HEFAT2008
6th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
Pretoria, South Africa
Paper number: LM2

MODELLING OF A THERMOSIPHON EFFECT IN A RADIATOR

M. Lahoubi, C. Pennel,
Institut Catholique d'Arts et Metier, Department of Energetics.
6, rue Auber. 59046 Lille Cedex. France.

D. Rousselle,
Zehndergroup, Electrical Product Development
17, rue des Parachutistes de la France Libre. BP F-02110 Vaux-Andigny. France

ABSTRACT

Heat exchangers have multiple applications. We find them in domestic heating, solar heating systems and also in many cooling systems. Our paper is concerned with oil filled electric radiators. This work gives our conclusion concerning a change in design of one model and its consequences on the internal fluid circulation.

INTRODUCTION

The radiators on which we are working are based on the principle of the thermosiphon. In 1823 Thomas Fowler patented the first convective heating system. When Fowler invented the Thermosiphon, the only other known convective heating system would have been the Roman hypocaust. Thermosiphon (alt. thermosyphon) refers to a method of passive heat exchange based on natural convection which circulates liquid in a vertical closed-loop circuit without requiring a conventional pump. Its purpose is to simplify the pumping of liquid and/or heat transfer, by avoiding the cost and complexity of a conventional liquid pump. Convective movement of the liquid starts when the bottom of the loop is heated, causing the fluid to expand and become less dense, and thus more buoyant than the cooler fluid in the top of the loop. Convection moves heated liquid upwards in the system as it is simultaneously replaced by cooler liquid returning by gravity. In many cases the liquid flows easily because the thermosiphon is designed to have very little hydraulic resistance. Our case is a simple thermosiphon but in other cases when the loop has more resistance to flow, the liquid may be heated beyond its boiling point, causing a phase change as the liquid evaporates.

The type of radiator that we studied uses radiation and convection to heat the surrounding. By combining these two modes of heat transfer, these radiators are known to have a smooth diffusion of heat due to their inertia. However, the inertia is inconvenient during the warm up period. We would like to have a faster increase in temperature and homogeneous surface temperatures. The distribution of temperature is important because we have to avoid hot spots. With this aim, this work consists in improving the radiator.

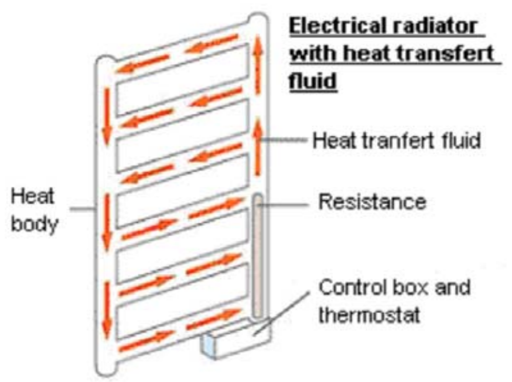

Figure 1 : Thermosiphon in a radiator filled with oil

Natural circulation is still the subject of research today. It refers to the ability of a fluid in a system to circulate continuously, with gravity and change in temperature being the only driving force.
One of the major drawbacks to natural circulation is that fluid does not typically move very fast; it is only capable of very low flow rates. In this radiator we have flow rates between 10^{-3} and 10^{-2} kg/s. The other problem is that if for some reason flow is stopped in the loop, there are cases in which flow could start up in the opposite direction. Another thing complicates our case since we have parallel flow paths so that the description of the

fluid flow is not obvious. Without the help of a numerical code it is too complicated to solve the problem.
Heat exchange in these radiators is complex ; conduction, convection and radiation are coupled. The emissivity E of the outer surface depends directly on the material which radiates. For oxidised mild steel the emissivity may be 0.5 whereas the metal used in the radiator is covered with a coat of white paint, giving an emissivity of about 0.95. With a matt black painting, the coefficient would be closer to unity. The distribution of the fluid and the temperatures inside the unit are not obvious when designing a new model. The idea is to create a CFD model in order to contribute to good design and optimization of new prototypes.

