Thermal-hydraulics of a homogeneous molten salt fast reactor concept – experimental and numerical analyses

PhD thesis

Bogdán Yamaji

Budapest
2016
CONTENTS

INTRODUCTION .......................................................................................................... 4

1 MOLTEN SALT REACTORS AND CONCEPTS, PROPERTIES OF MOLTEN SALTS AND STRUCTURAL MATERIALS ................................................................. 7

2 THE INVESTIGATED MOLTEN SALT REACTOR CONCEPT .................................. 18

3 APPLIED EXPERIMENTAL AND NUMERICAL METHODS ................................ 20

4 DESIGN AND CONSTRUCTION OF THE EXPERIMENTAL MODEL OF A MOLTEN SALT REACTOR CONCEPT ................................................................. 25

5 MEASUREMENT RESULTS .................................................................................... 41

6 UNCERTAINTY ANALYSIS .................................................................................... 60
6.3.2 Error caused by the image recording system ........................................... 65
6.3.3 Image processing, calculation of displacement ...................................... 66
6.4 Sources of error and sensitivity factors of ΔT time delay ........................... 66
   6.4.1 Error sources of the delay generator (timer) timing ................................. 66
   6.4.2 Error sources of the laser pulse timing .................................................. 66
6.5 Sources of error and sensitivity factors of δU velocity difference ............... 66
   6.5.1 Flow following ability of the particles (trajectory) ................................. 66
   6.5.2 Three-dimensional effects ......................................................................... 67
6.6 Sampling ........................................................................................................ 67
6.7 Quantification of uncertainty ......................................................................... 69

7 ENHANCEMENT OF THE CORE FLOW FIELD WITH MODIFICATION OF
   THE GEOMETRY ........................................................................................................ 73
   7.1 Design of a core flow distributor plate .......................................................... 73
   7.2 Measurement and simulation results of the modified geometry ................. 80
      7.2.1 CFX modelling of the experiment with modified geometry .................. 80
      7.2.2 Comparison of measurement data and CFX simulation results .......... 85
      7.2.3 Considerations and consequences on the MSFR concept .................. 106

8 SUMMARY ............................................................................................................. 107

9 PUBLICATIONS OF THE AUTHOR ................................................................. 109

10 ACKNOWLEDGEMENTS .................................................................................... 110

11 REFERENCES ...................................................................................................... 111

12 APPENDIX .......................................................................................................... 116
   12.1 Manufacturing and construction of the experimental model ...................... 116
   12.2 Measurements on the experimental model .................................................. 117
Electricity generation in nuclear power reactors results in large amount of highly radioactive actinides and fission products. If the spent nuclear fuel is considered not as waste but as a by-product, reprocessing and separation of fissile uranium and plutonium generates considerable amount of long-lived, strongly radioactive waste. Handling, treatment and disposal of spent nuclear fuel and the radioactive waste is a key issue of nuclear energy generation. Thermal nuclear reactors currently in operation utilize natural uranium with an approximate efficiency of 0.5-0.7%, thus the economically recoverable identified uranium resources are only available for about 120 years [1]. This calls for more efficient fuel consumption. One option is the utilization of excess military fissile stockpiles for electricity production [1], keeping non-proliferation as a top priority. Other possibilities are the application of fast breeder reactors and the conversion of thorium into fissile uranium. With the introduction of nuclear energy systems based on the latter two options uranium and thorium resources would provide fuel for nuclear reactors at least for several hundred or even thousand years. However these new nuclear systems should also include partitioning, reprocessing of spent fuel and the transmutation of radioactive waste.

Development of Generation IV nuclear reactors [2] aims to design new revolutionary reactors serving the aforementioned goals based on well known or experimental technologies currently available. One of the Generation IV reactor types under investigation is the molten salt reactor. Its main special attribute is that the fuel is liquid, which also functions as the heat carrier medium itself. Development of molten salt reactors started in the early years of the Cold War, first with the purpose of developing a small-sized nuclear reactor for aeroplane propulsion. Later the development shifted its focus to electricity production and the utilization of thorium, along with the breeding of fissile material. As a result the first and so far the only molten salt reactor was designed, built and operated during the Molten Salt Reactor Experiment (MSRE) programme at Oak Ridge National Laboratory in the 1960s [3].

Several molten salt reactors concepts were proposed in different national and international research projects. These include thermal reactors with graphite moderator and a channel type core similarly to the MSRE reactor. Other concepts have homogeneous single region cores, which are larger volumes that only include the molten salt fuel-coolant and very few structural elements (or control rods) if any. In case of these concepts the core is usually a cylindrical volume that is connected to coolant pumps, heat exchangers and other elements via pipes. A third type of the concepts is the multi-region molten salt reactor in which the core consists of two or more, generally radial regions [4]. Molten salt composition in the different regions may differ, and the regions can be homogeneous or could be channel type with graphite moderator.

It can be said about every concept that extensive research is needed for the investigation of their feasibility in terms of neutronics, thermal-hydraulics, materials research and chemistry. The liquid fuel-coolant raises fundamental issues to be solved concerning nuclear safety. For example reactivity loss due to the circulation of the fuel has to be considered, or that with liquid fuel instead of fuel melt down the freezing of the liquid could be a safety concern.

Subject of my dissertation is the experimental and computational thermal-hydraulics investigation of the homogeneous single region Molten Salt Fast Reactor (MSFR) concept proposed within the EU 7th Framework Programme international research project EVOL [5]. The MSFR is a fast reactor concept, it does not include graphite as moderator, and it does not have any internal structures in the core either, meaning that the core is a single continuous volume. Its liquid fuel-coolant – similarly to earlier molten salt reactors and concepts – is fluoride-based with thorium and fissile material dissolved in it. Advantages of such molten salts are high boiling point at low pressure, good solubility of thorium or actinides, fairly good
thermal properties and good characteristics from neutronics point of view. Size of the MSFR core is comparable to, but smaller than a typical light water reactor core. Main difference is that instead of having solid fuel cooled by water coolant the core is filled with a homogeneous continuum of the molten salt. Since the MSFR core does not contain either graphite moderator blocks or any other structural elements, from material compatibility and heat load point of view the material (special steel alloys) of the core vessel could be the most important. It also has to bear significantly different chemical (corrosion) and thermal conditions (higher temperatures) and neutron irradiation (as the MSFR is a fast reactor) than a typical light water reactor pressure vessel. Along with that the reactor pressure vessel of such a molten salt reactor does not need to be designed for high pressures as in case of a pressurized water reactor or even a boiling water reactor.

In my dissertation I investigated the feasibility of experimental modelling of the MSFR concept using water as substitute working fluid. I had to consider modelling issues such as scaling and segmenting the geometry and the modelling and measurement possibilities and constraints as well. Bearing in mind these considerations I designed and built a unique scaled and segmented experimental model based on the design and operational parameters of the MSFR. Applying Particle Image Velocimetry (PIV) measurement method I completed multiple measurement series on the experimental model. Results of these measurements made it possible to comprehend the fundamental thermal-hydraulics behaviour of the concept.

One of the goals of the EVOL project was to optimize the geometry in order to achieve a more uniform flow field in the core region of the MSFR. The proposed experimental model gave an excellent opportunity to experimentally investigate the flow characteristics of the concept, and carry out measurements to study a solution of optimization by modifying the geometry. With the design and addition of a perforated flow distributor plate I successfully examined the possibility of modifying and improving the core geometry. By this modification I managed to enhance the uniformity of the flow field in the core region, thereby presenting a solution to the selected optimization problem.

The experimental results not only allowed me to draw conclusions concerning the thermal-hydraulics behaviour of the MSFR core, but the experimental data were also applicable for the validation of numerical models of Computational Fluid Dynamics (CFD) analyses. Following the completion of a comprehensive uncertainty analysis of the complex of the measurement system and the experimental setup validation of the numerical model of the modified MSFR core model geometry became possible.

Throughout the whole work I continuously optimised and refined the PIV measurement system and the experimental setup itself in order to enhance measurement results, improve and optimize the measurement procedure.

In my dissertation first I present a short summary on the history of molten salt reactors and of some of the proposed molten salt reactor concepts. Following that I summarise the most important physical properties of applied and proposed molten salts together with the properties of structural materials. Knowledge of such properties is essential for experimental and numerical modelling of molten salt reactors. Then I describe the investigated molten salt reactor concept and its features in detail, and discuss the possibilities and constraints of the experimental modelling of the selected molten salt reactor concept by presenting the results of extensive numerical investigation using Computational Fluid Dynamics (CFD). Following this I describe the design based on the findings of the aforementioned analyses and construction of the actual experimental mock-up. It is followed by a detailed breakdown of selected results of an extensive series of measurements of steady state operation conditions, and the discussion of the thermal-hydraulic characteristics of the proposed molten salt reactor concept. Together with the measurement results I present the comprehensive uncertainty analysis of the complex of the measurement system and the experimental setup, which is
essential for proper evaluation of the measurements. Following that I recommend a
modification of the experimental geometry, that would allow me to investigate the feasibility
of enhancing the thermal-hydraulics characteristics of the proposed concept, and also present
and discuss the measurement results of the modified model geometry. Together with the
presentation of these results I discuss the possibility to improve the core geometry of the
molten salt reactor concept, and suggest further directions for investigating and optimizing the
geometry of the core.
1 MOLTEN SALT REACTORS AND CONCEPTS, PROPERTIES OF MOLTEN SALTS AND STRUCTURAL MATERIALS

1.1 MOLTEN SALT REACTORS AND REACTOR CONCEPTS

Research of liquid fuelled nuclear reactors utilizing molten salts started in the 1940s in the United States. The first experimental system was the Aircraft Reactor Experiment (ARE) developed and operated for nine days in 1954 at Oak Ridge National Laboratory (ORNL) [3] (see Figure 1). Between 1954-57 ARE was followed by the design of Aircraft Reactor Test (ART, also known as Fireball [6] – see Figure 2). The aircraft propulsion program was shut down in 1961 [7], but a research program was already started in 1960 to develop molten salt reactors for energy production and breeding of fissile material. In the framework of the Molten Salt Reactor Experiment (MSRE, 1960-69) program ORNL designed and built the first molten salt reactor (see Figure 3 and 4) that was operated for longer periods. Based on the graphite moderated reactor with a channel-structured core, in 1972 ORNL proposed a design of the so-called Molten Salt Breeder Reactor (MSBR), which was a 1000 MWe breeder power reactor [3, 6, 7] (see Figure 5). Very similar to the latter is the 1000 MWe Molten Salt Reactor (MSR) concept selected by the Generation IV International Forum to be developed as a generation IV type reactor [2] (also see Figure 5). It has to be noted that during the thirty years between MSBR and the proposal of the Generation IV MSR there was virtually no or very little progress in the development of molten salt reactors [8].

![Diagram of ARE core cross section and ARE moderator blocks](image1.jpg)

Figure 1: Left: ARE core cross section, right: ARE moderator blocks [7]
Figure 2: Left: ART cross section [9], right: plastic ART core model with stroboscopic particle photography measurement system [6]

Figure 3: Left: MSRE reactor vessel [10], right: graphite core of the MSRE [7]
Figure 4: Left: schematic of the MSRE [7] right: inside view of the MSRE test cell [11]

Figure 5: Top: The MSBR scheme [12], bottom: 1000 MWe MSR concept by GIF [2]
The aforementioned experimental reactors and concepts all considered graphite moderator in the core. In these designs the graphite slabs (stringers) had slots carved into each face that formed the vertical flow channels for the molten salt fuel-coolant. Other concepts, however, do not have internal structures or moderator in the core. The removal of the graphite moderator shifts the core neutron spectrum close to fast spectrum, thus such a reactor can be operated as actinide burner or thorium breeder. It would also eliminate the utilisation of the moderator structure with a relatively short lifespan [13].

One such concept is the Molten Salt Actinide Recycler and Transmuter (MOSART) [14] (see Figure 6). It has a cylindrical core, the molten salt enters through the lower cross section and leaves at the top through a conical-shaped outlet toward a distribution region. Below the core a reflector plate is located, the liquid medium passes it in an azimuthal mixing region. Below this and above the outlet two collector regions connect four coolant loops to the core.

Another single region homogeneous concept – which is the actual subject of my research – is the Molten Salt Fast Reactor (MSFR) [5]. It also has a cylindrical core (diameter: D = 2.255 m, height: H = 2.255 m) in which the molten salt flows upward from the inlet nozzles located at the bottom of the core to the outlet nozzles at the top of the cylinder. The core does not contain any structural elements, and it does not have graphite structures either, as it is a fast reactor concept. The core is one continuous single volume (unlike the MSRE reactor or the proposed MSBR and the Generation IV MSR, in which the molten salt flows through coolant channels in the graphite structure). Sixteen pair of inlet and outlet nozzles placed at the perimeter of the core dividing it virtually into sixteen identical segments. The liquid is transferred by pumps to internal heat exchangers, bubble separators and then back to the inlet nozzles. The primary loops are connected to an intermediate heat transfer loop by internal
heat exchangers. Below and above the core axial reflectors can be found, the core is surrounded by a fertile blanket (for detailed description and figures see Chapter 2).

In case of molten salt reactors with homogeneous core it is very important to achieve the proper flow characteristics and thermal-hydraulics behaviour in the core volume. The graphite structure in the core of MSRE not only acted as moderator but also strongly influenced the flow field of the core by directing it through the fuel channels formed by the shape of the adjacent blocks. This resulted in a strongly vertical flow in the core. However since the concepts with homogeneous core do not include graphite moderator blocks or any other flow-straightening structural elements, the thermal-hydraulics of such a core will be definitely characterised by three-dimensional flow field. Therefore in case of such cores it is essential to know the flow behaviour and to achieve the proper velocity distribution.

Generally these molten salt reactors were designed with three coolant circuits: the primary fuel-coolant circuit in which the medium passes through the reactor core. Through (often called intermediate) heat exchangers heat is transferred to a secondary molten salt cooled circuit. The secondary medium does not contain any fissile material. Purpose of such a secondary circuit is [15]:

- to provide additional barrier between the radioactivity in the fuel salt and the steam system in case of a heat exchanger failure, and to provide barrier to radioactive tritium migration to the steam system,
- to serve as thermal link between the fuel salt and the steam system in order to reduce the possibility of freezing the fuel salt,
- to isolate the high pressure steam system from the low pressure primary circuit, thus reducing the likelihood of subjecting the primary system to high pressure in case of steam generator tube failure,
- to guard against the entry of water into the primary system in order to avoid oxidation and precipitation of uranium and thorium,
- and to provide additional degree of freedom of system control by allowing the change of secondary circuit flow rate.

Commonly the third circuit is water cooled, and heat is transferred from the secondary circuit through steam generators. In case of the Molten Salt Reactor Experiment tertiary air cooling was provided by large fans [7, 10].

1.2 PROPERTIES OF MOLTEN SALTS AND STRUCTURAL MATERIALS

In the ORNL reactor development programs focus was on the application of fluoride-based molten salt compositions because these had high solubility for uranium, were stable chemical compounds, had low vapour pressure and reasonably good heat transfer properties [16]. Aspects of selecting the proper fuel-coolant material can be summarised as follows. The reactor is supposed to have large enough power output while having as small volume and mass as possible. The material is expected to have relatively high specific heat in order to minimize volume, size of pipes and pumps. One of the main aspects is to have a liquid that does not require high pressure in order to achieve high temperatures while not having phase change (namely boiling). This would allow high operating temperatures and high thermal efficiency without the need of large, heavy vessels designed for high pressure. In terms of applying in nuclear reactors the following properties are expected from fuel-coolants:

- fissile and fertile isotopes (uranium, thorium, actinides) should be reasonably soluble in the applied composition;
- neutron capture cross section of the molten salt carrier should be minimal at the applied energy spectrum;
- the fuel-coolant should be radiation-tolerant;
the generation of fission product and actinides should not degrade or significantly modify physical and chemical properties of the medium;
- stability at the given operating temperature and conditions.

In fluorides gaseous fission products do not dissolve, which makes it easier to separate and remove them from the system. That would result in higher achievable burn-up. Further properties that make fluoride-based molten salts advantageous for nuclear reactors are:
- in theory by using liquid fuel it is easier to refuel and reprocess;
- contrary to solid fuel elements fuel melt-down is not considered to be a risk in molten salt reactors;
- heat conduction of the liquid fuel is not a limiting factor;
- fuel rod and fuel assembly fabrication is not needed, however this is balanced out with the need of advanced chemical processing.

Besides the above mentioned advantageous properties molten salts raise several questions concerning material difficulties. Such a corrosive medium needs vessels made of special steels. Besides corrosion compatibility with other structural materials – for example graphite in case of channel type reactor concepts – has to be considered as well.

During the ORNL programs a comprehensive investigation of different molten salt compositions focused on the measurement and determination of as much physical properties as it was possible. In the 1980-90s research projects were initiated for example in Russia [17] that focused on the determination of molten salt physical properties, especially of fluorides, and later experimental loops were built and operated in the Czech Republic [18] as well.

It has to be noted that other molten salt compositions, for example chlorides and nitrites have also been considered or applied for other purposes such as heat carrier and thermal storage media in solar power plants [19] or in metallurgy [20]. In the future the experience from these applications may also be utilized in nuclear reactors, too.

1.2.1 Physical properties of molten salts

Highest priority issues concerning liquid molten salt fuel-coolants are the following: chemistry of the composition, solubility of actinides and lanthanides, compatibility of the irradiated molten salt with structural materials of the reactor vessel and the primary loop and – in case of it being present – with graphite [2].

For thermal-hydraulics investigations it is essential to know the thermo-physical properties of the selected molten salt composition such as density, specific heat, viscosity, thermal conductivity, and how these properties depend on other parameters like temperature.

The investigated or proposed molten salt compositions can be divided into several groups. The two main groups would be fuel-coolant salt and coolant salt, subgroups of these can be defined by the number of components: most of the compositions are two-, three- or four-component-materials, but there are some five-component-salts as well.

In order to carry out either experimental or numerical modelling of molten salt reactors, it is essential to know the physical properties of these materials, and the behaviour of the physical properties (e.g. temperature dependence). In the following sections the most important physical properties and their characteristics are summarised. In case of every property I present a few examples selected from the very large number of fluoride-based molten salt compositions that can be found in the international literature. These examples are presented together with the physical properties of the molten salt composition proposed for the MSFR (see Chapter 2 for details).

As it will be shown, many of the proposed molten salt compositions include light elements lithium and beryllium in order to achieve high conversion rate [2]. In these cases lithium is always considered to be highly enriched Li-7 because Li-6 has high neutron capture cross section, and the reaction generates large amount of radioactive tritium [21].
This summary presents an overview of the typical values considered in thermal-hydraulics analyses of molten salt reactors, and can give an insight into the great many compositions considered. In the summary I focus only on molten salt compositions as fuel-coolants and properties relevant to thermal-hydraulics analyses.

1.2.1.1 Specific heat

In general specific heat of different fluoride-based molten salts depends on the average atomic mass, i.e. the ratio of the average molar mass and the average atomic number:

$$\varepsilon_p = A \cdot \left( \frac{M}{N} \right)^B$$

(1)

where A and B are constants, M is the molar mass, N is the atomic number. A and B can be determined by measurement for a given compound, and applying the formula specific heat of other compositions can be predicted [22].

According to experimental data the specific heat of a given molten salt changes only slightly with temperature, therefore in many cases an average value is given for certain temperature ranges. Typical values range between 1000-3000 J/kgK for different molten salts, as for selected examples it is shown in Table 1.

Table 1: Specific heat of molten salt compositions

<table>
<thead>
<tr>
<th>Composition (mol%)</th>
<th>Specific heat [J/kgK]</th>
<th>Value at T = 700°C [J/kgK]</th>
<th>Ref.</th>
</tr>
</thead>
<tbody>
<tr>
<td>15 LiF - 27 BeF₂ - 58 NaF</td>
<td>2090</td>
<td>2090</td>
<td>[14]</td>
</tr>
<tr>
<td>46.5 LiF - 11.5 NaF - 42 KF</td>
<td>1769</td>
<td>1769</td>
<td>[23]</td>
</tr>
<tr>
<td>several multi-component fluorides</td>
<td>1046-2386*</td>
<td></td>
<td>[24]</td>
</tr>
<tr>
<td>77.5 LiF - 22.5 (Act-Th)F₄</td>
<td>(-1.111 + 0.00278×T[K]) × 1E+03</td>
<td>1594</td>
<td>[5]</td>
</tr>
</tbody>
</table>

*range

1.2.1.2 Density

Density of molten salts increases with higher molecular mass. Density decrease with increasing temperature is often given in linear form as it follows:

$$\rho = A - B \times T$$

(2)

where $\rho$ is density, A and B are constants for a given composition and T is temperature. Typical values range between 2000-5000 kg/m³ for different molten salt compositions at typical operation temperatures as shown for selected examples in Table 2. For four molten salt compositions Figure 7 shows the temperature dependency of density at typical operating temperatures.

Table 2: Density of selected molten salt compositions

<table>
<thead>
<tr>
<th>Composition (mol%)</th>
<th>Density [kg/m³] (=) temperature function</th>
<th>Value at T = 700°C [kg/m³]</th>
<th>Ref.</th>
</tr>
</thead>
<tbody>
<tr>
<td>15 LiF - 27 BeF₂ - 58 NaF</td>
<td>2163-(4.06×10⁻¹(T[C]-601.4)</td>
<td>2123</td>
<td>[14]</td>
</tr>
<tr>
<td>60 LiF - 40 NaF</td>
<td>2420 – 6.8×10⁻¹T [°C]</td>
<td>1944</td>
<td>[24]</td>
</tr>
<tr>
<td>18 LiF - 58 NaF - 24 BeF₂</td>
<td>2532 - 5.18×10⁻¹T [K]</td>
<td>2028</td>
<td>[25]</td>
</tr>
<tr>
<td>77.5 LiF - 22.5 (Act-Th)F₄</td>
<td>4094-8.82E-01×(T[K]-1008)</td>
<td>4125</td>
<td>[5]</td>
</tr>
</tbody>
</table>
1.2.1.3 Viscosity

According to experimental data the viscosity of different molten salt compositions increases with higher molecular mass. General temperature dependence of dynamic viscosity can be expressed by the following formula [24]:

\[
\mu = A \times \exp\left(\frac{B}{T}\right) (3)
\]

where A and B are constants for a given composition, T is temperature [K]. Typical values of dynamic viscosity of molten salts range between 1-10 g/ms at normal operating temperatures as shown for selected examples in the table below. Figure 8 compares the dynamic viscosity-temperature function at typical operating temperatures for four molten salt compositions.

