The application of computational fluid dynamics within the automotive industry

Benjamin, S.F. and Hickman, C.

Published version deposited in CURVE January 2014

Original citation & hyperlink:
Benjamin, S.F. and Hickman, C. (1989). The application of computational fluid dynamics within the automotive industry. In *Proceedings of IMechE Auto Tech 89, volume C399/18: Computational Simulation and Information Management*. IMechE.
http://books.google.co.uk/books/about/Autotech_89_Seminar_Papers.html?id=z7fGtgAACAAJ&redir_esc=y
http://www.imeche.org/

The application of computational fluid dynamics within the automotive industry

by Mr S F Benjamin and C Hickman

SYNOPSIS The application of computational fluid dynamic (CFD) techniques for the solution of engineering problems is rapidly gaining acceptance within the automotive industry.

This paper explores the possibilities of CFD and provides an overview of the current status and future trends within this area. In particular examples are shown of the application of CFD within MIRA.

1 INTRODUCTION

Computational fluid dynamics (CFD) provides a means of predicting the fluid flow field via solution of the transport equations for mass, momentum and energy. Hence CFD offers the design engineer a powerful analysis tool to be used at various stages of the design and development process. In this paper it is intended to examine the role and application of CFD within the automotive industry and in particular demonstrate its implementation within MIRA. It is perhaps useful to first discuss where and how CFD can feature within the design/development process.

2 THE ROLE OF CFD IN THE DESIGN/DEVELOPMENT PROCESS

2.1 Front-end analysis

Ideally CFD should feature as an integral part of the first stage of the design process. It is at this stage where numerous designs can be simulated prior to expensive prototype manufacture and the most promising selected. Many variants can thus be simulated both in terms of geometrical configuration and operating conditions. This offers the prospect of providing better designs, a reduced test programme and subsequent savings in time and cost.

2.2 Complement to test programme

Rig testing of prototypes will always remain an essential part of the development programme. At this stage CFD can complement the testing by:

(a) providing an interpretation of performance data in terms of the fundamental flow processes.

(b) supplementing flow measurements which may only be available at a few locations by providing predictions throughout the whole flow domain; this also offers the prospect of suggesting optimal siting for measurements in critical regions.

In many complex geometries it can be extremely difficult to measure the flow field due to access difficulties or a hostile flow environment; in-cylinder flows are a case in point. In this situation CFD may be the only practical way to gain information on the flow and thus a deeper understanding of the physical processes involved.

2.3 Trouble-shooting

Problems which arise after manufacture can be diagnosed using CFD to identify causes and offer potential solutions. Problems may arise, for example, where the operational range of the equipment has been extended beyond its origin specification. Parametric studies using CFD could then be used to examine operational limits and design modifications.

Hence CFD can be used throughout the design/development programme from front-end analysis through to prototype refinement.

3 AUTOMOTIVE APPLICATIONS

Specific applications of CFD within the automotive industry can broadly be classified into three main areas; vehicle related, engine related, and components.

(i) Vehicle related areas include

External flows – aerodynamics, dirt deposition

Underbonnet flows - Cooling systems
In-vehicle flows - Passenger comfort

(ii) Engine related areas include

Cylinder head cooling
Manifold and port flows
In-cylinder flows and combustion

(iii) Components

Heating and ventilation units
Radiators
Bearings (lubrication)
Fans, pumps, turbochargers etc

Examples of CFD applications in some of these areas can be found in reference (1).

4 CFD CONSIDERATIONS FOR THE DESIGN ENGINEER

For sucessful application of the CFD technique it is important to understand its current limitations and the resource requirement. Ideally CFD would not only accurately simulate fluid phenomena but also be capable of practical implementation. The latter requires consideration of available software, hardware and the necessary human resources needed for successful application. To discuss these issues it is perhaps useful to first consider the nature of CFD.

4.1 CFD methodology

The flow equations comprise mathematical descriptions of the transport equations of mass, momentum and energy in a continuum. For all but the simplest flows these equations must be solved numerically.

The numerical procedure normally takes the form of solving the equations at a finite number of points within the flow domain. This is accomplished by subdividing the domain into a mesh of elements or cells each containing an unknown value of the field variables (pressure, velocity etc). The flow equations are discretised providing a set of algebraic equations which describe the relationships between variables in neighbouring cells. These equations are then solved numerically in an iterative fashion.

For most applications the flow is turbulent and therefore varies in space and time from relatively long to extremely small scales. To resolve down to the finest turbulent scales would require fine meshing and a large computational effort. Whilst such techniques are being used they are at present impractical for engineering application. Rather the codes model these small scale turbulent effects and solve for the average or mean values of the flow variables.

Turbulent flow modelling presents the major challenge to providing accurate descriptions of the physical processes. Many codes offer a range of turbulent models and a knowledge of what works best for a particular type of flow is a pre-requisite for appreciation of the limitations of the chosen system is desirable.

4.2 Software considerations

New codes are appearing at an unprecedented rate and faced with choosing a CFD package several key points need to be addressed.

(i) Problem type

Certain CFD codes may be termed 'general purpose' in that they were developed to apply to as wide a range of applications as possible. If the user's intended application is multifarious then these codes should be considered.

Other codes have been developed for particular problems. Examples of this are the in-cylinder codes eg KIVA (2) and aerodynamic codes VSAERO (3). These codes, by their nature, will contain specific attributes (eg fuel spray mixing in KIVA) which may not feature in the general purpose software. Also because these codes are specific they will be expected to perform more efficiently than the general purpose software for their particular applications.

For the general purpose codes the user must first establish that the code attributes include all the physics pertaining to the problem. For example, transient solvers would be needed for vehicle warm up simulations.

(ii) Geometric flexibility

This relates to the ease with which complex geometries can be simulated. The cell topology comprising the mesh largely dictates this.

Many codes now feature cells which can be non-orthogonal and therefore capable of significant distortion to fit complex boundaries. These are often referred to as body fitted co-ordinate codes. Other codes require orthogonal brick type elements and therefore cannot readily match complex boundaries. However where boundary influenced flows are not of interest these codes may be adequate.

(iii) Mesh generation and post-processing facilities

In practice, by far the most time consuming activity is generating the geometry and meshing the flow domain. Many codes feature their own mesh generators. However a trend is developing whereby 'links' are provided to CAD/CAE software many of which feature advanced geometric and mesh generating modules. This can be an attractive route especially when a user's data base may already reside on a CAD/CAE system which links into the fluid flow software. Similar considerations apply to the post processing software.

(iv) User interface

Ease of use is an attractive feature of any turbulent modelling, for example, is a developing science and users may wish to incorporate new models which, from their experience, are known to more accurately represent the flow of interest. Hence a degree of code 'openness' may be desirable so that such changes can be implemented. This may compromise 'ease of use' but may be unavoidable.

(v) Code robustness and numerical efficiency

This is difficult to judge except by experience. The flow equations are non linear and simulations may fail to converge due to the particular geometry (mesh) and type of flow. In practice the user is obliged to monitor progress of the solution and 'fine tune' solver parameters until convergence is obtained. This may necessitate modifying the geometry in certain problem areas. Experience with the code is perhaps the only way to judge this and indeed other aspects of its implementation.

(vi) Accuracy

Clearly, evidence of code performance against experimental data is desirable. Many code vendors will provide such examples as may exist. Where the user's problem is sufficiently removed from standard test cases it may be necessary to test the codes for the case in point. This is a non-trivial exercise but where a long-term commitment to a special application is required it may be unavoidable.

4.3 Hardware considerations

The CFD simulation requires essentially four distinct stages namely;

(i) Generating the geometry
(ii) Creating the mesh
(iii) Performing the simulation
(iv) Post-processing the results

Generating the geometry and meshes in a CAD/CAE environment can be a most attractive route provided links to the fluid flow software exist. A good colour graphics terminal linked to a mainframe or as part of a workstation is essential.

CFD simulations fall within the category of large 'number crunching' activities with computing times dependent on the number of variables to be solved, the number of cells, the number of time steps (for transient problems) and the tolerances set for convergence.

Large 3D simulations (upwards of say 100 000 cells) will typically require 2-6 hours on the largest mainframes (1). Where access to such computing power is difficult or costly then consideration should be given to first performing the coarse grid trial runs and initial convergence tests on a local machine; the complete simulation using a fine mesh can then be performed remotely on the large mainframe.

appropriate overnight and weekend scheduling.

Finally, for large simulations, considerations should be made for in-core memory requirement. Finite element type CFD codes can, for example, be quite demanding in this respect. Clearly, memory requirements should be established early on as this will impact both on software and hardware decisions.

4.4 Other considerations

Finally mention should be made of other factors which normally feature in the decision making process.

Commercial considerations are always important. Efficient user support is desirable especially where fast response is needed in a commercial environment. Considerations of code enhancement and development plans are important especially in such a rapidly developing field as CFD.

Having discussed the potential for CFD and considerations for its implementation the remainder of this paper will describe MIRA's experience which it is trusted will be beneficial to other users in the automotive industry.

5 APPLICATION OF CFD AT MIRA

MIRA has long recognised the potential for CFD applied within the automotive industry.

Applications software was perceived as falling into two distinct categories.

(a) Vehicle aerodynamics
(b) Other applications

5.1 Vehicle aerodynamics

Whilst general purpose CFD codes offer the prospect of predicting aerodynamic forces, at this stage of their development it is clear that much computing effort would be needed to simulate the detailed features which are known to influence the drag on modern vehicles. For example, a simulation which will resolve down to drip rails and mud flaps would require an extremely fine mesh and awesome computing effort.

MIRA have therefore adopted two distinct approaches.

