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Information Integration for Finite Element Analysis at JET

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ABSTRACT

An information flow has been defined in order to permit finite element analysis of JET components with data stored in the CAD system, thus reducing the time involved and allowing for better geometry characterisation and boundary conditions of the desired component.

INTRODUCTION

In a complex machine like JET, finite element analysis is a common tool for analysis and simulation of components under various loads, i.e., thermal, mechanical, electromagnetic, or a combination of them.

In practice, the designer has to deal with both the generation of the geometry and the later analysis. In the generation phase for complicated components and/or complex loads the normal approximation would use the simplest geometry model to reduce man and machine time. As a consequence, this approximation can reduce the quality of the simulation. This paper explains how the geometry stored in the JET CAD system can be exported to a finite element pre-processor code and from here the analysis can be performed with the closest desired geometry and boundary conditions.

Although there are several commercial codes that convert CAD information into a format ready to be treated for the finite elements code pre-processors, they are usually expensive and the limited use envisaged at JET make them uneconomical. Instead, an accepted standard for interchange of information between the CAD codes and machine-tools, the IGES format, will be used.

FLOW OF INFORMATION

Fig 1 shows the desired path for the graphical information flow between the JET CAD system, CATIA (1), at one end and the most frequently used finite element code, ABAQUS (2), at the other. In the middle lies the pre and post-processor P3/PATRAN (3) for generating the finite element model and for the results visualisation. Although the information interchange between P3/PATRAN and ABAQUS is, in their current versions, fully integrated, this is not the case between CATIA and P3/PATRAN, thus losing the possibility of a complete path for information interchange. Consequently the proper implementation of the specification for information flow between CATIA and P3/PATRAN was the main aim of this work.

Traditionally, engineering drawings and associated documentation are used to communicate product definition data. Commercial interactive graphics systems, developed as aids to produce these two dimensional drawings, rapidly developed sophisticated three-dimensional solid modelling. This three-dimensional capabilities lead to CAM applications utilising product definition data in manufacturing (e.g. numerical control machining and computer-controlled co-ordinate measurements). In order to permit the compatible exchange or product definition data used by Computer Aided Design and Computer Aided Manufacturing, the IGES (Initial Graphics Exchange Specification) format was designed (4).

The IGES format is the format chosen for information interchange between CATIA and P3/PATRAN. This format is supported by all major mainframe CAD systems and a number of PC based CAD systems including AutoCAD. CATIA data exchange via IGES is achieved using the CATIA Interfaces module. Within this module there is CATMOD, an optimisation program used to optimise the CATIA model for the receiving system before the IGES file is written. A number of data transfer utilities have been developed, using the CATIA Interfaces program, to meet the data transfer requirements at JET (5).

On the other hand, the CATIA system is oriented also towards the generation of graphical entities for manufacturing. CATIA solids are generated as the volume resulting from the intersection of a number of planes defining surfaces and edges, fig 2. This arrangement is very useful for manufacturing because machine-tools can use those generating planes as cutting planes. Unfortunately this particular arrangement for geometry generation means that some CATIA entities are not directly useful to transfer to P3/PATRAN as it is mainly designed as a pre- and post-processor for finite-element analysis. Consequently, a number of geometrical entities related to solid modelling have to be modified in CATIA before they would be ready to transfer in order to avoid the limitations of both CATIA and the IGES format for our purpose.

Geometrical entities available to transfer are those currently supported by IGES, but IGES does not support solids or volumes. Therefore any 3D models have to be transferred as wireframe models or edge surfaces. This is a major problem as most entities to be analysed at JET are in fact 3D solids. As a result, some final treatment has to be done using P3/PATRAN in converting the transferred model into solids.

At CATIA level the main steps are :

1) Delete all entities not supported by IGES, i.e. solids, volumes, etc. Leave only the elements required, that is, the model for transfer should contain only surfaces, faces and geometry (lines, curves, grids, etc.)

