CFD DATA TRANSFER TO STRUCTURAL ANALYSIS

W. Scott Taylor

Sverdrup Technology, Inc.
620 Discovery Drive
Huntsville, Alabama 35806

ABSTRACT

This paper describes the development of a procedure to transfer computational fluid dynamics (CFD) results entities to structural analysis. This procedure is a subset of a larger effort at NASA/Marshall involving interdisciplinary data transfer between a number of traditionally somewhat isolated disciplines. A brief discussion of that effort will also be included. The specifics of translating CFD structured grid results entities in Plot-3D binary format to a dissimilar finite element mesh for load re-interpolation permitting subsequent structural analysis is demonstrated. MSC/PATRAN Command Language was used to automate various features of this capability. The procedure resulted in a major productivity enhancement due to the fact that previously, there was no convenient method to get vehicle or other complex geometry CFD results on to structural analysis models. The procedure is being used routinely for similar interdisciplinary data transfers.

INTRODUCTION

This report describes the process developed at NASA/MSFC by Sverdrup Technology, Inc., to electronically transfer computational fluid dynamics (CFD) results to other disciplines such as structural analysis [1]. The process described in this report utilizes the finite element pre/post processor MSC/PATRAN. It is based on a MSC/PATRAN Command Language (PCL) routine written to facilitate the selection and translation of node points for CFD results recovery. The transfer process utilizes a code called INTFEM[2], a generalized multi-zone 2-D & 3-D interpolation code. Once the node point result values have been interpolated they are written out in MSC/PATRAN results format. Those results are then read into MSC/PATRAN and displayed as fringe contour plots. Immediately afterwards a 'FEM Field' is created which defines a table of values from which appropriate boundary conditions can be interpolated.

Recently, NASA/MSFC has conducted a series of interdisciplinary data transfer investigations aimed at fostering communication between the various disciplines involved in the design, analysis, and test of spacecraft launch and deployment structures. As is typical with large organizations a certain amount of redundant effort has accrued during the normal exercise of doing business. The interdisciplinary data transfer program teams initially gathered information about the software tools and methods employed in each discipline. For example, design, CFD, thermal, stress, life assessment, dynamics, and structural test. Of primary interest was how each discipline was sharing pertinent data with other related disciplines. A great deal of information was obtained and data transfer methods where either developed or optimized. A case in point is the subject of this report.

Ideally, structural design transfers appropriate surface definition to the CFD group via IGES. Most if not all the software tools employed by CFD includes IGES support. After reading the IGES surfaces into a structured grid generation package the CFD analyst proceeds to generate a 2-D or 3-D grid. The grid generation usually follows a strict 'ijk' numbering permutation. A view showing the CFD SSTO model and one 'i-j' plane of a 3-D grid network is shown in Figure 1. Our goal is to scavenge result entities on the vehicle exterior and re-apply those values to a dissimilar finite element nodal/elemental network.

This process would be very simple if the structural analyst was able to use the same IGES surfaces to generate a finite element mesh as the CFD analyst. The case that is demonstrated next does not enjoy any of the ideal conditions that would make this type of data transfer straight forward. It does demonstrate the robustness of the procedure.

PROBLEM DEFINITION

In the present case, a finite element model of a SSTO Vehicle was delivered by NASA/Langley to the Dynamics group at NASA/MSFC. The NASA/MSFC CFD analyst had IGES geometry of a similar but different configuration of the above SSTO Vehicle. The finite element mesh is shown in Figure 2. Noticeably absent are the payload bay doors and fuel tank access doors. These where added later. Since the goal is to get the external field variables from the CFD analysis on to the finite element mesh a group containing the external structural elements and nodes of the SSTO was created. That group is shown in Figure 3. As was mentioned above, it was known in advance that the two models contained slightly dissimilar geometries especially along the exterior surface of the vehicle. In an effort to determine the quantitative difference between the two a FORTRAN program was written to read the CFD grid point data in Plot3-D format and write them out into MSC/PATRAN readable Neutral File format (ASCII based). Those grid points were then read into the data base that contained the finite element mesh and examined for disparities. A front view of the SSTO vehicle with the CFD grid points superimposed on the mesh is shown in Figure 4. The offset is clearly visible. For the interpolation program to generate accurate data it is essential to have surface definitions that are as nearly coincident as possible. Due to symmetry considerations the CFD model only contains half of the SSTO outer surface. It can also be seen that the wing length of the CFD model is shorter than the structural model and that it is displaced downward.

The problem is now two-fold. The general problem (assuming coincident exterior surface geometry) is to take the finite element node points and use their geometric positions as interpolation points in the Plot-3D binary CFD results file. It is clear from Figure 4 that to position the finite element nodes in the CFD results field properly they will have to be displaced upwards some distance. In so doing, the winglet region will be moved further away from the corresponding CFD grids. To alleviate this problem it was decided to break the selection of finite element nodes into two sets. The first would contain the fuselage and wing nodes (without the winglet). The second would have only the winglet nodes. The vertical offset distance was taken as the distance from the lowest CFD grid point to a point at the base of the wing at the wing-to-fuselage interface. The horizontal offset was determined as the distance from a CFD grid point at midspan on the inboard side to a corresponding node on the finite element mesh.

METHODOLOGY

A MSC/PATRAN PCL was written to facilitate the selection of node sets and to write them out in any format in a robust manner. The MSC/PATRAN Command Language can be used to generate form widgets with various types of user interaction features, e.g. toggle buttons and select databoxes. Also, there are a number of means through which a user written PCL can be invoked. In this instance, it was decided to provide an access point in the top level menu bar in MSC/PATRAN3 under the ‘Shareware’ pull-down menu.

In Figure 5 the ‘Shareware’ pull-down menu is shown invoked with the first cascade item displaying the label ‘Nodes Out -->‘. The resulting form widget is shown partially covered by the pull-down menu. In Figure 6 the ‘Nodes Out Form’ has been moved up with the mouse for clarity. This is now a user interface allowing access to the MSC/PATRAN3 database for selecting any arbitrary set of nodes even if there are other entity types in the display. In the first text entry area the user is prompted to give a file name for the nodes to be written to. In the next text entry area surrounded by a ‘select box’ is where the nodes that have been selected will be arranged in ascending order. Recall that two files need to be created because of the nature of the discrepancies between the two geometries. The file name in this case was chosen to be ‘FuseNwing_Nodes’. By placing the mouse in the select data box and then moving over the location of the desired fuselage and wing nodes those entities can be selected. A triangular polygon has been created around the nodes of interest. When the polygon is closed any node inside that region is selected and displayed in the select data box text area. Once the user is satisfied that all the nodes of interest have been selected the apply button can be activated. At that point the nodes are written out to a file in ASCII format with a file header, date, time stamp, and number of nodes written. A similar procedure is repeated for the winglet nodes as shown in Figure 7. In that case a new file name is given, 'Winglet_Nodes' and only the winglet nodes were selected.

These two files containing the fuselage-wing and winglet nodes are used by INTFEM as input. INTFEM is also given the offsets that were found previously. INTFEM then takes each node position data and finds which cell it is in, projects it onto the external surface and then interpolates to the nearest grid point. The node ID and result entities are then written back out in a MSC/PATRAN results file ASCII format. This is done for each file of nodes. The files can then be concatenated using a standard editor or appropriate UNIX commands to create one file that has both the fuselage-wing and winglet node ID's and results entities. In this case the pressures on the vehicle at Max Q where of interest. A Plot3-D image is show in Figure 8 where the pressure contours are given in psf.

The user must now read the results file into the MSC/PATRAN database and display them in a fringe contour plot. In this case the pressures are read via a displacement template. Figure 9 displays a fringe contour plot of the x component of the displacement template results data. Typically, a displacement results format contains 6 columns of data, 3 translations and 3 rotations. For this procedure two data types were actually recovered from the Plot3-D results file. They were pressure and temperature, both scalar variables. If the temperature profile was desired then the y component would be plotted.

