# LOGOS CFD Software Application for the Analysis of Liquid Metal Coolants in the Fuel Rod Bundle Geometries

D.Fomichev[1]

[1]JSC N.A. Dollezhal Research and Development Institute of Power Engineering
(JSC NIKIET), Moscow, Russian Federation

E-mail contact of main author: fomichev@nikiet.ru

**Abstract**. In this paper, the SST (Shear Stress Transport) *k-ω* model as well as combination of this model with turbulent heat transfer models, such as AKN and TMBF, released in the LOGOS software within framework of Federal Target Program "New Generation Nuclear Energy Technologies for the period of 2010-2015 and for the future up to 2020" and project "Development of Supercomputers and Grid Technologies", is applied for analysis of liquid metal coolants in the fuel rod bundle geometries. Results of numerical simulations of the LBE flow around a heated rod in an annular cavity (THESYS project), sodium flow in the tightly packed parallel rod-bundle (TEGENA experiment) and LBE flow in the 19-pin hexagonal rod bundle with support grids (THINS project) are described. Comparative analysis of calculated and experimental data is presented, as well as cross-validation with the results obtained using the CFD code ANSYS FLUENT 16.1.

**Key Words**: LOGOS CFD, Liquid Metals, Verification and Validation, Fuel Rod Bundles.

## 1 Introduction

Liquid metal (LM) such as sodium, lead or lead-bismuth eutectic are the most preferred coolants when developing innovative types of fast nuclear reactors. Despite the comprehensive amount of experimental and calculated data, obtained by Russian as well as foreign scientists over the past 30-40 years, the investigation of hydraulic and heat transfer characteristics of the fuel pin bundles is one of the key issue under the reactor design [1, 2].

Due to the development of the computational technologies, the thermal-hydraulic behavior of LM coolants, specifically in wire-spaced fuel assemblies (FA), as well as smooth rod bundles, can be studied by means of numerical modeling using the CFD codes, thus reducing the amount of expensive full-scale experiments.

The LOGOS CFD software was developed at FSUE RFNC-VNIIEF (Sarov, Russia) [3-5] within the frameworks of the project "Development of Supercomputers and Grid Technologies". At present time, it is being verified for the purposes of LM coolant flows modeling as part of the Federal Target Program "New Generation Nuclear Energy Technologies for the period of 2010-2015 and for the future up to 2020".

This paper presents the use of LOGOS CFD software to the numerical simulation of the following cases: LBE flow around a heated rod in an annular cavity (THESYS project [6, 7]), sodium flow in the tightly packed parallel rod-bundle (TEGENA experiment [8, 9]) and LBE flow in the 19-pin hexagonal rod bundle with support grids (THINS project [10, 11]). Brief consistent descriptions of experimental facilities and methodology of measurements, mathematical models, solver parameters and results are provided. The obtained results compare to an experimental data.

## 2 Mathematical model and solver parameters

The analyzed cases were solved using the RANS approximation in steady-state formulation. A SST k-ω model (Re-All) in combination with TMBF [12] and AKN [13] turbulent heat transfer models realized in the LOGOS software was selected for turbulence models.

When building the mathematical model both ANSYS FLUENT 16.1 and LOGOS CFD the following solver parameters were used: solution algorithm – COUPLED, the Courant number (initial, maximum) – 200, the Courant number (minimum) – 10, accuracy of solution – $10^{-6}$. The scheme parameters are as follows: convection – Upwind Difference (UD), pressure – STANDART, gradient – Green-Gauss method. Additionally, non-orthogonal correction for velocity, pressure, temperature and turbulent parameters is used [3-5]. Usage first order UD scheme for convection terms allows obtaining acceptable calculation accuracy at the lowest computation cost.

The thermal properties of the coolants used in the analyzed problems are described by functional temperature dependencies [14].

## 3 Fluid dynamics and heat transfer of lead-bismuth eutectic flow in annular channel

### 3.1 Description of Experimental Facility

The experimental investigation of characteristics of the longitudinal lead-bismuth eutectic flow past a single fuel rod was carried out in the THESYS2 test section [6, 7] of KALLA facility.

