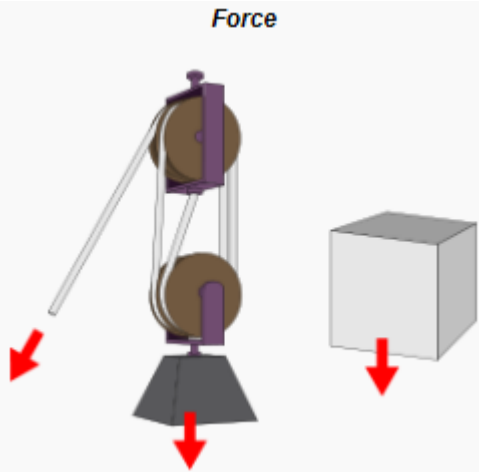
Week 12 - Lecture
Mechanical Event Simulation

Lecture Topics

- Mechanical Event Simulation Overview
- Additional Element Types
- Joint Component Description
- General Constraint Refresh
- Mesh Control

Force Estimation Methods

- Experience
  - Engineers reply on past experience from similar projects to estimate forces. Often results in an over designed product.



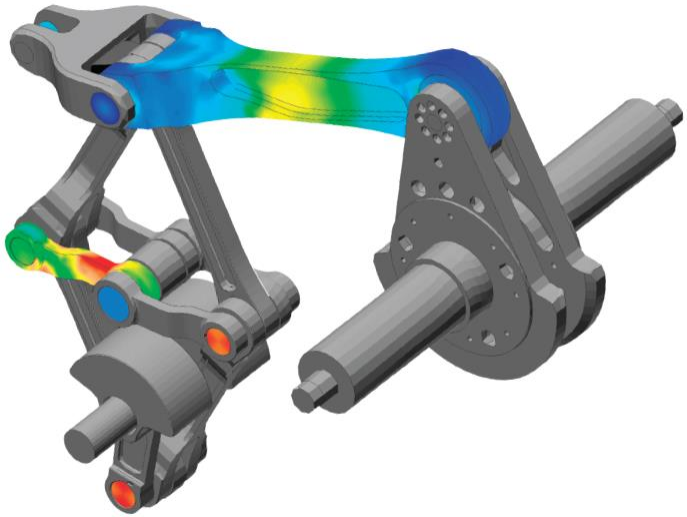
- Rigid-Body Dynamics
  - Leverage the 3D design data to perform a motion simulation using rigid bodies to gain insights into force values.
- Physical Experimentation
  - Experiment with prototypes or past products to obtain accurate force values.

Let’s Keep Exploring Simulation

Mechanical Event Simulation (MES)

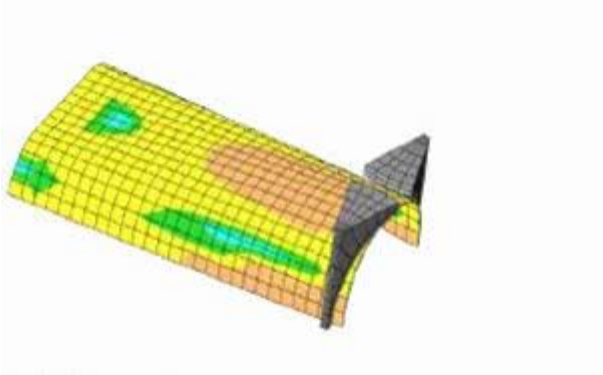
Mechanical Event Simulation (MES) combines kinematic, rigid, and flexible-body dynamics, and nonlinear stress analysis.

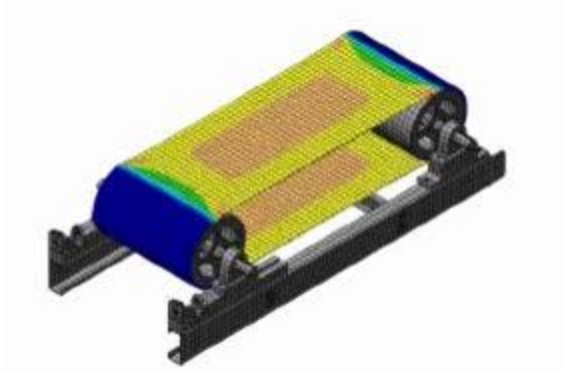
- Events with Large Deformations
- Nonlinear Material Properties
- Kinematic Motion
- Forces Caused by Motion
- Stress Results / Motion Forces



MES Use Cases

- Complete FEA on part throughout complete loading cycle.
- Conduct a impact test on a product.
- Validate the motion of flexible body components.

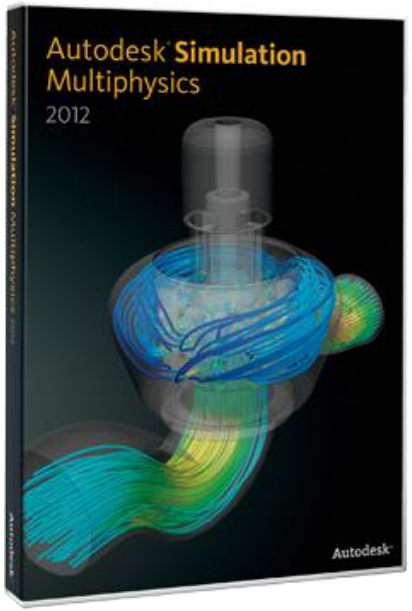




Autodesk Simulation

Autodesk Simulation helps designers and engineers make decisions earlier and predict product performance.

- Linear and Nonlinear Static Stress
- Fatigue Analysis
- Linear Dynamic
- Mechanical Event
- Heat Transfer
- CFD
- Multiphysics

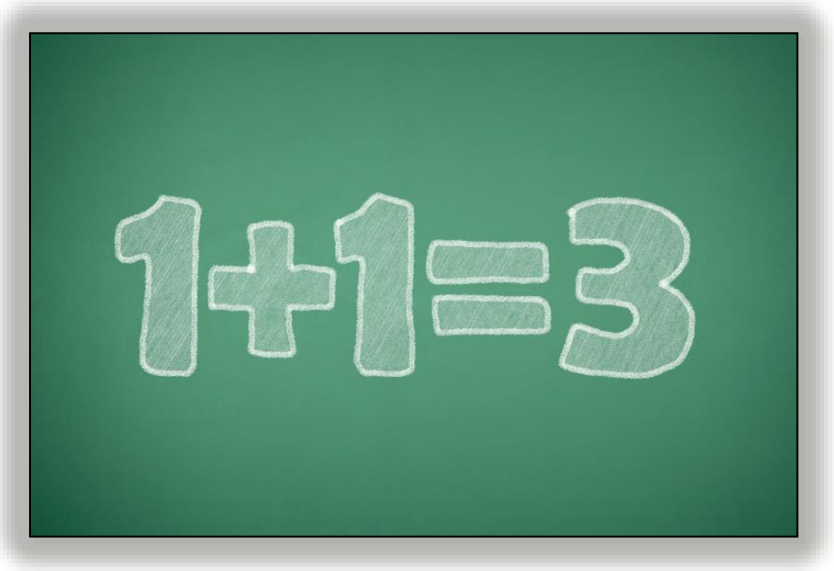


MES High Level Process

- Setting up the Model
  - Setup the complete physical event within the model using loads, boundary conditions, contact, and etc.
- Analyzing the Model
  - Solve the simulation frame by frame for the duration of the event. *(Note: Computer Resources Required)*
- Results Evaluation
  - Examine the results of the analysis throughout the timeline of the study.

