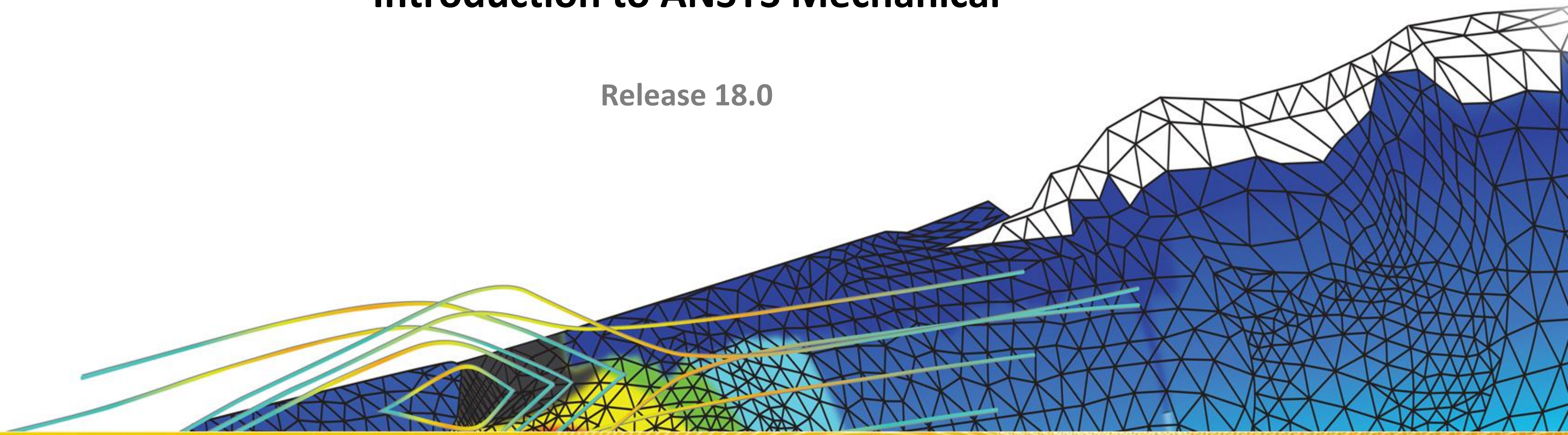
Realize Your Product Promise®

Module 08: Eigenvalue Buckling and Submodeling

Introduction to ANSYS Mechanical

Release 18.0



Module 08 Topics

This module covers introductory topics for eigenvalue buckling and submodeling:

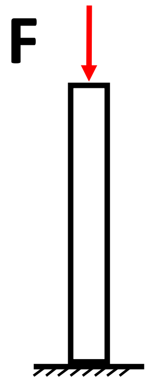
1. Eigenvalue Buckling Overview
2. Eigenvalue Buckling Geometry and Material Properties
3. Eigenvalue Buckling Pre-Stress Analysis
4. Eigenvalue Buckling with Linear Pre-Stress: Loads and Supports
5. Eigenvalue Buckling with Linear Pre-Stress: Solution
6. Eigenvalue Buckling with Linear Pre-Stress: Results
7. Submodeling Overview
8. Submodeling Procedure
9. Workshop 08.1: Eigenvalue Buckling with Linear Pre-Stress
10. Workshop 08.2: Submodeling

08.01 Eigenvalue Buckling Overview

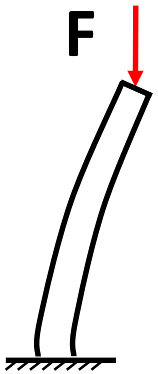
Many structures require an evaluation of their structural stability. Thin columns, compression members, and vacuum tanks are all examples of structures where stability considerations are important.

At the onset of instability (buckling) a structure will have a very large change in displacement {Δx} with essentially no corresponding change in the load (beyond a small load perturbation).

Stable



Unstable



08.01 Eigenvalue Buckling Overview

Eigenvalue buckling analysis predicts the theoretical buckling strength of an *idealized* elastic structure.

This method corresponds with the textbook approach to elastic buckling analysis; the eigenvalue buckling analysis of an Euler column will match the classical Euler solution.

Imperfections and nonlinear behavior prevent most real world structures from achieving their theoretical elastic buckling strength; thus, eigenvalue buckling generally yields *non-conservative* results because it does not account for these effects.

Although non-conservative, eigenvalue buckling analysis offers the advantages of being computationally inexpensive compared to nonlinear buckling solutions and providing approximate, albeit non-conservative, prediction of the buckling conditions

08.01 Eigenvalue Buckling Overview

In an eigenvalue buckling analysis, the following eigenvalue problem is solved to obtain the buckling load multiplier λ_i and buckling modes ψ_i:

$$([K]+\lambda_i[S])\{\psi_i\}=0$$

In this solution, the [K] and [S] matrices are *assumed to be constant*, which means that the solution is linear. It is possible, however, to include all types of nonlinearities in the "pre-stress" static structural environment (more on this below) that is used to form the [K] and [S] matrices for the subsequent eigenvalue solution. This possibility may allow for a more accurate representation of nonlinearities in the physical system in comparison with a traditional linear pre-stress state.

08.02 Eigenvalue Buckling Geometry and Material Properties

Any type of geometry supported by Mechanical may be used in buckling analyses:

- Solid bodies
- Surface bodies (with appropriate thickness defined)
- Line bodies (with appropriate cross-section defined)
  - Only buckling modes and displacement results are available for line bodies.
- Although Point Masses may be included in the model, they will not have any effect in an Eigenvalue Buckling analysis, as there are no inertial loads; thus, the applicability of this feature may be limited.

For material properties, *Elastic Modulus* and *Poisson’s Ratio* are required as a minimum.

08.03 Eigenvalue Buckling Pre-Stress Analysis

An eigenvalue buckling analysis must be preceded by a separate Static Structural analysis known as the "pre-stress analysis":

- **The primary purpose of the pre-stress analysis is to supply the [K] and [S] matrices to the eigenvalue buckling analysis.**
- **Although the eigenvalue buckling solution is always linear, the static pre-stress system can be either linear or nonlinear. This is an important distinction, as *the procedure for computing the buckling loads is dependent upon whether the pre-stress system is linear or nonlinear*.**

08.03 Eigenvalue Buckling Pre-Stress Analysis

The pre-stress analysis will be nonlinear if any of the following are present in the model:

- Nonlinear geometry (Large Deflection = On)
- Nonlinear materials
- Nonlinear joints (only Bushing or General joints currently have the possibility of being nonlinear)
- Nonlinear springs
- Any contact regions *other than*:
  - Bonded Type *with the MPC Formulation*
  - No Separation Type *with the MPC Formulation*

08.03 Eigenvalue Buckling Pre-Stress Analysis

The last bullet on the previous slide is important to understand. It means that Bonded contact using the Pure Penalty formulation (for example) will cause the pre-stress system to be interpreted as nonlinear, and thus will require a different method for computing the buckling loads that the one presented below. The interpretation of certain Bonded and No Separation contact formulations as nonlinear differs from their typical treatment in other analysis types, and therefore can be a source of confusion.

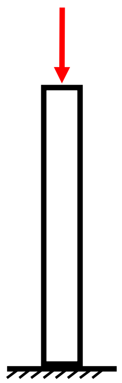
All subsequent development in this module applies to eigenvalue buckling analysis with linear pre-stress systems. Eigenvalue buckling analysis with nonlinear pre-stress systems is covered more fully in the training course ANSYS Mechanical Basic Structural Nonlinearities.

08.04 Eigenvalue Buckling with Linear Pre-Stress: Loads and Supports

All loads must be applied in the static pre-stress analysis. At least one structural load that is expected to cause buckling should be applied.

- *All* structural loads will be multiplied by the computed load multiplier (λ) to determine the buckling load (more on this below).
- Compression-only supports are not recommended.
- As with any structural solution, the structure should be fully constrained against rigid-body motion.

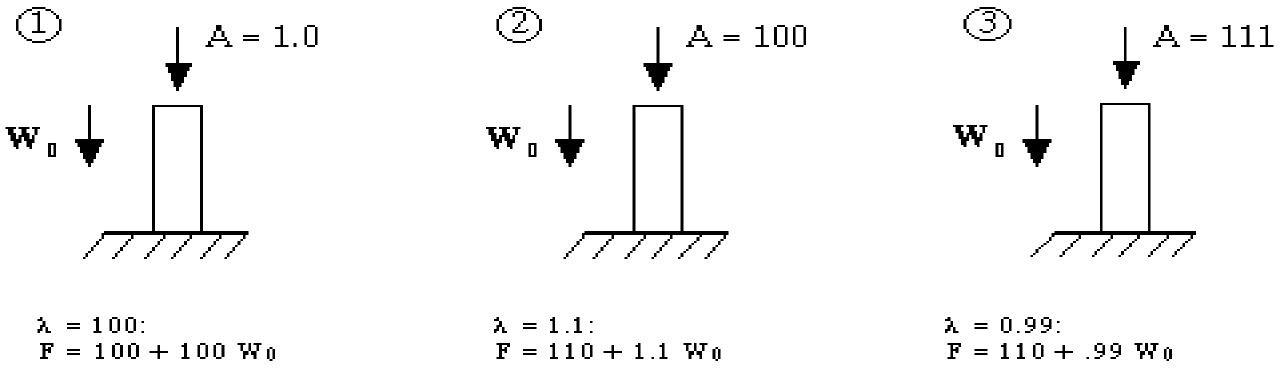
F x λ = Buckling Load



08.04 Eigenvalue Buckling with Linear Pre-Stress: Loads and Supports

Special considerations apply if both constant *and* proportional loads are present.

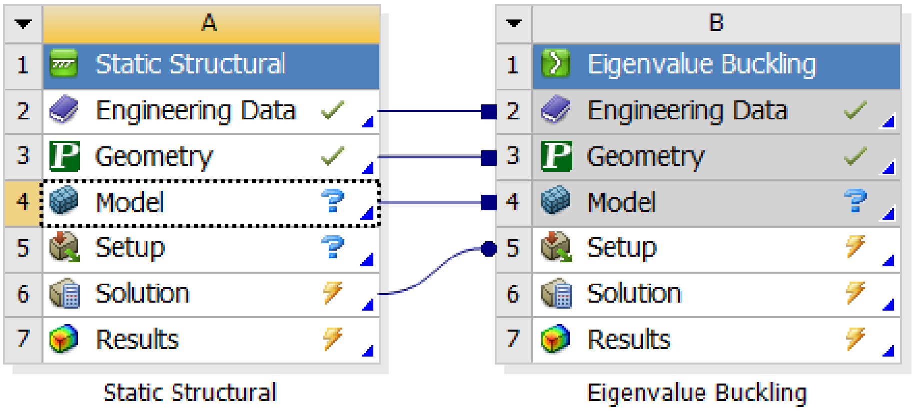
- The user must iterate on the buckling solution, adjusting the variable loads until the load multiplier becomes negligibly close to 1.0.
- Consider the example of a column with self-weight W_0 and an externally applied force A: a correct solution can be reached by iterating the buckling analysis while adjusting the value of A until $\lambda = 1.0$. This insures that self-weight = actual weight or $W_0 * \lambda = W_0$.

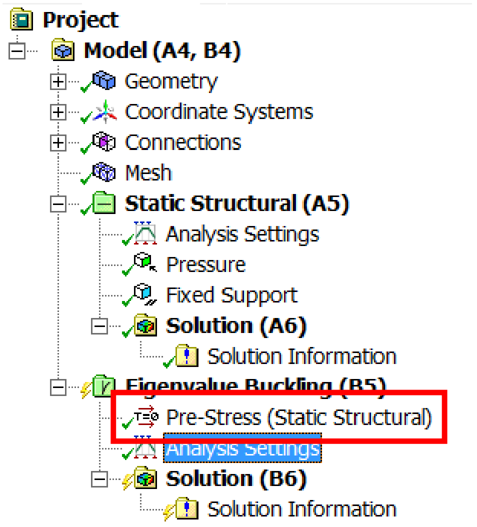


08.05 Eigenvalue Buckling with Linear Pre-Stress: Solution

An Eigenvalue Buckling analysis must be coupled to a structural analysis in the project schematic.