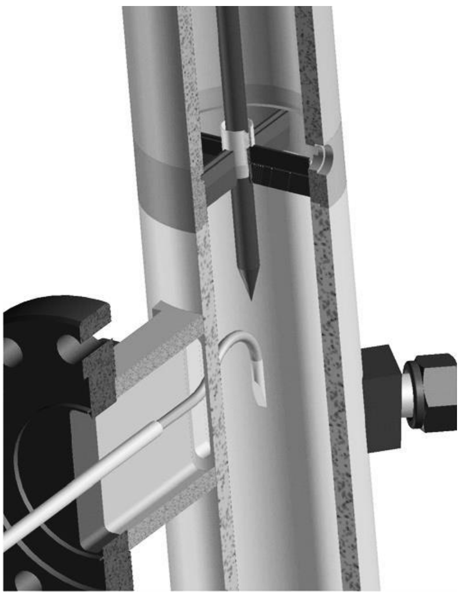


*FIG. 1. Geometrical model of rod with spacer grids.*

The test section consists of a moving cylindrical rod with a conic tip placed concentrically within a cylindrical channel. The diameter of the power generating rod, $d_{rod}$, is 8.2 mm, the inside diameter of the channel, $D$, is 60 mm. The length of the power generating region of the rod, $L_{heat}$, is 860 mm. When power of the rod is at its maximum, 22.4 kW, surface power generation reaches ~ $10^6$ W/m$^2$.

The central concentric placement of the rod within the channel is provided by means of three spacer grids, placed with constant pitch, $P$, of 370 mm.

The velocity profile measurement is carried out using a moving Pitot tube (see Fig. 1) with inside diameter of 0.5 mm, placed between the second and the third grids. The tube has two thermocouples situated 0.5 mm upstream, which enables simultaneous measurements of flow temperature and pressure.

The coolant flow through the working section is measured by four independent flowmeters, namely: electro-magnetic, vortex ones, as well as by differential pressure measurement and by a permanent magnet. The error of the average flow measurement is 0.3 %. The temperature measurement error is ± 0.1 K.

### 3.2 Methodology of flow parameter measurements

The coolant velocity measurements are taken at the following relative axial coordinates: $x^*$ = 1.56, $x^*$ = 9.29 and $x^*$ = 17.01, with $x^* = x/d_h$, where x is the axial coordinate, mm; $d_h$ is the hydraulic diameter calculated as $d_h = D - d_{rod}$, mm.

At coordinate $x^*$ = 15.87 the radial annular channel temperature is measured. The relative transverse coordinates $r^*$, where experimental measurements are taken, are presented in Table I.

TABLE I: RELATIVE TRANSVERSE COORDINATES
OF TEMPERATURE MEASUREMENT POINTS.

| **Measurement point number** | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 |
|---|---|---|---|---|---|---|---|---|---|---|---|
| **Relative transverse coordinate, *r**** | 0 | 0.01 | 0.03 | 0.05 | 0.1 | 0.14 | 0.2 | 0.3 | 0.34 | 0.47 | 0.64 |

### 3.3 CFD model

The geometrical model reproduces the test section. A computational grid (see Fig. 2) is a multi-block structure, which consists of tetrahedral elements in the regions, where the grids are positioned, and hexagonal elements in the coolant inlet area and in the regions between the grids.

According to the results of the grid convergence analysis, a grid consisting of ~ 20 million control volumes was selected. A dimensionless distance from the wall, $y^+$, is ~ 30.

During the numerical modeling, the boundary conditions were specified as given in Table II.

TABLE II: THESYS2 BOUNDARY CONDITIONS.

| **Parameter** | **Value** |
|---|---|
| Average velocity, $U$, m/s | 0.786 |
| Inlet temperature, $T_{in}$, K | 573 |
| Volumetric power generation, $q_v$, W/m$^3$ | $2 \cdot 10^8$ |

Average velocity at the computational model inlet corresponds to volumetric flow rate of 8.0 m$^3$/h (Reynolds number, Re, is equaled $2.67 \cdot 10^5$).

### 3.4 Results

Fig. 3 shows dimensionless velocity of the eutectic alloy flow in the vicinity of the conic tip of the rod in wall parameter coordinates $u^+(y^+)$. The value of the dimensionless velocity was calculated as $u^+=U/\sqrt{\tau_w/\rho}$, where $U$ is average flow velocity m/s; $\tau_w$ is shear stress on the wall (of the rod or the channel), Pa; $\rho$ is the coolant density, kg/m$^3$.

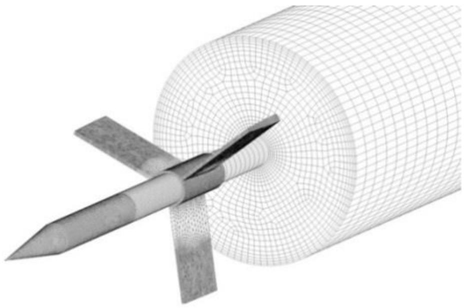


*FIG. 2. Meshing model fragment.*

The comparative analysis of the experimental and calculated values of $u^+$ shows that the extreme deviations of the calculated values of dimensionless velocity, $u^+$, in the vicinity both of the rod and the channel wall, are no more than 10 %, whereas the SST *k-ω* turbulence model without the additional turbulent heat transfer model shows better agreement with the experiment. Calculated values of $u^+$ when using the additional heat transfer models (TMBF and AKN) are similar to each other and exceed the experimental values of $u^+$ by no more than 5 %.

