Case Study

Conjugate Heat Transfer Simulation for a Fuel Element of an Experimental Nuclear Reactor

Dr. Yuxuan Liu, Professor Brendan Kochunas, University of Michigan

Edited by the Ansys Academic Development Team

education@ansys.com

Summary

Ansys Discovery is a simulation-driven design tool that combines instant physics simulation, high fidelity simulation and interactive geometry modeling in a single easy-to-use experience. This enables designers and engineers to illustrate a wide variety of concepts in the fields of design, structures, fluids, and heat transfer with the help of simulation.

This case study demonstrates how a conjugate heat transfer analysis can be performed for a fuel element in an experimental nuclear reactor, including geometry importing and simplification, meshing, simulation and post-processing, all being carried out using Ansys Discovery.

Table of Contents

1. Introduction

A nuclear reactor is a device used to initiate and control a fission nuclear chain reaction. Nuclear reactors are often used at nuclear power plants for electricity generation and in nuclear marine propulsion. Some reactors are also used to produce isotopes for medical or industrial use, or for radiation relevant to research and experiments. The heat from nuclear fission is often passed to a working fluid (*e.g.*, water or gas).

The aim of this case study is to simulate the heat transfer of a fuel element in an experimental reactor. Compared to the general heat transfer applications, the fuel element of an experiment reactor often includes unique geometrical characteristics, such as very thin fuel plates, that increase the complexity of modeling and simulation for geometry repairing, heat source set up, and mesh creation. This case study walks through the steps of using Ansys Discovery to perform the heat transfer calculation of the nuclear fuel element.

2. Experimental Nuclear Reactor

The Ford Nuclear Reactor (FNR) was a research reactor that operated at the University of Michigan from September 1957 until July 2003. As shown in Figure 1, the FNR was an open-pool experimental reactor. The core suspended six meters below the pool surface on a movable support structure. The highly demineralized primary coolant in the pool served as a radiation shield, a neutron moderator and reflector, the reactor core coolant, and as a viewing window to permit visual observation of the reactor core. The control of the reactor was achieved through the use of three shim safety rods and one regulatory rod.

Figure 1. Photograph of the FNR facility

A typical core configuration used in the FNR, shown in Figure 2, consisted of 36 plate-type fuel elements. Overall fuel elements dimensions were approximately 8.10 cm x 7.71 cm x 88.34 cm. Standard fuel elements contained 18 aluminum clad fuel plates. Each plate was nominally 7.06 cm wide and 0.1524 cm thick. Plates were fabricated in sandwich fashion with aluminum cladding on both sides of a layer of intermetallic uranium aluminide. The thickness of fuel meat was 0.0762 cm and the thickness of cladding is 0.0381 cm.

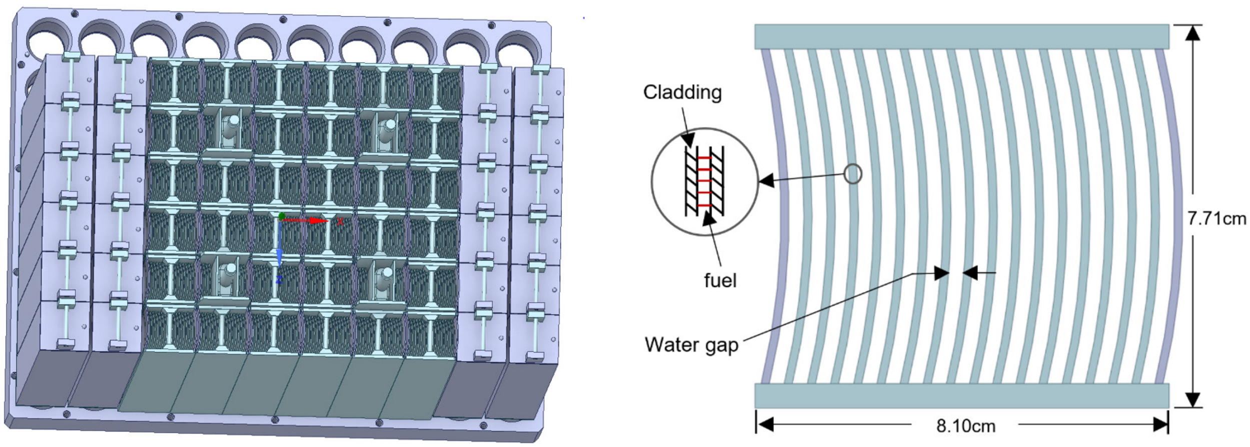

Figure 2. FNR core configuration (left) and fuel element geometry (right)

