EN1740 Computer Aided Visualization and Design

Spring 2012

4/26/2012

Brian C. P. Burke

Last time:

- More motion analysis with Pro/E

Tonight:

- Introduction to external analysis products
  - ABAQUS

External Analysis

Advanced analysis typically is done in problem-specific packages

Moldflow®

SIMULIA

ABAQUS

ANSYS®

MSC Software®
Simulating Reality, Delivering Certainty™

Adams
Multibody Dynamics

MSC Nastran
Accurate, Efficient & Affordable Finite Element Analysis

LS-DYNA®

Structural Analysis

Utility and Applications

- **Structural Analysis is used to predict the response of a solid to an applied load**
- **Response is typically linked to some performance criteria of component**
    - **Yield**
    - **Fatigue**
    - **Rupture**
    - **Frequency response**
- **Predictions need to capture correct physics**
- ***Correct predictions (or “virtual prototyping”) can have a huge impact on product development***

Structural Analysis

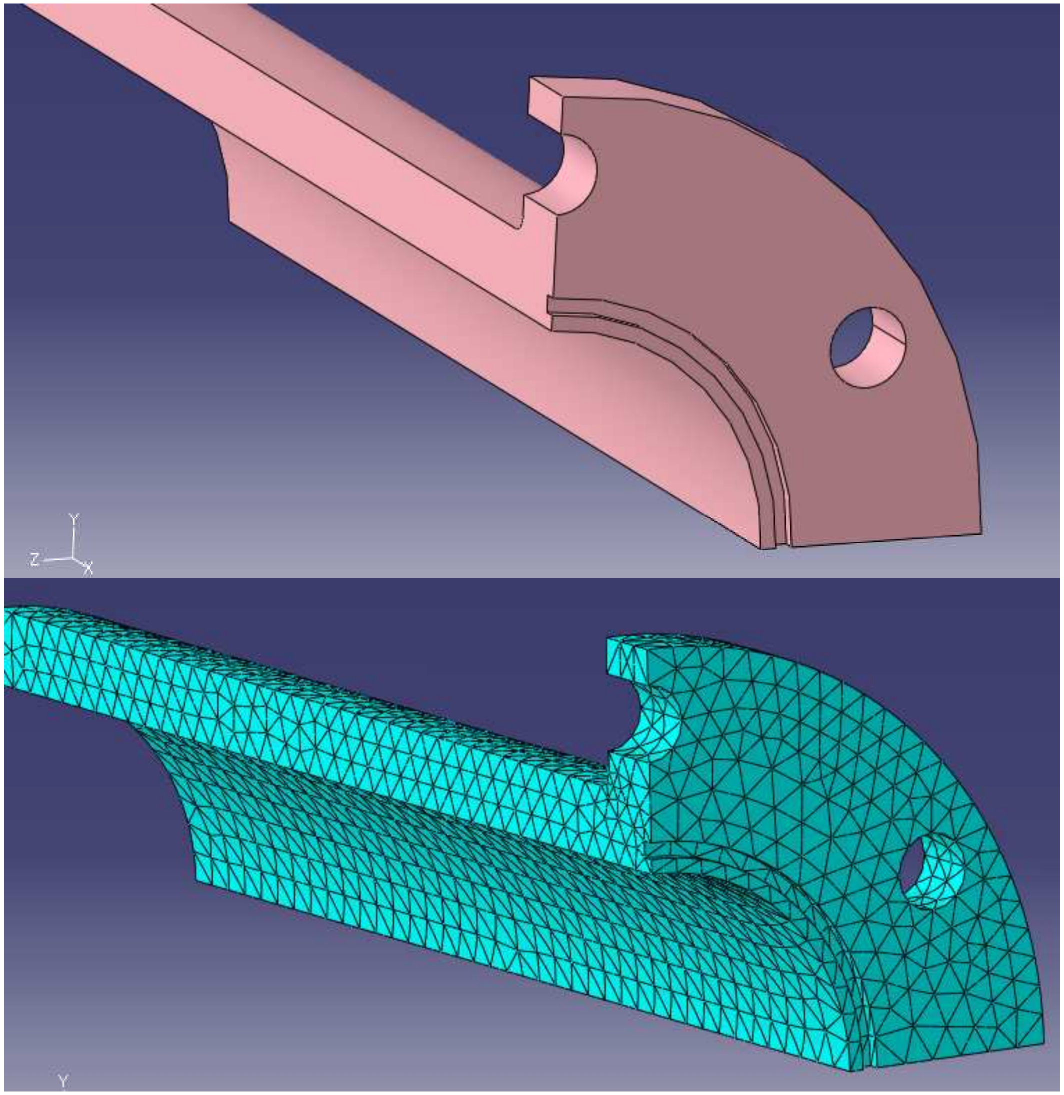
Key Concepts

- **Solids and structures are analyzed in terms of stresses and strains**
  - **Stress – measure of the internal forces acting in a solid**
  - **Strains – describes deformation of a solid**
- **Stresses and strains provide a mathematical language to quantify material response**

Structural Analysis

Finite Element Method

- **FEM uses a divide and conquer approach**
    - **Approximate the geometry using a collection of nodes and elements**
    - **Satisfy equilibrium on each element**
- **Displacements are calculated directly**
- **Stresses and strains are calculated from displacements**



Structural Analysis

Finite Element Method – Process layout

- **Establish the geometry (CAD)**
- **Pre-processing**
  - **Material definition**
  - **Discretization (meshing)**
  - **Establish boundary conditions (loads and displacements)**
  - **Solver settings**
- **Solution**
- **Post-processing**
  - **Evaluating the results**
  - **Interpreting the results – *MOST IMPORTANT PART!!***

Structural Analysis

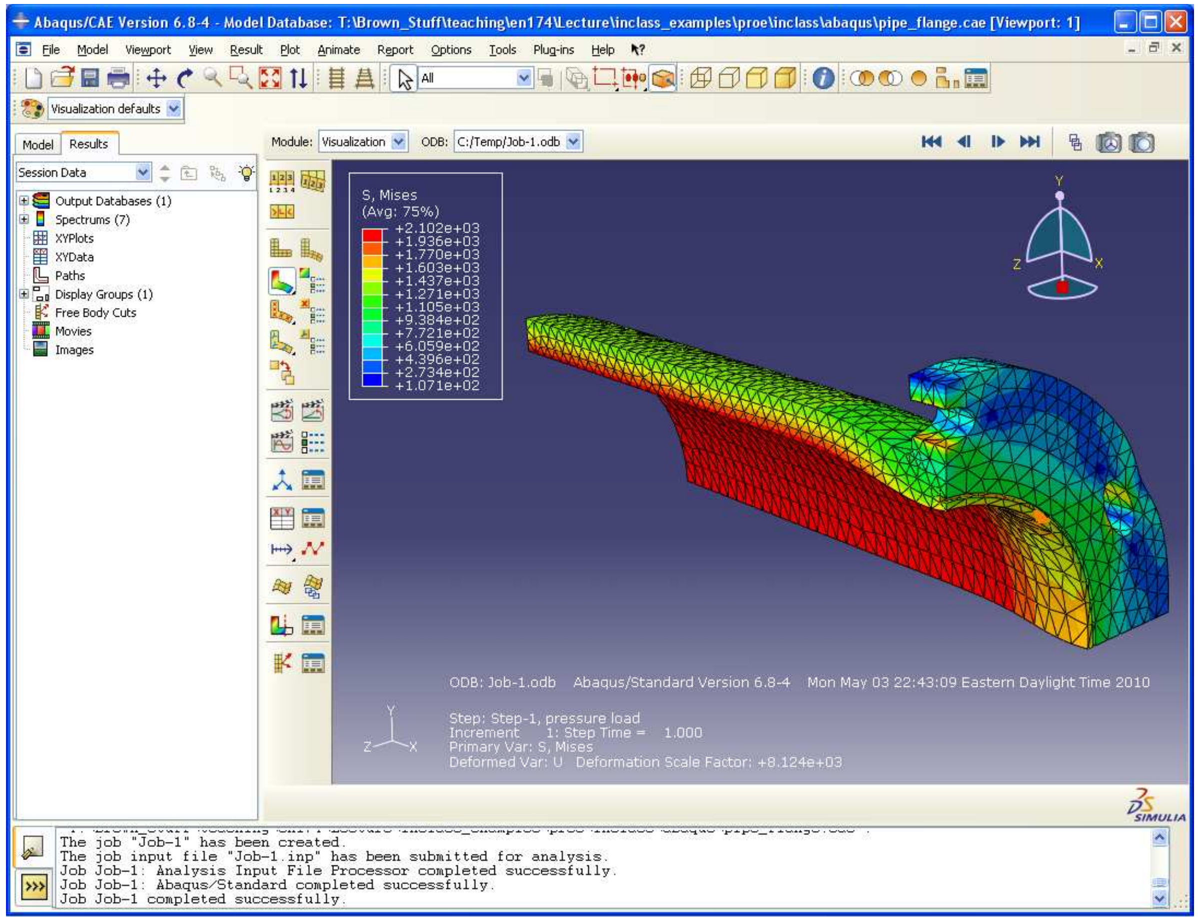
Finite Element Method – Packages

- **Many CAD systems come with internal analysis software**
  - **Typically very limited**
  - **True mathematics and physics are often "package" to broaden number of potential users**
- **There are very strong third party packages available that specialize in this type of analysis**
  - **ABAQUS**
  - **ANSYS**
  - **ALGOR**
  - **….many others**
  - **These types of codes provide the largest flexibility and versatility**

EXERCISE - Structural Analysis

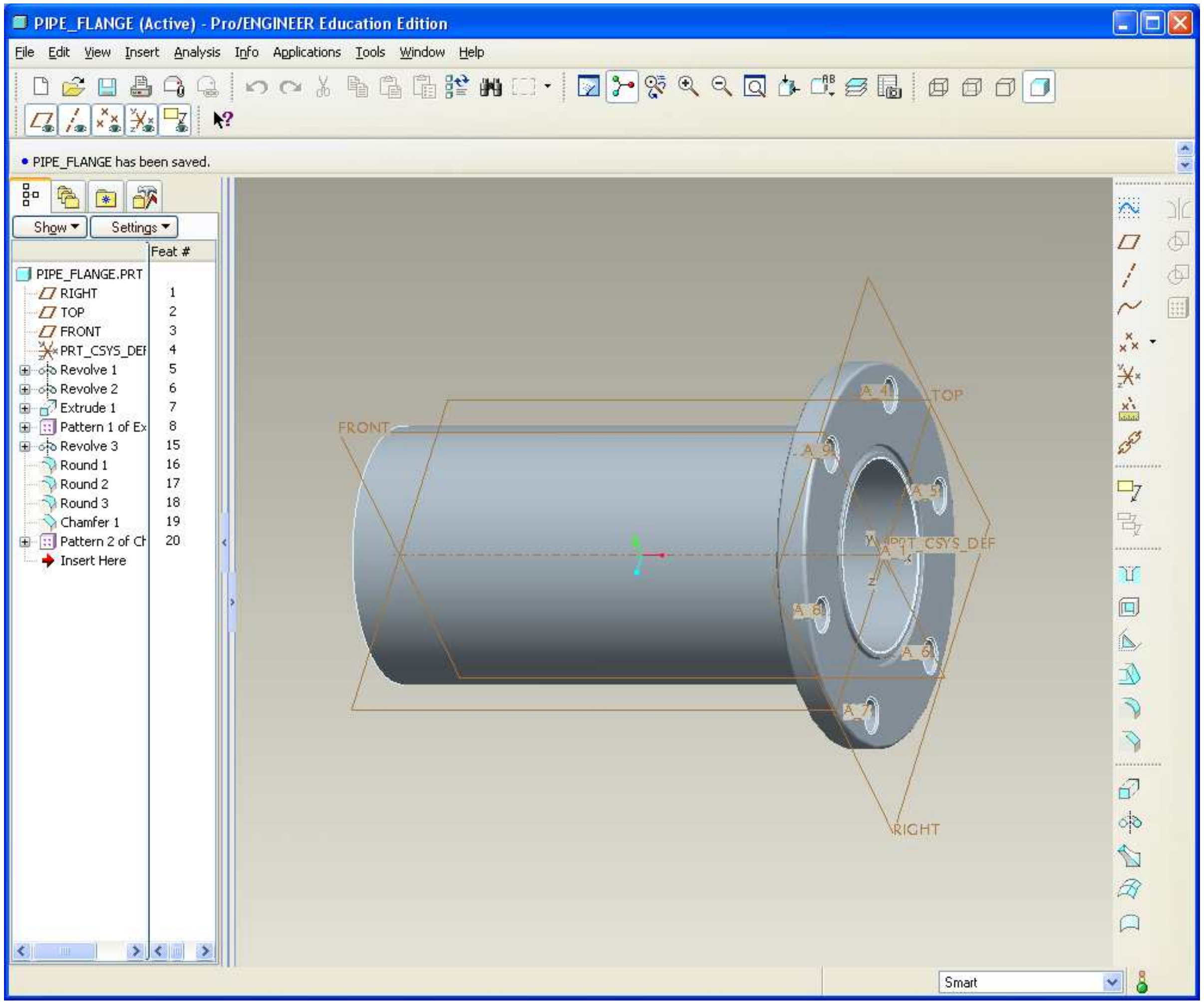
Employing Finite Element Method

- **Using FEM assess whether the pipe section shown can support an internal pressure of 500psi**
- **Assume:**
  - **Symmetry can be employed**
  - **Component is 1018 steel**



EXERCISE – Structural Analysis

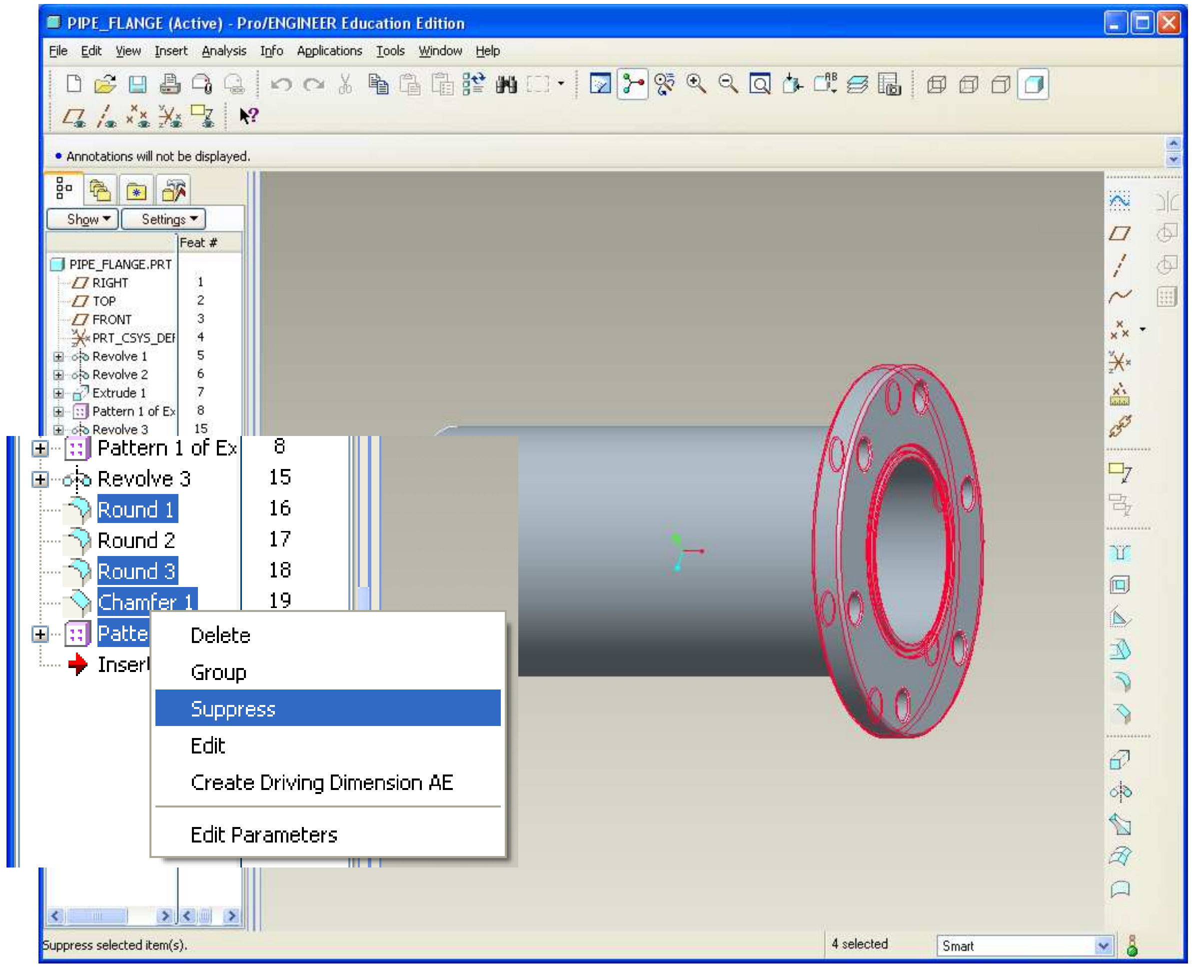
- **Download pipe_flange.prt from the Supporting Materials page of the web site**
- **Open the component in Pro/Engineer**



EXERCISE – Structural Analysis

De-feature CAD model

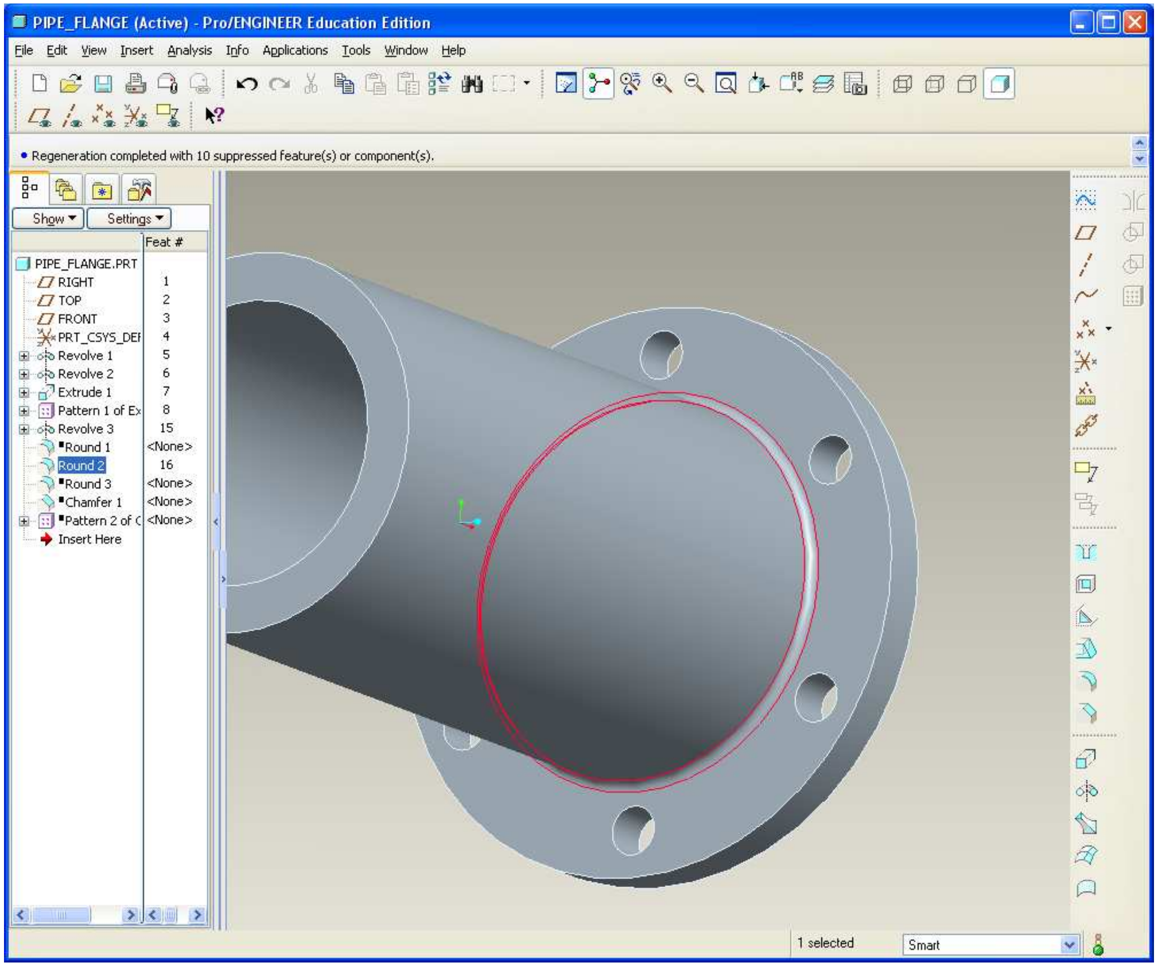
- We don't need this level of detail to perform the analysis we're interested in
- ***Best practice >***
    - Wherever possible suppress instead of delete
- Suppress chamfers and rounds that are not structural
- ***Note: Keep*** **Round 2**



