GLOBAL-TO-LOCAL ANALYSIS WITHOUT USING SUPERELEMENTS

KLAUS OTTO SCHWARZMEIER, Eng.

EMBRAER - Empresa Brasileira de Aeronáutica S/A

Av. Brigadeiro Faria Lima, 2170

12227-901 - São José dos Campos - SP

BRAZIL

ABSTRACT

Detail analysis of a structure in general is a problem at least workful. Frequently this type of analysis requires very refined finite elements meshes being unfeasible to include all regions of interest in an unique model. This work can be simplified when we substitute the unique model by a set of interrelated models. We consider in the methodology presented in this paper, one model called *primary model* to analyze the global behavior of the structure and some other models, *the secondary models*, to do the detail analysis of some regions based on the results of the primary model analysis. The technique presented here does not use the Superelements feature of MSC/NASTRAN, neither mesh transitions. Therefore, it can be easy used with MSC/NASTRAN for Windows. The use of this technique by means of MSC/PATRAN will also be discussed.

INTRODUCTION

The analysis of structures is a complex and sometimes very interesting activity. It requires the determination of some global characteristics such as stiffness, load distributions between main components, vibration modes and the study of secondary effects such as stress concentration around holes and other geometric discontinuities.

To make studies of stress concentrations it is necessary that the Finite Element Mesh be a much finer mesh. In a structure composed by many components and a large amount of discontinuities, considering such effects in a unique model, becomes a non practical task (considering the limitations of the modeler, of the solver, or the archiving resources available). The main limitation that normally is forgotten is the time variable. The greater the number of nodes involved in the mesh (more refined meshes), the bigger are the involved matrices and therefore more time will be spent solving the problem. As someone said sometime that "Time is money", if we consider only the increase of the time needed by the solver to solve the finer mesh, we have an increase of the analysis cost. Classically, references state that the time to solve the equations is proportional to the square of the rigid matrix order ($t \quad n^2$). Other times that have to be considered are the time of meshing a finer mesh, the time to prepare the run (only to mention some of them).

Working with smaller problems is ever more easier. Thus, we can use a *primary model* that consider only the main components of the structure. This model is used to analyze the global behavior of the structure, revealing its *critical regions*. This regions will be analyzed in more detail with *secondary models* with a much more refined mesh where are introduced all types of discontinuities and details that are required.

Nowadays, the modeling of such primary and secondary models are not painful than times ago, because of the CAD/CAE graphical softwares that are available. The main problem is the determination of the *loadings and boundary conditions* to be applied to the *secondary model*. A large amount of such informations can be obtained with the analysis of the primary model.

This paper presents an *automated procedure* to use *primary* and *secondary models*. This automatism is obtained by the use of resources available in MSC/NASTRAN associated with a postprocessor that could be easily developed using conventional programming languages. Finally, the usage of this methodology with MSC/NASTRAN for Windows and MSC/PATRAN will be discussed.

METHODOLOGY DESCRIPTION

The keypoint in the proposed methodology is essentially transform a unique complex and difficulty problem in a sequence *of easier to solve and smaller problems*. In fact, it is a different way to solve the problem, but with more chance to succeed and with lower costs. It organizes the work in *steps with clear and well defined targets*, and gives the integration means of each step. An additional advantage is the increased confidence on the results of the *secondary model*, because several input data are obtained directly from the *primary model*.

STEP 1 - THE PRIMARY MODEL

Construct a model the more simple as possible, capable to accurately represent the *global characteristics* and *behavior* of the structure. In this step we are not concerned to analyze the details, having no need to refine the mesh around discontinuities, that simplifies and speeds up its generation. Consequently the results around this regions are valid only at *a certain distance* of them (Principle of Saint-Venant). The details will be analysed, if necessary, by the *secondary models*.

The Fig.1 shows a schema of a primary model of an structure. Notice that there were no concern in refine the mesh around discontinuities.
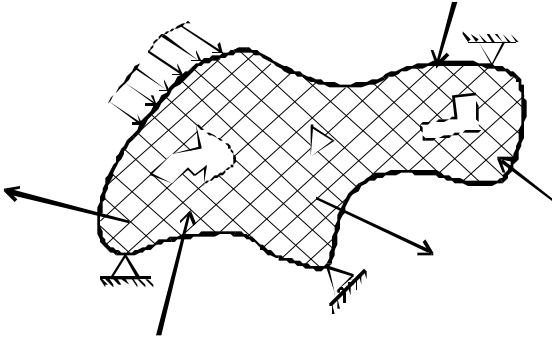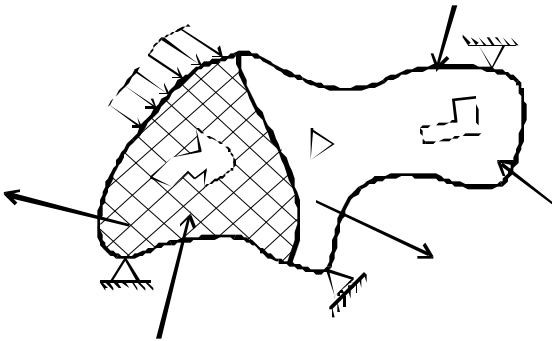
Figure 1 - Primary Model

This step does not require any special feature of MSC/NASTRAN. It is recommended, however, after its validation, to keep a *database* that contains the stiffness matrix already decomposed. This allows to save time when you need to analyze new loading cases using the automatic restart capability of MSC/NASTRAN.

