Performance Of Interface Elements In The Finite Element Method

2004

Kairas Rabadi

University of Central Florida

Find similar works at: http://stars.library.ucf.edu/etd

University of Central Florida Libraries http://library.ucf.edu

Part of the Mechanical Engineering Commons

STARS Citation


This Masters Thesis (Open Access) is brought to you for free and open access by STARS. It has been accepted for inclusion in Electronic Theses and Dissertations by an authorized administrator of STARS. For more information, please contact lee.dotson@ucf.edu.
© 2004 Kairas S. Rabadi
The objective of this research is to assess the performance of interface elements in the finite element method. Interface elements are implemented in the finite element codes such as MSC.NASTRAN, which is used in this study. Interface elements in MSC.NASTRAN provide a tool to transition between a shell-meshed region to another shell-meshed region as well as from a shell-meshed region to a solid-meshed region. Often, in practice shell elements are layered on shell elements or on solid elements without the use of interface elements. This is potentially inaccurate arising in mismatched degrees of freedom. In the case of a shell-to-shell interface, we consider the case in which the two regions have mismatched nodes along the boundary. Interface elements are used to connect these mismatched nodes. The interface elements are especially useful in global/local analysis, where a region with a dense mesh interfaces to a region with a less dense mesh. Interface elements are used to help avoid using special transition elements between two meshed regions. This is desirable since the transition elements can be severely distorted and cause poor results. Accurate results are obtained in shell-shell and shell-solid combinations. The most interesting result is that not using interface elements can lead to severe inaccuracies. This difficulty is illustrated by computing the stress concentration of a sharp elliptical hole.
ACKNOWLEDGEMENTS

I would like to take this opportunity to express my sincere appreciation to my professor and advisor, Dr. David Nicholson, who has helped and guided me throughout the course of this research.

I would like to thank Dr. Richard Zarda, who suggested this research topic, and has helped me at every step of this research.

I would also like to thank my committee members; Dr. Alain Kassab and Dr. Quan Wang for being on my thesis defense committee, and my colleague Eric Teuma for his constant support and encouragement.

Last but not the least, I would like to thank my parents, my family, and the Dashtaki family, for their enduring support in my academic pursuit.
# TABLE OF CONTENTS

LIST OF TABLES ......................................................................................................................... vi
LIST OF FIGURES ........................................................................................................................ vii
CHAPTER 1 INTRODUCTION ........................................................................................................ 1
CHAPTER 2 LITERATURE REVIEW ............................................................................................... 4
CHAPTER 3 MATHEMATICAL FORMULATION OF INTERFACE ELEMENTS ......................... 5
CHAPTER 4 SHELL-TO-SHELL INTERFACE ELEMENT ............................................................ 8
  4.1 Introduction .......................................................................................................................... 8
  4.2 Quarter-Plate Model ........................................................................................................... 10
    4.2.1 Mesh of the Quarter-Plate with Element Ratio 1:1 ...................................................... 11
    4.2.2 Mesh of the Quarter Plate with unequal Elements ...................................................... 18
    4.2.3 Mesh of the Quarter Plate Model using Transition Elements...................................... 21
  4.3 Conclusion .......................................................................................................................... 27
CHAPTER 5 SHELL-TO-SOLID INTERFACE ELEMENT ............................................................. 28
  5.1 Introduction ......................................................................................................................... 28
  5.2 Cantilever Beam Model ...................................................................................................... 29
  5.3 Quarter Plate Model ........................................................................................................... 33
    5.3.1 Quarter Plate Model - Case 1 ......................................................................................... 33
    5.3.2 Quarter Plate Model - Case 2 ......................................................................................... 42
    5.3.3 Quarter Plate Model - Case 3 ......................................................................................... 49
    5.3.4 Transition Elements ....................................................................................................... 56
    5.3.5 Conclusion ................................................................................................................... 63
  5.4 Non-linear Model ................................................................................................................ 64
CHAPTER 6 DISCUSSION .............................................................................................................. 74
  6.1 For the shell-to-shell interface element – GMINTC ........................................................... 75
  6.2 For the shell-to-solid interface element – RSSCON ............................................................ 76
CHAPTER 7 CONCLUSION ......................................................................................................... 77
REFERENCES ............................................................................................................................. 78
LIST OF TABLES