EXERCISE – Structural Analysis

De-feature CAD model

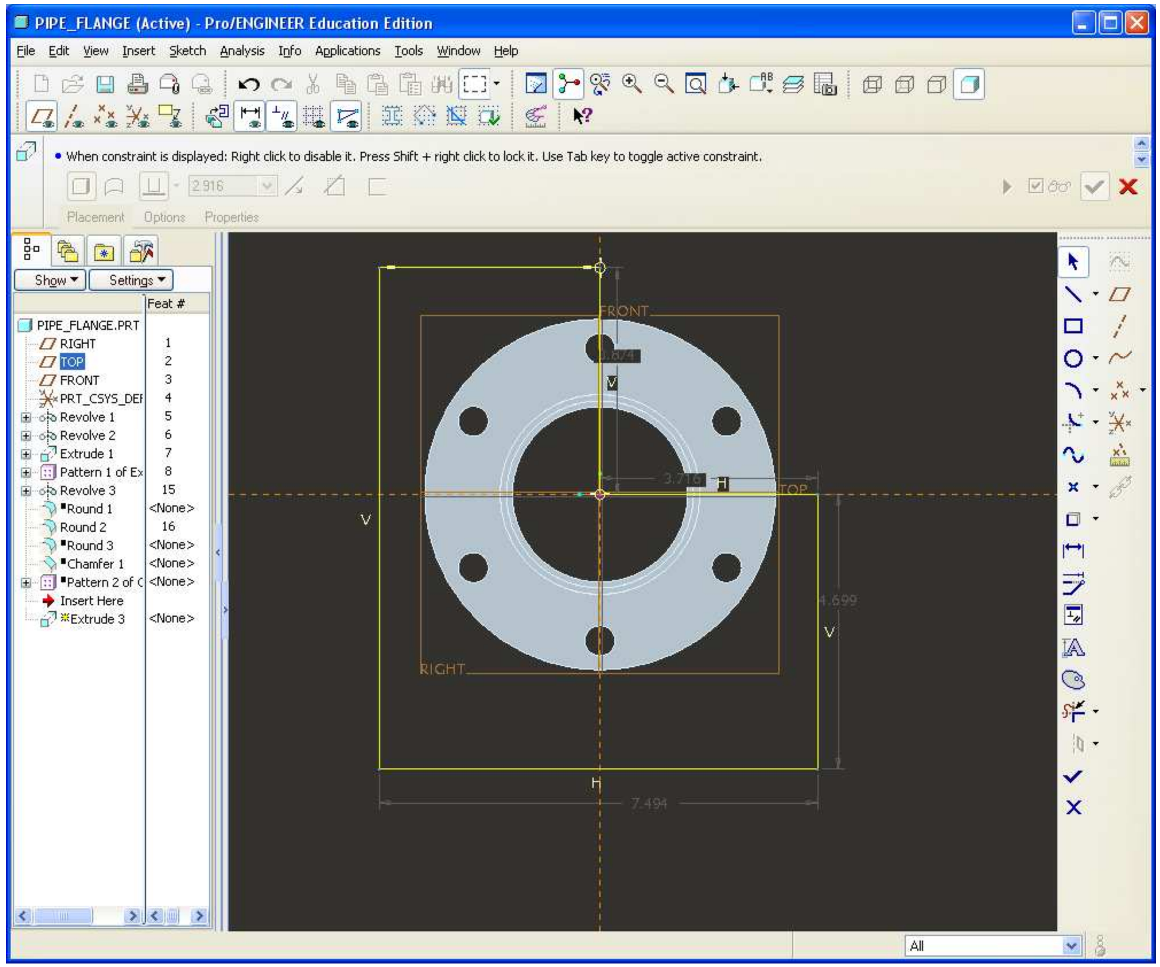
- Round 2 will definitely have a non-negligible effect on the stress in the component
- Can't get rid of this!



EXERCISE – Structural Analysis

Symmetry

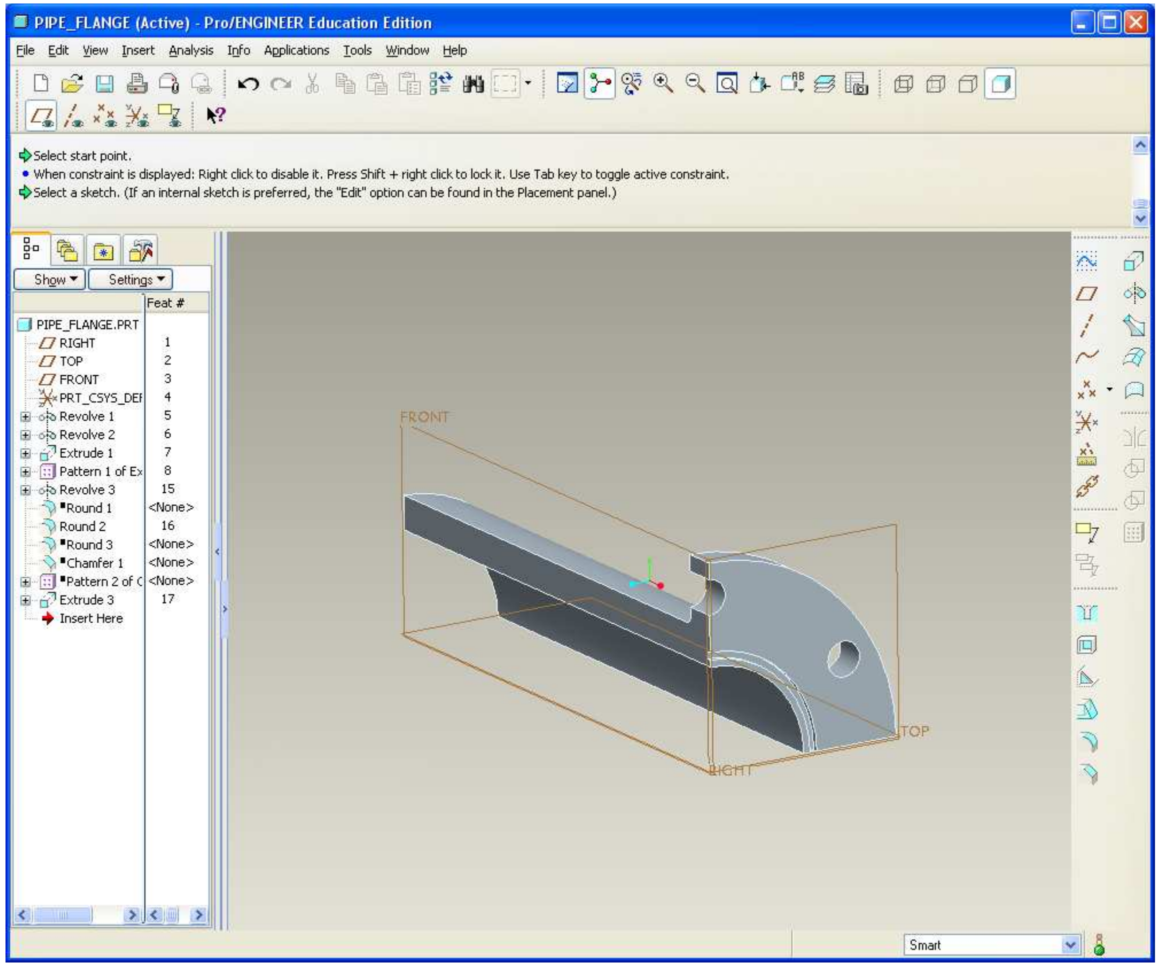
- **When possible, symmetry can aide an analysis substantially**
    - **Ease application of BC's**
    - **Reduces computation time required**
- **Cut the model along the two datums shown**



EXERCISE – Structural Analysis

Symmetry

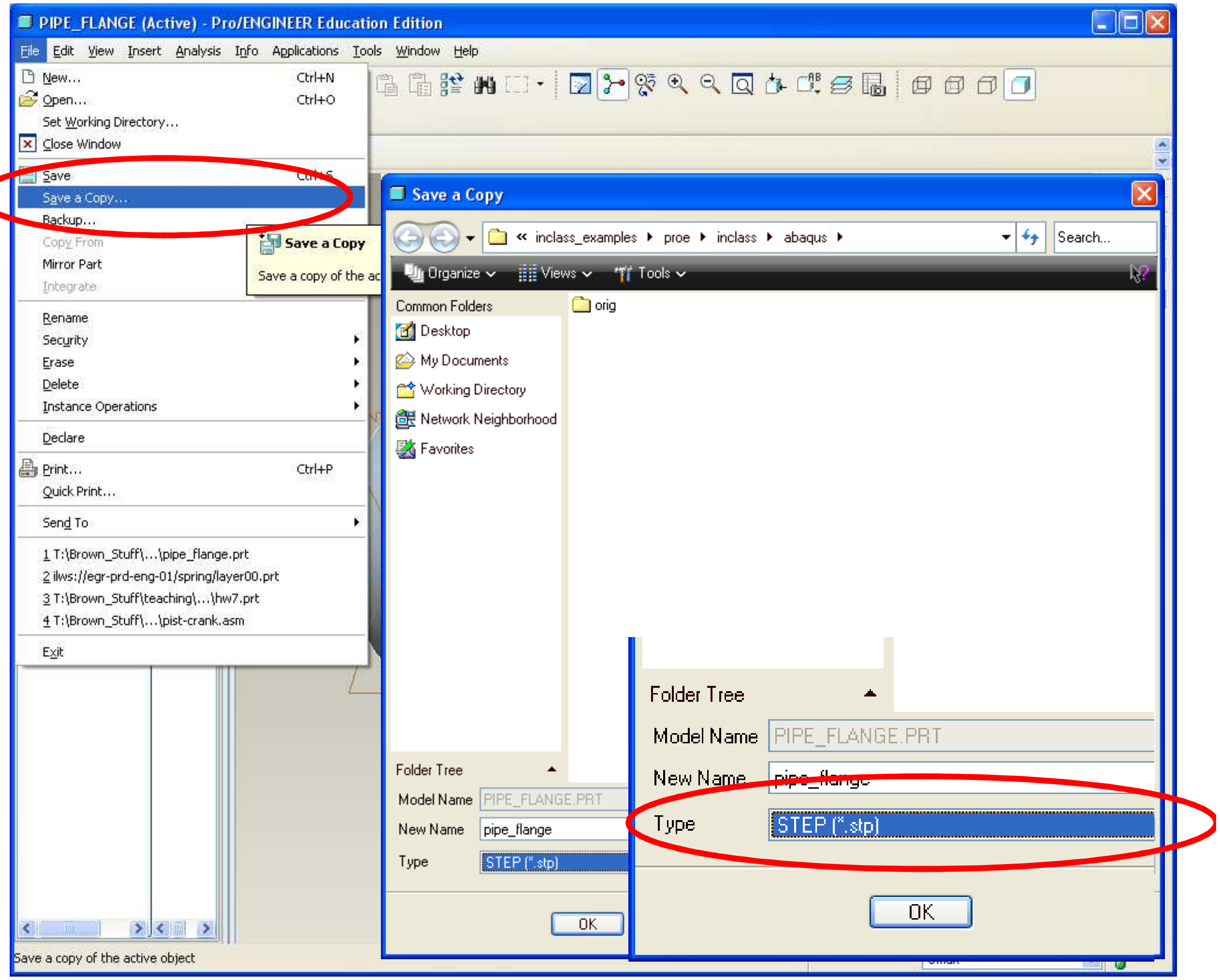
- **This should be all that's left**
- **If you're here, you're ready to export the model**



EXERCISE – Structural Analysis

Export Geometry

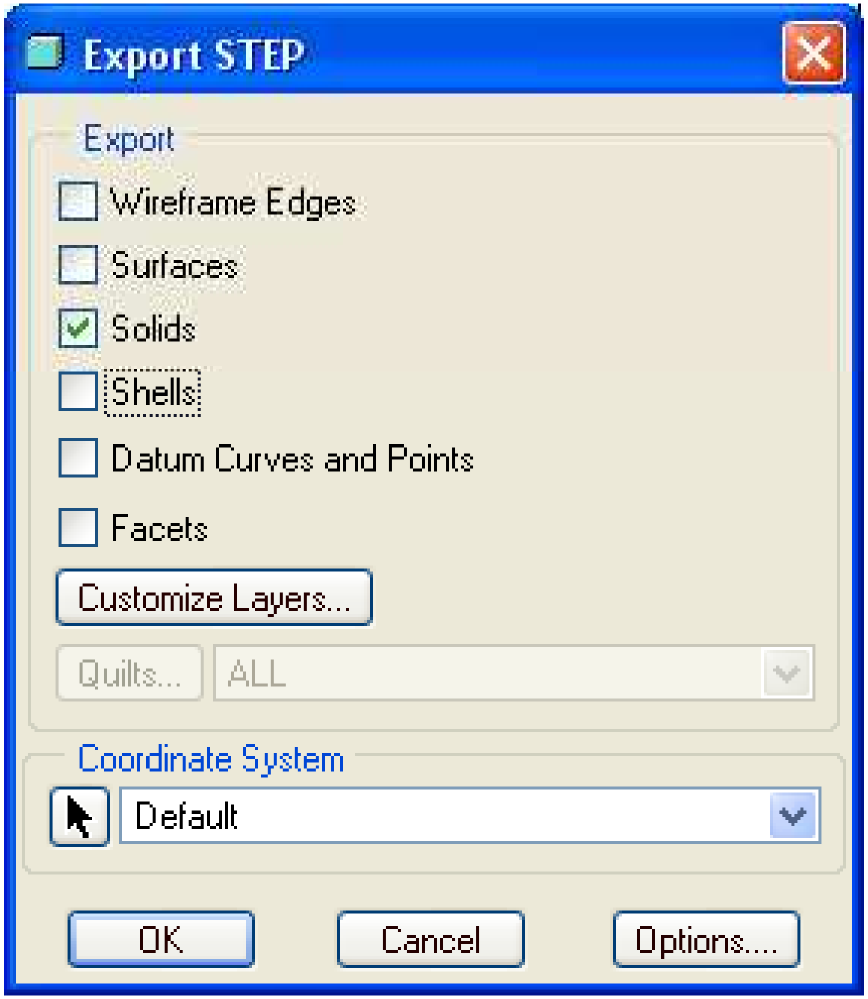
- File > Save a Copy…
- Select STEP as the file type
- Keep the same file name



EXERCISE – Structural Analysis

Export Geometry

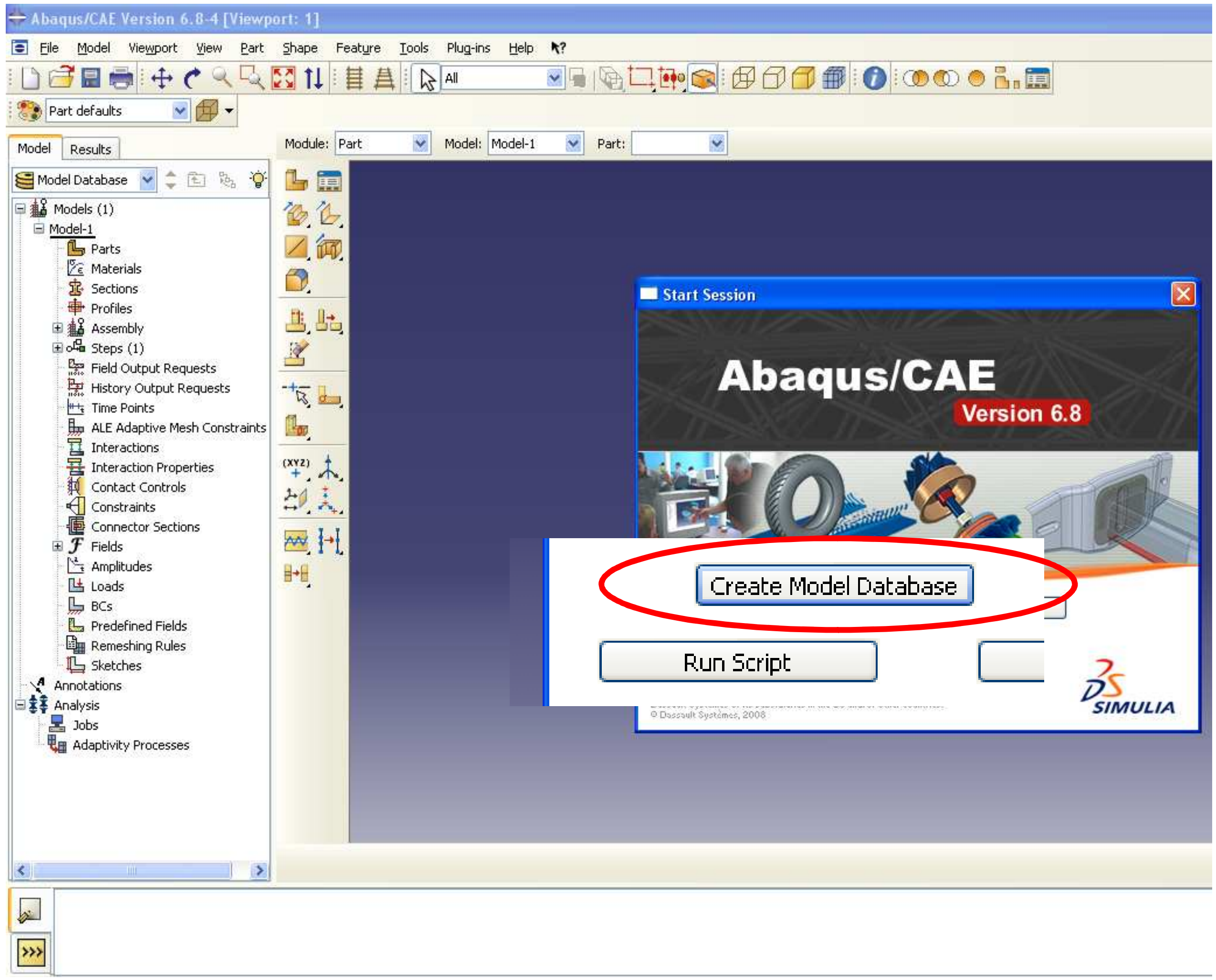
- From the Export STEP options dialog box
  - Un-check Shells
  - Make sure Solids are checked
  - Keep all other default preferences
- Click OK



