Using FloEFD as an **Engineering Tool**

By Karl du Plessis, Senior Engineer, ESTEQ

What do you do when faced with analyzing a shell and tube heat exchanger as in the model shown in Figure 1? I can already hear you saying "you want to 'C..F..D' this thing!? There's like a thousand meters worth of piping..?" Quite literally in fact, approximately 1km in total with a 1mm wall thickness and 800 bends. Thoughts that run through my mind are: "How big is this mesh going to be? How long is it going to take to solve? I only have a quad core laptop (at least with 32GB of memory which helps)". And if I were to use anything other than FloEFD I'd also think with all those bends I'm probably going to have to remodel the piping so that I can HEX-mesh it... It seems overwhelming at first because most of the time, us engineers simply don't have time for all that, we need answers and we needed them yesterday!

Fortunately this is exactly where FloEFD starts to make a lot of sense, especially for the internal pipe flow, where the SmartCells™ technology within FloEFD really comes into play. SmartCells will recognize directly from the CAD geometry if it is a pipe or a channel, and decide based on the number of cells across this pipe or channel to apply a textbook or engineering calculation (1D) for the pressure drop and heat transfer when there is insufficient cells across the pipe to numerically resolve the flow. Alternatively, when there is indeed a sufficient number of cells across the pipe, SmartCells will then automatically switch to resolving the flow (3D) with the numerical grid. But, if you've ever wondered exactly how well FloEFD performs in this regard, perhaps the following observations may be very beneficial.

Part I: Internal Pipe Flow

Consider for example the internal flow of a single 10-pass pipe layout as relevant to the heat exchanger at hand. The FloEFD model of the pipe is shown in Figure 2. Heat transfer to the fluid is modeled with an external HTC applied to the outer wall, to allow for the calculation of conduction through the wall along with the conjugate heat transfer at the fluid-solid interface on the internal pipe surface. Radiation is neglected for this example. The mesh was generated such that the characteristic number of cells across the diameter of the pipe was gradually increased, starting with as little as 2x2 cells across the pipe diameter up to 6x6 cells.

