A PRACTICAL APPROACH TO WATER JACKET FLOW ANALYSIS

Sunil K. Jain
Performance Engine Products Division
Caterpillar Inc.
Mossville, IL 61552

Abstract

The process of optimization of the cooling system in a diesel engine involves the computation of the coolant flow field in the jacket. This field not only helps in identifying potential problem areas in the jacket, but also provides the convective heat transfer coefficient field, one of the most important boundary conditions for the thermal analysis of the head. This field can be obtained only through three–dimensional flow analysis of the jacket. This paper describes the use of VECTIS, a general purpose computational fluid dynamics software from RICARDO, to obtain such a field in a cost effective manner for three different jackets. Sample results in the form of velocity vectors and convective heat transfer contours are given for one of the jackets. However, more emphasis is placed on the methodology than on the results.

Introduction

The market trend towards the higher ratings of a diesel engine is pushing the cooling system to its limit. This necessitates the use of a non–traditional method to further improve its performance. Traditionally the optimization of a cooling system is done on a test bench through a trial and error method. This method is not only costly and time consuming, but also can not provide the details of the velocity field in the jacket. The only practical method to achieve this field is through three–dimensional computational fluid dynamics (CFD) simulation of the jacket. This field helps the designer in further improving the system by making sure that there is high flow only where it is required which leads to efficient management of temperature field with the minimum volume of the coolant.

Designers have been aware of the advantage of CFD in the process of optimization of a cooling system. However, its use is hampered by very long turnaround time.

This long turnaround time is generally caused by the grid generation process. Hence, a faster grid generation technique was needed. This need was addressed by RICARDO through the introduction of an automatic grid generator in their commercial CFD software VECTIS. Following sections describe the use of this software to perform the water jacket flow analysis.

Technical Approach

The VECTIS package consists of a pre–processor, a solver and a post–processor. It solves the three dimensional flow equations governing conservation of mass, momentum and energy for laminar or turbulent, compressible or incompressible, and steady or unsteady flows. Standard k–e model is used for turbulent flows. The salient features of the code are the use of stereolithographics (STL or SLA) format for surface definition, its automatic grid generation capability and the interface with various finite element analysis (FEA) software to map the heat transfer coefficient for thermal analysis.

Surface Definition:

The advantage of STL format for surface definition can not be over–emphasized. In this format, the whole surface is defined by a set of triangular faces. Because of its generic nature, it becomes extremely easy even for an inexperienced user to manipulate the files in the VECTIS pre–processor in order to completely close the triangulated surface model for the mesh generator.

Figure 1 shows the level of difficulty to obtain the STL definition from the available surface description in three different cases.

In the first case, there was a solid model available for the head and the block water jacket, and it took less than an hour to obtain the required STL file.

In the second case, the available CAD model was not completely closed and there were lots of overlapping surfaces. The model could not be fixed in the CAD package because of the incompatibility with the current version and the one it was developed on. In this case, the file was exported to STL format and was read into VECTIS pre–processor. Here the triangular surfaces were manipulated to obtain a perfect surface definition of the model. Though it took more than a month to stitch the file, the analysis may not have been possible without the STL manipulator.

In the third case, because of the absence of a CAD model, eight sand cores sections of the cooling passage were digitized by RICARDO to obtain its STL definition.

Though a good STL model was obtained for the complete passage, its size was more than 600 MB. These files were further compressed to approximately 60 MB using another software. This process took nearly one month. But again it was the STL format which made the manipulations, like merging different sections of the head, easy. This may not have been possible though the use of another format, like IGES, as for each section, the size of an IGES file would have been more than 100 MB.