The first is an empirical method for the prediction of drag (4). The vehicle is subdivided into a number of geometric features eg edge radii dimensions, windscreen rake angles, mud flap areas etc. Extensive wind tunnel testing has been conducted and correlations with these individual geometric features established. Once details are known for a particular vehicle a programme called DRAG integrates these individual contributions to provide a prediction of the total drag force. This has worked successfully and drag predictions usually within ±5% can be

In parallel with this activity a CFD programme called AIRFLOW has been developed which is based on the panel technique. The purpose of this programme was to provide a technique capable of predicting the pressure distribution over the 'attached flow' part of the body. The technique is useful for example for siting of vents, and for front end design. Similar programmes have been developed elsewhere (3).

AIRFLOW assumes that outside the thin boundary layer on the vehicle the flow can be considered inviscid. Because of this the flow field can be completely determined by solving for the velocity potential only on the body surface. This is the virtue of the panel technique which is essentially two-dimensional and therefore inherently more economical than the fully viscous CFD codes.

To solve for the velocity potential, the body surface is first subdivided into an array of triangular panels and the velocity potential is numerically calculated within each panel (see fig (1)). Hence velocities and pressures are derived. Separation lines can be prescribed based either from experience, wind tunnel testing or from examination of the pressure distribution. Once identified an initial wake shape is assigned. A final wake shape is computed through an iterative procedure which relaxes the wake shape to a stream surface.

AIRFLOW has been linked into MIRA's CAE environment where meshing and post-processing is carried out.

5.2 Other applications

It was clear that to cover the range of applications envisaged in the automotive industry a general purpose CFD code was needed. A software survey identified the STAR-CD code as one which met MIRA's long term requirements.

STAR-CD is based on the finite volume method. It presently uses a structured grid whose cells are (non-orthogonal) hexahedral elements, ie topologically the mesh equates to a rectangular block.

The procedure adopted for performing simulations is described below and the application to three distinct cases aims to demonstrate the capabilities of the system.

Geometric definition and mesh generation

STAR-CD provides links to MIRA's CAE system (I-DEAS) and this was an important consideration both in terms of MIRA's client data base and the sophisticated mesh generating facilities that were available.

Fig (2) illustrates the procedure. The I-DEAS software encompasses several modules for the design and analysis of structures. These include Solid Modelling and Engineering Analysis, the latter incorporating a mesh generator, and geometry definition task.

The first stage of the analysis is the definition of the geometry of the flow volume to be simulated. Once defined a mesh is constructed within this volume.

The process normally proceeds along three possible routes:

(i) Geometric data is passed between the clients CAD system to MIRA using the IGES file format. This is an industry standard format and contains geometric data (points, curves, surfaces etc). These files are passed to MIRA's host CAD system and are then read by the I-DEAS software.

Within I-DEAS the mesh generator suite of the Engineering Analysis modules is used to generate the mesh which can be read by STAR-CD.

(ii) The geometry of the component can be modelled using the Solid Modeller module of I-DEAS. Once defined, mesh generation and the link to STAR-CD proceed as above.

(iii) Simpler geometries (normally 2D) can be generated in the Geometry Definition task of the Engineering Analysis modules with subsequent meshing as above.

The meshing procedure is best illustrated by consideration of fig (3) which shows the mesh generation on a curved surface.

Fig (3a) shows a curved surface which is composed of four curved edges. In figure (3b) a mesh area is defined by the four edges and the surface. The specification has also been assigned to edge 1 (3 cells) and edge 2 (4 cells). Fig (3c) shows the 'mapped' mesh generated with elements lying on the original curved surface.

The same idea applies to generating solid elements where six surfaces and six mesh areas (forming a mesh volume) are defined and the mesh specification is set along three edges. The cell density can also be biased towards the ends or the centres of the edges thus providing considerable freedom in mesh composition. This is useful where higher cell densities near flow boundaries are desired.

Meshes can be generated in complex volumes by subdividing the domain into several mesh volumes, meshing these separately and then joining them up.

Flow down an inlet manifold

This case demonstrates the methodology for investigating the flow down part of an inlet manifold.

The manifold was generated by solid modelling in I-DEAS. Sixteen manifold sections were created along a given centreline as shown in fig (4a). These sections were then 'skinned' such that a solid (surfaced) model was generated fig (4b). Once defined the model was read into the mesh generator of I-DEAS.

Eight sided mesh volumes were generated along the pipe length and these were automatically meshed once edge specifications had been assigned (fig (4c)). The mesh was read by STAR-CD and a simulation performed once initial and boundary conditions had been specified (fig (4d)).

Flow over a windscreen wiper

This simulation was performed to demonstrate how CFD could be used to investigate wiper lift at high speed.

Fig (5a) shows a two dimensional idealised representation of a windscreen wiper. (The simulation assumes that the wiper is normal to the incident flow field). The 2D geometry was generated in the Geometry Definition task of the Engineering-Analysis module.

Fig (5b) shows a portion of the mesh which was subsequently generated. Parts of the mesh have been 'blocked off' to represent the ground, the wiping element and its harness. These blocked-off or 'dead ' cells are defined by STAR-CD as impermeable to the flow.

It is noteworthy that the non-orthogonality of the cells permits such geometries to be simulated quite accurately.

Fig (5c & 5d) show the velocity and pressure distribution predicted over the wiper. The velocity field is complex with a large recirculating eddy formed downwind of the wiper element. The pressure distribution indicates that positive lift is generated on the blade.

This type of analysis permits the exploration of, for example, alternative harness shapes for generating negative lift at high speed.

Flow in a heating and ventilation unit

This simulation was performed to investigate the flow and heat performance of a car heating and ventilation unit.

Figure (6a) shows a solid model of the unit created in I-DEAS. The main casing was generated by extruding a 2D profile to half model depth. Plane cuts using boolean operations were performed to shape the casing. The various outlets were similarily generated and joined to the main casing using boolean operations. In such a way complex models can be generated.

This model was however simplified to permit a two-dimensional flow simulation.

Prior to performing the simulation various internal features need to be described. These are shown in fig (6b).

The radiator was modelled as two permeable baffles with a resistance a function of the airflow as derived from wind tunnel tests on the core matrix.

The temperature door directs varying amounts of air flow through the radiator depending on its position. The mode door directs flow by varying amounts to the floor or the screen; again depending on its position. In fig (6b) it is positioned to direct flow towards the floor.

Various other baffles are shown to direct air around the unit.

Figure (6c) shows the two dimensional mesh generated for this geometry. Note that the area above the mode door has been 'blocked-off' with dead cells. The baffles are modelled by assigning zero porosities to cell faces.

It is worth noting that for the mesh structure required by STAR-CD (and other codes) high cell deformation is unavoidable in complex geometries especially where boundaries (or baffles) meet at acute angles. This may cause convergence problems.

Also where high mesh densities may be required to resolve small scale features they are 'propagated' throughout the flow domain due to the structured nature of the mesh. This causes inefficiency.

These type of problems are being resolved by introducing 'indirect addressing' techniques in such codes.

Fig (6d) shows a three dimensional mesh which was generated by sequentially extruding a two dimensional mesh. In this mesh the central register of the heater unit has also been generated.

Figs (6e-g) show velocity predictions for 2D and 3D simulations for various geometric configurations. These types of simulations enable the design engineer to explore many variants for optimising the flow and heat performance of these units. In particular interest focusses on providing minimum flow loss for a given heat output and in obtaining a linear temperature profile against door position.

6 CONCLUSIONS

(i) The field of CFD is a rapidly developing one and there exists great potential for simulating a wide range of automotive related flows.

(ii) The major limitations are

* turbulence modelling
* restrictions on geometric complexity
* computational resource for large 3D simulations.

(iii) Many complex flows however can be simulated using current technologies. This is expedited when interfaces exist with sophisticated mesh generators.

(iv) Advances in the field of turbulence modelling, numerical techniques and computing hardware will [illegible] made promoting the use of CFD for an ever wider range of applications and users.

7 REFERENCES

(1) MARINO C(Ed) Supercomputer Applications in Automotive Research and Engineering Development 'Proceedings of the Second International Conference on Supercomputing Applications in the Automotive Industry'
Seville, Spain October 1988.

(2) AMSDEN A A O'ROURKE PJ BUTLER TD 'KIVA-II: A Computer Programme for Chemically Reactive Flows with Sprays' LA-11560-MS, UC-96 May 1989

(3) SUMMA J M and DVORAK FA 'Computing Automotive Aerodynamics by an Integral Method', Proceedings Supercomputer Application in Automotive Research and Engineering Development, October 1986.