The generation of heat in the FNR reactor core is primarily dependent upon the interaction of thermal neutrons and fissile materials in the core. The plate-type fuel elements use an enriched U-235 as the primary fissile isotope. The fuel is assumed to be evenly distributed in the fuel meat, thus the heat generation is then dependent mainly upon the distribution of the neutron flux. The reactor operates at a full power level of 2MW. At this full power condition, the primary coolant system removes 2MW of heat from the core by forced circulation and maintains a bulk pool temperature of less than 47oC. In forced circulation, the primary coolant flows down through the water gaps between the fuel plates in the fuel elements.

The conjugate heat transfer problem to be solved here via simulation is a single fuel element from the FNR. Several assumptions are made to obtain the heat sources and boundary conditions of the problem.

1) The power of a fuel element is averaged from the full power level (2MW divided by 36).
2) A typical forced convection coefficient of water (3000 $W/m^2 \cdot °C$) is used for the outer solid surfaces of the fuel element.

3. Geometry Importing and Simplification

The fuel element CAD model is extracted from the full core FNR model, as shown in Figure 2. Starting from the fuel element model, the steps shown in Figure 3 are performed to create a CFD-ready model for the analysis with Ansys Discovery. Instructional videos on how to use Discovery for this purpose can be found at this link: Geometry Preparation for Fluids Simulation. A copy of both the original and the simulation ready CAD model are available in the downloaded case study folder.

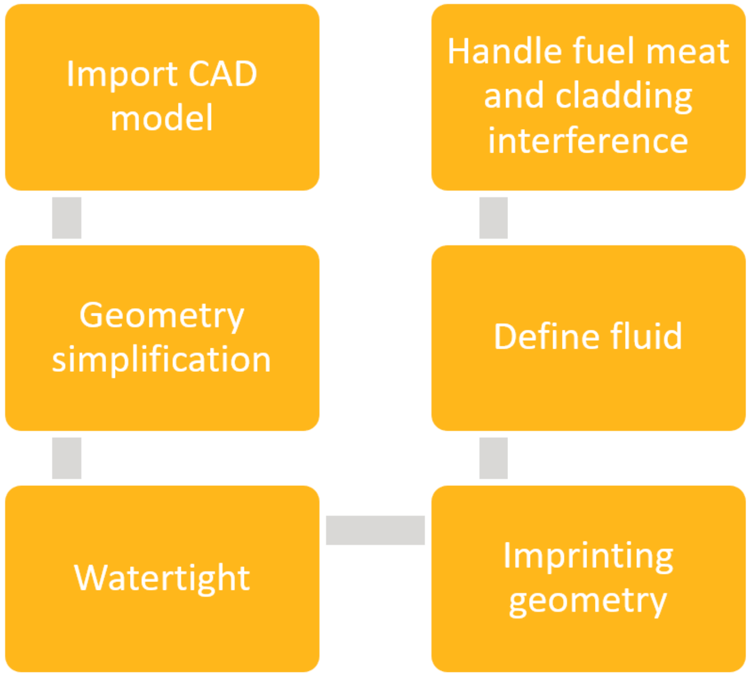

Figure 3. Geometry simplification and tuning

Geometry simplification

The original CAD model includes excessive geometrical details at the inlet and outlet of the fuel element. The nozzles and bars, along with the small holes/gaps in the supporting structures, create an overly complicated geometry for a demonstration calculation of the fuel element, where the focus is on the heat transfer inside the fuel element (*i.e.* heat conduction in fuel plates and heat convection between fuel and coolant). Therefore, the geometrical details of the inlet and outlet of the fuel element are eliminated, as shown in Figure 4.

