THE USE OF AUTOMATIC TET MESHING WITHIN A CONCURRENT ENGINEERING ENVIRONMENT

C.P. Griffiths
British Aerospace Plc
Military Aircraft & Aerostructures
Brough, East Yorkshire, England
e-mail:- Paul-C.Griffiths@bae.co.uk

ABSTRACT

The use of CATXPRESS to enable CATIA solid geometry to be transferred to MSC/PATRAN has been investigated. This geometric data has then been used to generate a finite element model using the automatic meshing routines available in MSC/PATRAN. New working methods have been developed which make use of this facility and three examples of 'real' engineering problems are presented.

1.0 INTRODUCTION

Traditionally, finite element analysis (FEA) has been used during the certification of a design or as an aid in solving particular design problems after manual methods have failed. FEA was rarely, if ever, used during the concept definition and detailed design stages of the design process because conventional model building techniques were labour intensive, slow and could only be carried out by 'experts'. However competitive pressures are now forcing British Aerospace to use numerical simulation and optimisation methods throughout the entire design process to help reduce time-to-market and improve product quality.

Efficient concurrent engineering and teamworking skills will be critical if companies such as British Aerospace are to maintain their competitive edge. Due to the complex, multi-technological nature of aeronautical products, opportunities for improving existing technologies/design tools in isolation are limited. However significant improvements to the design process can be gained from the use of fully integrated technologies within a concurrent engineering environment.

At British Aerospace, the concurrent engineering environment is assuming a greater significance within all new project teams as new design methodologies are introduced and the concept of the 'paperless' design becomes a reality. With structures, design, systems and production engineers all working in close proximity to one another a requirement to achieve a rapid analysis capability was identified as offering efficiency improvements to the design process. In order to satisfy this requirement, the use of CATXPRESS to transfer CATIA solid geometry to MSC/PATRAN has been developed. CATIA is the CAD/CAM/CAE system currently in widespread use at British Aerospace and enables both drafting and solid modelling to be carried out. The geometric data transferred into MSC/PATRAN has then been used to generate a tetrahedral finite element model using the automatic meshing routines that are available.

These new working methods have been used on a number of production analyses alongside more traditional model building techniques, and have been shown to offer significant time and cost savings. In addition, the ability to include detailed geometric features in the finite element model has lead to more representative analyses being performed.

Three examples of 'real' engineering analyses are presented:

i) avionics bay door hinge bracket
ii) tailplane skin
iii) flight refuelling probe casting

Significant savings in time, and hence cost, can be demonstrated for each example given. At the same time solution accuracy has been maintained and in the case of the hinge bracket has actually improved due to the inclusion of detailed geometric features which would otherwise have been omitted from the analysis.

2.0 CATIA →CATXPRESS→MSC/PATRAN

MSC provide a facility called CATXPRESS which integrates into the CATIA environment and allows the user to generate a file containing geometric data which can subsequently be imported into MSC/PATRAN. At the time this work was carried out MSC/PATRAN Version 6 was used. The method developed and currently adopted by British Aerospace (Brough) for the analysis of solid components is to automesh this imported data using the meshing routines available in MSC/PATRAN and analyse the finite element model using the MSC/NASTRAN solver.

The integrity of the imported geometric data is of prime importance if a rapid analysis is to be achieved. A number of CATIA solid models, ranging from simple to complex in detail, were used to test CATXPRESS and its ability to consistently transfer 'useable' data between CATIA and MSC/PATRAN. Eight models were used during this exercise, a selection of which are shown in figures 1 to 4.

Only that solid shown in figure 2 would not automesh. Examination of this solid in both MSC/PATRAN and CATIA gave no clues as to why a problem should exist with this model. Close examination of the constituent surfaces in MSC/PATRAN revealed a surface that was totally corrupt. Surprisingly, it was one of a pair of similar surfaces either side of the 'symmetry' line (the solid is not exactly symmetrical), the opposite surface being perfectly OK. To further complicate matters this surface could not be deleted (in MSC/PATRAN) or a new surface constructed in its place.

Examination of the construction history in CATIA indicated that the solid had been generated in the 'standard' way with nothing to indicate that a problem would exist with data transfer. This example illustrates the point that even relatively simple geometric entities may cause problems with no logical explanation. Clearly there is still much work to be done in the area of data transfer.

Under normal circumstances the most common problem with the imported data was associated with topological errors. The information which informs MSC/PATRAN that a surface edge is connected to an adjoining surface edge is either incomplete or corrupt. MSC/PATRAN then interprets this to mean that a gap (in many cases of zero length) exists between the two surfaces, and as these surfaces are used in the definition of the solid, MSC/PATRAN is unable to construct the solid geometry.

When this situation occurs the user has a number of options available. A capability to 'edge match' is available which automatically searches the database and creates the necessary topological information to allow the solid to be constructed. However, because a user definable geometric tolerance is used in deciding if surface edges are actually matched, it is possible that not all surfaces are edge matched. This option is very useful and is recommended in all situations where gaps exist between surfaces, approximately 80-90% of geometry transfer problems being solved in this way.

An alternative approach is to automesh the geometric surfaces and 'stitch' the elements together using the 'node equivalence' option. Assuming a continuous surface mesh can be achieved, the elements can be used to define the boundary of a solid for the purpose of solid automeshing. A possible disadvantage of this approach is that, assuming the edge match option has been tried unsuccessfully, the 'gap' could be so large that forcing the nodes to merge could change the geometry so much that the results are affected, hence this approach must be used with caution. An advantage of this option is the user can view the surface triangular mesh at an early stage which will give an indication of the quality of the final mesh before the solid model is generated.

