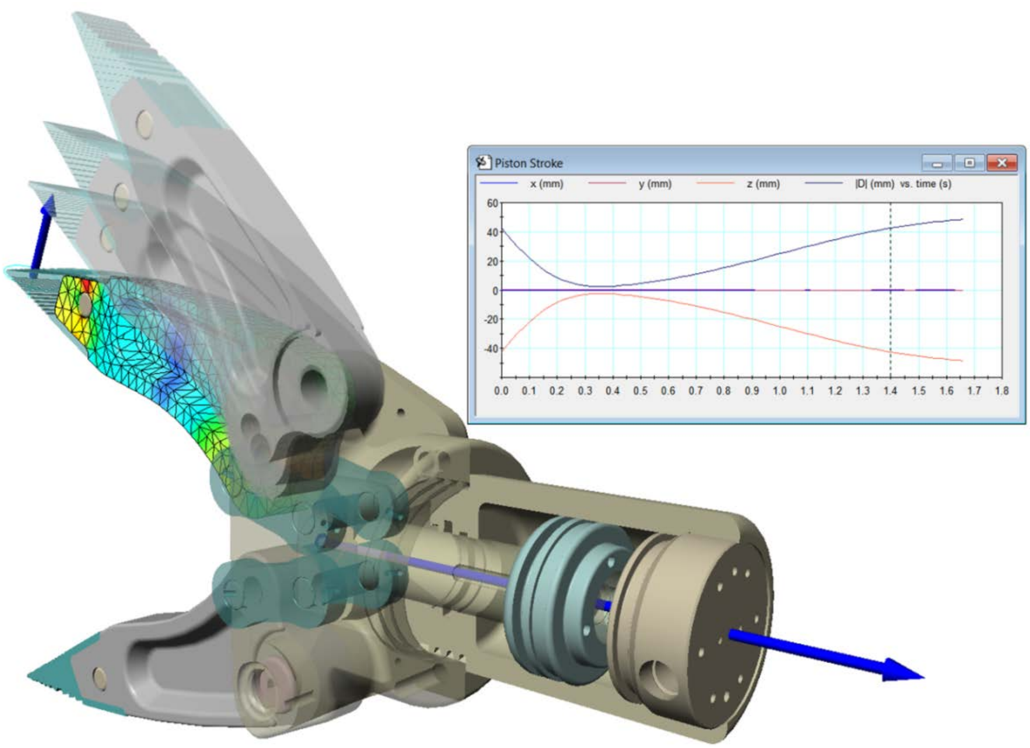
User Training Resource

SimWise 4D

Click to Get Started



Design Simulation Technologies

Welcome

This training resource guide is intended to serve both as a hands-on learning tool in becoming proficient with SimWise and also as a on-demand reference guide for accessing information on a specific feature or topic. Additional help can be found in the program under the Help menu.

The exercises that are blended within the content of this guide may be completed in any order desired, However, to get the most benefit from the exercises, it is suggested you start at the beginning of this document, read each topic and do the exercises in the order listed.

Use the Back , Forward and Table of Contents buttons to navigate around the guide. You can also use the scroll wheel on your mouse or the keyboard arrow keys.

Navigation tools

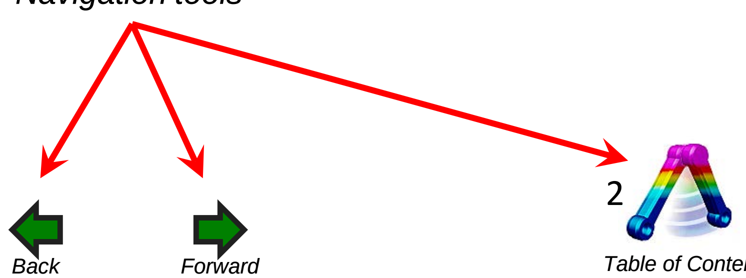


Table of Contents I

Table of Contents II

Table of Contents III

Simulation Settings

FEA Modeling

Table of Contents IV

Exercises

Click on an exercise below to see what is covered in that exercise.

Appendix A

Using the documentation

- There are two products that make up the motion simulation and FEA tools in SimWise: **SimWise Motion** and **SimWise 4D**. This training document and all images within it were developed using SimWise 4D. If you are using SimWise Motion, you may see subtle differences in a few dialog boxes, butnot enough to affect the training experience.

