MŰEGYETEM 1782

BUDAPEST UNIVERSITY OF THECHNOLOGY AND ECONOMICS

Institute of Nuclear Techniques

Investigations of thermal hydraulics of ALLEGRO fuel assemblies

Booklet of the PhD thesis

Gergely Imre Orosz

Supervisor: Prof. Dr. Attila Aszódi

Budapest

2023

1. INTRODUCTION

Gas-cooled Fast Reactor (GFR) is one of the generation IV nuclear reactor concepts selected by the Generation IV International Forum (GIF) for further analysis and implementation [1]. The helium coolant's core outlet temperature is up to 850 °C. With this high temperature condition, the GFR power plant can reach thermal efficiency up to 43-48% [2]. The introduction of these reactor types into the nuclear power plant fleet would bring huge economic and social benefits.

GFR core has approximately one order of magnitude higher power density (~100 MW/m^3) than the historical gas-cooled thermal reactors (5-10 MW/m^3) [3]. Because of the big power density and the lack of the thermal inertia the reliable heat removal from the core is a key issue of the GFR design.

Before the construction of a larger scale GFR of 2400 MWth, a reactor must be built that will prove the viability of the concept [4] [5]. In order to achieve the above goals, construction of an experimental gas-cooled fast reactor of 75 MWth is planned. The design thermal power of the ALLEGRO test reactor is $75MW_{th}$, but the power density of the core is 92 MW/m^3, which is high enough to investigate the performance of a GFR reactor.

The ALLEGRO reactor will demonstrate the concept and help to develop the required technologies. The main technical challenges include the fuel handling machine, decay heat removal heat exchangers, refractory fuel assemblies, helium purification technologies, etc. The ALLEGRO reactor design is currently being developed in an EU project called SafeG [6].

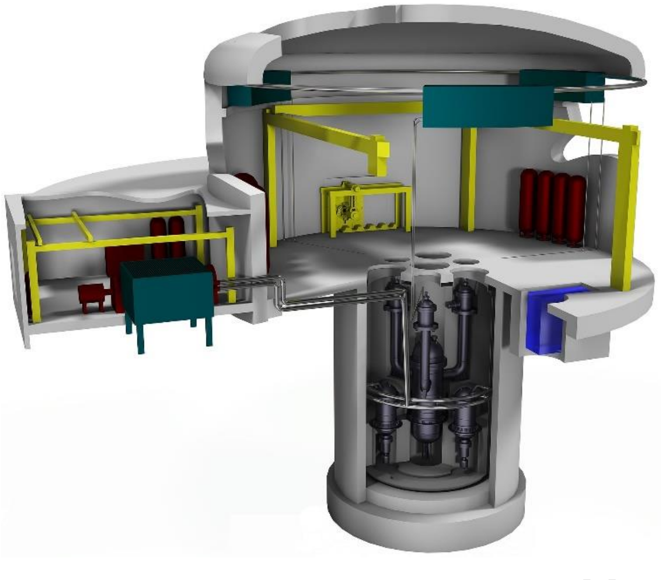

Figure 1 : GFR Demonstrator ALLEGRO [6].

2. OBJECTIVES

The aim of this PhD dissertation is to give as accurate description as possible of the thermal hydraulics processes, that take place in the ALLEGRO rector's ceramic fuel assembly.

In line with the plans, the initial core of ALLEGRO is set up with MOX (Mixed-OXide) fuel assemblies to investigate the performance of the main additional equipment (such as the heat exchangers and blowers). In a later stage of operation, a number of experimental ceramic fuel assemblies is planned to be loaded into the core in order to investigate the effects of irradiation. The MOX assemblies will work at lower temperature (530 °C average core outlet temperature).

After this starting phase, temperature resistant refractory (ceramic) assemblies will be used in the demonstration core at similar coolant temperature range as in the GFR concept, which requires the application of innovative UC-PuC fuel and SiC composite cladding [2] [7]. These materials are technically considered ceramics, which is why the assemblies are called ceramic assemblies.