Fig.4 shows the experimental and calculated values of the Nusselt number, Nu, along the length of the heated rod.

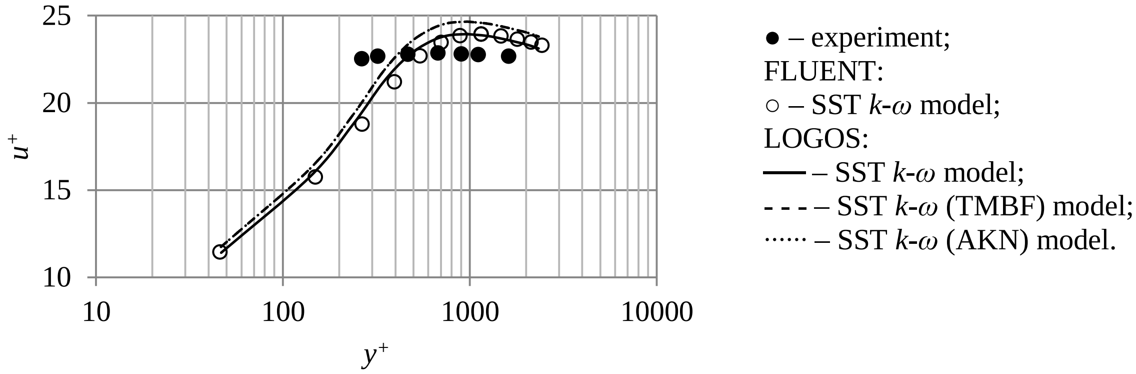



*FIG. 3. Distribution of dimensionless velocity $u^+$ ($x^*$=1.56).*

The calculated Nusselt number, Nu, values differ from the experimental ones in the regions between the grids. This is caused by the heated rod could move against the grids and the measuring instruments during the experiments, while when modeling the rod was fixed.

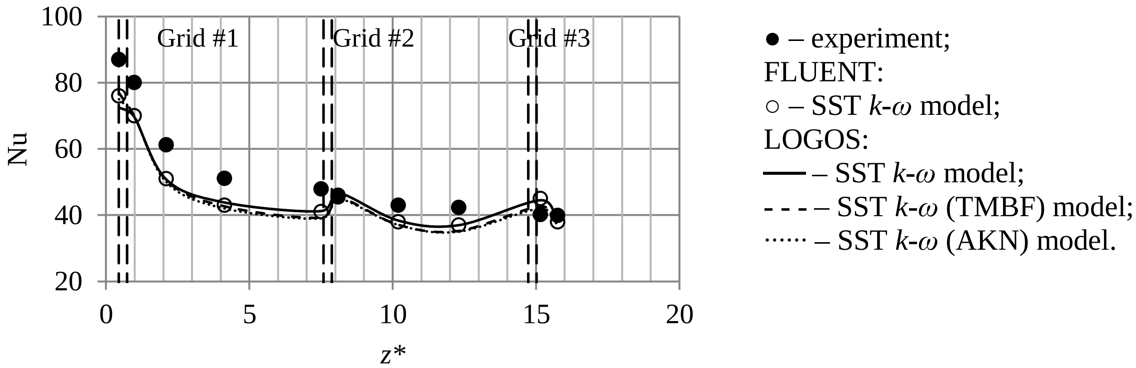
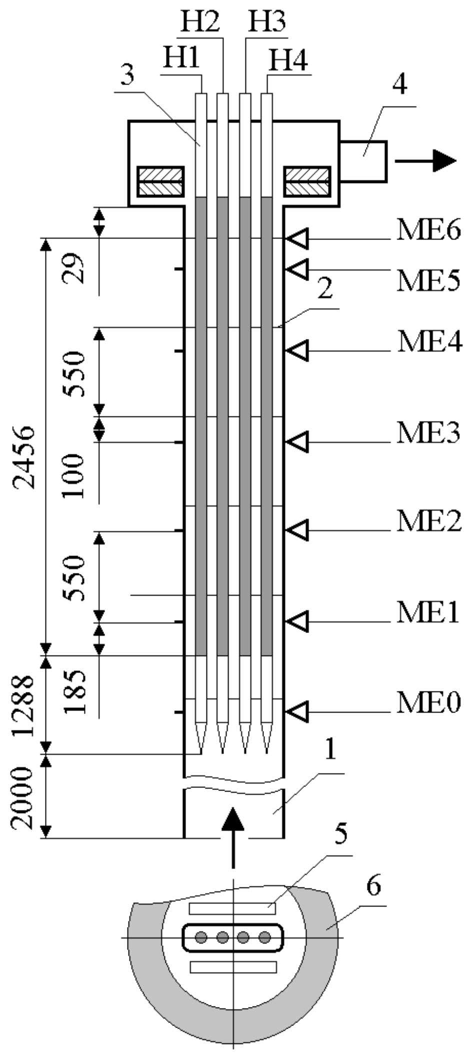