2) All faces should have a maximum of four edges. Faces with more edges have to be split into two or more until this requirement is met. This point, although not strictly necessary, is strongly recommended as it will enhance the capability of the mesh generator in producing a better parametric connectivity between surfaces and a mesh with less distortion elements.

3) Every two topologically connected faces have to share a full edge.

4) If in the end the model has to be a solid, then the faces have to be arranged in such a way that this arrangement improves the later solid generation in P3/PATRAN.

5) Geometry tolerances have to be set so that faces that are effectively connected through their respective common edges are still connected after transfer.

When all these requirements have been met, the models are usually transferred successfully from CATIA to P3/PATRAN using the IGES format. Once the model has been transferred, the solid geometry can be created from the proper combination of two-facing surfaces.

On average, the time involved in carrying out all these processes varies, according to our experience, between a few minutes up to one hour for complicated models. After that, the generation of the IGES file itself takes just a couple of minutes.

Figures 3-7 show a series of solid models from CAD data mostly used for thermal analysis of Mark-I and Mark-II divertors.

CONCLUSIONS

1) A complete path for information flow between the JET CAD system (CATIA) and the finite element code ABAQUS with P3/PATRAN as pre and post-processor has been implemented.

2) IGES files are used to link CATIA models with P3/PATRAN. The proper implementation of this specification for our particular purpose has been carried out.

3) The processing time involved is relatively small compared with the time required to generate the complete model in P3/PATRAN.

REFERENCES

[1] CATIA. IBM/DASSAUT SYSTEMS. Paris. France

[2] ABAQUS USER'S MANUAL, Version 5.3. Hibbitt, Karlsson & Sorensen, Inc. Pawtucket, RI. USA

[3] P3/PATRAN User Manual. PDA Engineering. Costa Mes, Ca. USA

[4] INITIAL GRAPHICS EXCHANGE SPECIFICATION (IGES). Version 4.0. U.S. Department of Commerce. National Bureau of Standars. USA

[5] C.A. Earl. CATIA DATA TRANSFER UTILITIES, User Guide. Revision 1. Joint European Torus, Abingdon. UK