- In most instances there is more than one way to perform a task or access a feature. For example, there are at least four different ways to create a constraint. The approach shown in the exercises may not be the only way to complete a task. Refer to the non-exercise content to review all possible options.
- Commands prompting for user action are listed in bold type. For example, **right-select**, **choose** and **drag**.

- Program feature names and objects that are to be accessed by the User are listed in bold blue type. For example, **Structural Load, Face Normal** and **Solve FEA.**

- General notes and tips, in smaller bold type, are found throughout the example problems. "Notes" give additional information or clarity on the particular task being performed. "Tips" offer alternative ways or short cuts to accomplish a task.

- You may find it easier and faster to manipulate the viewing of your model by using the mouse and keyboard. Shown below are standard commands while using a mouse with a scroll wheel:

  Pan: Ctrl + press & hold mouse scroll button
  Zoom: Crtl +Shift + press & hold mouse scroll button
  Rotate: Press and hold mouse scroll button

Product Overview

- ***SimWise Motion***
  - 3D Motion Simulation that evaluates the kinematic and dynamic performance of assemblies.

- ***SimWise FEA***
  - 3D FEA which evaluates structural characteristics such as stress and deflection.
  - Additional FEA analyses:
    - Linear-elastic Stress and Deflection (Static)
    - Steady State Thermal
    - Natural Frequency & Mode shapes
    - Linear-elastic Buckling

- ***SimWise 4D (Motion +FEA)***
  - Simultaneous solution of 3D motion and FEA.
  - A quasi-static stress analysis is performed at each motion frame using the dynamic loads calculated by the motion simulation. Product can be used for motion-only, FEA-only, or both.

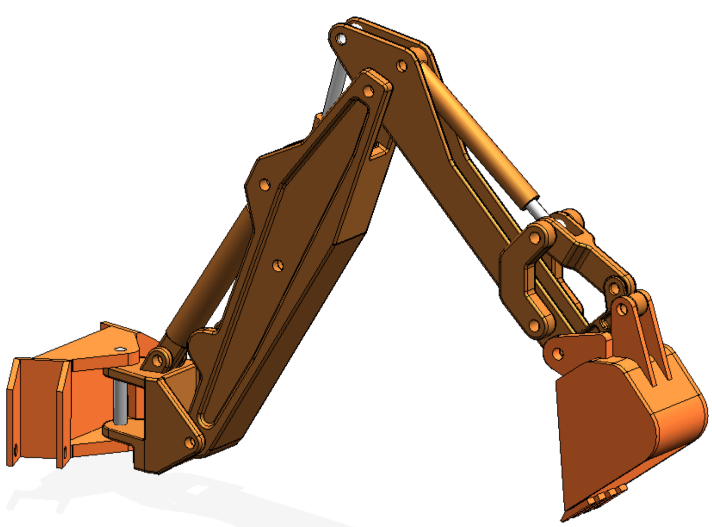
Classification of Mechanisms - Kinematics

Kinematic System

• System with 0 degrees of freedom (DOF)

• Regardless of the system's mass , gravity, inertia, and externally applied forces, this system is still restricted to a given range of motion.

• Kinematics is concerned with the motion of objects, such as displacement, velocity and acceleration, but **not** the forces that cause the motion



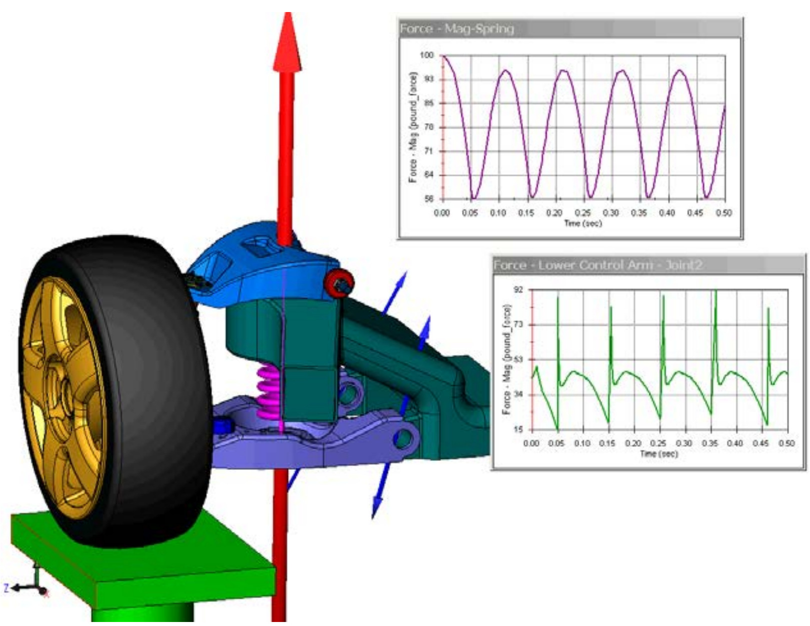
This system has 0 DOF, when motions are included on the actuators. Regardless of the mass of the system, the links will always move through a given (or defined) range of motion

