Preliminary CFD Investigations for the NIST Neutron Source

Idan R. Baroukh *†, Anil Gurgen*‡, Joy S. Shen*ⱡ, Abdullah G. Weiss*
** NIST Center for Neutron Research, 100 Bureau Drive, Gaithersburg, MD 20899, USA*
† NRCN Nuclear Research Center of the Negev, Israel
‡ Department of Materials Science and Engineering, University of Maryland, College Park, MD 20742, USA
ⱡ Department of Mechanical Engineering, University of Maryland, College Park, MD 20742, USA
idan.baroukh@nist.gov; anil.gurgen@nist.gov; joy.shen@nist.gov; abdullah.weiss@nist.gov

doi.org/10.13182/T127-39557

INTRODUCTION

Studies are currently ongoing to design a replacement for the National Bureau of Standards Reactor (NBSR) at the NIST Center for Neutron Research. The replacement reactor, currently dubbed as the NIST Neutron Source (NNS), is conceptualized as a 20-MW light water reactor with a compact core design. The compactness of the core presents a lot of challenges in maintaining appropriate cooling, especially since high fission rates are desirable to supply a stream of neutrons for neutron scattering and irradiation experiments.

This paper highlights the preliminary efforts towards the development of a computational fluid dynamics (CFD) model for detailed flow behavior investigation. A 2-dimensional representation of the core's flow geometry is described alongside model specifics such as the expected inlet conditions. The expected physical phenomena are discussed alongside some preliminary results for a simplified geometry of the core. Note that the scope of this paper is limited to the CFD modeling efforts, where the thermal-hydraulics and neutronics analysis activities are described in other works [1,2].

METHODOLOGY

Geometry & Expected Physics

The pre-conceptual NNS core consists of nine fuel assemblies (FA) that are arranged in a 3x3 lattice, where each FA houses 21 curved fuel plates cooled by upwards-flowing light water (H_2O). The average power of a fuel plate in the core is expected to be ~106 kW, which can be used to understand the heating applied on the flow. Taking a section view of the core reveals that the fluid will occupy the geometry shown as in Fig. 1, where 3 distinct regions are visible: 1. the inlet, 2. the active height, and 3. the outlet regions. It is helpful to distinguish these 3 regions in order to predict the likely flow phenomena across the core. Distinguishing the expected flow phenomena is important because it guides decisions regarding how a CFD model can be validated. As is, the core is a unique geometry, and validating its CFD model would require a specialized experimental effort with a replica of the geometry. However, when the core is divided into the regions in Fig. 1, fundamental flow physics are apparent, and allow for the ability to validate them with classical experiments (channel flow, mixing, etc…), which are abundant in literature.

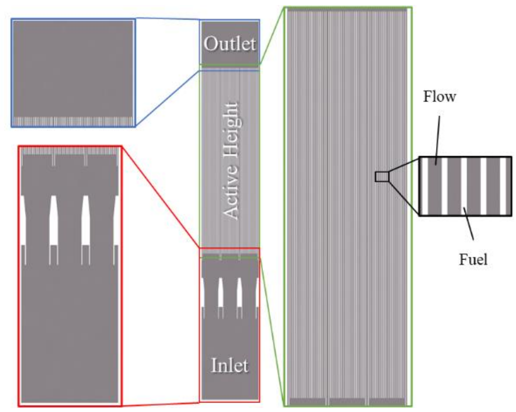

Fig. 1. The two-dimensional core geometry (side view) with the distinct regions. Note the presence of unique coolant channels between each two adjacent fuel assemblies.

For example, take the active height region where an array of separated flow channels form between the fuel plates in the core. Fundamentally, each flow channel resembles a classical channel flow (when considering the viewpoint in Fig. 1). This makes it straightforward to compare and validate models dealing with the active height, where channel flow experiments can provide sufficient validation metrics, particularly heated channel flows [3].

The outlet region is a case of channel flow mixing [4], albeit with significantly more channels than what is found in classical fluid mechanics literature, making it very similar to parallel flow studies [5]. As such, significant shear stresses are expected at the outlet due to the development of multiple mixing layers caused by flows from channels attempting to combine. This could also yield considerable pressure gradients that will affect the flow distribution based on the outlet temperatures from each channel.