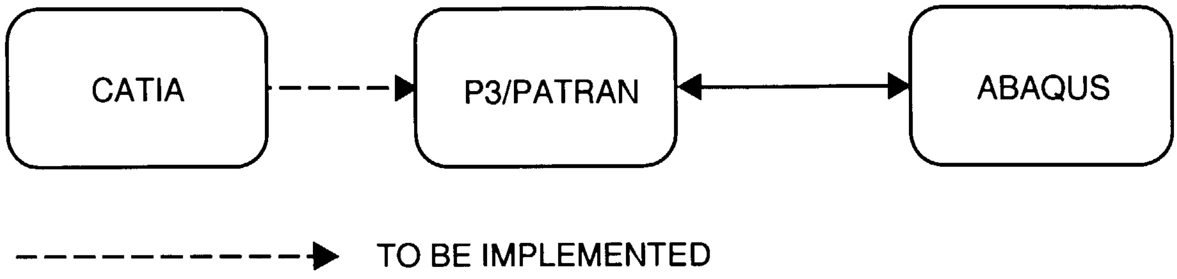

Fig.1 Graphics Information Flow

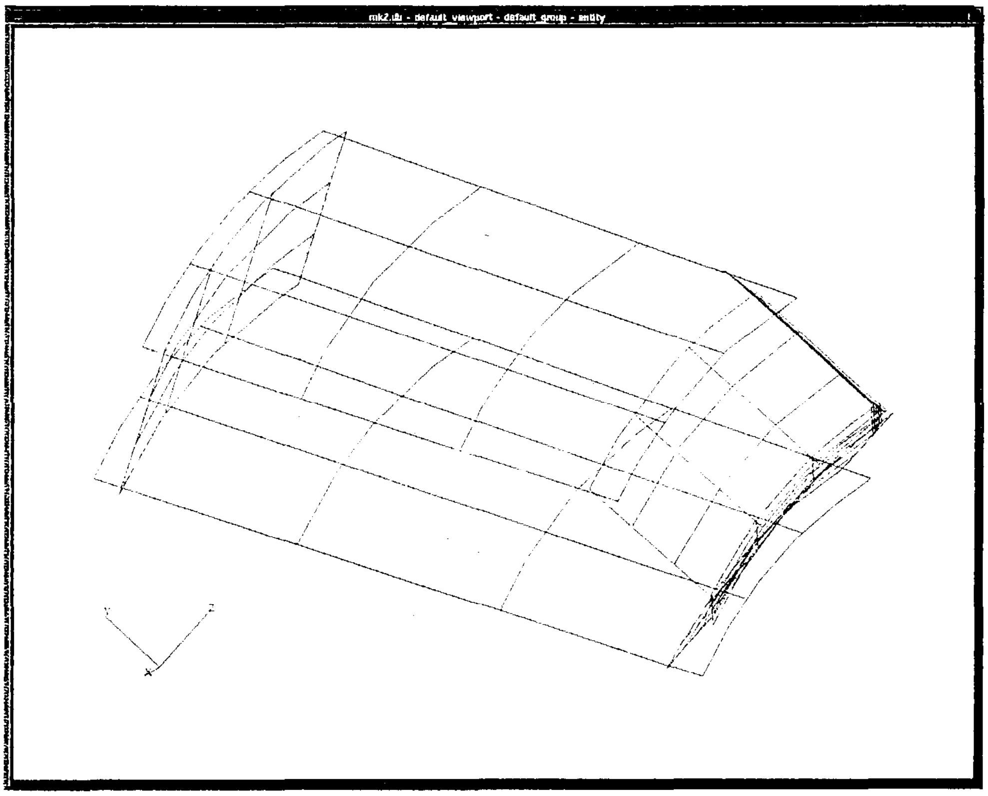

