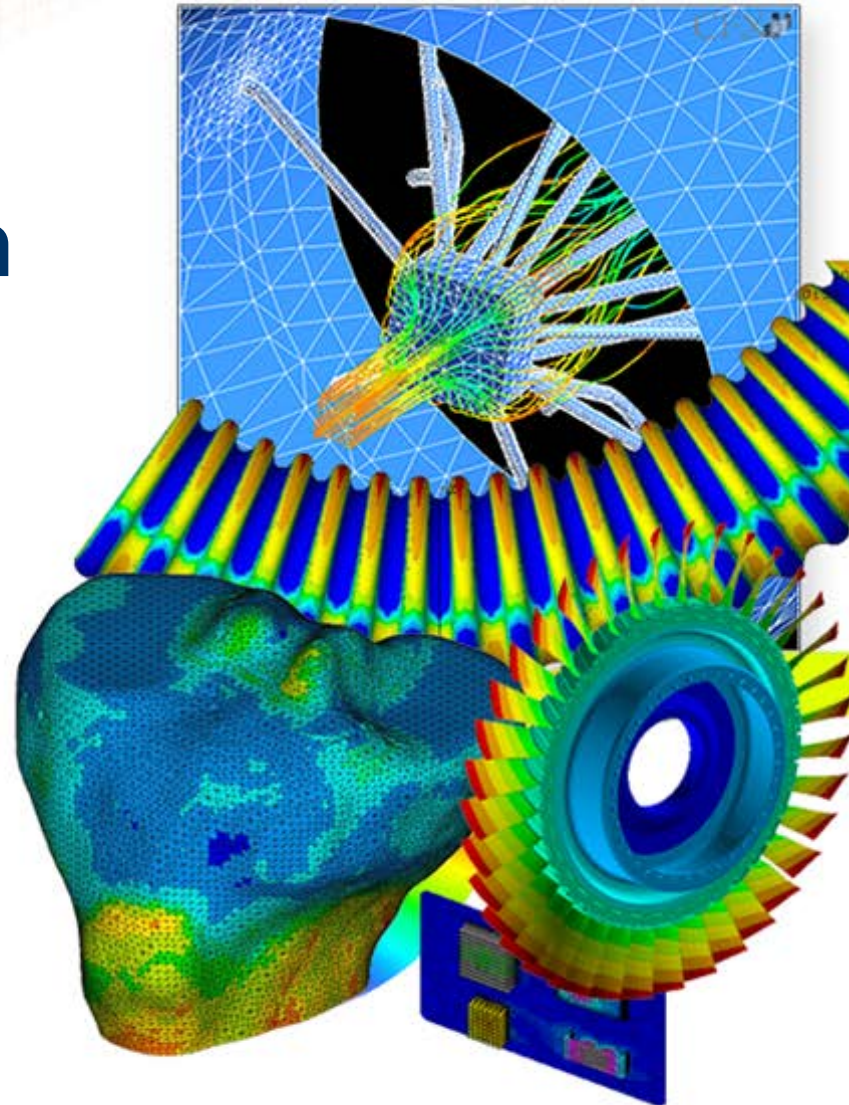


Submodeling with ANSYS v14.5

Patrick Cunningham

May 2013



- This presentation is part of a series of E-Learning webinars offered by CAE Associates.
- Other E-Learning webinar topics we have covered include:

Parametric Modeling Best Practices

Advanced Loads in ANSYS Workbench

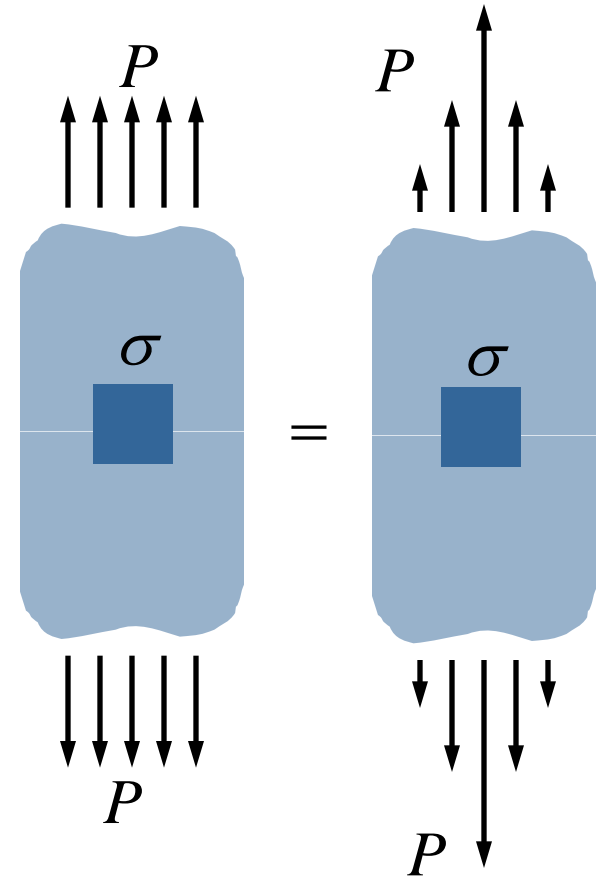
Managing Material Property Data

Working with MAPDL Commands in Workbench

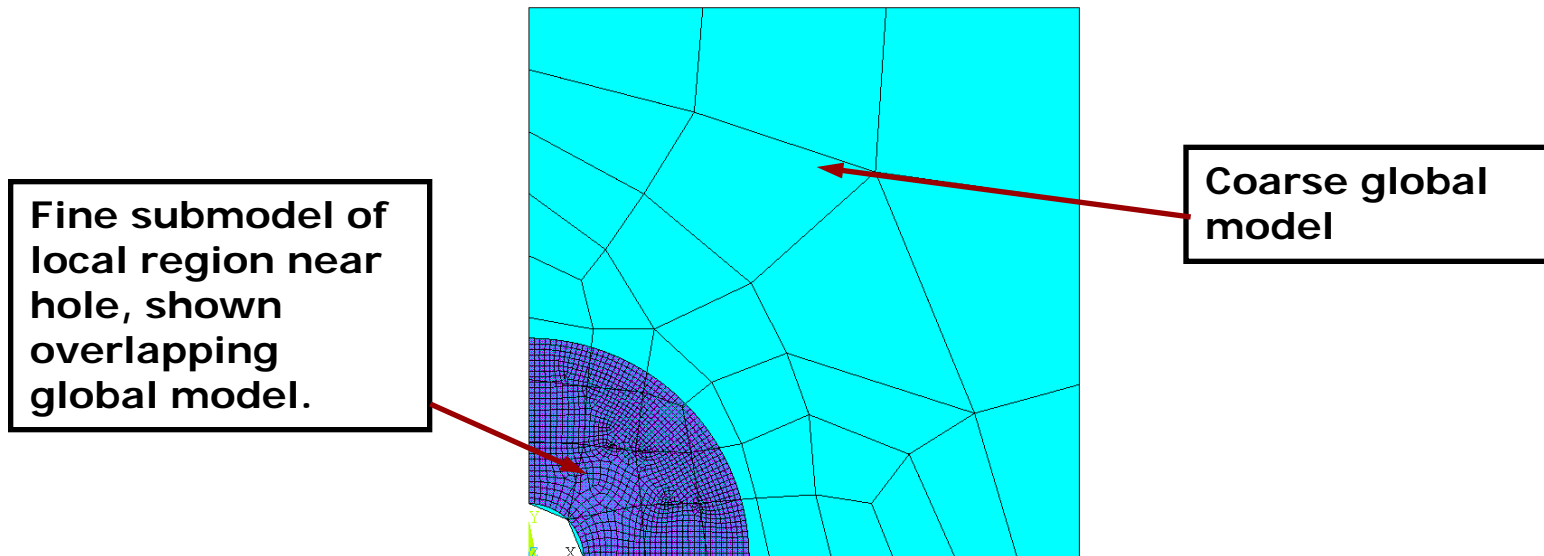
Visit www.caeai.com and click on the 'Reference Library' link to view webinar recordings and download presentation files for these and other topics.

- If you are a New Jersey or New York resident you can earn continuing education credit for attending the full webinar and completing a survey at the conclusion of the presentation.

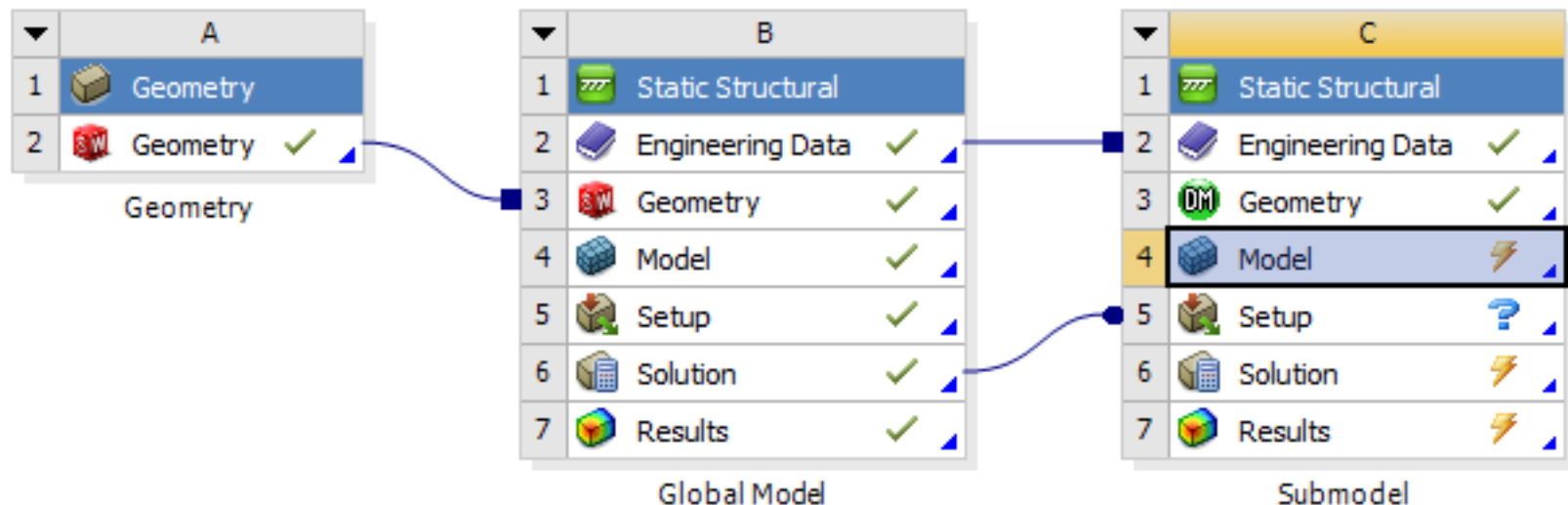
- Saint-Venant's principle states that if two different load distributions are statically equivalent, the effects on a location a sufficient distance away will be the same.
- Therefore, if the global model displacement field is used to define boundary conditions on a detailed submodel, the submodel will be an equivalent but more accurate representation of the local region.



- The basic characteristics of **submodeling**:
 - The submodel is a separate analysis from the global model.
 - The submodel includes only the local region of interest.
 - The submodel has its own geometry, so local features that may have been omitted in the global model can be included in the submodel.
 - The submodel has its own mesh, so a much finer mesh and even different element types can be used to obtain more accurate results.

