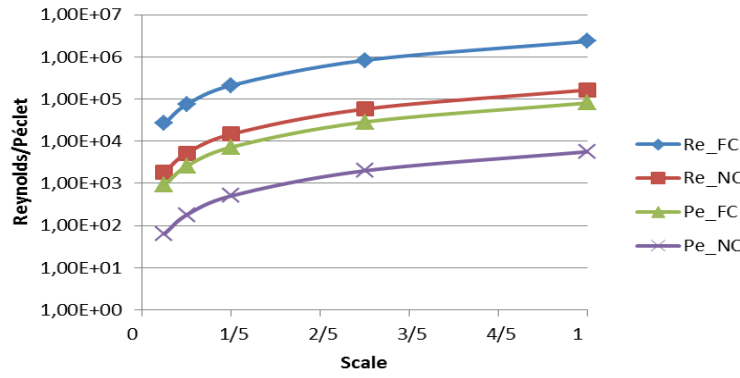


Figure 2 Evolution of Reynolds and Péclet numbers for Froude/Richardson scaling

3.1 Computational domain, boundary conditions, fluid properties and numerical approach

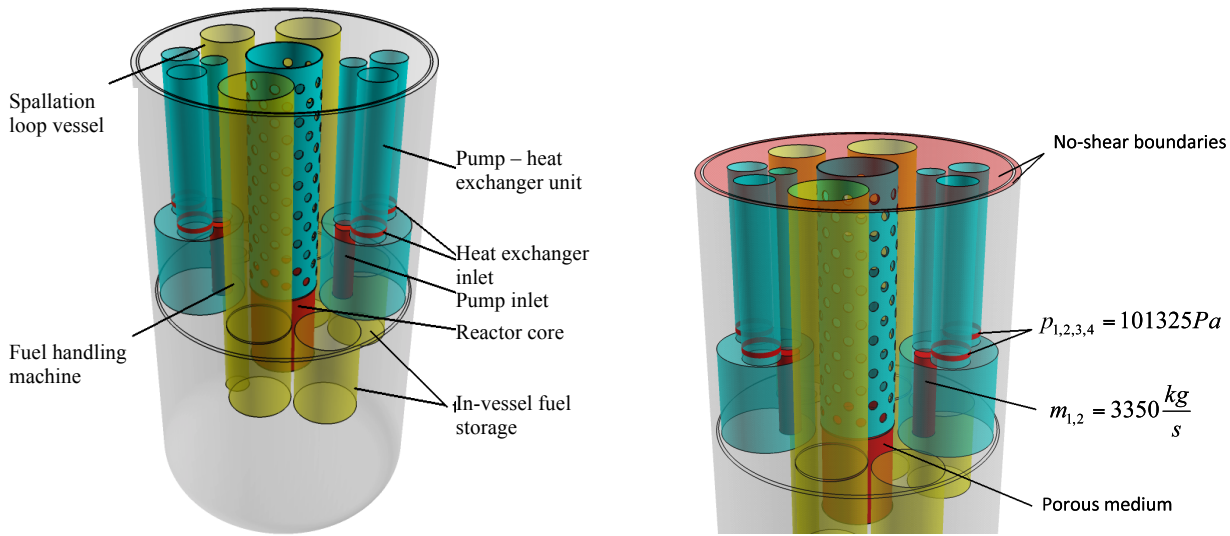


Figure 3 The computational domain and boundary conditions for CFD simulations

The computational domain is shown in Figure 3. The two pump inlet ducts represent the inlet boundary of the computational domain, the four boiler inlets represent the outlet boundary. Since in the current MYRRHA design the spallation loop vessel is no longer present, it was discarded in the CFX simulations. All LBE properties are taken from the LBE Handbook [10] at a temperature of 335°C. The boundary conditions are shown in Figure 3. The free surfaces are modelled as no-shear boundaries. The position of the boundary is at the free surface level expected in the upper plenum. This simplifies calculations while test calculations have shown that the influence on the flow pattern is marginal: low vertical velocity components below the surface lead only to small and local free surface deformations. The inlet mass flow rates are specified at the pump inlet ducts. Turbulent intensities are set at 10% at the inlets. A reference pressure of 101325 Pa is set at the heat exchanger inlets. The nuclear core is modelled as porous medium with 4 different resistance zones corresponding to different zones in the real core. The resistance coefficients in the transversal direction were taken two orders of magnitude higher than the axial ones.