Computational Domain:

Figure 2 shows the computational domain used for the analysis. It consists of a block water jacket and a head water jacket of a six cylinder diesel engine. This computational domain was obtained by merging a Pro/Engineer generated STL model of the block water jacket with the UG generated STL model of the head water jacket. The decision to perform the analysis on the complete jacket instead on a section of the head was made because of the nature of the cooling circuit. Any simplification to the extent of the domain would have given misleading results. As seen, this model is extremely large, and in our experience, this was the first time a CFD analysis was performed on an actual geometry of this size. The whole process of merging and stitching different STL files took less than a day.

Mesh Generation:

This step has been extremely simplified through the availability of an automatic grid generator in VECTIS. Generally this step is the bottle neck while performing CFD analysis for a complex configuration. This generator requires the user to define a global Cartesian mesh over the whole domain. It took nearly 4 hours to define the global mesh for the domain shown in Figure 2. Local refinements were defined near the zones of interest which are around injector sleeve and valve bridge areas. Different boundaries, like the wall, inlet and outlet, were defined further. Thereafter, it took nearly 30 hours on a HP 755 system to generate the computed mesh through the grid generator. Figure 3 shows untrimmed cells on the surface of a section of the domain, while a CFD mesh on a small section is shown in Figure 4. The local refinement near valve bridges and injector sleeve can clearly be seen in Figure 4. The grid generation process generated 380,000 CFD cells.

Boundary Conditions & Solver:

Mass flow rates of 4.0 kg/s were defined at the inlet as well as at the outlet. The properties of a 50/50 mixture of water and ethylene glycol at 110 degree C were used for the analysis which was assumed to be isothermal with wall temperature fixed at 110 degree C. PISO scheme was selected for velocity – pressure coupling. The analysis took 15 days on a HP755 system with 480 MB RAM.

Results & Discussions

The VECTIS postprocessor was used to obtain pressure contours, velocity vectors and heat transfer coefficient field, and sample results are shown in Figures 5 – 7. Figure 5 shows the velocity vectors on a plane passing through the bottom deck of the head. Coolant velocity ranging from .5 m/s to 2. m/s through different valve bridges can be seen here. Higher coolant flow is also seen away from the bridge area. Figure 6 shows velocity vectors on a plane passing through the valve bridges as well as the injector sleeve area. The figure shows many recirculation zones around the sleeve area. Both these figures showed insufficient cooling capability of the jacket. Based on these results, a redirecting flow strategy was successfully developed to obtain the desired cooling characteristics of the jacket.

Figure 7 shows the heat transfer coefficient on the bottom deck of the jacket. This type of field on the modified design was mapped to an ABAQUS model and the thermal analysis was subsequently performed.

Conclusion:

CFD analysis on a complete water jacket of a diesel engine has been performed successfully . The use of triangulated surface definition in the mesh generation made it possible to perform analysis on the cooling circuit where no CAD model was available. The availability of an automatic grid generator further accelerated the analysis process. In conclusion, it was possible to perform CFD analysis on a large complex configuration, like the water jacket of a diesel engine, in a timely, as well as cost effective, manner through the use of the general purpose CFD software VECTIS.