Table 3: Dynamic viscosity of selected molten salt compositions

<table>
<thead>
<tr>
<th>Composition (mol%)</th>
<th>Dynamic viscosity (µ) [g/ms] – temperature [K] function</th>
<th>Value at T = 700°C [g/ms]</th>
<th>Ref.</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 UF₄ - 50 NaF - 48 ZrF₄</td>
<td>0.098×exp(3895/T[K])</td>
<td>5.36</td>
<td>[24]</td>
</tr>
<tr>
<td>14.33 LiF - 59 NaF - 26.67 BeF₂</td>
<td>\log_{10}\mu = -1.0018+(1617.4)/T[K]</td>
<td>4.57</td>
<td>[25]</td>
</tr>
<tr>
<td>29.21 LiF - 11.7 NaF - 59.09 KF (wt%)</td>
<td>1.633×exp(-2762.9/T[K]) + 3.1095×10⁻²T⁻²[K])</td>
<td>2.54</td>
<td>[26]</td>
</tr>
<tr>
<td>77.5 LiF - 22.5 (Act-Th)F₄</td>
<td>density×5.54E-05×exp(3689/T[K])</td>
<td>10</td>
<td>[5]</td>
</tr>
</tbody>
</table>
Dynamic viscosity of molten salts

![Graph showing dynamic viscosity vs temperature for different molten salts.]

Figure 8: Dynamic viscosity temperature dependence of selected molten salts [5, 14, 24, 25]

1.2.1.4 Thermal conductivity

Thermal conductivity of different molten salts change similarly to viscosity, with increasing density (or molecular mass) the heat conduction coefficient increases. The following empirical formula is proposed for the determination of the heat conduction coefficient of different molten salt compositions [27]:

$$\lambda = -0.34 + 5 \cdot 10^{-4} T + 32 / M$$

(4)

where $\lambda$ is thermal conductivity [W/mK], $T$ is temperature [K] and $M$ is molar mass [g/mol]. This formula shows that temperature dependence of thermal conductivity is almost negligible, and in most cases constant values are given. According to data published in literature typical values for different compositions range between 1-5 W/mK for different molten salts at typical operation temperatures as shown for selected examples in the table below.

Table 4: Thermal conductivity coefficient of molten salt compositions

<table>
<thead>
<tr>
<th>Composition (mol%)</th>
<th>Thermal conductivity ($\lambda$) [W/mK]</th>
<th>Value at T = 700°C [W/mK]</th>
<th>Ref.</th>
</tr>
</thead>
<tbody>
<tr>
<td>15 LiF - 27 BeF$_2$ - 58 NaF</td>
<td>0.838 + 0.0009(T[°C] - 601.3)</td>
<td>0.927</td>
<td>[14]</td>
</tr>
<tr>
<td>4 UF$_4$ - 50 NaF - 46 ZrF$_4$</td>
<td>2.25</td>
<td>2.25</td>
<td>[24]</td>
</tr>
<tr>
<td>66 LiF - 34BeF$_2$</td>
<td>1</td>
<td>1</td>
<td>[28]</td>
</tr>
<tr>
<td>77.5 LiF - 22.5 (Act-Th)F$_4$</td>
<td>0.928 + 8.397E-05×T[K]</td>
<td>1.0097</td>
<td>[5]</td>
</tr>
</tbody>
</table>

1.2.1.5 Melting point and boiling point

From feasibility point of view melting point and boiling point of the liquid fuel-coolants are very important. As the examples show in Table 5 melting point can be as high as 565 °C, which implies that unintentional cooling or freezing of the fuel-coolant should be avoided. Molten salts have the advantage of very high boiling point at atmospheric operating pressure,
which ensures that such a system would avoid bubble formation or heat transfer crisis due to boiling. Consequently the achievable high operating temperatures can lead to high thermal efficiency of a molten salt reactor.

Table 5: Melting point and boiling point of selected molten salt compositions (at 1 atm) [29]

<table>
<thead>
<tr>
<th>Molten salt (mol%)</th>
<th>Melting point [°C]</th>
<th>Boiling point [°C]</th>
</tr>
</thead>
<tbody>
<tr>
<td>59.5 NaF - 40.5 ZrF&lt;sub&gt;4&lt;/sub&gt;</td>
<td>500</td>
<td>1290</td>
</tr>
<tr>
<td>46.5 LiF - 11.5 NaF - 42 KF</td>
<td>454</td>
<td>1610</td>
</tr>
<tr>
<td>31 LiF - 31 NaF - 38 BeF&lt;sub&gt;2&lt;/sub&gt;</td>
<td>315</td>
<td>1400</td>
</tr>
</tbody>
</table>

1.2.2 Molten salts compared to other coolants

In general molten salt can be considered to be better coolant media than liquid metals. Molten salts often have higher specific heat capacity than liquid lead or sodium, which allows the reduction of total volume and the reduction of the size of pumps and pipings. Generally heat conduction of molten salts is worse than that of liquid metals, but it also means lower thermal shock to structures in case of sudden temperature changes of the coolant. Molten salts do not react with water or air, unlike sodium, which reacts with both in a highly exothermic way. Compared to water based systems (aqueous liquids) the main advantage is that with molten salts very high temperatures (for example 500-800 °C) are achievable at atmospheric pressure without the risk of boiling. Density of molten salts is generally higher than the density of water, and these liquids have higher thermal conductivity, too. Table 6 compares the advantages and disadvantages of different coolant types, while in Table 7 physical properties (at operating temperature) of sodium, water and a selected molten salt are compared.

Table 6: Comparison of liquid fuelled systems (+/– : advantageous/disadvantageous) [31]

<table>
<thead>
<tr>
<th>Medium</th>
<th>Compactness</th>
<th>Radiation resistance</th>
<th>Chemical stability</th>
<th>Vapour pressure</th>
<th>Actinide solubility</th>
<th>Corrosion</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aqueous</td>
<td>–</td>
<td>+/–</td>
<td>+/–</td>
<td>–</td>
<td>+/–</td>
<td>+</td>
</tr>
<tr>
<td>Chloride</td>
<td>+</td>
<td>+</td>
<td>+/–</td>
<td>+/–</td>
<td>+</td>
<td>–</td>
</tr>
<tr>
<td>Fluoride</td>
<td>+</td>
<td>+</td>
<td>+</td>
<td>+</td>
<td>+</td>
<td>+</td>
</tr>
<tr>
<td>Liquid metals</td>
<td>+</td>
<td>+</td>
<td>+/–</td>
<td>+/-</td>
<td>+</td>
<td>–</td>
</tr>
</tbody>
</table>

Table 7: Comparison of physical properties of sodium, FLIBE and water at typical operating temperatures

<table>
<thead>
<tr>
<th>Physical property</th>
<th>Medium</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Melting point [°C]</td>
<td>Na</td>
<td>97</td>
</tr>
<tr>
<td>Operating temperature [°C]</td>
<td>66% LiF - 34% BeF&lt;sub&gt;2&lt;/sub&gt; (FLIBE) [32, 33]</td>
<td>500</td>
</tr>
<tr>
<td>Specific heat [J/kg°C]</td>
<td>458</td>
<td>700</td>
</tr>
<tr>
<td>Density [kg/m&lt;sup&gt;3&lt;/sup&gt;]</td>
<td>1300</td>
<td>2340</td>
</tr>
<tr>
<td>Thermal conductivity [W/mK]</td>
<td>720</td>
<td>300</td>
</tr>
<tr>
<td>Dynamic viscosity [g/ms]</td>
<td>2.21</td>
<td>0.558</td>
</tr>
<tr>
<td>Water</td>
<td>841</td>
<td>2050</td>
</tr>
<tr>
<td></td>
<td>5.6</td>
<td>0.087</td>
</tr>
</tbody>
</table>
1.3 STRUCTURAL MATERIALS FOR MOLTEN SALT REACTORS

Research programs of the 1950s and 1960s demonstrated the applicability of nickel-based structural materials (INOR-8, Hastelloy B and N, Inconel) and Nb-Ti alloys, but their suitability for a lifetime of a reactor was not proven [2]. During the 1980s extensive research at the Kurchatov Institute in the Soviet Union developed a similar nickel-based alloy, HN80MT with several subtypes. During the 1990s in the framework of a national transmutation research program Skoda developed the MONICR alloy with high nickel content. The MONICR was applied in an experimental molten salt test loop [21].

As reported in [2] INOR-8 is a highly durable, stable, corrosion resistant alloy, ductile and easy to weld. It is fully compatible with graphite, with sodium-free salts up to 815 °C and with sodium-based salts below 700 °C. The modified Hastelloy N developed for the application with fluoride-based molten salt at high temperatures (up to 800 °C) proved its corrosion resistance, however its long-term applicability required further research. It has to be taken into account that molten salts (typically fluoride-based compositions, without fissile or fertile material) used in the secondary circuit can be more corrosive than the primary fluids. This difference has to be considered in the selection of structural materials as well [21].

In case of cores without graphite attention needs to be paid to the fact that nickel-based alloys are sensitive to helium-induced irradiation embrittlement, which reduces creep ductility of the material. Investigations have shown that low percentage (below 2%) addition of titanium into the alloy solves this problem [21].

Besides nickel-based alloys stainless steels are also applicable for structural materials. These have better radiation resistance, but their applicability is limited to 650-700 °C. Since the nickel content of these steels is lower, helium-induced irradiation embrittlement is less problematic.

In case of molten salt reactors with graphite moderator radiation damage requires the replacement of graphite slabs after 4-10 years [2], so development of graphite with better properties is still necessary. Applying graphite with longer life-time would be a significant improvement in terms of economic competitiveness, especially in case of molten salt reactors, which are supposed to run continuously without refuelling outages [21]. Evidently in case of MSFR this is not a matter of concern since the concept does not include internal elements or solid moderator structure made of graphite.
2 THE INVESTIGATED MOLTEN SALT REACTOR CONCEPT

The Molten Salt Fast Reactor (MSFR) concept is a single region homogeneous reactor with fluoride-based molten salt as primary coolant and fuel proposed within the so-called EVOL international EU FP7 research project initiated in December 2010. The basic concept was introduced previously with different nominal values concerning thermal power, inlet and outlet temperatures [13, 34 and 35]. Purpose of the EVOL project was to investigate the feasibility of such a homogeneous molten salt fast reactor. In the EVOL project multiple aspects of the feasibility of the proposed concept were investigated, including fuel composition and actinide solubility, interaction between molten salt and structural materials, neutronics characteristics and burn-up of the core. My experimental and numerical investigations focus on the flow phenomena inside the core of the reactor; external parts, such as pumps, the fertile blanket or the heat exchangers are not modelled. The parameters and design properties of the MSFR used in my investigation are based on the EVOL MSFR benchmark proposal [5].

The geometry of the core is cylindrical; the fluoride-based fuel-coolant molten salt flows upward from the inlet nozzles located at the bottom of the core to the outlet nozzles at the top of the cylinder. Sixteen pair of inlet and outlet nozzles placed at the perimeter of the core dividing it virtually into sixteen identical segments. The liquid salt is transferred by pumps to internal heat exchangers, bubble separators and then back to the inlet nozzles. The primary loops are connected to an intermediate heat transfer loop by internal heat exchangers. Below and above the core axial reflectors can be found, while the core is surrounded by a fertile blanket that acts as radial reflector (see Figure 9). The radial fertile blanket is also filled with molten salt, has an independent cooling system and is surrounded by boron carbide neutron shielding. Top and bottom axial reflectors are proposed to be made of nickel-based alloys [5].

Nominal thermal power of the MSFR concept is 3000 MW, nominal inlet temperature is 650 °C and outlet temperature is 750 °C. Main thermal parameters of the MSFR concept are summarised in Table 8. The fuel-coolant molten salt composition is fluoride-based with
Table 8: MSFR concept design parameters [5]

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal power [MW]</td>
<td>3000</td>
</tr>
<tr>
<td>Electric power [MW]</td>
<td>1500</td>
</tr>
<tr>
<td>Initial fuel salt composition [mol%]</td>
<td>77.5% LiF - 22.5% (U/Th/Pu)F₄</td>
</tr>
<tr>
<td>Initial blanket salt composition [mol%]</td>
<td>77.5% LiF - 22.5% ThF₄</td>
</tr>
<tr>
<td>Inlet temperature [°C]</td>
<td>650</td>
</tr>
<tr>
<td>Outlet temperature [°C]</td>
<td>750</td>
</tr>
<tr>
<td>Core diameter and height [m]</td>
<td>2.255</td>
</tr>
</tbody>
</table>

Table 9: Physical properties of 77.5% LiF - 22.5% (Act-Th)F₄ molten salt [5]

<table>
<thead>
<tr>
<th>Property</th>
<th>Function of temperature</th>
<th>Value at 700 °C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density [g/cm³]</td>
<td>4.094 - 8.82E-04 × (T[K]-1008)</td>
<td>4.1249</td>
</tr>
<tr>
<td>Kinematic viscosity [m²/s]</td>
<td>5.54E-08 × exp(3689/T[K])</td>
<td>2.46E-06</td>
</tr>
<tr>
<td>Dynamic viscosity [kg/ms]</td>
<td>density × 5.54E-08 × exp(3689/T[K])</td>
<td>1.01E-02</td>
</tr>
<tr>
<td>Thermal conductivity [W/mK]</td>
<td>0.928 + 8.397E-05 × T[K]</td>
<td>1.0097</td>
</tr>
<tr>
<td>Specific heat [J/kgK]</td>
<td>(-1.111 + 0.00278 × T[K]) × 1.0E+03</td>
<td>1594</td>
</tr>
</tbody>
</table>

Based on the thermal power, desired inlet and outlet temperatures, considering the physical properties of the proposed molten salt general hydraulic properties of the MSFR core can be determined. The flow in the core is highly turbulent, the Reynolds number exceeds 1,000,000 (see Table 10). The flow inside the inlet nozzles is highly turbulent as well (Reynolds number is around 500,000).

Table 10: Nominal hydraulic properties of the MSFR concept

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Core thermal power, Q [W]</td>
<td>3.0E+09</td>
</tr>
<tr>
<td>Average temperature [°C]</td>
<td>700</td>
</tr>
<tr>
<td>Temperature difference (inlet-outlet), ΔT [°C]</td>
<td>100</td>
</tr>
<tr>
<td>Specific heat, c [J/kgK]</td>
<td>1594</td>
</tr>
<tr>
<td>Density at 700 °C, ρ [kg/m³]</td>
<td>4125</td>
</tr>
<tr>
<td>Core mass flow rate, ̇m = Q/(c×ΔT) [kg/s]</td>
<td>3.0E+09/(1594×100) = 18,820</td>
</tr>
<tr>
<td>Single inlet mass flow rate, ̇m/16 [kg/s]</td>
<td>18,820/16 = 1176.33</td>
</tr>
<tr>
<td>Core diameter, D [m]</td>
<td>2.255</td>
</tr>
<tr>
<td>Core cross section, A [m²]</td>
<td>3.994</td>
</tr>
<tr>
<td>Core mean velocity, ̇v = m/(A×ρ) [m/s]</td>
<td>1.14</td>
</tr>
<tr>
<td>Core Reynolds number, Re_core = ̇v×D/ν</td>
<td>1.05E+06</td>
</tr>
<tr>
<td>Inlet nozzle diameter, d [m]</td>
<td>0.3</td>
</tr>
<tr>
<td>Inlet nozzle average velocity, ̇v_in [m/s]</td>
<td>4.03</td>
</tr>
<tr>
<td>Inlet nozzle Reynolds number, Re_inlet = ̇v_in×d/ν</td>
<td>4.93E+05</td>
</tr>
</tbody>
</table>

Fundamental difference in the design of this concept compared to MSRE and the other main molten salt reactor concepts (except for the MOSART concept) is that it does not include any internal structures in the core region therefore the reactor core flow domain is one large homogeneous volume. The MSRE and the proposed MSBR, together with the Generation IV MSR concept all have a channel-type flow domain in the core formed by the
graphite moderator blocks in the core. Flow characteristics of such core geometry are expected to be fundamentally different from that of a large homogeneous volume.

Neutronics of the MSRE reactor was thoroughly investigated previously [36] using numerical methods, and with simplifications three-dimensional CFD analyses of the MSRE were also carried out before [37, 38]. I selected the proposed MSFR for my investigations because of it being a rather well defined concept, and because such a homogeneous molten salt reactor core has not yet been investigated before thoroughly using both numerical (CFD) and experimental methods (i.e. PIV) together, and it is essential to evaluate the feasibility of such a concept in every aspect.

3 APPLIED EXPERIMENTAL AND NUMERICAL METHODS

In nuclear engineering numerical methods are widely used in different analyses. One of the most important tools is Computational Fluid Dynamics (CFD) with which detailed three-dimensional investigation of different thermal-hydraulics problems in complex geometries is possible. In many cases the investigated problem allows only numerical analyses since the conditions, geometries, etc cannot be reproduced in the real equipment, or the goal of the examination is such a state, condition or behaviour of a system that cannot be reproduced in full-scale tests. However the applied numerical methods, tools also require validation, and for that purpose experimental models are built and different measurement methods are applied. For examples of scaled experimental models with the purpose of analysing mixing phenomena in nuclear reactors, and to help validating numerical codes and models, see Chapter 4.1.2. Other applications focus on the flow dynamics in rod bundles since it is an integral part of nuclear reactors, where flow and heat transfer phenomena have an essential role [39]. Others investigate larger domains, coolant mixing in different parts of the reactor pressure vessel [40, 41 and 42]. In certain cases CFD analyses help the design of experimental mock-ups of future nuclear systems [43].

For molten salt reactors and concepts main application of CFD was the optimisation of geometry concepts [44], or the numerical investigation of specific subsystems of proposed molten salt reactor concepts, such as heat exchangers [45], and basic heat transfer phenomena in simple geometries, such as pipes [46]. As part of the EVOL project in [44] Rouch et al proposed a modification of the core vessel geometry based on preliminary CFD calculations.

In this chapter fundamentals of the applied experimental measurement method are described along with the introduction of the actual measurement system. Then I will give a brief introduction of the numerical code used for CFD calculations.

3.1 INTRODUCTION OF PARTICLE IMAGE VELOCIMETRY (PIV) AND THE APPLIED MEASUREMENT SYSTEM

3.1.1 Particle Image Velocimetry (PIV) [47]

As it is described in detail in [47] development of the Particle Image Velocimetry (PIV) measurement technique, which allows for capturing velocity information of whole flow fields, has begun in the 1980s. By 1998 PIV technique emerged from laboratories to applications in fundamental and industrial research, in parallel to the transition from photographic to video recording techniques. The first applications of PIV in wind tunnels, performed in the mid-eighties, were characterized by such time scales as two to three days to set up a system and to obtain well focused photographic PIV recordings. Time required to process the film was up to
one day, to evaluate a single photographic PIV recording by means of optical evaluation methods took 24 or up to 48 hours. From the late nineties with electronic cameras and computers it became possible to focus on-line, to capture multiple recordings per second, and to evaluate a digital recording within seconds. Since then there was a fast development of computer hardware and software for the PIV measurement method. High power and frequency lasers, optics together with improved digital cameras and software lead to a drastic increase in performance. Thus the range of possible applications increased rapidly, and today PIV is used in several different areas ranging from biology to aerodynamics, in fundamental turbulence research, combustion or two phase flows, and also in micro devices and systems. Thanks to the variety of different applications of PIV and the vast number of different methods to illuminate, to record and to evaluate, many different modifications of the PIV measurement technique have been developed.

Particle Image Velocimetry is based on the concept of indirectly measuring fluid flow by the measured displacement and velocity of seeding particles added to the fluid flow. Based on the applied measurement configuration two and three-dimensional PIV can be distinguished. The latter requires the use of two recording devices (cameras) and an illumination sheet positioned perpendicular to the flow direction. Hereinafter this description will be limited to two-dimensional PIV as this was the method I used.

For seeding particles with density close to the density of the fluid are supposed to be applied, especially in case of liquids (i.e. water). Depending on the fluid seeding could be polystyrene, polyamide (for water), oil droplets (for air), or other metallic, or coated hollow glass sphere particles of small diameter. In my measurements I applied polyamide seeding particles (PSP) of \( d = 50 \mu m \) diameter designed for use with water [48].

In a PIV measurement using an appropriate light source – in my case a pulsed Nd:YAG dual laser – the flow domain is illuminated for two short subsequent periods, and the light scattered on the seeding particles is recorded by a (digital) camera. Comparing the location of the particles in the two images their displacement can be determined, and by knowing the time delay between the pulses instantaneous velocity values can be derived.

Theoretical arrangement of PIV measurements is shown in Figure 10-11. In order to be able to achieve two laser pulses with short time delay and independent of the pulse strength generally double oscillator lasers are applied for light sources. (Illuminations are always doubled, of the two laser oscillators the first always provides the first, the second oscillator provides the second pulse.) Using mirrors and polarisers beams of the two oscillators are combined into one within the housing of the laser. The output beam is guided and shaped by optical devices, such as a light guide arm and light sheet optics. Using the light sheet optics the circular beam is shaped into a conical, two-dimensional light sheet that allows the illumination of a two-dimensional region of the flow domain. Ideally the recording device (digital camera) is positioned in a way that the recording sensor and the illumination sheet is parallel (i.e. the optical axis of the camera is perpendicular to the light sheet). On the lens of the camera a band-pass filter is used, so that only the light scattered by the seeing particles enter the camera.

In PIV image or data processing the detected region (the recorded image) is divided into smaller, so-called interrogation areas. Size of the interrogation areas depends on particle density, flow velocity and the delay between the illumination pulses, and it is assumed that the velocity (direction and magnitude) is homogeneous within one interrogation area (the displacement of the particles is space-independent within the interrogation window). A velocity vector can be associated to each interrogation area, and it gives us a two-dimensional velocity field of the recorded region [47].
3.1.2 The applied PIV measurement system

For the measurements I applied the PIV system installed at the Institute of Nuclear Techniques, Budapest University of Technology and Economics (BME NTI). Components and features of the PIV system are described in this section.

For seeding \( d = 50 \, \mu m \) polyamide seeding particles (PSP) were used. Illuminating device is a Litron Nano L PIV double oscillator Nd:YAG laser (peak energy: 135 mJ, wavelength: 532 nm, pulse length: ~6 ns, maximum repetition rate: 15 Hz) [50]. Because of the actual measurement arrangement (for details see Figure 12, Chapter 4.3 and Chapter 5) it was not possible to position the laser directly as an illumination source, since the laser is mounted horizontally, and it was not possible to align it vertically above the experimental model. I opted to apply a flexible light guide arm attached to the laser, and at the other end of the arm the light sheet optics are mounted. The beam inside the light guide arm is directed by special optical mirrors. The light guide arm has multiple joints that made it possible to position the ending and the light sheet optics in the desired location. This way I could arrange the light sheet optics above the experimental model tank, and managed to set up vertical illumination planes (see Figure 12 and Figure 30). For image recording a FlowSense 2M two-megapixel
(1600x1200) digital camera is used [51]. The light source and the camera is synchronised by a Dantec Timer Box (80N77) [52]. Images recorded by the camera are buffered, saved, and the velocity fields are calculated by the Dantec DynamicStudio software package on a PC [53].

