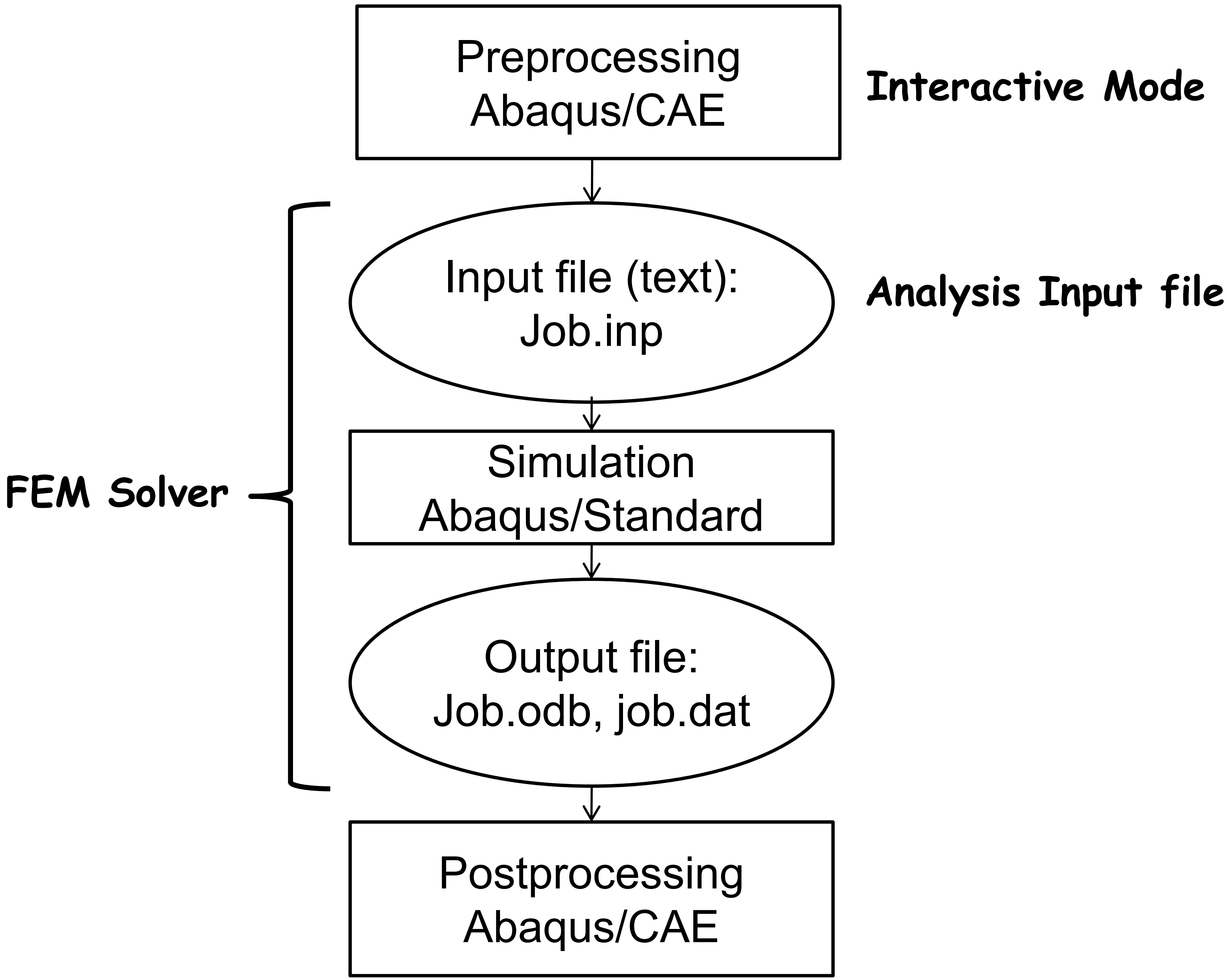
Finite Element Analysis

Using Abaqus

Instructor: Nam-Ho Kim (nkim@ufl.edu)

Abaqus Basics



Methods of Analysis in ABAQUS

- Interactive mode
  - Create an FE model and analysis using GUI
  - Advantage: Automatic discretization and no need to remember commands
  - Disadvantage: No automatic procedures for changing model or parameters

- Python script
  - All GUI user actions will be saved as Python script
  - Advantage: Users can repeat the same command procedure
  - Disadvantage: Need to learn Python script language

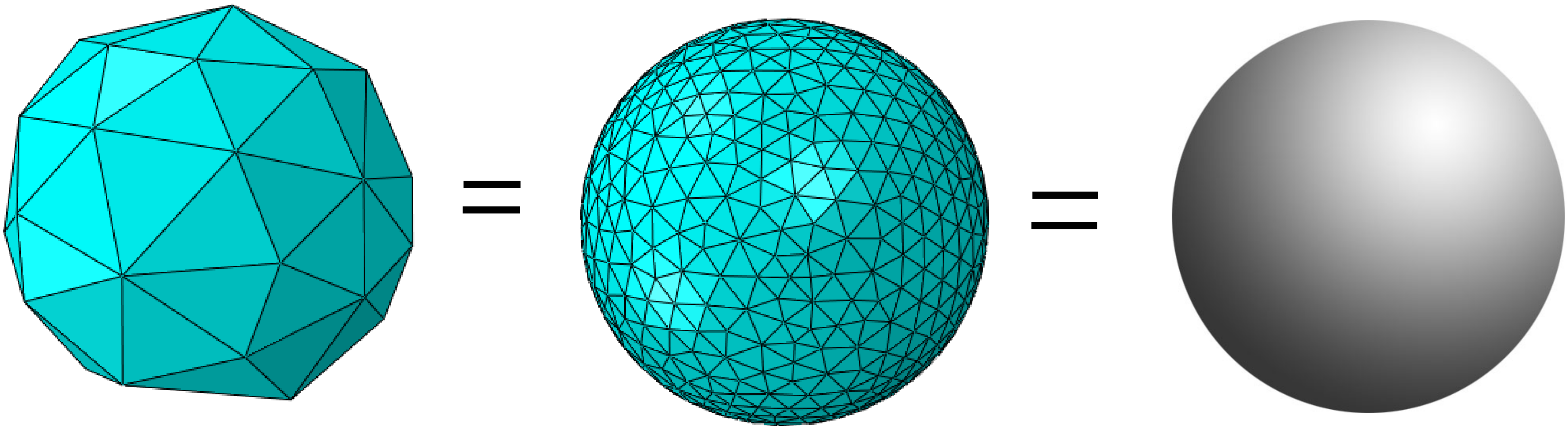
Methods of Analysis in ABAQUS

- Analysis input file
  - ABAQUS solver reads an analysis input file
  - Possible to manually create an analysis input file
  - Advantage: Users can change model directly without GUI
  - Disadvantage: Users have to discretize model and learn ABAQUS input file grammar

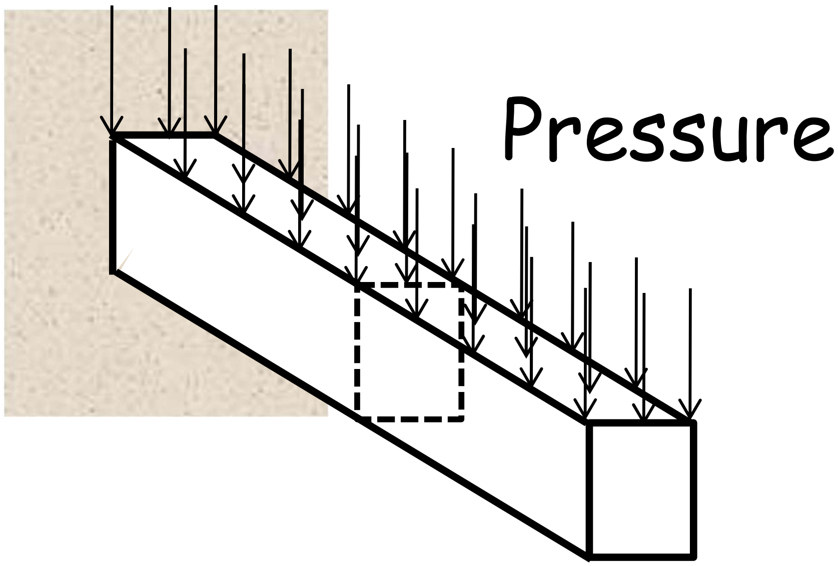
Components in ABAQUS Model

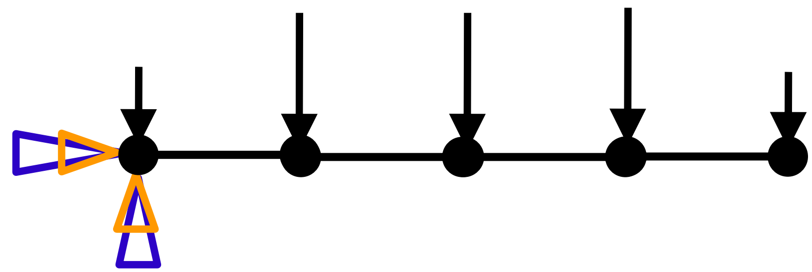
- Geometry modeling (define geometry)
- Creating nodes and elements (discretization)
- Element section properties (area, moment of inertia, etc)
- Material data (linear/nonlinear, elastic/plastic, isotropic/orthotropic, etc)
- Loads and boundary conditions (nodal force, pressure, gravity, fixed displacement, joint, relation, etc)
- Analysis type (linear/nonlinear, static/dynamic, etc)
- Output requests

FEM Modeling

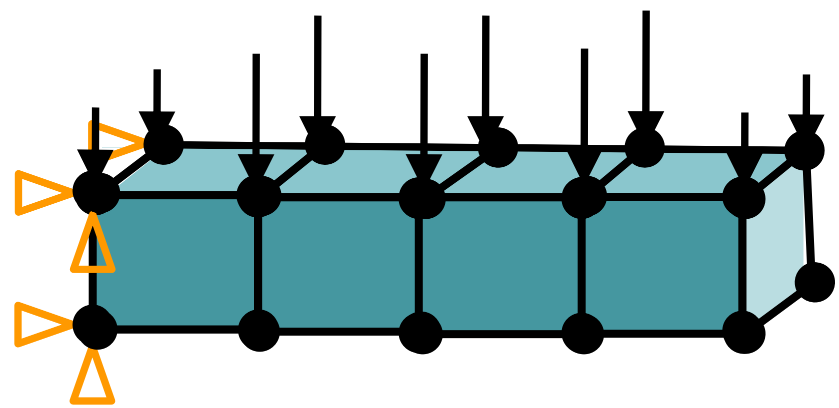


FEM Modeling





Beam element



Solid element

- Which analysis type?
- Which element type?
  - Section properties
  - Material properties
  - Loads and boundary conditions
  - Output requests

FEM Modeling

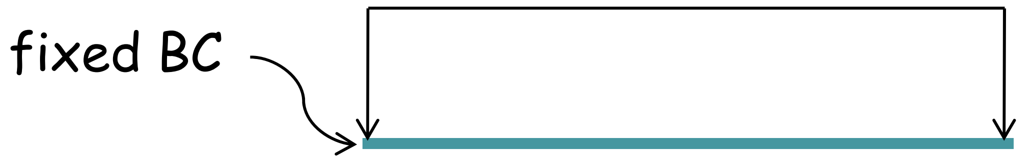
Line (Beam or truss element)

- Assign section properties (area, moment of inertia)
- Assign material properties

Volume (Solid element)