What is Being Introduced in MES

- Additional Element Types
- Joint Components
- Load Curves
- Gravity



Additional Element Types

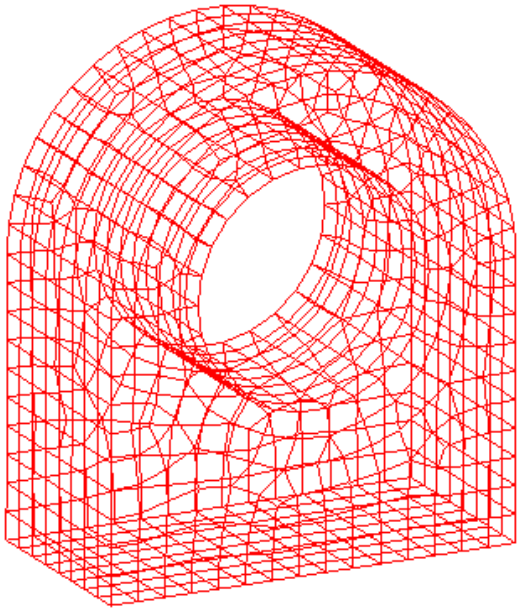
The following additional element types apply to the MES environment.

- Beam / Truss
- Spring
- Actuator
- Pulley
- Slider
- 3-D Kinematic

3D Kinematic Element

3D Kinematic elements do not experience strains and as a result do not report or calculate stresses.

- Elements Have Mass
- Apply Loads
- Experience Motion
- Acts like a Rigid Body



Spring Element

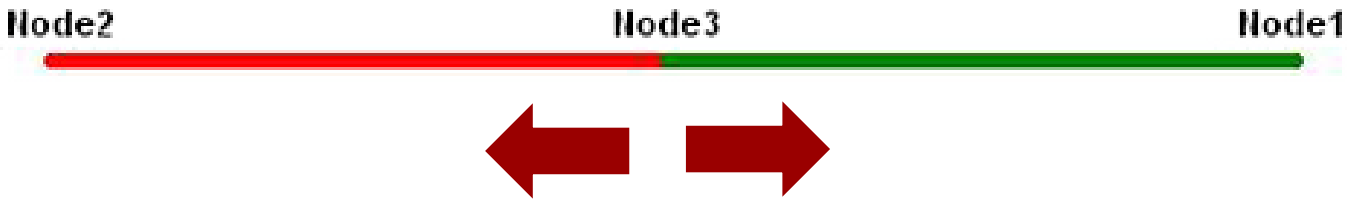
A spring element can connect two nodes on parts with a defined stiffness value.

- Axial or Rotational
- Displays as a Line or 3D Object

Slider Element

The slider element is used to translation a single node along a defined axis.

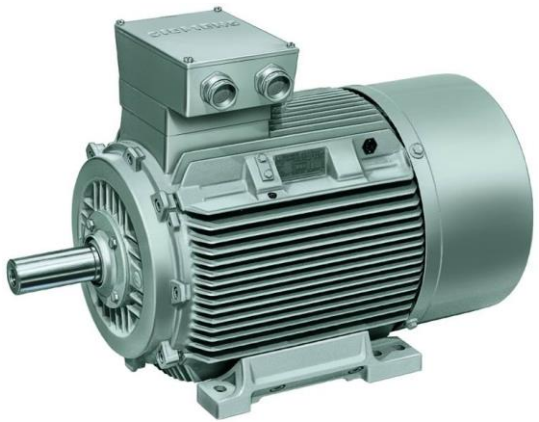
- Requires 3 Connection Nodes
- Node 3 Slides between Node 1 and 2
- Two Lines must be Connect and Parallel
- Slider Elements can Attach to Beam, Truss, 2D, Brick



Actuator Element

An actuator element is a two-node element that can either change length or rotate during an analysis.

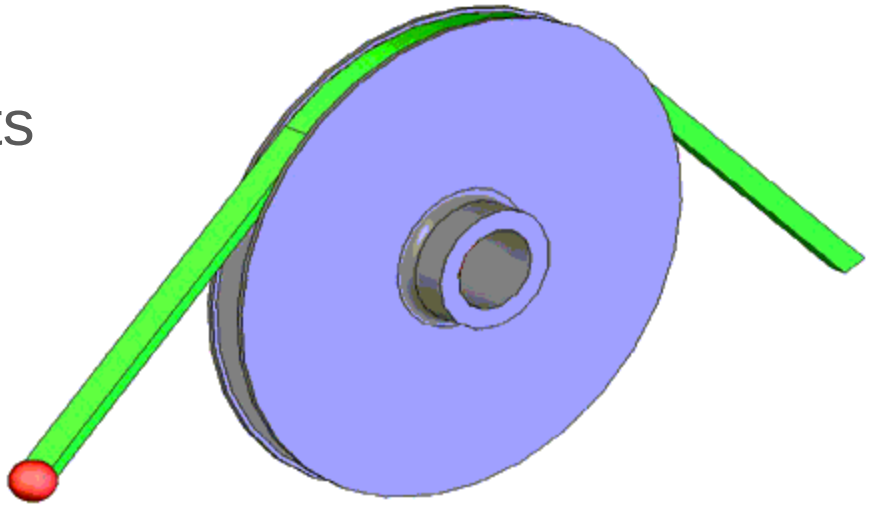
- Uses a load curve to control the action.



Pulley Element

The pulley element is used to represent simple pulleys to rotate objects in most cases.

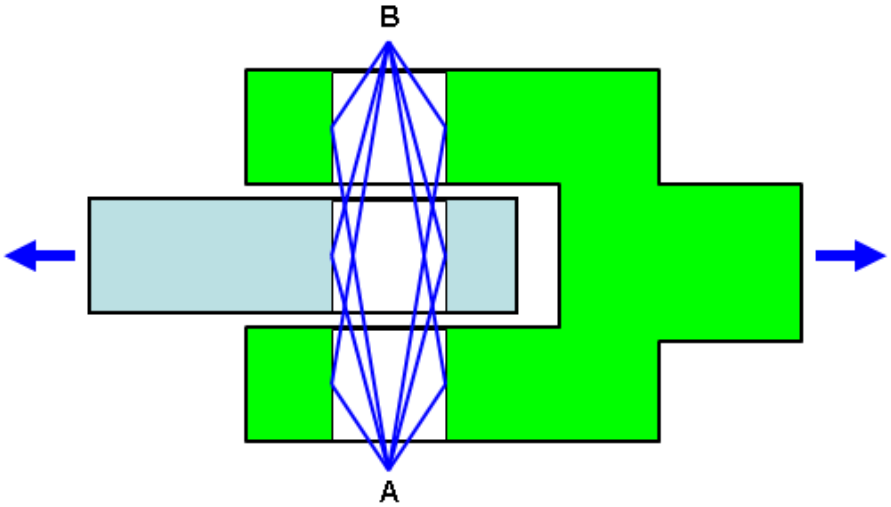
- Consists of Three Nodes (Driver, Pivot, and Slack)
- Can Stay Stationary
- Can Move with Other Parts



