Baffle Design: Tube-in-Shell Electrical Gas Heaters

K. Lund[1] and S. Lord*[2]
[1]COMSOL Consultant, Kurt Lund Consulting, San Diego, CA, USA
[2]President, SML Associates, Encinitas, CA, USA
*Corresponding author: 109 Peppertree Lane, Encinitas, CA 92024, slord@smlassociates.com

Abstract: Two- and three-dimensional analyses are performed with COMSOL on heat exchangers where the hot-tube fluid is replaced by electrical heater-rods. Heat transfer from the rods to the tubes is by radiation and conduction, and turbulent fluid flow for the shell-side gas-flow uses the k-epsilon theory. Special, thru-flow baffles that maintain tube spacing are designed to limit local temperature hot-spots. Novel turning baffles at the inlet are designed and evaluated for beneficial flow development. Tie-rods that are always provided to position and support the baffles are further used to improve flow distribution and heat transfer by radiation.

The results show that the serpentine flow design of a typical shell and tube heat exchanger is inherently unsuitable for electrical heating, and can lead to burn-out issues; however, the COMSOL modeling shows that converting to well-distributed axial flows solves these issues.

Keywords: Axial Flow, Heater, Shell and tube, Electric, Heat Exchanger, Turbulent Flow.

1. Introduction

It is frequently necessary to heat gases, and a convenient and safe geometry is the familiar shell and tube heat exchanger. Could this geometry also be used for electrical heating, as indicated in Fig. 1 where the tube-side is modified for electrical heater-rods, and the baffled shell-side contains the flowing gas? Or would the fixed heat flux or heater-rod bending cause a problem?

Figure 1: Heater Geometry.

This modification changes the essence of the thermal boundary conditions near the baffles from being temperature-limited to being flux-limited; special, detailed considerations are then necessary to avoid burn-out.

The results show that the serpentine flow design of a typical shell and tube heat exchanger is inherently unsuitable for electrical heating, and can lead to burn-out issues; however, the COMSOL modeling shows that converting to well-distributed axial flows solves these issues. Also, establishment of the distributed axial flow down the exchanger is studied in detail; it is shown that proper inlet and outlet baffle designs can produce a "turning baffle" which converts the sideways inlet and outlet flows to smooth axial flow down the heat exchanger length.

The detailed radiation analysis shows that the tie-rods also provided an "extended area" surface which could provide further heat transfer. This extended area is unique in that the mechanism of heat supply to the extended area is via radiation rather than conduction as in a conventional fin.

1.1 Conventional Heat Exchanger

Standard tube and shell heat exchanger designs have the shell-side fluid as a mixed cross/axial flow over the tube-bank, alternating back and forth in serpentine fashion between baffles having opposing cutouts. Part of such a design was constructed, as the symmetry-section in Fig. 2, which contains a typical baffle at mid plane; flow boundary conditions were applied at top and bottom of the model, which resulted in the streamlines and flow arrows in Fig. 2.

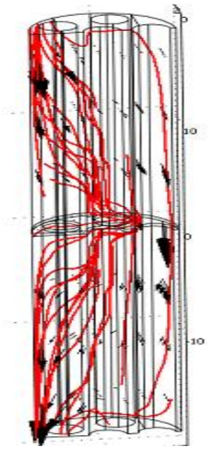

Figure 2: Serpentine Gas Flow.

It is seen that there is a stagnation region opposite the baffle cut-out, which is inconsequential for fluid-to-fluid heat exchange that is temperature limited. However, when the tube-internal fluid is replaced by electrical heaters the boundary condition is changed to a specified heat flux, and this stagnation will result in a local hot spot that may have an excessive temperature and cause burn-out. To avoid this situation a conventional heat exchanger was redesigned for pure axial flow, with the baffles replaced by spider-type rod-spacers. Extensive COMSOL modeling was required to limit temperatures.