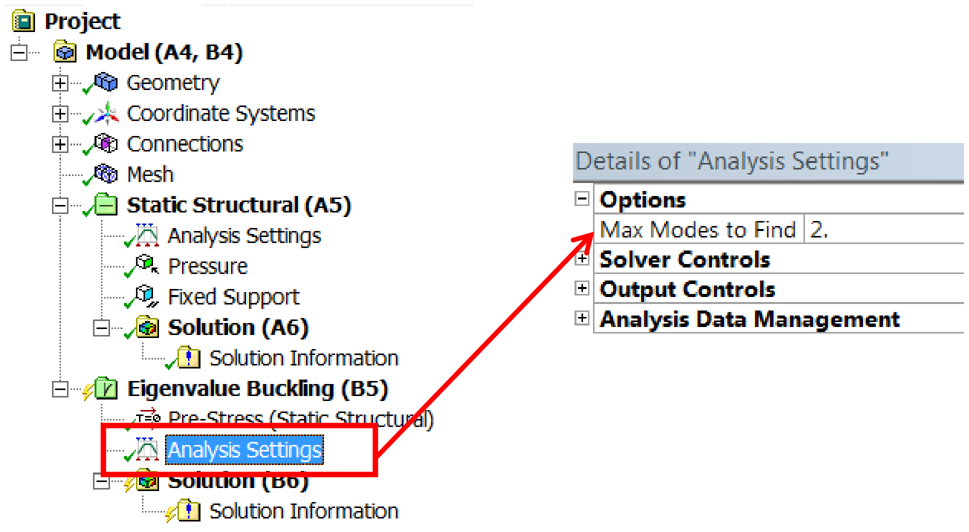
- **The Static Structural analysis system is the pre-stress analysis.**
- **The “Pre-Stress” object in the Eigenvalue Buckling environment references the results from the Static Structural analysis.**





08.05 Eigenvalue Buckling with Linear Pre-Stress: Solution

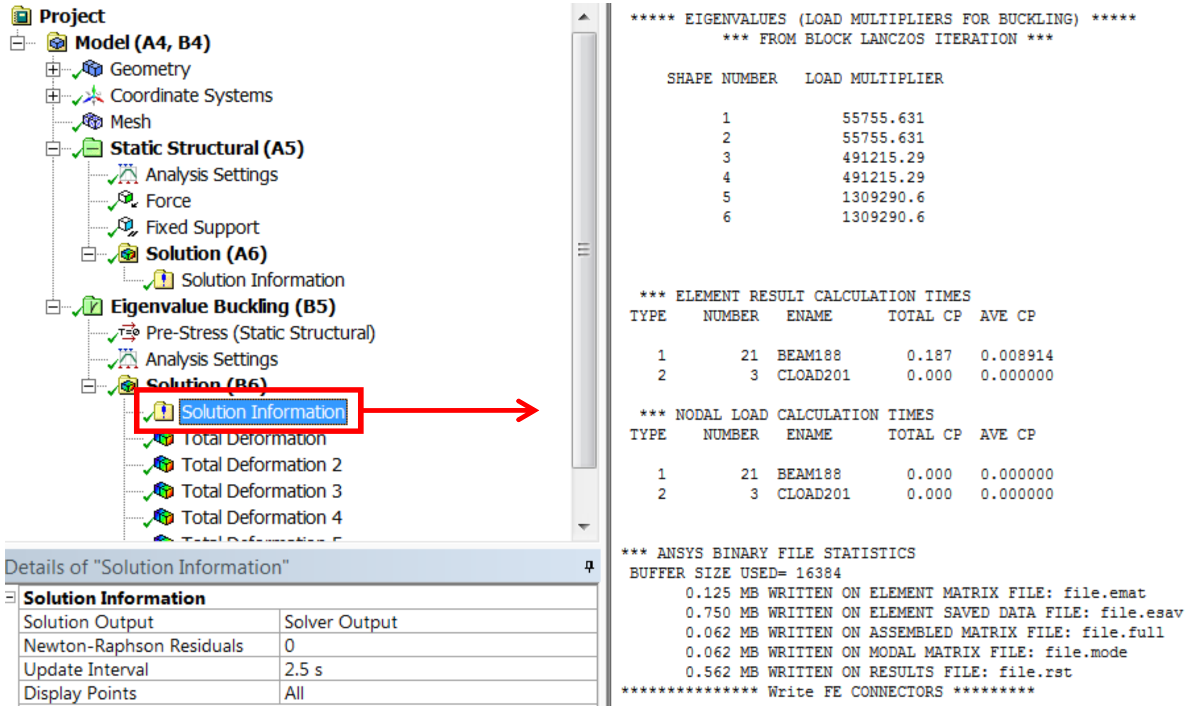
- The Analysis Settings Details view in the Eigenvalue Buckling environment allows specification of the number of buckling modes to find. Although a very large number of buckling modes / buckling load factors may be computed for most finite element models, it is typical that only the smallest load factor is of interest, so not many modes need be computed. The default value is 2.



08.05 Eigenvalue Buckling with Linear Pre-Stress: Solution

After model setup is complete, the Static Structural pre-stress analysis and the Eigenvalue Buckling analysis can be solved together.

- A eigenvalue buckling analysis is more computationally expensive than a static structural analysis of the same model.
- The "Solution Information" branch provides the solver output listing.

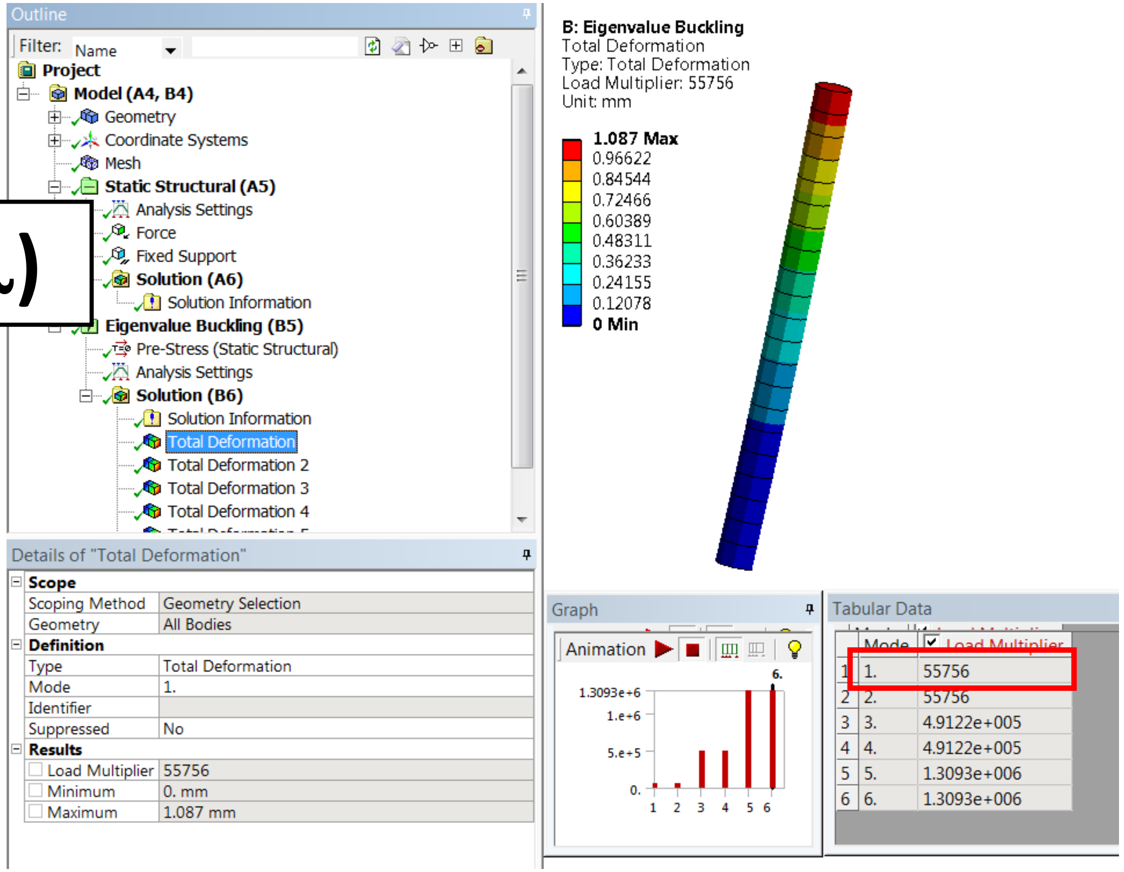


08.06 Eigenvalue Buckling with Linear Pre-Stress: Results

After the solution is complete, the buckling modes can be reviewed:

- The Load Multiplier for each buckling mode is shown in the Details view as well as the graph and chart areas. The load multiplier times the applied loads represent the predicted buckling condition.

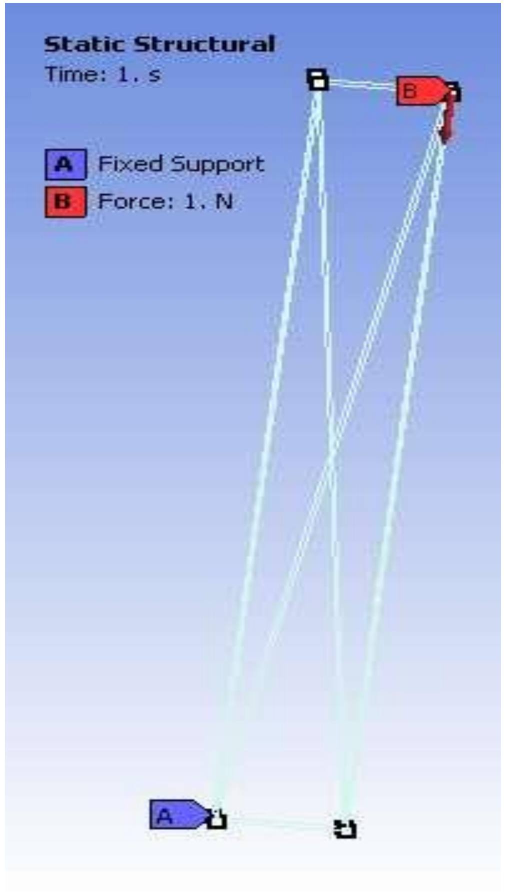
$$F_{buckle} = (F_{applied} \times \lambda)$$

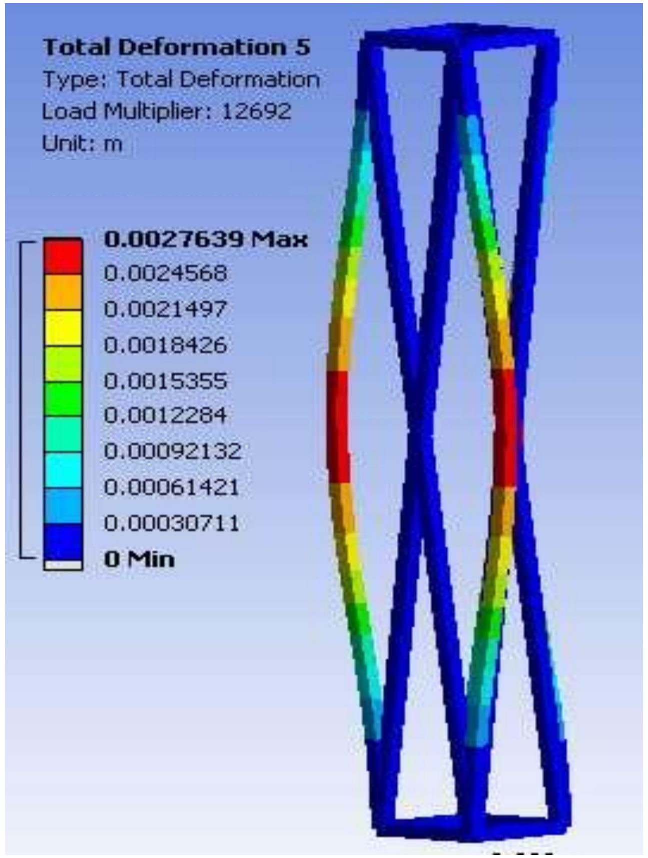


08.06 Eigenvalue Buckling with Linear Pre-Stress: Results

Interpreting the Load Multiplier (λ):

- The tower model below has been solved twice. In the first case, a unit load is applied. In the second case, a realistic design load is applied (continued on next slide).