Joint Component

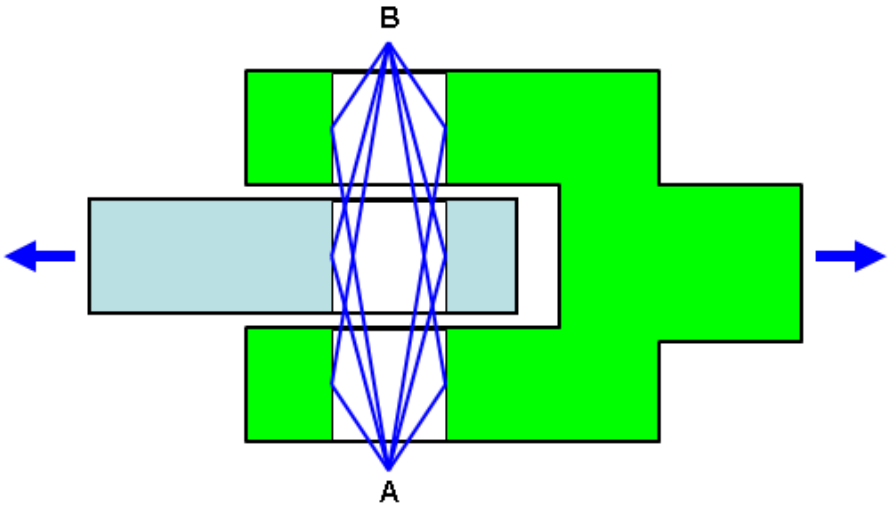
Joint components are used to simulate joints and pinned connections for rotational purposes.

- Pin Joint
  - Lines to Axis Endpoints
- Universal Joint
  - Lines to Axis Midpoint



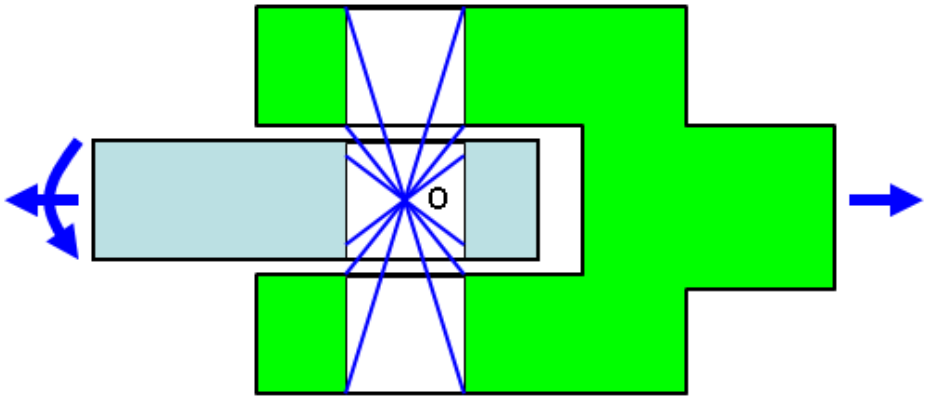
Pin Joints

- Used instead of modeling a true 3D pin part to allow rotation about the defined axis.
- Creates truss elements that use pinned connections to allow the bodies to rotate.



Universal Joint

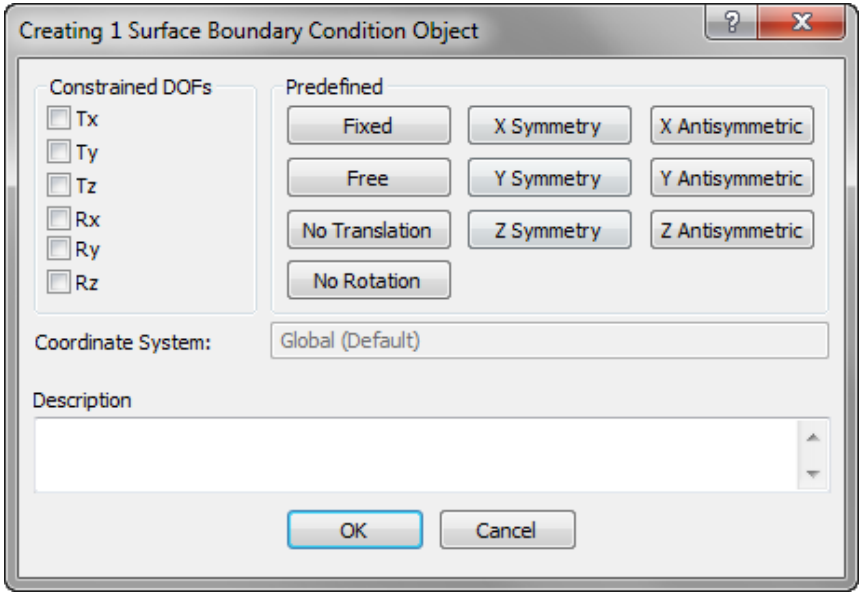
- Creates a part that allows the model to rotate about the axis and swivel about the center point of the axis.
- All of the space truss elements connect to the center.



General Constraint Boundary Condition

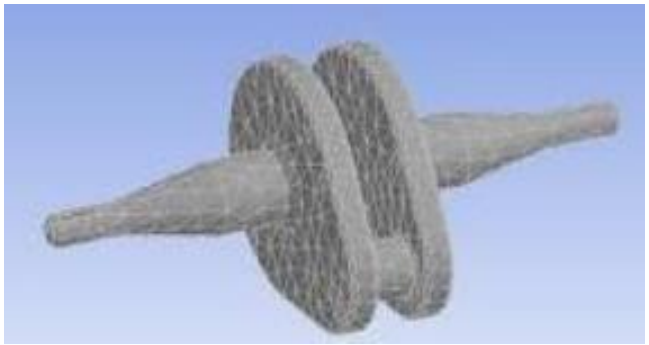
The General Constraint establishes boundary conditions to constrain DOF's.

Note:
Similar to the Autodesk Inventor Professional standard joint features in Dynamic Simulation.



Model Mesh Control

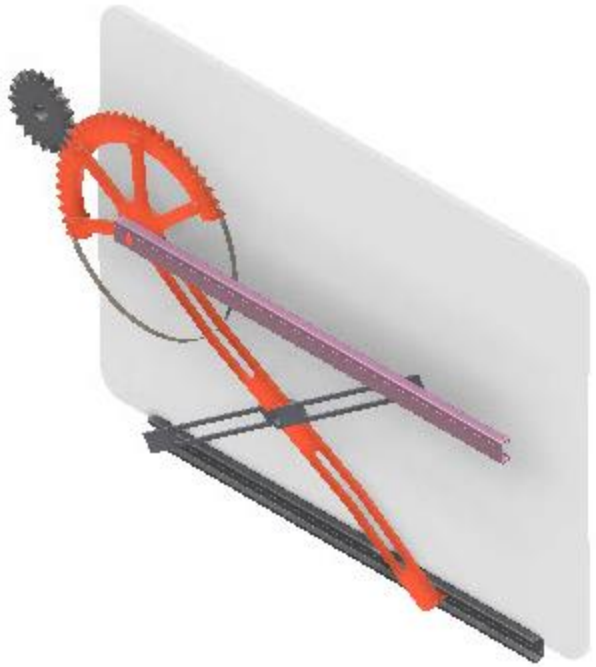
- CAD model mesh options can be controlled at a part level override.
- This provides the ability to streamline the mesh for the most optimal results for larger assembly based models.



