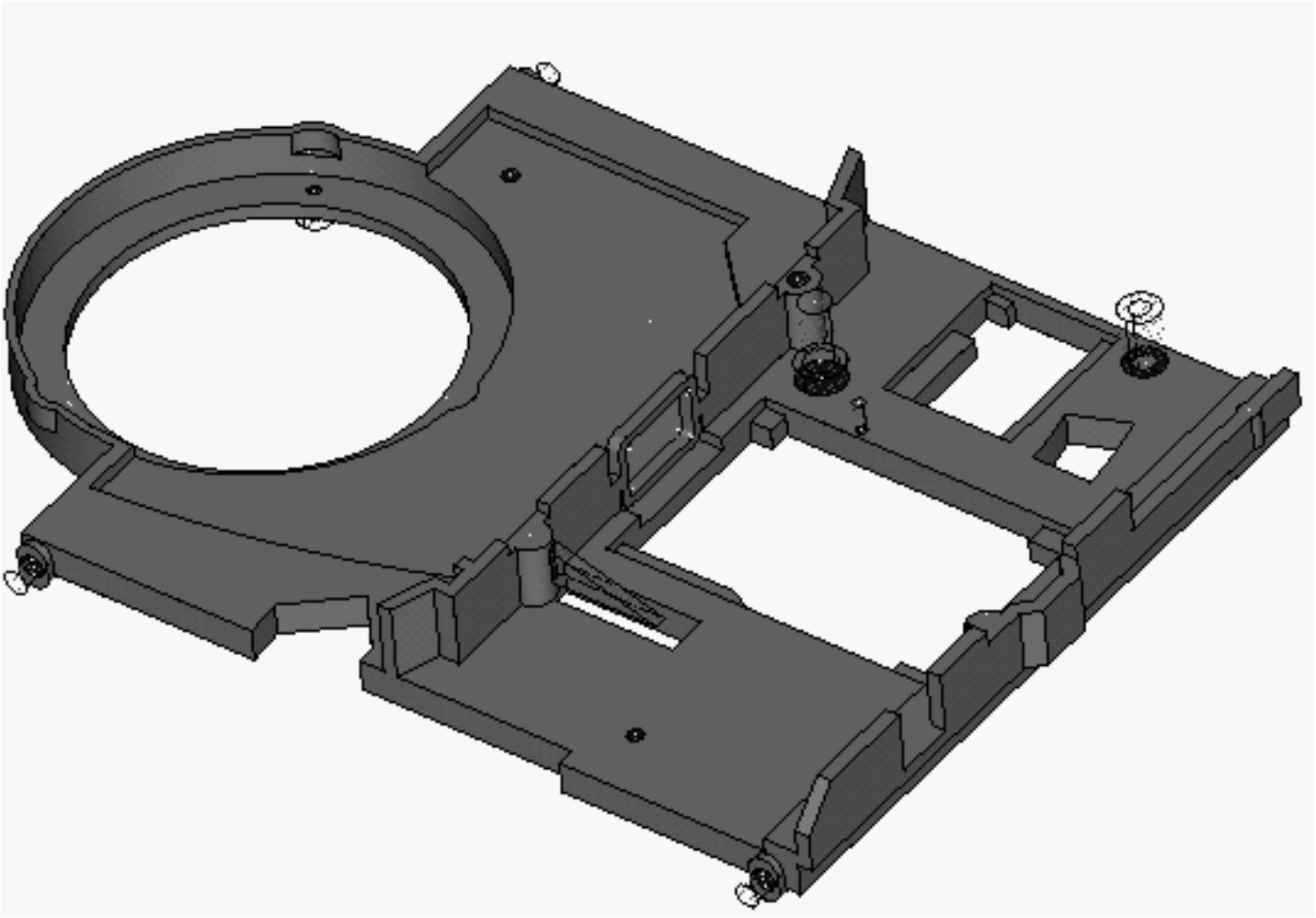
Zip Demo
FE Section Vignette



CHECKLIST

Preparation

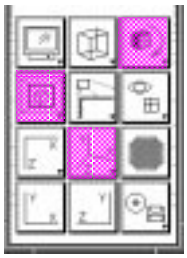
1.0 Make sure Master Modeler and Simulation are loaded on your machine
2.0
3.0
4.0

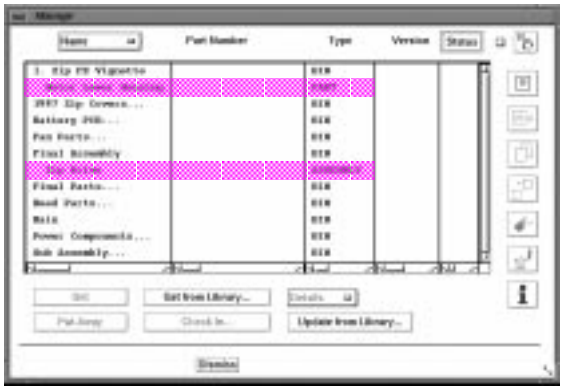
Demonstration Installation and Setup

- Copy or unload the demo files to a local directory
- cd to the directory containing the demo files
- Start I-DEAS
  Project = **Any**
  Model File = **No model file**
  Application = **Design**
  Task = **Master Assembly**

- File, Import, IDEAS Universal File
  'Zip_FE_Section_Vignette.unv'
- **Design ... Simulation**
- Do the following manually or run
  'Zip_FE_Section_Vignette.prg'

- **Options, units, MM**
- **Display Filter Part, toggle Off centerlines, centerpoints, coordinate systems, local origin, OK**
- **Display Filter Assy, toggle Off Assy Name (Top) and Constraints, OK**
- **Display Filter, toggle Off Workplane, OK**
- **Collapse all bins on Manage Bins form except for the 'Zip FE Vignette' bin and the 'Final Assembly' bin.**
- **Get 'Zip Drive' Assy and 'Motor Lower Housing' part to workbench, Dismiss**
- **Iso View**
- **Zoom All**
- **Shaded Display**





- **Save – use any name you like, i.e., 'Zip_FE_Section_Vignette'**

- The demonstration is now ready. The above setup only needs to be done once (don't save over your start model file, during the demonstration, make incremental saves under a new name).

Demonstration Positioning

This presentation is intended for an analysis orientated audience. The techniques illustrated are intended to give the analysist a feel for ONE of the many tools available for mesh abstration.

You might ask why we don't perform the same task using feature supression. You can. You might ask why we don't just mesh the part in all its detail. You can. You might ask why we don't make an ACF and make changes on the associative copy. You can. The point is to show **A** method. Keep this in mind if the designer/analyst–wanne–be starts asking questions.

Additionally, section meshing encompases much more that just feature abstraction. It is very useful for 'healing' dirty geometry that arrives via IGES.

What is Section Meshing

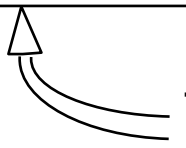
Section meshing allows you to:

1. Create meshable areas from wireframe geometry that does not 'close' or contains duplicates. The benefit is you don't have to spend time cleaning the wireframe.

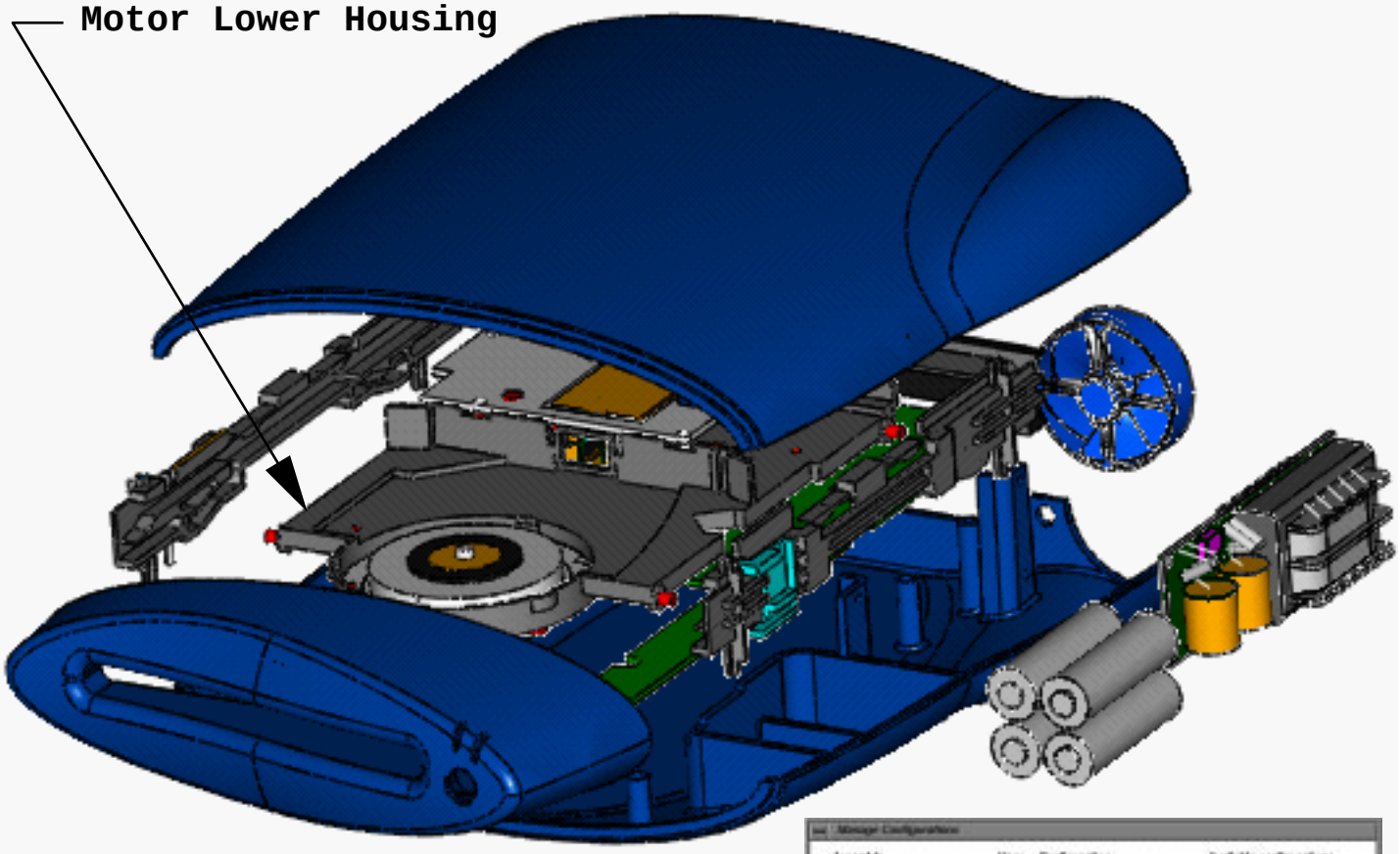
2. Combine many (or few) splinter/fragment surfaces into one topologically correct meshable area that gets meshed independentably from the small artifact edges of the parent surfaces. The benefit being that micro elements don't get created yet the nodes sit on the original surfaces (no loss of accuracy).

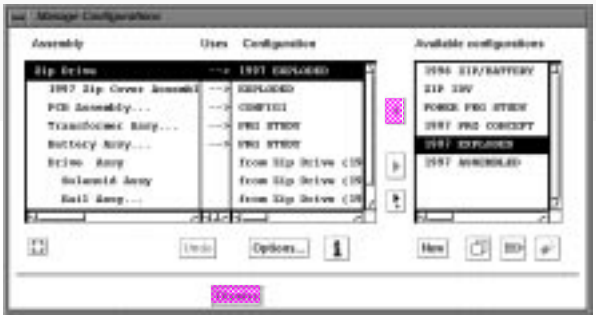
3. Sections that replace/superseed topologically challenged surface combinations can be used in volume meshes. The benefit is you don't have to spend time repairing surfaces to get a volume suitable for meshing

4. Features that result in inner loops on the section can be ignored by the mesher. The benefit being a quick way of doing (limited) feature abstraction/suppression on referenced parts.