08.06 Eigenvalue Buckling with Linear Pre-Stress: Results

Interpreting the Load Multiplier (λ):

Details of "1st Buckling Mode"	
Scope	
Geometry	All Bodies
Results	
Load Multiplier	29008

$$[\text{buckling load}] = \lambda * [\text{unit load}]$$

$$\Rightarrow [\text{buckling load}] = \lambda$$

Details of "1st Buckling Mode"	
Scope	
Geometry	All Bodies
Results	
Load Multiplier	1.0003

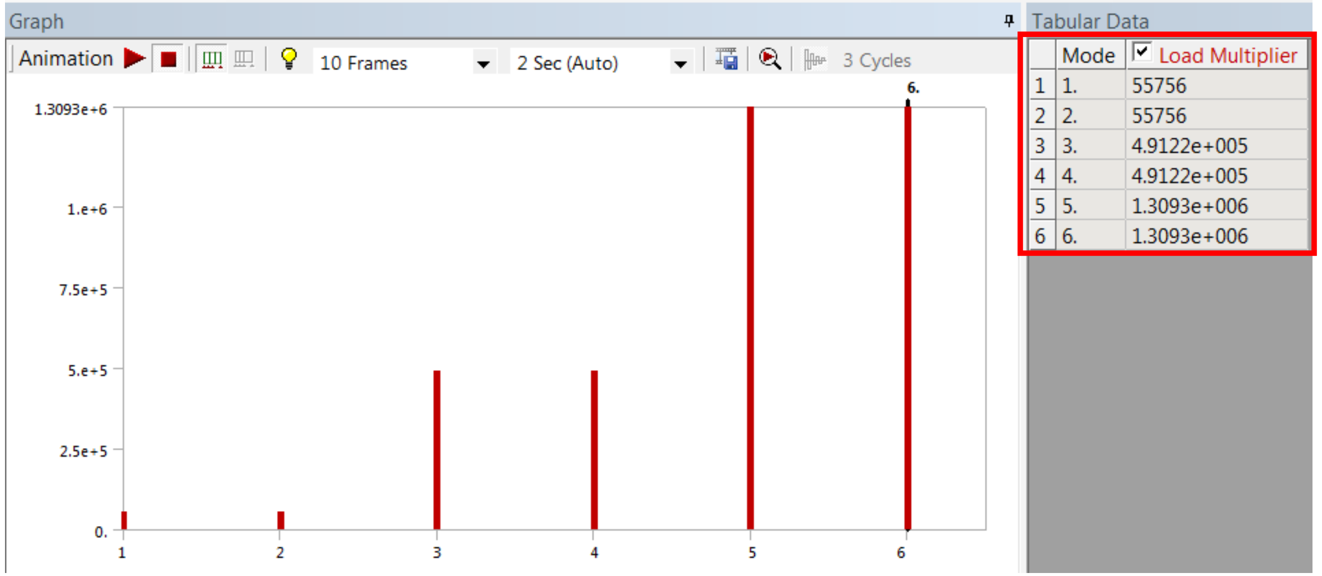
$$[\text{buckling load}] = \lambda * [\text{actual load}]$$

$$\Rightarrow [\text{buckling load}]/[\text{actual load}] = \lambda = [\text{safety factor}]$$

08.06 Eigenvalue Buckling with Linear Pre-Stress: Results

The buckling load multipliers can be reviewed in the "Timeline" section of the results under the "Linear Buckling" analysis branch.

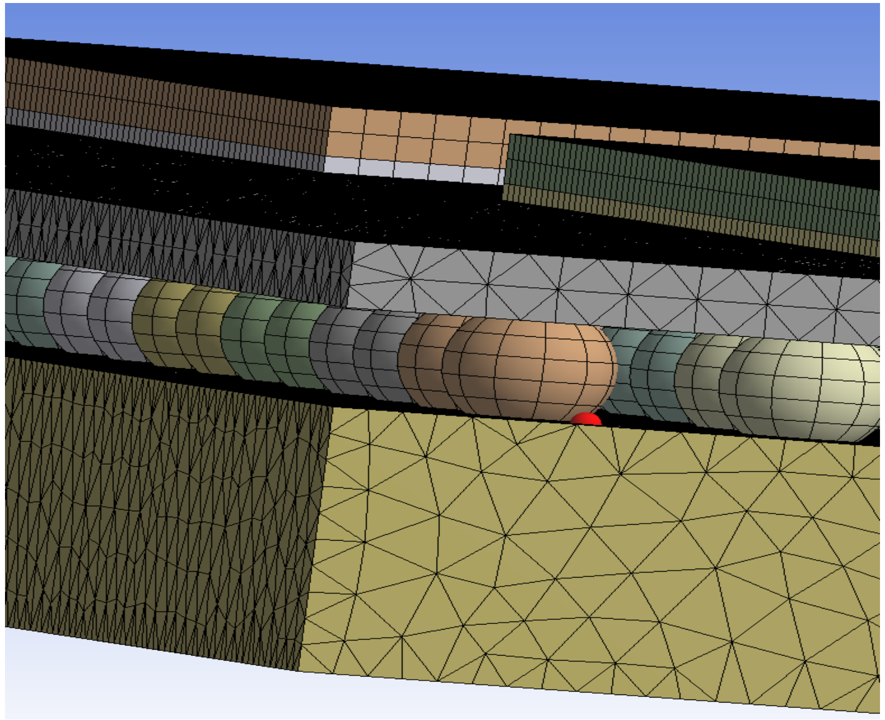
- **It is good practice to request more than one buckling mode to see if the structure may be able to buckle in more than one way under a given applied load.**
- **As is the case for modal analysis, the buckling mode shapes *do not represent actual displacements*, but are representative of the *shape* that a structure will take on when it buckles.**



	Mode	Load Multiplier
1	1.	55756
2	2.	55756
3	3.	4.9122e+005
4	4.	4.9122e+005
5	5.	1.3093e+006
6	6.	1.3093e+006

08.07 Submodeling Overview

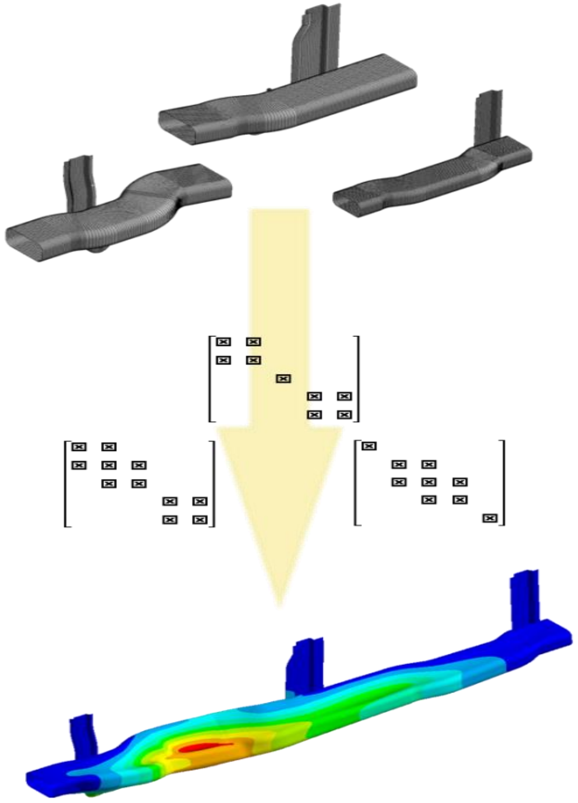
Current trends in simulation show an increased need for the computation of large models.



Statistics	
Bodies	1552
Active Bodies	1552
Nodes	3964827
Elements	1756665

08.07 Submodeling Overview

Various reduction or acceleration techniques may be used for faster and/or more accurate solutions.



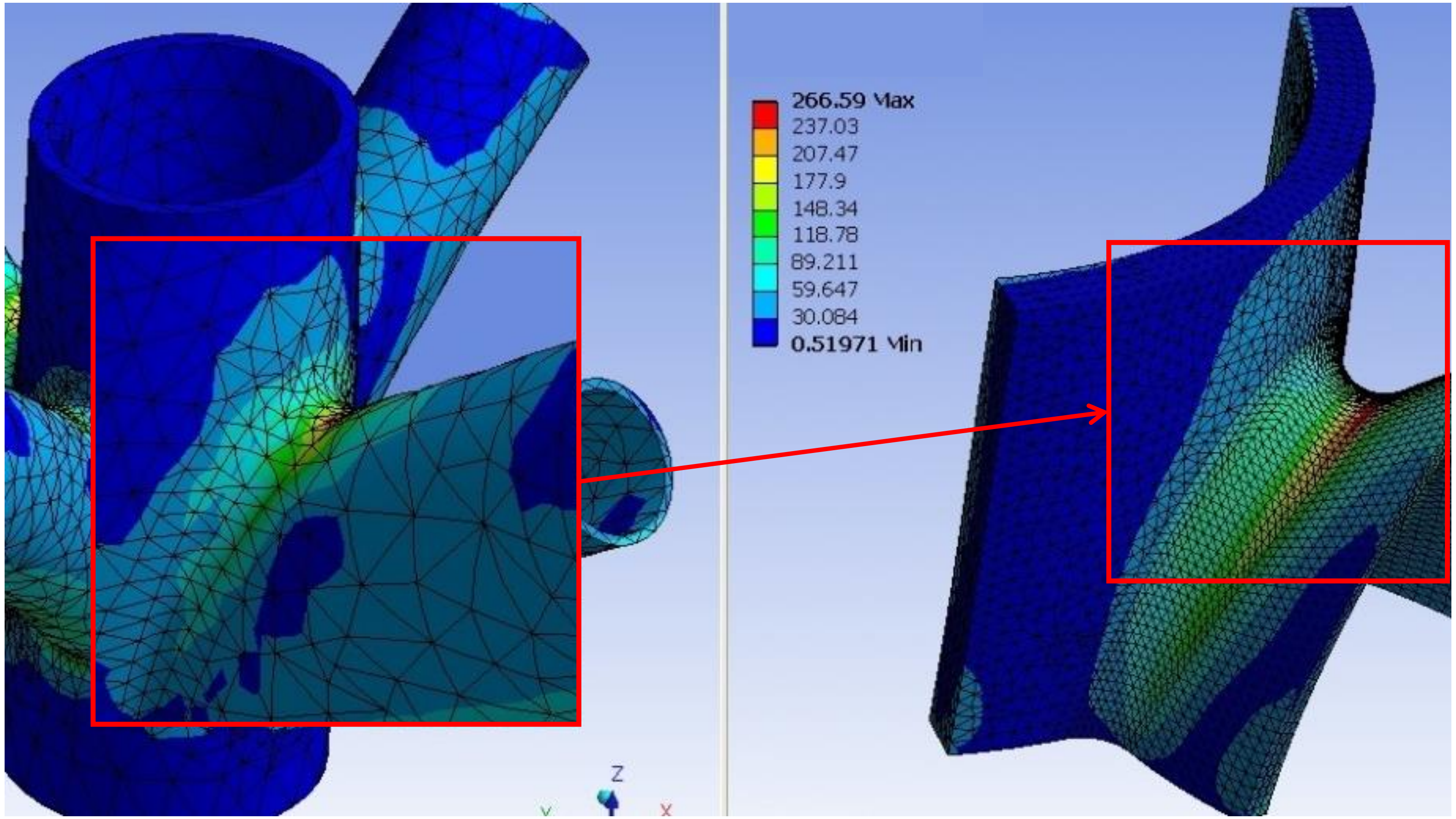
ANSYS

08.07 Submodeling Overview

Submodeling is a technique that may be applied in situations where high accuracy is required in a relatively small region of a larger model:

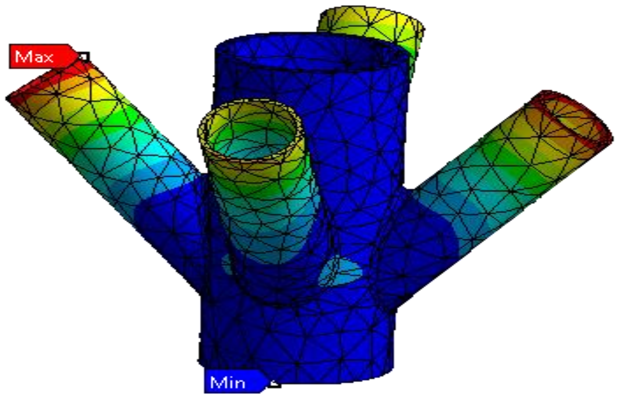
08.07 Submodeling Overview

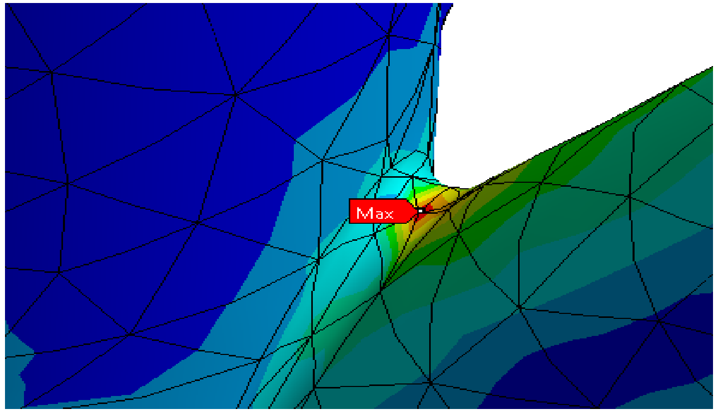
The procedure begins with a coarse-mesh solution of the entire geometry. This, then, is followed by a fine-mesh solution of the smaller region of interest, known as a *submodel*. The smaller region may also include geometric detail that was not included in the coarse region.



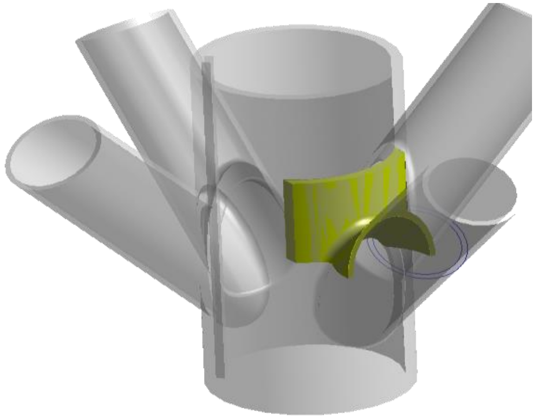
08.07 Submodeling Overview

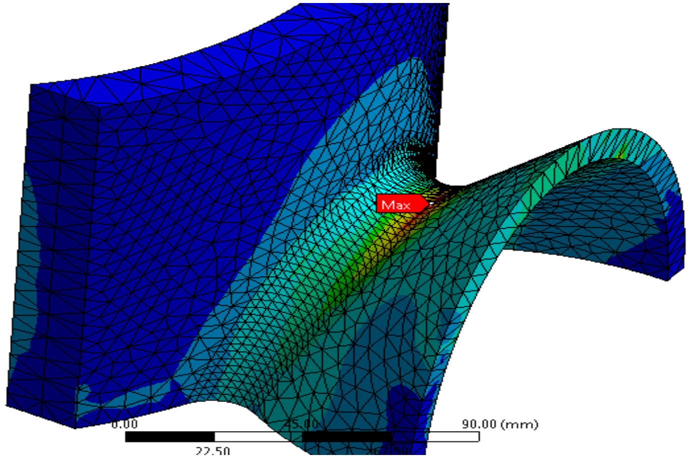
The coarse model provides accurate deformations but inaccurate stresses in some areas.





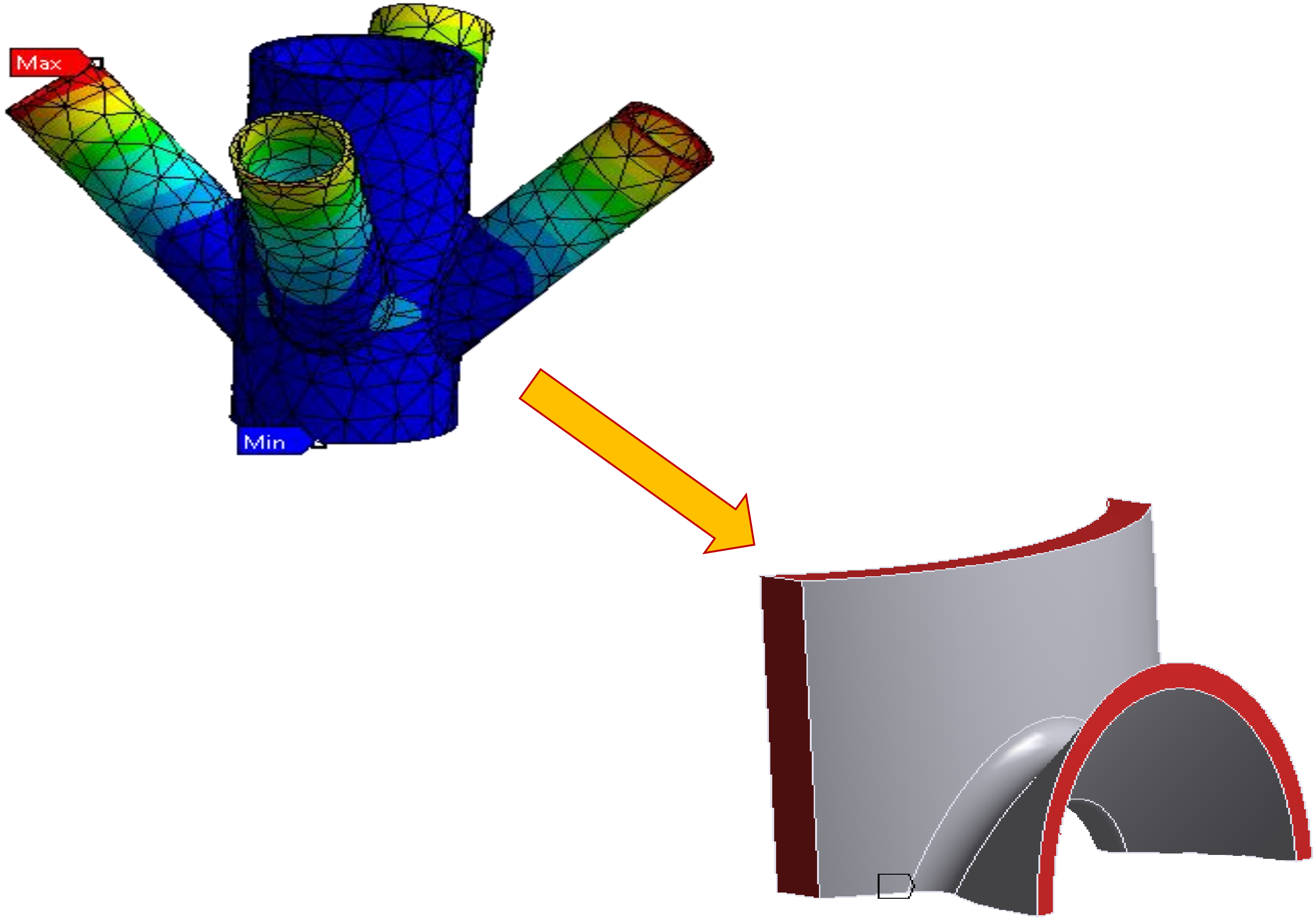
The submodel can provide more accurate stresses in a specific region of interest.





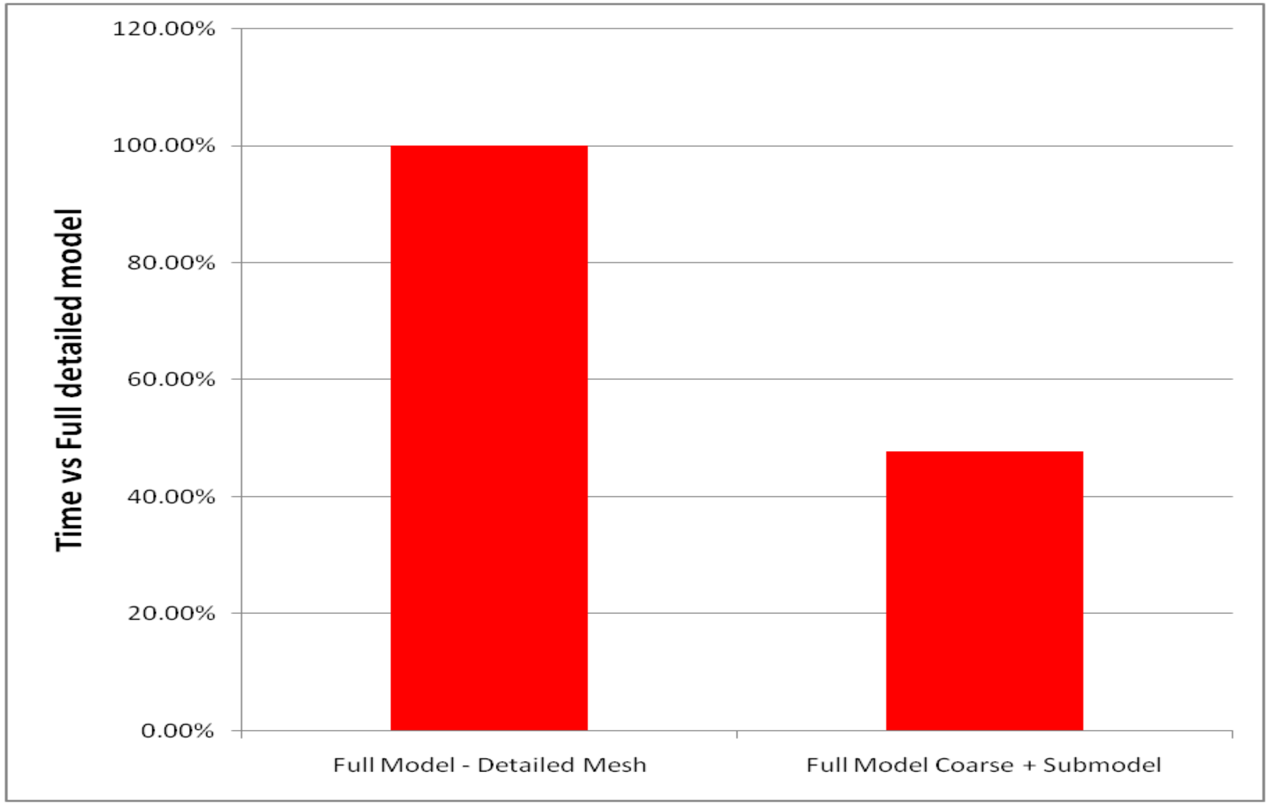
08.07 Submodeling Overview

Information is passed from the coarse model to the submodel (fine model) through results mapping. Displacements from the coarse model are mapped to specified cut boundaries on the submodel.

