A NEW APPROACH TO THE MODELLING OF ENGINE COOLING SYSTEMS*

P. MENEGAZZI and J. TRAPY

Institut français du pétrole[1]

UNE NOUVELLE APPROCHE POUR LA MODÉLISATION DES CIRCUITS DE REFROIDISSEMENT DES MOTEURS

L'optimisation des circuits de refroidissement internes est devenue une étape indispensable lors de la conception des moteurs actuels. Bien que la modélisation numérique tridimensionnelle des phénomènes hydrodynamiques se soit révélée un outil bien adapté à ce type d'application, elle implique la nécessité de consacrer un temps très long à la phase préliminaire de génération des maillages. Ceci reste certainement un obstacle majeur à l'intégration systématique de ces méthodes dans les processus de développement des moteurs.

Cet article propose une approche originale du problème, permettant de générer des maillages 3D pour des circuits internes de formes complexes, dans des cas où les méthodes traditionnelles de maillage s'avèrent inapplicables dans des temps raisonnables. Les maillages non structurés de tétraèdres obtenus de cette manière, sont directement utilisables pour des calculs d'hydrodynamique au moyen du code N3S, mettant en œuvre des méthodes numériques par éléments finis. Pour illustrer cette méthode, un exemple complet est présenté, depuis l'étape de génération des maillages jusqu'au calcul hydrodynamique.

A NEW APPROACH TO THE MODELLING OF ENGINE COOLING SYSTEMS

Optimizing the internal cooling circuits has become vital for the design of modern engines. Although three dimensional (3D) Computational Fluid Dynamics (CFD) proves to be a powerful tool, well suited to this kind of application, a very long time is still spent in the mesh generation step. This is certainly a major obstacle to its systematic use in the engine development process.

This paper proposes an original approach enabling the 3D grid generation of complex internal circuits where traditional methods prove to be unusable within reasonable time. The unstructured meshes of tetrahedra obtained in this way are usable for fluid-dynamics calculation with N3S, a finite element code. As an illustration of the method, a complete example is presented from the mesh generation step up to the CFD calculation.

NUEVO ENFOQUE PARA LA MODELIZACIÓN DE LOS CIRCUITOS DE ENFRIAMIENTO DE LOS MOTORES

La optimización de los circuitos de enfriamiento internos ha llegado a ser indispensable al proceder al diseño de los motores actuales. Aun cuando la modelización tridimensional de los fenómenos

---

[1] 1 et 4, avenue de Bois-Préau,
92852 Rueil-Malmaison Cedex - France

* This article was presented at the 3rd IMechE "Computers in Reciprocating Engines and Gas Turbines", London, January 9-10 1996.

físicos mediante ordenados ha manifestado ser una herramienta potente y perfectamente adaptada para este tipo de aplicación, ello presupone, no obstante, la necesidad de dedicar un tiempo sumamente dilatado en la etapa preliminar de generación de las reticulaciones y sigue siendo un obstáculo destacado para la integración sistemática de estos métodos en los procesos de desarrollo de los motores.

Se propone en este artículo un enfoque original del problema, que permite generar reticulaciones 3D para los circuitos internos de formas complejas, en aquellos casos en que los métodos tradicionales de reticulación demuestras ser inaplicables respetando tiempos de duraciones razonables. Las reticulaciones no estructuradas y compuestas por tetraedros obtenidos de este modo se pueden utilizar directamente para los cálculos hidrodinámicos por medio del código N3S, sistema de permite poner en aplicación métodos digitales por elementos finitos. Se presenta un ejemplo completo destinado a ilustrar este método, desde la etapa de generación de las reticulaciones hasta los cálculos de hidrodinámica.

INTRODUCTION

In recent years, most engine developments have been directed towards producing more reliable cars with lower fuel consumption and pollutant emissions. In addition, there is an increasing demand for engines with higher output leading to the use of turbochargers and multivalve technology.

