Thermal-Fluid Modeling of HFIR Using RELAP5 and COMSOL, Hot Spot Influence On Wall Temperature

Lee Tschaepe

ABSTRACT

Thermal-fluid simulations of a HFIR (High Flux Isotope Reactor) sub-channel are developed using the finite difference code RELAP5 and the finite element code COMSOL. A 1-D analytic solution is determined in order to provide a baseline for comparison between the COMSOL and RELAP5 predictive outcomes. A typical HFIR flux profile is considered, and a HFIR flux profile with hot spot. Channel bulk temperature profile, and the clad/coolant interface temperature profiles predicted by the analytic solution, RELAP5 and COMSOL are compared. The comparisons show the influence of multi-dimensional fuel conduction, and thermal boundary layer development and axial thermal diffusivity in the coolant, all modeled in COMSOL, on the wall temperature prediction near the hot spot. In this way the benefit of using the more complex multi-physics COMSOL simulation method can be quantified.

INTRODUCTION

The HFIR is a beryllium-reflected, light water cooled, high enriched uranium (HEU) fueled research reactor. Figure 1 (All figures will be displayed in the appendices) is a dimensioned cut away view of the current HFIR HEU core. This figure is taken from McLain, 1967. The height of the core shown in Fig. 1 is 0.6096 m. The fueled height is 0.508 m, with the fueled region centered along the core height. The core is composed of two elements. The outer diameter of the outer element is 0.42 m. The inner element contains 171 fuel plates. The outer element contains 369 fuel plates. Fuel plate thickness and spacing are each 1.27 mm. The current operating power of the reactor is 85 MW and the heat load attributable to the fuel plate is 80.7 MW. The remaining heat is deposited in the target, control cylinders, and reflectors (Morris and Wendel 1993).

Figure 2 is an enhanced view of a radial slice of the current HFIR core. The cladding in the HFIR fuel plates remains predominately uniform through the fabrication process; while the poison-bearing region and the fuel are graded. The poison, B_4C, is only present in the inner element fuel plates. The fuel and poison grading is shown in Fig. 2. The fuel to poison ratio varies from near zero to greater than 0.9 based on the relative radial position within each fuel plate. The direction of the coolant flow in Fig. 2 is into the page. It is important to note that the thickness of the fuel plate and coolant channel is greatly enhanced in Fig. 2. The combined half-fuel plate and half-coolant channel thickness is 1.27 mm. This is roughly equivalent to 10 sheets of paper.

The poison grading shown in Figure 2 will be neglected in the sub-channel analysis and the COMSOL and RELAP5 simulations. Additionally, the involute shape depicted in the radial slice will not be modeled. Instead, a uniformly thick half fuel plate and half coolant channel will be the subject of the analytic development and subsequent COMSOL and RELAP5 simulations. Figure 3 is a graphical representation of the relation between the radial slice of the HFIR fuel plate depicted in Figure 2 and the COMSOL and the RELAP5 simulated domain. The fuel region will incorporate axially varying power densities. Figure 4 demonstrates the modeled power density profile from the channel inlet to the channel outlet.

A hot spot with amplification factor 1.2 is also incorporated near the flux peak in the nominal HFIR power profile, and the influence of this spot on bulk temperature and

wall temperature are examined using all three methods. The influence of axial conduction, boundary layer development in the coolant, and thermal diffusivity in the coolant on the wall temperature prediction is evident in the COMSOL simulation. This comparison shows the impact of these combined effects and helps determine when use of these more complex computational approaches is justified.

SCENARIOS

Two thermal-fluid models are developed: A nominally heated HFIR sub-channel, and a nominally heated HFIR sub-channel with a hotspot near the flux peak at the core mid-plane. A diagram of a nominally heat HFIR sub-channel is presented in Figure 5. Each fuel regions has a power density and they are enumerated in Figure 4. The channel hydraulic diameter is 0.00248m. The flow in the sub-channel is 15.8 m/s in the negative y-direction. Following a fluid element's progression through the sub-channel; the fluid element's properties vary with the local bulk temperature. In order to capture this variation the density, specific heat, thermal conductivity, and viscosity are functionalized over the temperature range of interest. These functionalized material properties and their corresponding r-square values are displayed in Figures 6-9. RELAP5 and COMSOL both use higher order functions to model the coolant properties. The clad conductivity and fuel conductivity are simulated in the analytic and COMSOL models as constant values, 181.3 W/(m*K) and 176.95 W/(m*K) respectively. Due to the fact that the RELAP5 model is based on a model built by Morris and Wendel the clad and fuel conductivities are functionalized for the RELAP5 model.

THE ANALYTIC MODEL

Performing a basic energy balance across the heated channel, the bulk temperature change across the channel can be determined.

$$Q_i = \dot{m} c_{p_i} (T_{out} - T_{in}) \qquad \text{Eq. 1}$$

where Q_i is the local nominal core power, $\dot{m}$ is the nominal mass flow rate through the fuel region, c_{p_i} is the local specific heat for water evaluated at the coolant inlet temperature, T_{out} is the coolant exit temperature, and T_{in} is the coolant inlet temperature. The coolant mass flow rate is 140.83 kg/s and the coolant channel inlet temperature is 321.9 K. In order to fully solve Equation 1, the functionalized specific heat must be employed.

$$y = 0.445238x + 4037.583333 \qquad \text{Eq. 2}$$

where *x* is the exit temperature and y is the local specific heat. The coolant bulk exit temperature is determined to be 343.9 K. Determination of the clad/coolant interfacial temperature is achieved,

$$q'' = hA_{ht}(T_{wall} - T_{Bulk}) \qquad \text{Eq. 3}$$

where q" is the local heat flux, h is the heat transfer coefficient, and A_{ht} is the local area for heat transfer. , the heat transfer coefficient between the cladding surface and the coolant flow is examined. The equation for calculating the heat transfer coefficient is:

$$h = \frac{k(Nu)}{D_H} \qquad \text{Eq. 4}$$

where k is the coolant thermal conductivity, Nu is the Nusselt Number, and D_H is the hydraulic diameter.

Numerous engineering heat transfer models have been developed to quantify the Nusselt Number in fully developed turbulent internal flows. One of the most commonly used models for determining the Nusselt Number is the Dittus-Boelter model. This model has no explicit correction for thermo-physical property variations across the thermal boundary layer,

$$Nu_{db} = 0.023\,\mathrm{Re}^{0.8}\,\mathrm{Pr}^{0.4} \qquad \text{Eq. 5}$$

where Re is the Reynolds Number, and Pr is the Prandtl Number. The exponent associated with the Prandtl Number is depended upon whether the fluid is heated (0.4) or

the fluid is cooled (0.3). The use of this particular model in the analytic model is convenient because it is also the same model that is used in RELAP5 for determining the heat transfer coefficient for single phase fluid flow. The calculated Dittus-Boelter heat transfer coefficient varies with position within the channel but an average value is ~81,000 W/(m^2K). Equations 1 through 5 constitute the analytic model implemented in EXCEL. This model does not include the pump heating of the fluid, which is near 2% of the enthalpy addition to the fluid in HFIR, but RELAP5 and COMSOL do include these terms.

RELAP5

RELAP5 employs a finite difference method in its solution of the differential equations that govern thermal-fluid behavior. A finite difference method solves differential equations by replacing the differential equation with an approximate difference equation. The validity of this method can be demonstrated by the use of a Taylor expansion in the solution of a first order differential equation. As a consequence of using a finite difference method for the solution to differential equations, two types of error are inherent: round off error and truncation error. The round off error is a consequence of the computer memory limitations and cannot be affected without altering the computer hardware. The truncation error is a product of the domain discretization employed in the solution of a given problem. In order to reduce the truncation error a finer nodalization of the domain space must be employed.