Finally individual problem surfaces can be deleted and re-constructed using the geometry modification/creation tools in MSC/PATRAN. For the user to consider this course of action the surface is usually so distorted that it may well be difficult to maintain the accuracy of the original solid geometry in the modified region. A preferable option would be to trace the error in the original CATIA model and attempt to 'clean' the geometry in the CATIA environment. This also has the advantage of helping to educate the CATIA user in the strict requirements of the data transfer process and will help to improve the future quality of CATIA solid geometry.

CATXPRESS can translate both solids and volumes produced by CATIA. It was quickly recognised that volumes generally produced more 'useable' geometry in MSC/PATRAN than did the solid type entity. As an example, figure 5 shows the surfaces which define a solid in MSC/PATRAN. These surfaces originated from a CATIA solid. When this solid is automeshed the elements will respect the surfaces, thus, in the area highlighted, elements with very acute internal angles will be generated. This may, depending on the mesh density used, cause the MSC/NASTRAN solver to fail. If a volume is extracted from the solid in CATIA and then transferred to MSC/PATRAN this acute angle disappears as new faces are generated to define the volume, resulting in better quality elements being produced.

A major fault with the MSC/PATRAN/NASTRAN system was that on several occasions models containing linear tet elements solved successfully while the same geometry meshed with parabolic tets failed. This was due to the presence of very acute (almost zero) internal geometric angles which cause the mesher to generate 'flat' elements. Parabolic tets are much more susceptible to this type of problem because they more accurately model the geometry. Linear tets, in many instances, only approximate the geometry and thus do not suffer the same problems. This is cause for concern as the accuracy of linear tets (as discussed in section 3) is not as good as parabolic tets. It is noted that MSC/NASTRAN SYSTEM(213) = 1 will ignore the element check but it would be short-sighted to use this as the misgivings surrounding the use of tets would be further reinforced.

If the solver fails due to the presence of 'flat' (parabolic) tets there are several options open to the analyst. Obviously the analyst can revert to a fine mesh of linear tets. Alternatively the solid can be meshed with linear tets which can then be converted to parabolic tets and, usually, solved successfully. The drawback to this option is that the geometry is not as accurately represented when meshed with linear elements but will result in more accurate results. A further alternative is to identify the failed elements and convert them to linear formulation and re-run the analysis. This mix of 4- and 10-noded tets is certainly not recommended as effectively 'cracks' are present in the model. However, when the 'flat' elements are away from any critical regions of the model this technique can be successfully employed with little detrimental effect on the overall results. This approach has been surprisingly successful.

[A note of caution, these techniques have until now been used by very experienced analysts and as such the use of them is encouraging in achieving the stated goal of rapid analysis. However the situation may change when occasional analysts start to use these techniques on a regular basis.]

Another disadvantage of using the MSC/PATRAN automesher is the lack, at the present time, of an 'octree' type mesher. Although one is available in MSC/PATRAN, it can only be used with Unigraphics CAD data. An octree mesher allows refined meshes to be generated in areas of interest whilst generating coarse meshes in other regions of the model. The advantage of this is that many elements can be placed in critical areas whilst keeping the model size down to a minimum, with a consequent saving in computer resources and an increase in the solution accuracy. A triangular surface mesh can be generated which will help produce a more refined mesh in areas of interest but it has been our experience that the solid automesher tends to fall over if the element sizes are vastly different. In addition it was felt that this was getting away from the rapid meshing that was originally desired. (It is noted that MSC/PATRAN Version 7 offers significant improvements in this area)

In spite of the above comments the MSC/PATRAN/NASTRAN system has been used successfully on a number of projects, as reported in sections 4.1, 4.2 and 4.3.

3.0 TET ELEMENT ACCURACY

Traditionally structural analysis of solid components at British Aerospace Brough has been done using 8-noded hexahedron brick elements (it is recognised that 20-noded elements are more usual but for reasons outside the scope of this paper 8-noded bricks have until now been used). Tetrahedral elements, either 4-noded or 10 noded, have a (arguably undeserved) reputation for inaccuracy. However the current state of meshing technology does not, and will not for the foreseeable future, allow automeshing with brick elements. Therefore a method by which consistently accurate solutions can be obtained with tets must be developed. It has been the authors experience that tet elements will produce perfectly acceptable results if they are used correctly.

To illustrate the accuracy of tet elements a simple lug type model, shown in figure 6, was developed in MSC/PATRAN and analysed using the MSC/NASTRAN solver. This model was analysed using shells, bricks and tets. The shell and brick mesh were developed such that, based on the authors experience, accurate results would be obtained and these models were compared with the several tet mesh models that were generated. The tet meshes were based on target element edge lengths of 1.0mm, 0.5mm and 0.25mm.

The results of this comparative exercise are shown in table 1, and indicate that tet elements will give accurate results if used correctly, a statement which may also be applied to finite element analyses involving brick elements. It is noted that parabolic tets offer superior performance over linear tets and as such are the preferred element to use. However as noted elsewhere this is not always possible.

It is also noted that the parabolic tet mesh, based on an edge length of 0.25mm, produces results approximately 2.5% greater than the brick mesh. It is believed that this result is the more accurate as it represents the geometry most accurately.

Mesh	Target element edge length	Degree of freedom	Normalised stress[1]
brick	-	17955	1.0
shell	-	2398	0.9967
tet (parabolic)	1.0	2001	0.8322
tet (parabolic)	0.5	8061	1.0103
tet (parabolic)	0.25	41664	1.0254
tet (linear)	1.0	354	0.3251
tet (linear)	0.5	1269	0.6061
tet (linear)	0.25	6111	0.8367

Table 1

[1] Normalised stress based on unaveraged Von Mises stress at node points.

4.0 ENGINEERING EXAMPLES

4.1) Avionics Bay Door Hinge Bracket

In addition to the simple lug model used to test solver accuracy, a test of the accuracy of the tet mesher was also carried out using the 'swan neck' hinge fitting. The hinge bracket is shown in figure 7, which also shows the loads and restraints that were applied for the analyses. Typical meshes (brick and tet) are shown in figures 8 and 9.