*FIG. 4. Distribution of Nusselt numbers, Nu, along the length of annular gap.*

Values of Nu obtained using ANSYS FLUENT 16.1 and LOGOS differ by no more than 2 %. The extreme deviation of the calculated Nu values obtained with LOGOS from the experimental data is no more than 12 %.




1 – rectangular channel; 2 – spacers; 3 – heated rod (heated zone); 4 – coolant outlet; 5 – radiation heater; 6 – channel insulation;

ME 0…6 – locations of temperature measurement points

*FIG.5. TEGENA experiment.*

## 4 Fluid dynamics and heat transfer of sodium coolant flow in heated rod bundle

### 4.1 Description of experimental facility

The RIG WÜP II experimental facility at which the TEGENA (TEmperatur- und GEschwindigkeitsverteilungen in Stabbündel-Geometrien mit turbulenter NAtriumströmung) experiment was carried out [8, 9], is a closed circulation loop with sodium coolant. The main objective of the experiment was to measure the coolant temperature in the gaps between the parallel heated rods simulating fuel elements.

The working section of the facility (see Fig. 5) is a rectangular channel with dimensions of 118.5×32.4 mm, inside of which four heated rods are positioned in parallel to each other. The rods are spaced by spacers with a width of 3.7 mm and a pitch of 550 mm. The outer rod diameter is 25 mm, the rod length is 3744 mm. In order to provide homogeneous coolant supply to the bundle, the rods have conic rod extension.

There is a possibility of uniform electrical heating of the rods along the 2456 mm length.

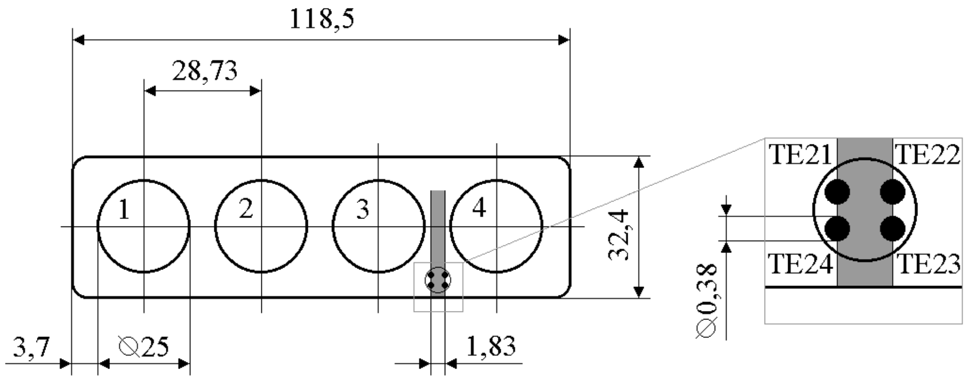
Flow temperature measurements were made by means of four thermocouples TE21…TE24, mutual arrangement of which is schematically shown in Fig. 6.

This article presents the experimental data on the flow temperature measurements in the gap between the third and the fourth rods along the paths corresponding to the locations of thermocouples TE 21 and TE 22. The coolant flow parameters and the surface heat flux values, $q_{rod}$, for each rod are given in Table III.

TABLE III: TEGENA BOUNDARY CONDITIONS.

| **Reynolds number, Re** | **Mass flow, *G*, kg/s** | **Inlet temperature, $T_{in}$, °C** | **Rod surface heat generation, $q_{rod}$, W/cm$^2$** | | | |
|---|---|---|---|---|---|---|
| | | | No.1 | No.2 | No.3 | No.4 |
| 60100 | 3.12 | 257.98 | 49.38 | 49.0 | 48.42 | 49.74 |

The detailed description of the experimental arrangement is given in [8].




TE21…24 – thermocouples

*FIG. 6. Coolant flow temperature measurement path.*

### 4.2 CFD model

The computational CFD model is a fragment of the test section consisting of heated rods and an outer channel with rectangular cross-section. Spacers, coolant inlet and outlet areas of the rod bundle were eliminated from the computational model.

The computational grid is block-structured and formed on the basis of hexagonal control volumes (see Fig. 7). According to the results of a sensitivity analysis (convergence analysis) of the grid, a grid consisting of ~ 60 million control volumes was selected. Dimensionless distance from the wall, $y^+$, is ~ 1.