A user of the RELAP5 code is able to interface with the code through the use of various components including: Heat structures, hydrodynamic components, and various types of junctions. Within each of these components is a plethora of options allowing the user to manipulate the code in such a manner as to best model the physics of interest. The components employed in the present simulations consist of the tmdpvol (time dependent volume), tmdpjun (time dependent junction), pipe, sngljun (single junction), and a heat structure. Figure 10 is a graphical representation of the RELAP5 model used in all simulations for flow in a heated sub-channel. The time dependent volume1 acts as a reservoir of fluid that provides the simulation with the necessary initial conditions. This fluid is then transmitted through the tmdpjun into pipe1. Pipe1 is used as a link to

the sngljun1, where the desired inlet flow conditions to the section of interest, pipe2, are initiated. The sngljun1 is used to set the inlet pressure loss coefficient. Pipe2 is the sub-channel of interest and is directly related to the coolant channel control volume displayed in Figure 4. Pipe2 is nodalizied into seven parts. Sngljun2 carries the fluid out of pipe2 while applying the pressure loss coefficient at the pipe exit. Pipe3 then carries the fluid to the final junction before it enters tmdpvol2, where the simulation is concluded. The heat structure displayed is thermally insulated on its left boundary, allowing heat to only be transmitted through the right boundary and into the fluid within pipe2.

COMSOL

COMSOL is a commercial finite element based equation solver built on the MATLAB programming language. MATLAB is built on the C programming language. Thus, COMSOL is a rather high level programming environment, driven through a graphical user interface that is also accessible through MATLAB and internal scripting language and a batch-mode environment.

The COMSOL finite element method solves the differential equations by breaking the continuum of interest into a mesh with mesh intersection points represented locally by scalar property values. This allows the rendering of the governing differential equation system into a linear algebraic equation system suited to solution using matrix algebra. The size of the matrices being handled during solution is proportional to the number of pieces into which the continuum was partitioned. This can lead to large memory requirements. The finite element method suffers from similar sources of error to those in the finite difference codes.

The COMSOL model employed for the simulation of the nominally heated sub-channel is depicted in Figure 5. The COMSOL model employs three thermal fluid modules to simulate the sub-channel: General Heat Transfer (htgh), General Heat Transfer (htgh2), and k-ε Turbulence Model (chns). The boundary conditions for each package are displayed in Figure 11-13. The General Heat Transfer modules simulate heat conduction in both x and y directions. Two dimensional conduction models are not possible in the current model of RELAP5. The k-ε Turbulence Model allows for the

simulation of the fluid flow in two dimensions. An example of the mesh size used in the COMSOL models is displayed in Figure 14. COMSOL performs integrations over a mesh element by taking a weighted sum of the integrand evaluated at a maximum of 30 points within each triangular mesh element. The number of elements and the number of degrees of freedom in the COMSOL simulation for the nominally heated sub-channel are 133,019 and 886,732.

The hotspot simulations, for both RELAP5 and COMSOL, are setup in an identical fashion to the nominally heated channel with the difference of an additional 0.02m volume near the mid-plane of the channel. The hot-spot model is depicted in Figure 15. This hotspot has an elevated volumetric power of 1.2 times the original local volumetric power.

RESULTS

Each scenario is compared based on the predicted outcome of the bulk temperature profile across the channel and the outcome of the predicted clad/coolant temperature profile. The COMSOL channel centerline temperature is used in place of a bulk temperature, so this value should be near 2% greater than the COMSOL predicted bulk temperature. The analytic solution is also provided for each scenario. The bulk temperature profile and the clad/coolant interface temperature profile for the nominally heat sub-channel are displayed in Figure 16-17. It is important to note that the coolant flows from the right to the left in these figures. For the nominally heated channel COMSOL predicts an exit centerline temperature that is slightly larger than the RELAP5 exit bulk temperature, as would be expected. However, for a majority of the distance through the channel, the COMSOL predicted centerline temperature is less than the RELAP5 predicted fluid bulk temperature. RELAP5 trends closely to the analytic solution for the bulk temperature profile. RELAP5 prediction of the clad/coolant, interfacial temperature profile is also very close to the analytic prediction. COMSOL required 1354 seconds to solve this scenario, and with Y^+ set to 10 for the first node in the fluid mesh, a recommended value for implementation of law of the wall, COMSOL predicts a heat transfer coefficient well in excess of that suggested by Dittus Boelter, as

indicated by the wall temperature predictions in Figure 17. Doubling the position to the first node in the mesh had very little influence on the wall to fluid heat transfer predicted in COMSOL. RELAP5 required 48 seconds to solve this scenario.

The bulk temperature profile and the clad/coolant interface temperature profile for the sub-channel with a hotspot are displayed in Figure 18-19. It is important to note that the coolant flows from the right to the left in these figures. For the channel containing the hotspot COMSOL predicts an exit centerline temperature that is larger than the RELAP5 exit bulk temperature prediction. However, for a majority of the distance through the channel COMSOL predicts a centerline temperature that is less than the fluid bulk temperature predicted by RELAP5. Figure 20 displays the COMSOL predicted turbulent thermal conductivity at three axial positions. These turbulent conductivity values are consistent with those developed for lateral conduction in the HFIR fuel cooling channel from simulations performed by Ruggles in 1997 using legacy empirical models for this parameter. RELAP5 trends closely to the analytic solution for the bulk temperature profile and for the clad/coolant, interfacial temperature profile. Again, the heat transfer coefficient predicted in COMSOL is dramatically larger than that predicted by Dittus-Boelter.

The disparity in heat transfer coefficient between the COMSOL and RELAP5 makes detailed examination of the influence of the multidimensional fluid flow and heat transfer modeling on the hot spot thermal performance. For this reason, the analytic solution is modified to have heat transfer coefficient equal to that predicted by COMSOL. This simulation of wall temperature with hot spot is compared to the COMSOL prediction of wall temperature with hot spot in Figure 21. Figure 21 indicates the effect of the hot spot is significantly reduced for the circumstances simulated here when the axial fuel conduction and fluid boundary layer development and thermal diffusivity are resident in the simulation.

CONCLUSION

RELAP5 and COMSOL both predict of the fluid bulk temperature with reasonable accuracy. COMSOL predicts heat transfer coefficient well in excess of that

expected using legacy semi-empirical models based on data. Several methods exist to force the COMSOL prediction toward conventional expectations, but these have not been implemented here. With adjusted heat transfer, the influence of axial conduction in the fuel, and boundary layer development in the coolant, along with thermal diffusivity in the coolant effectively reduces the effect of the hot spot on the fuel/coolant interface temperature. However, the details of this multidimensional simulation of the hot spot are not validated. Challenges lie ahead validating these multi-physics simulation techniques to where they are reliable for reactor fuel simulation, safety and licensing evaluations.

REFERENCE

Morris, D. G. and Wendel, M. W. *High Flux Isotope Reactor System RELAP5 Input Model,*ORNL/TM-11647, January 1993.

Ruggles, A. E., "Techniques for Consideration of Spatial Flux Perturbations due to Fuel Manufacturing Tolerances in Plate Fueled Reactors." Proceedings of NURETH 8, pp. 1823-1830, Sept. 30-October 4, 1997.

