EXPERIMENTAL MODELLING AND NUMERICAL ANALYSIS OF A MOLTEN SALT FAST REACTOR

Bogdán Yamaji, Dr. Attila Aszódi
Institute of Nuclear Techniques (NTI)
Budapest University of Technology and Economics (BME)
1111 Budapest, Műegyetem rkp. 3., Hungary
yamaji@reak.bme.hu
aszodi@reak.bme.hu

ABSTRACT

In this paper experimental and numerical investigation of the MSFR (Molten Salt Fast Reactor) concept will be presented. This homogeneous, single region liquid fuelled fast reactor concept uses fluoride-based molten salts with fissile uranium and thorium and other heavy nuclei content with the purpose of applying the thorium cycle and the burn-up of transuranic elements. Molten salt reactors with liquid fuel have a unique safety related property that needs clear understanding. In the core neutron flux and fission distribution is determined by the flow field through distribution and transport of fissile material and delayed neutron precursors. Since the MSFR concept has a single region homogeneous core without internal structures, it is a difficult task to ensure stable flow field, which is also strongly coupled to the volumetric heat generation. These considerations suggest that experimental and numerical modelling (including the option of coupled neutronics-thermal-hydraulics) would be needed to better understand the flow phenomena in such geometry.

A scaled and segmented experimental mock-up of MSFR was designed and built at BME NTI with the purpose of investigating the flow behavior inside the core region using particle image velocimetry. Not only the basic flow behavior inside the core region can be investigated but measurement data can also provide resource for the validation of computational fluid dynamics models, specific problems or phenomena (for example inlet geometry, optional internal structures, mixing) may be studied as well. Measurement results of steady state conditions will be presented with comparison of measurement data and results of numerical analyses.

Key Words: experimental modelling, MSFR, PIV, CFD

1. INTRODUCTION

The MSFR (Molten Salt Fast Reactor) concept is a single region reactor with homogeneous core utilizing fluoride-based molten salt as liquid fuel-coolant [1]. Purpose of the concept is to utilize thorium, and to be capable of burning actinides and breed fissile material. The concept has no internal structures in the core, and unlike the MSRE [2] or the reference MSR of the Generation IV International Forum [3] it has no graphite moderator with coolant channels. The molten salt flows upward in the cylindrical core from the inlet nozzles located at the bottom of the core toward the outlet nozzles at the top of the core. The sixteen identical loops consist of internal heat exchangers, pumps and the interconnecting pipes. The core is surrounded by axial and radial reflectors and a fertile blanket. Nominal thermal power of the MSFR is 3000MW, nominal inlet and outlet temperatures are 650 °C and 750 °C respectively. Based on the MSFR concept a scaled and segmented experimental mock-up was designed and
built in order to experimentally investigate the flow behavior in the core of such geometry. Results published earlier indicated that a limited reduction of flow rates – maintaining the core Reynolds-number over $1-2 \times 10^4$ in order to stay in the fully turbulent regime – and the scaling of the geometry will not affect significantly the flow behavior in the core region, and velocity fields could be reproduced with water as working fluid and with neglecting volumetric heat generation in the core. Those results also showed that a quarter segment of the original model could be representative for the bulk of the flow, however marginal regions near walls had to be left out of consideration [4].

Using particle image velocimetry (PIV) [5] measurement data would also help validating computational fluid dynamics (CFD) models and it also can be used for the investigation of design modifications or enhancements, such as inlet geometry, application of internal structures in the core. In the present work the measurement setup will be introduced, measurement results will be presented and comparison to preliminary CFD calculations will be discussed.

2. EXPERIMENTAL MODEL AND MEASUREMENT SETUP

A scaled and segmented experimental model of the MSFR core with water as working fluid was designed and built based on preliminary CFD calculations and detailed discussion of modelling goals and constraints [4]. Scaling ratio of the experimental model is 1:6 and only one quarter of the cylindrical core geometry is modelled. This ratio corresponds to model inlet diameter $d = 0.05 \text{ m}$. Inner diameter and height of the core cylinder is $D = 376 \text{ mm}$, other dimensions and the final design of the model tank is shown in Figure 1.

