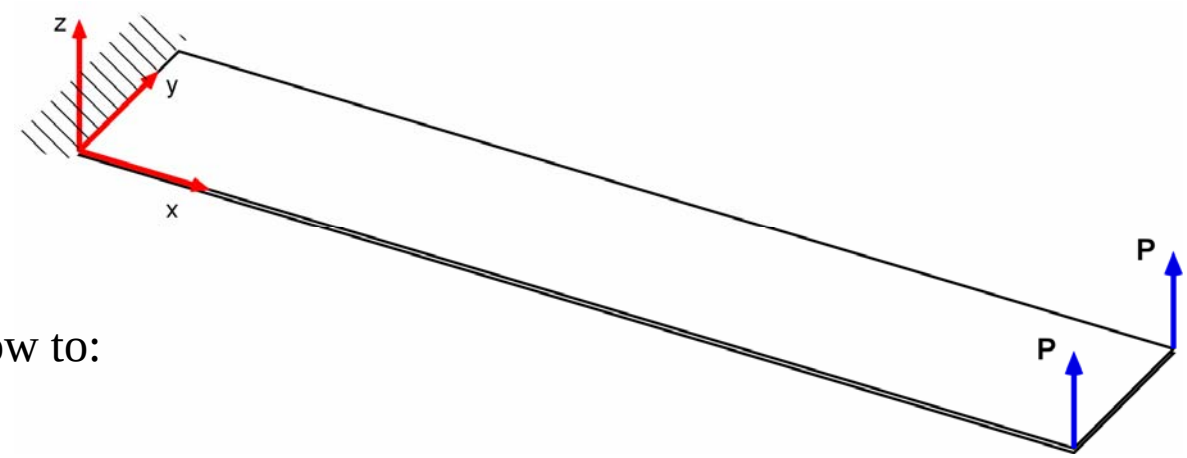
# ABAQUS/CAE Tutorial:
## Large Deformation Analysis of Beam-Plate in Bending

Hyonny Kim
September 28, 2004




In this tutorial, you'll learn how to:

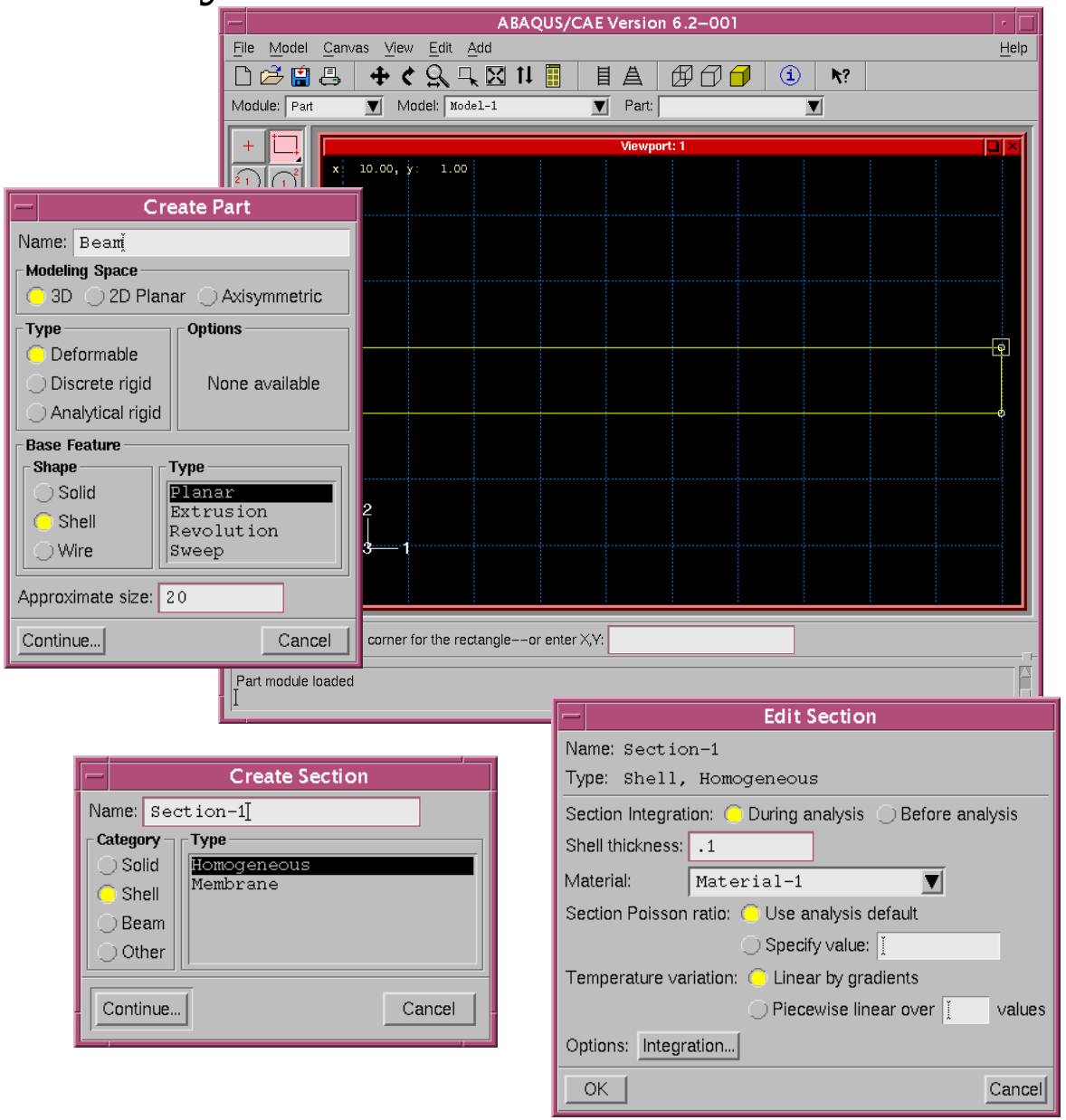
- Create a 3D model using shell elements.
- Conduct a geometrically nonlinear analysis (using *Nlgeom** option).
- Plot different FEA output with respect to each other (e.g., force vs. displacement).