(4) CARR G W New MIRA Drag Prediction Method for Cars' Automotive Engineer June/July 1987 pp 34-38

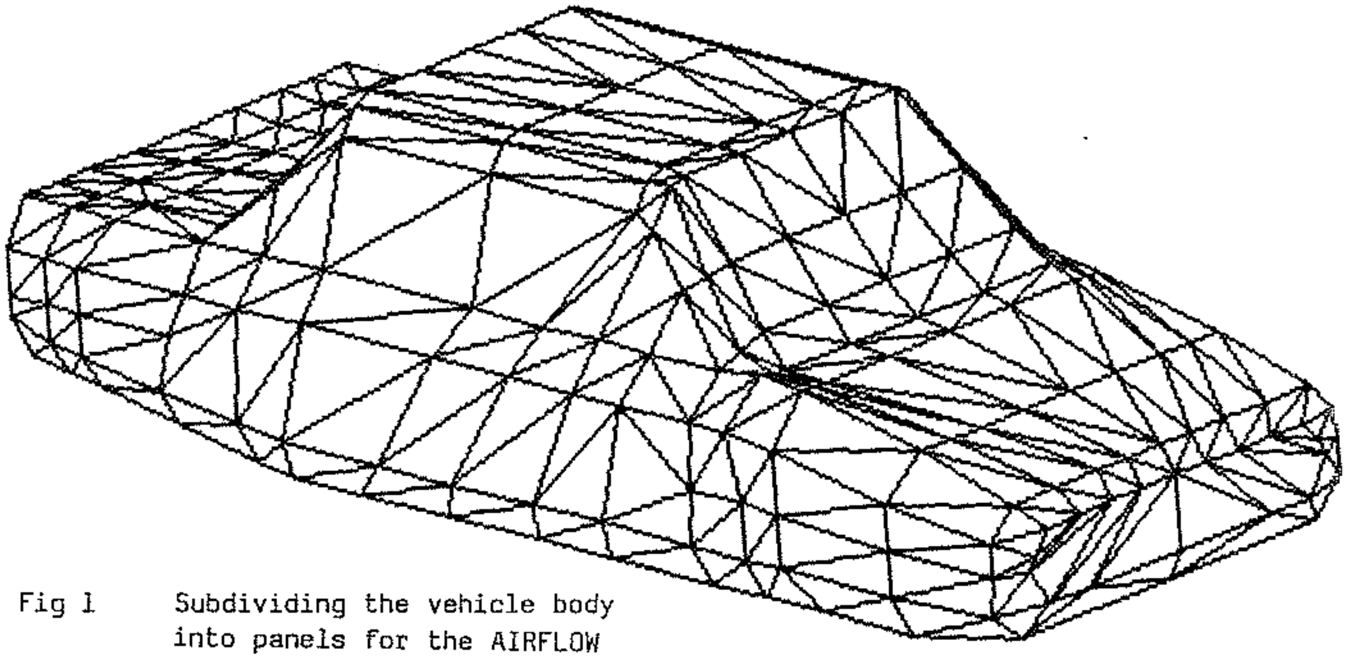

Fig 1 Subdividing the vehicle body into panels for the AIRFLOW programme

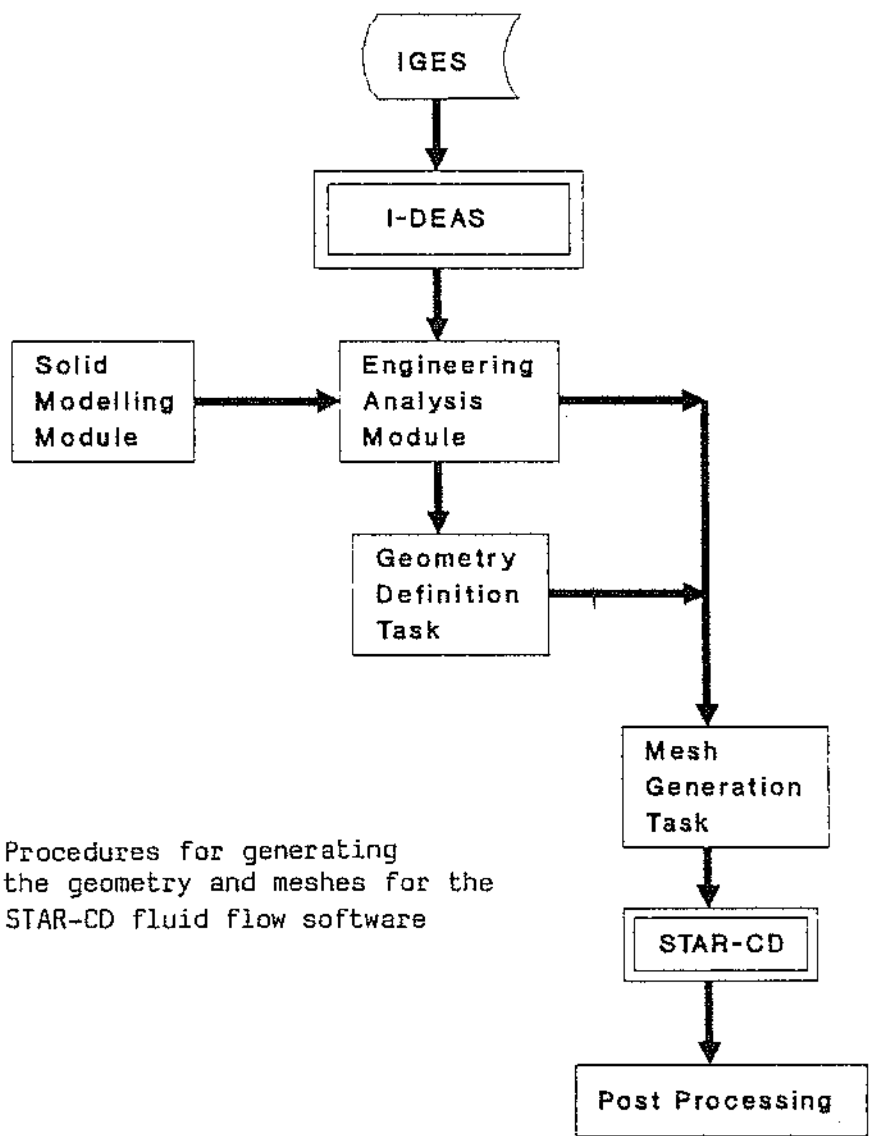

Fig 2 Procedures for generating the geometry and meshes for the STAR-CD fluid flow software