Fig.2a Mk-II tile as defined by CATIA

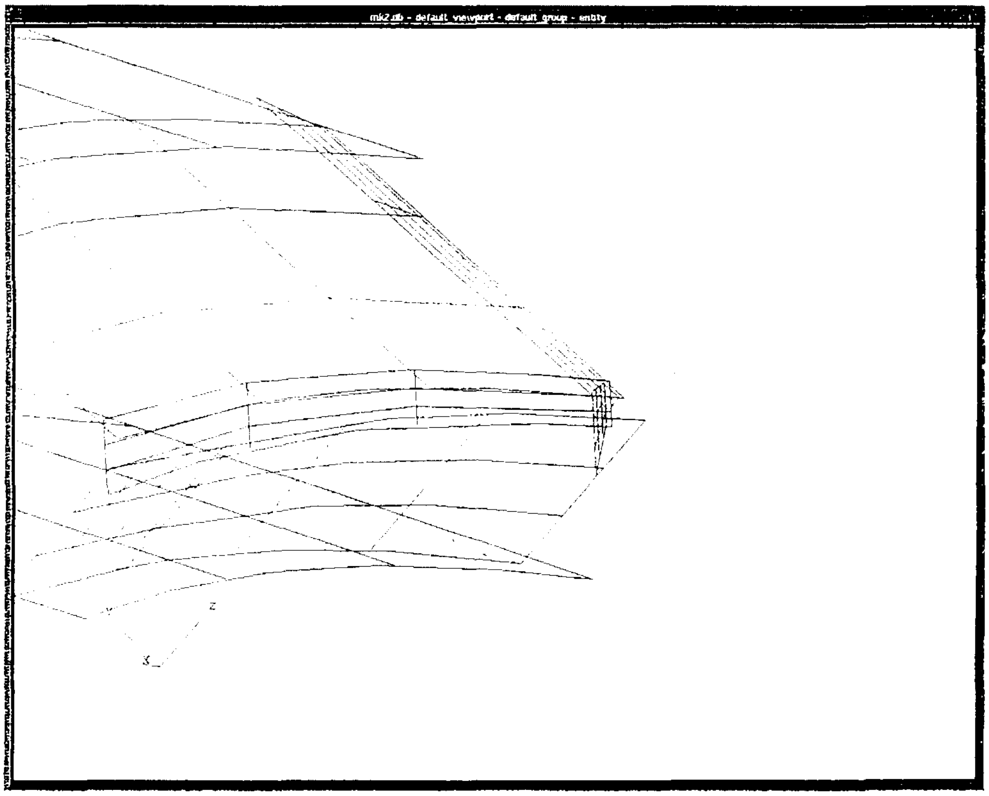

Fig.2b Corner Detail of Fig. 2a

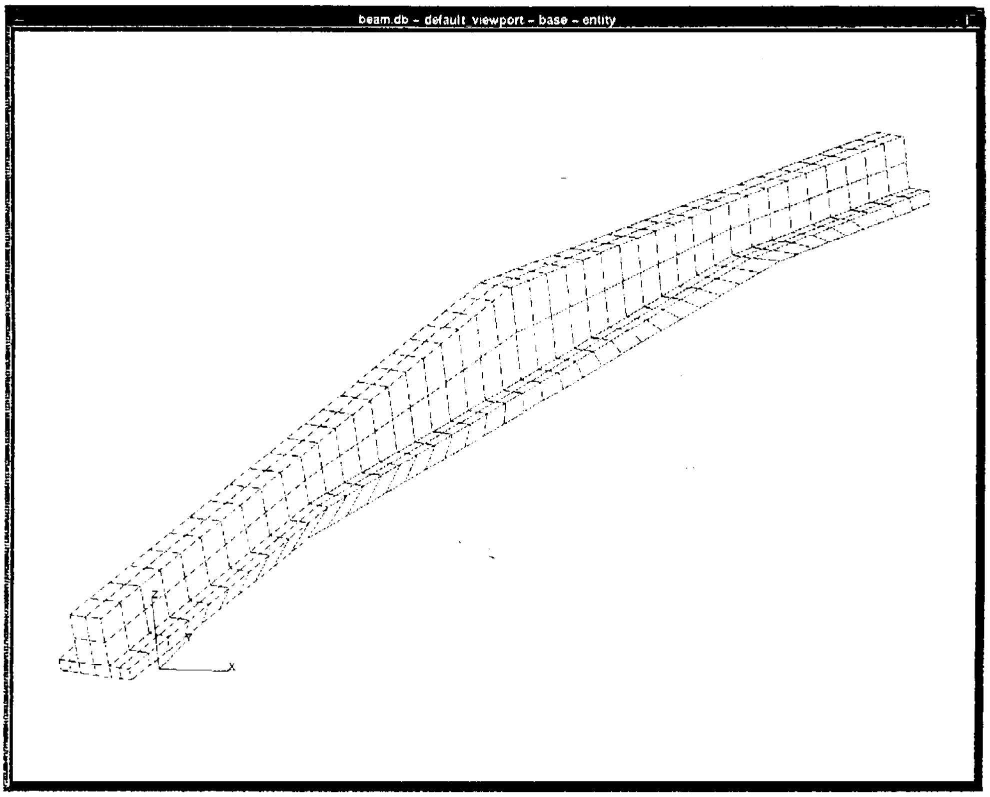