Finally, the inlet region is the most complex region with a separation (due to the presence of a lower core plate) and mixing directly upstream of the fuel plates and their respective coolant channels. Multiple phenomena are on display here including flow separation commonly seen in flows over bluff bodies [4, 6], mixing [4, 5], recirculation and entrainment in the more confined portions of the geometry [7]. This region alone justifies the need for a CFD model, where inhomogeneity in the flow due to either thermal or momentum-based pressure gradients can cause improper cooling for the fuel plates. As such, it is important to validate models simulating that region (and the other regions) and breaking down the flow behavior into classical flow phenomena is monumental in finding appropriate validation metrics. However, it is also important to simply develop a CFD model for the core, and this paper details those preliminary efforts with the utilization of a simplified geometry, particularly when considering the inlet region in Fig. 1. Note that the CFD efforts outlined in this work deal only with the behavior of the flow in the core, and not the thermal behavior of the fuel plates themselves. The fuel plates are represented as thermal boundary conditions based on the average power of a fuel plate in the core.

Simplified Core Model Setup

The model with the simplified inlet takes on the geometry shown in Fig. 2, where the flow separation and mixing upstream of the parallel channels in the FAs are all collapsed into a single channel flow. This reduces most computational difficulty in simulating the inlet region and allows for computationally inexpensive simulations, which is desirable in this early stage of core design, when the whole core is still being optimized. Note that a y^+ treatment is implemented here, where y^+ is a non-dimensional wall distance for a wall bounded flow and is generally defines as the ratio of friction velocity multiplied by the nearest distance to the wall to the local kinematic viscosity of the fluid [8]. A y^+~10 is implemented in the FA coolant channels, and y^+<200 at the outer plate of the core, which is acceptable per studies in literature. Future simulation will focus on improved Y^+ treatment, where greater mesh refinement will be pursued near the walls.

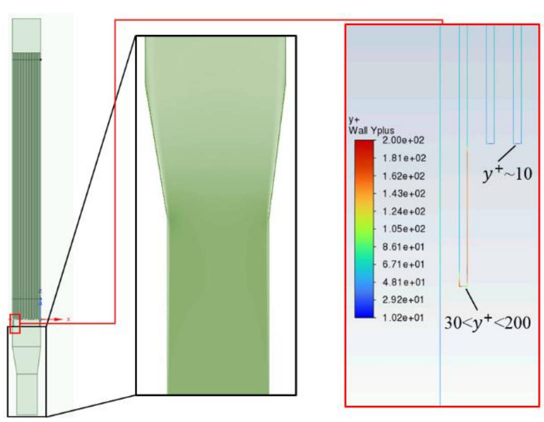

Fig. 2. The simplified geometry. Note the use of different y^+ values to reduce computation costs where reasonable.

In this simplified 2D model, the mesh contains 69,700 quad/tri cells, and a standard k-ϵ turbulence model [9] is utilized. Considering that the parallel channels are the majority of the model, k-ϵ is expected to perform adequately for most of the geometry, but higher uncertainties are expected at the outlet (mixing) and inlet (separation). The coolant (H_2O) has an inlet temperature of 316.5 K and a bulk velocity of 12.78 m/s. This model is only used for preliminary analyses to better understand the performance of k-ϵ in the core. Another study is conducted in order to understand the flow behavior in a single channel only.

Single Channel Study Setup

To better understand the impact of the mesh on the flow behavior in a single channel, multiple models are explored and compared. The idea is to have the best possible accuracy with the lowest possible computational cost. Fig. 3 shows four 3D meshes that were investigated in this work, where the curve is due to the curved fuel plates and is only visible when viewing the FAs from a top view. Axially, they represent vertical channels as shown in Fig. 1 and Fig. 2. Here, the effects of y^+ is investigated with a standard k-ϵ turbulence model, an inlet temperature of 300 K, and inlet velocity of 12.78 m/s. Table I compares the features of the meshes in Fig. 3.

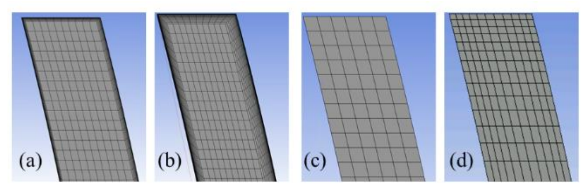

Fig. 3. The investigated meshes in the single channel study.

Table I. The details of each mesh used in the single channel study (Fig. 3).

Mesh	# Of Cells	Inflation Layers	y^+
(a)	4.9×10^6	Yes	0.8-5
(b)	8.2×10^6	Yes	1.2
(c)	1.3×10^6	No	103-174
(d)	0.14×10^6	Bias	31-80

Analytical Reference Solution

As a comparison and benchmark to the CFD results, an analytical 1D thermal-hydraulics model has been developed and is detailed in another work [1]. This model can provide flow rate and temperature distributions throughout the core and can act a reference solution to check the CFD results.

