PSEUDO-3D STEADY STATE THERMAL-HYDRAULIC MODELING OF THE ADVANCED HIGH TEMPERATURE REACTOR FUEL ASSEMBLY

P. Avigni, B. Petrovic
Nuclear and Radiological Engineering, Georgia Institute of Technology
770 State St., Atlanta, GA, 30332-0745, USA
pietro.avigni@me.gatech.edu, Bojan.Petrovic@gatech.edu

ABSTRACT

The Advanced High Temperature Reactor (AHTR) is one of the most promising advanced designs, since it allows low pressure operation, implements passive safety with large safety margins, and heats the coolant to high temperatures resulting in high plant efficiency. The currently considered fuel design employs fuel plates made of TRISO particles dispersed in a graphite matrix, so the fuel density and consequently heavy metal loading is relatively low; several options for improving the fuel utilization and cycle length are under evaluation, among which there are variations of the fuel assembly design and online refueling.
The fuel assembly of the AHTR uses fuel plates enclosed into a hexagonal graphite box, and it is cooled by Flibe, which flows through thin rectangular channels located between fuel plates. This innovative fuel assembly design requires optimization from the thermal and neutronic standpoints.
The work developed recently aims to provide a better understanding of the flow in the channels and the 3D temperature distribution within the assembly. In particular, the following features are being evaluated: power removal by the inter-assembly flow, flow distribution for the intra-assembly channels, effects of the thin internal channels on the 2D temperature distribution of the assembly, maximum fuel temperature. This work will provide indications how to improve the overall design of the assembly, regarding the flow distribution and the geometry of the system. The ultimate aim is to provide a model of the assembly, as simple as possible while providing adequate results, which will be integrated in a full core model for the evaluation of the operating conditions and transients, particularly the online refueling.

KEYWORDS
AHTR, LSCR, FLiBe, TRISO fuel plate, liquid salt thermal-hydraulics

1. INTRODUCTION

This report presents the evaluation of the steady state 3D thermal-hydraulic behavior of the AHTR fuel assembly. Since the problem is relatively large from both thermal and hydraulic standpoints, some approximations will be assumed in the development of the model in order to allow faster evaluation of the results. The paper is structured in two sections: the first section presents the methodology along with the approximations and assumptions of the model; the second section shows the results and the conclusions. The approach is based on a 2D finite volume model of the horizontal cross section of the assembly for the evaluation of the 2D temperature distribution, given the boundary conditions. Neglecting the axial component of heat conduction, the 2D finite volume model is integrated with an axial calculation of the flow distribution as well as the temperature of the coolant for the internal and external channels, allowing a pseudo-3D evaluation of the thermal performance of the assembly.

2. METHODOLOGY

Figure 1 shows the 3D model considered for the calculation. Since the fuel assembly presents a hexagonal symmetry, in order to reduce the computational requirements, only one third of the fuel assembly has been considered. In addition, only the active region of the fuel (5.5 m tall) is represented in the model, this implies that the pressure drop is not the total pressure drop of the core, but the pressure drop of the active region only (which anyways should be the main contribution to the core pressure drop).
Figure 1 shows a drawing of the model, focusing the detail on the horizontal cross section, which includes six fuel plates, seven internal channels, the Y-shaped support and the box wall.
The model accounts also for the power that is transferred from the box walls to the external channels. The total mass flow rate that flows in the z direction splits between the internal and external flow; the fraction of the coolant that flows in the external channels can be changed in order to analyze the effects of the bypass flow on the temperature distribution of the assembly.
The dimensions of the assembly parts refer to the standard AHTR reactor design.[1]

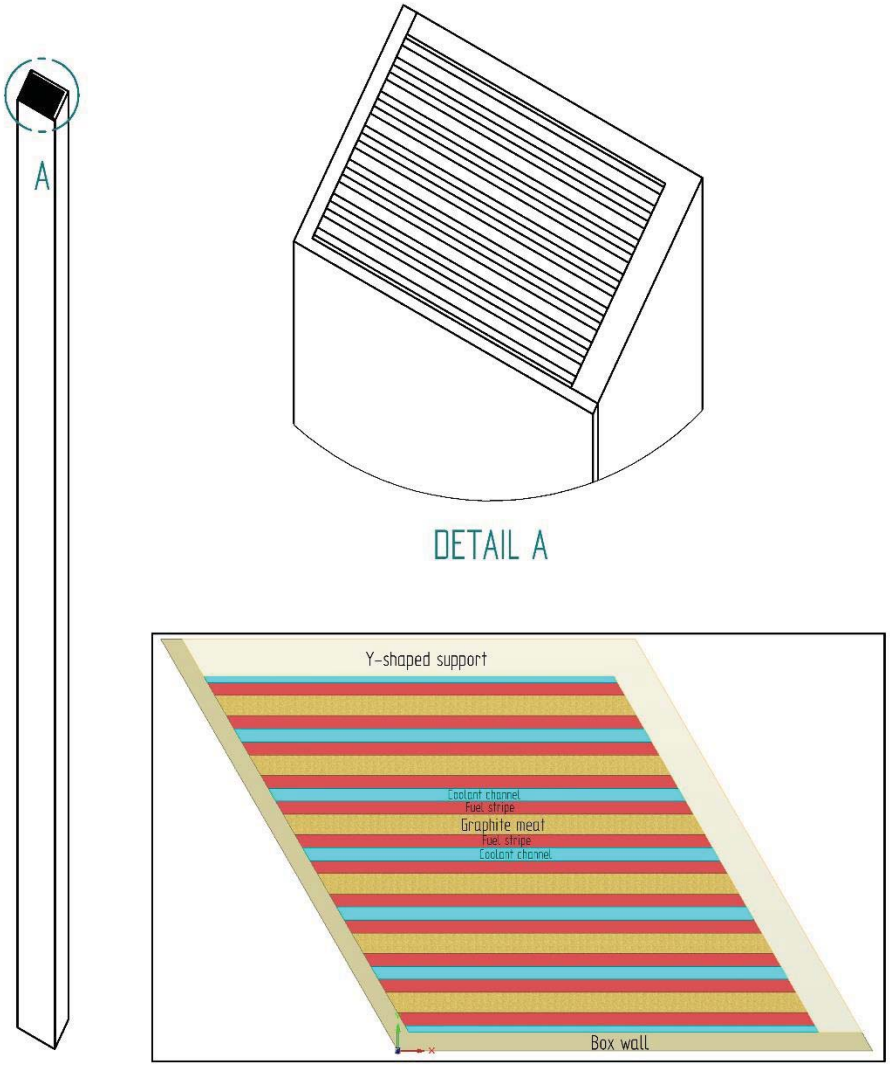

Figure 1. Details of the fuel assembly model

2.1. Assumptions and Approximations

The following sections present the main approximations assumed in the model:

- Neglect axial heat conduction in the thermal structures
- Approximate the coolant as 1D incompressible with constant properties
- Use the fully-developed flow solution for the friction factor (hydrodynamically developed flow)
- Use the fully-developed flow solution for the heat transfer coefficient (thermally developed flow)

2.1.1. Axial heat conduction

The fuel plate of the AHTR is a 3D conducting structure, but the heat conduction prevails in the horizontal plane. This analysis aims to estimate the error that results in treating the heat conduction as monodimensional. If the axial heat conduction is negligible, a much simpler treatment of the heat conduction is allowed, a lot less computationally demanding. We expect the axial conduction to be negligible, since the temperature gradient in the vertical direction is 1000 times smaller than the one in the horizontal one, but we want to obtain a numerical estimate of this effect.
The monodimensional approach is compared to a 2D approach that accounts for axial conduction. The model represents a section of the fuel plate, with the z being the vertical direction (from the inlet to the outlet of the core) and x being the horizontal direction (from the centerline of the graphite meat to the surface in contact with the coolant).
The solution of the heat conduction is performed through a 2D finite volume approach, which in the case of the 1D approximation neglects the heat conduction in the z direction. This is the only difference in the two solutions.
The problem is solved using a MATLAB code; the solution of the 1D linear system is obtained through the inversion function provided by MATLAB, while the solution for the 2D model is obtained through a Gauss-Seidel iterative procedure. This is due to the fact that the exact solution for the 2D model is hard to find, while an iterative procedure is a lot less computationally demanding.
The plate is 5.5 m high and 2.6 cm thick, but the study considers only 1.3 cm thick half of the plate for symmetry. Of the 1.3 cm, 7 mm are fuel and 6 mm graphite meat; the “cladding” (outermost thin layer of graphite) is not considered for simplicity.
The 2D domain is discretized and the following energy balance can be written for the general node (i is the index for the z direction and j is the index for the x direction):

$$k\bullet\frac{T_{i,j-1}-T_{i,j}}{dx}\bullet dz\bullet l+k\bullet\frac{T_{i,j+1}-T_{i,j}}{dx}\bullet dz\bullet l+k\bullet\frac{T_{i-1,j}-T_{i,j}}{dz}\bullet dx\bullet l+ +k\bullet\frac{T_{i+1,j}-T_{i,j}}{dz}\bullet dx\bullet l+q'''\bullet dx\bullet dz\bullet l=0 \quad (1)$$

Where T is the temperature, k the thermal conductivity and l the plate width.
For the right boundary the conduction term from the node j+1 can be canceled; for the lower boundary the i-1 flux is canceled and for the top boundary the i+1 flux is not considered. This holds because on these boundaries insulated boundary condition is used.
For the 1D model, for the general node, the following equation is used (no contribution from the i+1 and i-1 nodes, meaning no axial conduction):

$$k\bullet\frac{T_{i,j-1}-T_{i,j}}{dx}\bullet dz\bullet l+k\bullet\frac{T_{i,j+1}-T_{i,j}}{dx}\bullet dz\bullet l+q'''\bullet dx\bullet dz\bullet l=0 \quad (2)$$

In this case for the right boundary condition the j+1 flux cancels. The top and bottom boundary conditions do not affect the equations.

In both cases the left boundary condition is convection to the coolant. The equation for the 2D model becomes:

$$h\bullet(T_{i,j-1}-T_{i,j})\bullet dz\bullet l+k\bullet\frac{T_{i,j+1}-T_{i,j}}{dx}\bullet dz\bullet l+k\bullet\frac{T_{i-1,j}-T_{i,j}}{dz}\bullet dx\bullet l+ +k\bullet\frac{T_{i+1,j}-T_{i,j}}{dz}\bullet dx\bullet l+q'''\bullet dx\bullet dz\bullet l=0 \quad (3)$$

The conductivity of graphite is considered constant (k=15 W/(mK)) for simplicity.
The axial profile of the power density for this study is a cosine shaped function with a 1.3 peaking factor at the center.
Finally, an equation for the coolant is required. In order to make the Gauss-Seidel method stable, forward differencing has been used for the coolant. The following equations are used, for the inlet and the following nodes, respectively:

$$2\bullet m\bullet cp\bullet\left(T_{i,j}-T_{in}\right)=h\bullet\left(T_{i,j+1}-T_{i,j}\right)\bullet dz\bullet l \quad (4)$$

$$m\bullet cp\bullet\left(T_{i,j}-T_{i-1,j}\right)=h\bullet\left(T_{i,j+1}-T_{i,j}\right)\bullet dz\bullet l \quad (5)$$

Where h is the heat transfer coefficient, m is the mass flow rate and cp is the specific heat.
All the properties, such as specific heat cp and heat transfer coefficient are constant. The mesh used for the calculation is relatively fine, dx=dy=0.2mm.
As expected, the 2D solution is lower where the temperature is higher and higher where the temperature is lower, because of slightly enhanced heat conduction. Anyways the temperature difference at the maximum is about 0.003°C, which is a completely negligible difference, considering for example the uncertainties on k. Plus, using the 1D solution would mean being conservative.
Regarding the coolant temperature profile, the results behave as expected: a little more power is delivered where the coolant is colder and a little less where the coolant is hotter, but the difference is absolutely negligible, of the order of 10^{-4}.
The 1D approach for the evaluation of the axial plate temperature distribution is very close to the 2D solution and it is conservative; it substantially reduces the computational requirements, which makes it suitable for many purposes.
The conclusion of this analysis is that the axial conduction in the plate can be neglected.