Fig.3 Mk-1 tiles support beam

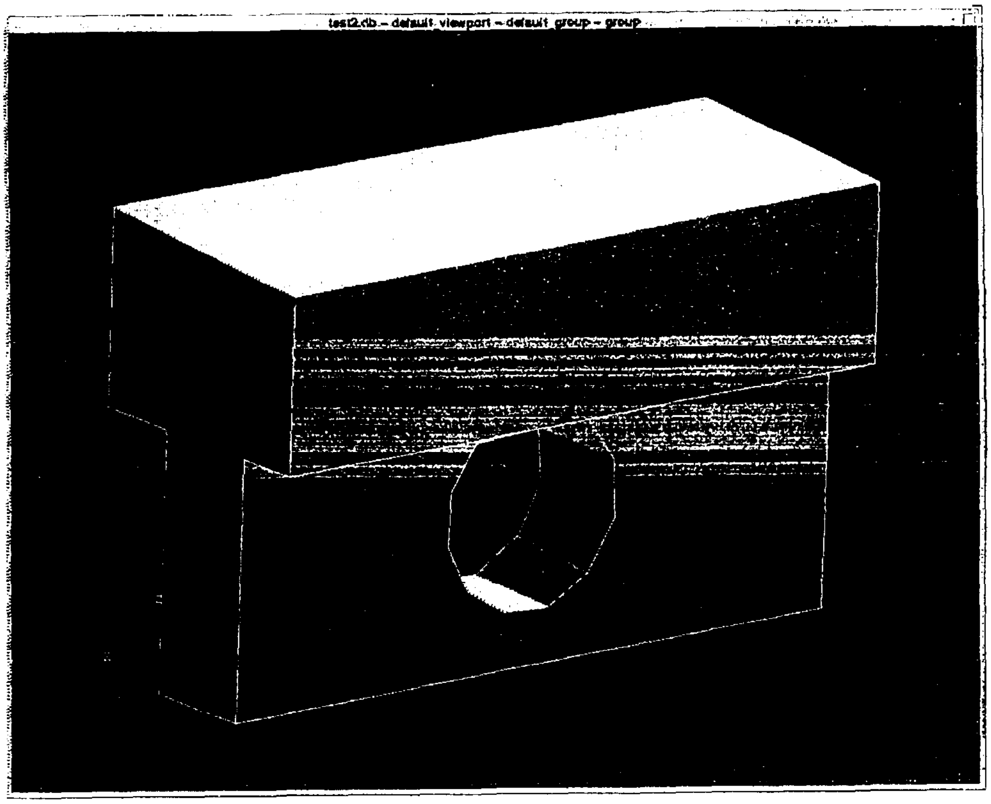

Fig.4a Solid Mk-I tile model