The initial analyses suggested that the 'automated' analysis was underestimating the stress levels by approximately 15%. This discrepancy was caused by the omission of the pocket fillet radii from the brick element model. This was done to simplify the modelling process and is an accepted modelling practice. When the fillet radii were removed from the MSC/NASTRAN tet model and the analysis re-run, the calculated stresses were more comparable with those generated by the brick mesh. This illustrates an important point, that a benefit of the new automated methods which include geometric features which otherwise would be omitted, may in certain circumstances give more representative results.

The hinge bracket problem was solved in approximately one hour, this being from starting with the CATIA solid geometry to actually plotting the Von Mises stresses on screen. The brick element model took approximately one and a half days to complete.

This example also serves to illustrate a current weakness with the MSC/PATRAN meshing routines in that approximately 15,500 tet elements were used in this problem because of the lack of either an octree type mesher or a suitable alternative. This meant that to achieve acceptable accuracy a fine mesh was generated in the entire solid rather than in the area of expected high stress, which for this example is easily predicted in advance. Version 6.2 of MSC/PATRAN will allow mesh seeds on solid edges and will go some way to improving the situation, although by how much remains to be seen.

Analysis Method	Von Mises Stress Mpa (with fillet)	Von Mises Stress MPa (without fillet)	Number of degrees of freedom
MSC/NASTRAN brick	-	1033	27126
MSC/NASTRAN tet	906	1017	86316

Table 2

4.2) Tailplane Skin

The second example to illustrate the use of the automatic tet mesher is that of part of a wing skin structure. For reasons outside the scope of this paper a radius had to be altered to one of three that were proposed, and analysis was deemed the best method by which the best solution could be identified. A typical CATIA solid model of the feature is shown in figure 10, which also shows the boundary conditions used during the analysis and a typical tet mesh is shown in figure 11. Three different radii were analysed using tet meshing and shell elements. The analysis using the MSC/NASTRAN shells was carried out as this is the approved method at the present time and was required for structural acceptance. In addition it would provide baseline results against which the tet element results could be compared.

It is noted that this example, that of a 'thin' solid may not be suitable for modelling with any type of solid element. However the purpose of using the new methodologies was to take advantage of existing CATIA solid geometry to reduce analysis times whilst still maintaining accuracy of results, hence the analysis was carried out to see what improvements, if any, could be gained. The number of elements thru' thickness to model bending effects was not an issue in this example as all loads were in-plane. It is realised that this could be a possible source of error if out-of-plane loads were applied.

Table 3 shows the results that were obtained for each option. The tet element results were obtained using linear tet elements as a fatal error occurred if parabolic tets were used. This error was due to the existence of a near zero angle between surfaces at a number of locations in the model. This is a not an uncommon occurrence and raises questions about the accuracy of the analysis if linear (4-noded) tets are used. Previous experience in similar situations meant that the mesh generated for these models was excessive in that approximately 80,000 elements were created for each model to ensure accuracy of the results. This does not cause problems with model size or run times as the analyses were all run on a Cray so times are measured in minutes rather than hours.

To further complicate matters the area of interest in this particular problem is made up of several curved segments which should not really be meshed using linear tets. The meshes that are obtained are shown in figures 12a and 12b and show how the 10-noded tets more accurately represent the geometry than the 4-noded tets. In this case it was not too much of a problem as a set of baseline results were available for comparison, however the mesh containing approximately 20,000 linear tets (figure 12b) produced results that were 40% in error of the expected results. This exercise re-inforces the caution that should be used when using linear tet elements.

This example was also solved using a mixture of (mainly) parabolic tets and (a few) linear tets. The number of elements in this analysis was approximately 20,000 and the stress results were better (from the point of view of comparison with shell results) than the analysis involving only linear tets. [Note the number of elements converted from 10-noded to 4-noded was approximately ten]

Analysis Method	Option 1 Stress MPa	Option 2 Stress Mpa	Option 3 Stress MPa
MSC/NASTRAN shells	64.7	46.7	55.6
MSC/NASTRAN tets	63.8	42.9	51.9

Table 3

The problem with using linear tet elements is that doubt exists in the mind of the analyst as to the accuracy or otherwise of the results. Obviously the models could be successively re-meshed with more refined meshes until a converged solution is reached but that would defeat the object, which is to obtain quick, right first time analyses. The solution to this particular problem is still the subject of much discussion, at present the only answer would appear to be to 'flood' the model with many thousands of elements which to the traditionalists is a less than elegant solution. Other options would be some kind of adaptive solution, either h-, p- or hp adaptivity, or again the availability of an octree type mesher would possibly help.

The time taken to complete all three analyses using automatic meshing was less than three hours, which is the time taken from starting with a CATIA solid through to producing stress plots of the three options, allowing a choice to be made as to which was the best option. The time taken to do the shell element analyses was two days.

Note: p-adaptivity is available in MSC/NASTRAN and was actually used successfully in the avionics bay door hinge problem. However it is not yet a mature technology and does suffer problems in several areas. In addition it is not yet fully supported by MSC/PATRAN which makes it cumbersome to use and not ideal for the occasional user.

4.3) In-Flight Refuelling Probe

The IFR casting is a more complex solid than either the hinge fitting or tailplane skin and is not readily analysed using traditional methods. Typical CATIA solids as imported into MSC/PATRAN are shown in figures 13a and 13b with a typical mesh shown in figure 14. The complexity of the component means that significant time (approx. two to three weeks) would be required to generate a brick finite element model using the MSC/PATRAN pre-processor. The potential savings of being able to analyse this type of component in days rather than weeks is obvious, assuming that the results obtained using tet elements are accurate.

