An Interactive Computer Aided Design System for Cut-outs in Pressurized Aircraft Fuselages

M.E. Heerschap

Delft University of Technology, Faculty of Aerospace Engineering,
P.O. Box 5058, 2600 GB Delft, The Netherlands

ABSTRACT

Cut-outs in pressurized aircraft fuselages are very sensitive to fatigue. This explains the need for a design tool to enable the designer to perform a comprehensive design of these "difficult" structures. A menu-driven, highly interactive system for the design of the reinforcement around a cut-out in a pressurized fuselage is presented. The design system is set up in such a way that maximum use is made of the combination of MSC/NASTRAN and MSC/PATRAN. There are six basic options offered by the system: 1) fast, easy initial model generation; 2) fully interactive, user-friendly model editing; 3) preparation for both geometrically linear and geometrically non-linear finite element calculations; 4) preparation for a sensitivity analysis and graphical display of the results of such an analysis; 5) carrying out "what-if" studies; 6) preparing and performing a design optimization.

The system is set up as a special-purpose design tool. This makes a considerable improvement in the design process, because the time needed for the modelling of the complex structure is greatly reduced. The design system is based on the pre- and post-processor MSC/PATRAN. The necessary software is written almost entirely in MSC/PATRAN Command Language (PCL) which implies that the generated code can be run "inside" MSC/PATRAN. Maximum use is made of the graphical capability of this software package. The finite element calculations, including sensitivity analysis and optimization, are performed with MSC/NASTRAN.

List of symbols

A	cross-sectional area
$[K]$	stiffness matrix
$\underline{P}$	force vector
r_i	structural response number i
$\bar{r}_i$	allowable structural response number i
t	thickness
$\underline{U}$	displacement vector
$\underline{x}$	design variable vector
x_j	design variable number j
Ψ_i	structural response of constraint number i
Δ	small change

1 Introduction: The Structural Design Problem

Cut-outs are inevitable in all aircraft fuselages. In particular large cut-outs in the pressure cabin, like those for doors, remain very fatigue sensitive. This is due not only to the fact that the cut-out causes a large stress concentration, but is also due to the use of the door in service. This considerably increases the chance of accidental damage. When a small dent or other damage has occurred, a fatigue crack is only a matter of time. Fatigue damage can endanger safety, or at the very least lead to expensive repairs. For these reasons stress levels must be kept below carefully defined maximum values. In spite of the addition of all kinds of reinforcement, full-scale fatigue tests often demonstrate fatigue cracking at the corner of a cut-out. This is partly due to the fact that the methods of structural analysis used in the (initial) design of the cut-out may not be very accurate, because of the assumptions necessary to perform those calculations by hand [1, 2, 3]. When a more detailed finite element calculation can be performed, special attention must be paid to the area near the corner of the cut-out because of the very high stress gradient at that location. Furthermore, it is common practice in aircraft manufacture that some aircraft must be delivered with modifications to suit customers' wishes. One of the structural items most frequently modified are the dimensions of the door, and its position in the fuselage. This kind of modification must, in general, be designed in a very short period of time. It is clear that there exists a need for a design system which offers a quick design cycle, as well as an accurate calculation of the stress levels in the structure.

The fuselage is almost invariably built up as a stiffened shell. The stiffening elements are frames in the circumferential direction and stringers in the axial direction. The typical structure of an aircraft fuselage with cut-out is drawn in figure 1 (taken from [1]). Near the cut-out the frames are being heavier than in the undisturbed part of the fuselage. These frames, besides being axially loaded due to the hoop stress, are also highly loaded in bending due to the presence of the cut-out and have to react concentrated forces on a number of doorstops connected to this frame as well. The eccentricity of the door stops, with respect to the shear centre of the frame, causes a twisting moment. Because the frame itself generally has an open cross-section of limited torsional stiffness, so-called intercostals are placed between the frame next to the door and the adjacent frame. These intercostals are beams with high bending stiffness and carry the moments caused by the eccentricity of the loads on the doorstops. The stringers above and below the cut-out are replaced by beams with high bending stiffness, the upper and lower sills. All reinforcing members have gradually varying cross-sections for a proper load transfer. The skin around the cut-out is loaded in biaxial tension due to pressurization, carries large shear loads due to redirection of loads around the cut-out, and is locally subject to high bending stresses also due to the presence of the cut-out. The skin around the cut-out, especially near the corners of the cut-out, is usually thickened to several times the thickness of the skin in the undisturbed part of the fuselage.

Finite element methods are widely accepted as analysis tool for structural design. The design procedure is normally based on (a series of) finite element calculations, with resizing until the desired goal is achieved. In its simplest form the design procedure consists of adding material in the highly stressed regions until all stresses are below a certain acceptable level. When material is added the assumption is generally made that the structure behaves like a statically determinate one. This implies that the stress is inversely proportional to the cross-sectional area or thickness. Furthermore, it is assumed that a change in a structural member affects only the member itself. It will be clear that these assumptions do not hold for a statically indeterminate structure such as a fuselage, especially around cut-outs. The redesign of the structure could be greatly improved if so-called sensitivity data were available. These data describe the change in structural response (stress, strain, displacement, etc.) in the complete structure due to a change in one of the properties of a particular structural member. Design sensitivity data provides a very useful way to gain insight into the behaviour of the structure. According to Santos et al. [4, 5], these data provide valuable information about the design variables that are most critical to some constraint, and can supply information concerning the constraints which are most affected by a particular design variable. Using this sensitivity data the designer can make a more accurate prediction of the effect of structural changes. To explain the usefulness of sensitivity data, it is obvious that adding material will reduce the stress locally, but due to the changed internal load distribution stresses might be increased in other parts of the structure. Prediction of the structural response in this way greatly improves the efficiency of the design procedure, because far less design iterations are needed. The calculated sensitivity data is also the basis for "what-if" studies, which can be an excellent tool for the "fine-tuning" of a design. Furthermore, sensitivity data is required for a first-order gradient based optimization, as used in MSC/NASTRAN [6]. Optimization employs the same sensitivity data and steers the design towards a certain objective while satisfying the chosen constraints. The task of gradually improving the design is now performed by the computer instead of by the designer, in this way saving much time. Furthermore, the optimization needs typically less iterations than an "optimization by hand", since it computes the maximum improvement of the design in each iteration.

1.1 Computer Assisted Design

Most finite element codes offer the means to perform both geometrically linear and non-linear calculations. Some also offer options to perform structural optimization and/or sensitivity analysis. The "ordinary" finite element calculations are nowadays well accepted in industry, but the use of sensitivity analysis and optimization is still treated with some suspicion in spite of the potential advantages stated in the previous section. Partly this is due to lack of accurate loading information in an early stage of the design, but also due to lack of proper means of pre- and post-processing the data generated by a sensitivity analysis or optimization. The first problem cannot be solved here but, because the load distribution and the design are to some extent related to each other, a fast design method will assist in a fast determination of the loading as well. The problem of a suitable means to post-process the data generated by a sensitivity analysis or optimization is solved by the design system proposed here. In particular, post-processing of a sensitivity analysis requires a well considered, graphical presentation because of the huge amount of data generated. The same could be said for structural optimization but here is also the danger of arriving at a local optimum, or even no convergence at all, both at the expense of a large amount of computing time. This has largely prevented its acceptance by industry. Still, it should be noted that a properly used optimization makes maximum use of the generated sensitivity data and in general arrives at a better design, or at least it will reach an optimum faster, than can be achieved by "optimization by hand".

In this paper a highly interactive design system is proposed, offering the means of performing finite element calculation, sensitivity analysis and structural optimization in an easy, user-friendly way. The system provides several means to post-process sensitivity data, including use of a graphical display of these data, and "what-if" studies. The finite element model for the analysis is easily generated and preprocessed by the design system. To achieve the goals of the design system, a menu-driven, graphical environment is used. Maximum use is made of commercially available software. Furthermore, it should be noted that all software written for use with this design system is in addition to the available software. That means that no modification of the source code of the commercial software is needed; the design system is intended to make the best use of the combination MSC/PATRAN and MSC/NASTRAN. Most of the extra computer code is written in PCL, which is a structured programming language available in MSC/PATRAN [7]. The main reason for programming in PCL is to make use of the graphical user interface in MSC/PATRAN for the design system. Furthermore MSC/PATRAN offers the possibility of creating a user-defined menu, which is a powerful means of creating a user-friendly design system.

