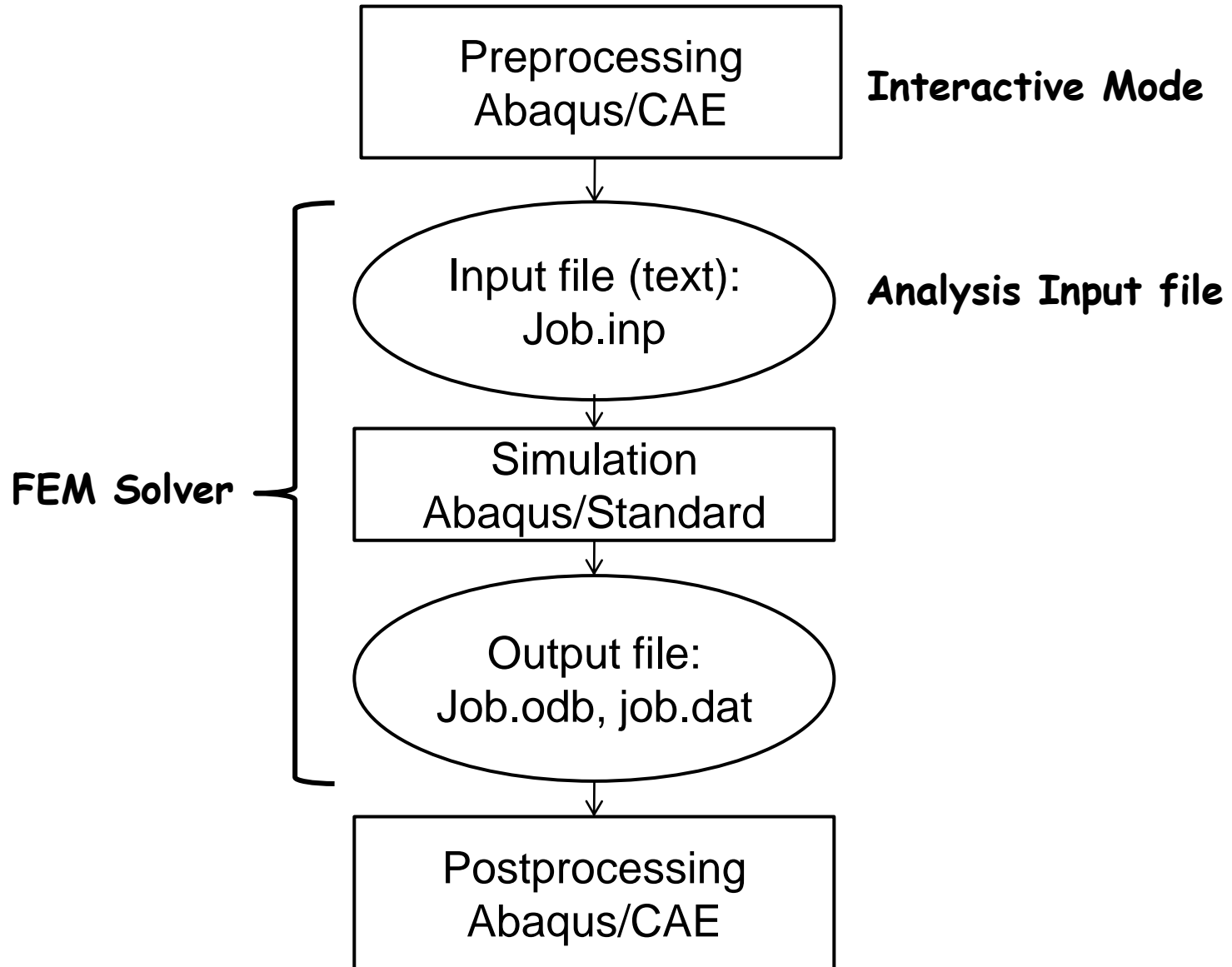


# Finite Element Analysis

## 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

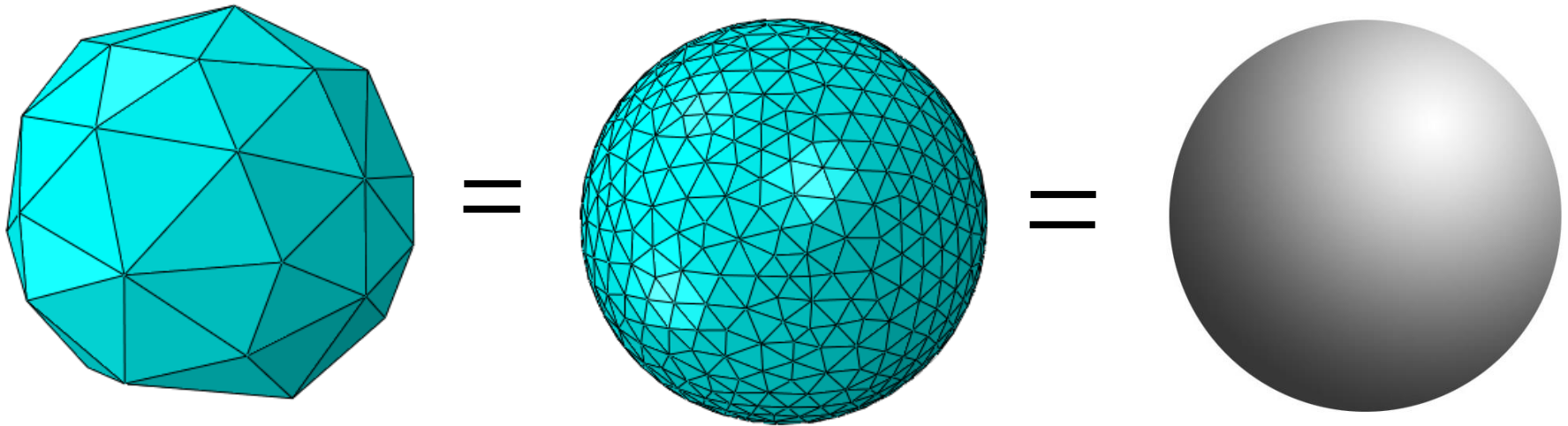
# Methods of Analysis in ABAQUS

- 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

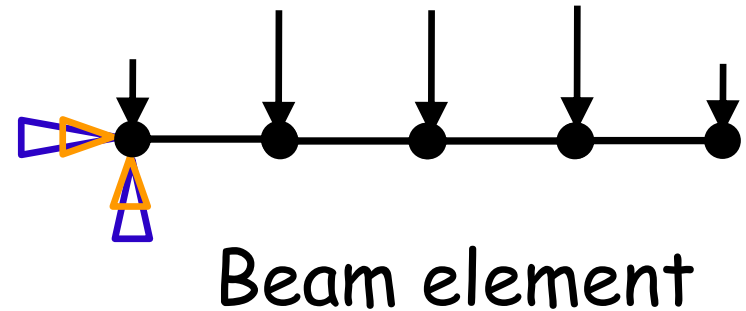
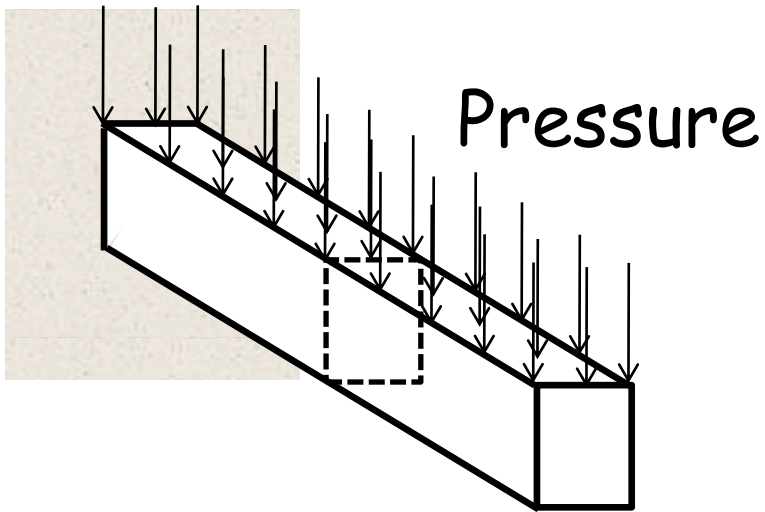
# Components in ABAQUS Model