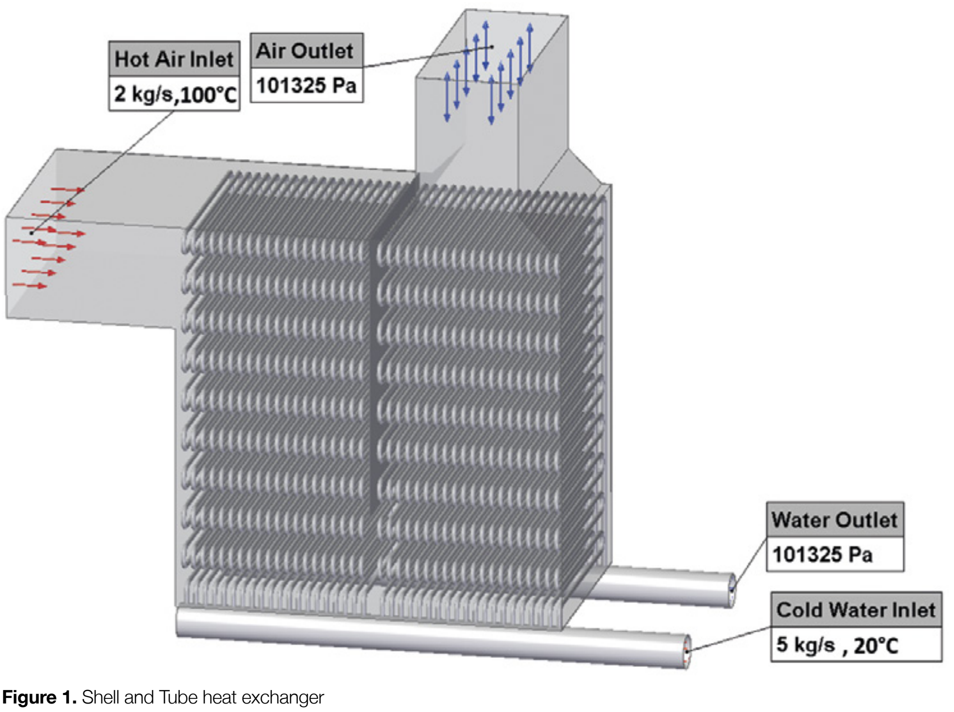

Figure 1. Shell and Tube heat exchanger

Now let us compare the results from FloEFD with that of the very reliable 1D thermal hydraulic system solution called Flownex (developed locally here in South Africa). The Flownex model of the same pipe layout is shown in Figure 4. Graphs of the pressure drop and total heat transfer are presented in Figure 5. The FloEFD results are displayed with respect to the increasing mesh density and compared to the Flownex result. A band of +10% and -10% about the Flownex result is also shown to add some perspective to the comparison.

Consider first of all the pressure drop comparison. One can see a very good agreement between FloEFD and Flownex, albeit a slightly lower pressure drop being predicted by FloEFD. Please note that for the sake of brevity I only display the results for meshes up to six characteristic cells, since further investigations showed that the FloEFD pressure drop result becomes mesh independent from around six cells for this example and eight to ten cells for an example with air at much higher flow velocities. Furthermore, it was also established that there seems to be a consistent difference in pressure drop prediction between Flownex and FloEFD, which is mainly attributed to the possible differences in the calculations for both primary (wall friction) losses and secondary (bend) losses used within Flownex and FloEFD, so we will not focus our attention on this.

What I would rather prefer to focus on is the much more fascinating heat transfer result. It runs out that for this example the heat transfer prediction by FloEFD is always within the +/-10% band compared to Flownex, regardless of the mesh density.

Again I am only showing one example here, but an extensive study of first comparing 1-pass, 2-pass and then 10-pass pipe layouts, with flows at varying Reynolds numbers (as high as Re=600,000 with air at 45 to 50m/s), all produced very similar behavior. For the sake of everybody's curiosity I want to make the following interesting observation: It seems the switch over point from the engineering calculation to the fully resolved pure CFD calculation, happens at around eight to ten cells. Beyond this point one could see a sudden jump in heat transfer prediction as the mesh resolution is increased, all the while remaining within the +/-10% band compared to Flownex, see Figure 6.

Part II: External Flow over Heated Cylinder

So, what about the flow on the outside of the pipes, i.e. the shell side of the heat exchanger in question? To represent the 'shell side' flow we will consider the standard validation example of external flow over a heated cylinder. In this analysis only the heat transfer behavior is considered, and not the drag, per sé. Again the mesh is set up