The CATXPRESS method of transferring the geometric data of this component from CATIA into MSC/PATRAN suffered problems throughout that were never fully resolved. The IFR casting is a very complex geometry and as such the integrity of the geometric data was expected to cause problems, especially as several difficulties were encountered during the solid modelling process in CATIA. Typical problems were due to errors in topology as detailed in section 2 and the existence of extremely thin (sliver) surfaces. However due to the geometry tools available in MSC/PATRAN, only one version (out of ten attempted) of the IFR casting was not analysed.

Depending on the quality of the geometric data that was imported and the amount of 'cleaning' that had to be performed, a MSC/NASTRAN model was generated in usually between one and three days, the longest time being five days and the shortest half a day. Each model contained approximately 14000-24000 elements and had five loadcases applied to each analysis, typical run times being of the order 1000 cpu seconds on a Cray super computer.

The first analysis that was completed was compared with the manual calculations that had been used to originally size the IFR casting. The comparison indicated that there was broad agreement between the manual calculations and the finite element results. After this first analysis the solid geometry changed on a regular basis, essentially invalidating the original calculations. When each change was completed the geometry was transferred to MSC/PATRAN and meshed, loaded and re-analysed. These analyses, carried out relatively quickly, were of great assistance to the design process, providing confidence that as the design evolved it still satisfied the structural requirements.

The reason for so many design iterations and consequently the ten analyses being attempted was that a number of compromises had to be made to satisfy several different design requirements. These requirements came from mass, manufacturing, stress and mechanical design. The pilots who advised on the project also had an input into the positioning and length of the IFR probe which dictated the magnitude of the applied loads and the restraint conditions which changed significantly during the design cycle. However the speed with which design changes and re-analysis could be carried out in a multi-disciplinary environment was far superior to the previous situation where individual departments existed and effectively designs were 'thrown' over the fence between departments for comment and analysis.

The target mass for this particular component was 8.10kg, the initial design being over 10kg. The final design had a mass of 7.95kg and due to the analyses that had been completed was thought to be structurally acceptable. The testing that is due to be carried out on this component will show whether the analysis using automatic tet meshing is valid or not!

SUMMARY

Automatic tet meshing is gaining acceptance as a valuable tool in the overall design process. It has been used successfully on a number of occasions when alternative methods would have been prohibitively expensive in time and cost. In several instances it has been the only viable method of analysis other than a build and test approach. In addition the ability to include detailed features has led to more representative analyses being carried out.

Initial feedback from the concurrent engineering teams suggest that the new methodologies being adopted are significantly improving the design process. The turn around time for several analyses that have been carried out have been very impressive. This has led to more 'what-if' type questions being asked as engineers begin to realise the benefits of being able to perform quick analyses of different design solutions.

Despite the successes that have been achieved to date there are still several improvements than can be made by MSC. As previously mentioned an octree type mesher (or equivalent) would be desirable. Improvements to the MSC/PATRAN/NASTRAN interface to enable p-element type input decks to be generated without the present need for manual editing of the input file would also be welcomed. This would be an alternative to an octree type mesher if a converged solution could be almost guaranteed with the use of some kind of mesh adaptivity. Further improvements to the meshing routines would also be welcomed, especially in the areas of speed and the quality of elements produced. A method of identifying, at the earliest opportunity, areas where elements with acute angles cause failure of the solver and a solution to this problem would also be desirable, possibly with some kind of special element which is a solution favoured by at least one of MSC's competitors.