The light sheet optics head is fixed and positioned with a wall-mounted positioning frame that allows shift displacement in three directions and the light sheet head can be rotated along the vertical axis in the head mount.

The image recording camera is mounted on a precision positioning system frame built with linear motion glide systems that allow fine positioning along three axes in three dimensions. This system allows the precise re-positioning of the camera when moving to new measurement positions or reproducing earlier measurements. The measurement system with the experimental model is shown in Figure 12.

Figure 12: The experimental setup: scaled and segmented model, laser light source with beam guide and light sheet optics, digital (CCD) camera
3.2 The CFX three-dimensional code [54]

As numerical tool I used the ANSYS CFX three-dimensional Computational Fluid Dynamics (CFD) code. The code numerically solves the unsteady Navier-Stokes equations in their conservation form using the finite volume method. The three conservation equations can be written in the following form:

the continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla (\rho \mathbf{U}) = 0$$ (5)

the momentum equations:

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla (\rho \mathbf{U} \otimes \mathbf{U}) = -\nabla p + \nabla \cdot \mathbf{\tau} + \mathbf{S}_M$$ (6)

where \(\mathbf{\tau}\) is the stress tensor:

$$\mathbf{\tau} = \mu \left( \nabla \mathbf{U} + (\nabla \mathbf{U})^T - \frac{2}{3} \nabla \cdot \mathbf{U} \right)$$ (7)

and the energy equation:

$$\frac{\partial h_{\text{tot}}}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho U h_{\text{tot}}) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot (\mathbf{U} \cdot \mathbf{\tau}) + \mathbf{U} \cdot \mathbf{S}_M + \mathbf{S}_E$$ (8)

where \(h_{\text{tot}}\) is the total enthalpy. The term \(\nabla \cdot (\mathbf{U} \cdot \mathbf{\tau})\) represents the work due to viscous stresses. This models the internal heating by viscosity in the fluid, and is negligible in most cases. The term \(\mathbf{U} \cdot \mathbf{S}_M\) represents the work due to external momentum sources and is currently neglected. In these equations \(\rho\) is density, \(t\) is time, \(\mathbf{U}\) is velocity, \(\lambda\) is thermal conductivity and \(p\) is pressure.

After setting material properties (as direct input of data, or functions or from standard tables), boundary and initial conditions and convergence criteria in the Pre (pre-processing) module can be set. Steady state or transient calculations can be run using the Solver module. The CFX code utilises different turbulence models (for example \(k-\varepsilon\), \(k-\omega\), etc) for calculation of non-laminar flows. CFX Post is used for evaluation and post-processing of the results.

For definition of the investigated flow domain the geometry I utilised ANSYS ICEM CFD, which was also applied for the volumetric mesh generation that is fundamental for the finite volume method. ICEM CFD is applicable for the generation of multiblock structured or unstructured hexahedral and hybrid volumetric meshes, quadrilateral and triangular surface meshes. Pyramidal and/or prismatic elements can be generated for boundary layer mesh [55].

3.3 Other modelling tools

During the design phase of the experimental setup the SolidWorks three-dimensional solid modelling computer-aided design and engineering (CAD/CAE) tool was extensively used. For data analysis and processing OriginLab OriginPro 8 and Microsoft Excel 2003 were used.
4 DESIGN AND CONSTRUCTION OF THE EXPERIMENTAL MODEL OF A MOLTEN SALT REACTOR CONCEPT

In order to be able to experimentally and numerically investigate the MSFR concept described in Chapter 2 I had to evaluate the options and possibilities of experimental modelling. In this chapter I present the considerations and constraints of the design of such an experimental model. In the chapter I describe the different cases that were investigated numerically in order to help the design of the experimental mock-up. On one hand the desired model had to be of reasonable size, considering the limitations in laboratory capacity (area, necessary power, etc) and that the purpose of the model was to carry out PIV measurements. On the other hand geometrical scaling and segmenting of the model and the utilisation of water as substitute fluid could be applied in a manner that the flow characteristics in the model domain remain representative of the flow in the core of the modelled reactor concept. Although it would be required that besides appropriate geometric scaling the dimensionless numbers (in this case the Reynolds number) are the same for an experimental model but often in similar (research or industrial) cases this requirement cannot be fully satisfied. In this chapter I summarise the options and constraints of the experimental modelling along with examples of other experimental facilities. In the chapter I present the CFD calculations that helped to find the appropriate scaling ratio with the necessary inlet conditions. In the third part of the chapter the final design of the experimental setup is described in detail.

4.1 OPTIONS OF EXPERIMENTAL MODELLING OF MSFR

4.1.1 Fluid selection for experimental modelling

Objective of the experimental model was to investigate the thermal-hydraulic behaviour of the MSFR core under laboratory conditions using the PIV measurement system installed at BME NTI. As working fluid water applied between room temperature and a maximum of about 60 °C was considered during the design of the experimental setup. Experience with water and its application for PIV with the suitable seeding particles suggested its use for the experimental investigation of the MSFR.

Table 11 shows the necessary water flow rates at different temperatures considering the original MSFR geometry and nominal core Reynolds number. One can see that in order to keep the nominal Reynolds number very high flow rates would be needed even at higher temperatures though increased temperature reduces viscosity. This also indicates that having a certain flow rate higher water temperature means achieving higher Reynolds number. However the probable complexity of an isothermal experimental system operated at elevated temperatures would need much more effort in design and could mean extra difficulties in operation. In my investigation I discuss the possibility of reducing the size of the experimental model while maintaining the nominal core Reynolds number, and will also investigate the application of lower Reynolds numbers that are still in the fully turbulent regime, namely greater than $(1.0-2.0) \times 10^4$. \[56\]

Table 11: Flow rates at different temperatures, water, Re=1.05×10^6

<table>
<thead>
<tr>
<th>Temperature [°C]</th>
<th>density [kg/m³]</th>
<th>Dynamic viscosity [g/ms]</th>
<th>Kinematic viscosity [m²/s]</th>
<th>Average core velocity [m/s]</th>
<th>Mass flow rate [kg/s]</th>
<th>Volumetric flow rate [l/min]</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>998.16</td>
<td>1.003</td>
<td>1.0048E-06</td>
<td>0.47</td>
<td>1869</td>
<td>112,347</td>
</tr>
<tr>
<td>50</td>
<td>988.02</td>
<td>0.547</td>
<td>5.5363E-07</td>
<td>0.26</td>
<td>1019</td>
<td>61,899</td>
</tr>
<tr>
<td>80</td>
<td>973.18</td>
<td>0.355</td>
<td>3.6509E-07</td>
<td>0.17</td>
<td>662</td>
<td>40,819</td>
</tr>
</tbody>
</table>
4.1.2 Scaled reactor models with water as working fluid

Worldwide several different experimental setups were built and are operated to investigate the thermal-hydraulics behaviour of nuclear power reactors. The ROCOM test facility [57] is a scaled plexiglas model of the Konvoi type pressurized water reactor, and is used to simulate mixing processes in the primary loop and the reactor pressure vessel (see Figure 13). The Vattenfall test facility is a model of a three-loop Westinghouse PWR, made of plexiglas as well, and there is the Gidropress test facility made of metal [58]. All three are 1:5 scaled models of power reactors and use water at room temperature (around 20°C), except for the Vattenfall test facility, which uses water at 53.6 °C in order to achieve lower viscosity [59]. These experimental mock-ups are widely used for the investigation of mixing processes in reactor pressure vessels and other important thermal-hydraulics phenomena. The measurement data provided by these experimental facilities are also of great importance for validation of Computational Fluid Dynamics (CFD) models and codes. With these three models the achieved Reynolds numbers in the reactor pressure vessel model sections are around $10^5$, which are two orders of magnitude lower than the nominal values at the nozzles or in the downcomer in the given power reactors, where the Reynolds number is around $10^7$.

![Figure 13: Left: vessel of the ROCOM test facility [57], middle: Vattenfall 1:5 scale PWR model [59], right: schematic view of the Gidropress test facility [58]](image)

In the Molten Salt Reactor Experiment (MSRE) programme at Oak Ridge National Laboratory during the design phase in the 1960s two mock-ups of the MSRE reactor were built for the investigations of coolant flow in the inlet region, in the downcomer and in the lower plenum [60]. The first one was a 1:5 scale plastic model in which water was used as working fluid. This model was able to reproduce the Reynolds number in the above mentioned regions of the MSRE reactor. The second mock-up was 1:1 scale and made of metal with plastic windows that allowed looking into the model during experiments. In case of this second model the Reynolds number was not reproduced exactly but was “well into the turbulent range” [61]. Both mock-ups were operated at isothermal conditions with water temperature at 24-27 °C for the full-scale model, and at 35 °C for the 1:5 scale model. Figure 14 shows the two MSRE mock-ups. As it was already indicated in Chapter 1.1 during the design phase of ART plastic models were used with stroboscopic particle photography method in order to determine the flow behaviour of the ART core domain [6, 9]. These examples confirm that geometrical scaling of the experimental model is plausible, and reduction of the Reynolds number within the turbulent range can be acceptable if reproduction of the nominal value is not feasible.
4.2 Preliminary CFD analyses for the support of the experimental modelling of MSFR

4.2.1 Set of cases analysed

I carried out a set of preliminary CFD calculations in order to identify the possibilities of designing a linearly scaled experimental model of the MSFR with the purpose of carrying out PIV measurements. For these calculations I used the ANSYS CFX 13 and 14 three-dimensional CFD codes. Based on the design concept of MSFR I created the computational geometry and generated the volumetric mesh of the CFD model. With this geometry and the set of input data I defined the following groups of cases to be analysed:

- CFD modelling of the MSFR reactor concept defined with the molten salt fluid properties, calculation with and without volumetric heat generation in the core. For heat generation modelling uniform distribution and $\text{SIN}(z)$ axial distribution was considered using the following formula:

$$Q(z)=Q_0 \times \text{SIN}(z\pi/2.255m)$$ (9)

- CFD modelling of water-based scaled experimental models: water at 20 °C as working fluid, full scale and scaled complete (360°) geometry, quarter segment of the original geometry (full scale and scaled versions), at nominal and reduced inlet water flow rates corresponding to the nominal and reduced Reynolds numbers.

The applied axial power distribution described by (9) is an exact solution for a non-reflected core. I applied this together with the assumption of radially uniform distribution as an approximation and omitted the effect of the reflectors surrounding the core.

Inlet flow rates were set according to the nominal value (Re=1.05×$10^6$), and in subsequent calculations flow rates were defined corresponding to Reynolds numbers of 6.0×$10^5$, 1.5×$10^5$ and 5.0×$10^4$. I investigated not only the reduction of the inlet flow rate but the reduction of the extent of the model. Therefore a set of calculations were carried out with 1:2 and 1:4.51 and 1:6 linearly downcaled geometries with corresponding core diameters of D = 1.1275 m, 0.5 m and 0.375 m. The last case corresponds to d = 0.05 m inlet diameter, for this case results of calculations with Re = 1.0×$10^5$ are also included.
Another set of calculations were carried out in which I investigated the possibility of segmenting the geometry. Using only one quarter of the full geometry would further reduce the necessary inflow and pumping power. With these calculations the effects on the flow in the core region of heat generation, flow rate reduction, linear geometric scaling and segmenting were analysed. Geometry and computational domain are shown in Figure 15.

Figure 15: Dimensions and boundary conditions of the full size CFD model, black arrow: inlet, yellow arrow: outlet

For the steady state calculations inlet boundary conditions were set as mass flow rates (according to the desired Reynolds number), for turbulence modelling the k-ε turbulence model was applied. Tetrahedral volumetric mesh included 1.36 million elements, in case of the quarter segment models the element number was 0.36 million. For the scaled cases transformed (rescaled) versions of the original geometry and mesh were used. Calculations carried out were named according to the following alphanumeric symbols:

- character 1-2:
  - FU: full geometry
  - Q: quarter segment geometry
- character 3-5:
  - 100: 1:1 geometrical scale (100%),
  - 050: 1:2 geometrical scale (50%),
  - 022: 1:4.51 geometrical scale (22%)
- character 7-9: modelled material:
  - SL: salt
  - W: water
- in case of SL character 11-13: power generation modelling:
  - SIN: sinusoidal distribution,
  - UNI: uniform axial distribution,
  - ZER: without power generation („zero“)
- after „Re” last 4 characters indicate the core Reynolds number:
  - 1000 – Re=1.05E+06 (MSFR nominal value)
  - 0600 – Re=6.0E+05,
  - 0150 – Re=1.5E+05,
  - 0100 – Re=1.0E+5,
  - 0050 – Re=5.0E+4
- Names including DN50 correspond to 1:6 scaled final design.

This gives a total of forty different simulated cases, summarised in Table 12.
Table 12: Simulation matrix of the preliminary CFD analyses

<table>
<thead>
<tr>
<th>Heat generation</th>
<th>Full (360°) model</th>
<th>1:1 full scale</th>
<th>Re=1050000</th>
<th>Re=600000</th>
<th>Re=150000</th>
<th>Re=100000</th>
<th>Re=50000</th>
</tr>
</thead>
<tbody>
<tr>
<td>MSFR molten salt</td>
<td>FU100_SLT_SIN_Re1000</td>
<td>FU100_SLT_ZER_Re1000</td>
<td>FU100_SLT_UNI_Re1000</td>
<td>FU100_WAT_Re1000</td>
<td>FU050_WAT_Re0600</td>
<td>FU022_WAT_Re022</td>
<td>FU_DN50_WAT_Re1000</td>
</tr>
<tr>
<td>No heat generation</td>
<td>No heat generation</td>
<td>No heat generation</td>
<td>No heat generation</td>
<td>No heat generation</td>
<td>No heat generation</td>
<td>No heat generation</td>
<td>No heat generation</td>
</tr>
<tr>
<td>Water</td>
<td>FU100_WAT_Re1000</td>
<td>FU050_WAT_Re0600</td>
<td>FU022_WAT_Re022</td>
<td>FU_DN50_WAT_Re1000</td>
<td>FU_DN50_WAT_Re0600</td>
<td>FU_DN50_WAT_Re022</td>
<td>FU_DN50_WAT_Re022</td>
</tr>
<tr>
<td>1:2 scaled (D=1.1275 m)</td>
<td>FU100_WAT_Re1000</td>
<td>FU050_WAT_Re0600</td>
<td>FU022_WAT_Re022</td>
<td>FU_DN50_WAT_Re1000</td>
<td>FU_DN50_WAT_Re0600</td>
<td>FU_DN50_WAT_Re022</td>
<td>FU_DN50_WAT_Re022</td>
</tr>
<tr>
<td>1:4.51 scaled (D=0.5 m)</td>
<td>FU100_WAT_Re1000</td>
<td>FU050_WAT_Re0600</td>
<td>FU022_WAT_Re022</td>
<td>FU_DN50_WAT_Re1000</td>
<td>FU_DN50_WAT_Re0600</td>
<td>FU_DN50_WAT_Re022</td>
<td>FU_DN50_WAT_Re022</td>
</tr>
<tr>
<td>1:6.01 scaled (D=0.3755 m)</td>
<td>FU100_WAT_Re1000</td>
<td>FU050_WAT_Re0600</td>
<td>FU022_WAT_Re022</td>
<td>FU_DN50_WAT_Re1000</td>
<td>FU_DN50_WAT_Re0600</td>
<td>FU_DN50_WAT_Re022</td>
<td>FU_DN50_WAT_Re022</td>
</tr>
</tbody>
</table>

In the following sections I present the comparison of calculation results of the different groups of cases. For quantitative comparison radial distributions of the velocity and the axial velocity component along selected monitor lines are shown. The comparisons were done for every simulation case and at all elevations and angular positions (see Figure 16). In Chapter 4.2.2-4.2.4 a selection of these comparative analyses are presented in order to give an overview of the general conclusions concerning scaling and segmenting of the geometry, application of water as substitute medium and the omission of volumetric heat generation. The selected results presented here provide a comprehensive picture on the applicability of the geometry modifications and the modelling approximations.

4.2.2 MSFR calculation results (FU100_SLT series)

A series of calculations were carried out using the molten salt properties given in Table 9. In the cylindrical core (see Figure 16) volumetric heat generation was defined for two cases: sinusoidal axial profile along the vertical (z) axis (FU100_SLT_SIN) and uniform heat generation in the core domain (FU100_SLT_UNI). To see the effect of neglecting heat generation and the reduction of inlet flow rates the previously defined lower Reynolds numbers were set for the following calculations: FU100_SLT_ZER_Re1000, FU100_SLT_ZER_Re0600, FU100_SLT_ZER_Re0150 and FU100_SLT_ZER_Re0050.

Figure 17 shows the temperature and velocity distribution in the X-Z symmetry plane for calculation FU100_SLT_SIN_Re1000. A zone with higher temperature can be seen at the perimeter of the core, which is generated by the recirculation of the fuel salt near the wall. Qualitative and quantitative analyses (see Figure 18) of the flow field show that near the wall above the inlet nozzles a stagnating region and slow downward flow develops that leads to the formation of a local hot area near the wall. In the centre of the core the fluid flows upwards with higher local velocity resulting in lower molten salt temperature.
Figure 16: Top and side view with monitor lines (red): CS lines: parallel to nozzle axes, Y lines: parallel to inter-nozzle symmetry lines

Figure 17: Temperature and velocity distribution in the X-Z symmetry plane (FU100_SLT_SIN_Re1000)

Velocity and axial velocity component profiles were extracted along monitor lines located at H/4, H/2 and 3H/4 (where H is the MSFR core height 2.255 m) in “Y” and “CS” vertical symmetry planes (shown in Figure 16). These velocity profiles show that the definition of heat generation does not influence the flow field in the core (see Figure 18) and its shape is basically determined by turbulent forced convection. Therefore for further comparisons FU100_SLT_SIN_Re1000 and FU100_SLT_ZER_Re1000 were selected as reference results.
Velocity distributions shown in Figure 19 indicate that the reduction of the inlet flow rates, and therefore reduction of the simulated core Reynolds numbers from Re=1.05×10^6 to Re=6.0×10^5, 1.5×10^5 and 5.0×10^4 does not influence the shape of the velocity profiles and the general nature of flow inside the core. Values in Figure 19 are normalized to the maximum velocity value of the given distribution. It is also visible that the recirculation region – with downward flow, i.e. negative axial component values – is limited within the outer 10% of the diameter of the core. These results suggest that in nominal case the forced convection flow determines the flow distribution within the core and heat generation modelling might be neglected. It is also suggested by these results that the simultaneous reduction of dimensions and flow rate – keeping the values in the turbulent regime – will not alter the results significantly.

The objectives are not only to downscale the geometry and reduce the necessary inlet flow rate (and pumping power) but also to carry out experiments using water as working fluid instead of molten salt. Calculations for these cases are discussed in the following sections.
4.2.3 Results of modelling with water as substitute material, reduction of flow rates and scaling of the geometry (FU100_WAT, FU050_WAT, FU022_WAT AND FU_DN50 series)

In order to examine the feasibility of a resized water-based model of MSFR calculations were carried out simulating water as working fluid. Series of simulations were done for each size, 1:1 original (FU100), 1:2 scaled (FU050), 1:4.51 scaled (FU022) and the 1:6 scaled case referring to \( d = 0.05 \) m inlet nominal diameter (FU_DN50) with nominal and reduced Reynolds number as described in Chapter 4.2.1. Results presented in Figure 20 indicate that neglecting the volumetric heat generation in the core, using water as substitute fluid and the aforementioned reductions of inlet flow rate (FU100_WAT series) and geometrical scaling (FU050 and FU022 series) will result in flow distributions very similar to the distributions of the reference results. Values in Figure 20 are normalized by the maximum values of the given distribution. These distributions reconfirm that the general flow behaviour in the core region is not influenced by the geometric downscaling and by the reduction of Reynolds number. The flow field is not altered by these modifications, including the change of working fluid from molten salt to water.
4.2.4 Results of calculations with quarter segment model and water as working fluid (Q1100_WAT, Q1050_WAT, Q1022_WAT AND Q1_DN50 series)

In order to further reduce the necessary water flow rates for a possible experiment calculations were carried out simulating only one quarter segment (Q1) of the original geometry. Series of simulations were done for each size (1:1 original (Q1100), 1:2 scaled (Q1050), 1:4.51 scaled (Q1022) and 1:6 scaled case referring to d = 0.05 m inlet nominal diameter (Q1_DN50)) with nominal and reduced Reynolds number as described in Chapter 4.2.1. Results show that in the inner regions of the model, further away from the vertical bounding walls and away from the corner where these walls are connected the flow behaviour is not affected by the walls. Influence of the corner is limited only to a narrow
region near the corner. Wall effects can be neglected if we only investigate the flow between the four pairs of nozzles, the similarity is even more definite when only the region between the two inner nozzle pairs is considered (Figure 21 a-d).

![Streamlines at an inner pair of nozzles, axial velocity component (W) distributions at different elevations: b - H/4, c – H/2, d – 3H/4 (Q1100_WAT_Re1000)](image)

Figure 21: a, Streamlines at an inner pair of nozzles, axial velocity component (W) distributions at different elevations: b - H/4, c – H/2, d – 3H/4 (Q1100_WAT_Re1000)

Only close to the vertical walls and their connection at the centre appears significant difference between the flow distribution in the quarter segment models and the full model. Calculation results are shown in Figure 22-25, where values are normalized by the maximum value of the given distribution and for scaled geometries radial distance is normalized by the radius of the corresponding core model. The results show that differences in velocity values induced by the geometrical difference are very small and localized in a very narrow region near the vertical axis.
Figure 22: Normalized velocity and axial velocity component distribution, monitor line CS3 (3H/4) (Q1100_WAT series) in case of full and quarter model with different Reynolds numbers using water.

Figure 23: Normalized velocity and axial velocity component distribution, monitor line CS2 (H/2) (Q1050_WAT series) in case of full and downscaled quarter model with different Reynolds numbers using water.