All these trends raise the thermal load of the engine components and particularly of the cylinder head, where the local heat flux is maximum and where hot spots may appear. The efficiency of internal cooling circuits in term of reliability, is closely related to the local characteristics of the flow field: placing the cooling jackets only in the locations where they are needed will eliminate hot spots and allow reductions in coolant flow requirements and water pump horsepower.

Recent works suggest that numerical modelling is now able to provide a reliable and comprehensive description of flows in complex 3D geometries like cooling circuits [1] and [2]. Some of them even show that it has become a valuable tool for their optimization, [3] and [4].

The numerical modelling methodology is usually characterized by four main steps:

- describing the geometry to be modelled,
- meshing the geometry,
- solving the equations on the mesh,
- analyzing the solution (postprocessing).

Generally, the geometry to be modelled is first described using a Computer-Aided Design (CAD) tool. The mesh is then generated by importing the CAD data to an adapted software. In most cases this whole procedure turns out to be efficient. However two main restrictions should be mentioned:

- the CAD description doesn't exist in every case: many industries have a large catalogue of traditional parts created without CAD tools. Moreover, it may be quite interesting to analyze some parts created by others and whose CAD description is generally not available;
- in the case of very complex 3D geometries like the internal cooling circuits, the CAD description can consist of hundreds of CAD surfaces like NURBS, whose handling is very tedious within a meshing software.

In both cases, generating the mesh of the model becomes quite a critical task in terms of time consumption, particularly for structured grids, and can take several months, that is to say up to 80% of the whole processing time [5] and [6]. Of course, this strongly limits the applicability of 3D modelling within an industrial environment.

1 CREATING THE COMPUTATIONAL MESH

In order to reduce the time required for the generation of the computational mesh in the case of complex 3D internal circuits, a novel approach, based on the digitization of the object to be described, is developed at *IFP*. An overview of the general methodology is given in this section.

1.1 Describing the geometry

As mentioned before, the first problem to deal with is getting a description of the object to be meshed. When the object exists, several methods like mechanical touch probes or laser range scanners, enable to acquire the external object shape. However, in the case of internal geometries like cooling circuits, these traditional methods are unusable. Then, we opted for X-Ray Computed Tomography (X-Ray CT) scan digitization, a non-invasive technique for imaging the whole structure of 3D objects.

The X-Ray CT consists of making X-Ray projections from different angular positions around the object (Fig. 1). Each of these projections yields a one dimensional absorption profile. These profiles are then used to reconstruct a slice through the object. The same operation repeated along the axis normal to the cut plane, provides a set of serial slices that can be stacked into a true 3D image of the object.

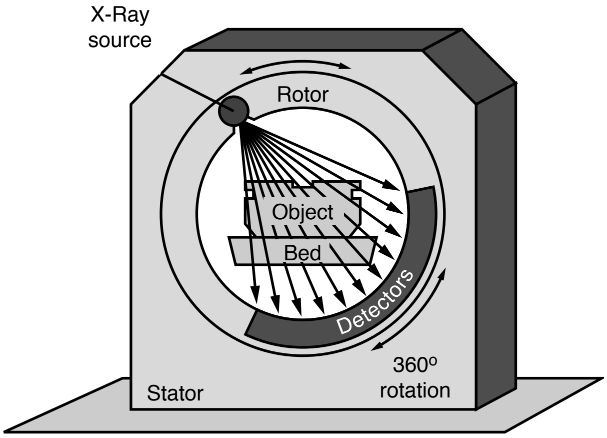

Figure 1

X-ray CT scan device principle.

The main parameters for the scan acquisition are the size of the acquisition field, the size in pixels of the images, the thickness of the slice and the distance between two slices. The values of these parameters depend on the type of scan device, the size of the object to be inspected as well as the precision required for the images. A concrete example of X-Ray CT acquisition is proposed in section 2.