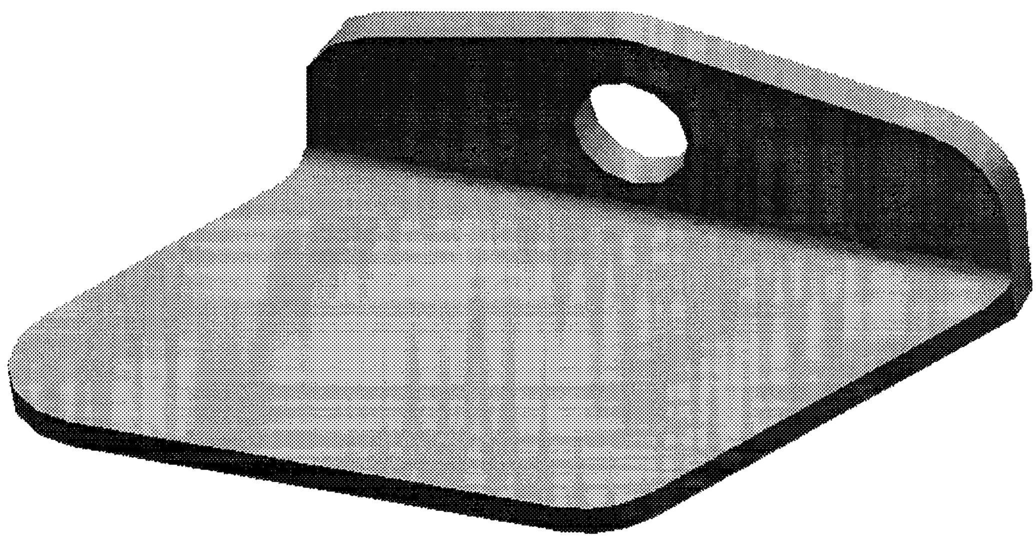

Figure 1 - simple bracket

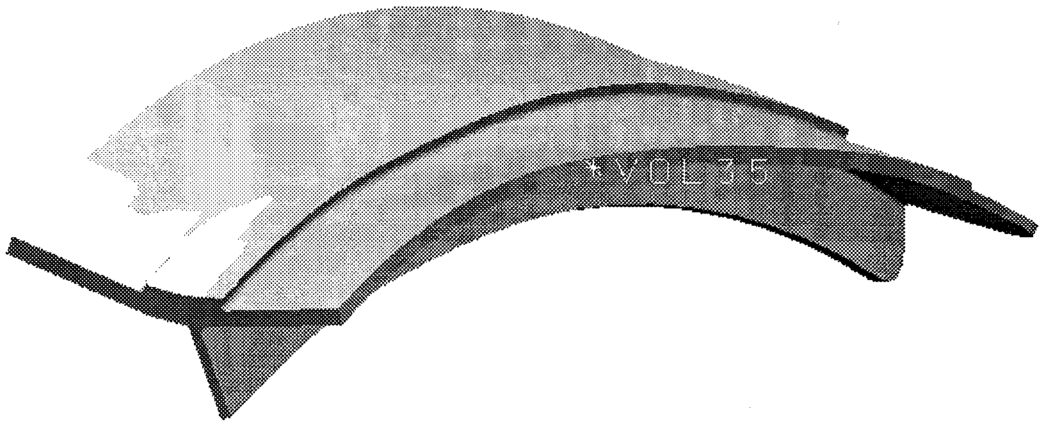

Figure 2 - curved 'I' beam

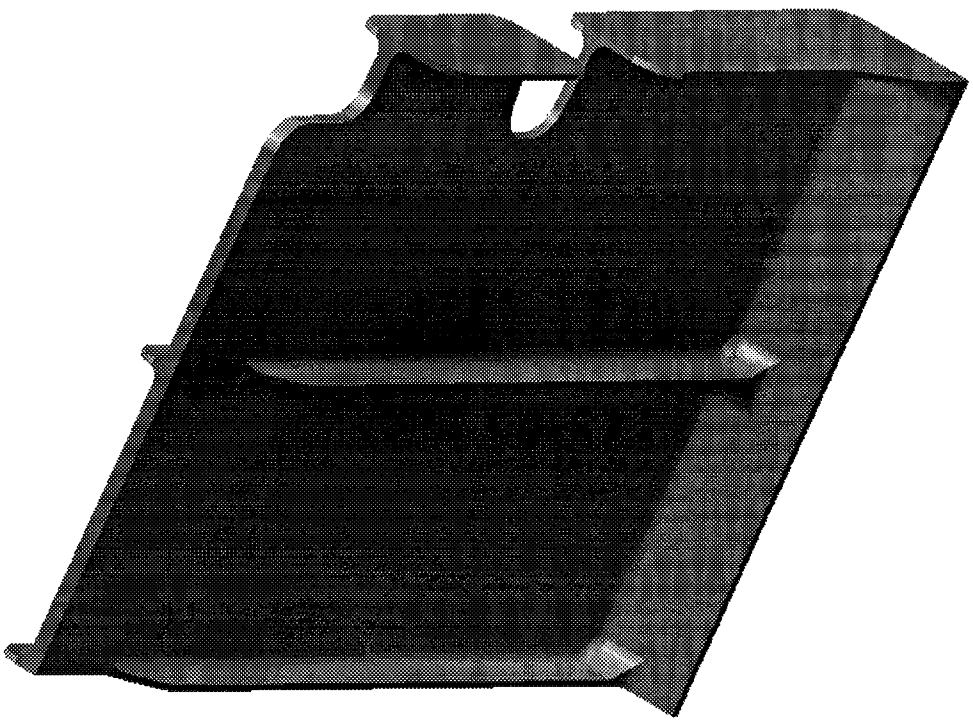

Figure 3 - shear web