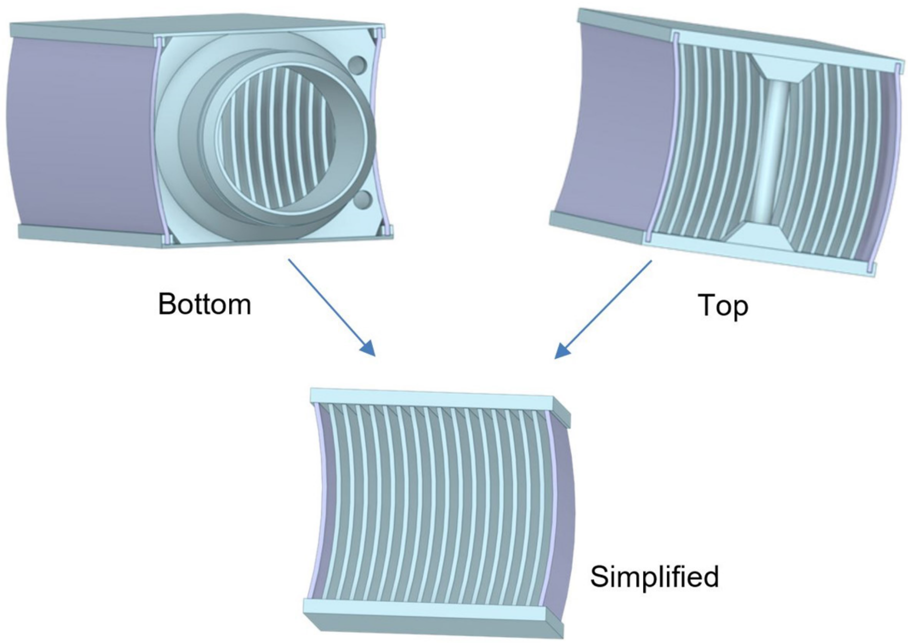

Figure 4. Geometry simplification of the fuel element

Watertight

A model is considered watertight if the faceting of all topologically linked surfaces is coincident. A close check of the CAD model reveals gaps when curved fuel plates are inserted into the side panel. These gaps are sealed through Ansys Discovery integrated geometry editing tools, in order to create the fluid volume using subtraction operation.

Imprinting

Imprinting detects coincident faces between bodies and imprints them onto the coincident face. The contact regions will be the same shape, and the resulting mesh on each face will be similar. This can be helpful for analyzing the heat conduction between two surfaces.

Defining fluid

The CAD model only defines the solid components. The fluid volume can be defined by Boolean operations provided by Ansys. From the user end, the surfaces that enclose the fluid region (*i.e.*, inlet and outlet fluid surfaces) should be specified. In addition, a seed surface (internal surface that the fluid flows on) is specified.

Fuel and cladding interference: In each fuel plate, the fuel meat and cladding are defined in the CAD model such that the volume of fuel meat is fully interfered with the cladding volume. To fix the overlapping problem, a subtraction of fuel meat from cladding is done to obtain the true cladding volume.

4. Fuel Element Heat Transfer Simulation with Ansys Discovery

The physics of heat transfer for fuel elements involves heat conduction in the solids of fuel and cladding, and heat convection in the fluid that cools the fuel plates. In this case study, the modeling and analysis process of a conjugate heat transfer model of a fuel element has been carried out using Ansys Discovery as shown in the following sections. A dedicated educator resource on teaching heat transfer fundamentals with Ansys Discovery can be accessed at this link: Lecture Unit: Introduction to Heat Transfer with Ansys Discovery while instructional videos on how to set up conjugate heat transfer simulations can be found at this link: Fluid Solid Heat Transfer Tutorial.

4.1 Material definition and boundary condition

Starting from the simplified CAD model described in Section 3, the materials are defined in the problem first. The fuel cladding and side plates are all aluminum, which is an available material in Discovery. The fuel meat is made of intermetallic uranium aluminide (about 85% of aluminum). To model the alloy, a pre-defined aluminum material is modified by its density, thermal conductivity and specific heat capacity, according to Ref. [1], as shown in Table 1.