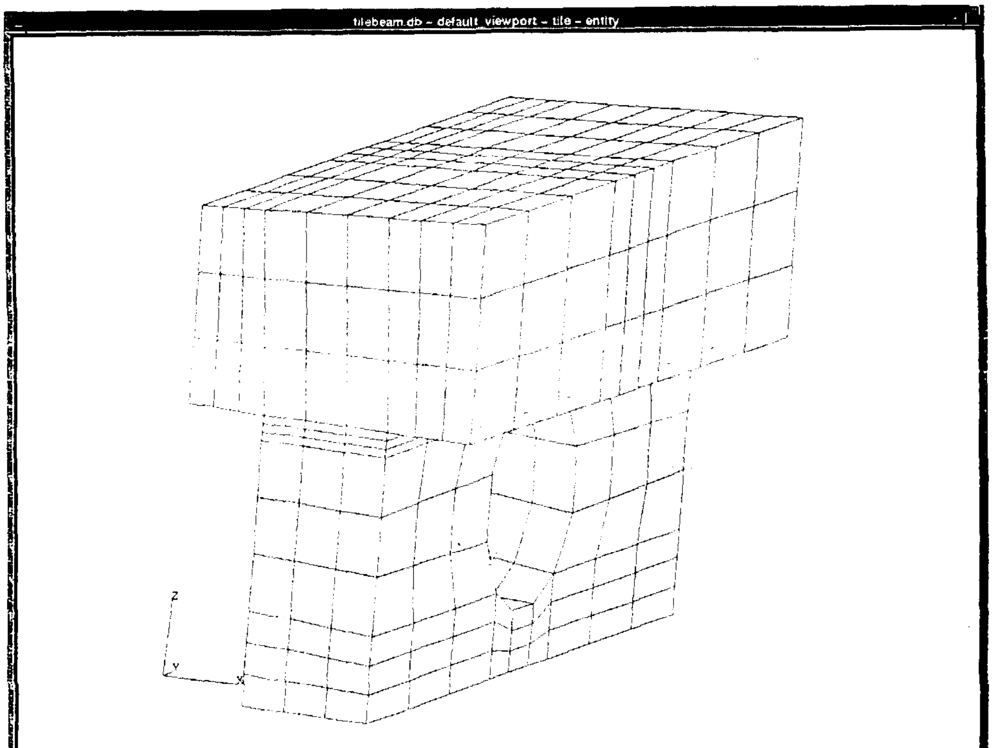

Fig.4b Meshed Mk-I solid tile model

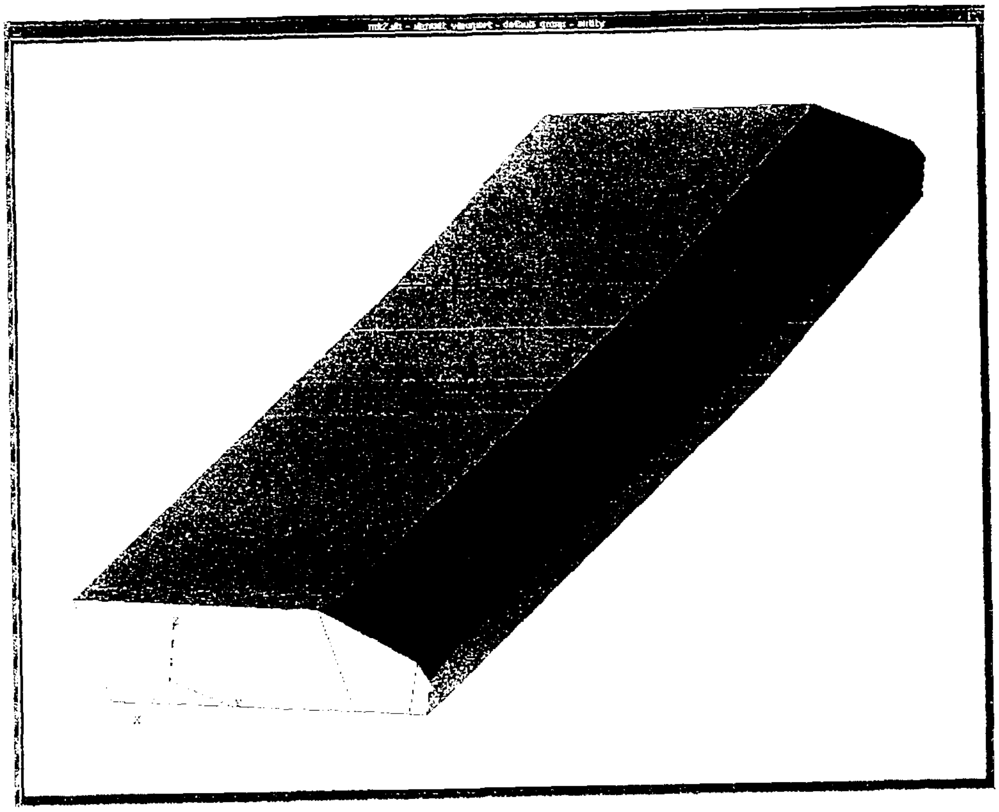

Fig.5a Solid Mk-II tile model