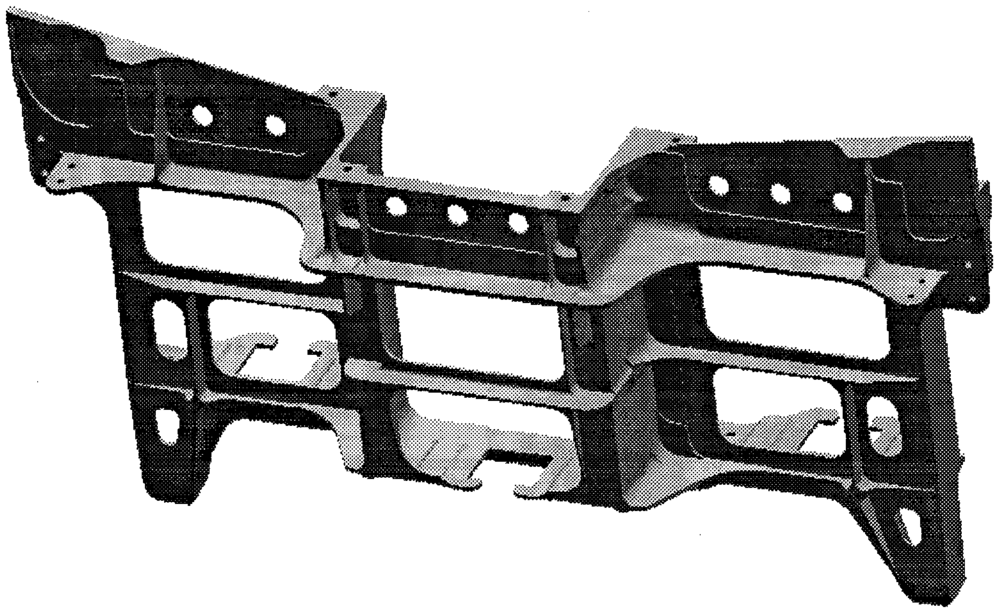

Figure 4 - equipment tray

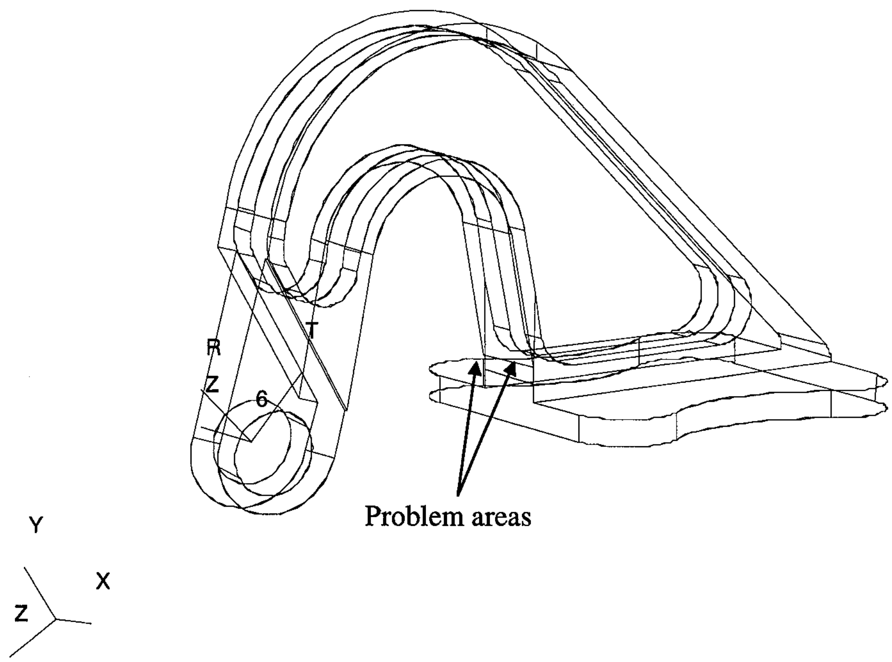

Figure 5 - CATIA geometry imported to MSC/PATRAN showing near zero angles between geometric entities