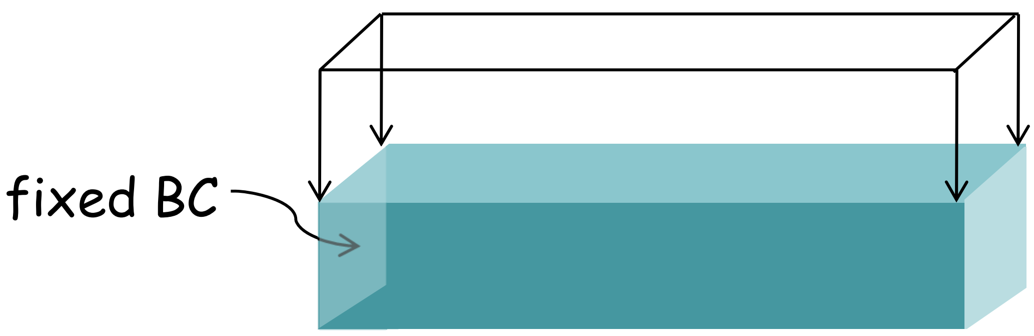
- Assign section properties
- Assign material properties

FEM Modeling



Line (Beam element)

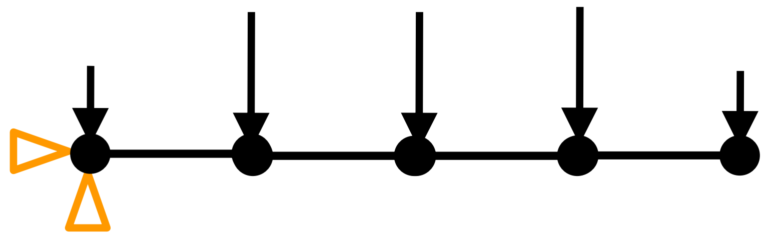
- Apply distributed load "on the line"
- Apply fixed BC "at the point"



Volume (Solid element)

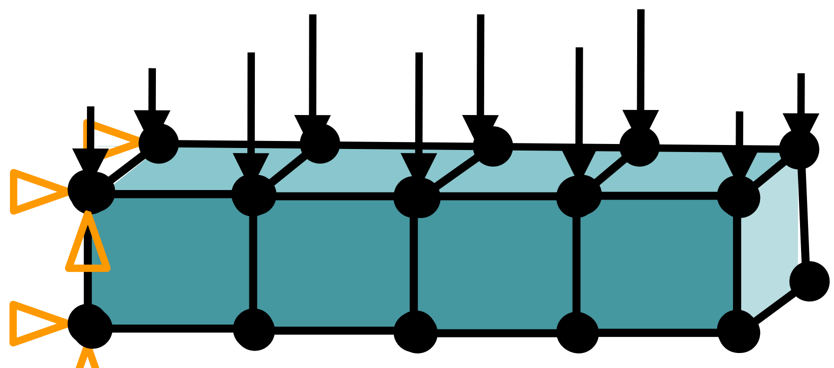
- Apply distribution load "on the surface"
- Apply fixed BC "on the surface"

FEM Modeling



Line (Beam or truss element)

- Discretized geometry with beam element
- Discretized BC and load on nodes

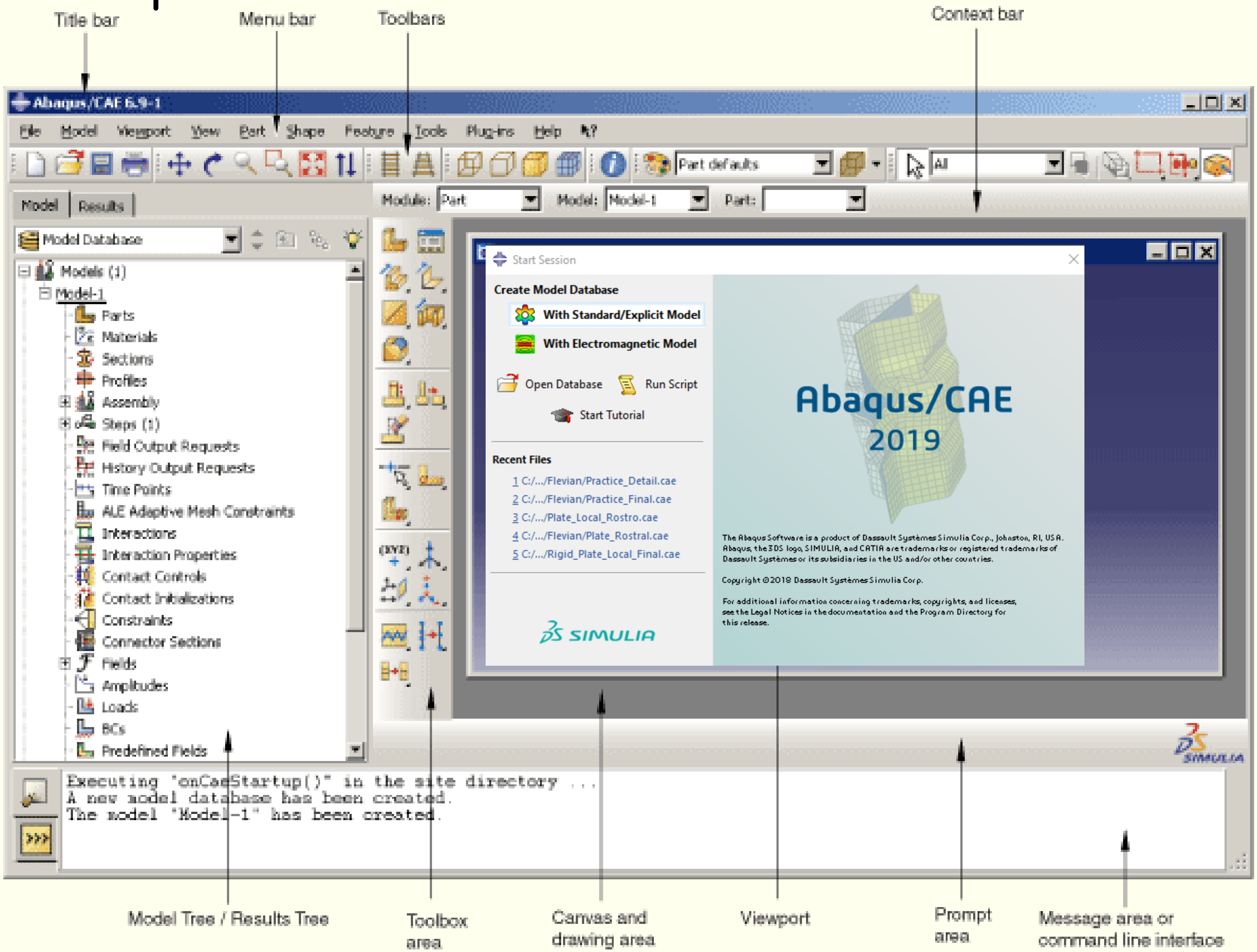


Volume (Solid element)

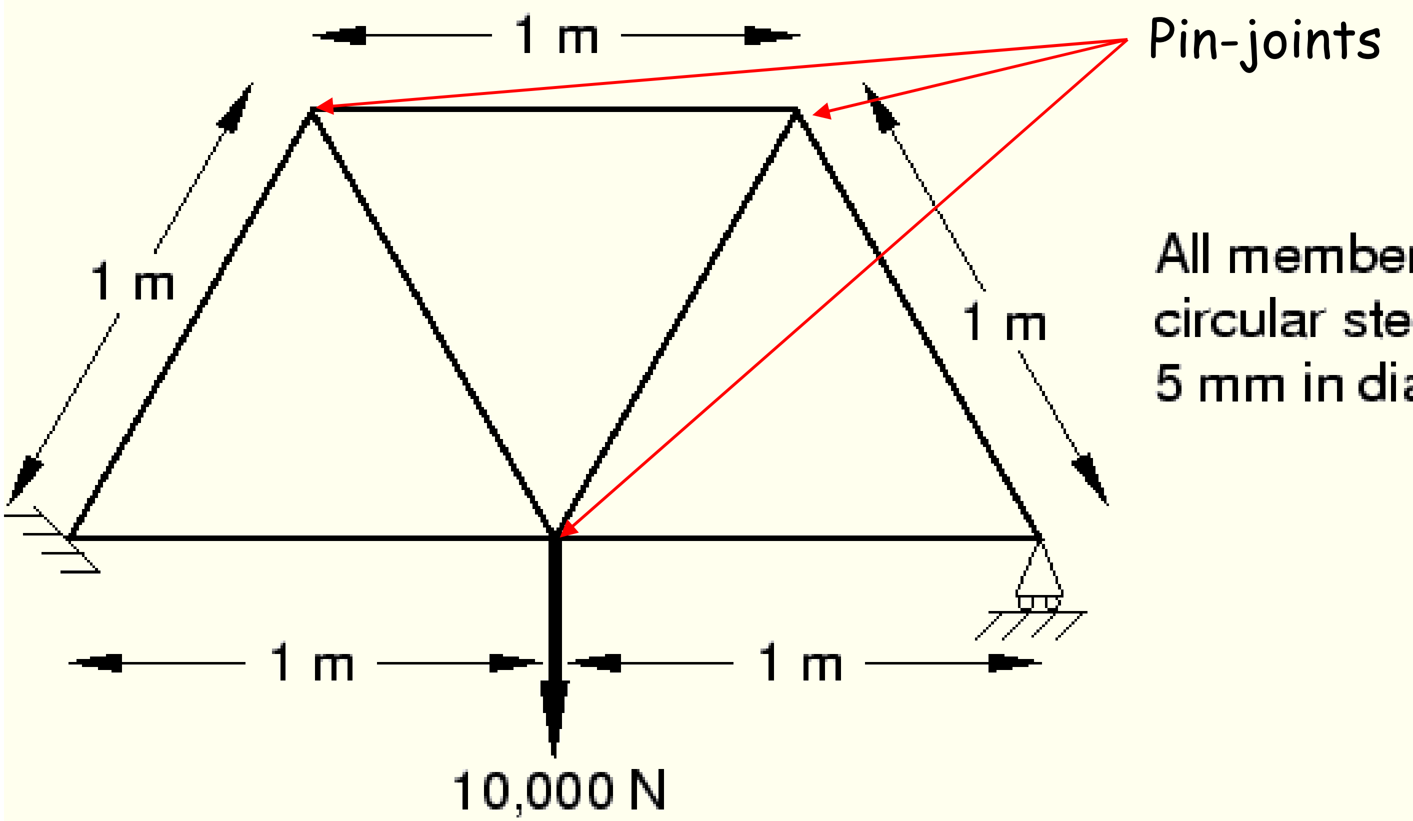
- Discretized geometry with solid element
- Discretized BC and load on nodes

Start Abaqus/CAE

- Startup window



Example: Overhead Hoist



All members are circular steel rods, 5 mm in diameter.