Table 1. Properties of uranium aluminide

Property	Value
Density [g/cc]	3.0701
Thermal conductivity [W/(cm·°C)]	1.7
Specific heat capacity [J/(kg·°C)]*	435

* Use rule of mixtures to calculate specific heat of alloy:
Aluminum specific heat: 500 J/(kg·°C) w/o 85%
Uranium specific heat: 67 J/(kg·°C) w/o 15%

For the boundary conditions, we apply a typical convection coefficient (3000 W/m²·°C) to the outer surfaces. These surfaces are supposed to be cooled by the water outside the fuel element, which is not included in single fuel element model.

4.2 Setting up heat sources and fluid flow

For this demonstration problem, uniform heat sources are assigned to the 18 fuel meats inside the fuel clads. This can be done by the following steps: 1) hide the fluid and clad volumes; 2) triple click a fuel meat body to select it; 3) use the power select tool to simultaneously select all the 18 bodies of fuel meats, as shown in Figure 5; 4) Apply a total heat of 55.6kW (2MW/36 fuel element) to the selected bodies.

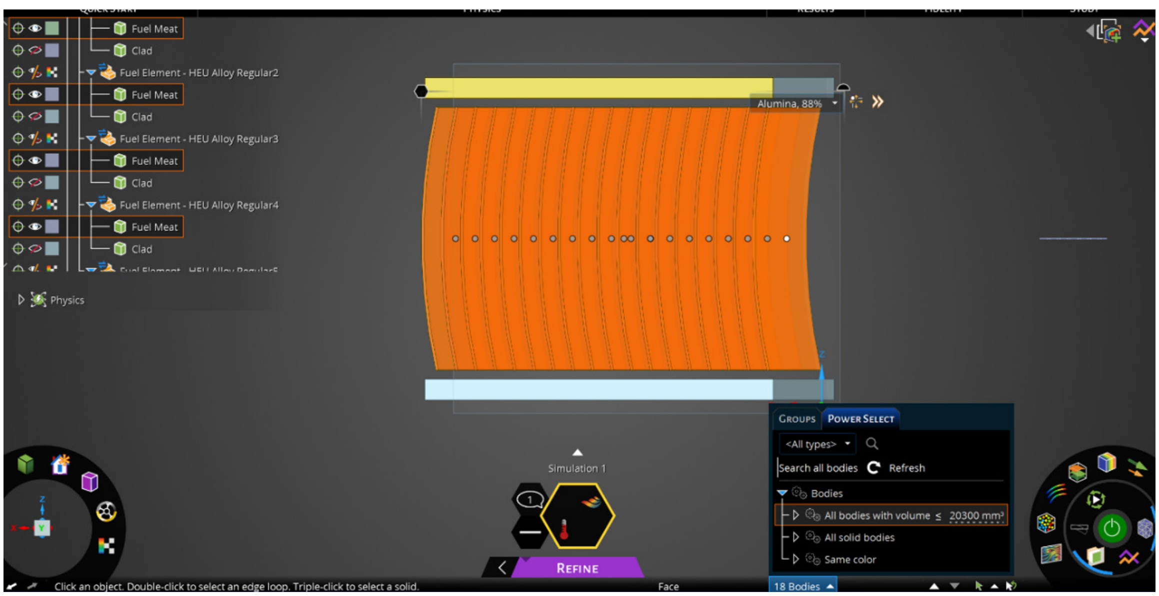

Figure 5. Power select all 18 fuel meat bodies

The fluid flow is set by specifying the inlet temperature to be 40 °C, *i.e.*, the pool temperature, and the coolant mass flow-rate (per fuel element) to be 2.44 kg/s [2].

4.3 Solid connections and fluid-solid interfaces

To perform the heat transfer calculation, solid connections (for conduction) and fluid-solid interfaces (for convection) should be identified. This information is automatically processed by Discovery according to a parameter "detection distance" – the connection of two surfaces is detected if they are within the detection distance. In our problem, since the scale of fuel meat/cladding are less than 1mm, a smaller "detection distance" must be used. Otherwise, those surfaces without direct connection (*e.g.* an inside fuel meat surface and an outside cladding surface adjacent to water) would be recognized as connected.

