RAL
DESIGN & DISCOVERY
Open Days July 1990
RUTHERFORD APPLETON LABORATORY
SCIENCE AND ENGINEERING RESEARCH COUNCIL

STRUCTURAL ANALYSIS

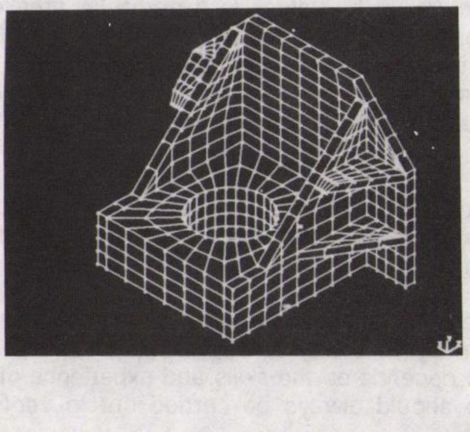
Introduction

The structural design of an instrument intended to be used in space has many constraints placed upon it. With the high cost of launching payloads perhaps the most severe constraint is one of mass. Therefore when designing a structure it has to be optimised such that it is able to withstand the loads imposed by launch as well as meet the dynamic requirements, without too much redundant mass. An over conservative structure will take up valuable mass which could be used for the scientific payload.

To assist in the design and optimisation of space structures a technique known as finite element analysis is commonly used. Here in the Space Science Department at RAL this technique is used extensively and to reduce the time required for this type of analysis, sophisticated software is available.

Finite Element Analysis

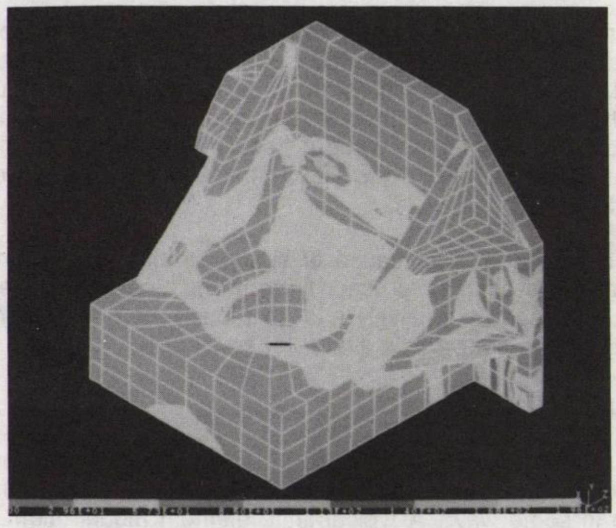
Finite element analysis is a method of analysing structures by producing a mathematical model, which can then be used to predict how a structure might behave when acted upon in some way. A typical finite element model might look something like that shown in the picture below.



The structure is divided up into a number of elements (a process known as meshing or discretisation), where each element defines the stiffness of the structure at that point and is dependent on the geometry at that location and on the material used for the structure. In this way the stiffness of the whole structure is described mathematically and can be stored in a computer.

This mathematical model can have representative loads and boundary conditions applied and may then be solved for the required unknown parameters. Typically a designer would be interested in the deflections of a structure under a load, the stresses produced and the natural (resonant) frequency of the structure.

For each element in the model these required values are calculated. A typical model could contain several thousand elements, so a large amount of data results from this type of analysis. To assess these data quickly and accurately, we employ a software package called I-DEAS™. This presents the data in a visual format; the picture below shows some typical output from this software.



The stress contour plot (shown above) is a very effective way of determining high stress areas in a structure. If the values in a particular area exceed a certain level, then a failure might occur when the structure is in service. However the designer, on seeing this sort of information would be able to modify his design to increase the strength in that area and so prevent any failure occurring. Similarly if the structure has areas of low stress, it may be possible to remove some material and hence lighten the structure.

In a similar manner deflections, resonant frequencies and mode shapes(the distorted shape of the structure at the resonant frequency) are needed to determine whether the dynamic performance of the structure will meet the specification.

All of this work can be done without having to manufacture any part of the structure. However this type of analysis is only an approximation to the real hardware, and the accuracy of the results depends on the skills and experience of the analyst. Also tests on the final hardware should always be carried out to verify the results from the finite element analysis.

For further information contact : Nigel Morris Tel: (0235) 821900 ext 5210

SERC TWENTY FIFTH ANNIVERSARY 1965-1990