- 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

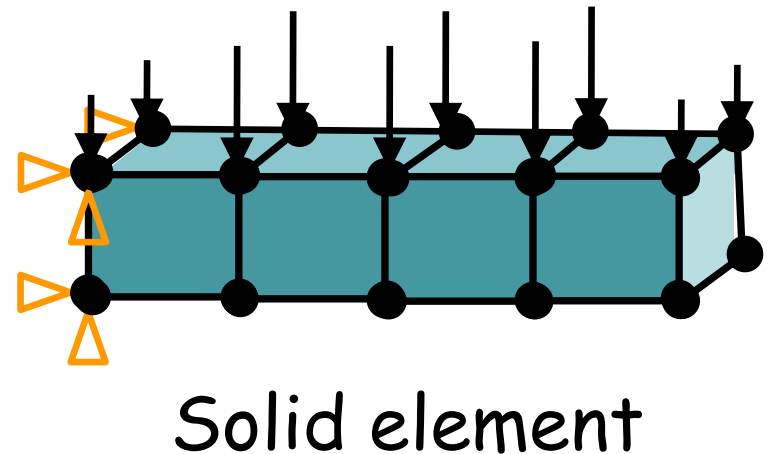
# FEM Modeling



# FEM Modeling



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



# FEM Modeling

---

## Line (Beam element)

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



## Volume (Solid element)

- Assign section properties
- Assign material properties

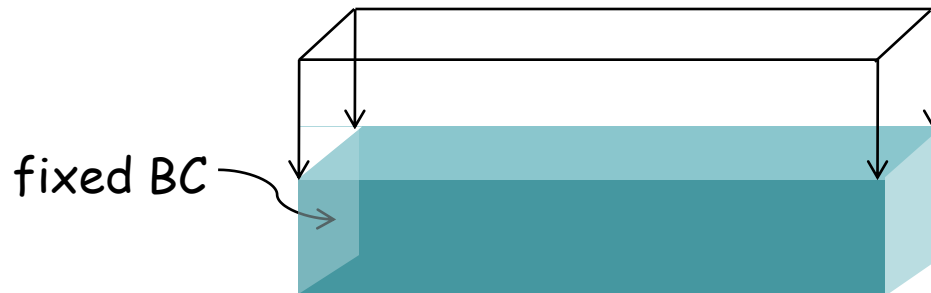


# FEM Modeling



## Line (Beam element)

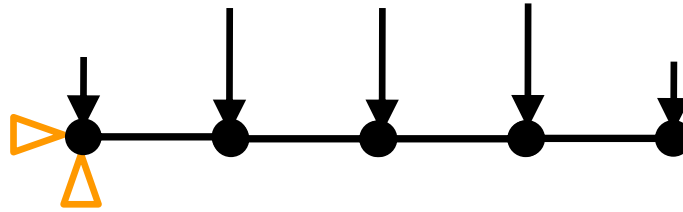
- Apply distributed load "on the line"
- Apply fixed BC "at the point"



## Volume (Solid element)

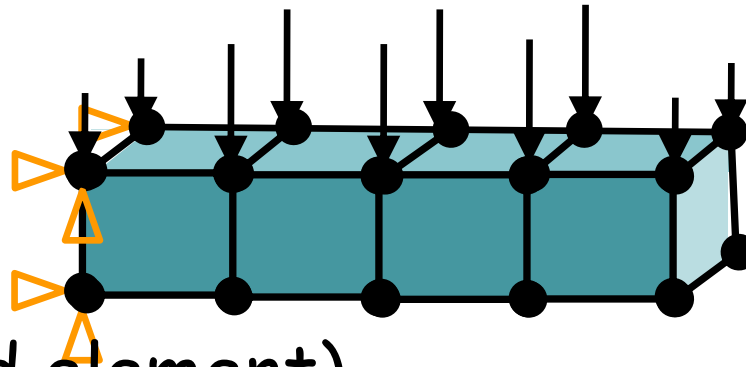
- Apply distribution load "on the surface"
- Apply fixed BC "on the surface"

# FEM Modeling



Line (Beam element)

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

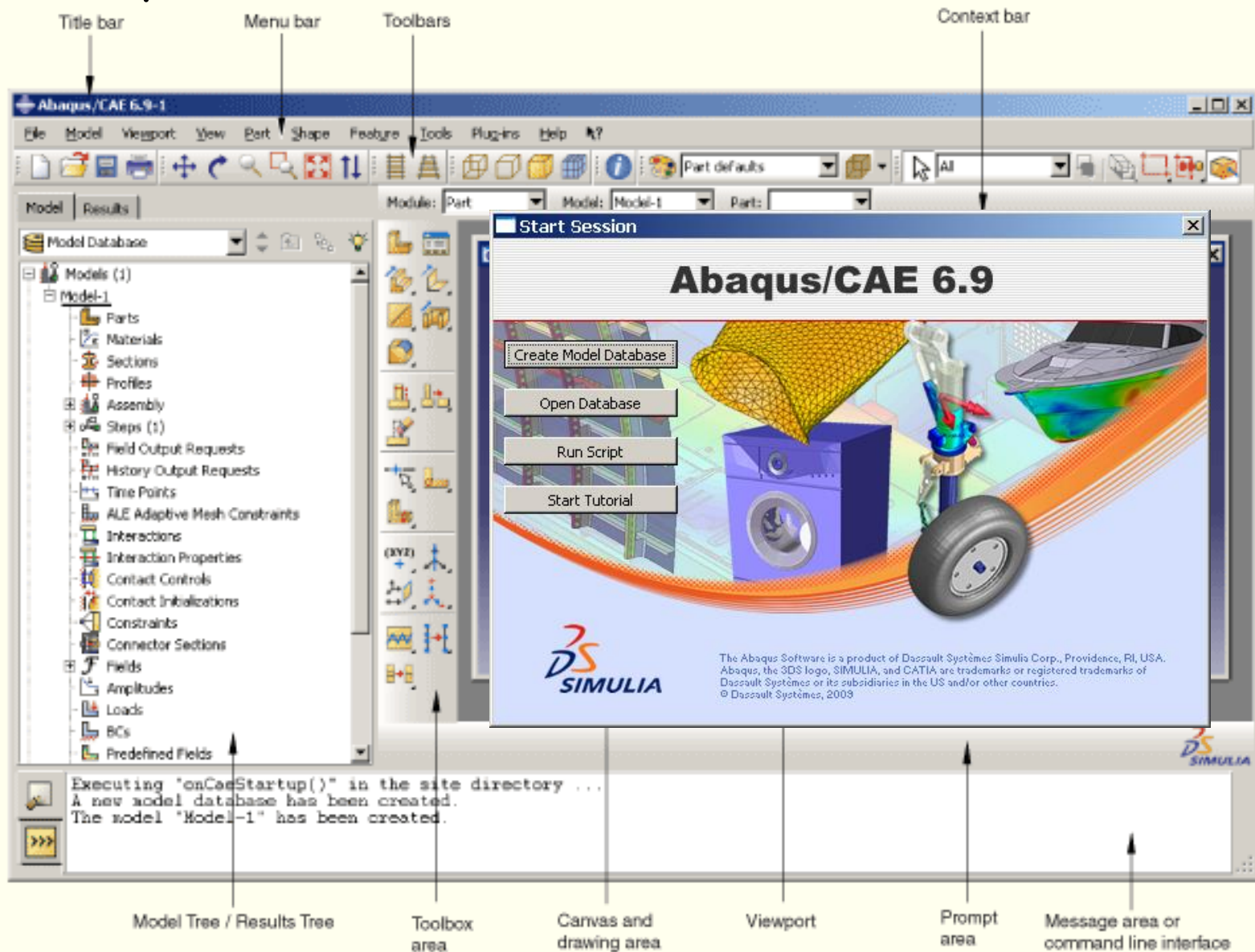


Volume (Solid element)

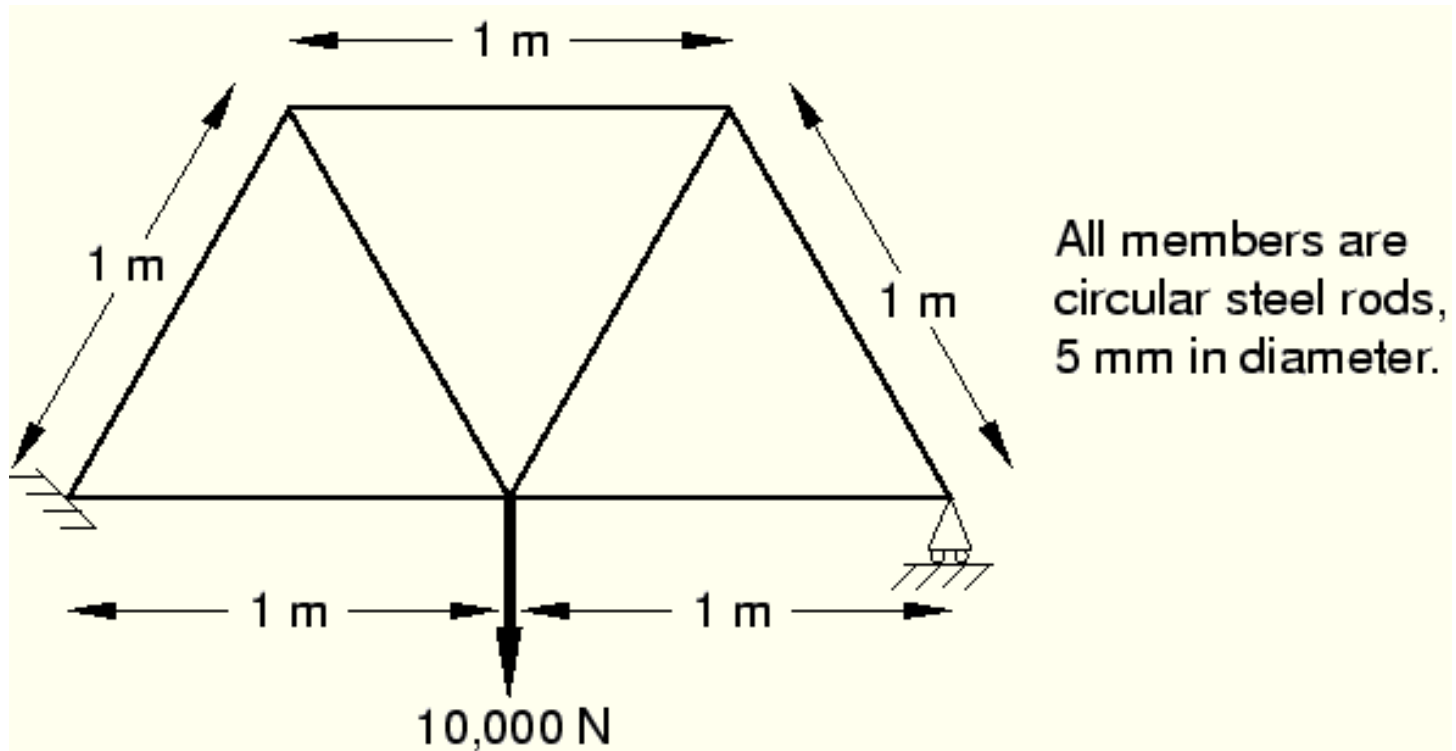
- Discretized geometry with solid element
- Discretized BC and load on nodes