The focus of this vignette



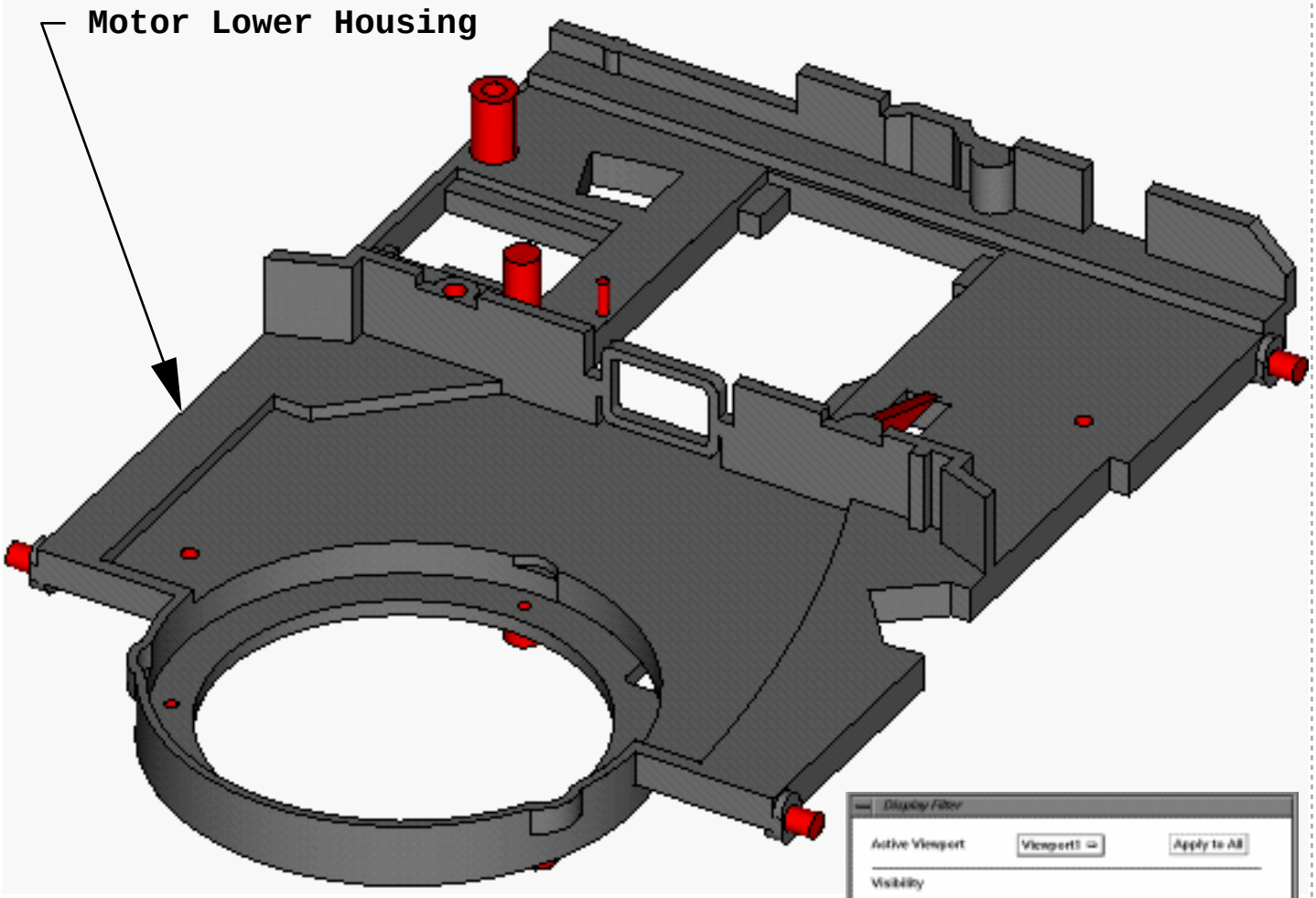


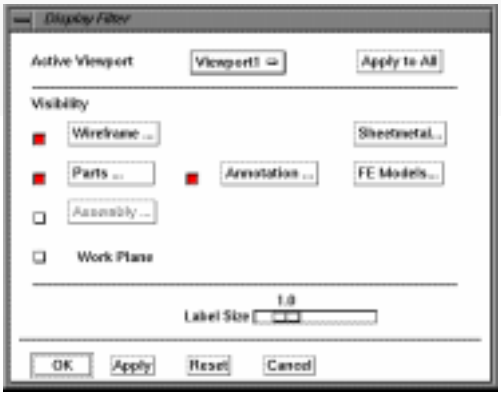
The demonstration starts at the end of the Zip demo.

Master Assembly...

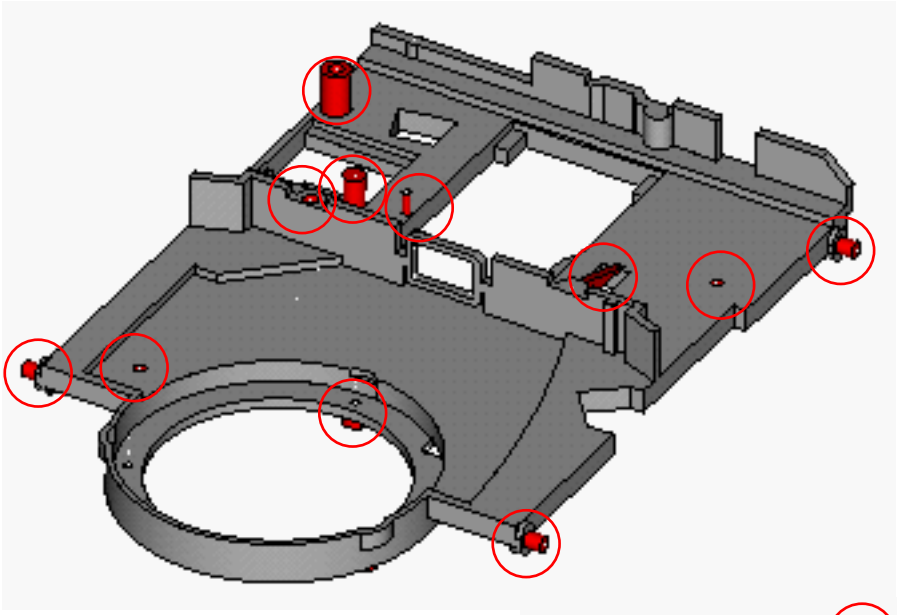
- **Manage Configurations**
  Change from ’1997 ASSEMBLED’ to ’1997 EXPLODED’

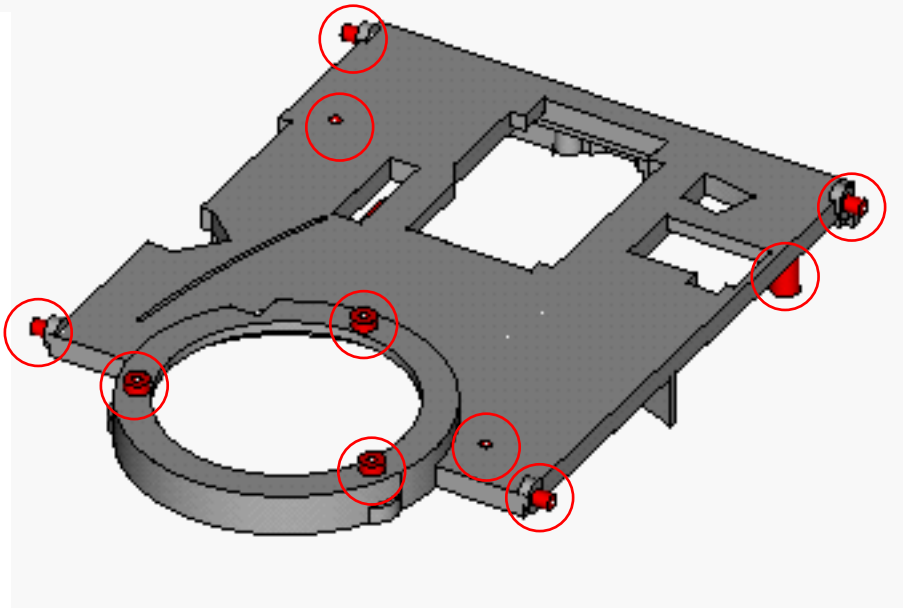
This will open the assembly. The focus of the presentation is the ’Motor Lower Housing’ part.





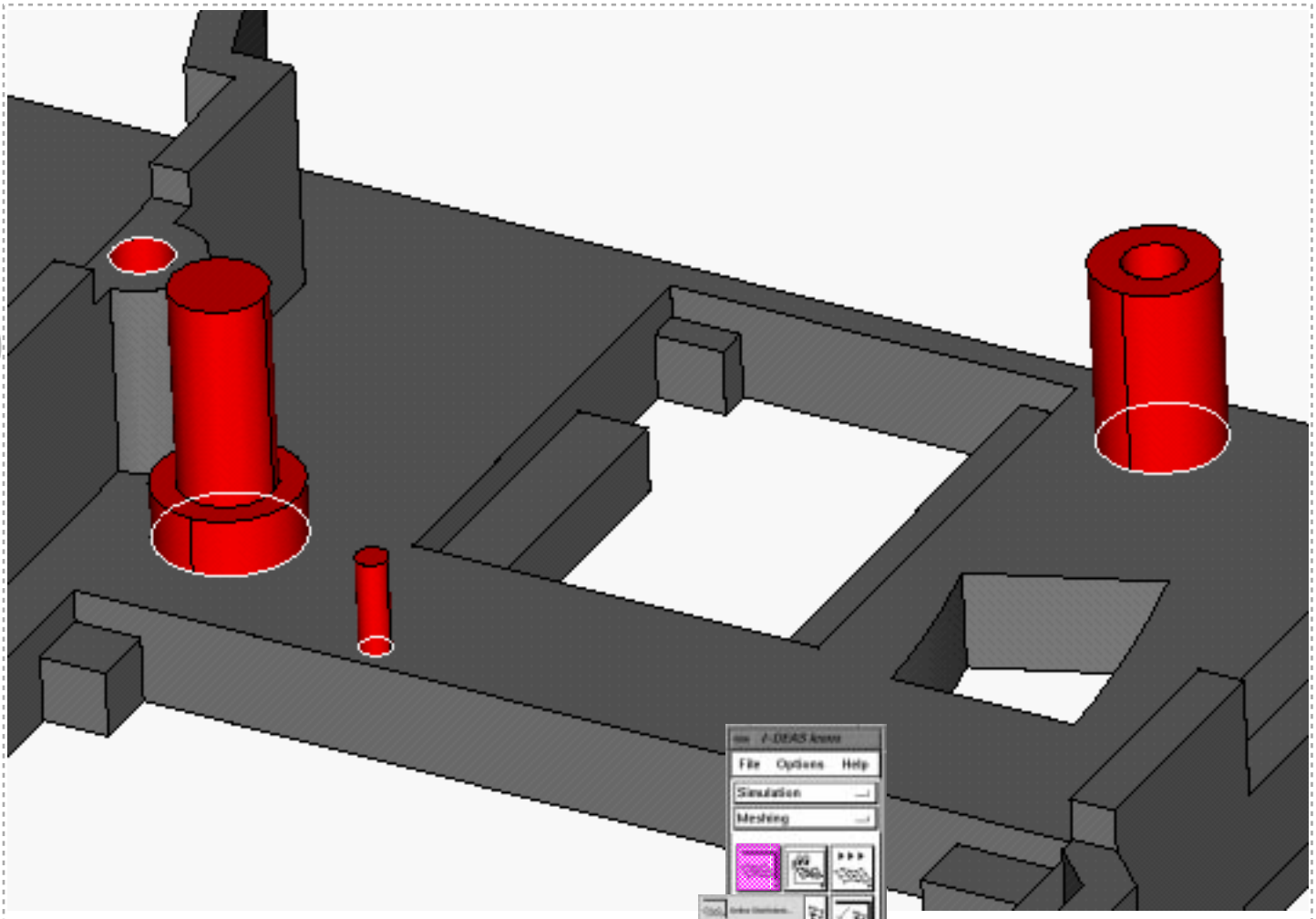
- **Display Filter**
  Toggle off Assembly Visibility