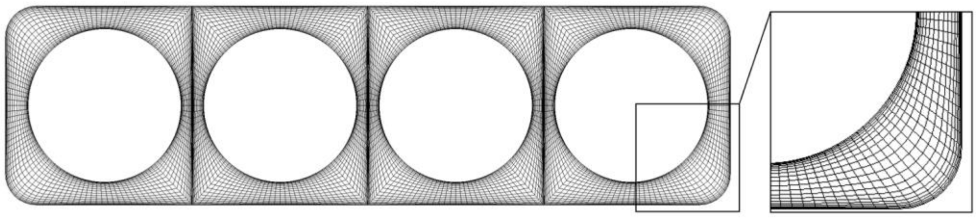


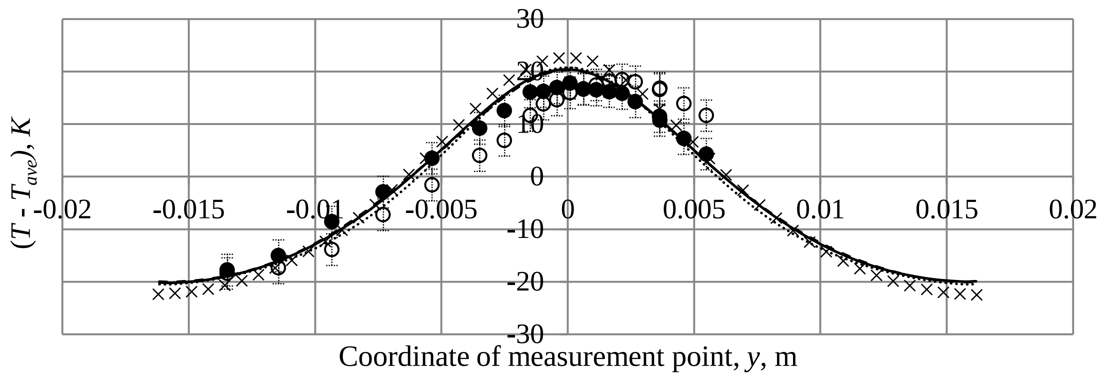
*FIG. 7. General drawing of computational grid.*

The boundary conditions used in the computational model description correspond to the experimental mode conditions: coolant mass flow, $G$, of 3.12 kg/s with uniform distribution of velocity over the cross-section and a temperature, $T_{in}$, of 257.98 $^oC$. For the heated walls, surface heat flux values, $q_{rod,}$ for each rod were specified (see Table III) and the flow no-slip condition, for the non-heated walls, adiabatic condition and flow no-slip condition were specified.

### 4.3 Results

Fig. 8 shows the distributions of the local temperature deviations from the average value ($T$-$T_{ave}$), calculated along path of TE21 located in the gap between the third and fourth rods.

Calculated values of temperature difference ($T$-$T_{ave}$) are slightly higher than the experimental data, which may be associated with a 10% error of the experimental coolant flow measurement. In this case the difference values ($T$-$T_{ave}$) obtained using LOGOS are within the deviation range of the experimental values ($T$-$T_{ave}$). The extreme deviation of the calculated values ($T$-$T_{ave}$) from the experimental ones is 3 %. The use of the additional turbulent heat transfer models did not cause significant changes in distribution of value ($T$-$T_{ave}$).




● – experiment (TE21);
○ – experiment (TE 24);
FLUENT:
× – SST *k-ω* model (TE21);

LOGOS:
——— – SST *k-ω* model (TE21);
- - - - – SST *k-ω* (TMBF) model (TE21);
········ – SST *k-ω* (AKN) model (TE21).

*FIG.8. Distribution of temperature deviations ($T$-$T_{ave}$) along paths of TE21 and TE24.*

## 5 Fluid dynamics and heat transfer of lead-bismuth eutectic flow in bundle with heated rods and spacer grids

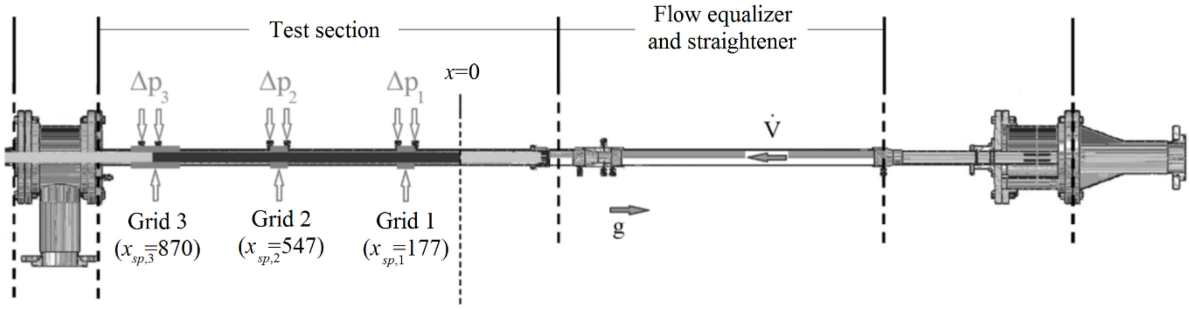
### 5.1 Description of experimental facility

The experimental work was carried out in the THEADES test section [10, 11] of the KALLA experimental facility. Test section arrangement with the installed FA is shown in Fig. 9.