EXERCISE – Structural Analysis

Open ABAQUS

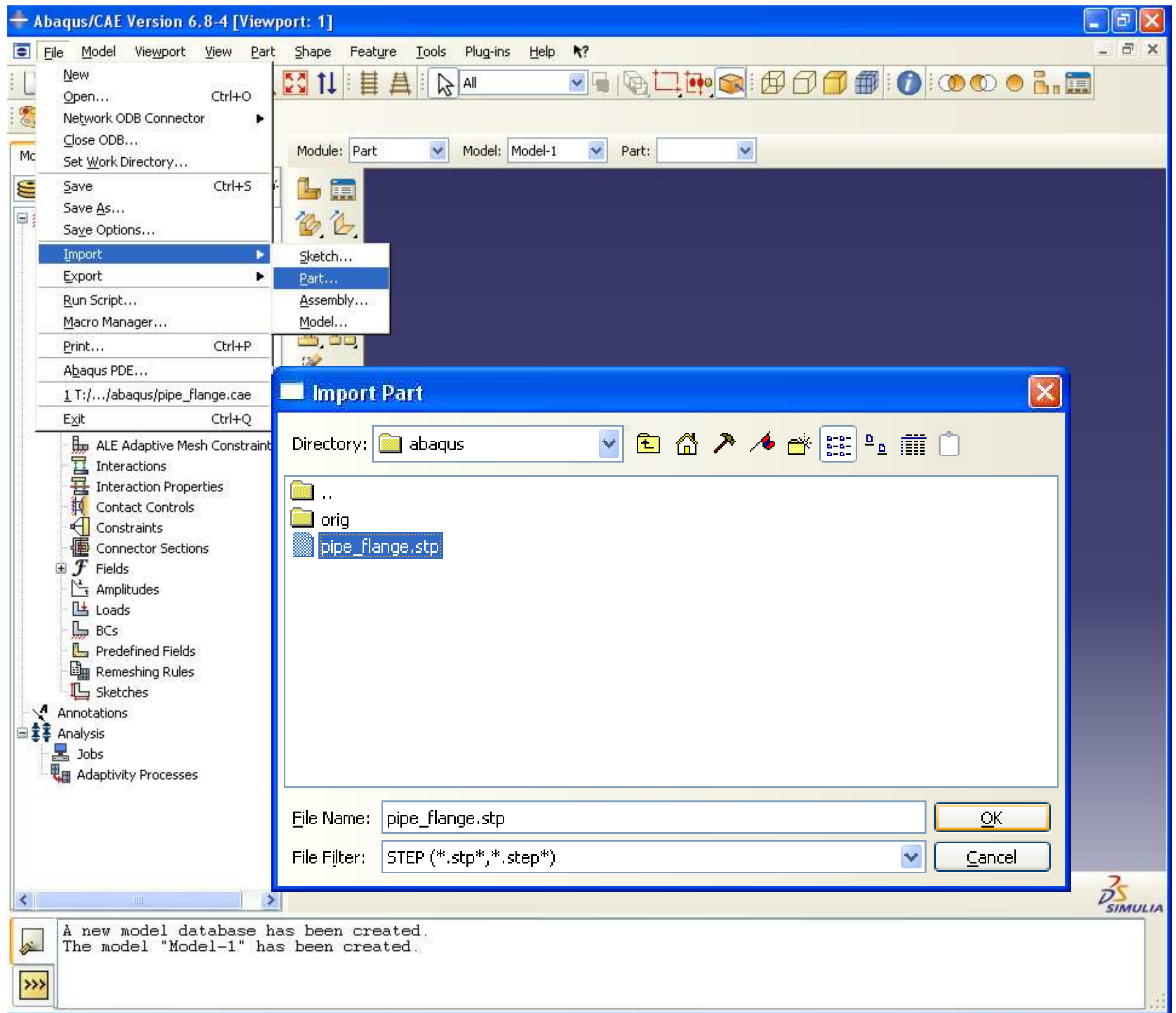
- ABAQUS is a third party Finite Element Analysis package
- Note the GUI layout > Very similar to Pro/E
  - Model Tree
  - Command Icons
- Click Create Model Database to get started



EXERCISE – Structural Analysis

Import our geometry

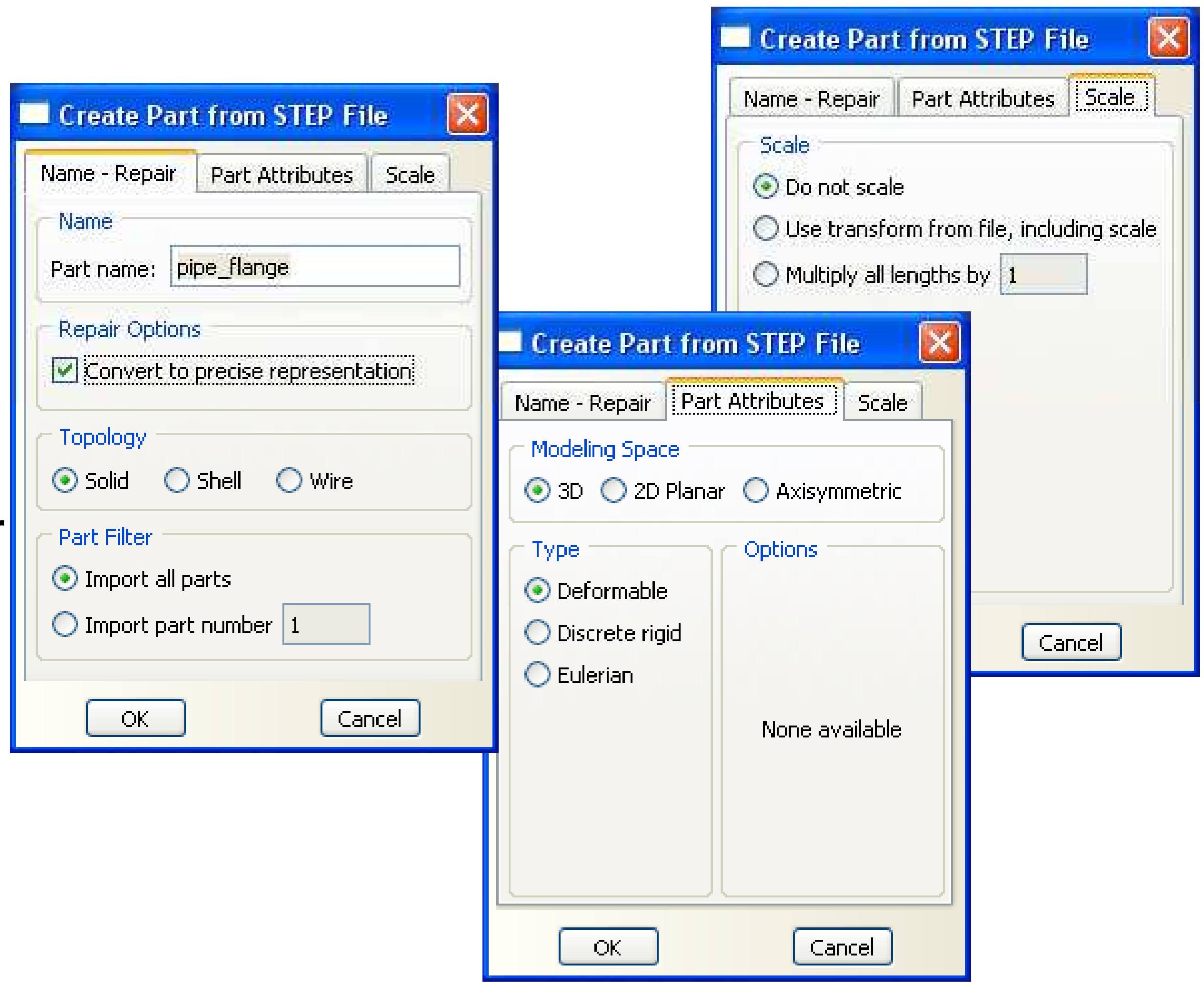
- **File > Import > Part**
- **Navigate over to our STEP file**
- **Click OK**



EXERCISE – Structural Analysis

Import our geometry

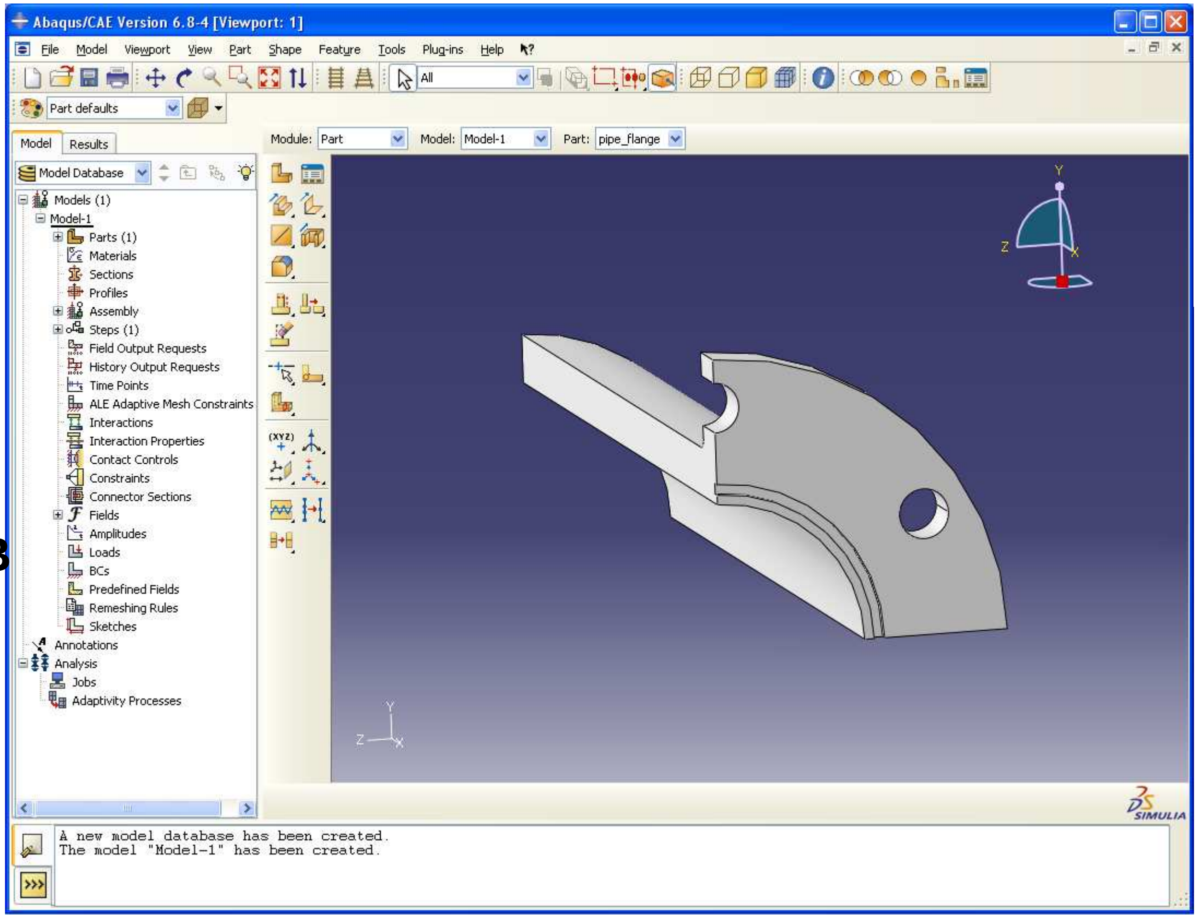
- **Create Part... Dialog pops up**
    - **On Name – Repair dialog check Convert to precise representation**
- **Keep the defaults on the other pages, but look over what is available**
- **Click OK**
    - **STEP is loaded**
    - **This will take a minute**



EXERCISE – Structural Analysis

Get familiar with the interface

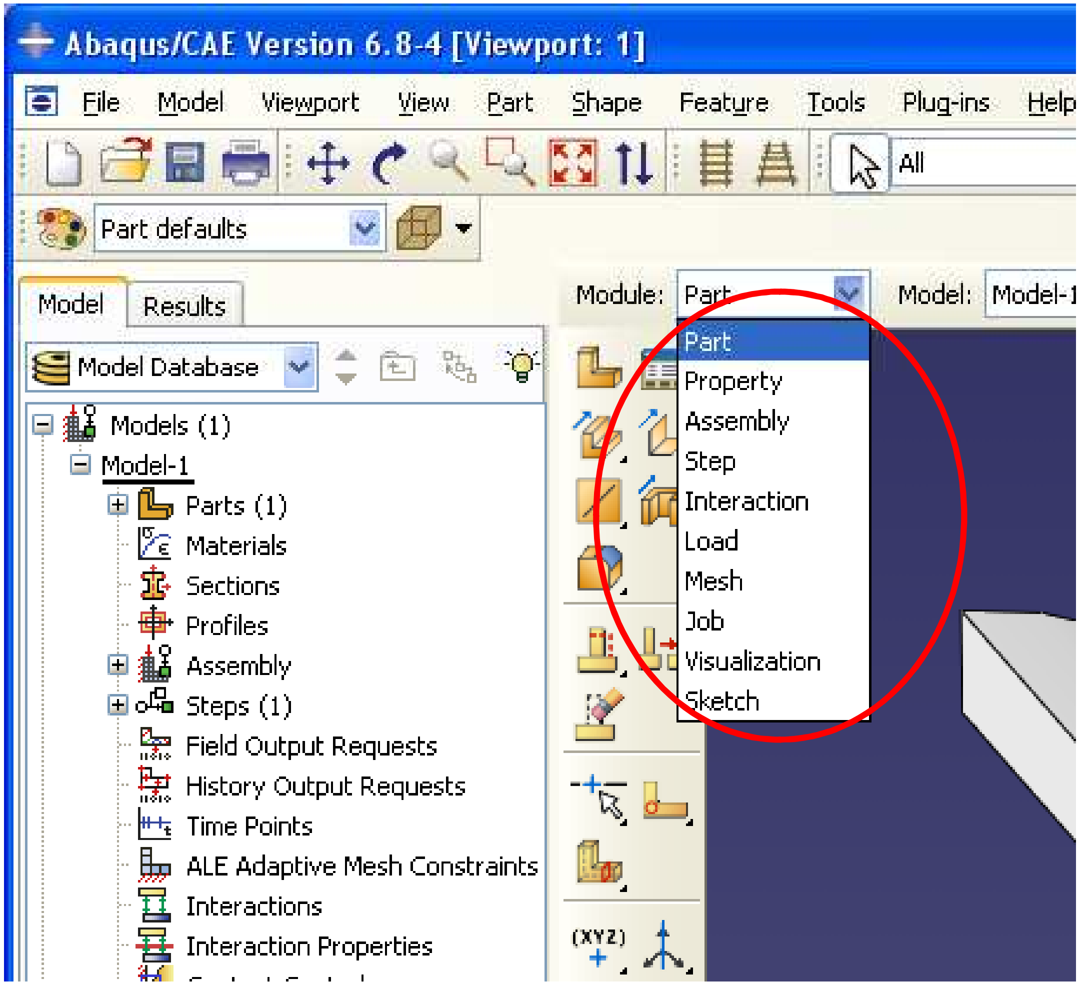
- **POSSIBLE THE MOST USED FUNCTION – *Spin, Pan, Zoom is DIFFERENT!***
  - **For either Spin, Pan or Zoom start by holding the Crtl + Alt Key**
  - **Spin – Crtl + Alt + LMB**
  - **Pan – Crtl + Alt + MMB**
  - **Zoom – Crtl + Alt + RMB**
- **Try this until comfortable**



EXERCISE – Structural Analysis

Get familiar with the interface

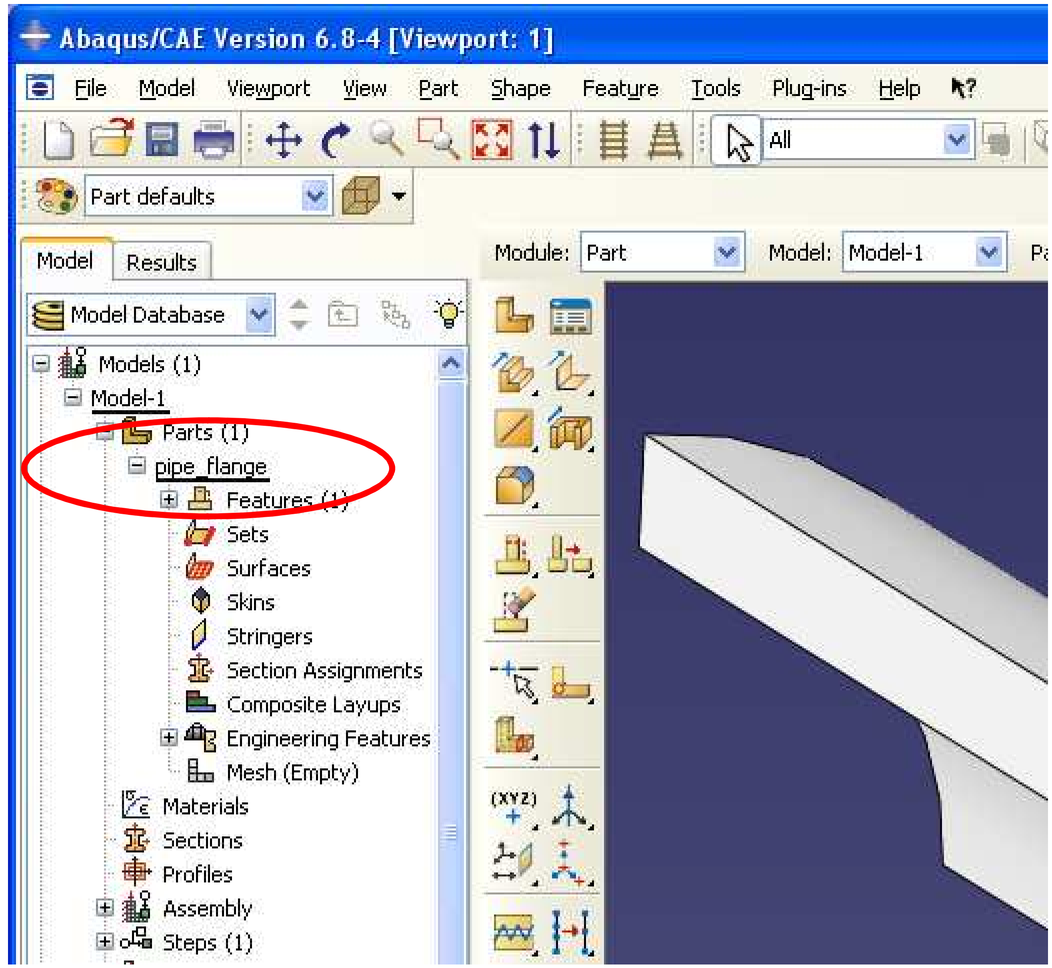
- **The ABAQUS/CAE interface is broken up into a number of modules**
- **Definitions established in these modules set necessary parameters and options for an analysis**
- **Working through these one at a time makes the setup process very orderly**