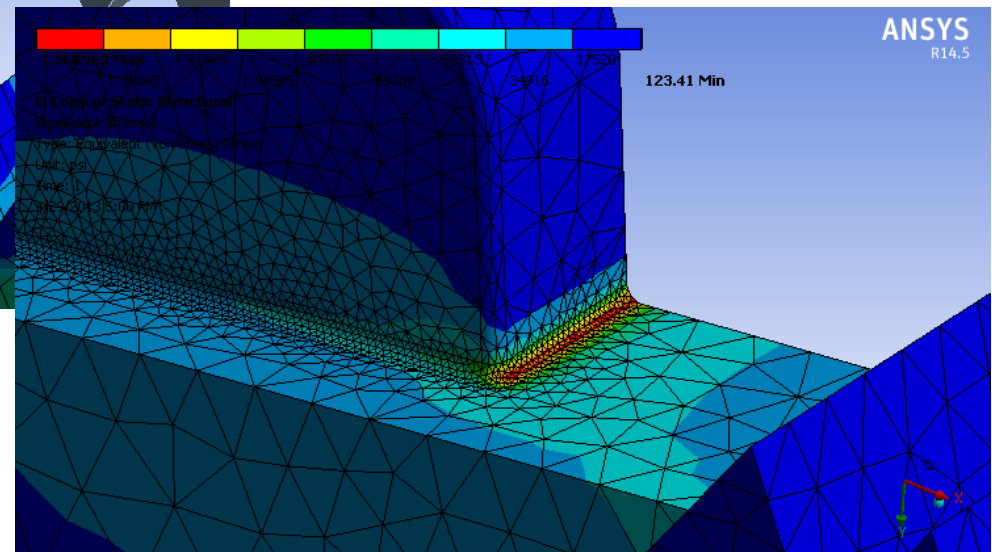
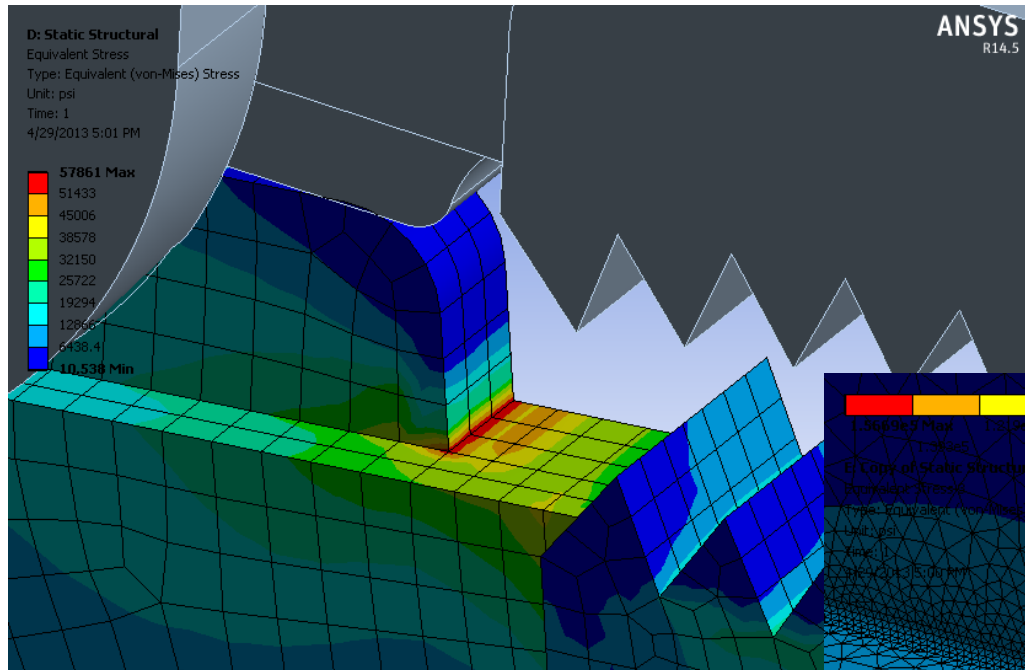


- Submodeling has been available in MAPDL (“ANSYS Classic”) for eons.
- Submodeling was possible in Workbench but required the use of APDL command blocks to accomplish the cut boundary interpolation of the displacements or temperatures.
- With **ANSYS v14.5** this effort has been reduced to simple drag and drop operations.

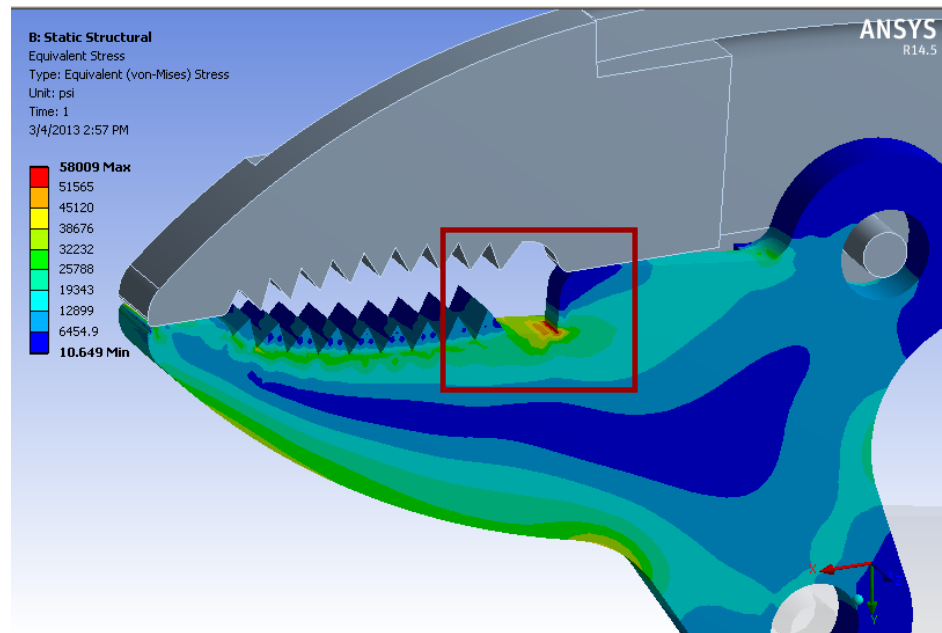


- In v14.5 the submodeling procedure consists of the following steps:
 1. Set up and run the global model.
 2. Create the submodel.
 3. Interpolate the global displacements onto the submodel at the cut boundary locations.
 4. Compare the force distribution and stress at the cut boundary locations between the submodel and the global model.
 5. Evaluate the submodel results.

