ACTIVITIES OF UPV/EHU CONCERNING THE TS-1 TARGET SYSTEM EXPERIMENTS

Alberto Peña, Alberto Abánades, Gustavo Esteban, Fernando Legarda and **Javier Sancho**
Nuclear Engineering and Fluid Mechanics Department, Universidad del País Vasco, Spain

Abstract

The cooling of spallation target windows for ADSs has been studied through the thermal experiments related to the TS-1 target (Obninsk, Russia). In the department of Nuclear Engineering and Fluid Mechanics of the University of the Basque Country, calculations have been made with FLUENT and STAR-CD codes. The comparison between the available experimental data and some preliminary analytical results obtained with the codes allows first conclusions for the reliability of computer simulations and contributes to the validation and improvement of these commercial Computational Fluid Dynamics codes in the application to ADSs target design.

Introduction

The target system (TS-1), in the frame of ISTC Project 559, is under development in the State Scientific Center of Russian Federation Institute of Physics and Power Engineering (SSC RF IPPE) and Joint Design Bureau "Gidropress" (JDB "Gidropress") for accelerator LANSCE (Los Alamos National Laboratory). Thermal experiments have already been performed at the IPPE.

The Department of Nuclear Engineering and Fluid Mechanics (UPV/EHU) has run the simulation of these thermal experiments using two CFD codes: FLUENT and STAR-CD. A particular CAD/CAM software: GAMBIT, completely compatible with FLUENT, has been applied to configure the geometry (3-dimenssion) and the mesh of the problem. The GAMBIT geometry and mesh has then been exported to STAR-CD.

The basic simulation was run on the basis of a 3D geometry, and some simplifications were necessary, at this stage of our calculation, in order to obtain suitable results at a reasonable simulation cost. The results were made for a benchmark exercise proposed for a IAHR workshop. [2] All the theoretical assumptions and simplifications are reported in detail in the next sections, as well as the new calculations.

The new simulation may be considered acceptable, in spite of being in its preliminary state, since convergence criterion has been fulfilled within a reasonable period of time, taking into consideration the large amount of cells used, the turbulence model that has been employed, and the hardware available. The results are compared with the experimental ones and the old calculations, and the main differences between them have been identified and explained.

In the following specification, the subject and the objectives of these calculations are explained, and the layout of the experimental device and initial conditions as far as necessary for performing the calculation.

Experiment description and benchmark exercise objective

The experimental device represents a tube-to-tube structure (Figure 1). NaK coolant enters through the inlet connection ∅68, and then it flows in the annular channel of the model between tubes ∅185 and ∅136 toward to the membrane positioned at the end of the annular channel. Here, in the form of coming together streams the coolant runs to the centre of the membrane, then it turns striking against the membrane surface and passes through the distributing grid. The membrane is heated by a copper block, simulating the energy deposited by a protom beam interacting with the target.

In order to get an axisymmetric flow in the membrane, a sickle-shaped plate is installed in the upper part of the annular channel. More details in reference [1].

The objective of the benchmark exercise is the comparison of the coolant and membrane temperature at different positions, analysing the temperature behaviour of liquid metal coolant and membrane of the target system, and examining the reliability of the thermal-hydraulics codes and their physical models.