# Start Abaqus/CAE

- Startup window



# Example: Overhead Hoist



## Material properties

General properties:

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

Elastic properties:

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

$$\nu = 0.3$$

# Units

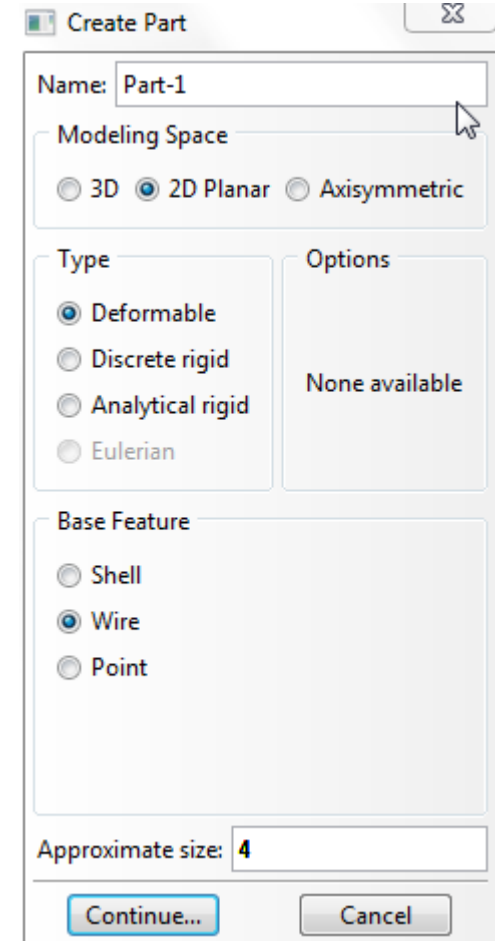
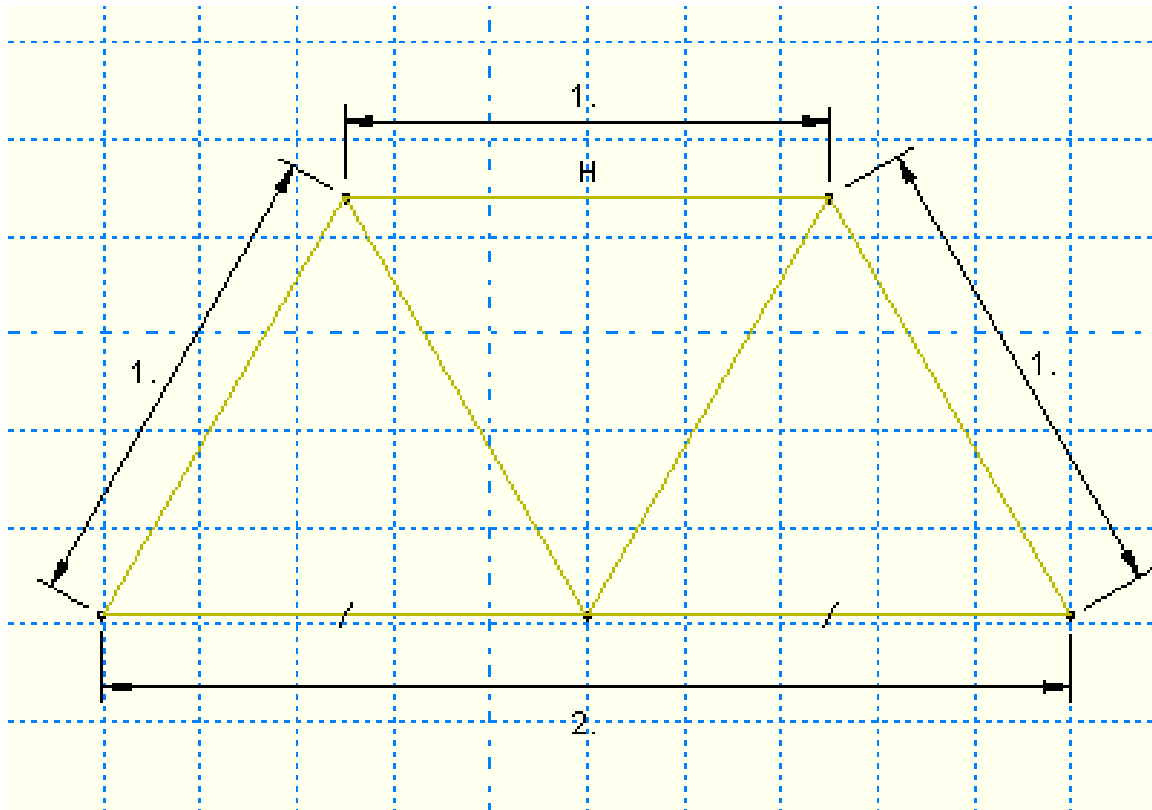
Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne ( $10^3$ kg)	slug	lbf s <sup>2</sup> /in
Time	s	s	s	s
Stress	Pa (N/m <sup>2</sup> )	MPa (N/mm <sup>2</sup> )	lbf/ft <sup>2</sup>	psi (lbf/in <sup>2</sup> )
Energy	J	mJ ( $10^{-3}$ J)	ft lbf	in lbf
Density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

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

# Create Part

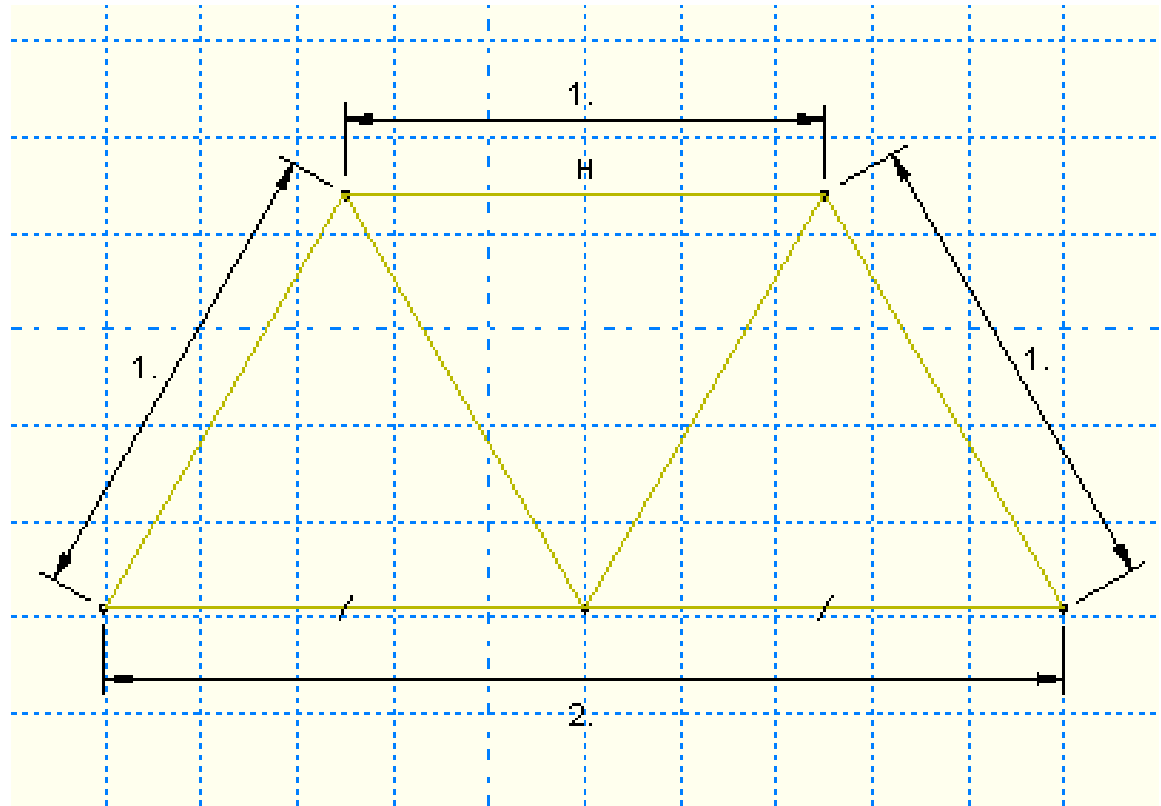
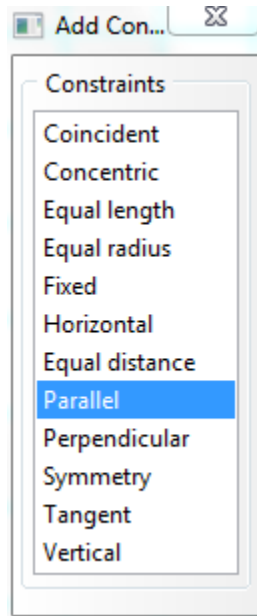
- Parts