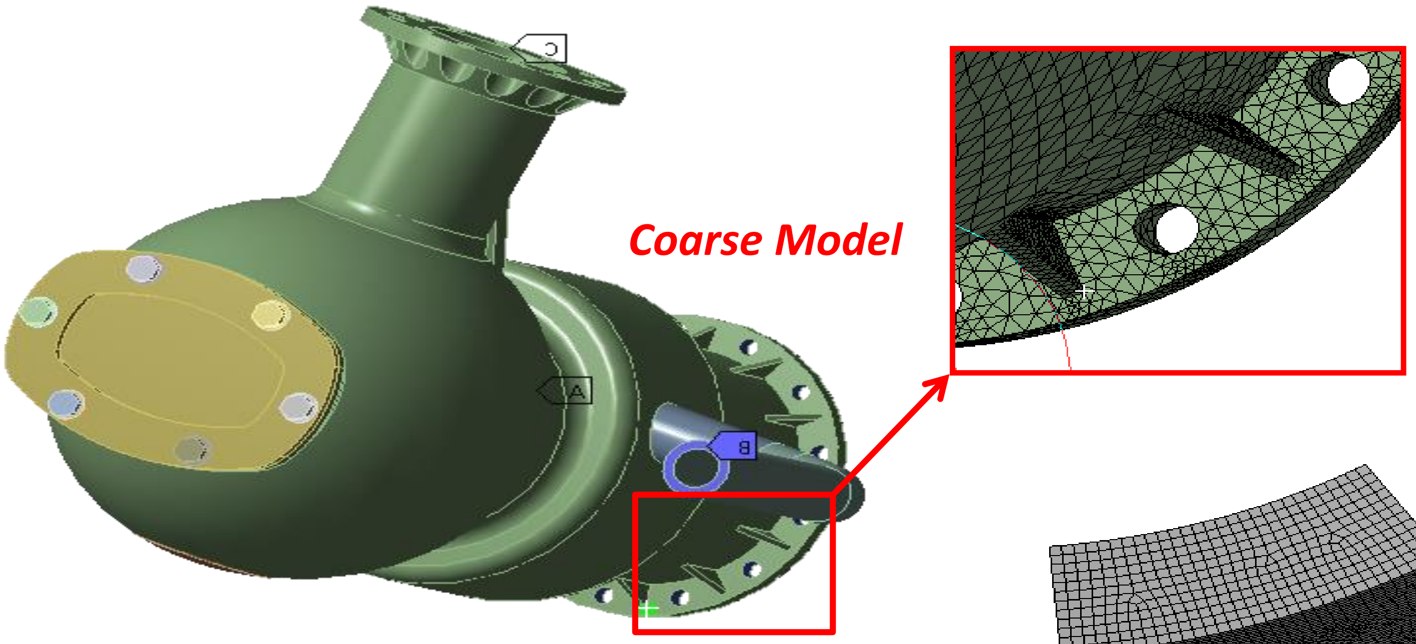


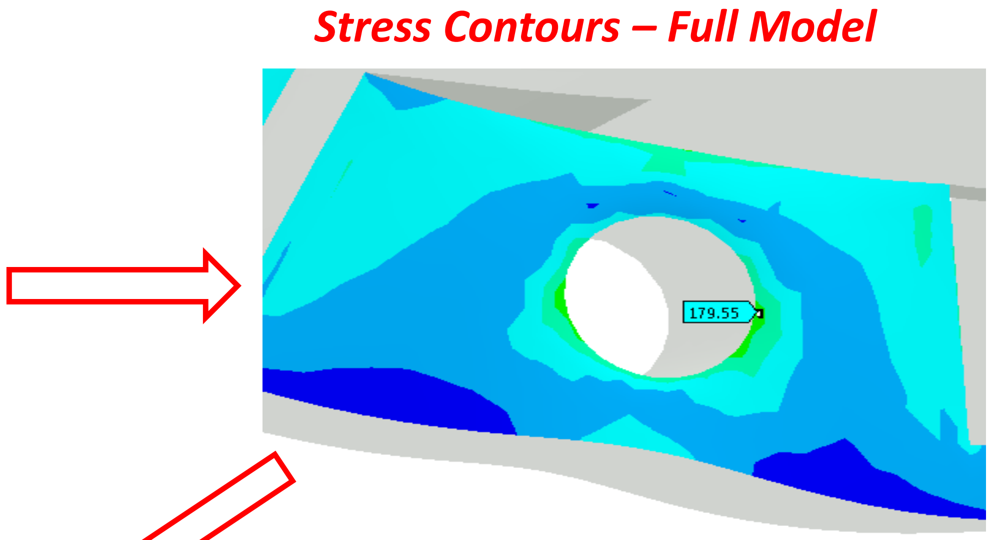
08.07 Submodeling Overview

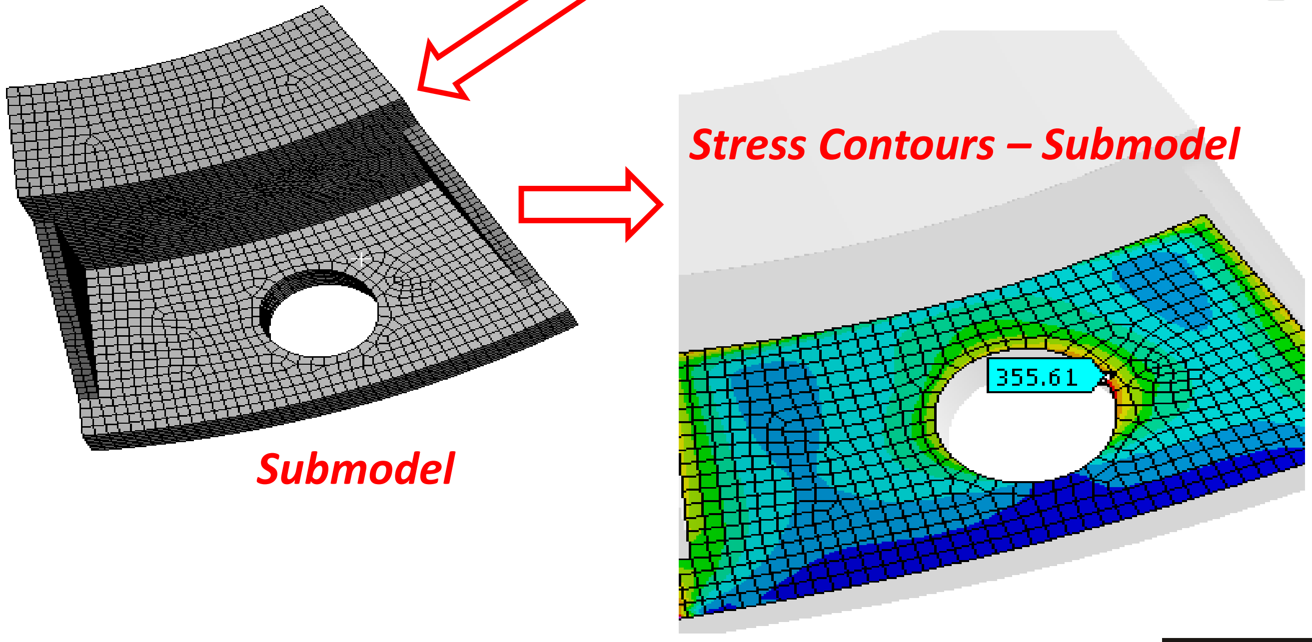
In many cases, solving the two models can be faster than solving the larger model with comparable mesh refinement.



08.07 Submodeling Overview



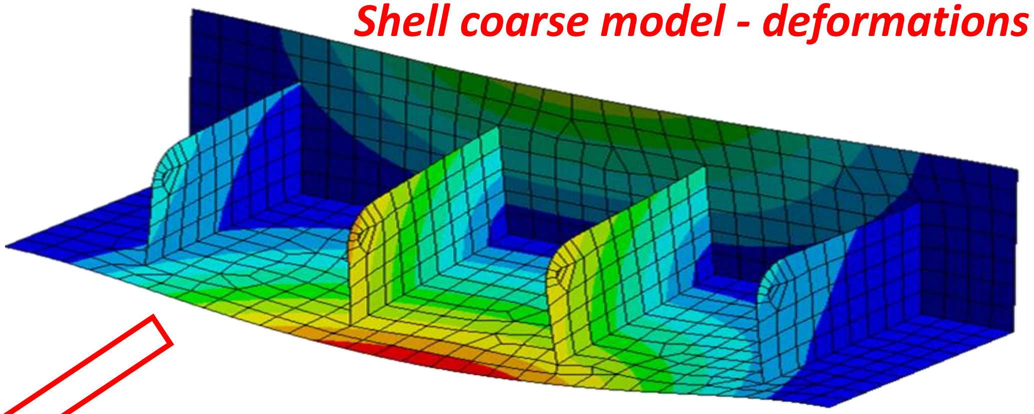


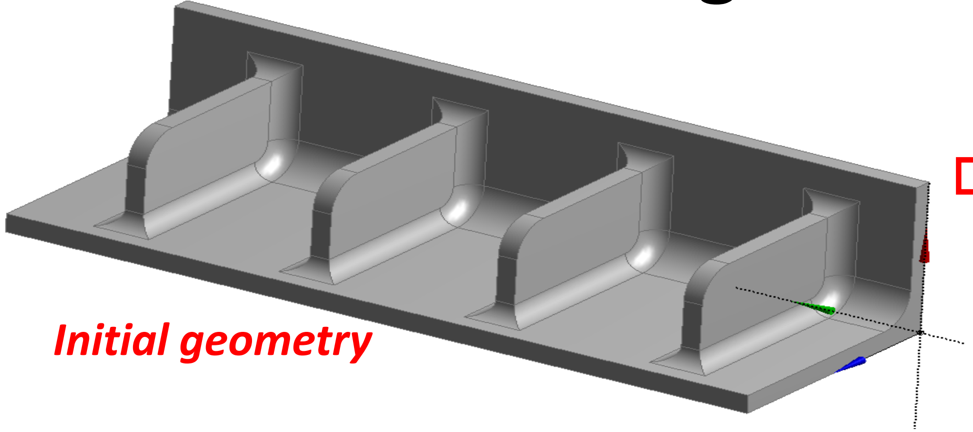


Example: solid coarse model to a solid fine model ("solid-to-solid submodel")