Table 4-1: Summary of Results obtained for the GMINTC element ................................................. 27
Table 4-2: Summary of Results obtained for the GMINTC element with Transition Elements .. 27
Table 5-1: Summary of Results for the Cantilever Beam Model ..................................................... 32
Table 5-2: Summary of Results obtained with the RSSCON element .......................................... 62
Table 5-3: Summary of Results obtained with the RSSCON element in the case of transition elements ................................................................. 62
Table 5-4: Summary of Results obtained with the RSSCON element .......................................... 63
Table 5-5: Summary of Results obtained with the RSSCON element in the case of transition elements ................................................................. 63
| Figure 1-1: Distinct models on different workstations                             | 2 |
| Figure 4-1: Two regions of unequal shell mesh densities                        | 8 |
| Figure 4-2: Three noded transition shell mesh                                  | 9 |
| Figure 4-3: Illustrating the use of Interface elements                         | 9 |
| Figure 4-4: Quarter Plate model with dimensions                                | 10 |
| Figure 4-5: Two shell meshed regions                                           | 11 |
| Figure 4-6: Two shell meshed regions with unmatched nodes                      | 12 |
| Figure 4-7: An enlarged view of the mismatched nodes on the top edge           | 13 |
| Figure 4-8: An enlarged view of the mismatched nodes on the right edge         | 13 |
| Figure 4-9: Input Panel in I-DEAS9 for Force                                   | 14 |
| Figure 4-10: Input Panel for Constraints on left edge                          | 14 |
| Figure 4-11: Input Panel for constraints on bottom edge                        | 15 |
| Figure 4-12: Plot for stresses in the y-direction                             | 17 |
| Figure 4-13: Mesh of the quarter plate with one region denser than the other   | 18 |
| Figure 4-14: A closer look at the denser mesh and the boundary of mismatched nodes | 19 |
| Figure 4-15: An enlarged view of the top edge of the boundary of the mismatched nodes | 19 |
| Figure 4-16: Plot of the Stress in the y-direction                             | 20 |
| Figure 4-17: Mesh of one region of the quarter plate                           | 21 |
| Figure 4-18: Mesh of second region of the quarter plate                        | 22 |
| Figure 4-19: Mesh of the two regions of the quarter plate                      | 22 |
| Figure 4-20: Mesh of the two regions brought together                          | 23 |
| Figure 4-21: An enlarged view of the mismatched nodes along the boundary of the two regions | 24 |
| Figure 4-22: An enlarged view of the transition elements with coincident nodes | 24 |
| Figure 4-23: Plot of the stress in the y-direction using interface elements     | 25 |
| Figure 4-24: Plot of the stress in the y-direction without interface elements   | 26 |
| Figure 5-1: Illustrating the position of shell and solid nodes                 | 28 |
| Figure 5-2: Cantilever beam model                                              | 29 |
| Figure 5-3: Displacement Plot in I-DEAS9                                       | 30 |
| Figure 5-4: Non-layered Stress plot in MSC.NASTRAN                            | 31 |
| Figure 5-5: Stress plot in MSC.NASTRAN                                         | 31 |
| Figure 5-6: Isometric view of the quarter plate showing the two regions; surface and volume | 34 |
| Figure 5-7: Showing the front and isometric views of the meshed plate          | 35 |
| Figure 5-8: Meshed plate with Boundary conditions applied                      | 35 |
| Figure 5-9: Plot of the Stress in the y-direction                             | 36 |
| Figure 5-10: Plot of the stress in the x-direction                            | 37 |
| Figure 5-11: View of the mesh without interface elements                       | 38 |
Figure 5-12: Stress Plot in the y-direction without Interface Elements ........................................... 39
Figure 5-13: Enlarged view of the area of concern ........................................................................ 40
Figure 5-14: Stress Plot; Without Interface element (left) and with interface elements (right) ... 40
Figure 5-15: Plot of the stress in the x-direction .......................................................................... 41
Figure 5-16: Quarter Plate Model for Case 2 ................................................................................ 42
Figure 5-17: Quarter Plate Model mesh with Boundary Conditions .............................................. 43
Figure 5-18: Stress Plot in the y-direction ...................................................................................... 44
Figure 5-19: Plot of the stresses in the x-direction ....................................................................... 45
Figure 5-20: Stress Plot in the y direction without interface elements ........................................... 46
Figure 5-21: Enlarged view of the area of concern ....................................................................... 46
Figure 5-22: Stress Plot; Without Interface element (left) and with interface elements (right) ... 47
Figure 5-23: Plot of stresses in the x-direction .............................................................................. 48
Figure 5-24: Quarter Plate model for Case 3 ................................................................................ 49
Figure 5-25: Mesh of Quarter Plate Model Case 3 with Boundary Conditions ............................ 50
Figure 5-26: Stress plot in the y-direction ...................................................................................... 51
Figure 5-27: Stress plot in the x-direction ...................................................................................... 52
Figure 5-28: Stress Plot in the y direction without interface elements .......................................... 53
Figure 5-29: Enlarged view of the area of concern ....................................................................... 54
Figure 5-30: Stress Plot; Without Interface element (left) and with interface elements (right) ... 54
Figure 5-31: Plot of the stresses in the x-direction ....................................................................... 55
Figure 5-32: Shell and Solid mesh along with transition shell elements ...................................... 56
Figure 5-33: Transition Elements stepping down from three shell elements to one shell element .......................................................................................................................... 57
Figure 5-34: Transition Elements stepping down from three shell elements to one shell element
in the case of interface elements .................................................................................................. 57
Figure 5-35: Stress plot in the y-direction without Interface Elements ........................................... 58
Figure 5-36: Stress plot in the x-direction without Interface Elements .......................................... 59
Figure 5-37: Stress plot in the y-direction in the case of Interface Elements .................................. 60
Figure 5-38: Stress plot in the x-direction in the case of Interface Elements .................................. 61
Figure 5-39: Solid Model used for Non-Linear Analysis ................................................................. 64
Figure 5-40: Mesh of the symmetric solid model .......................................................................... 65
Figure 5-41: Hyperelastic Material card used in MSC.NASTRAN .............................................. 66
Figure 5-42: Front view of the Cylindrical Model .......................................................................... 67
Figure 5-43: Showing the three surfaces for boundary condition application ............................. 67
Figure 5-44: Isometric View of the Mesh and Boundary Conditions ............................................ 69
Figure 5-45: Side view of the mesh and Force ............................................................................... 69
Figure 5-46: Plot for Maximum Displacement With Interface elements ..................................... 71
Figure 5-47: X-Y graph plot for Load vs. Displacement for Node # 81 .......................................... 72
Figure 5-48: Plot for Maximum Displacement Without Interface elements .................................. 73
CHAPTER 1 INTRODUCTION

When performing global/local analysis, the issue of connecting dissimilar meshes often arises, especially when refinement is performed. One method of connecting these dissimilar meshes is to use interface elements, which have been developed by the NASA Langley Research Center. In MSC.NASTRAN 2001 and thereafter these interface elements have been implemented.

The problem of connecting dissimilar meshes at a common interface is a major one in finite element analysis. Such interfaces can result from a variety of sources, which can be divided into two categories: generated by the analyst, and generated by the analysis program.

Dissimilar meshes generated by the analyst can occur with global/local analysis, where part of the structure is modeled as the area of primary interest, in which detailed stress distributions are required, and part of the structure is modeled as the area of secondary interest. Generally the area of primary interest has a finer mesh than the area of secondary interest, and therefore a transition area is required. Severe transitions generally produce elements that are heavily distorted, which can result in poor stresses and poor load transfers into the area of primary interest.

To illustrate the use of interface elements, an example of an airplane model is given below. Development of an airframe finite element model requires division of engineering labor. Models are often built by two or more companies remotely located,
which model specific sections of the airframe structure. These sections may also be subdivided and the resultant sub-sections assigned to different personnel. Such division of a model requires a final mesh assembly, which will result in having mismatched nodes at the interface of each meshed model.

Figure 1-1: Distinct models on different workstations

Interface elements have recently been implemented in MSC.NASTRAN 2004 to connect two dissimilar meshes. We seek to establish the performance of these elements. The transitions of interest include shell-to-shell or shell-to-solid.
As previously stated, the goal is to evaluate the performance of the interface elements implemented in MSC.NASTRAN 2004. I-DEAS9 in used for the initial model development and I-DEAS9 and PATRAN is used to portray the output graphically.