Material properties

General properties:

$$\rho = 7800 \text{ kg/m}^3$$

Elastic properties:

$$E = 200 \times 10^9 \text{ Pa}$$

$$\nu = 0.3$$

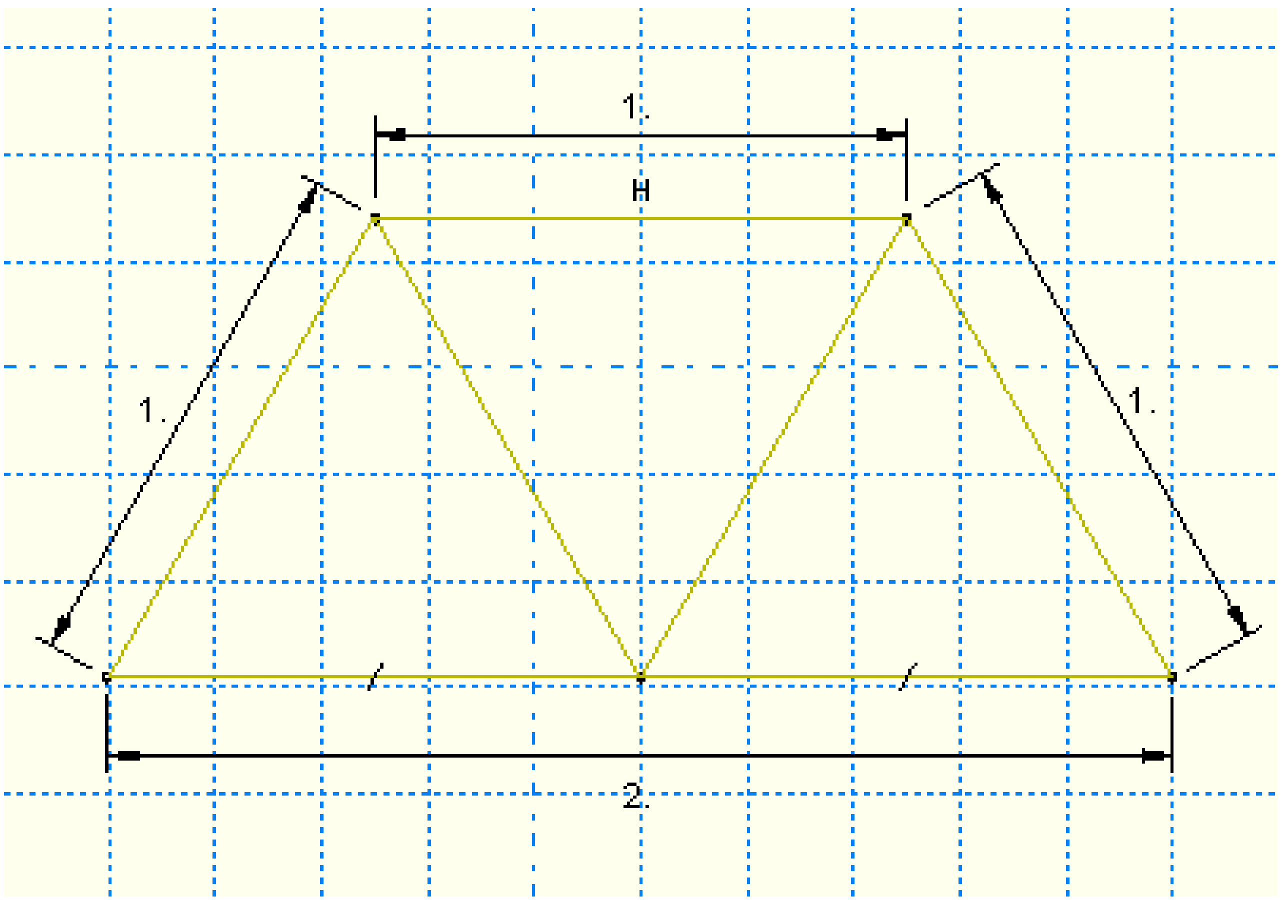
Units

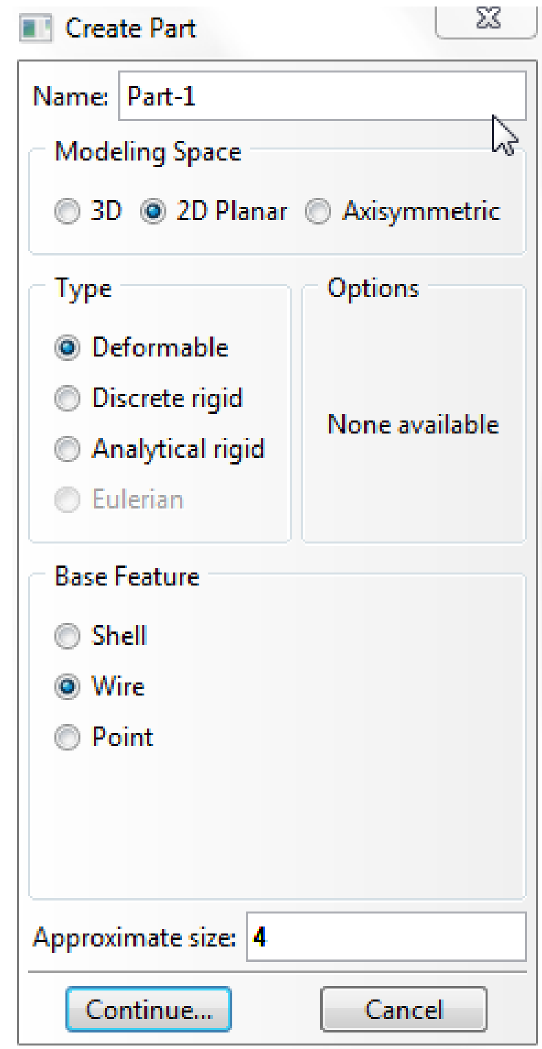
Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne (10^3 kg)	slug	lbf s^2/in
Time	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3} J)	ft lbf	in lbf
Density	kg/m^3	$tonne/mm^3$	$slug/ft^3$	lbf s^2/in^4

- Abaqus does not have built-in units
- Users must use consistent units

Create Part

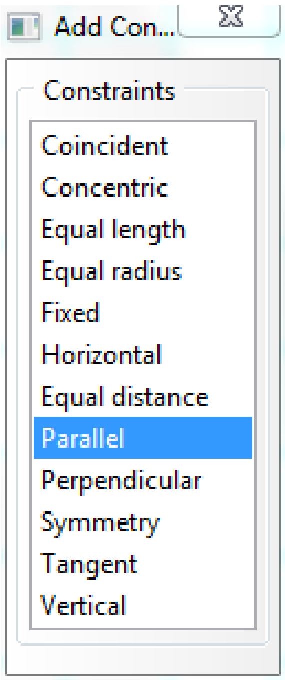
- Parts
  - Create 2D Planar, Deformable, Wire, Approx size = 4.0
  - Provide complete constrains and dimensions
  - Merge duplicate points