MODELISATION

The numerical calculation was carried out using the Fluent code, which employs the finite volume method. Parameters in Fluent were validated by comparing numerical results with experiments in the laboratory. In the laboratory we used an infrared camcorder and thermocouples on the surface of the radiator to make our measurements. This enabled us to adjust the parameters and so modify and improve the models.

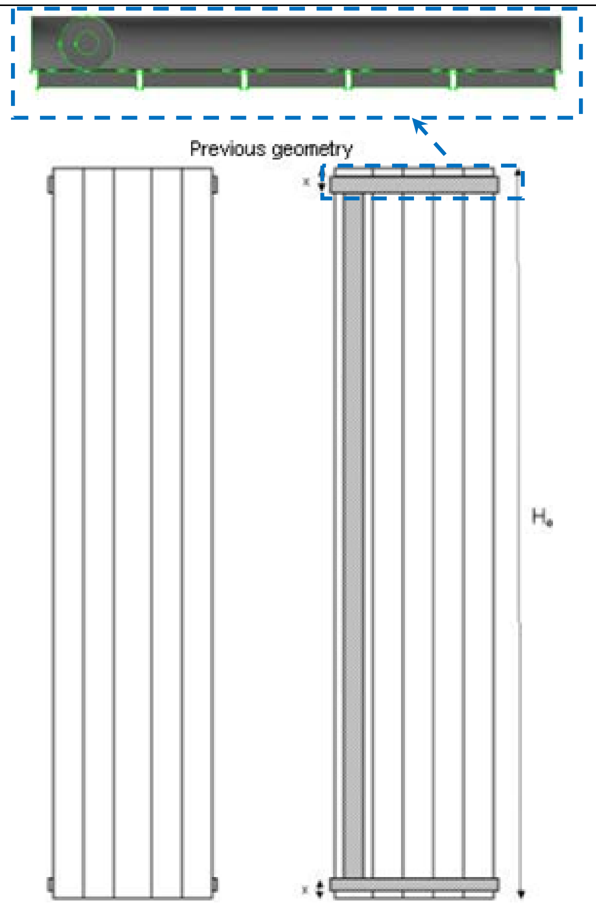

Figure 2: Layout of Heater

Geometry

The radiator is composed of five elements, with two horizontal manifolds and one vertical manifold with an electrical resistance. The five elements are disposed side by side and are connected to the manifolds by welding. Flow takes place through 7 mm holes. The connection is completed by bridges for the last element to enhance the fluid circulation. The electrical resistance is placed in the vertical manifold and gives a power of 750W.

Mesh

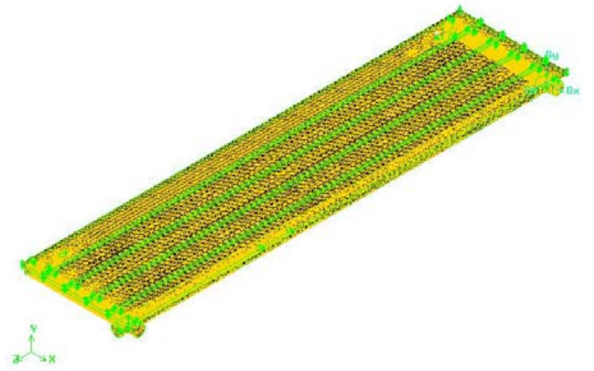

Figure 3: Radiator mesh

We meshed the radiator optimizing the number of volumes. We divided the geometry in zones to refine meshing especially in the critical parts like the junction between elements and manifolds. We ended with about 300 000 volume elements.

Boundary conditions

The radiator is a closed loop bounded by steel walls with emissivity 0.95. There is an electrical resistance in contact with the fluid; the fluid temperature rise was measured in experiments. We simulated the warm up according to the measured curve that we made with a resistance plunged into oil. These experiments led us to a representative equation of the warm up. Then we employed user-defined function or UDF files which make it possible to define personalized conditions of temperature.
An UDF is a programmed function which can be dynamically loaded with the FLUENT solver to enhance the standard features of the code. UDFs are written in the C programming language. They access data from the FLUENT solver using predefined macros and functions. UDFs are either interpreted or compiled and are hooked to the FLUENT solver using a graphical user interface panel. We used UDF to customize the boundary conditions of the resistance.