In Fluent two different three-dimensional grids were created for the full size simulations with similar structure but different hexahedral cell size to investigate grid sensitivity. Their main properties are shown in Table 2 and a sample of the grid is shown in Figure 4. The mean velocity

and turbulent kinetic energy profiles were compared at various sections of the geometry. Little variations were observed which led to conclude that a mesh size of 5.2 million cells was adequate. However, thanks to the availability of computer power, the scaled model simulations could be performed with the mesh of 8.8 million cells. The main properties of the unstructured tetrahedral grid used in CFX for the full size simulations are shown in Table 3. The total mesh size is limited by computational capacity. A local grid refinement at the pump exits proved to be necessary. Within a half sphere with radius 1.4 m, element sizes are reduced to 50 mm. A sample of the grid is shown in Figure 5. For the scaled calculations in CFX, elements are reduced to the values listed in Table 3.

	Grid 1	Grid 2
y^+	400 (1mm)	400 (1mm)
$\Delta_{x,y,z}^{HEX}$	50 mm	40 mm
$\Delta_{x,y,z}^{TET}$	30 mm	30 mm
Total number of cells	5,2M	8,8M

Table 2 Grid properties for Fluent

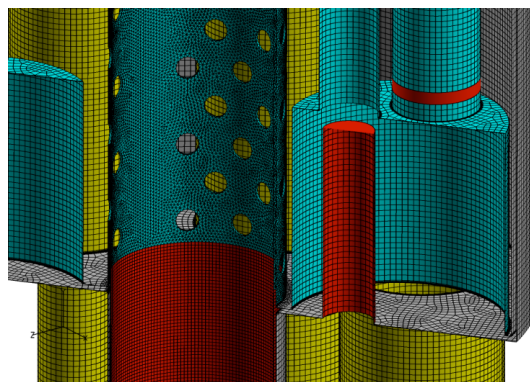


Figure 4 Sample of the grid for Fluent simulations

	1/1 Grid	1/5 Grid	1/8 Grid
y^+	< 400 (0.25 mm)		
$\Delta_{x,y,z}^{TET}$	max. 150 mm	max. 29 mm	max. 12,5 mm
$\Delta_{x,y,z,pump\ exit}^{TET}$	50 mm		
Total number of nodes	3,4M	3,8M	3,4M

Table 3 Grid properties for CFX

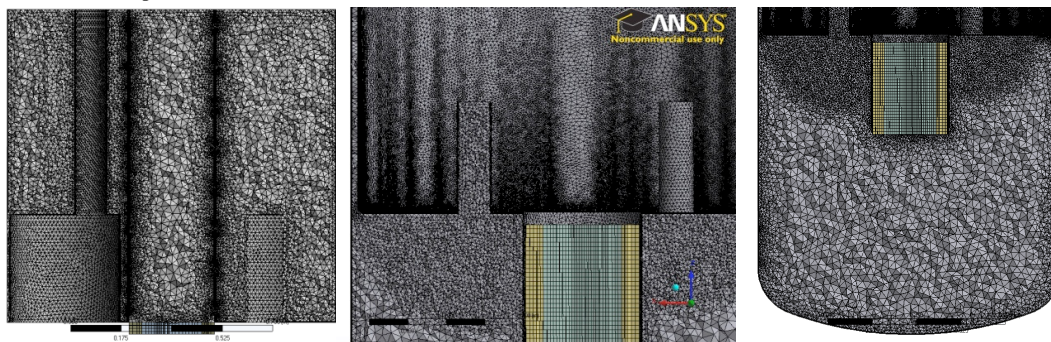


Figure 5 Sample of the grid for CFX simulations

Fluent and CFX are used for solving the steady-state RANS equations in combination with SST $k-\omega$ turbulence modelling. In Fluent, the PRESTO scheme is used for the pressure equation, first order upwind for momentum and turbulence quantities. Real velocity formulations are used in the porous zones. CFX solves the nodal-based, finite-volume hydrodynamic transport equations in a coupled way. A "high-resolution" differencing scheme is used for the momentum equations (i.e. a blend of first and second order upwind) whereas a first order scheme is used for the turbulent equations. Both steady-state and transient calculations were performed.