Beam Specifications
- Dimensions: 10 x 1.0 x 0.1 in.
- Material: Steel
- Loading: P = 100 lb (200 total)

* Note: the use of the *Nlgeom* option can result in a considerable increase in computation cost. Use this option only when it is necessary. When using *Nlgeom*, it is generally advised to first run your model with *Nlgeom* turned off so as to "de-bug" any problems.

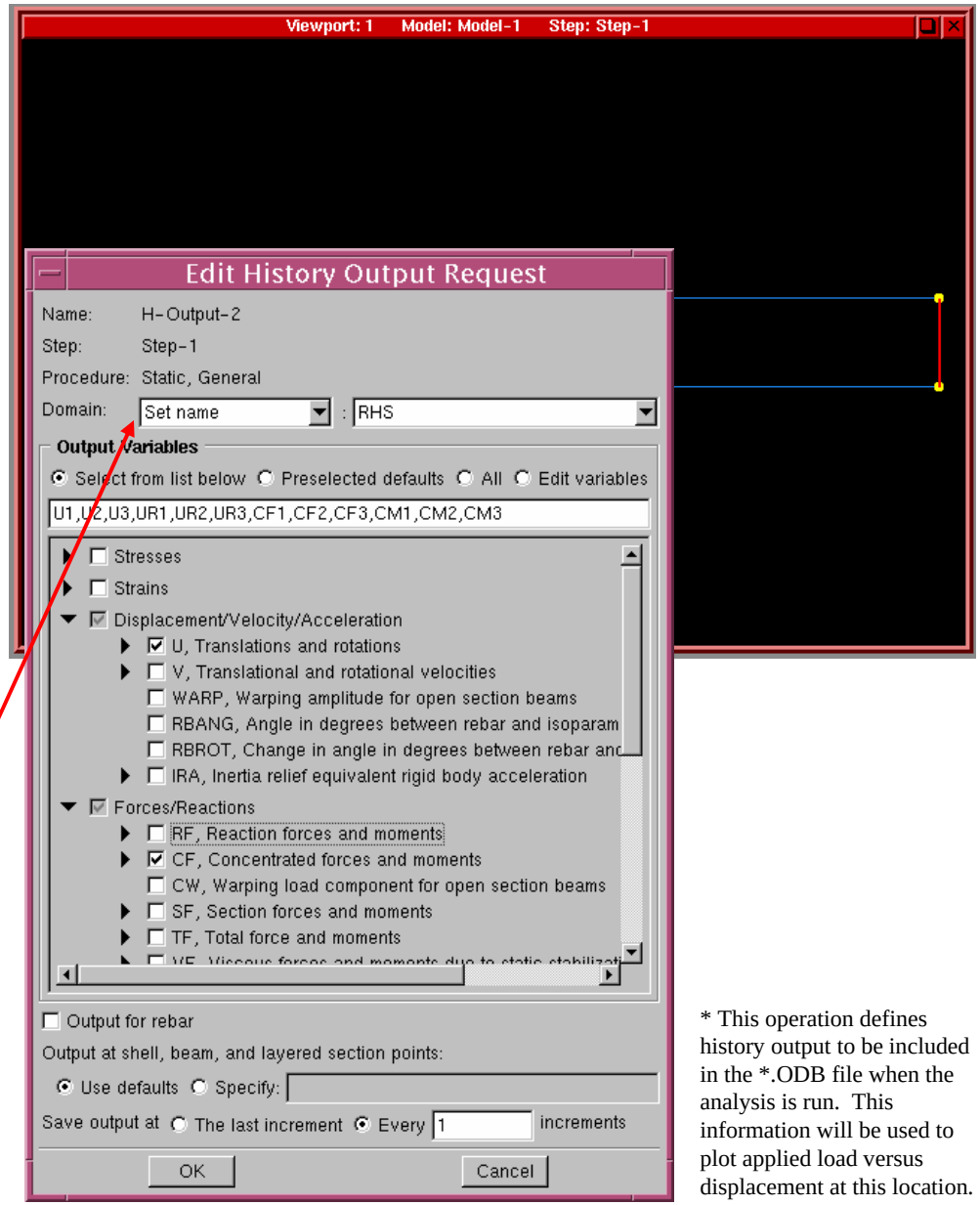
# Part, Property, and Assembly Modules

- Start ABAQUS/CAE
- Enter the **Part Module**
  - under **Create Part**,:
    - Enter a part name, e.g., *beam*
    - 3D
    - Deformable
    - Shell
    - Planar
    - Approximate Size 20
  - draw 10 in. x 1 in. rectangle by specifying corner coordinates (0,0) and (10,1).
- In the **Property Module**
  - specify steel: E = 30e6, ν = 0.3
  - when **Creating Section**, select **Shell,** click **Continue** then enter **Shell Thickness** value of 0.1