Experimental FA placed vertically in upward coolant flow contains 19 rods arranged in a triangular lattice. The rod bundle is enclosed in a channel with hexagonal cross-section. The rods heated electrically. The main geometrical parameters of the FA are given in Table IV.

TABLE IV: MAIN GEOMETRICAL PARAMETERS OF EXPERIMENTAL FA

| **Parameter** | **Value** |
|---|---|
| Outer diameter of the rod, $d$, mm | 8.2 |
| Rod length, $L$, mm | 1272 |
| Length of the heated region, $L_{heat}$, mm | 870 |
| Relative rod position diameter, $P/d$ | 1.4 |
| Hydraulic diameter of the bundle, $d_{h,bdl}$, mm | 7.7 |
| Hydraulic diameter of an inner sub-channel, $d_{h,sch}$, mm | 9.52 |
| Spacer grid height, $L_{sp}$, mm | 25 |
| Grid porosity, $\chi$ | 0.29 |




1 – Experimental FA, 2 – Coolant inlet channel, 3 – Grid 3, 4 - Grid 2, 5 – Grid 1, 6 – Spacer grid positions

*FIG. 9. Test section arrangement with the installed FA*

Coolant inlet channel (see Fig. 9) consists of a flow equalizer and straightener providing uniform flow supply directly to the rod bundle.

Due to a possible flow velocity perturbation when passing through the support grid, which fixes the bottom ends of the rods, isothermic flow stabilizing region in the rod bundle is $L_{stab}$ = 400 mm ($L_{stab}$ / $d_{h,bdl}$ ≈ 52; $L_{stab}$ / $d_{h,grid}$ ≈ 42).

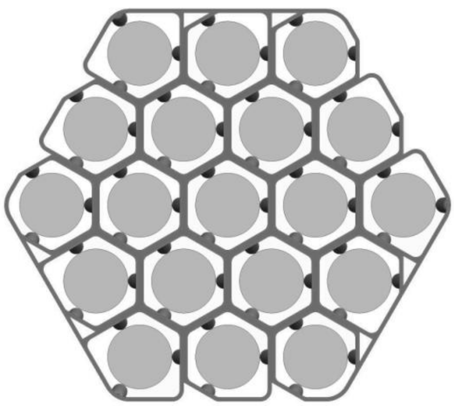


*FIG.10. Geometric model of spacer grid (hexagonal shroud is not shown).*

Spacer grids (see Fig. 10) are positioned throughout the height of the rod bundle, where the first two grids have fixed heights, and the third one, which is the closest to the outlet section, can move slightly. The axial coordinates of the grid centers adopted in the experimental and computational studies are as follows: $x_{sp,1}$ = 177 mm, $x_{sp,2}$ = 547 mm, $x_{sp,3}$ = 870 mm. The origin of coordinates is at the rod heating boundary $x$ = 0 mm.

### 5.2 Methodology of flow parameter measurements and their errors

Temperature measurements are taken at the following height coordinates: $x_{T,sp1}$ = 162 mm $x_{T,sp2}$ = 532 mm and $x_{T,sp3}$ = 855 mm. Error of the temperature measurement is ± 0.1 K.

Pressure is measured using 6 pressure taps. Each pair of taps is located at a distance of ± 50 mm from the center of the corresponding grid. Error of the pressure measurement does not exceed 0.15 %.

More detailed descriptions of the test section and the rod bundle, procedures for the experiments and the experimental data processing techniques are given in [10].

### 5.3 CFD model

The inlet section is located at a distance of 400 mm from the area where the rod heating begins, which is in the agreement with the experimental data. The outlet section is located at a distance of 180 mm downstream of the spacer grid. Thus, the total length of the rod bundle model is 1450 mm.

The multi-block computational grid of the rod bundle model is a combination of polyhedral-type grids in the spacer grid areas and hexagonal-type grids at straight-line portions of the bundle. Total amount of the grid control volumes is 37.68 millions. The computational grid parameters were selected in such a way that when the Reynolds number is at its maximum (regime 6, Table V), the dimensionless value $y^+ \approx 1$.