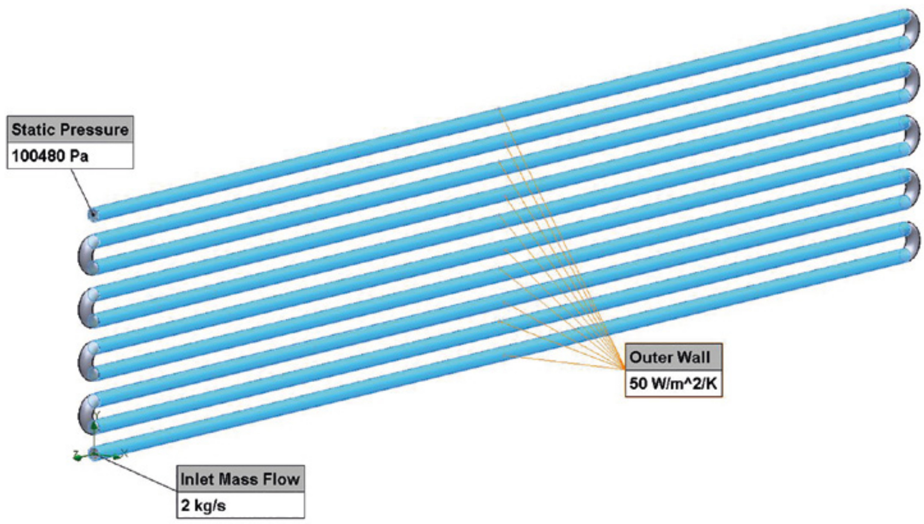

Figure 2. FloEFD model of 10-pass pipe layout

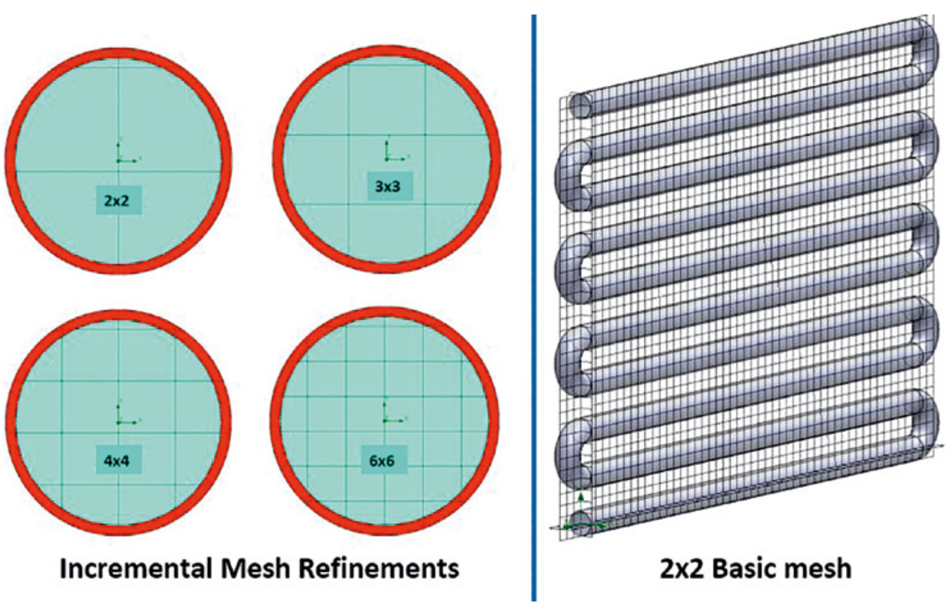

Figure 3. FloEFD mesh resolution

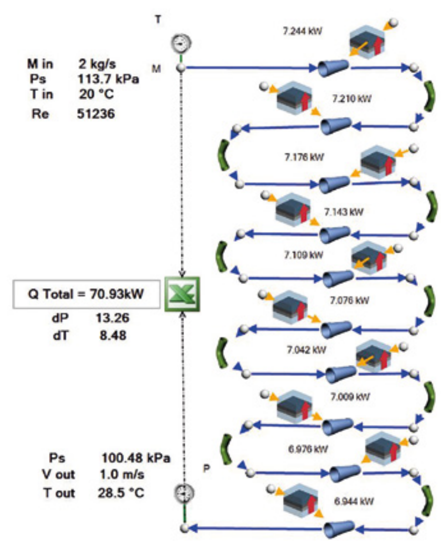

Figure 4. 1D Flownex model of 10 pass pipe layout

This article was originally published in Engineering Edge Vol. 7 Iss. 1

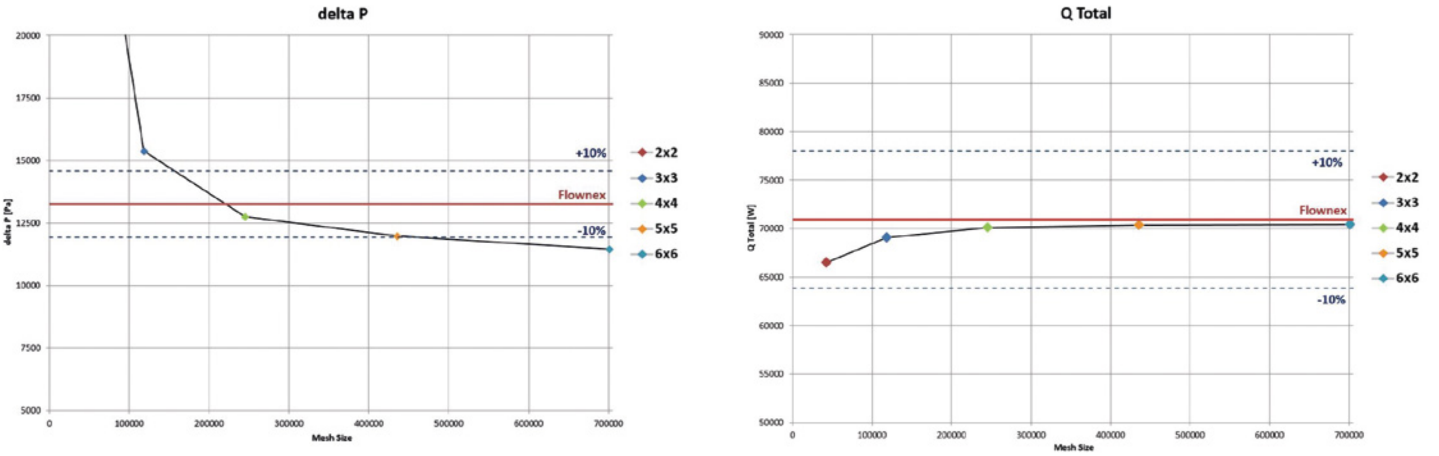

Figure 5. FloEFD versus Flownex results comparison.

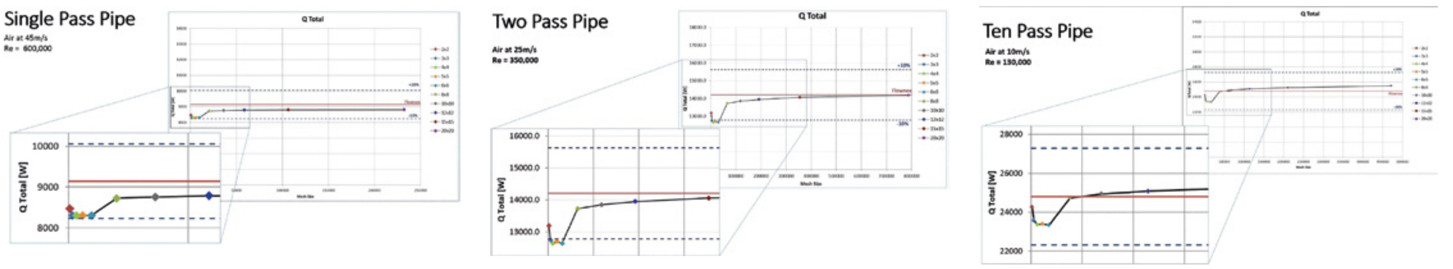

Figure 6. FloEFD versus Flownex results comparison: High Reynolds number example

such that the characteristic number of cells across the diameter was varied incrementally. Consider the graph in Figure 7 which shows the Nusselt number prediction for several mesh densities across a range of Reynolds numbers. It is evident from the graph that regardless of the mesh density the FloEFD prediction is very good, always within the scatter of the experimental data, even for extremely coarse meshes in CFD terms (four to ten cells per diameter). See especially the close-up image showing the 4x4 and 6x6 mesh results. This observation fortunately aligns very well with that of the internal flow results in that one should be able to generate very useful engineering results with meshes as coarse as just four characteristic cells across the pipe diameter (six cells would be ideal for this 'engineering' approach).

Part III: Full Heat Exchanger

The reason for all of this is basically to establish just how coarse a mesh one could dare to use when having to analyze large or complex heat exchangers like this. Since, you might find yourself in the same position as many engineers in South Africa, usually required to make do with limited computer resources. Therefore, it would be very beneficial if you can use CFD software that can double-up as an engineering tool to solve large problems on your standard issue laptop or desktop machine. And this is exactly where FloEFD starts to make a lot of sense.