2. Tube-Shell Analysis

2.1 Axial-Flow Models

We consider the geometry in Fig 3, shown as a radial cross section and assume that the flow is even and axial. Here the open circles represents the tubes (to have heater rods inside), and the closed circles are the baffle tie-rods; the placement of the latter serves to enhance the turbulent flow distribution between the tubes, rods and shell, shown as the gray area:

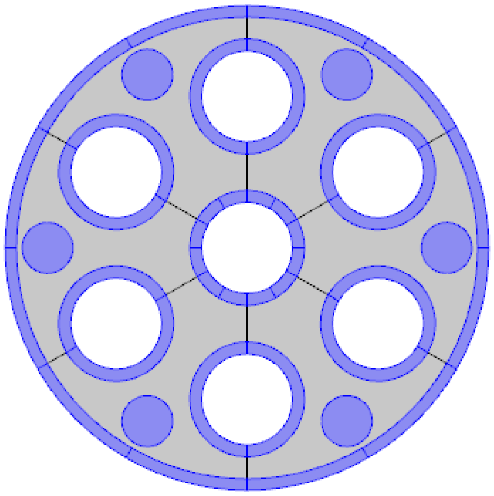

Figure 3: Cross Section of Flow Area

For analysis, a 60 degree symmetry sector was chosen, and extruded over a length, as shown for the Reynolds numbers in Fig. 4:

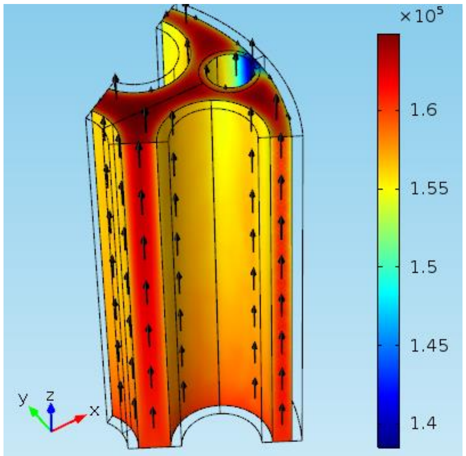

Figure 4: Flow computation

For *Re* based on the hydraulic diameter and the computed local velocity, it is clear that this flow is very turbulent. A uniform velocity was specified at the bottom of the diagram, with flow going axially upwards, as indicated by the arrows. The top surface has uniform pressure and shows the well-distributed flow over the cross section. The *k-ω* turbulence model was used in this model, which computes the velocity field outside the logarithmic layer at the walls; therefore, there appears a non-zero flow at the surfaces, even though zero velocity is specified.

For the thermal model, heat generated in the rods (not shown) becomes a specified heat flux at the tube inner surfaces; a uniform gas temperature is specified at the model-inlet. These conditions resulted in the gas temperature increases in Fig. 5:

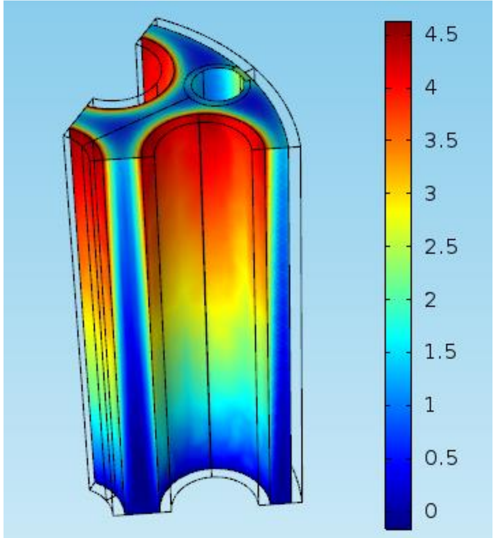

Figure 5: Gas temperature increases (^{O}C).

It is seen that there is a substantially uniform temperatures from the bulk (blue) through the surface layers (yellow) to the heater-tubes outside surface (red). This results in fairly uniform solid temperatures (which are the limiting conditions for the heater rods), as seen in Fig. 6:

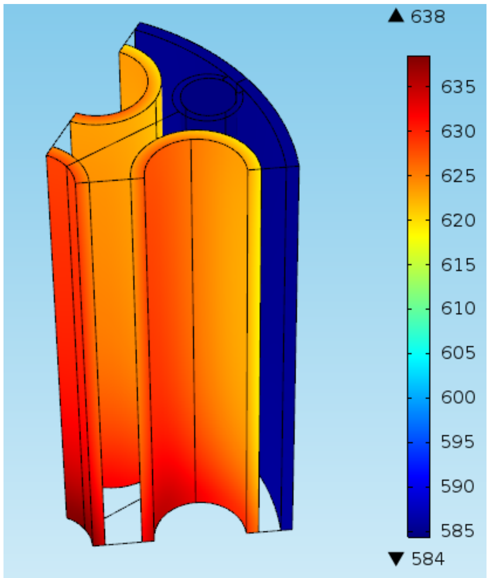

Figure 6: Computed solid temperatures (^{O}C).