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



# Geometry Constraint

- Define exact geometry
  - Add constraints 

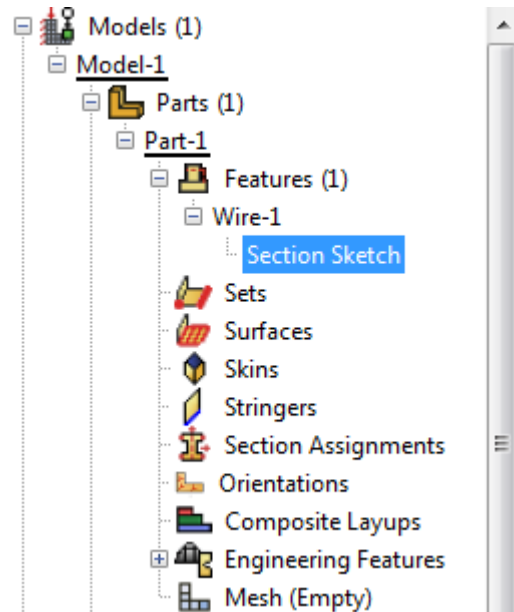


- Add dimension 
- Over constraint warning

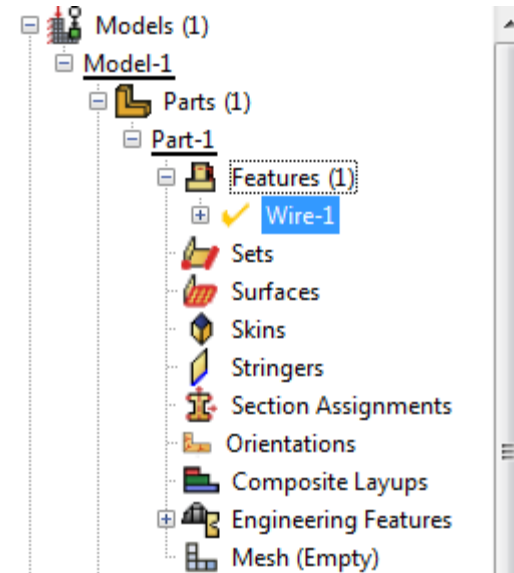
# Geometry Modification

- Modify geometry modeling

1. Go back to the sketch



2. Update geometry





# Define Material Properties

- Materials
  - Name: Steel
  - Mechanical
    - Elasticity
    - Elastic

**Edit Material**

Name:

Description:

Material Behaviors

- Elastic**

**Elastic**

Type:

☐ Use temperature-dependent data

Number of field variables:

Moduli time scale (for viscoelasticity):

☐ No compression

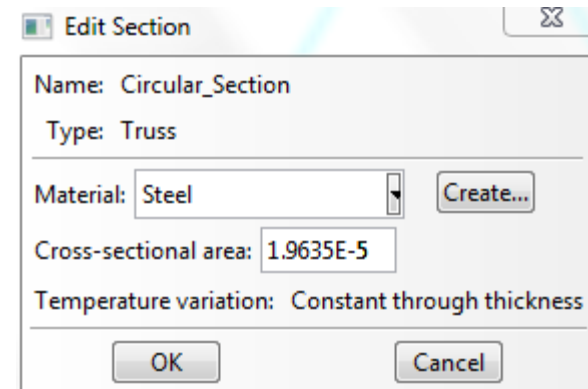
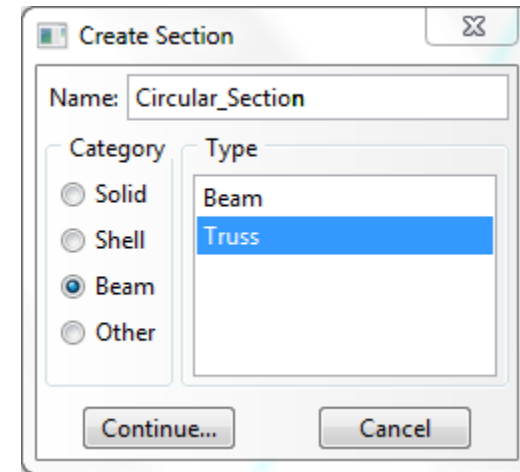
☐ No tension

**Data**

	Young's Modulus	Poisson's Ratio
1	200E9	0.3

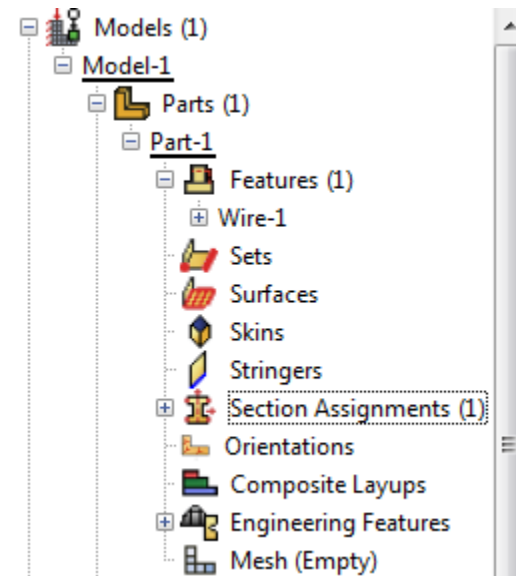
# Define Section Properties

- Calculate cross-sectional area using CLI (diameter = 5mm)
- Sections
  - Name: Circular\_Section
  - Beam, Truss
  - Choose material (Steel)
  - Write area