1.2 Illustrative Example

A specific example is presented throughout this paper and discussed in detail, to illustrate the use of the program. Furthermore this example shows the potential offered by an integrated design system. The example is a circular fuselage of radius 1400 mm, subject to an internal pressure of 0.05 MPa. The frame pitch is 500 mm and the stringer pitch 150 mm. The cut-out for the door is 1300 mm high (measured around the contour of the fuselage) and 800 mm wide. The corner radius of the corner of the cut-out is 100 mm. The maximum stresses are limited to a major principal stress of 90 MPa and to a shear stress of 70 MPa, which are typical values for fatigue stress levels in a pressurized fuselage. The fuselage is initially dimensioned according to Niu [1]. This sizing method assumes that the skin carries only shear stress, no normal stress. Furthermore the corner radius of the cut-out is not taken into account in this initial design. The reason for the maximum allowable shear stress requirement is primarily to have a design constraint for the plate thicknesses. The result of this preliminary sizing is shown in figure 2. Figures 3a and 3b show the finite element calculation of the principal stress in the initial design. Figure 3a shows that the principal stress at the outer surface of the shell, at the corner of the cut-out, reaches 285 MPa while only 90 MPa is allowed. Figure 3b shows the shear stress in the fuselage. In the area where the frame next to the cut-out changes in cross-sectional area, the peak shear stress of 68.8 MPa is just below the maximum allowable shear stress of 70 MPa. It can be concluded that a simple sizing by hand such as by the methods of [1, 2, 3] does not guarantee a design which properly meets the requirements.

2 General System Description

In this section the overall program capability as defined in figure 4 is discussed. The figure shows a condensed user-defined menu system; the complete menu system can be found in the User's Manual "Cufus" [8]. Menus are presented in the most right-hand column of the MSC/PATRAN-window, as shown in various figures, 7, 8, 9a and 9b. Roughly speaking the design session will consist of two main phases, first the finite element modelling phase (pre-processing) and second the post-processing phase. The finite element modelling phase itself can be divided into several stages, as shown in figure 4. These stages are: 1) initial finite element modelling; 2) model editing (both geometry and mesh adaption); 3) property assignment (including definition of design variables and constraints); 4) application of boundary conditions and loads; 5) preparation of MSC/NASTRAN input files. These different stages will be discussed in the following sections.

As well as the regular types of post-processing, not shown in the special user-defined menu (figure 4) because they are covered in the standard MSC/PATRAN menu [9], the post-processing phase has a number of extensions for post-processing of a structural optimization and sensitivity analysis. These additional types of post-processing are also discussed in more detail in the following sections.

2.1 Finite Element Model Generation

For the finite element model generation two separate stages should be recognized. First there is an initial finite model generation, which is only slightly interactive, and secondly a finite element model editing phase which is highly interactive. The initial model generation is done by use of a small number of key dimensions of the structure, as shown in figure 5. These dimensions refer only to the general geometry of the model. The dimensions used at this stage can be entered via a file or interactively. Thicknesses and other element properties are assigned at the model editing stage. The limitation on the number of key dimensions also implies that there are limitations on the finite element model that can be generated. First of all the cross-section of the fuselage can be single or double circular. This means that both circular and "double-bubble" fuselages can be generated (see figure 6). Furthermore there are no other disturbances (such as windows) except for a floor with struts and/or longitudinal floor beams, in the direct neighbourhood of the cut-out. However, it should be stated that this last limitation can be overcome by use of some "tricks". These limitations arise from the specification for the design system, and are therefore not fundamental limitations but rather arbitrary.

MSC/PATRAN makes a geometry model and a finite element model generated on top of the geometry model. The geometric entities created by MSC/PATRAN which are important for the finite element model are patches (surface definitions) and lines. Aa single patch is placed between two stringers and two frames. Lines are generated along the frames. The mesh of shell elements is generated on each patch separately. The mesh over each patch is uniform, sometimes with the exception of the boundary of the patch where a small transition region can be present. This also limits the possibilities for mesh adaption. Stringers are generated on two opposite edges of the patch. Beam elements for the frames are generated along the previous mentioned lines.

The mesh density of the initial model is fixed. The mesh is based on "reasonable" dimensions of a cut-out in a fuselage. The mesh in this initial stage is quite coarse, but can be adapted very easily during model editing. Based on the dimensions of the fuselage in the example (section 1.2) the initial mesh is generated as shown in figure 7. The generation of this finite element model took about 6 minutes and 30 seconds (see also table 1). During the initial modelling some extra data concerning mesh density and geometric properties is saved in a local database. This local database is necessary to speed up the interactive model editing. This is done by storing a part of the standard MSC/PATRAN database in the local database, stored in the core memory of the computer at the start of each design session, eliminating the need for time consuming disk access.

2.2 Interactive Model Editing

Much attention is paid to interactive model editing, because this consumes a great deal of user time. This is also the reason that user-friendliness is emphasized at this stage of the design cycle. User-friendliness also implies that the system response must be quick. Especially for interactive work, long response times can be a source of irritation. Model editing usually starts with mesh adaption. The menu system is constructed in such way that all options, such as mesh adaption, are placed into a logical, sequential order. The user, by simply clicking a patch, can double or halve the mesh density in different directions. The program takes care of mesh transition regions. The result of this interactive mesh adaption is shown in figure 8, for the same fuselage example. The only possibility for mesh adaption is to double or to halve the mesh density. This can be a disadvantage in regions of the finite element model where only rather modest mesh adaption is required. However, due to the type of structure analysed here, this limitation on mesh adaption is not considered to be a serious disadvantage because it enables the user to create in an easy way wide variations in mesh density. At the stage of interactive model editing it is also easy to re-dimension the geometry of the cut-out within certain limits. This re-dimensioning requires a partial regeneration of the geometry and the finite element model. The variation in mesh density is still present after this regeneration, but property data and data concerning the design variables and constraints must be assigned again.

2.3 Property Assignment

When the mesh adaption is completed, properties can be assigned. This can be done in several ways. Property data can be chosen from a predefined property database, or in some cases from internally available properties. These internally available properties consist of a number of standard beam cross-sections. By clicking through some menus a beam property is generated (see figures 9a and 9b). Properties can be assigned to all elements which lie on geometric entities, like patches or lines, or to elements directly. Furthermore some typical structural members, like frames and stringers, can be used for property assignment. In this case the user clicks on a member of the frame or stringer and the program assigns those properties to all elements belonging to that frame or stringer. Combinations of property assignments, e.g. a few patches in combination with a number of elements, can also be used. The assigned properties can be visualized in a colour plot.

If the only goal is to perform a static finite element calculation the interactive modelling is now finished, but if the goal is to perform a sensitivity analysis or optimization it is necessary also to define design variables and constraints. The definition of design variables is very similar to the property assignment; the property data is copied to a new, reserved entry which will be used as a design variable. The part of the program which prepares the MSC/NASTRAN input files scans the MSC/PATRAN data base for these reserved entries and uses data from these entries for construction of design variables instead of for regular property entries. The constraints defined by the user are normally active for both optimization and sensitivity analysis. However, the definition of constraints for both types of analysis does not completely match. A number of parameters needed for optimization are unnecessary for a sensitivity analysis and vice-versa. The design system will guard against incomplete data for the two types of analysis. To use the constraints for both optimization and sensitivity analysis, care should be taken to make sure that the necessary data is entered for both. Stress and strain constraints are defaults applying to the whole model. Constraints can be further defined by the menu for forces, stresses, strains and displacements. The menu also offers the possibility to choose another structural less commonly used response for the constraints by directly entering a response code for the MSC/NASTRAN DCONSTR-card (optimization) or DSCONS-card (sensitivity analysis) [10].

2.4 Preparation of MSC/NASTRAN Input Files

The translator between MSC/PATRAN and MSC/NASTRAN [11] does not have the capability to transfer optimization and sensitivity analysis data. Only a *Bulk Data Deck* is generated. Therefore the design system must generate the necessary parts of the input not provided by the standard translator, such as a *Case Control Deck*. If a *Case Control Deck* already exists, the translator can include it in the MSC/NASTRAN input file. For a sensitivity analysis the extension of the standard input deck includes a very short so-called *rf-alter*, which causes MSC/NASTRAN to write the calculated sensitivity data both to the standard output files and to a separate *punch* file, which is used for post-processing. The necessary extensions to the *Bulk Data Deck* are generated by design system Cufus. If a small program is started then the output of the MSC/PATRAN translator and the additional are merged into one MSC/NASTRAN input file.

3 Post-processing

Several aspects of post-processing can be distinguished. First of all the post-processing of a static (non-)linear finite element calculation, which consists of the regular post-processing such as contour plotting of stresses and the displacements. Secondly, post-processing of sensitivity data, including a "what-if" analysis, and thirdly post-processing of optimization data. The first option, post-processing of regular data, will not be discussed further because only standard options of MSC/PATRAN are used. This is discussed thoroughly in the user's manual [9]. The second and third options are dealt with in separate sections here. Discussion of these types of post-processing will again be done on the basis of the previously mentioned fuselage example.

3.1 Post-processing of Sensitivity Data

In using sensitivity analysis and optimization a huge amount of output is generated. The usefulness of this output depends mainly on how well the engineer can process the data. Therefore it is considered necessary to exploit the graphical capability of MSC/PATRAN as much as possible. This way of presenting results can be extremely helpful in finding those structural elements that have the greatest influence on the behaviour of the structure.