3.2 Results

The results of the Fluent calculations by NRG are presented in Figure 6 to Figure 9. The results of the CFX calculations by SCK•CEN are presented in Figure 10 to Figure 12. The calculations show a flow regime which is mainly determined by the pump jets and the small jets from the holes in the suspension tube.

In CFX the oscillation of the RMS values of the residuals after a number of iterations indicated that the solution might be unsteady. Indeed, transient simulations showed an oscillatory jet behaviour with a period of 150 seconds. This is shown schematically in Figure 11. The time interval between successive images is 30 seconds. This timing of this phenomenon corresponds with the oscillation frequencies observed for confined jets due to cross flow. Due to this transient behaviour, DES or SAS turbulence modelling might be more appropriate. This was however beyond the scope.

Two scaling approaches were considered by NRG for a scale factor of 1/5: Froude scaling and maximum velocity scaling. SCK•CEN considered four scaling approaches at two scaling factors 1/5 and 1/8: Froude scaling, time scaling, Reynolds scaling and maximum velocity scaling.

Despite the limitations of the models (oscillatory jet behaviour, porous medium and no-shear boundary approach), one can conclude from the results that:

- Qualitative comparisons show very similar flow patterns for all scaling strategies and both scaling factors when compared to the unscaled flow.
- The Reynolds scaling leads to unacceptable high flow velocities and free surface deformations. Although free surfaces were not modelled, the likeliness of large deformations can be deduced from the large pressure variations observed at the no-shear boundary.

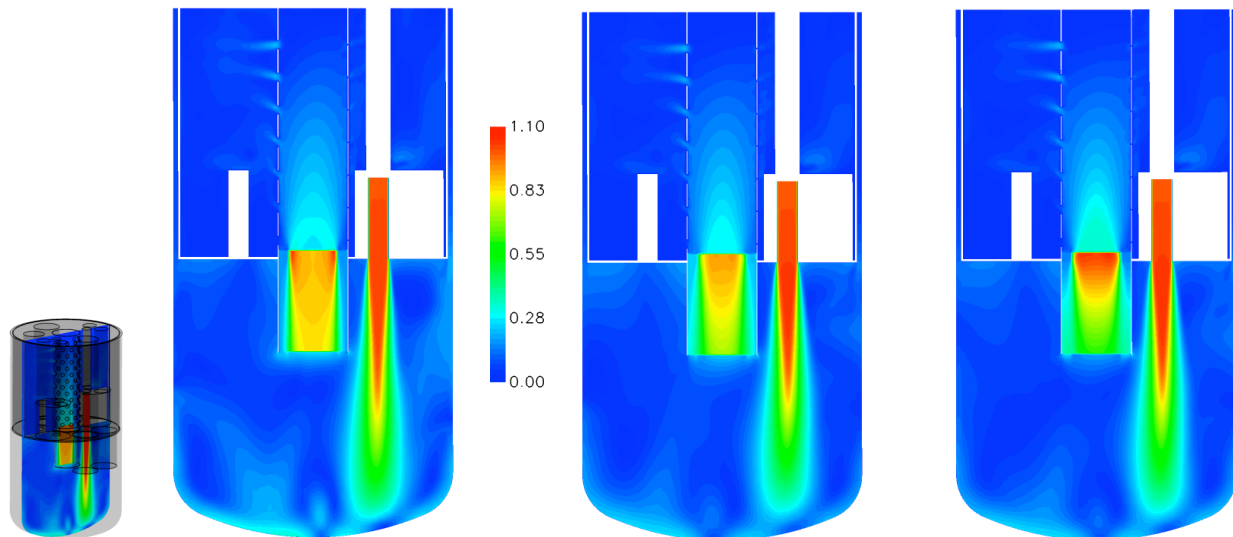


Figure 6 Dimensionless velocity patterns for the full scale model (left), the 1/5- Maximum velocity scale model (middle) and the 1/5- Froude scale model (right) in a vertical plane through pump 1