Objective

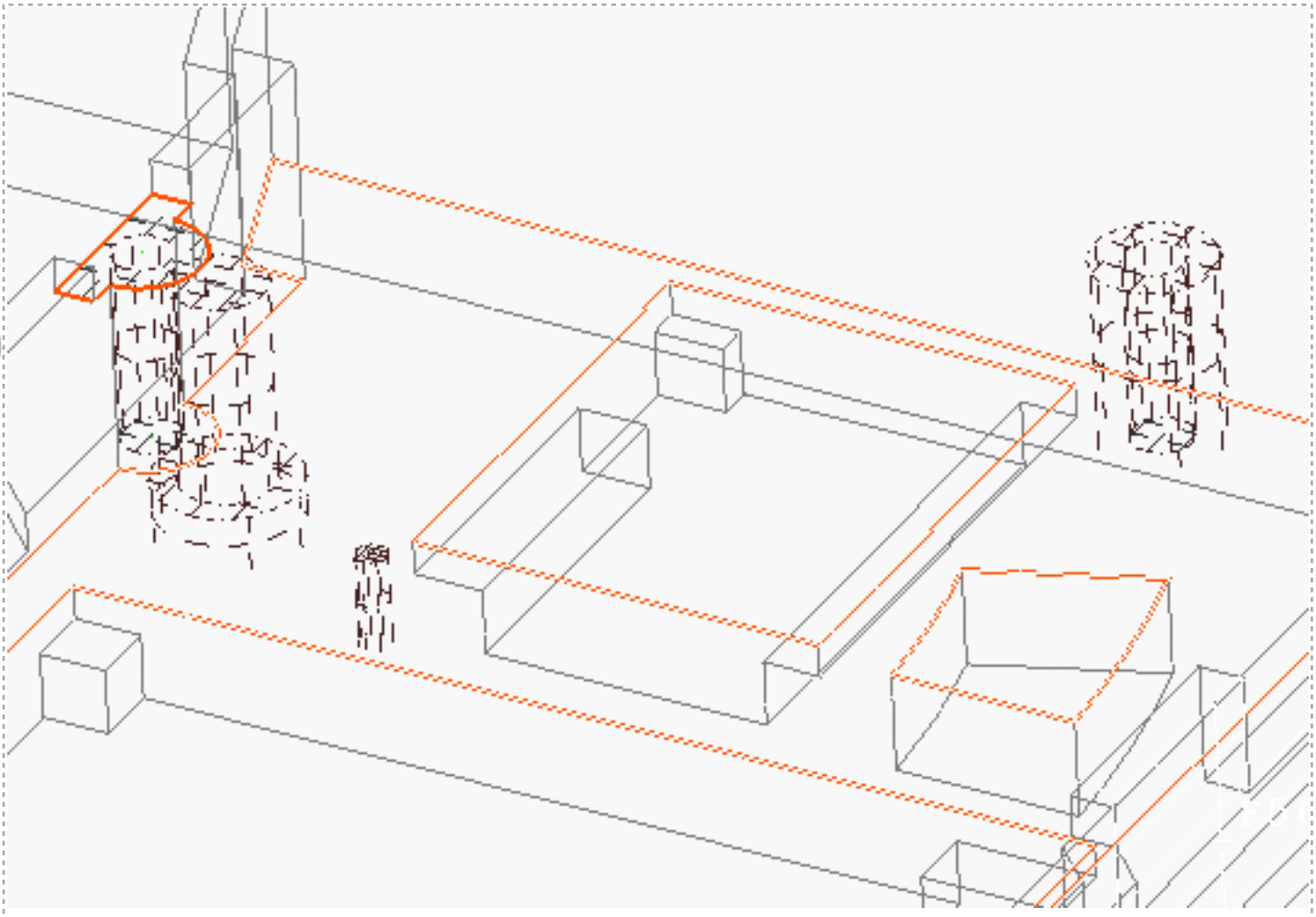
We want to use the mesh abstraction tools to create a volume mesh that does not contain the features shown in red. These features are holes and non-structural protrusions that we don't think are pertinate to our analysis at this point.



- **Master Modeler ... Meshing**
- **Section Create**
- **Manual Suppression**

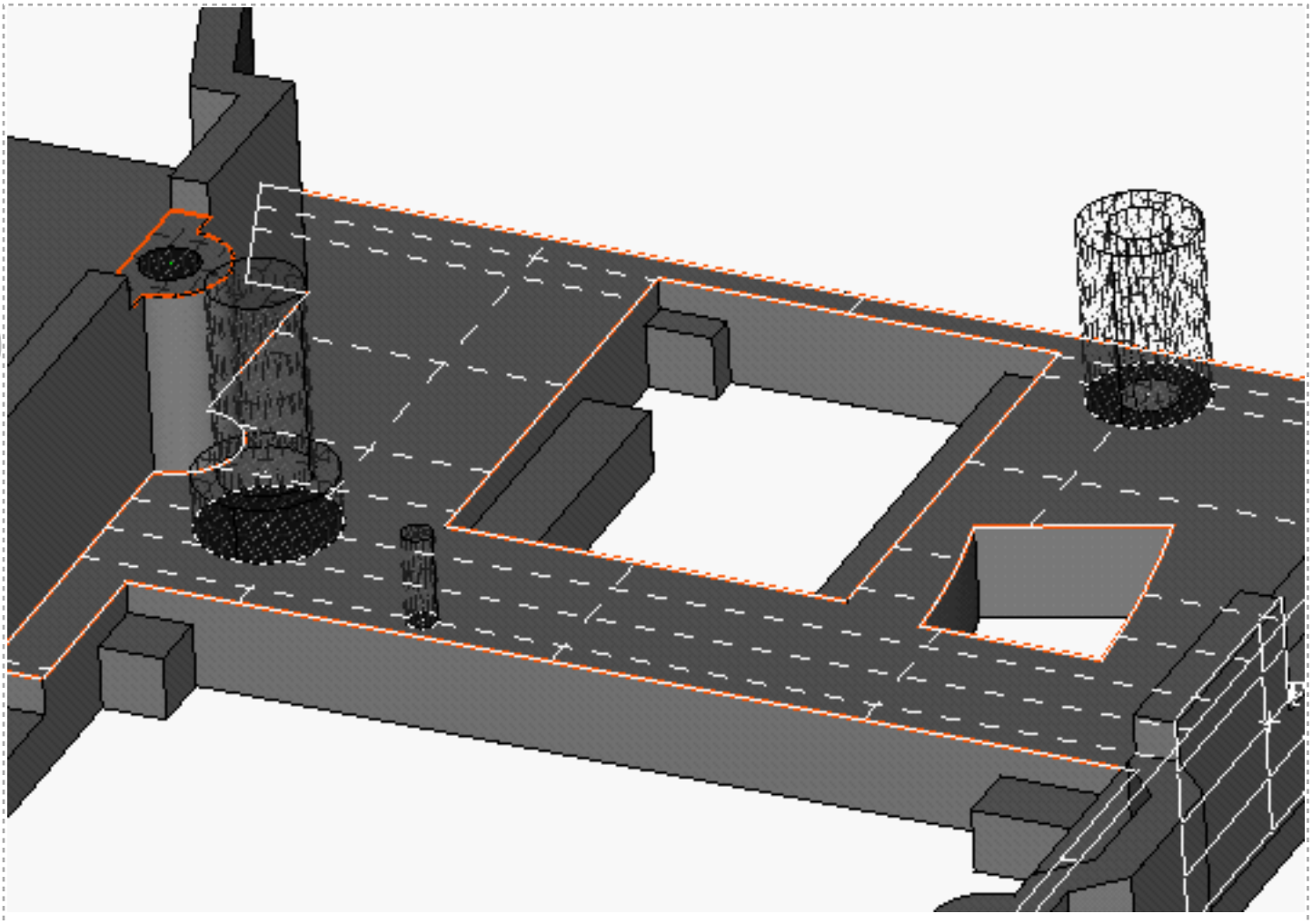
Create the FE model by highlighting the ’All Sections’ button as shown

At the ’Pick inner loops/vertices to suppress/unsuppress’ prompt, Pick the 4 edges shown, Done



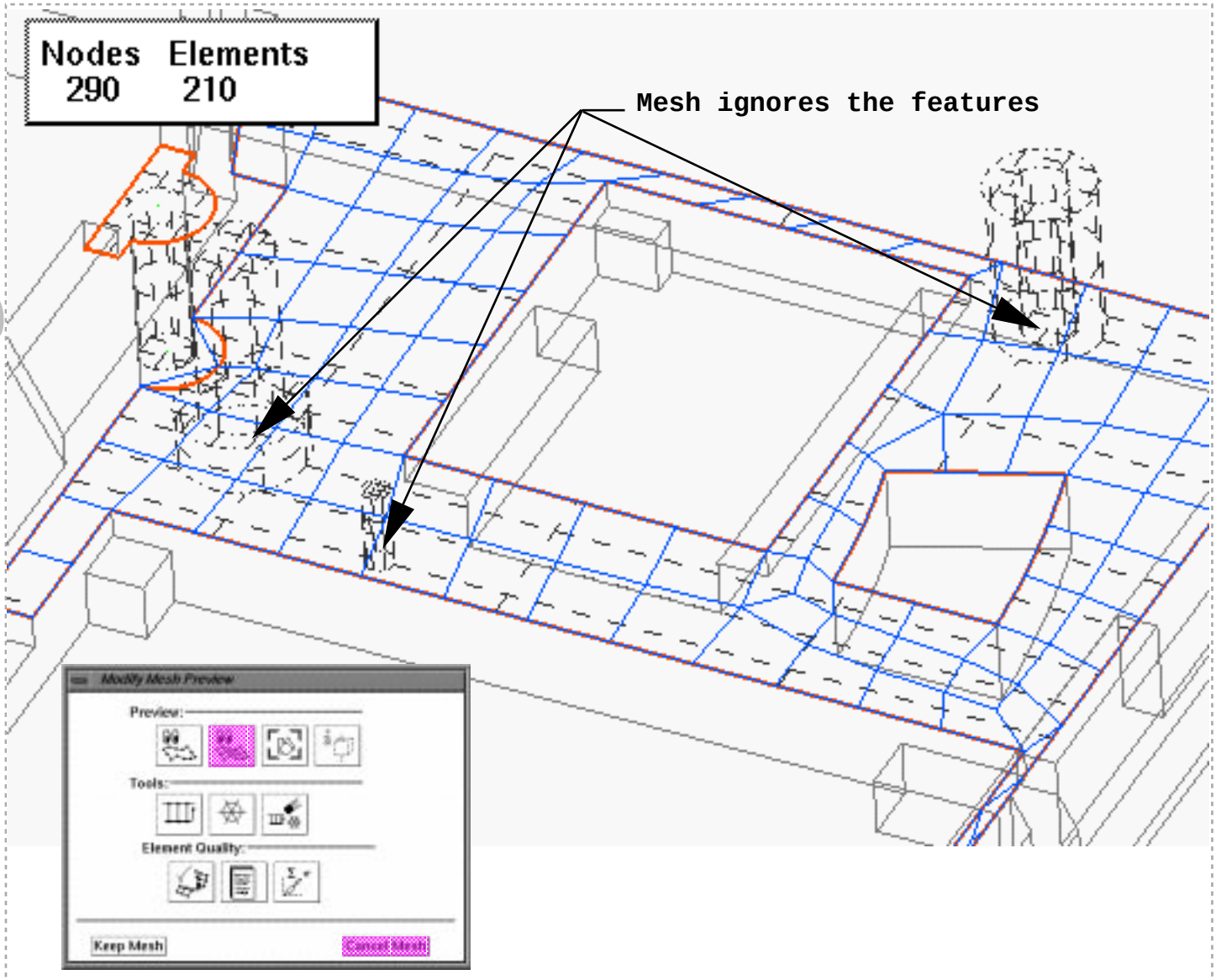
Notice that the 3 bosses and the thru hole are now shown dotted and an orange section was created. This section contains all the information on its parent surface less the hole and the 3 bosses (the 4 inner loops were not created).

What just happened? Illustrate by applying a surface mesh.



