CHAM Limited
*Pioneering CFD Software for Education & Industry*

CHAM Case Study – Numerical Simulation of a Air-to-air Cross-flow Heat Exchanger

PHOENICS demonstration case

A CFD model of a cross-flow heat exchanger was created following receipt of a specification from the Roads and Maritime Service (RMS) of New South Wales. RMS personnel were carrying out a numerical simulation study to predict the air flow and temperature distribution in the air-to-air type heat exchanger installed in their Variable Message Sign (VMS) system.
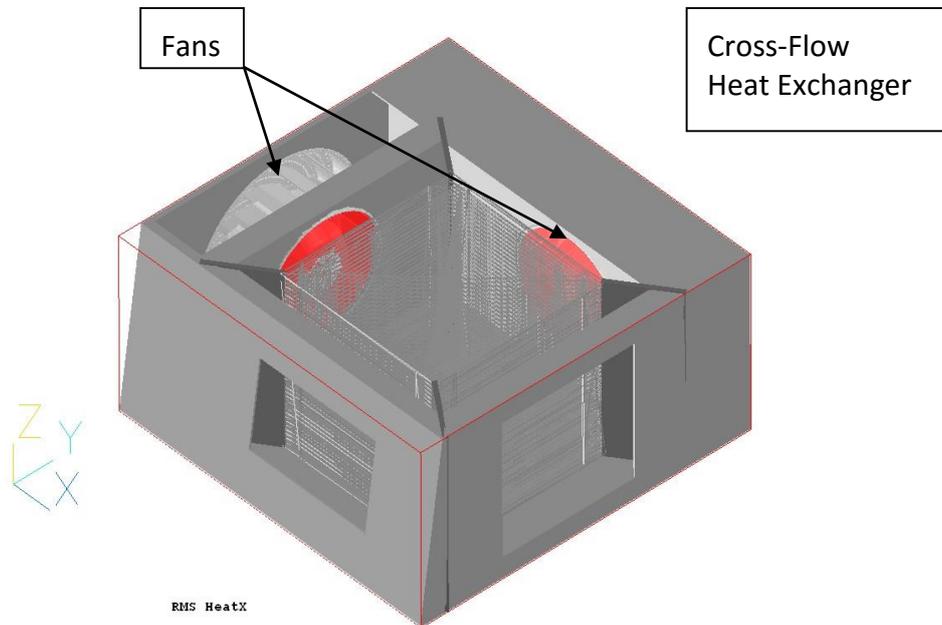
The electronics inside the VMS generate heat due to energy losses under normal operating condition. During the summer time, the temperature inside the VMS can be as high as 75˚C. The internal re-circulating air and the external cooling air are circulated by fan at a flow rate of 600m^3/h. The ambient temperature is 25˚C and the temperature of internal inlet air is 75˚C. A 2% inlet turbulence intensity was specified. The size of the heat exchanger element is 0.2m X 0.2m X 0.2m and it is epoxy coated aluminium. The rest of the enclosure is aluminium alloy grade 5005 H34 with 2.0mm thickness.

Cross-Flow Heat Exchanger

Arrayed Thin Plate objects

HeatX
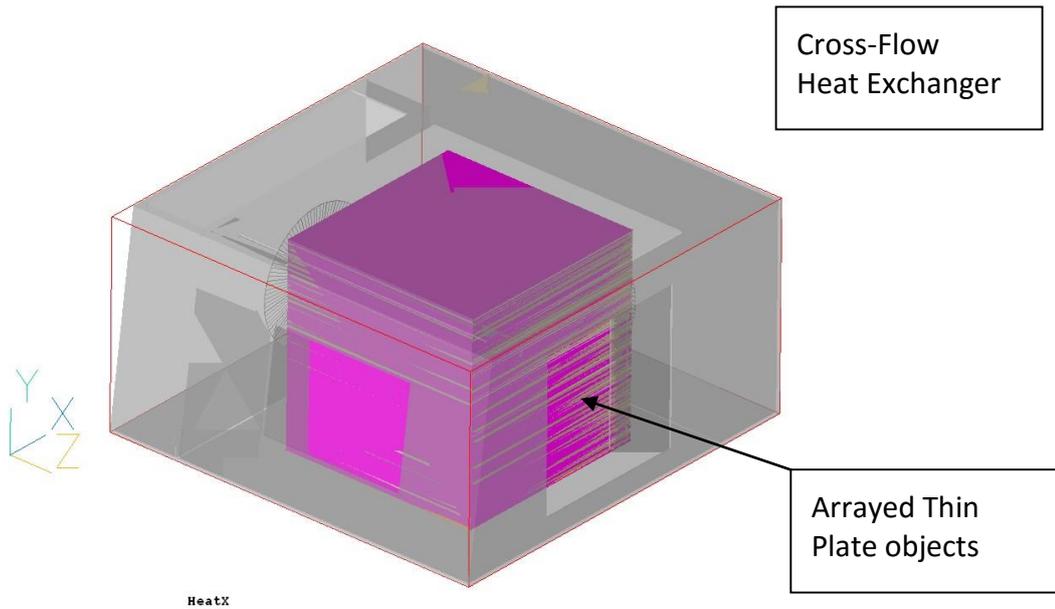
RMS engineers were primarily interested in using the CFD model to calculate and predict the thermal performance and effectiveness of the VMS heat exchanger under different boundary conditions (such as different internal or external inlet temperature, different air flow rate for exhaust fan etc) and how would they affect the results.