Beyond the *global characteristics* of the structure, this model allows to reveal the critical regions for that loads and boundary conditions. If this regions include one or more discontinuities, secondary models would be constructed for them.

Figure 2 - Detail of a critical region with one discontinuity

The Fig. 2 shows a supposed critical region detected during the analysis of the primary model.

It is recommended that no *relevant changes* will be done on the structure, very common during the preliminary design phase, that certainly will invalidate the detailed analysis of the region. It could even be not critical anymore.

Once the primary model is *consolidated*, we could go to the *detail analysis* step.

STEP 2 - THE SECONDARY MODEL

Figure 3 - Base model of the critical region extracted from Primary Model.

The Secondary Model is used to analyze the *stress concentration* effects that appear due to discontinuities. As this effects are *local effects*, it is not necessary to model large portions of the structure. In general, the modeling of the critical region detected in the previous step is sufficient to evaluate these effects.

With the primary model available, the generation of the secondary model is made easier because it is contained into the primary model. So, one utility of the primary model in this step is to *serve as reference* to make secondary models.

The Fig.3 shows one possible critical region just as modeled on the previous step. The discontinuity, in this case, is represented by an hole of irregular form. This mesh is not proper to detect the high stress gradient due to the hole. To analyze this local effect we need to construct a *more refined* mesh.

The new mesh must include the nodes that exist on the borders of the primary model (the reason for this will be made clear in the next items). To make the mesh more regular, *additional nodes* are created on the borders. The resulting mesh is shown in the Fig.4.

● AdditionalGRIDsattheborders.

○ GRIDsfromprimarymodel

The next step consists in applying the loading and the boundary conditions. This is the more developed part of the process and will be presented in detail next.
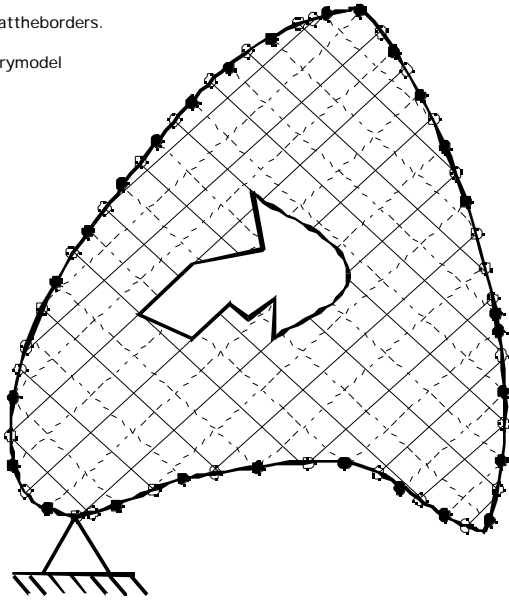
Figure 4 - Secondary Model of the critical region.

STEP 3 - THE APPLICATION OF THE LOADING AND BOUNDARY CONDITIONS

One advantage of primary and secondary models is its *integration*. Part of this integration was shown above where the primary model is used to generate the secondary model.

This item shows another integration aspect involving the two models: the evaluation of the loadings for the secondary model.

As the secondary model is the representation of one detail of the whole structure, it does not have individual loading conditions. The loading conditions are *derived* from that applied to the primary model and were critical during the analysis process.

LOADING DEFINITION

The loadings are divided in two categories:

- **Internal loading**: refers to the loads applied directly to the primary model on the region of the secondary model. This loads must be applied the *usual way*: on the corresponding elements and nodes of the new mesh.

- **Interface loading**: refers to the efforts that act along the interface border of the secondary model. This efforts must be applied to *re-establish the equilibrium* between internal and interface forces (and moments) due to the separation of the secondary model from the rest of the structure. There are two ways to do this: applying interface loads to the nodes or forcing the interface border of the secondary model to stay at the same deformed position of the primary model, *by enforcing the displacements*. In this methodology we use the second one (enforced displacements) because it is easier to automate.

The facts that justify this substitution are:

- When the border is sufficiently distant from the discontinuity, it is not influenced by it. Therefore, the displacement field at the border *remains the same* obtained from the primary model.

- The displacements and stresses that result from the simultaneous application of the loading conditions is equivalent to the algebraic sum of each corresponding load condition applied separately (Principle of the Effects Superposition).