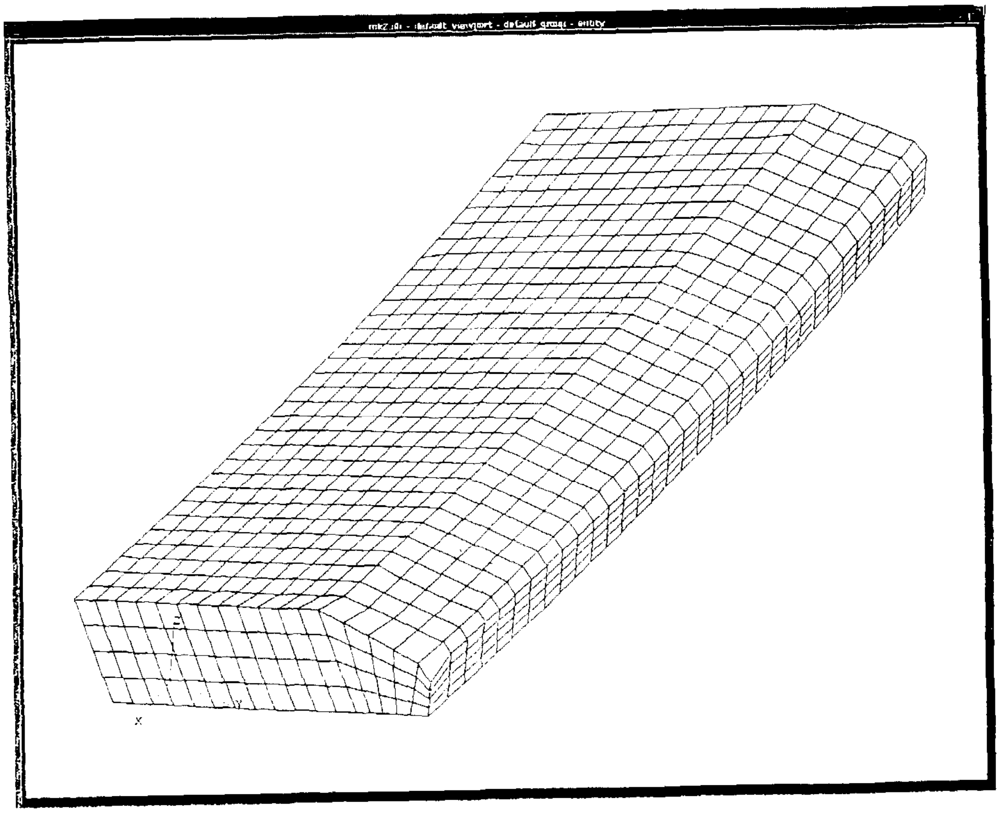

Fig.5b Meshed Mk-II solid tile model

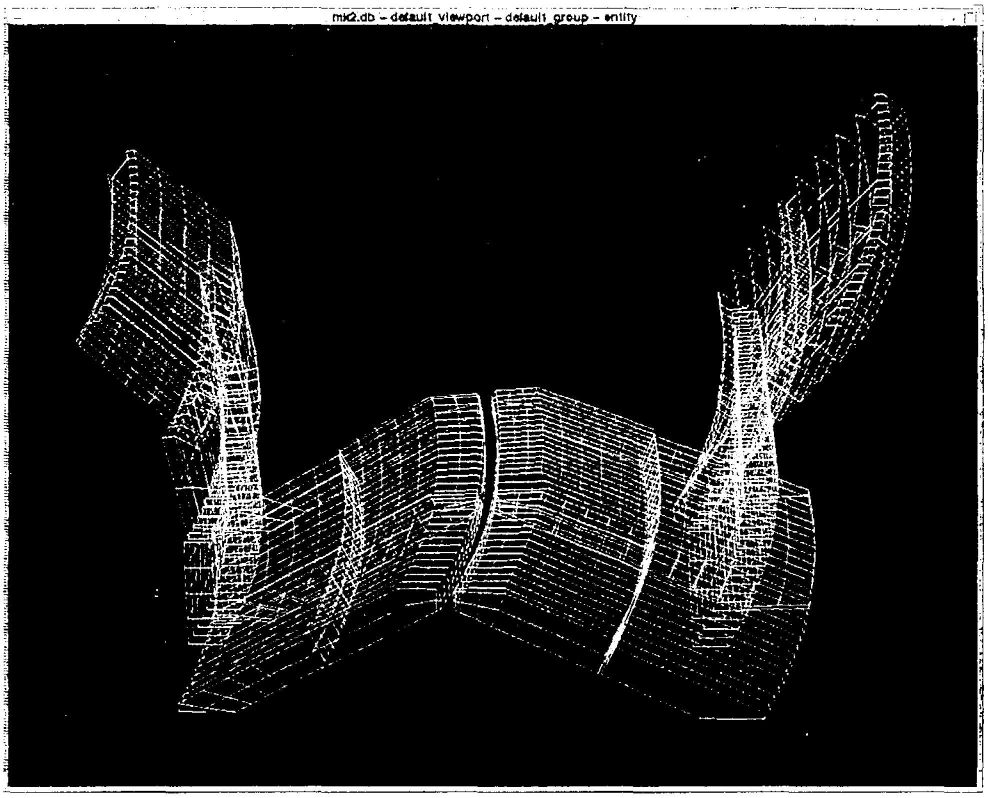