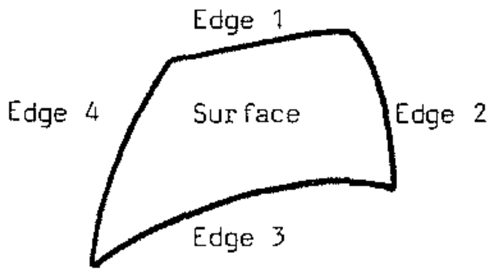

Fig 3a Curved surface with four edges

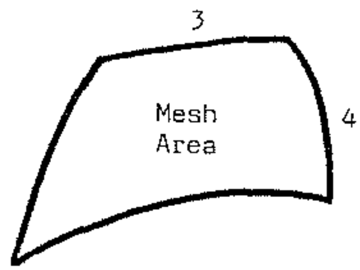

Fig 3b Defining the mesh area and cell specification

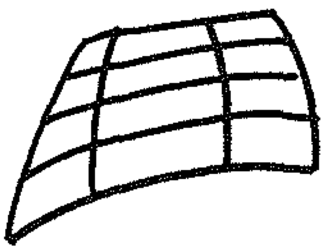

Fig 3c Generation of cells on the surface

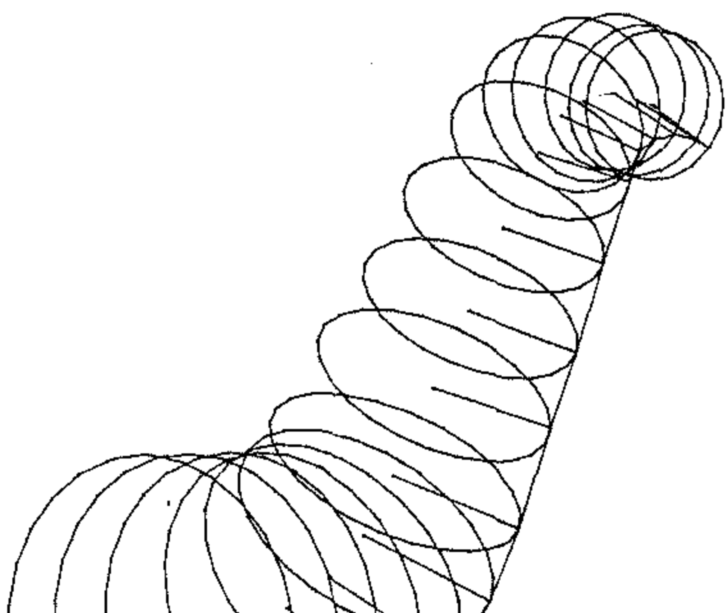

Fig 4a Sections used to construct the manifold