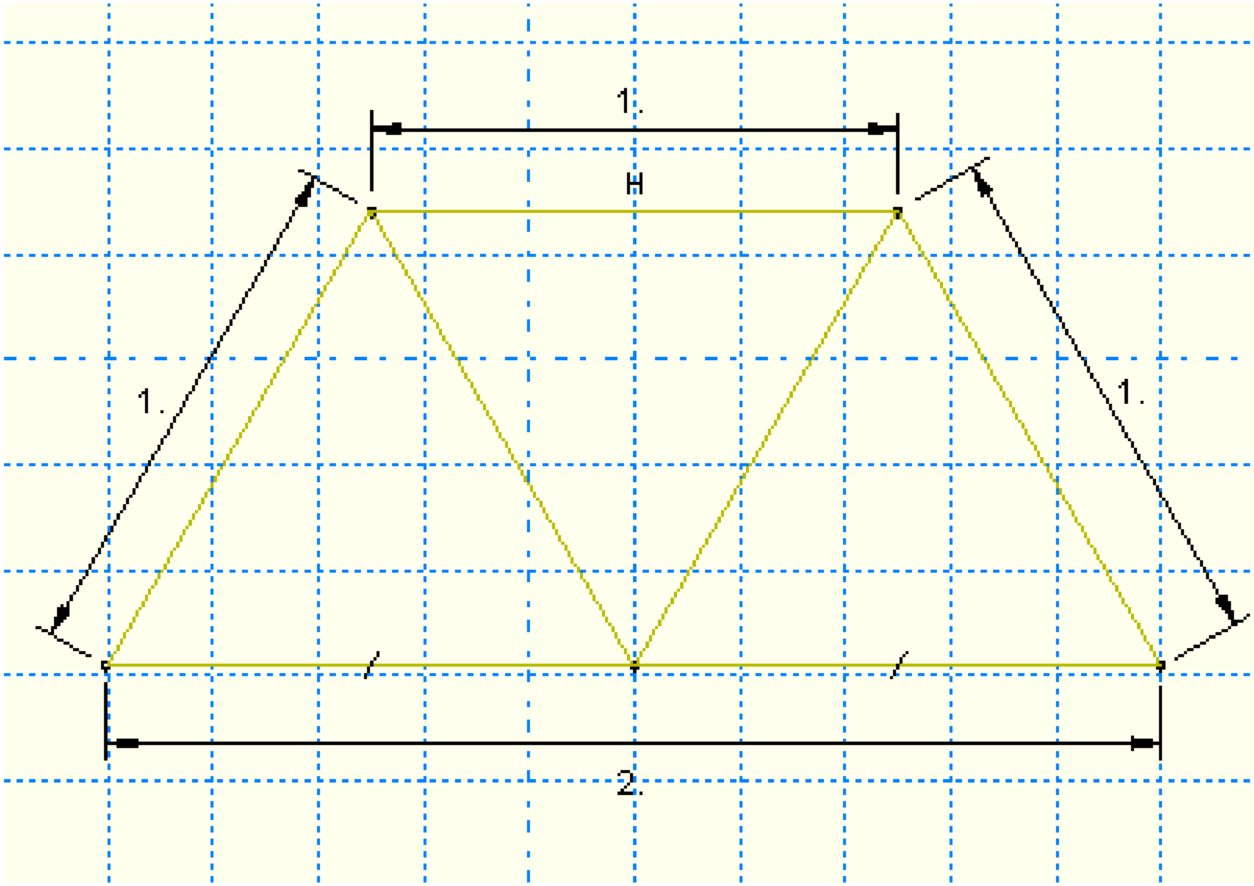




Geometry Constraint

- Define exact geometry
  - Add constraints



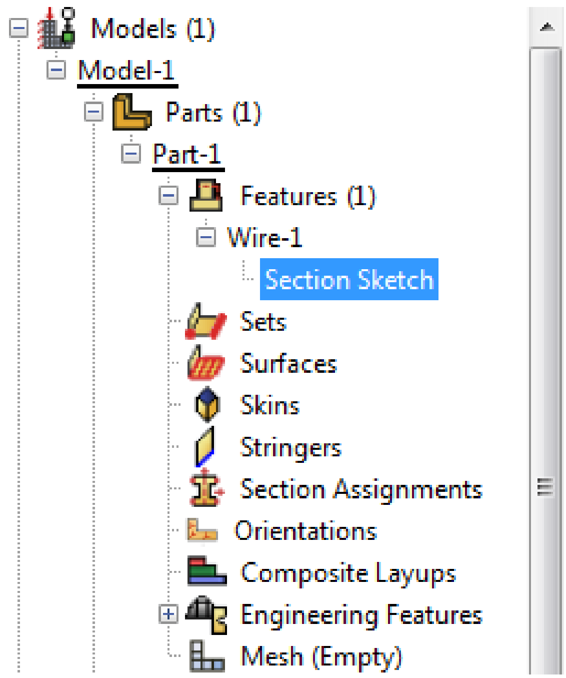


  - Add dimension
  - Over constraint warning

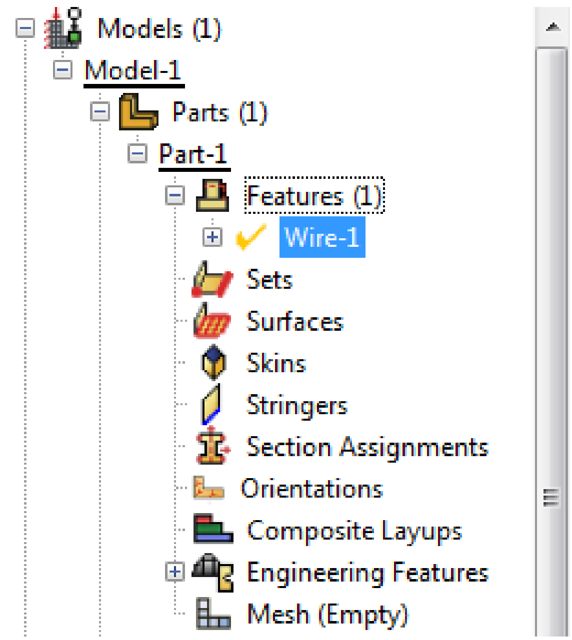
Geometry Modification

- Modify geometry modeling

1. Go back to the sketch

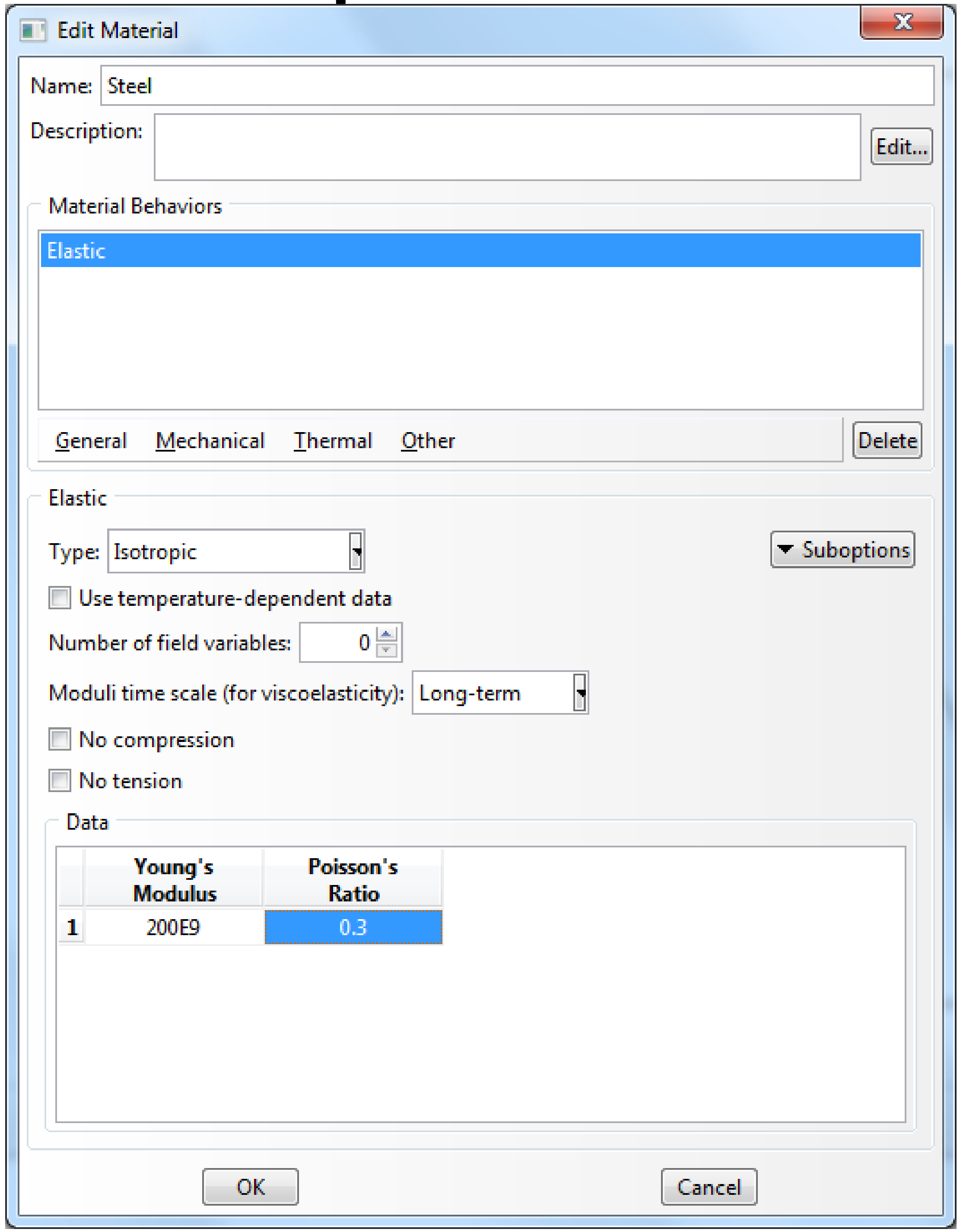


2. Update geometry



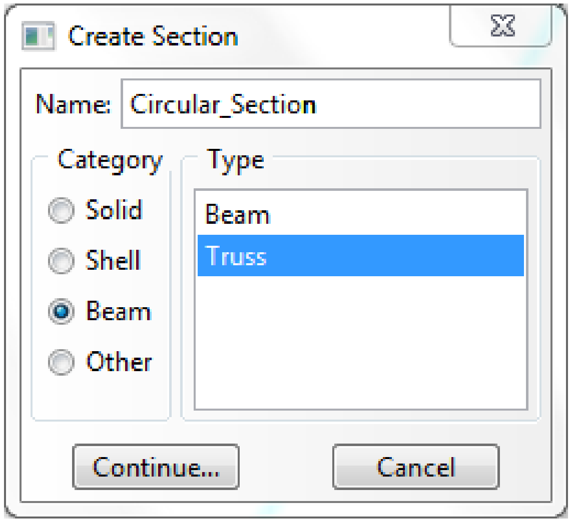
Define Material Properties

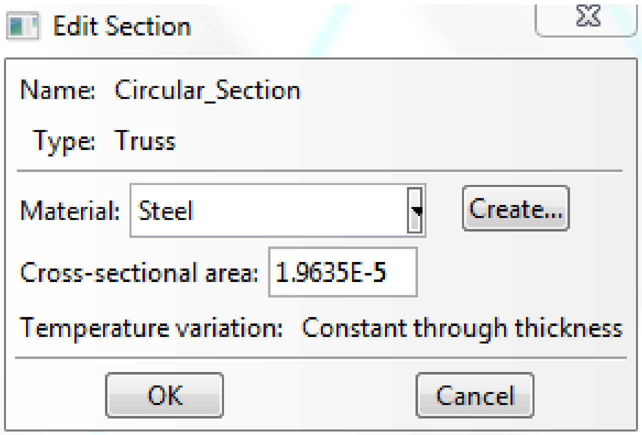
- Materials
  - Name: Steel
  - Mechanical
    Elasticity
    Elastic



Define Section Properties

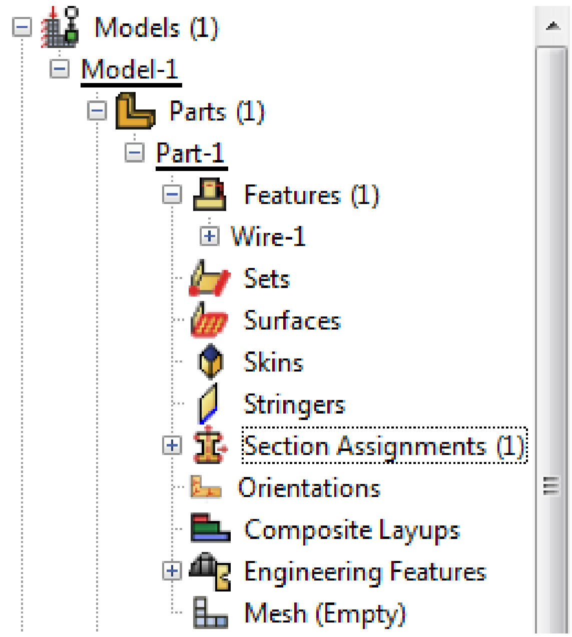
- Calculate cross-sectional area (diameter = 5mm)
- Sections
  - Name: Circular_Section
  - Beam, Truss
  - Choose material (Steel)
  - Write area



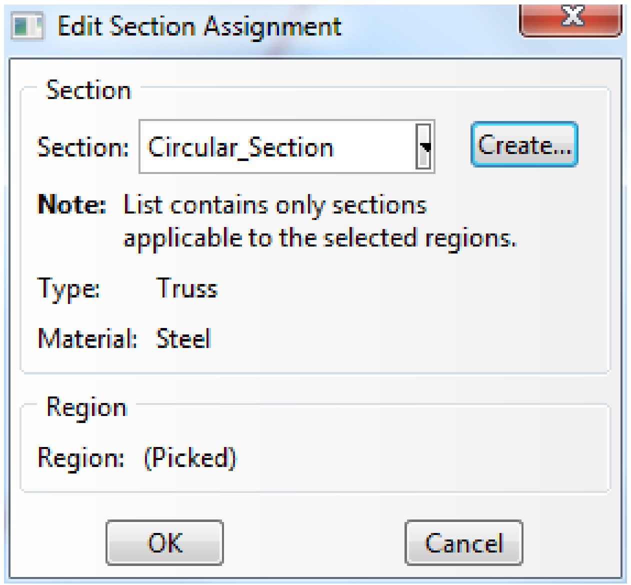


Define Section Properties

- Assign the section to the part
  - Section Assignments

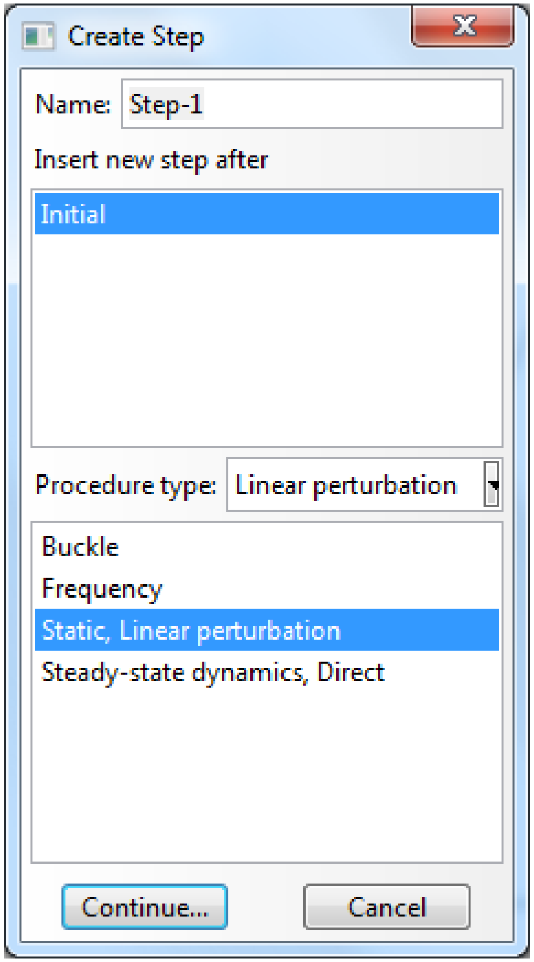


  - Select all wires
  - Assign Circular_Section



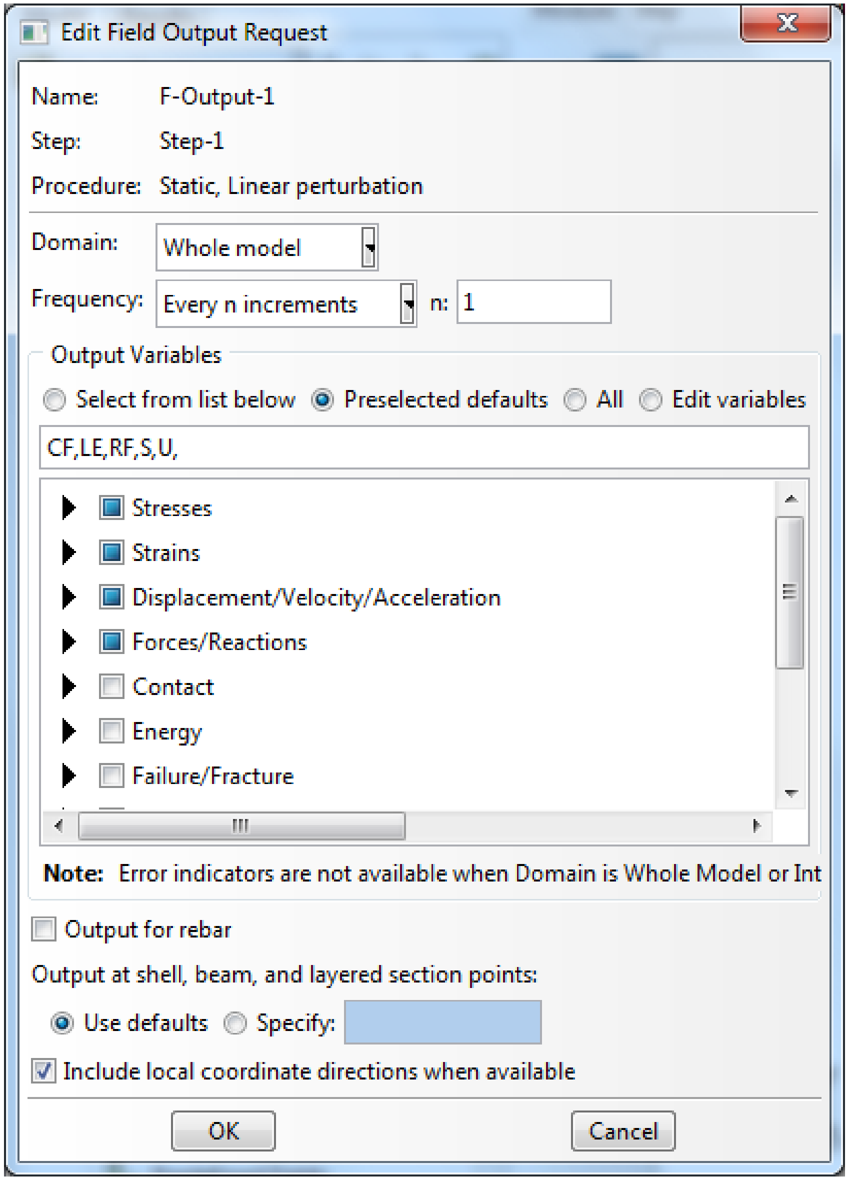
Assembly and Analysis Step

- Different parts can be assembled in a model
- Single assembly per model
- Assembly
  - Instances: Choose the frame wireframe
- Analysis Step
  - Configuring analysis procedure
- Steps
  - Name: Apply Load
  - Type: Linear perturbation
  - Choose Static, Linear perturbation