- For one stiffness and one determined loading condition corresponds only one displaced configuration and vice-versa.
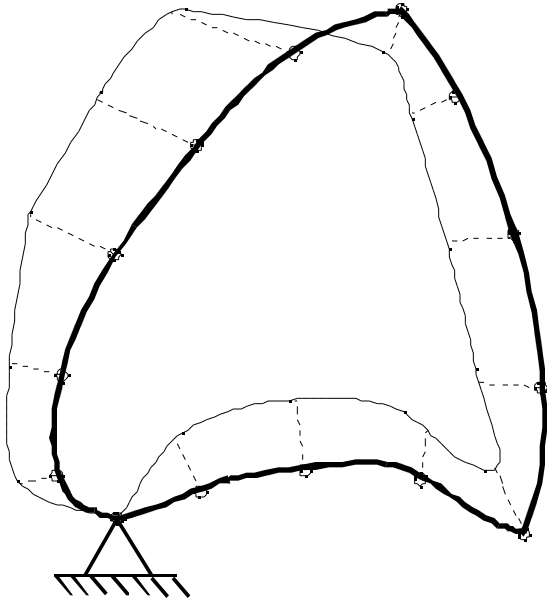

Figure 5 - Displaced condition of the Primary Model at the region corresponding to the boundary of the Secondary Model.

Notice that the last two assertions implicitly suppose that we are working with small displacements (Linear Analysis). This do not invalidate the use of the methodology for non-linear problems. In this case, we must *additionally* assure that all the border is far from the region under non-linear effects.

The Interface Loading is applied using special boundary conditions at the border (Fig.5). All other boundary conditions *inherit* from the primary model must be applied normally to the corresponding nodes of the secondary model. Notice the presence of one of the supports acting on the critical region at Fig.3 and 4.

MIGRATING DISPLACEMENTS ON THE BORDER

In the previous section it was shown that the internal forces effect at the border that come from the rest of the structure can be obtained applying prescribed displacements to the border nodes. This displacement values are obtained directly from the corresponding nodes of the primary model.

In this phase we will use a computer program to reformat the displacements from the MSC/NASTRAN punch file to generate a file with SPCD bulkdata cards with large field format to prevent precision lost. This is done only for the nodes of the primary model that correspond to the border of the secondary model.

The use of the SPCD bulkdata card is fundamental in case of the need to analyze multiple loading conditions. The enforced displacements are that way, treated by MSC/NASTRAN as a loading condition, i.e, they are selected as a load case with a LOAD card in the Case Control Section and not with a SPC card. This difference implies that we can also use the RESTART technique to analyze the secondary model, allowing to save processing time according to the model size and the number of loading cases to be analyzed.
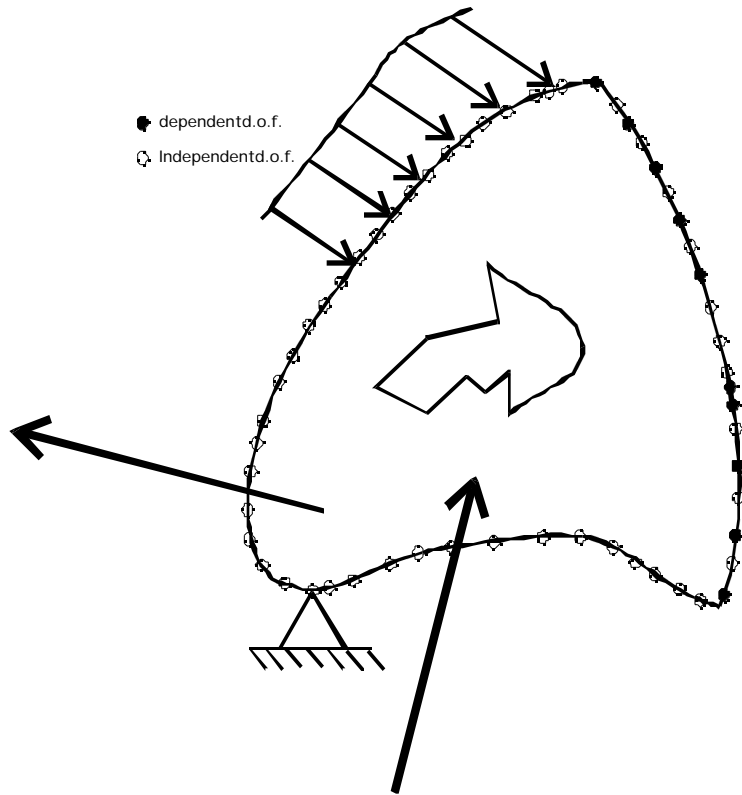
DISPLACEMENT INTERPOLATION ON THE BORDER

The border of the secondary model has in general more nodes than in the original primary model due to the mesh refinement. This additional nodes and the original ones are represented with different markers in the Fig. 4.

The additional nodes must also satisfy certain boundary conditions, as follow:

In the primary model the displacements of a region on the border between two consecutive nodes are determined by the displacements of this two nodes (Fig. 5). This condition must also be satisfied by the secondary model. As the primary model gives displacements only for some nodes, it is necessary to use an interpolation process to obtain the displacement values for the other secondary model additional nodes.

This interpolation can be made by MSC/NASTRAN itself through the use of RSPLINE *Interpolation Constraint Elements*. For that, it is enough to connect the border nodes with one or more of this elements. The original nodes and the additional nodes out of the interface border are declared *independent* and the additional nodes at the interface border are declared *dependent* (Fig. 6).