By generating a fringe contour plot an internal table or array is created by MSC/PATRAN which can then be used to create a 'FEM Field'. All that is necessary to accomplish this is shown in Figure 10. By selecting the 'Fields' radio button from the MSC/PATRAN top level menu bar the 'Fields' form is displayed. The user merely gives the field a name, selects 'Continuous' in the 'Fem Field Definition' and 'Scalar' in the 'Field Type' regions. Finally, the appropriate group is selected and then 'Apply' is mouse activated. That completes the generation of the 'Fem Field' which will be used as a look-up table for pressure load generation.

To generate the pressure load condition the user selects the 'Load/BC's' button in the top level menu bar show in Figure 11. Once the Load/Boundary Conditions form is displayed the user toggles to the 'Pressure' object and then enters a name in the 'New Set Name' text data box. The ‘Target Element Type’ is: 2-D. The previously created field gets referenced in the next step. Select the ‘Input Data...’ button near the bottom of the form. The ‘Input Data’ form appears to the left of the first form. The user places the mouse in the ‘Top Surf Pressure’ text entry region and then clicks on the ‘Pres_From_CFD string in the ‘Spatial Fields’ box. Select the ‘Select Application Region’ button below the ‘Input Data...’ button. The left most form appears as show in Figure 11. The ‘Fem’ button is shown selected and all the elements in the display have been selected. The user must push the ‘Add’ button for those elements to actually be selected. Once those elements show up in the ‘Application Region’ box the user completes the pressure load generation by selecting the ‘Apply’ button on the main form.

To verify that the fem field interpolation was completed correctly the user can select the ‘Plot Contours’ in the Action button and ‘Pressure’ in the Object button. Select the appropriate Set and Data Variable (see Figure 12.) and Group. Hitting the ‘Apply’ button initiates the pressure load interpolation from the fem field nodal data to element centroids. A carpet plot of the pressure values is shown and the values are given in psi.

DISCUSSION

A close inspection of the color pressure contours in Figure 8 against those in Figure 12. indicates that a faithful mapping of the pressure values and gradients are maintained in the translation process.

CONCLUSIONS

A robust procedure has been developed to translate CFD results data to MSC/PATRAN structural analysis models. The example demonstrated takes only 20 minutes start to finish. Compared to the laborious manual effort required previously this new procedure constitutes a time savings by a factor of 10.

As a result of NASA initiated interdisciplinary team investigations set up to foster communication and optimize data transfer an existing FORTRAN code called INTFEM was identified. Originally, it was created to interpolate CFD solutions or other functions from single- or multiple-zone structured grids of arbitrary size. A team member of that committee was also a proficient MSC/PATRAN user and saw the possibly of efficient data transfer. INTFEM was subsequently slightly modified for I/O and translations of input nodes. A MSC/PATRAN PCL was written to facilitate the selection and translation of structural analysis data to INTFEM for results interpolation. The procedure is finally successful due to the built-in capability in MSC/PATRAN called 'FEM Fields' which allows for re-interpolation of externally generated results quantities.

ACKNOWLEDGMENTS

The author wishes to thank Larry Pearce of The MacNeal-Schwendler Corporation who's expertise in PCL helped this effort.

REFERENCES

[1] Taylor, W. Scott, "CFD Data Transfer To Structural Analysis," Sverdrup Technology, Inc. Report No. 612-016-94-004.

[2] Fay, John F., "Programs 2DINTERP And 3DINTERP User's Manual," Sverdrup Technology, Inc. Report No. 631-003-035.

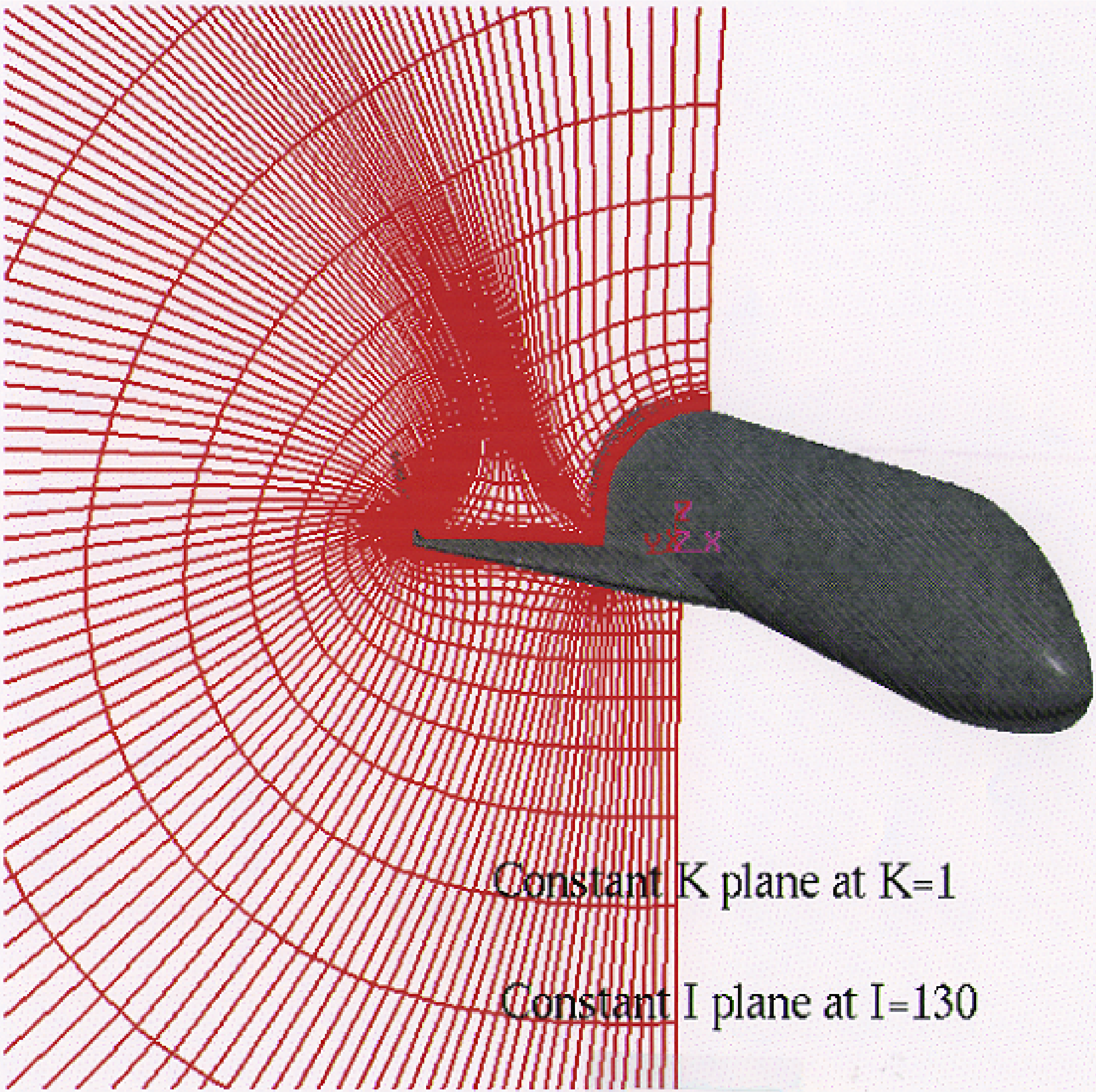


Figure 1.