1.2 Reconstructing the geometry

From the set of slices previously generated, the surface of the object is extracted through a 3D reconstructing process. Figure 2 illustrates one of the usable reconstruction methods based on the sequential extraction of the object contours and the approximation

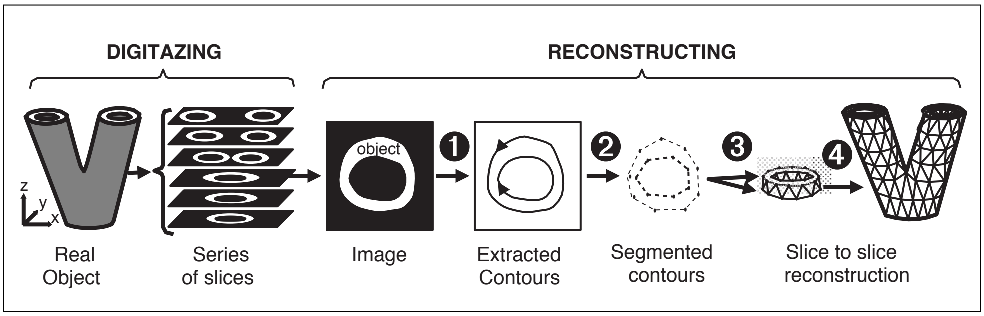

Figure 2

3D reconstruction process.

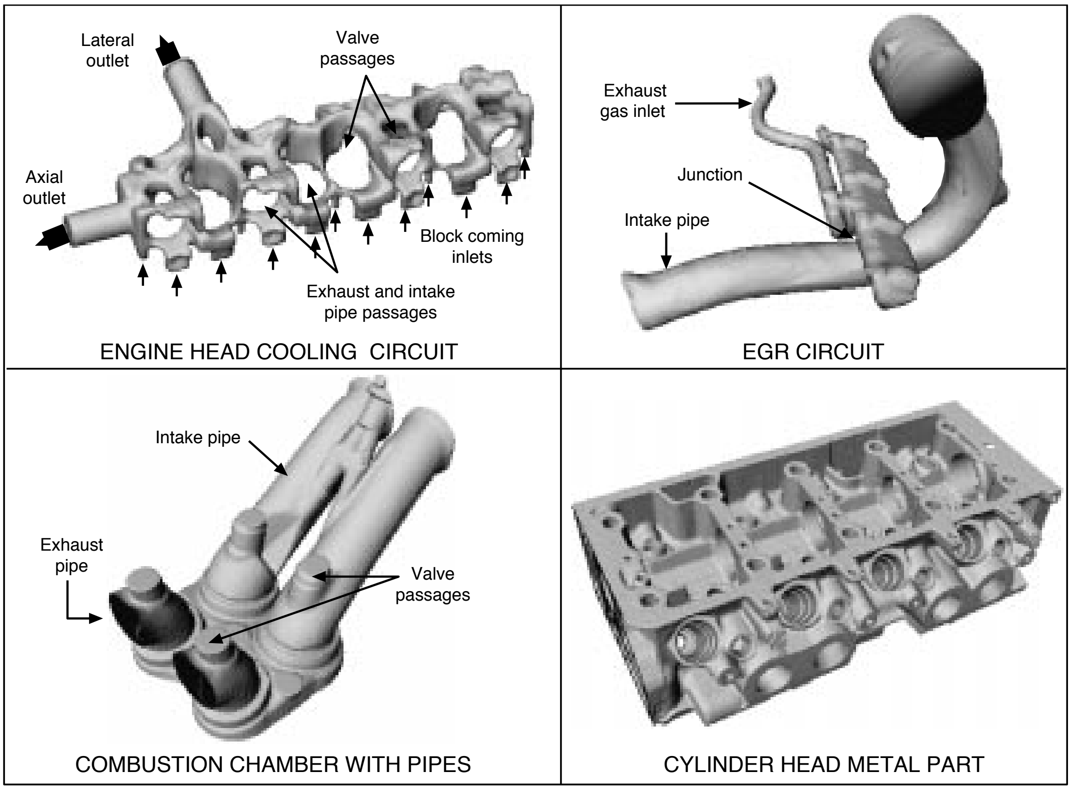

Figure 3

Examples of engine part reconstructions.

to each contours by closed polygons for each image (step 1 and 2). The object external surface is then obtained by connecting the contours with triangles two slices at the same time (step 3 and 4) [7] and [8].