Figure 6 - Border GRIDs connected by RSPLINEs

The Fig. 7 and 8 illustrate the difference between the displacements when we leave the additional nodes free and when the additional nodes are forced to follow the displacements of the original nodes by means of the RSPLINE *Interpolation Constraint Element*.
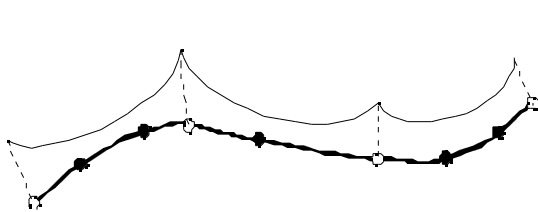
Figure 7 - Effect of Enforced displacements applied only at the border GRIDs corresponding to the Primary Model. OBS.: The results in this case are not correct.
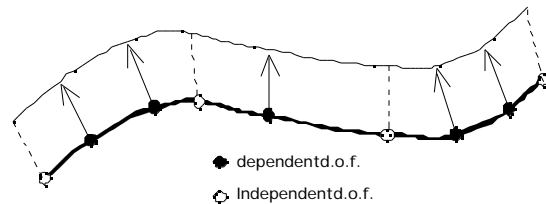
Figure 8 - Effect of Enforced displacements applied at the border GRIDs connected with RSPLINEs.

STEP 4 - THE SECONDARY MODEL ANALYSIS

With the displacements of the original nodes formatted as SPCD cards and all the interface border nodes connected by RSPLINE elements, we need now completely constrain the original nodes on the interface border with SPC or SPC1 bulkdata cards.

In the case there are multiple loading conditions it is recommended to perform a MSC/NASTRAN run only with the constrained border and the original boundary condition without the enforced displacements application, to obtain a database with the decomposed stiffness matrix. This procedure intends to save time in the next runs.

In this runs we can use the RESTART capability of MSC/NASTRAN to efficiently use the decomposed stiffness matrix saved in the database. The only additional data needed for this runs are the sets of SPCD cards corresponding to the loading conditions to be analyzed.

HOW TO USE THIS METHODOLOGY WITH MSC/NASTRAN FOR WINDOWS

This methodology can be used directly with MSC/NASTRAN for Windows as it its described in this paper. As the FEMAP modeler does not support the RSPLINE *Interpolation Constraint Element*, this element need to be given by means of a file in conjunction with the SPCD Enforced Displacements file generated by the in house postprocessor (see MIGRATING DISPLACEMENTS ON THE BORDER above), when the analysis of the secondary model is run, using the INCLUDE option to include this file in the BULKDATA Section of the MSC/NASTRAN input deck.

HOW TO USE THIS METHODOLOGY WITH MSC/PATRAN IN CONJUNCTION WITH MSC/NASTRAN

MSC/PATRAN has a feature to transform the results (displacements in this case) of an analysis run in a Continuous Displacement Field () that can be applied to the interface border of the secondary structure region. By means of the definition of this Displacement Field the interpolation of the displacements at the additional nodes on the secondary model border is made by MSC/PATRAN itself, and it is not necessary to use the RSPLINE *Interpolation Constraint Element*.

The MSC/NASTRAN analysis deck is generated for the secondary model to be analyzed with MSC/NASTRAN.

CONCLUSIONS

The usage of this technique of primary and secondary models demands a *change in the philosophy* of how structures are modeled for the finite element method. Instead of trying to construct a unique model that totally covers all needs and for this reason would be very complex, it can be chosen a *set of more simple models with specific finalities*. This different way to attack the problem results in some advantages:

• Each model has a defined finality that reflects directly on how much refinement is needed for the finite element analysis. That way, *more regular meshes* are obtained, that reduce the occurrence of *numerical errors* during problem resolution.

• The time needed to construct, check and analyze the primary model is *sensibly reduced* due to detail elimination that do not contribute on the global behavior of the structure. This time reduction is fundamental during the initial phase of a project because it allows the *study of a higher number* of solutions, rising up the probability to obtain a better definitive solution.

• The primary model, without unnecessary details for a preliminary static analysis, can be integrally used (or serve as a basis) for dynamic analysis identically important in determining the global structure behavior. The use of a complex unique model prevents this integration (or minimizes the possibility of integration), demanding to make another more simplified model for this purpose. The disadvantages on doing an unique model are:

- For each project modification, it is necessary to update at least two different models, one for static analysis and other for dynamic analysis.

- This updates use to be as time consuming and hard to do as (or more), the bigger the model is.

- Being the models different, the management of successive modifications can be a difficult task. It can be said that, after a certain number of modifications, probably the models *will not be representing the same structure*.

• Allows saving a lot of time and resources in the detail analysis, because:

- Postpone the detail analysis to a later phase of the project, and therefore *not so susceptible to modifications.*

- Until this point the primary model have already be *sufficiently tested and checked* and therefore the results are more confident, being able to be directly used as boundary conditions for the secondary models derived from it.

- Through the results of primary models, the *critical regions* can be identified. The number of secondary models needed can so be defined, avoiding unnecessary detailments of the structure.