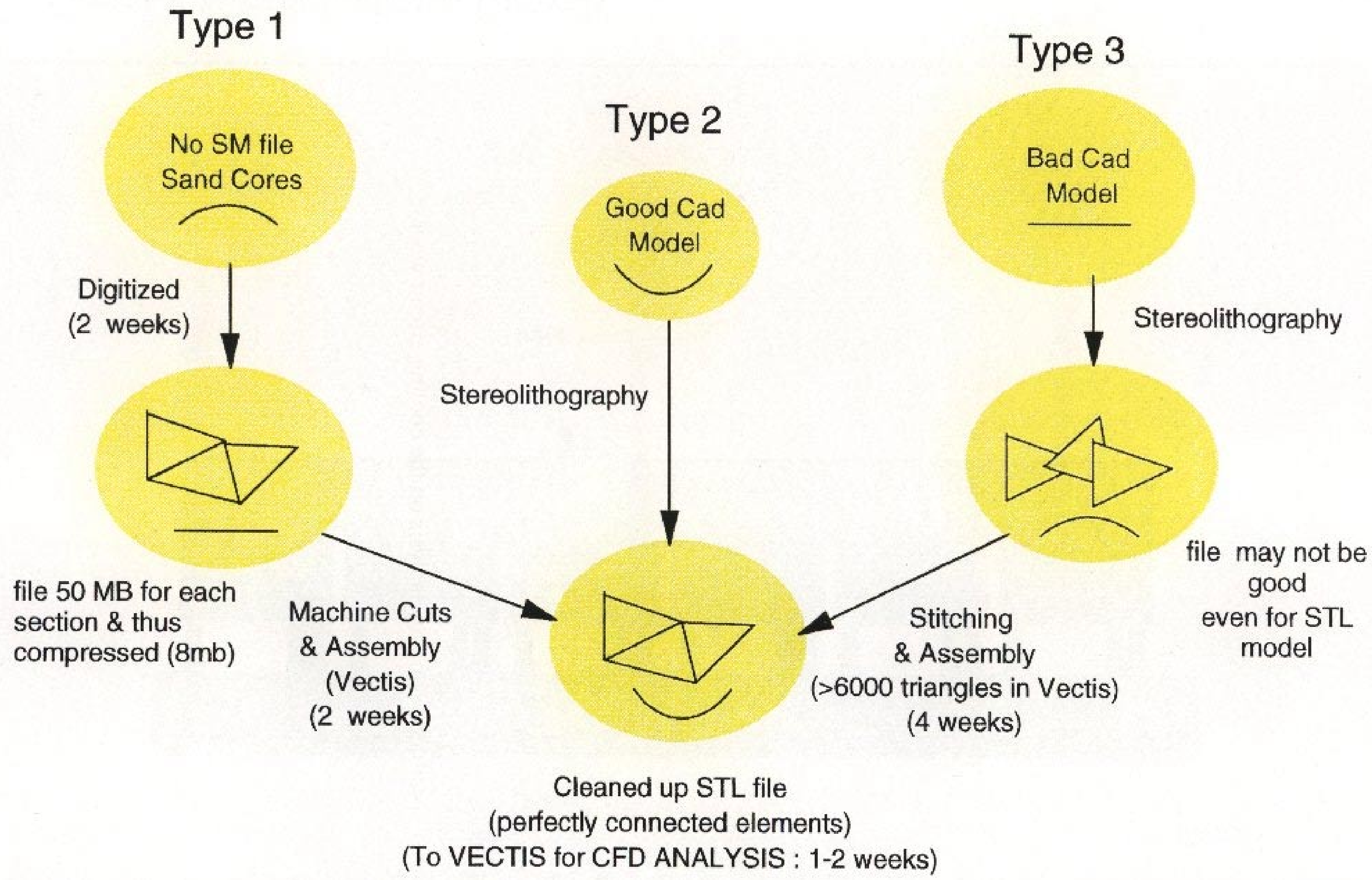

Figure 1: Typical Steps from Available Surface Definition To STL Definition

