RANS simulation of the thermal mixing in HTTF LP during normal operation conditions – High Temperature Test Facility at Oregon State University

Malwina J. Gradecka, Brian Woods
Warsaw University of Technology
Nowowiejska 21/25, 00-665 Warsaw, Poland
phone: +48-739-212603, malwina.gradecka@itc.pw.edu.pl

Oregon State University, 3451 SW Jefferson Way, Corvallis, OR 97331, US

Abstract – Since High Temperature Gas-cooled Reactors are being considered as the most promising design of upcoming IV Gen reactors, key research areas were identified to address safety aspects of this design. A number of simulations and experiments need to be conducted in this field. In this paper, thermal-hydraulics aspects of coolant flow through Lower Plenum (LP) of HTGR were considered, specifically flow characteristics to identify the risk of temperature stratification in LP and hot spotting on LP floor. Local temperature gradients can cause material degradation. As the power profile is non-uniform across the core, jets of coolant exit the core region at different temperatures and enter the LP impinging on LP floor causing hot spots at LP structure and temperature stratification. To address those issues numerical simulation and an experiment are being developed. The numerical simulation provides coolant flow velocity and temperature fields. The purpose of this study is to investigate the mixing phenomenon in the LP due to the risk of the hot streaking and thermal stratification phenomena during normal operation of HTTF. The following aspect are being examined: identification of gas flow behavior in lower plenum of HTTF based on CFD simulations, identification of hot streaking issue in the HTTF lower plenum using CFD tools, and computational investigation of gas mixing efficiency. This paper includes a description of experimental setup of HTTF, guidance for LP CFD modeling, and the results and analysis of CFD simulation.

I. INTRODUCTION
Currently Generation II and III nuclear power plant designs offer a secure and stable-cost electricity supply in many markets. However, further advances in the nuclear industry offer to broaden the prospects of nuclear energy use. To explore new opportunities, governments, industry, and the research community worldwide are engaged in a comprehensive discussion of the development of next generation nuclear energy systems known as Generation IV.[1]

The High Temperature Test Facility (HTTF) is being constructed at Oregon State University (OSU). HTTF is one of the facilities related to High Temperature Gas Reactor (HTGR) development. The High Temperature Test Facility was designed in order to simulate scaled coolant flow and heat transfer of the Modular High Temperature Gas Reactor (MHTGR) during the depressurized cooling conduction event, pressurized cooling conduction event and normal operation. The HTTF design (table 1) proposed by OSU is a quarter height, 1/8 pressure, full temperature model of the MHTGR. Thus it will support R&D needs for the Next Generation Nuclear Plant (NGNP) project. Tests performed at HTTF will
be used for reference modeling, analysis and validation of computational tools which will be crucial to the licensing process for performance prediction at normal and accident conditions in the HTGR. [2][3]

The HTTF is non-nuclear facility. TRISO fuel is replaced by graphite heater rods and the graphite prismatic block structure is replaced by ceramic blocks to capture prototypical temperature profiles. The working fluid in the MHTGR reaches very high temperatures on the hot side of the loop, which can be very beneficial for industrial applications, however, it also creates potential threats for material integrity. Helium as a non-reactive gas has good chemical properties for this type of application. However, it can cause structural problems if the power distribution across the core is not uniform. Helium, while cooling core sections with higher power density (hot channels), achieves significantly higher temperatures than in other channels/areas.[4][6]

There is a need for assuring that the facility will maintain integrity, function safely and remain intact during normal operation conditions for various levels of power generation. To safely operate the MHTGR-like reactor, the issue of hot streaking in the LP must be examined and if needed, resolved.[4]

Table 1: Parameter comparison between prototype MHTGR and scaled model HTTF[5]