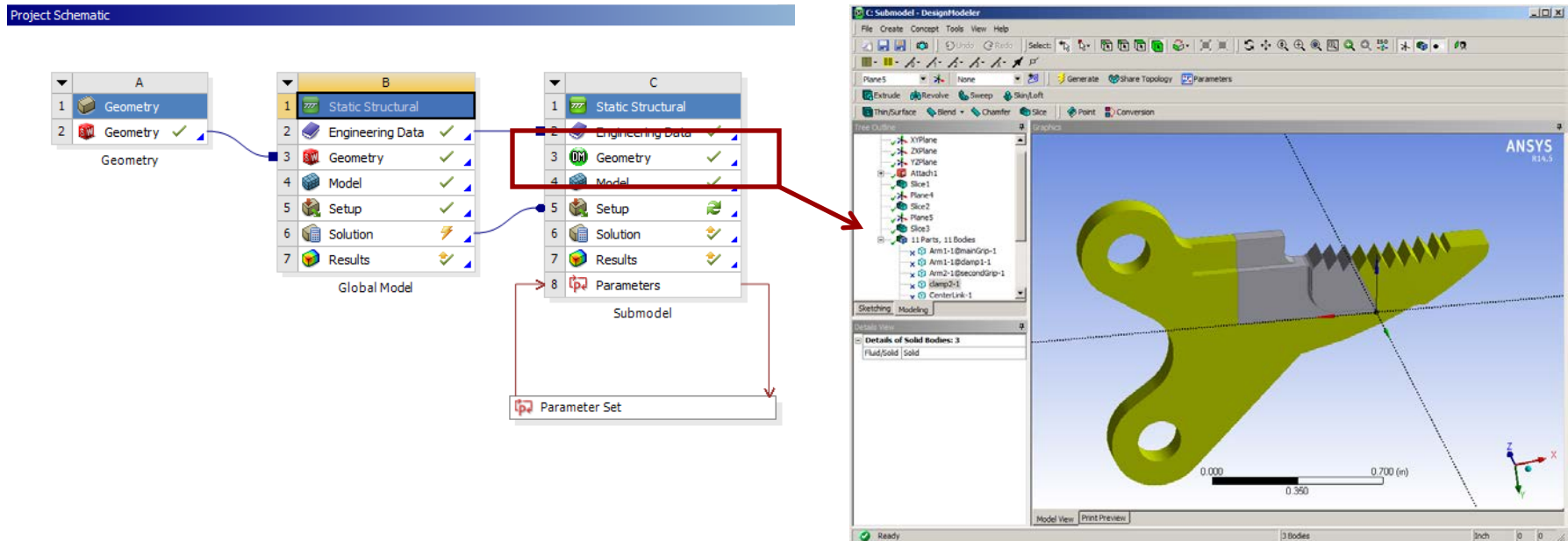
- You can omit local geometry features (fillets, rounds, chamfers, etc.) in the global model if you plan to evaluate the local region using a submodel.



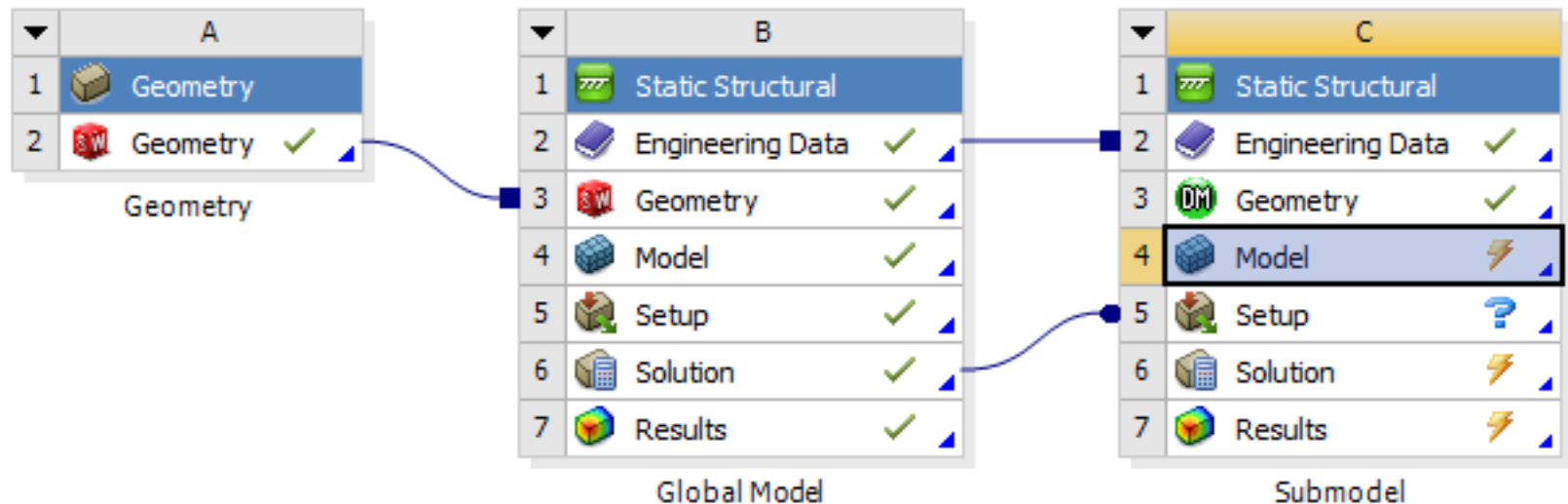
- In the analysis of a plier assembly we need to resolve the singular stress solution in the lower jaw.
- Our choices are:
 - Add a fillet to the lower jaw in the assembly model and resolve the entire model.
 - Extract a submodel of the region in question, add the fillet to the submodel and generate a local stress solution.