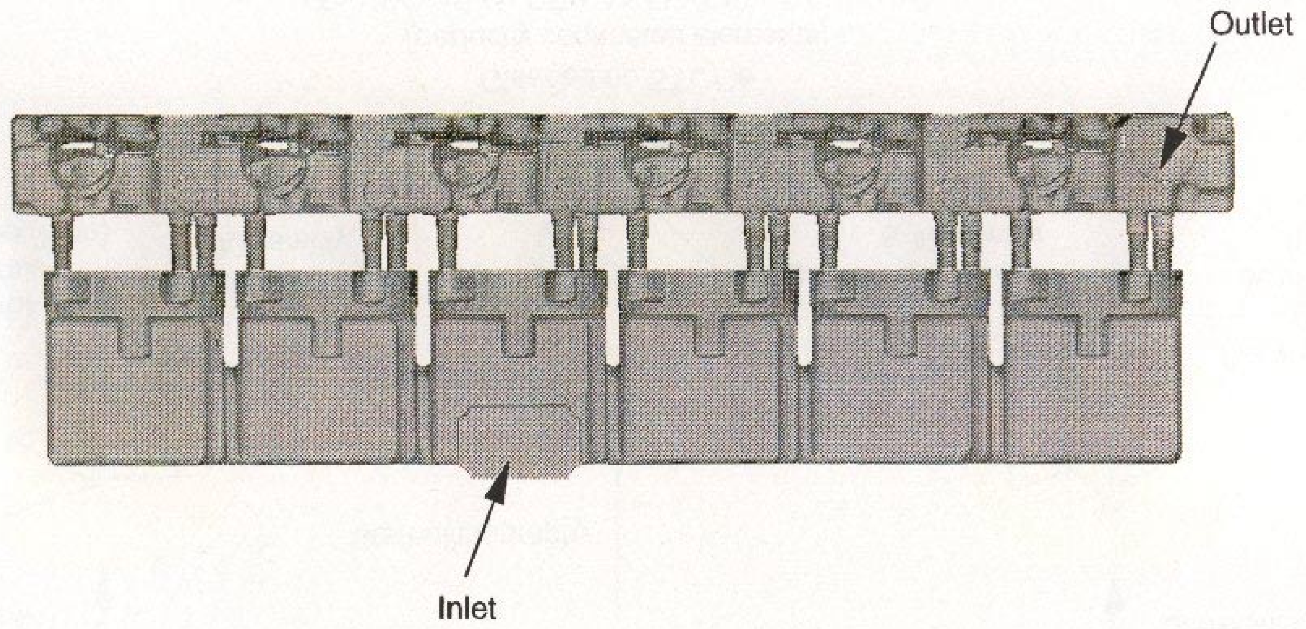

Figure 2 : Computational Domain

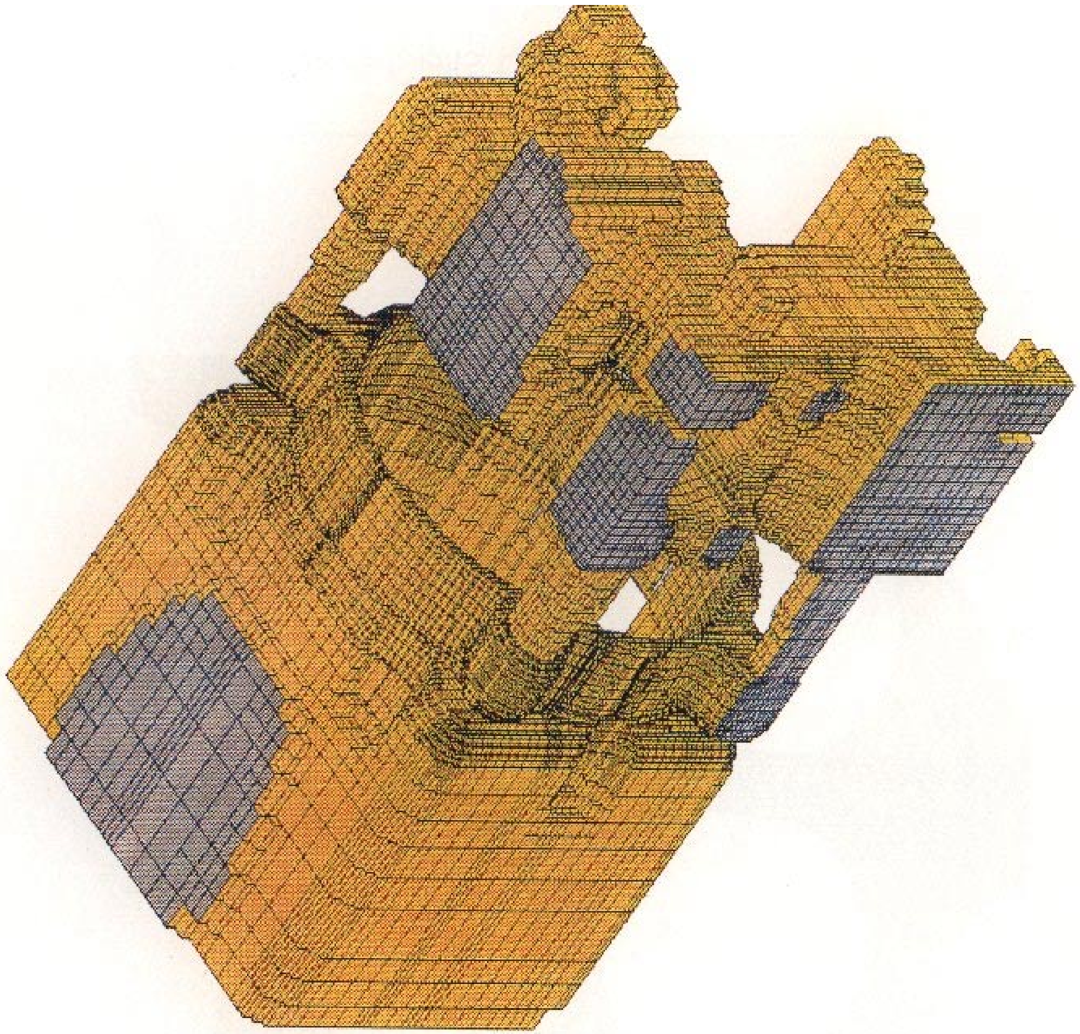