The major conclusions are as follows:

- Interface elements provided accurate results in the case of a sharp elliptical crack. However, not using interface elements as is common practice gave very inaccurate results. Good practice calls for using interface elements.
- The shell-shell and shell-solid cases gave accurate results using interface elements, provided there is no discontinuity in nodal density.
- If there is a discontinuity in nodal density, very inaccurate results were obtained despite using Interface elements.
- Accurate results were obtained in a nonlinear problem using interface elements.
CHAPTER 2 LITERATURE REVIEW

Since the introduction of the finite element method there has been the need to connect mismatched nodes along the boundary of the element interface. Previously the issue was addressed by moving the nodes or writing multi-point constraint equations on the interfaces. Moving the nodes is very cumbersome and may heavily distort the element. The biggest restriction of moving the nodes is that both sides of the interface must have the same number and type of elements. The other approach of writing multi-point constraint equations has its disadvantages as well. Multi-point constraints equations are used for a node between two nodes, where the node in the center is allowed to slide between these two nodes. However, multi-point constraint equations provide additional relationships for the existing degrees of freedom on the interface, and in the process may result in additional local stiffness.

The new method of connecting mismatched nodes using Interface Elements developed by NASA Langley Research Center has been implemented in MSC.NASTRAN.
CHAPTER 3 MATHEMATICAL FORMULATION OF INTERFACE ELEMENTS

The formulation of the interface elements, which is a hybrid variational formulation using Lagrange multipliers, is shown below.

The displacement vector \( \{v\} \) on the interface is expressed in terms of node and edge coefficients \( \{q_{s}\} \), which are defined on the interface elements, and interpolation function \([T]\), which is a matrix containing the function for each field of the interface displacement vector:

\[
\{v\} = [T] \{q_{s}\}
\]

The displacement vector \( \{u_{j}\} \) on each subdomain \( j \) is expressed in terms of the node and edge coefficients \( \{q_{j}\} \) and interpolation functions \([N_{j}]\), which is a matrix containing the functions for each field of the subdomain displacement vector:

\[
\{u_{j}\} = [N_{j}] \{q_{j}\}
\]

The Lagrange multiplier vector \( \{\lambda_{j}\} \) on each subdomain \( j \) is introduced in terms of the node and edge coefficients \( \{a_{j}\} \) and interpolation functions \([R_{j}]\), which is a matrix containing the functions for each field of the Lagrange multiplier vector:

\[
\{\lambda_{j}\} = [R_{j}] \{a_{j}\}
\]

Defining the combined operator and material matrix \([B_{j}]\), the density \( \rho \), and the surface tractions \( \{t_{j}\} \), and considering the potential energy for all the subdomains \( j \)
together with the internal energy, internal forces, and applied forces, and associating the interface \( I \) with the Lagrange multiplier gives the potential energy as:

\[
\Pi = \sum_j \left[ \frac{1}{2} \int_\Omega u_j^T B u_j \, dA + \frac{1}{2} \int_\Omega u_j^T \rho_j u_j \, dA - \int_\Gamma u_j^T \lambda \, ds + \int_\Gamma \tilde{A}^T_j (v - u_j) \, ds \right]
\]

The internal body forces:

\[
F_j = -\rho_j \ddot{u}_j
\]

have been multiplied by a factor of one half since they are proportional loads. Using the standard assumption of simple harmonic motion for the frequency \( \omega \):

\[
\ddot{u}_j = -\omega^2 u_j
\]

and expanding the vectors into their coefficients and interpolation functions gives:

\[
\Pi = \sum_j \left[ \frac{1}{2} \int_\Omega q_j^T N_j^T G_j N_j q_j \, dA - \frac{1}{2} \int_\Omega q_j^T N_j^T \rho_j \omega^2 N_j q_j \, dA - \int_\Gamma q_j^T \lambda \, ds + \int_\Gamma (q_j^T T - q_j^T N_j^T) R_j \alpha \, ds \right]
\]

Defining the matrices of interpolation functions as

\[
M_j = -\int_\Gamma N_j^T R_j \, ds
\]

\[
G_j = \int T^T R_j \, ds
\]

and substituting them, together with the standard definition of stiffness matrix \([k]_j\), mass matrix \([m]_j\), and load vectors \([f]_j\), into the potential energy gives

\[
\Pi = \sum_j \left[ \frac{1}{2} q_j^T k_j q_j - \frac{1}{2} \omega^2 q_j^T m_j q_j - q_j^T f_j + (q_j^T G_j + q_j^T M_j) \alpha_j \right]
\]

Partitioning the \( q \) into \( q_j \), the node and edge coefficients on the interface, and \( q_o \), the coefficients other than on the interface, gives:
\[ \Pi = \sum_j \left\{ \frac{1}{2} q_j^o q_j^o T \left[ k_j^o \ k_j^i \ k_j^o \ k_j^i \right] q_j^o \right\} - \frac{1}{2} \alpha^2 \left\{ q_j^o q_j^o T \left[ m_j^o \ m_j^o \ m_j^i \ m_j^i \right] q_j^o \right\} \left\{ q_j^r q_j^r T \left[ f_j^r \ f_j^r \right] \right\} \right\} + \left\{ q_j^T \left[ G_j + \left[ q_j^T \left[ M_j \right] \right] \alpha_j \right] \right\} \]

Deriving the Euler equations by taking the variations of the potential energy with respect to the four groups of variables \( q_j^o \), \( q_j^i \), \( q_j^s \), and \( \alpha_j \) gives

\[ \frac{\partial \Pi}{\partial q_j^o} = \left( k_j^o - \omega^2 m_j^o \right) q_j^o + \left( k_j^i - \omega^2 m_j^i \right) q_j^i - f_j^o = 0 \]

\[ \frac{\partial \Pi}{\partial q_j^i} = \left( k_j^o - \omega^2 m_j^o \right) q_j^o + \left( k_j^i - \omega^2 m_j^i \right) q_j^i - f_j^i + M_j \alpha_j = 0 \]

\[ \frac{\partial \Pi}{\partial q_j^s} = \sum_j G_j \alpha_j = 0 \]

\[ \frac{\partial \Pi}{\partial \alpha_j} = G_j^T q_j + M_j^T q_j^i = 0 \]

This system of equations is symmetric, but not positive definite. All the interface terms \([M_j]\) and \([G_j]\) appear in the stiffness matrix, with none in the mass matrix.
CHAPTER 4 SHELL-TO-SHELL INTERFACE ELEMENT

4.1 Introduction

The Shell–to–Shell Interface Elements in MSC.NASTRAN are intended to avoid errors resulting from transitions between two shell meshed regions. The interface elements are introduced at the boundary where the two meshes meet.