Figure 24: Normalized velocity and axial velocity component distribution, monitor line CS2 (H/2) (Q1022_WAT series)
The general profile of the flow at different elevations is well reproduced with the segmented geometry even at reduced dimensions which was confirmed with calculation series Q1050_WAT, Q1022_WAT and Q1_DN50_WAT. The shapes of the velocity profiles are in good agreement, however it can be observed that the velocity decrease (axial component) toward the cylindrical wall of the core region increases with smaller models, but significant difference is limited to a region defined by about a maximum of 10% of the radius. It is also clear that the inner region closest to the corner (connections of vertical boundaries) does not represent well the flow behaviour of the original model, and will have to be neglected in comparisons of scaled and segmented water based experiments.
4.3 DESIGN OF THE EXPERIMENTAL MODEL

4.3.1 Design of the model tank, considerations and constraints

Based on the preliminary simulation analyses I designed and built a segmented and scaled experimental model of the MSFR core. For PIV measurement method the investigated flow domain has to be transparent therefore a plexiglas tank representing a scaled and segmented model of the MSFR geometry was designed and manufactured. The connecting loops that provide the necessary water flow rate were also designed and built together with flow measurement and data acquisition system. Along with these I had to consider the following aspects for the selection of the model tank size:

- smaller size for easier handling, lower necessary pumping power,
- capability to use pumps that provide the desired flow rate and Reynolds number,
- a size that allows the use of standard size pipes, tubes and system components (i.e. valves, flow meters, pumps, etc),
- large enough for handling, PIV calibration, and image recording.

I selected the scale of 1:6 for downscaling the geometry of the original MSFR reactor core described in Chapter 2. This scale corresponds to \( d = 0.05 \text{ m} \) as nominal inlet and outlet nozzle diameter of the experimental model. This inner nominal diameter is standard both for plexiglas tubes and copper pipes that I selected for the construction of the circulating loops.

Figure 26: Dimensions of the scaled and segmented plexiglas tank, final design with flanges
The plexiglas tank (see Figure 26) has a removable top with connection to an expansion tank. The removable top provides access into the tank (even in cases when it is filled with water) for PIV calibration and removal of any contamination that might disturb the optical measurement. Table 13 shows the necessary inlet flow rates corresponding to given Reynolds numbers. At this size (inlet diameter \( d = 0.05 \text{ m} \)) about 10 m\(^3\)/h water flow rate (at room temperature) in one loop would result in \( Re_{core}=1.5E+05 \) simulated core Reynolds number. In order to maintain this value under operational conditions, and also to be able to achieve higher flow rates and Reynolds numbers the pump was selected accordingly.

### Table 13: Water material properties, Reynolds numbers and corresponding flow rates. Green indicates the desired design operation range

<table>
<thead>
<tr>
<th>Water 20°C, core diameter ( D=0.375 \text{ m} )</th>
<th>1.05E+06</th>
<th>6.00E+05</th>
<th>1.50E+05</th>
<th>1.00E+05</th>
<th>5.00E+04</th>
</tr>
</thead>
<tbody>
<tr>
<td>core Reynolds-number</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>core diameter [m]</td>
<td>0.375</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>core cross section ([\text{m}^2])</td>
<td>0.110938</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>core average velocity ([\text{m/s}])</td>
<td>2.81</td>
<td>1.60</td>
<td>0.40</td>
<td>0.27</td>
<td>0.13</td>
</tr>
<tr>
<td>single inlet flow rate ([\text{m}^3/\text{h}])</td>
<td>70.031</td>
<td>40.042</td>
<td>10.011</td>
<td>6.674</td>
<td>3.337</td>
</tr>
<tr>
<td>single inlet flow rate ([\text{l/s}])</td>
<td>19.45</td>
<td>11.12</td>
<td>2.78</td>
<td>1.85</td>
<td>0.93</td>
</tr>
<tr>
<td>inlet nozzle diameter [m]</td>
<td>0.05</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>nozzle cross section ([\text{m}^2])</td>
<td>0.001963</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>inlet average velocity ([\text{m/s}])</td>
<td>9.91</td>
<td>5.66</td>
<td>1.42</td>
<td>0.94</td>
<td>0.47</td>
</tr>
<tr>
<td>inlet Reynolds-number</td>
<td>4.93E+05</td>
<td>2.82E+05</td>
<td>7.05E+04</td>
<td>4.70E+04</td>
<td>2.35E+04</td>
</tr>
</tbody>
</table>

In order to have a fully developed turbulent flow at the inlets the minimum inlet pipe length is defined by the following formula [62]:

\[
 l_0/d = 4.4 \times Re_{in}^{1/6} \tag{10}
\]

where \( l_0 \) is the minimum pipe length (or entrance length), \( d \) is the pipe inner diameter, and \( Re_{in} \) is the inlet Reynolds number. Taking the 10 m\(^3\)/h inlet flow rate for each inlet nozzle the values would be the following: \( Re_{in} = 7.05E+04 \) and \( l_0 = 0.05 \times 4.4 \times 6.42 = 0.05 \times 28.248 \). This means that a ratio of \( l_0/d = 30 \) would provide enough length to have fully developed flow in the inlet pipes, therefore \( l_0 = 1.5 \text{ m} \) was selected for construction. Fully developed turbulent flow is essential for CFD validation, which is an objective of the experimental model.

### 4.3.2 Loop design considerations

In each loop the following elements were considered in design:

- high power pump (single phase, 230V DC),
- isolation valves on the inlet and the outlet sides of each loop,
- regulator valves on the inlet and the outlet sides of each loop,
- flow rate measurement on the inlet side of each loop.

Hydraulic isolation of the plexiglas tank is necessary for the optical calibration of PIV and would also provide the possibility to investigate cases where one or more loops are closed. For optical calibration a filled but open tank is needed, and the isolation valves help to keep the water in the tank. For pump regulation one regulator valve is needed on the inlet side. On the outlet side a second regulating valve provides necessary fine tuning of each loop. With this valve setup hydraulically identical loops can be configured as well. In each loop a flow meter is installed after the pump to determine the loop flow rate. Schematic layout of one loop of the experimental setup is shown in Figure 27.
The expansion tank is located geodetically above the experimental tank, it is open to the atmosphere and has an overflow line connected to a drain tank. The pump discharge is vertical, and the connection of the flow meter into a vertical section of the loops provides better air removal from the section of the flow meter and the pump. After the flow meter the loop bends horizontally and through a regulator and an isolation valve it is connected to the MSFR model tank using flanges with DN50. It is also visible in Figure 27 that the inner nominal diameter of the pipes and the other elements on the pump side of the isolation valves is \( d = 0.032 \) m (DN32). The diameter change between DN32 and DN50 is located at the isolation valves.

### 4.3.3 System components and final design

For hydraulic isolation Oventrop Optibal ball valve was selected (DN50), as regulator valve Oventrop Hydrocontrol VTR (DN32) was built in. The regulating valve provides the capability to fine tune loop flow rate and pressure loss of the outlet leg. In each loop a Wilo Economy MHIL 903 pump provides a nominal delivery head of 26 m at 10 m\(^3\)/h flow rate. It guarantees that the desired high Reynolds number can be reproduced based on the predicted pressure loss of one loop including the model tank. For flow measurement MOM Hydrus ultrasonic flow meter (DN32) was selected. It provides low pressure loss (0.35 bar) and high nominal flow rate (10 m\(^3\)/h). For piping copper components were selected. The configuration of a single loop is shown in Figure 28 together with dimensions and elevations. Figure 29 shows the assembled experimental model during fill-up.
Figure 28: Components and layout of a loop (top), dimensions and elevations (bottom)

Figure 29: The assembled experimental mock-up
In conclusion purpose of the scaled and segmented model was to create a mock-up with which I could experimentally investigate the general flow behaviour in a geometry that is similar to the MSFR core and to produce measurement data for the optimization of the numerical (CFD) model of the experimental mock-up. In order to reduce necessary pumping power and volume, I applied linear scaling of the geometry. I also applied segmentation of the geometry: a quarter, 90-degree segment of the cylindrical core geometry was modelled. The preliminary calculations have shown that segmenting the investigated domain would not result in fundamentally different core flow field compared to the whole 360-degree geometry. Previous examples of scaled experimental models showed that a limited reduction of the core Reynolds number was applicable in the case of modelling the MSFR core region.

5 MEASUREMENT RESULTS

Purpose of the measurements was to investigate and understand the fundamental steady state behaviour of the flow field in the modelled core region. In one set of measurements identical inlet flow rates were set for each loop. With such setup it can be assumed that in the selected measurement planes described below the flow will be dominantly two-dimensional, namely the velocity components in the measurement planes are much larger then the out-of-plane components. For these sets of measurements I applied different inlet flow rates set by the regulator valves on the inlet side of each loop, setting the same flow rate for each loop in each case. This way I investigated the fundamental thermal-hydraulics characteristics of the modelled geometry, and by applying the inlet flow rate as experimental parameter I also managed to examine the effect of reduced inlet flow rate (or modelled inlet or core Reynolds number) on the modelling method.

In the experimental geometry detailed in Chapter 4 four vertical measurement planes were selected for two-dimensional PIV measurements. Two are defined by the centrelines of the inlet and outlet nozzles (11.25° and 33.75°) and two are the symmetry planes between the neighbouring nozzle pairs (22.5° and 45°), as it is shown in Figure 30. 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.

In the second part of the chapter I present another set of measurements in which only two of the four loops operate. This way I managed to examine operational states different from nominal, such as an asymptotic steady state case of loss of coolant pumps. However in this case only the following states can be considered to be symmetrical in the modelled geometry: Loop 2 and Loop 3 in operation, measurement plane is Plane 4 (45°) or Loop 1 and Loop 4 in operation, measurement plane is once again Plane 4 (45°). Measurement results of the first case (Loop 2 and 3 operate) are presented and discussed in the second half of chapter.
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 refraction can be avoided. A dot matrix reference target is used for calibration and so-called image de-warping, which is applied to remove the perspective distortion caused by the off-axis camera position. With de-warping pixel positions of the recorded images are converted into metric data, too (see Figure 31). Calibration is carried out before a measurement series for every measurement plane and each measurement position. The dot matrix calibration target is put in the experimental tank filled with water in the position of the selected measurement plane (see Figure 30). Then the camera is positioned accordingly with the use of the positioning system (described in Chapter 3.1.2) in a way that the largest possible area is recorded by the camera. In each measurement location the camera is moved to two or three additional elevations so that the total height of the experimental tank is covered. By recording the image of the calibration plane, the calibration routine of the software recognizes the dot matrix target and for the determination of the pixel position-metric data correlation a third order XYZ polynomial imaging model fit is used [63]. This process is repeated for the other measurement plane positions while the measurement arrangement is kept identical for the corresponding cases.
5.1 UNIFORM 4-LOOP OPERATION MEASUREMENTS AND RESULTS

For the measurement of steady state turbulent cases at each position 1500-1600 image pairs were recorded at a rate of 15 Hz. Number of recorded image pairs was determined based on preliminary inspections and the image buffer limitations of the recording system. At least 400 recorded image pairs were found sufficient enough for averaging, however when going over 1600 pairs buffer lag often resulted in the unintended termination of recording due to buffer limitations. In every measurement the first 15 image pairs were left out of the analysis because of the run-up of the laser intensity right after the start of the acquisition. Vector maps were calculated using the remaining 1485-1585 image pairs. For evaluation and analyses the average of these instantaneous velocity fields were used. Figure 32 shows a typical averaged velocity field in Plane 3 (33.75°) at the elevation of the inlet nozzles.
Measurements were carried out along the full height of the model tank in all four measurement planes. Measurements for each position were repeated with flow rates set for the four loops presented in Table 14. The different inlet mass flow rates correspond to different simulated core Reynolds numbers ranging between $Re_{\text{core}} = 1.17E+05 - 1.82E+05$. Based on the MSFR nominal parameters the original nominal core Reynolds number is $Re_{\text{MSFR}} = 1.06E+06$, but it was not possible nor intended to maintain or fully reproduce it with the scaled experimental model. However the achieved simulated core Reynolds numbers are well within the turbulent range and the highest achieved value is up to 1/6th of the nominal MSFR core Reynolds number. In this section I present and discuss selected results measured in Plane 2 (22.5°) and Plane 3 (33.75°). These two planes are further from the side wall of the tank and represent one plane in the centrelines of the nozzle pairs (Plane 3 (33.75°)) and one in the symmetry plane between nozzles (Plane 2 (22.5°)). For quantitative analysis radial velocity distributions of the two velocity components ($U$ – radial component, $V$ – axial component) and the absolute value of the velocity (or length of the velocity vector – $L$) were extracted along monitor lines shown in Figure 33.
Figure 33: Position and elevation of the monitor lines for radial distribution extraction

Figure 34-37 show radial distributions of (radial (U) and axial (V)) velocity components and velocity length (L) extracted from measurements in Plane 2 (22.5°) taken at different inlet flow rates (refer to Table 14). Distributions are given against radius, where $R = 188$ mm is the position of the cylindrical wall of the geometry, and $R = 0$ m is the location of the 90° corner of the quarter sector cross section. 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 38-41 show similar radial distributions for Plane 3 (33.75°).

Figure 34 ($z = 110$ mm) shows that slightly above the top of the inlet nozzles (nozzle centreline elevation: $z = 41$ mm) the velocity field is mainly defined by the axial (vertical - V) component and the upward flow in the inner part of the geometry ($R = 0-100$ mm). Distributions reach maximum at about $R = 80$ mm while the maximum of the radial component (U) slightly shifts toward the outer region, from $R = 70$ mm to $R = 90$ mm with increasing inlet flow rate. At higher elevations between the inlet and outlet nozzles such as $z = 180$ mm (see Figure 35 and Figure 39) and $z = 280$ mm (see Figure 36 and Figure 40) the radial components are practically reduced to zero, which means almost pure vertical flow. The flow direction is upward in the inner region mentioned above, and the axial component (V) decreases below 0.2 m/s or even reaches zero or negative values that means slight downward flow at the cylindrical wall. At $z = 180$ mm in Plane 2 (22.5°) and in Plane 3 (33.75°) the absolute value of the radial component (U) does not exceed 0.12 m/s and the distributions are
very similar. At \( z = 280 \) mm the radial component in both planes reaches very low values not larger than 0.16 -0.25 m/s, and the direction of the flow has changed, water flows toward the outlet nozzles. At the highest elevation presented (Figure 37 and 41) the distributions are flattened and the radial and axial components are in similar range of extent, 0.2-0.7 m/s for the radial components (U) and 0-0.8 m/s for the axial components. As it can be expected at this elevation increase in inlet mass flow rate will result in larger radial components i.e. increased outward flow.

In Plane 2 (22.5°) at \( z = 180 \) mm the radial component (U) is very small (between -0.08 and 0.04 m/s) compared to the axial component (V) that is practically represented in the velocity length (see Figure 35). A similar dominance of the axial component is visible for Plane 3 (33.75°) at \( z = 280 \) mm where again the velocity length is practically represented by the axial (V) component (see Figure 40).

Between the elevation of the inlet nozzles and the outlet nozzles (see elevations \( z = 110 \) mm, 180 mm and 280 mm, Figure 34-36 and 38-40) the flow field is separated into two main regions, one close to the centreline is characterized by strong upward flow and the other one is near the cylindrical wall, where the vertical velocity components are very small, only a fraction of the main upward velocity values, or even switch into the negative range that means downward flow. With the increase of the inlet flow rate the flow conditions (therefore by getting closer to the original MSFR core flow conditions) this separation of the flow is more significant since the increase of the upward component is larger in the region of the main stream then the increase of the same component in the stagnating part. For example in Plane 2 (22.5°) at \( z = 280 \) mm (Figure 36) the maximum of the axial velocity component is increased from about 0.8 m/s to 1.3 m/s while in the region close to the cylindrical wall it only changes from around 0 m/s to 0.2 m/s by increasing the inlet flow rate. In Plane 3 (33.75°) at \( z = 180 \) mm (Figure 39) maximum of the same velocity component is increased from 1 m/s to over 1.4 m/s while in the region close to the cylindrical wall it is increased from around 0.05 m/s to 0.2 m/s. In both examples the experimentally simulated core Reynolds number is increased from 1.3E+05 to 1.81E+05 through the increase of the inlet flow rate.

It is important to point out that at elevations between the inlet and outlet nozzles the radial velocity component is generally one order of magnitude lower than the axial component along the whole radius (see Figure 35-36 and Figure 39-40). This also identifies the stagnating fluid in this region of the geometry near the cylindrical wall. Naturally close to the inlet or outlet nozzles the radial component will be much higher, and will be comparable with the axial component. It has to be pointed out, that as it will be described in detail in Chapter 6 the velocity uncertainty for low velocities is comparably large and result in large relative uncertainties. This is because the velocity independent component of measurement uncertainty results in a lower threshold, under which the velocity measurement is characterised by large uncertainties. This also explains the seemingly large differences in the resulting radial distributions of the radial velocity component (see Figure 35, 39 and 40).

These measurement results show that this simple core design (a cylindrical core with radial inlet and outlet nozzles at the bottom and the top) would not be preferable, since in case of this molten salt reactor concept such a separation in the flow field would result in significant differences in the local neutronics characteristics leading to large differences in heat generation. Such differences in the local heat generation would result in large temperature differences that would strongly affect the reactor vessel, too. The results indicate that in order to achieve better uniformity in the core flow field different core geometry or flow enhancing internal structures might be necessary.
Figure 34: Radial distribution of radial (U), axial (V) velocity component and absolute velocity value (L), velocity field
Plane 2 (22.5°), z = 110 mm, different inlet flow rates
Figure 35: Radial distribution of radial (U), axial (V) velocity component and absolute velocity value (L), velocity field Plane 2 (22.5°), z = 180 mm, different inlet flow rates
Figure 36: Radial distribution of radial (U), axial (V) velocity component and absolute velocity value (L), velocity field
Plane 2 (22.5°), z = 280 mm, different inlet flow rates
Figure 37: Radial distribution of radial (U), axial (V) velocity component and absolute velocity value (L), velocity field Plane 2 (22.5°), z = 330 mm, different inlet flow rates
Figure 38: Radial distribution of radial (U), axial (V) velocity component and absolute velocity value (L), velocity field Plane 3 (33.75°), z = 110 mm, different inlet flow rates
Figure 39: Radial distribution of radial ($U$), axial ($V$) velocity component and absolute velocity value ($L$), velocity field
Plane 3 (33.75°), $z = 180$ mm, different inlet flow rates
Figure 40: Radial distribution of radial (U), axial (V) velocity component and absolute velocity value (L), velocity field
Plane 3 (33.75°), z = 280 mm, different inlet flow rates
Figure 41: Radial distribution of radial (U), axial (V) velocity component and absolute velocity value (L), velocity field 
Plane 3 (33.75°), z = 330 mm, different inlet flow rates
5.2 Measurement of Symmetrical 2-Loop Operation States

Measurement results of a two-loop operation mode are presented in this section. Here I investigate the case when the two inner loops (Loop 2 and 3) are in operation while the pumps of Loop 1 and Loop 4 are switched off (see Figure 42). This way I could examine a state similar to an asymptotic steady state case of loss of coolant pump. For the PIV measurements the symmetry plane, Plane 4 (45°, see Figure 43) was selected.

![Figure 42: Operation mode of pumps and loops for symmetrical two-loop cases](image)

In the measurements 815 image pairs were recorded at a rate of 15 Hz at each position. (The necessary number of image pairs were determined after and extensive uncertainty analysis that is described in detail in Chapter 6.) The first 15 image pairs were left out of the analysis because of the run-up of the laser intensity at the beginning. Vector maps were calculated using the remaining 800 image pairs. Once again, for evaluation and analyses the average of these instantaneous velocity fields was used. Measurements were carried out along the full height of the model tank in the selected Plane 4. Measurements for each position were repeated with different flow rates set for the two loops according to Table 15. The different inlet mass flow rates correspond to different simulated core Reynolds numbers ranging between $Re_{core} = 1.31E+05 - 1.81E+05$. Radial distributions of velocity and velocity components were compared along selected monitor lines (see Figure 43).

<table>
<thead>
<tr>
<th>Table 15: Loop flow rate settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet valve set [%]</td>
</tr>
<tr>
<td>Inlet volumetric flow rate [l/s]</td>
</tr>
<tr>
<td>Inlet velocity [m/s]</td>
</tr>
<tr>
<td>Inlet Reynolds number [-]</td>
</tr>
<tr>
<td>Simulated core Re-number [-]</td>
</tr>
</tbody>
</table>
Figure 43: Elevations of the monitor lines for radial distribution extraction

Figure 44-46 show velocity length (L), radial (U) and axial (V) velocity component radial distributions extracted from the two-dimensional velocity fields.

At the elevation midway between the inlet and outlet nozzles (z = 180 mm, see Figure 45) the radial component has increased, while the axial component has decreased compared to the values shown at higher elevation, especially in the outer region (R > 90 mm), which indicates stagnation and recirculation. Once again it is also visible, that with increasing inlet flow rates the velocity maximums shift toward the corner region.

Recirculation is more visible at z = 80 mm (see Figure 44), which shows further increased radial component, especially in the inner region (R < 75 mm) together with the axial component reaching almost zero in the outer part (R > 90 mm).

Figure 46 (z = 280 mm) shows that below the outlet nozzles (nozzle centreline elevation: z = 334 mm) the velocity field is mainly defined by the axial (vertical) component and the upward flow in the inner part of the geometry (0 < R < 90 mm). With increasing inlet flow rate the radial component slightly increases as well, indicating intensified flow towards the outlet nozzles. However this change in the values is very limited. Radial distributions of the axial component (V) and the absolute value (L) have a maximum close to the corner (0 < R < 90 mm), and with increasing inlet flow rate this peak shifts toward the corner, while the increase in the outer region is moderate. It shows that with increasing inlet flow rate the separation of the inner, high velocity jet and the outer, low speed region is more significant.

These results indicate the same flow behaviour in the core domain that was presented in Chapter 5.1 for four-loop-operation states. Without any significant modification of the core geometry the flow distribution in the core domain will be dominated by the separation of the stagnating outer part and the inner stream characterised by high velocity in the corner region. It is once again demonstrated that with increasing inlet flow rates this separation is even more evident. Higher flow rates mean that the experimental mock-up is operated at a state even closer to the actual flow conditions of the modelled molten salt reactor concept.
Figure 44: Radial (U), axial (V) velocity component and velocity (L) distributions at different inlet flow rates, Plane 4 (45°), z = 80 mm
Figure 45: Radial (U), axial (V) velocity component and velocity (L) distributions at different inlet flow rates, Plane 4 (45°), z = 180 mm
Figure 46: Radial (U), axial (V) velocity component and velocity (L) distributions at different inlet flow rates, Plane 4 (45°), z = 280 mm
6 UNCERTAINTY ANALYSIS

6.1 ASSESSMENT OF UNCERTAINTY

As it was introduced in Chapter 3.1.1 in Particle Image Velocimetry instead of measuring directly the fluid flow velocity the velocity of seeding particles is measured [47]. Seeding particles mixed into the fluid have approximately the same density as the liquid in question. The particle diameter may range between 5-100 micrometers. Two digital images of the particle distribution are taken and based on the displacement of the particles a two-dimensional velocity field is calculated. Based on the actual physical flow velocity time delay between the recorded images can vary between several tens of microseconds up to several hundred milliseconds. Applied values also depend on seeding density (number of particles added to unit volume or number of recorded particles in one interrogation area) and the size of the interrogation area. General PIV arrangement is shown in Figure 10-11 in Chapter 3.1.1.