  - assign the section to the beam
- In the **Assembly Module**, instance the beam



# Step Module – Activate Nlgeom and Specify History Output

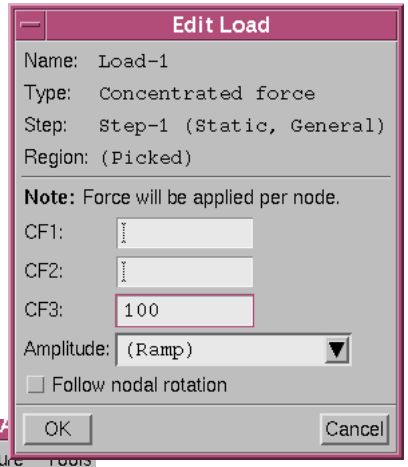
- Go to the **Step Module**
- Click **Create Step**
  - select *Static,General* then **Continue**
  - under the **Basic** tab, be sure **Time Period** of **1** is set, and that **Nlgeom** is on
- Go to Menu Bar -> Tools -> Set -> Create
  - enter a name, e.g., **RHS**, click **Continue**
  - pick the right hand side edge of the beam, it will turn red, click **Done**
- Click **Create History Output***
  - give name (use default), be sure **Step-1** is chosen, **Continue**
  - under the *Domain*, choose **Set name**, and be sure that ***RHS*** is chosen
  - from list below, activate:
    - *"U, Translations and rotations"*
    - *"CF, Concentrated forces and moments"*
  - click **OK**