Computer-Cluster Projects (CP12)

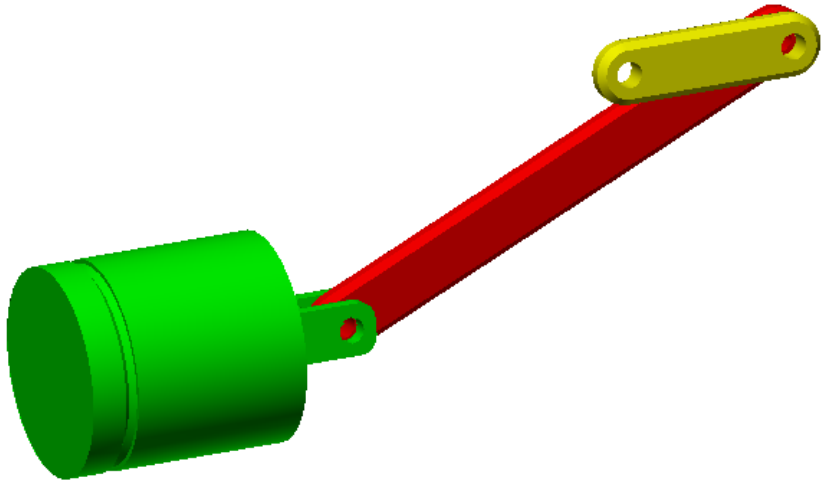
Guided Lab Project 1

Guides instructions for transferring loads from a Dynamic Simulation to a linear FEA.



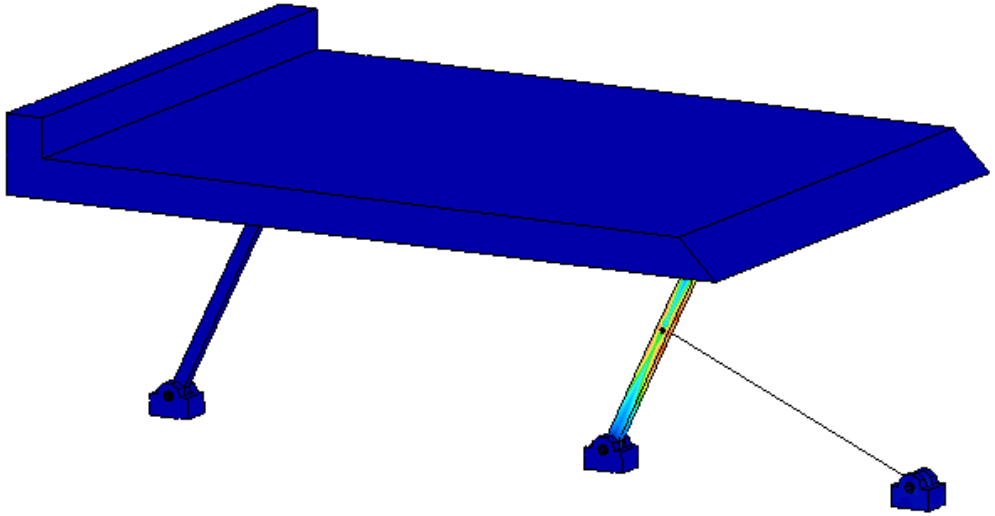
Guided Lab Project 2

Guided instructions for simulating a piston and crank using Mechanical Event Simulation (MES) with nonlinear materials.



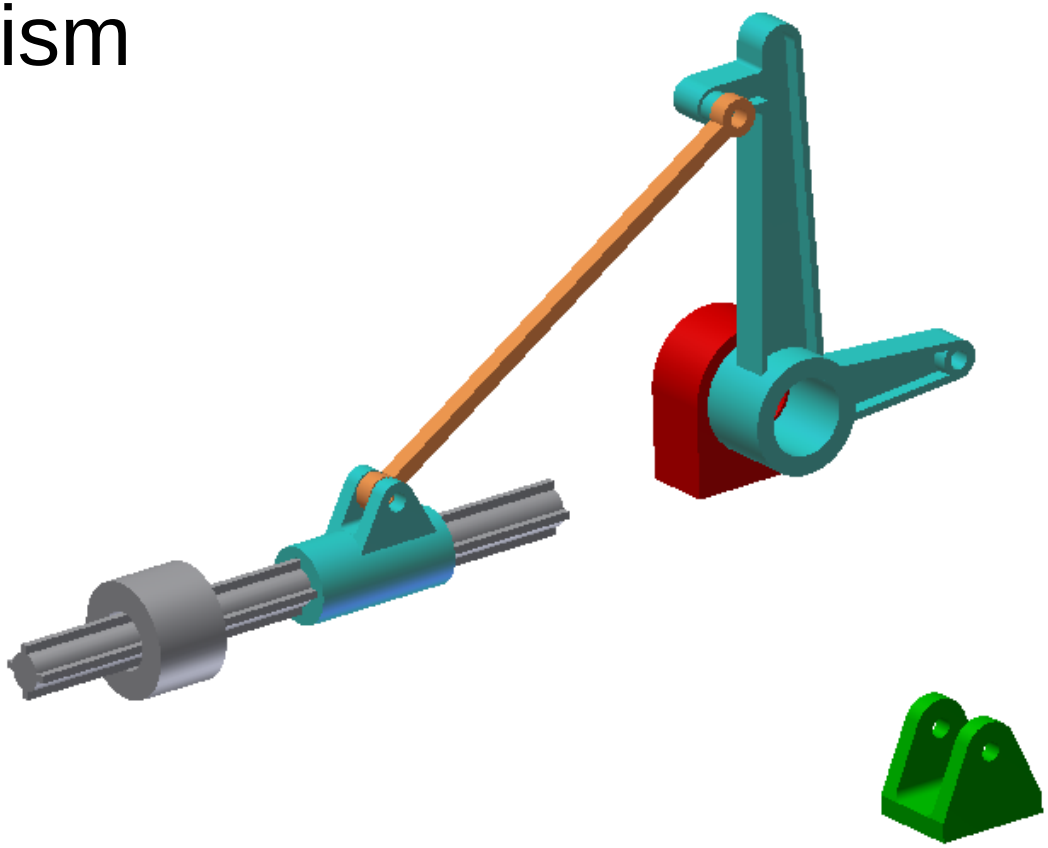
Guided Lab Project 3

Guided instructions for completing a MES analysis with additional element types and joints.