APPENDICIES

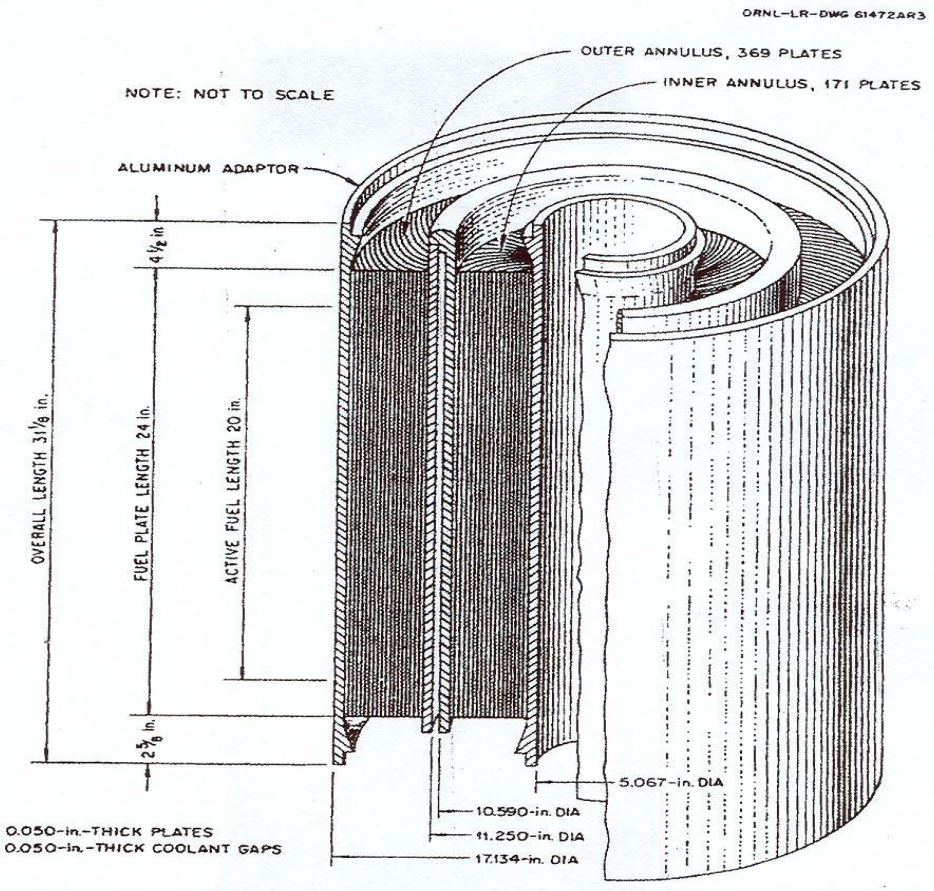

Fig. 3. HFIR Fuel Assembly.

Figure 1: HFIR core dimensions.

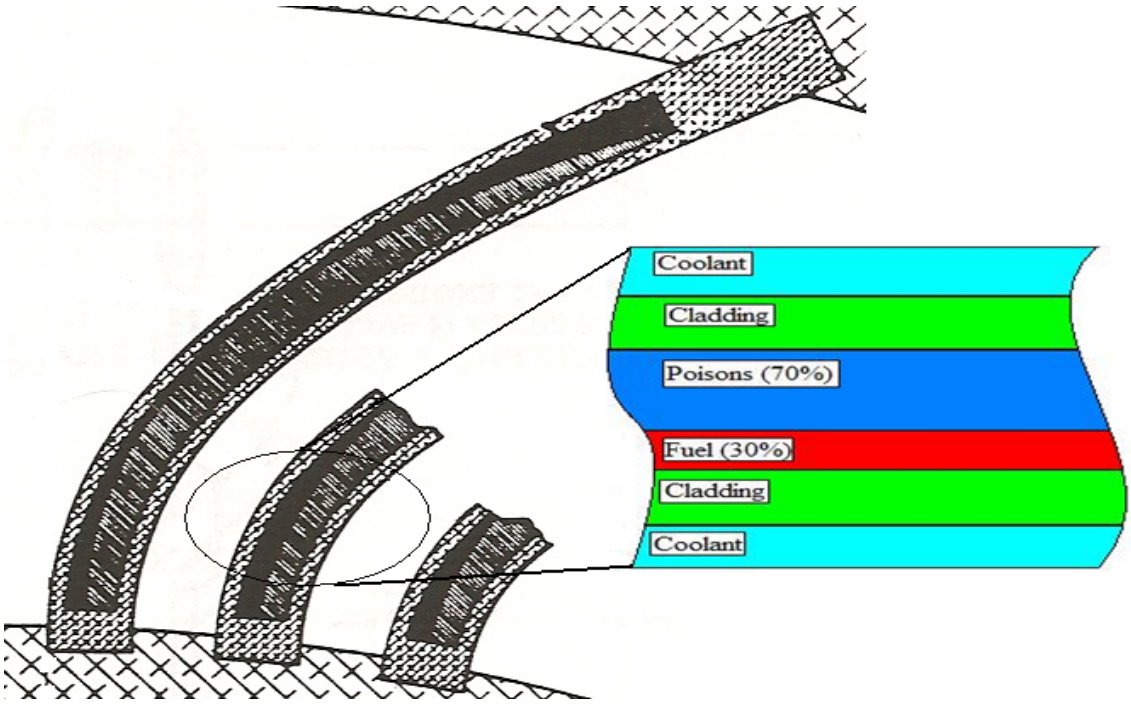

Figure 2: Enlarged view of coolant channels, not to scale.