2.1.1. Coolant properties

The AHTR uses FLiBe as a coolant. The properties of this salt are reasonably well known and the most recent formulas for the interpolation of experimental data are provided in Ref. 2. This fluid can be approximated as incompressible and most of its properties are not strongly dependent on the temperature. For analysis purposes, it is frequently easier to assume a fluid with constant properties. This section provides a quick overview of the averages of these properties in the operating temperature range of the AHTR, with particular focus on the interval between 650 and 700°C. The following considerations refer to the interpolation functions presented in Ref. 2.
The mass density has a linearly decreasing trend with respect to temperature; also the function has a very small uncertainty, meaning that the curve fits very well the experimental data.
If we take the average value of the density over the operating range we obtain 1950 kg/m^3. This number is 0.3% far, on average, from the actual value predicted by the formula. This error is acceptable because, in general it is relatively small; it might be not negligible in some specific conditions.

The average value of the *dynamic viscosity* is about 6 Pa*s and the error of the constant approximation with respect to the actual value is 6% on average. The constant approximation in this case is acceptable because the uncertainty of the formula is relatively large, about 20%.
Considering the *thermal conductivity*, the use of a constant value is justified for two reasons: the uncertainty of the actual value is large (15%) and the average error of the mean value with respect to the actual value is small (0.66%). The mean value is about 1.1 W/(mK).
The *heat capacity* is constant. Its uncertainty is 2%, relatively small. The reference value is 2416 J/(kgK).
The *volumetric heat capacity* is the heat capacity times the mass density. The average value in this range is 4.71e6 J/(m^3K).

2.1.2. 2D and 3D comparison

The CFD and neutronics modeling of the AHTR core cannot be performed without making simplifying assumptions, because the domain is too large. Focusing on the CFD modeling of the assembly, the first assumption that one could make is the 2D treatment of the flow in the coolant channel. The coolant channels are of three types:

- Thick intra-assembly channel: 7 mm
- Thin intra-assembly channel: 3.5 mm
- Inter-assembly channel: 1.8 cm

The width of these three channels (x direction in Figure 1) is about 20 cm; this analysis aims to investigate the effects of treating these channels as a flow between two infinite parallel plates, instead of considering their full 3D geometry.
The modeling approach is based on ANSYS Fluent. The main idea is that we want to compare the solution for a 2D and a 3D model. In order to do this we first need to have a good 2D model that allows us to find the best mesh and the optimal length to capture all the involved aspects.
Starting from the geometric description of the problem, the 2D model is represented as a rectangle with a 7 mm base and 50 cm height. The 3D model is identical but with z dimension added (the thickness is 23 cm). We must note that the cross section of the real coolant channel is not a rectangle, but a parallelepiped: the short edges are inclined at 30°. In any case this should not make a relevant difference, since we are mainly interested in capturing the effect of adding a 3rd dimension.
Regarding the meshing, the first aspect is the fact that these channels are very thin, so the grid spacing must be fine enough to capture the flow distribution. Also the flow is normally turbulent so the boundary layers must be able to accurately describe the steep velocity gradient.
Just to have an idea, in order to have an accurate modeling of the problem, about 40 to 60 intervals are required in the y direction (see Figure 2); a deeper analysis of the meshing is presented afterwards.

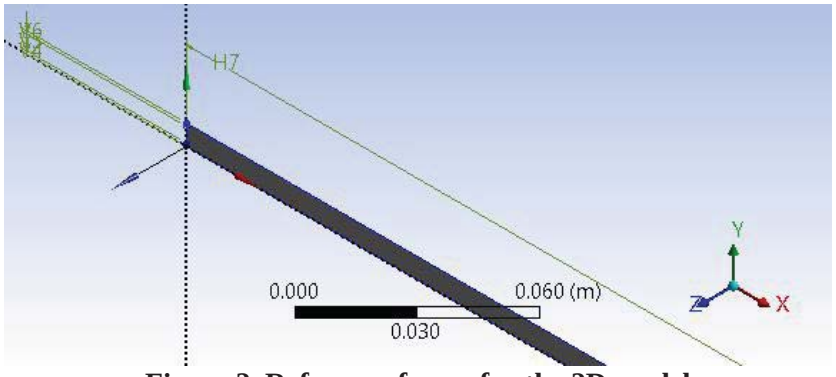

Figure 2. Reference frame for the 2D model

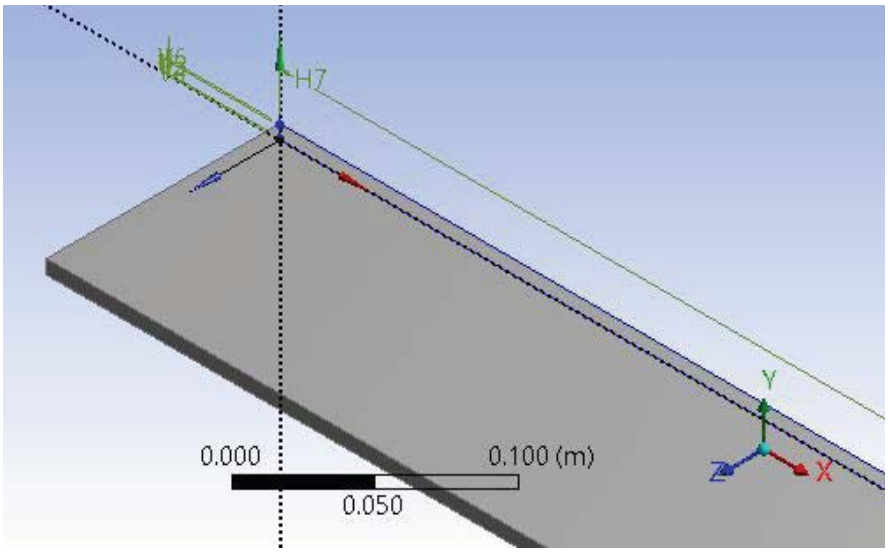

Figure 3. Reference frame for the 3D model

Figure 2 and Figure 3 show the geometry model for the 2D and 3D model respectively. Note that the 3D model represents only one half of the thickness (11.5 cm instead of the full 23 cm) because of the symmetry. The symmetry plane is located at z=0.

The X-Y cross section is the same for the two models (7mm*Ly), where Ly can be changed, depending on the length required for the flow to become fully developed. Just to give an idea, the fully-developed condition is reached after 30-40 cm in the 2D case, and after about 1.5 m in the 3D case. Further details will be presented later.