Problem Set Assignment

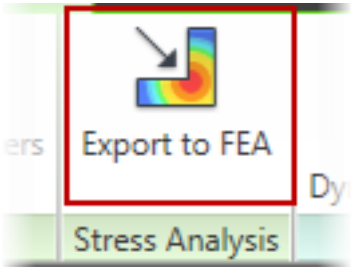
Conceptual Design Validation of Crank Slider Mechanism

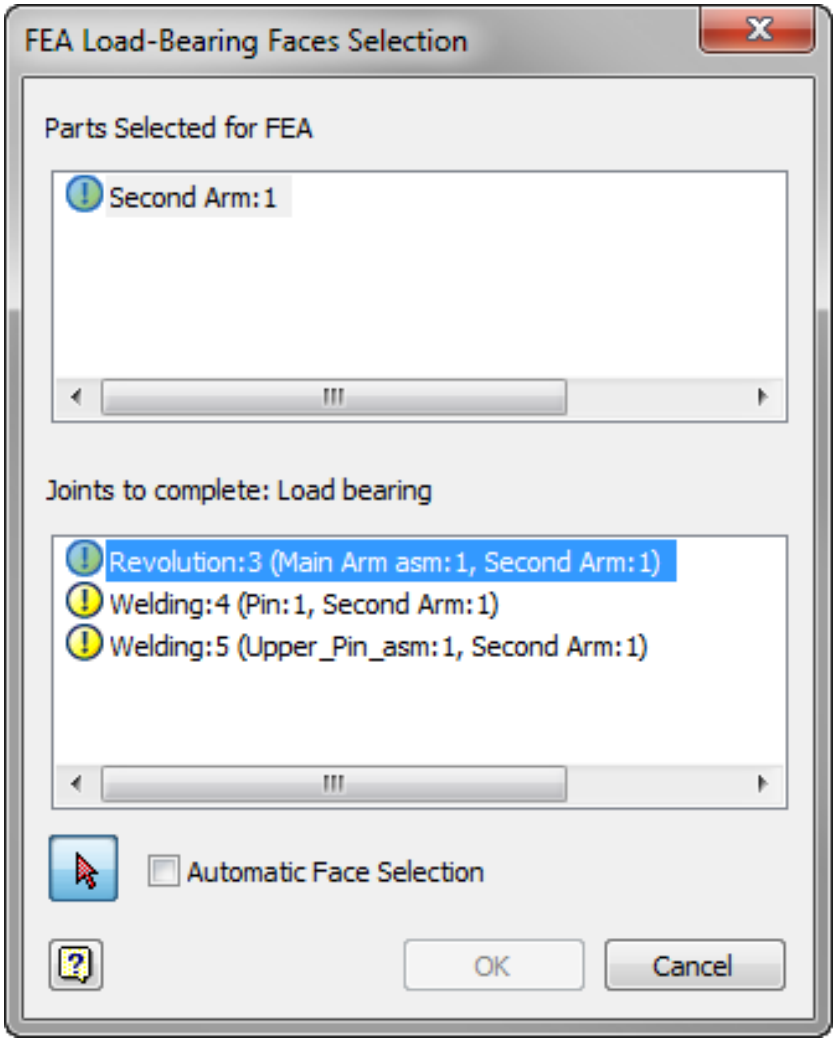


Demo Topics

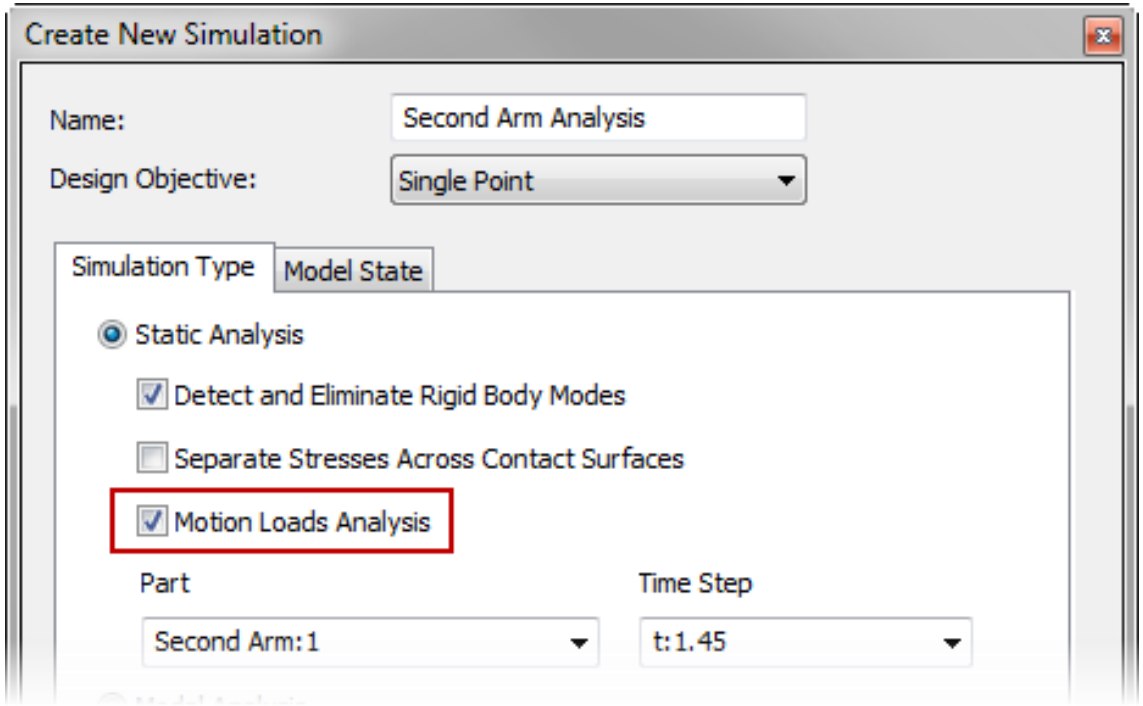
Exporting Dynamic Simulation Results

Time (s)	Force (Revoluti…	Force (Welding:…
1.45000	161.21000	80.39210
1.46667	161.20900	80.39160
1.48333	161.19900	80.38640
1.50000	161.17923	80.37650