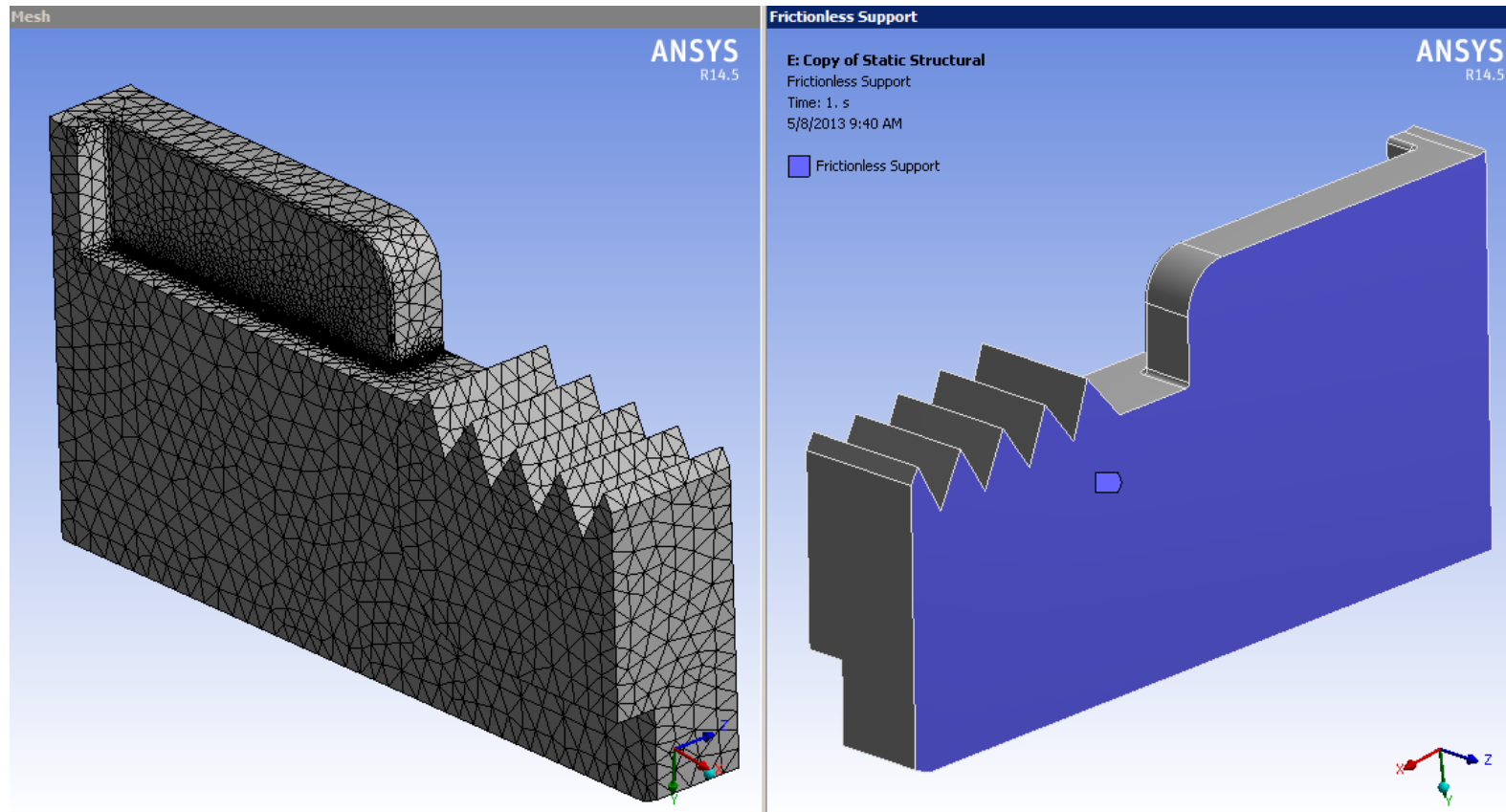
- Cut boundary displacement interpolation is a new feature in Workbench v14.5.
- DesignModeler can be used to extract the submodel from a duplicate of the global geometry. Simply create a copy of the analysis system and modify the submodel geometry locally in DesignModeler.
- In this fashion both the assembly (global) model and the submodel can reside in the same project.



