Pressure Vessel Design Optimization of The Transformational Challenge Reactor

Nathan D. See*, Benjamin R. Betzler, Sacit Cetiner, Phillip Chesser, Jesse Heineman

*Oak Ridge National Laboratory, One Bethel Valley Road, Oak Ridge, TN 37830-6170, *seend@ornl.gov*
https://dx.doi.org/10.13182/T123-32981

INTRODUCTION

Increasing operation, construction, and engineering costs challenge the economic viability of nuclear reactors in the United States, thereby reducing reactor deployments and lengthening the timeline to critical operation. Many advanced reactor development activities focus on addressing some of these challenges through the design process, whereas others accept materials and manufacturing limitations. The US Department of Energy's Office of Nuclear Energy (DOE-NE) Transformational Challenge Reactor (TCR) program looks to change this focus with a manufacturing-informed design approach. The DOE-NE TCR program aims to fundamentally change the way nuclear reactors are developed, built, and certified. Bypassing traditional manufacturing techniques, the TCR program is taking advantage of advancements in additive manufacturing (AM) to design, certify, and operate a modern reactor over the course of years, not decades. By leveraging the design freedom that AM allows, traditional issues like cooling system design can be more precise and innovative in shape, placement, and performance. Advanced modeling and simulation tools are necessary to predict the behavior and performance of these more precise geometric configurations.

This work documents computational fluid dynamics (CFD) design optimization studies performed at Oak Ridge National Laboratory. Two areas of the pressure vessel were analyzed: the inlet plenum region and the outlet plenum region. Due to the limitations imposed by conventional fabrication techniques, historically, within the nuclear industry, these areas have not been specifically designed to significantly reduce their contribution to the overall pressure drop while still maintaining their uniformity of flow, vertical velocity, and/or temperature.

The TCR is characterized as a 3MWth helium gas–cooled reactor [2] with an advanced fuel form (uranium nitride TRISO embedded in 3D printed SiC [3]. Core design and analysis activities in the TCR program are driven by manufacturing with a focus on rapid prototyping and testing to support near-term deployment. Ongoing development activities such as those discussed herein will supply additional data and experience to support reactor deployment.

Computational simulations were performed with Siemens' STAR-CCM+ CFD Package and controlled by Siemens' HEEDS Design Optimization Package. The SHERPA algorithm within HEEDS was used to automatically execute STAR-CCM+ runs and select the optimum design parameters.

METHODS

Within the pressure vessel and outside of the core region, there are two areas of focus for design optimization: the inlet plenum and the outlet plenum. The inlet plenum is an annulus shape at the bottom of the pressure vessel, and the outlet plenum takes up the interior void left behind. Helium coolant flows from the bottom of the pressure vessel, up the outside, and then back down through the core in the center of the pressure vessel. After leaving the core, the helium coolant flows down and out of the pressure vessel through the outlet plenum.

Geometry

The inlet plenum consists of two inlets that bring the cold helium into the pressure vessel. From the inlets, the helium flows through the inlet plenum region and up into the riser section. The riser section is characterized by structural ribbing that rotates in a helical pattern up the riser section, as seen in Fig. 1.

The outlet plenum consists of inlet coolant flow coming through a structural plate and into an open funnel-shaped cavity that directs the helium coolant out of a single outlet tube in the bottom of the pressure vessel, as seen in Fig. 1.

Computational Flow Regime

The TCR is a 3MWth gas-cooled reactor. Pressurized helium is used as the coolant with a mass flow rate of 2.892kg/s at 7MPa.

As the inlet plenum is dominated by the coolant entering the pressure vessel, a constant density physics model is a suitable, lower computational expensive, choice. A constant density of 4.9435 kg/m^3 and constant dynamic viscosity of 3.5E-5 Pa-s were utilized [5]. While these values represent an older inlet temperature design point of 400°C, the best design was re-evaluated and deemed suitable at the current design temperature of 300°C.