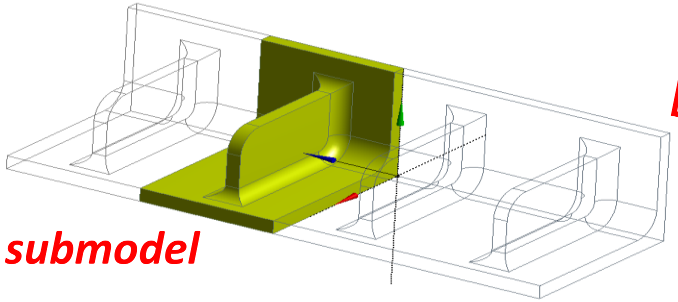
08.07 Submodeling Overview

Shell coarse model - deformations

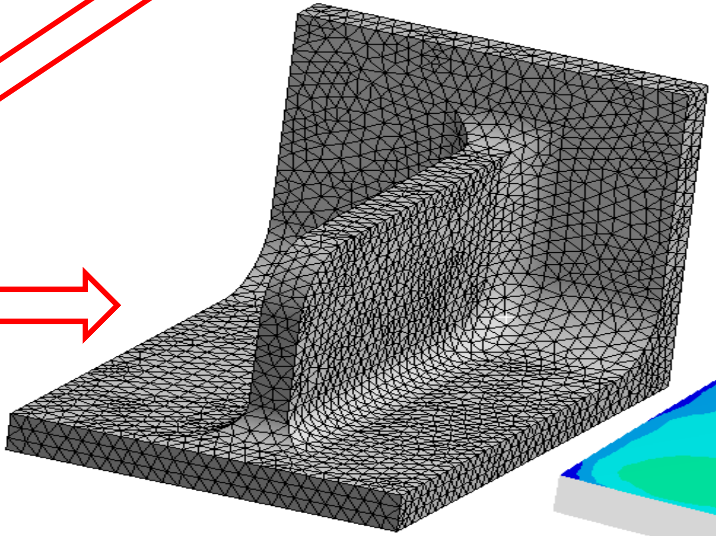


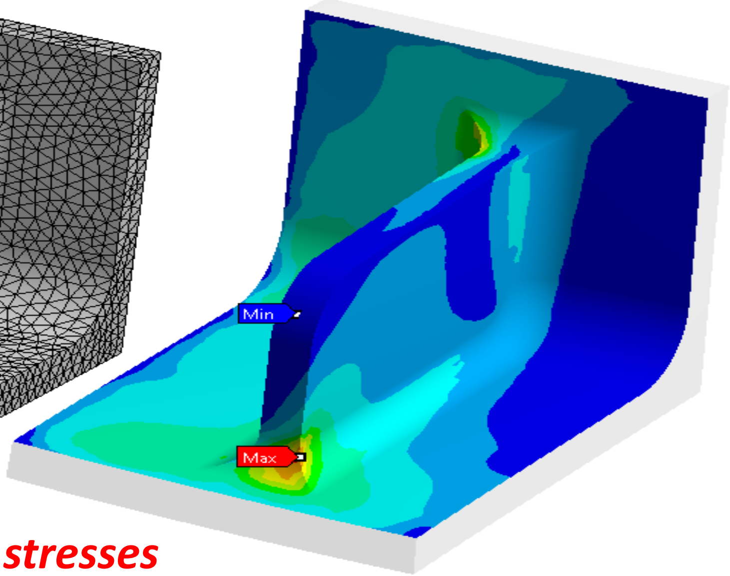


Initial geometry



Solid submodel





Solid submodel - stresses

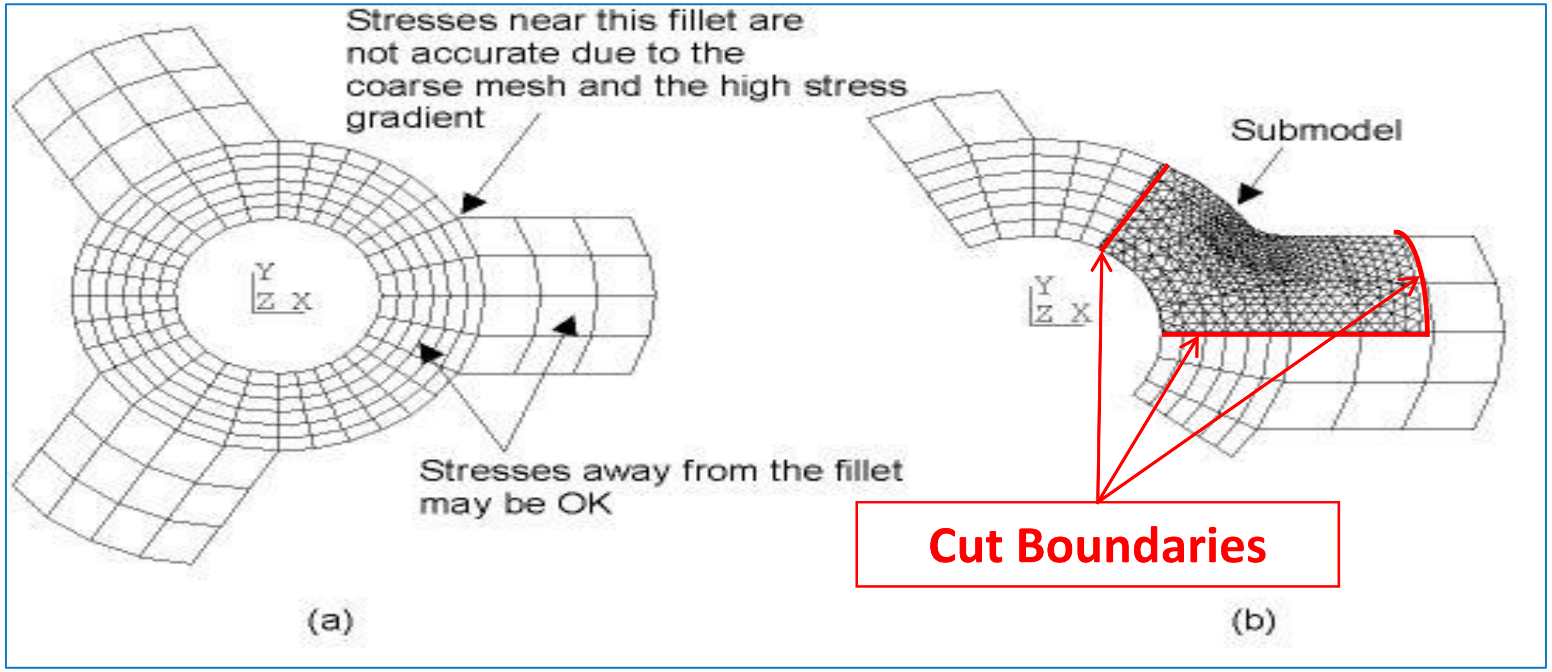
Example: shell coarse model to a solid fine model ("shell-to-solid submodel")

Beam-to-solid and beam-to-shell submodels are also available.

08.08 Submodeling Procedure

Submodeling is a technique where a coarsely meshed model can be solved followed by a subsequent solution using only a portion of the coarse model with a more refined mesh. Submodeling is available for structural and thermal analysis types.

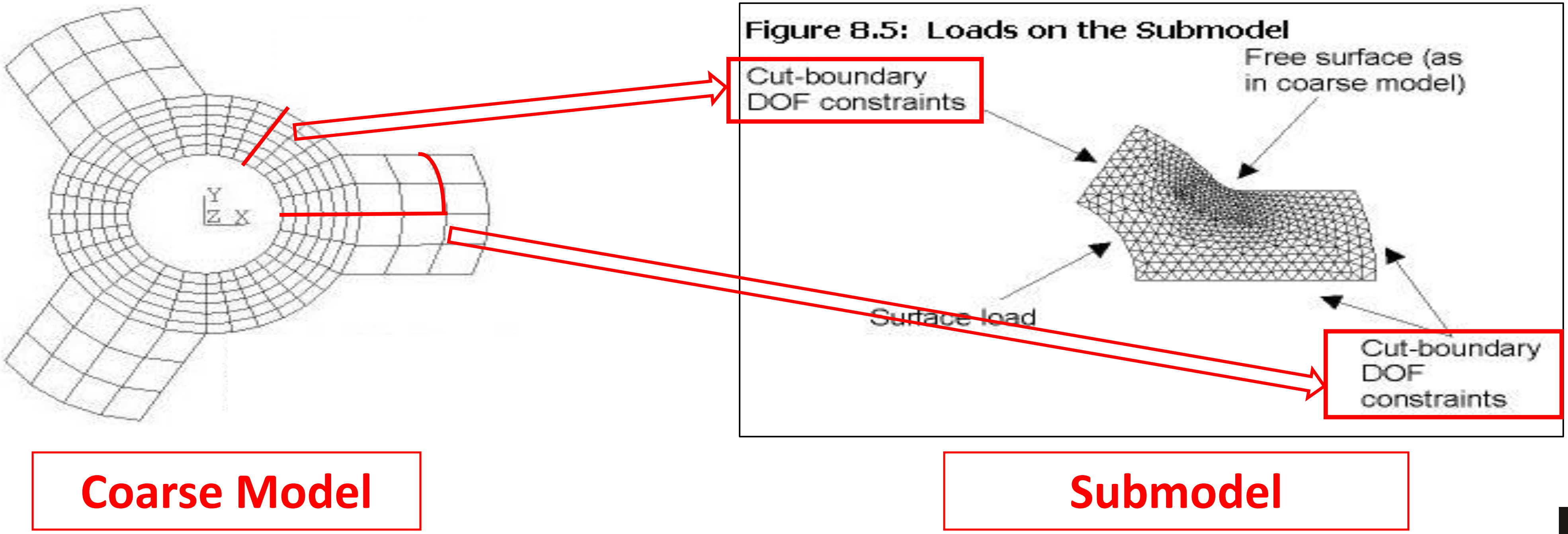
As shown in the example below, and explained shortly, one of the key concepts in submodeling is the designation of the “cut boundaries” defining the submodel.



08.08 Submodeling Procedure

As the figures below show, the displacements from the coarse model are mapped to the cut boundary locations on the submodel in the form of DOF constraints (i.e., applied displacements).

- Note: If the cut boundaries are too close to the stress concentrations of interest, the accuracy of the submodel can be degraded. A results comparison can be used to verify the cut boundary location (this will be detailed in an upcoming example).

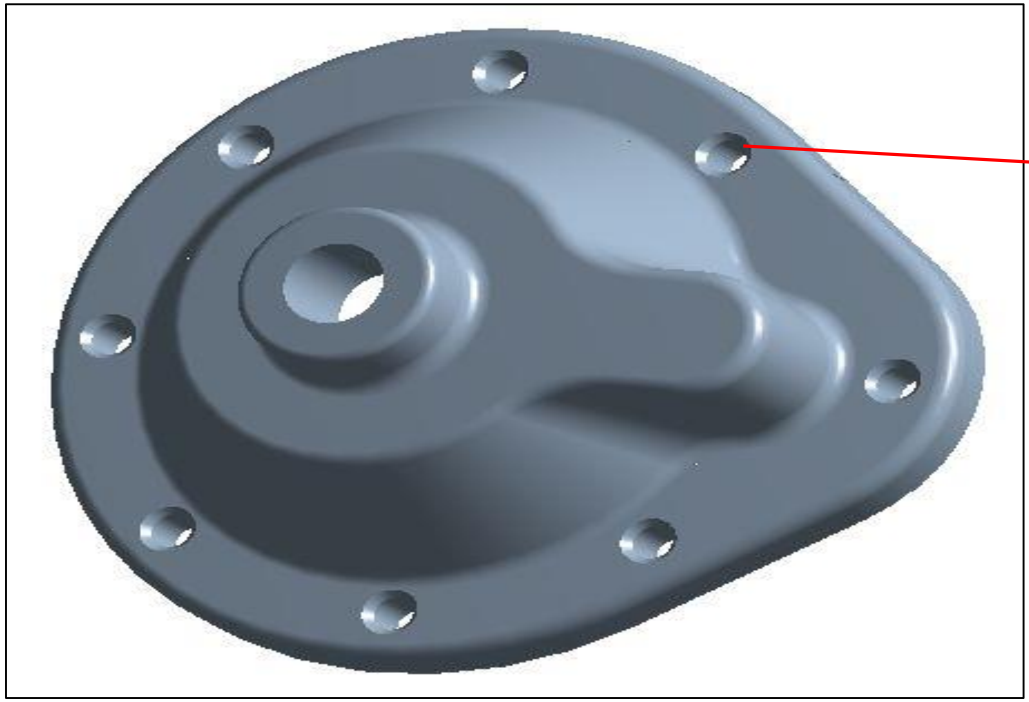


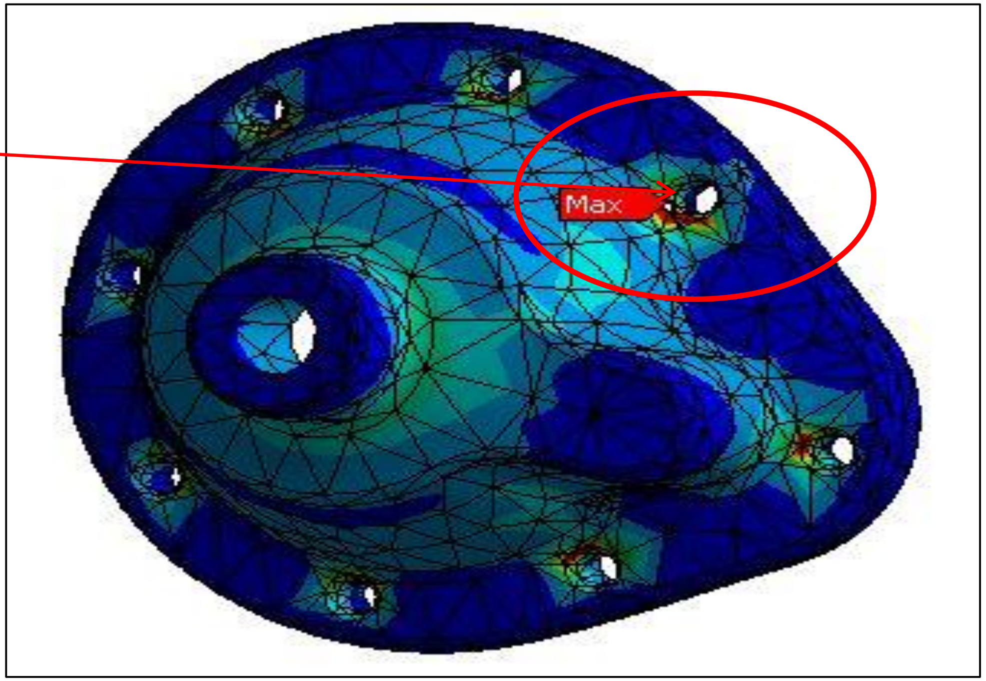
  April 4, 2017

08.08 Submodeling Procedure

Submodeling Example:

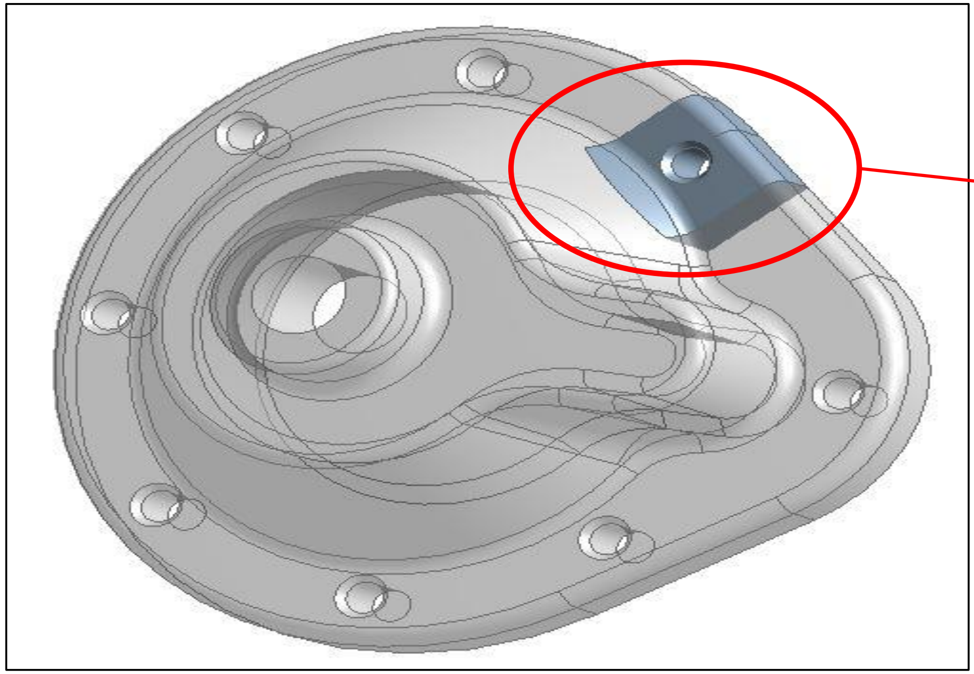
- The model shown below is initially solved using a coarse mesh.
- As expected, we see stress concentrations in regions containing detailed geometry along with a coarse mesh.
- Based on these results we choose to create a submodel to explore the region circled in more detail.

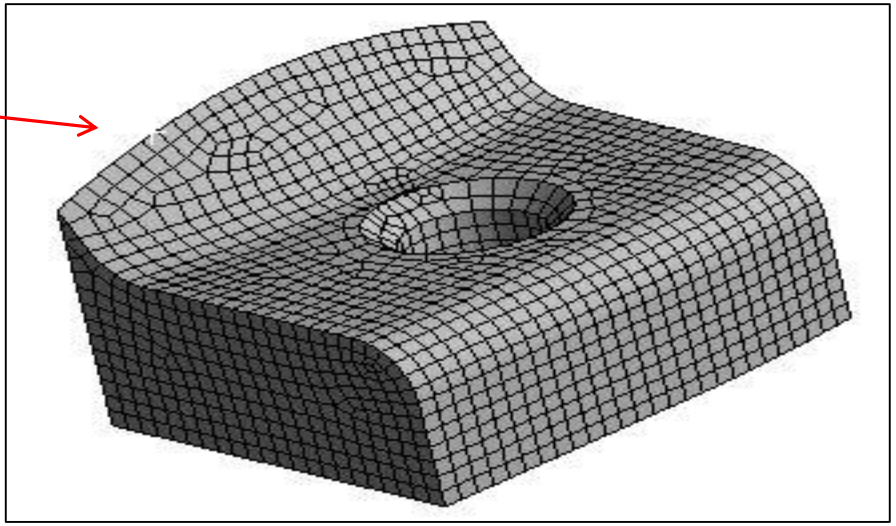




08.08 Submodeling Procedure

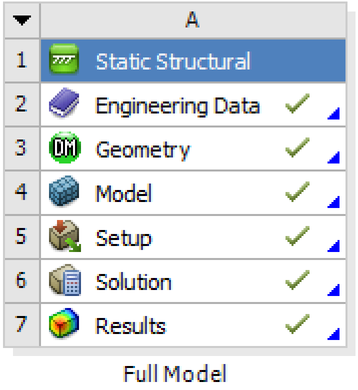
Although any available geometry modeling technique can be used to create the submodel, we have chosen to slice a body from the full model using ANSYS SpaceClaim. This new body becomes our submodel, and a more refined mesh is created on the submodel in the Mechanical application.



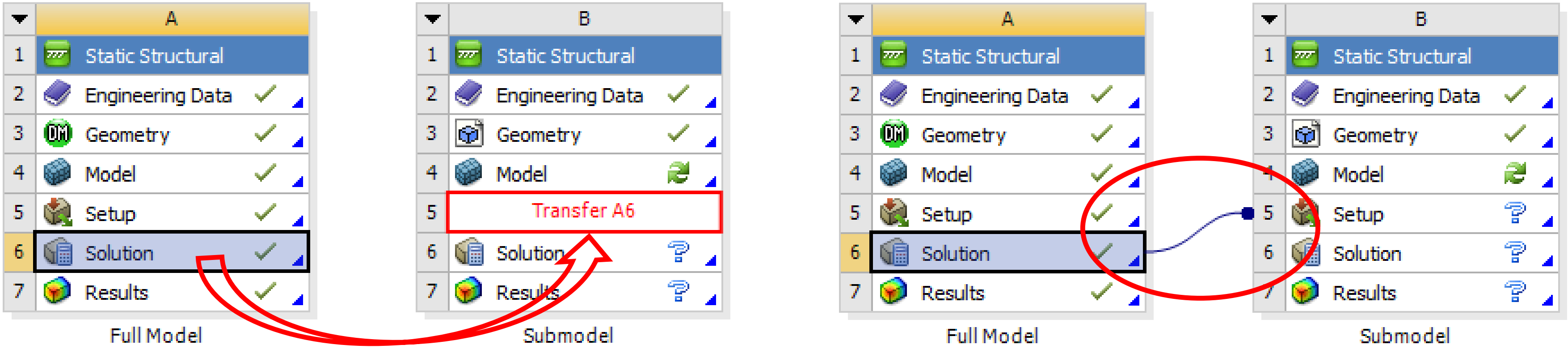


08.08 Submodeling Procedure

The submodel schematic is set up as shown here:



Original Full Model



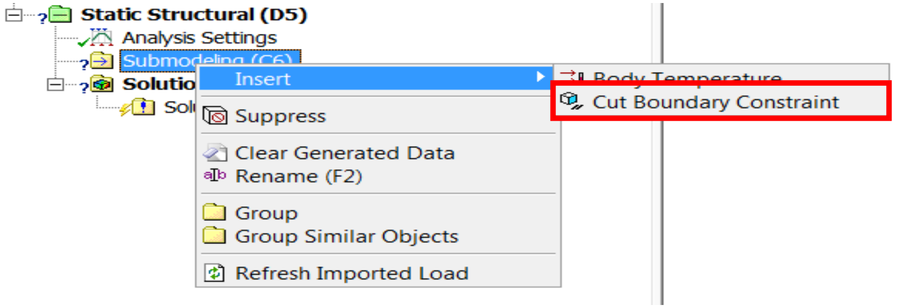
Since the full model and submodel are comprised of different geometries, we typically don't drag and drop a new structural system onto the existing one, as this would link the geometries. Instead, we create a new stand-alone system, drag the full model solution cell onto the submodel's setup cell, and then create or import the desired submodel geometry.

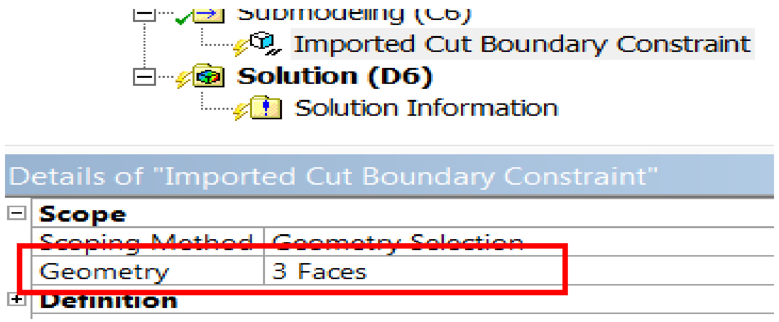
08.08 Submodeling Procedure

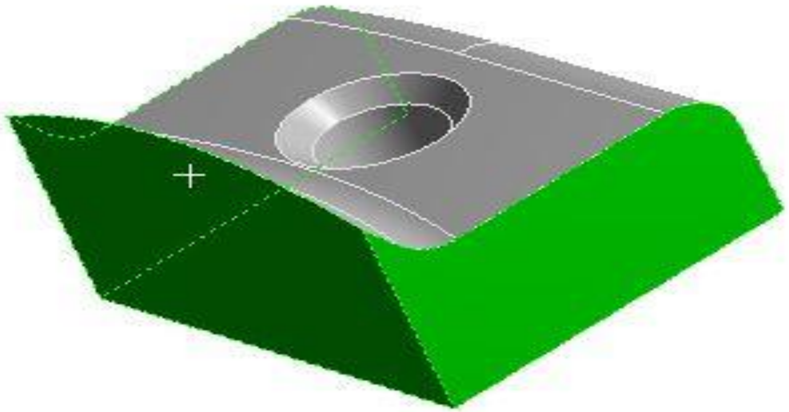
After opening the new (submodel) analysis system in Mechanical, we see a new “Submodeling” branch has automatically been inserted into the tree. If we RMB on this branch we can choose the type of result to import (displacement, in this example).

In the details of the imported load, we scope the cut boundaries of the submodel.

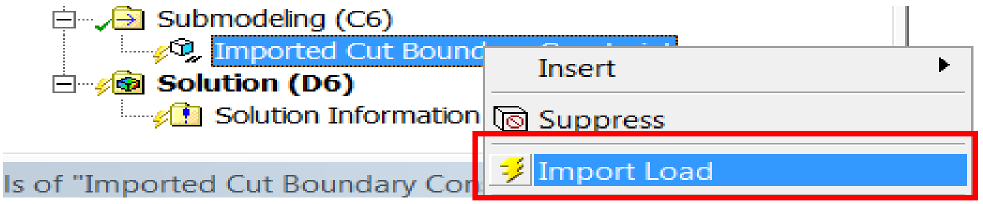
Note: There are numerous mapping options available when transferring loads, not all of which apply to submodeling. For a complete discussion, see the following reference in *ANSYS Help*: ANSYS Documentation > Workbench > User’s Guide > ANSYS Workbench Systems > Component Systems > External Data.