Define Shell Mesh...
on the surface shown

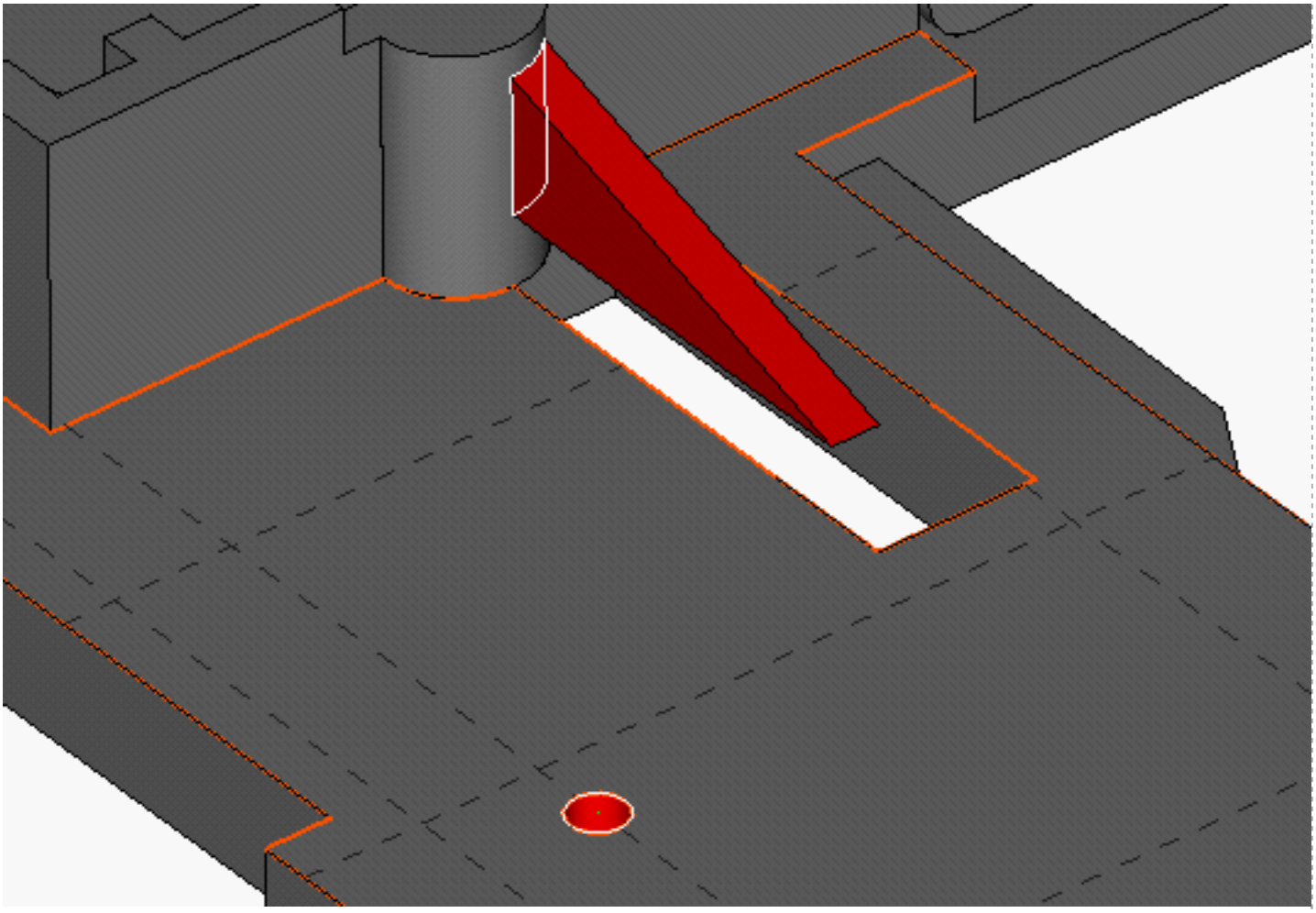
Modify Mesh Preview... 

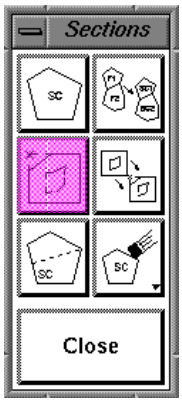


•Notice that the mesh preview shows that the elements ignore the suppressed features.

•Cancel Mesh, Cancel (do not keep the mesh)

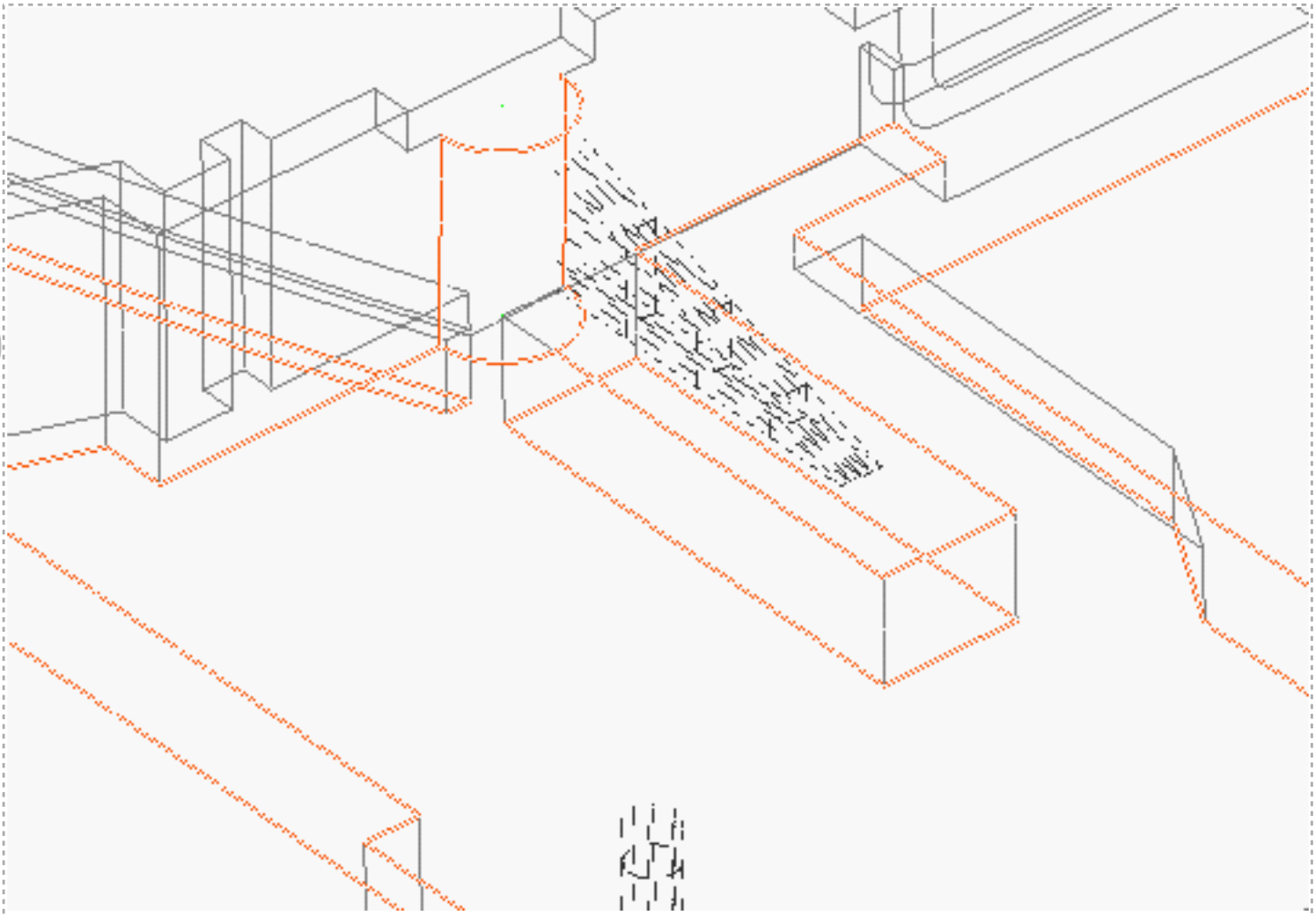
•Continue suppressing features



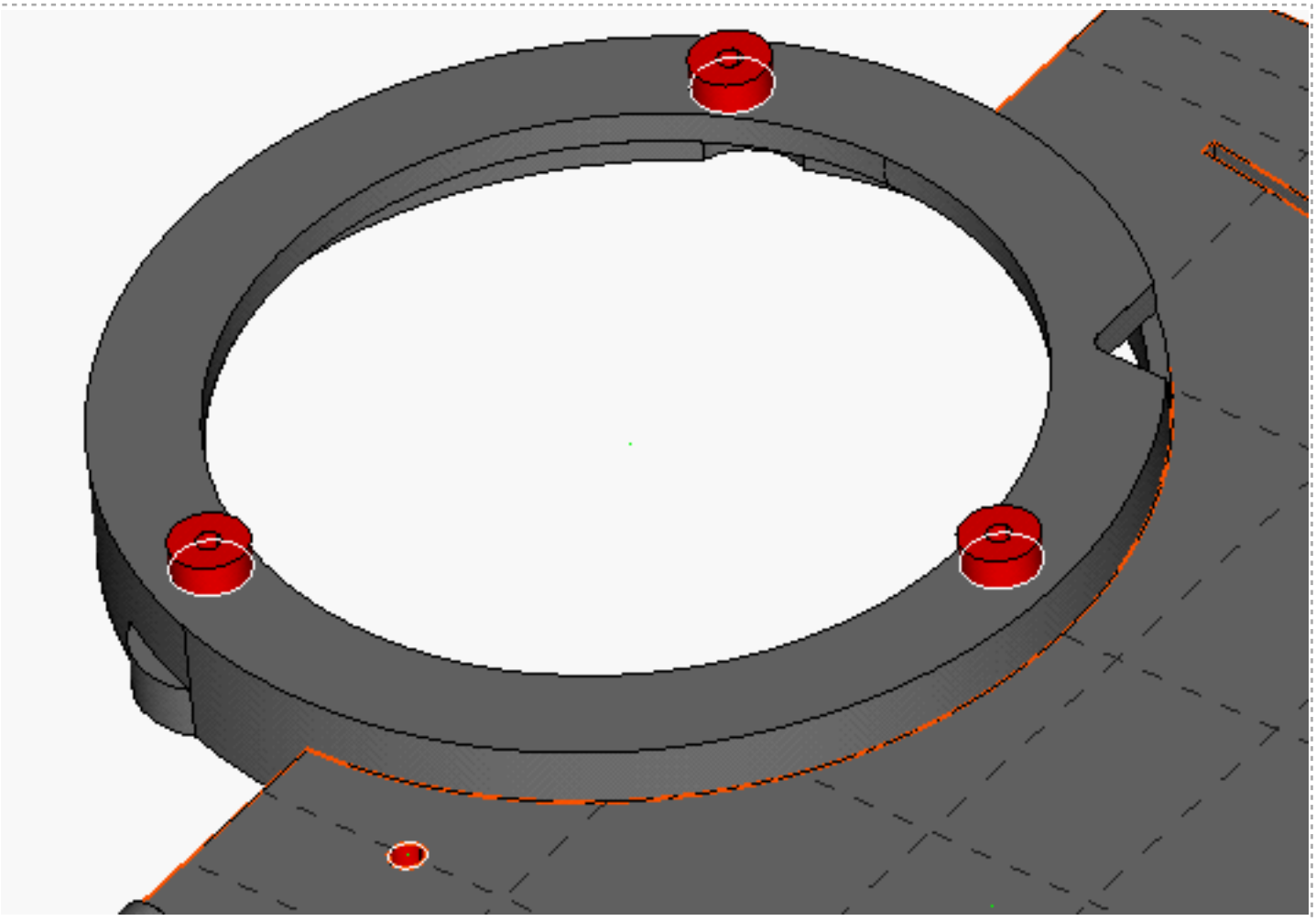


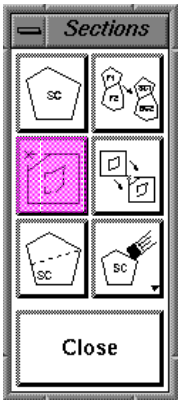
Continue selecting features to suppress.

•Manual Suppression...
Pick the two loops shown. Done.



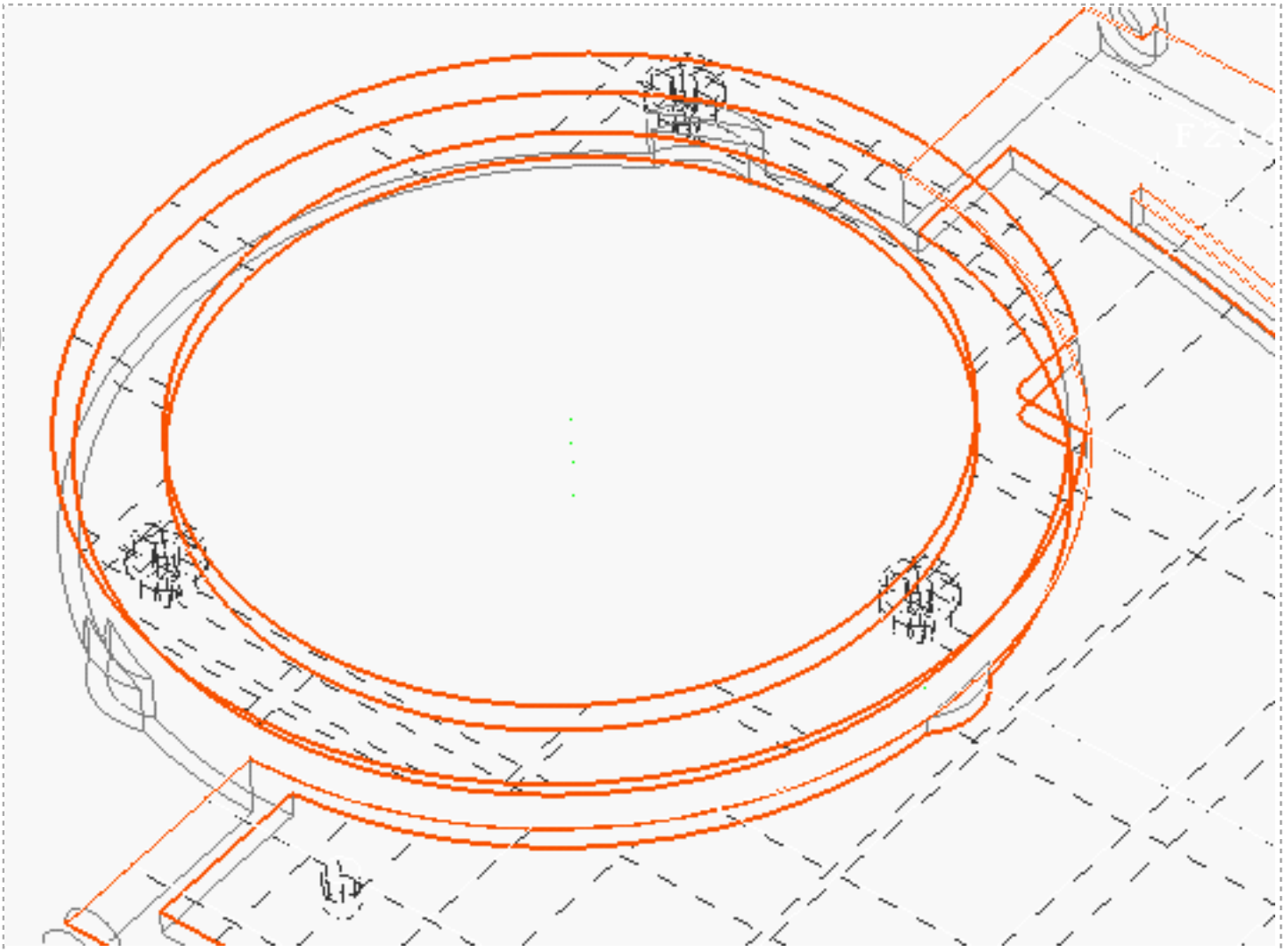
These 2 features are now shown dotted.