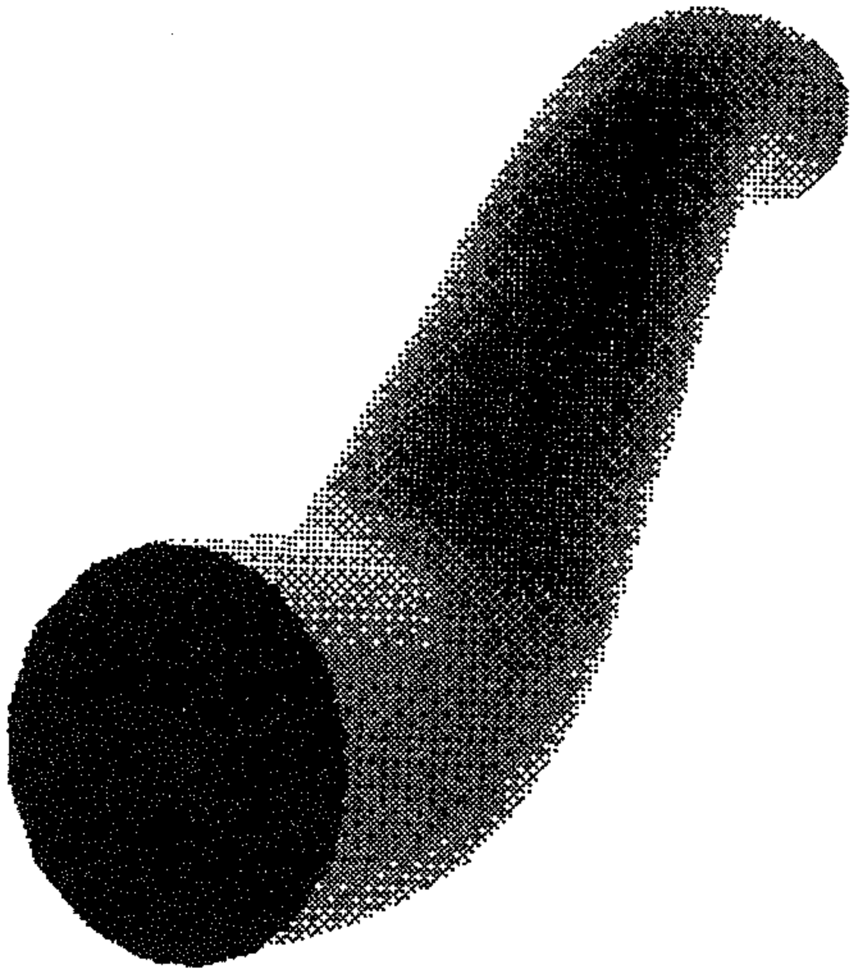

Fig 4b   Solid model of manifold generated by skinning sections

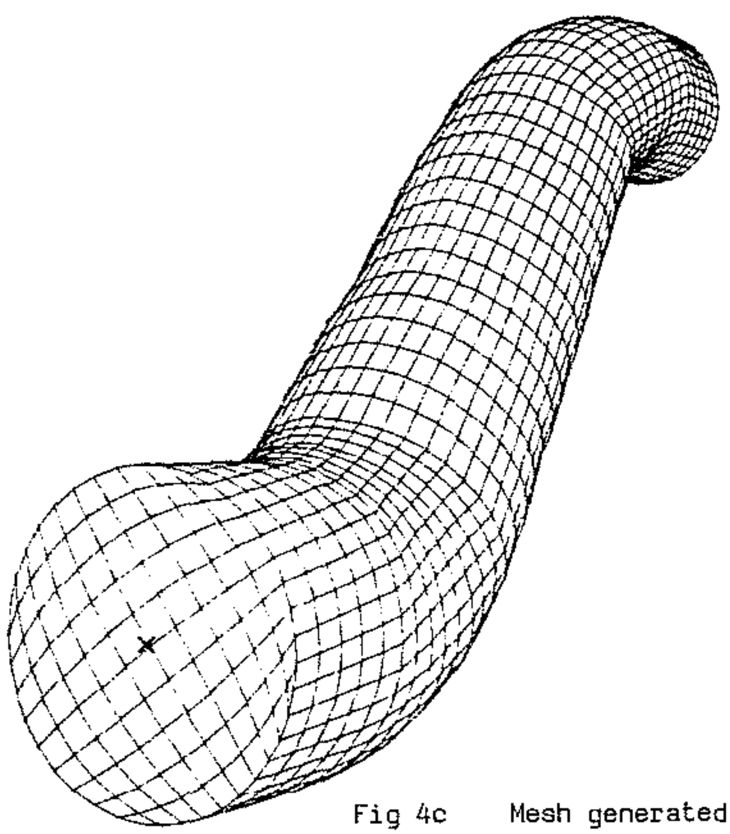

Fig 4c   Mesh generated in the manifold

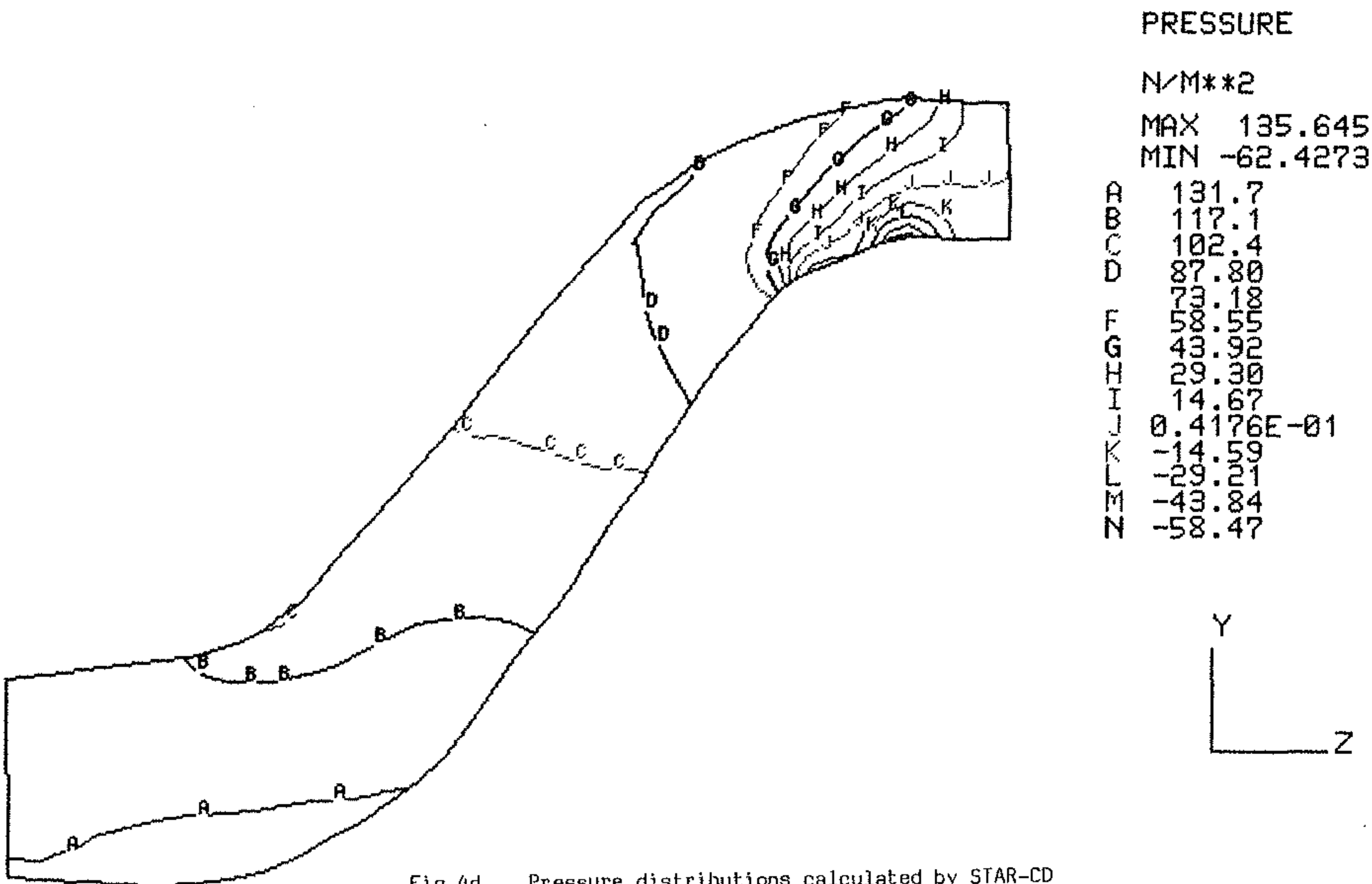

Fig 4d Pressure distributions calculated by STAR-CD

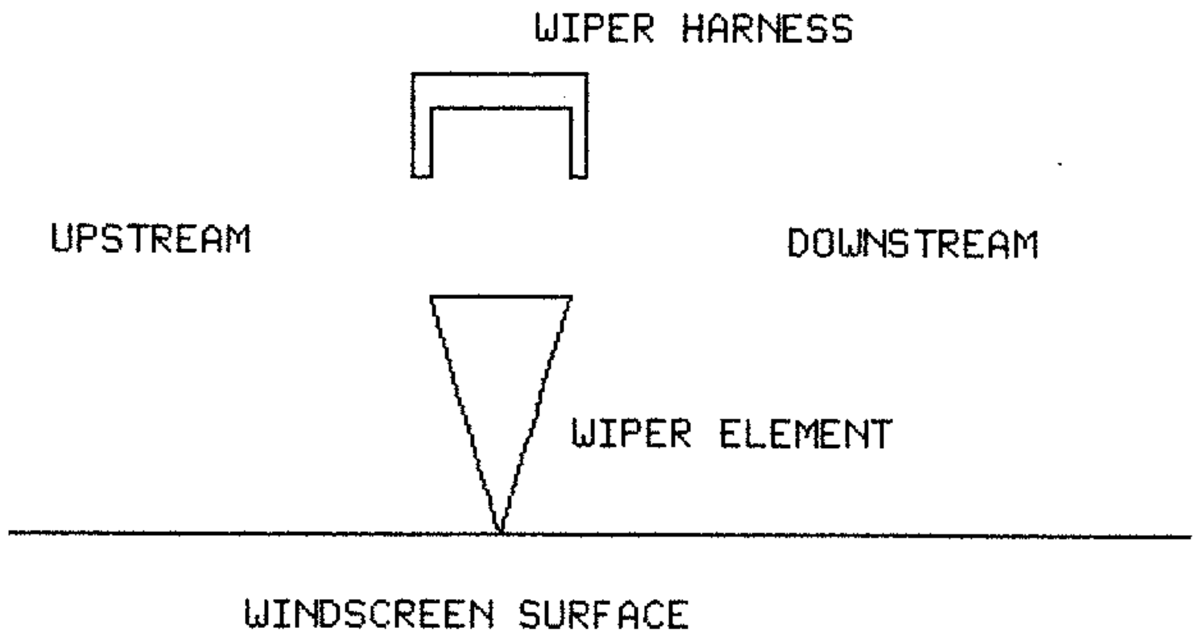