The tubes are of metal with moderately high conductivity, so the tube temperatures appear fairly uniform. In this model there is also surface-to-surface radiation (essentially from the tubes to the shell), so the central tube temperature runs slightly higher than the others, due to a reduced view factor. With convection only, and without the radiation, the temperatures were about 10 ^{O}C higher (the exterior of the shell was considered insulated).

2.1 Tube Interior Model

There was concern that a bowed or eccentrically placed heater rod could cause excessive temperatures. However, at high temperatures there is significant radiation between the tube and rod surfaces; therefore, there is minimal effect on the rod and tube temperatures due to rod bowing, as seen in Fig. 7, where the temperatures are to the same scale, and a typical convection condition was placed on the outside tube surface, and a specified heat flux on the rod inside.

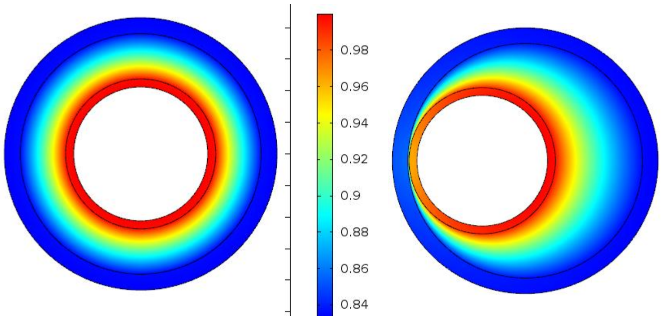

Figure 7: Temperature Ratios, T/T_{max}.

This axial-flow heat exchange was considered adequate for the application; thus, attention could be directed to the baffle design and evaluation.

3. Baffle Analysis

3.1 Conventional Baffle Model

To illustrate the effect of a conventional baffle, a model was built as in Fig. 8, where the purple color is the fluid region, and the baffle is axially midway. A downward inlet velocity was specified at the top, and pressure at the bottom outlet. The resulting turbulent velocity distribution is shown in Fig. 9.

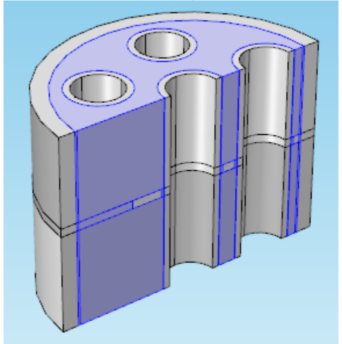

Figure 8: Conventional Baffle Geometry.

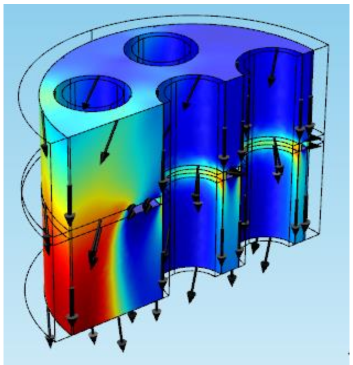

Figure 9: Flow Distribution.

Here, the red color is for the maximum velocity, and the blue is near zero; it is seen from the colors and the arrows that there is significant bypass flow, and very little flow around the tubes to the right in the diagram.

When a heat flux is applied to the tube inner surfaces, the flow blockage results in excessive temperatures (dark red, Fig. 10). Although there is flow through a narrow gap surrounding the tubes, as seen by the yellow streamlines, this is insufficient for good heat transfer.

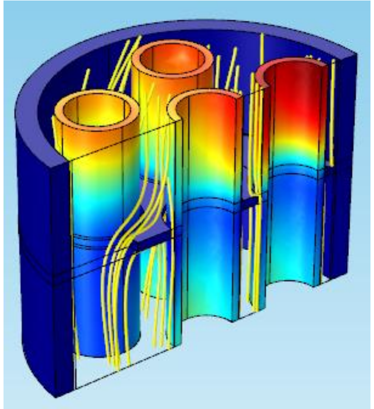

Figure 10: Temperatures near baffle.

This flow pattern can be seen in more detail in Fig. 11, where now the streamlines are red. There appears to be strong circulation turbulence below the baffle; but on the approach-side, there is reduced flow and excessive temperatures.