Shown in Figure 4-1 are two regions, each having a 4-node quad shell mesh. The region on the left hand side has five elements on its boundary and the region on the right hand side has two elements.

![Figure 4-1: Two regions of unequal shell mesh densities](image)

For purpose of validation, a transition mesh has been created between these two regions to connect all the boundary nodes. As shown in Figure 4-2, 3 node triangular shell elements are used to transition between the five element region and the two element region.
The three node triangular shell elements are known to be stiffer compared to the four noded quad shell elements, and therefore provide conservative results. In addition, if the gradients at the interface are high the transition elements will experience severe distortion and may cause error.

With the implementation of interface elements, the two regions are joined without using transition elements, as illustrated in Figure 4-3.

The interface elements are intended to provide an accuracy-preserving transition between two mismatched nodes at the boundary of two shell-meshed regions.
4.2 Quarter-Plate Model

As a first example for validation, a 0.25 in thick quarter-plate having the dimensions shown below is modeled in I-DEAS9 (Figure 4-4).

![Quarter Plate model with dimensions](image)

**Figure 4-4: Quarter Plate model with dimensions**

The quarter plate FEA model is chosen as it provides an analytical solution, which can be used to compare with the computational results obtained by using interface elements.
4.2.1 Mesh of the Quarter-Plate with Element Ratio 1:1

Four node quad shell elements are used to mesh the plate. Two shell-meshed regions are created in the quarter plate model so as to generate a dissimilar mesh for the use of interface elements.

Figure 4-5: Two shell meshed regions

The boundary of the two regions show mismatched nodes, as shown in Figures 4-6, 4-7, and 4-8.
Figure 4-6: Two shell meshed regions with unmatched nodes
An enlarged view of the mismatched nodes is shown in Figure 4-7.

Figure 4-7: An enlarged view of the mismatched nodes on the top edge

Figure 4-8: An enlarged view of the mismatched nodes on the right edge
4.2.1.1 Boundary Conditions

4.2.1.1.1 Force

An In-plane force of 25 lbf/in is applied on the Element Free Edge at the top edge of the plate. I-DEAS9 input panel is shown in Figure 4-9.

![Figure 4-9: Input Panel in I-DEAS9 for Force](image)

4.2.1.1.2 Constraints

Symmetric conditions are applied on the two edges of the quarter-plate. For the left hand side edge, the constraint specified in the input panel is shown in Figure 4-10.

![Figure 4-10: Input Panel for Constraints on left edge](image)
For the bottom edge of the plate, the applied constraint is shown in the Figure 4-11.

![Figure 4-11: Input Panel for constraints on bottom edge](image)

### 4.2.1.2 Material

Isotropic Steel material is specified for the model, for which the properties are:

- Modulus of Elasticity: \(1\times10^7\) psi
- Poisson's Ratio: 0.333
- Shear Modulus: \(3.75939\times10^6\) psi
- Mass Density: \(0.000732\) lb/ft\(^2\)/in\(^4\)

### 4.2.1.3 Physical Properties

A thickness of 0.25 in is specified for the shell thickness.

The file is now exported to MSC.NASTRAN.
4.2.1.4 Introduction of Interface Elements in Bulk Data Entry in MSC.NASTRAN

After the I-DEAS9 model file is exported to NASTRAN it is necessary to modify the input file within NASTRAN to exercise the interface elements. The following cards are introduced in the Bulk Data Entry of MSC.NASTRAN.

GMBNDC
GMINTC
PINTC

Two interface elements are used in this model because of the sharp 45° boundary at the meeting of the two models.

4.2.1.5 Results

The plot for the stress in y-direction is shown in Figure 4-12.

\[ \sigma_{yy} = 305 \text{ Psi} \]

at the right edge of the hole
The results obtained are valid as they are confirmed by the analytical results.

\[
\frac{\sigma_{yy}}{\sigma_0} = \left(1 + 2 \frac{a}{b}\right) = 3
\]

where;

\(\sigma_0 = 100\)
\(a = b = 0.5\)

Otherwise stated, the interface elements served to compute the correct stress at the edge of the hole. However, the above plots show discontinuity in the stress contours in the region where interface elements are used. It appears that the stresses predicted in this vicinity are unreliable.
4.2.2 Mesh of the Quarter Plate with unequal Elements

The above case is used again, this time, one the regions have a denser mesh as compared to the other. There are 12 elements on the edge of the denser mesh to 8 elements on the corresponding less dense mesh.

Figure 4-13: Mesh of the quarter plate with one region denser than the other
Figure 4-14: A closer look at the denser mesh and the boundary of mismatched nodes

Figure 4-15: An enlarged view of the top edge of the boundary of the mismatched nodes
4.2.2.1 Results

The stress in the y-direction ($\sigma_{yy}$) is obtained, and the results are compared to those obtained in the earlier model as well as the exact solution. The plot of the stress in the y-direction is shown in Figure 4-16.

$$\sigma_{yy} = 462 \text{ Psi}$$

![Figure 4-16: Plot of the Stress in the y-direction](image)

In this case the elements were not in the same ratio along the interface and the results obtained were not accurate.
4.2.3 Mesh of the Quarter Plate Model using Transition Elements

The above quarter plate model is meshed using transition elements. Transition elements help step down from three elements to one element, allowing use of regions of varying nodal densities.

One region of the model is meshed with 8 elements line up on one edge, as shown in Figure 4-17.

![Figure 4-17: Mesh of one region of the quarter plate](image)
The second region is meshed with 24 elements on one edge, as shown in Figure 4-18.

Figure 4-18: Mesh of second region of the quarter plate

The two separately meshed regions are shown in Figure 4-19.

Figure 4-19: Mesh of the two regions of the quarter plate
The two meshed regions are then oriented to line up at their end nodes.

Figure 4-20: Mesh of the two regions brought together
In the first case, the nodes are mismatched and interface elements are used along the boundary of the mismatched nodes. The mismatched nodes are shown in Figure 4-21.