Figure 3 : Untrimmed Surface Cells over a Section of the Domain

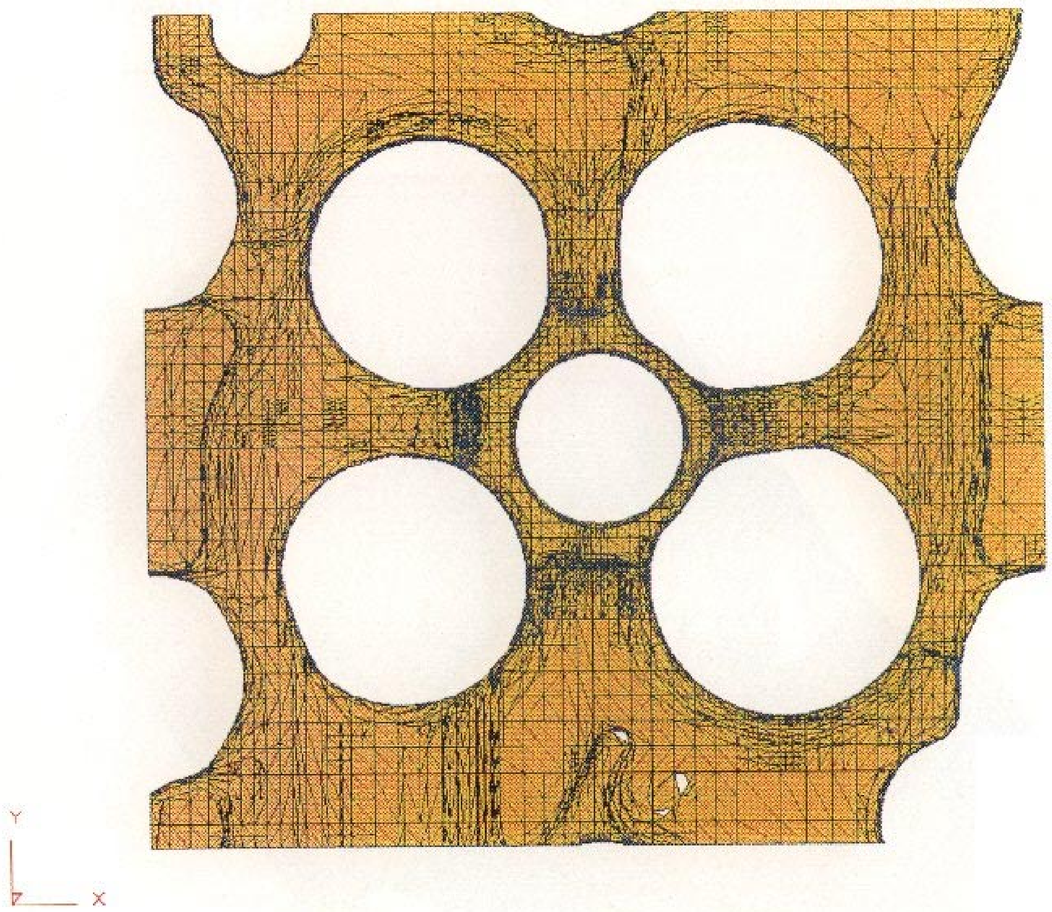

Figure 4 : Trimmed Surface