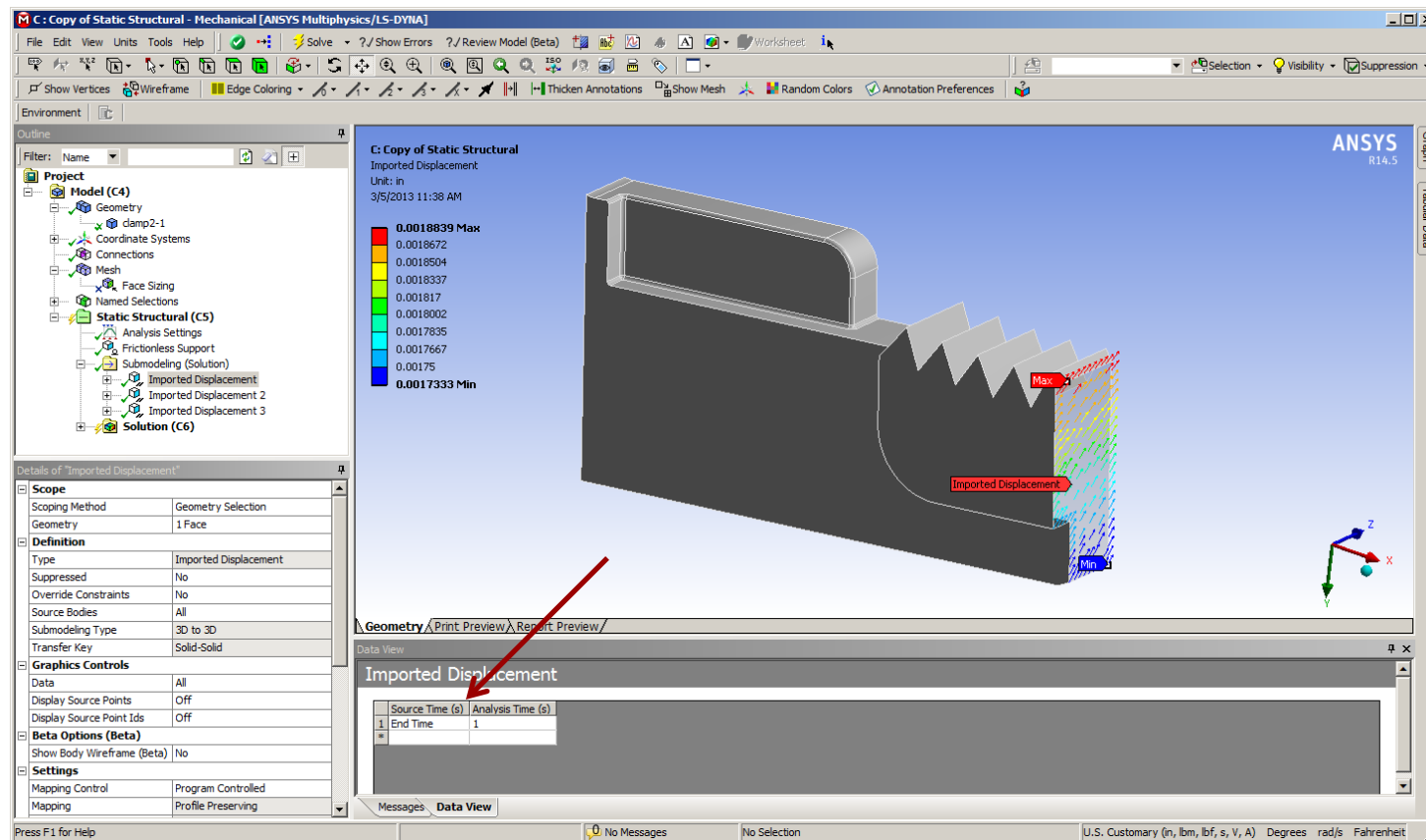
- RMB click on the Geometry row (B3) and select Duplicate.
- Drag and drop the Solution row from the global model onto the Setup row of the submodel to import the global displacements.
- Update the Solution row (B6) and Refresh the Setup row (C4).
- Open the submodel in Mechanical



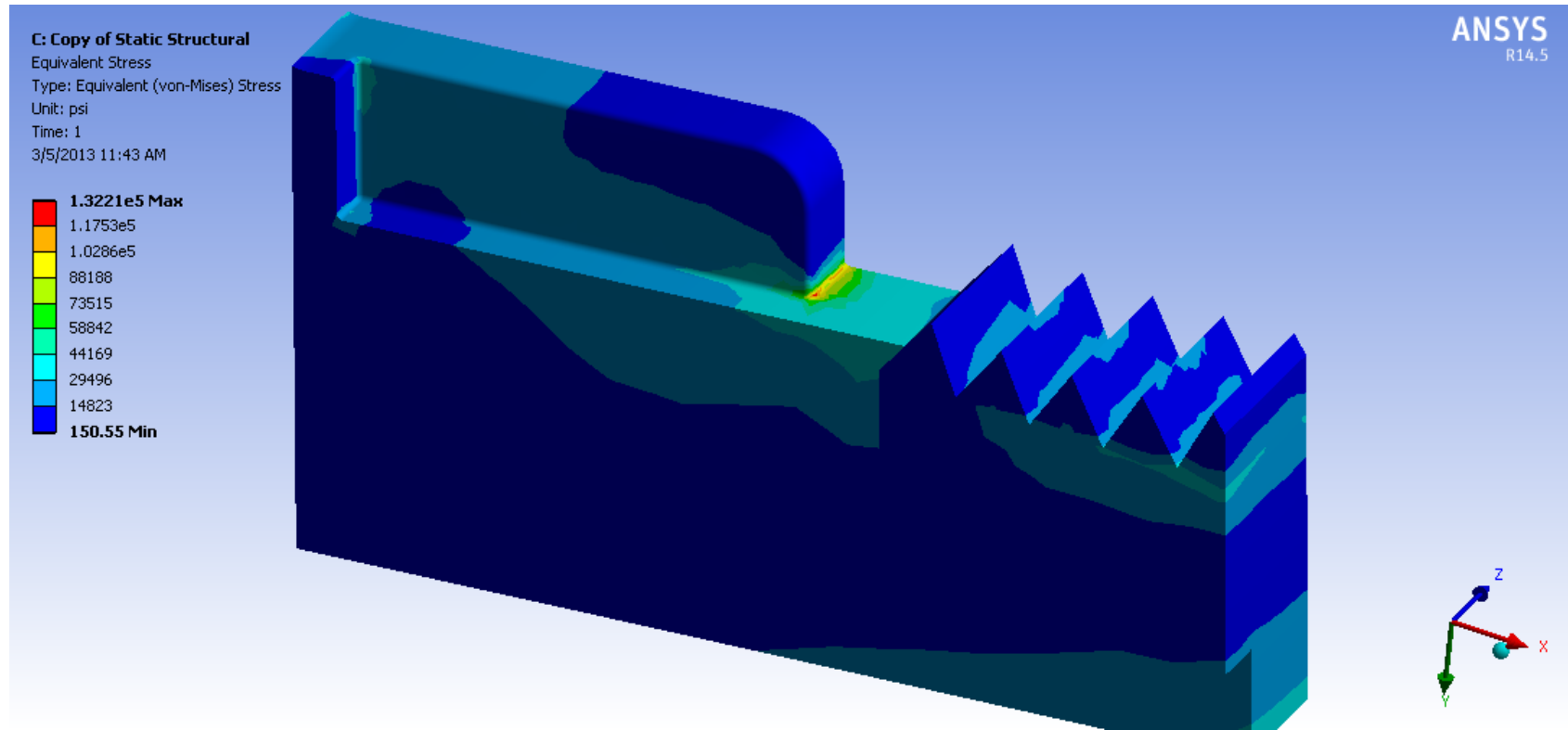
- Add mesh refinement controls to the region of interest.
- Make sure that all loads applied to the same region in the global are applied to the submodel.