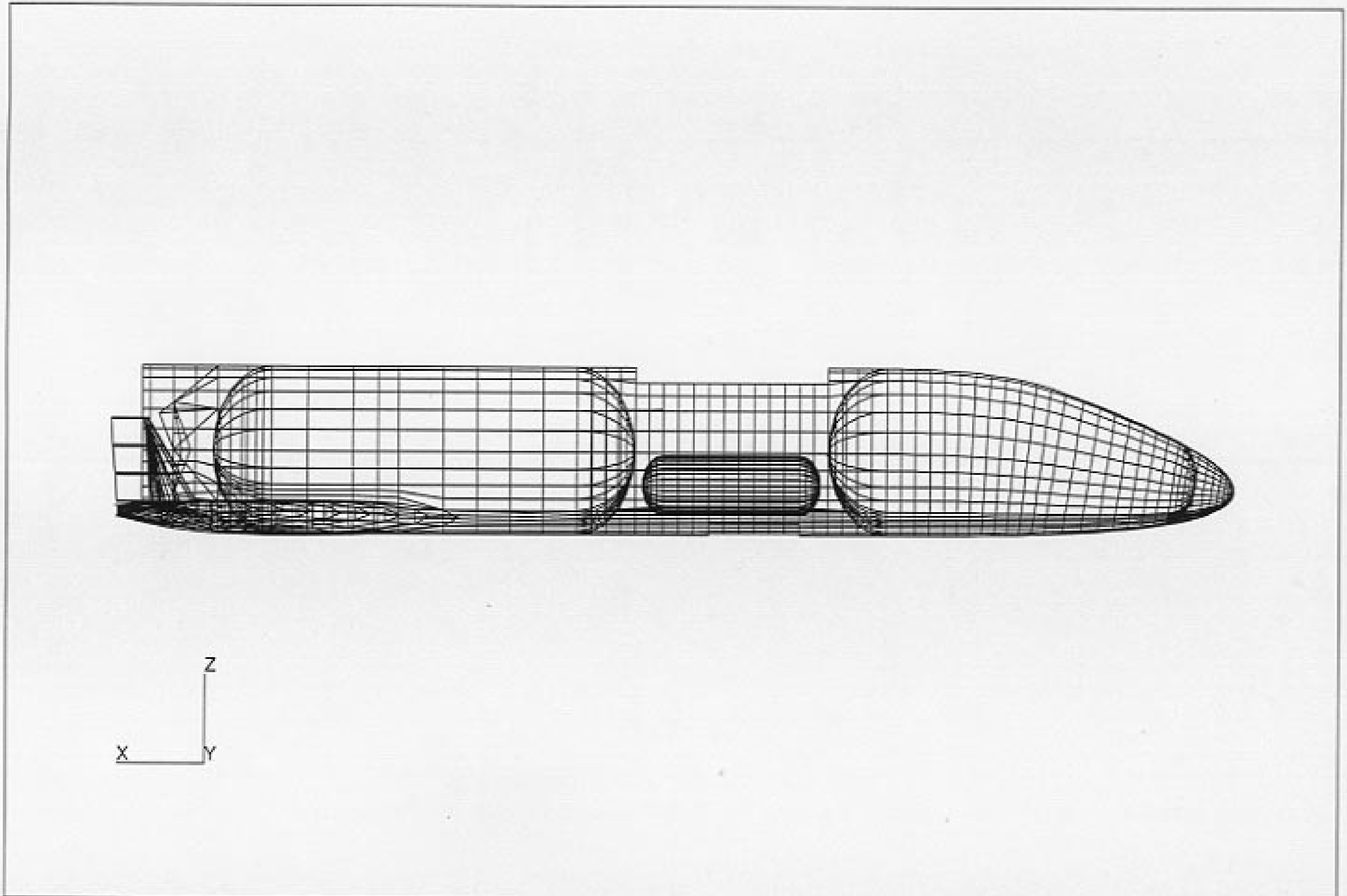


Figure 2.

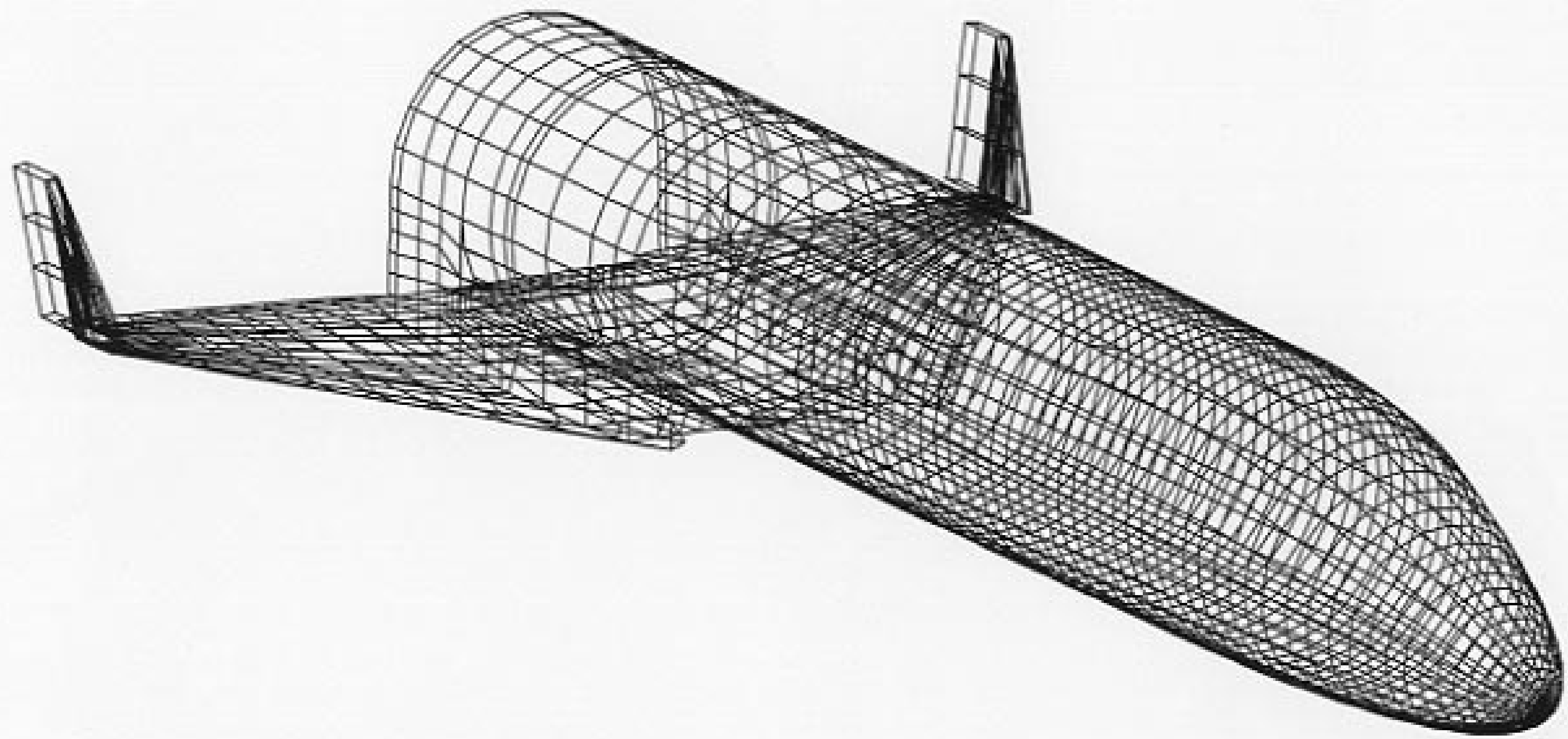
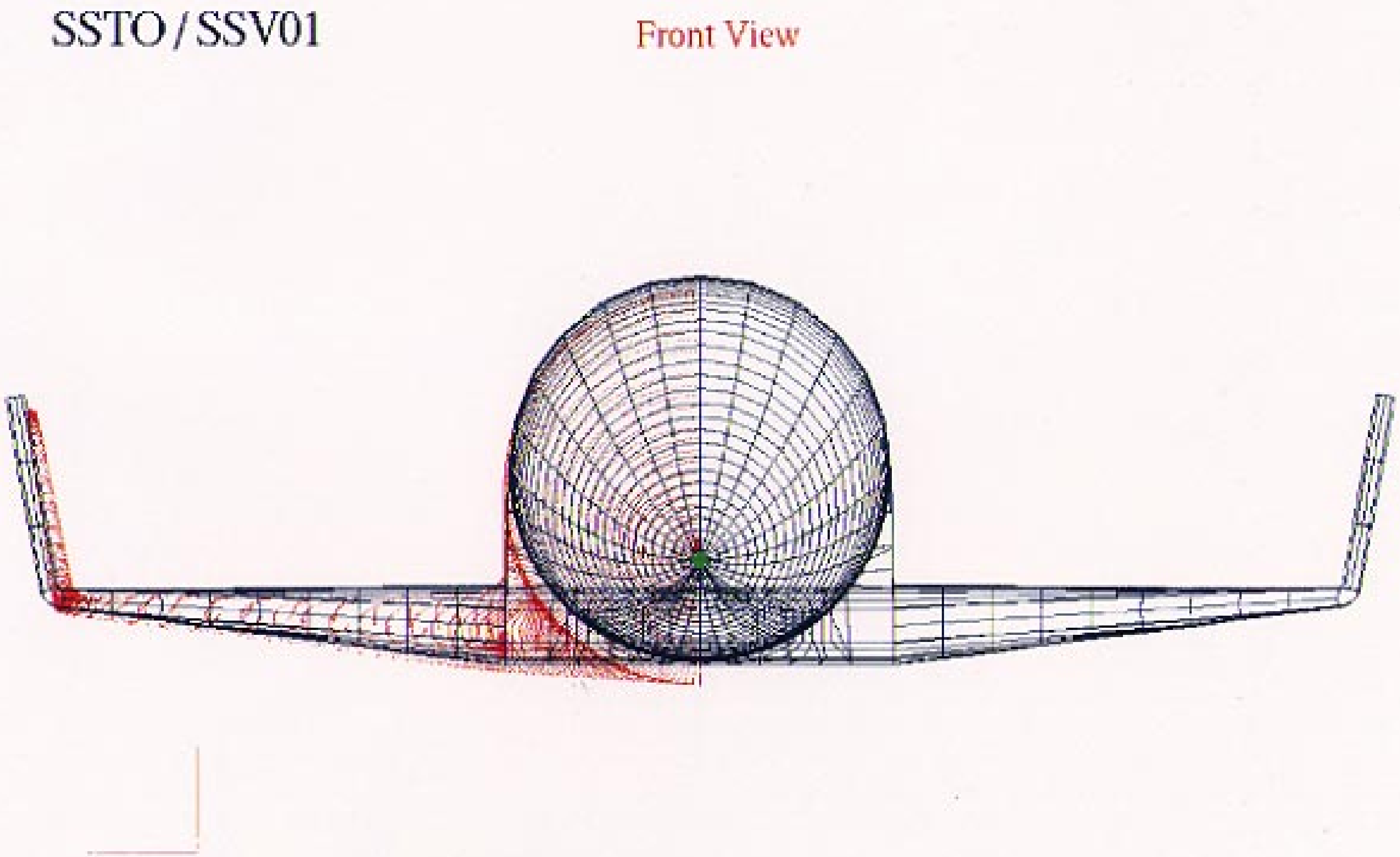