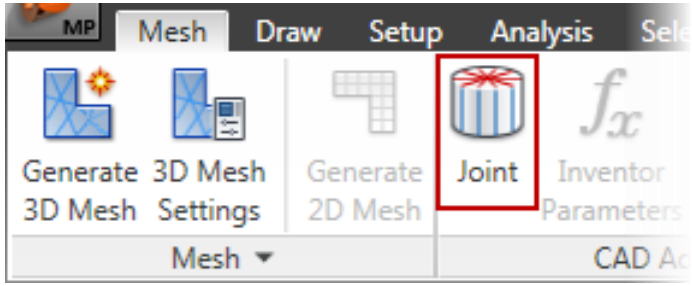


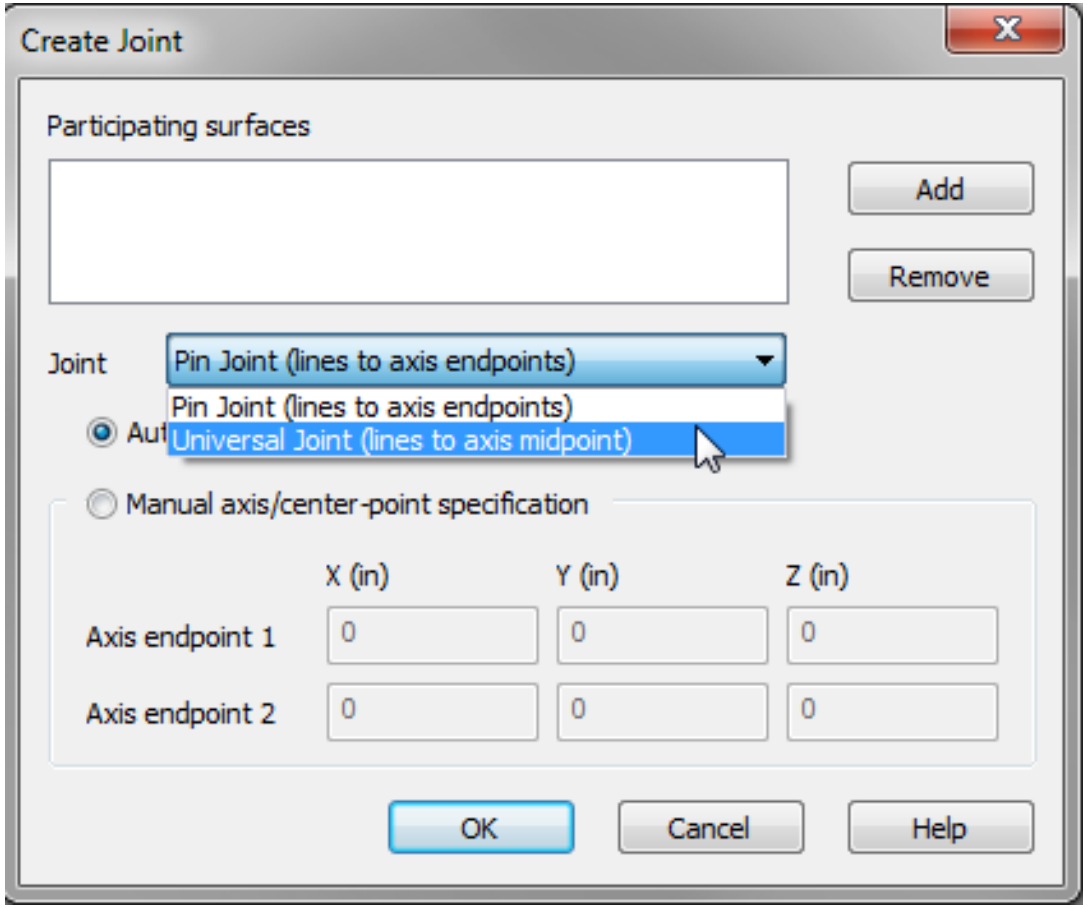


Create Simulation with Motion Loads

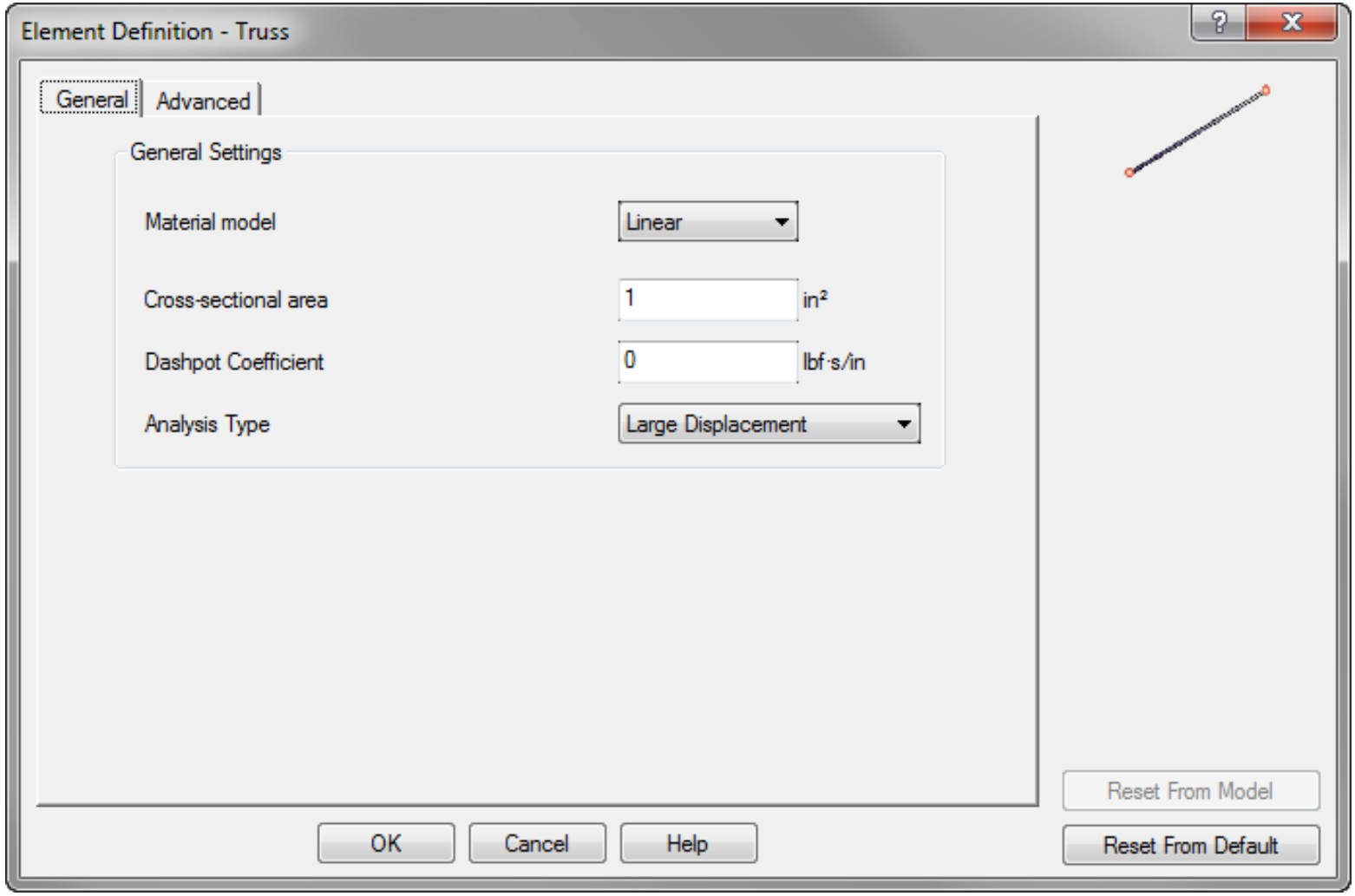


Pin and Universal Joints

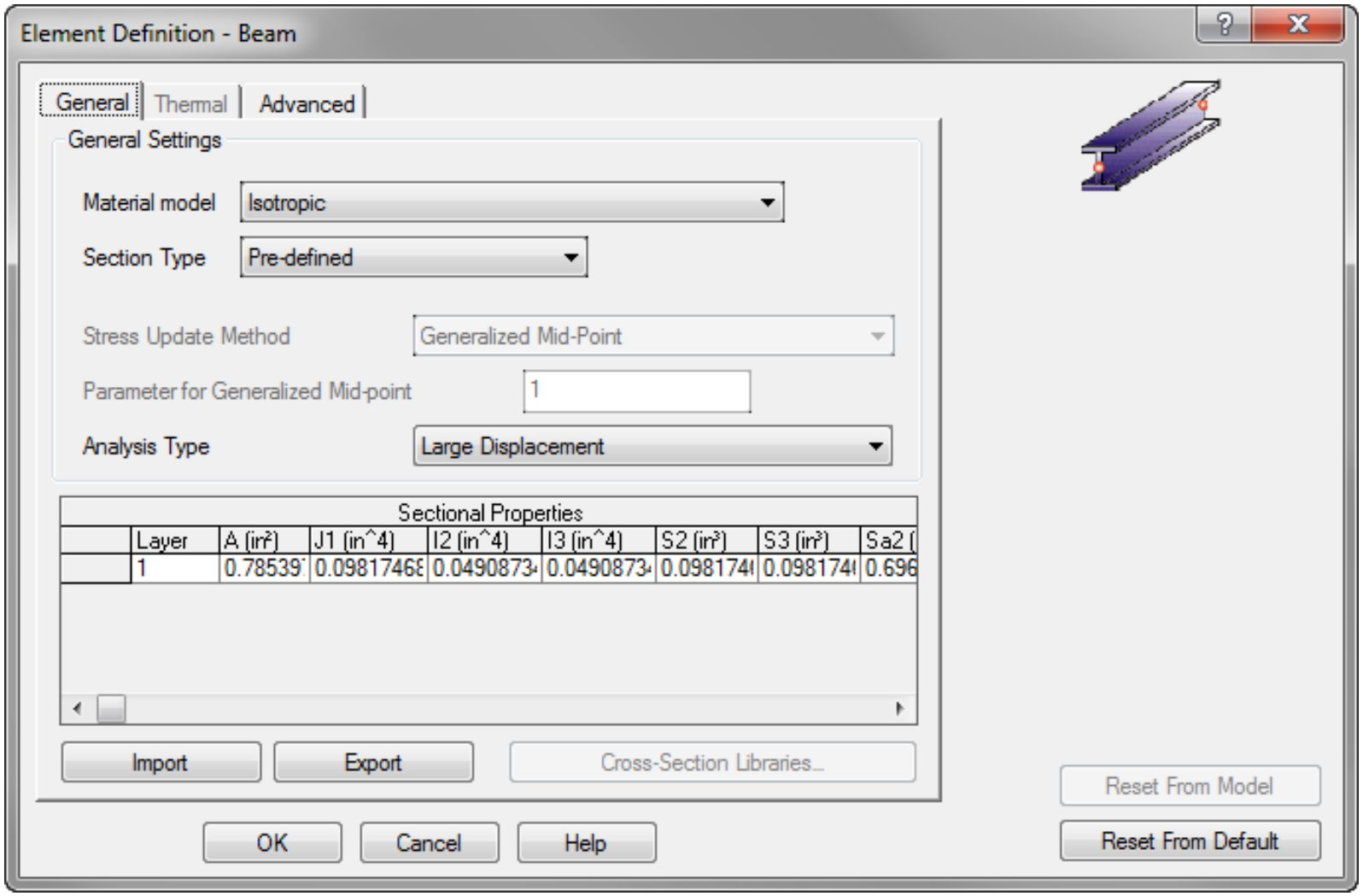




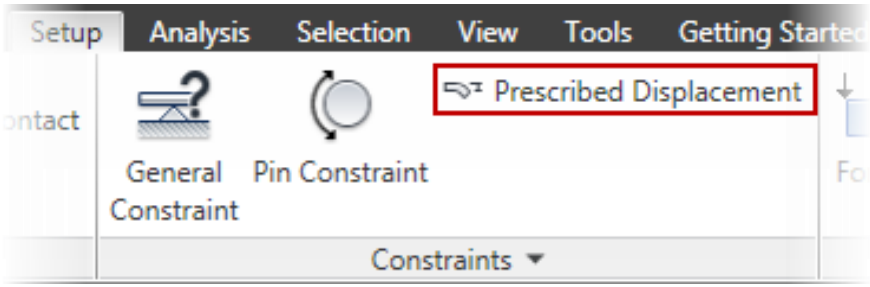
Truss Elements