In the interrogation areas the velocity is assumed to be uniform during the image pair recording period. By knowing the time delay between the recorded images, together with the displacement of the seeding particles in the image plane velocity vectors can be associated to the interrogation areas using correlation methods [47]. With calibration the displacement in the image plane (measured in pixels) can be converted into metric values according to the following formula [64]:

\[ u = \alpha \frac{\Delta X}{\Delta t} \]  

(11)

where \( u \) is the physical velocity [m/s], \( \alpha \) [m/pixel] is the conversion factor or magnification, \( \Delta X \) [pixel] is the recorded particle image displacement in the image plane and \( \Delta t \) [s] is the time delay between the recording of the two images. Magnification \( \alpha \) is determined by calibration (see Chapter 5).

Similarly to the method proposed by VSJ [64] and ITTC [65] the measurement configuration can be divided into four subsystems:

- calibration: conversion of displacement in the image plane to metric displacement,
- visualisation: seeding particles, illumination,
- image recording: digital camera,
- image processing: cross correlation methods, calculation of the vector field, etc.

Target variables of the measurement are defined by Equation 11, and each variable is affected by the error sources of the four subsystems. Similar division of the measurement is done by Lazar et al. [66], however in their case the particle dynamics have a great contribution since their experiment applied supersonic air flow with oil droplets as particles. Physical quantities considered in their analysis are physical calibration length (l [m]) and image length (L [pixel]), derived from these is magnification or conversion factor (\( \Psi = l/L \)), image recording delay (\( \Delta t \) [s]).

According to VSJ [64] and ITTC [65] the four target variables (\( u, \alpha, \Delta X \) and \( \Delta t \)) are influenced by the followings.

Magnification \( \alpha \) is influenced by physical, geometrical properties and accuracy of the calibration target, the actual position and the positioning accuracy of the calibration target, the recording accuracy of the imaging system (digital camera, optical lens system, etc), and the physical properties and actual position of the illuminating light sheet.

\( \Delta X \) displacement in the image plane is affected by the followings: through illumination and the properties of the imaging system the actual position of the particles, the imaging devices and methods will determine the errors.

\( \Delta t \) time delay between the subsequent images has error sources due to the following components: jitter of the illuminating device (i.e. the laser), which means the timing error of
the illuminating pulses, and the jitter of the timer, which determines the error of the delay interval.

In addition flow velocity $u$ has error sources because of the measurement principle. In general the seeding particles have a certain flow following capability and a tendency of gravitational settling. Both of these are functions of the closeness of the density of the particle to the density of the fluid, and also determined by the flow conditions. Additional effects are: distortion by the two dimensional measurement of a possibly three-dimensional flow field, or the optical perspective. The $\delta u$ velocity difference therefore has errors because of the visualisation subsystem as well.

The reference measurement presented by VSJ [64] (and ITTC [65]) was water flow in a rectangular tank. Inlet was horizontal on the side, outlet was vertical through a nozzle located at the bottom of the tank. Dimensions of the tank were 300x300x50 mm, in which water height was 160 mm, therefore free surface flow was observed (see Figure 47). Inlet velocity was 400 mm/s. The geometry was smaller but very similar to the mixing tank investigated at BME NTI [67, 68] (see Figure 48). The applied laser was a 20 mJ Nd:YAG, the applied delay was $\Delta t=2.4$ ms. For seeding 50 $\mu$m diameter nylon particles were used, applied lens was a Nikon 60 mm f/2.8 macro (exact same as the one applied at BME NTI), image recording frequency was 30 Hz, size of the interrogation area was 32x32 pixels.

Figure 47: Experimental tank, dimensions and measurement result, VSJ PIV Handbook [64]

Figure 48: Mixing tank, dimensions of the tank investigated at BME NTI [67, 68]
Gui, Longo and Stern [69, 70] present towing tank measurements in which the PIV system moves along with the model vessel. For seeding 15 µm average diameter coated hollow glass spheres were used. Characteristic velocity (towing speed) was $U_c = 1.53$ m/s, resulting in a $Re = 5.2 \times 10^6$. The following variables were considered to be influenced by error sources: the physical calibration length ($L_{obj}$ [m]) and the image calibration length ($L_{img}$ [pixel]), time delay between the subsequent images, displacement (by components) in the image plane, physical coordinates. In their case $L_{obj}/L_{img}$ ratio is equal to magnification.

S. Longo et al. [71] investigated water flow around a hydrofoil using PIV (see Figure 49). For seeding $d_p = 10$-20 µm sand was used, elements of the PIV system were: TSI PIV 4 MP camera with a Nikkor AF D 50 mm lens, focal length extended by 14 mm. As light source Solo Nd:YAG III dual 15 Hz 50 mJ laser was applied. Image recording frequency was 3.75 Hz, delay was 2000 µs. In the uncertainty analysis two components were considered: reading of calibration target (1 pixel as bias) and random uncertainty of 0.28 pixels from sub-pixel analysis. Error sources of timing were considered negligible.

![Figure 49: Overview of the hydrofoil experimental system by S. Longo et al [71]](image)

For the uncertainty assessment of the PIV measurement I followed the method proposed by VSJ [64] that applies the ANSI/ASME PTC 19.1-1985 standard [72, 73] and this will be presented in this current Chapter 6.1.

According to this each component of Equation 11 is associated with systematic error and random error, which result in the bias and random uncertainty of the measured values. These quantities can be described as follows:

- Precision is the closeness of multiple observations to one another, or the repeatability of a measurement. Precision is characterised by precision index or random error, often denoted with $S$.
- Accuracy is the closeness of a measurement (or set of observations) to the true value. The true value – therefore accuracy – is unknown, however estimation could be made by determining the bias error or bias limit, often denoted with B.
According to the standard [72, 73] random error and systematic error of a measured value should be specified at 95% confidence level and the root sum square (RSS) should be formed. This will give us the measurement uncertainty corresponding to 95% confidence level.

The measurement result \( r \) is a function of \( X_i \) measured parameters: \( r = f( X_1, X_2, \ldots, X_J) \). In the following example I will consider two measurement parameters: \( x \) and \( y \). Both have statistical error \( \varepsilon \) with Gauss distribution around \( \mu_x \) and \( \mu_y \) average values. Systematic error is \( \beta \), and is unknown and not measurable, however it can be estimated based on assumptions and past experience. If it can be assumed that the systematic error changes during repetition of measurements, it also may have a distribution, either Gaussian, uniform or other. Considering one selected random sample of a measurement series it can be written for two measurement parameters:

\[
x_i = x_{\text{true}} + \beta_{xi} + \varepsilon_{xi} \\
y_i = y_{\text{true}} + \beta_{yi} + \varepsilon_{yi}
\]

and introducing

\[
\delta_i = r(x_i, y_i) - r(x_{\text{true}}, y_{\text{true}})
\]

and using the Taylor series of \( r \) around the true value, the error of \( r \) measurement result can be written as

\[
\delta_i = \theta_x \beta_{xi} + \theta_y \varepsilon_{xi} + \theta_x \beta_{yi} + \theta_y \varepsilon_{yi}
\]

where \( \theta_x \) and \( \theta_y \) are the sensitivity factors of \( x \) and \( y \), respectively. In general sensitivity factors represent the contribution of the given error source to precision and accuracy, and can be written as

\[
\theta_i = \frac{\partial r}{\partial X_i}.
\]

Taking the square of both sides of Equation 14 and summarising it for \( N \) samples (\( N \) goes to infinity), and substituting the statistical parameters we get

\[
u_r^2 = \theta_x^2 b_x^2 + \theta_y^2 b_y^2 + 2 \theta_x \theta_y b_{xy} + \theta_x^2 S_x^2 + \theta_y^2 S_y^2
\]

where \( u_r \) is the combined standard uncertainty, \( b_x^2 \) and \( b_y^2 \) are the variances of systematic errors, \( b_{xy} \) is the covariance, and \( S_x^2 \) and \( S_y^2 \) are the standard deviation of random errors. Generalized form of the formulation above, by expressing the variance of the function, would be the following:

\[
D^2 g(\xi) = M\left((\Delta g)\right)^2 = \sum_{i=1}^{n} \left(\frac{\partial g}{\partial \xi_i}\right)^2 D^2 (\xi_i)
\]

where \( \xi_i \) are the directly measured quantities, \( g(\xi) \) is the function that links the directly measured values with the target values of a measurement and \( D^2 \) is the variance of the value of the function at \( \xi_i \). This is a general formula to describe error propagation [74].

From now on the covariance of random errors is going to be considered to be zero. The Standard divides Equation 16 as

\[
B_r^2 = \theta_x^2 B_x^2 + \theta_y^2 B_y^2
\]

and

\[
S_r^2 = \theta_x^2 S_x^2 + \theta_y^2 S_y^2.
\]

The ASME standard [72, 73] considers the covariance of the systematic errors \( b_{xy} \) to be zero. Factors of systematic error is not known, therefore estimation has to be made considering all the error sources and all the variables: the high estimate of an elemental systematic error \( B_{jk} \) is called bias limit. Estimation of bias limits is supposed to be made at 95% confidence level. Total systematic error of parameter \( X_j \) will be

\[
B_j = (B_{j1}^2 + B_{j2}^2 + B_{j3}^2 + \ldots + B_{jM}^2)^{1/2}.
\]
$S_x$ (or $S_y$) will be the standard deviation of the average of the given parameter:

$$S_{\bar{X}} = \left( \frac{1}{N(N-1)} \sum_{i=1}^{N} (X_i - \bar{X})^2 \right)^{1/2}. \quad (20)$$

According to the standard at 95% confidence level the random and systematic uncertainties can be summarised as

$$U_{RSS} = [B_r^2 + (t_{95}S_r)^2]^{1/2} \quad (21)$$

or at 99% confidence level

$$U_{ADD} = B_r + t_{95}S_r \quad (22)$$

where $t_{95} = 2$ (N>30). If the systematic or the random component is negligible compared to the other, both Equation 21 and 22 will result in 95% confidence interval.

### 6.2 Sources of Error and Sensitivity Factors of Magnification $\alpha$

Determination of magnification factor $\alpha$ is determined by placing a calibration target at the measurement position, in the same plane as the light sheet is used for two-dimensional PIV measurement. Image of the calibration target is recorded by the image recording device (CCD camera) and the magnification factor can be derived by relating the recorded metric information to the pixels of the image.

#### 6.2.1 Reference length identification

Reading of the reference length is carried out by identifying the location of two points with the accuracy of 0.5 pixels each, therefore the total accuracy will be 0.707 pixels. The sensitivity factor is defined by the following equation [64]:

$$\frac{\partial \alpha}{\partial L_{sel}} = -\frac{l_{sel}}{L_{sel}} \frac{1}{L_{sel}} [\text{mm/pixel}^2] \quad (23)$$

where $L_{sel}$ [pixel] is the length on the image plane and $l_{sel}$ [mm] is the real physical length. The Dantec calibration board is 200x200 mm dot matrix with 5 mm pitch. Manufacturing accuracy is assumed to be 0.1% = 0.02 mm. Sensitivity factor is [64]:

$$\frac{\partial \alpha}{\partial l_{sel}} = 1/L_{sel} [1/pixel]. \quad (24)$$

#### 6.2.2 Error caused by the image recording system

There are two main parts of the image recording device: the recording (digital) camera and the attached lens. Distortion of a quality optical lens is approximately 0.5 % [64] of the total recorded image width (therefore the calibration length). This is in line with the same suggested value from Lazar et al. [66]. In my case the applied lens is exactly the same that was referenced by VSJ [64], a Nikon 60mm f/2.8 Micro-NIKKOR AF-D, and it is widely used with PIV systems. Therefore accuracy due to lens distortion is considered to be 0.005×$L_{sel}$, sensitivity factor is defined in Equation 23. $L_{sel}$ image length has to be acquired for each measurement plane.

In case of digital cameras the recording sensor is usually a CCD in which the recording pixels are located in rows and columns. Distortion is due to the manufacturing inaccuracy of the image recording sensor and is in the range from 0.0033 pixels (or 1/300th pixel, ∼50 nm [75]) to 100 nm or 0.006 pixels, while the latter value can be considered to be an upper limit [76]. The value of 0.0056 pixels [64] is going to be used here as accuracy. Sensitivity factor is defined by Equation 23. As suggested by [64] further error sources originating from the design and operation features of a CCD are considered to be included in the reference length identification accuracy.

If the calibration plane and the light sheet (measurement plane) are at a small angle to each other, this slight rotation will introduce bias. For small angles up to $\Theta = 0.035$ rad (=2°) the $l_{sel}$ recorded distance is reduced by $l_{sel}\Theta^2$ and the magnification will be [64]:

Bogdán Yamaji PhD thesis 64/118
\[ \alpha = \frac{l_{sel} \Theta^2}{L_{sel}} \]  
and the sensitivity factor will be
\[ \frac{\partial \alpha}{\partial \Delta \Theta} = -2l_{sel} \Theta / L_{sel} \text{ [mm/pixel].} \]  

6.2.3 Error due to de-warping

Measurement planes are defined as shown in Figure 30. These planes were selected according to the symmetry of the geometry and the flow conditions. In these measurement planes the flow is considered to be dominantly two-dimensional. The recording camera is placed perpendicular to the side wall of the measurement tank therefore the measurement planes are not parallel with the plane of the CCD sensor. The aforementioned dot matrix reference target (see Figure 31) 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 converted into metric data. Calibration is carried out for every measurement plane and the measurement geometry is kept the 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 [63]. The polynomial model is strictly empirical but the advantage of the method is that it is more robust than other methods. Accuracy of the de-warping is measured by the average re-projection error which is calculated for each measurement plane and each calibration. This value is taken into account as bias limit of de-warping, sensitivity factor is calculated as in Equation 23.

6.3 Sources of error and sensitivity factors of \( \Delta X \) image displacement

\( \Delta X \) displacement in the image plane is influenced by the actual positions of the seeding particles, particle image position, properties of the image recording devices and methods through properties of illumination and image recording.

6.3.1 Error due to illumination

Spatial and temporal fluctuations of the illuminating light sheet affect the detected position of the seeding particle. According to [64] this means a bias of about 1/10th of the particle diameter, in our case \( 0.1 \times d_p = 0.005 \text{ mm} \). Since two subsequent images have to be considered, the total bias limit will be \( \sqrt{2} \times 0.005^2 = 0.007 \text{ mm} \). From Equation 11 velocity is
\[ u = \frac{\Delta x}{\Delta t} = \alpha \frac{\Delta X}{\Delta t} \]  
and using \( \Delta x \) physical displacement the displacement in the image plane will be
\[ \Delta X = \frac{\Delta x}{\alpha}. \]  
With this sensitivity factor for the displacement will be
\[ \frac{\partial \Delta X}{\partial \Delta x} = 1/\alpha. \]

6.3.2 Error caused by the image recording system

Error due to the optical distortion of the imaging lens was already taken into account at the accuracy of magnification \( \alpha \) (see Chapter 6.2). CCD distortion accuracy – similarly as in Chapter 6.2 – is considered to be 0.0056 pixels. Sensitivity factor equals to 1 [64]. Further error sources originating in the design and operation features of a CCD are considered to be included in sub-pixel analysis accuracy [64].

Bias due to the displacement of the recording device (CCD sensor, i.e. the camera) is considered to be included in the bias of magnification \( \alpha \) as part of the de-warping accuracy.
6.3.3 Image processing, calculation of displacement

The method of determining the particle displacement uses sub-pixel analysis in order to specify the location of the correlation peak with a resolution under 1 pixel. Precision and accuracy of the sub-pixel analysis is a function of multiple factors. According to the results of the analysis presented in [64] uncertainty precision and accuracy of the sub-pixel analysis is considered to be 0.033 pixels and 0.017 pixels respectively with a sensitivity factor of 1 [64]. Smoothing due to interrogation is considered as bias, accuracy is 0.008 pixels and the sensitivity factor is 1 [64].

6.4 Sources of error and sensitivity factors of Δt time delay

In case of the Δt delay between the subsequent images recorded during a PIV measurement there are two sources of error. Precision and accuracy of the timing of the light pulses and the delay is determined by the properties of the delay generator and the laser.

6.4.1 Error sources of the delay generator (timer) timing

For precision and accuracy of the delay generator (timer) one should refer to the reference manual of the device. If the accessible data is unclear or underspecified, it is suggested to consider the information for precision and accuracy as well [64]. In case of the applied Dantec Timer Box 80N77 pulse positioning accuracy is 12 ns [52]. Sensitivity factor will be 1 [64].

6.4.2 Error sources of the laser pulse timing

Lasers are characterised by so-called jitter, which is the timing uncertainty of the illuminating laser pulse. Information on the jitter should be referenced from the manual of the applied light source. Once again, if the accessible data is unclear or underspecified, it is suggested to consider the information for precision and accuracy as well [64]. Jitter of the Litron Nano L 135-15 PIV laser applied in my measurements is less than 0.5 ns [50], therefore 0.5 ns is considered as precision and accuracy, sensitivity factor is 1 [64].

6.5 Sources of error and sensitivity factors of δu velocity difference

6.5.1 Flow following ability of the particles (trajectory)

Flow following capability of seeding particles added to the fluid is composed of two phenomena, namely acceleration response and gravitational settling. According to the literature acceleration response is generally investigated by the frequency response of the seeding particle, which means that how a particle follows a sinusoidal change of flow velocity. This capability is basically determined by the specific density of the particle, although other factors, such as particle diameter, and the properties of the flow and the fluid also has to be considered. Gravitational settling (or elevation) can be considered as negligible except for the lowest flow velocities, therefore this effect is usually neglected. According to Adrian and Westerweel [77 (pp. 40-50)], when the relative density is in the range between 0.56<ρ’<1.62 (where ρ’=ρp/ρf=1, ρp: particle density, ρf: fluid density) the Hp complex frequency response will not cut off, therefore such particles have good acceleration response. Raffel et al. [47 (pp. 15-22)] state that if the density of the applied seeding particle is close to the fluid density, then flow following ability of the particles will be good. In my case the polyamide seeding particle diameter is dp=50 µm and the density is ρp=1.03 g/cm³, therefore ρ’=1.03 in water medium, the applied particle can be considered as good flow follower.
According to Stokes’ law velocity component $U_g$ due to gravity can be written as follows [47 (p. 15)]:

$$\dot{U}_g = \frac{d_p^2 (\rho_p - \rho)}{18 \mu} g$$  \[(30)\]

where $g$ is gravity, $\mu$ is dynamic viscosity of the fluid and $d_p$ is the particle diameter. Applying this formula for the Dantec PSP 50 particles the settling velocity will be $U_g = 4.0 \times 10^{-5} \text{ m/s} = 0.04 \text{ mm/s}$. In case of $d_p=5 \mu\text{m}$, $\rho_p=1.05 \text{ g/cm}^3$ polystyrene latex particles the settling velocity would be $0.68 \mu\text{m/s} = 0.00068 \text{ mm/s}$ [77]. Extrapolating this value using second order polynomial formula for $d_p=50 \mu\text{m}$ diameter settling velocity would be $67 \mu\text{m/s} = 0.067 \text{ mm/s}$. [64] and ITTC [65] calculated the velocity due to gravitational settling to be $0.05 \text{ mm/s}$. In my case the applied value will be $0.04 \text{ mm/s}$, considering it as bias, but should only be taken into account for the component parallel to $g$, and sensitivity factor would be 1.

### 6.5.2 Three-dimensional effects

Movement of a seeding particle in a light sheet of finite thickness may have a velocity component perpendicular to the light sheet. Because of such a velocity component the recorded particle displacement will be different, and this effect is called perspective due to the three-dimensional nature of the flow. Systematic error due to perspective could be up to 15%, but it is commonly neglected since it does not alter the general characteristics of the measured flow, and total elimination of this error source is only possible with three-dimensional stereoscopic PIV [47 (p. 59)].

### 6.6 Sampling

In case of PIV measurements results are generally ensemble or time averages of instantaneous values [66]. Many publications discuss the necessary sample (two-dimensional instantaneous velocity field) number for a result with appropriate standard deviation. Gui et al. [69] use $N=1200$ instantaneous velocity fields for averaging. Uzol and Camci [78] (air flow seeded with fog particles, inlet Reynolds number of 10,000) compares the average of a total of $N=2750$ instantaneous velocity fields with the average of $N = 5, 10, 25, 50, 100, 250, 500, 750, 1000$ samples taken from the same set of measurement. Their finding was that $N=1000$ samples are sufficient to minimise the variations. They also add that deviations from the average value strongly depend on turbulence intensity and flow velocity especially in case of low sample numbers. Similar investigation is presented by INL [79] through presenting the results of measurements carried out with mineral oil as fluid, seeded with $14 \mu\text{m}$ silver coated hollow glass spheres. In their case the PIV system consisted of a Big Sky 532 nm Nd-YAG laser and LaVision cameras (2-15 Hz). In this analysis the total number of samples was $N=4000$, and for averaging $N=5-4000$ samples were used. They found that in case of $N=500$ the average will represent velocities and the turbulence quantities well.

I present the examination of effect of sample number on the average result through the analysis of a selected measurement. In this example case total number of instantaneous velocity fields was $N_0=2411$, the average of all the samples was used as reference. For the investigation $N=50, 100, 250, 500, 1000, 1500$ samples were used for averaging and the difference from the reference value of the measured velocity components was analysed.

Figure 50 and 51 show that for these measurement conditions with a sample number over $N=500$ the relative difference between the average and the reference value will be less than 5% and by further increasing the sample number the relative difference will not decrease significantly. Recording a maximum of 1000 image pairs is determined to be sufficient for averaging, and considering other factors, such as buffering of measurement images, power
input of the pumps, I selected N=800-815 to be the length of measurement, namely the recorded sample number for further measurements.

Figure 50: Relative difference of averages (U velocity component) as a function of sample number

Figure 51: Component U moving average as a function of sample number compared to the reference average
6.7 QUANTIFICATION OF UNCERTAINTY

In order to quantitatively demonstrate the uncertainty analysis method a calculation for one measurement point will be presented in this section. Table 16 shows the properties of the selected point.