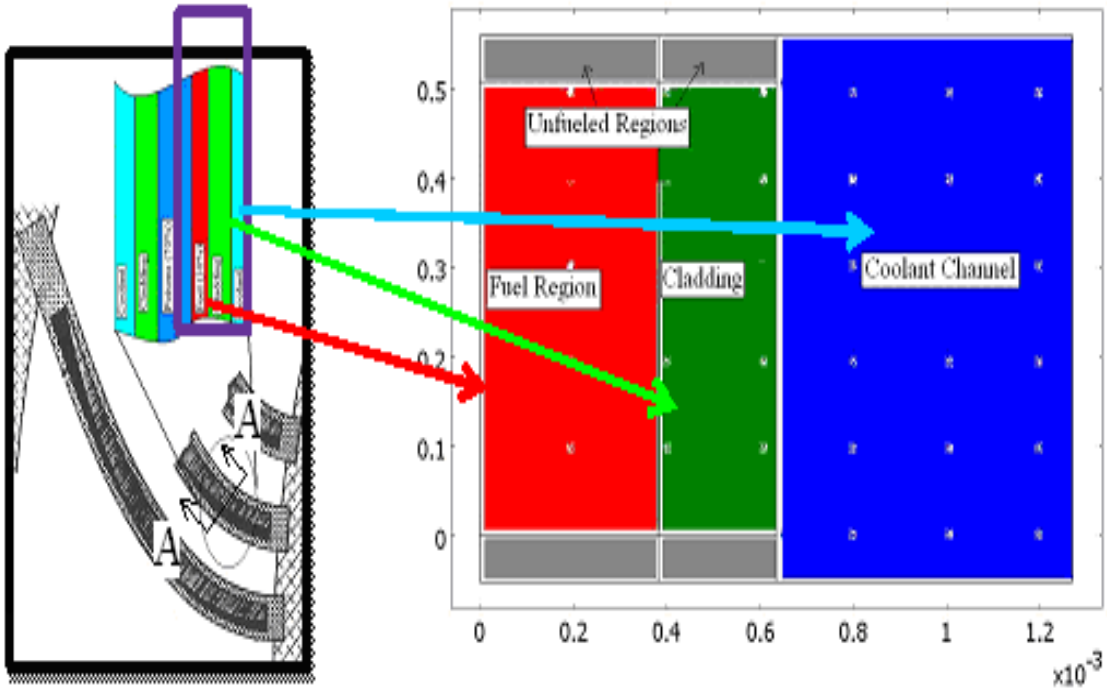

Figure 3: Sub-Channel Model

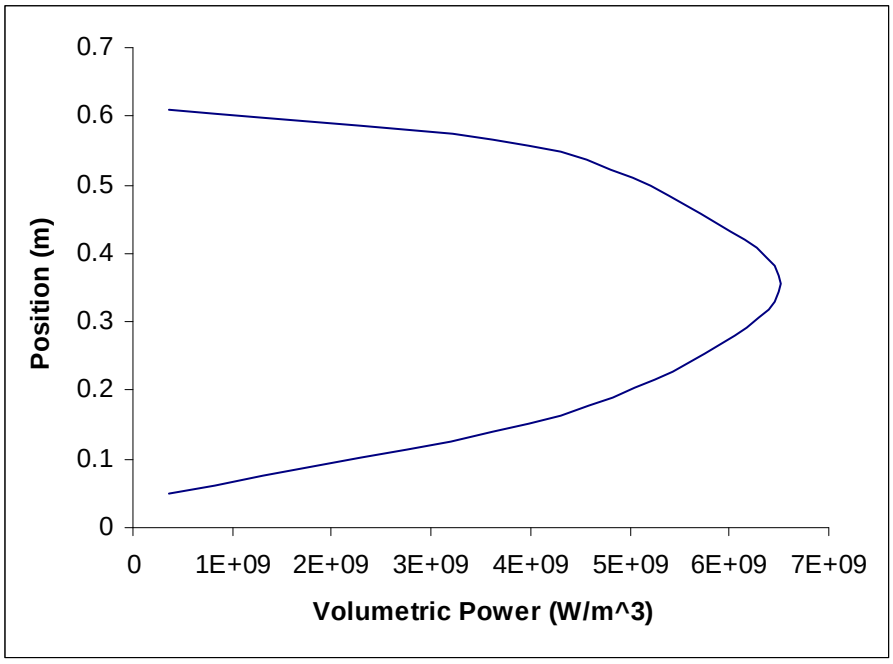

Figure 4: Power Profile

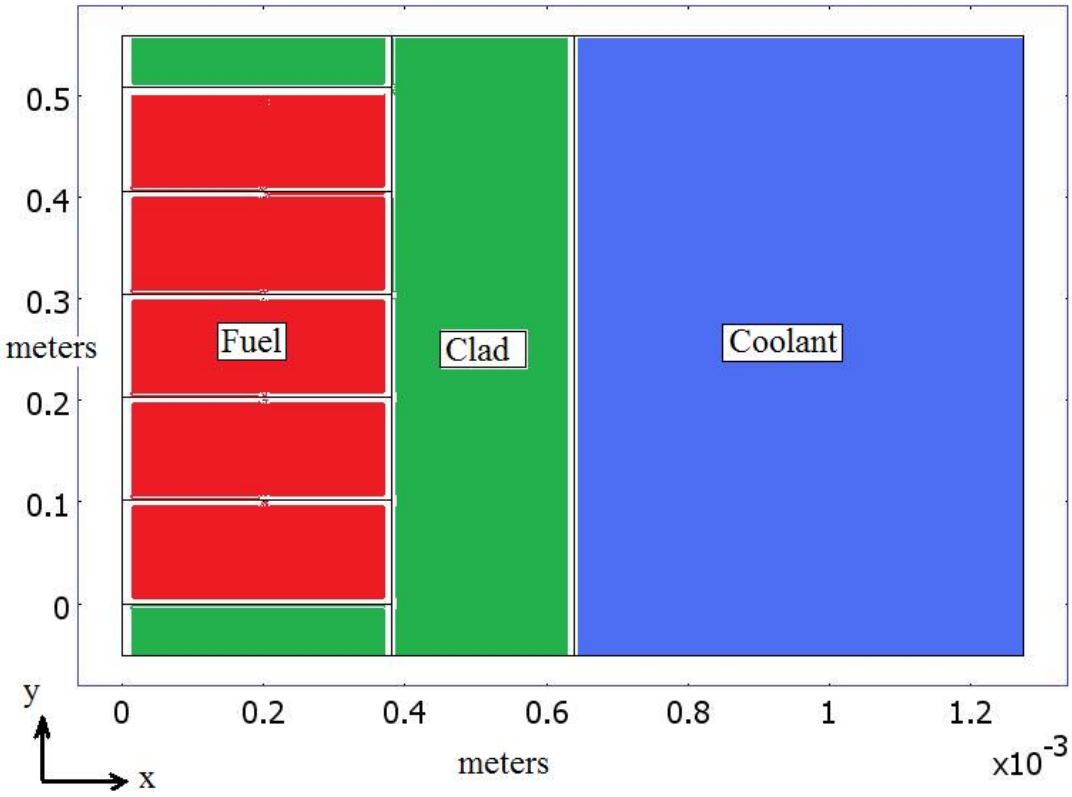

Figure 5: Nominally Heated Sub-channel

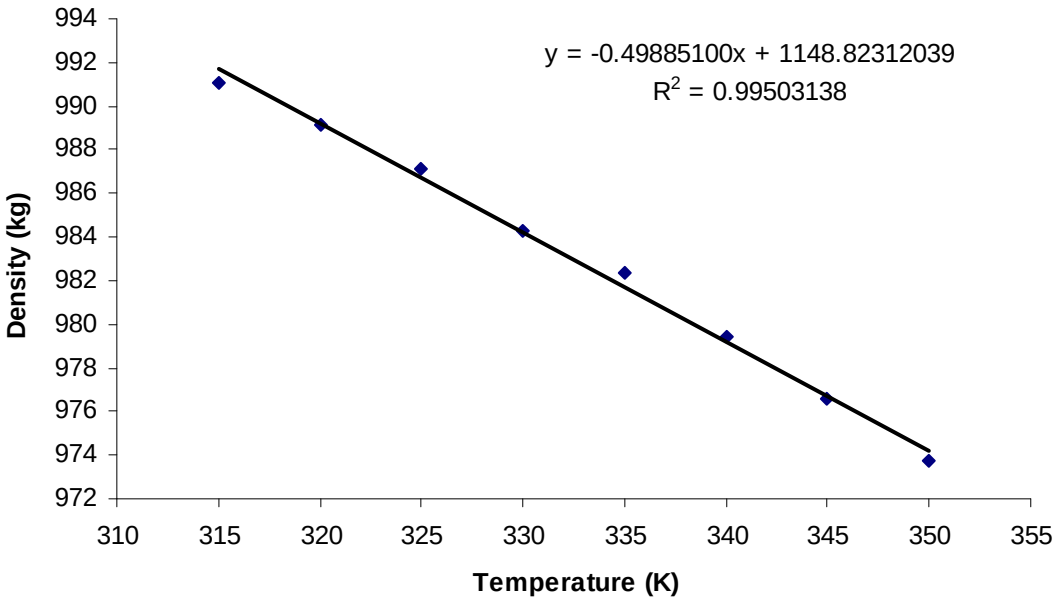

Figure 6: Functionalized Density

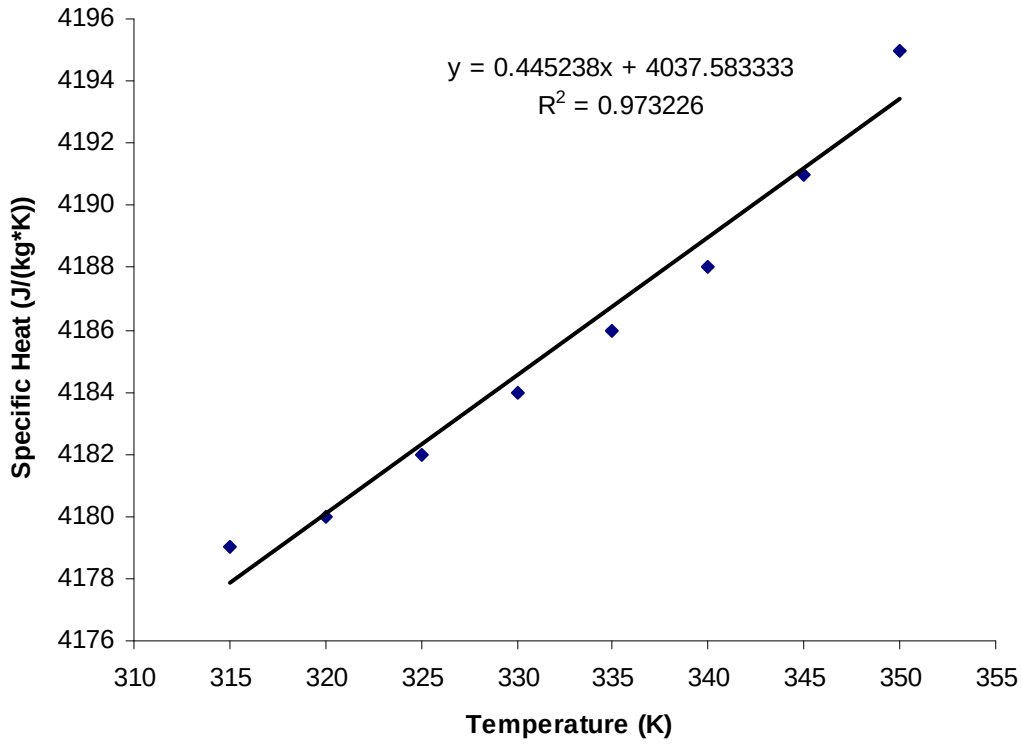