*FIG. 11. Fragment of computational grid (some of the rods are not shown)*

A fragment of the surface grid in the vicinity of the grid spacer is shown in Fig. 11.

The following conditions were taken as the boundary ones: for the inlet section – uniform flow distribution with constant mass flow value, corresponding to the design flow regime (Table V); for the outlet section, uniformity of a differential static pressure of 0 Pa is specified; for the non-heated rigid surfaces of the model, the adiabatic condition and the flow no-slip condition are specified; for the rod heating area, constant heat flux is specified.

Several modes within the wide range of volumetric coolant flows were selected from the matrix of experimental modes for numerical modeling. The considered design modes are given in Table V.

TABLE V: FLOW REGIMES MATRIX.

| **Flow regime number** | **Volumetric flow, $\dot{V}$, m$^3$/h** | **Temperature at model inlet, $T_{in}$, °C** | **Thermal power, $Q$, kW** | **Reynolds number, Re, $10^4$** |
|---|---|---|---|---|
| 1 | 1.00 | 200 | 50 | 0.966 |
| 2 | 2.00 | | 100 | 1.932 |
| 3 | 4.00 | | | 3.863 |
| 4 | 6.00 | | | 5.795 |
| 5 | 8.00 | | | 7.726 |
| 6 | 10.00 | | | 9.658 |

### 5.4 Results

Fig. 14 shows distributions of hydraulic friction coefficient of the rod bundle, *f*.

The numeric modeling results, obtained using the SST $k$-$\omega$ turbulence model, are in good agreement with the experimental data. The use of TMBF and AKN turbulent heat transfer models results in an increase in the friction coefficient, *f*, of a rod bundle, and the calculated values are within the margins of error of experimental determination of coefficient *f*.

The extreme deviation of the calculated values of friction coefficient, *f*, from the experimental ones is equaled 15 %.

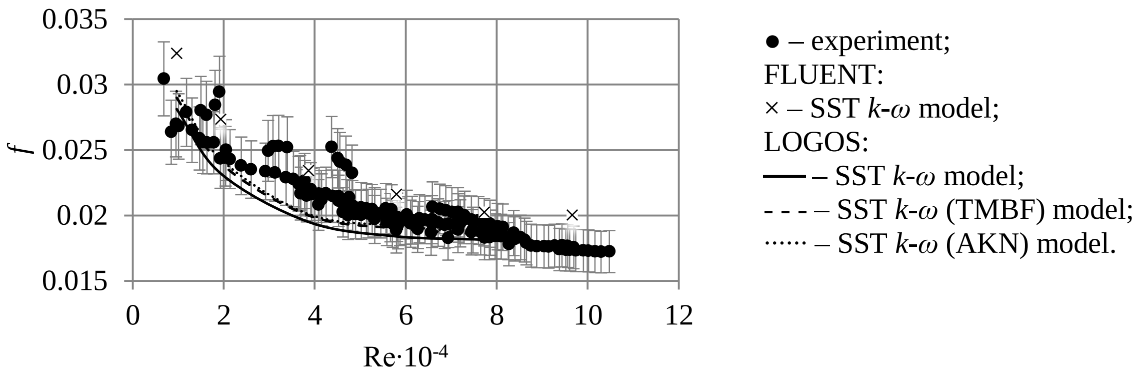



*FIG. 14. Friction coefficient, f, of rod bundle.*

Fig. 15 shows the experimental and calculated values of Nusselt number, Nu, as a function of Peclet numbers, Pe.

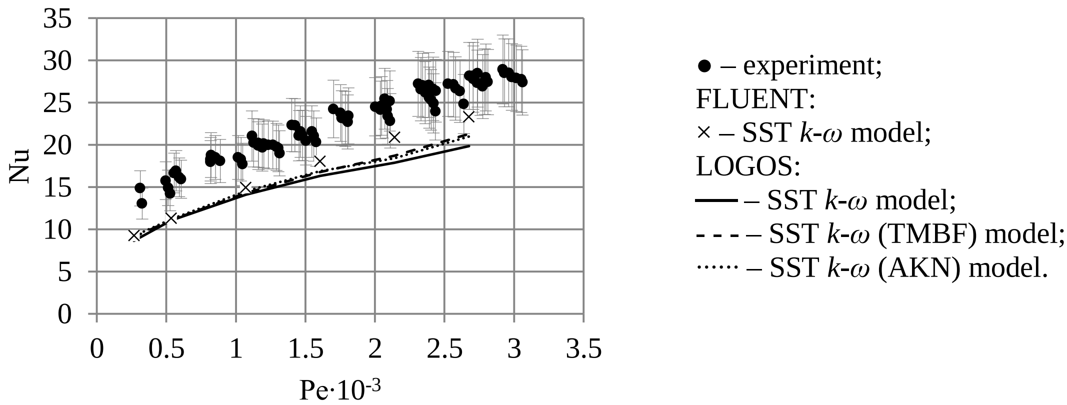



*FIG. 15. Distribution of Nusselt number, Nu, as a function of Peclet number, Pe.*