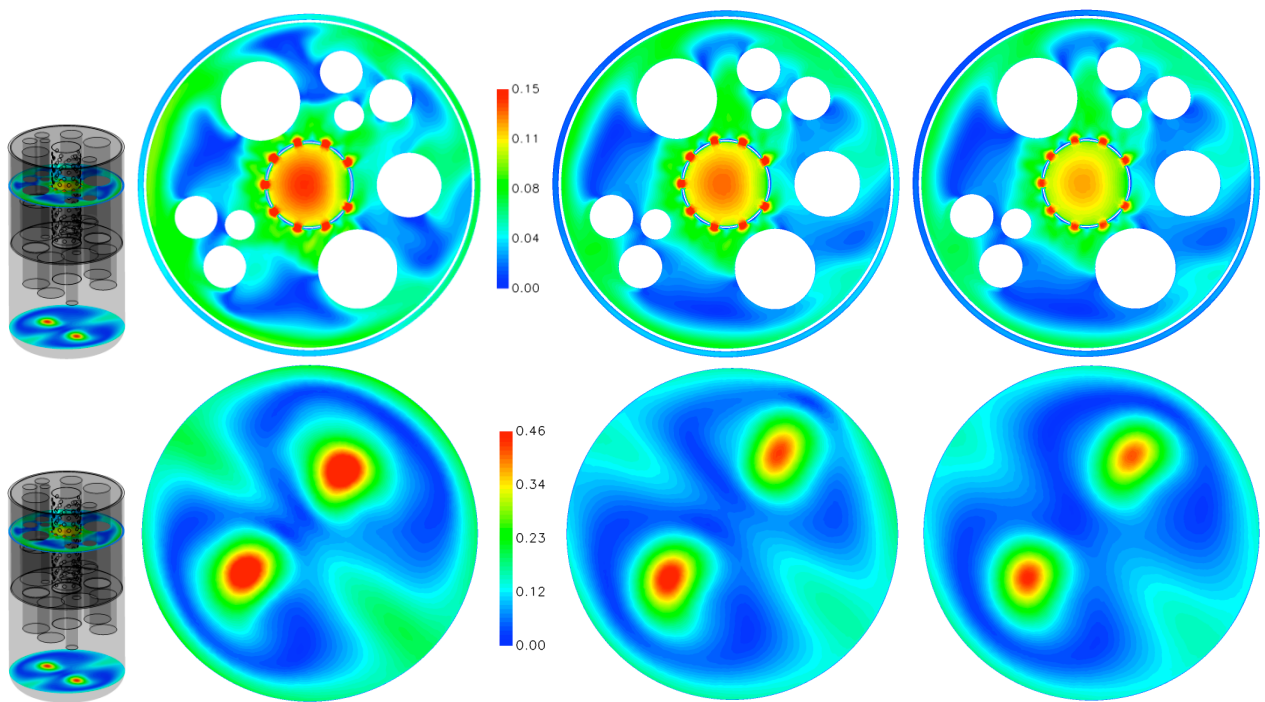


Figure 7 Dimensionless velocity patterns for the full scale model (left), the 1/5- Maximum velocity scale model (middle) and the 1/5- Froude scale model (right) in two horizontal planes