Dynamic System

• System with more than 0 degrees of freedom (DOF)

• The system's mass , gravity, inertia, and externally applied forces will govern the time-response of the system and how the system can move

• Dynamics is concerned with the motion of objects **and** the forces that cause the motion



This system has more than 0 DOF. The automobile frame and wheel experience different motions due to the presence of the spring and shock. The movement of the spring and shock is governed by the accelerations and mass of the system. If the mass of the system was changed, the motion would also change.

Supported File Types (CAD Associativity)

- Alibre Design have plug-in or associative data support with SimWise

- User initiates data transfer from within CAD system

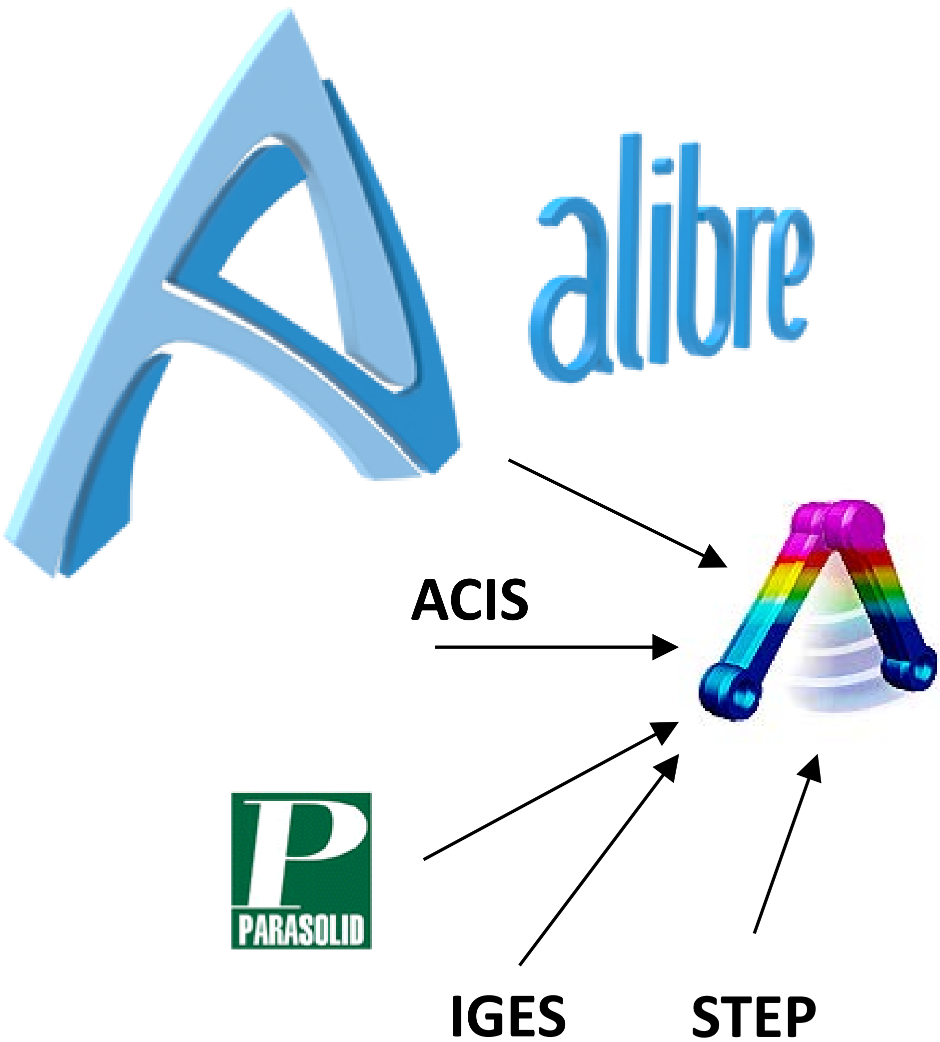
- Geometry and constraints are transferred to SimWise