EXERCISE – Structural Analysis

Part Module

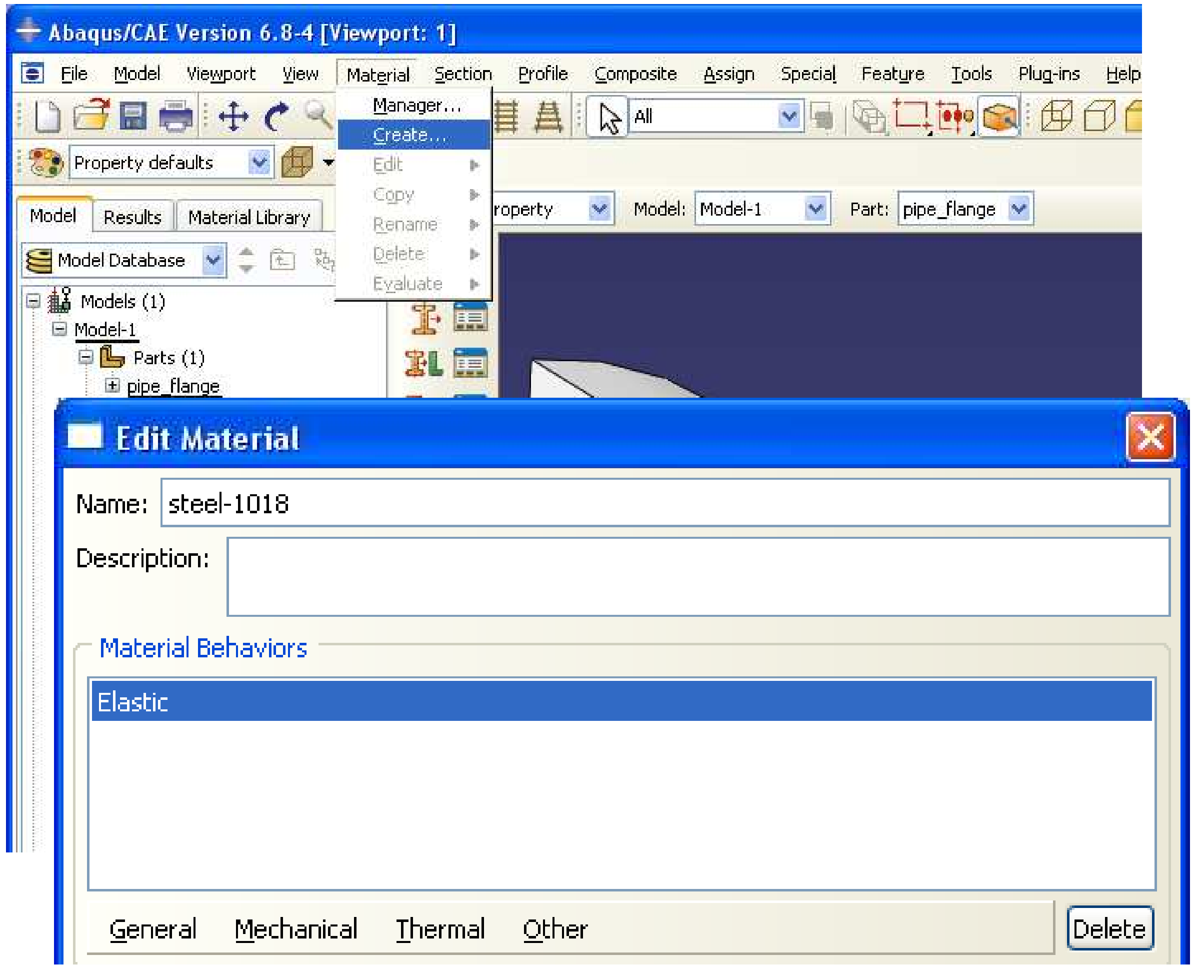
- **Importing the STEP file completes everything we need in the part module**
- **Expand the Model Tree to see the STEP file name**
- **Take a look at the geometry tools in ABAQUS**
    - **You can create solid models right in ABAQUS**
    - **If you know Pro/E, *don't do this***



EXERCISE – Structural Analysis

Property Module

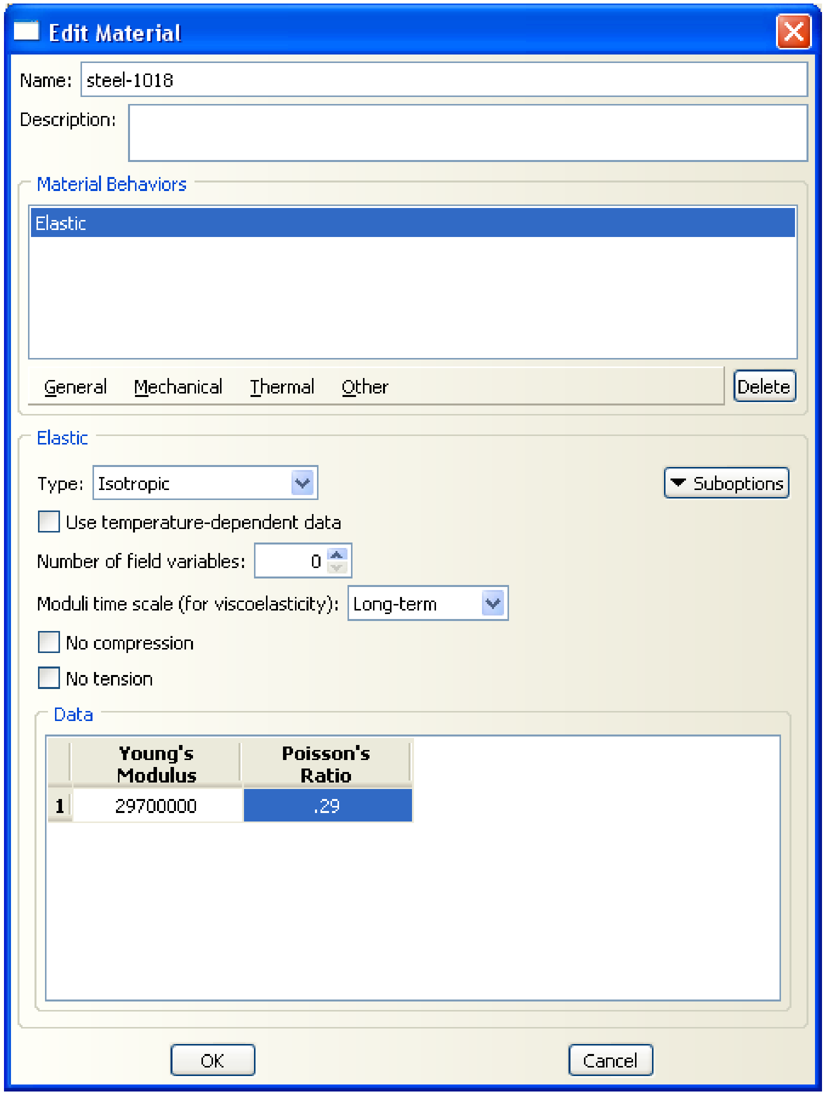
- **Define material properties**
  - **Material > Create**
  - **Rename the material 1018-steel**
  - **For Material Behaviors select Mechanical > Elasticity > Elastic**



EXERCISE – Structural Analysis

Property Module

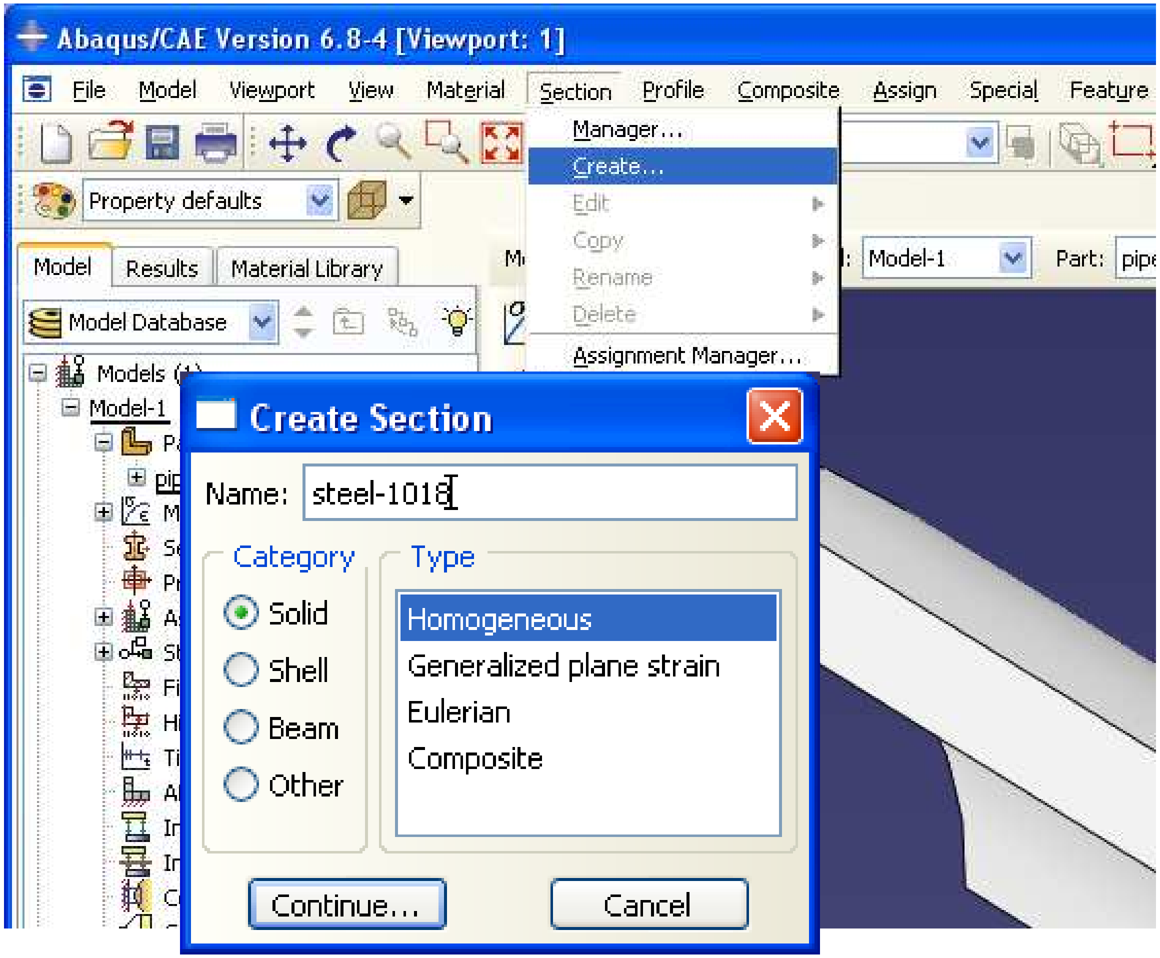
- **Define parameters**
  - **Leave Type as Isotropic**
  - **Enter 29.7e6 psi as Young's Modulus**
  - **Enter .29 as Poisson's Ratio**
- **NOTE: ABAQUS does *NOT* keep track of units. It's your responsibility to make sure the units are coordinated**



EXERCISE – Structural Analysis

Property Module

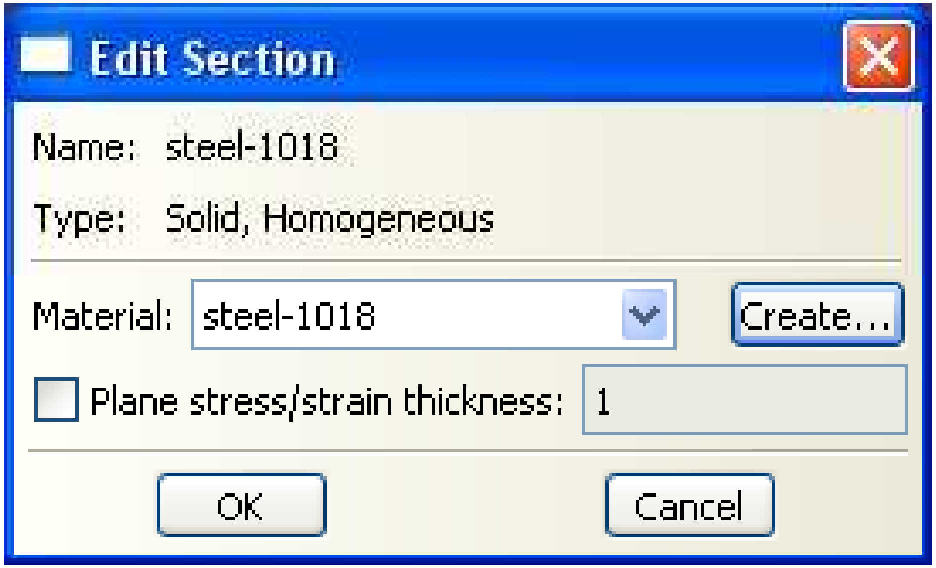
- **Define Section properties**
  - **A Section is a set of parameters that defines how a geometry should be treated**
  - **Solid, homogeneous, material, etc.**
  - **Sections are then assigned to geometry**
- **Name the section steel-1018**
- **Keep defaults of Solid and Homogeneous**
- **Click Continue…**



EXERCISE – Structural Analysis

Property Module

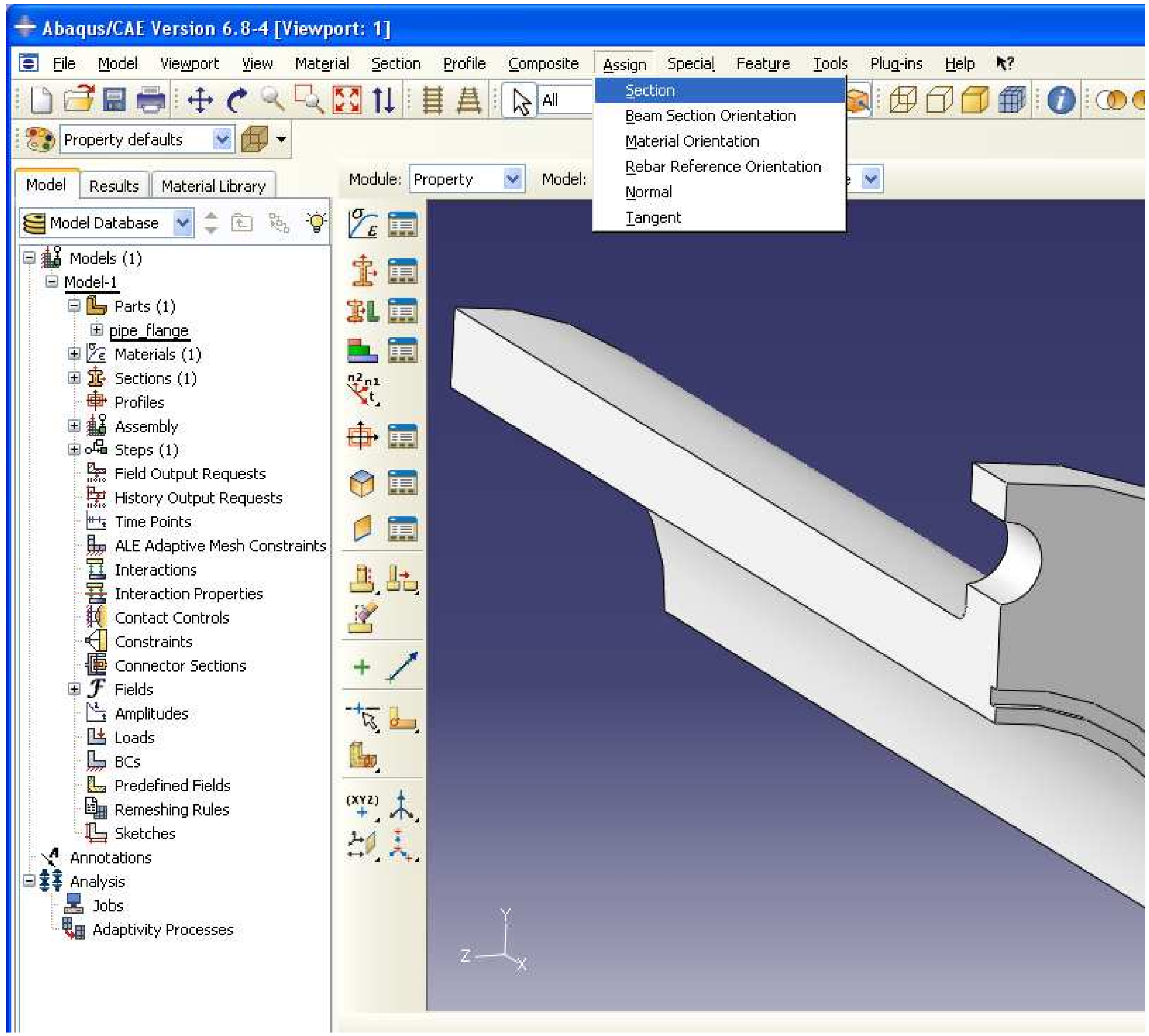
- **In the next dialog**
- **Make sure steel-1018 is selected as material**
- **Un-check Plane stress/strain**
- **Click OK**



EXERCISE – Structural Analysis

Property Module

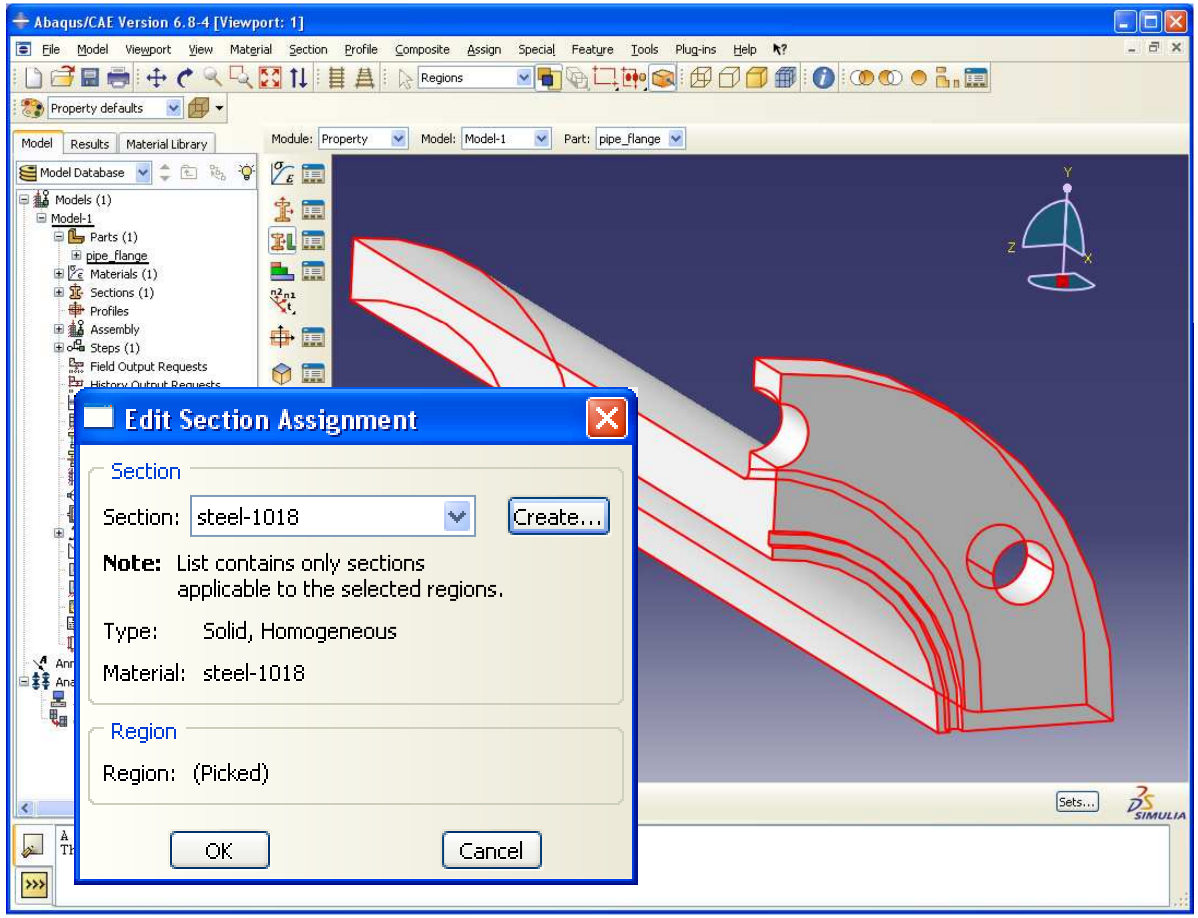
- **Assign the Section**
    - **The Section properties must be applied to the geometry**
- **Assign > Section**