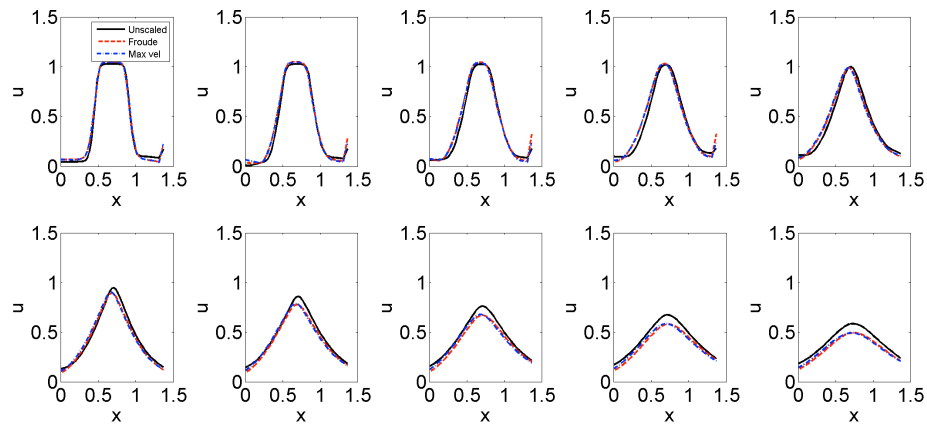


Figure 8 Dimensionless transverse velocity profiles at 1 to 10 pump diameters below pump 1

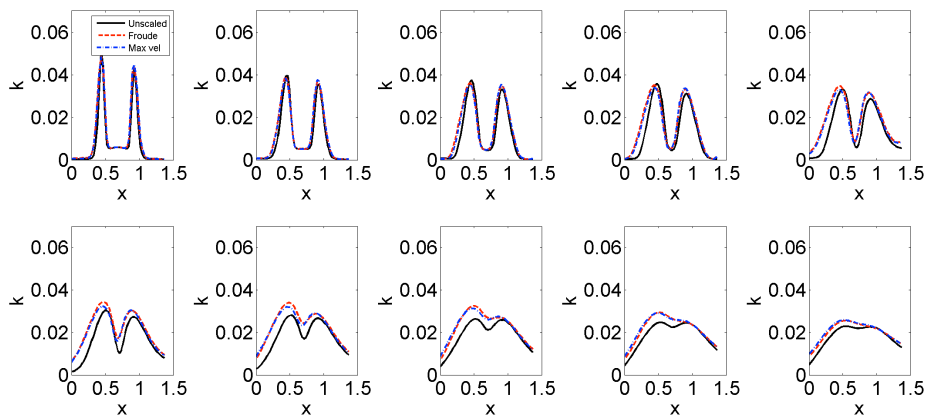


Figure 9 Dimensionless transverse TKE profiles at 1 to 10 pump diameters below pump 1

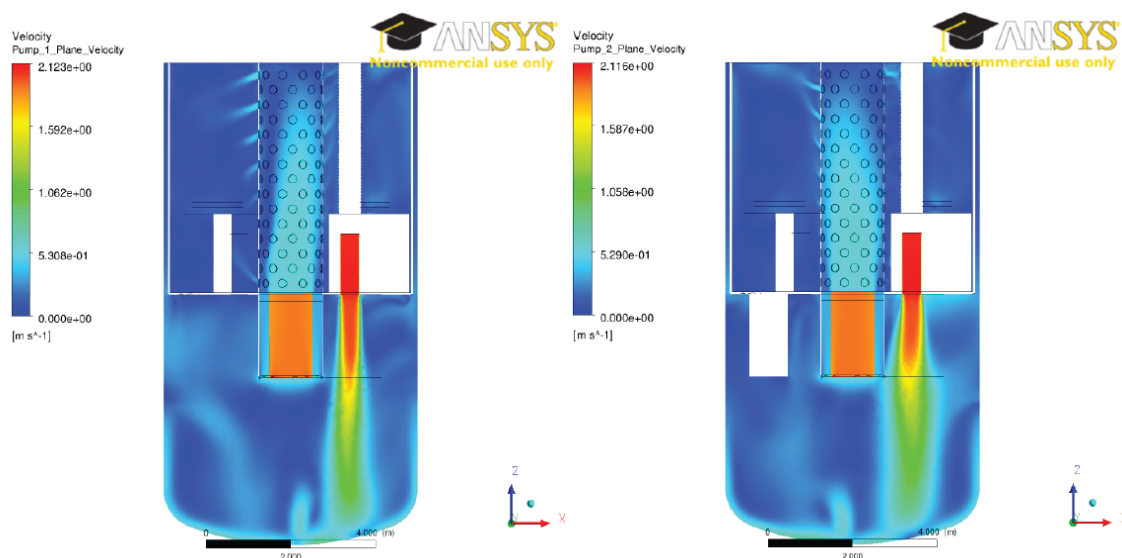


Figure 10 Velocity patterns in a vertical plane through pump 1 (left) and pump 2 (right)