My dissertation deals with the challenges of the thermal hydraulics of the aforementioned ceramic fuel assembly.

I have used two different methodologies for this, 3D Computational Fluid Dynamics (CFD) models and real physical measurements applying Particle Image Velocimetry (PIV). Both methodologies can provide relevant results separately, but together I can also confirm their validity.

As a first step, it is necessary to understand the limitations of CFD modelling and to verify the reliability of the CFD models built for the ALLEGRO assembly by validation methods. The validated models can be used to find the hot spots of the assembly. In the meantime, I can get an accurate picture of the different heat transfer processes in the assembly to support further design work.

Hot spots are limiting factors for safe operation, so my aim is to find a way to reduce the thermal maximum of the assembly without reducing the reactor power. By improving the heat transfer, the thermal maximum can be reduced. A known solution to increase the heat transfer in a nuclear fuel assembly is the use of so-called spacer grids with mixing vanes. There has been no known precedent for the usage of these mixing vanes in gas-cooled reactors. My goal is to develop a viable Mixing Vane with spacer Grid (MVG), in the ALLEGRO reactor using CFD methods.

As the field is completely unique, I needed a series of validation experiments to verify my CFD results. For validation, I developed an experimental test facility to demonstrate the effectiveness of the mixing vanes I have designed. The experimental device can be used not only for testing the ceramic assembly of the ALLEGRO reactor, but also for testing other types of assemblies with triangular grid spacing. The experimental test facility is called PROUETTE (PIv ROd bUndlE Test faciliTy at bmE). I have used optical based PIV (Particle Image Velocimetry) methodology to investigate the flow processes in rod bundles.

3. INVESTIGATION METHODOLOGY

It was presented in the previous Chapter that the ALLEGRO reactor has very special thermal hydraulics conditions. In the field of nuclear energy there are several possibilities to model or simulate thermal hydraulics process in fuel assemblies. The easiest and most commonly used method is computer simulation. Depending on how detailed we want to describe the chosen problem, we can create coarser system-code models, or if we want to examine processes in more details, we have to limit the investigation domain to a smaller segment of the facility and a high-resolution 3D CFD (Computational Fluid Dynamics) simulation model can be used. In my case, the goal was to describe the fuel assembly as accurately as possible, for which I have developed several 3D models using the ANSYS CFX code.

CFD codes can provide fast results for fluid flow conditions that we would not be able to measure in reality. These codes and software packages are generally well validated, but in some special cases it is necessary to confirm the results. In the nuclear sector, it is particularly important to decide to what extent the calculated results cover the real physical processes. To do this, a measurement method with a resolution close to the CFD codes discretisation must be used. Fortunately, the LDA (Laser Doppler Anemometry) and PIV (Particle Image Velocimetry) measurement techniques allow extremely detailed flow measurements in a non-intrusive manner. At the BME NTI, we have a PIV measurement system at our disposal, with which it was possible to perform such validation measurements.

3.1. Computational Fluid Dynamic approach

The Computational Fluid Dynamics (CFD) summarises all the methodologies, for the numerical calculation of flow and related transport processes (heat, mass). Therefore, CFD is an interdisciplinary scientific method. Practitioners of the discipline simulate flows in complex geometries with a precision that most of the flow experiments would not be able to reproduce. Another advantage over the experimental method is that there is no need to design expensive and time-consuming experiments.

Nowadays, the results produced by a carefully designed CFD simulation are considered to be equivalent to actual physical measurements. The primary reasons for this have been the dramatic increase in the computational speed of computers and the development of widely validated universal CFD codes. But I should mention that in the case of new a physical phenomenon, complex geometries, special media or topics of high safety relevance, it is particularly appropriate to verify and validate simulation models and methods, which require experimental results at a resolution comparable to that of the CFD method.

Most CFD codes use the Finite Volume Method (FVM). The method consists in decomposing the part of space (domain) under study into a finite number of cells of finite sizes. These cells form a so-called mesh. When creating a mesh, distinction is made between surface and volume elements.