The *practical viability* of this methodology presented here depends fundamentally of the *special modeling resources* (RSPLINE), solution algorithms (SPCD) and the RESTART capability, that are available with MSC/NASTRAN associated with the development of the simple postprocessor described (reformatter).
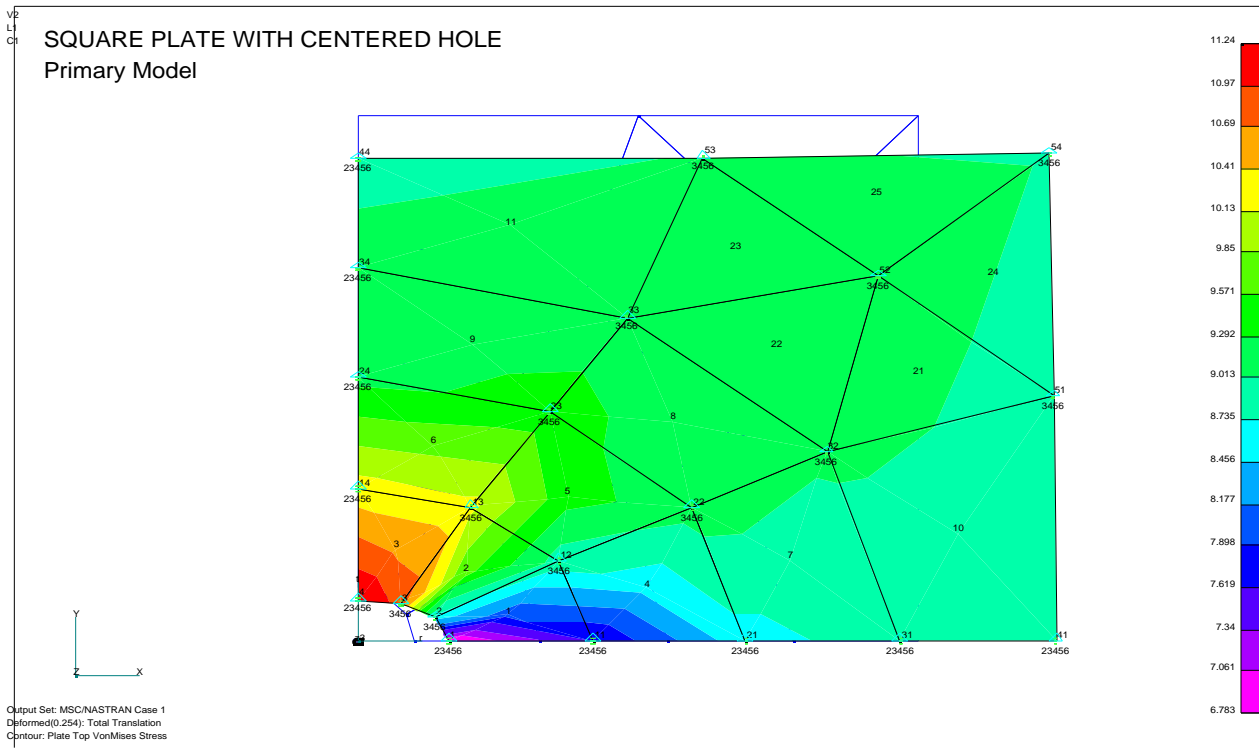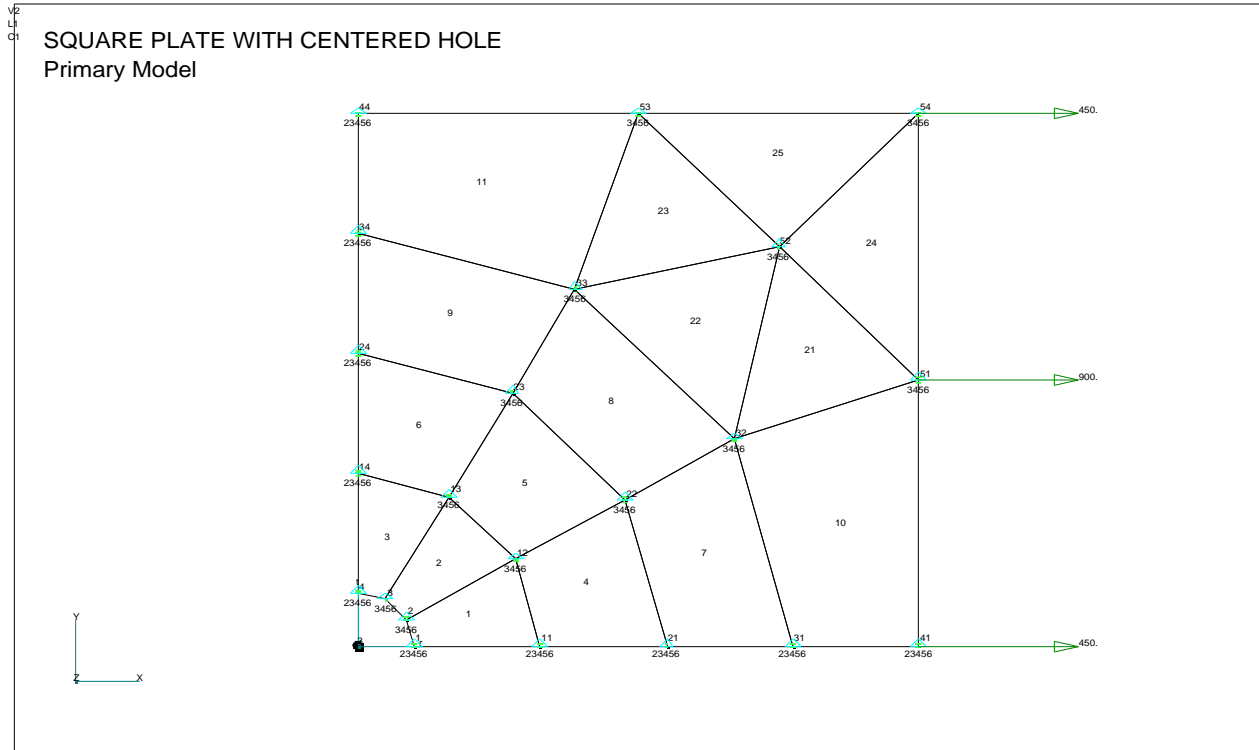
REFERENCES