The outlet plenum, unlike the inlet plenum, has a much more dynamic mixture of temperatures, due to the core being upstream. Therefore, a polynomial fitted density, specific heat, and thermal conductivity was utilized alongside a constant dynamic viscosity, 1.716E-5 Pa-s [5]. This setting of a constant dynamic viscosity was an oversight and instead it should have also been set as a fitted polynomial. However, the best design was re-evaluated with the appropriate settings and the trends held regardless of viscosity. Therefore, the study was deemed suitable.

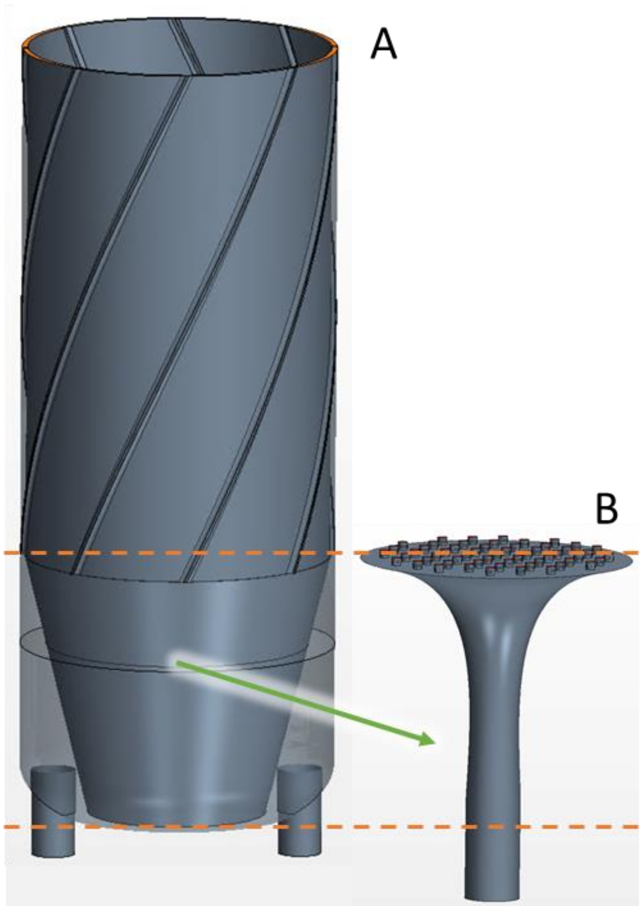

Fig. 1. A: Inlet plenum design geometry with transparent exterior pressure vessel wall. B: Outlet plenum fluidic design geometry, located in the internal, bottom center of conical inlet plenum.

Computational Setup

An internal high performance computing (HPC) cluster known as *Libby* was used for all simulations. Libby is a 28 compute node cluster with 512GB DDR4 RAM per node. Each compute node used for this research has 2 Intel Xeon CPUs with 16 physical cores each. Libby uses a rack-mounted InfiniBand Switch with speeds of 56Gb/s between compute nodes.

Siemens STAR-CCM+ flow solver (v15.02.007) was used for all CFD simulations. Simulations were performed with one to six nodes, and each node using all 32 physical cores.

Computational Mesh

For both the inlet plenum and the outlet plenum optimization studies, a trimmed cell hexahedral mesh was used with a surface remesher and a prism layer mesh near the body. The choice of the trimmed cell hexahedral mesh over the polyhedral mesh was simply due to the speed at which the volume grid could be grown, lending itself to the quicker turnaround time for a design optimization case.

A mesh independent study was performed for both the inlet plenum and outlet plenum. A base cell size was varied to control the bulk mesh size, and prism layer controls. The resulting prism layer height ratio was defined by mesh M_n/M_{n+1}. Table I and Table II detail these studies and show that the pressure drop was within 5%. Figure 2 plotted the pressure drop against the prism layer height demonstrating that the slope was zero and therefore acceptable.