Table 16: Properties of the selected point

<table>
<thead>
<tr>
<th>Measurement plane</th>
<th>Plane 2 (22.5°)</th>
</tr>
</thead>
<tbody>
<tr>
<td>R (x) [mm]</td>
<td>160</td>
</tr>
<tr>
<td>z (y) [mm]</td>
<td>329.9</td>
</tr>
<tr>
<td>Vector length (velocity) ( U_0 ) [mm/s]</td>
<td>421</td>
</tr>
<tr>
<td>Time delay ( \Delta t ) [s]</td>
<td>0.00025</td>
</tr>
<tr>
<td>Displacement in the image plane ( \Delta X ) [pixel]</td>
<td>0.888</td>
</tr>
<tr>
<td>Calibration width (range of applicability) ( l_{sel} ) [mm]</td>
<td>185</td>
</tr>
<tr>
<td>Calibration width in the image plane ( L_{sel} ) [pixel]</td>
<td>1561</td>
</tr>
<tr>
<td>Magnification ( \alpha = l_{sel}/L_{sel} ) [mm/pixel]</td>
<td>0.1185</td>
</tr>
<tr>
<td>Average re-projection error [pixel]</td>
<td>0.389</td>
</tr>
</tbody>
</table>

Table 17 summarises the error sources of each parameter distinguishing by subsystem as described in Chapters 6.2-6.5. In Table 17 the values of each accuracy and precision component are shown as well, together with the applied sensitivity factors. Using these for each subsystem the absolute accuracy and precision values can be calculated. In Table 18 values of accuracy and precision are given, and the total absolute velocity uncertainty at the selected measurement point is presented.

Table 17: Summary of the PIV measurement subsystem parameters, error sources, accuracy and precision values, sensitivity factors for the measurement setup

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Subsystem</th>
<th>Error source</th>
<th>Absolute accuracy</th>
<th>Absolute precision</th>
<th>Sensitivity factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha ) magnification</td>
<td>Calibration</td>
<td>Reading of reference length</td>
<td>( B_{11} = 0.7 ) pixel</td>
<td>-</td>
<td>( \theta_{11} = l_{sel}/L_{sel}^2 = 7.6E-05 ) mm/pixel^2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Physical distance</td>
<td>( B_{12} = 0.02 ) mm</td>
<td>-</td>
<td>( \theta_{12} = 1/L_{sel} = 6.4E-04 ) 1/pixel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Optical distortion</td>
<td>( B_{13} = 0.005 \times L_{sel} )</td>
<td>-</td>
<td>( \theta_{13} = l_{sel}/L_{sel}^2 = 7.6E-05 ) mm/pixel^2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>CCD distortion</td>
<td>( B_{14} = 0.0056 ) pixel</td>
<td>-</td>
<td>( \theta_{14} = l_{sel}/L_{sel}^2 = 7.6E-05 ) mm/pixel^2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Angle of calibration plane and illuminated plane</td>
<td>( B_{15} = 0.035 ) rad</td>
<td>-</td>
<td>( \theta_{15} = 0.0083 ) mm/pix</td>
</tr>
<tr>
<td></td>
<td></td>
<td>De-warping</td>
<td>( B_{16} = 0.389 ) pixel</td>
<td>-</td>
<td>( \theta_{16} = l_{sel}/L_{sel}^2 = 7.6E-05 ) mm/pixel^2</td>
</tr>
<tr>
<td>( \Delta X ) displacement in the image plane</td>
<td>Data recording</td>
<td>Spatial and temporal fluctuation of the laser CCD distortion</td>
<td>( B_{21} = 0.00707 ) mm ( B_{22} = 0.00056 ) pixel</td>
<td>-</td>
<td>( \theta_{21} = 1/\alpha = 8.44 ) [pixel/mm] ( \theta_{22} = 1 )</td>
</tr>
<tr>
<td></td>
<td>Data processing</td>
<td>Displacement calculation</td>
<td>( B_{23} = 0.017 ) pixel</td>
<td>( S_{13} = 0.033 ) px</td>
<td>( \theta_{23} = 1 ) ( \theta_{13} = 1 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Smoothing by interrogation</td>
<td>( B_{23} = 0.008 ) pixel</td>
<td>-</td>
<td>( \theta_{23} = 1 ) ( \theta_{3} = 1 )</td>
</tr>
<tr>
<td>( \Delta t ) time delay</td>
<td>Data recording</td>
<td>Timer jitter</td>
<td>( B_{31} = 1.25E-08 ) s ( B_{32} = 5.00E-10 ) s</td>
<td>( S_{2} = 5E-10 ) s</td>
<td>( \theta_{31} = 1 ) ( \theta_{32} = 1 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Laser jitter</td>
<td>( B_{32} = 1.25E-08 ) s</td>
<td>-</td>
<td>( \theta_{32} = 1 ) ( \theta_{3} = 1 )</td>
</tr>
<tr>
<td>( \delta u ) velocity difference</td>
<td>Measurement principle</td>
<td>Gravitational settling</td>
<td>( B_{24} = 0.04 ) mm/s</td>
<td>-</td>
<td>( \theta_{24} = 1 )</td>
</tr>
</tbody>
</table>

Bogdán Yamaji PhD thesis 69/118
With the values given in Table 17:

\[ B_\alpha = \left[ (B_{11}\theta_{11})^2 + (B_{21}\theta_{21})^2 + (B_{31}\theta_{31})^2 + (B_{41}\theta_{41})^2 + (B_{51}\theta_{51})^2 + (B_{61}\theta_{61})^2 \right]^{1/2} = \] 
\[ = \left[ (0.7\times(-7.6E-05))^2 + (0.02\times6.4E-04)^2 + (0.005\times1561\times(-7.6E-05))^2 + 
\right.\] 
\[ + (0.0056\times(-7.6E-05))^2 + (0.035\times(-0.0083))^2 + (3.89E-01\times(-7.6E-05))^2 \right]^{1/2} = \] 
\[ = 6.63E-04 \text{ mm/pixel} \quad (31) \]

\[ S_\alpha = 0 \] 
(32)

\[ B_\Delta X = \left[ (B_{12}\theta_{12})^2 + (B_{22}\theta_{22})^2 + (B_{13}\theta_{13})^2 + (B_{23}\theta_{23})^2 \right]^{1/2} = \] 
\[ = \left[ (0.00707\times8.44)^2 + (0.005\times1)^2 + (0.017\times1)^2 + (0.008\times1)^2 \right]^{1/2} = 0.063 \text{ pixel} \] 
(33)

\[ S_\Delta X = \left[ (S_{13}\theta_{13})^2 \right]^{1/2} = \left[ (0.033\times1) \right]^{1/2} = 0.033 \text{ pixel} \] 
(34)

\[ B_\Delta t = \left[ (B_{42}\theta_{42})^2 + (B_{52}\theta_{52})^2 \right]^{1/2} = \left[ (1.25E-08\times1)^2 + (5.0E-10\times1)^2 \right]^{1/2} = 1.25E-08 \text{ s} \] 
(35)

\[ S_\Delta t = \left[ (S_{52}\theta_{52})^2 \right]^{1/2} = \left[ (5.0E-10\times1) \right]^{1/2} = 5.0E-10 \text{ s} \] 
(36)

\[ B_\delta u = \left[ (B_{24}\theta_{24})^2 \right]^{1/2} = \left[ (0.04\times1) \right]^{1/2} = 0.04 \text{ mm/s}. \] 
(37)

Following the method proposed by VSJ [64] relative bias limit values and relative random uncertainty values will be:

\[ B_{\alpha}/\alpha = 6.63E-04 \text{ mm/px}/0.1185 \text{ mm/px} = 0.0056 = 0.56\% \] 
(38)

\[ B_{\Delta X}/\Delta X = 0.063 \text{ px}/0.888 \text{ px} = 0.07 = 7\% \] 
(39)

\[ B_{\Delta t}/\Delta t = 1.25E-08 \text{ s}/0.00025 \text{ s} = 0.00005 = 0.005 \% \] 
(40)

\[ B_\delta u/\delta u = 0.04 \text{ mm/s}/421 \text{ mm/s} = 9.5E-05 = 0.0095\% \] 
(41)

\[ B_u/U = \sqrt{\left( B_\alpha/\alpha \right)^2 + \left( B_{\Delta X}/\Delta X \right)^2 + \left( B_{\Delta t}/\Delta t \right)^2 + \left( B_\delta u/\delta u \right)^2} = 0.07 = 7\% \] 
(42)

\[ S_{\alpha}/\alpha = 0 \] 
(43)

\[ S_{\Delta X}/\Delta X = 0.033 \text{ px}/0.888 \text{ px} = 0.037 = 3.7\% \] 
(44)

\[ S_{\Delta t}/\Delta t = 5.0E-10 \text{ s}/0.00025 \text{ s} = 0.00002 = 0\% \] 
(45)

\[ S_\delta u/\delta u = 0 \] 
(46)

\[ S_u/U = \sqrt{\left( S_\alpha/\alpha \right)^2 + \left( S_{\Delta X}/\Delta X \right)^2 + \left( S_{\Delta t}/\Delta t \right)^2 + \left( S_\delta u/\delta u \right)^2} = 0.037 = 3.7\%. \] 
(47)

Using the values for the selected measurement point (see Table 16) the relative combined (root sum square) uncertainty can be calculated as follows:

\[ U_{uRSS}/U = \sqrt{\left( B_u/U \right)^2 + \left( 2\times S_u/U \right)^2} = 0.1018 = 10.18\%. \] 
(48)
Table 18: Total absolute velocity uncertainty at the selected measurement point

<table>
<thead>
<tr>
<th></th>
<th>Absolute accuracy $B_0$</th>
<th>Absolute precision $S_0$</th>
<th>Velocity sensitivity coefficient $\theta_i$</th>
<th>Accuracy [mm/s]</th>
<th>Precision [mm/s]</th>
<th>Uncertainty [mm/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$=0.1185 mm/px</td>
<td>6.63E-04 mm/px</td>
<td>-</td>
<td>3552 px/s</td>
<td>6.63E-04 mm/px×3552 px/s= 2.35 mm/s</td>
<td>-</td>
<td>43.25</td>
</tr>
<tr>
<td>$\Delta x$=0.888 px</td>
<td>0.063 px</td>
<td>0.033 px</td>
<td>474 mm/px/s</td>
<td>0.063 px×474 mm/px/s= 29.86 mm/s</td>
<td>0.033 px×474 mm/px/s= 15.64 mm/s</td>
<td></td>
</tr>
<tr>
<td>$\Delta t$=0.00025 s</td>
<td>1.25E-08 s</td>
<td>5.0E-10 s</td>
<td>-1.68E+06 mm/s²</td>
<td>-0.021 mm/s</td>
<td>-0.00084</td>
<td></td>
</tr>
<tr>
<td>$\delta u$ (u=421 mm/s)</td>
<td>0.04 mm/s</td>
<td>1</td>
<td>0.04 mm/s</td>
<td>0.04 mm/s</td>
<td>0</td>
<td></td>
</tr>
</tbody>
</table>

An additional component of uncertainty is the precision calculated from the averaging. In this example $N=785$ image pairs were processed, which resulted in the same number of instantaneous vector fields, all of these were used in the averaging process. Using the result value for the selected point defined in Table 16 standard deviation of the velocity vector was $\sigma = 0.22$ m/s. Precision of sampling $S_s$ can be approximated by calculating the standard deviation of the mean as

$$SD_{mean} = \sigma / \sqrt{N}. \quad (49)$$

where $N$ is the sample number (in this example case $N = 785$). Thus $S_s = ((0.22 \text{ m/s}) / \sqrt{785}) = 0.0078 \text{ m/s} = 7.8 \text{ mm/s}$. Adding this to the result given in Table 18 the combined absolute uncertainty for the selected example point will be $U_S = 43.42 \text{ mm/s}$, and relative combined uncertainty is $U_{rel,S} = U_S / U_0 = 43.42 / 421 = 0.1031 = 10.31\%$.

The methodology described above was applied for each measurement point (each measured velocity vector or interrogation area) of a complete two-dimensional velocity field. Figure 52 represents the distribution of relative combined velocity uncertainty (in Figure 52 denoted with $U_{rel}$) over the whole measurement area. It is visible that for the most of the recorded area relative uncertainty is under 10% and only in a small segment will it increase over 30%. In Figure 53 radial distribution of the relative velocity uncertainty is presented along lines at three elevations ($z = 275$ mm, 305 mm and 375 mm, corresponding to 1/4, 2/4 and 3/4 of the total height of the measurement domain shown in Figure 52). As it is visible the lower outer section of the measurement domain the relative velocity uncertainty can go over even 50%, however these extreme values limited only to a small part of the whole recorded area. This area with higher relative uncertainty is the outer part, close to the cylindrical wall, and as it is shown in Figure 30 where for the measurement the lowest velocity values were recorded. In this region the velocity independent component of the total uncertainty will result in a large relative velocity uncertainty.
Figure 52: Relative velocity uncertainty ($U_{\text{rel}}$) distribution over the measurement area

Figure 53: Radial distribution of relative velocity uncertainty ($U_{\text{rel}}$) at selected elevations
7 ENHANCEMENT OF THE CORE FLOW FIELD WITH MODIFICATION OF THE GEOMETRY

7.1 DESIGN OF A CORE FLOW DISTRIBUTOR PLATE

In order to enhance the flow distribution in the core domain separation of the stagnating, recirculating region and the region with strong upward flow should be avoided. Therefore in this section I propose the application of a perforated flow distribution plate. In case of the MOSART single region molten salt reactor concept Ignatiev et al. suggested a perforated plate with a volume porosity $\phi=32\%$ designed to be placed under the core region [14]. Details of the proposed distributor plate were not defined, therefore here I discuss examples from industrial solutions. In case of VVER-440 type pressurised water reactors two hole diameters ($d_{440,1}=40$ mm, $d_{440,2}=70$ mm) are used in two separate perforated structures [41]. In case of VVER-1000 $d_{1000}=40$ mm is used in a certain structure, too [42]. In order to experimentally investigate the effect of such a perforated plate the $\phi=32\%$ volume porosity recommended for the MOSART concept and the two aforementioned hole diameters are analysed here. In case of the 1:6 linearly scaled segment model the relevant hole diameters would be $d_{\text{modell},1}=6.7$ mm and $d_{\text{modell},2}=11.67$ mm. For future manufacturing reasons $d_{\text{modell},1'}=7$ mm and $d_{\text{modell},2'}=12$ mm will be considered. Table 19 shows the necessary hole numbers and pitch values for the two hole diameters and two lattice structures for approximately $\phi=32\%$ volume porosity.

Table 19: Possible parameters of the perforated plate in the scaled and segmented model

<table>
<thead>
<tr>
<th>Lattice structure and diameter</th>
<th>Theoretical porosity</th>
<th>Hole diameter</th>
<th>Pitch</th>
<th>Number of holes (theoretical)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Square lattice, diameter 1</td>
<td>31.81 %</td>
<td>7 mm</td>
<td>11 mm</td>
<td>228</td>
</tr>
<tr>
<td>Triangular lattice, diameter 1</td>
<td>30.86 %</td>
<td>7 mm</td>
<td>12 mm</td>
<td>221</td>
</tr>
<tr>
<td>Square lattice, diameter 2</td>
<td>31.33 %</td>
<td>12 mm</td>
<td>19 mm</td>
<td>76</td>
</tr>
<tr>
<td>Triangular lattice, diameter 2</td>
<td>32.65 %</td>
<td>12 mm</td>
<td>20 mm</td>
<td>80</td>
</tr>
</tbody>
</table>

The experimental geometry has a 90° rotational symmetry so the application of the triangular lattice would result in a geometric difference between loops 1-2 and 4-3 respectively. Therefore in my preliminary CFD investigation square lattice arrangements are used. Also for later manufacturing reasons thickness of the perforated plate for the experimental model is selected to be $h=12$ mm, which is a standard size for plexiglas plates.

Figure 54 and Figure 55 show the geometry, dimensions and position of the proposed perforated flow distributor plates.
Figure 54: Geometry and dimensions of the modelled perforated plate designs with hole diameters of 7 mm and 12 mm

Figure 55: Three-dimensional geometry model with the perforated plate (diameter: 12 mm, pitch: 19 mm)
Figure 56-59 show a comparison of previous measurement and CFD simulation results of the original geometry (without perforated plate) with simulation results of the models that include perforated plate geometry. The figures show comparisons of the radial distributions of the radial (U) and axial (V) velocity component and the absolute velocity value (L) at selected elevations in Plane 3 (33.75°) and Plane 4 (45°). The results clearly show that the implementation of such a flow distributor plate would fundamentally modify and improve the flow field in the core region. The axial velocity component (V) in both planes and both elevations has a much more uniform, flat shape compared to the original case. It means that the flow direction is practically upward along the radius in the core domain, any velocity decrease is limited to the corner region and the difference between the maximum and the minimum values are moderate. For the axial velocity component the region with negative values (i.e. downward circulation) is almost completely removed by the flow distributor plate in both Plane 3 and Plane 4, at the presented elevations. It means that the recirculation near the cylindrical wall is successfully avoided in most of the geometry, and the flow field is much better balanced in the core region. As a result the radial velocity component (U) also has a less hectic radial distribution. At the elevation of the outlet nozzles (z = 331 mm, see Figure 57 and Figure 59) the recirculation is significantly reduced, the negative (downward) peak of axial velocity component at the top corner region of the geometry is reduced from around -0.4 m/s to about -0.1 m/s, the width of this region has also decreased. With the perforated plate the radial distribution of the axial velocity component reaches 0 m/s at R = 10-14 mm while without the plate the point of change is over R = 20 mm. Magnitude of the change is also moderate compared to the original design. Without the perforated flow distributor the axial component (V) increases from about -0.4 m/s up to over 0.7 m/s while with the new structure the maximum is around 0.2 m/s and the shape of the distribution is practically flat between R = 20 mm and R = 180 mm. Recirculation at the cylindrical wall between the inlet and outlet nozzles is completely removed by the application of the flow distributor. It is also visible that both investigated diameter and pitch sizes result in very similar velocity distributions in both measurement planes, and there is no significant difference due to these geometrical parameters.
Figure 56: Radial distributions of velocity components (U, V) and absolute value (L): measured and simulated (CFX) without perforated plate (original geometry); simulation results with perforated plate (PERF 7-11: diameter: 7 mm, pitch: 11 mm; PERF 12-19: diameter: 12 mm, pitch: 19 mm), Plane 3 (33.75°), z = 265 mm
Figure 57: Radial distributions of velocity components (U, V) and absolute value (L): measured and simulated (CFX) without perforated plate (original geometry); simulation results with perforated plate (PERF 7-11: diameter: 7 mm, pitch: 11 mm; PERF 12-19: diameter: 12 mm, pitch: 19 mm), Plane 3 (33.75°), z = 330 mm
Figure 58: Radial distributions of velocity components (U, V) and absolute value (L): measured and simulated (CFX) without perforated plate (original geometry); simulation results with perforated plate (PERF 7-11: diameter: 7 mm, pitch: 11 mm; PERF 12-19: diameter: 12 mm, pitch: 19 mm), Plane 4 (45°), z = 265 mm
Figure 59: Radial distributions of velocity components (U, V) and absolute value (L): measured and simulated (CFX) without perforated plate (original geometry); simulation results with perforated plate (PERF 7-11: diameter: 7 mm, pitch: 11 mm; PERF 12-19: diameter: 12 mm, pitch: 19 mm), Plane 4 (45°), z = 330 mm
Based on the preliminary design analysis I found that the application of such a perforated plate would significantly increase the uniformity of the flow in the modelled core domain. It suggests that the application of a similar flow distributor in the full scale molten salt reactor concept would also enhance the flow field in the core resulting in smaller temperature differences. Based on the presented preliminary calculations I designed a perforated plate that was manufactured (see Figure 60) and was placed in the mock-up in order to carry out measurements for validations purposes. For manufacturing reasons the final hole arrangement is slightly modified compared to the model applied in the preliminary calculations. The pitch was reduced to $p = 18.5$ mm in order to avoid manufacturing problems with peripheral holes being too close to the edge of the plate. Hole diameter was kept to be $d = 12$ mm.

![Figure 60: The manufactured perforated plate for PIV measurements; hole diameter: 12 mm, pitch: 18.5 mm](image)

### 7.2 Measurement and Simulation Results of the Modified Geometry

In order to investigate the effect of the installed perforated plate I carried out measurements of steady state, uniform inlet operation modes. Using the measurement results and applying the uncertainty estimation method described in Chapter 6 I also carried out a comprehensive CFD model development and validation.

For the measurements the design operational state was used setting the inlet flow rate to be 2.76 l/s ($\sim 10$ m$^3$/h) on all four inlets. Measurements were done along the length of the model tank from the top of the perforated plate ($z = 92$ mm) up to the top of the model in all four aforementioned measurement planes (see Figure 55). In this chapter I selected to present the results of measurements and simulations in Plane 2 (22.5°) and Plane 3 (33.75°).

#### 7.2.1 CFX Modelling of the Experiment with Modified Geometry

The CFX model was built using the original scaled and segmented geometry with the addition of the perforated plate region, modelling precisely the actual geometry of the set of perforations. Different volumetric meshes were generated in order to investigate the effect of mesh resolution and achieve mesh independency. Three different turbulence models were used in the comparison of the numerical simulations. The $k$-$\varepsilon$ turbulence model is a widely known and one of the most commonly used two-equation eddy-viscosity models [80]. The second model was the Shear Stress Transport (SST), also a two-equation model based on the $\omega$ frequency formulation [81]. In order to test the applicability of Reynolds-stress turbulence models the Baseline (BSL) Reynolds Stress model [54] was selected for the analyses.

In the CFX simulations walls were handled as no-slip boundaries. For inlets mass flow rate values were given at the cross section of the extended inlet nozzles, and for the outlets zero relative pressure was defined. The length of the inlet nozzles were extended to 1500 mm.
(the necessary minimum entry length) in order to achieve fully developed flow at the inlet cross section of the model tank (see Chapter 4.3.1 and Equation 10). The outlet nozzles were also extended (to 666 mm) in order to move the outlet surface far away enough from the outlet cross section of the model tank. This way it can be assumed that the closeness of the outlets does not cause convergence problems nor interfere with the actual flow field inside the tank. Other parts of the experimental setup (e.g. valves and further components of the loops, see Figure 27) were not included in the CFX models (see Figure 61).

The nozzle extensions were made by extruding the surface mesh of an internal cross section of the nozzles, in the actual direction of each given nozzle (see Figure 62). This way the resolution of the tetrahedral volumetric mesh was kept and the mesh in the extensions did not require impractically large number of additional volumetric elements.