Besides regular graphical post-processing of data, such as contour plotting of stresses and displacements, graphical post-processing of the sensitivity analysis is added. To enable this kind of post-processing, the sensitivity data is translated to a file with a similar format to the stress result files. The standard MSC/NASTRAN sensitivity coefficient is:

$$\frac{\Delta\Psi_i}{\left(\frac{\Delta x_j}{x_j}\right)} = x_i \frac{\partial\Psi_i}{\partial x_j} \qquad (1)$$

Here Ψ_i is the structural response of constraint number i, and x_j is design variable number j. The absolute change of the structural response of the constraint is related to the relative change in the design variable. Sensitivity data can be presented in a number of different ways, which implies that the sensitivity data are normalized in different ways. Other user-defined normalizations are prepared but not yet implemented.

The sensitivity is calculated here for the example referred to in previous sections. The design variables for the skin and frames are defined in figure 10. The design variables for the skin are the thicknesses of the shell elements, and for the frame and sill are the cross-sectional areas of the beam elements. As grey-scale imported graphics is hard to read, the figure is reproduced using hatching instead of grey-scale. Figure 11 displays the change in major principal stress distribution when design variable number 2 is changed. The major principal stress displayed is the stress at one of the surfaces of the shell elements, and the sensitivity coefficient is chosen to be the standard MSC/NASTRAN sensitivity coefficient definition (equation 1). The particular design variable (number 7 in figure 10) is chosen because it emphasizes the use of sensitivity analysis. The change in the design variable shows that in the whole region of the variable itself the stress is decreased due to increase of thickness, as expected. The figure shows a considerable decrease in some parts of the finite element model of maximum 71.6 MPa when the thickness is doubled. The area in the corner of the cut-out, shows a *decrease* of 12 MPa due to the doubled thickness of the design variable 7, even though this design variable is a considerable distance away from the corner of the cut-out.

In general it can be stated that in a structure such as a cut-out in a pressurized fuselage most changes in structural members result in considerable changes in stress in other parts of the structure. This implies that "normal" re-design of a structure, consisting of some kind of fully-stressed design cycle on the basis of an independent behaviour of the design variables, will result in a very slowly convergent design process.

3.2 Post-processing of a Structural Optimization

Post-processing of an optimization task can be divided into three parts. First the regular post-processing, such as visualizing stresses and displacements. Secondly, display of the last computed sensitivity data, needed for the optimization. Thirdly, display of the history of the design variables, constraints and objective function during optimization, in two-dimensional plots. The first two parts need no further discussion, because the post-processing of sensitivity data is similar to the post-processing of the results of a sensitivity analysis, and post-processing of regular data is described in detail in the MSC/PATRAN user's manual [9]. Display of the optimization history of design variables, constraints and objective function as x-y plots are made by P/PLOT, which is a separate program in MSC/PATRAN. Because P/PLOT does not have the same customizing capability as MSC/PATRAN, it is not possible to make an user-defined menu in P/PLOT. Display of x-y plots is started as a child process out of MSC/PATRAN; after displaying the plots, it returns automatically back to MSC/PATRAN. The design system takes care of the data recovery from MSC/NASTRAN and preparation for P/PLOT.

The main reason for displaying optimization history data is that it enables the engineer to make a quick and easy check on the optimization. It is possible that the optimizer has stopped prematurely, or even stops before reaching a feasible design. To check for these (more or less common) errors, x-y plots can be drawn for the design variables, objective function and maximum constraint violation. Plots for design variables are made separately for different groups of variables, i.e. thickness design variables in one group, cross-sectional area variables in another group. An example of these plots is shown in the figure 12. The optimization for which these plots are generated is performed for a quarter model of fuselage under internal pressure. The constraints used keep both the maximum tensile principal stress and the maximum compressive principal stress below 90 MPa, while the maximum shear stress is limited to 70 MPa. These constraints act on both surfaces of the shell elements, which implies that bending is taken into account. For the beam elements, constraints are defined to limit both the maximum tensile stress and the maximum compressive stress also to 90 MPa. The design variables are the same as shown in figure 10. The design variables for the skin elements are subject to a lower limit of 0.8 mm and a upper limit of 14 mm. As can be seen in figure 12 the initial design is infeasible. The constraint violation at the beginning of the analysis is around 1.12, where constraints are defined as:

$$\psi_j(\underline{x}) = \frac{\bar{r}_j - r_j(\underline{x})}{|\bar{r}_j|} \tag{2}$$

A constraint violation of 1.12 implies that the maximum allowable stress is exceeded by approximately 112%. Figure 12 shows that at the beginning of the optimization some weight is added to cure the constraint violation. It can be seen that the maximum constraint violation rapidly decreases to approximately zero. This figure also shows clearly that the initial design variables have changed considerably during optimization. This emphasizes the need for structural optimization. The figure also shows that some of the design variables reach their lower limits. This is an indication that some extra weight saving could be realized if the limits on those variables were relaxed. It can happen that the optimizer arrives at a local optimum. To avoid this, it is common practice to repeat the optimization a number of times, starting from different initial designs. If the optimizer reaches the same optimum a few times, it is likely that the optimum found is the right one. The design system also offers a way to visualize the results of a few different optimizations. The objective functions of three optimizations, starting from different initial designs but subjected to the same constraints, are displayed in figure 13. From this figure it is clear that the objective function in optimizations 2 and 3 reaches approximately the same value. Optimization 1 terminates at a prematurely, however. This example emphasizes the need to perform an optimization several times with different initial values for the design variables. The premature termination of optimization 1 is caused by difficulty with the finite difference step. This is discussed in more detail in section 4.

3.3 Performing a “What-if’ Analysis

A “what-if” analysis can probably best be described as a hand driven optimization. The user can prescribe a change in one of the design variables and the program predicts the new stress distribution due to that changed design variable. The prediction is made as an extrapolation of the last finite element calculation and the sensitivity data. The sensitivity analysis provides only the first derivatives of the structural responses with respect to a change in design variable. Therefore it will be clear that the change in a design variable during a “what-if” analysis must remain small, because extrapolation of the structural responses is linear while the structural response itself is in general not linear. The prediction is calculated using:

$$\underline{U} + \Delta\underline{U} = \underline{U} + \frac{\partial \underline{U}}{\partial x_i}\Delta x_i \qquad (3)$$

The “what-if” analysis is also intended as an intelligent round-off routine after an optimization. The structural optimization in MSC/NASTRAN is unfortunately not capable of performing an optimization using discrete values for the design variables, where the designer is normally restricted to the use of structural members with standard dimensions (e.g. standard sheet thicknesses). The “what-if” analysis shows the changed response of the structure without starting a new finite element calculation. Furthermore the “what-if” analysis can also be used as an extension of the post-processing of the sensitivity data. In this light it can be seen as a tool to gain insight into the behaviour of a structure subject to specific changes in design variables. The design system offers the opportunity to update properties during a “what-if” analysis. This means that changes made in the design during such an analysis can be made permanent, so it is quite easy to start a new finite element analysis using the updated properties. It is always advisable to perform an extra analysis subsequent to a “what-if” analysis, to ensure that the errors made due to linearization of the structural response are within reasonable limits.

Figure 14 shows the start of a “what-if” analysis. In this example design variable 1 (see figure 10) is changed from 1.3 to 1.4 mm and subsequently the value of design variable 2 is changed from 2.7 to 3.0 mm, i.e. changes of about 7.7% and 11% respectively. The design system predicts the stresses after both changes, as shown in figure 15. To check the error due to linearization of the structural response a new finite element calculation is performed for the updated design. The results of this calculation are shown in figure 16. It is clear that the differences between figures 15 and 16 are quite small, i.e. the error of the *maximum* stress is less than 1%, indicating that the behaviour of the structure is sufficiently accurately approximated by a linear extrapolation of the response for a change in the design variable of 7.7% and 11%. To limit errors due to linear extrapolation, the change in the variables should generally be limited to less than 20% as rule of thumb. The time needed for a structural modification during the what-if analysis is typically between 10 and 15 seconds. It should be noted the major part of this time is used by displaying the graphics, which implies that the time of 10 to 15 seconds depends heavily on the available graphics hardware.