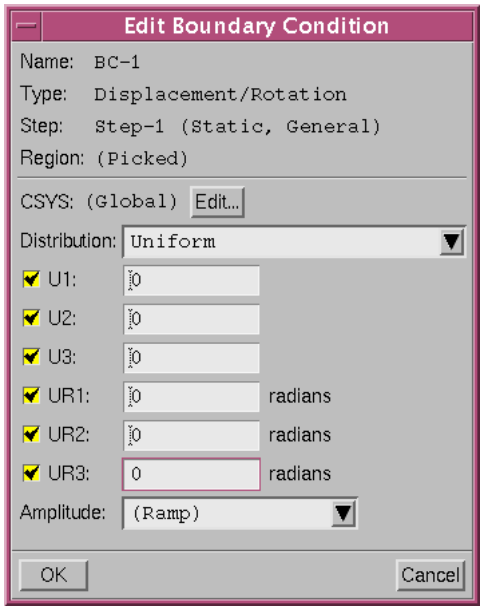


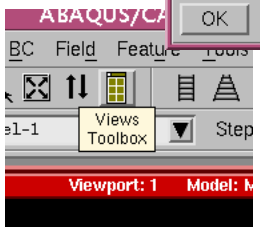

* This operation defines history output to be included in the *.ODB file when the analysis is run. This information will be used to plot applied load versus displacement at this location.

# Load & Mesh Module, Submit Job

- In the **Load Module**
  - apply **Concentrated Force** to the right hand corners of the beam.
    - enter a value of 100 in the CF3 box (forces act in 3-direction)
  - set all six d.o.f. at the left hand edge to be zero
  - view the beam in isometric view through the **Views Toolbox**

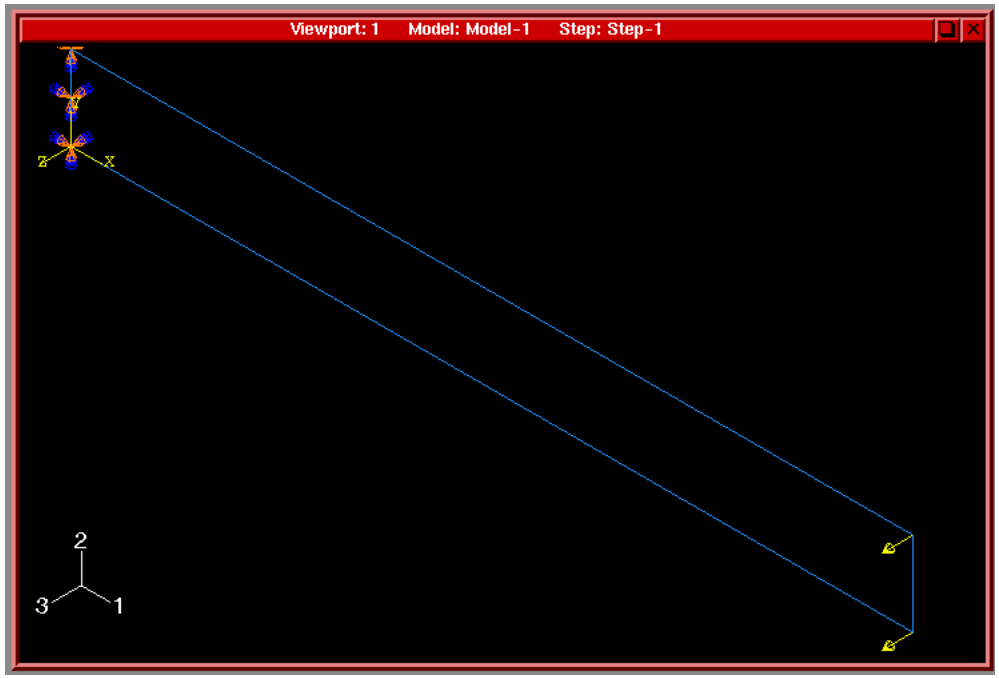



- In the **Mesh Module**
  - set a *Mesh Seed Size* of **1.0***
  - specify the use of quadratic, 6DOF per node shell elements, **S8R**
  - mesh the beam