Figure 7: Functionalized Specific Heat

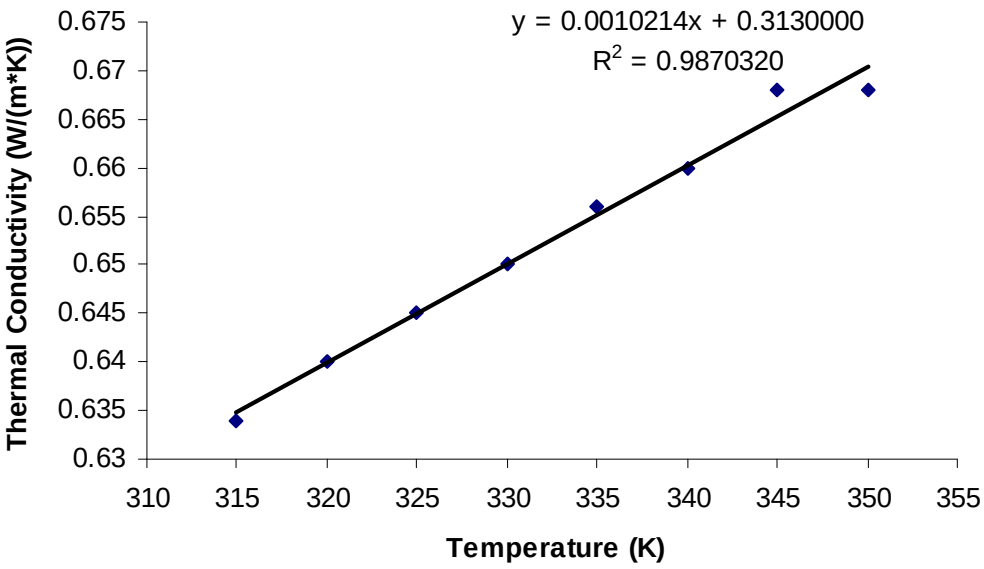

Figure 8: Functionalized Thermal Conductivity

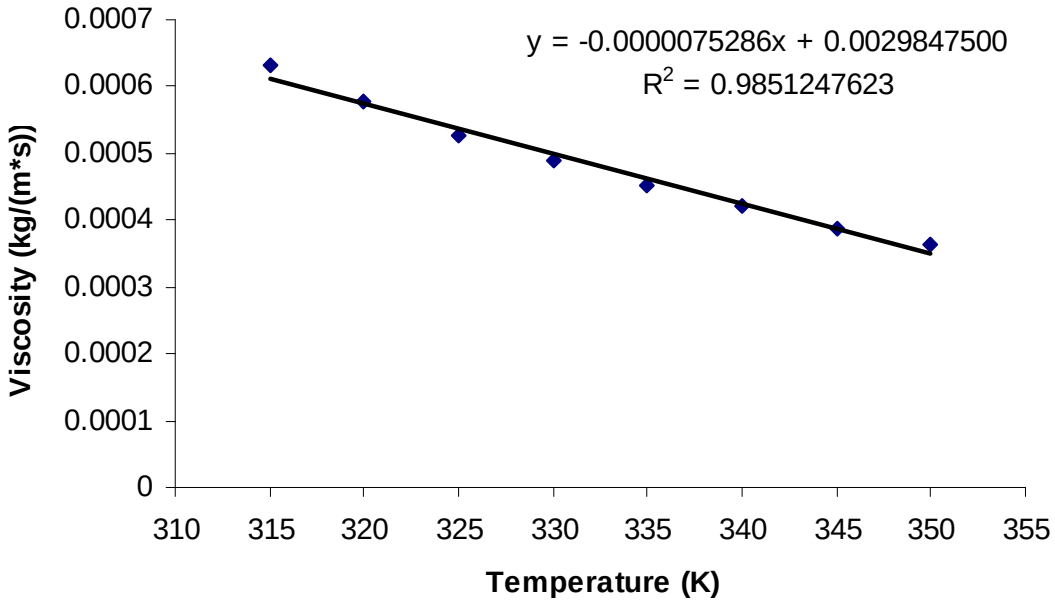

Figure 9: Functionalized Viscosity

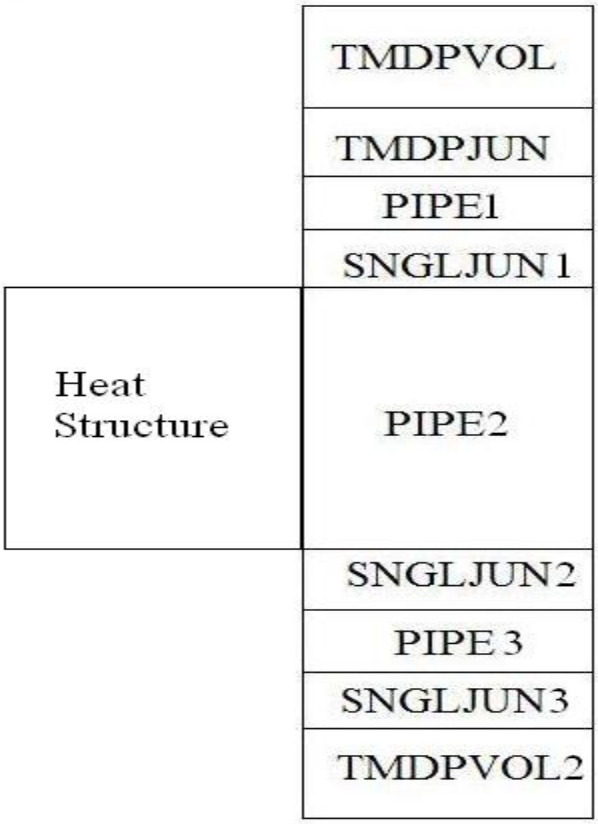

Figure 10: RELAP5 Simulation of Single Phase Flow

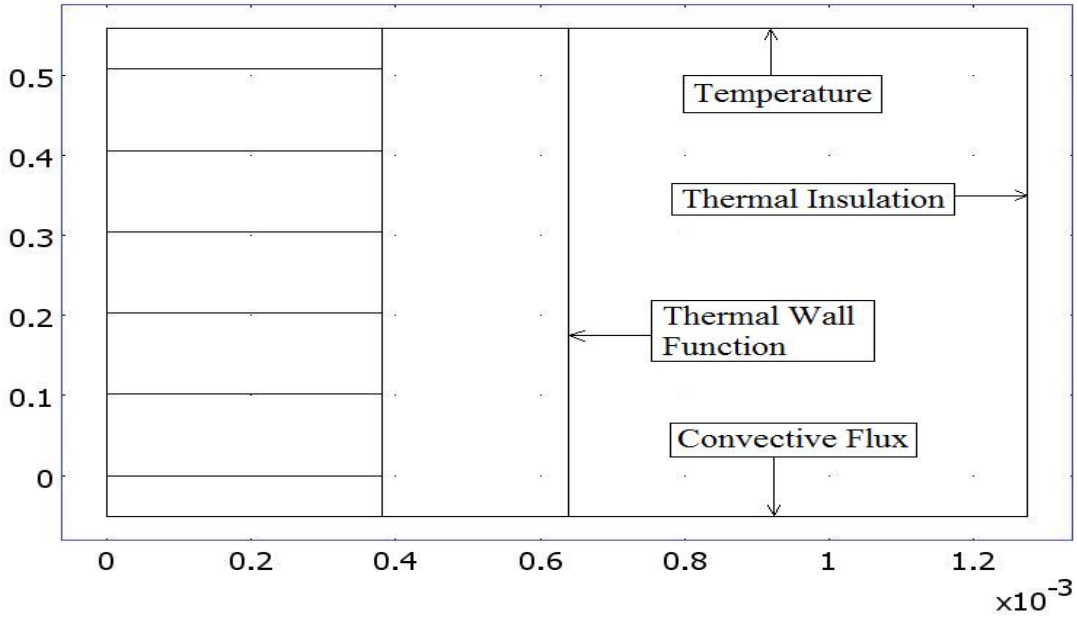