Fig 5a   Cross section of an idealised windscreen wiper

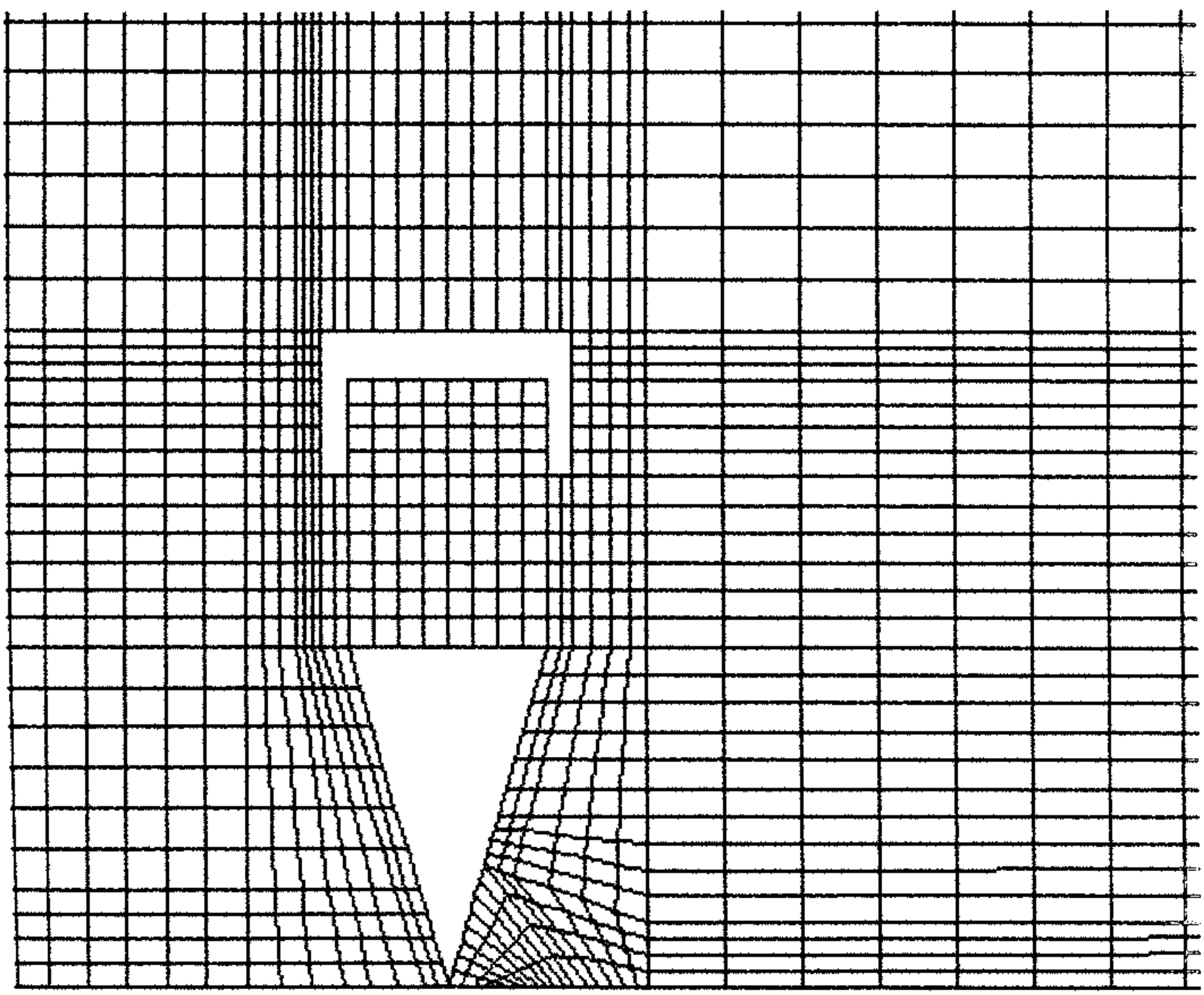

Fig 5b   Two dimensional mesh

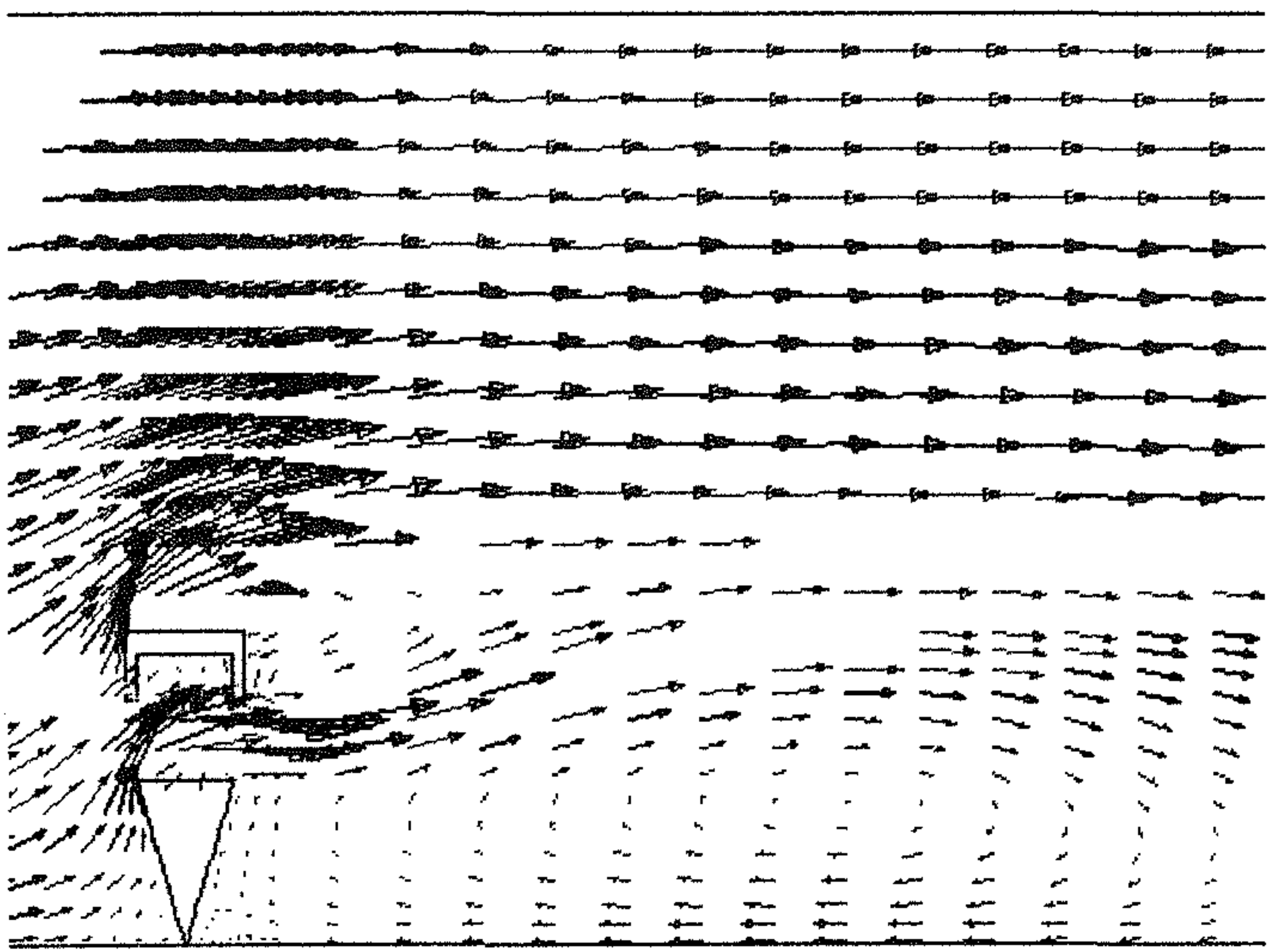

Fig 5c   Velocity distribution around the wiper

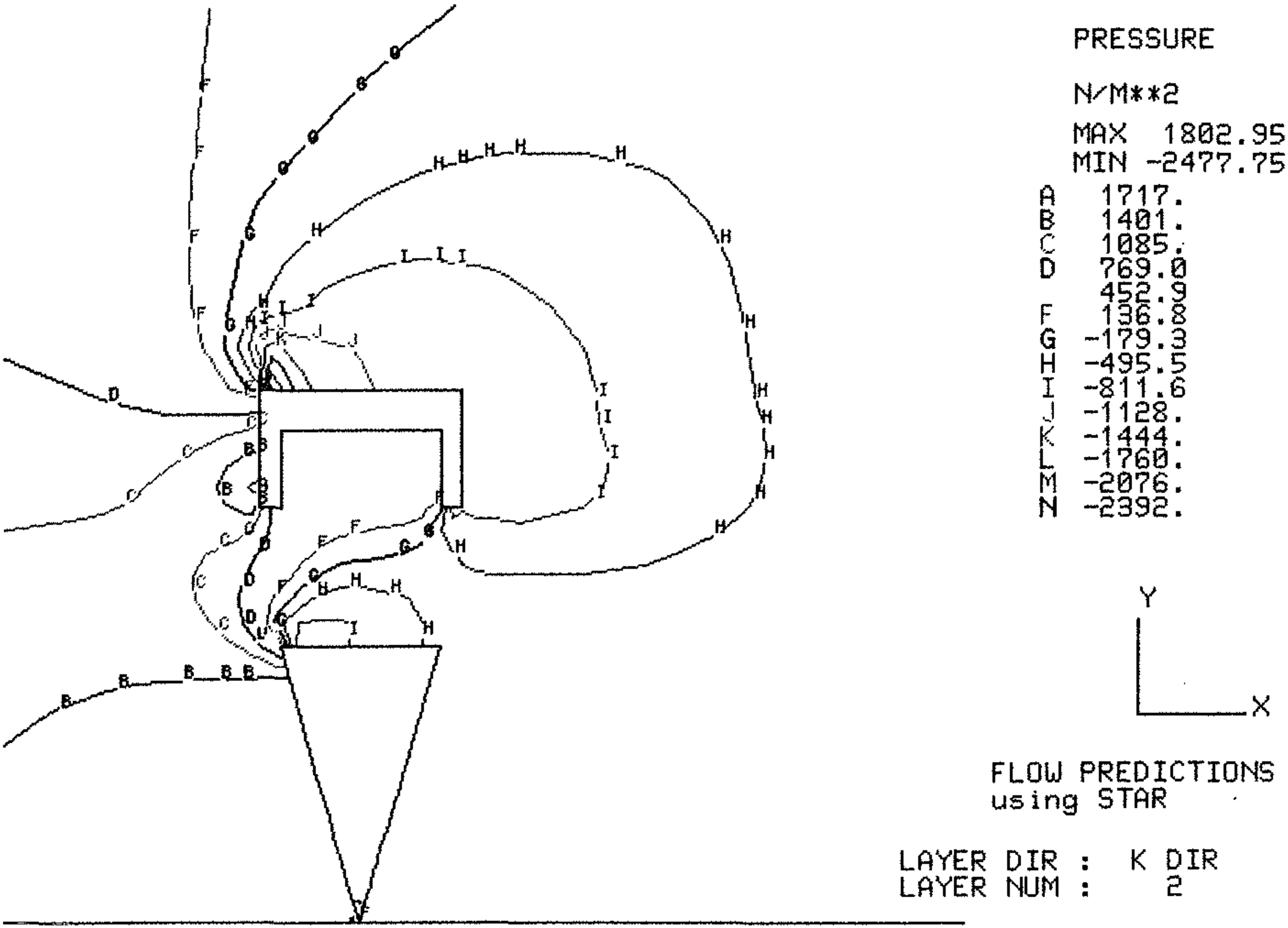

Fig 5d Pressure distribution around the wiper

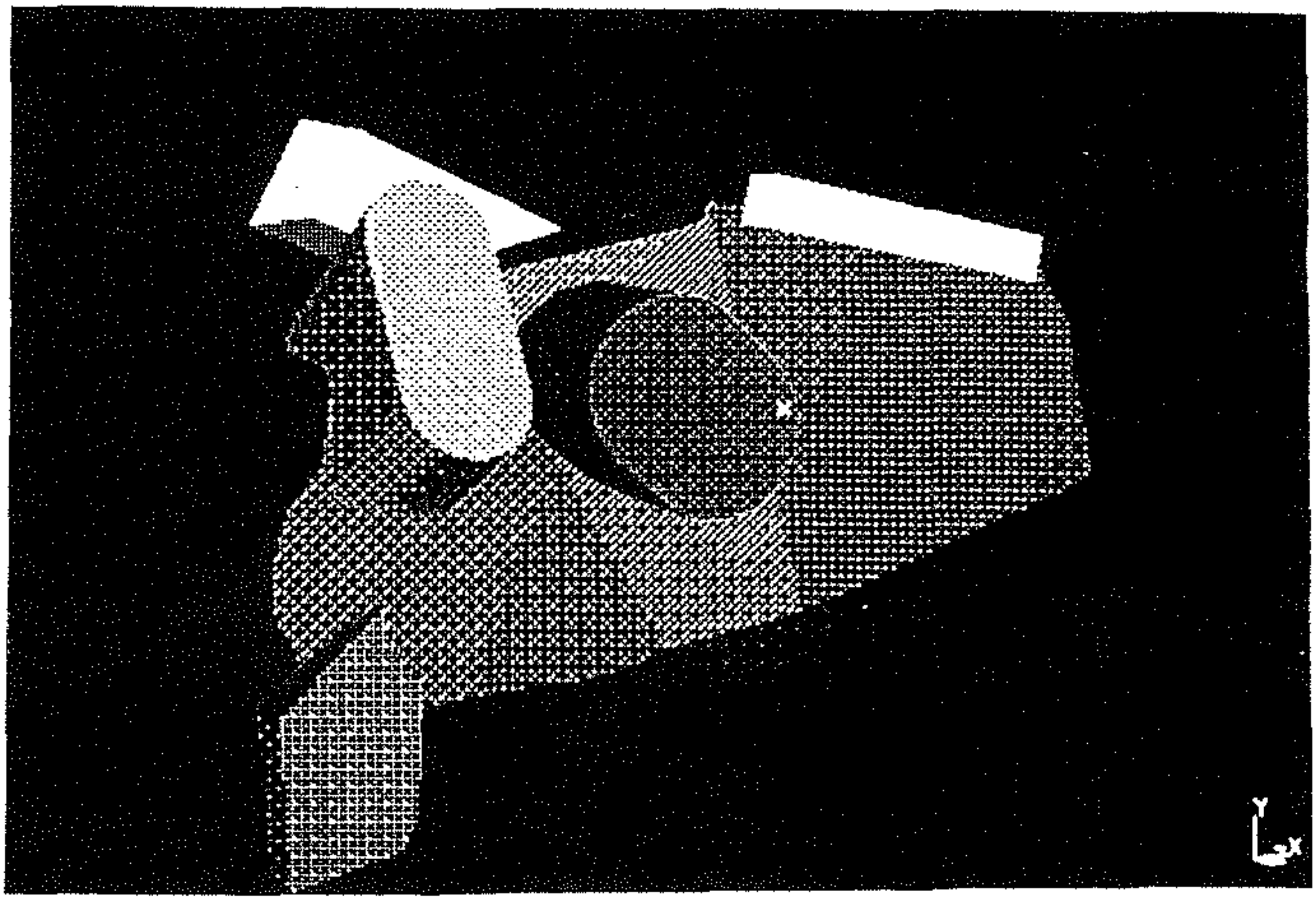