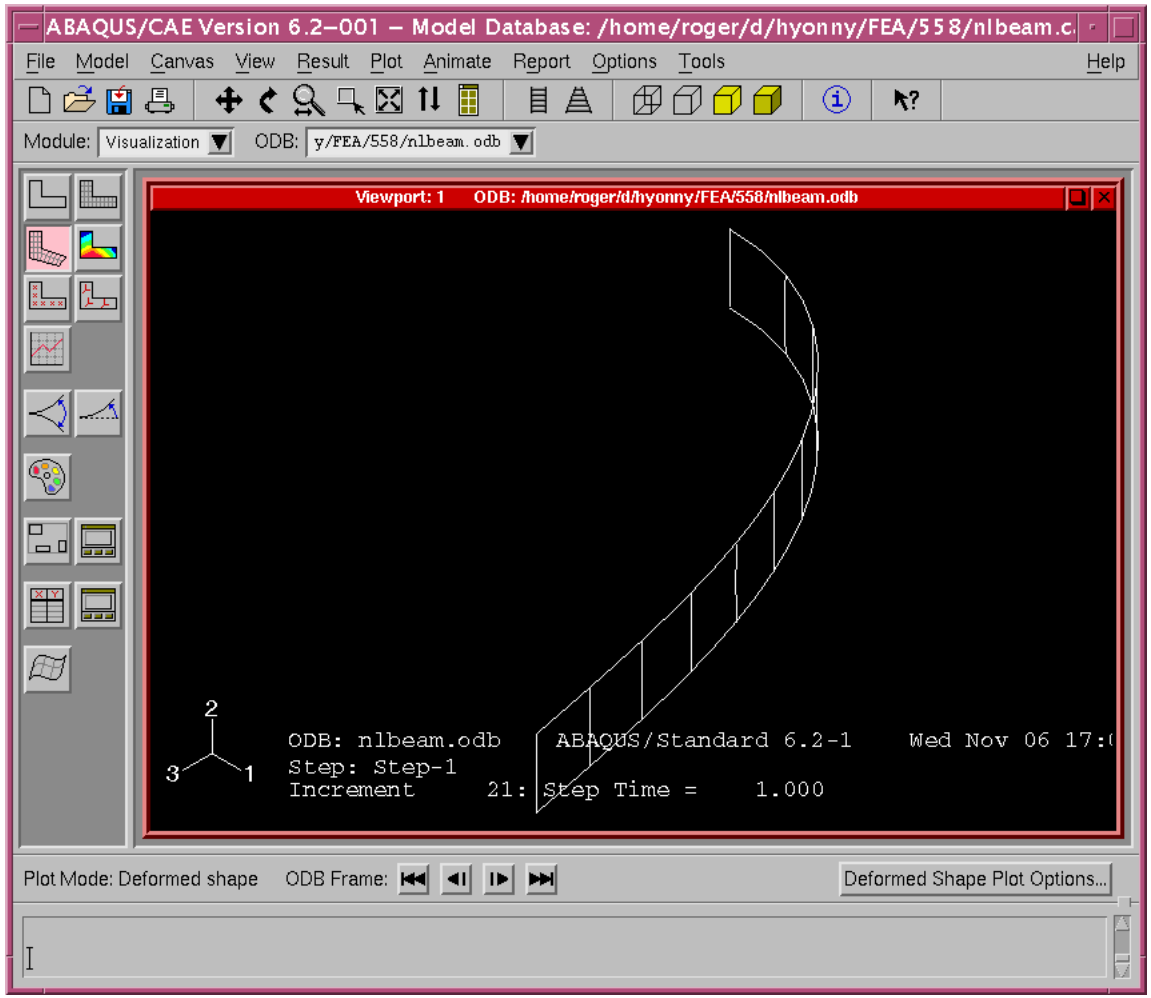
- In the **Job Module**
  - submit the job, click on **Results** or go to the **Visualization Module** when the analysis has completed



# Advanced Post-Processing – Load vs. Deflection

- In the **Visualization Module**, you should get the following displacement profile

- Note that when **Nlgeom** is turned on, the displacement scale factor during post-processing is set to a value of **1.0**, so the displacements shown on the screen are NOT exaggerated.