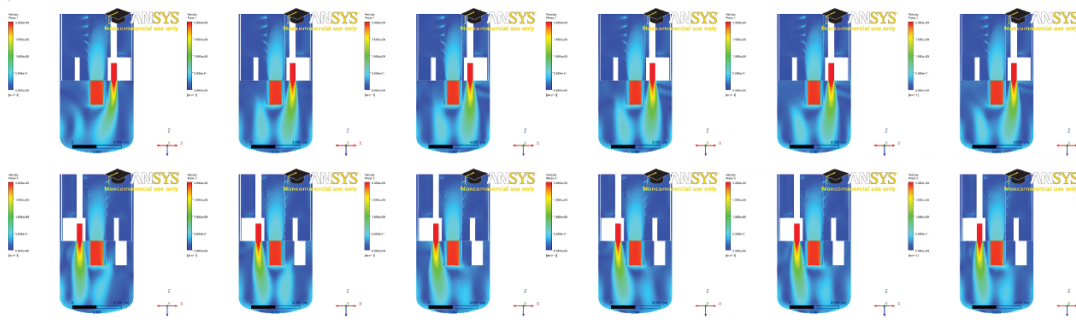


Figure 11 Oscillatory pump jet behaviour for pump 1 (top) and pump 2 (bottom)

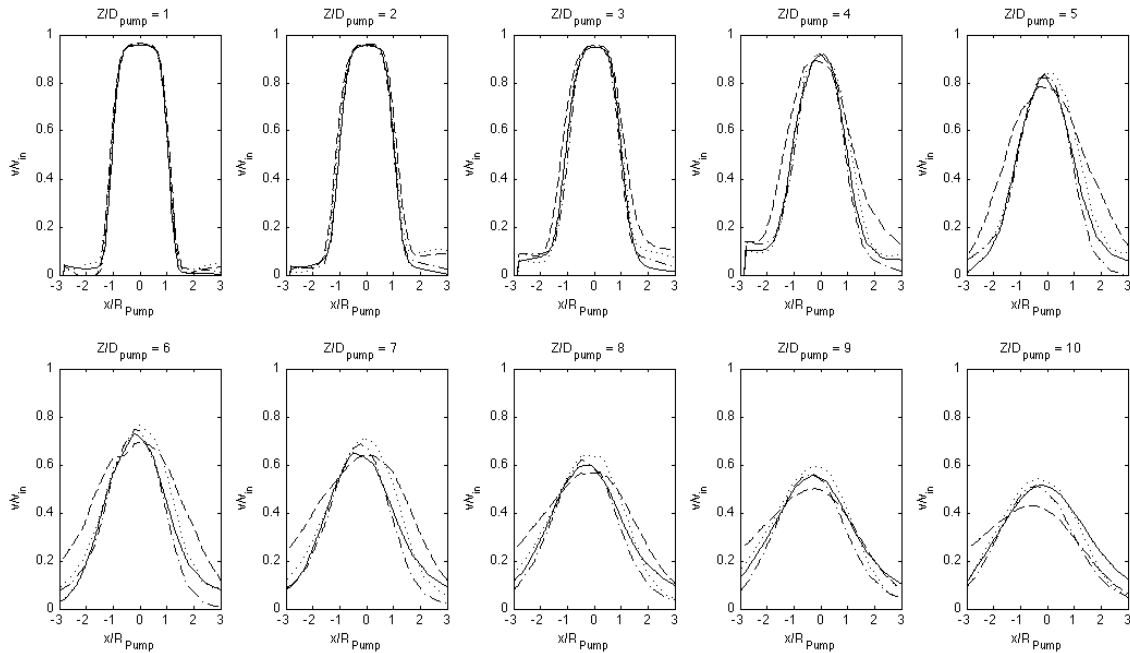


Figure 12 Dimensionless transversal velocity profiles at 1 to 10 pump diameters below the exit of pump 1 (— full scale / ---- 1/5 Froude / 1/5 Maximum velocity / -.- 1/5 Reynolds)