The mesh generator software commonly prefers the following discretization elements: triangle and quadrilateral elements cover the surfaces, while in the inner regions tetrahedron, hexahedron, pyramid or prism type of elements are usually used. My most detailed CFD models (see in Chapter 6 of my dissertation) consisted of more than 197 million elements. An example of my model discretisation can be seen on Figure 2.

For the resulting cells, the codes iteratively solve the key conservation equations of hydrodynamics (heat, mass, and momentum). Iterative computations are very resource-intensive, so simulations are often performed on a high-performance cluster computer. Most of the real-life applications are based on turbulent flows, for which special turbulence models are applied in CFD simulations. These turbulence models are simplified constitutive equations that predict the statistical evolution of turbulent flows. Without these models, steady-state calculations would also be very time consuming. However, it should be noted that turbulence models are not universal, but rather problem-specific. Their uncertainties should be assessed and considered. In my research, I used several turbulence models, which are described in the respective chapters. The applications were based on the many years of experience of the CFD research group of Institute of Nuclear Techniques and the Best Practice Guidelines [8] used in the nuclear industry.

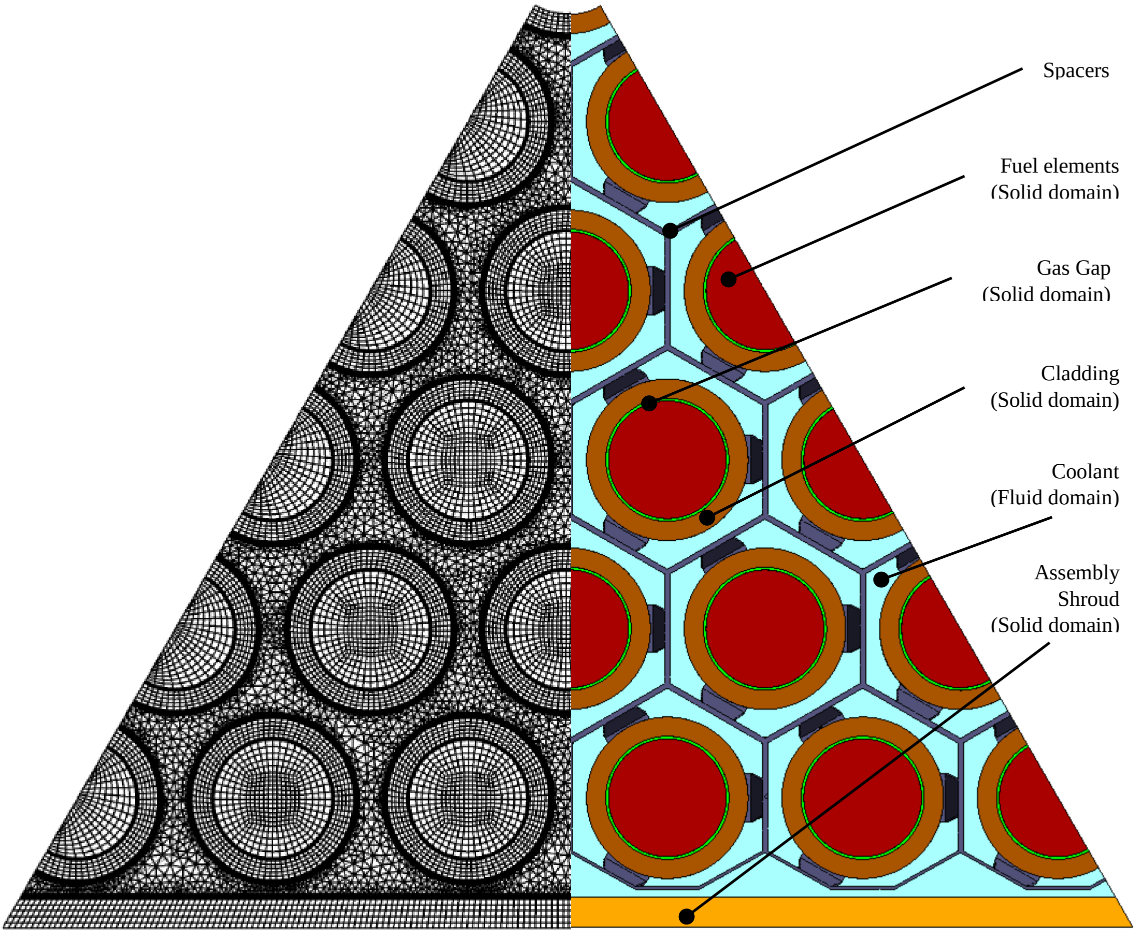

Figure 2: Mesh resolution at the outlet and the composition of the OS2_CMPL CFD model of the ALLEGRO GFR fuel assembly

3.2.Particle Image Velocimetry approach

Particle Image Velocimetry (PIV) is an optical measurement technique for the non-invasive analysis of flows, which has been under development since the 1980s [9]. By the end of the 1990s, the technique was becoming more widespread in basic research and industrial applications, partly due to the replacement of photographic image capture by the possibility of video recording and the proliferation of electronic and then digital cameras.