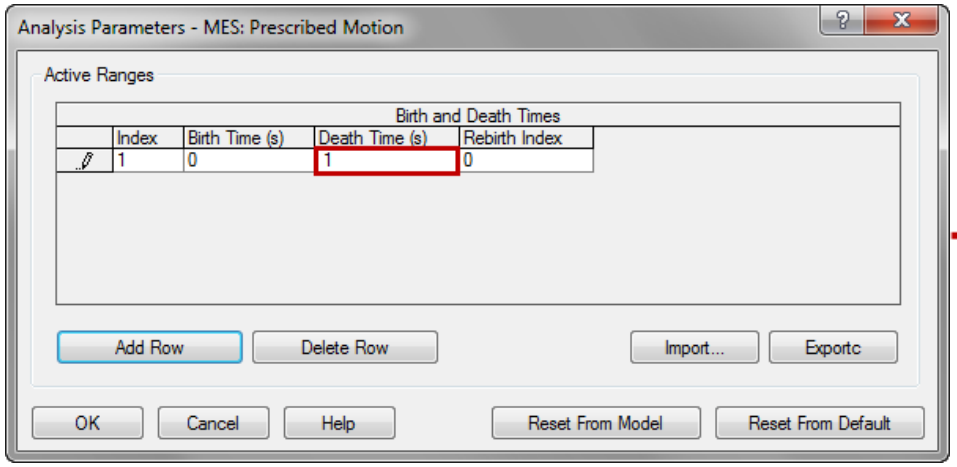


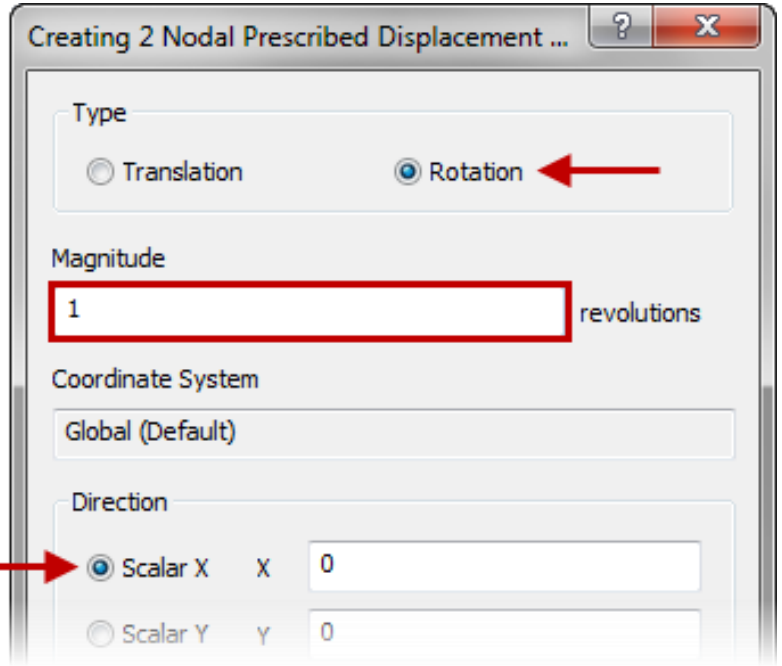
Beam Elements



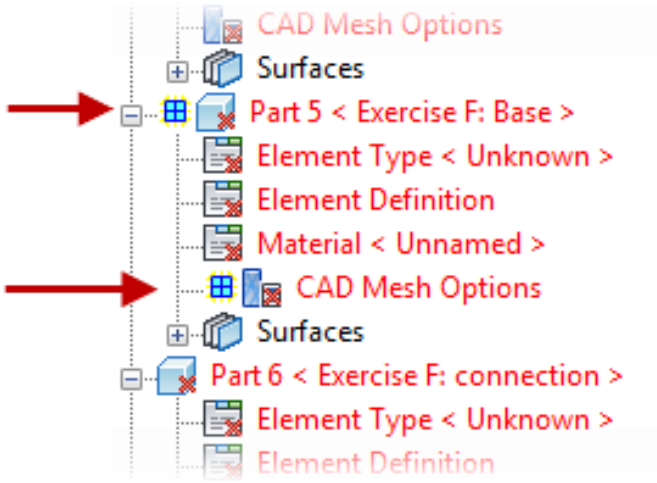
Rotational Prescribed Displacement

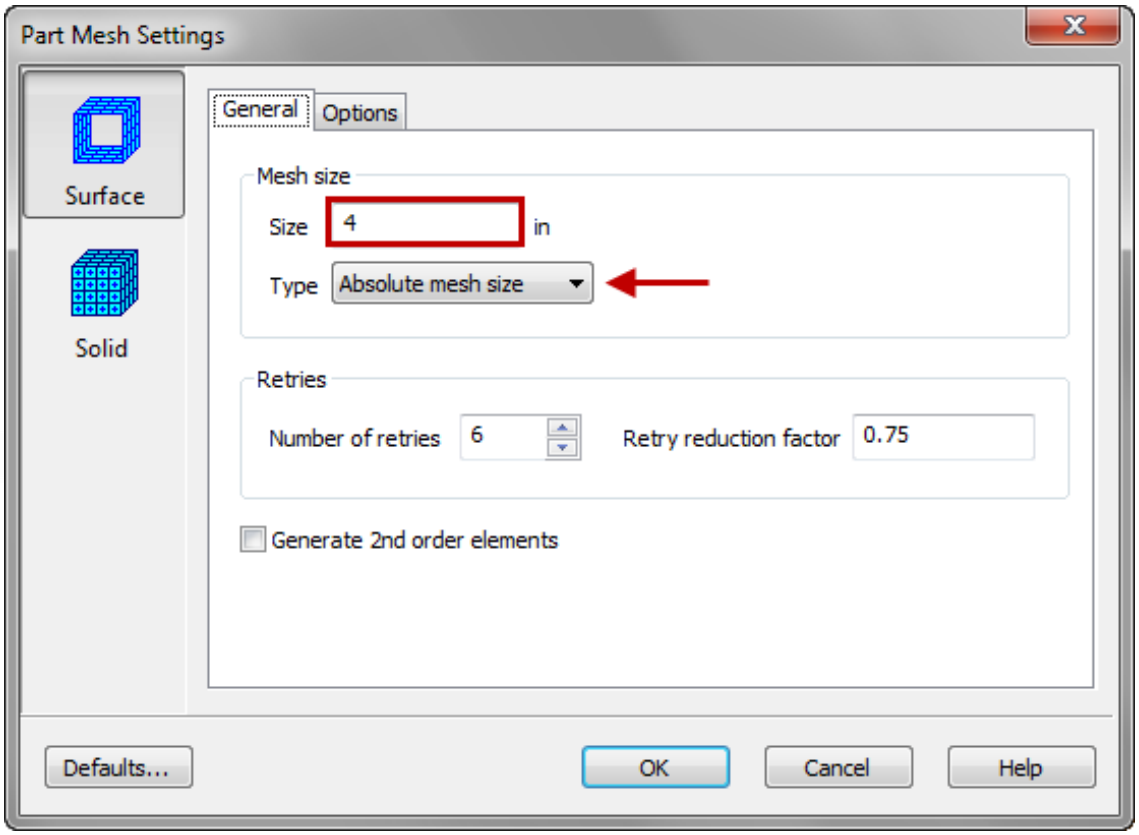




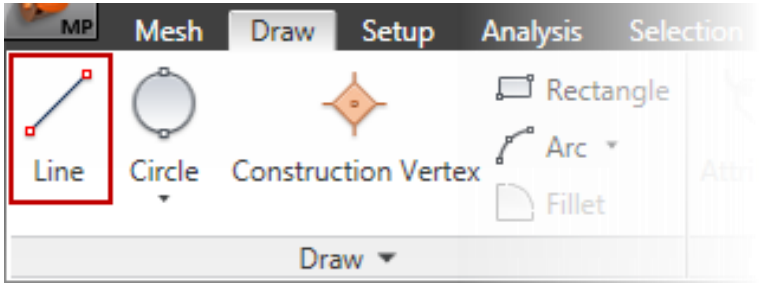


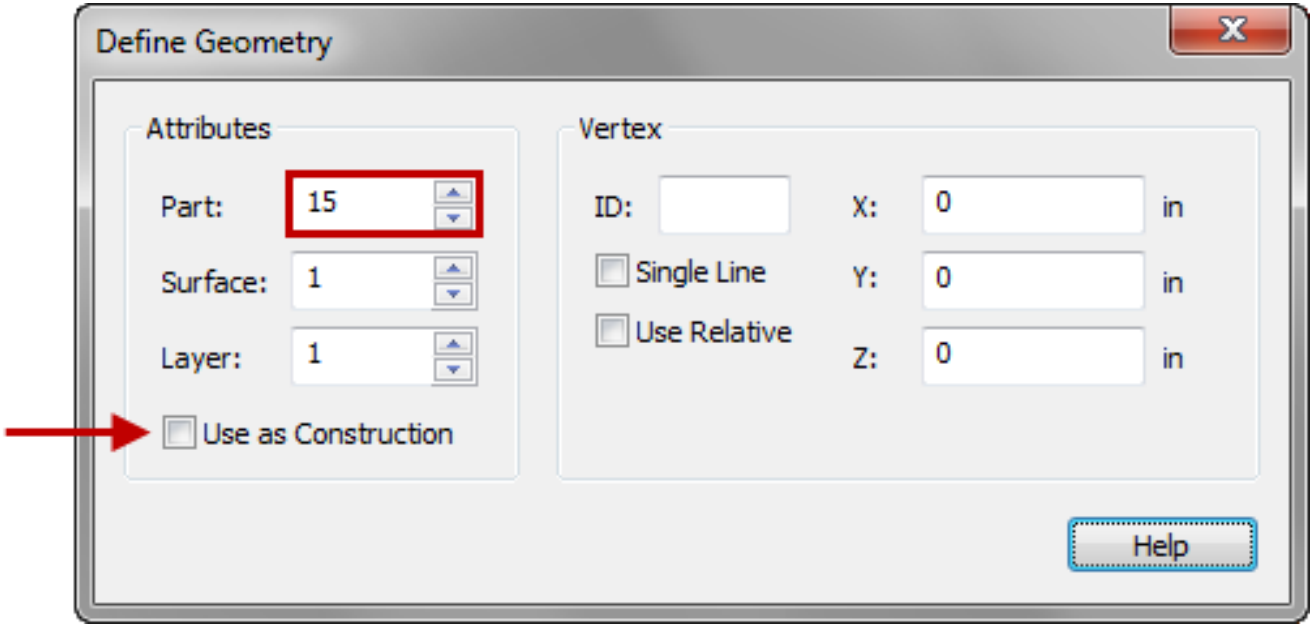
Part Mesh Settings





Draw Line


MP
Mesh
Draw
Setup
Analysis
Selection
Line
Circle
Construction Vertex
Rectangle
Arc
Fillet
Draw


Define Geometry
Attributes
Part:
15
Surface:
1
Layer:
1
Use as Construction
Vertex
ID:
Single Line
Use Relative
X:
0
in
Y:
0
in
Z:
0
in
Help

Actuator Elements