CONFERENCE PAPER

(1) Sauer, P. H., "Metodologia Automatizada para Análise de Modelos Secundários utilizando Resultados de Modelos Primários, 1º Encontro Nacional de Usuários MSC/NASTRAN, 1992
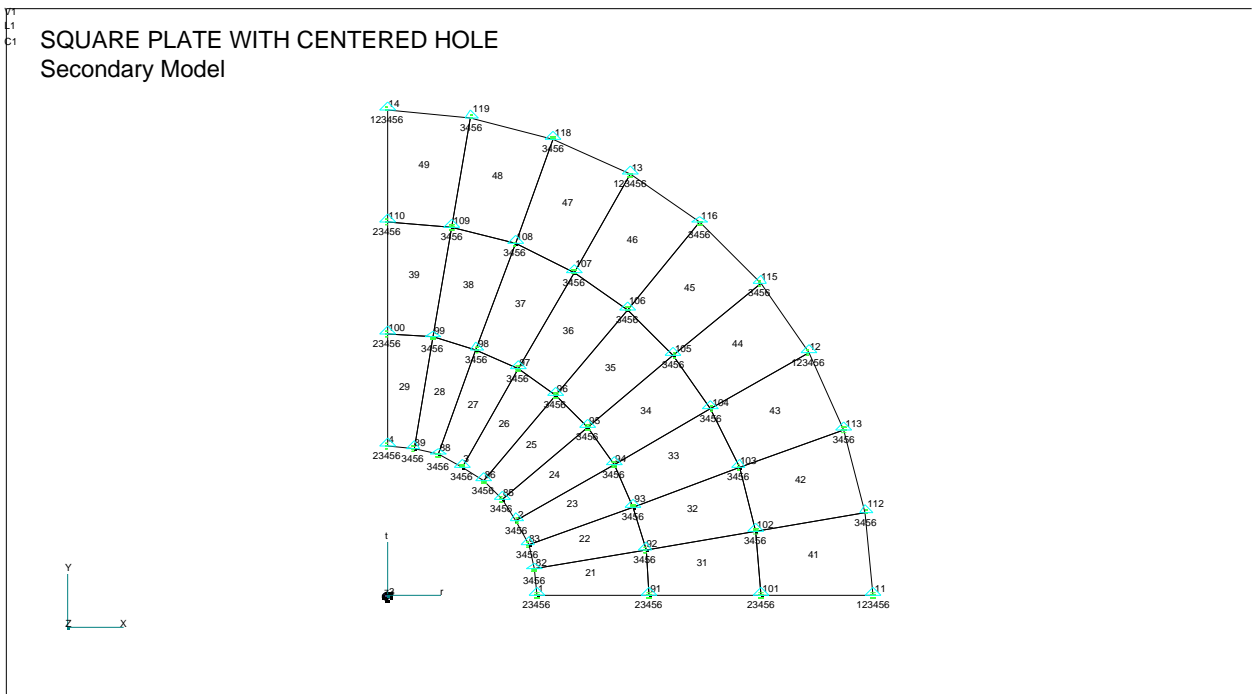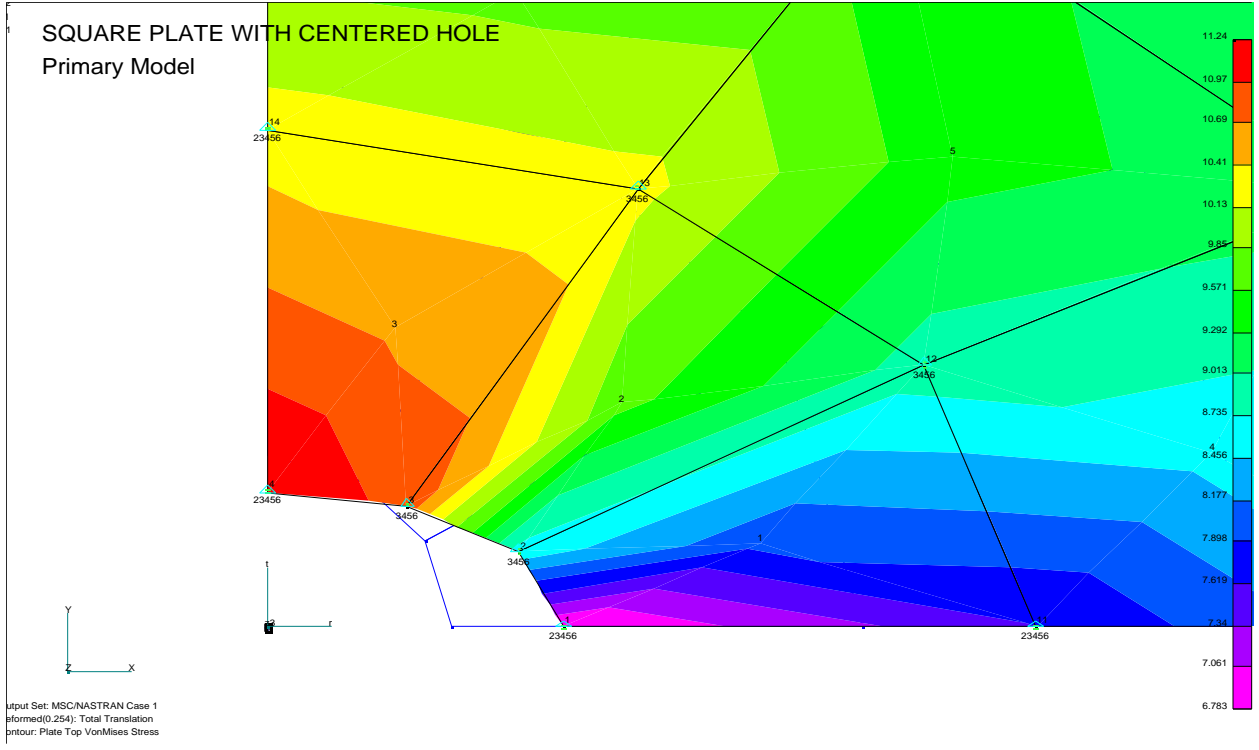
MSC/PRODUCTS