At the same time, the necessary computing facilities were developing at a rapid pace, enabling the capture of large quantities of digital images and their increasingly rapid processing and analysis. An important element of optical metrology is the illuminating light source, and as the size of lasers has decreased and their parameters have improved, PIV systems have become increasingly applicable to a wider range of applications.

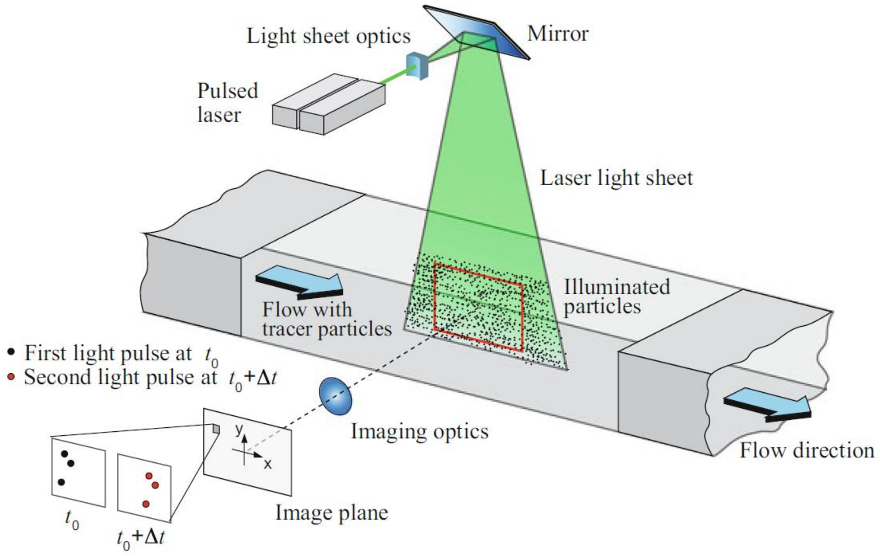

Figure 3: Experimental arrangement for planar 2D PIV in a wind tunnel [9]

The PIV measurement technique is based on recording the displacement of the tracer particles mixed with the flowing medium and moving with it, and inferring the velocity distribution in the flow. The flowing medium can be gas, water or other optically transparent materials. Optimally, the tracer particle should have a similar density as the flowing medium, and its size and shape should ensure good flow tracking and good reflectivity. Particles moving with the flow are illuminated at two successive time instants, the light scattered by the particles is recorded (i.e. photographed), and the direction and extent of movement of the identical particles identified in the two images is determined. Using appropriate mathematical procedures, this grain identification can be solved and, knowing the time elapsed between the two images, the flow velocity field can be determined. With proper calibration, the true physical content of the image displacement, i.e. the pixel-to-metre conversion, can be determined, and the 2D physical velocity field can be calculated from the picture pairs captured by one camera.