TABLE I. Inlet Plenum Mesh Independence Study

Mesh	Prism Layer Height	Prism Layer Height Ratio	Δp (psi)	% Change
M1	0.08	-	0.632	-
M2	0.15	1.9	0.659	4.26%
M3	0.19	1.3	0.661	0.34%

TABLE II. Outlet Plenum Mesh Independence Study

Mesh	Prism Layer Height	Prism Layer Height Ratio	Δp (psi)	% Change
M1	0.15	-	0.252	-
M2	0.20	1.3	0.257	1.89%
M3	0.25	1.3	0.245	-4.63%

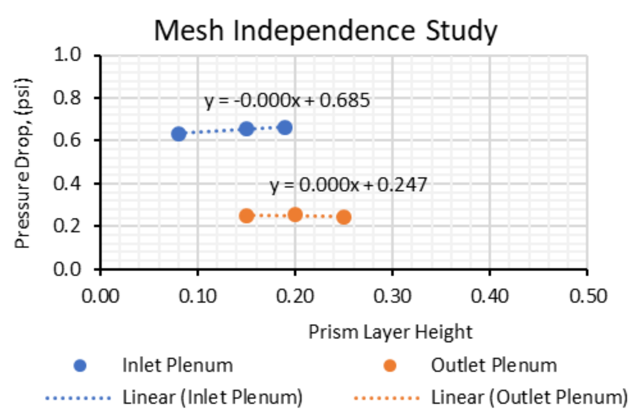

Fig. 2. Mesh independence study, prism layer height vs. pressure drop.

Simulation Strategy

The inlet plenum and outlet plenum studies followed the same simulation strategy. A quality steady-state run was performed initially as determined by Wall-Y^+ regimes and based on engineering judgement gained from experience working on the TCR project. Then the simulation settings—predominantly the number of prism layers and the prism layer height—were adjusted until a lower volume cell count was achieved. This reduction of volume cell size reduced the amount of time to mesh and run the solution, lending itself to a design optimization study. Then the design optimization study was performed. To find the best design, 100–150 simulations where performed.

Design Optimization Setup

The goal of the inlet plenum design study was to reduce the pressure drop across the region while maintaining a high level of vertical velocity uniformity up the riser section. As the pressure drop was not to be sacrificed to the expense of the vertical flow uniformity, a two-point weighted objective function was used, weighting the minimization of the system pressure drop with a factor of 1.0, and weighting the maximization of the vertical riser velocity uniformity with a factor of 0.25. STAR-CCM+ has a volume uniformity report that was utilized by using the vertical component of velocity, v_y, over slices along the vertical y-axis.

$$\overline{v_y} = 1 - \frac{\sum_c \left| v_{y,c} - \overline{v_y} \right| V_c}{2 \left| \overline{v_y} \right| \sum_c V_c} \tag{1}$$

Where, $\overline{v_y}$ is the volume average of v_y, $v_{y,c}$ is the cell value, and V_c is the cell volume.

These objectives were realized by modifying two variables that allowed the inlet tubes to rotate about their radial axis, x and y. The x-axis was allowed to rotate between 0 and 35 degrees, pitching the inlet tubes towards the slope of the helical structural supports within the riser section. The y-axis was allowed to rotate between -5 and +5 degrees, pitching the inlet tubes towards and away from the pressure vessel's exterior wall.

With the outlet plenum design study, the TCR instrumentation and control (I&C) team is placing thermowells within the outlet plenum to measure the average core exit temperatures. To do this, a radial plane must be determined for which the measured temperature is within ±5°C of the average core exit temperature. Previous insights from design studies on the outlet plenum led to the selection of the current I&C plane.

The goal of this outlet plenum design study was to reduce the standard deviation of the coolant helium temperature on the I&C plane. This allows the thermocouples to more accurately measure the average core outlet temperature. A single point optimization was used with two design constraints: (1) the average coolant temperature on the I&C plane had to be within ±5°C of the average core exit temperature, i.e. temperature uniformity, and (2) the outlet plenum pressure drop was limited to 0.5 psi. The second constraint was due to the design limit of the overall pressure drop on the entire pressure vessel. If the outlet plenum were to exceed the 0.5 psi limit, then a larger circulator would be needed to compensate.