In addition to this type of approach, highly automatic methods, based on a specific 3D process of the slices, are developed at *IFP* to reconstruct a wide range of complex engine parts including internal circuits (cooling, EGR, intake pipes, etc.) as well as solid parts (Fig. 3).

1.3 Calculation meshes

The surface meshes of triangles obtained after the reconstruction step, require particular conditioning before being used in numerical application: the meshes are checked and regularized in size and in quality on the entire surface. The unstructured meshes of tetrahedra are then generated in the internal volume using automatic meshing algorithms [9]. After referencing the surface nodes needed to impose the boundary conditions, the final 3D meshes can be used for CFD calculations.

2 APPLICATION EXAMPLE

The object of this section is not to make a comprehensive analysis of a flow in a given cooling circuit but to provide a practical illustration of the implementation of the methodology described in the previous section, within a computational procedure. A complete study example can be found in reference [10].

The following example is extracted from a general study made at *IFP* on the comparison of several cylinder head cooling circuits of existing V6 engines. In this study, we considered the case of a non-isothermal turbulent incompressible flow coming from the engine block circuit and being distributed to the 22 inlets of the cylinder head circuit. As the thermal problem is assumed not to be coupled to the hydrodynamic one, the temperature is considered as a passive effluent and the physical properties of the working fluid, considered constant in temperature, are those of water at 20°C.

2.1 Mesh generation

Before being scanned, the aluminum alloyed engine head is prepared as follows: the cooling circuit is filled with water to be easily distinguished from the other internal circuits on the images. Moreover, the heavy metalled parts (valves guides, valves, etc.), being too absorbent for the X-Ray beam, are extracted so as not to create disturbances on the slice images. Figure 4 shows a short sequence of slice images as well as the detailed image obtained from the engine head. The acquisition field is 315 mm and the reconstruction matrix (that is to say the slice image) is 512 x 512 pixels which results in a pixel size of 0.615 mm. About 400 slices are necessary to describe the whole object. The duration of the acquisition is about 5 hours with the scan device available at IFP.

Using the procedure described in the preceding section, the coolant circuit is reconstructed. In Figure 5 an exhaust side view of the reconstructed circuit is presented as well as a view from below showing the above mentioned 22 inlets coming from the cylinder block and being calibrated by the resizable head gasket holes.