During my PhD research I have designed and constructed a measurement system called PIROUETTE, which is described in detail in Chapter 7.1 of the PhD dissertation and can be seen in Figure 4. The measurement system is designed to circulate water at isothermal conditions. There is a one-meter-long test section in the facility with a seven pin rod bundle simulating the flow conditions in the ALLEGRO core. The flow conditions of the water is set to achieve Reynolds-numbers similar to the core conditions of ALLEGRO in nominal state, allowing the investigation of the flow in the rod bundle and around the spacer grids, important for the design of the reactor core.

The water flow is provided by the main centrifugal pump. The fine control of the volumetric flow rate was controlled by a valve, followed by three ultrasonic flowmeters. Multiple volumetric flow meters can increase the accuracy of the volumetric flow measurement, which is very important for setting the inlet boundary condition for CFD calculations.

A flow straightener is necessary to reduce disturbances caused by mechanical, measuring and pipe lining equipment. The 1-meter-long seven-rod bundle was installed in the test channel section. A removable roof has been designed on the test channel section for ease of access. This is necessary, to allow the change of the rod geometry, for example different type of spacer grids and mixing vanes. In the test section, the rod bundle is made of FEP (Fluorinated Ethylene Propylene) to meet the MIR (Matching Index of Refraction) criteria. The FEP polymer has a refractive index of 1.33 which is nearly the same as the refractive index of the working medium (water).

The spacers were designed according to the ALLEGRO GFR assembly spacers, with slight modifications to fit the measurement requirements. The wall thickness of the spacers is 0.8 mm. The grids were made by a high-resolution 3D printing method (SLA) with special rigid composite resin. Thanks to 3D printing, spacer geometries can be easily varied and extended with different types of mixing vane (e.g. spacer grid without vane, spacers with SPLIT and TWISTED type mixing vanes). Preventing deformation of MVGs is a complex task. It is necessary to keep the working fluid temperature low with a heat exchanger unit (~30 °C) and prevent the MVG material from coming into chemical interaction with the working medium.

