INVESTIGATION OF MESH INFLUENCE FOR HEAT TRANSFER IN PIPE FLOW

[1]Björn Jedvik*
[1]Semcon Caran AB, Sweden

KEYWORDS –
CFD, heat transfer, meshing, inflation layers, y+

ABSTRACT –

In order to improve performance of electric vehicles, a research project called Eldrivet was launched in 2011 where participants such as Volvo, Scania, Kongsberg Automotive, Lund University and Semcon gather and contribute with their respective competence.

Within the frame of Eldrivet project, Semcon have studied the performance of a fluid cooled inverter.

Meshes have been made using ANSA and different meshing parameters have been investigated such as y+-value, number of inflation layers, element type together with different CFD parameters in various CFD softwares.

Validation has been performed in Semcon's laboratory and an examination has been made in order to get an understanding of when simulations seem to correspond well to the experiments and when not.

TECHNICAL PAPER -

1. INVESTIGATION OF MESH INFLUENCE FOR HEAT TRANSFER IN PIPE FLOW

Electric vehicles have lately become serious competitors to vehicles with combustion engines, but their more powerful engines, batteries and power electronics demand better cooling than before. Therefore, an imitation of an inverter was produced as an aluminum piece (67x67x500 mm^3) with a drilled pipe (20 mm) covered with 6 thermal resistors generating heat of 1 kW (imitation of transistors). The aluminum piece is cooled with a 50/50 water-glycol mix with a flow range of 4-22 l/min.

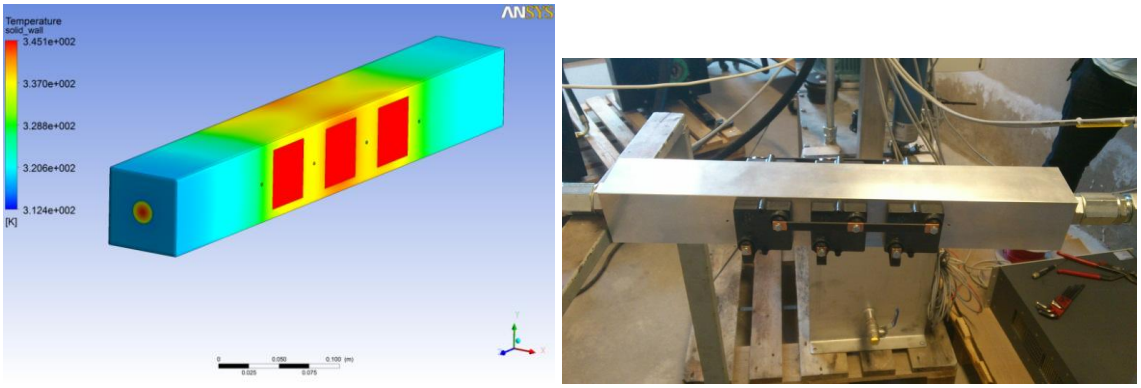

Figure 1 – Left: CFD simulation, Right: Experiment

Turbulator

In order to improve the cooling, a static-mixer turbulator was mounted in the pipe, see Figure 2.

Figure 2 – Static-mixer turbulator

2. SIMULATIONS

The simulations were made using CFX and different options in the software were tried out in order to achieve the best result.

Meshing

The meshing was made in ANSA were different strategies were tested.

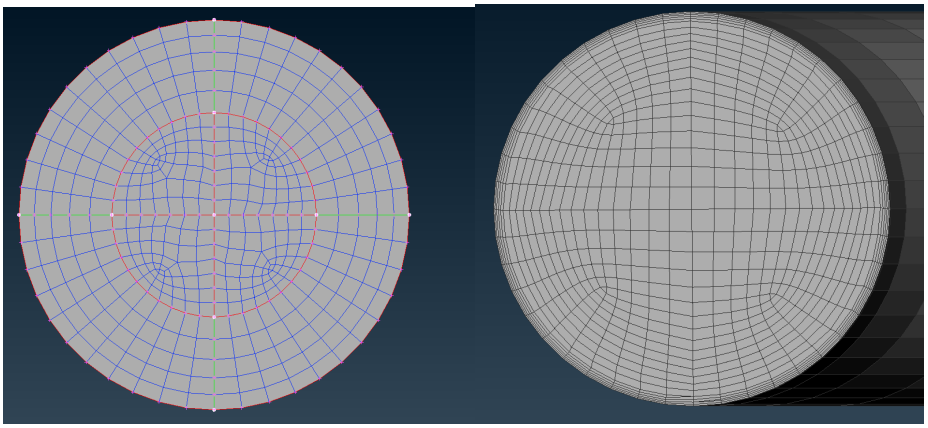

Figure 3 – Left: y+~40, Right: y+~1

3. RESULTS

When a mesh with y+~1 with 8 prism layers and hexahedras, the simulations correspond well to the experiments for Re > 5000, but for flows in the laminar or transitional regime, the CFD simulations under-predict the cooling, see Figure 4.

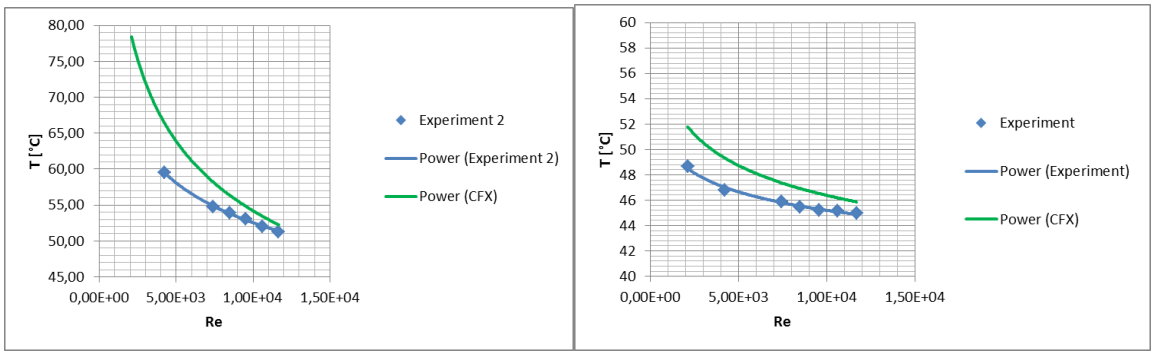

Figure 4 – Left: Without turbulator, Right: With turbulator

3. CONCLUSIONS

For flows in the fully turbulent regime, CFD simulations seem to work well to predict heat transfer in a pipe flow given that a proper mesh is made and proper options are used in ANSYS CFX. Anyhow, in the laminar or transitional the simulations under-predict the turbulence and hence the cooling effect.