Velocity, m/s

| 30.61792 |
| 28.70430 |
| 26.79068 |
| 24.87706 |
| 22.96344 |
| 21.04982 |
| 19.13620 |
| 17.22258 |
| 15.30896 |
| 13.39534 |
| 11.48172 |
| 9.568099 |
| 7.654480 |
| 5.740860 |
| 3.827240 |
| 1.913620 |
| 0.000000 |

Hot In

Velocity

Cold In

HeatX
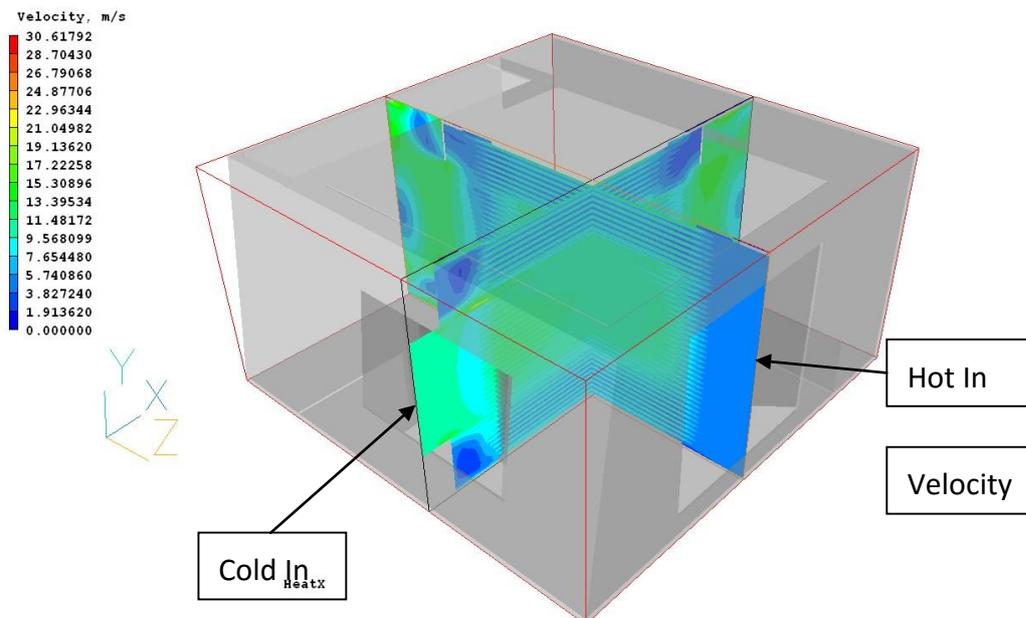
The above plot shows velocity of the hot air (near right to far left) and the cooling air (near left to far right). The next plot shows the temperatures of the two air streams, and shows the heat-exchange mechanism clearly.
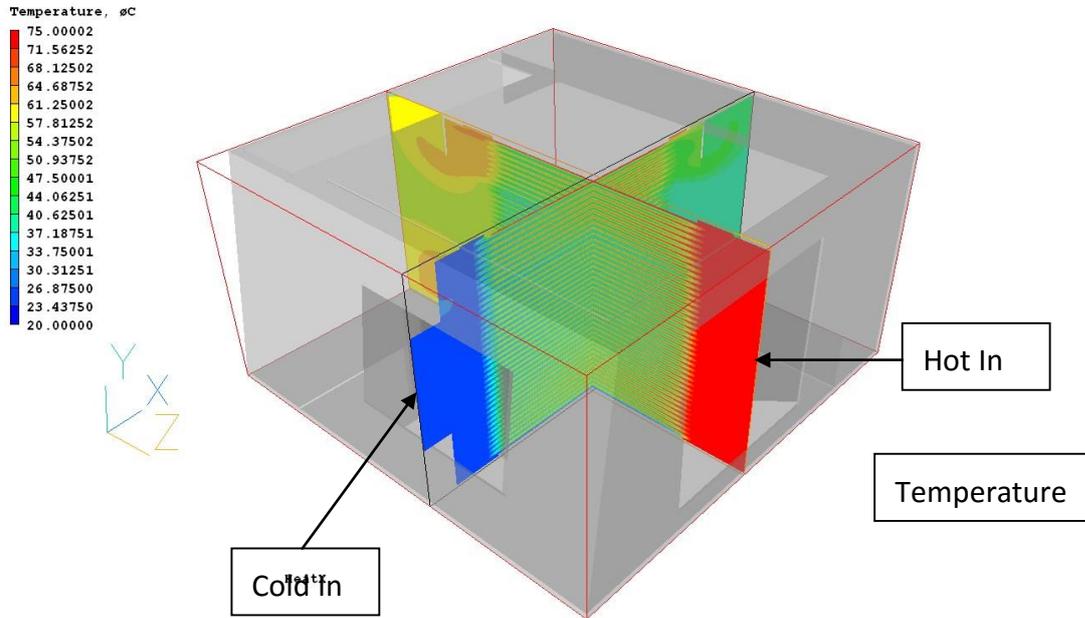