EXERCISE – Structural Analysis

Property Module

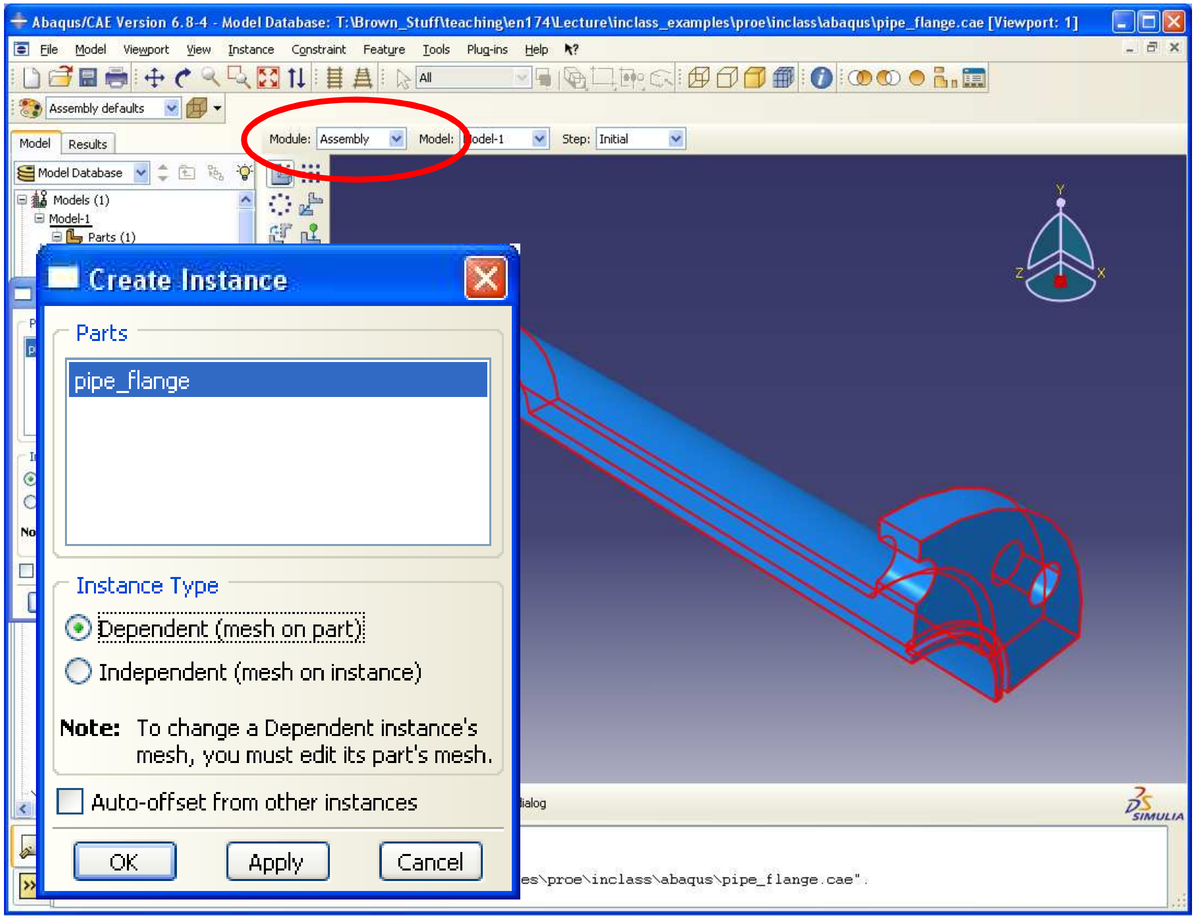
- **Select the region as shown**
- **The Edit Section Assignment dialog comes up**
- **Make sure the steel-1018 section is in the Section block**
- **Click OK**



EXERCISE – Structural Analysis

Assembly Module

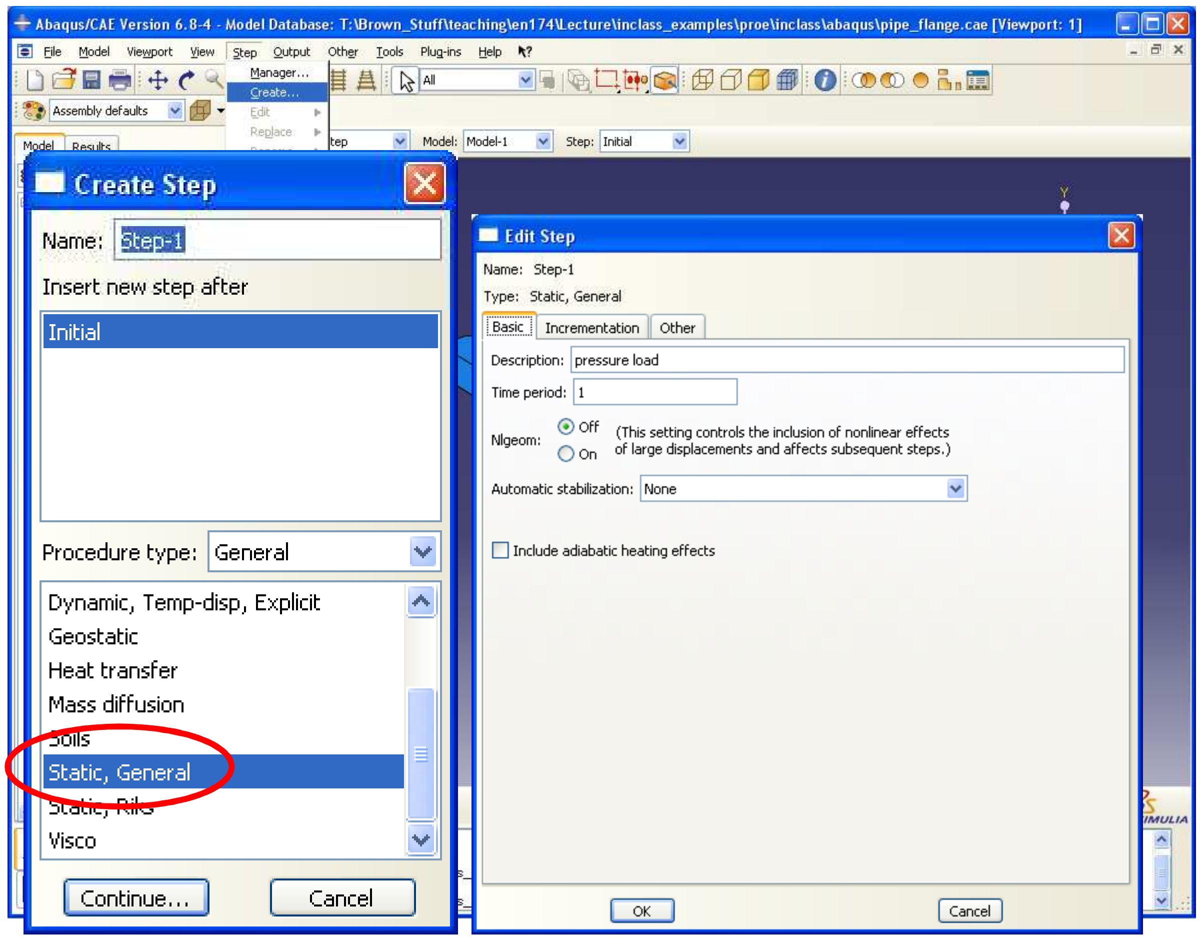
- Switch to the Assembly Module
- Create an Instance of the pipe_flange
    - Instances allow for multiple uses of geometry in assembly
    - If there's only one part, this isn't a meaningful step
- Click on Instance Part icon
- Leave Instance Type as Dependent
- Click OK



EXERCISE – Structural Analysis

Step Module

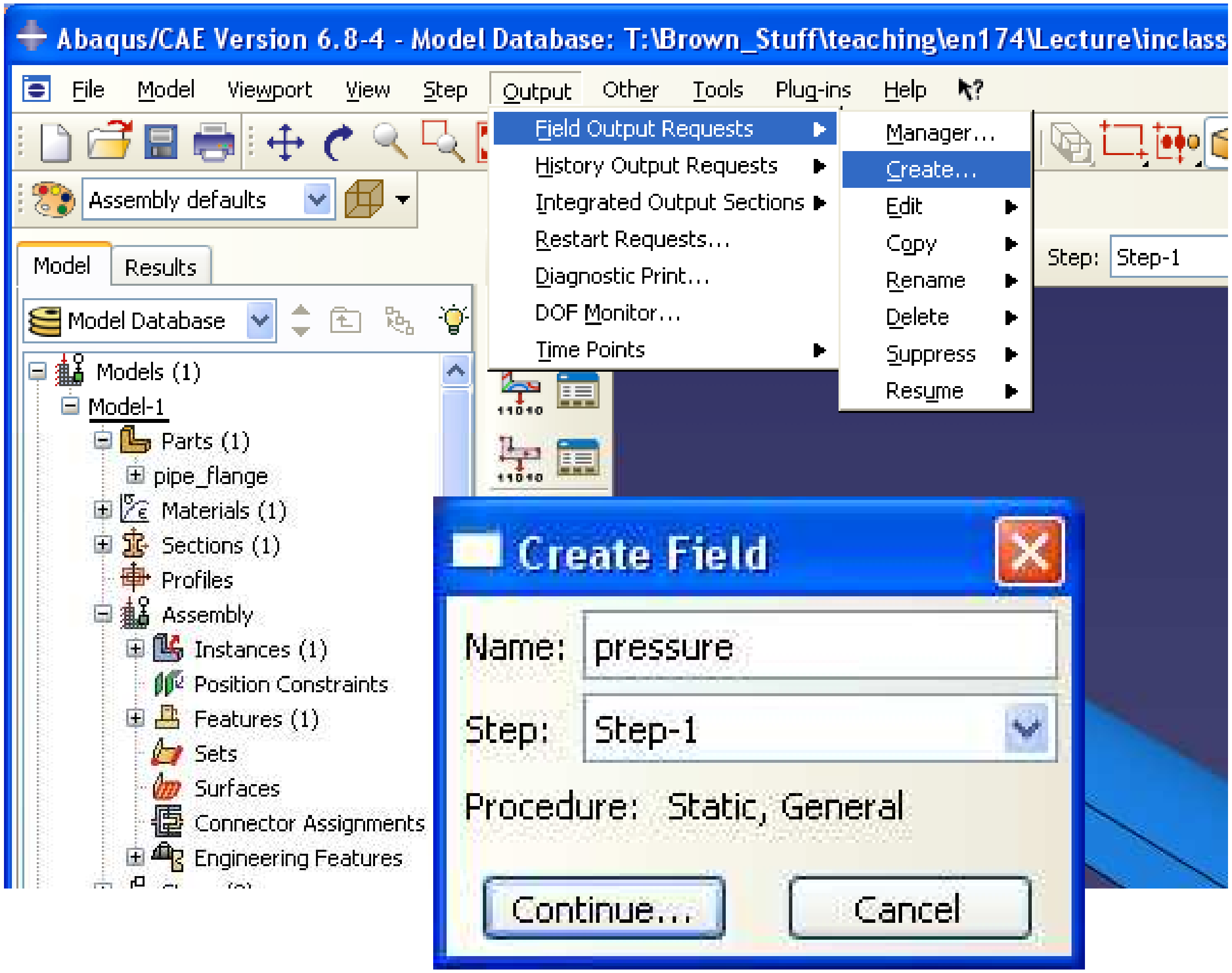
- **Switch to the Step Module**
- **Create a new Load Step**
  - **Step > Create**
- **For Procedure type select Static, General**
- **Click Continue**
- **We'll use the defaults for the step > Click OK**



EXERCISE – Structural Analysis

Step Module

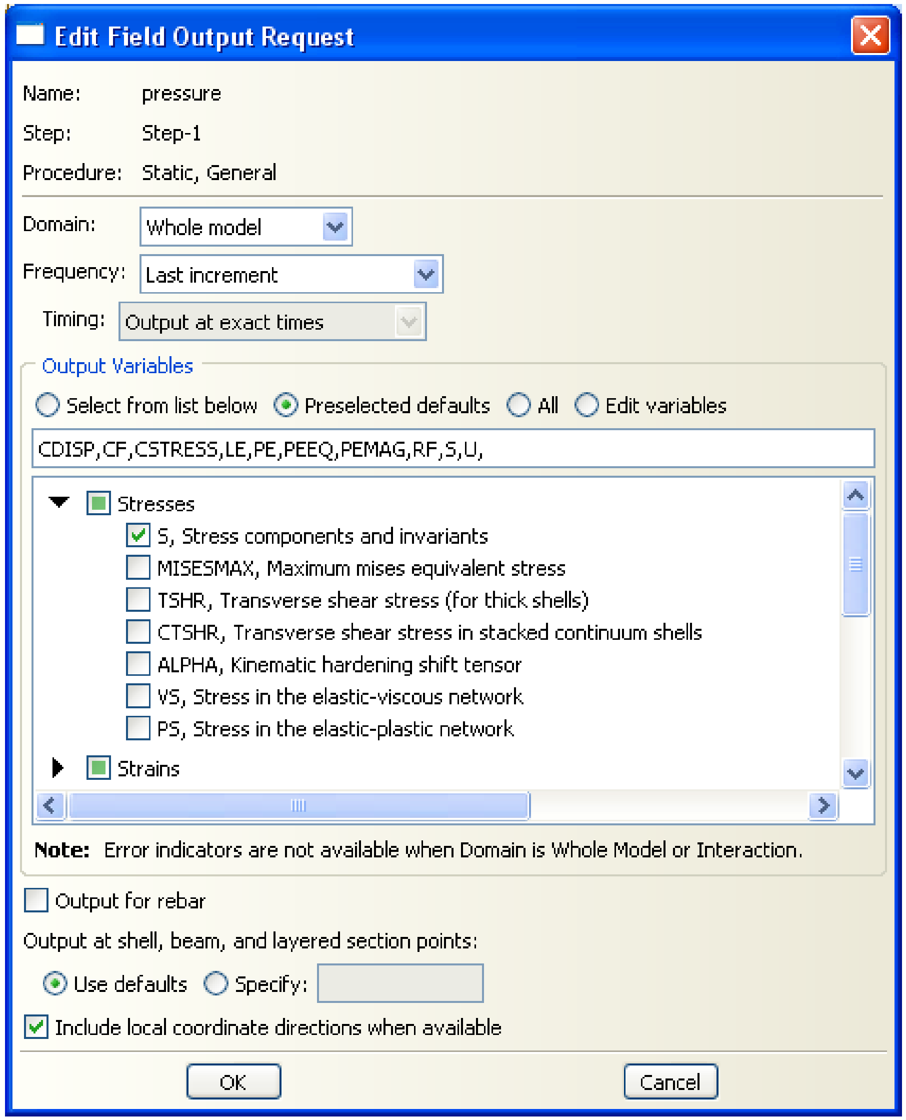
- **Select what variable should be written to output**
  - **Field Output Requests - written at the end**
  - **History Output Requests - written at sub-steps**
- **Click Field Output Requests > Create**
- **Use pressure as the name for the results set**
- **Click Continue…**



EXERCISE – Structural Analysis

Step Module

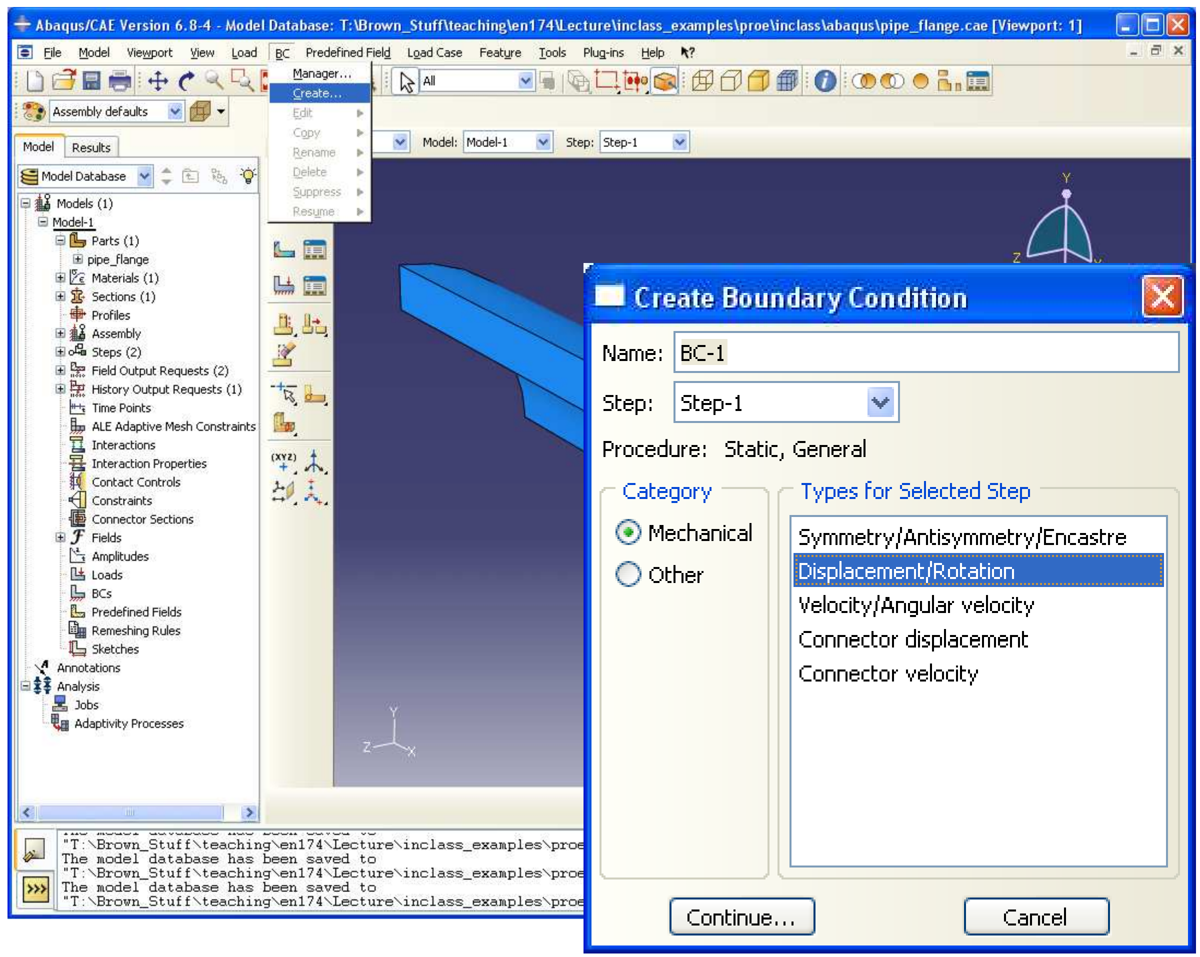
- There's a lot of options here – let's stay general
  - Domain – Whole model
  - Frequency – Last increment
  - Use Preselected defaults for the variable selection