Figure 3.



CFD Grid Superimposed on FEM

Figure 4.

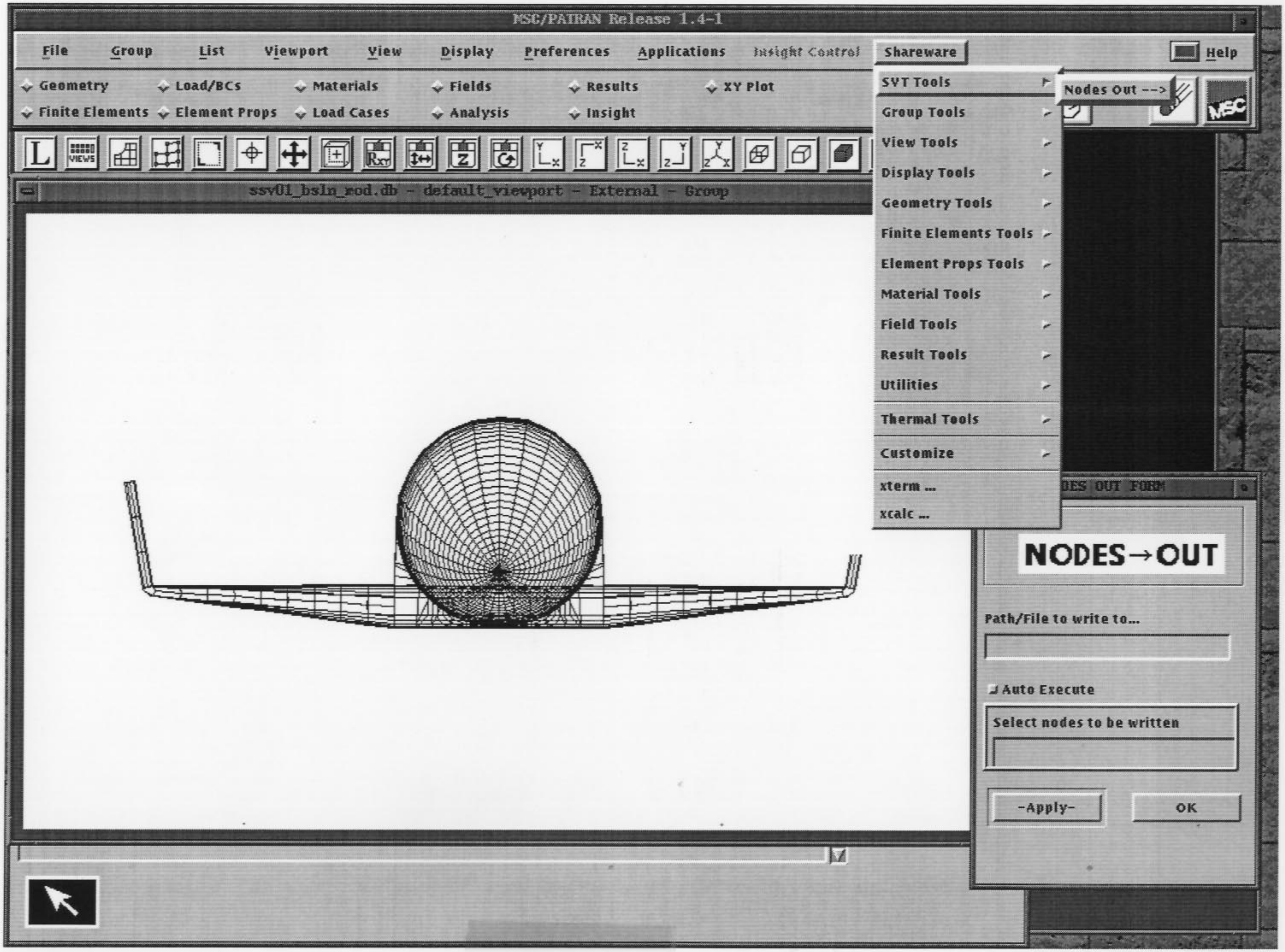

MSC/PATRAN Release 1.4-1
File
Group
List
Viewport
View
Display
Preferences
Applications
Insight Control
Shareware
Help
Geometry
Load/BCs
Materials
Fields
Results
XY Plot
Finite Elements
Element Props
Load Cases
Analysis
Insight
SVT Tools
Nodes Out -->
Group Tools
View Tools
Display Tools
Geometry Tools
Finite Elements Tools
Element Props Tools
Material Tools
Field Tools
Result Tools
Utilities
Thermal Tools
Customize
xterm ...
xcalc ...
ssv01_bsln_mod.db - default_viewport - External - Group
NODES→OUT
Path/File to write to...
Auto Execute
Select nodes to be written
-Apply-
OK

Figure 5.