Regarding the Fluent setup, the main features are the use of a turbulent model(k-ε) and the iterative SIMPLE method solver. The meshing step is very important in order to obtain a correct solution; the main reason is the relatively steep gradient of the velocity at the boundary.

The first step is the definition of the meshing for the 2D model. This splits into two aspects: the definition of the Y mesh, that describes the velocity profile and the velocity steep gradient, and the X mesh, which gives the location at which the flow is fully developed.

The second step is the definition of the mesh in the Z direction for the 3D geometry. This step is relatively simple, since the general size of the meshing is set by the first step.

Considering the meshing in the XY plane, a bias is used in order to reduce the size of the mesh at the boundary. A preliminary analysis shows that 40 intervals can provide a very accurate velocity profile description, producing a 0.3% error on the velocity gradient at the boundary, which is fully acceptable.

Using 40 segments in the Y direction, a similar analysis was performed for the X direction, considering a 50 cm long channel. The bias of the cell dimension is 50 (outlet/inlet). The fully developed flow, as mentioned before, is reached at about 30 cm from the inlet. Also the results show that a 100 interval meshing on 50 cm is adequate.

For the Z direction 40 segments are used with a bias of 10, in order to match the grid spacing of the other two directions.

In the 3D case, the total length of the model was increased to 2 m because the fully developed flow for the 3D model is expected to be reached at a higher distance from the inlet.

The results show that the velocity distribution of the 3D solution requires more space to reach the fully developed region and that, at the centerline (z=0) it is slightly higher than the solution of the 2D case. This last aspect is due to the fact that the velocity of the flow is smaller when z approaches the sides of the channels in the z direction (z=11.5 cm).

Regarding the fully developed velocity profile, the curve for the 3D solution is higher (at the centerline) compared to the one of the 2D solution. The maximum velocity is 2.31 m/s for the 3D case and 2.28 m/s for the 2D case. The slope of the curve at the edges is about 2% different from one case to the other. Regarding the length at which the flow is fully developed we obtain that for the 2D case it is about 30 cm, while for the 3D case it is significantly longer, at least 1.5 m, requiring a much larger simulation domain. The friction factor has been derived from the two solutions, as a function of z. The two friction factors, in the central region, are very close; the difference is about 0.8%. Moreover, the 2D friction factor is the highest, so the 2D approximation can be considered a very good assumption, because it is also conservative.
It should also be noted that the difference is even more negligible in that interval where the 2D solution is developed and the 3D solution is still developing. This means that for the evaluation of the fully developed solution a relatively short length can be used, reducing the computational demand.

2.1.3. Friction factor

This section presents the evaluation of the friction factor for the coolant channels of the AHTR.
The friction factor correlations are required in order to get an adequate distribution of the flow and the temperature in the AHTR fuel assembly. The modeling approach is based on ANSYS Fluent. The geometry of the Fluent model is presented in Figure 2.
The main geometric features of the model are the following:

- The 2D model is represented as a rectangle with a 7 mm base and 40 cm height.
- The meshing is 40 segments in the y direction, appositely biased; the meshing in the x direction is 100 segments.
- The inlet velocity can be changed and the skin friction coefficient of the fully-developed region is calculated as a function of the velocity.

In the Fluent setup, the main aspects are that the k-ε model is used for the turbulent region and the solver is the iterative SIMPLE method for both cases. The only difference between the laminar and turbulent calculations is in the method settings.
The laminar region is analyzed first. The analytic solution for two infinite parallel plates is:

$$f=\frac{24}{Re} \quad (6)$$

The analytic and numerical solutions are in very good agreement.
In the turbulent region, the results have been interpolated with a power function plus a constant, providing the following result:

$$f=2.821096*Re^{-0.710987}+0.00510267 \quad (7)$$

This interpolation provides an R^2 very close to 1 (error is less than 1e-4), meaning that the fit can provide a very good prediction of the results in this range of Re number.
The transition region is the zone where 2000<Re<4000. Since no data are available in that zone, as initial guess, a linear function has been assumed, that connects the upper boundary of the laminar to the lower boundary of the turbulent region. The result is:

$$f=4.2721e\text{-}7\bullet Re+0.011146 \quad (8)$$

The behavior of the skin friction coefficient as a function of the Reynolds number is shown in Figure 4.

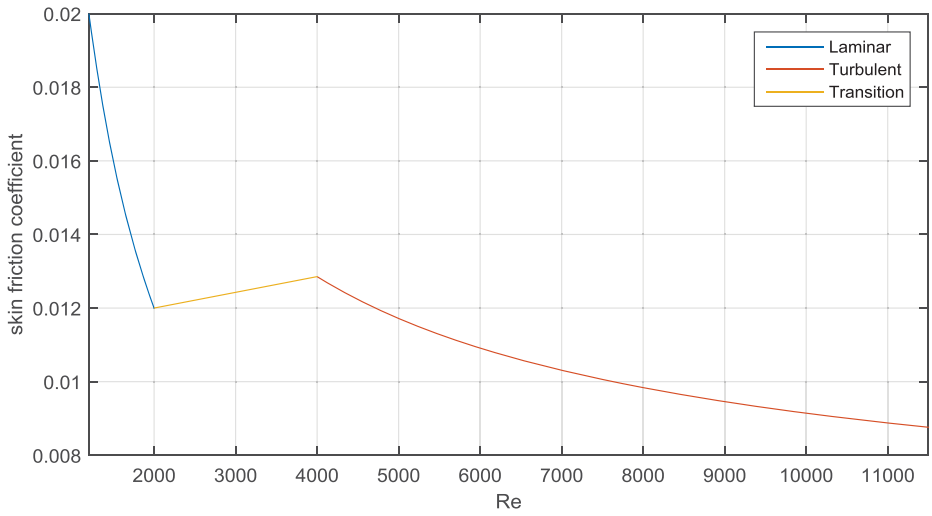

Figure 4. Skin friction coefficient as a function of Re

The overall behavior of the skin friction coefficient can then be expressed as:

$$\begin{gathered} f=\frac{24}{Re},\ Re<2000 \\ f=4.2721e\text{-}7\bullet Re+0.011146,\ 2000<Re<4000 \\ f=2.821096\bullet Re^{-0.710987}+0.00510267,\ Re>4000 \end{gathered} \quad (9)$$