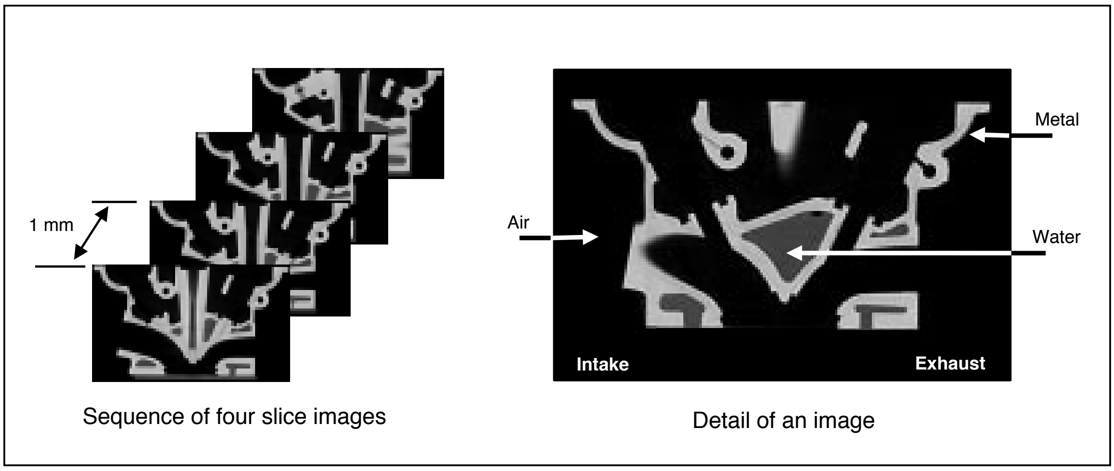

Figure 4

X-ray CT scan sections of the engine head

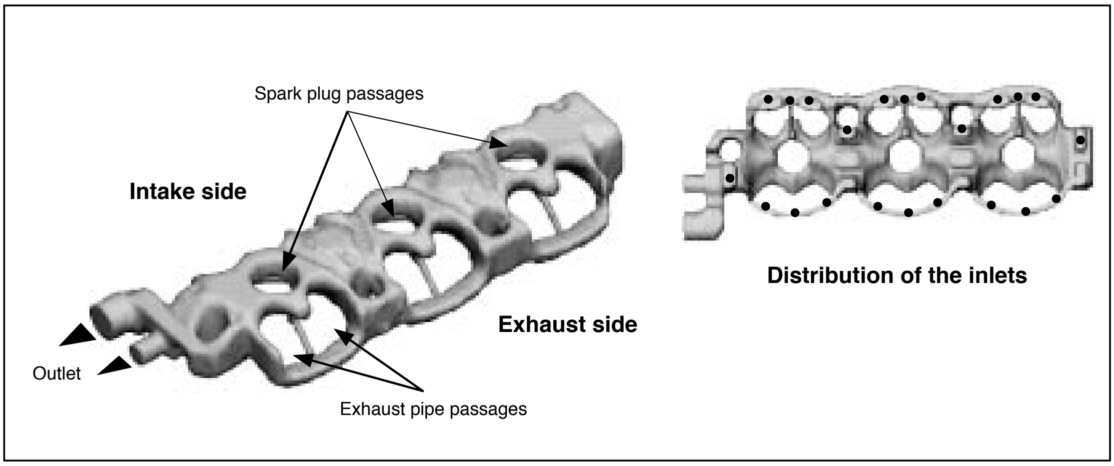

Figure 5

Three dimensional images of the circuit.

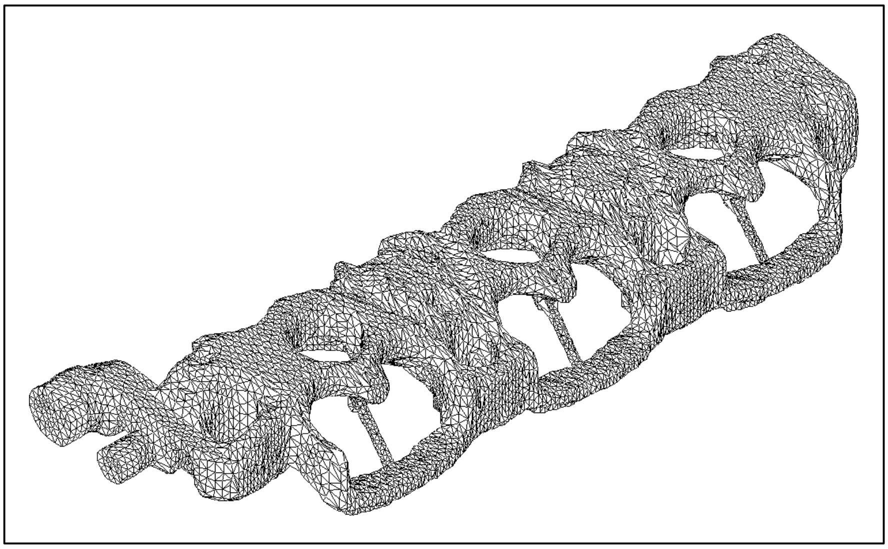

Figure 6

Computational mesh of tetrahedra.