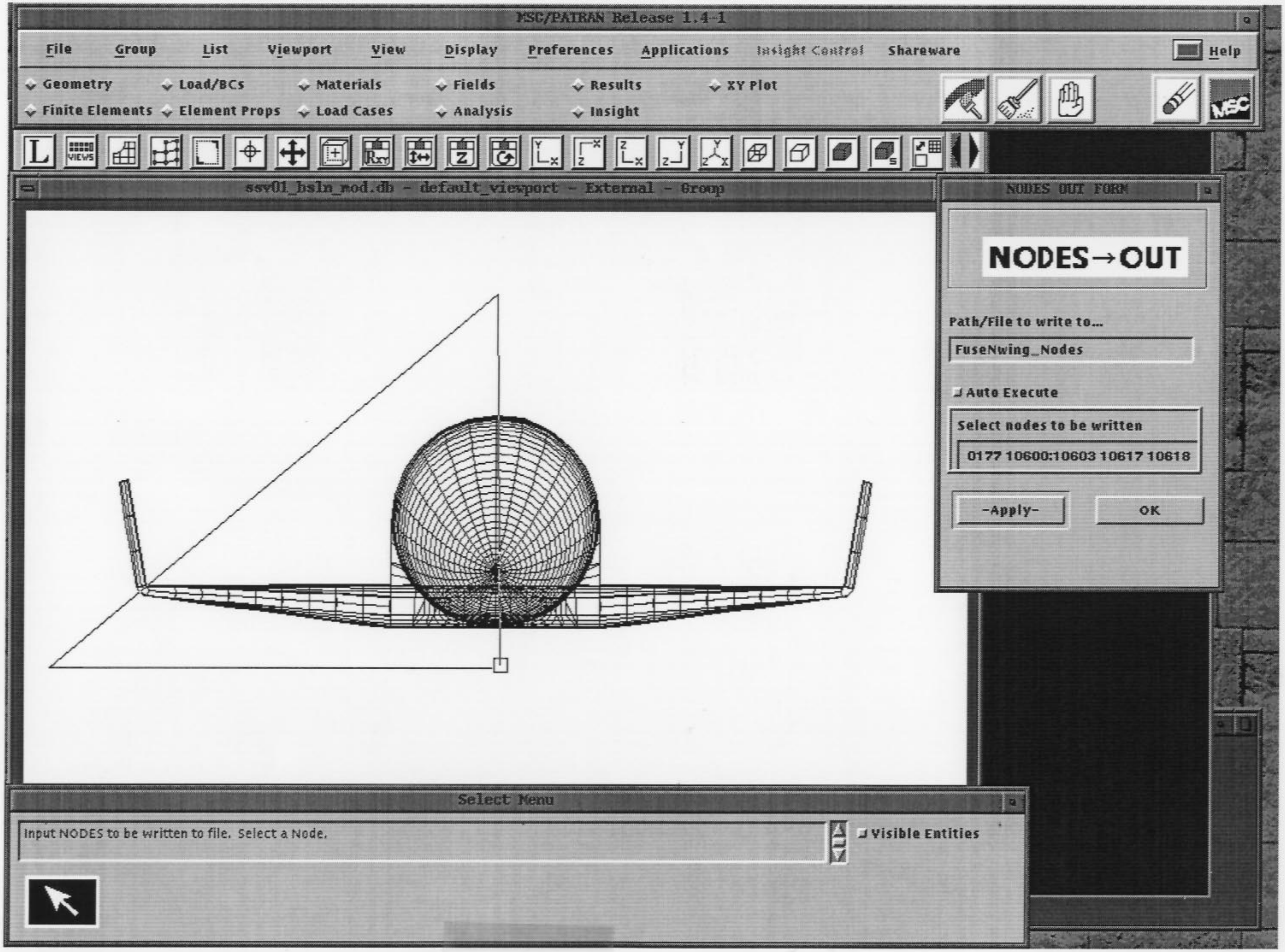

MSC/PATRAN Release 1.4-1
File
Group
List
Viewport
View
Display
Preferences
Applications
Insight Control
Shareware
Help
Geometry
Load/BCs
Materials
Fields
Results
XY Plot
Finite Elements
Element Props
Load Cases
Analysis
Insight
ssv01_bsln_mod.db - default_viewport - External - Group
NODES OUT FORM
NODES→OUT
Path/File to write to...
FuseNwing_Nodes
Auto Execute
Select nodes to be written
0177 10600:10603 10617 10618
-Apply-
OK
Select Menu
Input NODES to be written to file. Select a Node.
Visible Entities

Figure 6.