![Figure 4-21: An enlarged view of the mismatched nodes along the boundary of the two regions](image1)

In the second case, the nodes are coincident, so no interface elements are used in this case. The coincident nodes at the boundary are shown in Figure 4-22.

![Figure 4-22: An enlarged view of the transition elements with coincident nodes](image2)
4.2.3.1 Results

The stress for the y-direction in the case of Interface elements is plotted. The plot is shown in Figure 4-23.

\[ \sigma_{yy} = 524 \text{ psi} \]

Figure 4-23: Plot of the stress in the y-direction using interface elements

The stresses not only show discontinuity in the contours, as well as do not provide good results either.
The stress in the y-direction in the case of without Interface elements and using Transition elements is plotted. The plot is shown in Figure 4-24.

\[ \sigma_{yy} = 307 \text{ psi} \]

**Figure 4-24: Plot of the stress in the y-direction without interface elements**

The stress contours are smooth and continuous and good results are obtained.
The summary of the results obtained is shown in Table 4-1.

Table 4-1: Summary of Results obtained for the GMINTC element

<table>
<thead>
<tr>
<th>Stress Direction</th>
<th>Exact Results – Analytical Solution (psi)</th>
<th>Results obtained using Interface Elements (psi)</th>
<th>% Error</th>
<th>Results obtained using Interface Element (nodal density discontinuity)</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma_{yy}$</td>
<td>300</td>
<td>305</td>
<td>1.6 %</td>
<td>462</td>
<td>35 %</td>
</tr>
</tbody>
</table>

Summary of Results with Transition Elements:

Table 4-2: Summary of Results obtained for the GMINTC element with Transition Elements

<table>
<thead>
<tr>
<th>Stress Direction</th>
<th>Exact Results – Analytical Solution (psi)</th>
<th>Results obtained using Interface Element and Transition Elements (Ratio of 3:1)</th>
<th>% Error</th>
<th>Results obtained using Transition Elements only (Ratio 3:1)</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma_{yy}$</td>
<td>300</td>
<td>524</td>
<td>42 %</td>
<td>307</td>
<td>2.2 %</td>
</tr>
</tbody>
</table>

### 4.3 Conclusion

The interface element improved accuracy when the two connected regions have the same mesh density. When there is a density mismatch, we did not obtain good results, nor did we obtain good results after introducing transition elements to overcome the mesh discontinuity.
5.1 Introduction

Shell–to–Solid Interface Elements have been introduced in MSC.NASTRAN to achieve an accurate transition between shell and solid elements. The ability to do so is a major potential benefit, since, in practice, shell and solid elements are often connected without use of interface elements. Eight node brick solid elements and a four node quad shell elements are shown in Figure 5-1. It should be noted that the shell element nodes and the solid element nodes do not 'match up'.

The shell nodes are located at the mid points of two solid nodes.

![Figure 5-1: Illustrating the position of shell and solid nodes](image)

Interface elements are used between two solid nodes and one shell node.
5.2 Cantilever Beam Model

For performance evaluation by comparison with an exact solution, an FEA model consisting of solid and shell elements in the form of a cantilever beam is created, with one end clamped and load imposed on the end node. The block on the right hand side has the dimensions 6x6x2 inches and is meshed with four node brick solid elements. The 40 in long cantilever beam is meshed with shell elements.

Figure 5-2: Cantilever beam model

On one far side at the end of the solid elements the nodes are clamped, and at the tip of the cantilever beam a net force of 100 lbs is applied in the downward direction. Two interface elements are placed between the two solid nodes and the shell node.
5.2.1 Results

The results obtained are listed below:

Maximum Stress; $\sigma_{\text{max}} = 930 \text{ psi}$

Maximum Deflection; $\delta_{\text{max}} = 0.0183 \text{ in}$

The Displacement Plot is depicted in Figure 5-3.
MSC.Patran is used for post-processing to visualize the stress contours.

Figure 5-4: Non-layered Stress plot in MSC.NASTRAN

Figure 5-5: Stress plot in MSC.NASTRAN
5.2.2 Verification of Results

The Stress and the Displacement values are verified analytically by Classical beam theory. The applicable formulae are as follows

**Stress**

\[
\sigma_{\text{max}} = \frac{MC}{I}
\]

\[
\sigma_{\text{max}} = \frac{F d C 12}{bh^3}
\]

\[
\sigma_{\text{max}} = \frac{(100)(40)(1)(12)}{(6)(2)^3}
\]

\[
\sigma_{\text{max}} = 1000 \text{ psi}
\]

Percent Error = 7 %

**Deflection**

\[
\delta_{\text{max}} = \frac{PL^3}{3EI}
\]

\[
\delta_{\text{max}} = \frac{(100)(40)^3(12)}{(3)(30 \times 10^9)(6)(2)^3}
\]

\[
\delta_{\text{max}} = 0.0177 \text{ in}
\]

Percent Error = 3%

Table displays the summary of the above results.

<table>
<thead>
<tr>
<th></th>
<th>Interface Elements</th>
<th>Analytical Solution</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stress (psi)</td>
<td>930</td>
<td>1000</td>
<td>7 %</td>
</tr>
<tr>
<td>Deflection (in)</td>
<td>0.0183</td>
<td>0.0177</td>
<td>3 %</td>
</tr>
</tbody>
</table>
5.3 Quarter Plate Model

The quarter plate discussed in the earlier chapter is again invoked for the RSSCON (shell-solid interface element). Here, three cases as follows are presented.

Case 1: a = 0.5, b = 0.5  
Case 2: a = 0.25, b = 0.5  
Case 3: a = 0.05, b = 0.5

where, ‘b’ and ‘a’ are the major and minor axes to the circular/elliptical hole in the center of the 12 x 12 inch symmetric plate. Note that the ellipse approximates a crack in Case 3.

As earlier, owing to symmetry the quarter plate is meshed and symmetric conditions are applied to the two edges of the plate. A force of 25 lbs is applied on the top edge of the plate, and isotropic steel material properties are used.

5.3.1 Quarter Plate Model - Case 1

The 6 x 6 in quarter plate with a center hole cut-out of 0.5 in radius is shown in Figure 5-6.
Figure 5-6: Isometric view of the quarter plate showing the two regions; surface and volume

The 2 x 2 in area surrounding the circular hole in the plate is meshed with 4 node solid brick elements, and the remaining area is meshed with four node quad shell elements. Two interface elements are placed on the boundary where the shell elements meet the solid elements.