Figure 1. **TS-1 Device**

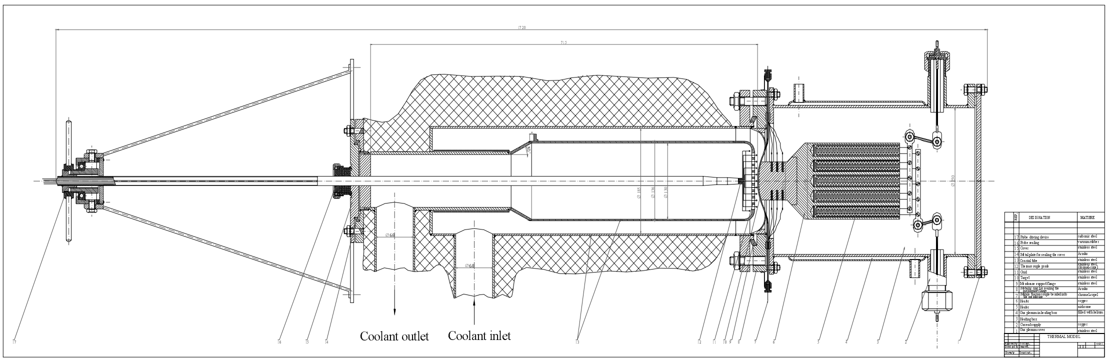
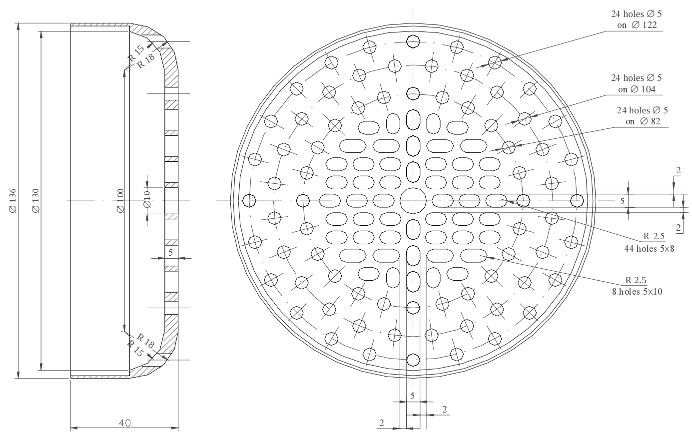


Figure 2. **Distribution grid**



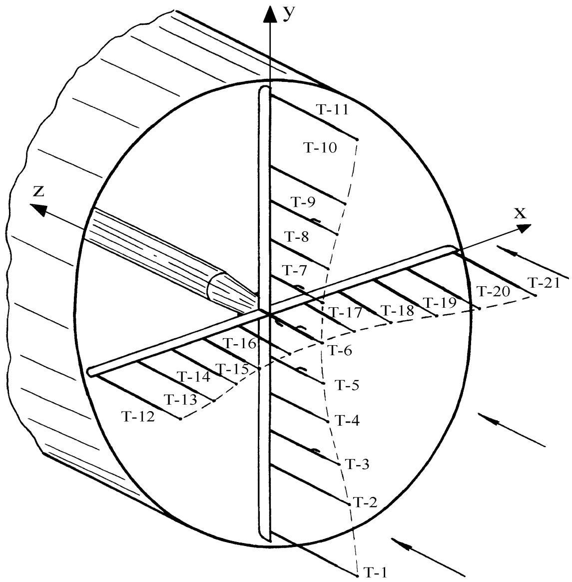
Modelling assumptions

The modelling assumptions in this simulation have followed the criterion of minimal resources consumption for a 3D approach in order to obtain reliable results in the minimum time schedule.

Geometry

The distribution grid is one of the most critical elements in the design of the target, as it affects strongly the behaviour of the cooling performance. It has been substituted by a perforated plate with only one central hole, and characterised as a porous medium with pressure loss properties equivalent to the remaining peripheral holes. The porous medium zone is divided in two volumes to take into account the different hole distribution in the grid (see Figure 2 for distribution grid geometry). This simplification allows gaining speed in the simulation in relation to what the real geometry had involved. Simultaneously, this assumption prevents distorting the fluid behaviour in the proximity of the distributing grid because it was designed in order to achieve the main fluid flow through the central round hole of the grid. In addition, the rounded contours have been neglected and perfect cylindrical shapes have been considered. Nevertheless, a complete geometry nodalisation for the grid is under development to include the flow discontinuity that a real grid produces.

Figure 3. **Thermocouples distribution**



Nodalisation

Using GAMBIT in a 3-D scheme (Fig. 4), the general meshing scheme followed throughout the whole volume has been mixed (tetrahedral, hexahedral, pyramid), preserving the skewness (<0.8) and smoothness (size ratio<20%) quality criteria. The total number of cells computed has been 231 525, while in the basis calculations 101 390 cells were used.

To this extent a first-order accuracy has been desired for all the equations evaluated. Consequently, the First-Order-Upwind scheme has been applied for the momentum, energy, k and ε transport equations. In the discretisation of the continuity equation the pressure-velocity coupling is achieved by means of the algorithm SIMPLE, the pressure interpolation scheme used here being the Body-Force-Weighted.