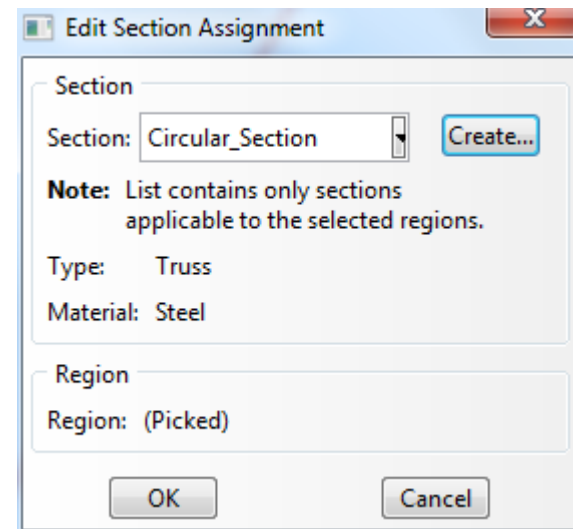


# Define Section Properties

- Assign the section to the part
  - Section Assignments

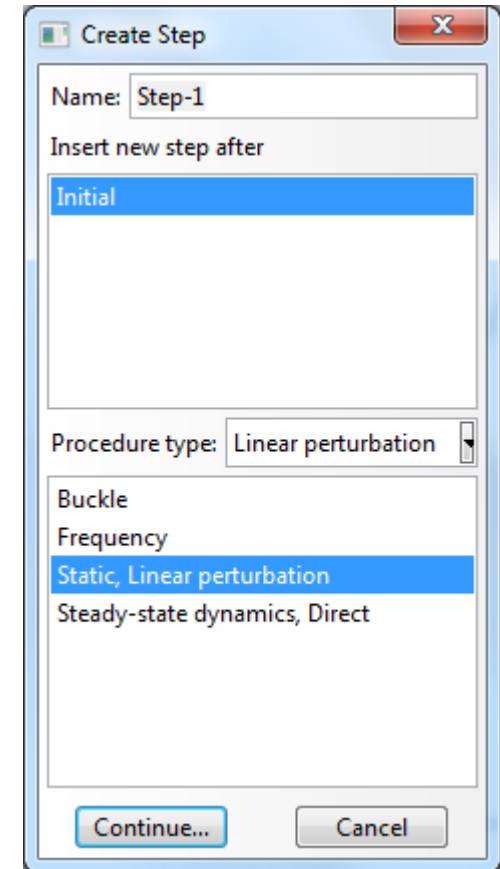


- Select all wires
- Assign Circular\_Section



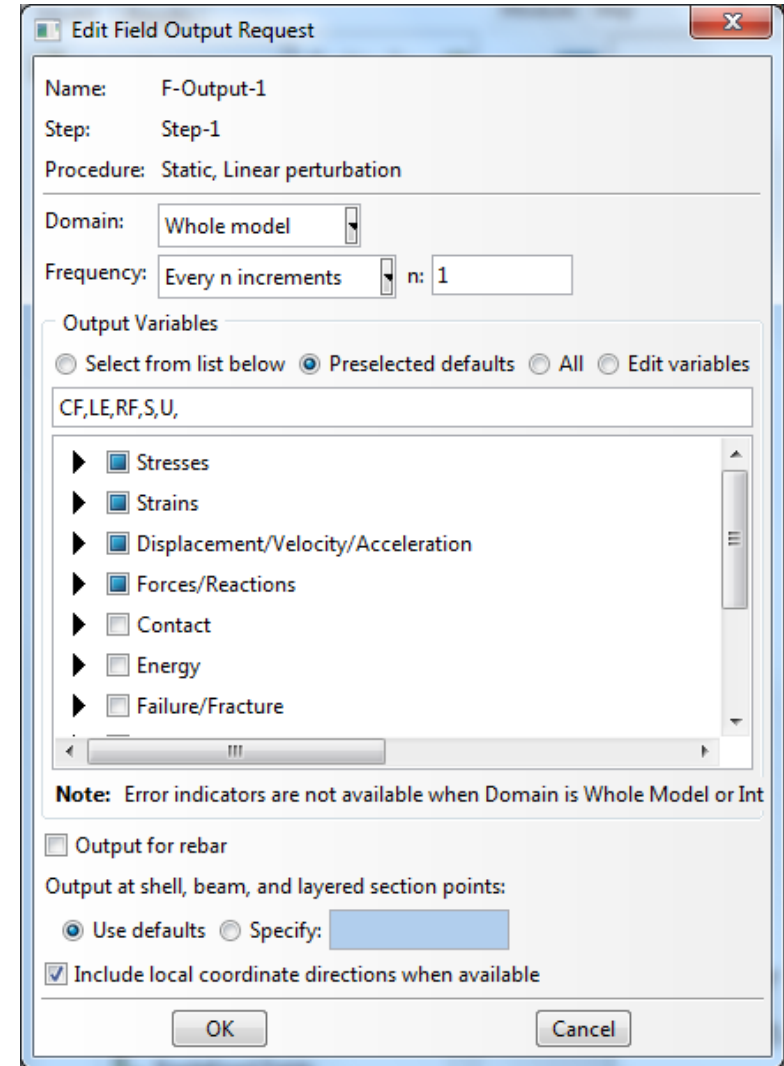
# Assembly and Analysis Step

- 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



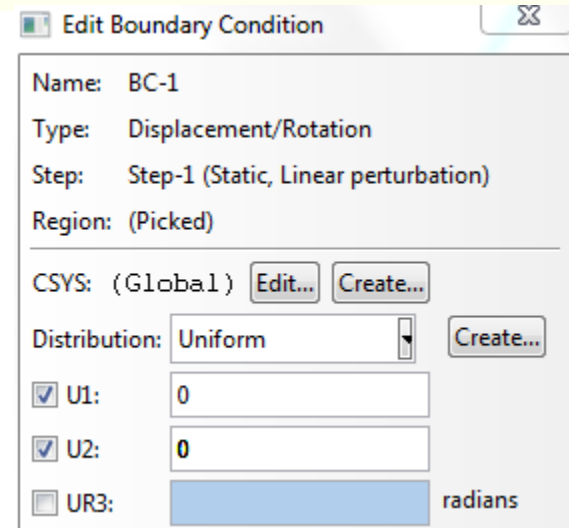
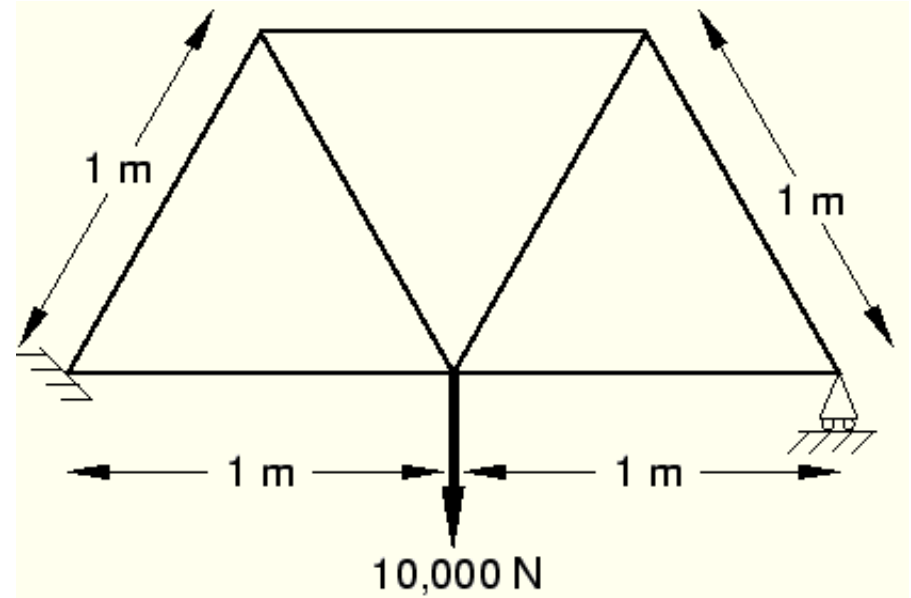
# Assembly and Analysis Step

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



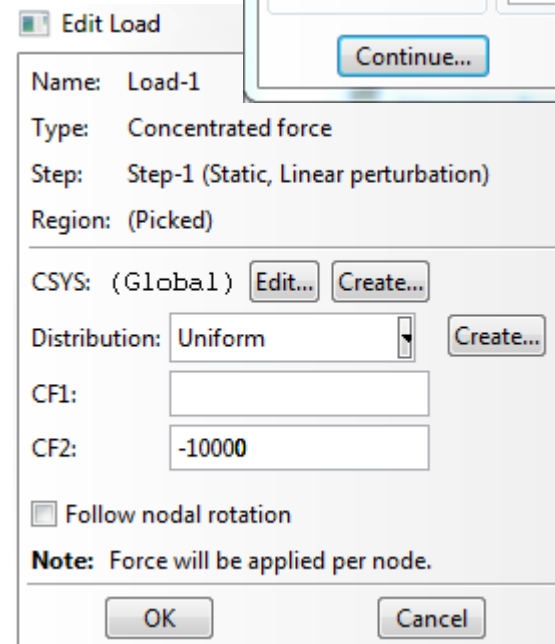
# Boundary Conditions

- 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



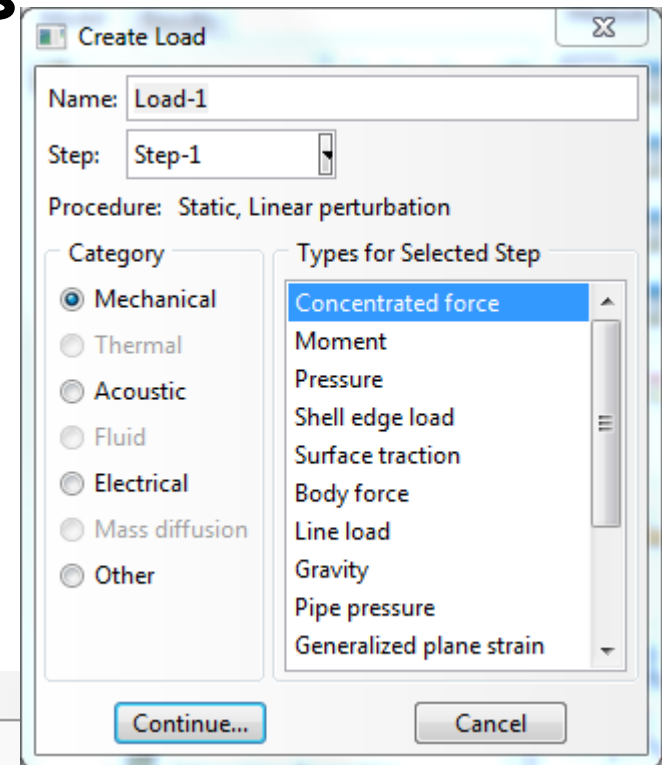
# Applied Loads

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



The 'Edit Load' dialog box is shown below the 'Create Load' dialog. It contains the following fields and options:


- Name: Load-1
- Type: Concentrated force
- Step: Step-1 (Static, Linear perturbation)
- Region: (Picked)
- CSYS: (Global) with 'Edit...' and 'Create...' buttons
- Distribution: Uniform with a 'Create...' button
- CF1: (empty text box)
- CF2: -10000
- ☐ Follow nodal rotation
- Note: Force will be applied per node.
- Buttons: OK, Cancel

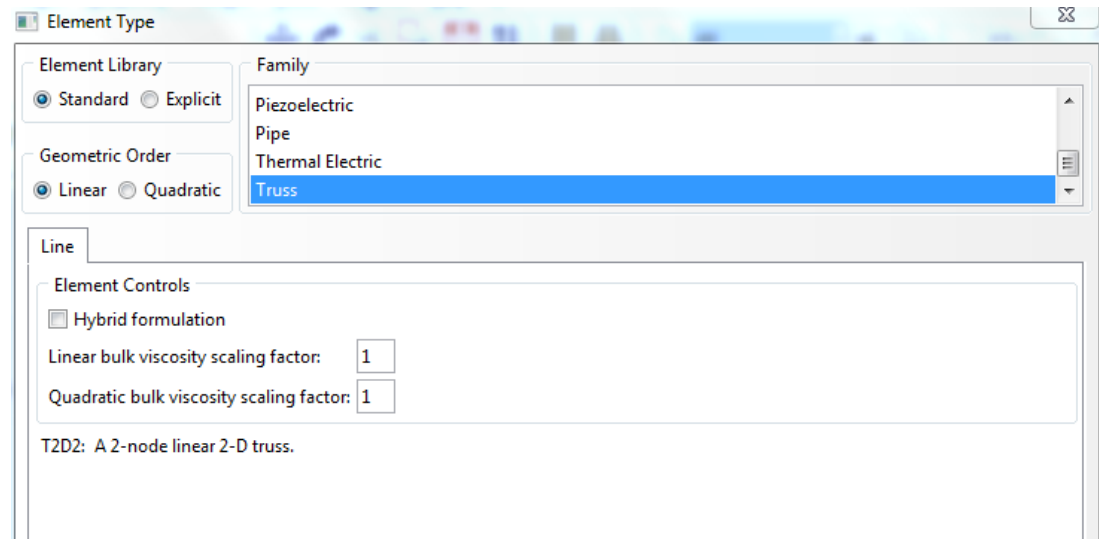


The 'Create Load' dialog box is shown above the 'Edit Load' dialog. It contains the following fields and options:

- Name: Load-1
- Step: Step-1
- Procedure: Static, Linear perturbation
- Category: Mechanical (selected)
- Types for Selected Step: Concentrated force (selected), Moment, Pressure, Shell edge load, Surface traction, Body force, Line load, Gravity, Pipe pressure, Generalized plane strain
- Buttons: Continue..., Cancel



# Meshing the Model

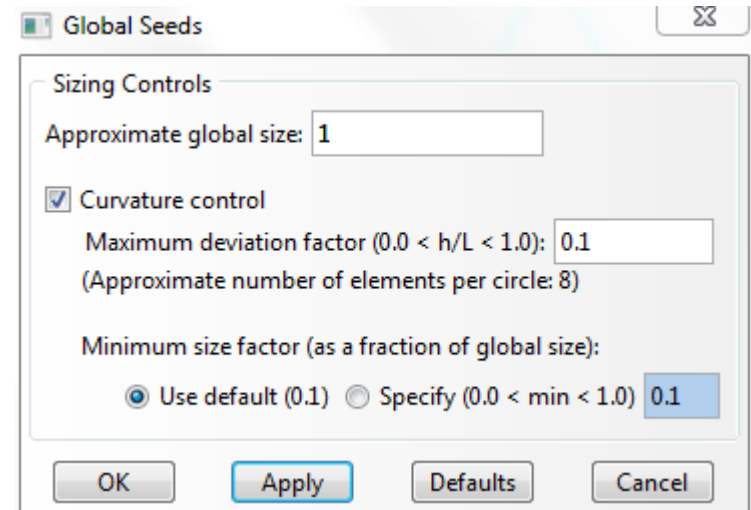
- Parts
  - Part-1, Mesh
- Menu Mesh, Element Types (side menu )
- Select all wireframes
- Library: Standard
- Order: Linear
- Family: Truss
- T2D2: 2-node linear 2-D truss








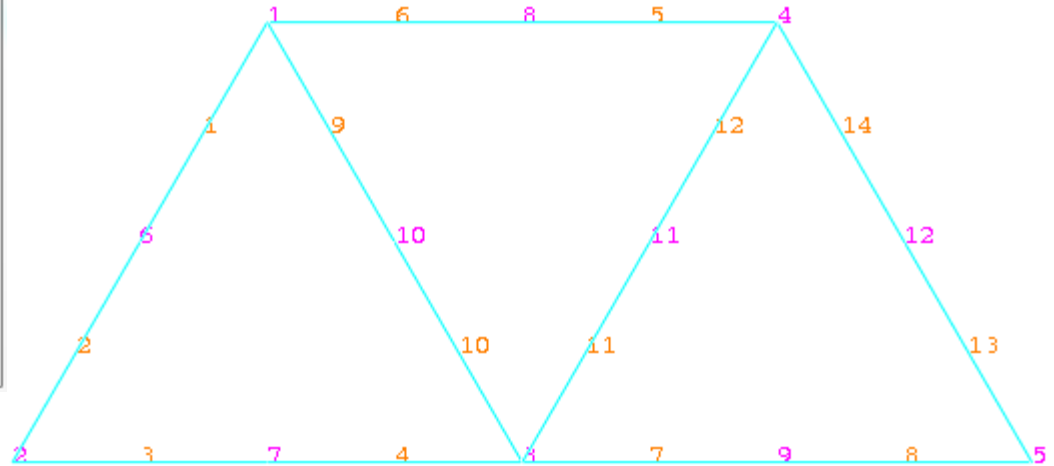
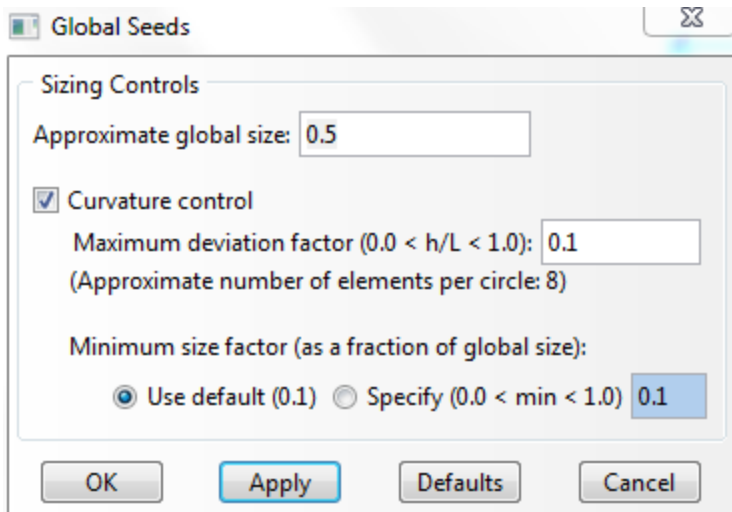
# Meshing the Model

- 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)

The screenshot shows the 'Edit Job' dialog box for a job named 'Truss'. The 'Name' field is 'Truss', the 'Model' is 'Model-1', and the 'Description' is 'Truss under center load'. The 'Submission' tab is selected, showing options for 'Job Type' (Full analysis, Recover (Explicit), Restart) and 'Run Mode' (Background, Queue). The 'Submit Time' section has options for 'Immediately', 'Wait' (with hours and minutes fields), and 'At' (with a date field and a 'Tip...' button). The 'Continue...' button is highlighted in the top right corner.

**Create Job**

Name: Truss

Source: Model

Model-1

Continue... Cancel

**Edit Job**

Name: Truss

Model: Model-1

Description: Truss under center load

Submission General Memory Parallelization Precision

Job Type

☒ Full analysis

☐ Recover (Explicit)

☐ Restart

Run Mode

☒ Background ☐ Queue: [ ] Host name: [ ]

Type: [ ]

Submit Time


☒ Immediately

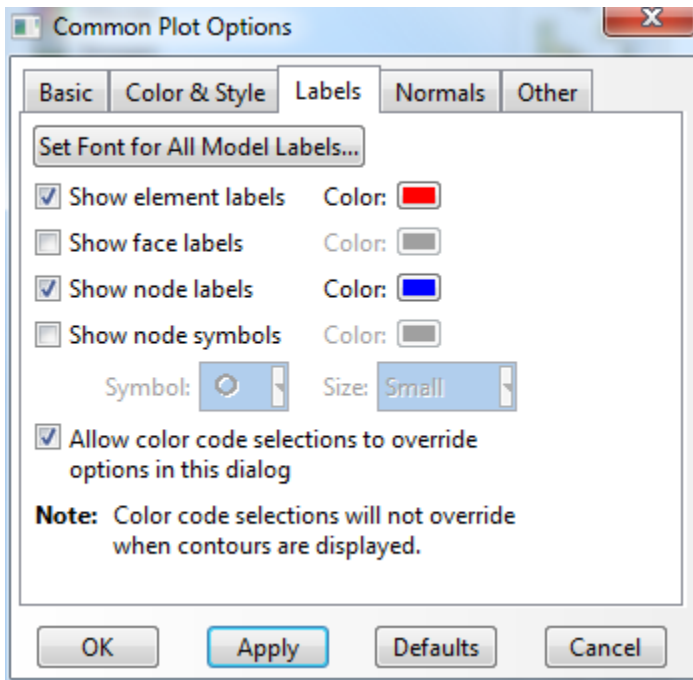
☐ Wait: [ ] hrs. [ ] min.