Figure 61: Simulation domain and boundary conditions

Figure 62: Surface mesh representation and the extruded mesh in the nozzles
Volumetric meshes were generated in the same geometry in order to examine the effect of different resolutions used, starting from a very coarse mesh and an y+ around 30 to finer meshes and y+ about 10 and 1 for the finest meshes. The near wall resolution was intended to be kept for the whole geometry, however in the perforated region it was necessary to generate finer mesh structures, otherwise it was not possible to generate the mesh, or meshes without problems (e.g. negative volumes). Table 20 shows the main parameters of the five different volumetric meshes generated for the numerical analyses.

Table 20: Parameters of the different meshes applied in numerical modelling

<table>
<thead>
<tr>
<th></th>
<th>C1</th>
<th>C2</th>
<th>F</th>
<th>VF1</th>
<th>VF2</th>
</tr>
</thead>
<tbody>
<tr>
<td>y+, nozzle</td>
<td>30</td>
<td>30</td>
<td>10</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>y+, tank</td>
<td>30</td>
<td>30</td>
<td>10</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>y+, perforation</td>
<td>10</td>
<td>10</td>
<td>10</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>max. edge length [mm]; global</td>
<td>20</td>
<td>5</td>
<td>5</td>
<td>5</td>
<td>4</td>
</tr>
<tr>
<td>max. surface edge length [mm]; tank wall</td>
<td>10</td>
<td>5</td>
<td>5</td>
<td>5</td>
<td>4</td>
</tr>
<tr>
<td>max. surface edge length [mm]; perf. plate</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>num. of inflation layers; nozzles, tank</td>
<td>5</td>
<td>5</td>
<td>10</td>
<td>20</td>
<td>24</td>
</tr>
<tr>
<td>num. of inflation layers; nozzles, perf. plate</td>
<td>5</td>
<td>5</td>
<td>5</td>
<td>10</td>
<td>12</td>
</tr>
<tr>
<td>total number of volume elements</td>
<td>2 078 516</td>
<td>4 291 474</td>
<td>4 990 757</td>
<td>7 725 516</td>
<td>9 017 769</td>
</tr>
</tbody>
</table>

Figure 63 and Figure 64 show cross-sectional cut and surface mesh representations at different locations of the volumetric meshes used. One can see the improvement of the mesh in the core volume starting from mesh C1 and the refinement of the connection between the near wall inflated layers and the inner volume region.

Table 21 shows the application matrix of volumetric meshes and turbulence models. I selected the coarser meshes (mesh C1 and C2) to be used with k-ε and SST turbulence models only since the more sophisticated BSL Reynolds stress model requires proper volumetric resolution and fine resolution of the wall boundary regions. The SST model was selected to be used with all five meshes, and the BSL Reynolds stress turbulence model was applied with the three finest numerical grids only. Convergence criterion was to be to have the RMS residuals below $10^{-5}$. Comparison of the measurement data and the simulation results are presented here by comparing radial distribution of the two velocity components, namely the radial (U) and the axial (V) velocity component, extracted along radial monitor lines described previously (see Figure 33).

Table 21: Turbulence models applied in numerical modelling for different meshes

<table>
<thead>
<tr>
<th></th>
<th>C1</th>
<th>C2</th>
<th>F</th>
<th>VF1</th>
<th>VF2</th>
</tr>
</thead>
<tbody>
<tr>
<td>k-ε</td>
<td>k-ε</td>
<td>k-ε</td>
<td>k-ε</td>
<td>k-ε</td>
<td></td>
</tr>
<tr>
<td>SST</td>
<td>SST</td>
<td>SST</td>
<td>SST</td>
<td>SST</td>
<td></td>
</tr>
<tr>
<td>BSL</td>
<td>BSL</td>
<td>BSL</td>
<td>BSL</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Figure 63: Volumetric mesh in a vertical cross section of the core, different mesh resolution
Figure 64: Surface mesh on the core domain and at an outlet, different mesh resolution
7.2.2 Comparison of measurement data and CFX simulation results

Below I present selected simulation results compared to measurement data. For the presented results the investigated state was nominal operation, i.e. uniform, same ∼2.75 l/s (∼10 m³/h) inlet flow rate on all four inlets. Radial distribution of velocity components are extracted, measured data and simulation results are compared. Here measurement and simulation results from Plane 2 (22.5°) and Plane 3 (33.75°) are presented, data extracted from four monitor lines in these planes at four different elevations: z = 110 mm (right above the perforated plate), 180 mm and 280 mm (between the elevation of the inlet nozzles and the outlet nozzles) and at 330 mm, near the centreline of the outlet nozzles (see Figure 33). In both planes I will present the radial distribution of the radial (U) and axial (V) component at four different elevations. First I present the results of CFX simulations carried out using the k-ε turbulence model, while applying different volumetric meshes. Similar comparison is made for simulation results with the SST turbulence model and different volumetric meshes. As an illustration I also include the radial distributions of the velocity length (L) at two elevations (z = 180 mm and 280 mm) in Plane 2 (22.5°) compared with the k-ε turbulence model results, and at another two (z = 280 mm and 380 mm) in Plane 3 (33.75°) compared with the SST turbulence model results. In Figure 65-69, Figure 71-75 and Figure 76-79 measurement data are presented with error bars indicating the estimated absolute uncertainty (in [m/s]) based on the uncertainty analysis method presented in Chapter 6. Since I applied the same measurement planes in the region above the perforated plate, the modified geometry did not alter the illumination conditions or the general camera positioning. Therefore I could calculate standard deviation of the data points applying the same method. For better visualisation only every second measurement point is labelled with an error bar.

For quantitative comparison of the obtained results the two following metrics will be applied together.

In order to evaluate the extent of difference between the measurement results and the calculated values, [82, 83] proposed the application of the following M metric:

$$M = \frac{1}{N} \sum_{i=1,N} |C_i - D_i|$$

where N is the number of data pair compared, C_i is the i^{th} data point from a CFD calculation and D_i is the i^{th} measurement data point for a radial distribution of a velocity component. In this case index “i” will represent the radial position of the actual data point, and the sum M will represent an average deviation between the measured values and the calculated values along the radius for the given radial distributions. In my comparison I will use this metric with a modification that utilises that the measurement data is also provided with absolute uncertainty values. The modification applies the following cut-off:

$$|C_i - D_i| \equiv 0 \text{ if } |C_i - D_i| \leq \sigma_i$$

where σ_i is the absolute uncertainty of the i^{th} measurement point determined by the method presented in Chapter 6. This modified metric is going to be calculated for every radial distribution comparison for each data point pairs. In order to generate the necessary data point pairs the simulations results will be interpolated onto the actual physical locations of the measurement data points. This interpolation is necessary since the spatial distribution of the measurement data is determined by the size of the recorded area and the applied size of interrogation windows. However the spatial resolution of the simulation results is determined by the applied volumetric mesh combined with the method of post-processing. Therefore the actual locations of the data will not necessarily coincide.

In order to get a single index for each numerical simulation the metrics obtained for the radial velocity component (U) and the axial velocity component (V) will be combined as follows:

Bogdán Yamaji PhD thesis 85/118
\[ \Delta L = \sqrt{M_U^2 + M_V^2} \]  \hspace{1cm} (52)

where \( M_U \) and \( M_V \) is the modified metric for the two velocity components.

A second metric will also be used for quantitative comparison, with which the difference between the measured and the calculated local maximum of the radial distribution of the axial velocity component, normalized by the measured value is calculated:

\[ \Delta V_{\text{max}} = \frac{| C_{\text{max}} - D_{\text{max}} |}{D_{\text{max}}} \]  \hspace{1cm} (53)

where \( C_{\text{max}} \) is the local maximum of the calculated distribution and \( D_{\text{max}} \) is the local maximum of the measured distribution. With this metric the estimation of the peak values of the axial velocity component will be measured at elevation \( z = 180 \text{ mm}, 280 \text{ mm} \) and \( 330 \text{ mm} \) at the inner region of the geometry, near the corner of the vertical side walls of the model tank. Since the measurements did not cover the modelled region all the way up to the corner of the vertical walls due to the shadow of the overflow nozzle, this metric will give an evaluation of the estimated local peaks in the radial distributions of the axial velocity component given by the simulations.

Beside the detailed qualitative evaluation and graphic comparison of the measurement data and the calculation results the application of these two metrics will provide a quantitative assessment of the numerical models as well.

Based on the graphical comparison of the measured values and the values from simulations it is clear that the use of \( k-\varepsilon \) turbulence model suggests the application of mesh F while the use of the finest mesh VF1 applied with the \( k-\varepsilon \) model would not provide significantly better results. As it can be seen in Figure 65 and Figure 66 in Plane 2 (22.5°) there is a difference between the measured and calculated axial velocity component values in the inner region of the geometry, near the corner of the vertical side walls of the model tank. It is also clear that with the application of finer volumetric meshes (from C1 to F) this difference reduces at elevations \( z = 330 \text{ mm}, 280 \text{ mm} \) and \( 180 \text{ mm} \) (see Figure 65-66). Where the radial direction of the flow is dominant (e.g. at the elevation of the outlet nozzle) the radial component is fairly well reproduced by either simulation model. At lower elevations (Figure 65) it can be seen that the measured maximum values are generally underestimated by the simulations although the general shape of the radial distributions are well reproduced. Right above the perforated plate the location and width of the small jets formed at the perforations are well reproduced however the highest peak values (both for the radial and axial component) are clearly underestimated by the simulations. In Plane 3 (33.75°) (Figure 68-69) similar observations can be made. At \( z = 180 \text{ mm} \) the shape of the velocity component profiles are fairly well reproduced by the simulations however the local maximums are generally underestimated. At \( z = 280 \text{ mm} \) the same can be seen however the radial position second local maximum of the axial component at \( R = 138 \text{ mm} \) is shifted to \( R = 145 \text{ mm} \) in the simulations. At this elevation the profile of the radial component is reproduced by the simulations however the maximum value is under predicted (-0.15 m/s instead of -0.2 m/s). At \( z = 330 \text{ mm} \) the local maximum of the axial component in the inner region (\( R<100 \text{ mm} \)) is underestimated by each model, while the values and the shape of the profile is well reproduced by the simulations in the outer region (100 mm <\( R \)).

Values of the metrics of comparison for results with the \( k-\varepsilon \) turbulence model are presented in Table 22. In both planes results with mesh F provide better (lower) scores at elevation \( z = 110 \text{ mm} \) and \( z = 330 \text{ mm} \), while at \( z = 180 \text{ mm} \) and \( z = 280 \text{ mm} \) mesh C2 have better score, although at \( z = 280 \text{ mm} \) the margin of difference between the scores of mesh C2 and mesh F is very low. Similar can be found for the relative \( \Delta V_{\text{max}} \) metric, in most cases mesh F provide better values, and where mesh C2 produced a better score the margin of difference is extremely limited.
Figure 65: Radial (U) and axial (V) velocity components, model with perforated plate, measured (PERF) and CFX simulation results with k-ε turbulence model, different meshes, Plane 2 (22.5°), z=110 mm and z=180 mm.
Figure 66: Radial (U) and axial (V) velocity components, model with perforated plate, measured (PERF) and CFX simulation results with 
k-ε turbulence model, different meshes, Plane 2 (22.5°), z=280 mm and z=330 mm
Figure 67: Radial distribution of velocity length (L), model with perforated plate, measured (PERF) and CFX simulation results with k-ε turbulence model, different meshes, Plane 2 (22.5°), top: z=180 mm, bottom: z=280 mm
Figure 68: Radial (U) and axial (V) velocity components, model with perforated plate, measured (PERF) and CFX simulation results with k-ε turbulence model, different meshes, Plane 3 (33.75°), z=110 mm and z=180 mm
Figure 69: Radial (U) and axial (V) velocity components, model with perforated plate, measured (PERF) and CFX simulation results with k-ε turbulence model, different meshes, Plane 3 (33.75°), z=280 mm and z=330 mm
### Table 22: Metrics of comparison for results with k-ε turbulence model

<table>
<thead>
<tr>
<th>Plane 2 (22.5°)</th>
<th>mesh</th>
<th>U</th>
<th>V</th>
<th>ΔL</th>
<th>ΔV(_{\text{max}})</th>
</tr>
</thead>
<tbody>
<tr>
<td>z = 110 mm</td>
<td>C1</td>
<td>0.03708</td>
<td>0.18274</td>
<td>0.18646</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>C2</td>
<td><strong>0.03525</strong></td>
<td>0.16512</td>
<td>0.16884</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>F</td>
<td>0.03540</td>
<td><strong>0.16367</strong></td>
<td><strong>0.16746</strong></td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>VF1</td>
<td>0.04057</td>
<td>0.16484</td>
<td>0.16976</td>
<td>-</td>
</tr>
<tr>
<td>z = 180 mm</td>
<td>C1</td>
<td>0</td>
<td>0.08004</td>
<td>0.08004</td>
<td>0.28626</td>
</tr>
<tr>
<td></td>
<td>C2</td>
<td>0</td>
<td><strong>0.07093</strong></td>
<td><strong>0.07093</strong></td>
<td>0.19623</td>
</tr>
<tr>
<td></td>
<td>F</td>
<td>0</td>
<td>0.08534</td>
<td>0.08534</td>
<td><strong>0.17134</strong></td>
</tr>
<tr>
<td></td>
<td>VF1</td>
<td>0</td>
<td>0.09794</td>
<td>0.09794</td>
<td>0.24441</td>
</tr>
<tr>
<td>z = 280 mm</td>
<td>C1</td>
<td>0.00569</td>
<td>0.05260</td>
<td>0.05291</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>C2</td>
<td>0.00305</td>
<td><strong>0.04570</strong></td>
<td><strong>0.04581</strong></td>
<td>0.29539</td>
</tr>
<tr>
<td></td>
<td>F</td>
<td>0.00193</td>
<td>0.05042</td>
<td>0.05046</td>
<td><strong>0.28157</strong></td>
</tr>
<tr>
<td></td>
<td>VF1</td>
<td>0</td>
<td>0.06742</td>
<td>0.06742</td>
<td>0.35287</td>
</tr>
<tr>
<td>z = 330 mm</td>
<td>C1</td>
<td>0.00841</td>
<td>0.07275</td>
<td>0.07323</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>C2</td>
<td>0.00532</td>
<td>0.06840</td>
<td>0.06861</td>
<td><strong>0.41528</strong></td>
</tr>
<tr>
<td></td>
<td>F</td>
<td>0.00548</td>
<td><strong>0.06027</strong></td>
<td><strong>0.06052</strong></td>
<td>0.41943</td>
</tr>
<tr>
<td></td>
<td>VF1</td>
<td><strong>0.00479</strong></td>
<td>0.07460</td>
<td>0.07475</td>
<td>0.47720</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Plane 3 (33.75°)</th>
<th>mesh</th>
<th>U</th>
<th>V</th>
<th>ΔL</th>
<th>ΔV(_{\text{max}})</th>
</tr>
</thead>
<tbody>
<tr>
<td>z = 110 mm</td>
<td>C1</td>
<td>0.06052</td>
<td>0.17453</td>
<td>0.18473</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>C2</td>
<td><strong>0.05048</strong></td>
<td>0.16657</td>
<td>0.17406</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>F</td>
<td>0.05494</td>
<td><strong>0.16130</strong></td>
<td><strong>0.17041</strong></td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>VF1</td>
<td>0.06107</td>
<td>0.17417</td>
<td>0.18457</td>
<td>-</td>
</tr>
<tr>
<td>z = 180 mm</td>
<td>C1</td>
<td><strong>0.00443</strong></td>
<td>0.08653</td>
<td>0.08665</td>
<td>0.18270</td>
</tr>
<tr>
<td></td>
<td>C2</td>
<td>0.00453</td>
<td><strong>0.04175</strong></td>
<td><strong>0.04200</strong></td>
<td>0.06738</td>
</tr>
<tr>
<td></td>
<td>F</td>
<td>0.00461</td>
<td>0.06156</td>
<td>0.06173</td>
<td><strong>0.05680</strong></td>
</tr>
<tr>
<td></td>
<td>VF1</td>
<td>0.00538</td>
<td>0.08048</td>
<td>0.08067</td>
<td>0.13632</td>
</tr>
<tr>
<td>z = 280 mm</td>
<td>C1</td>
<td>0.01228</td>
<td>0.04561</td>
<td>0.04723</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>C2</td>
<td>0.00947</td>
<td><strong>0.03181</strong></td>
<td><strong>0.03319</strong></td>
<td>0.17013</td>
</tr>
<tr>
<td></td>
<td>F</td>
<td>0.00694</td>
<td>0.03922</td>
<td>0.03984</td>
<td><strong>0.16738</strong></td>
</tr>
<tr>
<td></td>
<td>VF1</td>
<td><strong>0.00691</strong></td>
<td>0.04876</td>
<td>0.04926</td>
<td>0.24349</td>
</tr>
<tr>
<td>z = 330 mm</td>
<td>C1</td>
<td>0.03471</td>
<td>0.04654</td>
<td>0.05806</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>C2</td>
<td><strong>0.02557</strong></td>
<td>0.06030</td>
<td>0.06550</td>
<td><strong>0.33307</strong></td>
</tr>
<tr>
<td></td>
<td>F</td>
<td>0.02706</td>
<td><strong>0.03755</strong></td>
<td><strong>0.04629</strong></td>
<td>0.35885</td>
</tr>
<tr>
<td></td>
<td>VF1</td>
<td>0.02805</td>
<td>0.04667</td>
<td>0.05445</td>
<td>0.41235</td>
</tr>
</tbody>
</table>

The application of the Shear Stress Transport (SST) turbulence model suggests the use of mesh VF1 or VF2, but if computational resources are also considered the use of the finest mesh is not necessary. As it can be seen in Figure 71 and Figure 72 in Plane 2 (22.5°) the general radial distributions of the two velocity components are qualitatively well reproduced by the simulations, however the local minimums and maximums are usually underpredicted. By looking at the axial component (V) at z = 180 mm, 280 mm and 330 mm the shape of the radial distributions are very close to each other (simulation and measurement) however at z = 280 mm the radial position of the second, outer local maximum is slightly shifted toward the cylindrical wall in the simulations. It is also visible that recirculation, downward flow between the inlet and outlet nozzles near the cylindrical wall is nearly eliminated, the simulations suggest a slight recirculation in the top corner of the geometry, which can be considered a modelling effect of the segmented geometry of the experimental mock-up. Once
again the distributions of the radial velocity component are mostly well reproduced with the maximum being underestimated at \( z = 280 \) mm (around -0.15 m/s instead of -0.2 m/s).

In Plane 3 (33.75° – see Figure 73-75) at \( z = 180 \) mm the radial distribution of the radial component is fairly well reproduced by the simulations, model with mesh VF1 being the best approximation. Right above the perforated plate (\( z = 110 \) mm) the simulations suggest a slight recirculation near the cylindrical wall (see Figure 70), however the extent of it is much smaller than without the perforated plate. The model with mesh VF1 overpredicts the component values after the local maximum at \( R = 150 \) mm. The shape of the axial component distribution is mainly well reproduced by the simulations, but maximum value of the outer local maximum (at \( R = 125 \) mm) is underestimated by all models. At higher elevation (\( z = 280 \) mm) the radial distributions of the two components are very well reproduced in the simulations however for the radial component the maximum at \( R = 140 \) mm is generally underestimated (-0.15 m/s instead of -0.2 m/s) and once again, the radial position of the outer local maximum in the axial component distribution is shifted toward the cylindrical wall by about 10 mm in case of the simulation results.

![Figure 70: Vector fields from CFX simulation with SST turbulence model, mesh VF1, Plane 2 (22.5° – left) and Plane 3 (33.75° – right)](image)

Values of the metrics of comparison for results with the SST turbulence model are presented in Table 23. In both planes the lowest scores are divided between results with mesh C2, F and VF1, providing better (lower) scores at elevation \( z = 110 \) mm and \( z = 330 \) mm, with very limited differences between the metric values. At \( z = 180 \) mm and \( z = 280 \) mm mesh C2 have better scores, although at \( z = 280 \) mm mesh VF1 and VF2 resulted in the lowest \( M_u \) values while the \( \Delta L \) and \( \Delta V_{max} \) scores put mesh C2 at the top. At \( z = 180 \) mm in both planes mesh C2 have the best scores for the difference metrics while for the maximum relative difference metric (\( \Delta V_{max} \)) are best for mesh VF2 (Plane 2) or mesh C2 and VF2 (Plane 3).
Figure 71: Radial (U) and axial (V) velocity components, model with perforated plate, measured (PERF) and CFX simulation results with SST turbulence model, different meshes, Plane 2 (22.5°), z=110 mm and z=180 mm
Figure 72: Radial (U) and axial (V) velocity components, model with perforated plate, measured (PERF) and CFX simulation results with SST turbulence model, different meshes, Plane 2 (22.5°), z=280 mm and z=330 mm
Figure 73: Radial (U) and axial (V) velocity components, model with perforated plate, measured (PERF) and CFX simulation results with SST turbulence model, different meshes, Plane 3 (33.75°), z=110 mm and z=180 mm
Figure 74: Radial (U) and axial (V) velocity components, model with perforated plate, measured (PERF) and CFX simulation results with SST turbulence model, different meshes, Plane 3 (33.75°), z=280 mm and z=330 mm
Figure 75: Radial distribution of velocity length (L), model with perforated plate, measured (PERF) and CFX simulation results with SST turbulence model, different meshes, Plane 2 (22.5°), top: z=280 mm, bottom: z=330 mm
In Figure 76-79 simulation results with BSL Reynolds stress turbulence model with three mesh resolutions (mesh F, VF1 and VF2) are compared to the measurement results and the simulation results of models using k-ε (mesh F) and SST (mesh VF1) turbulence model. The

<table>
<thead>
<tr>
<th>Plane 2 (22.5°)</th>
<th>mesh</th>
<th>M_U</th>
<th>M_V</th>
<th>ΔL</th>
<th>ΔV_{max}</th>
</tr>
</thead>
<tbody>
<tr>
<td>z = 110 mm</td>
<td>C1</td>
<td>0.03512</td>
<td>0.18591</td>
<td>0.18920</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>C2</td>
<td>0.03932</td>
<td>0.17576</td>
<td>0.18011</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>F</td>
<td>0.04460</td>
<td>0.17350</td>
<td>0.17915</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>VF1</td>
<td>0.03788</td>
<td>0.17586</td>
<td>0.17990</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>VF2</td>
<td>0.03390</td>
<td>0.17934</td>
<td>0.18253</td>
<td>-</td>
</tr>
<tr>
<td>z = 180 mm</td>
<td>C1</td>
<td>0.03932</td>
<td>0.17576</td>
<td>0.18011</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>C2</td>
<td>0.05683</td>
<td>0.17915</td>
<td>0.18253</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>F</td>
<td>0.03788</td>
<td>0.17586</td>
<td>0.17990</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>VF1</td>
<td>0.03932</td>
<td>0.17576</td>
<td>0.18011</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>VF2</td>
<td>0.03390</td>
<td>0.17934</td>
<td>0.18253</td>
<td>-</td>
</tr>
<tr>
<td>z = 280 mm</td>
<td>C1</td>
<td>0.04460</td>
<td>0.17350</td>
<td>0.17915</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>C2</td>
<td>0.05683</td>
<td>0.17915</td>
<td>0.18253</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>F</td>
<td>0.03788</td>
<td>0.17586</td>
<td>0.17990</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>VF1</td>
<td>0.03932</td>
<td>0.17576</td>
<td>0.18011</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>VF2</td>
<td>0.03390</td>
<td>0.17934</td>
<td>0.18253</td>
<td>-</td>
</tr>
<tr>
<td>z = 330 mm</td>
<td>C1</td>
<td>0.04460</td>
<td>0.17350</td>
<td>0.17915</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>C2</td>
<td>0.05683</td>
<td>0.17915</td>
<td>0.18253</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>F</td>
<td>0.03788</td>
<td>0.17586</td>
<td>0.17990</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>VF1</td>
<td>0.03932</td>
<td>0.17576</td>
<td>0.18011</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>VF2</td>
<td>0.03390</td>
<td>0.17934</td>
<td>0.18253</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 23: Metrics of comparison for results with SST turbulence model
latter two are presented here as the best estimation based on the previously presented comparisons.