4 Experiences with the MSC/NASTRAN Optimizer

The optimizer included in MSC/NASTRAN is generally very effective. It requires only a limited number of iterations to reach an optimum solution. However, some attention should be paid to the way the sensitivity data for the optimizer is calculated. To calculate sensitivity data, the following equation has to be solved:

$$[K]\frac{\partial \underline{U}}{\partial x_i} = \frac{\partial \underline{P}}{\partial x_i} - \frac{\partial}{\partial x_i}[K]\,\underline{U} \tag{4}$$

The derivatives $\frac{\partial \underline{P}}{\partial x_i}$ and $\frac{\partial}{\partial x_i}[K]$ must be determined before equation 4 can be solved. These derivatives are approximated by the first forward finite differences [6]:

$$\frac{\partial}{\partial x_i}[K] \approx \frac{[K]\,(\underline{x}+\Delta x_i) - [K]\,(\underline{x})}{\Delta x_i} \tag{5}$$

$$\frac{\partial \underline{P}}{\partial x_i} \approx \frac{P(\underline{x}+\Delta x_i) - P(\underline{x})}{\Delta x_i} \tag{6}$$

Solving equations 5 and 6 results in a pseudo-load vector which can be used in equation 4. The derivative of the load vector with respect to the design variable, as shown in equation 6, is frequently zero. However, calculation of the finite differences can easily result in an underflow if the finite difference step is chosen too small. With the default settings for the finite difference step ($\Delta x_i = 0.01\ x_i$, DOPTPRM-card [10]) one of the optimization jobs run on 5 different machines resulted in 5 significantly different answers. The difference in internal number representation caused in some cases an underflow in the calculation of the finite differences, resulting in a premature optimum. Increasing the finite difference step to $0.05x_i$ gave good agreement. This has also been reported by other users [12]. The effect of the finite difference step on optimization 1 in figure 13 is shown in figure 17. However, it should be emphasized that behaviour such as shown in figure 17 is exceptional, in most cases a true optimum will be reached using the default settings.

5 Discussion and Conclusion

The design system presented in this paper performs a complete design cycle, including optimization. The time needed for the design cycle is minimized through tuning of the different parts of the system. Furthermore, user-friendliness also reduces the time needed to prepare a design. The time needed for one complete design cycle, for the example treated in this paper, is given in table 1. The excellent graphics of MSC/PATRAN make it possible to visualize the design sensitivity of the structure. Display of design sensitivity is a powerful tool for the designer because it gives a clear insight into how changes in individual parts of the structure affect the structure as a whole. "What-if" studies are shown to be an easy-to-use tool for manually modifying a structure. As long as the structural changes remain small this method is very fast and cheap. The response of the "what-if" analysis is so fast that it may be regarded as an interactive, "real-time" structural modification.

Incorporating several existing analysis techniques into one design system adds extra value to the separate parts. Communication through the design system, and the data preparation and translation by the system, enables the engineer to use several different analyses in MSC/NASTRAN without knowing all details of the analysis. Furthermore, automation of the data transfer between the basis of the design system MSC/PATRAN and the analysis part MSC/NASTRAN provides a huge gain in time in the design cycle. The system presented in this paper is meant as a "special-purpose" system. However, the special-purpose aspect is restricted mainly to the model generator. This implies that it is very easy to modify the program for another type of structure, with the same advantages as shown here.

Current research is focused on creating a so-called "tool-box". This implies that the future development of new design tools will require only the creation of a model generator. The other program modules can be taken directly from this general purpose tool-box. The "tool-box" is also usable for a "handmade" finite element model, so the ease of property definition, preparation of several analysis types and post-processing of several analyses can be made available for virtually every finite element model. Conversion from MSC/PATRAN version 2.5 to MSC/P3-PATRAN is (still) in progress. This program offers a graphical user interface far more sophisticated than its predecessor. MSC/NASTRAN version 68 also offers a shape optimization, which can also be included in a future release of the design system "Cufus".

Acknowledgement

The work presented here is part of a research project carried out with the financial assistance and technical collaboration of Fokker Aircraft BV. The author wishes to thank Fokker for their support. The author also wishes to thank Prof. Dr. A. Rothwell and Prof. Ir. P.A. van der Schee, both of the Faculty of Aerospace Engineering at Delft University of Technology, and Ir. C. Schimmel of Fokker Aircraft BV for the many helpful discussions.

References

1 M.C.Y. Niu: *Airframe Structural Design.* Conmilit Press Ltd., Hong Kong, 1990.

2 E. F. Bruhn: *Analysis of and Design of Flight Vehicle Structures.* Tri-State Offset Company, Cincinnatti, Ohio, USA, 1965.

3 P. Kuhn: *Stresses in Aircraft and Shell Structures.* McGraw-Hill Book Company, Inc. New York, USA, 1956.

4 J.L.T. Santos, M.M. Godse and Kuang-Hua Chang: An Interactive Post-Processor for Structural Design Sensitivity and Optimization: Sensitivity Display and What-If Study. *Computers and Structures*, vol. 35, no. 1, pp. 1-13, 1990.

5 J.L.T. Santos and K.K. Choi: Integrated Computational Considerations for Large Scale Structural Design Sensitivity Analysis and Optimization. *Discretization Methods and Structural Optimization Procedures and Applications,* Proceedings of a GAMM-seminar, Siegen, FRG, Oct. 5-7, 1988.

6 G.J. Moore: *User's Guide MSC/NASTRAN Design Sensitivity and Optimization Version 67.* The MacNeal-Schwendler Corporation, Los Angeles, USA, 1993.

7 *PATRAN Command Language (PCL) Guide.* Release 2.5, PDA Engineering, Costa Mesa, California, USA, 1990.

8 M.E. Heerschap: *User's Manual CUFUS, quick CUt-out FUSelage design,* LR-report, Delft University of Technology, The Netherlands, to be published.

9 *PATRAN Plus User Manual.* Release 2.5, PDA Engineering, Costa Mesa, California, USA, 1990.

10 *MSC/NASTRAN User's Manual version 67.* The MacNeal-Schwendler Corporation, Los Angeles, USA, 1991.

11 *PAT/MSC/NASTRAN Application Interface*. Release 3.0, PDA Engineering, Costa Mesa, California, USA, 1991.

12 R.J. Yang: Shape Sensitivity Analysis and Optimization Using NASTRAN. *Mechanical Structures & Machines*, vol. 19, pp. 281-300, 1991.

Program module	Required time [hours: minutes: seconds]
File preparation	0:10:00
Initial model generation	0:6:30
Mesh adaption	0:5:00
Node equivalencing and checking for duplicate elements	0:2:00
Loads and boundary conditions	0:0:50
Property definition	0:9:10
Definition of design variables	0:8:10
Definition of sensitivity constraints	0:9:10
Generation of MSC/NASTRAN Bulk Data Deck (sensitivity analysis)	0:6:45
Generation of MSC/NASTRAN input files	0:3:00
MSC/NASTRAN sensitivity analysis	0:11:00
Translation of results from MSC/NASTRAN to MSC/PATRAN	0:0:35
Post-processing of sensitivity results (generation of sensitivity result files)	0:2:35
Optimization (single run, including post-processing)	0:17:40
Optimization (2 "check" runs, including post-processing)	0:40:00
What-if analysis (post-processing previous run)	0:5:00
Total time	2:17:25

Table 1: Required time for different program modules.

Note: MSC/PATRAN/Cufus was run on a SUN Sparc 2 and MSC/NASTRAN on a CONVEX C-3840 computer. The times in this table are typical, not absolute times.

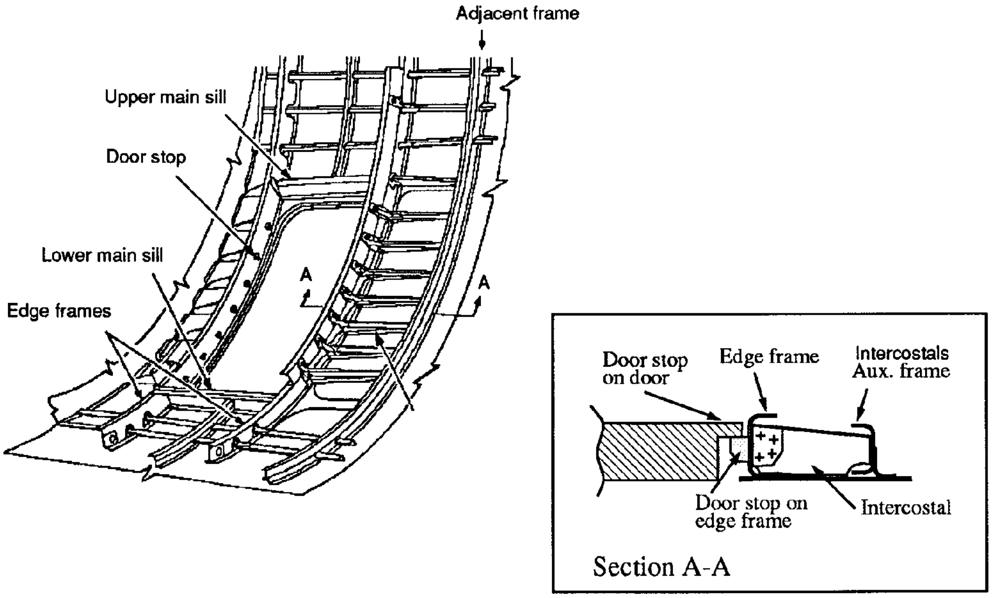


Figure 1: Typical cut-out reinforcement. (Figure 6.5.20 of [1])