Boundary conditions

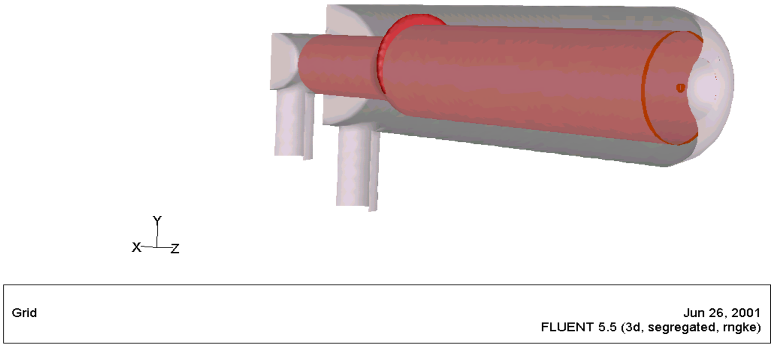
The boundary conditions are imposed by the characteristics of the inlet coolant flow and the heat generation device welded to the membrane. The NaK loop has been designed to have a symmetric velocity distribution on the annular channel and the cooling on the membrane, The coolant flow rate is 7 m^3/h, with a variable inlet temperature depending on the test case to simulate. The heat generation device implies a heat flux through the membrane of 8 600 kW/m^2, over an spherical area of 75 mm of diameter, and the results where the obtained in this report.

The internal walls have been considered, allowing a free heat exchange between both flows: the annular one and the inner one.

A velocity inlet boundary have been imposed, u_i= 0.535 m/s (flow 7m^3/h and 68mm-diametre) at a constant temperature of 35.3ºC. This fact is a simulation assumption, because the temperature varied during the performance of the thermal experiments from 33ºC to 37.4ºC.

All the physical properties of the fluid (NaK) and steel have been introduced in the model as reported in [1].

Figure 4. **Internal view of the GAMBIT geometry**

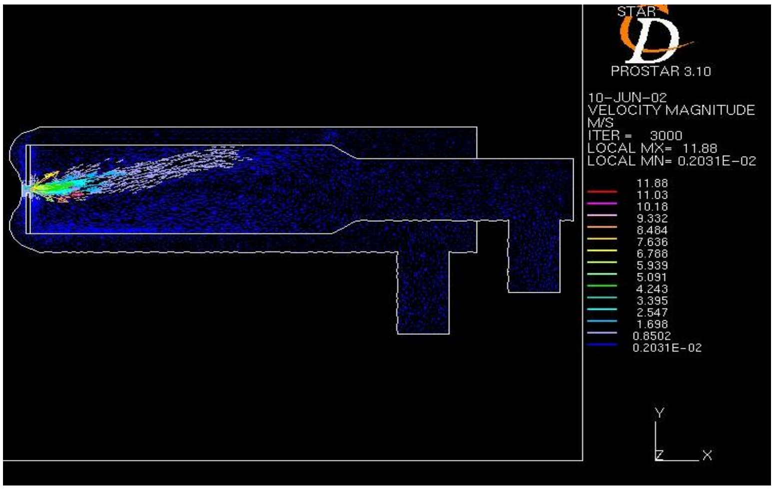


Analysis of the results

The first results of the analysis with STAR-CD are shown in Figure 5, where vectors of velocity are shown. The FLUENT temperature contours are depicted in Figure 6.

The temperature profile at each distance from the distribution grid is provided in the tables versus the data obtained with FLUENT and STAR-CD and the experimental data, in reference [2].

Figure 5. **Velocity in the display obtained with STAR-CD**



In the simulations, death zones, where liquid mobility is close to zero are foreseen (see Figure 7, for FLUENT calculation), together with some accelerated flux lines have been discovered close to the entrance and exit sections; the effort of modelling a certain length of those tubes has been worthwhile.

Attending the specific task, we have analysed the temperatures in the target and in the membrane In general, the tendencies of the variables that have been controlled during the simulation have matched the experimental values in terms of asymmetry in both the two principal axis of the geometry (and the physical distribution of the thermocouples can be seen in Figure 3). Although the values are, in general, higher than the experimental ones (see Figures 8, 9 and 10), the comparison between the coarser and the finer grid, shows that refining the meshing in the gap between the membrane and the distribution grid, could lead to very good agreement. However, the computation time and the required memory resources are important limitations.