Figure 11: General Heat Transfer (htgh) Fluid Boundary Conditions

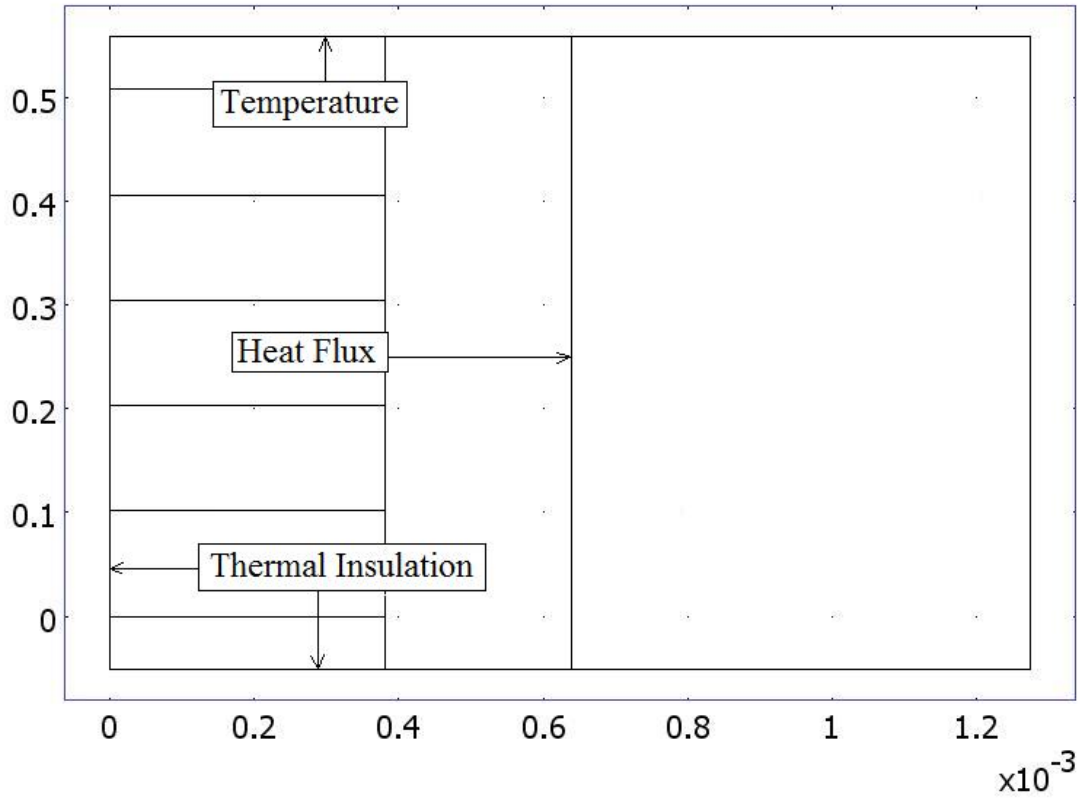

Figure 12: General Heat Transfer (htgh2) Solid Boundary Conditions

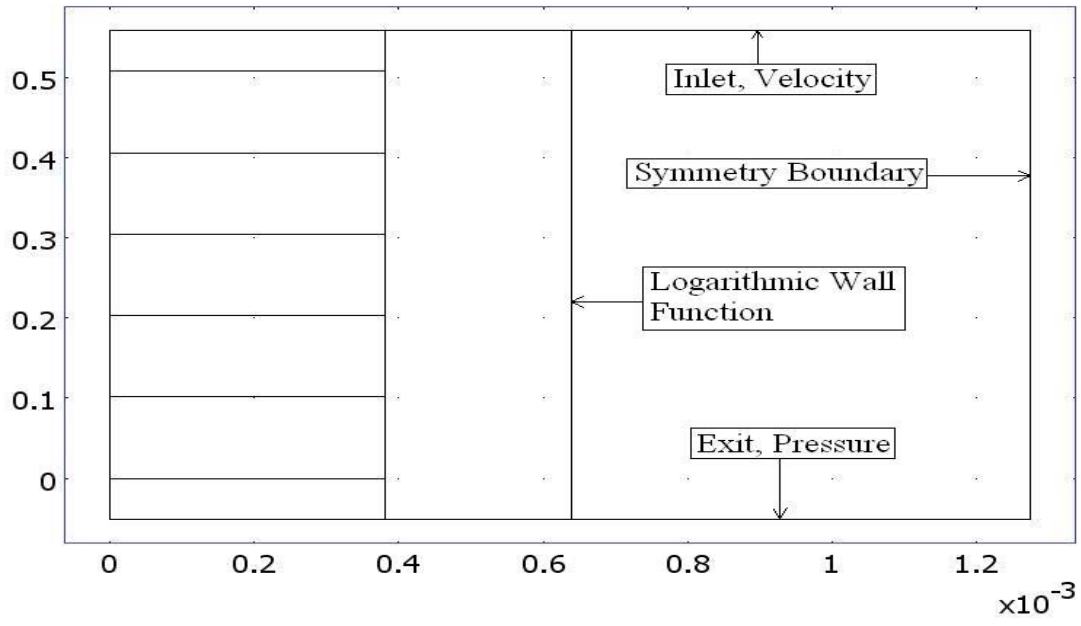

Figure 13: k-ε Turbulence Model (chns) Boundary Conditions

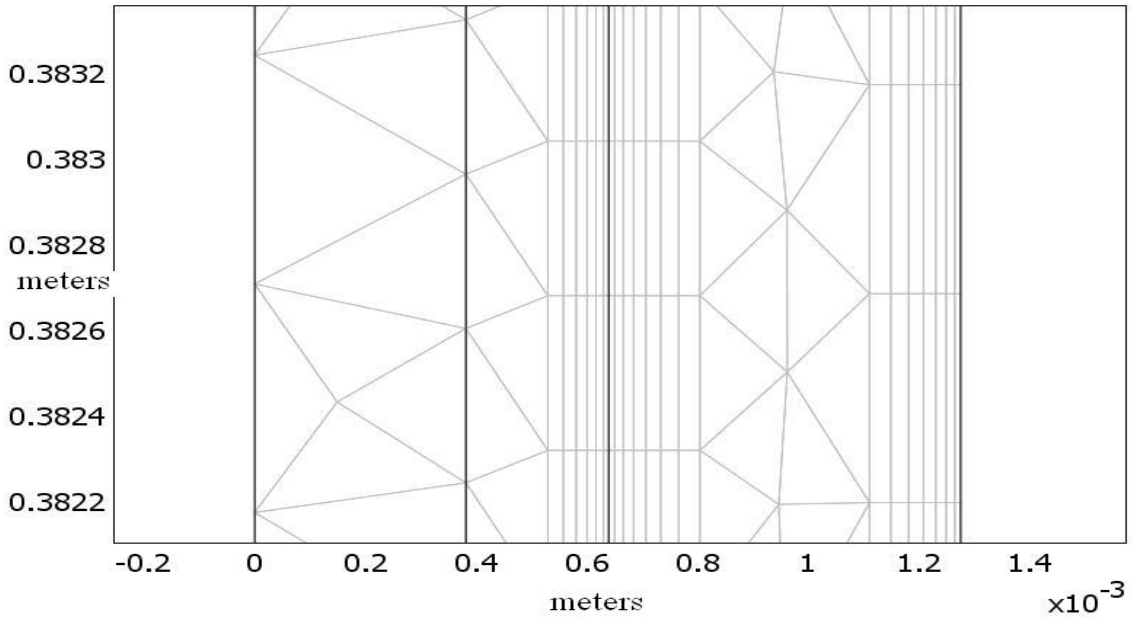

Figure 14: COMSOL Mesh Density