Solver model

We chose the method of solution as "Coupled". This option allows the controlling equations to be solved simultaneously. Initially, this mode was conceived for compressible flows at high speeds. This gives it an advantage for the treatment of highly coupled flows (strong interdependence between mass, momentum and energy) with body forces (e.g. buoyancy and rotation). It should be noted that the implicit coupled solver requires almost double the memory that would be used with the isolated solver, whereas the explicit coupled solver comes in between, in term of resource requirements. This, however, converges more slowly than the implicit formulation and is advised only for non-stationary flows.

Computing

The simulation represents the behaviour of the radiator during warm up and cool down, a period of about two hours. We divided the time period into time steps adapted to the difficulty of convergence. For the beginning of warm up we had to use very small steps of less than a second and a large number of iterations to achieve convergence. It is important to obtain convergence because the final result of a time step is used to compute the next one and so on. Later in the transient the time steps were 5 to 10 seconds.

MODEL AND PARAMETERS VALIDATION

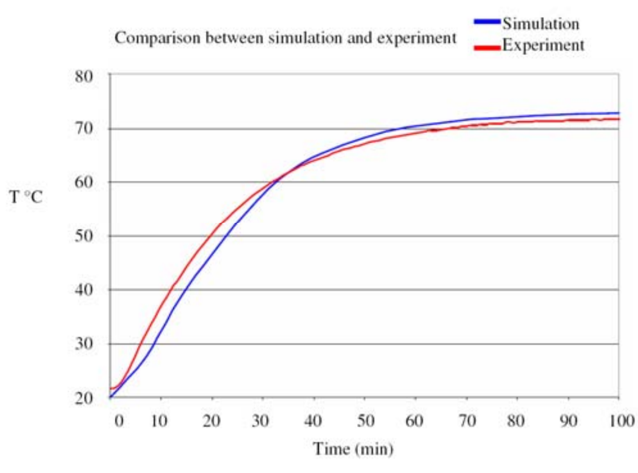

Figure 4: Evolution of temperature in the previous geometry, comparison between experimental data and computational (CFD) during 100 minutes.

We showed that the variation between the model and the experiment is less than 10%, which is satisfactory taking into account various uncertainties of measurement. Results from the model and real radiators are close enough to validate the design assumptions. On these curves we can also notice the strong inertia of the radiator since it is necessary to wait more than one hour to reach the stabilized maximum temperature. Can we reduce this inertia?

Design of a new geometry

The idea of the new model is to increase the internal flows within the radiator in order to facilitate the distribution of heat, especially by reducing the pressure losses. We study modifications of the manifold including resistance according to the following geometry:

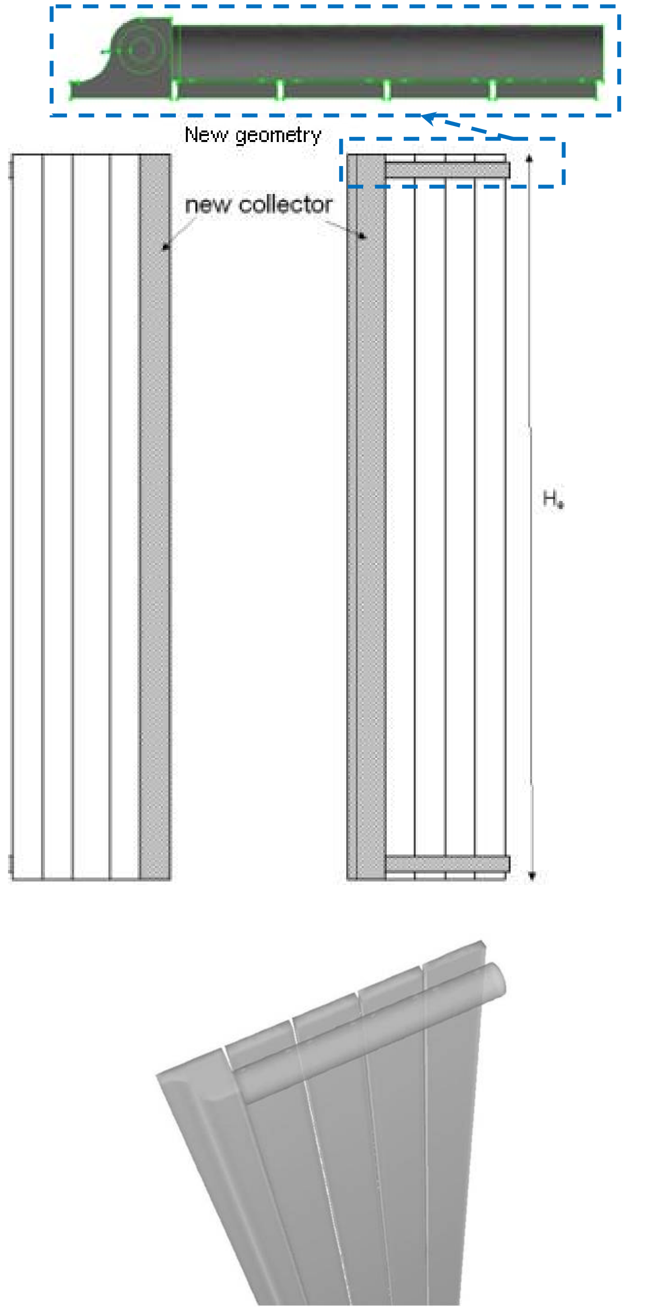

Figure 5: New radiator design

In the new model, the heater housing is combined with the first radiator element.

This avoids the fluid by passing the 7 mm openings, which generate the pressure losses. Before any calculations are performed we can be confident that the radiator will heat up more quickly. With our model, we could test this new arrangement.

Problems of circulation are caused only by the small holes. They create too much head loss and disturb the thermosiphon. We just act on four holes so we can't expect big changes in the whole radiator.

TRENDS AND RESULTS

During the period of test, the new heater element shows that the effects of the new design are limited. The hole suppression in the elements should be carried for the rest of the radiator. But the radiator has to be produced in large quantities and the production tools don't allow bigger holes between elements.

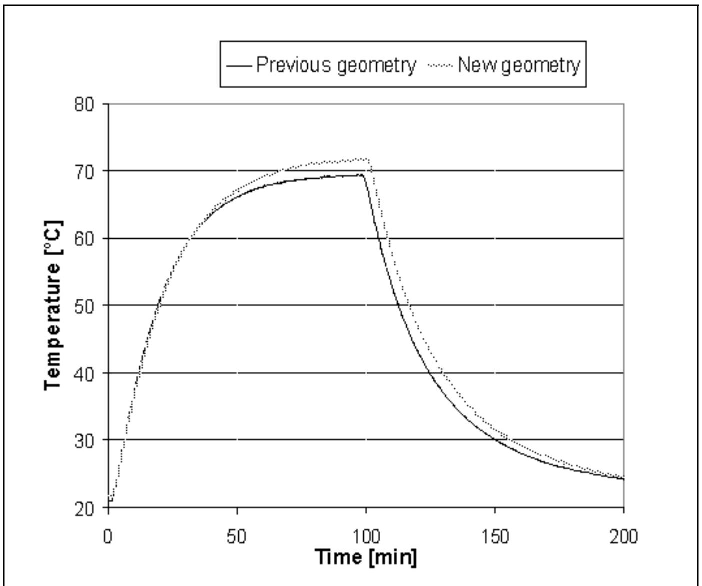

Figure 6: Comparison of the evolution of temperature between the previous and the new design

We obtain higher temperature during the same delay of the warm up and a better inertia during the cooling period. We effectively obtain very little improvement because of the problem of small holes.