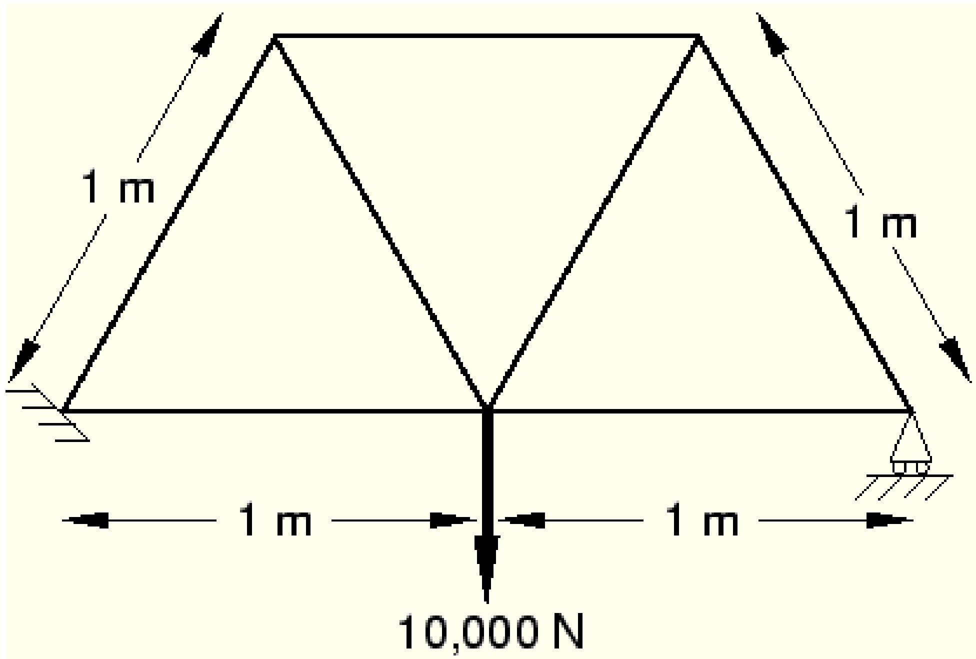
Assembly and Analysis Step

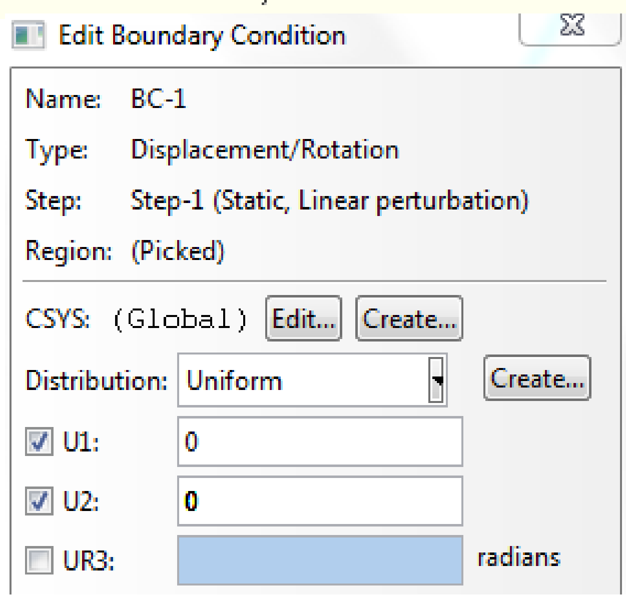
- Examine Field Output Request (automatically requested)
- User can change the request



Boundary Conditions

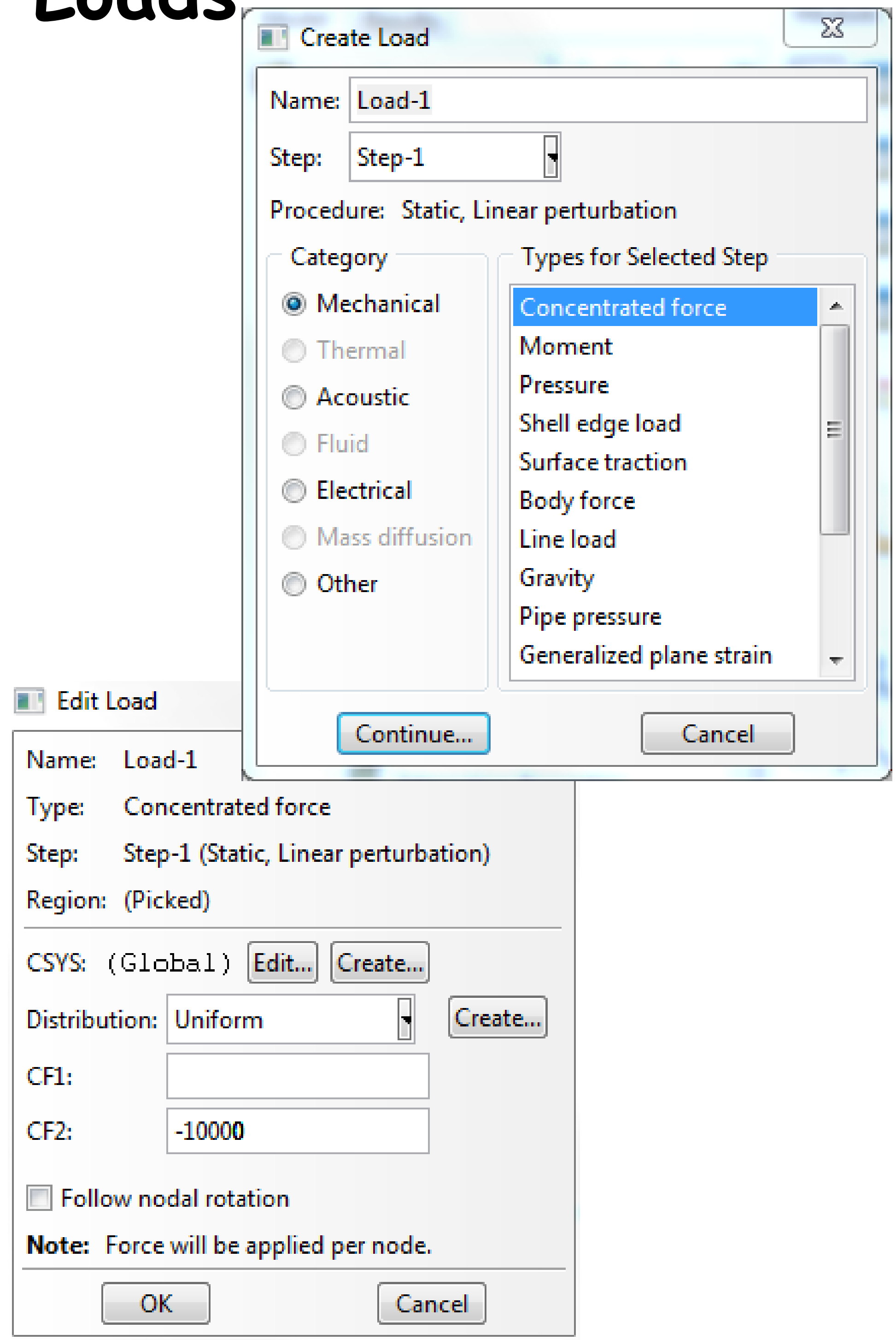
- Boundary conditions: Displacements or rotations are known
- BCs
  - Name: Fixed
  - Step: Initial
  - Category: Mechanical
  - Type: Displacement/Rotation
  - Choose lower-left point
  - Select U1 and U2
- Repeat for lower-right corner
  - Fix U2 only



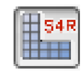


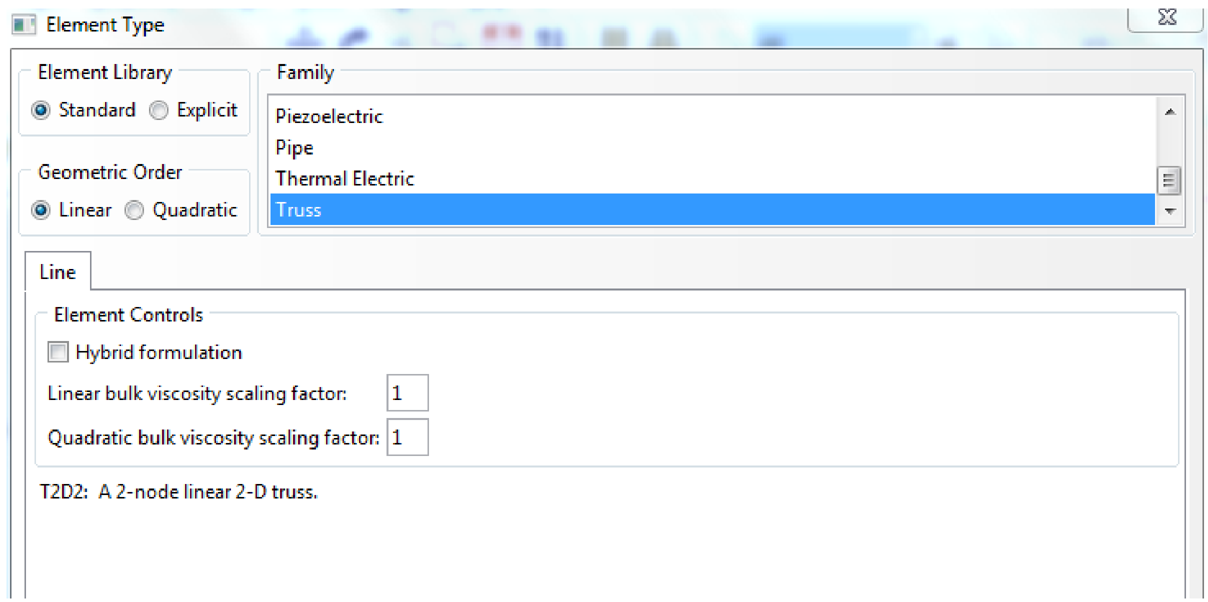
Applied Loads

- Loads
  - Name: Force
  - Step: Applied Load
  - Category: Mechanical
  - Type: Concentrated force
- Choose lower-center point
- CF2 = -10000.0