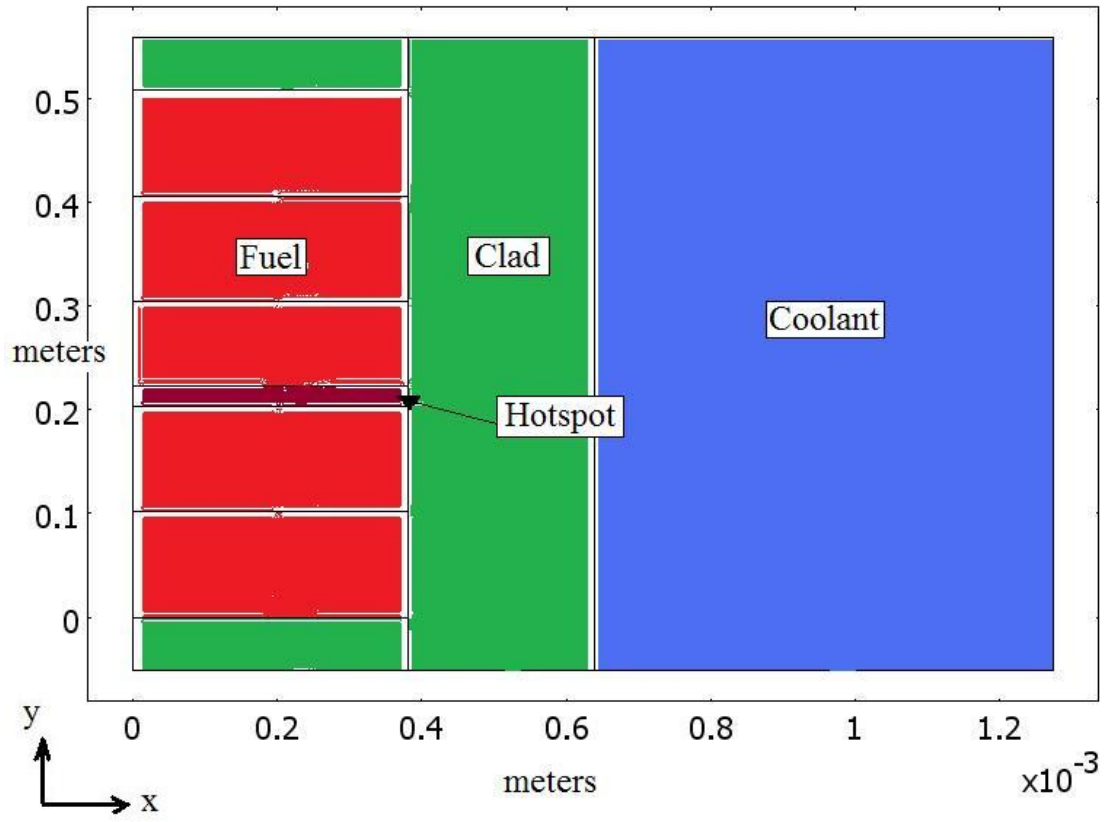

Figure 15: Hotspot Simulation

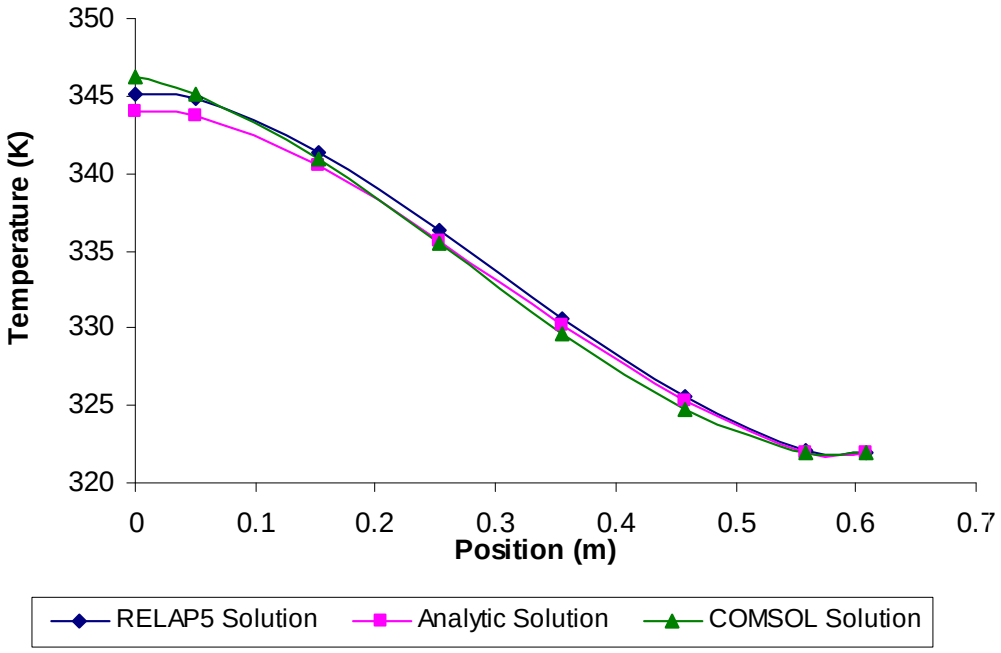

Figure 16: Bulk Temperature for Nominally Heat Channel

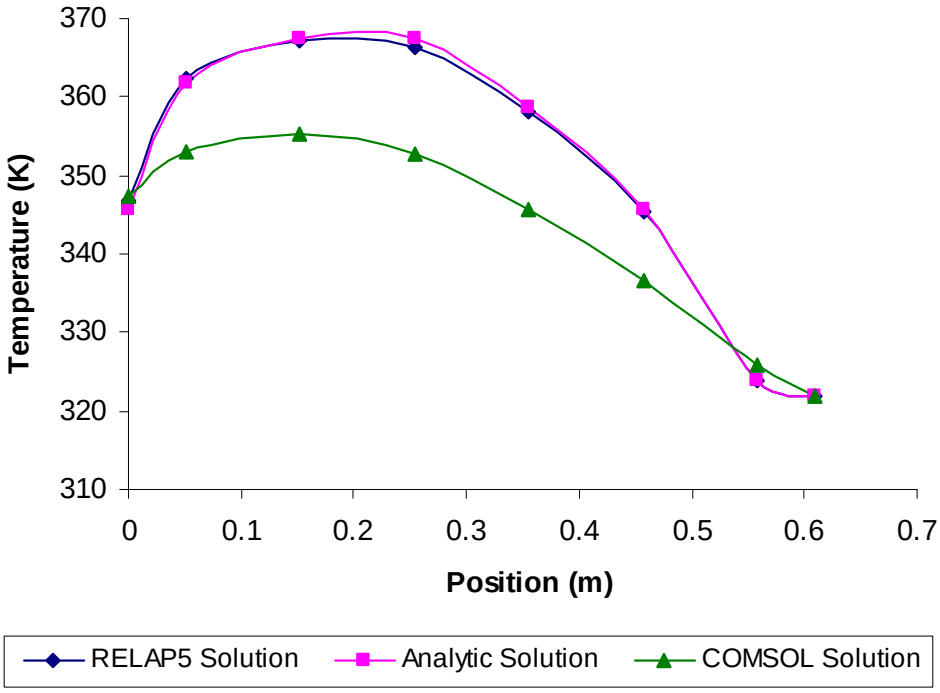

Figure 17: Clad/Coolant Temperature for Nominally Heat Channel

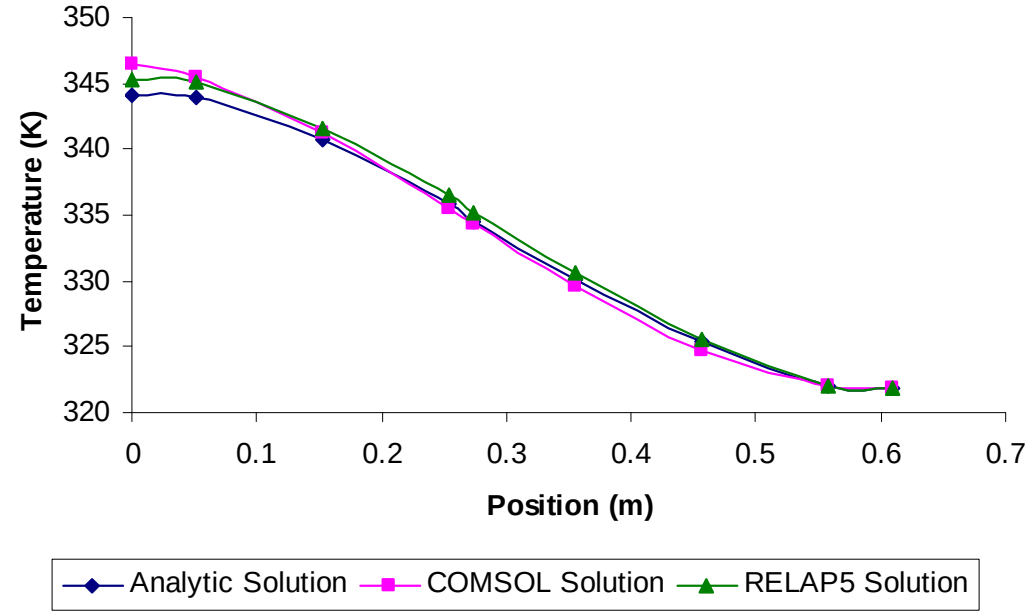