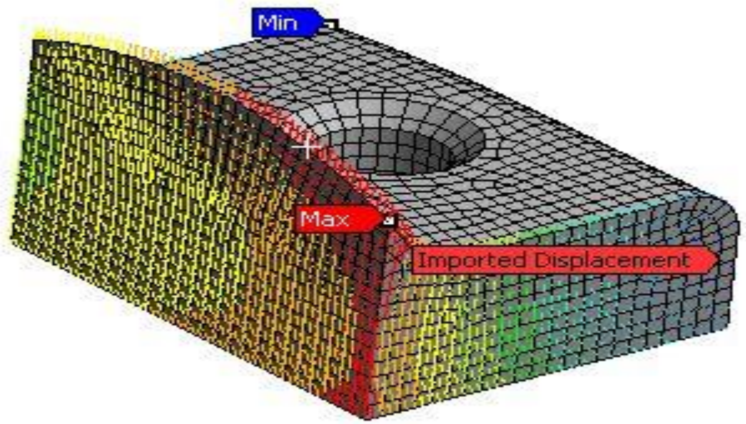




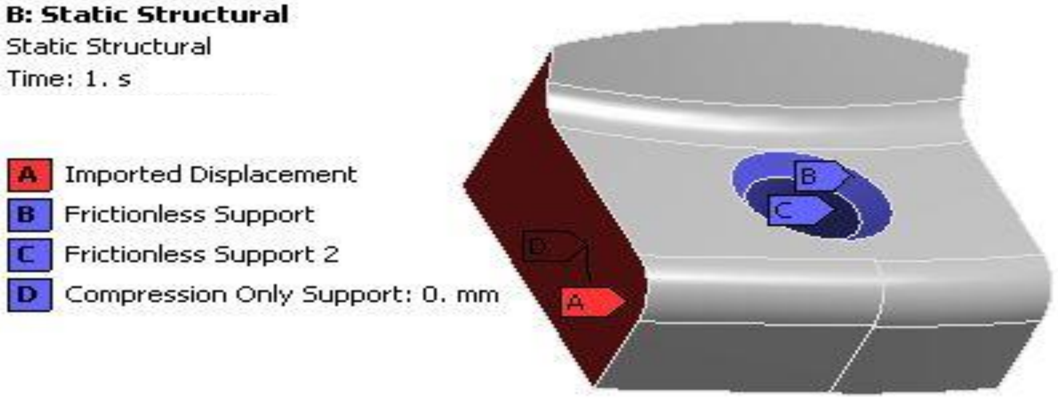
08.08 Submodeling Procedure



RMB to import the DOF constraints from the full model. When completed, the import can be reviewed graphically.



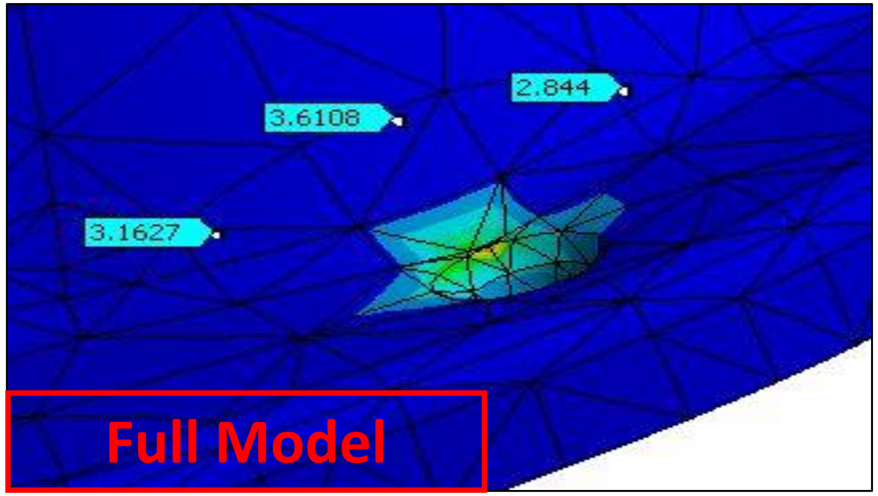
***Add any additional loads and/or boundary conditions to the submodel that are necessary to match those applied to the full model* and solve.**

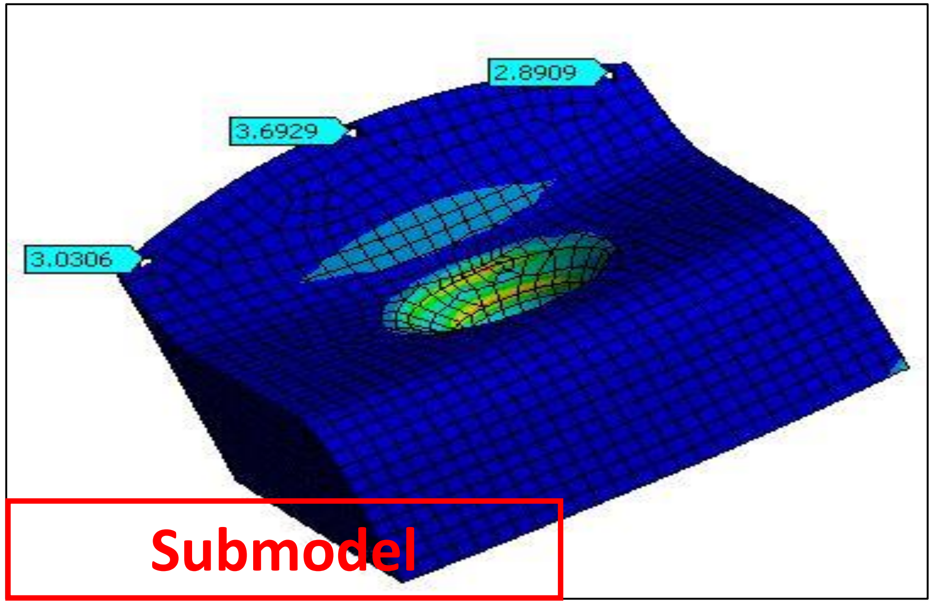


08.08 Submodeling Procedure

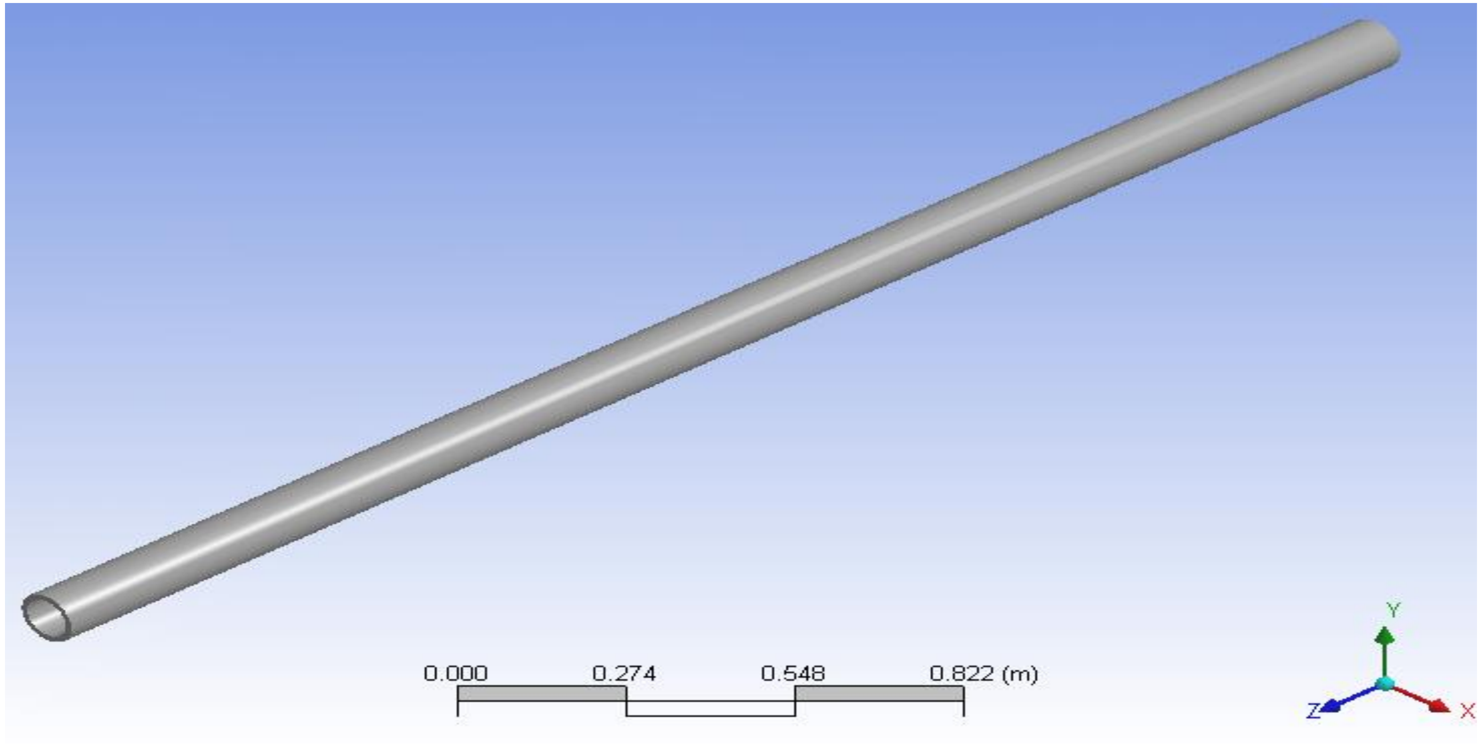
To insure that the cut boundary is far enough from the high stress region a check should be performed to compare full and submodel results near the cut boundary.

Here, an array of probes is used but path plots, surface plots, etc., are options as well. We simply want to verify that the results near the cut boundary are not significantly different between the full and sub models. If they are, it is usually an indication that the boundary needs to be moved further from the region(s) of high stress.

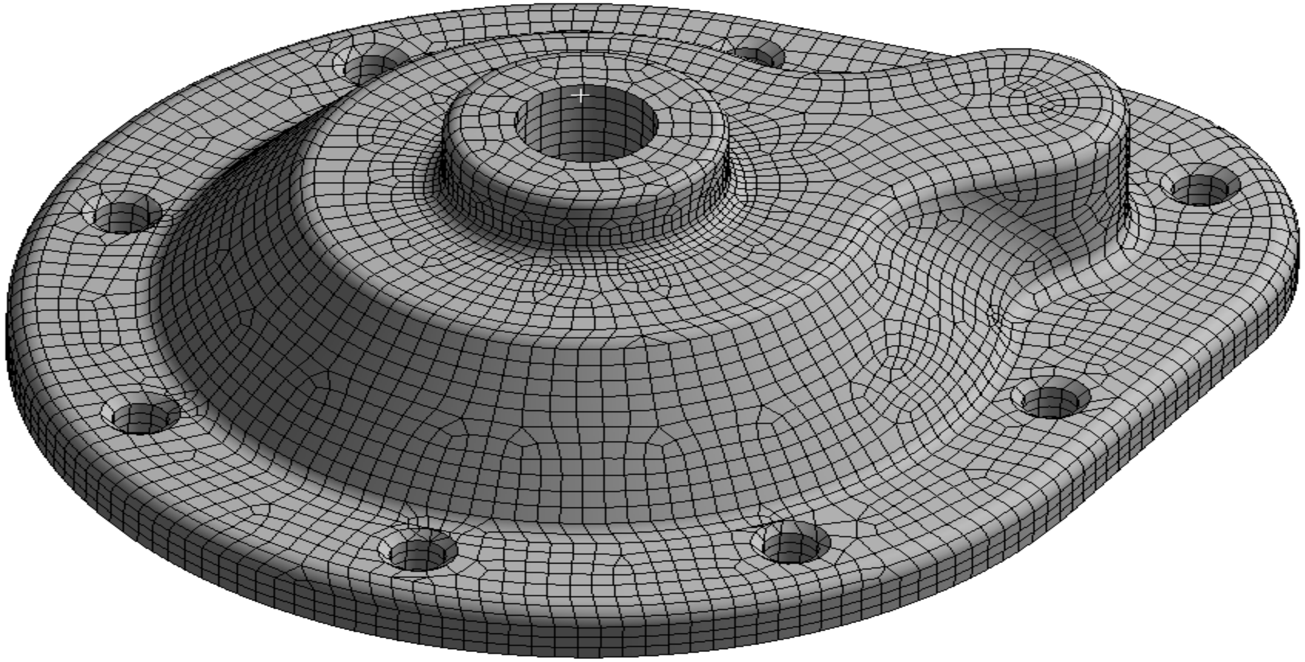




08.09 Workshop 08.1: Eigenvalue Buckling with Linear Pre-Stress



08.10 Workshop 08.2: Submodeling



Module 08: Eigenvalue Buckling and Submodeling

END