<table>
<thead>
<tr>
<th>Parameter</th>
<th>MHTGR</th>
<th>HTTF</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Core power</td>
<td>350.0</td>
<td>2.2</td>
<td>MW(t)</td>
</tr>
<tr>
<td>Core power density</td>
<td>5.9</td>
<td>-</td>
<td>MW/m2</td>
</tr>
<tr>
<td>Inlet helium flow</td>
<td>157.05</td>
<td>1</td>
<td>kg/s</td>
</tr>
<tr>
<td>Inlet helium temp</td>
<td>258.6</td>
<td>-</td>
<td>°C</td>
</tr>
<tr>
<td>Inlet helium pressure</td>
<td>6.38</td>
<td>0.8</td>
<td>MPa</td>
</tr>
<tr>
<td>Core outlet temp</td>
<td>687</td>
<td>-</td>
<td>°C</td>
</tr>
<tr>
<td>Average reactor pressure drop</td>
<td>31.4</td>
<td>-</td>
<td>kPa</td>
</tr>
<tr>
<td>Average fuel temperature</td>
<td>677</td>
<td>-</td>
<td>°C</td>
</tr>
<tr>
<td>Average graphite temperature</td>
<td>625</td>
<td>-</td>
<td>°C</td>
</tr>
</tbody>
</table>

Stage two contains the experimental part of the research, which will be conducted at the High Temperature Test Facility. This part of the research is not presented in this paper, since the experimental facility is not yet fully operational. However, the experimental setup is described in next section to clarify the modeling approach.

Table 2: Test cases boundary condition description

<table>
<thead>
<tr>
<th>Case name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base case</td>
<td>inlet velocity 20.05 m/s; temperature 950 K</td>
</tr>
<tr>
<td>Hot channel</td>
<td>inlet velocity 20.05 m/s; temperature 1000 K section of 6 channels has increased inlet temperature to 1000 K</td>
</tr>
<tr>
<td>Variation 1</td>
<td>as Base case + reduced mass flow by ~5%</td>
</tr>
<tr>
<td>Variation 2</td>
<td>as Hot channel case + reduced mass flow by ~5%</td>
</tr>
</tbody>
</table>

II. MATERIALS AND METHODS

This work is first out of three stages of study concerning optimization of the flow mixing in the lower plenum of HTTF during normal operation conditions. First stage of research contains preliminary calculations for the lower plenum mixing in the HTTF. Pretest calculations were prepared using best estimate methodology to obtain simulation as close to the reality as possible considering available computational resources. Cases described in table 2 were applied and analyzed to quantify characteristics of the coolant flow, thermal mixing and detect/confirm existence of the potential exploitation issues. STAR-CCM+ software was used for the computational fluid dynamics part of the research.

The experimental part of the research will be conducted at the High Temperature Test Facility at Oregon State University. The High Temperature Test Facility is a scaled – one quarter test installment model of Modular High Temperature Gas-cooled Reactor. The HTTF is a full temperature test facility operating at maximum temperature 850°C and operating pressure 0.8 MPa. Thus the margin between the maximum temperature and the operating temperature is over 160 degrees allowing to simulate the hot channel. The facility, equipped with a heater power of approximately 2.2 MWth, is cooled with helium; however, other gases can be used as well. [3]

The HTTF was designed to simulate a variety of accident conditions, transients and normal operation conditions. The main elements of the facility are:

- Vessel with 2.2 MW electrically heated prismatic block core simulator
- Ceramic reflector and core regions
- Gas circulator
- Forced flow cavity cooling system
- Break valves
- Confinement simulation tank
- Instrumentation package
- Complete data acquisition system [7]
core structure is supported by ceramic posts that rest on the bottom of the lower plenum. [3]

Jet mixing behavior in the LP will be examined using data collected from numbers of thermocouples and one pressure sensor. The static pressure sensor is placed opposite the plenum exit, near to the side of the LP. The reference line for this pressure tap will be the vessel inlet pressure. Coolant temperature characteristic data will be collected using a total of 62 thermocouples located within the LP structure. They can be divided into three types: gas thermocouples at certain height of the plenum, ceramic temperature thermocouples in the LP floor and gas thermocouples at the exit of transition blocks. Two primary phenomena are to be characterized by TC’ located in the LP: air ingress and jet mixing along with impingement behavior. [8]

II.B. CFD Simulation – PHASE I

The CFD simulation was modeled so it resembles the experiment as closely as possible considering the limited computational resources. Numerical calculations have been conducted using computational fluid dynamics software – STAR-CCM+ Adapco. The simulation was conducted for turbulent, incompressible gas flow through typical HTGR’s lower plenum geometry. Each case model was build using best estimate policy, therefore all phenomena occurring LP are modeled as close to reality as possible taking into account the limited computational time and capacity of available cluster computer.