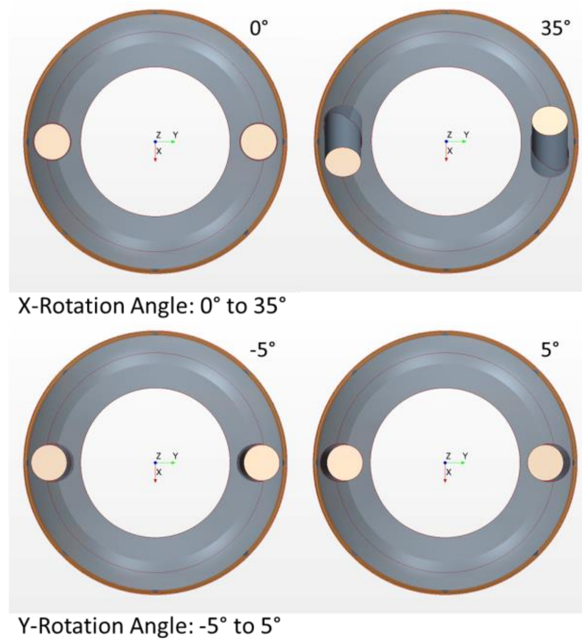

Fig. 3. Design parameters for inlet plenum.

By leveraging knowledge from previous outlet plenum design studies, it was determined that to gain temperature uniformity without causing a drastic pressure drop, the cross sectional area would need to be reduced. To do this, a spline was created with a single mid-point that would shape the structural outside wall. This spline would have two design variables: one controlling its vertical spacing, V, and one controlling its horizontal spacing, H. See Fig. 4.

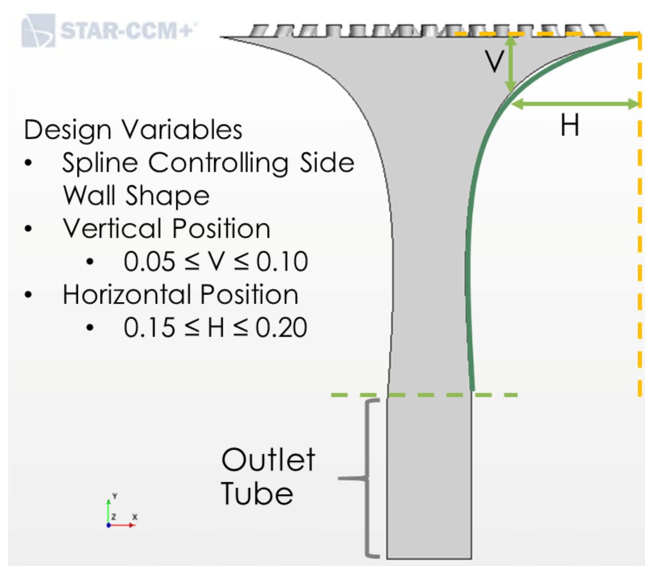

Fig. 4. Design variables for outlet plenum side wall.

Additional design variables were implemented to control the fluidic inlet channels through the structural support plate, just above the outlet plenum. These fluidic inlets are categorized by their neutronic ring numbers. Using a cylindrical coordinate system located at the center of the neutronic rings, each inlet was allowed radial motion, as well as rotation about theta, as shown in Fig. 5.

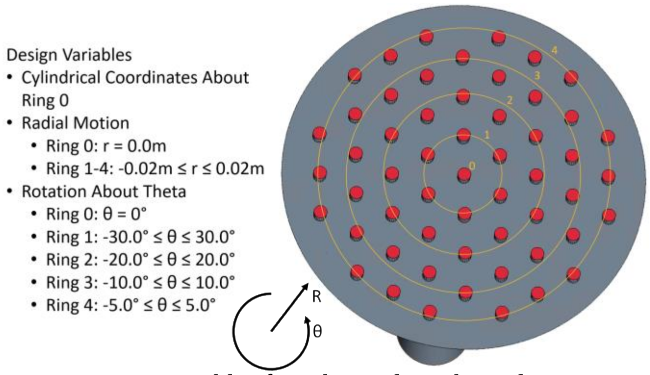

Fig. 5. Design variables for inlet cooling channels.