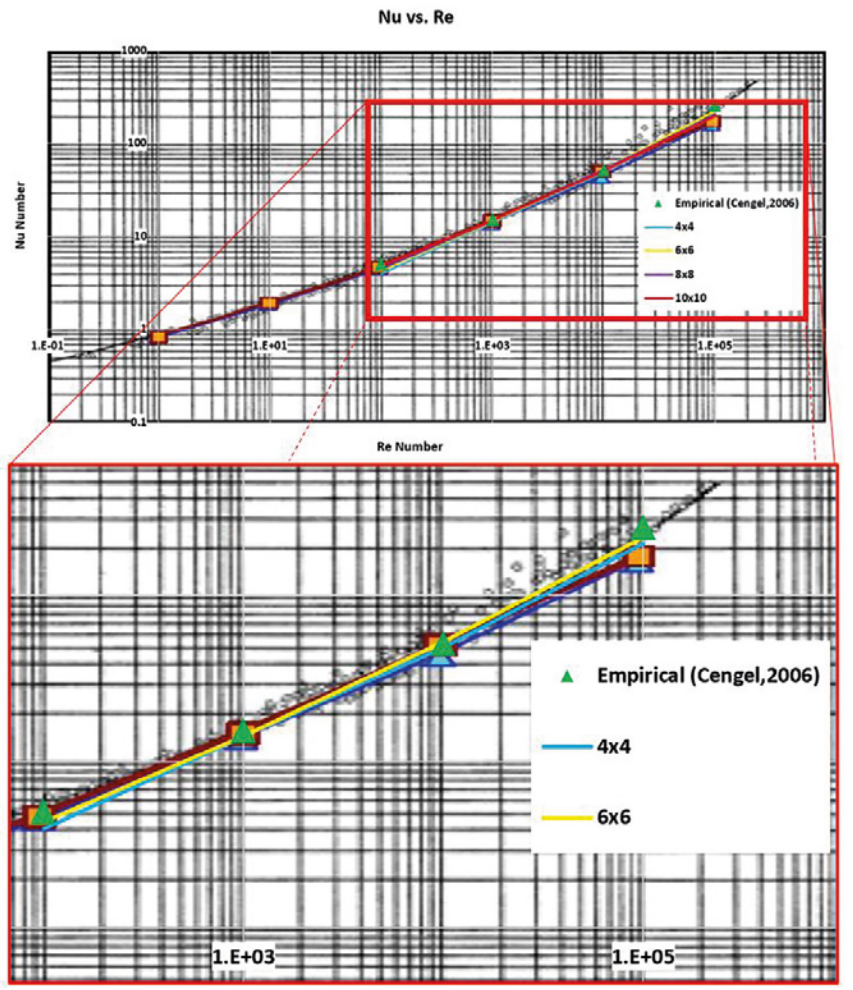

Figure 7. External Flow over a heated cylinder - Nu number comaprison.

In order to analyze the heat exchanger in question the only limitation would be the computer memory. By applying the knowledge gained from the preceding discussions, one could generate a mesh with four characteristic cells across the tube diameters and still have a high level of confidence in the 'engineering' answer. With the limit of 32GB memory, some stretching of the cells had to be applied to save a little on the memory requirements, thus stretching the cells away from the bends which resulted in a mesh size of approximately 5.7 million cells in total. See the resulting mesh on the tubes in Figure 8. Furthermore, on the question of the required calculation time, this particular model solved in a very respectable ten hours per travel with a mere quad core CPU, with the respective outlet temperatures already converged by 1.5 travels (flow freezing enabled). The resulting outlet temperatures obtained were, $T_{air,out}$ = 51.6°C and $T_{water,out}$ = 24.6°C. See the tube internal and shell temperatures in Figures 9 and 10. Based on the merits of the previous discussions, this result would already be very useful to base decisions on, especially when doing comparative studies of various baffle plate designs. It must also be stated that the solution runs very stable and convergence just happens. Quite astonishing considering the type of problem.