EXERCISE – Structural Analysis

Load Module

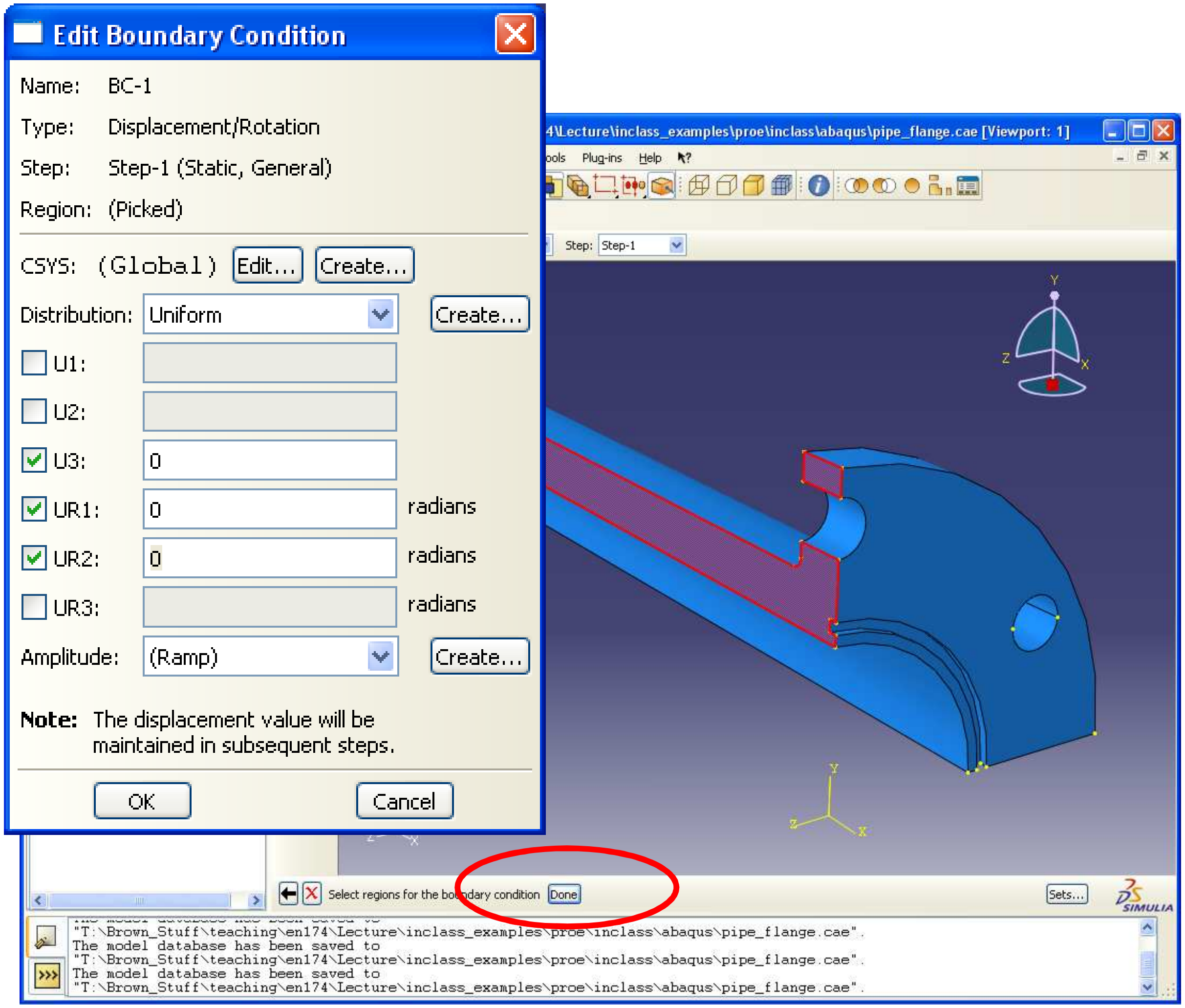
- In this module we establish loads and boundary conditions
- BC > Create
- Rename if desired
- Step is Step-1
- Category > Check Mechanical
- Types for Selected Step > Displacement/Rotation
- Click Continue…



EXERCISE – Structural Analysis

Load Module

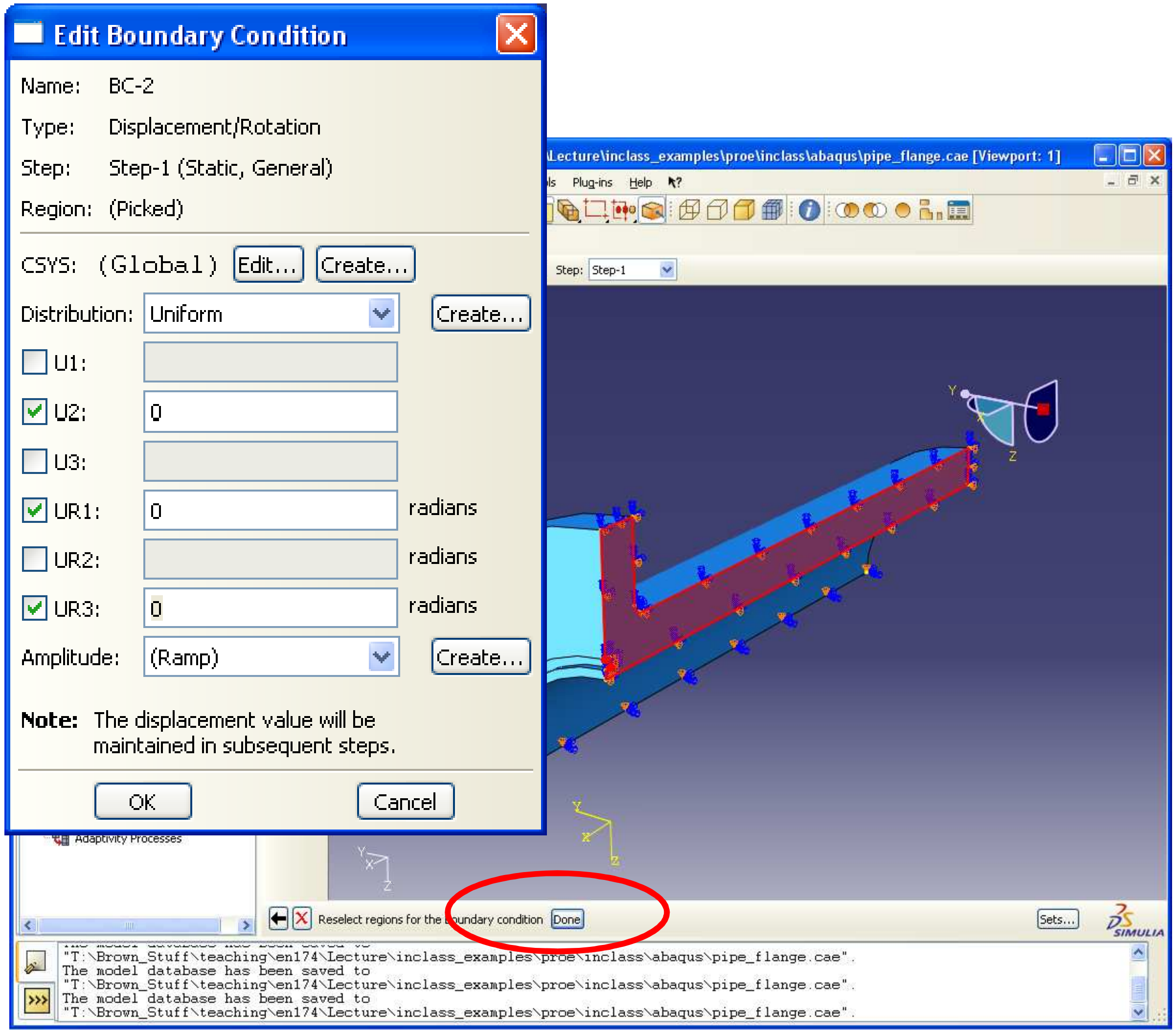
- **Select the highlighted regions**
- **Click Done**
- **Specify the following BC's:**
    - **Displacement u3=0**
    - **Rotations about u1, u2=0**
- **Click OK**



EXERCISE – Structural Analysis

Load Module

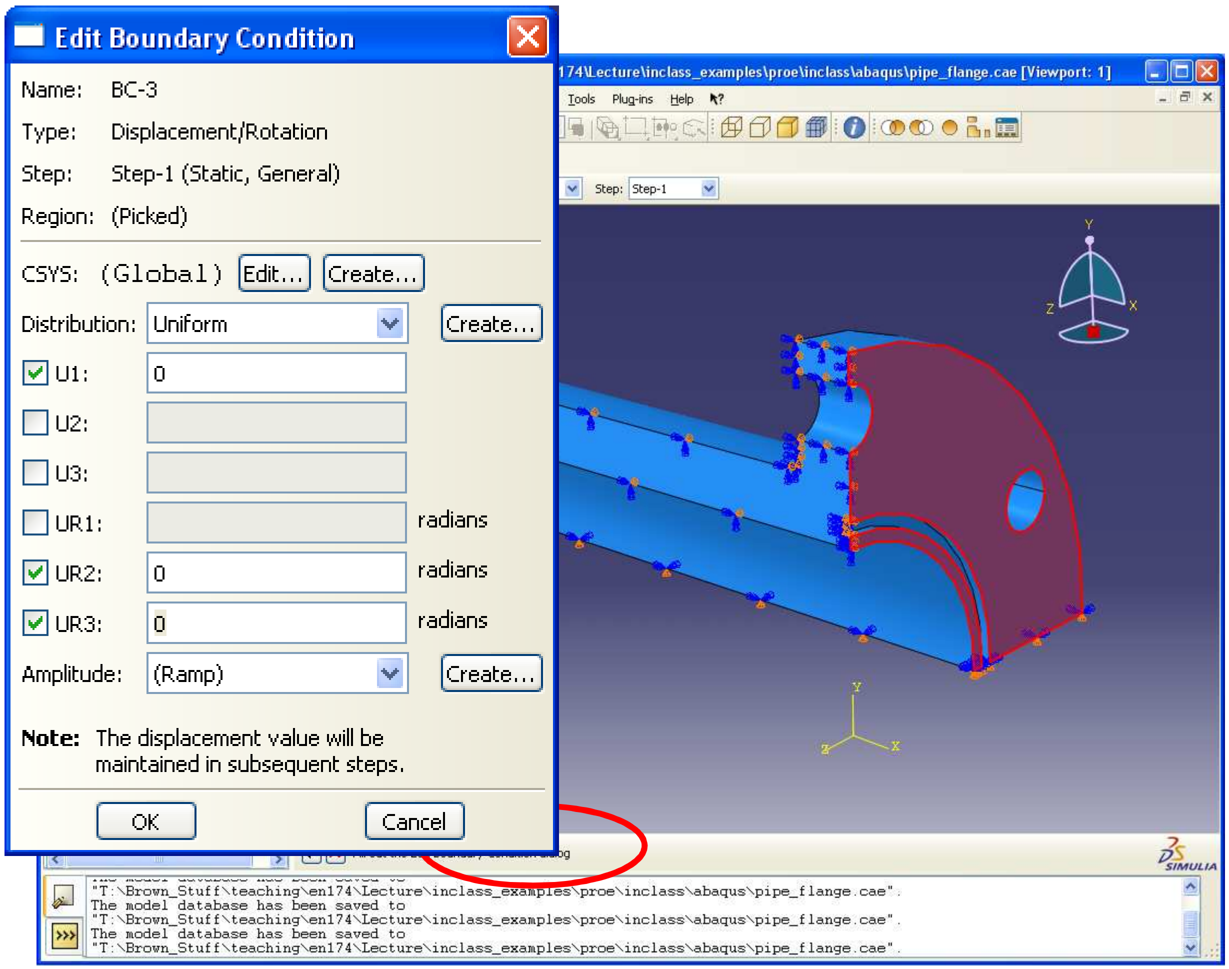
- **Select the opposite side**
- **Click Done**
- **Specify the following BC's:**
  - **Displacement u2=0**
  - **Rotations about u1, u3=0**
- **Click OK**



EXERCISE – Structural Analysis

Load Module

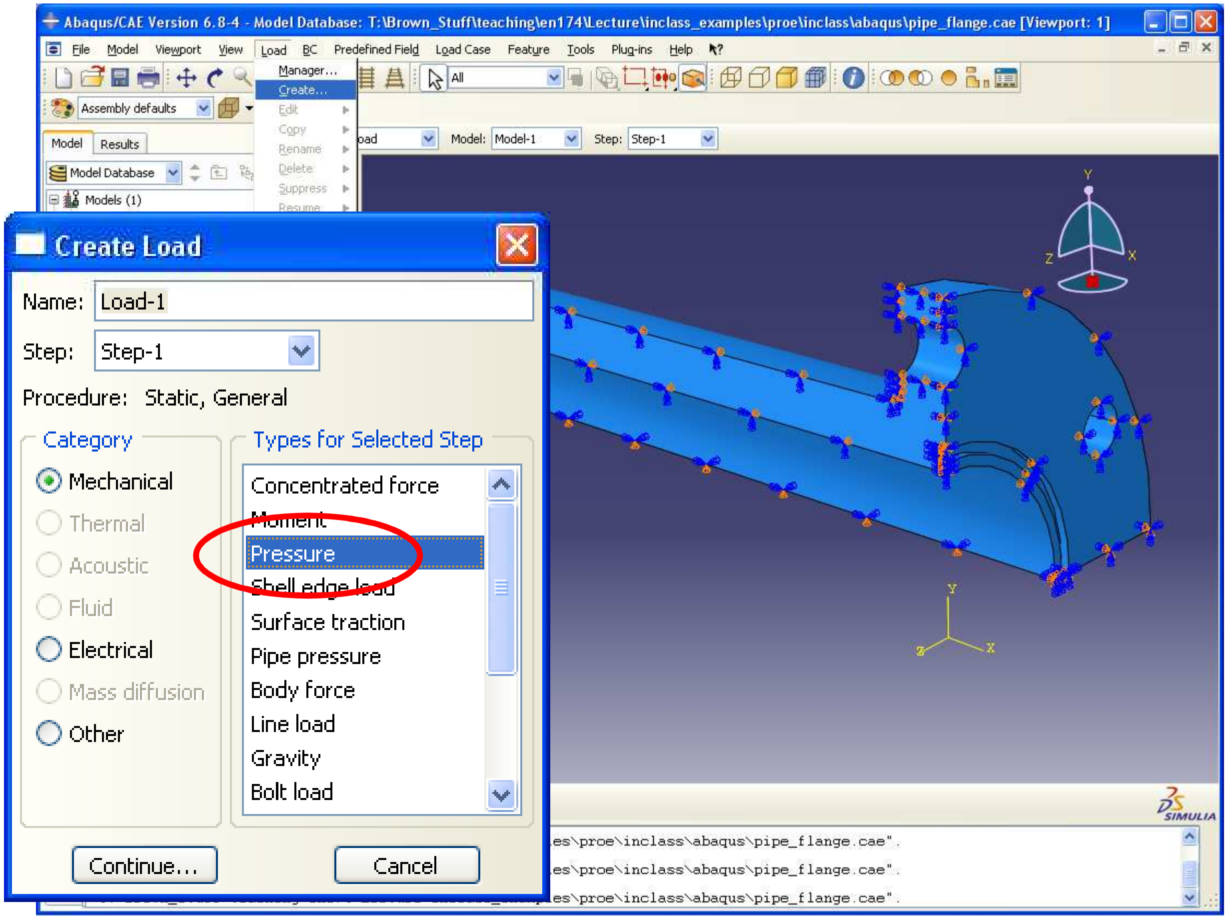
- **Select the flange face**
- **Click Done**
- **Specify the following BC's:**
  - **Displacement u1=0**
  - **Rotations about u2, u3=0**
- **Click OK**



EXERCISE – Structural Analysis

Load Module

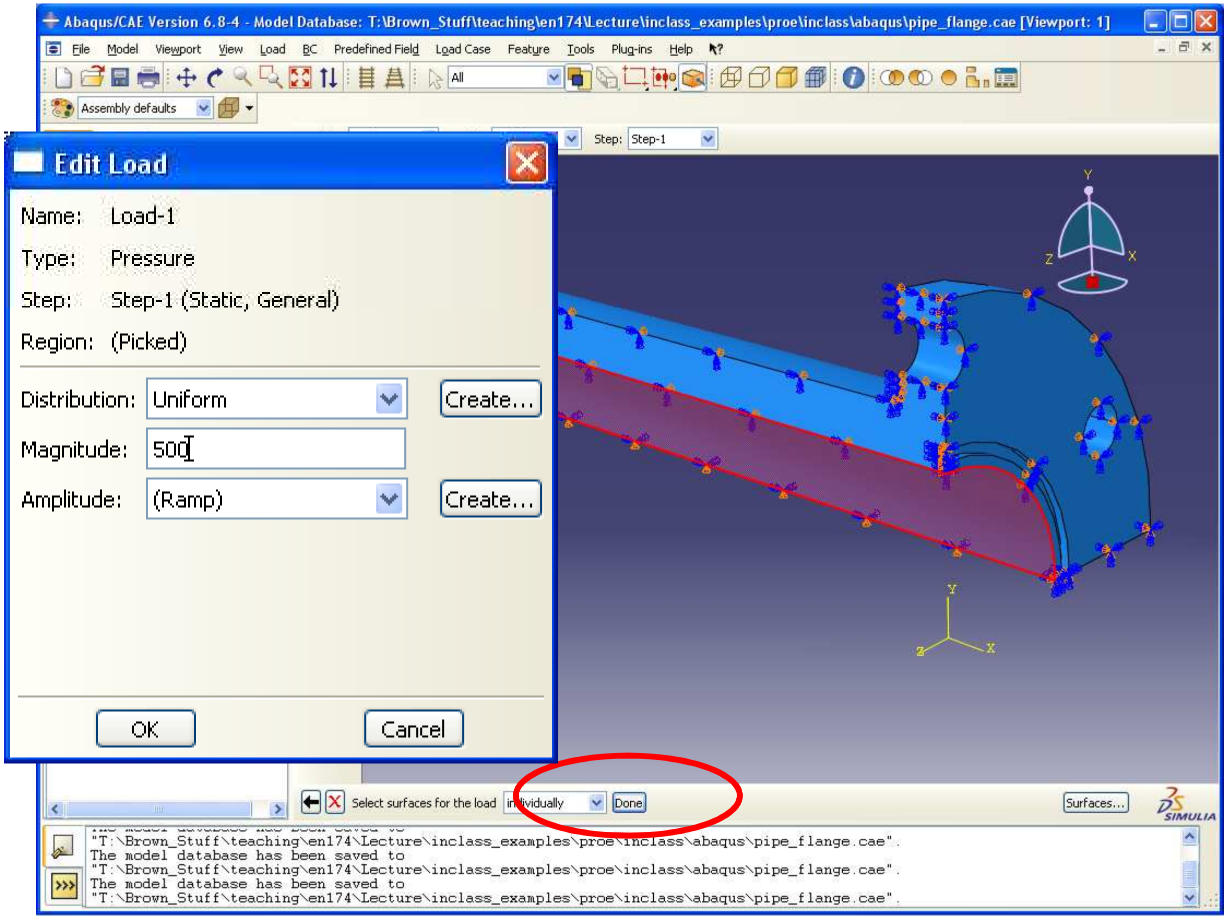
- **Create a pressure load on the inside of the pipe**
- **Load > Create**
- **Select Pressure for Types..**
- **Click Continue…**