(2) MSC/NASTRAN Linear Static Analysis User´s Guide, Versions 68 and 69, The MacNeal-Schwendler Corporation, Los Angeles, CA, 1995 and 1997 respectively.

(3) MSC/NASTRAN for Windows User´s Guide, Version 3.0.2, The MacNeal-Schwendler Corporation, Los Angeles, CA, October 1997

(4) MSC/PATRAN User´s Manual, Version 7.5, The MacNeal-Schwendler Corporation, Los Angeles, CA, October 1997

(5) MSC/PATRAN - PAT302 Exercise Workbook, Lesson 8, Release 5.0 Version, Revision 1.0, The MacNeal-Schwendler Corporation, Los Angeles, CA, April 1996.

FIGURES AND PLOTS

Using this methodology with MSC/NASTRAN for Windows

SQUARE PLATE WITH CENTERED HOLE
Primary Model

SQUARE PLATE WITH CENTERED HOLE
Primary Model

Output Set: MSC/NASTRAN Case 1
Deformed(0.254): Total Translation
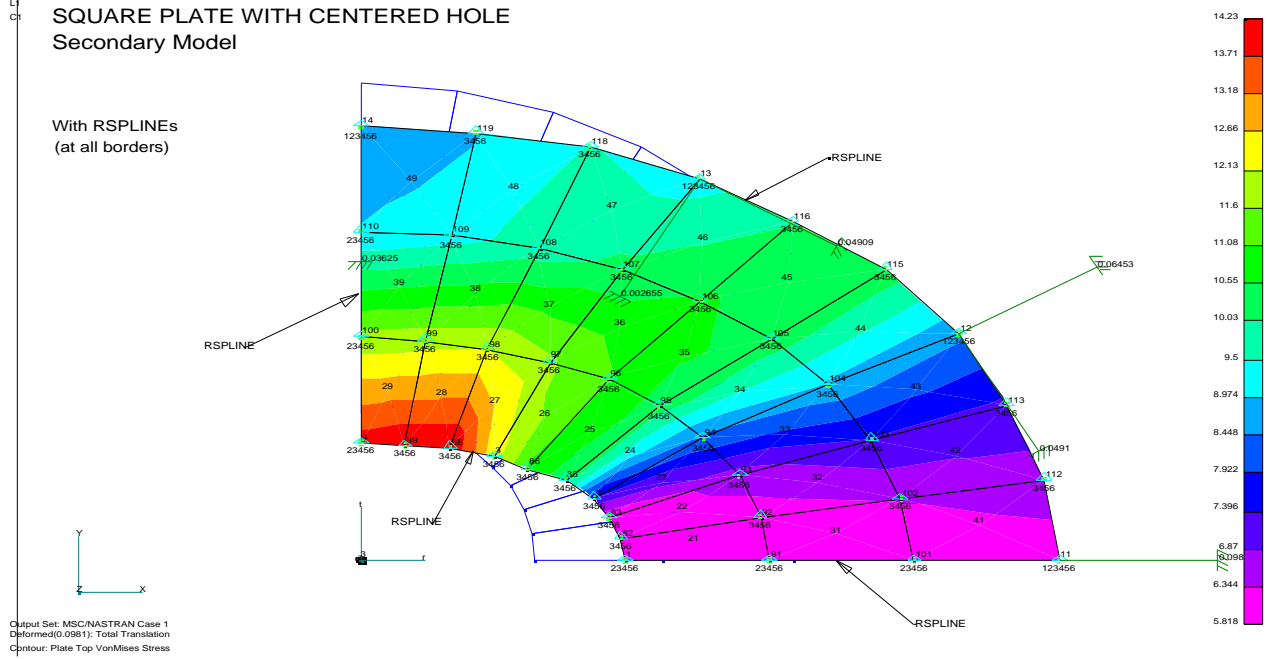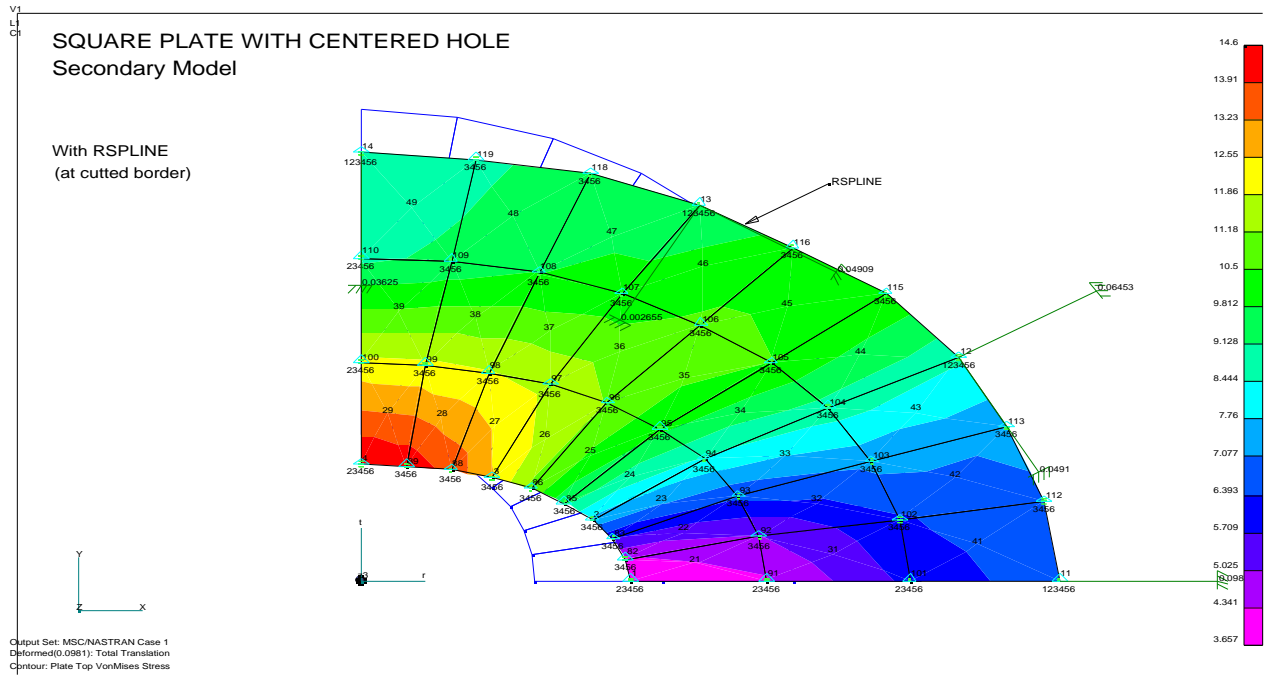Contour: Plate Top VonMises Stress

SQUARE PLATE WITH CENTERED HOLE
Primary Model

SQUARE PLATE WITH CENTERED HOLE
Secondary Model