Cells Over a Section of the Domain

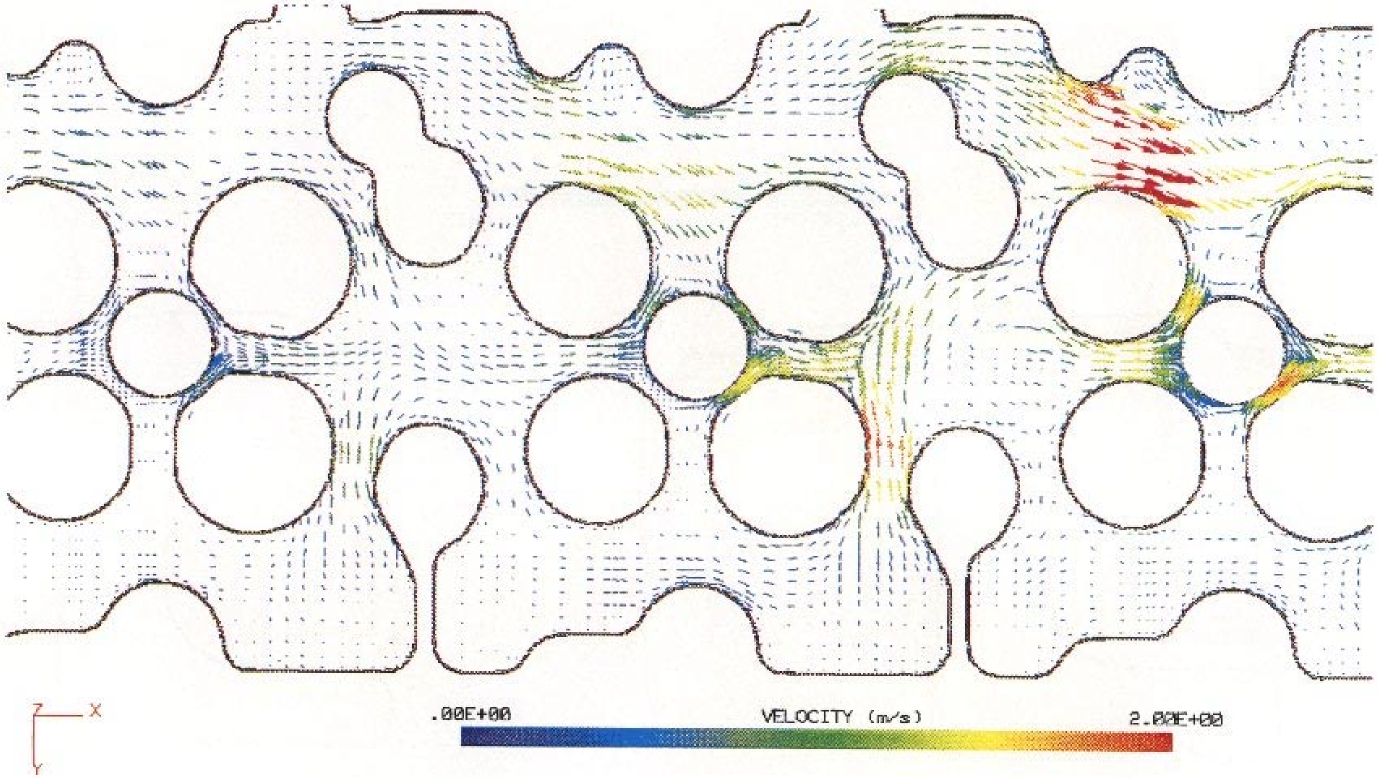

Figure 5 : Velocity Vectors on a Plane