We observe another phenomenon. Indeed, the observation presents a double circulation typical of convection in the manifold. The double direction of circulation in the manifold seems to occur because of the smalls holes not modified in the rest of the radiator. The thermosiphon occurs where it's easier to flow so it is like the convection happens only in a vertical long pipe. The fluid cools down in contact with the wall and runs between the wall of the manifold and resistance.

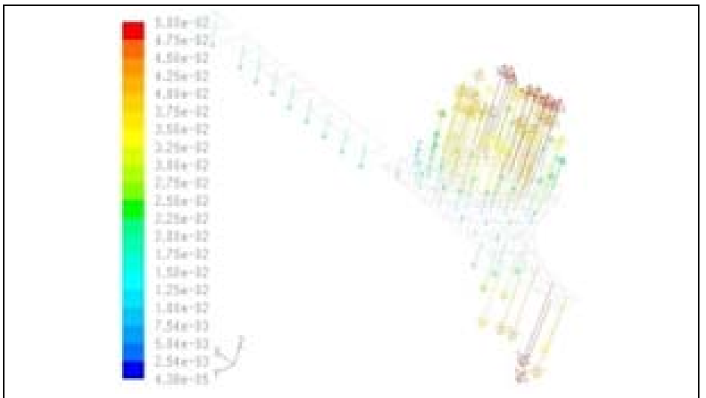

Figure 7: Double direction of circulation in the manifold.

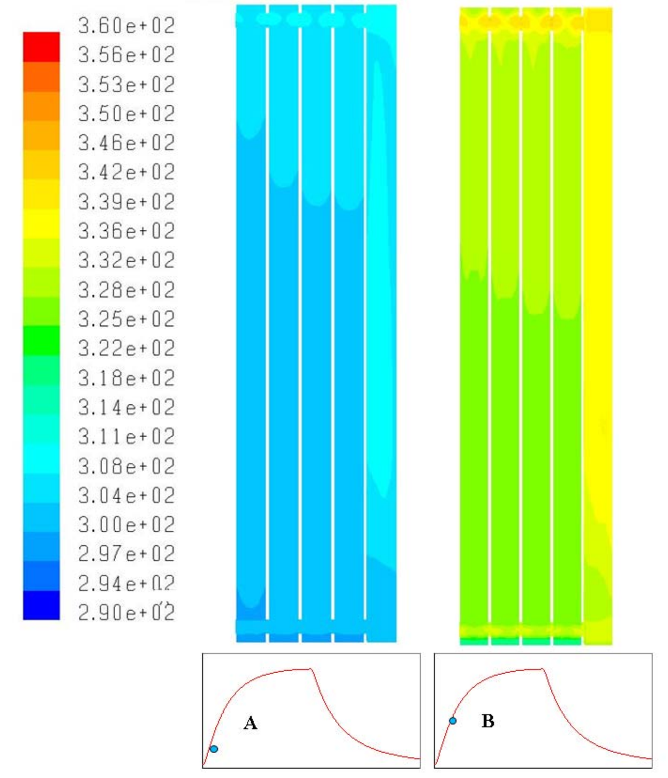

Figure 8.1: Contours of temperatures during heating in the new geometry.

A: t = 450sec. **B**: t = 1623sec.