After the conditioning step including the surface mesh regularizing and the surface nodes referencing, the final computational mesh of tetrahedra, shown in Figure 6, is generated. For that kind of circuit, the whole procedure can typically take less than two weeks.

The main features of the 3D tetrahedral mesh are given in Table 1 below.

TABLE 1

Features of the computational mesh

Number of tetrahedra	91 979
Number of vertices	22 677
Number of velocity nodes	149 951
Volume (l)	1.116
External surface (m^2)	0.206

2.2 Numerical method

For the CFD calculation, a commercial computer code, N3S-EF, developed by *Électricité de France (EDF)* is used. N3S is a 3D finite element (FE) code, for simulating incompressible turbulent flows in arbitrary complex geometries. It solves Reynolds-averaged Navier-Stokes equations coupled with the classical $k - \varepsilon$ model (in which k denotes the turbulent kinetic energy and ε the turbulent dissipation rate) and a thermal equation.

Time discretization is made by splitting the operators at first or second order in time. The advection step is solved using a characteristic method. The space discretization is based on a standard Galerkin finite element method leading to a generalized Stokes problem solved by a Chorin method.

2.3 Boundary conditions

The previous equations are generally completed by boundary conditions that depend on the type of boundary to be considered: for the inlet of the fluid domain, forced constrained conditions (Dirichlet) are used on all the variables, for the outlet, vanishing normal stress for the velocity and vanishing flux condition (homogeneous Neumann) for scalar quantities are used, for walls, models based on the generalization of the boundary layer on a flat plate is used. In this example, the scavenging of hot coolant by cold flow is modelled using two kinds of boundary conditions:

- Hydrodynamic conditions: a velocity corresponding to a flow rate of 100 l/min for a 4500 rpm engine speed, is imposed at the outlet, with standard Dirichlet conditions on k and ε at the inlet. The inlet pressure is imposed.
- Thermal conditions: Dirichlet conditions for the temperature are used,
  . inlet coolant temperature imposed at 90°C,
  . wall temperature imposed at 110°C everywhere.

Moreover, at the initial time, the fluid is supposed to be at 100°C.

As mentioned above, however unrealistic, these thermal conditions are only designed to provide results for comparing several circuits under similar conditions.

2.4 Computational details

The calculation are performed on the Fujitsu VPP500, a distributed-memory vector multi-processor computer. The VPP500 available at *IFP* consists of four processors having 512 Mo of local memory and a peak performance of 1.6 GFlops each. For the hydrodynamic calculation, the duration of the time step is about 65 s and 20 hours are necessary to reach the steady state. In comparison, the duration of the time step for the thermal problem (7s), is rather small but, due to the dominating diffusive effects in the stagnant areas of the circuit, several CPU hours are yet required to reach the thermal steady state.

2.5 Some results

3D simulations usually provide a great amount of data always difficult to be exploited. Fortunately, the use of adapted softwares on 3D graphic stations greatly eases the visualization of these data and the understanding of the physical phenomena they are supposed to describe. Even though it remains hard to reproduce them outside of the interactive software context, we have gathered in this section a few characteristic visual results as well as quantitative ones useful for the flow analysis.

2.5.1 Visual results

Figure 7 shows particle paths going from the inlet up to the outlet of the fluid domain. This kind of visualization is very convenient to highlight the preferential paths of the flow and particularly the influence of the inlet distributions on the structure of the flow.

Figure 8 presents a top view of the velocity and the temperature fields on the external surface of the circuit. We can note a funnel effect characterized by a longitudinal velocity gradient which yields an heterogeneous flow around the spark plug passages along the circuit. The velocity field also highlights the stagnant areas of the circuit mainly located on the intake side.