4. Conclusions

From dimensional analysis, different scaling options have been defined depending on the focus of the experimental program: Froude, time, Reynolds and maximum velocity scaling. These options were investigated for the scaled MYRRHA facility using two CFD codes, Fluent and CFX, for two different scaling factors 1/5 and 1/8. The Reynolds scaling was found to be impractical and leading to large deviations in free surface behaviour and was therefore discarded. The other scaling strategies lead to flow patterns that are close to the full scale situation for both scaling factors.

Economic considerations push towards smaller scales; practical considerations such as power density and instrumentation access demand for larger scales. For the E-SCAPE facility, a vessel diameter of 1.2 m was considered optimal in this respect. This corresponds to a geometrical scaling factor of 1/6.

It was decided to keep the three scaling possibilities open for forced convection experiments in the E-SCAPE facility: Froude, time and maximum velocity. This leads to total flow rates in forced convection varying from 2 l/s (time) over 5 l/s (Froude) to 12 l/s (maximum velocity). Depending on the phenomenon investigated, a specific flow rate will be set. Taking into account the interest in the free surface behaviour and to perform non-isothermal experiments in a second stage, the Froude scaling will however be taken as reference because its correspondence with the Richardson scaling.

The CFD calculations will be continued including simulation of the free surfaces, for non-isothermal flow cases in both forced-convection and natural-convection flow conditions. Eventually, transient calculations will be performed and compared to the full-scale case.

5. Acknowledgement

This work is supported by the 7th Framework Programme European Commission Collaborative Project THINS No. FP7-249337.

6. References

- [1] -, "ESNII – The European Sustainable Nuclear Industrial Initiative, A contribution to the EU Low Carbon Energy Policy: Demonstration Programme for Fast Neutron Reactors - Concept paper", Sustainable Nuclear Energy Technology Platform, www.snetp.eu, 2010.
- [2] H. Aït Abderrahim et al., "MYRRHA, a Multipurpose hYbrid Research Reactor for High-end Applications", Nuclear Physics News, Vol. 20, No. 1, 2010, pp. 24 - 28.
- [3] D. Tenchine, "Some thermal-hydraulic challenges in sodium cooled fast reactors", Nuclear Engineering and Design, Vol. 240, 2010, pp.1195-1217.
- [4] F. Roelofs, "Cross-cutting CFD Support to Innovative Reactor Design", Proceedings of ICAPP 2009, Tokyo, Japan, 2009 May 10-14.
- [5] F. Roelofs, E.M.J. Komen, P. Kupschus, K. Van Tichelen, "CFD Analyses of the Lead-Bismuth Flow Field in the Lower Plenum of the MYRRHA Pool", Proceedings of NURETH 11, Avignon, France, 2005 October 2-6.
- [6] M. Tarantino et al., "Natural circulation experiment in a heavy liquid metal loop", Proceedings of HeLiMeRT, ISSN 1782-2335, 2009 April 20-22.
- [7] Y. Eguchi et al., "Quantitative prediction of natural circulation in LMFR with a similarity law and a water test", Nuclear Engineering and Design, Vol. 178, 1997, pp. 295-307.
- [8] M. Ishii et al., "The three-level scaling approach with application to the Purdue University Multi-Dimensional Integral Test Assembly (PUMA)", Nuclear Engineering and Design, Vol. 186, 1998, pp. 177-211.
- [9] P.F. Peterson et al., "Scaling for integral simulation of mixing in large, stratified volumes", Nuclear Engineering and Design, Vol. 186, 1998, pp. 213-224.
- [10] C. Fazio (ed.), "Handbook on Lead-bismuth Eutectic Alloy and Lead Properties, Materials, Compatibility, Thermal-hydraulics and Technologies", OECD/NEA report n° 6195, 2007.