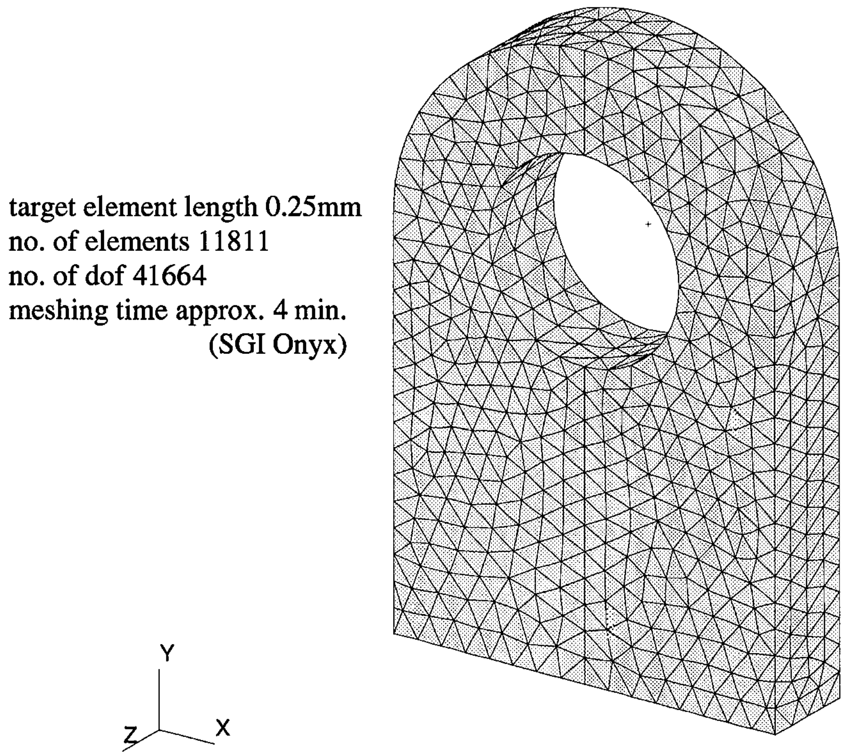

Figure 6 - typical mesh on lug component used for tet element accuracy tests

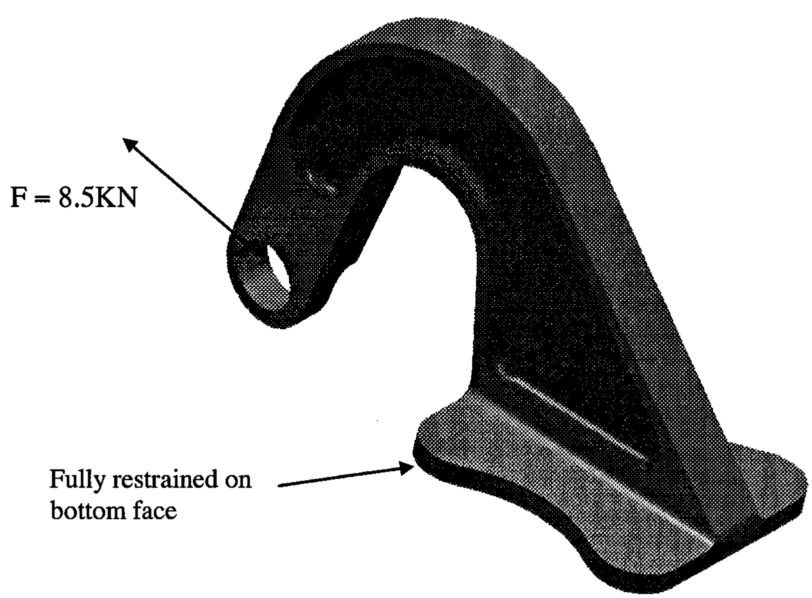

Figure 7 - Avionic bay door hinge bracket - CATIA solid

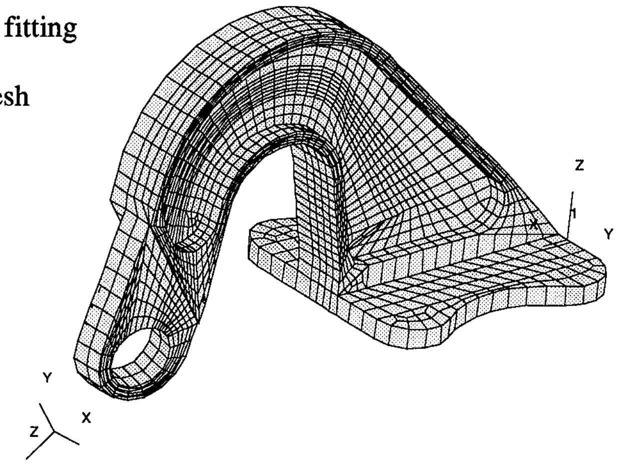

Figure 8 - Avionic bay door hinge bracket

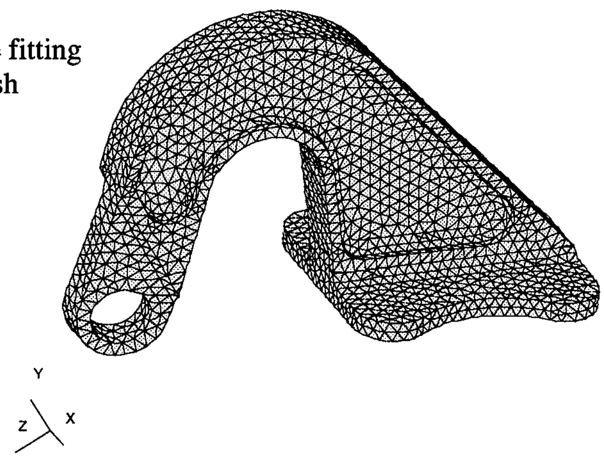

Figure 9 - Avionic bay door hinge bracket

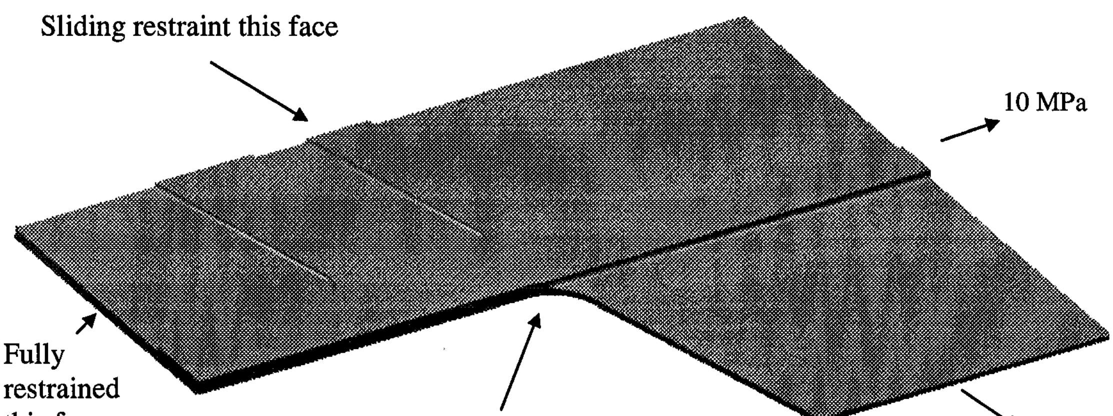

Figure 10 - Tailplane skin