The geometry (figure 1) of the model built in NX 7 consists of the lower plenum roof, the lower plenum floor, inlet channels, the outer reflector, support posts and the outlet duct (hot duct). Gas is delivered to the lower plenum by 234 inlet channels. Approximately 520 cooling channels of various diameters from the core area are merged in reflector nr 1 into 234 channels of constant diameter.[3]

Inlet channels can be divided in two categories due to their function. The majority of the channels (main channels - 192) cool the core, removing produced heat. Those channels are distributed by triangular array through the core. However, they are placed just in the heated area so they create an annular shape. The remaining 42 channels (bypass channels) are distributed along the inner (6 channels) and the outer boundary (36 channels) of the fueled area in the reflector. Those channels transport up to 11% of the flow rate and at the temperature of the coolant. In addition, the velocity in the bypass channels will be substantially lower than for regular channels. [3]

One outlet duct with an inner diameter of 0.298 m transports hot mixed helium out of the reactor vessel. The reactor core weight rests on 163 cylindrical posts of diameter 0.057 m. The posts obstruct the flow creating stagnation, recirculation and high turbulence zones. All structural elements in LP are made of ceramic material (Greencast 94-F). The surface of all elements is not perfectly smooth and may introduce considerable friction effects.[9]
The computational mesh was created using unstructured polyhedral cells with prism layer applied to the wall vicinity resulting in approximate number of 650 k, 2 million and 2.5 million cells.

The physical interior was established using the following models:
- Ideal gas
- 3 dimensional mesh
- Turbulent flow regime
- Reynolds-Averaged Navier-Stokes turbulence model

Boundary conditions – The boundary conditions applied vary for the different cases considered. Boundary conditions for each case are listed in table 2 along with description of phenomenon examined.

Figure 3: Computational domain with outlet duct, support posts and inlets.

The boundary conditions for the LP model were implemented in following manner. Inlet channels were divided into subsections/ zones which are controlled from level of the operating panel. Thus there are 10 different inlet zones of inlet channels and for every zone is capable of delivering different gas properties. These 10 zones contain 234 channels in total, providing in total flow rate of 0.9874 kg/s. About 11% of the flow is bypassed, gas which flow through the outer and central reflector, that is several degrees colder than gas in the other channels entering the LP. Flow rates, jet temperatures and velocities are summarized in table 2. [3]

The interior walls were modeled with no-slip condition, convection boundary as rough/smooth surfaces.

II.C. Test cases description for CFD simulation in PHASE 1 and experiment in PHASE 2

The several cases of operational conditions will be tested and analyzed, however the first two cases Base case and Hot channel case are more vital for the study results. Base case was used to obtain flow characteristics in the LP as well as validate data obtained in the first stage of research from computational analysis.

The second test – Hot channel indicates mixing characteristics in case of uneven power distribution and consequences of this operational condition, as well as provides answers for questions about the need for preventing hot spotting/thermal stratification. The two following tests (Variation 1 and 2) are designed to examine the influence of reduced coolant flow rate at temperature distribution on lower plate surface and temperature variations of the gas at the LP outlet.