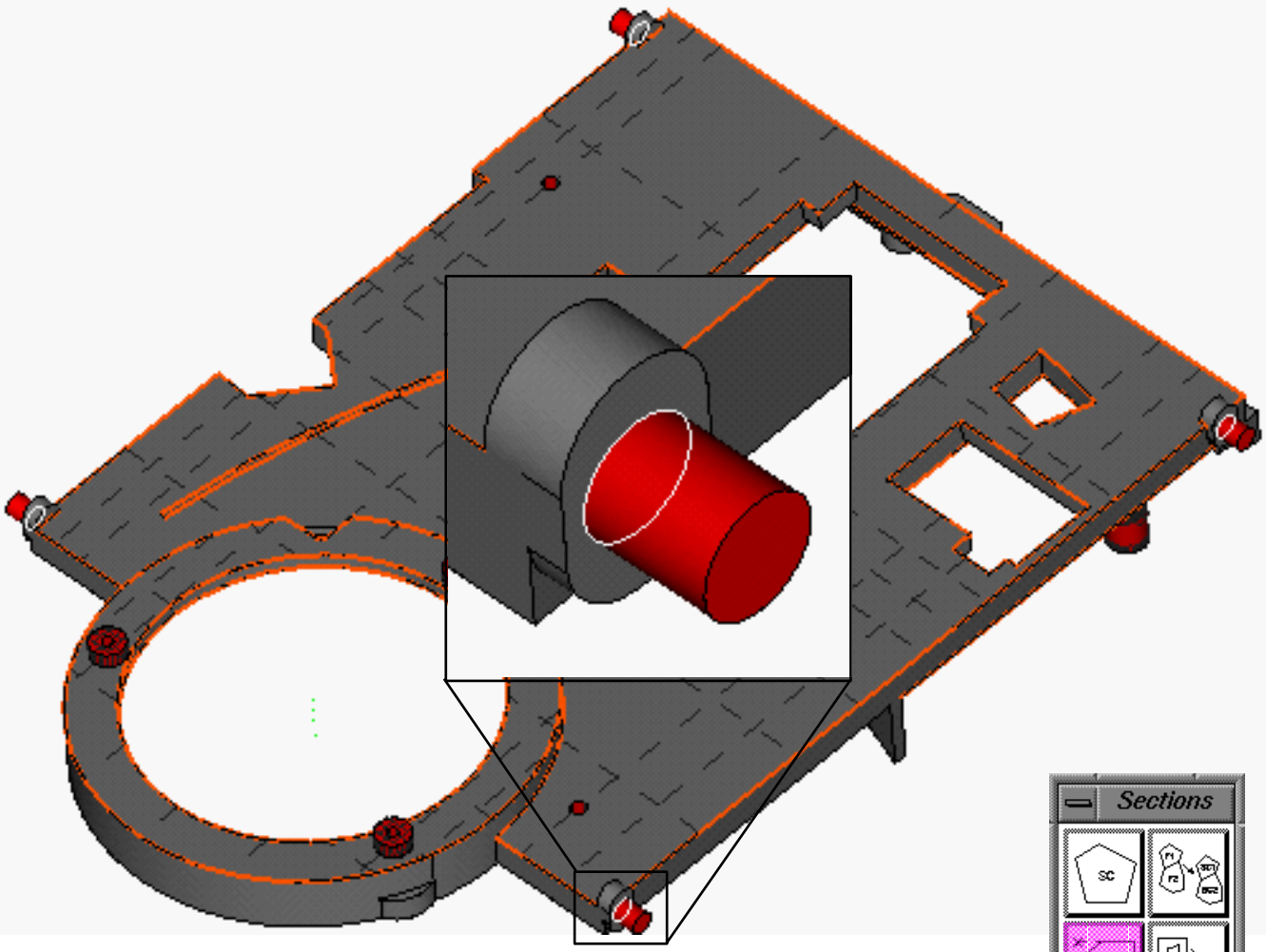


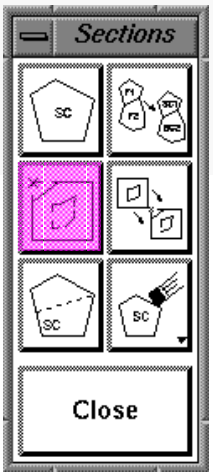
Continue selecting features to suppress.

Pick the 4 loops shown. Done



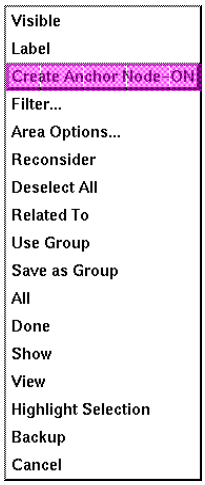
These features are now suppressed.

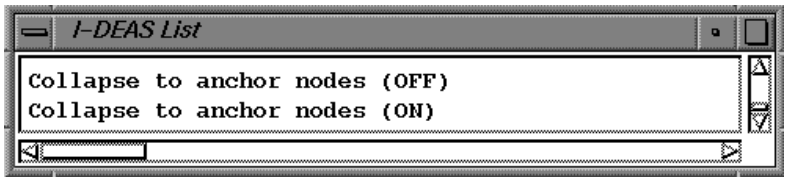


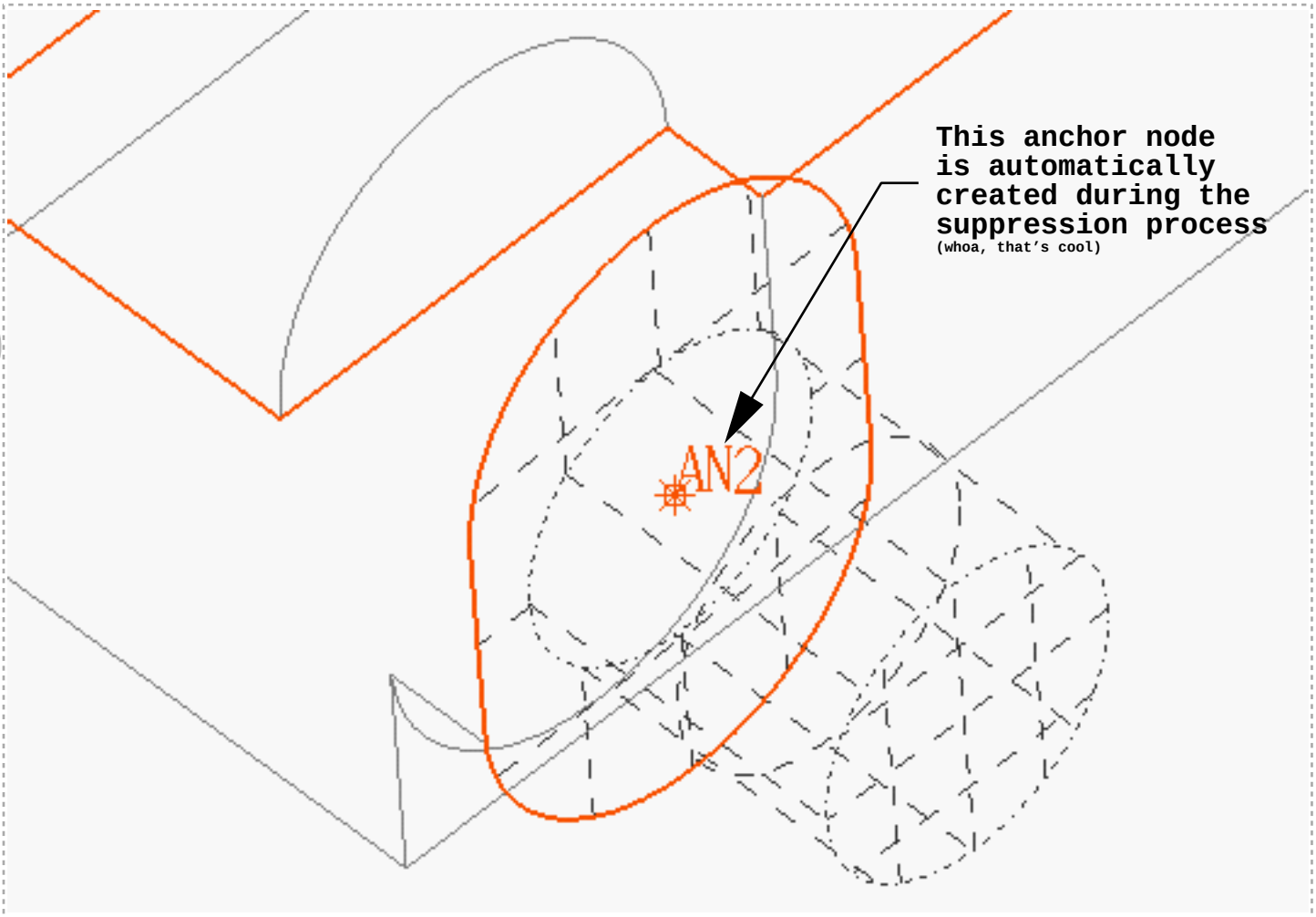


Continue selecting features to suppress.

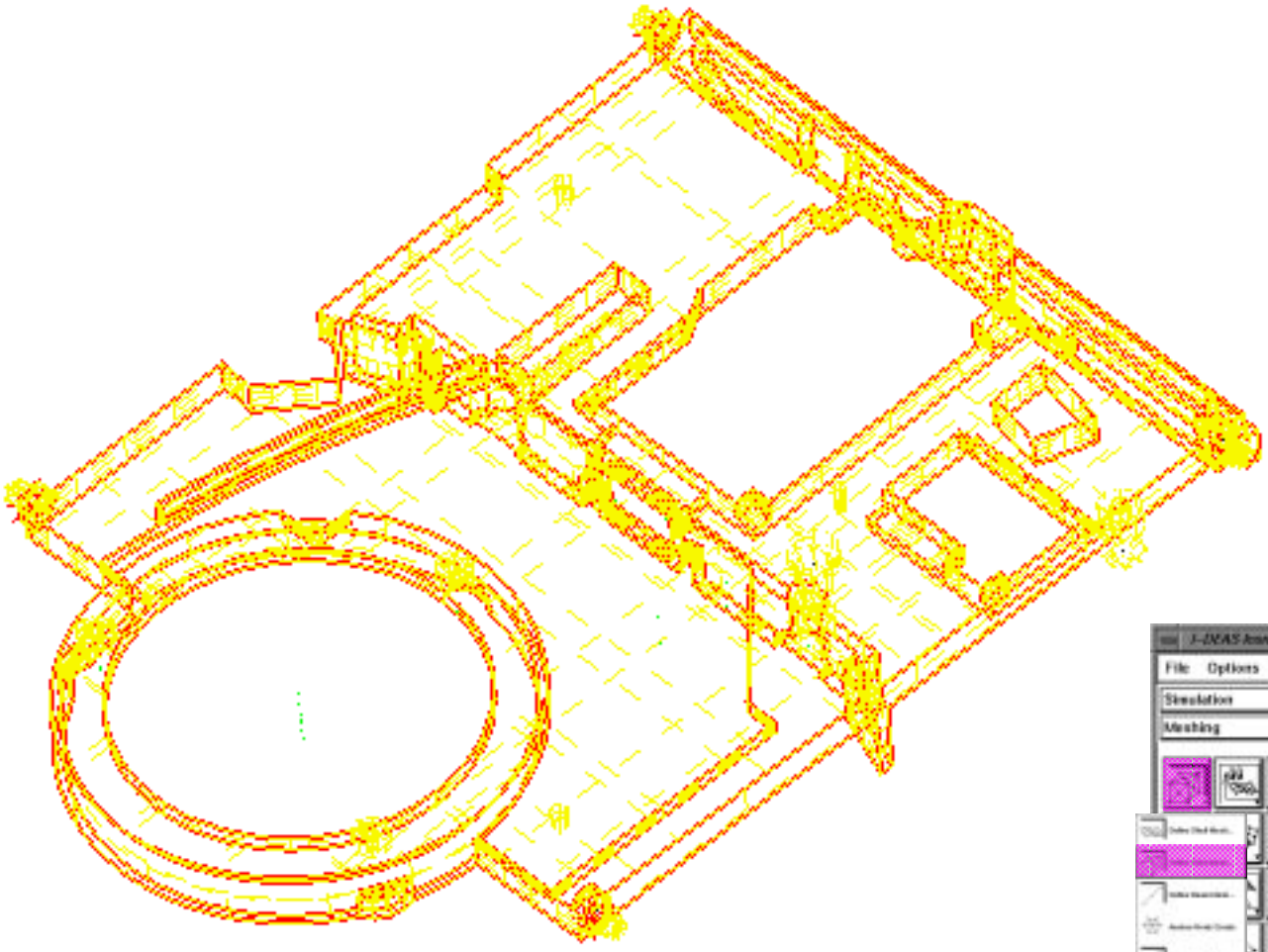
- **Manual Suppression...**
  MB3, toggle on 'Create Anchor Node'
  Pick the 4 edges of the cylindrical bosses that protrude from the side.