2.1.4. Heat transfer coefficient

This section presents the evaluation of the heat transfer coefficient and the Nusselt number for the coolant channels of the AHTR. The Nusselt number correlations are required in order to get an adequate distribution of the flow and the temperature in the AHTR fuel assembly.
The modeling approach is based on ANSYS Fluent. The geometry of the Fluent model is presented in Figure 2. The Fluent setup is the same as the one used for the evaluation of the friction factor. The behavior of the numerical solutions is slightly affected by the fact that the length of the pipe is adjusted every time in order to be able to reach the fully developed region.
The laminar region is analyzed first. The analytical solution for two infinite parallel plates is:

$$Nu=8.2353 \quad (10)$$

The analytic and numerical solutions are in very good agreement.
In the turbulent region, the results have been interpolated with a power function, providing the following result:

$$Nu=0.0378\bullet Re^{-0.8752} \quad (11)$$

This interpolation provides an R^2 of 0.9993, which is relatively good.
Equation (11) was considered acceptable since it is a pretty good fit of the Fluent results and can be directly compared to formulas from the literature (such as Dittus Boelter formula). The Dittus Boelter solution is normally used as an initial assesment when more accurate results are not available. The results show that the difference between the Dittus Boelter and the Fluent results is about 15% (in the considered flow regime); Dittus Boelter solution is smaller, so it is conservative.

A linear function has been assumed in the transition region, that connects the upper boundary of the laminar to the lower boundary of the turbulent. The result is:

$$Nu=0.0228 \bullet Re-37.265 \quad (12)$$

The Nusselt number as a function of the Reynolds number is shown in Figure 5.
The overall behavior of the Nusselt number as a function of Reynolds can be summarized as follows:

$$\begin{aligned} Nu&=8.2353, \ Re<2000 \\ Nu&=0.0228 \bullet Re-37.265, \ 2000<Re<4000 \\ Nu&=0.0378 \bullet Re^{-0.8752}, \ Re>4000 \end{aligned} \quad (13)$$

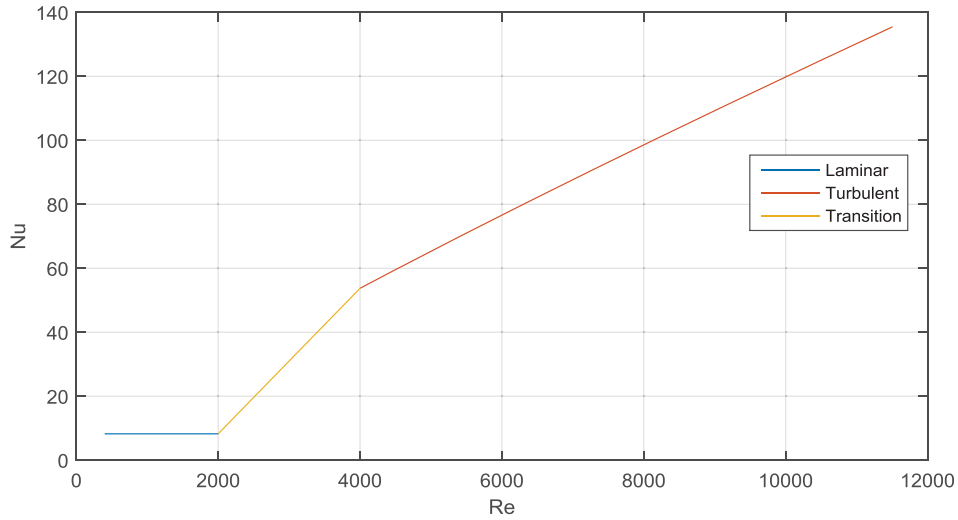

Figure 5. Nu as a function of Re

2.2. Modeling approach

This section presents the approach used to develop the assembly calculation. The aim is to create a fast tool that allows a correct representation of the temperature distribution of the fuel assembly and the coolant. The modeling is based on MATLAB and the code performs the operations explained in the following steps.

1- Definition of the mesh. Figure 6 shows a part of the mesh, particularly the portion of the horizontal cross that represents a single plate. The mesh presented in the figure is a lot coarser than the one used for the actual calculations.

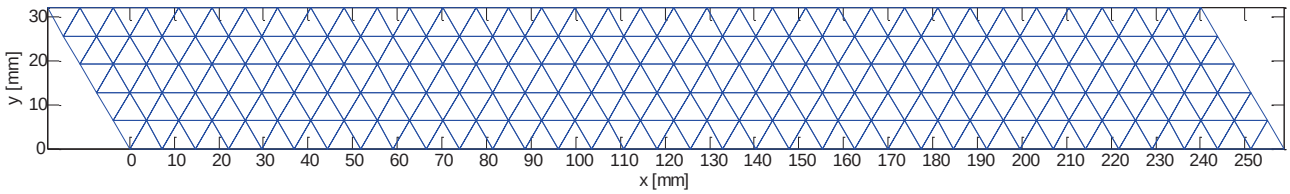

Figure 6. Portion of the mesh (plate)

A triangular mesh has been implemented, since the geometry of the assembly is hexagonal. Equilateral triangles have been used (this simplifies the treatment of the equations), even if this choice requires particular attention in order to accurately represent the body. The dimensions of the cross section in fact are slightly adjusted in order to correctly fit the mesh: the boundaries of different materials lie perfectly on the sides of specific triangles.

2- Definition of matrices for the following quantities:
   a. Direction of the triangle. For each node the direction (upward or downward) of the triangle is recorded;
   b. Distribution of materials. Figure 7 shows the material distribution for the plate: green for the box wall, blue for the fuel plate (light blue for the fuel stripe), orange for the support structure and red for the coolant. Note that the cladding is neglected.