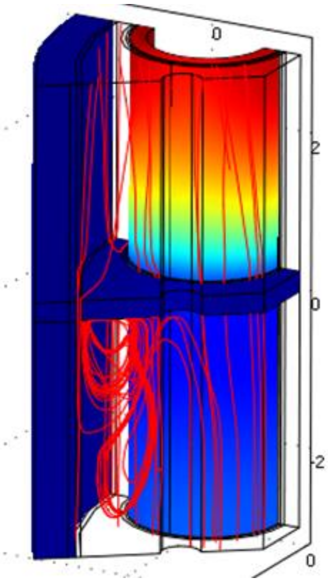

Figure 11: Flow detail.

3.1 Spider-Type Baffle Model

From the above analyses it is clear that a significant flow-through area surrounding the tubes at the baffle is required; this led to a spider-type design with stand-offs to center the tubes, as seen in Fig. 12, and its placement in the wedge model in Fig. 13:

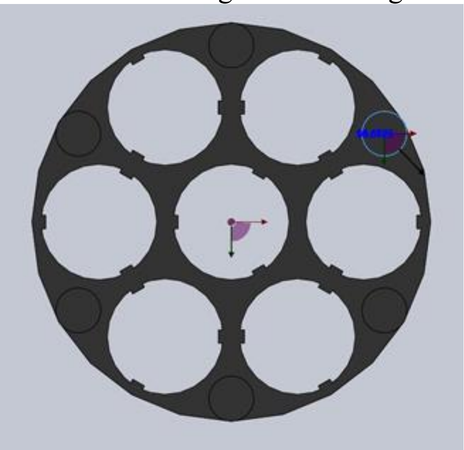

Figure 12: Spider-type Baffle or Spacer.

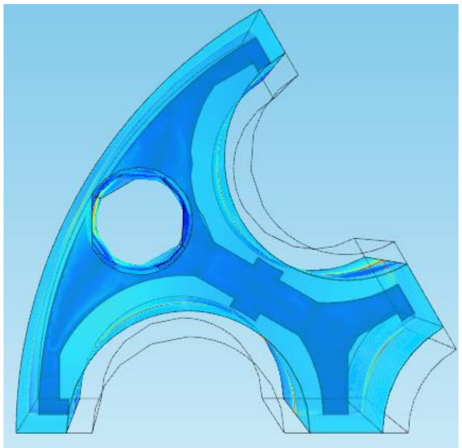

Figure 13: Spacer Placement in Model

With these radial gaps, there is good flow near the tube surface, as shown in Fig. 14 where the velocity is a factor of 2.5 greater (red color and proportional arrows) than at the model inlet, and the pressure drop is not excessive:

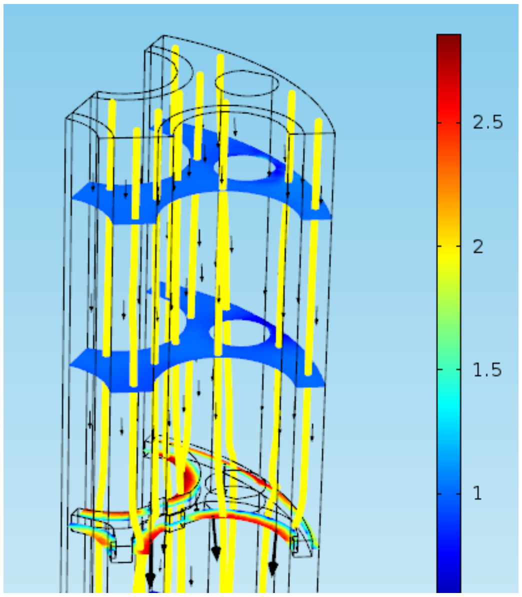

Figure 14: Velocity-ratio Increase at Spacer.

The solid temperature increases due to the baffle are shown in Fig. 15. These are only about 6 ^{O}C from top to bottom; thus, the spider-design keeps the tubes in place while permitting excellent heat transfer.