Conclusion

I have long since realized the value of FloEFD whenever it comes to solving heat transfer problems. However it has only now also become evident that FloEFD has made it possible for engineers to solve large problems like shell and tube heat exchangers with the minimum amount of effort and resources required, compared to 'old school' CFD programs, thanks to the underlying SmartCells technology and the ever-so-fantastic thin boundary layer approach. The only demand being placed on computer memory which limits the mesh size of these models. It is not just the ease of use and the minimal effort of setting up the model including the meshing, all of this would be useless were it not for the stability and robustness of the solver. FloEFD is simply the most reliable CFD software out there. From an Engineering in South Africa perspective, i.e. to be as resourceful as possible, FloEFD really resonates well with our kind of thinking.

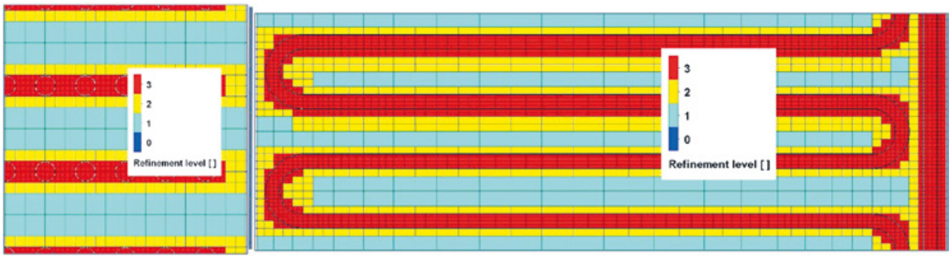

Figure 8. Heat exchanger tubes mesh refinement

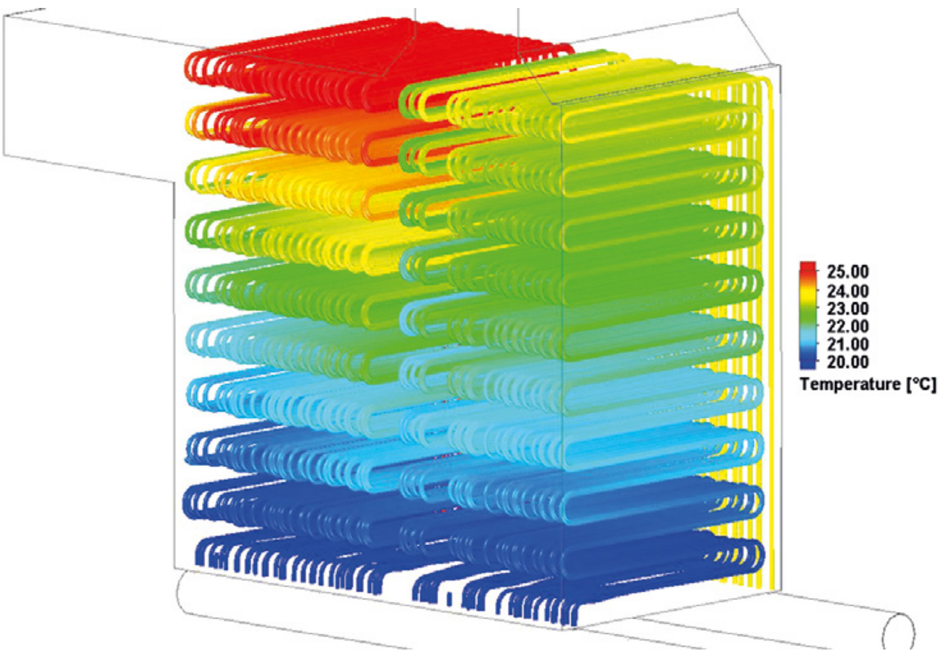

Figure 9. Tube-side water temperature flow trajectories

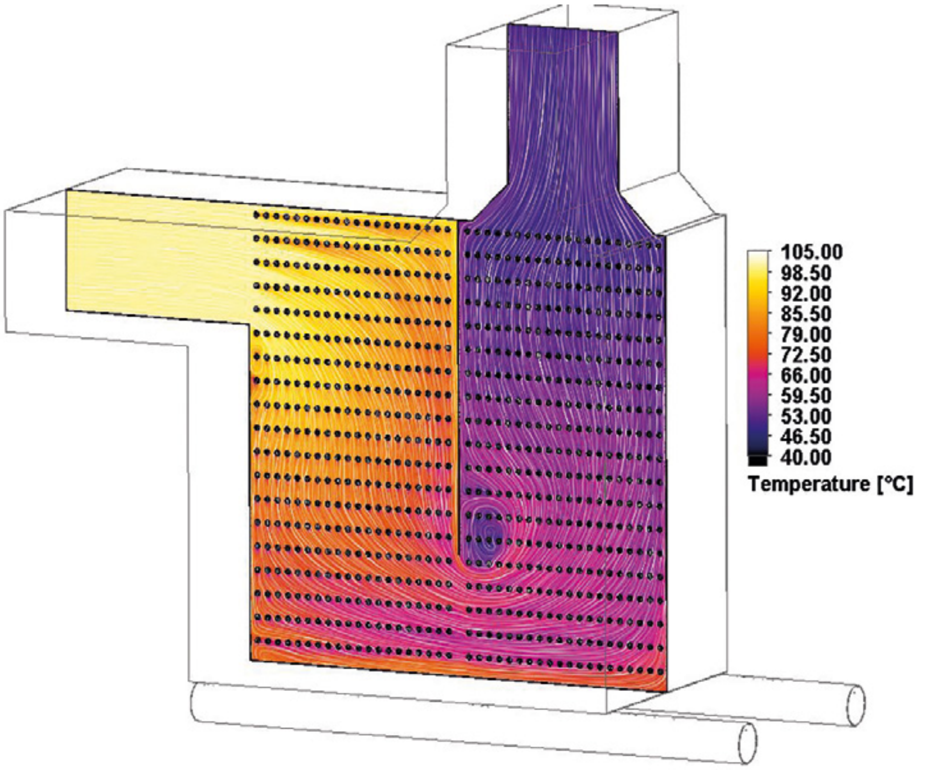

Figure 10. Shell-side air temperature cut plot

Mentor®
A Siemens Business

This article originally appeared in Engineering Edge Vol. 7 Iss. 1

Download the latest issue:
www.mentor.com/products/mechanical/engineering-edge