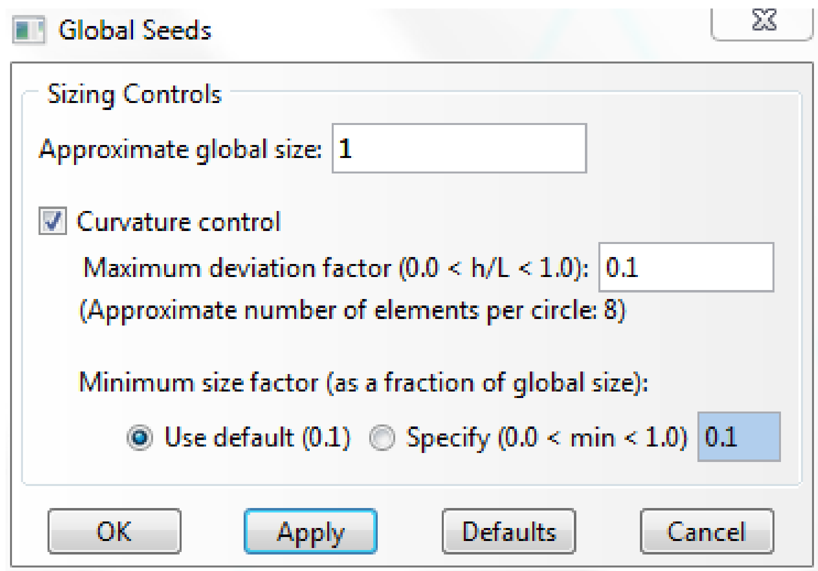
Meshing the Model

- Parts
  - Part-1, Mesh
- Menu Mesh, Element Types (side menu )
- Select all wireframes
- Library: Standard
- Order: Linear
- Family: Truss
- T2D2: 2-node linear 2-D truss

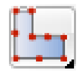
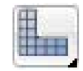


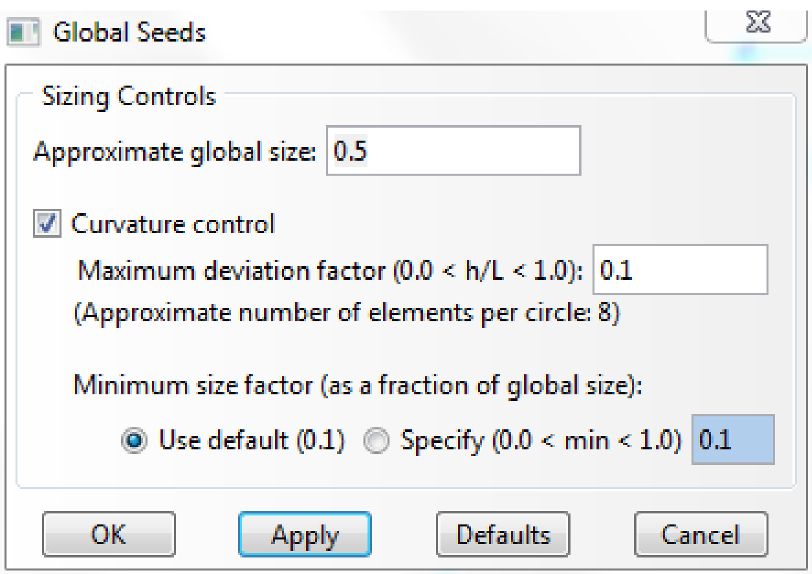
Meshing the Model

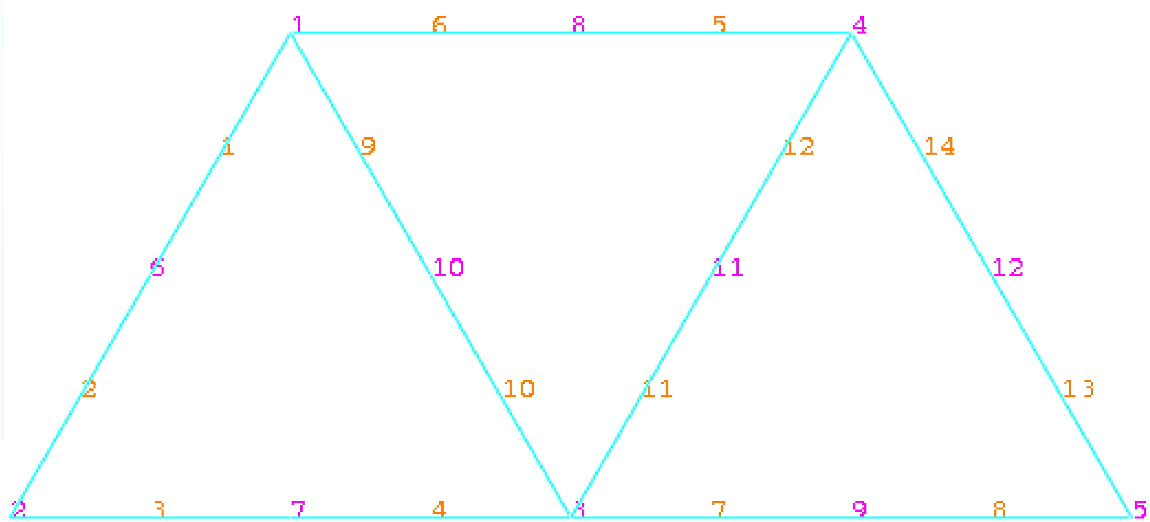
- Seed a mesh
  - Control how to mesh (element size, etc)
- Menu Seed, Part (side menu)
  - Global size = 1.0
- Menu Mesh, Part, Yes (side menu)
- Menu View, Part Display Option
  - Label on



Mesh Modification

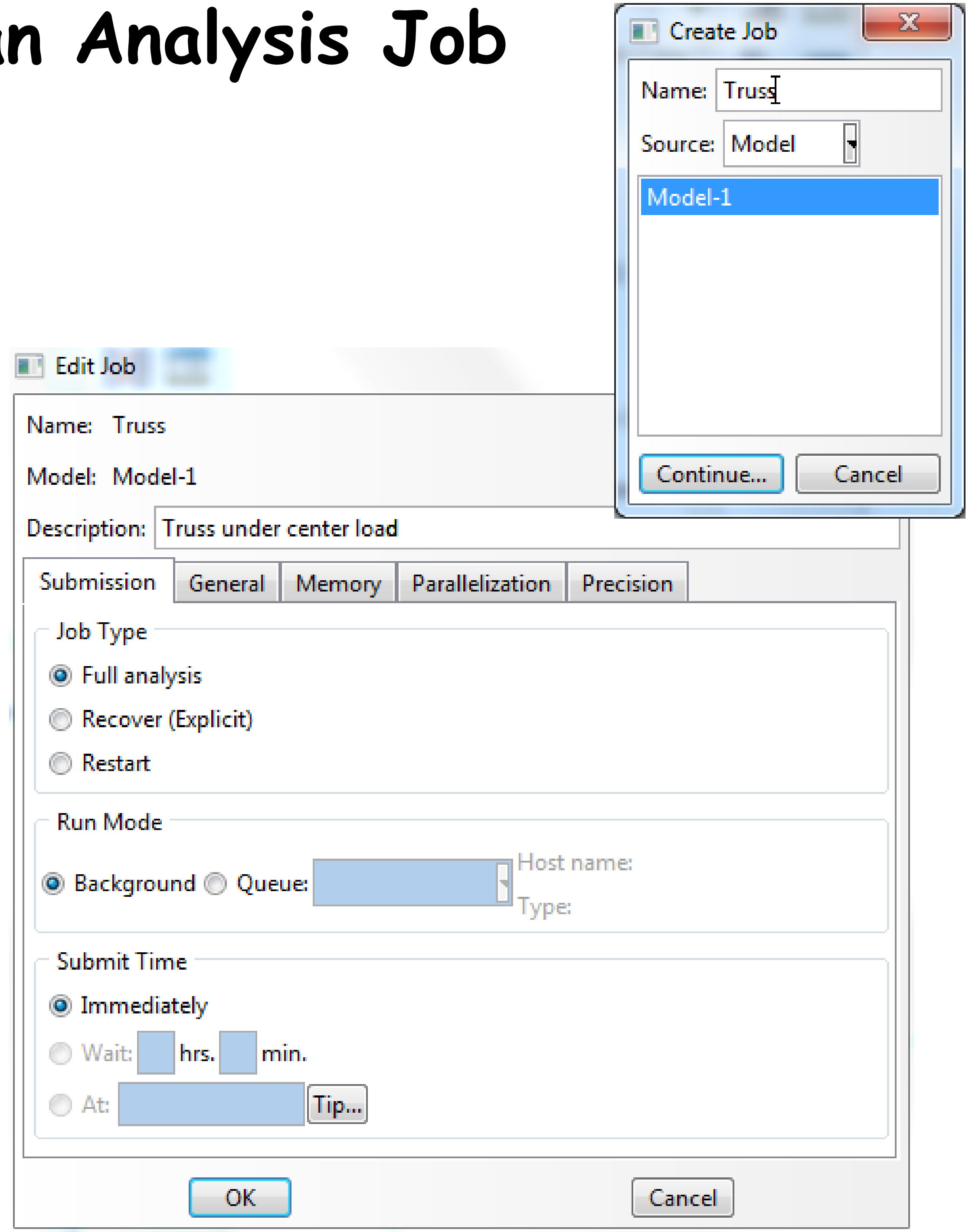
- Menu Seed, Part (side menu )
  - Change the seed size (Global size) 1.0 to 0.5
  - Delete the previous mesh 
- Menu Mesh, Part, Yes (side menu)





Creating an Analysis Job

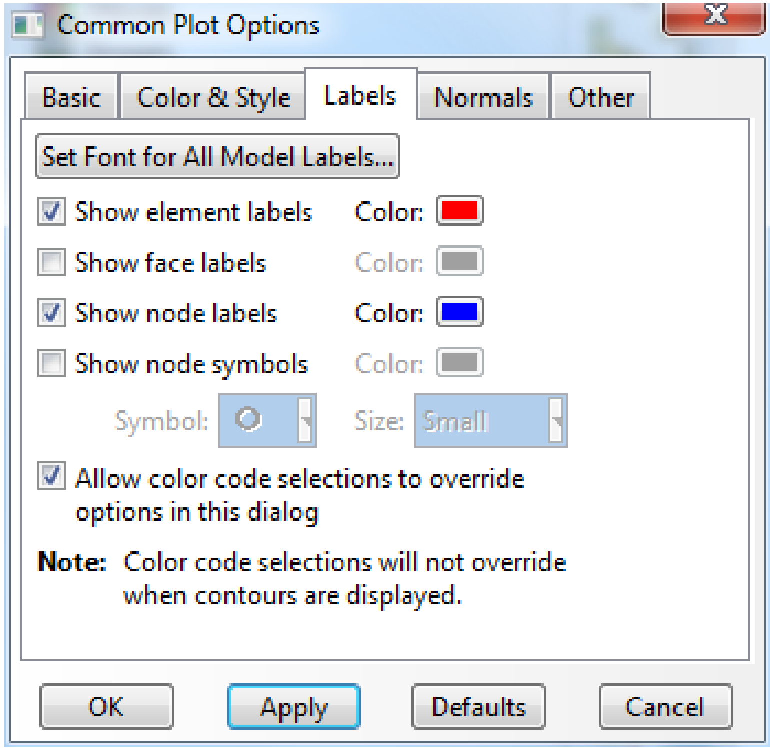
- Jobs
- Jobs, Truss
  - Data Check
  - Monitor
  - Continue (or, submit)