EXERCISE – Structural Analysis

Load Module

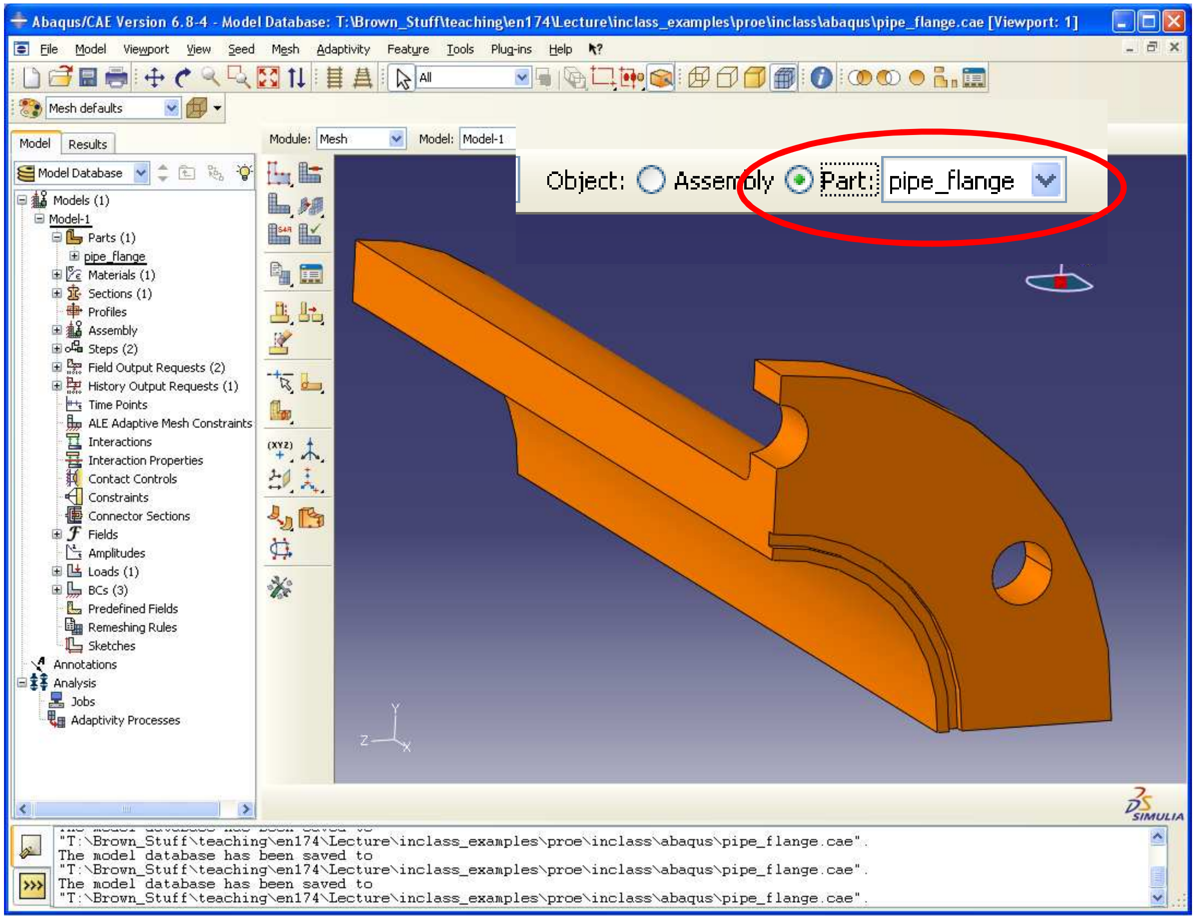
- Select the face shown
- Click Done
- Use 500 for Magnitude
- Click OK
- SAVE!



EXERCISE – Structural Analysis

Mesh Module

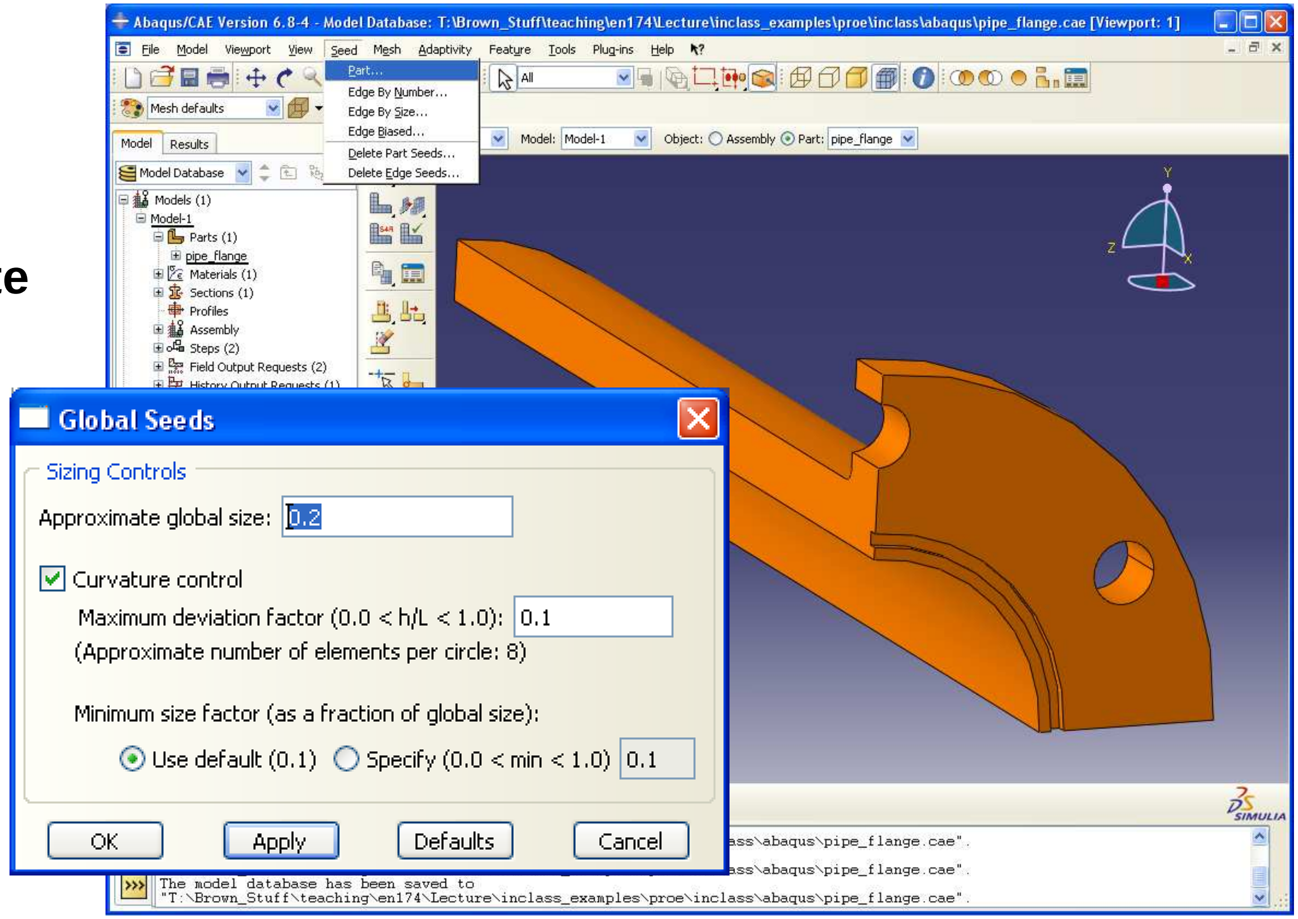
- **Switch to Mesh module**
- **Toggle to operate on the Part rather than Instance**



EXERCISE – Structural Analysis

Mesh Module

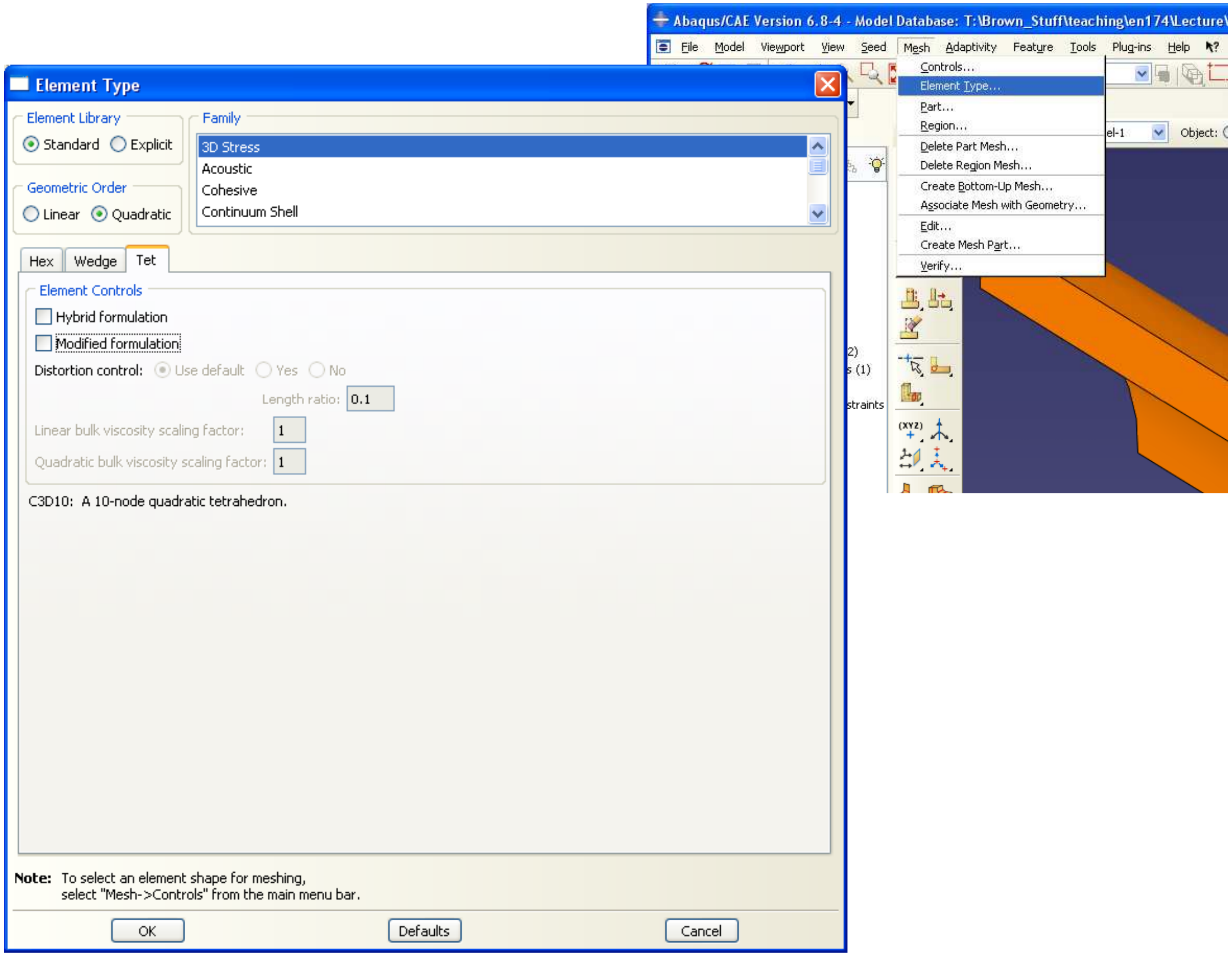
- **Prior to creating mesh, we need to tell ABAQUS approximately how big the elements should be**
- **Seed > Part**
- **Enter .2 as Approximate global size**
- **Click OK**



EXERCISE – Structural Analysis

Mesh Module

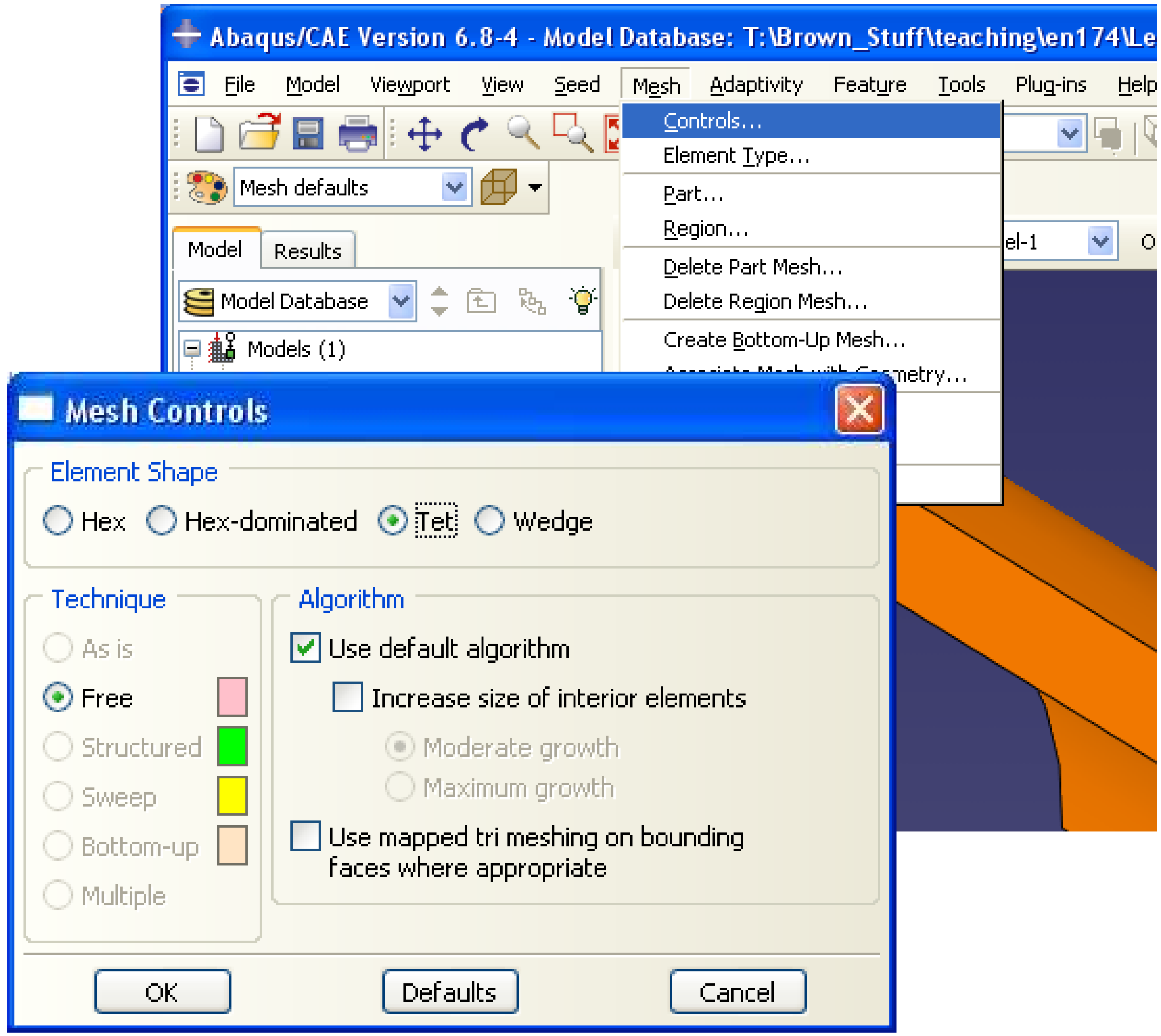
- We need to select the element type
- Mesh > Element Type
- Select >
    - Standard
    - 3D Stress
    - Quadratic
    - Tet
        - Uncheck formulation options
- Click OK



EXERCISE – Structural Analysis

Mesh Module

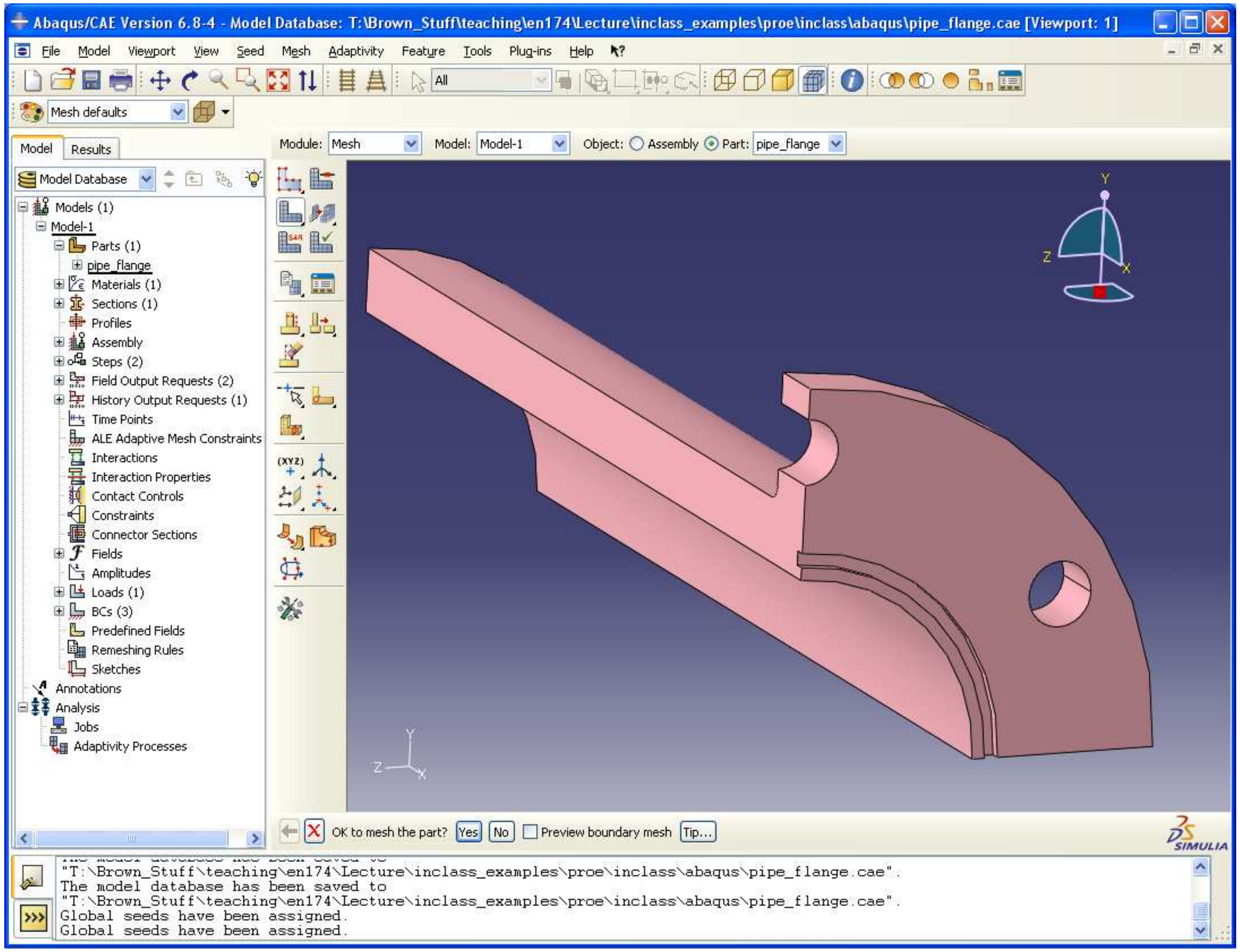
- We need to tell ABAQUS to use this element type
- Mesh > Controls…
- Click Tet
- Click OK