This is accomplished by using the appropriate cards in the Bulk Data entry of the .dat file for MSC.NASTRAN.
Symmetric boundary conditions enforced are shown in Figure 5-8.

Figure 5-8: Meshed plate with Boundary conditions applied
5.3.1.1 Results

Result obtained in I-DEAS for the Stress in the y-direction is shown in Figure 5-9 and are included in Table 5.2 at the end of this chapter.

\[ \sigma_{yy} = 296 \text{ Psi} \]

Figure 5-9: Plot of the Stress in the y-direction
The results obtained and the stress for the plot of the stress in the x-direction is shown in Figure 5-10.

\[ \sigma_{xx} = 90 \text{ Psi} \]

Figure 5-10: Plot of the stress in the x-direction
The same case is repeated, only this time interface elements are not used, and the solid elements are directly connected with shell elements, as none of the nodes are mismatched.

Figure 5-11: View of the mesh without interface elements
The stress plot in the y-direction is shown in Figure 5-12 as well as tabulated in Table 5.2.

\[ \sigma_{yy} = 300 \text{ Psi} \]

**Figure 5-12: Stress Plot in the y-direction without Interface Elements**

Though the stress results are the same as that obtained with the interface elements, it should be noted that the stress contours obtained with interface elements are far superior to those obtained without the interface elements. In the above plot there seems to be a discontinuity in the stress distribution along the line where the solid and shell elements are connected together.
An enlarged view of the area of concern is shown in Fig 5-13.

Figure 5-13: Enlarged view of the area of concern

For the sake of comparison between the stress contours for the interface elements to those without interface elements, the two contours are shown side by side in Fig 5-14.

Figure 5-14: Stress Plot; Without Interface element (left) and with interface elements (right)
The results obtained and the stress for the plot of the stress in the x-direction is shown in Figure 5-15.

\[ \sigma_{xx} = 94 \text{ Psi} \]

Figure 5-15: Plot of the stress in the x-direction
5.3.2 Quarter Plate Model - Case 2

The solid model of the 6 x 6 in quarter plate with an elliptical center hole cut-out of 0.5 in and a 0.25 in, major and minor axis respectively, is now depicted in Figure 5-16.

Figure 5-16: Quarter Plate Model for Case 2
The section closer to the hole is meshed with four node solid brick elements and the remainder is meshed with four node quad shell elements. The boundary conditions and the force are the same for all the three cases.

Figure 5-17: Quarter Plate Model mesh with Boundary Conditions
5.3.2.1 Results

The results obtained in I-DEAS for the stress in the y-direction are shown in Figure 5-18 and are tabulated in Table 5.2.

\[ \sigma_{yy} = 486 \text{ Psi} \]

Figure 5-18: Stress Plot in the y-direction
The results obtained and the stress for the plot of the stress in the x-direction is shown in Figure 5-10.

\[ \sigma_{xx} = 100 \text{ Psi} \]

**Figure 5-19: Plot of the stresses in the x-direction**
The above case is repeated without the use of interface elements.

\[ \sigma_{yy} = 530 \text{ psi} \]

Figure 5-20: Stress Plot in the y direction without interface elements

An enlarged picture of the area of concern is shown in Figure 5-21.

Figure 5-21: Enlarged view of the area of concern
For the sake of comparison between the stress contours for the interface elements to without interface elements, the two contours are shown side by side in Figure 5-22.

![Figure 5-22: Stress Plot; Without Interface element (left) and with interface elements (right)](image)

Similar discontinuity in the stress contours have been observed in the plots obtained from the models without interface elements.
The results obtained and the stress for the plot of the stress in the x-direction is shown in Figure 5-23.

\[ \sigma_{xx} = 106 \text{ Psi} \]

Figure 5-23: Plot of stresses in the x-direction
5.3.3 Quarter Plate Model - Case 3

The solid model of the 6 x 6 in quarter plate with an elliptical center hole cut-out of 0.5 in and a 0.05 in, major and minor axis respectively, is shown in Figure 5-24.

Figure 5-24: Quarter Plate model for Case 3
The mesh of the quarter plate with the symmetric boundary conditions and edge force is shown in Figure 5-25.

Figure 5-25: Mesh of Quarter Plate Model Case 3 with Boundary Conditions
5.3.3.1 Results

The result obtained in I-DEAS for the stress in the y-direction is depicted in Figure 5-26, and listed in Table 5.2.

\[ \sigma_{yy} = 2160 \text{ Psi} \]

Figure 5-26: Stress plot in the y-direction
The results obtained and the stress for the plot of the stress in the x-direction is shown in Figure 5-27.

\[ \sigma_{xx} = 102 \text{ Psi} \]

Figure 5-27: Stress plot in the x-direction
Similarly, for the third case, the interface elements were removed. The results obtained are shown in Figure 5-28.

\[
\sigma_{yy} = 1340 \text{ Psi}
\]

Figure 5-28: Stress Plot in the y direction without interface elements
An enlarged picture of the area of concern is shown in Figure 5-29.

![Figure 5-29: Enlarged view of the area of concern](image)

For the sake of comparison between the stress contours for the interface elements to without interface elements, the two contours are shown side by side in Figure 5-30.

![Figure 5-30: Stress Plot; Without Interface element (left) and with interface elements (right)](image)
Discontinuity in the stress contours obtained from the plots of the earlier two cases has again been observed from the model without interface elements.

The results obtained and the stress for the plot of the stress in the x-direction is shown in Figure 5-31.

\[ \sigma_{xx} = 105 \text{ Psi} \]

Figure 5-31: Plot of the stresses in the x-direction
5.3.4 Transition Elements

The quarter plate model – case 1; is meshed using transition elements on the side of the shell meshed region. Transition elements help step down from three elements to one element.