Through Bottom Deck

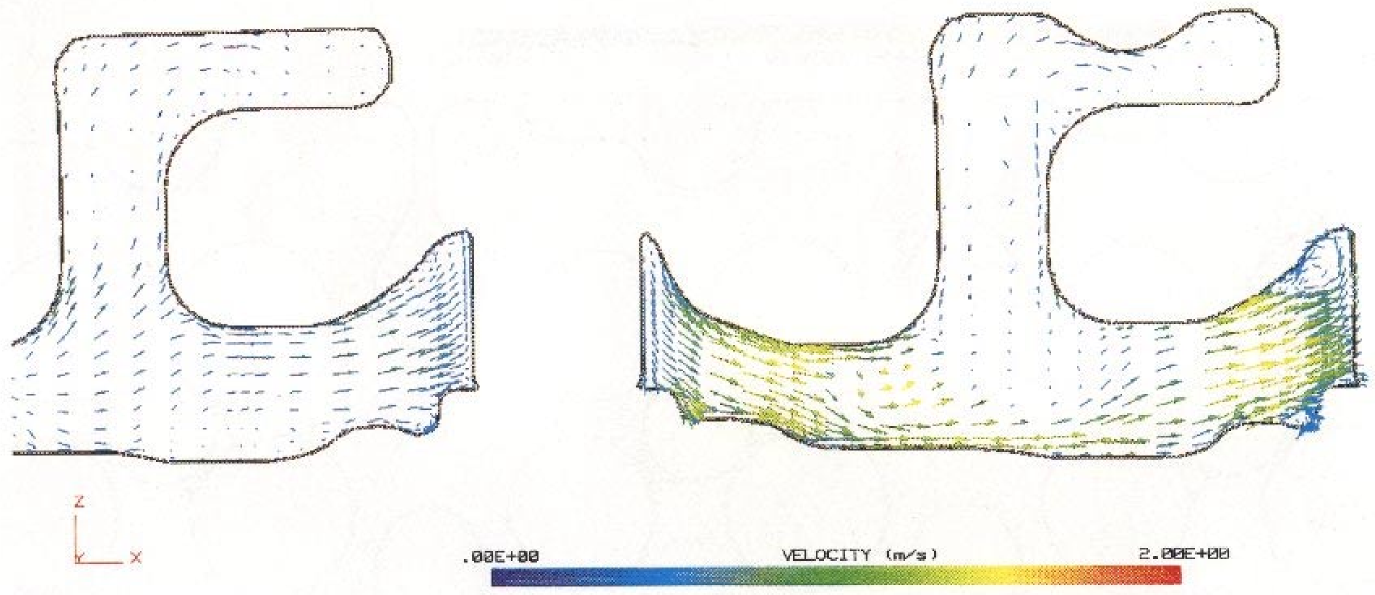

Figure 6 : Velocity Vectors on a Plane Through Valve Bridges