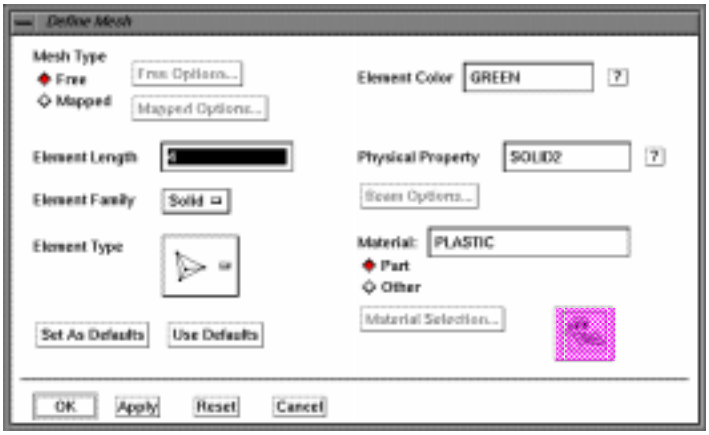


Notice that anchor nodes are created automatically for the suppressed features. This is a huge time saver. We have suppressed the feature yet retained a location that can be used to apply boundary conditions.

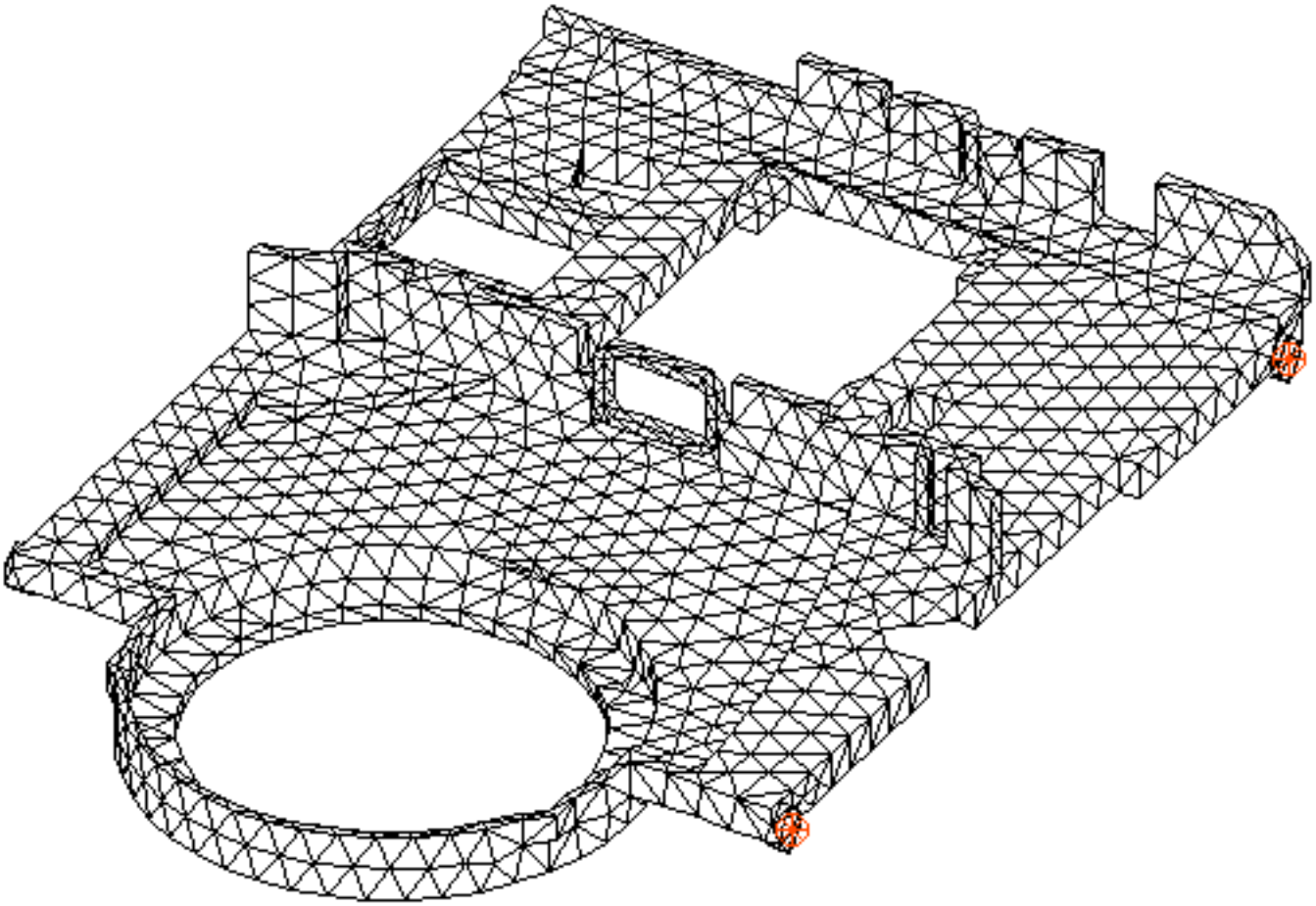


Define a Solid Element mesh

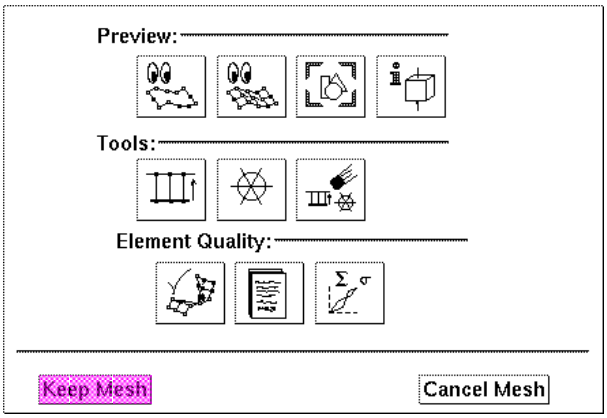
- **Define Solid Mesh...**
  Pick the part
  Fill the form out as shown.
  Use linear elements



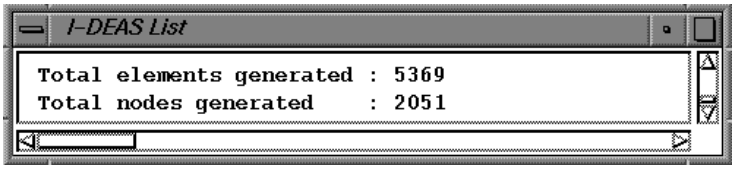
- **Modify Mesh Preview...**



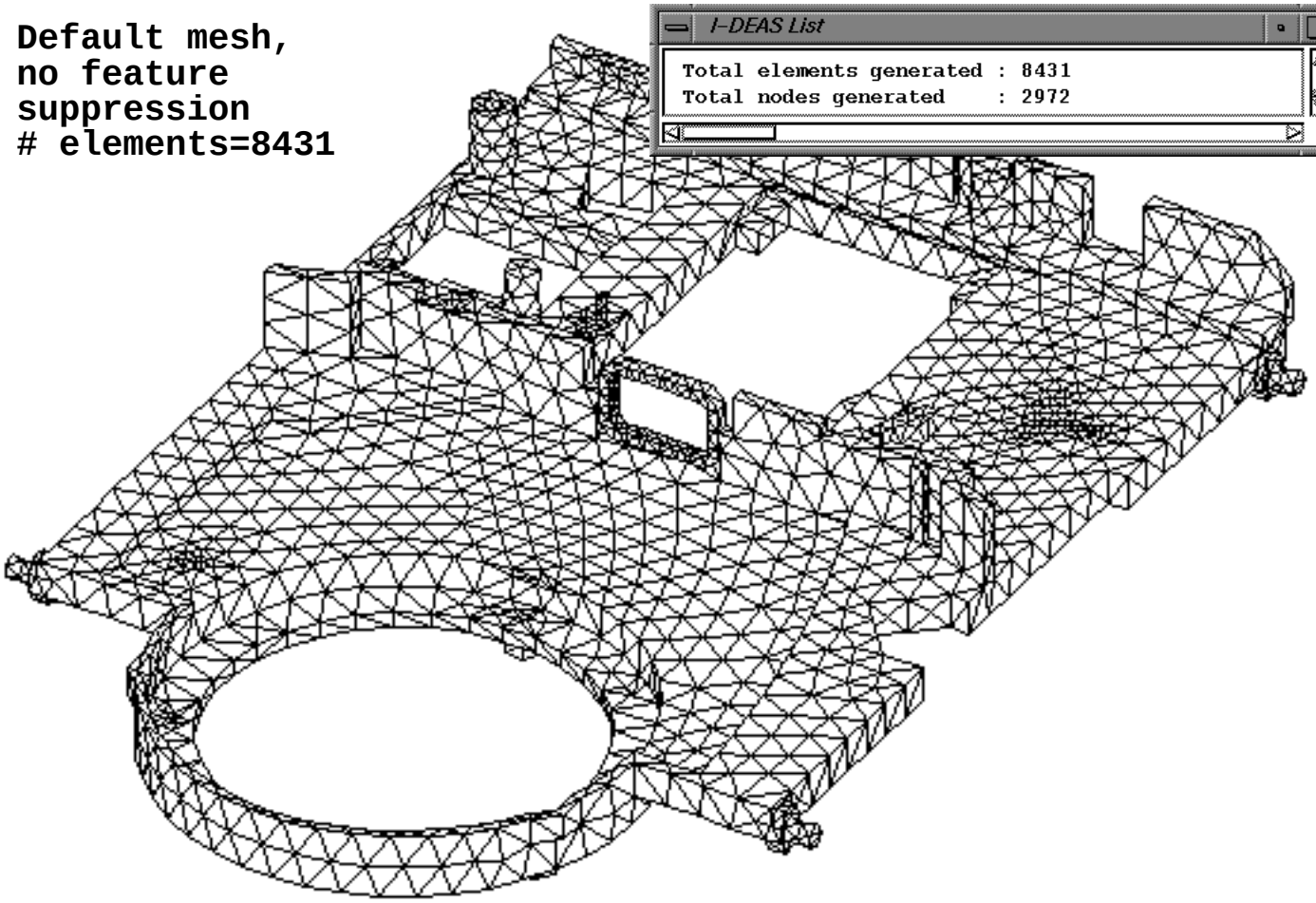
Mesh with suppression/abstraction



Notice that the suppressed features are not in the mesh. Only 5369 elements are generated. Compare this to the 8431 that are generated in the unabstracted part.