Figure 5-32: Shell and Solid mesh along with transition shell elements
In the second case, the transition shell element nodes are not coincident to the solid element nodes and therefore the interface element is used. An enlarged view of the mesh at the boundary is shown in Figure 5-34.
5.3.4.1 Results

The stress contours are depicted in Figure 5-35 and numerical values of $\sigma_{yy}$ at the tip of the hole are given in Table 5.2

$$\sigma_{yy} = 306 \text{ Psi}$$

Figure 5-35: Stress plot in the y-direction without Interface Elements
The results obtained and the stress for the plot of the stress in the x-direction is shown in Figure 5-36.

\[ \sigma_{xx} = 99.5 \text{ Psi} \]

Figure 5-36: Stress plot in the x-direction without Interface Elements
Stress produced in the y-direction in the case of Interface Elements is shown in Figure 5-37.

\[ \sigma_{yy} = 295 \text{ Psi} \]

Figure 5-37: Stress plot in the y-direction in the case of Interface Elements
Stress produced in the x-direction in the case of Interface Elements is shown in Figure 5-38.

\[ \sigma_{xx} = 94.7 \text{ Psi} \]

**Figure 5-38: Stress plot in the x-direction in the case of Interface Elements**
5.3.4.2 Verification

The following Tables show the comparison between the results obtained using interface elements and those obtained without interface elements to the analytical solution.

\[
\frac{\sigma_{yy}}{\sigma_o} = (1 + 2 \frac{a}{b}) \\
\sigma_{xx} = \sigma_o
\]

The summary of results for the stresses in the y-direction is shown in Table 5-2 and Table 5-3.

### Table 5-2: Summary of Results obtained with the RSSCON element

<table>
<thead>
<tr>
<th>CASE NO.</th>
<th>Exact Results – Analytical Solution (Psi)</th>
<th>Results obtained using Interface Elements (Psi)</th>
<th>% Error</th>
<th>Results obtained without using Interface Elements (Psi)</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>300</td>
<td>296</td>
<td>1.3 %</td>
<td>300</td>
<td>0 %</td>
</tr>
<tr>
<td>2</td>
<td>500</td>
<td>486</td>
<td>2.8 %</td>
<td>530</td>
<td>6.0 %</td>
</tr>
<tr>
<td>3</td>
<td>2100</td>
<td>2160</td>
<td>2.7 %</td>
<td>1340</td>
<td>36.2 %</td>
</tr>
</tbody>
</table>

In the case of using Transition Elements,

### Table 5-3: Summary of Results obtained with the RSSCON element in the case of transition elements

<table>
<thead>
<tr>
<th>CASE NO.</th>
<th>Exact Results – Analytical Solution (Psi)</th>
<th>Results obtained using Interface Elements (Psi)</th>
<th>% Error</th>
<th>Results obtained without using Interface Elements (Psi)</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>300</td>
<td>295</td>
<td>1.6 %</td>
<td>306</td>
<td>1.9 %</td>
</tr>
</tbody>
</table>
The summary of results for the stresses in the x-direction is shown in Table 5-4 and Table 5-5.

Table 5-4: Summary of Results obtained with the RSSCON element

<table>
<thead>
<tr>
<th>CASE NO.</th>
<th>Exact Results – Analytical Solution (Psi)</th>
<th>Results obtained using Interface Elements (Psi)</th>
<th>% Error</th>
<th>Results obtained without using Interface Elements (Psi)</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>100</td>
<td>90</td>
<td>10 %</td>
<td>94</td>
<td>6 %</td>
</tr>
<tr>
<td>2</td>
<td>100</td>
<td>100</td>
<td>0 %</td>
<td>106</td>
<td>6 %</td>
</tr>
<tr>
<td>3</td>
<td>100</td>
<td>102</td>
<td>2 %</td>
<td>105</td>
<td>5 %</td>
</tr>
</tbody>
</table>

In the case of using Transition Elements,

Table 5-5: Summary of Results obtained with the RSSCON element in the case of transition elements

<table>
<thead>
<tr>
<th>CASE NO.</th>
<th>Exact Results – Analytical Solution (Psi)</th>
<th>Results obtained using Interface Elements (Psi)</th>
<th>% Error</th>
<th>Results obtained without using Interface Elements (Psi)</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>100</td>
<td>94.7</td>
<td>5.3 %</td>
<td>99.5</td>
<td>0.5 %</td>
</tr>
</tbody>
</table>

5.3.5 Conclusion

Models with and without interface elements gave correct stress concentration factors (= 3) for the cases (1, 2). However the stresses without interface elements showed severe stress discontinuities along the interface. Furthermore, in Case 3, representing the highest stress concentration factor, the results without interface elements were very inaccurate, while the results with interface elements were quite good.
For Case 1, the circle, using transition elements provided accurate results with and without transition elements.

5.4 Non-linear Model

5.4.1 Introduction

The goal is to use interface elements in a non-linear analysis using Arc Length Method. The solid model shown below with its dimensions is modeled in I-DEAS. The Model consists of a large rubber cylinder having a rigid steel sleeve, along with a smaller steel cylinder (rod). The dimensions used towards the creation of the solid model are:

- Cylinder 1: \( r = 5 \) in, \( L = 10 \) in
- Cylinder 2: \( r = 4 \) in, \( L = 10 \) in
- Cylinder 3: \( r = 1 \) in, \( L = 20 \) in

Figure 5-39: Solid Model used for Non-Linear Analysis

The quarter of the solid model is used, making use of symmetry conditions.
5.4.2 Finite Element Model

The solid model is portioned such that mapped-meshing can be performed in I-DEAS9 on the quarter model. Mapped meshing is performed using solid and shell elements.

Total No of Nodes: 423
Total No of Solid Elements: 256
Total No of Shell Elements: 48

The outer cylinder or sleeve is meshed using shell elements, whereas the rest of the quarter model is meshed using solid elements. Interface elements are used at the front and rear of the model where the shell and solid elements meet.

Figure 5-40: Mesh of the symmetric solid model
5.4.2.1 Material Properties:

5.4.2.1.1 Rubber

The Mooney-Rivlin rubber material properties are:

\[ A_{10} = 25.67 \text{ Psi} \]
\[ A_{01} = 6.52 \text{ Psi} \]

The material card used in the analysis of fully non-linear hyperelastic materials like rubber is shown in Figure 5-41.

![Hyperelastic Material card used in MSC.NASTRAN](image)

The two Mooney-Rivlin constants related to distortional deformation are used.