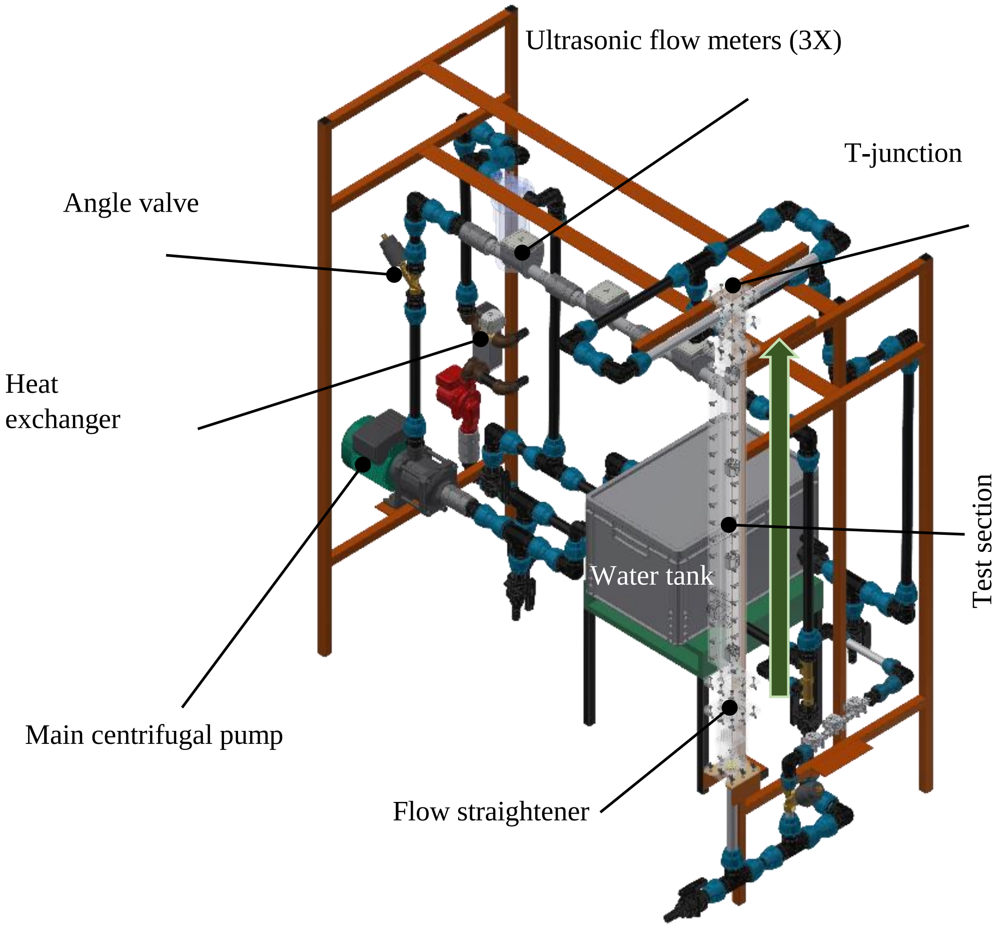

Figure 4: The structure of the PIROUETTE facility

4. THESIS STATEMENTS

I. **I developed a validated CFD model for the L-STAR benchmark facility, based on which I proved that radiative heat transfer in the ALLEGRO ceramic assembly is a significant physical process under normal operating conditions and should be modelled. Compared to the total thermal power of the rod, the radiative power varies from 0.71% to 9.7% for different rod positions** [P1],[P2],[P3].

II. I have developed a validated CFD model of the full cross-section of the ceramic assembly of the ALLEGRO gas-cooled fast reactor, describing all the structural elements of the active section of the assembly, their heat conduction and the heat transfer in the coolant and the gas gap. I **have found that in order to obtain more realistic model results, it is necessary to describe the heat conduction in the structural elements, which reduces the temperature maximum of the fuel assembly**. **I have identified that the large temperature differences between different subchannel types are caused by the lack of proper coolant mixing between the subchannels.** The highest rod cladding temperature is found in the corner subchannel [P1],[P2],[P3].

III. I have created a validated CFD model for the study of the mixing vanes in the ceramic assembly of the ALLEGRO gas-cooled fast reactor. In order to find the optimal geometry of the vanes, I have introduced dimensionless parameters. **By comparing the results of several types of mixing vanes, I concluded that the TWISTED type mixing vane is the most suitable for improving the temperature distribution of the assembly under the conditions typical for the ALLEGRO gas-cooled fast reactor.** I performed a sensitivity study to determine the optimal tilt angle for the TWISTED mixing vane, using my validated CFD models. **As a result of the study, I found that the optimum tilt angle of the TWISTED mixing vane is 30° under conditions typical of the ALLEGRO ceramic assembly** [P4].

IV. **For assemblies with a triangular grid lattice, a CFD specific method was developed to take into account the directional dependence of the coolant mixing between subchannels, as follows:**

$$Transverse\ flow_x = \frac{1}{s}\int \frac{v_{cross}}{w_{bulk}}\,\mathrm{d}y$$

where:

s – gap length between the subchannels [m],

v_{cross} – transversal velocity component [m/s],

w_{bulk} – bulk velocity of the coolant [m/s].

Thus, I have made the axial longitudinal variation of direction-dependent mixing function, which is particularly useful for the investigation and comparison of the secondary flows induced by different mixing vanes [P4].

V. I have developed a PIV experimental facility and several CFD models to investigate the effect of optimized TWISTED-type mixing vanes on the full length of the ALLEGRO gas-cooled fast reactor assembly. **By experimental and CFD studies, I have shown**

that the unwanted high temperatures of the corner rods can be avoided by using properly arranged mixing vanes. It is essential that the mixing vanes on both sides of the corner rod are not mirror symmetric, but rotated by 60°. I have shown that mirror-symmetrically positioned mixing vanes can increase the maximum temperature of the bundle at the hot spot by 15 °C, while in the 60° rotated arrangement they can reduce it by 29 °C [P5] [P4].