As shown in Figure 6, the maximum detection distance is set to 0.1mm. We can manually count the number of solid-fluid interfaces, then click the "no grouping" option and compare it with the number detected by Discovery. For example, there are 16 inside fuel plates with a recess at the top and bottom of the fuel element (inward and outward as in the figure), and two side fuel plates and two housing plats that have the same length as fluid, so the total number of solid-fluid interfaces should be 16 x 4 + 2 x 1 + 2 x 1 = 68. This number is consistent with the number detected by Ansys as shown in the figure. The confirmation can be done similarly for solid connections.

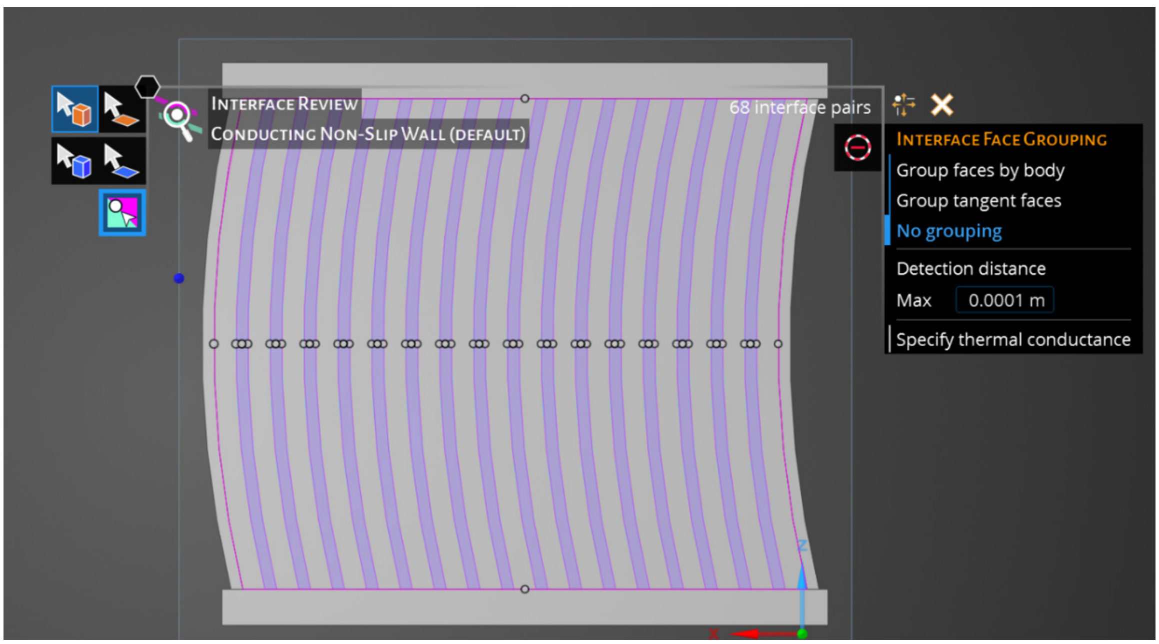

Figure 6. Setting fluid-solid interfaces

4.4 Solver and Meshing

The high-fidelity Refine mode, which uses the same solver technology as Ansys flagship software Fluent, has been used for this problem. In particular, the conjugate heat transfer problem was solved with a pressure based turbulent Omega SST solver which is the standard setting in Ansys Discovery. It is recommended to use the high fidelity "Refine" mode for this problem as it provides automatically local mesh refinement to accommodate the thin fuel plates (less than 1mm thickness of the fuel meat). Figure 7 shows the meshing of the full problem. It includes more than 20 million cells and requires substantial computer memory to solve. Therefore, as shown in Figure 8, we provide 3 versions of the model, *i.e.*, the full problem, the half-geometry with symmetry, and a simplified model with only 2 fuel plates for users to choose based on the available computing resources.