RESULTS

The inlet plenum design optimization study computed 100 design simulations in approximately 10 days on a single node. This would have taken approximately two months using traditional methods. This study was successful in saving 13% pressure drop while maintaining an 89% vertical riser flow uniformity. The final pressure drop was found to be 0.57 psi.

This design study was also found to be insensitive to the best design; all of the closest best designs had similar performances. This is a favorable feature to have in case reactor operating conditions vary from specified design parameters.

The outlet design study computed 150 designs in 2.25 days on 4 compute nodes; 24 of the 150 designs evaluated were feasible, capable of maintaining a pressure drop of less than 0.5 psi. This illustrates how difficult it is to gain a uniform temperature distribution in such a short flow distance without causing large scale mixing that would lead to a large pressure drop. The best design was able to maintain an average temperature of 497.3°C on the I&C plane while attaining a standard deviation of 1.03°C and limiting the pressure drop to 0.49 psi.

It is interesting to note that not all design variables were significant influencers to the overall outcome. To obtain the overall performance function, a low standard deviation of temperature and a pressure drop of under 0.5 psi were maintained, and the spline's horizontal variable and neutronic row 2's radial motion was important. However, when only considering the standard deviation of the I&C plane, neutronics row 1 theta rotation was singularly important, as illustrated in Fig. 6.

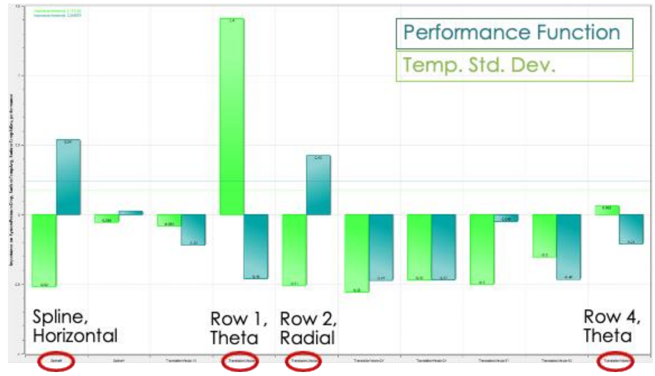

Fig. 6. Outlet plenum design variable influence.

CONCLUSION

Advancements in additive manufacturing, coupled with advanced computing techniques, bring a chance for further design gains where they were once unreasonable. The inlet plenum design study gained a 13% pressure drop improvement while the outlet plenum design study was able to reduce the standard deviation on the I&C plane to 1.03°C. With studies such as these, the TCR program showcases that these new design tools are ready for adoption in the design of the next generation nuclear fleet.

ACKNOWLEDGMENTS

This study was supported by the Transformational Challenge Program by the U.S. Department of Energy, Office of Nuclear Energy.

Peter Wang and Justin Weinmeister are also acknowledged for their contributions to this ongoing work.

REFERENCES

1. B.R. BETZLER et al., “Advanced Manufacturing for Nuclear Core Design,” ORNL/TM-2019/1258, Oak Ridge National Laboratory (2019).
2. B.R. BETZLER et al., “Transformational Challenge Reactor Preconceptual Core Design Studies,” Nuclear Engineering and Design, (2020). Submitted
3. G. VASUDEVAMURTHY et al., "Fabrication and Characterization Methodology of Transformational Challenge Reactor Fuel Form," *Trans. Am. Nucl. Soc.*, **June** (2020).
4. OAK RIDGE NATIONAL LABORATORY, “Transformational Challenge Reactor,” (Oak Ridge National Laboratory, https://tcr.ornl.gov/ (accessed June 23, 2020).
5. E.W. LEMMON, M.O. MCLINDEN, and D.G. FRIEND, “Thermophysical Properties of Fluid Systems,” NIST Standard Reference Database Number 69, National Institute of Standards and Technology (2018), doi.org/10.18434/T4D303.