Two methods of modelling the heat exchanger were considered. The first option was to simply import the geometry from CAD and then apply a sufficiently fine mesh to capture the 80 x 2mm cross-sectional slots in the heat exchanger. A second, more pragmatic, method was adopted involving the replacement of this section with an array of "thin plate" objects using the same dimensions. The advantage of doing so is to remove the possibility of incorrectly defined geometry and to ensure that the heat transfer is based on the correct plate thickness, whilst using a much smaller computational mesh.
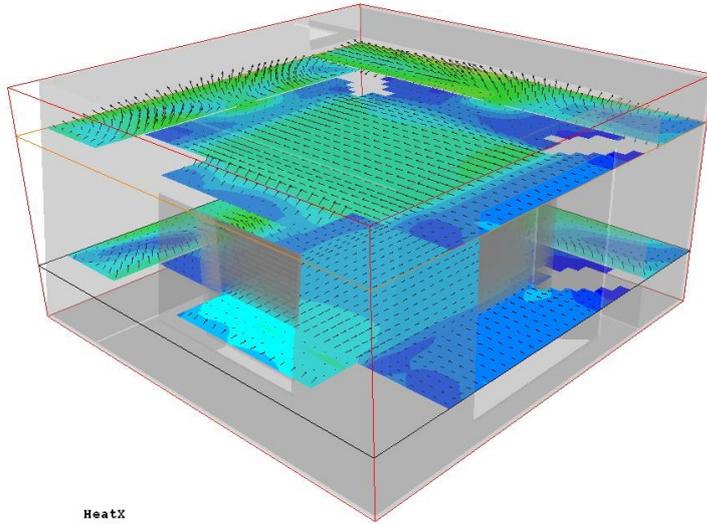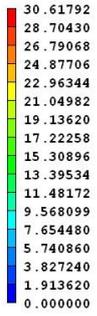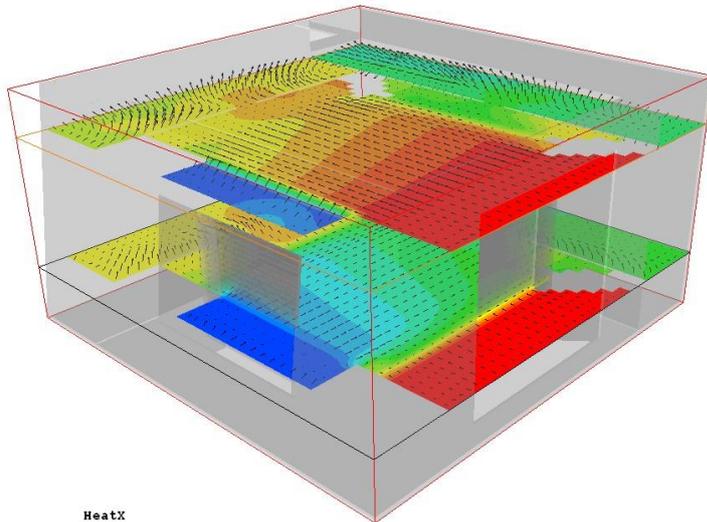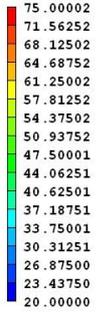
Velocity vectors and contours