Figure 18: Bulk Temperature for Hotspot Channel

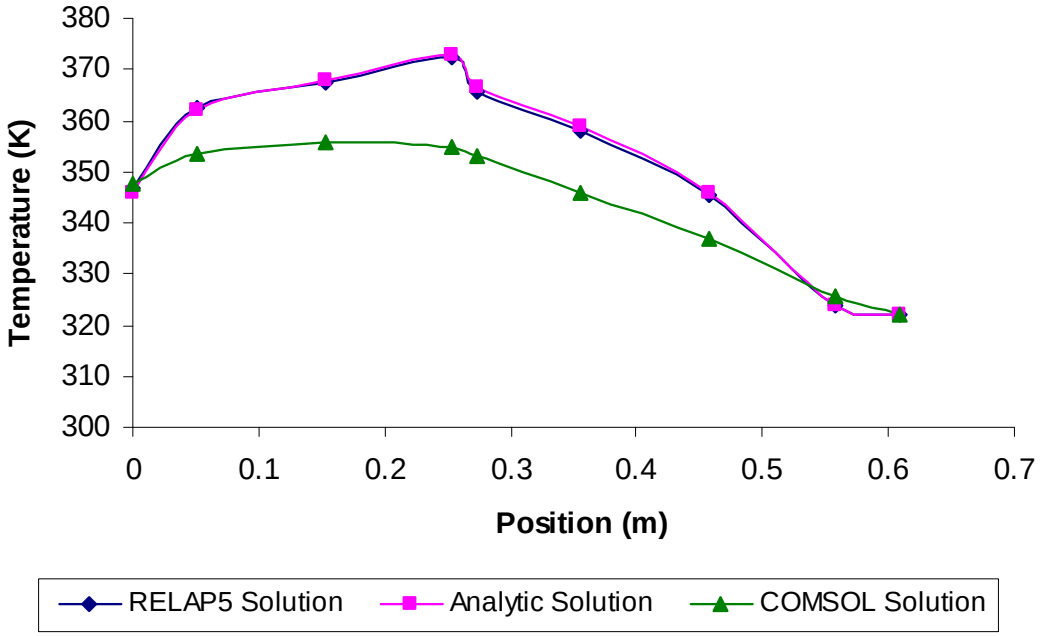

Figure 19: Clad/Coolant Temperature for Hotspot Channel

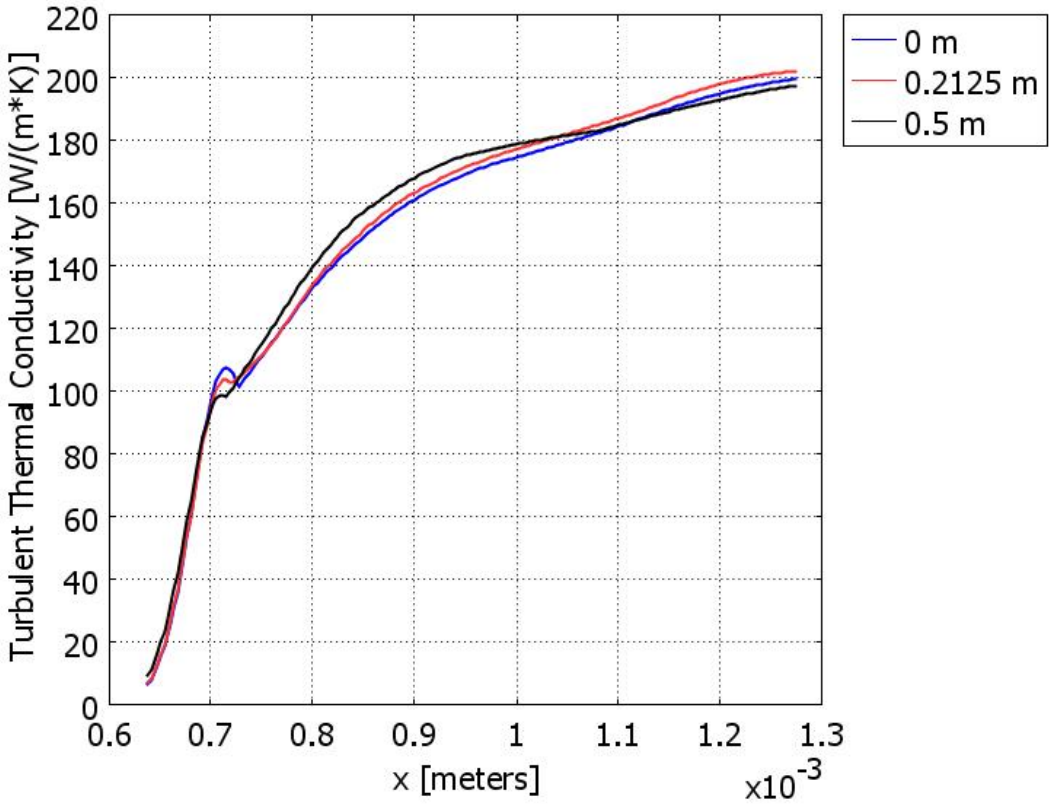

Figure 20: Turbulent Thermal Conductivity, COMSOL

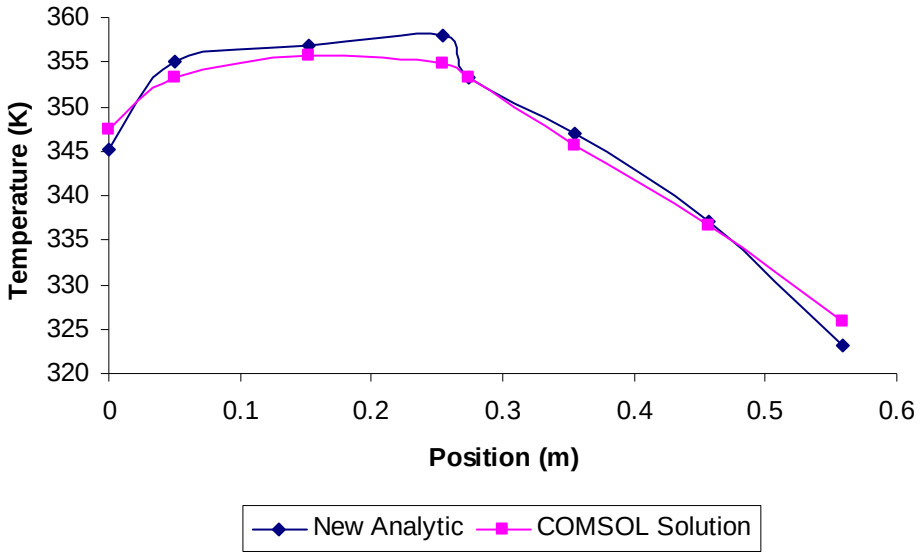

Figure 21: Wall Temperature w/ Analytic Heat Transfer Coefficient Increased by 54000