- RMB on the Imported Load folder and insert an imported displacement for each cut face.
- RMB click to import the loads.
 - Note that this can be done for any result step of the global solution.

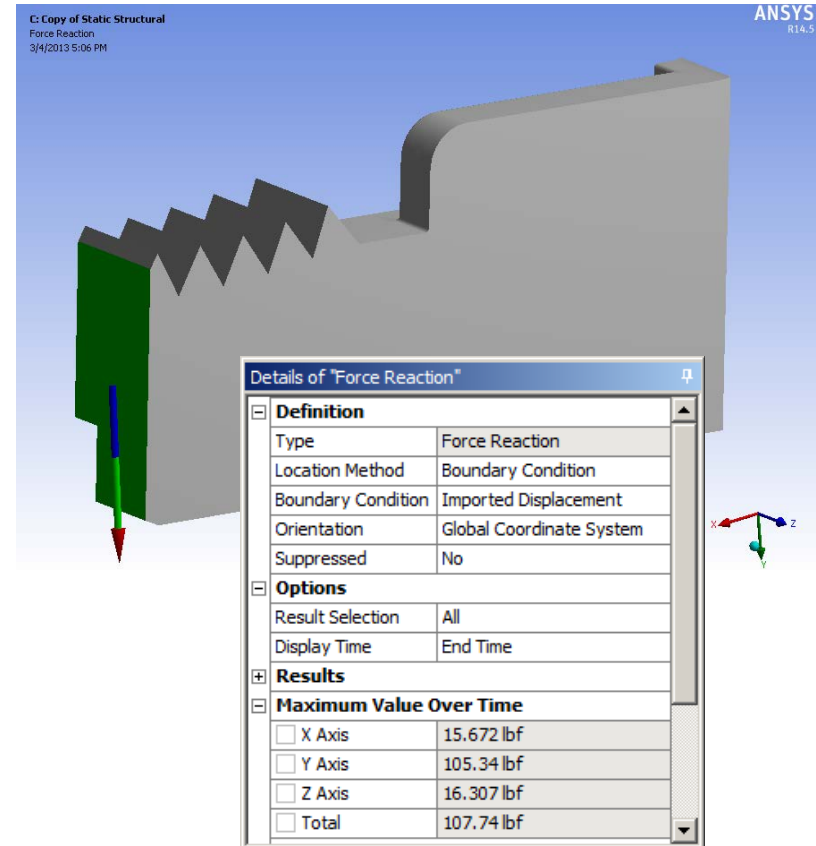
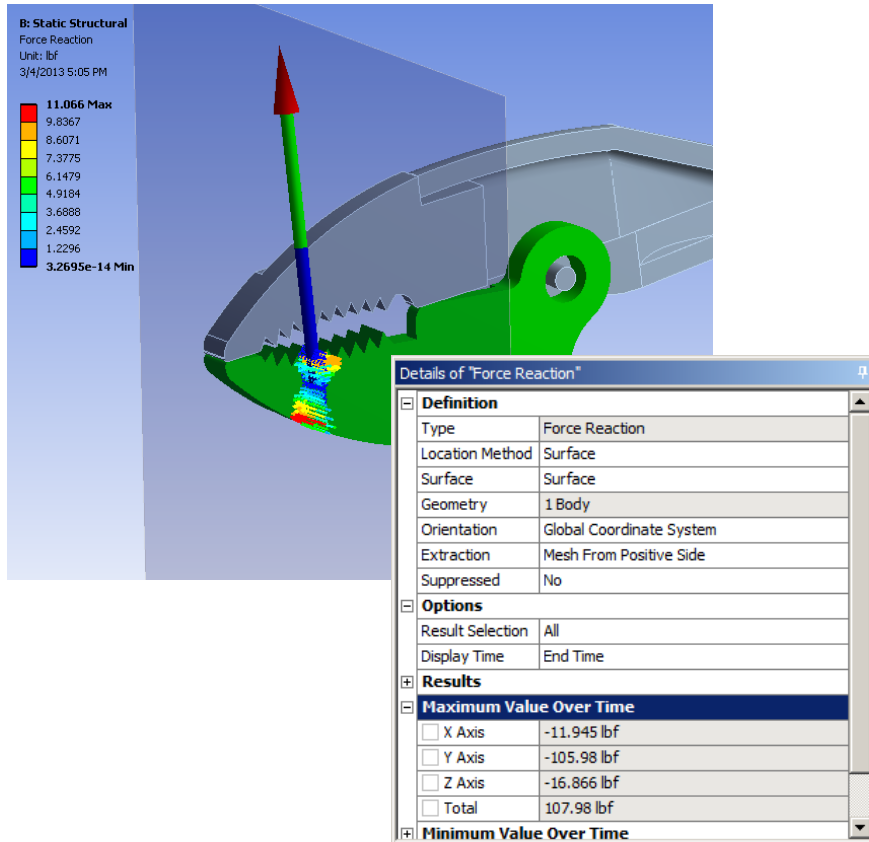


- Generate the initial solution.