☐ At: [ ] Tip...

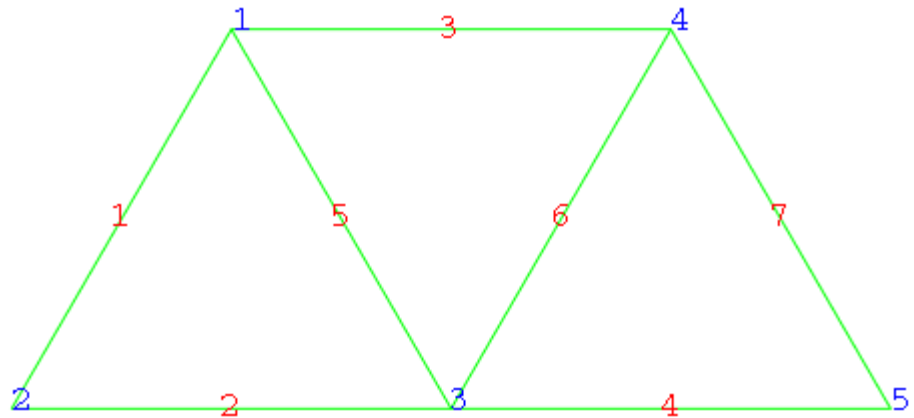
OK Cancel

# 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)

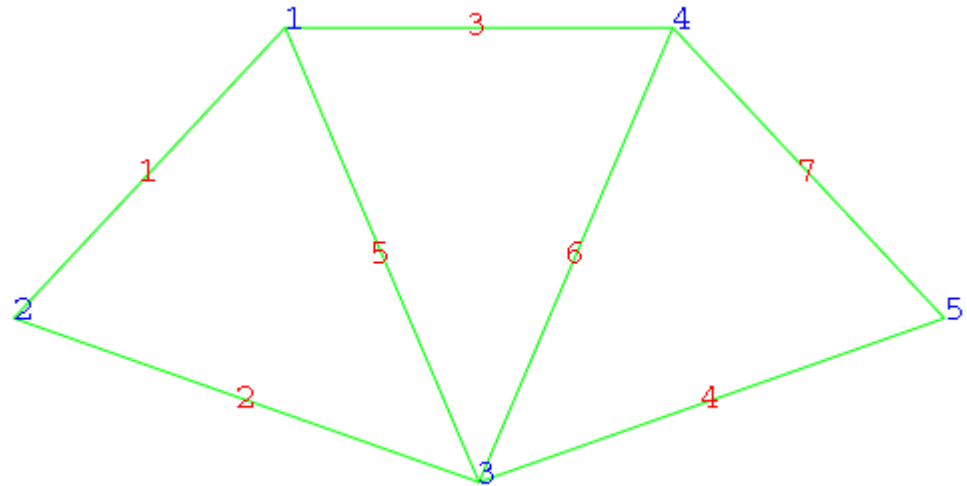
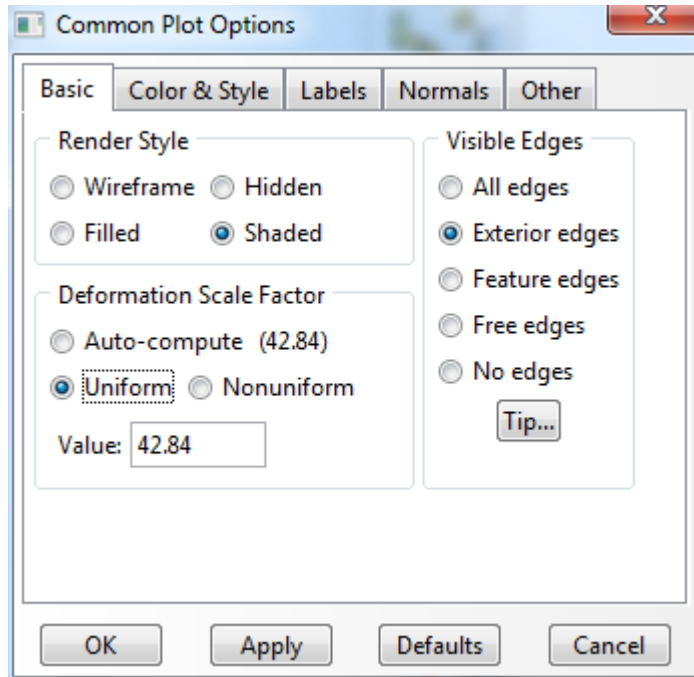


Set Font for All Model Labels...