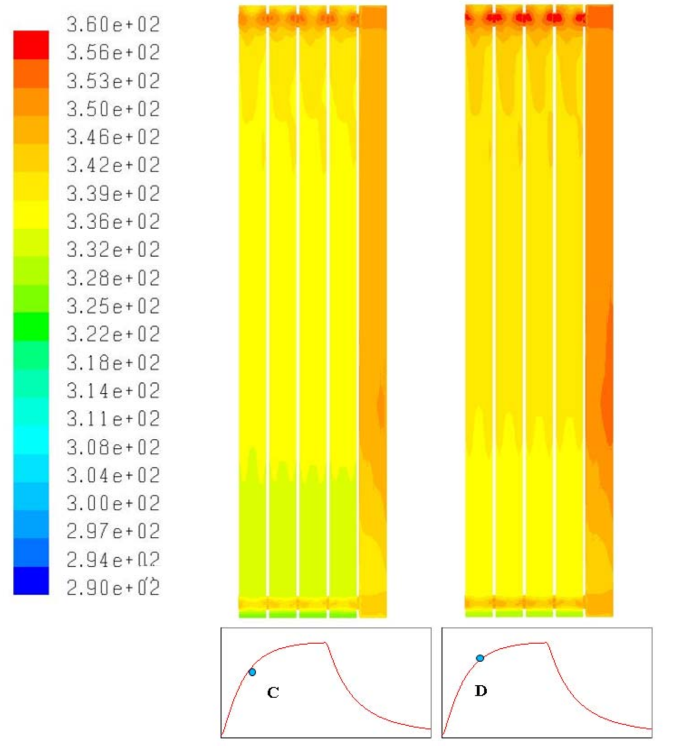

Figure 8.2: Contours of temperatures during heating in the new geometry.
C: t = 450sec. **D**: t = 1623sec.

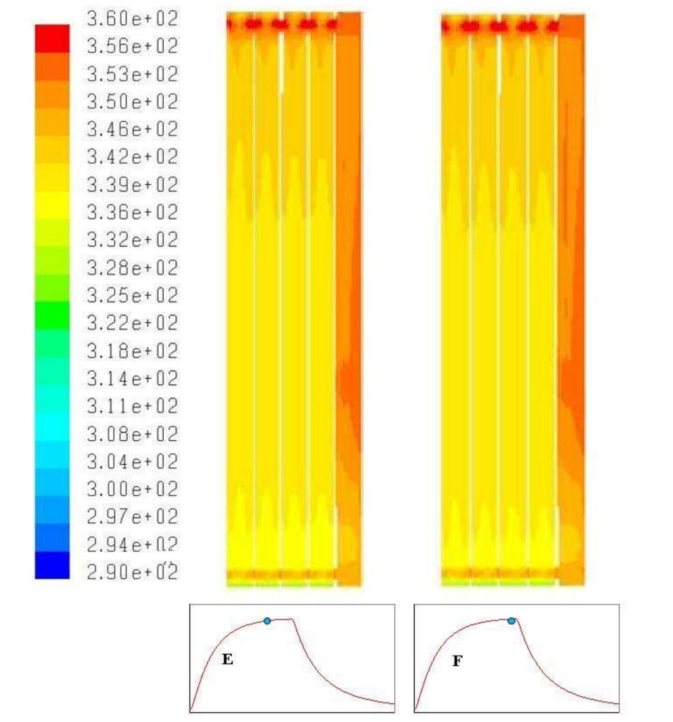

Figure 8.3: Contours of temperatures during heating in the new geometry.
E: t = 6623sec. **F**: t = 8822sec.

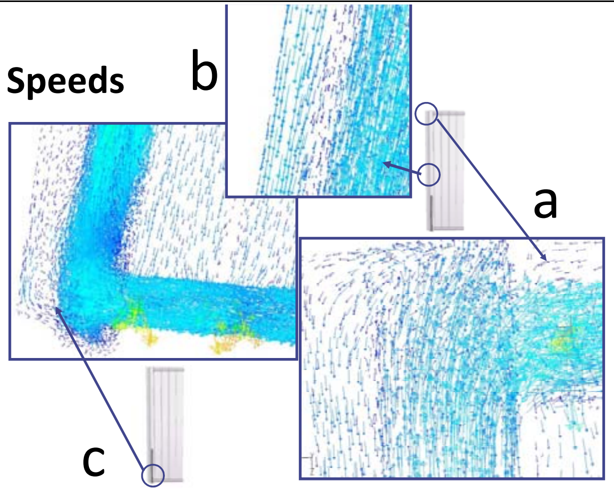

Figure 9.1: Velocities in the radiator.
(a): Recirculation flow at the bottom of the vertical collector.
(b): Fluid flow in the vertical collector.
(c): Velocities in the entrance of the horizontal collector.