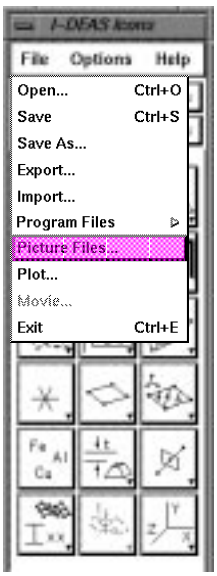
Default mesh, no feature suppression # elements=8431



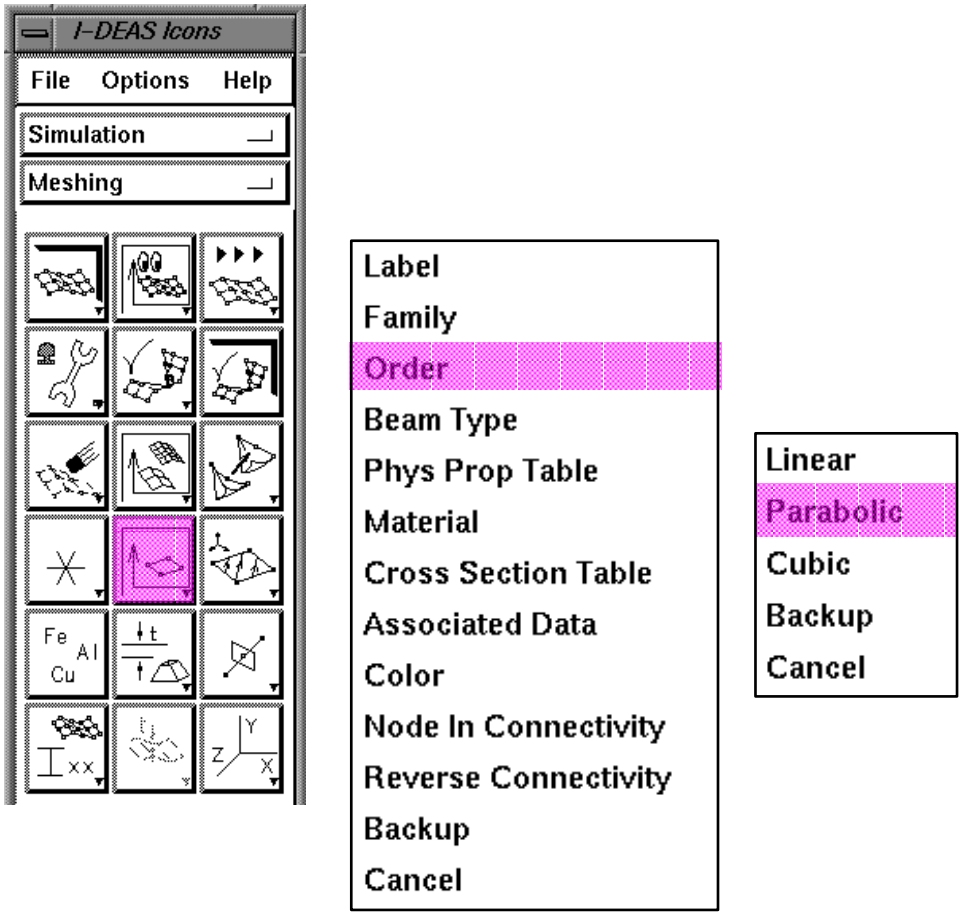
Mesh with no abstraction

Display picture file of default mesh...

•**File, Picture File**..., Display=On, Options..., SDRC Formatted=On, OK, 'Default_Mesh.pff', OK, OK.

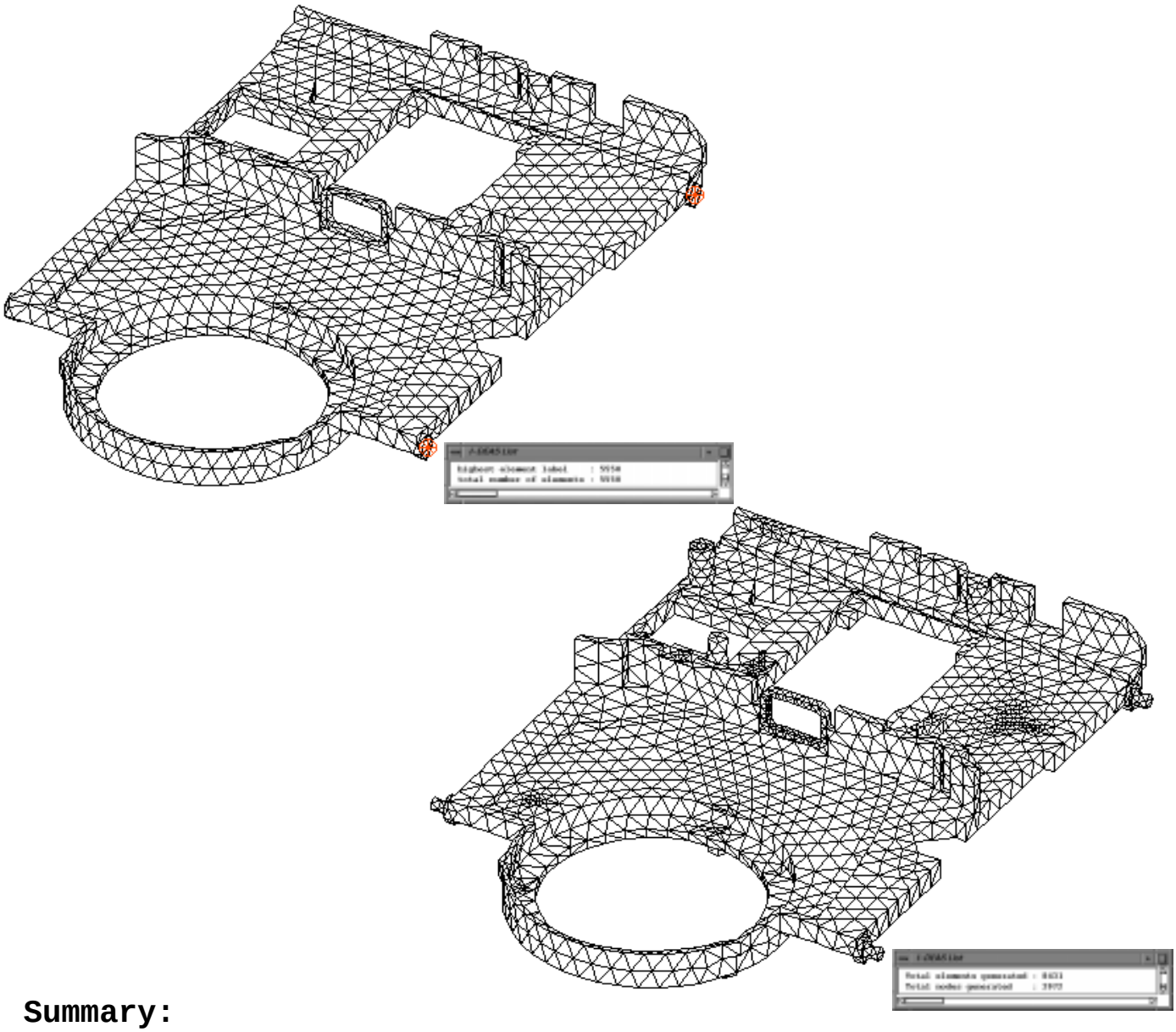


OPTIONAL



If desired (of if asked about parabolic elements), you can modify the order of the elements.

•Modify...MB3, All Done, Order, Parabolic, MB2, MB2

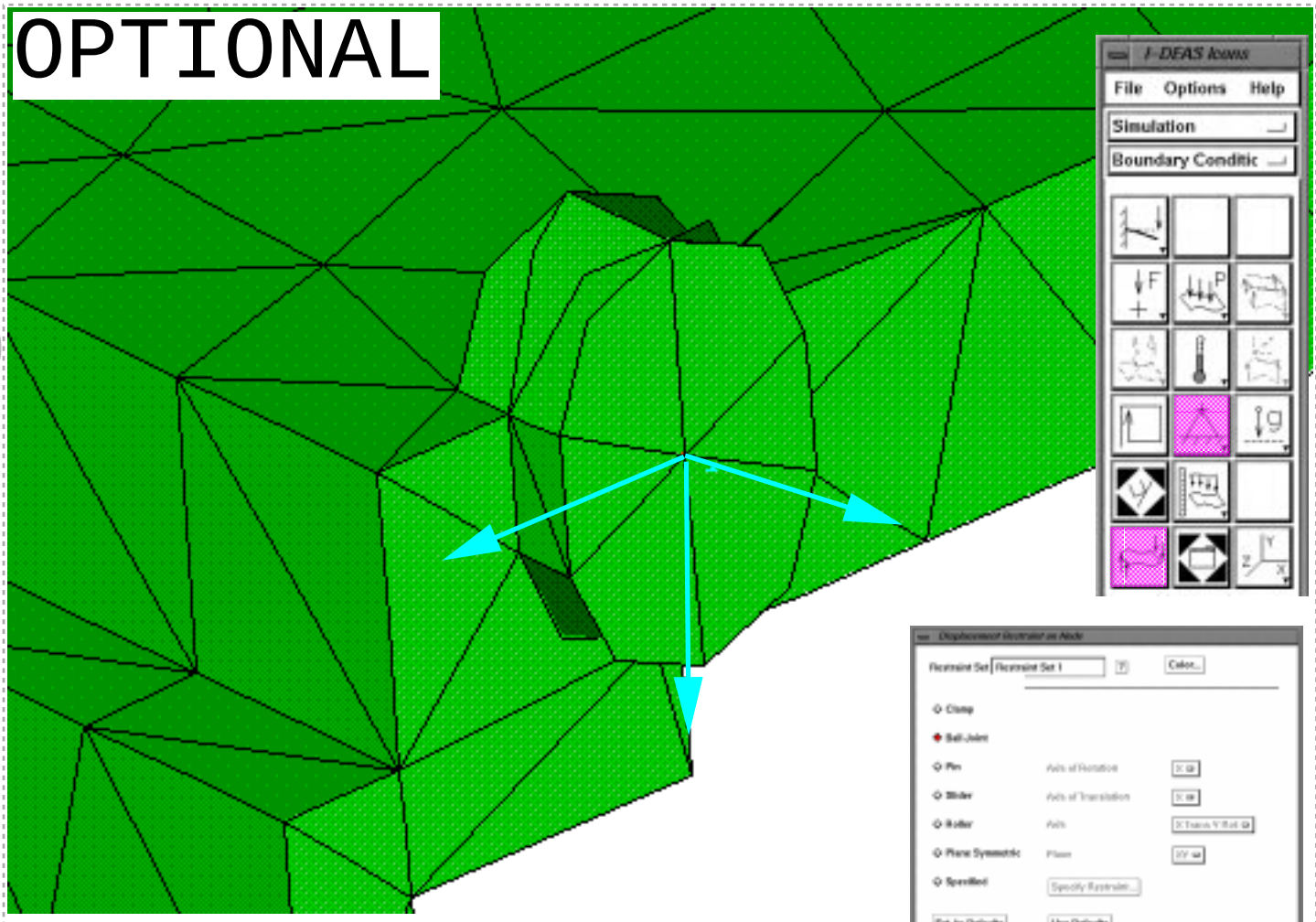


Summary:

Using suppression tools, you can quickly eliminate features from the mesh. The part can be referenced from a library. Eliminating features from the mesh results in a smaller mesh size and quicker solves. This may be insignificant for linear statics. However, a reduced mesh size may be significant for runs involving iterative solves (material non-linearities, large deformation, optimization problems).

Abstracting the four support bosses into anchor nodes keeps intact the location for the boundary conditions on the simplified model.

OPTIONAL



The next four (optional) pages show you how to perform a normal modes analysis on the part.