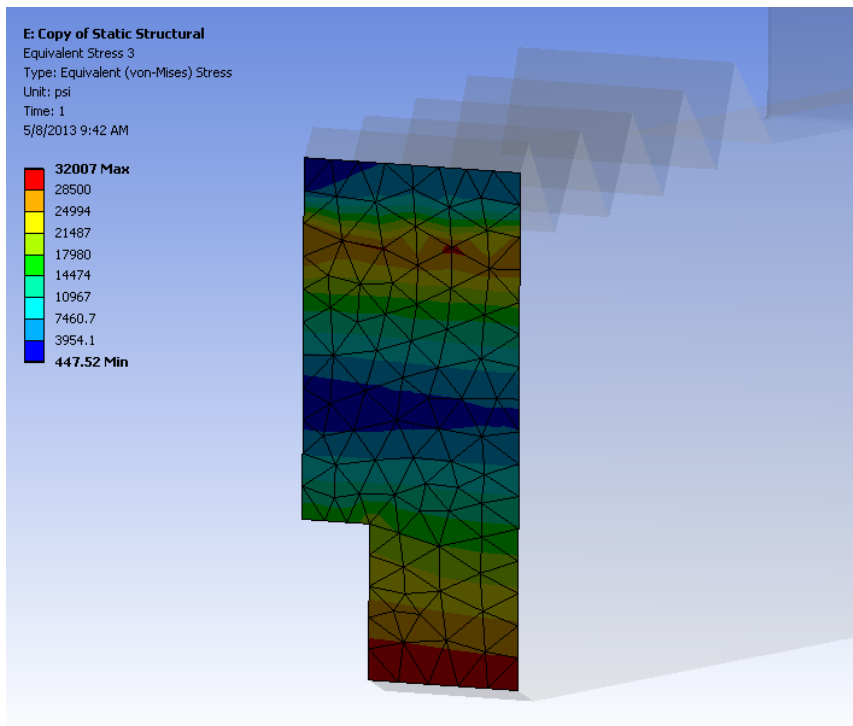
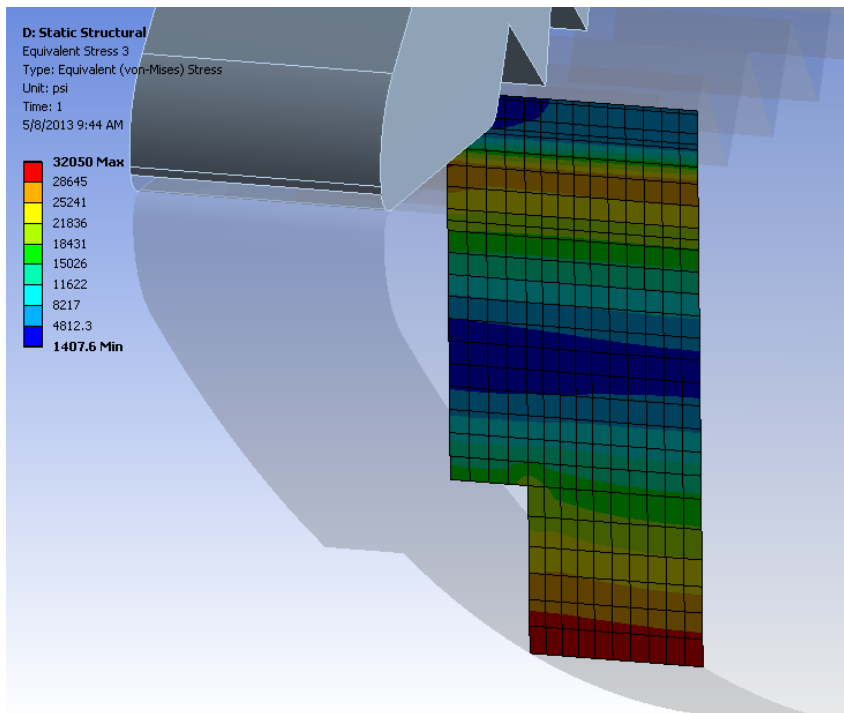


- Forces:

- Create a surface in the global model at the cut boundary using Construction Geometry.
- Scope a force reaction probe to the surface and compare with the submodel.



- Stress:
 - Use the surface in the global model to compare stress as well.



Submodel Mesh Refinement



- Use a parametric mesh study to refine the mesh to a converged stress solution.

The screenshot shows the ANSYS Workbench interface for a parametric study. The 'Table of Design Points' window displays the following data:

Name	P1 - fillet_radius	P2 - Face Sizing Element Size	P3 - Equivalent Stress 3 Maximum	Exported	Note
1					
2	Units	in	psi		
3	Current	0.005	1.368E+05		
4	DP 1	0.005	1.5156E+05		
5	DP 2	0.005	1.5633E+05		
6	DP 3	0.005	1.5774E+05		

The 'Parameter Chart 0' window shows a line graph of 'P3 - Equivalent Stress 3 Maximum [psi]' versus 'P2 - Face Sizing Element Size [in]'. The data points from the table are plotted, showing a decreasing trend in stress as element size increases, with a sharp drop at the final point.