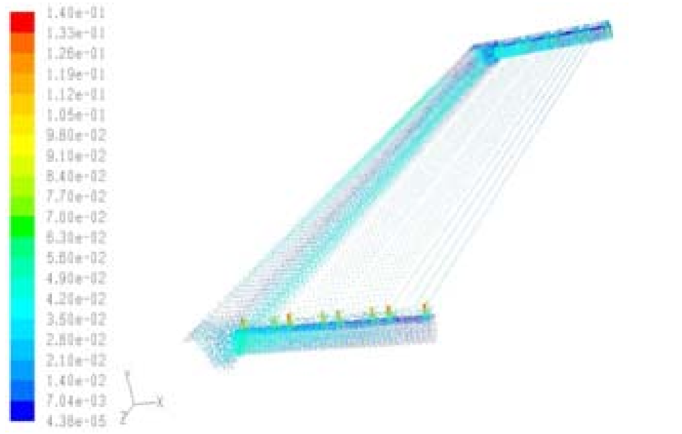

Figure 9.2: Overview of velocities in the radiator.

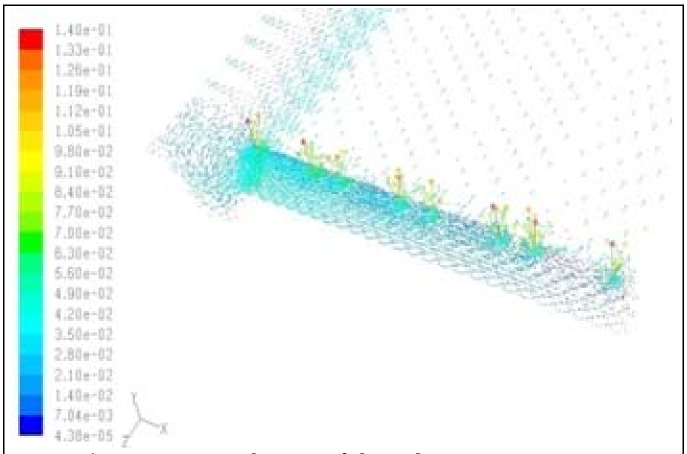

Figure 10: Distribution of the velocities at junction.

On the figure above we can distinguish the highest velocities ; (around 2e-1 m/s) they are caused by the reduction of section. A good way to model the radiator is to see it like an electric circuit comprising resistances in series and parallel. Our modifications were carried out only on the suppression of the first series of resistances; the total effects are visible but limited.

CONCLUSION

In this study, we observe that the circulation is ensured by the natural convection, the control of the pressure losses is of primary importance in order to obtain an homogenous surface temperature.

The numerical simulation is close enough to reality to avoid a lot of expensive prototypes. By designing the radiator to reduce loss of head, we manage to improve the performance of this system. The action on pressure losses produces good effects on inertia and temperature repartition.

REFERENCES

[1] Franck P. Incropera, David P. Dewitt, Theodore L. Bergman, *Introduction to Heat Transfer*, Fifth Edition, 2007 John Wiley & Sons,Inc.

[2] James R. Welty, Charles E. Wicks, Robert E. Wilson, *Fundamentals of Momentum, Heat and Mass Transfer*, Second Edition, John Wiley & Sons.

[3] W. M. Kays, A. L. London, *Compact Heat Exchangers*, Second Edition, Mc Graw-Hill book Compagny.

[4] Robert W. Fox, Alan T. Mc Donald, *Introduction to Fluid Mechanics*, Fourth Edition, John Wiley & Sons, Inc.

ACKNOWLEDGMENTS

We want to express our heart-felt thanks and gratefully acknowledge the contribution of Dr Brian Axcell, and of many ICAM and Erasmus students: Jasmina Titos Ruiz, Thibault Dumortier, Romain Chanzy, Alexandre Dutot, Mélanie Selosse, François Laridant.