<table>
<thead>
<tr>
<th>Case</th>
<th>Time dependence</th>
<th>Turbulence model</th>
<th>Mesh</th>
<th>Boundary conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base</td>
<td>Steady state</td>
<td>K-epsilon</td>
<td>2.5 mln</td>
<td>Hot channel</td>
</tr>
<tr>
<td>Hot</td>
<td></td>
<td></td>
<td>2 mln</td>
<td>Variation 1</td>
</tr>
<tr>
<td>Hot</td>
<td></td>
<td></td>
<td>650 k</td>
<td>Variation 2</td>
</tr>
<tr>
<td>Assumed</td>
<td>steady state</td>
<td>n/a</td>
<td>n/a</td>
<td>Base case</td>
</tr>
<tr>
<td>Hot section</td>
<td>Methodology will be selected basing on validation of Phase 1 with experimental results</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 3: Tests description

III. PRELIMINARY RESULTS

A number of simulations were conducted as described in table 2 and 3. Preliminary calculations – Base case was calculated to obtain a generic profile of the flow in order to address requirements for definition of next more advanced simulations. The temperature and velocity profiles were obtained using RANS k-ε model. Since it is a two equation model, complex flow occurring in LP may have poor performance on swirling flows in non-circular ducts, but for the purpose of preliminary analysis it is sufficient.

Figure 4: Percent amount of cells at selected temperatures in LP for different meshes

To examine the solution independence of the numerical grid, three mesh sizes were generated and solved. In figure 4, the percentage of cells under a certain temperature threshold is presented for all
calculated meshes. The amounts of cells at selected temperatures were varying by less than 1%. On average, the finest mesh contained the least amount of cells falling into the temperature threshold.

From perspective of mixing, the hot channel case is the most interesting thus its results are described in detail. In figure 5, the progress of gas mixing can be seen for the finest mesh consisting of 2.5 million cells. For the lowest value of threshold – 952 K, just 2 degrees over temperature in rest of the plenum, we can distinguish area where gas from hot channel influenced temperature rise on its way to the outlet. The next threshold of 962 K represents only fraction of cells amount from 1st threshold, indicating that primarily distinguished area of mixing represents insignificant level of actual thermal mixing between LP gas and hot inlet gas. The following illustrations show higher level of threshold. For the highest value containing temperatures from 992 to 1000 K, the amount of cells which is suitable for this threshold is rather low; however, gas at this temperature reaches over half of plenum height before mixing with lower temperature gases. This results in heating up the surface of lower plenum floor along the flow path of the hot gas. The maximum value of temperature in vicinity of the LP floor caused by hot channel presence was 970.7 K. This hot spot is localized in front half of the plenum.

The figures 7 and 8 display the velocity and temperature profile along probe line 1, which goes from a central point on the outlet along the x axis all the way through LP. This probe line, shown in figure 6, passes through few support columns, which is visible on the charts as blank gaps between data points. Since the hot section of inlets are not located in central part of the LP, for about half length of the plenum no significant temperature increase can be
seen. Subsequently, some variation in the temperature reading can be seen as initial hot gas mixes into the flow, and then temperature increases smoothly almost till the end of the probe line which is located at LP outlet.

For both profiles at figure 9 and 10 some discrepancies between results can be observed. This suggests that mesh independent solution has not been reached yet and further mesh refinement is advised, especially in order to achieve better agreement on flow velocity near and inside of outlet duct.

Comparison of flow and temperature fields for remaining calculated cases are shown in figures 11, 12, 14 and 15.

Figure 9: Temperature profile at probe line 1 for fine, medium and coarse mesh

Figure 10: Velocity profile at probe line 1 for fine, medium and coarse mesh

Figure 11: Temperature profile at probe line 1 for base case and variation 1

Figure 12: Velocity [i] profile at probe line 1 for base case and variation 1

Figure 13: Streamlines seeded from hot section
IV. CONCLUSIONS

Numerical model of LP was built in order to simulate and investigate gas mixing efficiency and assess probability of “hot spotting” on the LP structure. Application of one hot section with 50K temperature rise to the model caused LP floor temperature increase of over 20 K. Hot spot is present at the front part of the LP model, however still in area of LP.

The selected mesh sizing resulted with solutions which are somewhat grid dependent. Thus further mesh advancement is advised. Additionally applied k-ε turbulence model, as 2-equation model, doesn’t reflect advanced flow features as swirl flow and recirculation zones. It is advised to investigate application of more advanced models like RSM-RANS, LES.

REFERENCES