When Pe > 250, distributions of the calculated Nu numbers obtained using ANSYS FLUENT 16.1, are on the lower boundary of the confidence interval of the experimental Nu values, while the Nu values obtained using the LOGOS CFD software package are below the said interval. The extreme deviation of Nu values obtained from LOGOS CFD is ~ 20 % without the use of the turbulent heat transfer models, and ~ 15 %, when TMBF and AKN models are used.

## 6 Conclusions

The presented results of numerical modeling obtained as a part of the works on validation of the LOGOS CFD software showed, that this code can be used to simulate LM coolant flows in the channels of an irregular shape, specifically in FA rod bundles.

The turbulent heat transfer models, TMBF и AKN, implemented in the LOGOS CFD software used in combination with SST *k-ω* turbulence model make it possible to achieve better agreement with the experimental data.

The presented cross-validation analysis of the calculated data and the results obtained using the commercial CFD code ANSYS FLUENT 16.1 validate the use of the additional models.

**References**


[1] INTERNATIONAL ATOMIC ENERGY AGENCY, LMFR Core Thermohydraulics: Status and Prospects, IAEA-TECDOC-1157, Vienna (2000).

[2] ROELOFS, F., et al., "Status and perspective of turbulence heat transfer modelling for the industrial application of liquid metal flows", Nuclear Engineering and. Design (2015) 290, 99-106.

[3] KOZELKOV, A.S., et al., "Multifunctional LOGOS software package for computing fluid dynamics and heat and mass transfer using multiprocessor computers: basic technologies and algorithms", Supercomputing and mathematical modeling: Transactions of the XII International Workshop, Russia, Sarov, (2010) 215-230 (In Russian).

[4] KOZELKOV, A.S., et al. "Multifunctional LOGOS software package: physical and mathematical models for computing aerodynamics, fluid dynamics and heat transfer", preprint 111-2013, Russia, Sarov, FSUE RFNC-VNIIEF (2013) (In Russian).

[5] MELESHKIN, N.V., et al. "Verification of LOGOS software package for modeling of liquid metal coolant flow in fast reactor components", Innovations in nuclear power 2015 (Proc. Youth Conf. Moscow, 2015), Russia, Moscow, JSC NIKIET (2015) 261-272 (In Russian).

[6] LIFTIN, K., et al. "Investigation on heavy liquid metal cooling on ADS fuel pin assemblies", // J. of Nuclear Materials (2011) 415, 425-432.

[7] CHANDRA, L., et al. "A stepwise development and validation of a RANS based CFD modeling approach for the hydraulic and thermal-hydraulic analysis of liquid metal flow in a fuel assembly", Nuclear Engineering and Design (2009) 239, 1988-2003.

[8] MÖLLER, R. "TEGENA: Detaillerte experimentelle Untersuchungen der Temperatur- und Geschwindigkeitsverteilungen in Stabbündel-Geometrien mit turbulenter Natriumströmung", KfK 4491, Karlsruhe, Kernforschungszentrum Karlsruhe GmbH (1989).

[9] CHANDRA, L., et al. "CFD analysis of liquid metal flow in sub-channels for Gen IV reactors", Nuclear Engineering and Design (2011) 241, 4391–4403.

[10] PACIO, J., et al. "Heat transfer to liquid metals in a hexagonal rod bundle with grid spacers: Experimental and simulation results", Nuclear Engineering and Design (2015) 290, 27–39.

[11] PACIO, J., et al., "Heavy-liquid metal heat transfer experiment in a 19-rod bundle with grid spacers", Nuclear Engineering and Design (2014) 273, 33-46.

[12] CARTECIANO, L.N., et al. "Development and analysis of a turbulence model for buoyant flows", 1997 (Proc. 4th World Conference on Experimental Heat Transfer, Fluid Mechanics and thermodynamics Brussels, 1997).

[13] ABE, K., et al., "A new turbulence model for predicting fluid flow and heat transfer in separating and reattaching flows - 1. Flow field calculations", Int .J. Heat and Mass Transfer (1994) Vol. 37, No. 1, 139 -151.

[14] KIRILLOV, P.L., et al. "Nuclear power thermal-hydraulic analysis handbook. Volume 3. Thermal-hydraulic processes in transient and non-standard conditions. Severe accidents. Containment. Codes, their capabilities, uncertainties", Moscow, IzdAt, (2014) (In Russian).