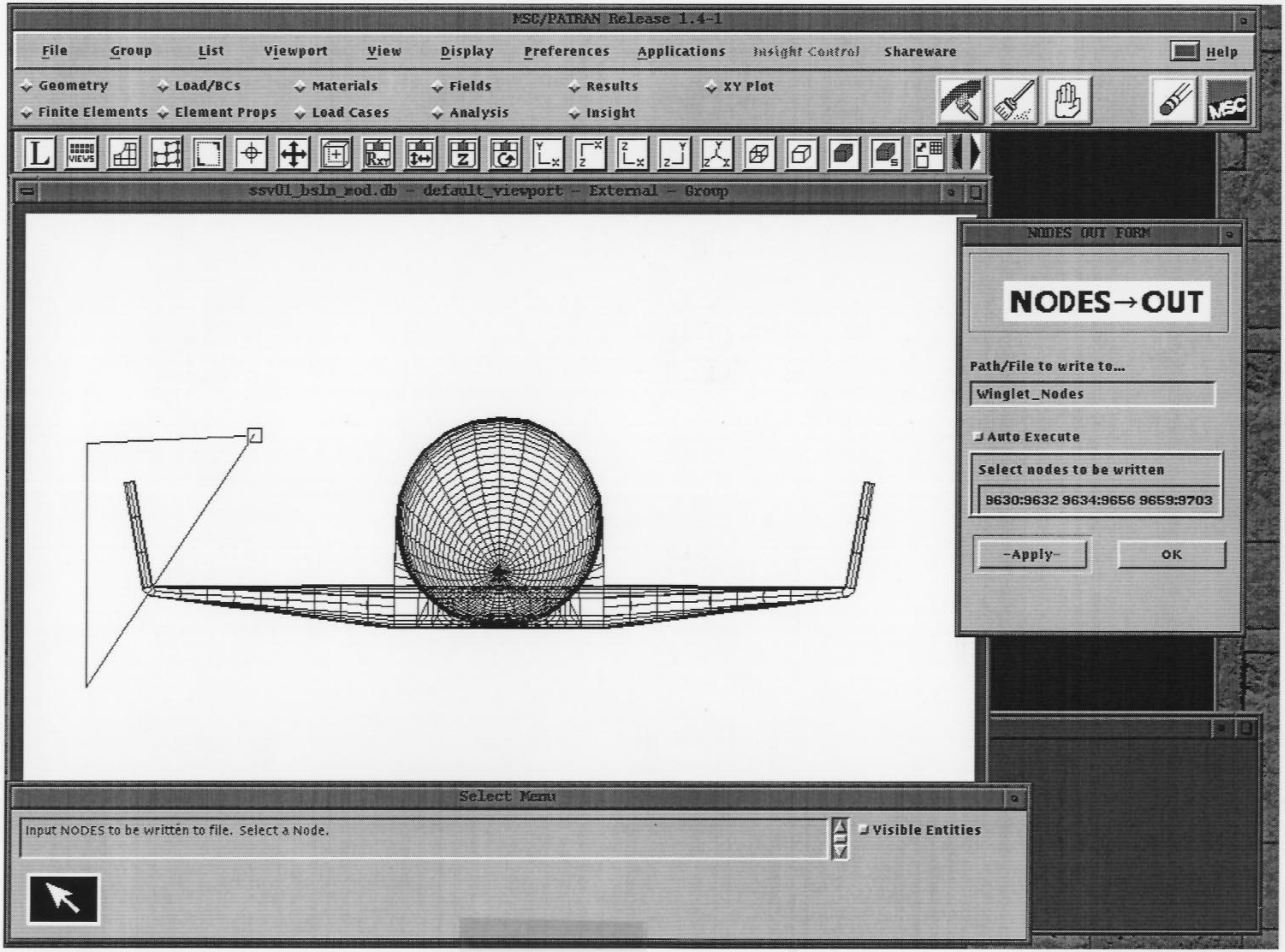
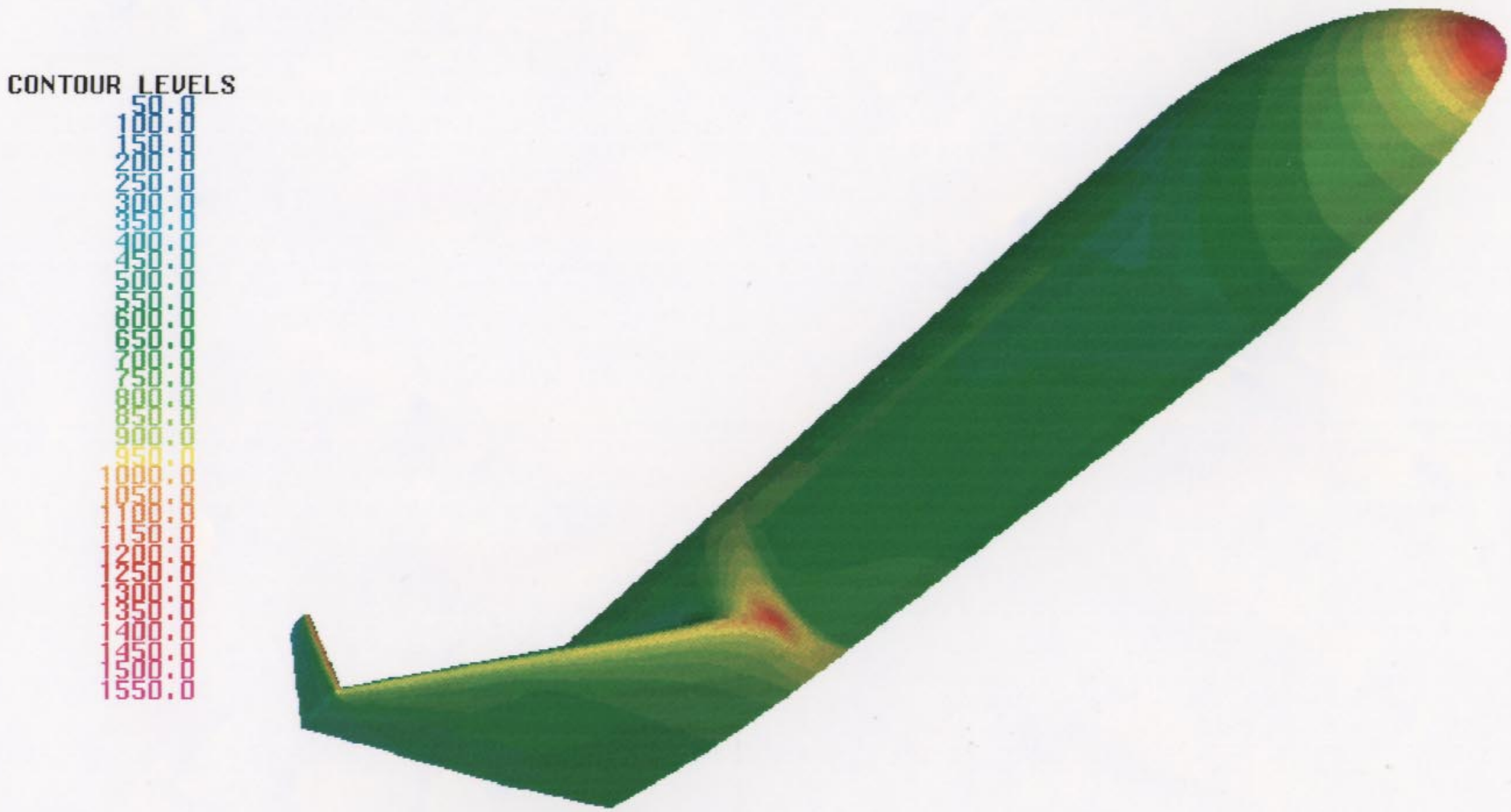
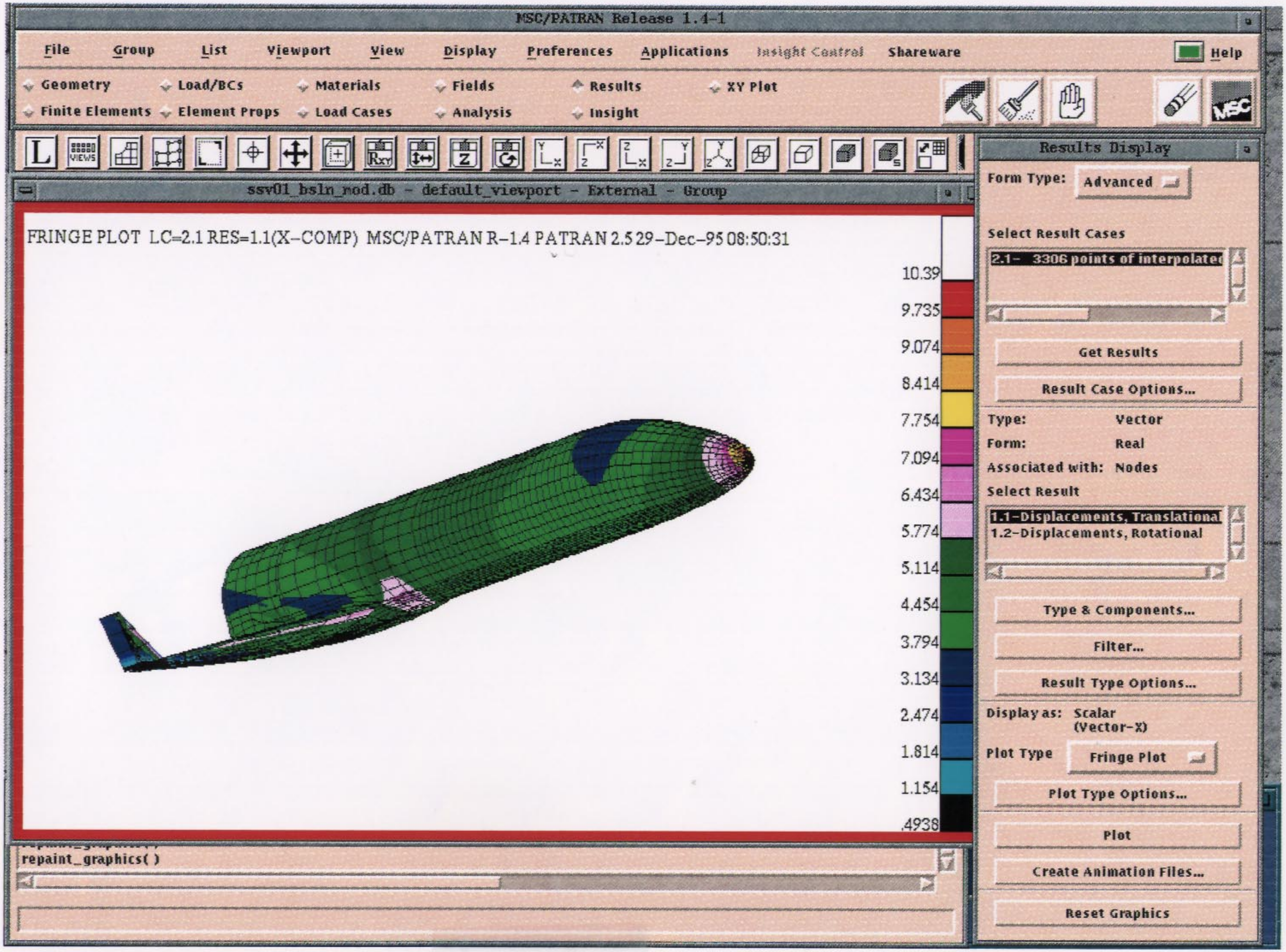

MSC/PATRAN Release 1.4-1
File
Group
List
Viewport
View
Display
Preferences
Applications
Insight Control
Shareware
Help
Geometry
Load/BCs
Materials
Fields
Results
XY Plot
Finite Elements
Element Props
Load Cases
Analysis
Insight
ssv01_bsln_mod.db - default_viewport - External - Group
NODES OUT FORM
NODES→OUT
Path/File to write to...
Winglet_Nodes
Auto Execute
Select nodes to be written
9630:9632 9634:9656 9659:9703
-Apply-
OK
Select Menu
Input NODES to be written to file. Select a Node.
Visible Entities

Figure 7.



SSTO CFD Plot3D Pressures (psf) - View From Below

Figure 8.