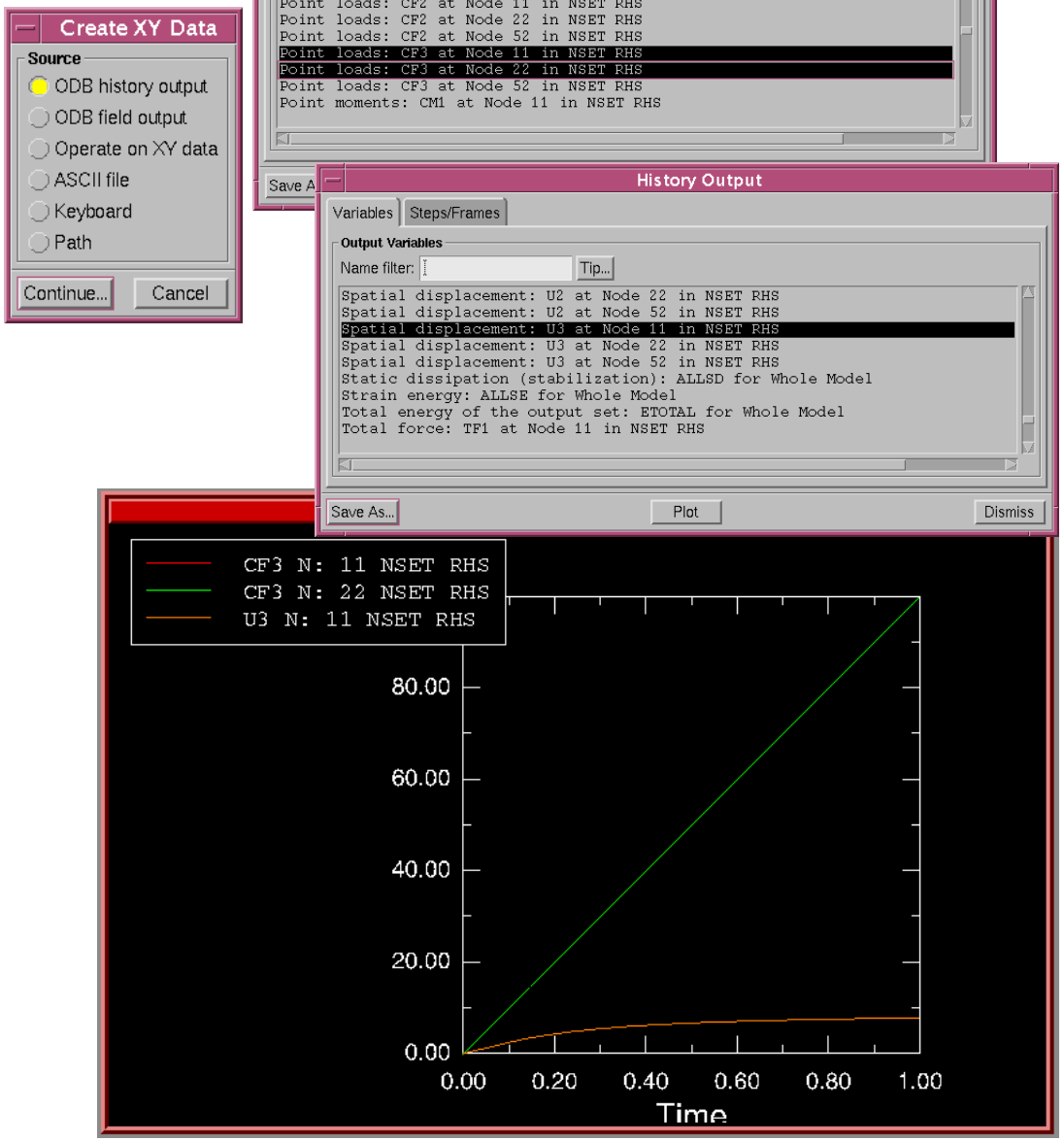
# Gathering History Output

- click the **Create XY Data** button 
- select **ODB history output**, **Continue**
- In the list that follows, select all three of the following so that they are highlighted (hold CTRL key):
  - Point Loads: CF3 at Node 11
  - Point Loads: CF3 at Node 22
  - Spatial displacement: U3 at Node 11

  Notes:

  1. you may have to resize the window to be able to read the entire line
  2. use SHIFT and CTRL keys while you click to select multiple items

  - click the **Plot** button, and you'll see the forces and displacements plotted versus a "Time" axis – this time axis is nondimensional, and is actually a scale factor of the loading (or prescribed displacement) such that at time of 1.0, the full load has been applied
  - click the **Save As** button and click **OK** to use default names, click **Dismiss**