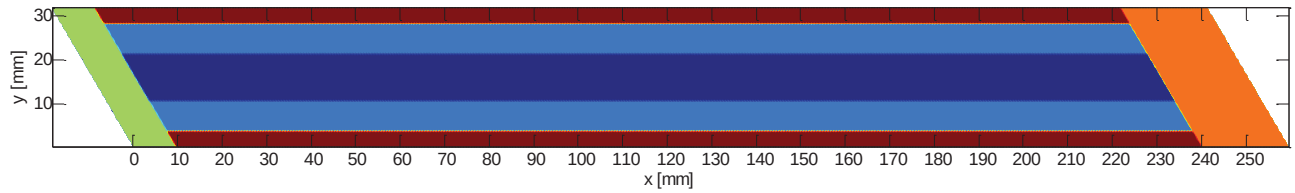

Figure 7. Distribution of materials for the fuel plate

   c. Power density distribution. The power is produced only into the two light blue stripes that we can see in Figure 7. Considering the axial distribution of the power, a chopped cosine profile has been used for the study (with a 1.3 peaking factor), but the model is capable of modeling any kind of profile.
   d. Thermal conductivity of the materials.

3- Evaluation of the boundary connections for every triangle. Since we want to use a finite volume method, we need to know for every triangle how it exchanges energy with the neighboring triangles. The model allows for three types of boundary connections:
   a. Insulated face. This means that there is no power exchange through that surface.
   b. Convection. This means that the face is connected to a certain channel at a given temperature. The heat transfer coefficient must be provided; the correlations provided in section 2.1.4 have been used for this purpose.
   c. Conduction. The thermal conductivity of graphite (about 15 W/mK) has been used.

   The boundary connections are used to build the matrix of the linear system that has to be solved in order to provide the temperature distribution. Also, a conductive connection has been modeled between the right and upper wall, in order to preserve the symmetry of the system.

4- Evaluation of the flow velocity for the internal channels. The total flow through the system is fixed; the ratio of the external and internal flow with respect to the total flow is defined in the input, while the splitting of the flow among the 7 internal channels is determined by the total pressure drop along the channels, which is the same for all of them. The correlations presented in section 2.1.3 have been used to evaluate the friction coefficient. The velocity of the flow in the 2 small channels (lower and upper) is about 1.5 m/s while the velocity in the other five channels is about 2 m/s.

5- The last step consists in the iterative procedure that provides the axial distribution of temperatures:
   a. The temperature of the coolant at the inlet of the channels is given (650°C);
   b. The horizontal temperature distribution of the assembly is solved for that axial level;
   c. The power delivered to each channel is obtained by integration over the boundary nodes for every channel;

d. The temperature of the coolant at the next axial step is evaluated through the following balance:

$$\dot{m}_{ch} \bullet c_p \bullet dT_c = \dot{q}' \bullet dz \qquad (14)$$

e. The procedure from point a is repeated until the top of the assembly is reached.

3. Results

3.1. Single plate temperature distribution

The single plate temperature distribution was first evaluated in order to verify the validity of the model. Figure 8 shows the temperature distribution within the plate when the coolant temperature is 675°C on both sides and in the external channel. The heat transfer coefficient is set to 7500 W/(m²K).

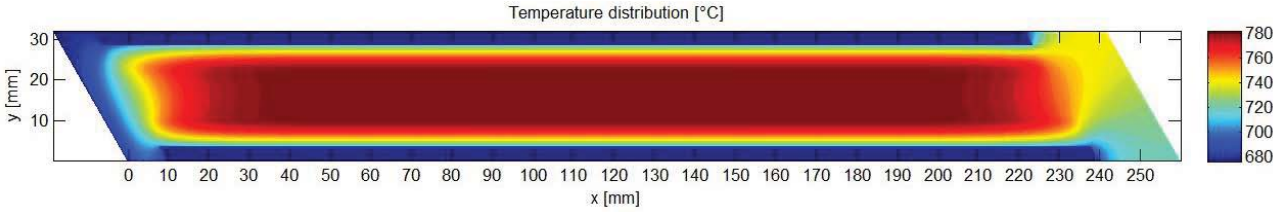

Figure 8. Temperature distribution of the single plate (1630x400 nodes)

It can be noted that the temperature distribution in the central portion of the plate does not depend on the x position. For this reason, the maximum temperature of the plate can be evaluated through a very simple 1D approximation of the heat conduction:

$$T_{max} = T_c + q''' \bullet \frac{t}{h} + \frac{q'''}{k} \bullet \frac{t^2}{2} = 783°C \qquad (15)$$

Where t is the thickness of the fuel stripe. The numerical result is 783.5°C, in very good agreement with the analytic result.
Furthermore, the temperature distribution in the box wall and in the support shape cannot be considered representative of the full assembly, since the symmetry condition is not correctly represented in the single plate model. The area in which this error is mostly evident is the right portion of the plate, the upper part of the Y-shaped support (the temperature is in the yellow zone). Here, the heat should flow out of the right boundary, but this would be more a global feature of the assembly than of the plate. This aspect is not correctly represented here, but it will be addressed in the full assembly analysis.

3.1. Full assembly temperature distribution

This section shows the temperature distribution of the fuel assembly. Figure 9 shows the temperature distribution in different configurations. The maximum fuel temperature is about 847°C and its location is slightly above the midplane (z=3.25 m), as shown in Figure 10. As mentioned before, the front and back plates are hotter (about 20°C) than the others, due to the low cooling capacity of the two thinner channels. On the XY plane, the maximum is located in the upper plate, between the graphite meat and the fuel stripe in contact with the upper channel. Figure 11 shows the XY temperature distribution of the plate at the location of the maximum.