- Constraints are mapped to corresponding motion constraints

- If the CAD model is updated, only the changes are transferred back to the motion model

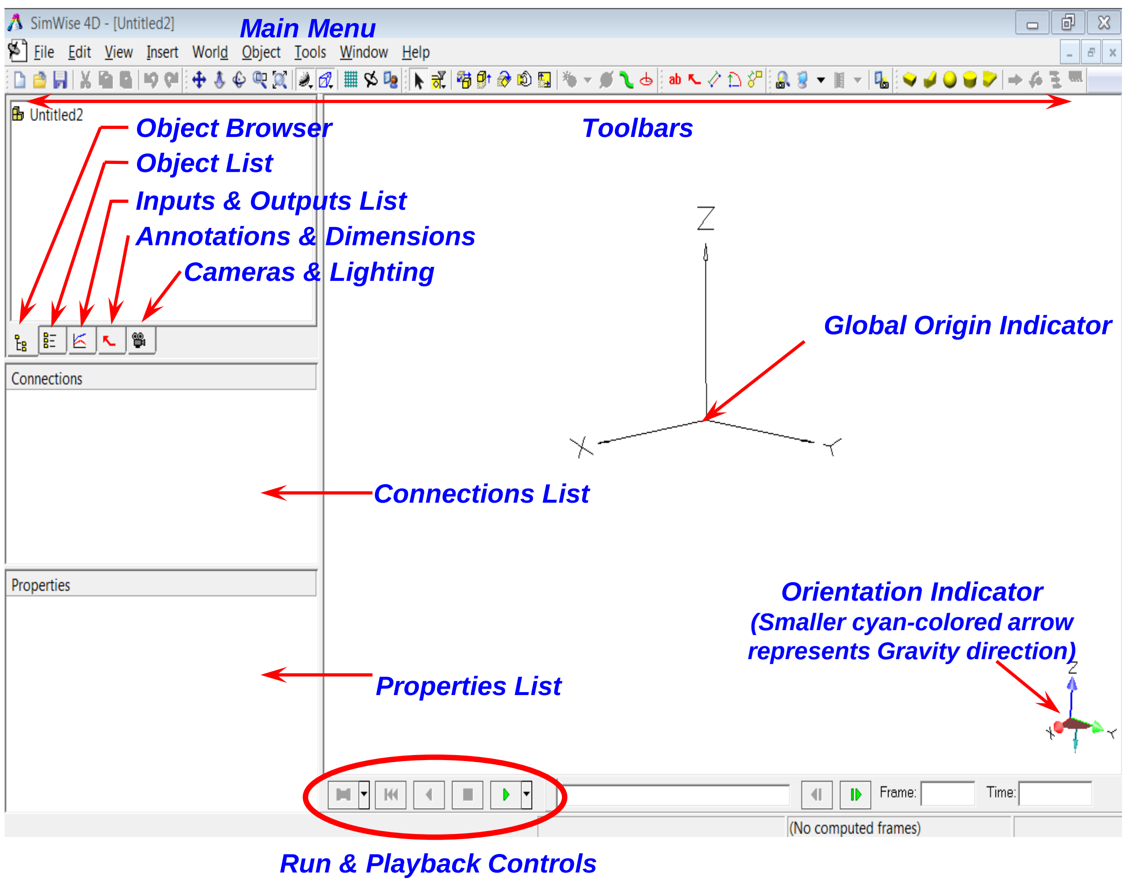
Supported File Types (Direct File Read)

- The following file formats can be read directly into SimWise



SimWise does not scale files. All units and correct dimensions must be set in the CAD system before exporting the above neutral filetypes and reading them into SimWise.