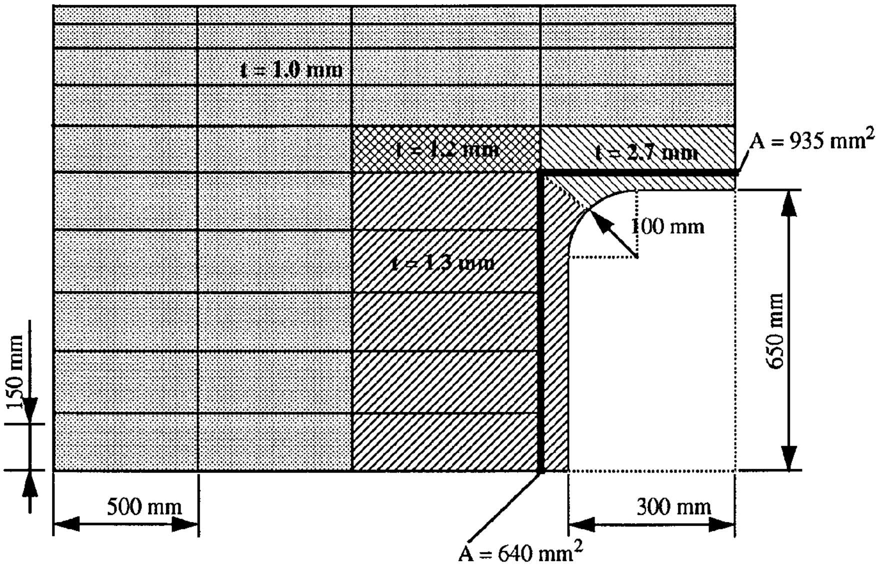


Figure 2: Dimensions used for the example, sized according to Niu [1]

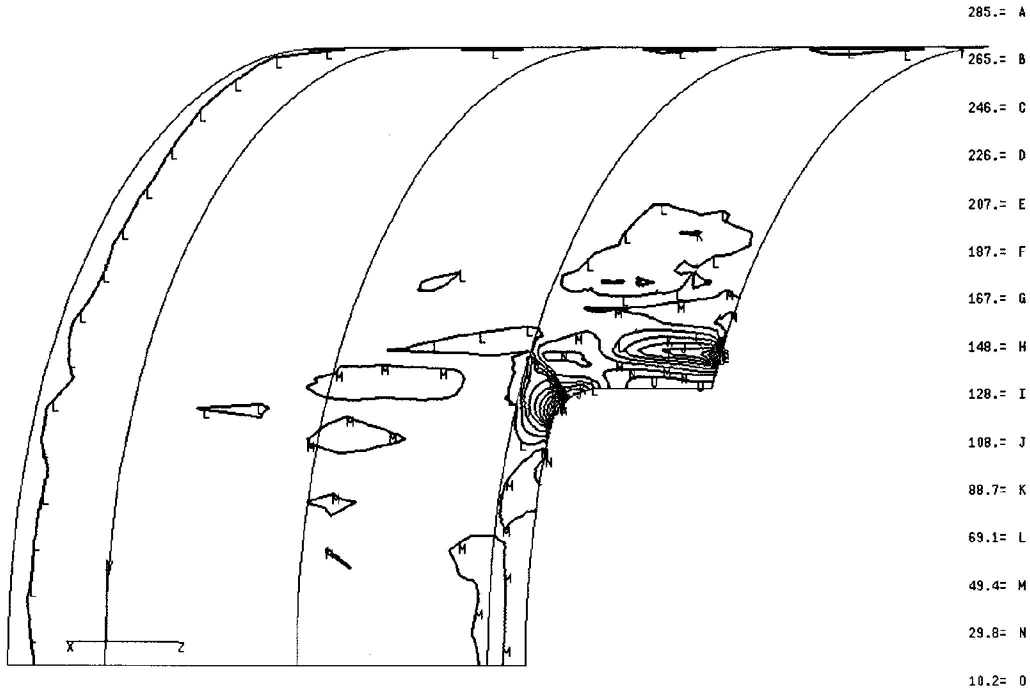


Figure 3a: Major principal stress [MPa] at one surface of the skin in the fuselage structure sized according to Niu [1]

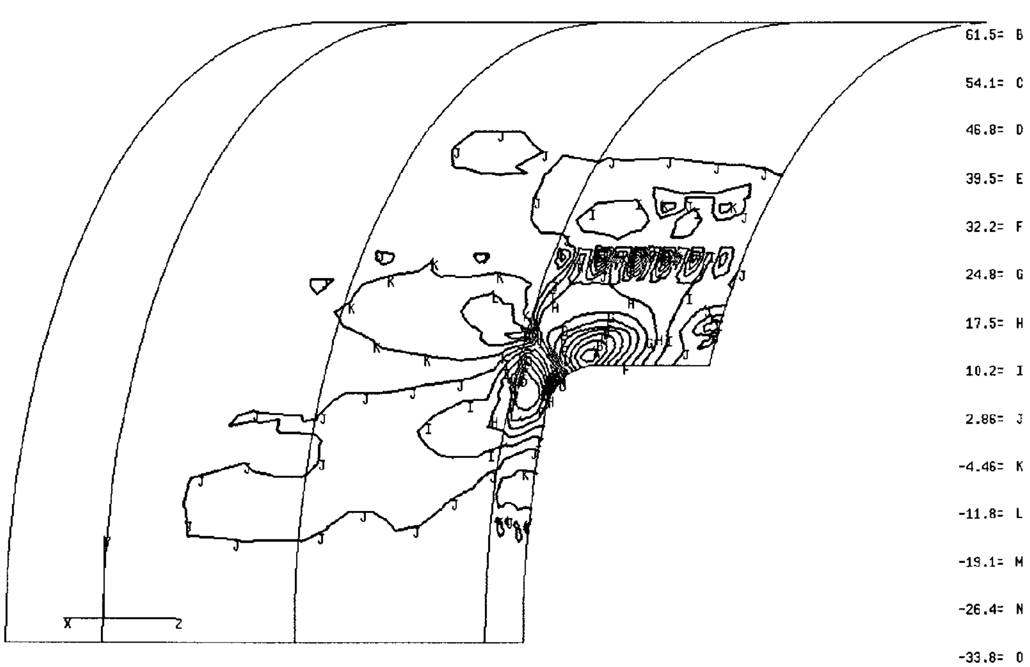


Figure 3b: Shear stress [MPa] at one surface of the skin in the fuselage structure sized according to Niu [1]