5.4.2.1.2 Steel

The default material property of Steel is used in I-DEAS

Elastic Modulus; \( E = 3 \times 10^7 \text{ Psi} \)
Poisson’s Ratio; \( \nu = 0.3 \)
Shear Modulus; \( G = 1.16 \times 10^7 \text{ Psi} \)
5.4.2.2 Boundary Conditions:

The Boundary Conditions applied on the quarter solid-model are as follows:

Figure 5-43: Showing the three surfaces for boundary condition application
Using the Front View of the model as shown in Figure 5-43, the boundary conditions applied on the nodes on each surface, listed above, are as follows:

**Surface 1**
Clamped Restraint:
Tx = Ty = Tz = Rx = Ry = Rz = 0

**Surface 2:**
Specified Restraint:
Tx = Tz = Ry = free
Ty = Rx = Rz = fixed

**Surface 3:**
Specified Restraint:
Ty = Tz = Rx = free
Tx = Ry = Rz = fixed

In which;
T = Translations in x, y, and z.
R = Rotations in x, y, and z.
Figure 5-44: Isometric View of the Mesh and Boundary Conditions

A net force of 5000 lbf is applied on the nodes is shown in Figure 5-45.

Figure 5-45: Side view of the mesh and Force
5.4.2.3 I-DEAS9 to MSC.NASTRAN

After meshing, applying material properties, and boundary conditions, the FE model is now exported from I-DEAS9 as a .DAT file.

The cards used for the non-linear analysis are as follows:

“Executive Control” Section of the .DAT file:

SOL 106

“Case Control” Section of the .DAT file:

MPC = 1
NLPARM = 121

“Bulk Data” Section of the .DAT file:

<table>
<thead>
<tr>
<th>NLPARM</th>
<th>121</th>
<th>480</th>
<th>AUTO</th>
<th>1</th>
<th>YES</th>
</tr>
</thead>
<tbody>
<tr>
<td>NLPCI</td>
<td>121</td>
<td>CRIS</td>
<td>0.25</td>
<td>4.0</td>
<td>10000.0</td>
</tr>
<tr>
<td>MATHP</td>
<td>2</td>
<td>25.67</td>
<td>6.52</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
5.4.2.4 Results:

A plot showing the maximum displacement is shown in Figure 5-46.

![Figure 5-46: Plot for Maximum Displacement With Interface elements](image)

The x-y graph for load vs. displacement is plotted for the node (# 81) experiencing maximum displacement in the model. The plot is shown in Figure 5-47.
Figure 5-47: X-Y graph plot for Load vs. Displacement for Node # 81

Though the x-y graph plot for load vs. displacement shows a linear curve, the visual inspection of the displacement shows a large deformation for the body with rubber material properties.

Interface elements were tested in the non-linear range and there seemed to be no conflict in the NASTRAN software in the use of interface elements for a non-linear solution.
5.4.2.5 Non-Linear model Without Interface Elements

In the next example, the interface elements were removed and the solid elements were directly connected to the shell elements.

![Plot for Maximum Displacement Without Interface elements](image)

Figure 5-48: Plot for Maximum Displacement Without Interface elements

Identical deformation results were obtained. This indicates that interface elements may be used without reservations in non-linear analysis.
CHAPTER 6 DISCUSSION

The results obtained from the shell-shell interface element GMINTC for the quarter plate model for the two regions having mismatched nodes show that when the regions have the same number of elements in both the regions provide valid results. The stress values in the case of the quarter plate model were verified by the analytical solution. In the case of interface elements used at the boundary of a dense mesh to a coarse mesh, valid results were not obtained.

- In the case of the shell-to-solid interface element RSSCON, for the cantilever beam model, the results obtained for the maximum stress and displacement were validated analytically by classical beam theory.

- For three cases for the quarter plate model having solid elements in the area of the cut and shell regions in the outer areas, the results were again compared with analytical solution. The stress contours showed a good distribution along the interface which was not achieved with the GMINTC element. Poor results were obtained in the sharp elliptical hole when interface elements were not used, but very good results when they were.

- The interface elements were also used in a non-linear model having Mooney-Rivlin rubber material properties and mismatched nodes. The interface elements gave the correct answer, validating that they are applicable to nonlinear analysis.
• Interface elements for solid to shell elements were found to provide more accurate results as compared to the interface elements for shell to shell elements.

• All boundary identification numbers must be unique.

• Interface elements may generate high or negative matrix/factor diagonal ratios. If there are no other modeling errors, these messages may be ignored and PARAM, BAILOUT,-1 may be used to continue the run, although this is highly discouraged, and this parameter was not used in any of the analyses carried out.

6.1 For the shell-to-shell interface element – GMINTC

The first and the last node on the interface boundary for two shell meshed regions should be the same in the case of GMINTC element.

For the shell-to-shell interface element, three new cards were introduced in the Bulk Data section, GMINC (Geometric Interface Curve), GMBNDC (Geometric Boundary Curve), and PINTC (Properties of Geometric Interface Curve), where they define the interface curves, nodes making up the boundary, and properties for the interface element.

The interface elements consists only of the difference in displacement components weighted by Lagrange multipliers, there are no conventional element or material properties. For the GMINTC element, the bulk data entry for property specifies a tolerance for the interface
element, which defines the allowable distance between the boundaries of the subdomains. If the distance is greater than the tolerance value, a warning message will be issued.

Interface elements should be applied along a straight edge. If there is a 45 degree edge, two interface elements should be used.

6.2 For the shell-to-solid interface element – RSSCON

For the shell-to-solid interface element, the RSSCON shell-solid element connector card is introduced in the Bulk Data section. For every two solid element nodes, one shell element node located in the middle is specified in this card.

The shell node musts lie between the two solid nodes for the RSSCON interface elements.
CHAPTER 7 CONCLUSION

The performance of interface elements was demonstrated in finite element methods on different models for varied cases. The results obtained were then validated by analytical results, and also each interface element model was compared to a similar model without using interface elements.

Interface elements generally gave better results and smoother stress distributions compared with not using interface elements when there were nodal or element mismatches.

We were not able to obtain accurate results when nodal density changes at an interface in the case of shell-shell.
REFERENCES


David Nicholson, “Finite Element Analysis Thermomechanics of solids”

I-DEAS Online Help Bookshelf

John E. Schiermeier, Jerrold M. Housner, M. A. Aminpour, W. Jefferson Stroud, “The application of interface elements to dissimilar meshes in global/local analysis”

M.A. Aminpour, Jonathan B. Ransom, Susan L. McCleary, “A coupled analysis method for structures with independently modeled finite element subdomains”


Michael Farley, “Redefining the Process of Airframe Finite Element Model Development using MSC/Super Model”

MSC.NASTRAN Quick Reference Guide

O.C. Zienkiewicz, “The Finite Element Method”