![Figure 1. Experimental model tank with flanges, dimensions of the model tank](image)

In the experimental setup design nominal flow rate for one inlet is $10 \text{ m}^3/\text{h}$, which corresponds to $Re_{\text{core}} = 1.5 \times 10^5$ Reynolds-number based on scaled cylindrical core geometry, with water at $20 \degree \text{C}$ temperature. In order to obtain fully developed flow at the inlets the total length of the horizontal inlet pipe is $l_0 = 1.5 \text{ m}$. Further parts of a loop connected to the model tank are a pump,
an ultrasonic flow meter, isolation valves and a regulator valves both on the inlet and the outlet side of the loop. At the discharge in the vertical section of the loop a straight section with a length of 20d before the flow meter, and with a length of 5d after the flow meter was incorporated. This is needed for the proper operation of the flow meter. At the top through a removable cover the model tank is connected to an overflow tank. This top cover also serves as the window for the light source of the PIV measurement system. Figure 2 shows the layout and components of one loop connected to the MSFR model tank.

![Figure 2. Components and layout of a loop connected to the model tank](image)

For PIV measurements the flow domain is seeded with polyamide particles (diameter $d = 50 \mu m$). The seeding particles are illuminated by a double cavity Nd:YAG pulsed laser (maximum pulse energy: 135 mJ, wavelength: 532 nm, pulse length: ~6 ns) equipped with a flexible light guide arm and light sheet optics. Scattered light is recorded on two subsequent image frames by a CCD camera. The velocity vectors are calculated from the displacement of the scattering particles based on the image pairs and the time delay of the recording. The image pairs are analyzed with adaptive correlation techniques to calculate velocity vectors [5]. Analysis and processing of recorded PIV image pairs were performed using the Dantec DynamicStudio package [6]. Figure 3 shows the experimental setup with the PIV measurement system.
3. MODEL FOR PRELIMINARY NUMERICAL ANALYSES

For numerical investigation a three-dimensional CFD model was built using ANSYS ICEM CFD. Hexahedral volumetric mesh was generated for the model; the calculations were carried out with ANSYS CFX 14.5 commercial code [7]. For the isothermal steady state calculations the k-ε turbulence model was applied. For each case the inlet boundary conditions were set according to the measured inlet flow rates. For outlet boundary condition zero relative pressure was set. The modelled domain included the full horizontal length of the inlet pipes, the quarter segment model tank and a short section of the outlet nozzles (see Figure 4).

Figure 4. CFX model and boundary conditions, green: inlet, orange: outlet
4. MEASUREMENT RESULTS AND COMPARISON

Four vertical measurement planes were defined for two-dimensional PIV measurements. Two are defined by the centerlines of the inlet and outlet nozzles (green – 11.25°, blue – 33.75°) and two are the symmetry planes between three neighboring nozzle pairs (red – 22.5°, orange – 45°), see Figure 6. Angle values assigned to each plane correspond to the angle between the vertical side plane of the tank and the measurement plane. The vertical side of the model tank acts as a window for the PIV recording camera.