Velocity

Temperature

Velocity vectors and temperature contours

The above two plots show velocity and temperature respectively, in a hot-air channel (above) and a cooling-air channel below. The cooling of the hot air and the warming of the cold air within the heat exchanger are clearly shown.

Details of the model set-up
A 3D Cartesian mesh was employed with 40 x 168 x 40 cells. PARSOL was not activated, as it is not required for the rectangular geometry of the heat exchanger. The LVEL turbulence model was used because of its suitability for flows through narrow channels, with a sparse grid resolution across the channels. The runs were reasonably converged after 1000 sweeps; this required a 75 minute run time on a
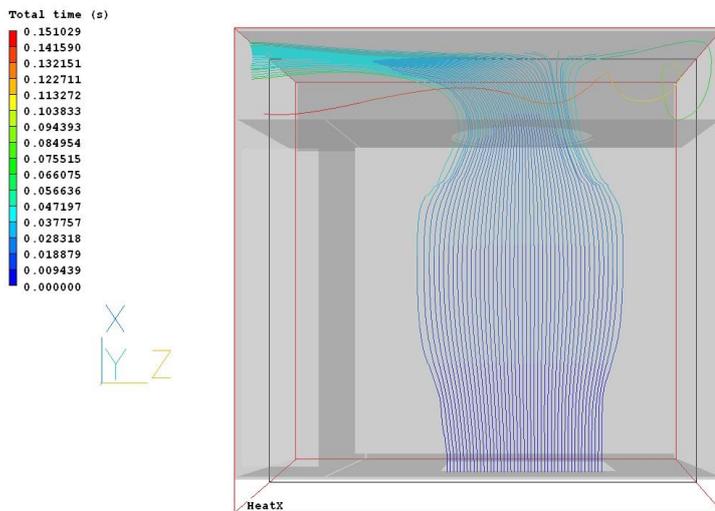
single-processor 3GHz system.

Streamlines X

Total time (s)

0.063302
0.059346
0.055389
0.051433
0.047477
0.043520
0.039564
0.035608
0.031651
0.027695
0.023738
0.019782
0.015826
0.011869
0.007913
0.003956
0.000000
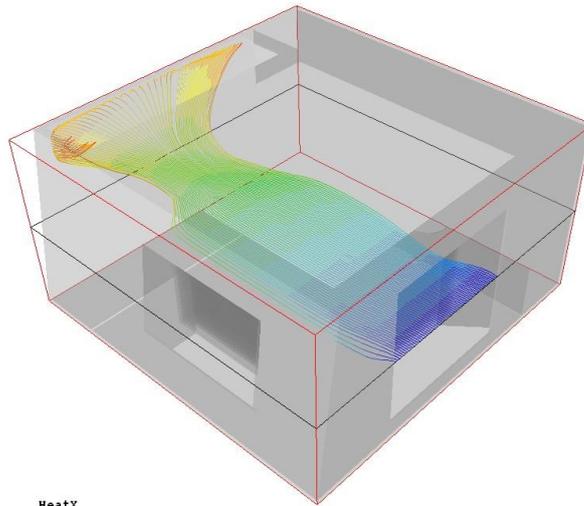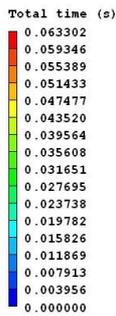
HeatX

Streamlines -Z

Conclusions

The model provides a good representation of the heat transfer processes in the heat exchanger. Close examination of the model predictions reveal non-uniformities within the heat exchanger. Air from the inlet plena impinges straight onto the central channels, while the outer channels receive air via a more circuitous trajectory. This means that the air velocities in the central channels are somewhat higher, giving better heat transfer. This can be observed directly in the temperature plots. If it were possible to spread the air more uniformly, the overall heat transfer efficiency might be improved. The CFD model can therefore function as a design testbed, which engineers can use to improve the performance of the unit.