It generally can be observed that the BSL model did not provide much better results than the best results of the simulations presented beforehand (SST turbulence model and mesh VF1). However it is also clear that the use of the SST or the BSL turbulence model will give better results than the use of k-ε model.

In Plane 2 (22.5°) at lower elevations (z = 110 mm and 180 mm – see Figure 76) the simulation results with BSL Reynolds stress turbulence model (with meshes F, VF1 and VF2) reproduce well the radial distributions of the radial and axial velocity component, however at z = 110 mm the local maximums are generally underestimated by the simulations. Radial position and jet width are fairly well reproduced in the simulations. At elevation z = 280 mm (see Figure 77) for the radial component the simulations well reproduced the radial distribution although the peak is systematically underestimated. However the difference between the measured and the simulated values is less than the error margin of the measured data. For the axial component same observations can be made as in the previous cases: general shape of the distribution is basically well reproduced by the simulations although the local minimum at R = 90 mm is underestimated, and the radial position of the second local maximum (at R = 150 mm) is slightly different between the measured data and the simulation results.

In Plane 3 (33.75° – see Figure 78-79) at z = 110 mm simulations generally gave similar results and while at some positions the velocity peaks are well reproduced, it can be said that the maximum values are mostly underpredicted. At z = 180 mm for both components the general shape of the radial distributions are fairly well reproduced by the simulations but the local maximum of the axial component is still underestimated by the computational models. Similar observations can be made in case of the comparisons of the distributions extracted at z = 280 mm and 330 mm. In case of the models with BSL Reynolds stress turbulence model the comparison of simulation results suggest that the finest volumetric mesh (VF2) did not provide better results, than the second finest (mesh VF1), and basically the three models gave results very close to each other.

By comparing the three turbulence models applied – considering the best results from the comparisons based on mesh resolution – the application of the k-ε turbulence model provided results least close to the measurement data. It also has to noticed that the largest differences can be seen at the inner region of the geometry (R < 80 mm), near the corner of the two vertical walls of the segmented geometry. The application of the more sophisticated, therefore more resource intensive BSL Reynolds stress model did not provide better results than the simulations with SST turbulence model. Considering that the use of SST turbulence model did not require the utilization of the finest volumetric mesh it can also be viewed as an advantage of the use of the SST model in the case of this problem.

Values of the metrics of comparison for results with the BSL Reynolds stress turbulence model are presented in Table 24. In both planes at most elevations the best (lowest) scores are given by the result with mesh F, with the exceptions of elevation z = 110 mm in Plane 2, where the lowest values correspond to mesh VF1, and elevation z = 280 mm and 330 mm in Plane3, where the lowest metrics are produced by the result with mesh VF1 and VF2.

Comparing the values in Table 21-23 it is clear that the lowest marks are provided by the result using the SST turbulence model or the BSL Reynolds stress model. Generally the use of the SST turbulence model would provide better score while applying a mesh with lower resolution, hence with less computational requirement. As a conclusion for the numerical investigation of the segmented model presented in my thesis, based on the previously presented comparisons I suggest the application of the Shear Stress turbulence model with the use of the appropriate spatial resolution, i.e. a fine volumetric mesh (in this case mesh VF1).
Figure 76: Radial (U) and axial (V) velocity components, model with perforated plate, measured (PERF) and CFX results with BSL turbulence model, different meshes compared to k-ε turbulence model/mesh F and SST turbulence model/mesh VF1, Plane 2 (22.5°), z = 110 mm and 180 mm
Figure 77: Radial (U) and axial (V) velocity components, model with perforated plate, measured (PERF) and CFX results with BSL turbulence model, different meshes compared to k-ε turbulence model/mesh F and SST turbulence model/mesh VF1, Plane 2 (22.5°), z = 280 mm and 330 mm
Figure 78: Radial (U) and axial (V) velocity components, model with perforated plate, measured (PERF) and CFX results with BSL turbulence model, different meshes compared with k-ε turbulence model/mesh F and SST turbulence model/mesh VF1, Plane 3 (33.75°), z=110 mm and 180 mm
Radial velocity component (U), with perforated plate, 33.75°, z=280 mm

Axial velocity component (V), with perforated plate, 33.75°, z=280 mm

Radial velocity component (U), with perforated plate, 33.75°, z=330 mm

Axial velocity component (V), with perforated plate, 33.75°, z=330 mm

Figure 79: Radial (U) and axial (V) velocity components, model with perforated plate, measured (PERF) and CFX results with BSL turbulence model, different meshes compared to k-ε turbulence model/mesh F and SST turbulence model/mesh VF1, Plane 3 (33.75°), z=280 mm and 330 mm
Considering the changes in the measurement results due to the modified geometry it is clear that the utilization of the perforated plate has practically eliminated the downward flow from the vertically central region of the core – between the inlet nozzles and the outlet nozzles (see Figure 65-66 and Figure 68-69). In the upper region (z = 280 mm and 330 mm) large radial differences of the axial component (see Figure 35-36) are removed by the application of the perforated plate. Although the radial profiles of the axial component did not become absolutely flat, the large difference between the inner region (R<60 mm) and the outer part is significantly reduced. With the application of the flow distributor plate there are two local maximums in the radial distributions of the axial component, with a typical maximum value of 0.3 m/s and 0.4 m/s at z = 330 mm and about 0.5 m/s at z = 280 mm, in Plane 2 (22.5°). In Plane 3 (33.75°) at the same elevations similar radial distribution and maximum values are measured. In these measurement planes and at these elevations the difference between the measured minimum and maximum values changed from 0.7-1 m/s (see Figure 35-36 and Figure 39-40) to about 0.2 m/s. At elevation z = 180 mm the radial distributions of V with one maximum between 0 mm < R < 40 mm in Plane 2 and 3 became distributions with two local maximums, one between 0 mm < R < 20 mm and one between 120 mm < R < 150 mm. Close to the cylindrical wall the measured values reach 0 m/s but only over R = 180 mm, while without the perforated plate this could have reached even at R = 140-160 mm (see Figure 34 and 36). The radial distributions are fundamentally different at z = 110 mm because of the presence of the perforated plate. The measurement data and the simulation results suggest that
the application of such a perforated plate as flow distributor would effectively modify the flow field in the core region of the investigated molten salt reactor concept. Reduction of large differences in the velocity profiles along the radius and the elimination of recirculation near the cylindrical wall are possible with such an internal structure.

7.2.3 Considerations and consequences on the MSFR concept

It has to be noted that the numerical modelling did not take into consideration neither other parts of a realistic MSFR coolant loop, namely pumps, heat exchangers, nor a possible loop layout with certain pipe lengths, elbows etc that could have an impact on flow behaviour at the inlets. The experimental model has closed loops with pumps, but obviously it did not include heat exchangers. It also can be suspected that a realistic loop layout in case of such a molten salt reactor would not be designed with fully developed flow at the inlets in mind; therefore the layout of a loop could be significantly different (for example shorter pipe lengths, external parts – pumps, etc – located close to the reactor vessel).

Focus of my investigation was to achieve more uniform velocity field in the core region, but other aspects could be considered as well. Suggestions for further experimental or experimental and numerical investigation of the optimization of the core flow of the MSFR may include the further optimization process to find a more suitable hole diameter and pitch, different layout of holes (i.e. triangular lattice), depending on the focus of the optimization, such as the achievement of thermo-dynamics optimum or the highest breeding ratio of the reactor concept. As an outlook it can be considered how each optimization goal could be fulfilled with different perforated plate designs, among which non-uniform, for example radially changing hole and pitch values may also be considered. The application of the proposed perforated distributor plate may also provide the possibility to relatively easily investigate the aforementioned problems, without having to modify the geometry of the MSFR core vessel and the layout of the inlet and outlet nozzles. Different perforated plate thickness or even the optimization of the perforated plate volume porosity could also be considered. It also may be necessary to investigate, what other implications would such a perforated plate have in terms of neutronics: considering applicable materials (steel, graphite, zirconium alloys, etc that are compatible with the proposed or applied molten salts) what thickness would best fit with reactor physics, with structural strength also taken into account. The material composition and volume ratios in the core (with the addition of a flow distributor plate) might need further evaluation as well.

It also needs to be considered, that the original MSFR concept does not include any internal structures inside the core, not even safety control rods. Safety shutdown of the reactor is provided by the draining of the core through a bottom nozzle by the melting of a freeze plug [5, 84]. It seems to be evident that with the addition of a flow distributor perforated plate (with optimized hole and pitch size, thickness and porosity) the transient behaviour of the core during SCRAM would be significantly different from the behaviour of the original concept, and it has to be investigated whether any additional safety or fast shutdown system would be necessary.
8 SUMMARY

First aim of my thesis was to design and construct an experimental model with which experimental thermal-hydraulics modelling and investigation the homogeneous single region Molten Salt Fast Reactor (MSFR) concept would be possible. With this in mind I investigated the feasibility of experimental modelling of the MSFR concept using water as substitute working fluid. I had to consider modelling issues such as scaling and segmenting the geometry, the modelling and measurement possibilities, requirements and constraints as well. By taking into account these considerations I designed and built a scaled and segmented experimental model of the MSFR based on the reference design, which is suitable for Particle Image Velocimetry (PIV) measurements. With this non-intrusive optical fluid flow measurement method it was possible to carry out multiple measurement series on the experimental mock-up. As a second objective results of the measurements made it possible to understand the basic thermal-hydraulics behaviour of the concept. With the design and addition of a perforated flow distributor plate I also successfully examined the possibility of improving the core geometry, thereby enhancing the flow field in the core region.

The experimental results not only allowed me to draw conclusions concerning the thermal-hydraulics behaviour of MSFR, but following the completion of a comprehensive uncertainty analysis of the measurement system and the experimental setup, it was possible to compare the collected experimental data with results of Computational Fluid Dynamics (CFD) analyses.

With the computational model of the modified core geometry (with the addition of the perforated flow distributor plate) I investigated the thermal-hydraulics behaviour of the MSFR concept, and also proposed certain modifications in order to enhance the design of the concept.

Throughout the whole work I also continuously optimised and refined the PIV measurement system and the experimental setup itself in order to enhance measurement results, improve the measurement procedure. It included the application of black paint on surfaces that did not need to be transparent, which helped to decrease unnecessary reflection of the illuminating laser light. The design and installation of the precision camera holder system and the positioning frame of the laser light sheet head significantly improved the applicability of the PIV system, and made it much easier to reproduce the measurements.

In my dissertation first I presented a short summary on the history of molten salt reactors and gave a brief description of some molten salt reactor concepts proposed by the international research community. Following that I summarised the most important physical properties of applied and proposed molten salts together with the properties of structural materials. Knowledge of such properties is essential for experimental and numerical modelling of molten salt reactors. In the following chapter I presented a detailed description of the investigated molten salt reactor concept and its features, and discussed the possibilities and constraints of the experimental modelling of the selected molten salt reactor concept. As a part of this I presented the results of extensive preliminary numerical investigation using Computational Fluid Dynamics (CFD). Based on the results of these analyses and considerations I presented the design based on the findings of the aforementioned analyses and construction of the actual experimental mock-up. It was followed by a detailed breakdown of selected results of an extensive measurement series of steady state operation conditions, and the discussion of the thermal-hydraulic characteristics of the proposed molten salt reactor concept. Together with the measurement results I presented a comprehensive uncertainty analysis of the measurement system and the experimental setup, which is essential for proper evaluation. Following that I proposed a certain modification of the experimental
geometry that allowed me to experimentally investigate the feasibility to enhance the thermal-hydraulic characteristics of the proposed concept, and also presented and discussed the measurement results of the modified geometry. With the help of results of the measurements and the comparison with the results of numerical modelling it became possible to evaluate the optimal CFD modelling parameters of the experimental mock-up. It also gave an opportunity to make propositions on further enhancement and optimisation of the full-scale MSFR core, too.

New results of my original research presented in this thesis can be summarized as follows:

1. I proved that for the thermal-hydraulics investigation of a homogeneous molten salt reactor concept core an appropriately scaled and segmented experimental model is suitable. I designed and built a unique scaled and segmented experimental model of the MSFR molten salt reactor concept to experimentally investigate and comprehend the fundamental thermal-hydraulics behaviour of MSFR. I designed and built the experimental model to be suitable for Particle Image Velocimetry (PIV) measurement technique, and through measurements I proved that the experimental model meets its goals and it works in the intended range of modelled core Reynolds number \( \text{Re} = (1.1 – 1.9) \times 10^5 \) \( \{1, 5, 7, 8\} \).

2. I was the first to experimentally prove that the original simple cylindrical core geometry of MSFR concept was not optimal in terms of thermal-hydraulics and, indirectly, neutronics. As I identified through a comprehensive series of measurements, the reason was that the flow separated into two main domains in the modelled region, namely a high velocity jet in the inner region, going from the inlets toward the outlets, and a stagnating, recirculation region of very low velocity near the cylindrical wall between the inlet and outlet nozzles. As a result large temperature differences can be expected in the MSFR core, which could lead to significant disadvantageous differences in burn-up and reactivity \( \{2, 5, 6\} \).

3. Through measurements performed in a wide range of Reynolds numbers I found that by increasing Reynolds number the flow behaviour is similar and the observed flow separation is even stronger at higher Reynolds numbers, thus I confirmed that similar separation should be expected in real MSFR operation conditions \( \{2\} \).

4. Taking into account the specific properties of the complex of the scaled and segmented experimental mock-up and the PIV measurement system I modified a general uncertainty analysis method to perform comprehensive uncertainty analysis, and determined the total combined uncertainty of the measurement, which ranges between 5% and 35% for the absolute value of velocity, only exceeding this range at very low velocities close to the cylindrical wall of the original model geometry \( \{3\} \).

5. In order to reduce the non-uniformities in the flow field of the core region, applying results of CFD analyses I proposed a flow distributor structure. Accordingly in order to achieve the objective a perforated plate has to be placed right above the inlet nozzles. By a thorough series of measurements I proved that such a perforated plate with the appropriate hole and pitch size was a feasible solution, as it significantly reduced radial differences in velocity profiles and the extent of recirculation areas in the modelled core region \( \{3, 4\} \).

6. I created an optimised numerical (CFD) model of the modified experimental geometry, and after comparison with measurement data I concluded, that in steady state calculations the best agreement between the experimental and numerical results can be obtained by the application of the Shear Stress Transport turbulence model and a mesh resolution that provides \( y+ = 10 \) or better (lower) and – where applicable – the global edge length is not more than 1.5% of the total extent of the geometry \( \{4\} \).
9 PUBLICATIONS OF THE AUTHOR


{5} Yamaji Bogdán, Aszódi Attila: Sóolvadékos reaktorkoncepció kísérleti vizsgálata, Nukleon, VII. 2. (2014) 161 (Experimental investigation of a molten salt reactor concept – In Hungarian)

{6} Bogdán Yamaji, Attila Aszódi: Experimental modelling and numerical analysis of a molten salt fast reactor, PHYSOR 2014 International Conference, 1106911, Sep 28 – Oct 3, 2014, Kyoto, Japan

{7} Bogdán Yamaji, Attila Aszódi: Experimental modelling of a molten salt reactor concept, 21st International Conference on Nuclear Engineering ICONE 21, July 29-August 2, 2013, Chengdu, China, ICONE21-16783

{8} Bogdán Yamaji, Attila Aszódi: Preliminary thermal-hydraulic analyses for designing an experimental model of a molten salt reactor concept, 21st International Conference on Nuclear Engineering ICONE 21, July 29-August 2, 2013, Chengdu, China, ICONE21-16592

{9} Yamaji Bogdán: Sóolvadékos reaktorkoncepció kísérleti termohidraulikai vizsgálatát megalapozó előzetes háromdimenziós CFD számítások, stacionárius állapotok kísérleti vizsgálata PIV módszerrel az MSFR kismint kísérleti berendezésen, kutatási jelentés, BME-NTI-691/2014 (Preliminary CFD calculations for the support of experimental investigation of a molten salt reactor concept, PIV investigation of steady state operation of the scaled MSFR model, Research Report) (in Hungarian)

{10} Yamaji Bogdán: Az MSFR reaktormodell kisminta kísérlet mérésibizonytalanság-elemzése (uncertainty analysis), újabb mérési eredmények, kutatási jelentés, BME-NTI-736/2015 (Uncertainty analysis of the MSFR scaled experimental model, new measurement results, Research Report) (in Hungarian)

{11} Allibert et al., Yamaji et al.: Recommendations for an MSFR demonstrator pre-conceptual design, Evaluation and Viability of Liquid Fuel Fast Reactor System EVOL, Deliverable D2.9, 01/17/2014


{13} Brovchenko et al., Yamaji et al.: Optimization of the pre-conceptual design of the MSFR, Evaluation and Viability of Liquid Fuel Fast Reactor System EVOL, Deliverable D2.2, 09/24/2013

10 ACKNOWLEDGEMENTS

I dedicate my thesis to my parents who always supported me and provided endless encouragement throughout my studies, job and the preparation of this work.

First I would like to thank Professor Dr. Gyula Csom, without whom there would not be molten salt reactor research at the Institute of Nuclear Techniques, and Professor Dr. Attila Aszódi, whose continuous, thorough and always motivating supervision made it possible to accomplish this work. I also have to be grateful for them for being my supervisors since my undergraduate years, for directing my interest toward molten salt reactors, thermal-hydraulics and computational fluid dynamics, and for the continuous help and instruction during my years as a student and as a member of the NTI staff.

I have to express my thankfulness to the late Dr. Zoltán Pandula (BME Department of Hydrodynamic Systems), whose help and practical advice was essential during the design phase of the experimental model, for the assistance with selecting the proper components and testing them at the HDS laboratory.

I am very thankful for the practical help of my colleagues at the NTI workshop: Gábor Szakács, Endre Gyimóthy, the late Géza Fekete, who helped me learn the crafts of copper soldering, working with metal and plexiglas parts and initially assisted in the assembly of the experimental mock-up.

I am grateful to my colleague Dr. Máté Szieberth whose persistence made it possible for the Institute and for me to participate in several international research projects focusing on the investigation of molten salt reactors.

I would like to thank my colleague Ildikó Boros, without whom there would not be Particle Image Velocimetry measurement system at our Institute.

I also would like to thank my former colleague at BME NTI, Dr. László Balázs (ELTE Department of Geophysics and Space Science) for his invaluable suggestions and advice on the uncertainty analysis of measurements.

I am thankful to my colleague Máté Kovács, whose basic training and help with the initial design allowed me to become able to use a three-dimensional CAD tool that was essential for the design and manufacturing of the experimental model.

I also would like to thank all my colleagues and students at BME NTI who in any way supported me in my work with their help, patience or with any kind of contribution.
11 REFERENCES


prospects and challenges); ЭНЕРГОATOMIZDAT, Москва, 1990. ISBN 5-283-03791-6 (in Russian)


[22] H. G. MacPherson: Molten-salt reactor program quarterly progress report for period ending July 31, 1959, ORNL-2799; Oak Ridge National Laboratory, 1959


[26] Torklep, Oye: Viscosity of the Eutectic LiF-NaF-KF Melt (FLINAK), Journal of Chemical and Engineering Data, 25, 1980, pp. 16-17


[28] Bardet, Peterson: Options for scaled experiments for high temperature liquid salt and helium fluid mechanics and convective heat transfer, Nuclear Technology, 163, 2008, pp. 344-357


[37] Bogdán Yamaji, Dr. Gyula Csom, Dr Attila Aszódi: Three-dimensional thermalhydraulic analysis of molten salt reactor concepts; Proceedings of the eleventh International Conference on Nuclear Engineering (ICONE-11), April 20-23, 2003, Tokyo, Japan, JSME-ASME No. 03-209


[41] B. Kiss, I. Boros, A. Aszódi: Recent results of three-dimensional CFD simulations of coolant mixing in VVER-440/213 reactor pressure vessel, 18th Symposium of AER on VVER Reactor Physics and Reactor Safety, Eger, Hungary, October 06-10, 2008


[50] Litron Lasers: Lamp Pumped lasers for PIV Applications from Litron, PB0101:3, 2010


10. Recommended Procedures and Guidelines, Uncertainty Analysis Particle Imaging Velocimetry, 7.5-01-03-03, Specialist Committee on Uncertainty Analysis of 25th International Towing Tank Conference (ITTC), 2008

[74] Szatmáry Zoltán: Mérések kiértékelése, egyetemi jegyzet, (Evaluation of measurements, lecture textbook), BME TTK, Budapest, 2010 (in Hungarian)


[76] Dr. Dániel Oszetzy: personal communication, Hungarian Academy of Sciences Wigner Research Centre for Physics, 2015. 03. 02.


12 APPENDIX

12.1 MANUFACTURING AND CONSTRUCTION OF THE EXPERIMENTAL MODEL

Figure A1: Copper parts soldering in the workshop of BME NTI

Figure A2: Manufacturing and gluing the plexiglas flanges in the workshop of BME NTI
Figure A3: Assembling the experimental model in the PIV laboratory

12.2 Measurements on the Experimental Model

Figure A4: Left: Data acquisition monitor, right: green laser illuminates the experimental tank
Figure A5: Left: Mobile laser platform, right: precision camera positioning frame

Figure A6: Left: Laser head positioning frame, right: light sheet collimator on the model tank

Figure A7: Installation of the perforated plate; filled model with the perforated plate