Two turbulent models have been used: the RNG k-ε and the Reynolds Stress Model (RSM) (for details about the models, see references [3] and [4]). The calculations seems to fit slightly better the experimental data with the RSM, as shown in Figure 11, and the time consuming is not very different, in spite of the use of seven extra equations in the RSM instead of two extra equations used in the RNG k-ε model.

In search of improving the quality of results, the main items to be re-considered in the future have been identified. Redistribution and refinement of the meshing in the critical points, as well as the use of the detailed geometry instead of simplified structures or the use of variable boundary conditions are some of the questions to analyse in a further study.

Figure 6. **Contours of temperature in a longitudinal section**

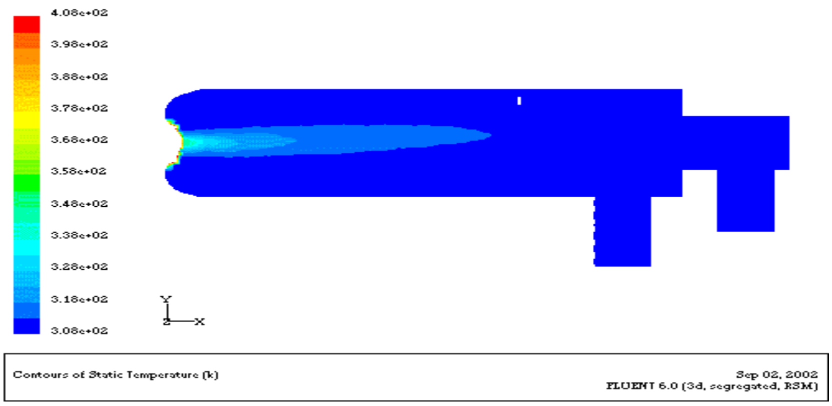


Figure 7. **Velocity vectors in the gap between the membrane and the distribution grid**