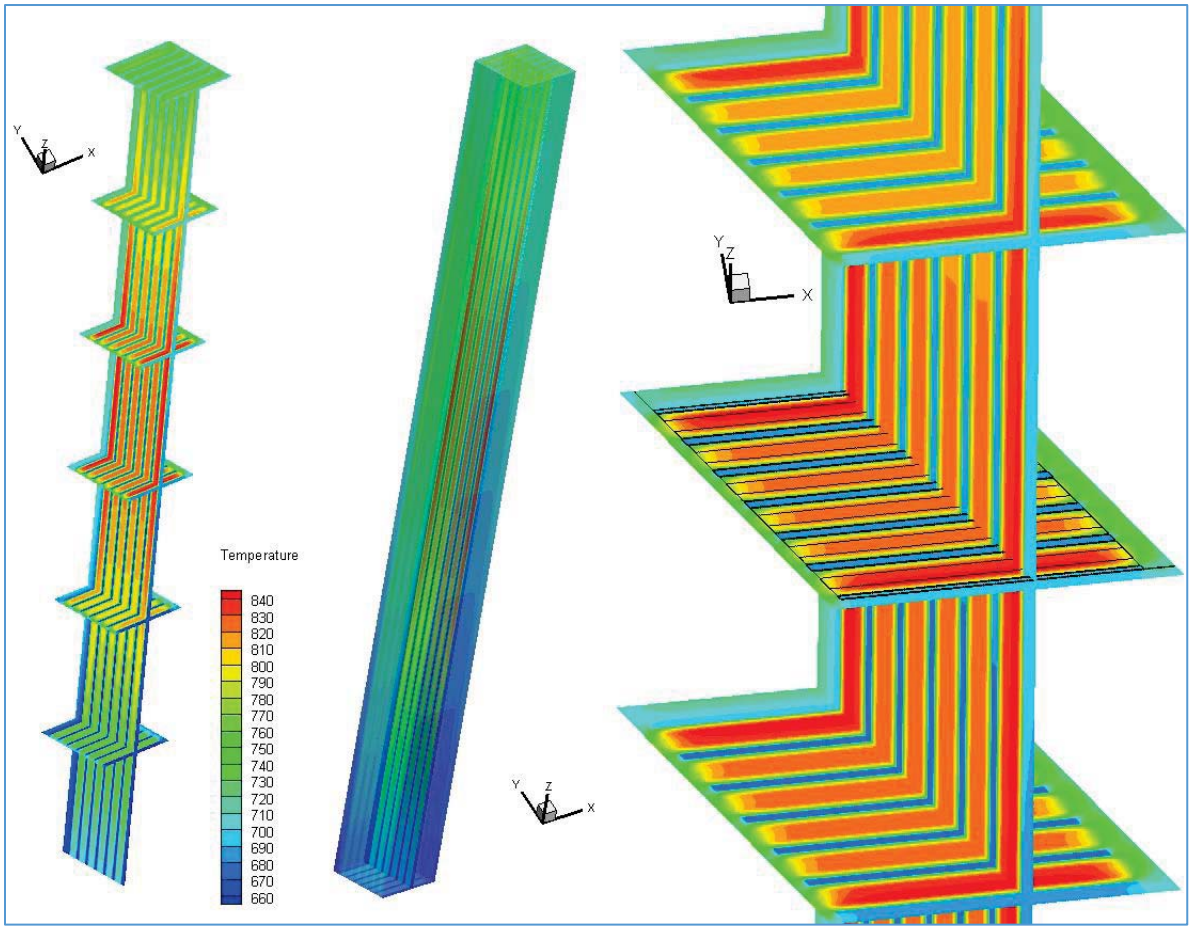

Figure 9. 3D Temperature distribution results

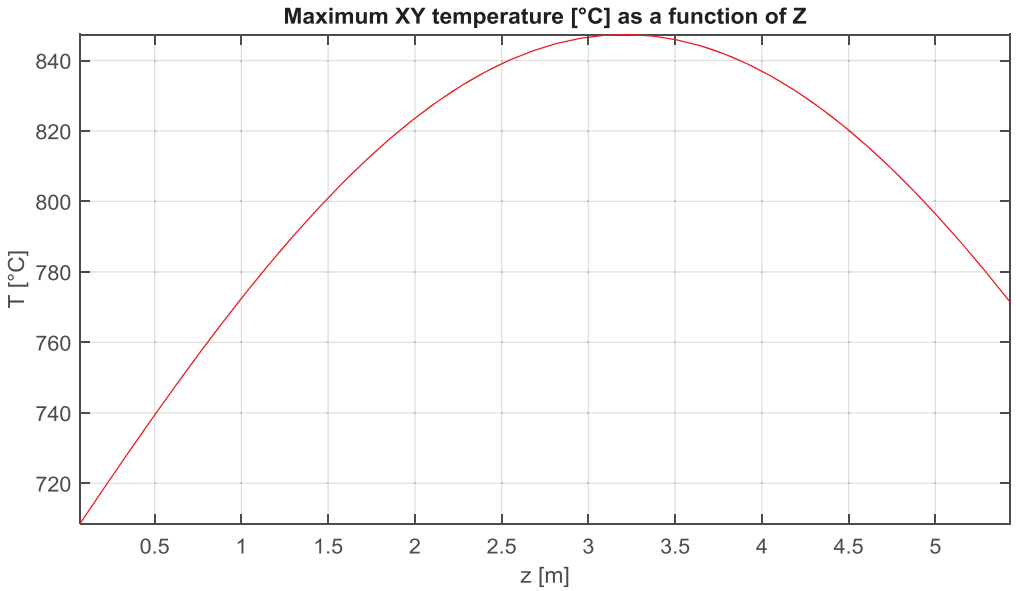

Figure 10. Axial distribution of the maximum fuel temperature

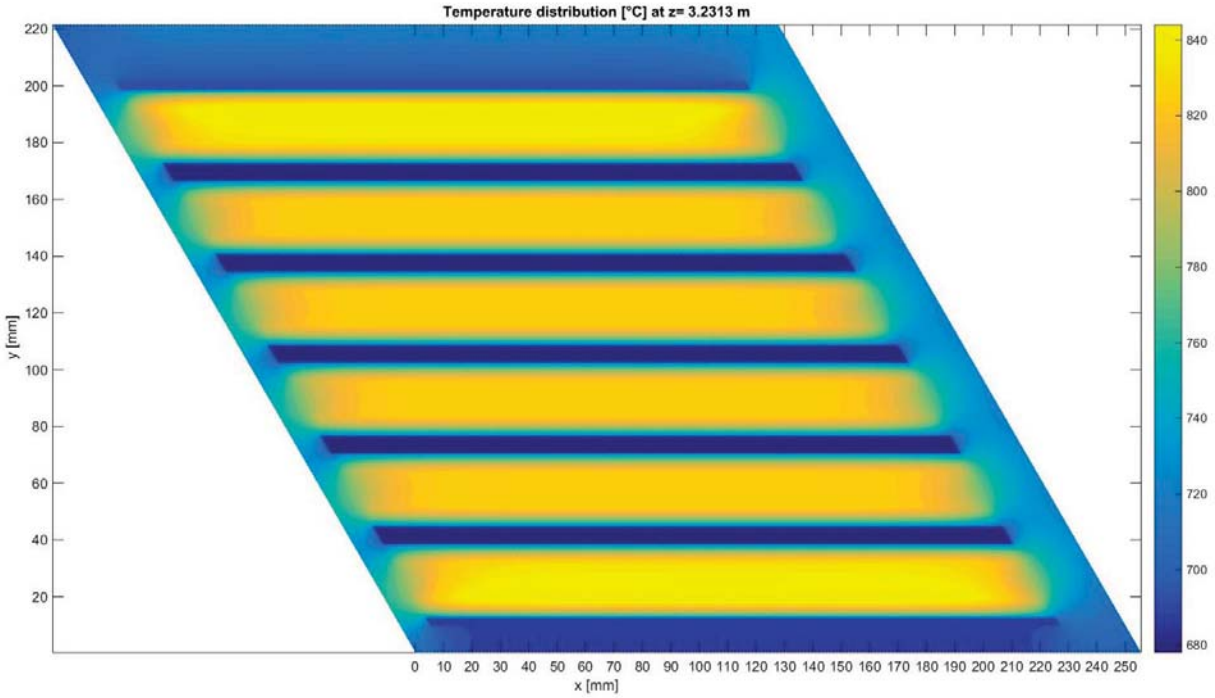

Figure 11. XY temperature distribution at the location of the maximum

Figure 12 shows the axial temperature distribution of the coolant for different channels. As we can see the external channel has a profile very similar to the one of the internal channels, so 0.5% bypass flow is an acceptable fraction.
The thinner channels, the front and back, are 20°C hotter than the average.

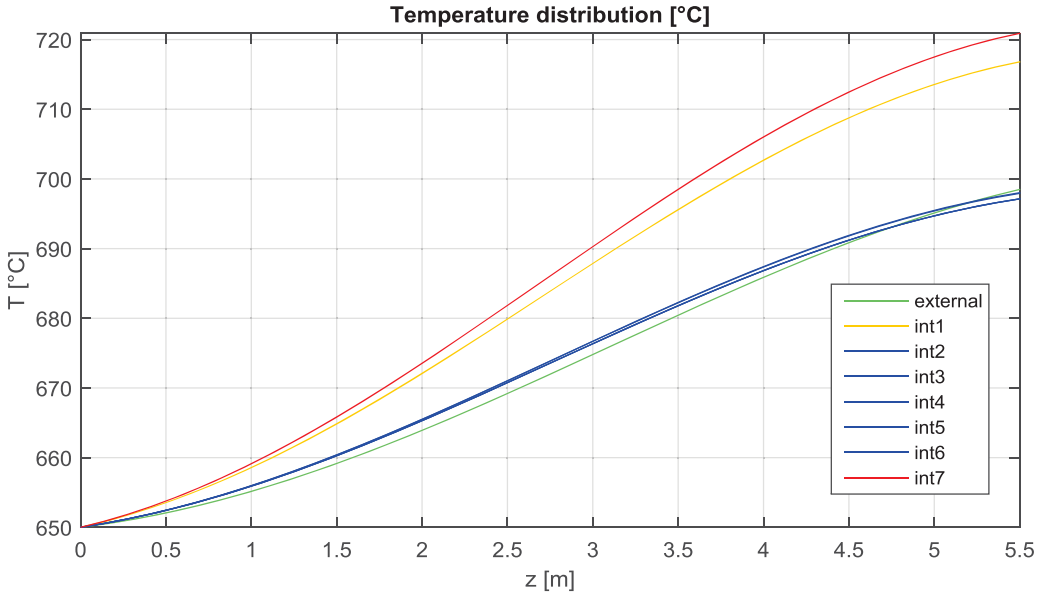

Figure 12. Axial temperature distribution of the channels

The same analysis has been performed, increasing the size of the smaller channels by 33% (from 3.5 mm to about 4.5 mm). In this case the temperature distribution is more uniform and the maximum temperature is 20°C lower. The lower and upper plate have about the same temperature distribution as the others. The temperature distribution in the new channels configuration brings lower temperature of the two thin channels; the outlet temperature is pretty uniform for the internal channels, and also the power delivered to the external channel is less.

4. CONCLUSIONS

The main finding of this simulation is that the outlet temperature of internal thin (narrow) coolant channels are about 20°C hotter than of the other channels, and this results in a higher temperature of the adjoining fuel plates. This is a purely thermal effect which can be further amplified by the fact that the neutron flux distribution (and so the power density) is not uniform in the fuel plates. In order to alleviate this issue a different geometrical design of the internal channels may be considered.