RESULTS & DISCUSSION

Single Channel Steady State Flow

Table II summarizes the results from the single channel study, where small changes can be seen in the total pressure drop (ΔP) and maximum wall temperature ($T_{w,max}$) across the channel. Of note is the fact that although mesh (d) has the lowest number of cells, its optimized inflation layer allows it to be in most agreement with the analytical reference solution (which is considered the benchmark in this study). Fig. 4 shows the results for mesh (d), where the wall temperature is maximum near the top of the channel, and uniform pressure, temperature and velocity distributions are observed. Considering the universally good agreement with the analytical solution, these results demonstrate that the standard k-ϵ turbulence model is sufficient for the channel flow, which is most of the core.

Table II. A comparison between the results obtained with each mesh for the single channel study. The % difference corresponds to the $T_{w,max}$ result.

Mesh	**ΔP [MPa]**	**$T_{w,max}$ [K]**	**% Difference to Analytical**
(a)	0.284	336.0	1.86%
(b)	0.295	335.0	1.56%
(c)	0.287	335.5	1.71%
(d)	0.291	334.2	1.33%
Analytical	0.299	329.8	-

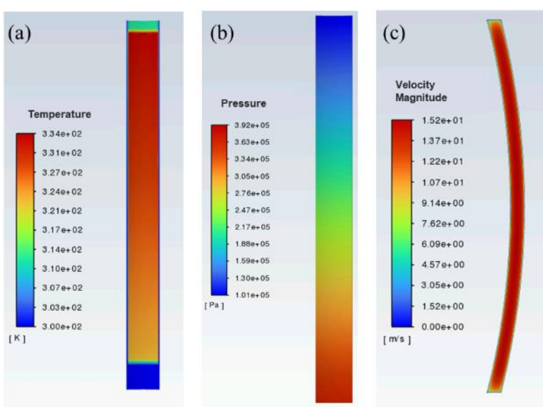

Fig. 4. The results for mesh (d), where (a) the wall temperature and (b) pressure axial temperature profiles are shown alongside (c) the radial profile of the velocity magnitude.

When comparing these results to a 2D equivalent (i.e., 2D mesh and problem setup), Table III shows nearly identical results to the 3D model with significantly less elements. This 2D model of the channel also uses an improved (lower) Y^+, which likely contributes to its good agreement with the 3D model. This demonstrates that a 2D model is sufficient to describe the fluid behavior in a fuel channel, and such notion is visually reiterated in Fig. 5, where both models show nearly identical pressure distributions.

Table III. A comparison between the 2D and 3D models of the single channel.

Parameter	**3D – mesh (d)**	**2D**
# Of Cells	140,000	36,000
Y^+	31-80	1.5
ΔP [MPa]	0.291	0.291
$T_{w,max}$ [K]	334.2	335

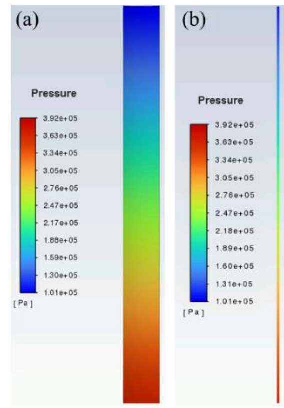

Fig. 5. A comparison between the (a) 3D and (b) 2D models of the single channel. Note that the same dimensions are used in both models, but the perspective differs.

Full Core Steady State Flow

The simplified 2D model of the core yields a pressure drop (ΔP) of ~0.322 MPa across the core, which is higher than the analytical ΔP shown earlier. This deviation is likely due to the added local ΔP at the inlet and outlet regions, which are neglected in the analytical ΔP.

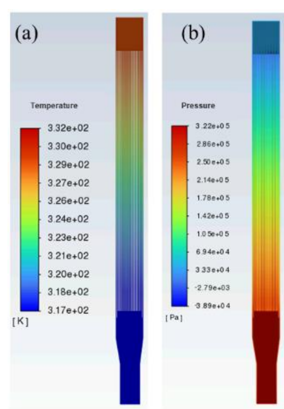

Fig. 6. The (a) temperature and (b) pressure profiles across the simplified 2D core model.

The temperature and pressure 2D profiles are shown in Fig. 6, where a uniform temperature profile is shown such that a temperature rise is evident as the flow progresses through the core. The pressure profile is interesting as it shows a uniform pressure drop as the flow progresses through the core until it reaches the outlet region, where non-uniformity emerges due to the mixing phenomenon. It is not clear whether this non-uniformity is characteristic of the physics or is simply a limitation brought by the usage of the k-ϵ turbulence model (additional studies are planned for this).