EXERCISE – Structural Analysis

Mesh Module

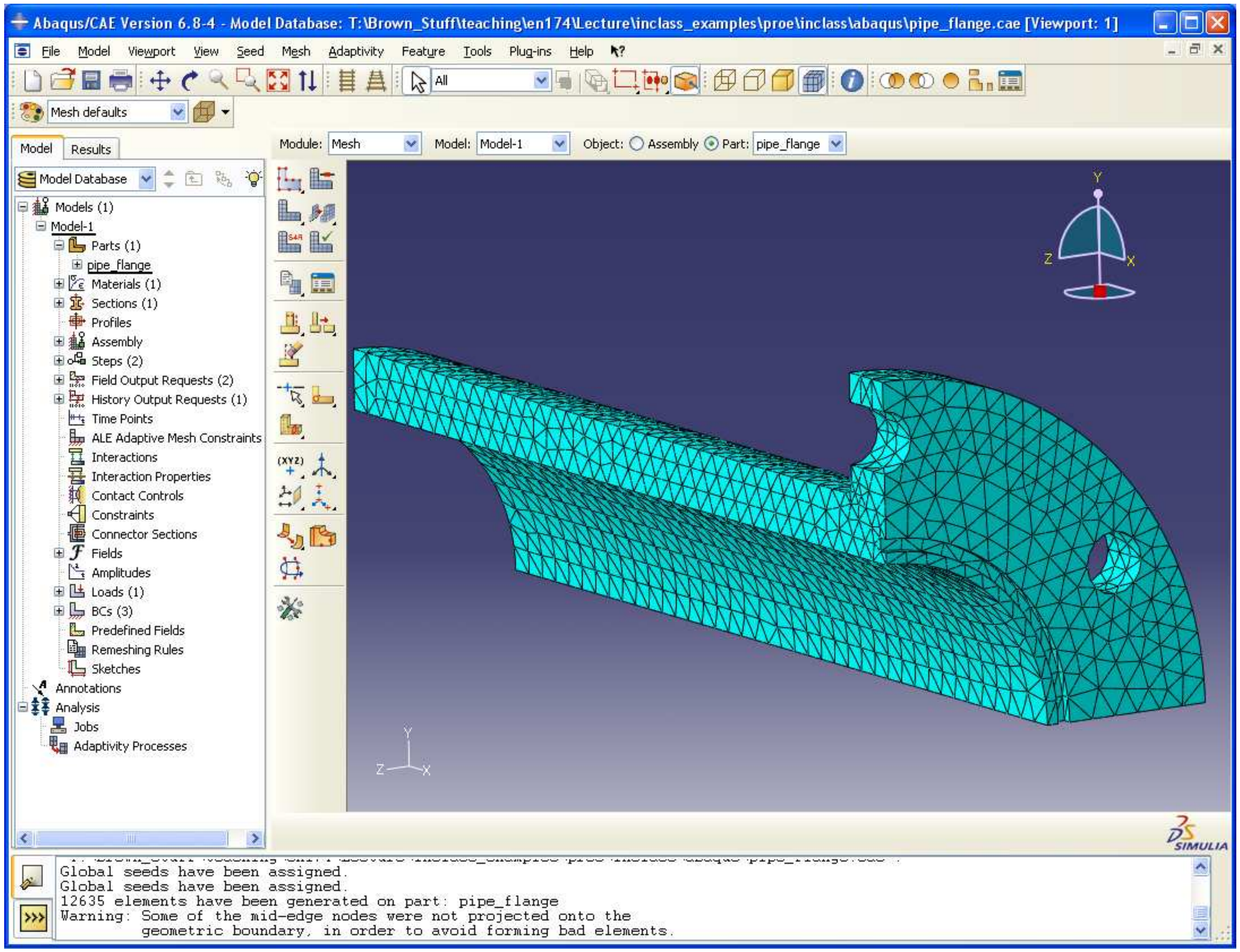
- **Create the mesh**
- **Mesh > Part**
- **Click Yes**



EXERCISE – Structural Analysis

Mesh Module

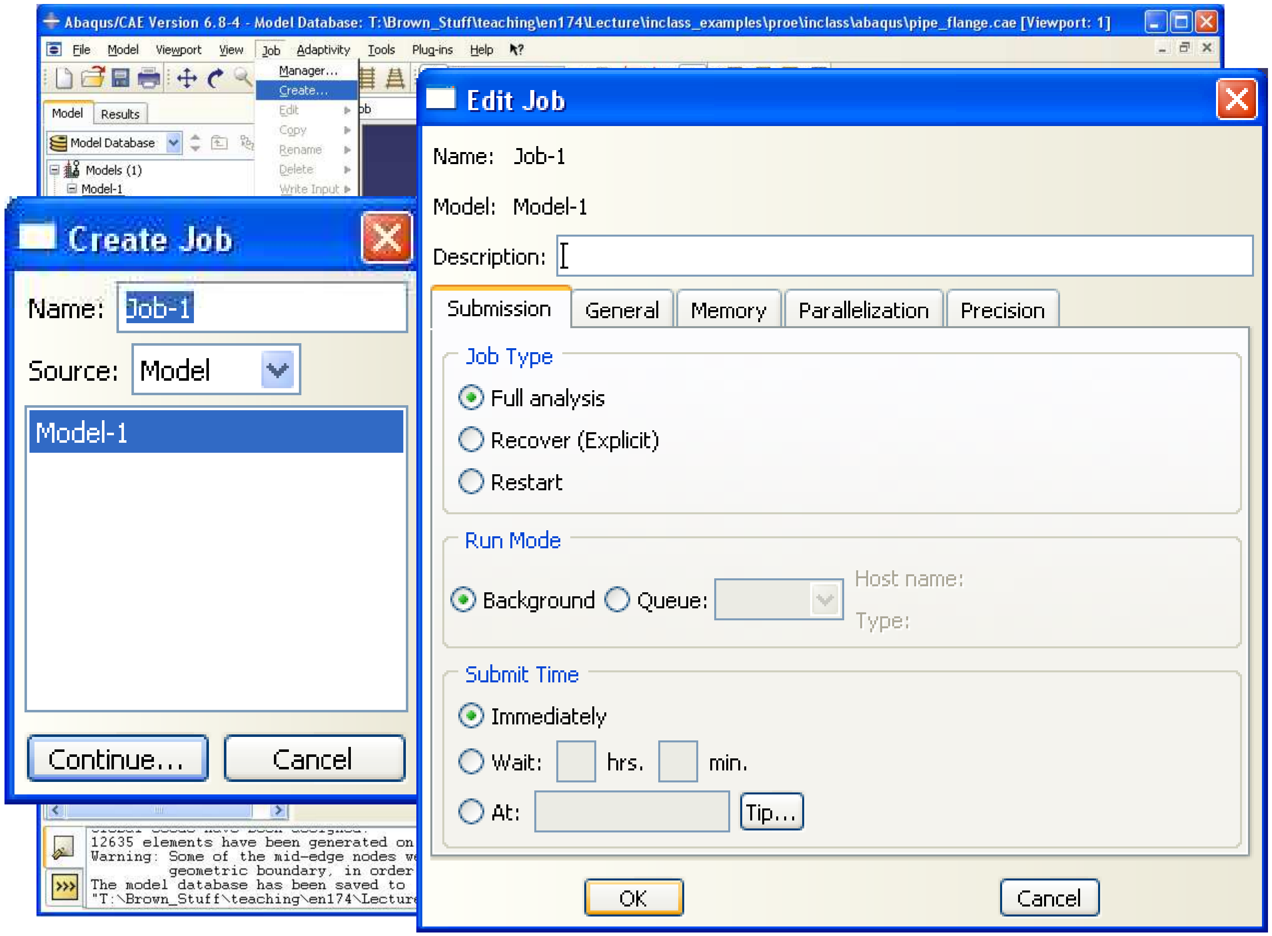
- **Take a look at the mesh we just created**
- **Save**



EXERCISE – Structural Analysis

Job Module

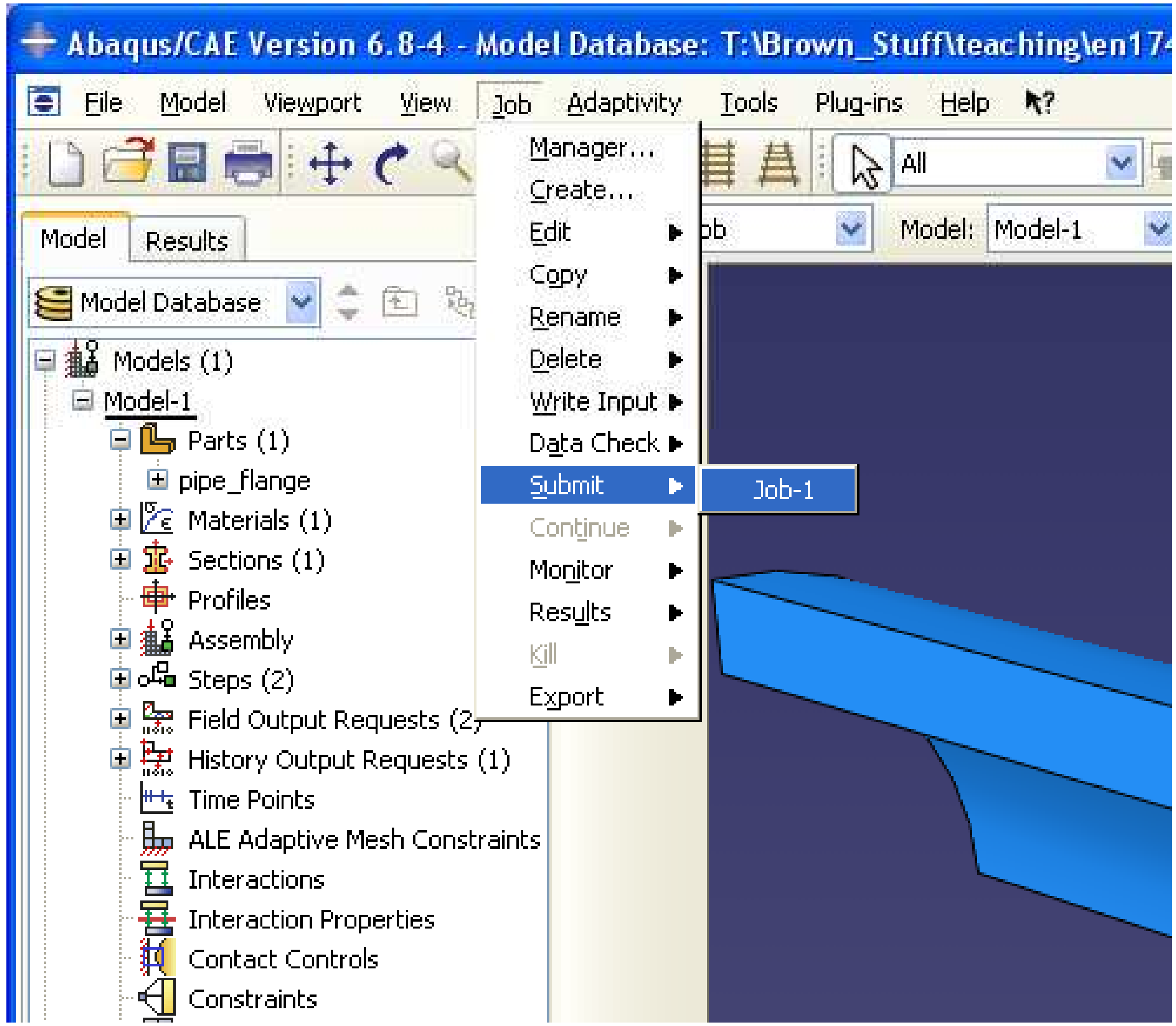
- **This module will create an input file and submit it to a solver**
- **Job > Create**
- **Click Continue**
- **Accept defaults and click OK**



EXERCISE – Structural Analysis

Job Module

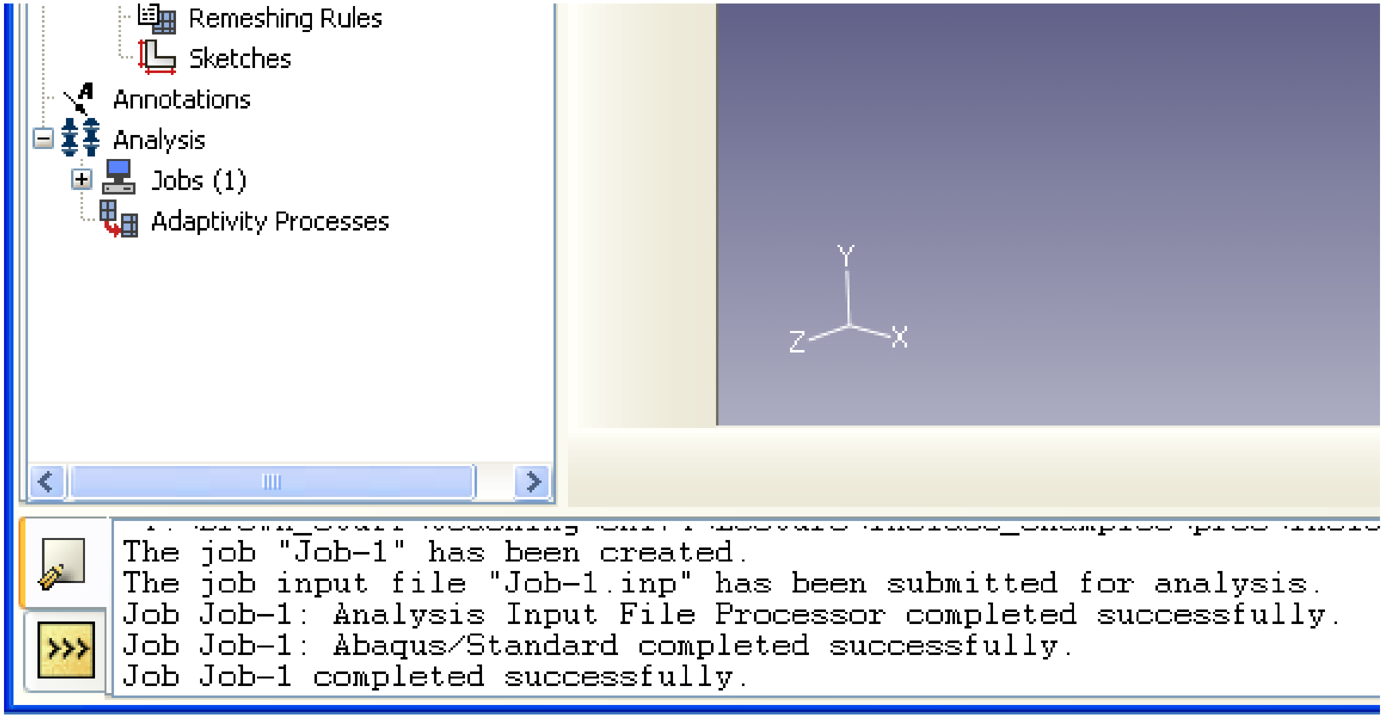
- **Solve the model**
- **Job > Submit > Job-1**



EXERCISE – Structural Analysis

Job Module

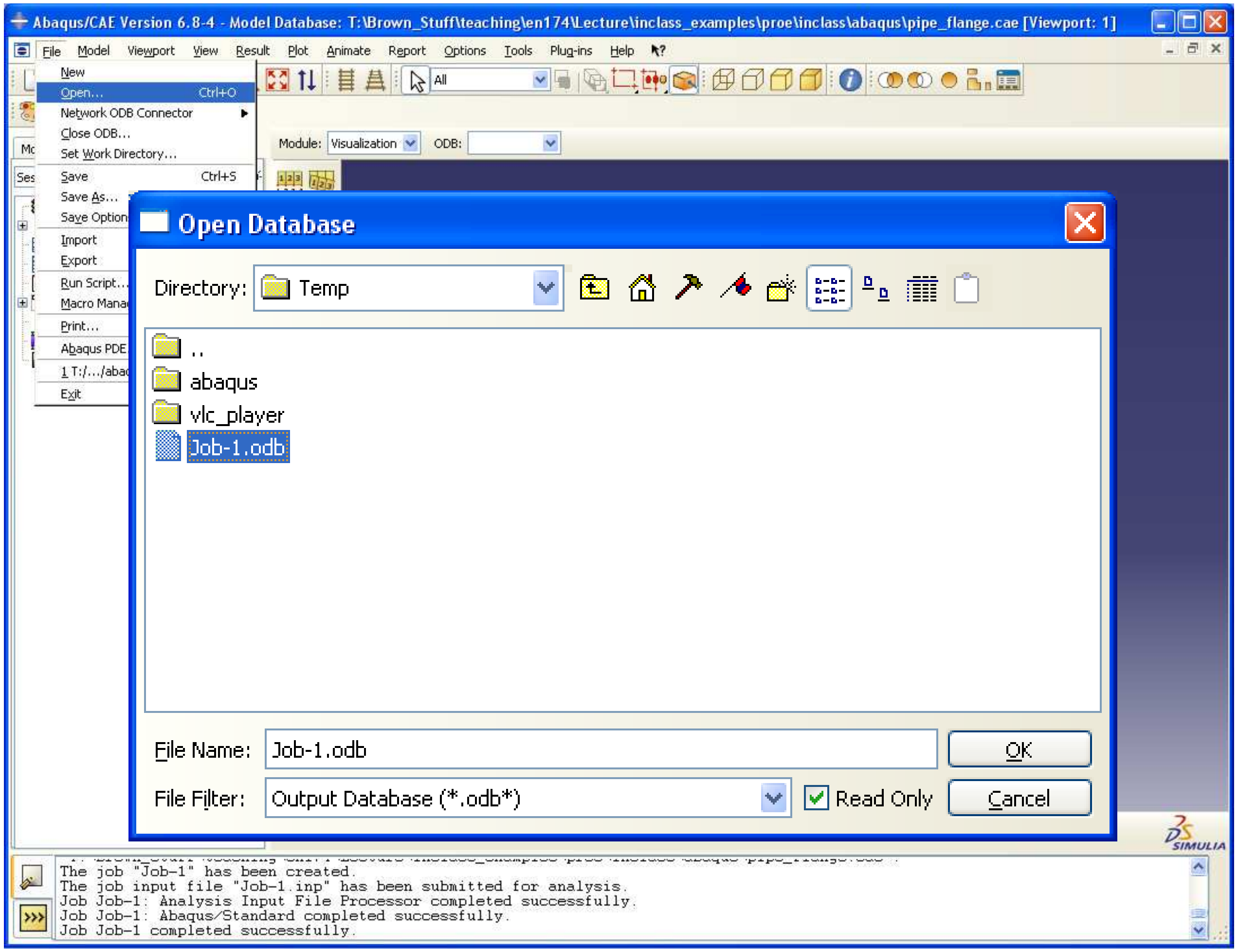
- **Hopefully you see this in the message frame**



EXERCISE – Structural Analysis

Visualization Module

- In this module you can visualize the results just computed
- File > Open
- Select Job-1.odb



EXERCISE – Structural Analysis

Visualization Module

- **Click the contour button to see the results**