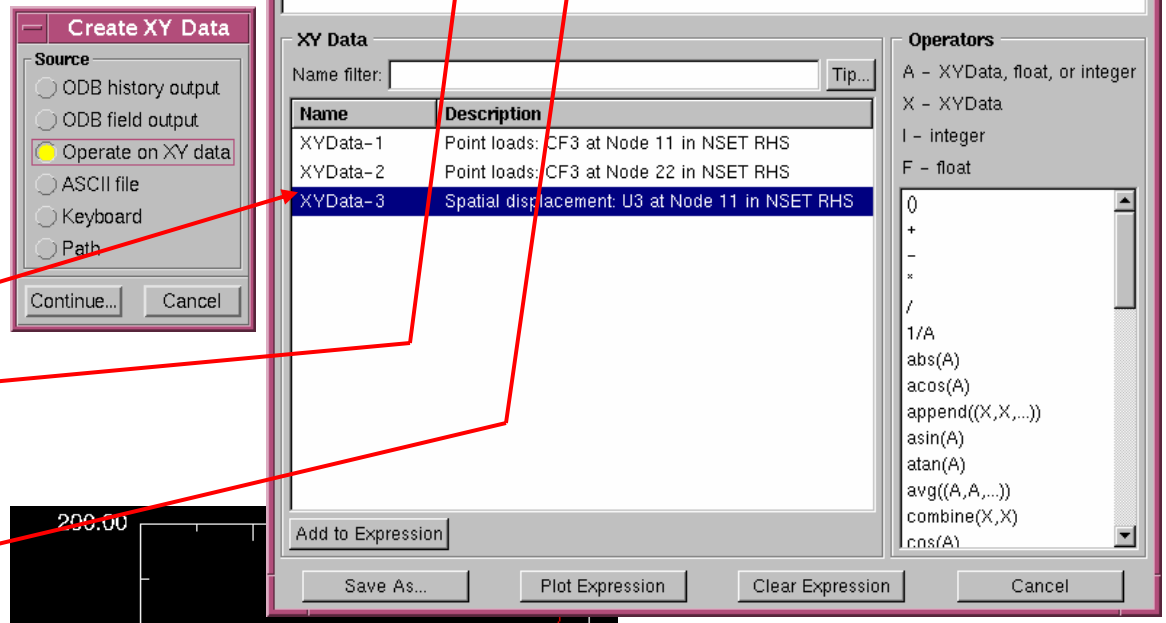


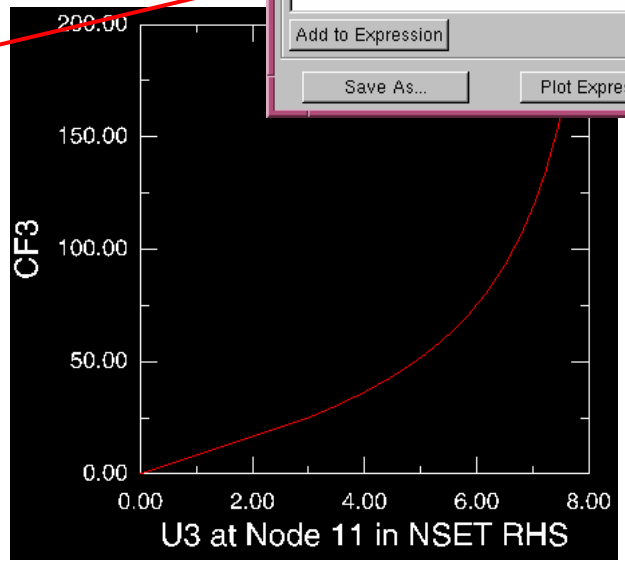
# Plot Load Versus Displacement at Beam Tip

- click the **Create XY Data** button 
- select **Operate on XY data, Continue**
- Do the following:

1. type in at the prompt: *combine(*
2. click **XYData-3** (U3 at Node 11) in the **XY Data** list
3. type in a comma: **,**
4. click **XYData-1** (CF3 at Node 11) in the **XY Data** list
5. type in a plus symbol: +
6. click **XYData-2** (CF3 at Node 22) in the **XY Data** list
7. type a close parenthesis: )
8. click **Plot Expression** at the bottom of the window

Note: **Combine** is one of many **Operators** that are available in the list on the right side of the window. You can click or type any of these commands in.




Observe that the plot of load vs. displacement is ***nonlinear***. This is due to the breakdown of linear beam theory for very large displacements; recall the beam length is only 10 in., displacement is nearly 8.0 in.

Try re-running the analysis with the **Nlgeom** option turned of (in the **Step Module**) and observing the results. You should find that the load vs. displacement behaves linearly, and in fact will be very closely predicted by the formula $\delta = PL^3/3EI$