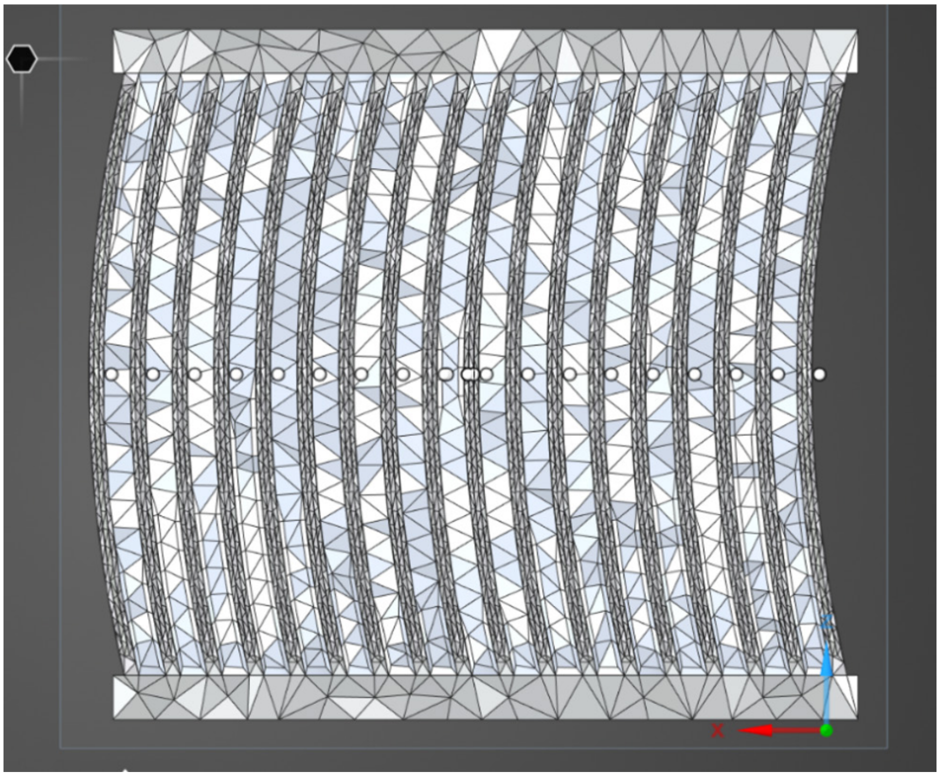

Figure 7. Problem meshing

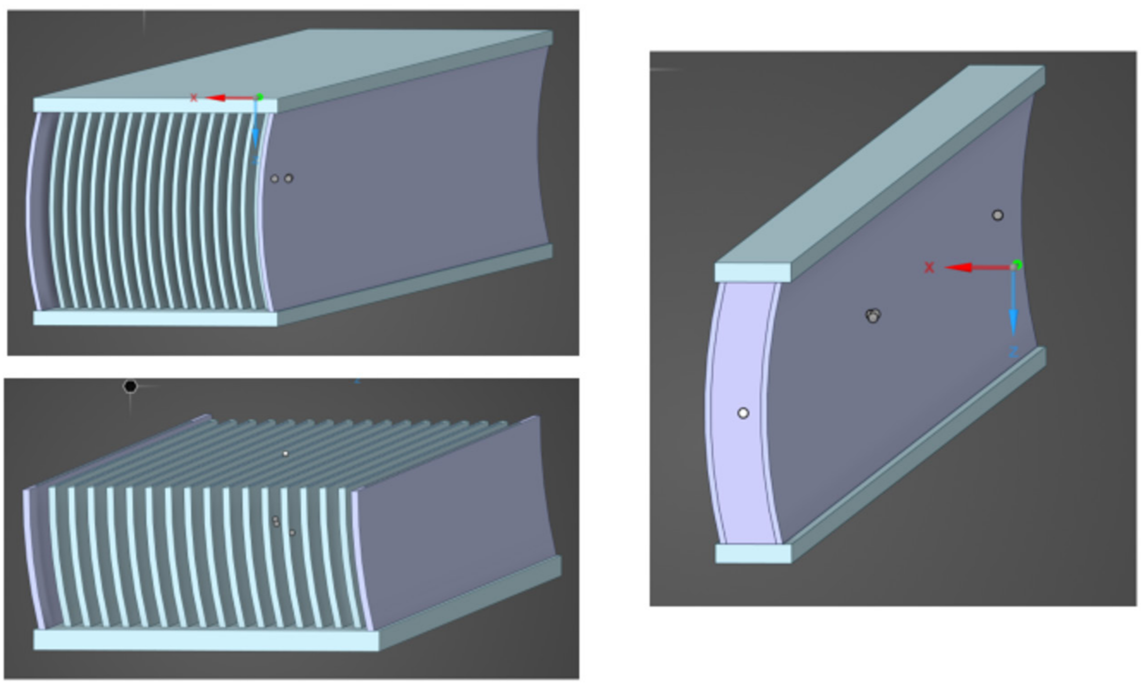

Figure 8. Three calculation models of the problem

4.5 Results of heat transfer

The fuel element is viewed in Figure 9 such that the water flows from the bottom left side (top of fuel) to the top right side (bottom of fuel). The temperature distribution of the model of the half-geometry with symmetry is presented. A uniform heat source was used in our case as a demonstration calculation, so both coolant and fuel temperatures monotonically increase through the fluid flow direction. In realistic reactor core conditions where the peak power shows up near the center of the fuel element, the maximum fuel temperature will instead be present near the location of peak power, although it would not be the same location due to the change of the coolant temperature. Note the maximum fuel temperature is significantly lower than the fuel temperature in commercial nuclear reactors (more than 1000 °C), since the purpose of the experiment reactor is to provide irradiation environment rather than power output.

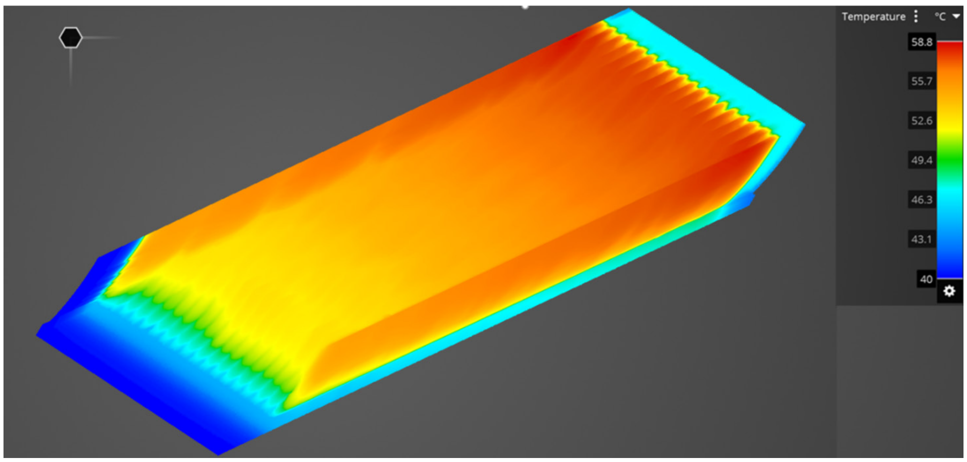

Figure 9. Temperature distribution of the half-geometry model with symmetry

5. Recommended further analysis with Ansys Discovery

For heat transfer calculations of practical nuclear reactor applications, the heat sources are often non-uniform due to the non-uniform distribution of fission energy release in the system. In our case, that means different heat sources should be assigned to different fuel plates. Also, there is a strong axial dependence of heat source along the long side of each plate, and it would be needed to break each plate into different bodies to assign a different heat source to each of them. This would be a tedious and error-prone process if everything were done by hand. The script editor recently added in Discovery may be leveraged to potentially set up the non-uniform heat sources. Figure 10 shows an example of writing a short python script to assign the uniform energy sources that is equivalent to what was done in Section 4.2. Note we loop over the 18 fuel plates, where the component x is each fuel plate and the “Bodies [0]” is the fuel meat. Such script can be adapted to handle non-uniform heat sources, *e.g.*, processed from files, which can be obtained by neutronics calculations. To learn more about the scripting capabilities the free course Scripting in Ansys Discovery is available on the Ansys Innovation Space.