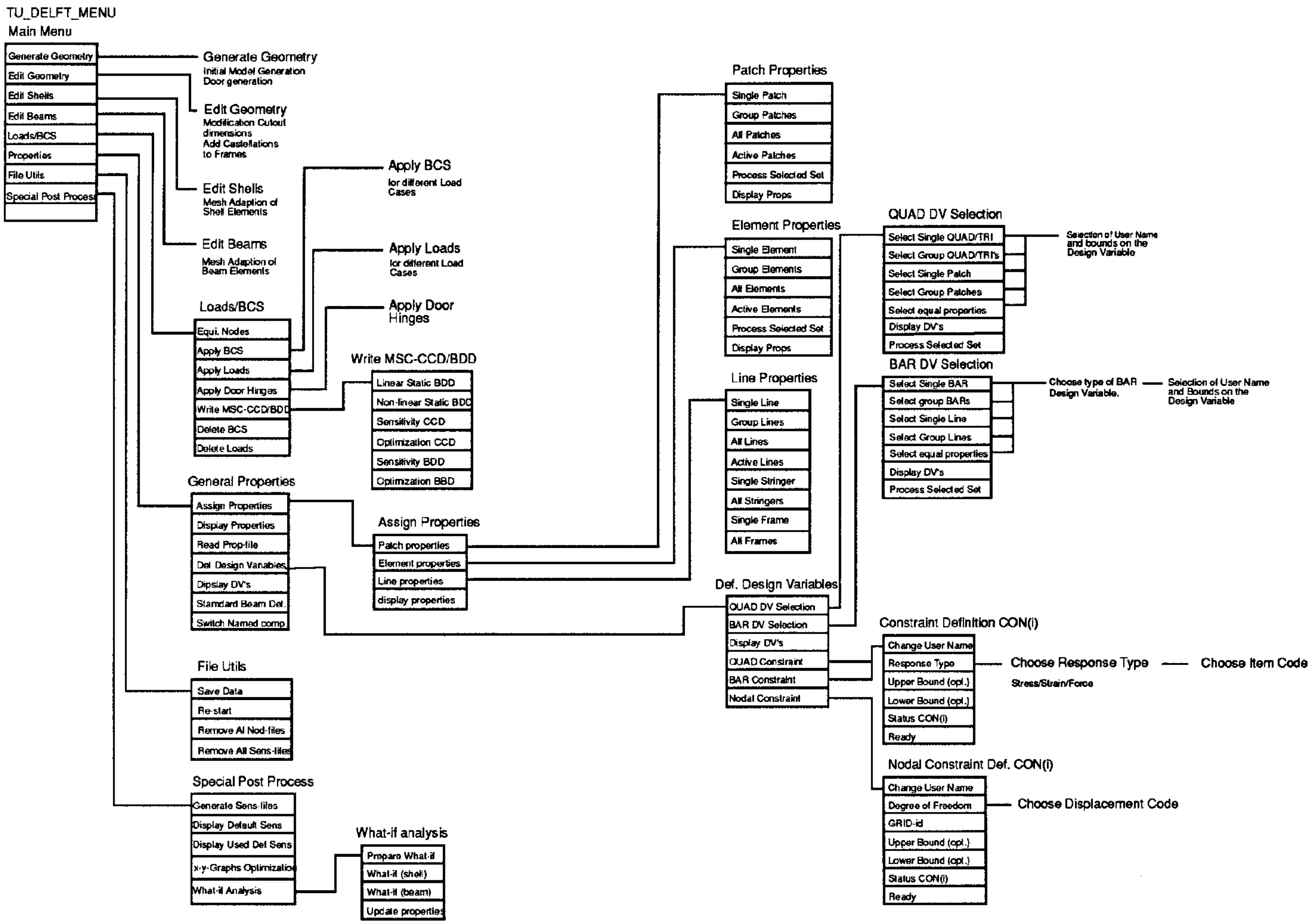


Figure 4: The condensed menu system

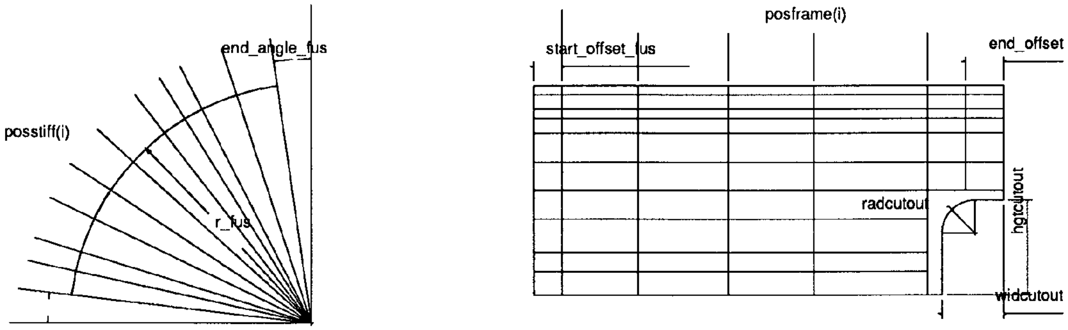


Figure 5: Necessary key dimensions for the geometry generation.

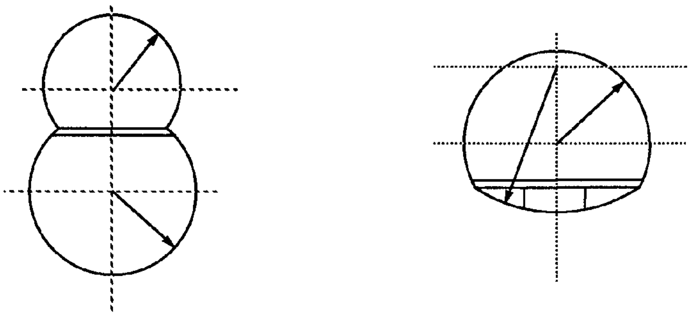
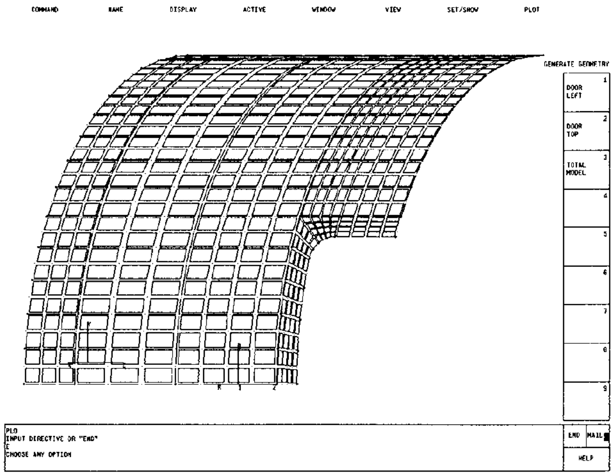
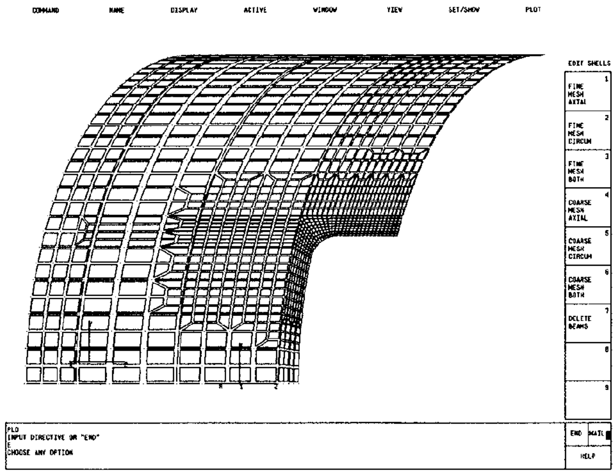


Figure 6: Two possible double-bubble fuselages



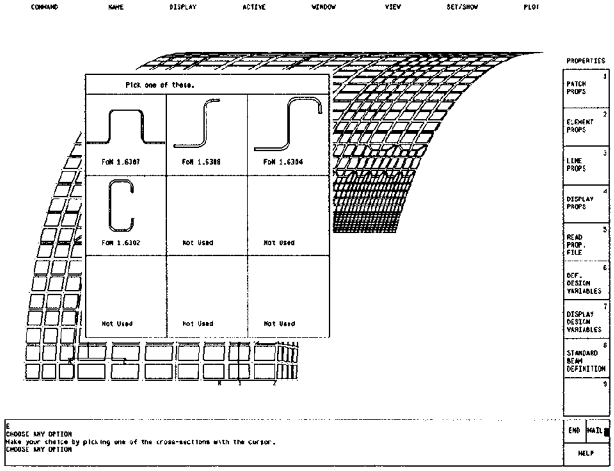
```
PLO
INPUT DIRECTIVE OR "END"
E
CHOOSE ANY OPTION
```

Figure 7: The initial finite element model.



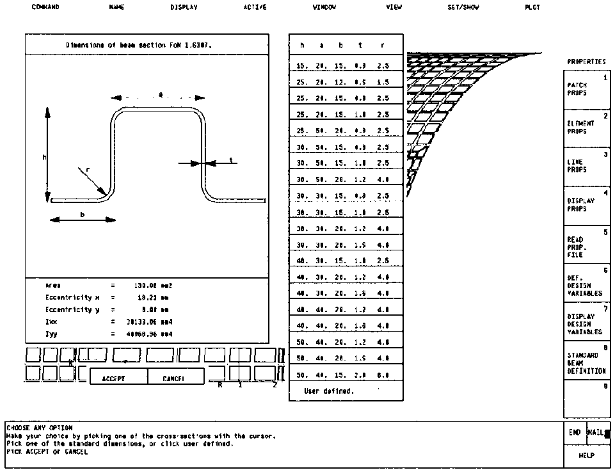
```
PLD
INPUT DIRECTIVE OR "END"
E
CHOOSE ANY OPTION
```

Figure 8: Mesh refinement in both directions on several patches.



```
E
CHOOSE ANY OPTION
Make your choice by picking one of the cross-sections with the cursor.
CHOOSE ANY OPTION
```

Figure 9a: Standard beam definition in main menu.