•**Meshing...Boundary Conditions** 

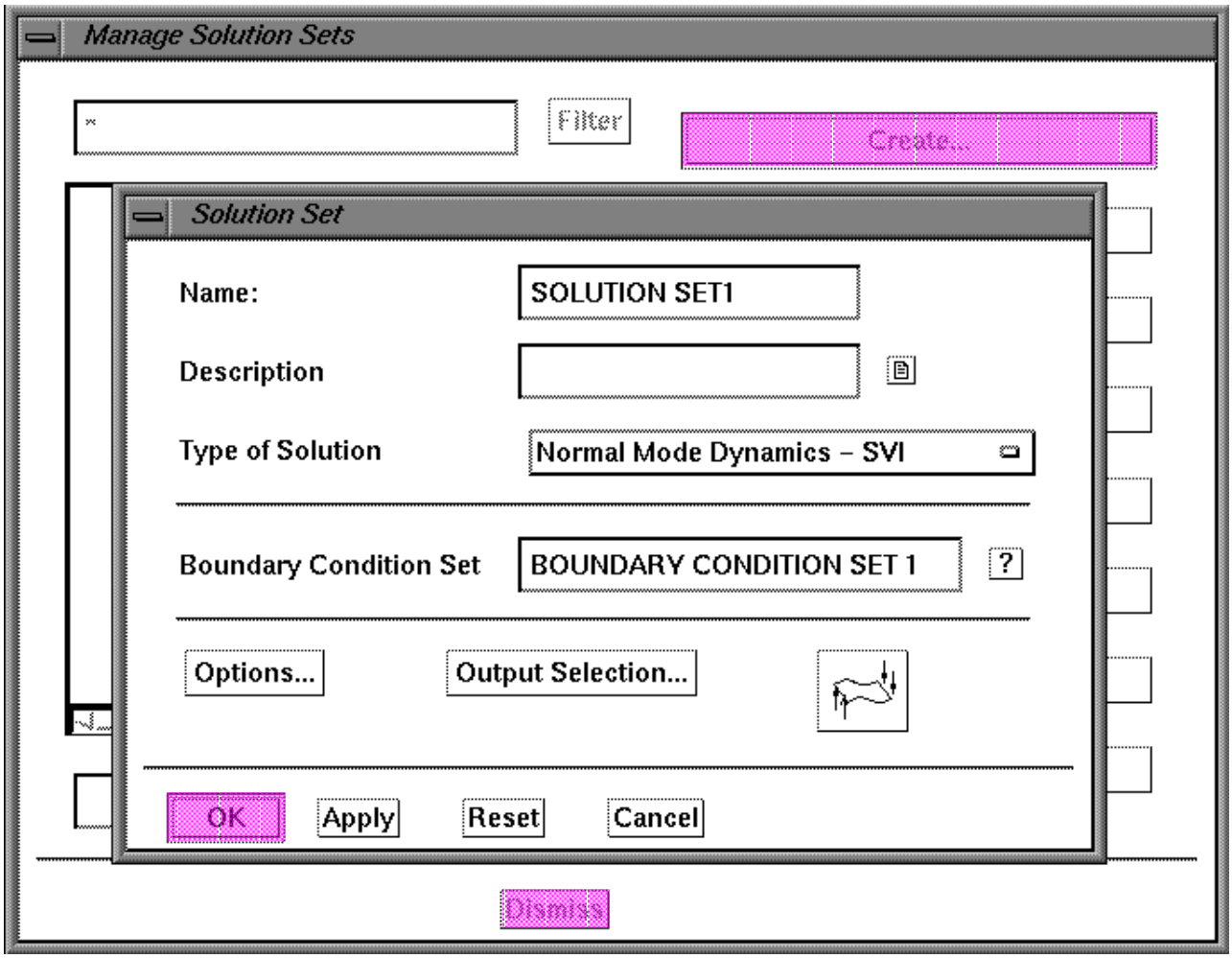
Pick the four anchor nodes and create a 'Ball Joint' type of restraint (pick nodes related to anchor nodes)..

•**Displacement Restraint...** 

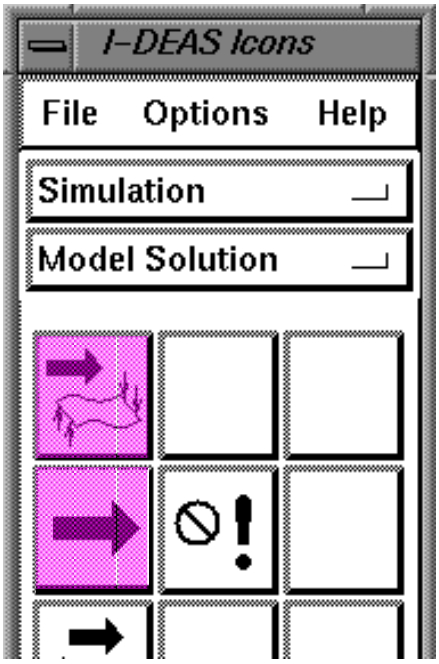
MB3, Related To, Node, Filter, Anchor Node, Pick only, *pick the 4 anchor nodes*, MB2, MB2, Ball Joint, OK.

•Create the boundary condition set

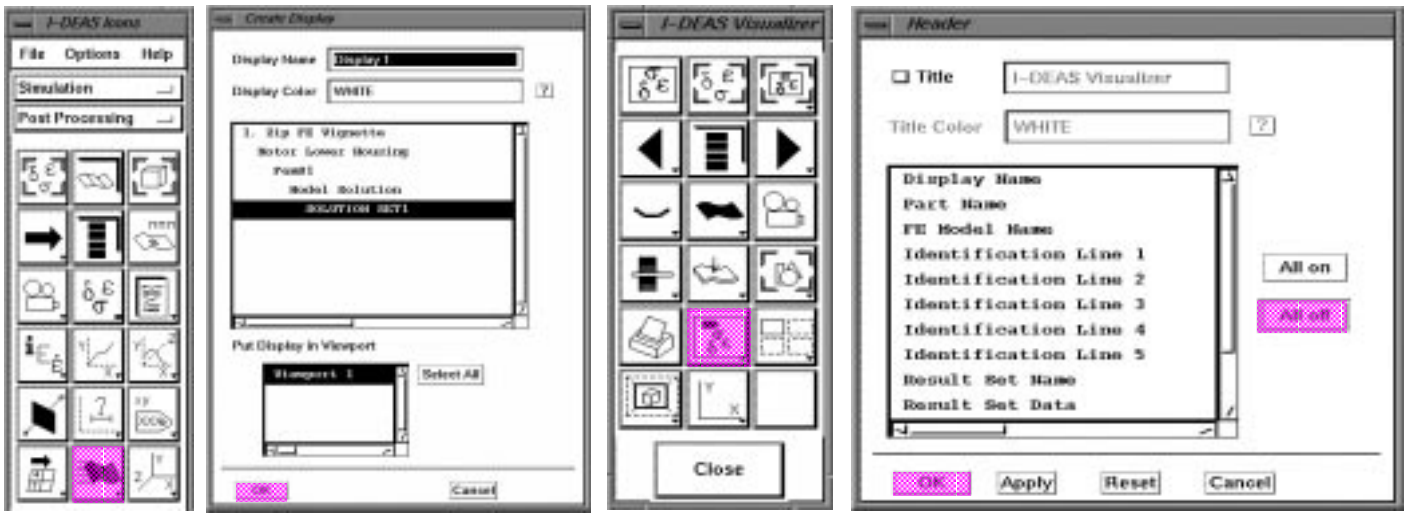
OPTIONAL

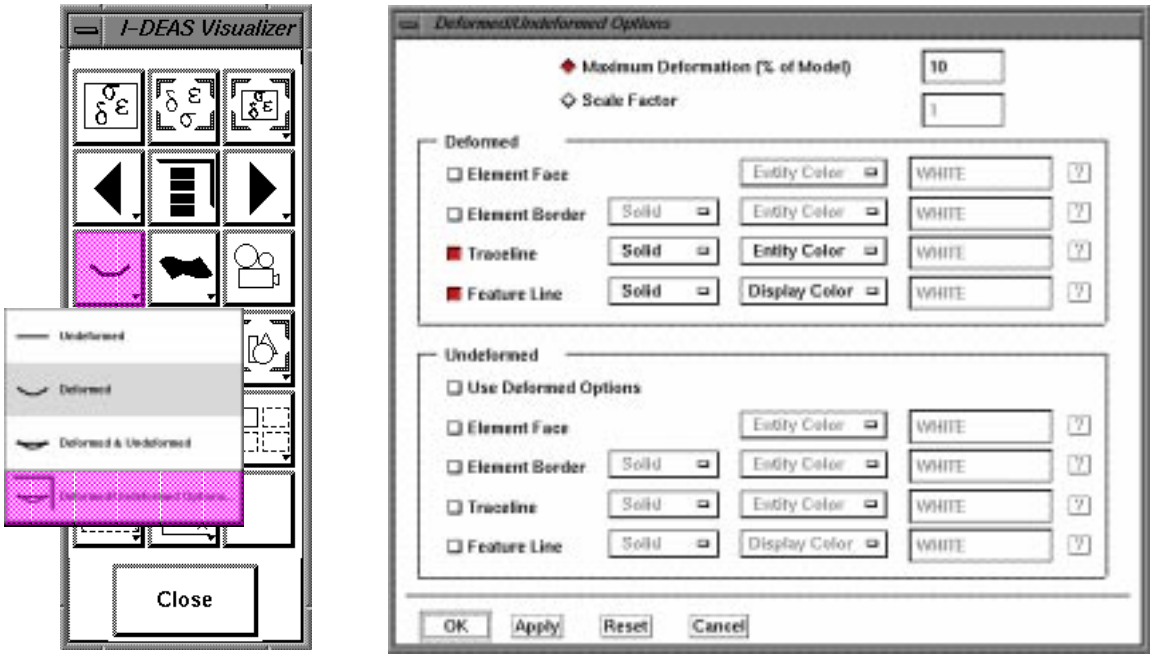


- **Boundary Conditions...Model Solution**
- **Solution Set...Create, Okay, Dismiss**
- **Solve**



OPTIONAL

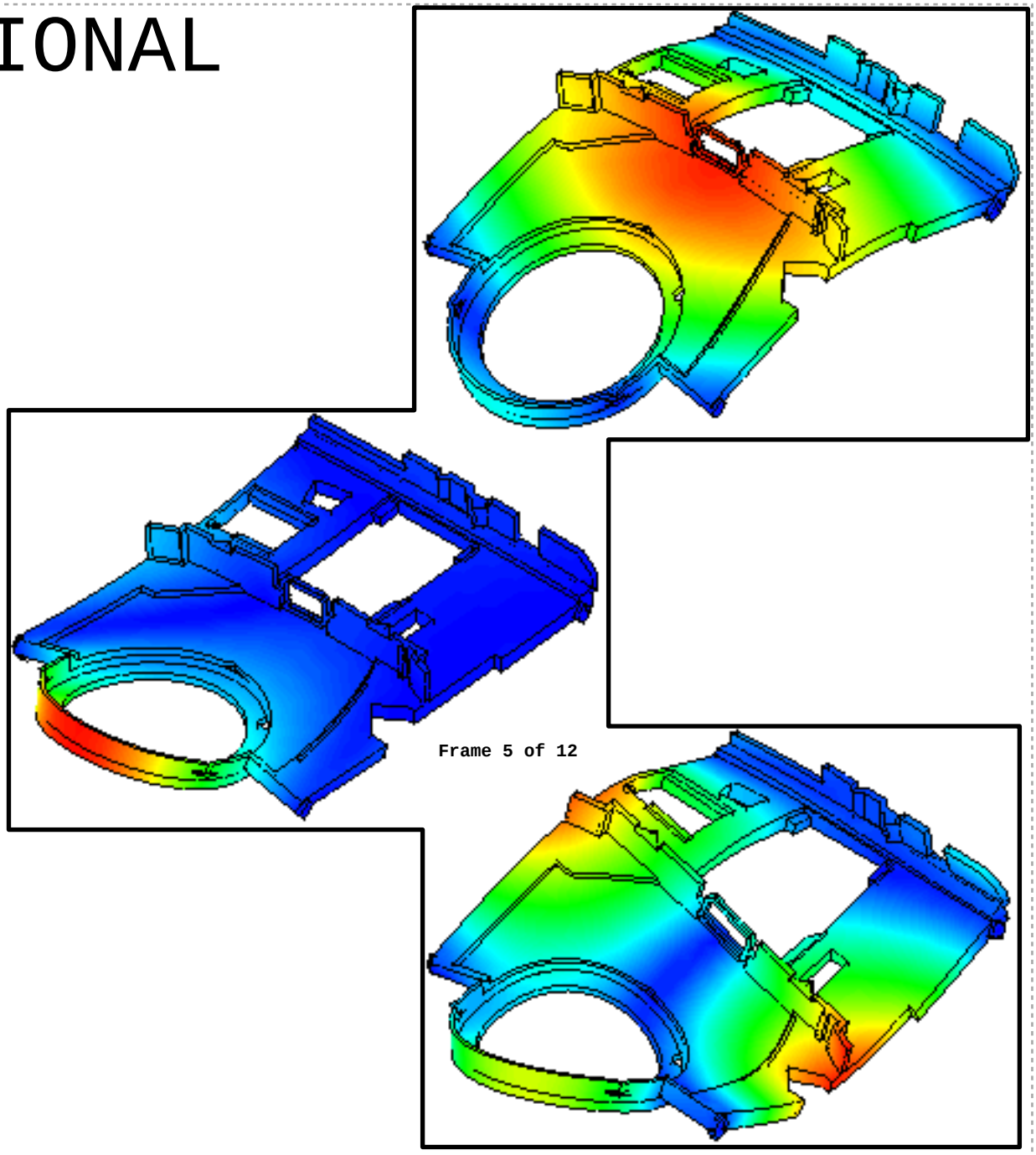




•Model Solution...Post Processing

Use the Visualizer to post the results of the 3 mode shapes. Set–up the options as shown.

OPTIONAL



Toggle through the three modes by hitting the arrow button.

•Animate a mode if desired. Set number of frames to 12.