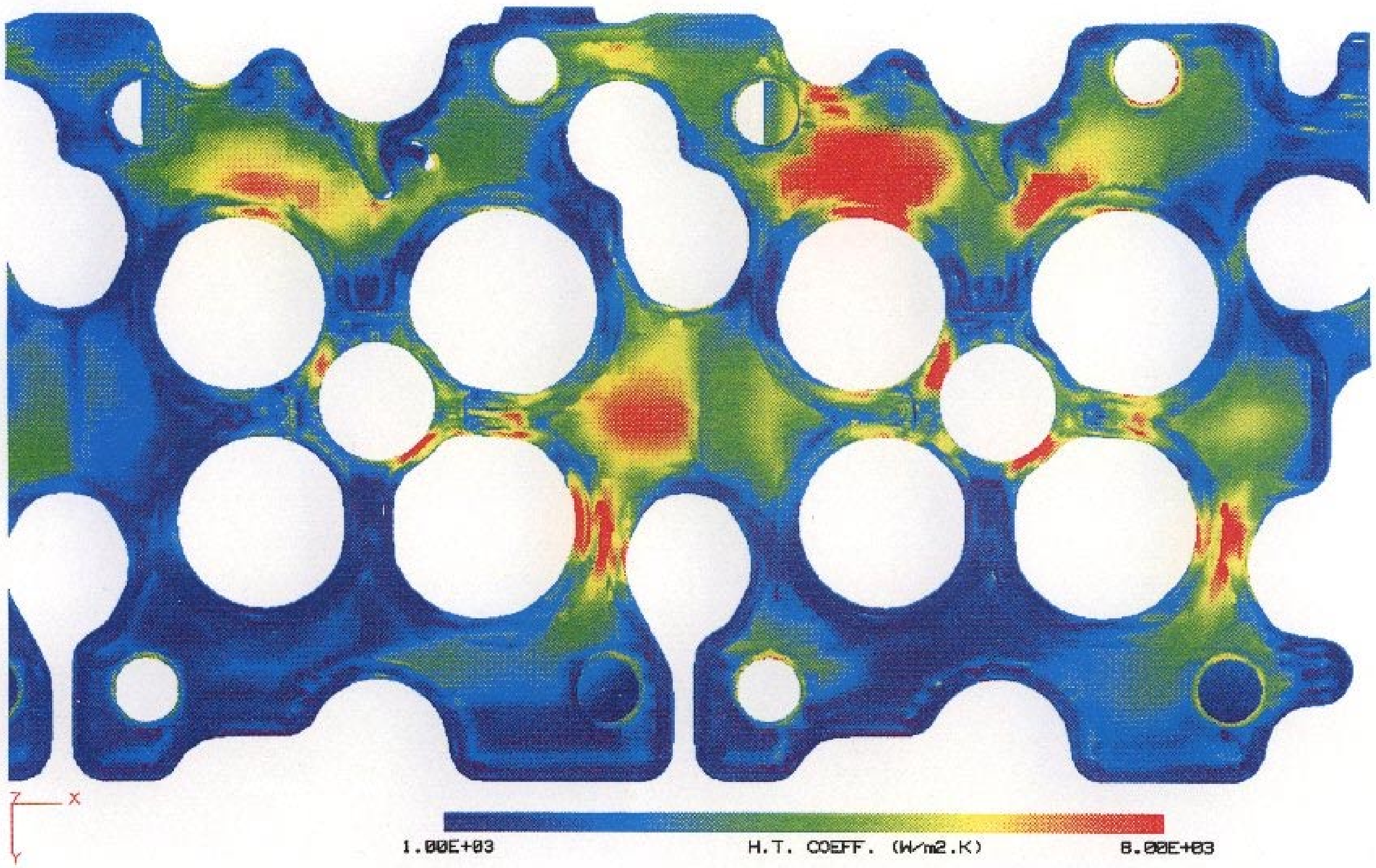

Figure 7 : Convective Heat Transfer Coefficient Field on Surface of Bottom Deck