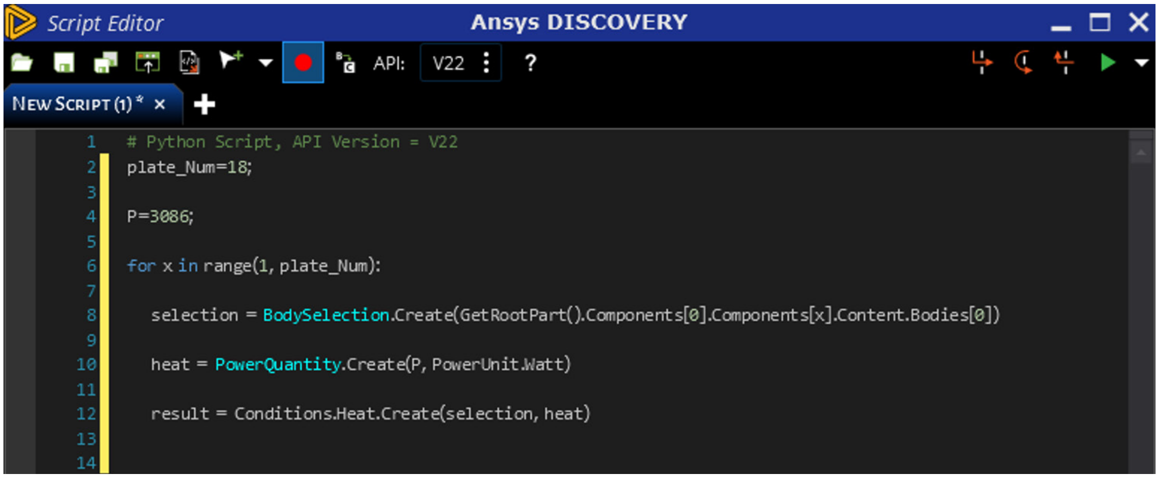

Figure 10. Example of using script editor for heat source assignment

6. Conclusions and Virtual Reality Class at the University of Michigan

This case study demonstrates how a conjugate heat transfer analysis can be performed for a fuel element in an experimental nuclear reactor, including geometry importing and simplification, meshing, simulation and post-processing, all being carried out using Ansys Discovery.

The high-fidelity heat transfer results calculated by Ansys Discovery have been integrated into the Extended Reality (XR) Nuclear Reactor Laboratory for the students at the University of Michigan to explore the reactor in the XR space. Students can 'see' the temperature distributions in a detailed level when 'swimming' inside the reactor core. Such details could help students understand more complex physics, for example, that the temperature hot spot sometimes differs from the power hot spot due to the fluid heating up along the flow channel. The simulated results can be easily visualized in the Unreal Engine used in the XR clinic for the project. Ansys Discovery is able to export the glTF/glb format, which can be directly imported by Unreal Engine. More information about the XR Nuclear Reactor can be found at this link: XR Nuclear Reactor.

The measurement techniques simply do not exist to capture this level of detail everywhere in a physical reactor. Consequently, incorporating simulated results into the XR environment provides important insights for student education into the dynamic responses of such systems. The importance of this type of internalized knowledge for a nuclear engineer cannot be overstated. More broadly digital twins and the development of cyber-physical systems is a growing area of interest in several fields of engineering. Wherever there is educational value in having students interact with a system that has any of the characteristics of being hazardous, expensive, remote, or otherwise exists on very large space and time scales (*e.g.* galaxies, climate change, *etc.*) is well suited for an XR environment to make the educational experience more active and engaging.

References

[1] Peacock, H. B., and Frontroth, R. L. Properties of aluminum-uranium alloys. United States: N. p., 1989. Web. doi:10.2172/5462232.

[2] Hartman, M., Development of a Model Characterizing Heat Transfer in the MTR Fueled Ford Nuclear Reactor. en. Tech. rep. University of Michigan, 1998, p. 10.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to structures, fluids, or heat transfer (physics areas supported by Ansys Discovery).

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at education@ansys.com.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com