Fig.6 Surface mesh of the Mk-II divertor tiles

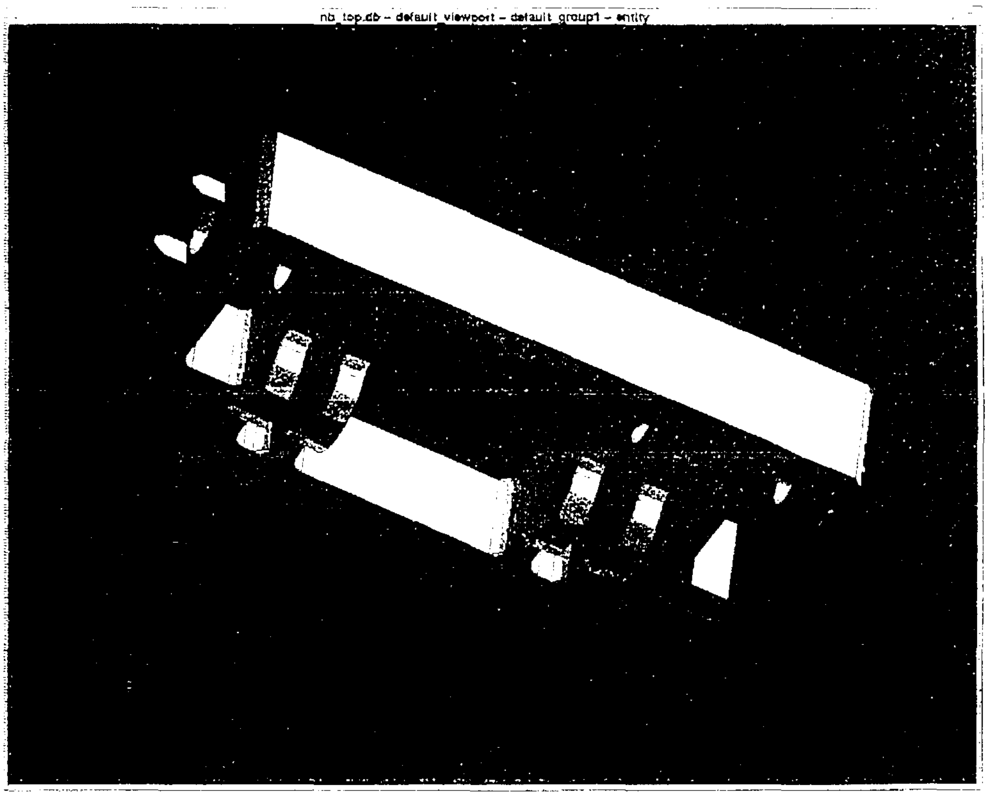

Fig.7 Top support of the poloidal limiter.