Fig 6a   Solid model of a heating and ventilation unit

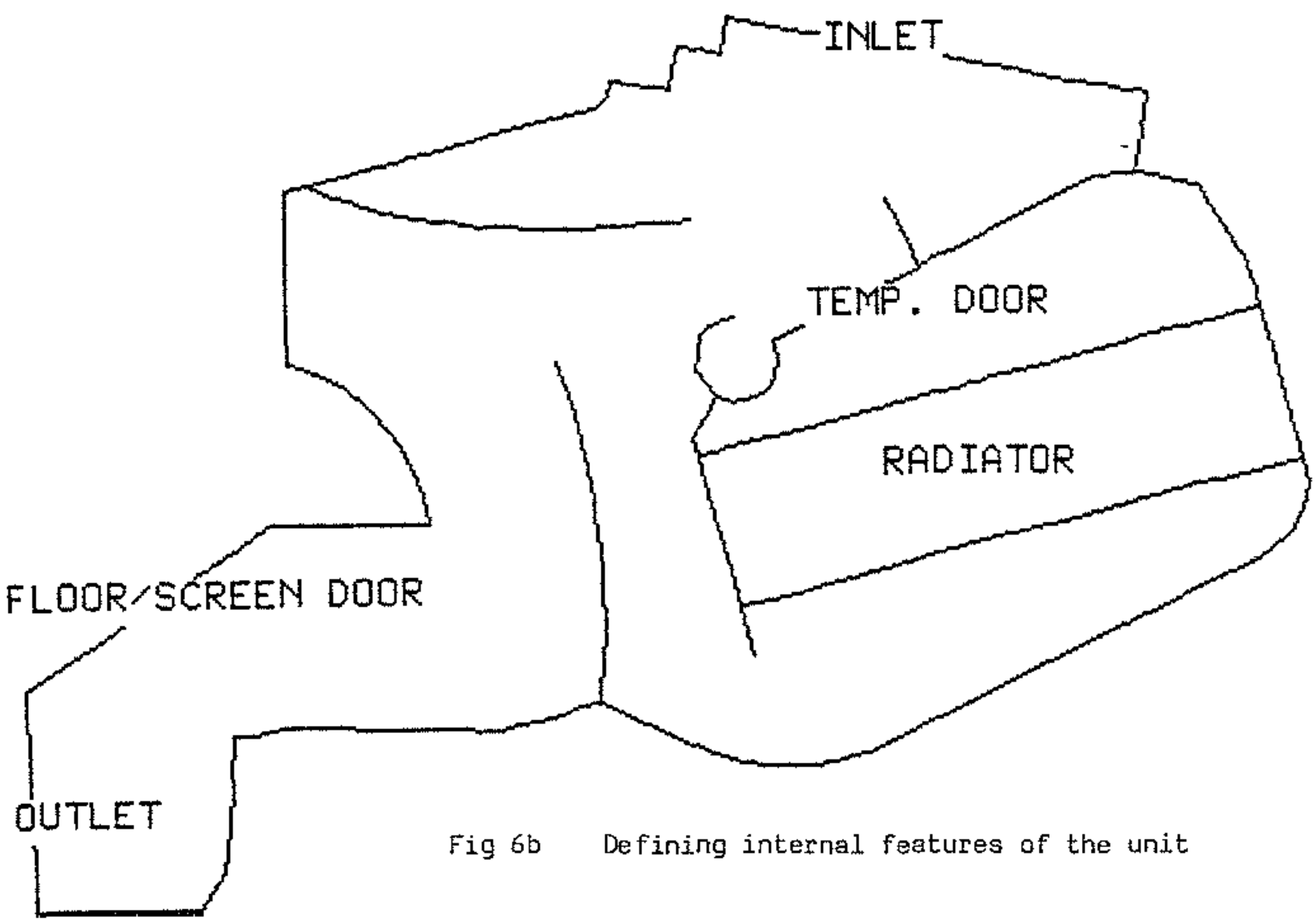

Fig 6b   Defining internal features of the unit

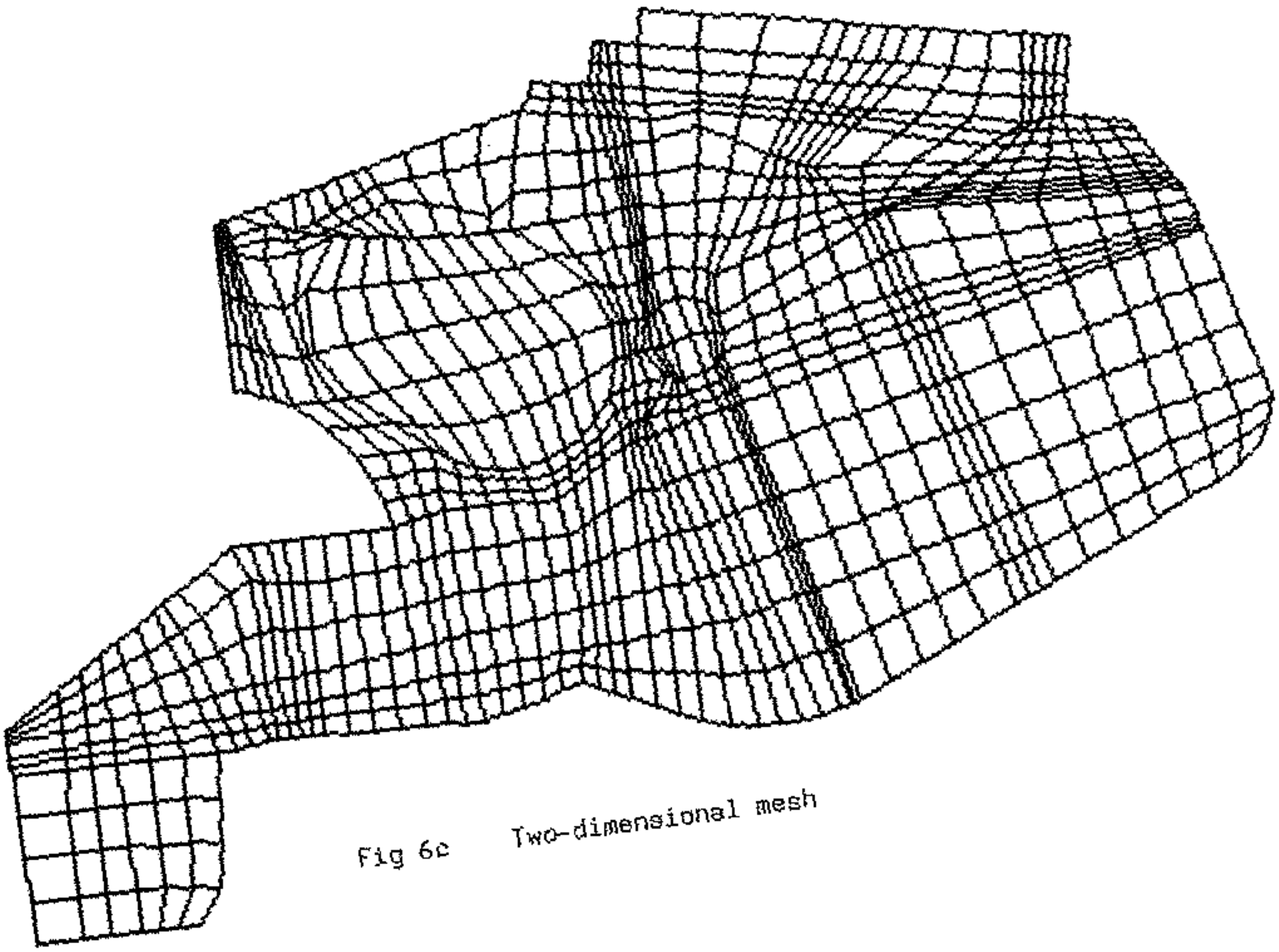

Fig 6c Two-dimensional mesh

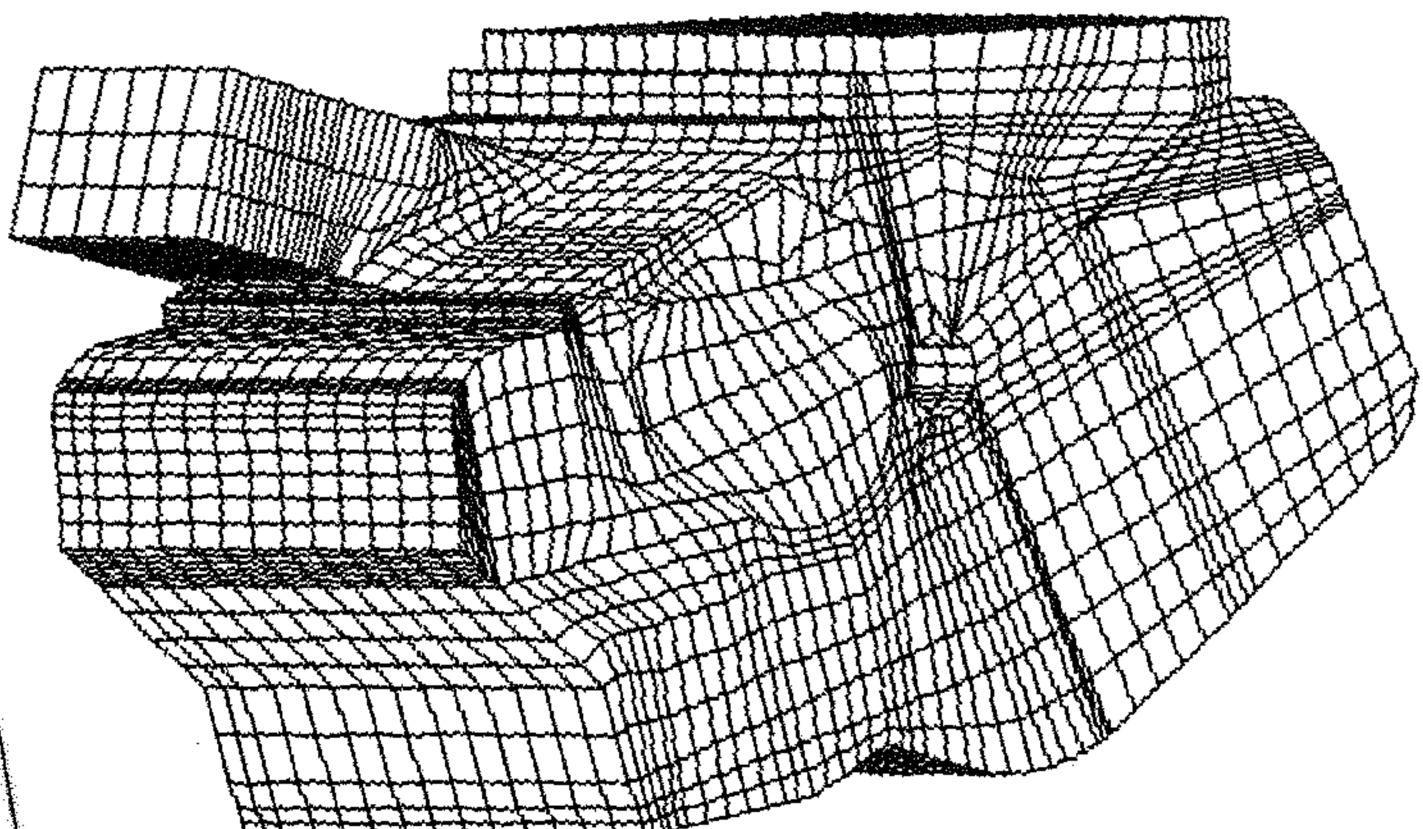

Fig 6d Three-dimensional mesh

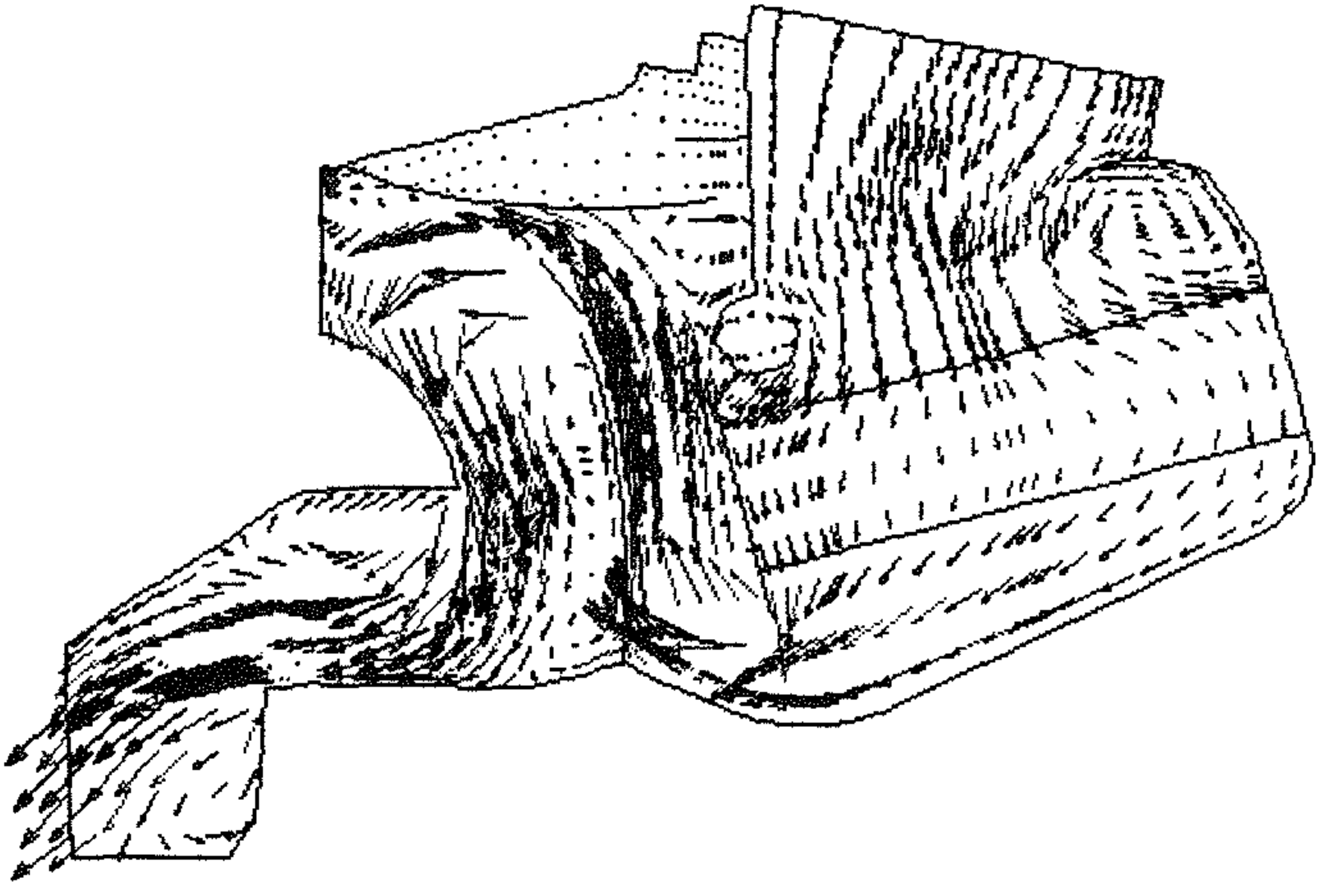