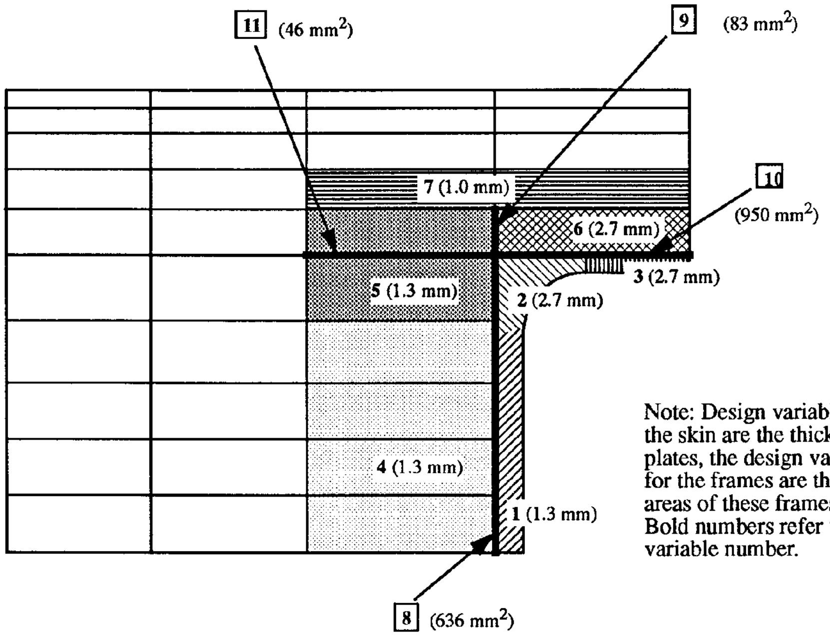


Area = 130.00 mm2
Eccentricity x = 10.21 mm
Eccentricity y = 0.00 mm
Ixx = 30133.06 mm4
Iyy = 40068.96 mm4

h	a	b	t	r
15.	20.	15.	0.8	2.5
25.	20.	12.	0.6	1.5
25.	20.	15.	0.8	2.5
25.	20.	15.	1.0	2.5
25.	50.	20.	0.8	2.5
30.	50.	15.	0.8	2.5
30.	50.	15.	1.0	2.5
30.	50.	20.	1.2	4.0
30.	30.	15.	0.8	2.5
30.	30.	15.	1.0	2.5
30.	30.	20.	1.2	4.0
30.	30.	20.	1.6	4.0
40.	30.	15.	1.0	2.5
40.	30.	20.	1.2	4.0
40.	30.	20.	1.6	4.0
40.	40.	20.	1.2	4.0
40.	40.	20.	1.6	4.0
50.	40.	20.	1.2	4.0
50.	40.	20.	1.6	4.0
50.	40.	15.	2.0	6.0
User defined.				

```
CHOOSE ANY OPTION
Make your choice by picking one of the cross-sections with the cursor.
Pick one of the standard dimensions, or click user defined.
Pick ACCEPT or CANCEL
```

Figure 9b: Second menu for standard hat stringers.



Note: Design variables used for the skin are the thicknesses of these plates, the design variables for the for the frames are the cross-sectional areas of these frames.
Bold numbers refer to the design variable number.

Figure 10: The design variables used for the plates and frames. The initial values of the design variables are put in brackets.

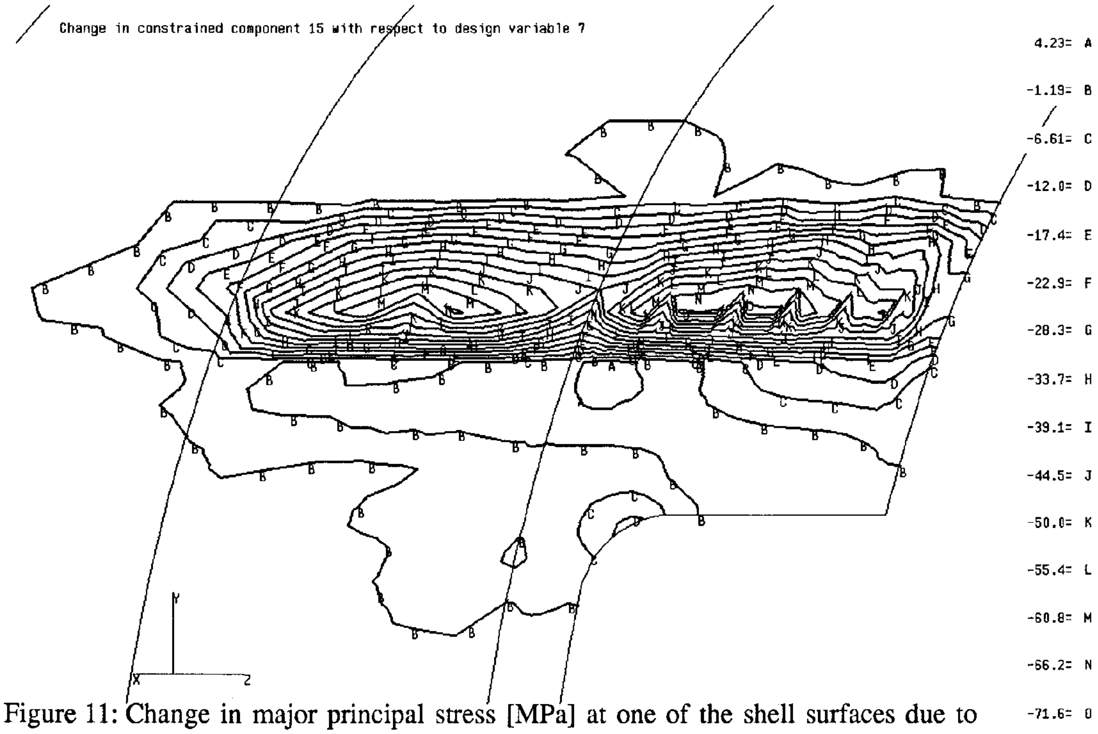


Figure 11: Change in major principal stress [MPa] at one of the shell surfaces due to doubling of the thickness in design variable 7 (see figure 10).

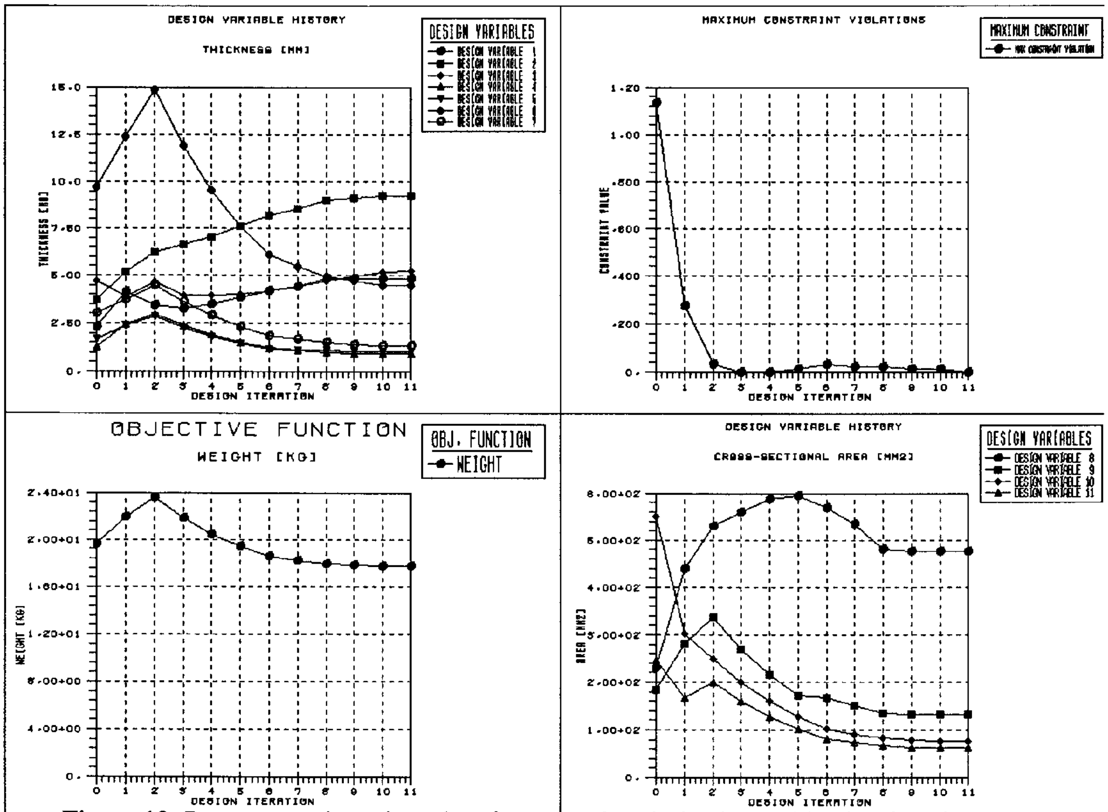


Figure 12: Post-processing of results of structural optimization - objective function.

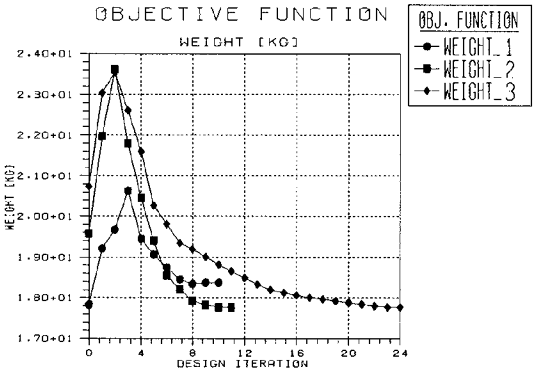


Figure 13: Post-processing the results of repeated structural optimizations, objective functions (weight).