VI. **I have designed and constructed an isothermal experimental facility with a 7-rod triangular rod bundle test channel, in which the rods are made exclusively of FEP material and the spacer grids are 3D printed, so that the effect of different spacer grid geometries on the flow can be investigated with PIV measurements. I have determined that the experimental setup should be placed vertically to minimize creep of the FEP rods and distortion of the 3D printed special spacer grid geometry.** The facility was designed and constructed with the purpose to produce high resolution flow data for CFD model validation of triangular rod bundles [P5].

VII. **I have demonstrated by measurements that the TWISTED-type mixing vane with 30° tilt angle is capable of generating the highest transverse velocity components in the region behind the spacer grid, therefore it is the most practical choice for the application in the ALLEGRO ceramic assembly** [P5].

VIII. I have recognised that in the narrow space between the rods there is a high velocity gradient area, where the finite thickness (~1.5 mm) of the laser sheet has to be taken into consideration, because the PIV technique cannot differentiate the seeding particles in depth within this volume. **For the proper comparison of the PIV and CFD results, the velocity distributions from the CFD calculations have to be based on local averaging of velocity along 1.5 mm long line segment perpendicular to the laser sheet.** My proposed method makes the line along velocity profiles from PIV and CFD studies directly comparable [P5].

5. POSSIBLE APPLICATIONS

The results of this thesis can be utilized by multiple sectors of the nuclear industry. The results of CFD calculations can be used as a basis for safety analyses of the GFR technology to prove its reliability. The dissertation highlights modelling challenges that should be considered in GFR technology.

Accurate knowledge of the heat transfer coefficients within the fuel assembly can serve as a basis for model parameter adjustments in system code analysis.

The direction-dependent mixing factors between subchannels, that can be extracted from the CFD models can be used as input parameters for subchannel codes. Subchannel codes widely used as the basis of the nuclear safety analysis.

The PIV measurement system presented in the dissertation is unique in its kind. It can be used to study the flow processes in the rod bundles of nuclear reactors with triangular fuel rod lattice. Triangular-grid fuel is typical for VVER reactors, and there are only few publications in the international literature that study their operation using PIV methodology. In the future, we will use the PIROUETTE system for flow measurements relevant to VVER reactors. Given the level of detail of the measurements, the results can serve as a basis of international CFD benchmark exercises, too [11].

6. SCIENTIFIC PUBLICATIONS RELATED TO THE THESIS STATEMENTS

P1. G.I. Orosz, S. Tóth, Thermal hydraulic investigations of ALLEGRO ceramic fuel assemblies, Annals of Nuclear Energy, 120 (2018) 570–580,

P2. G.I. Orosz, S. Tóth, A. Aszódi, Simulations for L-STAR experimental gas-cooled system, Kerntechnik 85 (2020) 5 326-335

P3. G.I. Orosz, S. Tóth, A. Aszódi, Detailed thermal modelling of the ALLEGRO ceramic assembly, Nuclear Engineering and Design, 376 (2021), 111127

P4. G.I. Orosz, A. Aszódi, CFD Modelling of Mixing Vane Spacer Grids in ALLEGRO relevant gas-cooled case, Annals of Nuclear Energy, 164 (2021), 108628

P5. G.I. Orosz, B. Magyar, D. Szerbák, D. Kacz, A. Aszódi, ALLEGRO Gas-cooled Fast Reactor Rod Bundle investigations with CFD and PIV method, Nuclear Engineering and Design, 400(2022), 112062

7. FURTHER SCIENTIFIC PUBLICATIONS

P6. Orosz G.I., Tóth S., CFD vizsgálatok az ALLEGRO kerámia kazetta belső szubcsatornájára, Nukleon, IX. évf. (2016) 193, (Hungarian)

P7. Investigation of corner subchannel of ceramic assembly of ALLEGRO reactor, In: IEEE, 2017 6th International Youth Conference on Energy (IYCE) Seattle (WA), New York, IEEE (2017) Paper: 17097401, 5 p.