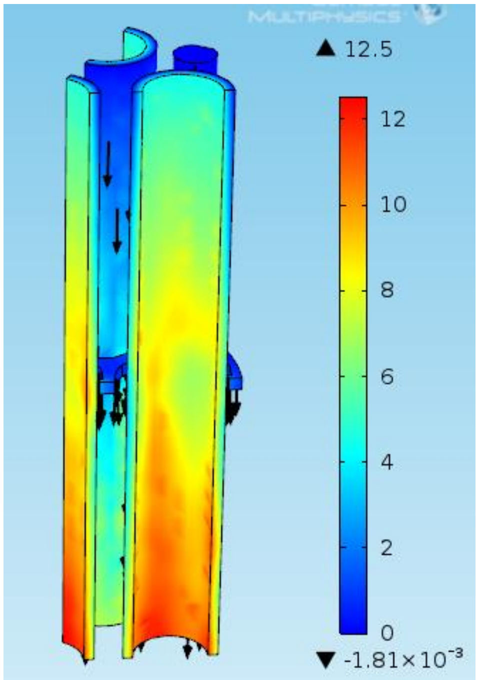

Figure 15: Temperature Increases (^{O}C).

3.2 Inlet Plenum Conditions

It would be preferable for the gas to enter the heat exchanger axially. However, for practical reasons, the axial location is needed for the electrical connections to the heater rods, as shown in Fig. 1; therefore a side entrance is used for the gas inlet, as indicated in Fig. 16, where purple indicates solids, and flow enters from the right.

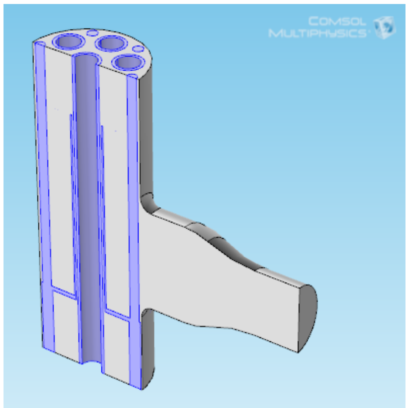

Figure 16: Gas Inlet Geometry.

Because of axial space limitations, a special design was needed to convert the radial in-flow to well-distributed axial flow over the tubes. This was achieved with sleeves surrounding the tubes, forcing the flow upwards (in Fig. 17) and distributing evenly into the annular resistance gaps between the sleeves and the tubes:

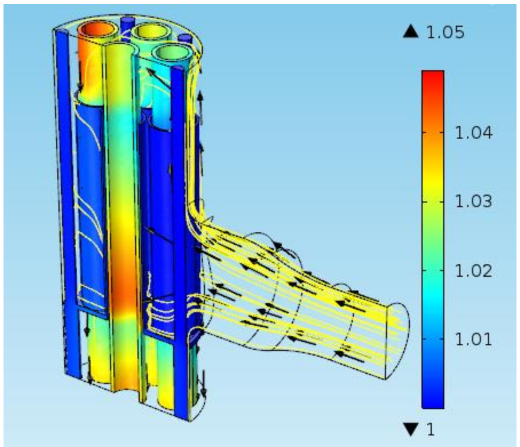

Figure 17: Inlet Temperature Ratios.

4. Discussion

COMSOL was very useful in addressing the two original questions posed by the new design and showed that heater tube bending was not a problem, but that flow stagnation was. It was very hard to get convergence in the flow stagnation problem because of meshing issues around the very small gaps in the baffles of conventional exchangers. However this simulation difficulty illustrated the physical flow problem. The progressive reduction in hole diameter required to redirect the flow from the serpentine fashion caused greater and greater simulation issues, showed the inherent problem, and pointed the way to using baffles with much larger holes and hence to axial flow.

Once this design change was made the flow was much better controlled; the simulations were much easier to converge, and also provided an unexpected extended area bonus. The last problem of the inlet turning baffles was easily modeled; additionally, the gas flow cooled the tube plate.

The practical, in-the-field, changes required to go from the unsuitable conventional design to a new high efficiency axial flow design were only in the baffles, which was an easy modification for the fabricator. This experience reinforced our long held opinion that “if it won’t simulate and converge it’s probably a bad design”

5. Conclusions

Use of COMSOL to analyze the conventional shell and tube heat exchanger was very beneficial in showing that the hypothesized hot spot problem was very real and inherent in the cross-flow design of a conventional shell and tube heat exchanger. The COMSOL capabilities in combined radiation and convection modeling were very valuable in showing that a modified design using axial flow was viable, provided high heat transfer, and the unexpected bonus of “extended area”. Practical considerations of heater layout required the invention and modeling of a “turning baffle” which could also be modeled by COMSOL. As a result of the innovations sparked by the modeling we were able to obtain worldwide patents on the axial flow heater: US #8260126.