MSC/PATRAN Release 1.4-1
File
Group
List
Viewport
View
Display
Preferences
Applications
Insight Control
Shareware
Help
Geometry
Load/BCs
Materials
Fields
Results
XY Plot
Finite Elements
Element Props
Load Cases
Analysis
Insight
ssv01_bsln_mod.db - default_viewport - External - Group
FRINGE PLOT LC=2.1 RES=1.1(X-COMP) MSC/PATRAN R-1.4 PATRAN 2.5 29-Dec-95 08:50:31
10.39
9.735
9.074
8.414
7.754
7.094
6.434
5.774
5.114
4.454
3.794
3.134
2.474
1.814
1.154
.4938
repaint_graphics( )
Results Display
Form Type:
Advanced
Select Result Cases
2.1- 3306 points of interpolated
Get Results
Result Case Options...
Type: Vector
Form: Real
Associated with: Nodes
Select Result
1.1-Displacements, Translational
1.2-Displacements, Rotational
Type & Components...
Filter...
Result Type Options...
Display as: Scalar (Vector-X)
Plot Type
Fringe Plot
Plot Type Options...
Plot
Create Animation Files...
Reset Graphics

Figure 9.

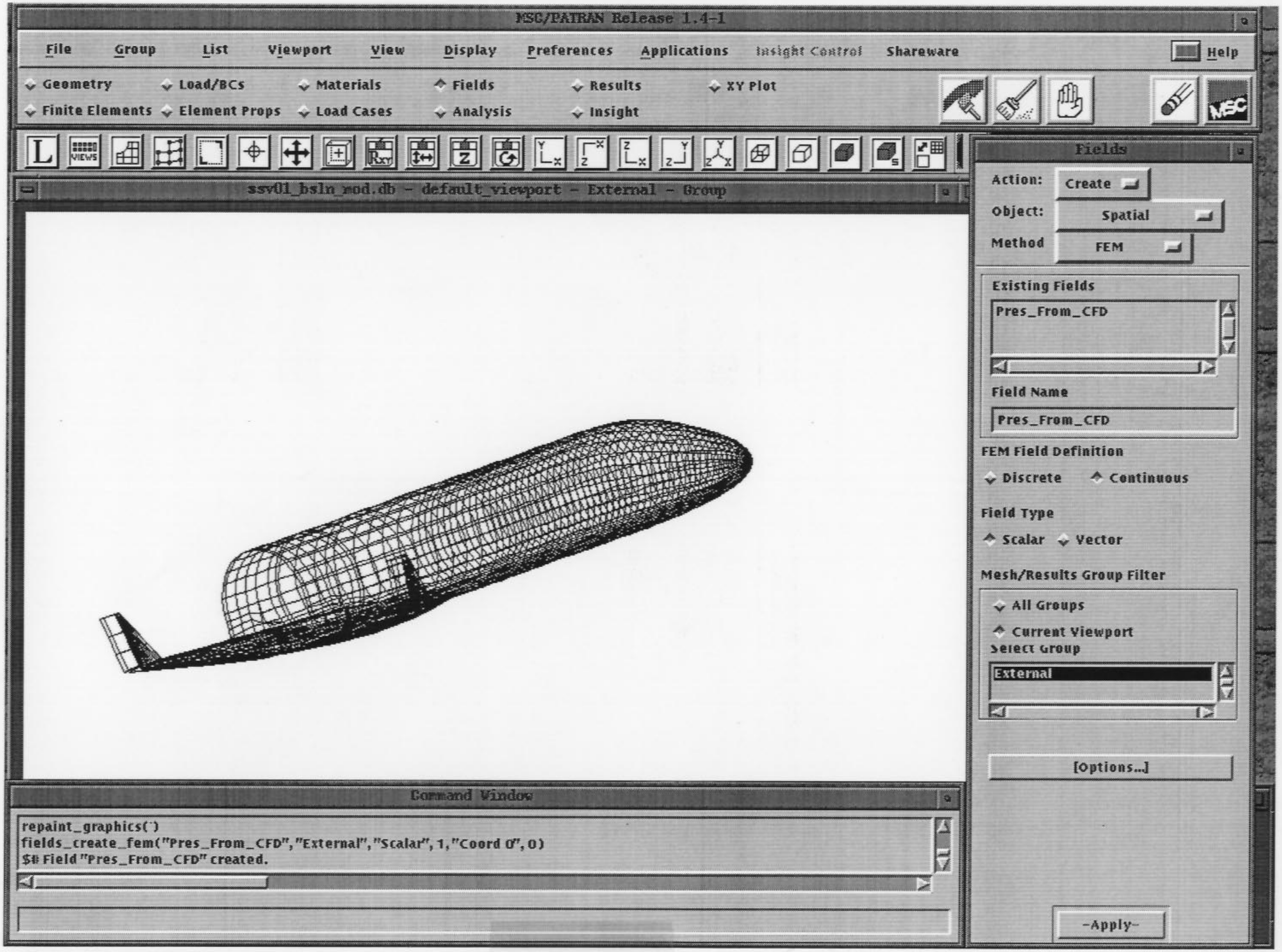

MSC/PATRAN Release 1.4-1
File
Group
List
Viewport
View
Display
Preferences
Applications
Insight Control
Shareware
Help
Geometry
Load/BCs
Materials
Fields
Results
XY Plot
Finite Elements
Element Props
Load Cases
Analysis
Insight
ssv01_bsln_mod.db - default_viewport - External - Group
Fields
Action:
Create
Object:
Spatial
Method
FEM
Existing Fields
Pres_From_CFD
Field Name
Pres_From_CFD
FEM Field Definition
Discrete
Continuous
Field Type
Scalar
Vector
Mesh/Results Group Filter
All Groups
Current Viewport
Select Group
External
[Options...]
-Apply-
Command Window
repaint_graphics()
fields_create_fem("Pres_From_CFD","External","Scalar",1,"Coord 0",0)
$# Field "Pres_From_CFD" created.

Figure 10.