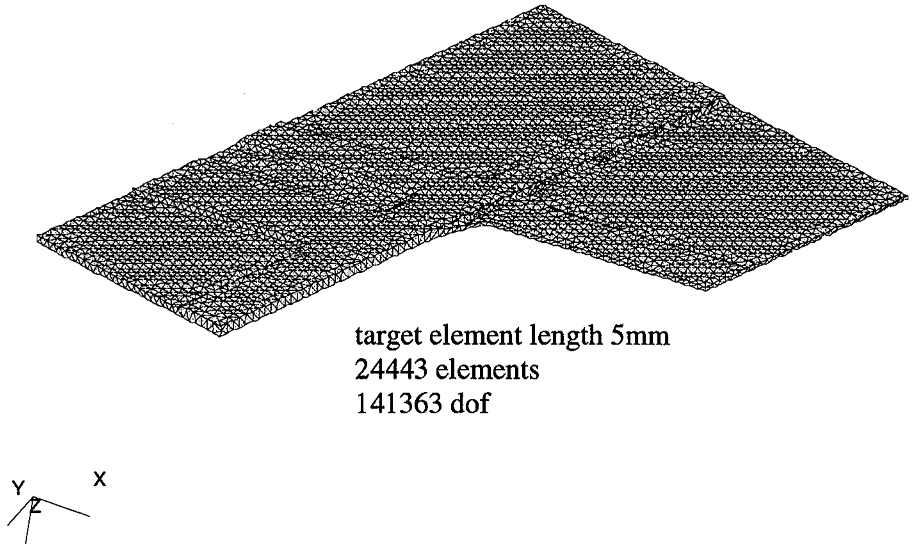

Figure 11 - Tailplane skin finite element mesh

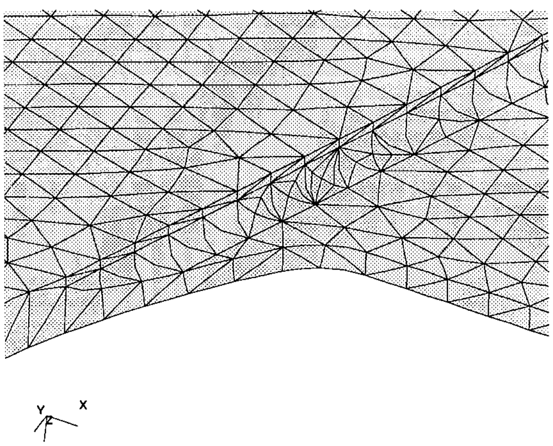

Figure 12a - 10-noded tets

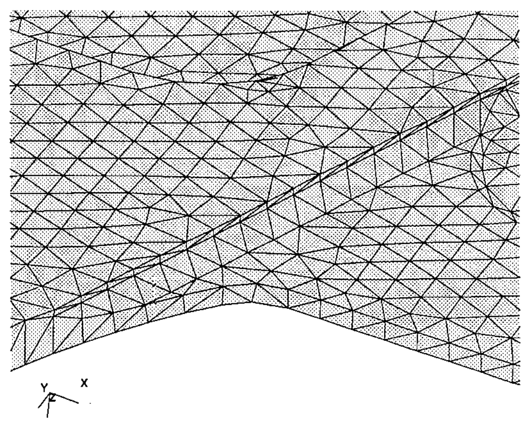

Figure 12b - 4-noded tets

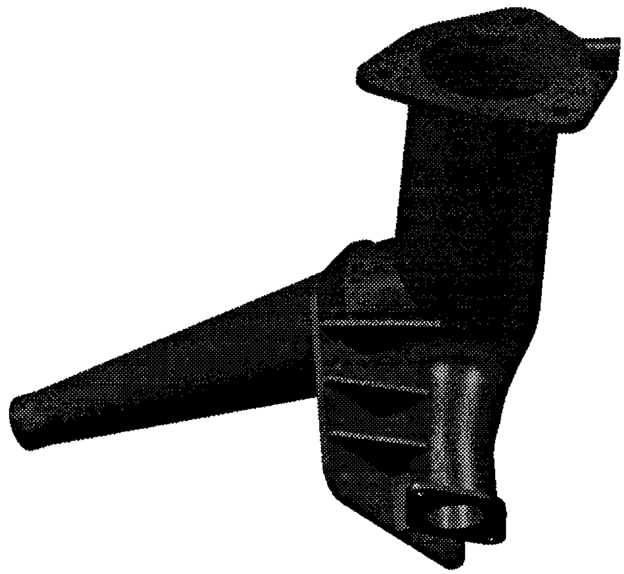

Figure 13a - IFR Probe casting

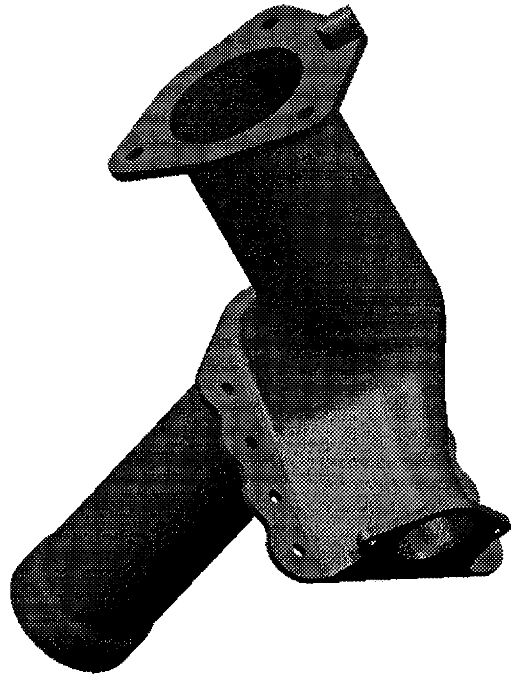

Figure 13b - IFR Probe casting

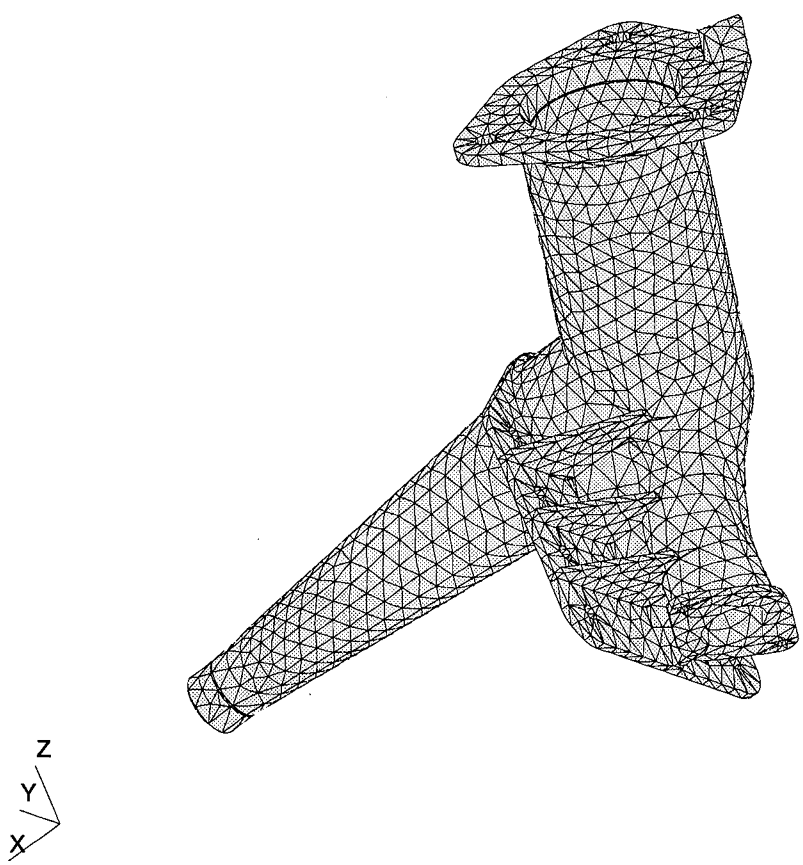

Figure 14 - IFR Probe casting