The velocity magnitude profile is shown in Fig. 7, where even with the simplified inlet region, the k-ϵ model appears to struggle in capturing the near-wall behavior at the channel inlets and, most notably, around the outer plates of the core. At the outlet region, it predicts a nearly uniform mixing with no bias, which is expected due to the symmetric nature of the geometry and the uniform inlet velocity. As for the flow distributions in the channels, most channels exhibit a nearly identical ~1.42 m/s except for the outermost channels, which exhibit ~1.33 m/s. Further investigations with other turbulence models are recommended.

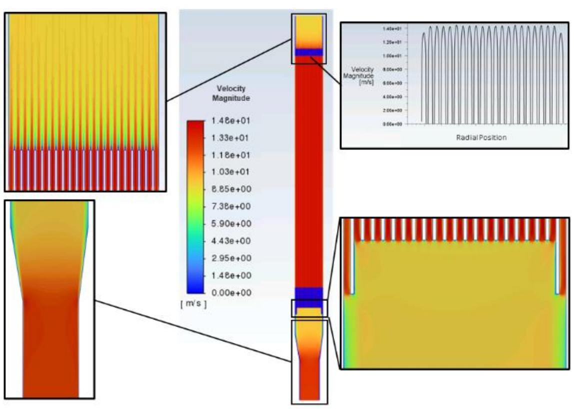

Fig. 7. The velocity magnitude profile across the simplified 2D core model.

CONCLUSIONS

This work demonstrates preliminary efforts towards the development of a CFD model for the NNS. Steps were taken to evaluate and break-down the expected physical phenomena, revealing three regions of interest across the core: the inlet, active height, and outlet regions. A limited mesh sensitivity analysis is shown for a single fuel channel model, where the y^+ treatment is found to be more important than the overall fine-ness of the mesh. The single channel analysis also demonstrated that a 2D model captures nearly identical physics to the 3D model of the channel. A simplified version of the full core is modeled in a 2D geometrical representation, which seemed to struggle in capturing the flow distribution when using simplified inlet near the walls and showed a non-uniformity at the outlet. The k-ϵ model is deemed appropriate for capturing the flow behavior in the channels, which spans most of the core, but a turbulence model sensitivity analysis is recommended to better understand the flow behavior in the inlet and outlet regions. A 3D model of the core is also recommended to provide a holistic understanding of the flow patterns and phenomena in the core.

DISCLAIMER

Certain commercial equipment, instruments, or materials (or suppliers, or software, ...) are identified in this paper to foster understanding. Such identification does not imply recommendation or endorsement by the National Institute of Standards and Technology, nor does it imply that the materials or equipment identified are necessarily the best available for the purpose.

REFERENCES

1. I.R. BAROUKH, A. GURGEN, J.S. SHEN, A.G. WEISS, “A Preliminary Thermal-hydraulics Analysis for the NIST Neutron Source”, Trans. Am. Nucl. Soc., 127, (2022).
2. O.S. CELIKTEN, D. SAHIN, A.G. WEISS, “Highlights of Neutronics Analyses for the Pre-conceptual NIST Neutron Source Design”, Trans. Am. Nucl. Soc., 127, (2022).
3. M. TEITEL, R.A. ANTONIA, “Heat transfer in fully developed turbulent channel flow: comparison between experiment and direct numerical simulations”, *International Journal of Heat and Mass Transfer*, **36 (6)**, 1701-1706 (1993).
4. A.G. WEISS, P.J. KRISTO, J.R. GONZALEZ, M.L. KIMBER, “Flow regime and Reynolds number variation effects on the mixing behavior of parallel flows”, *Experimental Thermal and Fluid Science*, **134**, 110619 (2022).
5. M. WEI, Y. FAN, L. LUO, G. FLAMANT, “CFD-based evolutionary algorithm for the realization of target fluid flow distribution among parallel channels”, *Chemical Engineering Research and Design*, **100**, 341-352 (2015).
6. R.L. SIMPSON, “Review–A Review of Some Phenomena in Turbulent Flow Seperation”, *Journal of Fluids Engineering*, **103 (4)**, 520-533 (1981).
7. Z. LI, Y. YUAN, B. GUO, V.L. VARSEGOV, J. YAO, “The Recirculation Zone Characteristics of the Circular Transverse Jet in Crossflow”, *Energies*, **13 (12)**, 3224 (2020)
8. J. NIKURADSE, “Laws of Flow in Rough Pipes”, National Advisory Committee for Aeronautics, Technical Memorandum 1292 (1950).
9. B.E. LAUNDER, B.I. SHARMA, “Application of Energy Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disc”, *Letters in Heat and Mass Transfer*, **1(2)**, 131-138 (1974).