SQUARE PLATE WITH CENTERED HOLE
Secondary Model

Without RSPLINE

Output Set: MSC/NASTRAN Case 1
Deformed(0.0981): Total Translation
Contour: Plate Top VonMises Stress

11.46
10.97
10.49
9.996
9.506
9.016
8.527
8.037
7.548
7.058
6.568
6.079
5.589
5.1
4.61
4.12
3.631

SQUARE PLATE WITH CENTERED HOLE
Secondary Model

With RSPLINEs
(at all borders)

Output Set: MSC/NASTRAN Case 1
Deformed(0.0981): Total Translation
Contour: Plate Top VonMises Stress

14.23
13.71
13.18
12.66
12.13
11.6
11.08
10.55
10.03
9.5
8.974
8.448
7.922
7.396
6.87
6.344
5.818

SQUARE PLATE WITH CENTERED HOLE
Secondary Model

With RSPLINE
 (at cutted border)

Output Set: MSC/NASTRAN Case 1
Deformed(0.0981): Total Translation
Contour: Plate Top VonMises Stress

Comparing the results:

Model	MAX
Primary Model	11.24
Secondary Model without RSPLINE	11.46
Secondary Model with RSPLINEs at all borders	14.23
Secondary Model with RSPLINE at interface border	14.60

Using this methodology with MSC/PATRAN