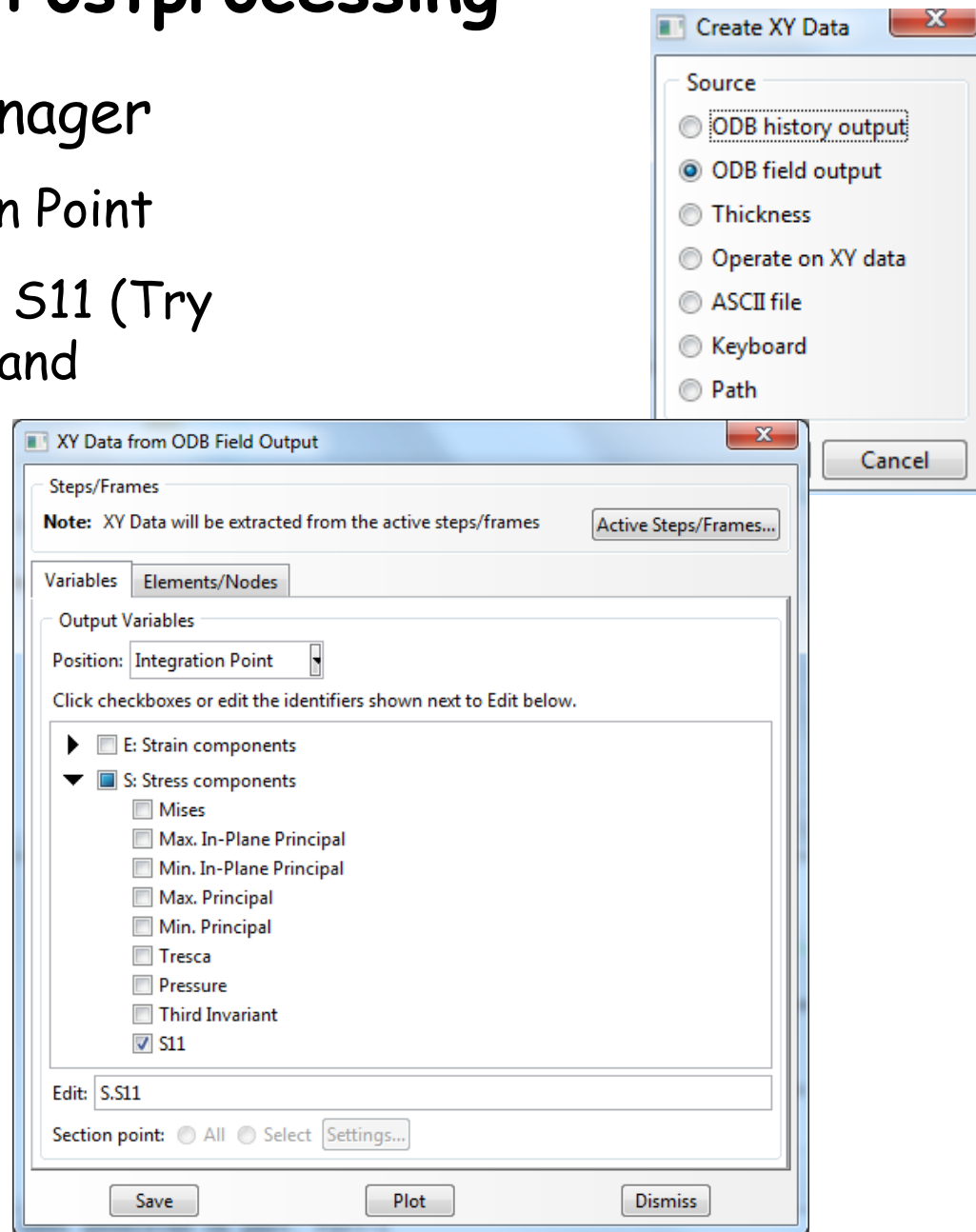
# Postprocessing

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



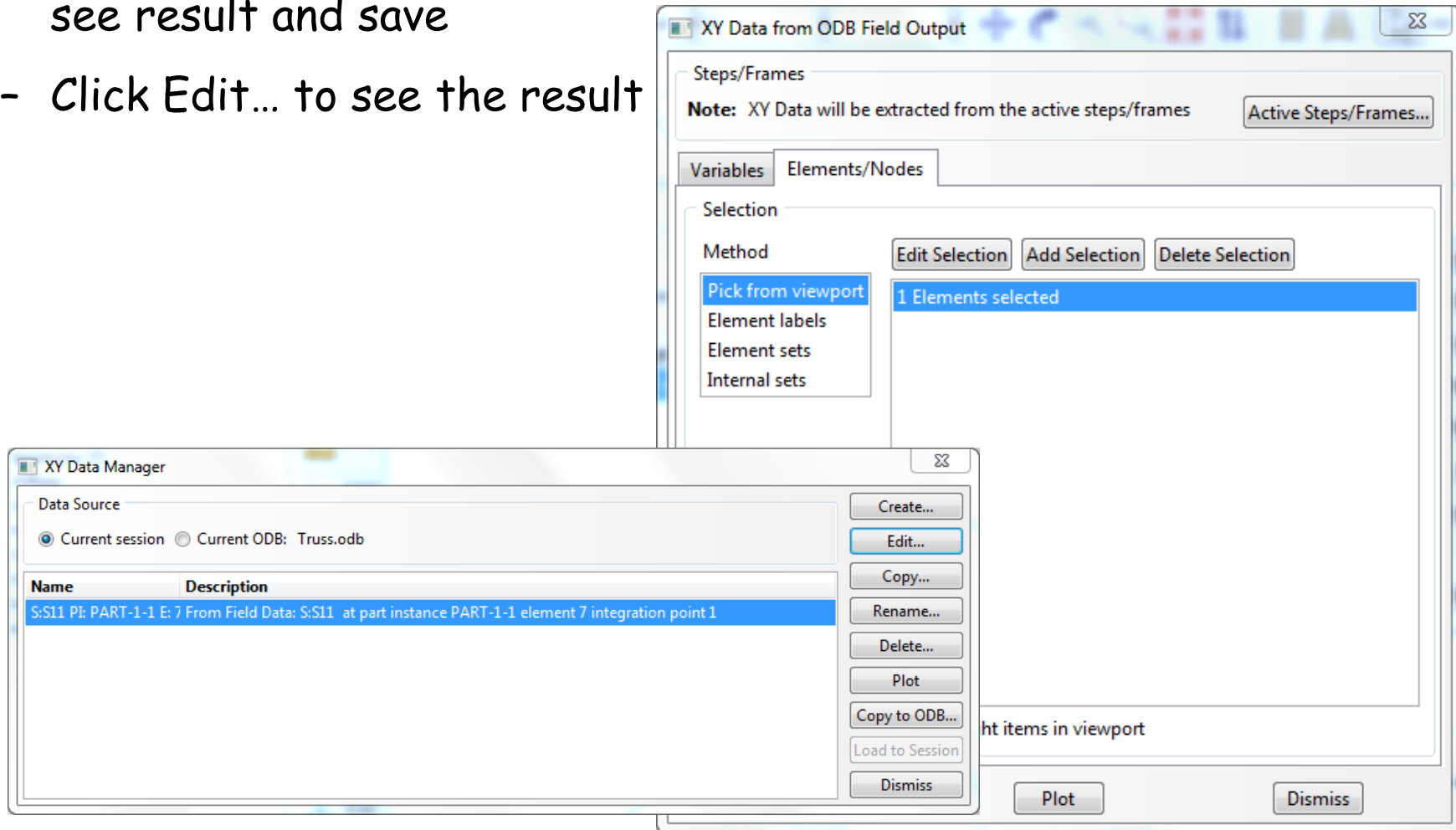
# Postprocessing

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



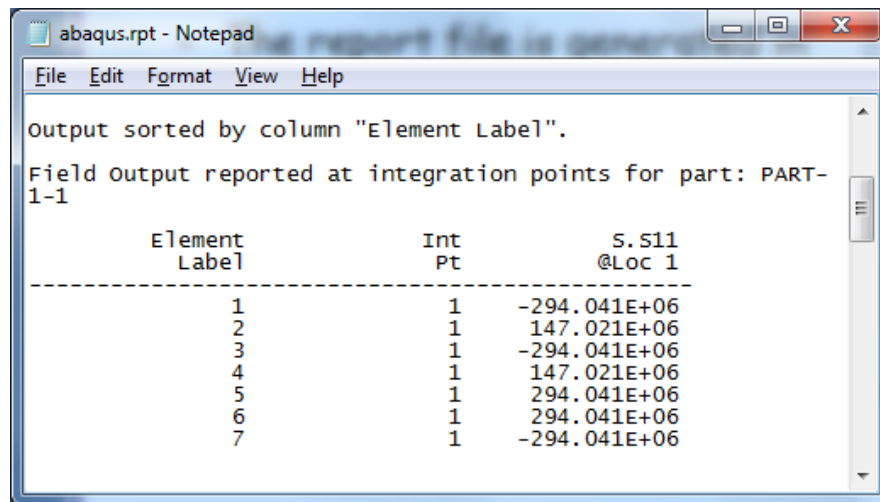
# Postprocessing

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

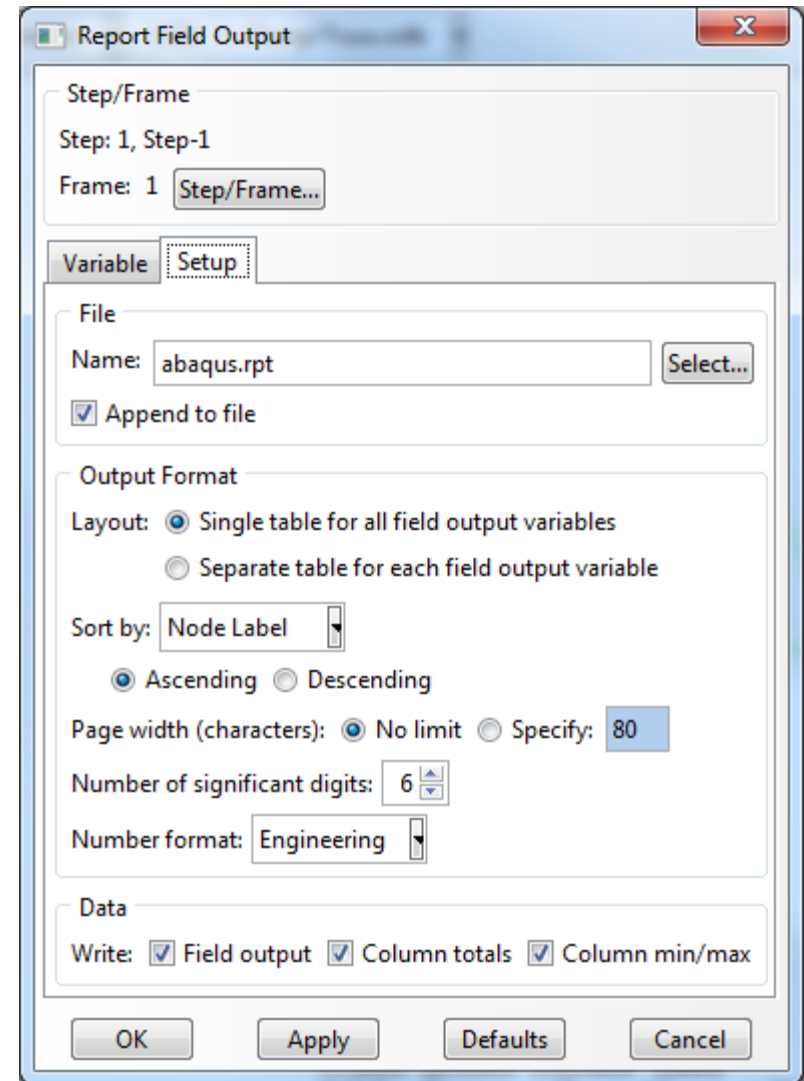


# Postprocessing

- 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



Element Label	Int Pt	S.S11 @Loc 1
1	1	-294.041E+06
2	1	147.021E+06
3	1	-294.041E+06
4	1	147.021E+06
5	1	294.041E+06
6	1	294.041E+06
7	1	-294.041E+06



Report Field Output

Step/Frame  
Step: 1, Step-1  
Frame: 1

Variable

File  
Name:    
☒ Append to file

Output Format  
Layout: ☒ Single table for all field output variables  
☐ Separate table for each field output variable  
Sort by:   
☒ Ascending ☐ Descending  
Page width (characters): ☒ No limit ☐ Specify:   
Number of significant digits:   
Number format:

Data  
Write: ☒ Field output ☒ Column totals ☒ Column min/max



# 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)