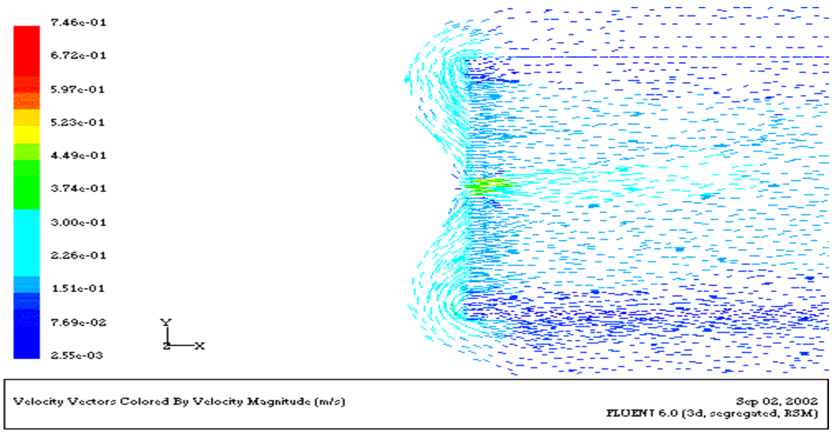


Figure 8. **Fluid temperature s=1**

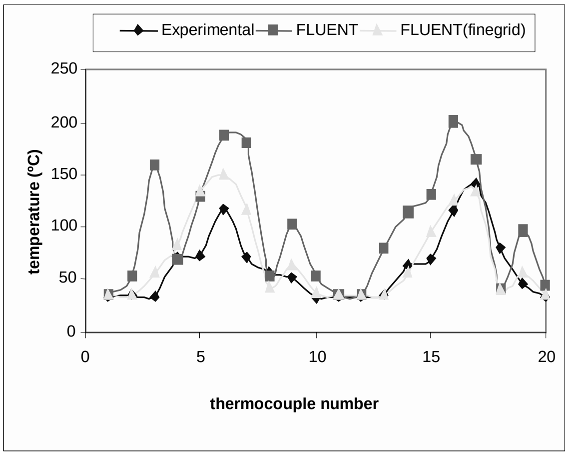


Figure 9. **Fluid temperature s=5**

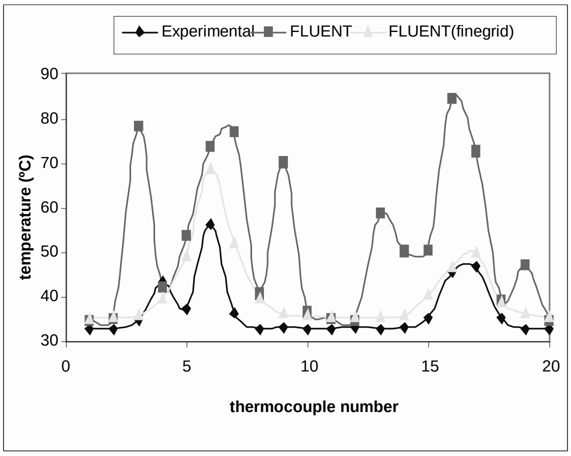


Figure 10. **Fluid temperature s=60**

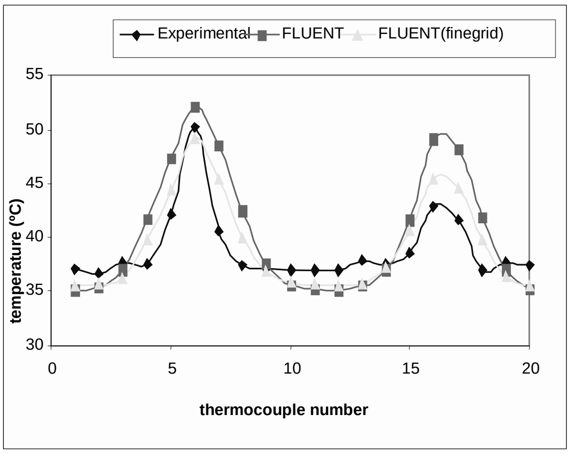
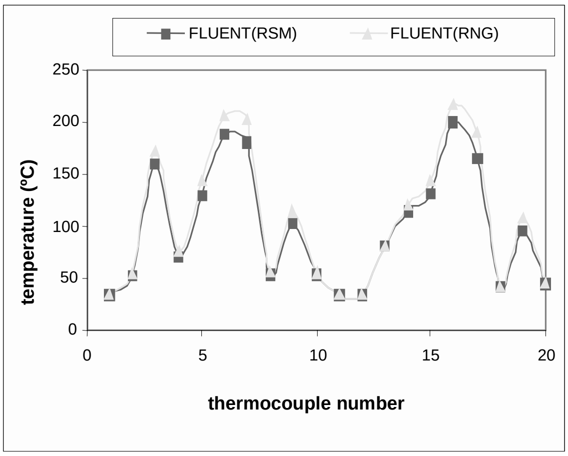


Figure 11. **Fluid temperature s=1**



In Figures 8, 9, 10 and 11 "s" is the distance of the thermocouple from the centre point of the membrane in mm. Figure 9 shows the temperature of the coolant just below the distribution grid, whereas in Figure 10, temperature inside the inner cylinder is depicted.

Conclusions

We have presented the status of the CFD analysis of the ISTC (TS-1) target that is being undertaken in the UPV/EHU. The preliminary results of our calculations have been shown, concluding that our approach seems to be correct for a preliminary CFD simulation of this target, but also suggests several improvements for a full simulation. There must be some more work to do with the nodalisation, above all, in the gap between the membrane and the distribution grid, and although exact temperatures are not reached, tendencies are good in these preliminary calculations. Both codes predict higher asymmetry that the experiment one, but cells are still too large for an adequate simulation. Computer hardware limits the total amount of cells in the problem description and forced to local grid optimisation.

Heat conduction in the solid walls, as well as the simplification of the plate (porous media model, instead of holes), are questions to consider in a future calculation, as well as the consideration of axis-symmetry.

REFERENCES

[1] Yu. Orlov, A. Sorokin, G. Bogoslovskaia, E. Ivanov, V. Malkov, M. Li (2001), *Specification of the Benchmark Problem: Thermal Experiment in the ADS Target Model*, Obninsk.

[2] A. Peña, A. Castro, G.A. Esteban, A. Abánades, F. Legarda (2001), *Benchmark Activities of UPV/EHU and LAESA Concerning the TS-1 Target System Experiments*. 10th International Meeting of the Iahr Working Group on Advanced Nuclear Reactors Thermal Hydraulics in July 17-19, Obninsk, Russia.

[3] "STAR-CD v 3,1 User Manual", Computational Dynamics Limited, 1999.

[4] "FLUENT 6 User´s Guide", FLUENT Incorporated, 1999.