ENGINEERING EDGE

Accelerate Innovation with CFD & Thermal Characterization

You might also be interested in...

Thermal Simulation of a SYN TRAC Powerpack

Edge Devices are Hot for IoT

Distributed Chilled Water Pumping AC System

Reliability Evaluation of Sintered Copper Die Bonding Materials

Virtual Reality and Modeling Tools

Ask the GSS Expert

Simulation of Hydraulic Servo Gimbal Control of a Space Launch Vehicle

How to… Convert CAD Geometry into a FloMASTER Sub-System

A Study of Aerothermal Loads

Development of a Turbocharger with FloEFD

Risk Prevention & Energy Saving in Ships

Interview: Takuya Shinoda, Denso Corporation

Three Thermal Simulation & Test Innovations for Electronics Equipment Design

Series Car Brake Cooling

Liquid Cooling Turned on its Side

Thermal Analysis of an ADAS Camera

Thermal Design Leading the Charge

Understanding Die Attach Thermal Performance

Rapid Beverage Cooling Analysis

Bitcoin Mining: A Thermal Perspective

Designing High Efficiency, Low-Cost Synchronous Reluctance Motors

From Measurements to Standardized Multi-Domain Compact Models of LEDs

A Cool Emulator

Achieving Precision in 1D Dynamic Models of Hydraulic Servo Valves

Comparison between Experimental & Computational Results

Design of a High Speed Decoy UAV

Thermal Interface for Pluggable Optics Modules

Geek Hub: What do Engineers see in Coffee Grounds?