Email: info@arcofluid.fr Web: http://www.arcofluid.fr

Supplementary images – Temperature & velocity @ Y & Z planes
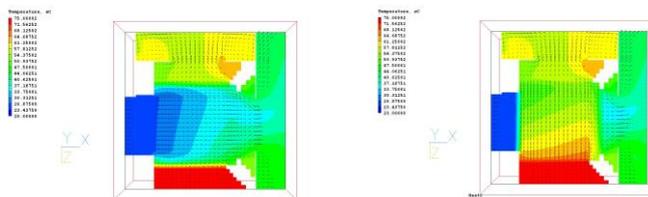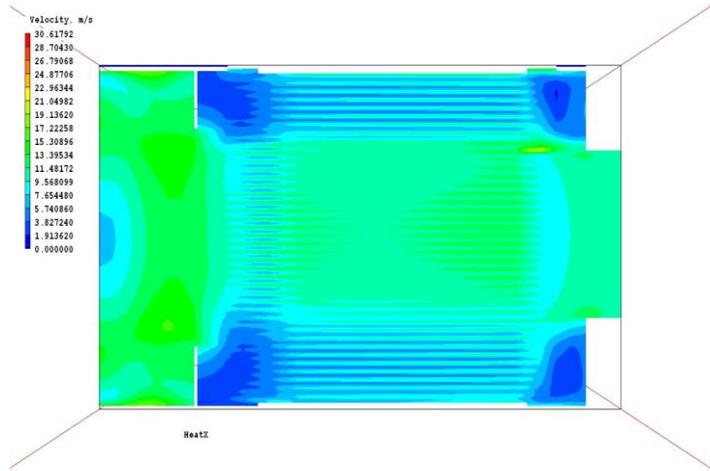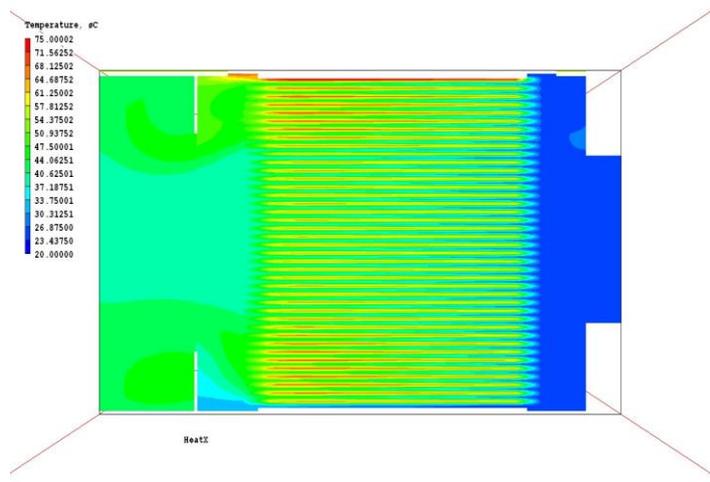
HeatX

HeatX

Temperature, øC
75.00002
71.56252
68.12502
64.68752
61.25002
57.81252
54.37502
50.93752
47.50001
44.06251
40.62501
37.18751
33.75001
30.31251
26.87500
23.43750
20.00000
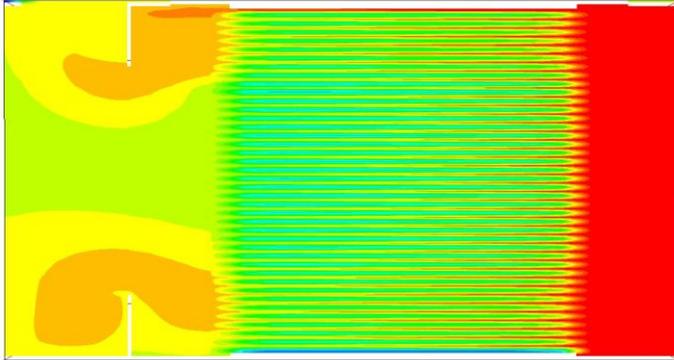
HeatX

Velocity, m/s
30.61792
28.70430
26.79068
24.87706
22.96344
21.04982
19.13620
17.22258
15.30896
13.39534
11.48172
9.568099
7.654480
5.740860
3.827240
1.913620
0.000000

HeatX