P8. Orosz G.I., Tóth S., Az ALLEGRO kerámia kazetta hőmérséklet-eloszlásának egyenletesebbé tétele, Nukleon, XI. évf. (2018) 211, (Hungarian)

P9. G. I. Orosz, M. Peireti, B. Magyar, D. Szerbák, D. Kacz, B. Kiss, G. Zsíros, A. Aszódi, Preliminary Thesis on the First Part of the ALLEGRO CFD Benchmark Exercise: ALLEGRO CFD BENCHMARK: PART 1: Flow Straightener Benchmark Description (2022), arxiv, https://doi.org/10.48550/arXiv.2203.03940

References

[1] OECD/NEA, Technology Roadmap Update for Generation IV Nuclear Energy Systems, January 2014.

[2] S. Tóth, B. Kiss, E. Gyuricza, and A. Aszódi, CFD Investigation of ALLEGRO Fuel Assemblies, The 15th International Topical Meeting on Nuclear Reactor Thermal - Hydraulics, in *NURETH-15*, Pisa, Italy, May 12-17, 2013.

[3] L. Bělovský, Project ALLEGRO, He-cooled fast reactor demonstrator, in *Nordic-Gen4 Seminar*, Lappeenranta, Finland, 2014.

[4] C. Poette, F. Morin, V. Brun-Magaud and J. Pignatel, ALLEGRO 75 MW cores definition at start of GOFAST, in *GoFastR-DEL-1.2-01, CEA*, Cadarache, France, 2010.

[5] P. Líška and G. Cognet, The ALLEGRO Project – European Project of Fast Breeder Reactor, in *1st International Nuclear Energy Congress*, Warsaw, Poland, May 23-24, 2011..

[6] Project SafeG, SafeG, [Online]. Available: https://www.safeg.eu/. [Accessed 09 06 2021].

[7] L. Charpentier, K. Dawi, M. Balat-Pichelin, E. Bêche, and F. Audubert, Chemical degradation of SiC/SiC composite for the cladding of gas-cooled fast reactor in case of severe accident scenarios, *Corrosion Science,* pp. 127-135, 2012 (59).

[8] OECD NEA, Best Practice Guidelines for the Use of CFD in Nuclear Reactor Safety Applications – Revision, www.oecd-nea.org: OECD NEA, February 2015.

[9] M. Raffel, C. Willert, S. Wereley, and J. Kompenhans, Particle Image Velocimetry – A practical guide, Springer: Berlin, Germany, 2007.

[10] G. I. Orosz and S. Tóth, CFD Investigation of Inner Subchannels of ALLEGRO Ceramic Fuel Assembly, *Nukleon,* May, 2016.

[11] G. I. Orosz, M. Peiretti, B. Magyar, D. Szerbák, D. Kacz, B. Kiss, G. Zsíros and A. Aszódi, Preliminary Thesis on the First Part of the ALLEGRO CFD Benchmark Exercise, *arxiv,* 2022.

[12] A. Vasile, B. Kvizda, S. Bebjak, G. Mayer, and P. Vácha, Thermal-hydraulics and Decay Heat Removal in GFR ALLEGRO, 2017.

[13] G. I. Orosz, S. Tóth and A. Aszódi, Simulations for L-STAR experimental gas-cooled system, *Kerntechninc,* 85(2020).

[14] G. I. Orosz, S. Tóth and A. Aszódi, Detailed thermal modelling of the ALLEGRO ceramic assembly, *Nuclear Engineering and Design,*376(2021).

[15] G. I. Orosz and A. Aszódi , CFD modelling of mixing vane spacer grids for ALLEGRO relevant gas-cooled reactor fuel geometry, *Annals of Nuclear Energy,* 164 (2021).

[16] G. I. Orosz, B. Magyar, D. Szerbák, D. Kacz, and A. Aszódi, ALLEGRO Gas-cooled Fast Reactor Rod Bundle investigations with CFD and PIV method, *Nuclear Engineering and Design,* 400(2022).