The amount of external flow does not have a strong influence on the global behavior of the system but it is still important for determining the temperature of the box wall and the support structure. One interesting aspect that comes out of this analysis is that with no external flow the temperature distribution of the structure is more uniform and this might be good from the mechanical standpoint. The option of not having any bypass flow might then be considered.
The approach we have developed in this project is simple and fast, but still provides an accurate full 3D description of the system at a very low computational cost.
The future work will include the following:

- Use of non equilateral triangles for the mesh. This would allow an even quicker evaluation of the temperatures.
- Evaluation of transient behavior.[3]
- Better characterization of the coolant through CFD models.

This assembly model will serve as a basis for the evaluation of the behavior of the AHTR full core thermal hydraulics.

ACKNOWLEDGEMENTS

This research is being performed using funding received from the U.S. DOE Office of Nuclear Energy's Nuclear Energy University Programs (NEUP).

REFERENCES

1. Varma K., Holcomb D., Peretz F., Bradley E., Ilas D., Qualls A., Zaharia, "AHTR Mechanical, Structural, and Neutronic Preconceptual Design," ORNL/TM-2012/320, September 2012.
2. Richard J., Wang D., Yoder G., Carbajo J., Williams D., Forget B. and Forsberg C., "Implementation of Liquid Salt Working Fluids Into TRACE," *Proceedings of ICAPP 2014*, Charlotte, USA, April 6-9, paper 14214 (2014).
3. Avigni P., Petrovic B., "Fuel element and full core thermal-hydraulic analysis of the AHTR for the evaluation of the LOFC transient," *Annals of Nuclear Energy* **64**, pp. 499-510 (2014).