Fig 6e 2D simulation - Temperature door open and flow directed to the floor

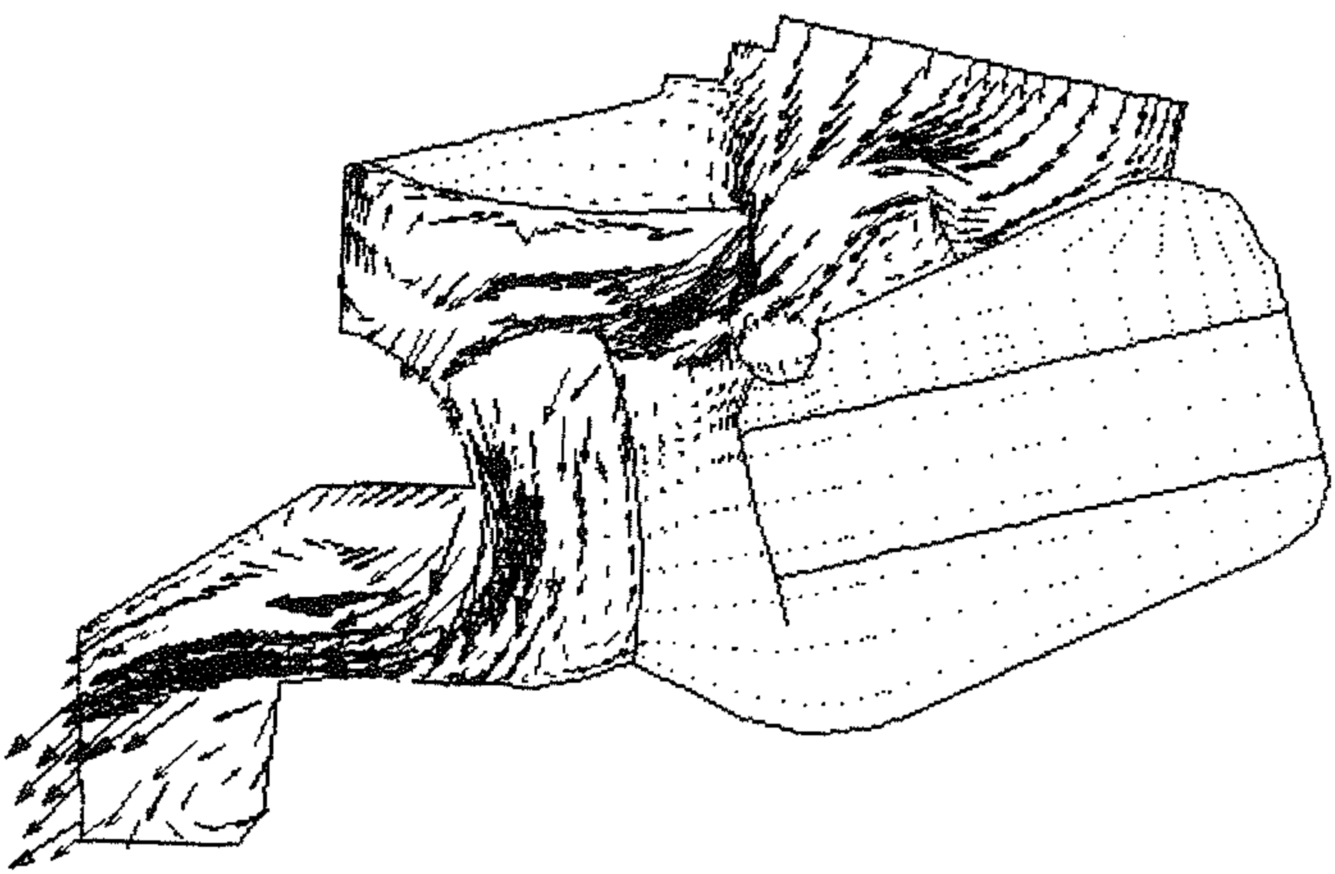

Fig 6f 2D simulation - Temperature door closed and flow directed to

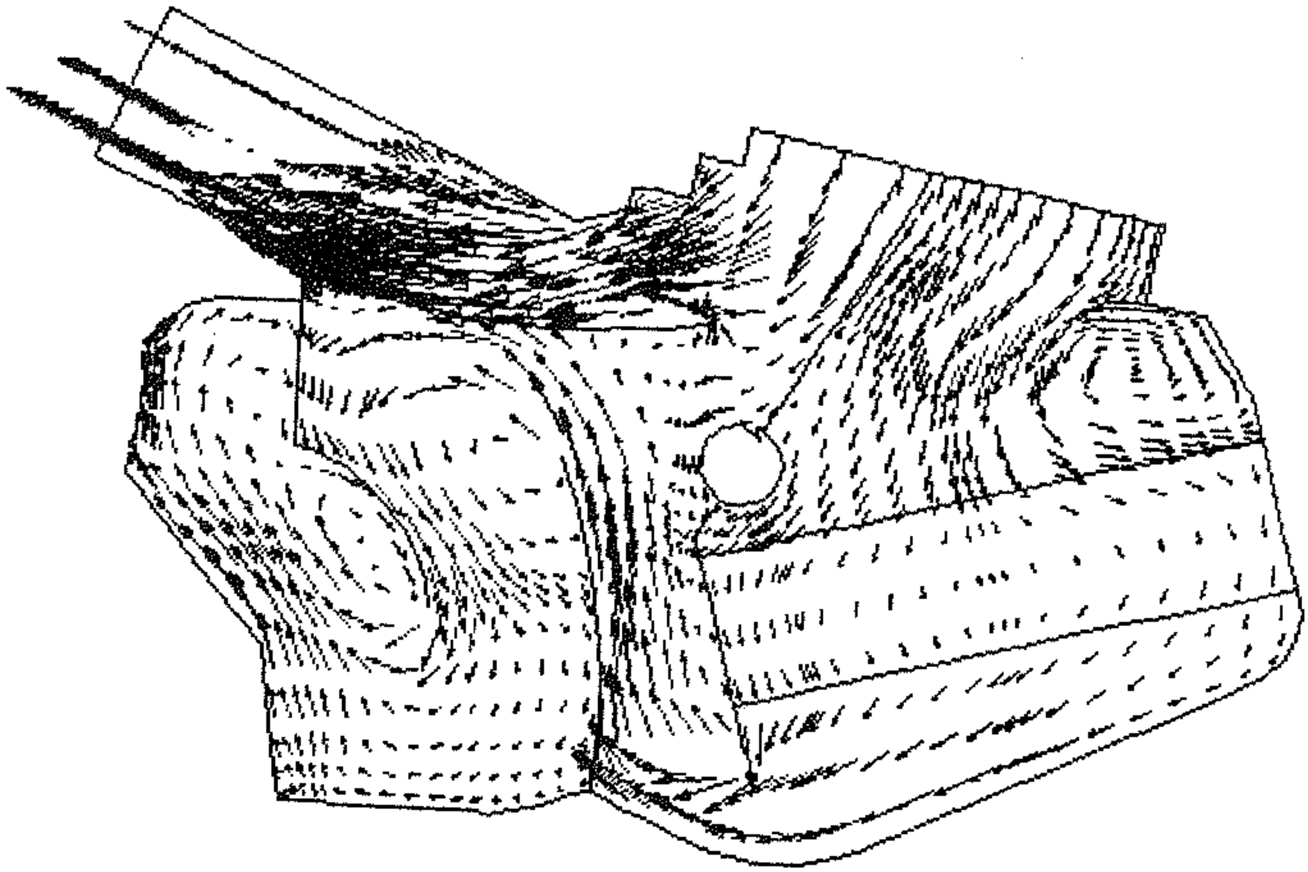

Fig 6g 3D simulation - Temperature door half open and flow directed to the screen and through the central register