The temperature field is particularly useful to locate the potential hot spots related to a small local velocity as well as those related to a local recirculating flow which confines a small quantity of fluid near the wall and that the velocity field is not able to show.

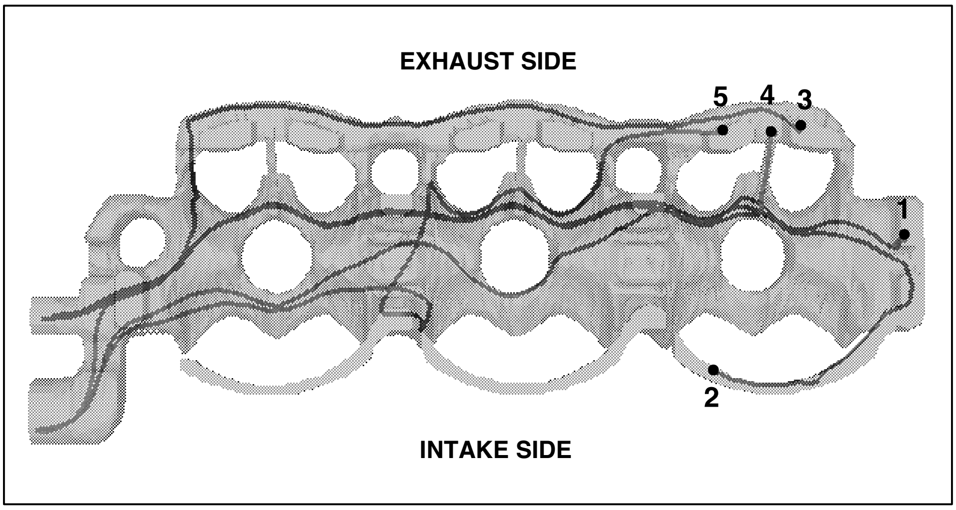

Figure 7

Particle paths from five inlets of the circuit.

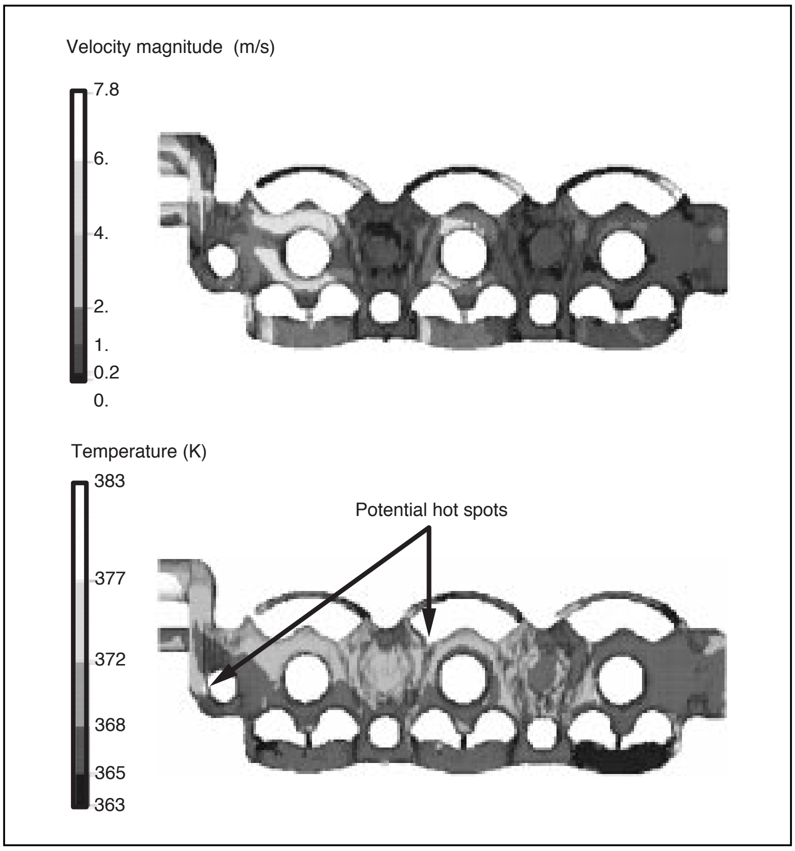

Figure 8

Top view of the velocity andtemperature fields on the external surface of the fluid domain.