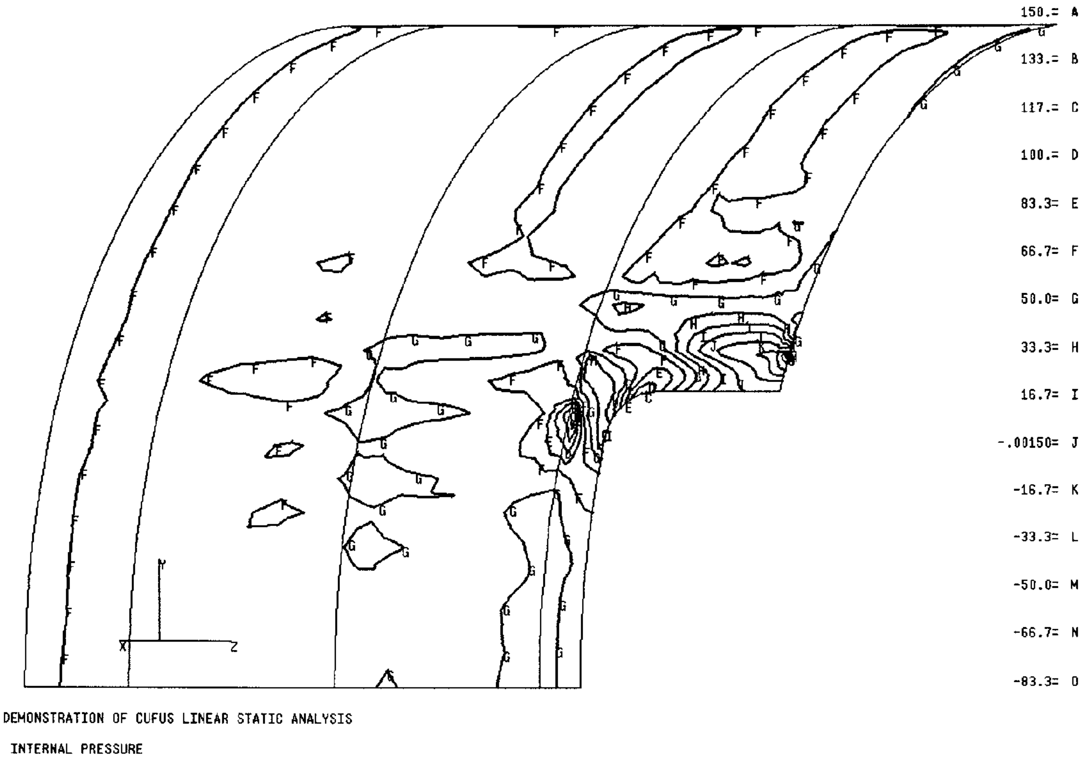


Figure 14: The starting point for the "what-if" analysis.

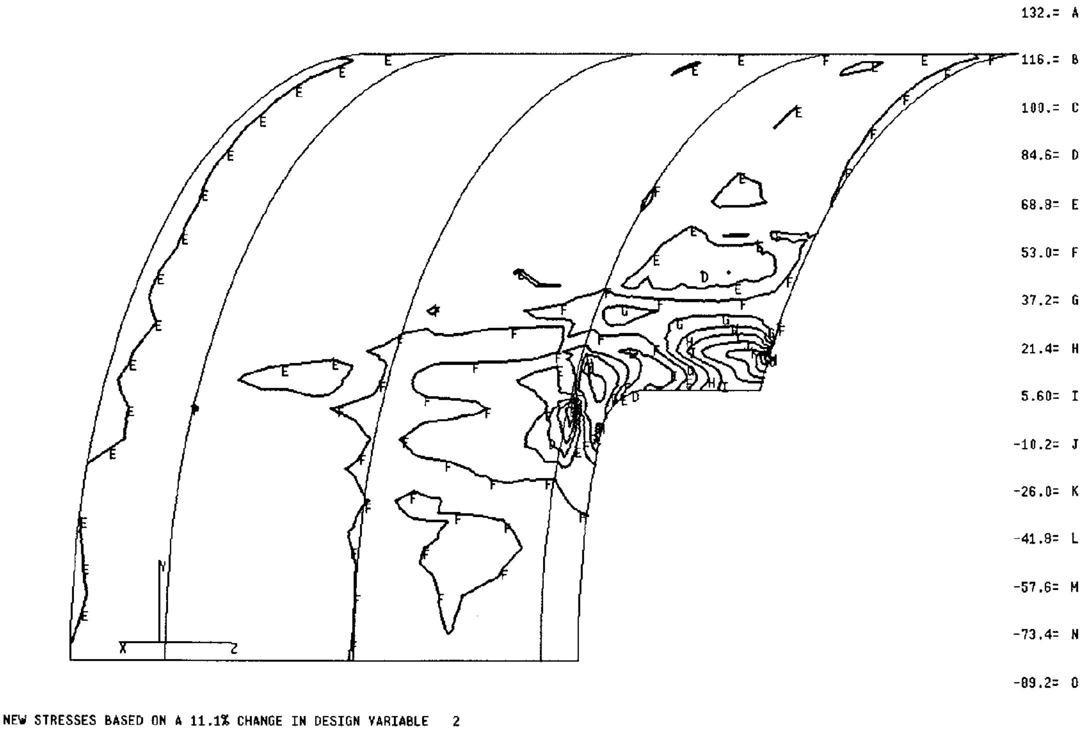


Figure 15: The modified design in the "what-if" analysis.

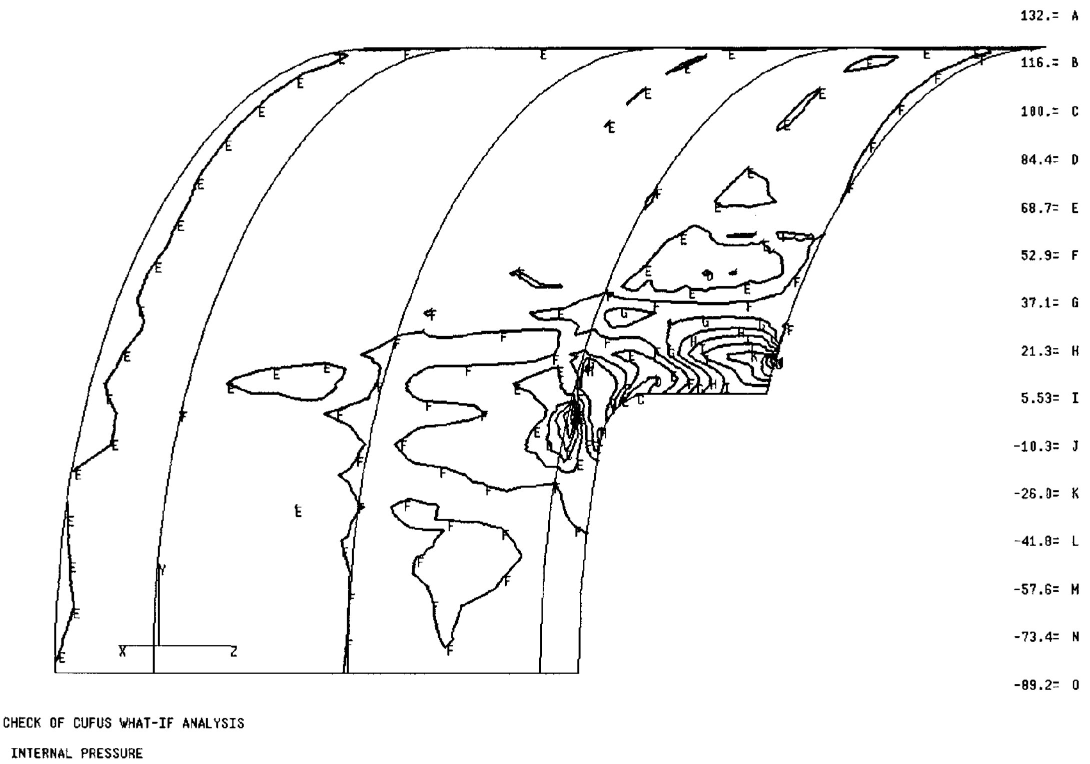


Figure 16: Check on the "what-if" analysis.

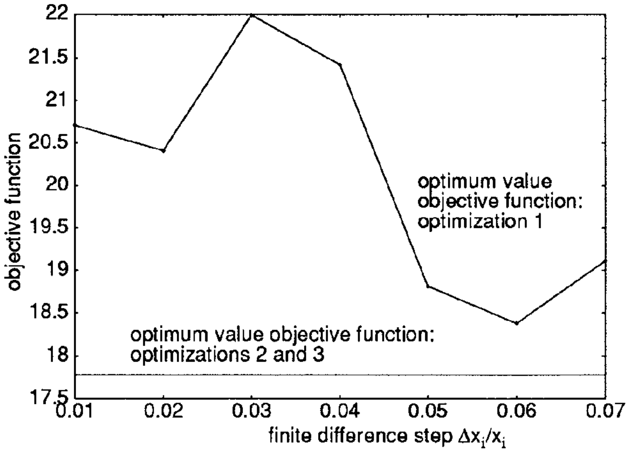


Figure 17: The effect of the finite difference step on the optimum value of the objective function for a specific optimization.