Figure 5. Surface mesh representation of the CFX model

Figure 6. Measurement planes for 2D PIV with corresponding angles
The PIV camera is positioned perpendicular to the side wall of the mock-up tank. That ensures that the recording plane (plane of the CCD sensor) and the side wall of the measurement tank are parallel and distortion caused by the refraction can be avoided. A dot matrix reference target is used for calibration and image de-warping to remove the perspective distortion caused by the off-axis position of the measurement planes. With de-warping pixel positions of the recorded images are transformed into metric data as well. Calibration is carried out for every measurement plane and the measurement geometry is kept same for the corresponding cases. For the determination of the pixel position-metric data correlation a third order XYZ polynomial imaging model fit is used [6]. For the measurement of steady state turbulent cases 400 image pairs were recorded at a rate of 15 Hz at each position. The first 15 image pairs were left out of the analysis because of the run-up of the laser intensity at the beginning. Therefore from 385 image pairs 385 instantaneous vector maps were calculated. For evaluation and analyses the average of these instantaneous velocity fields were used. Pump and valve setup was set to \( \text{Re}_{\text{core, model}}=1.5 \times 10^5 \) (single inlet flow rate \( q=2.7 \text{ l/s} \)). Measurements were carried out along the full height of the model tank in all four measurement planes. Measurements for each position were repeated multiple times in order to verify repeatability. In the following selected results will be presented and compared with the results of initial simulations. Figure 7-10 show radial velocity distributions extracted from repetitions of measurements (M) together with simulation results (CFX) for each different measurement position (i.e. different planes and elevations). The figures show radial distribution of absolute velocity value (\( \text{L} - \) length of the velocity vector) and the radial (U) and axial (V) velocity components. In these graphs the distributions are given against relative radius, where \( r/R=1 \) is the cylindrical side of the geometry, and \( r/R=0 \) is the location of the 90° angle corner of the quarter sector cross section. Elevation along the vertical axis (z) is measured from the bottom of the flow domain.

**Figure 7.** Velocity component (U, V), absolute value (L), radial distribution, 11.25°, \( z = 0.037 \text{ m} \)
**Figure 8.** Velocity component (U, V), absolute value (L), radial distribution, 22.5°, z = 0.111 m

**Figure 9.** Velocity component (U, V), absolute value (L), radial distribution, 33.75°, z = 0.265 m
Results show that the experimental system can be operated at the planned conditions, the measurements can be reproduced, and the obtained values are in accordance with the expectations. Figure 7 (z = 0.037 m) shows that slightly below the elevation of the inlet nozzles (centerline elevation: z = 0.0416 m) velocity values are mainly defined by the radial component as water enters the domain of the core region. Distributions reach maximum at r/R=1 while close to r/R=0 these are close to 0 and reach negative values that corresponds to downward flow close to the corner of the geometry. At higher elevations (see Figure 8-9) these distributions switch, in the corner between the vertical walls the flow is clearly directed upward. At these elevations water flows down near the cylindrical outer wall, this clearly means a recirculation region above the inlet nozzles. Figure 10 shows radial distributions at the elevation of the outlet nozzle centerlines, in the inner symmetry plane between two nozzle pairs (45°). Figures 7-10 also include graphs from the preliminary CFD simulations. These show a good agreement between the measurements and the calculations. The simulations reproduced the actual values and the shapes of distributions well. Not only the values, but the locations of maximums and the locations where the axial component changes direction are well reproduced in the numerical simulations.

5. CONCLUSIONS

Based on the MSFR concept a scaled and segmented experimental model was designed and built. With successful particle image velocimetry measurements velocity fields were obtained along the height of the experimental model tank. Measurements carried out multiple times justified the applicability of the system and the capability to reproduce measurements. The experimental
model system can be operated at the desired conditions. Based on earlier result the appropriate measurement conditions and parameters (such as light source delay, data acquisition frequency, etc.) were defined. Preliminary CFD calculations were carried out, the simulations results were in very good accordance with the measurement data. In the future the refinement of the CFD model is needed, and a series of measurements will be carried out at different operation conditions. With the experimental model not only symmetric steady state conditions but transient cases could be investigated as well. Based on the measurements the validation of the CFD models will be possible. This validation is essential for detailed CFD analyses with the aim to make suggestions for the refinement of the MSFR concept geometry.

ACKNOWLEDGMENTS

The research leading to these results has received funding from the European Community’s Seventh Framework Programme under grant agreement No. 249696 EVOL. This work is connected to the scientific program of the “Development of quality-oriented and harmonized R+D+I strategy and functional model at BME” project. This project is supported by the New Széchenyi Plan (Project ID: TÁMOP-4.2.1/B-09/1/KMR-2010-0002). The project received complementary financing from the Hungarian Development Agency in the frame of New Széchenyi Plan EU_BONUS_12 programme (contract No: EU_BONUS_12-1-2012-0003).

REFERENCES