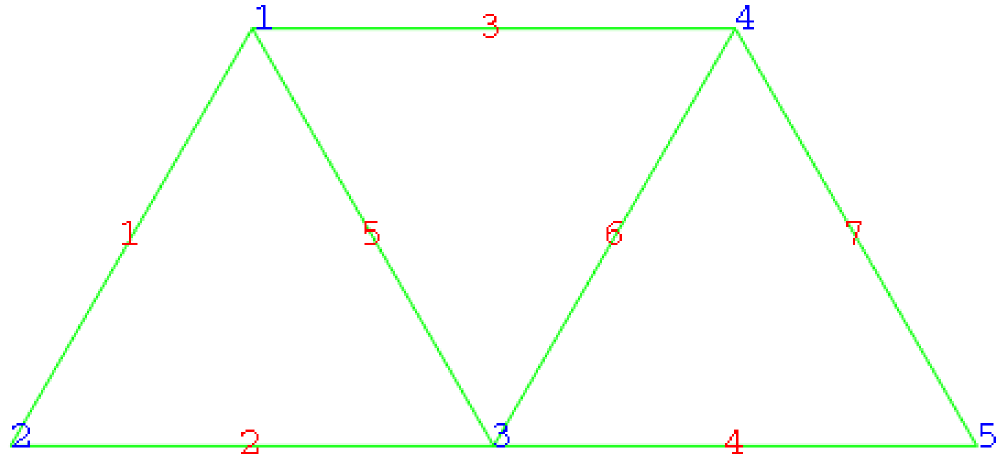
Postprocessing

- Change "Model" tab to "Results" tab
- Menu File, Open Job.odb file
- Common Plot Option (side menu ), click on the Labels tab

    (Show element labels, Show node labels)

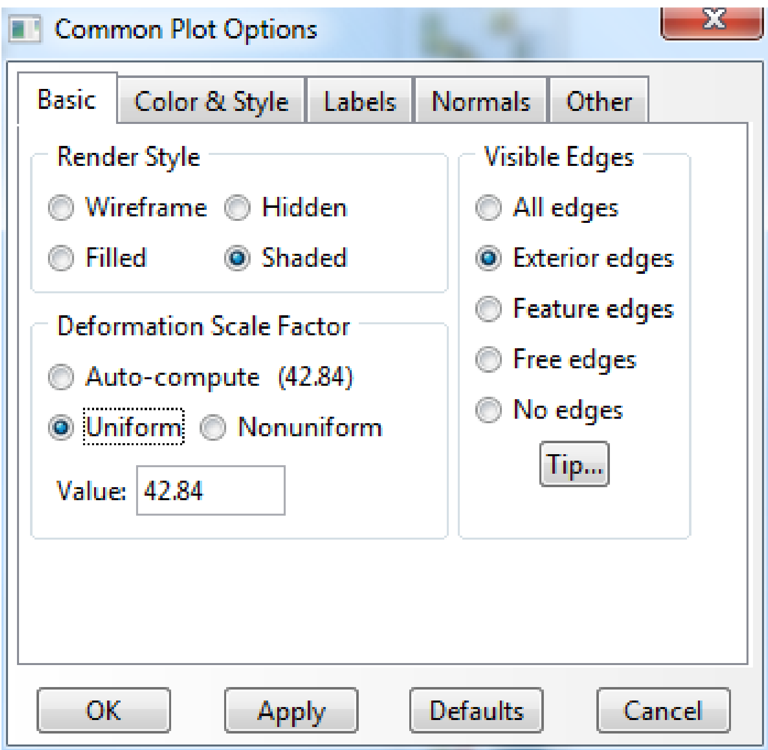


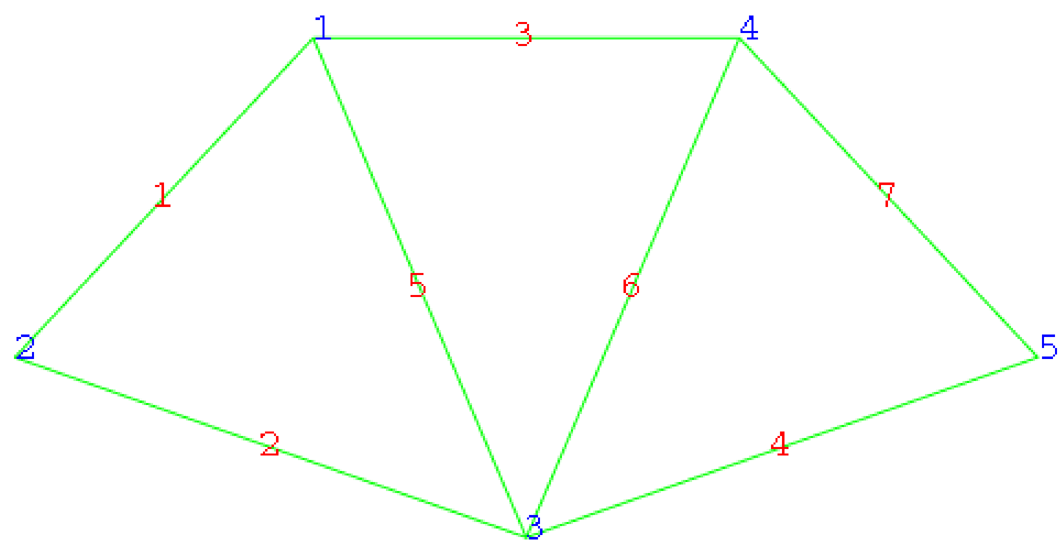
Set Font for All Model Labels...



Postprocessing

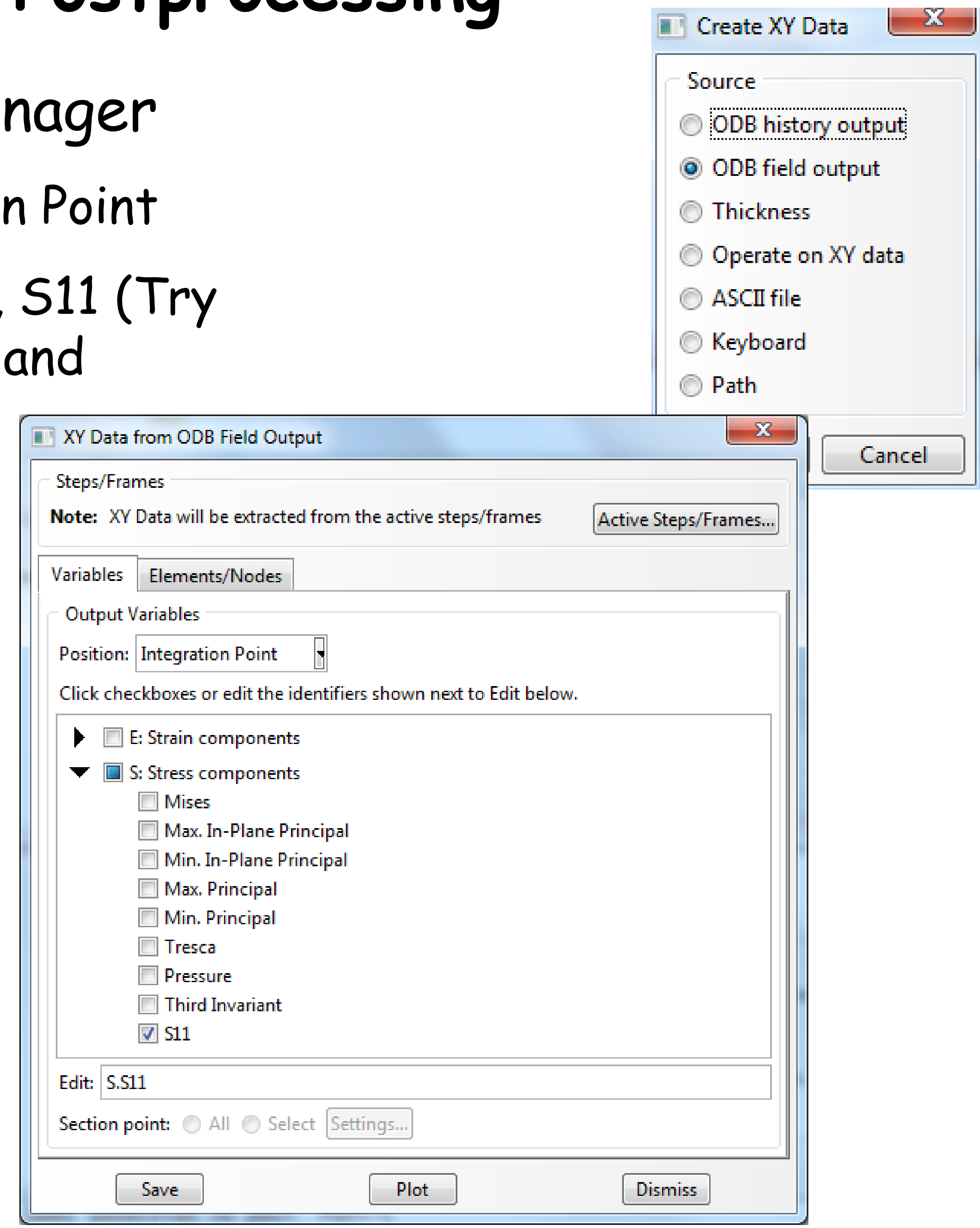
- Deformation scale
- Common Plot Option (side menu), click on the Basic tab, Deformation Scale Factor area





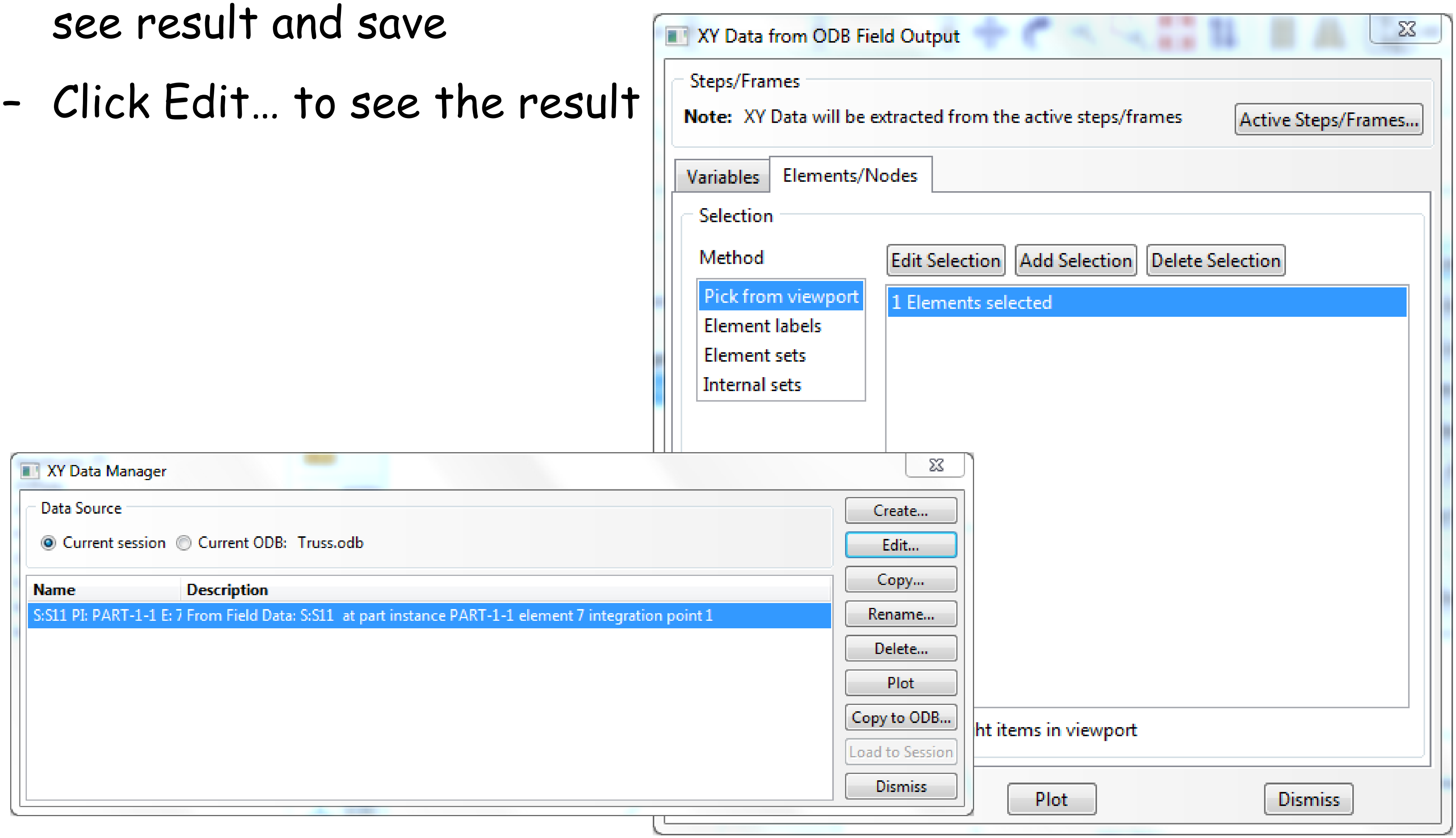
Postprocessing

- Tools, XY Data, Manager
  - Position: Integration Point
  - Stress components, S11 (Try with displacements and reaction)