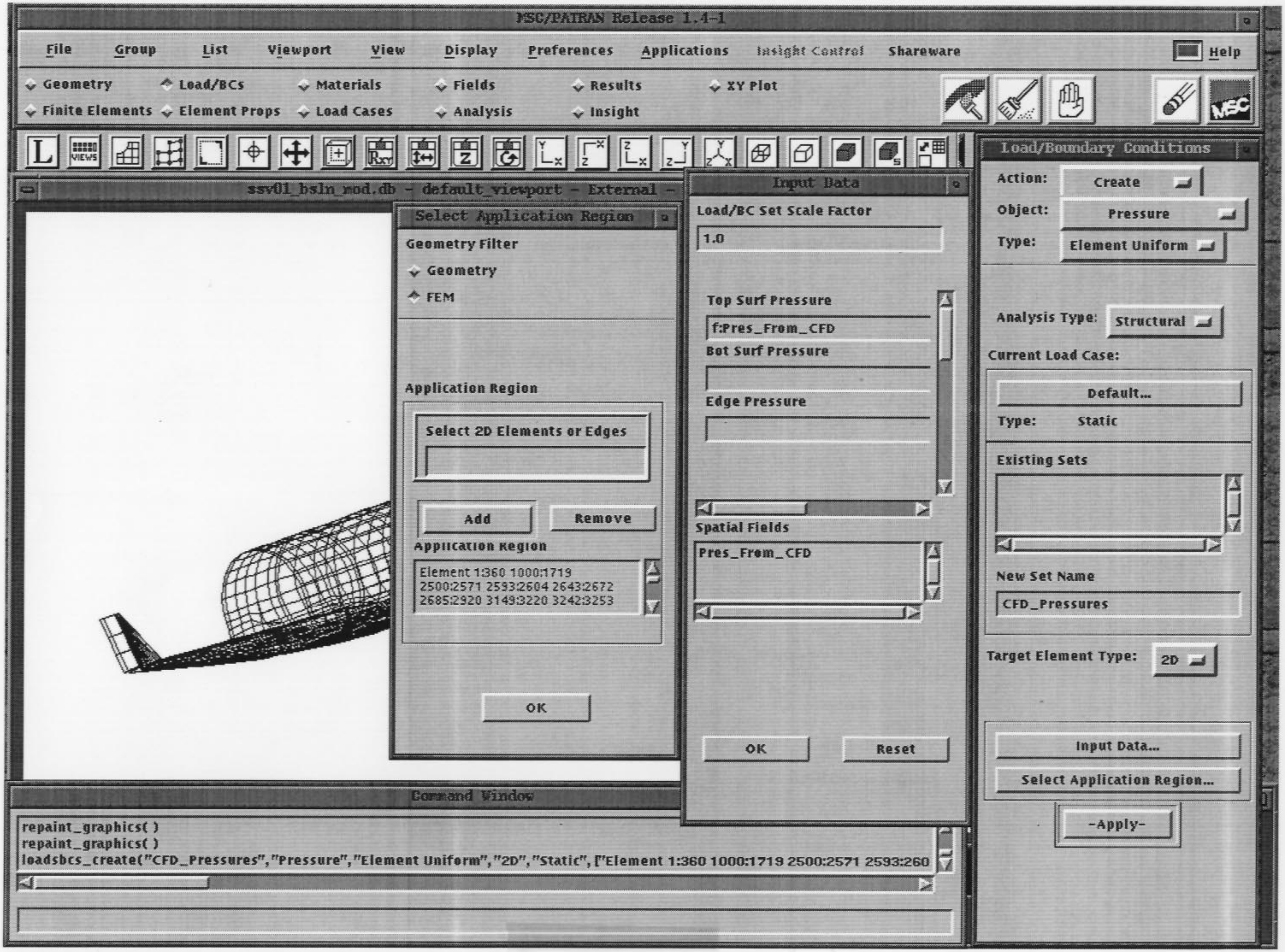

MSC/PATRAN Release 1.4-1
File
Group
List
Viewport
View
Display
Preferences
Applications
Insight Control
Shareware
Help
Geometry
Load/BCs
Materials
Fields
Results
XY Plot
Finite Elements
Element Props
Load Cases
Analysis
Insight
ssv01_bsln_mod.db - default_viewport - External -
Select Application Region
Geometry Filter
Geometry
FEM
Application Region
Select 2D Elements or Edges
Add
Remove
Application Region
Element 1:360 1000:1719
2500:2571 2593:2604 2643:2672
2685:2920 3149:3220 3242:3253
OK
Input Data
Load/BC Set Scale Factor
1.0
Top Surf Pressure
f:Pres_From_CFD
Bot Surf Pressure
Edge Pressure
Spatial Fields
Pres_From_CFD
OK
Reset
Load/Boundary Conditions
Action:
Create
Object:
Pressure
Type:
Element Uniform
Analysis Type:
Structural
Current Load Case:
Default...
Type:
Static
Existing Sets
New Set Name
CFD_Pressures
Target Element Type:
2D
Input Data...
Select Application Region...
-Apply-
Command Window
repaint_graphics( )
repaint_graphics( )
loadsbcs_create("CFD_Pressures","Pressure","Element Uniform","2D","Static",["Element 1:360 1000:1719 2500:2571 2593:260

Figure 11.

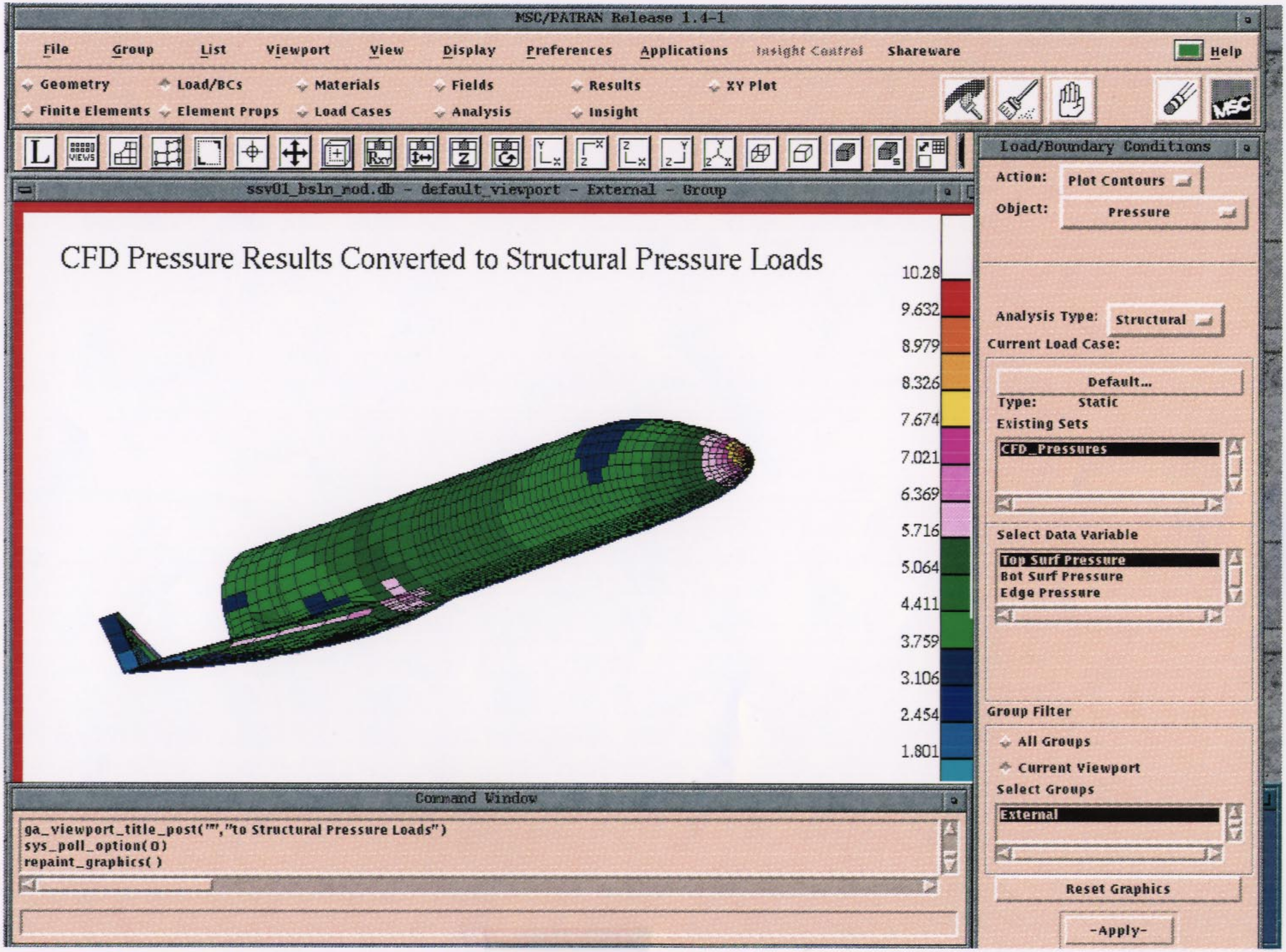

MSC/PATRAN Release 1.4-1
File
Group
List
Viewport
View
Display
Preferences
Applications
Insight Control
Shareware
Help
Geometry
Load/BCs
Materials
Fields
Results
XY Plot
Finite Elements
Element Props
Load Cases
Analysis
Insight
ssv01_bsln_rod.db - default_viewport - External - Group
CFD Pressure Results Converted to Structural Pressure Loads
10.28
9.632
8.979
8.326
7.674
7.021
6.369
5.716
5.064
4.411
3.759
3.106
2.454
1.801
Load/Boundary Conditions
Action: Plot Contours
Object: Pressure
Analysis Type: Structural
Current Load Case:
Default...
Type: Static
Existing Sets
CFD_Pressures
Select Data Variable
Top Surf Pressure
Bot Surf Pressure
Edge Pressure
Group Filter
All Groups
Current Viewport
Select Groups
External
Reset Graphics
-Apply-
Command Window
ga_viewport_title_post("","to Structural Pressure Loads")
sys_poll_option(0)
repaint_graphics()

Figure 12.