2.5.2 Global results

Table 2 presents a few global features of the circuit obtained numerically. The results concerning the power exchange give a good idea of the circuit efficiency in terms of cooling ability while the time to reach the steady state stands for the ability of the circuit to limit the duration of the engine warm-up.

TABLE 2

Global numerical features of the circuit

ΔT (outlet - inlet)	4.5
Total extracted power (kW)	31
Extracted power per surface unit (kW/m^2)	151
Bulk temperature (K)	366.3
Mean flow velocity (m/s)	0.43
Time to reach the steady state (s)	1.8

As mentioned above, all these results are mostly useful when comparing several circuits to one another. Future modelling work will concentrate on extending the thermal analysis to the whole cylinder head including the solid part.

CONCLUSION

In this paper, we have described an original procedure aiming to significantly reduce the mesh generation task in the case of complex 3D geometries when classical methods prove to be unefficient in reasonable time. The method, based on the X-Ray CT digitization of the object to be modelled, proves to be particularly efficient in applications like the characterization of internal engine circuits. This ability is clearly illustrated through a typical example in which the coolant flow in a cylinder head is investigated using CFD tool. Although promising results have been already obtained, there are a lot of developments in extending the current approach. First of all, the duration of the meshing procedure can be greatly reduced particularly by a higher automation of the mesh conditioning task and by taking into account the numerical constraints on the meshes in the reconstruction procedure. The problem of integrating the CAD tool in the process is also under consideration so as to extend the applicability of such an approach in industrial design.

REFERENCES

1 Arcoumanis C., J.M. Nouri and J.H. Whitelaw (1991), Coolant flowin the cylinder head/block of the Ford 2.5L DI Diesel engine. *SAE paper* 910300.

2 Sandford M.H. and I. Postlethwaite (1993), Engine coolant flow simulation - A correlation study. *SAE paper* 930068.

3 Jae-In Lee and Nam-Hyo Cho (1995), Numerical analysis of gasoline engine coolant flow. *SAE paper* 950274.

4 Colleoc A.(1996), Engine coolant flow optimization: numerical simulation. *Fisita Conference*, Prague.

5 Boretti A.A, M.G. Lisbona, P. Milazzo and P. Nebuloni (1992), The use of computational fluid dynamic based tools in engine design. *2nd International Conference on Fluid Mechanics, Combustion Emissions and Reliability in Reciprocating Engines,* pp. 69-80, Capri, Italy.

6 O'Connor L. (1992), Giving a boost to engine design, *Mechanical Engineering*, pp. 44-50.

7 Boissonnat J.D. (1988), Shape reconstruction from planar cross-sections. *Computer Vision, Graphics and Image Processing,* **44**, pp. 1-29.

8 Geiger B. (1993), Construction et utilisation des modèles d'organes en vue de l'assistance au diagnostic et aux interventions chirurgicales. *Thesis* of the École des Mines, Paris.

9 Francez L. and P.L. George (1991), Description of a 3D mesh generator. *European Conference on New Advances in Computational Structural Mechanics,* Giens (France).

10 Porot P., P. Menegazzi and N.S. Ap, Understanding and improving evaporative engine cooling at high load, high speed by engine test and 3D calculation. *VTMS Conference*, Indianapolis, *SAE* 971792.

11 Chabard J.P., B. Metivet, G. Pot and B. Thomas (1992), An efficient finite element method for the computation of 3D turbulent incompressible flows. *Finite Element in Fluids,* Hemisphere Publishing Corporation, **8**.

Final manuscript received in August 1997