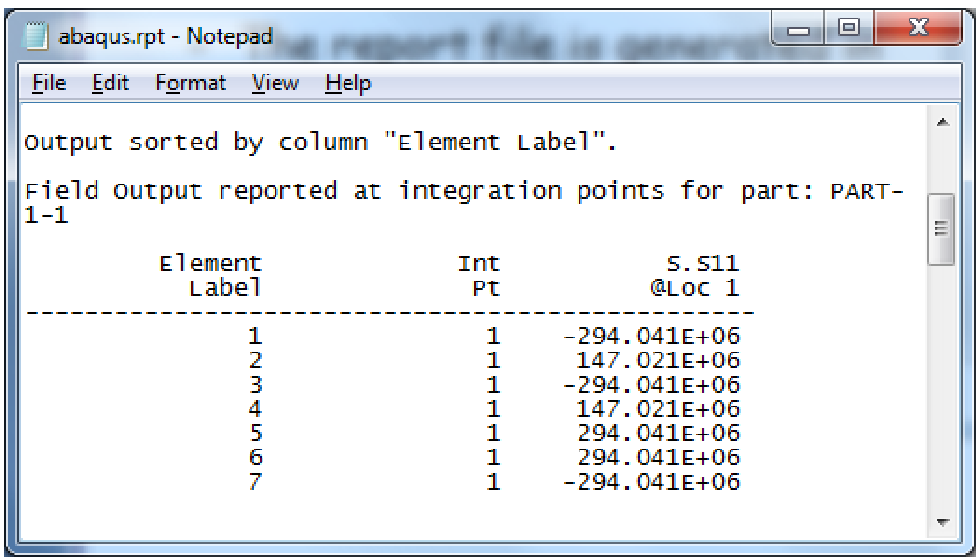
Postprocessing

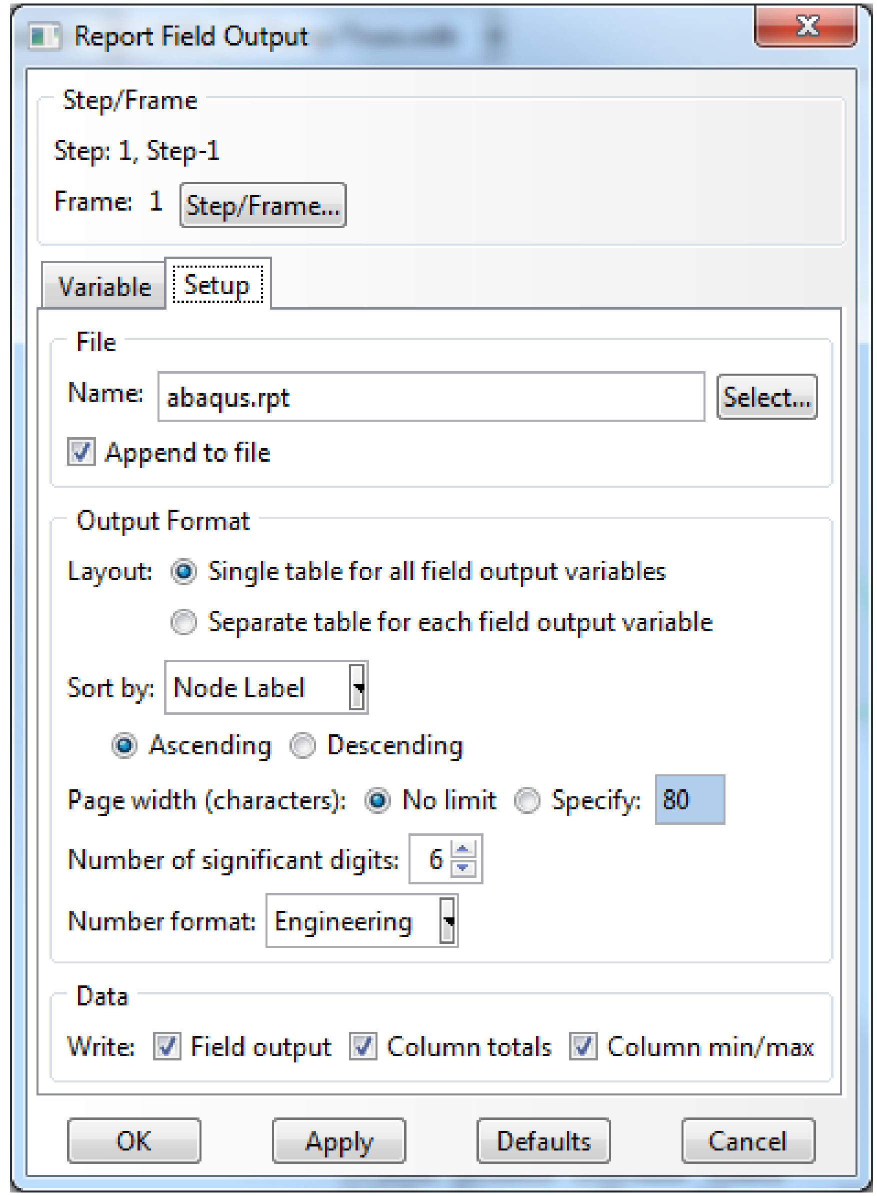
- Click on the Elements/Nodes tab
- Select Element/Nodes you want to see result and save
- Click Edit… to see the result



Postprocessing

- Report, Field Output
  - Position: Integration Point
  - Stress components, S11 (Try with displacements and reaction)
  - Default report file name is "abaqus.rpt"
  - The report file is generated in "C:\temp" folder



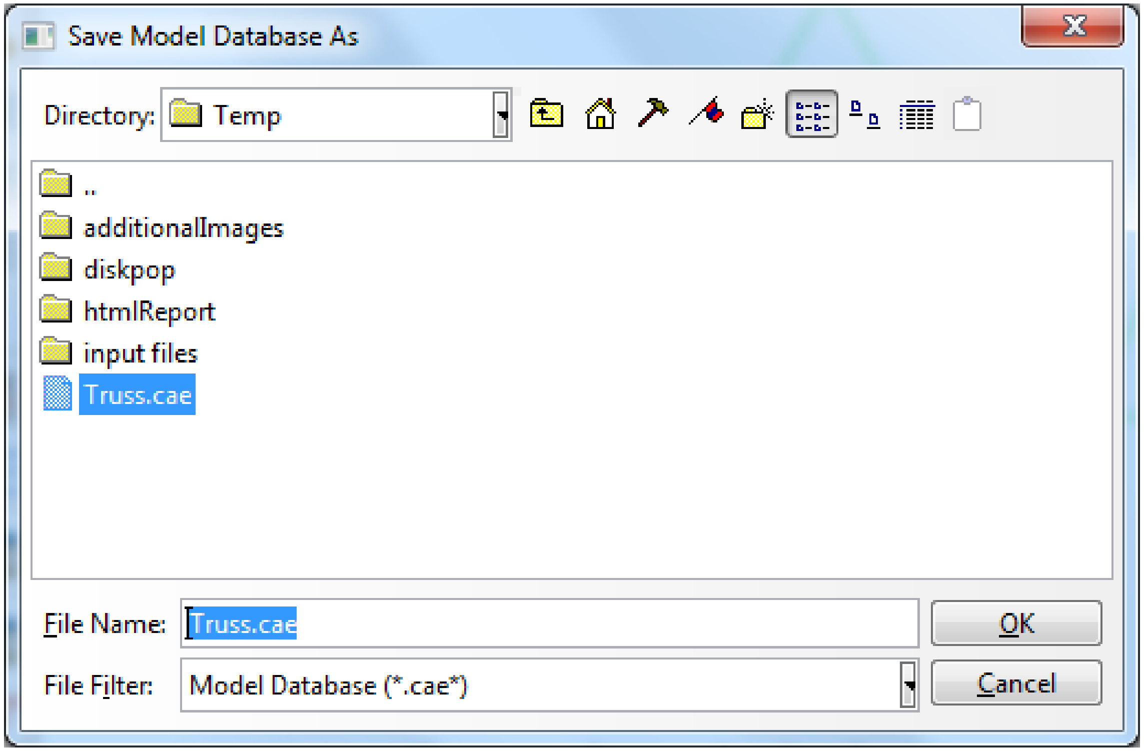


Save

- Save job.cae file
- Menu, File, Save As…

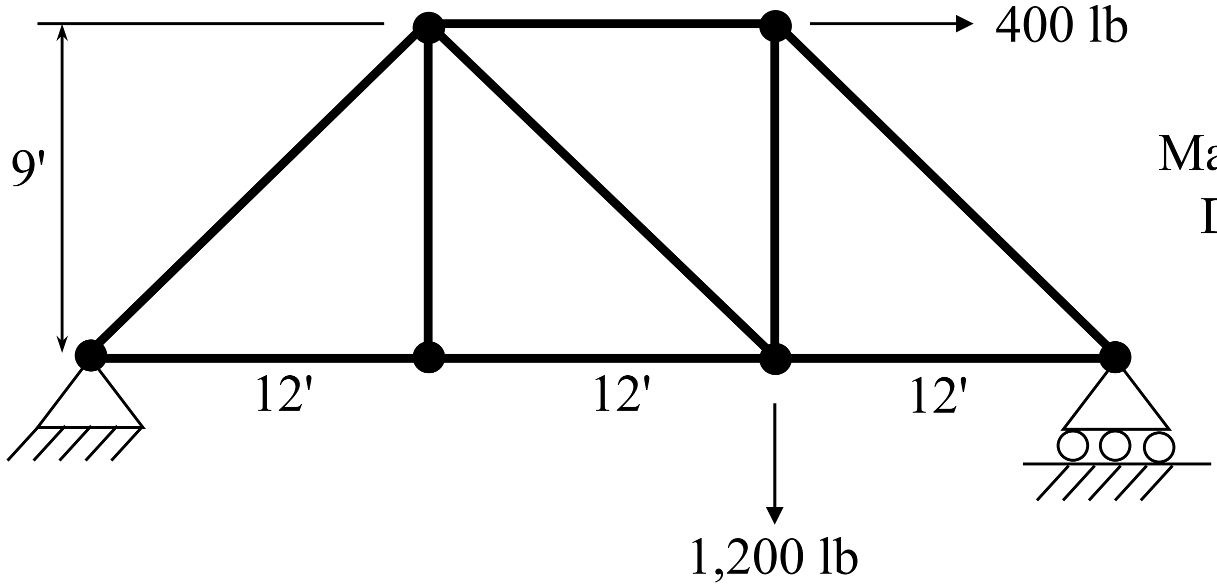
  - job.cae file is saved

  - job.jnl file is saved as well (user action history, python code)



Practice

- Calculate member stresses and displacements of truss shown in the figure using FEA software. Use Young's modulus = 2.9×10^{7} psi. Submit CAE file and report file (word or PDF) on Canvas



Material: Steel rod
Diameter: 0.5"