User Interface (Overview)



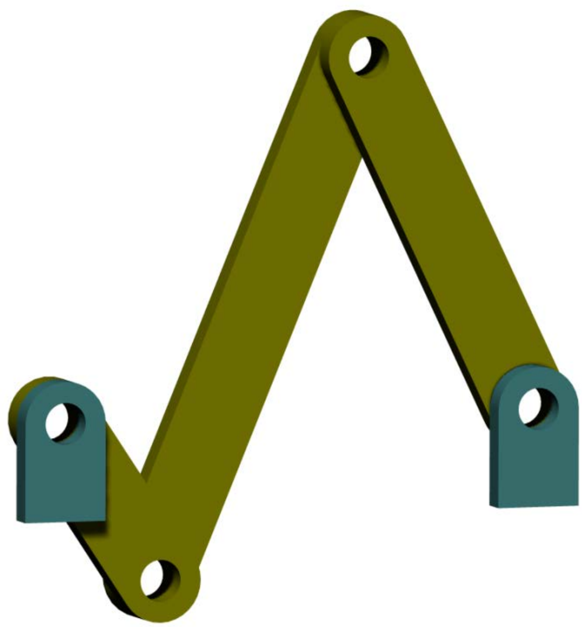
Exercise - FourBar

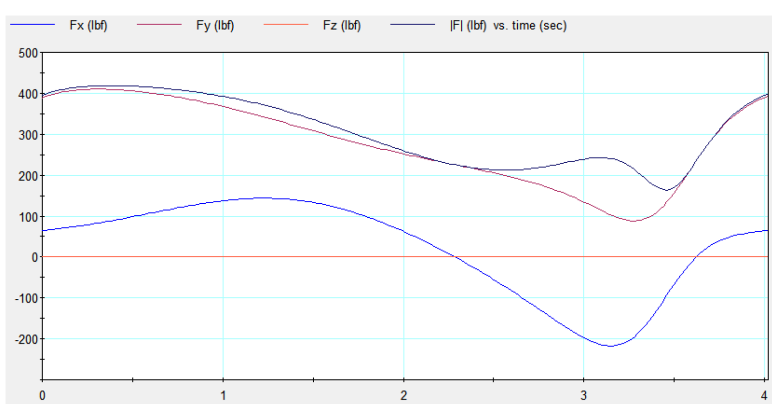
Simulation Objectives:

- *Get familiarized with the SimWise user interface*
- *Run a basic motion simulation*

Features Covered:

- *Gravity*
- *Unit Settings*
- *Fixing bodies*
- *Add a motor*
- *Running a Simulation*
- *Creating a Meter*





Open the SimWise file

1. ***Start SimWise***

2. *Select **File, Open** and **browse and locate** the file called "**SimWiseTutorial – Fourbar.wm3**".*

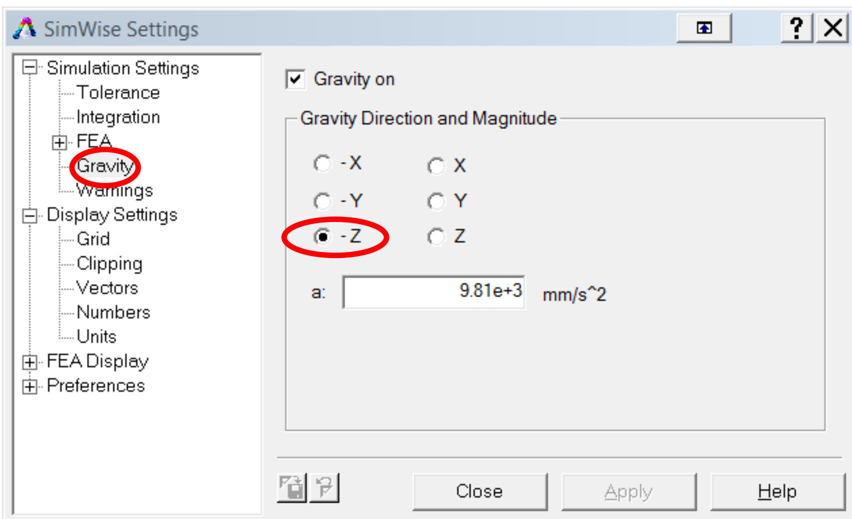
3. ***Select Open***

There are three links and two ground blocks. Constraints have already been added.

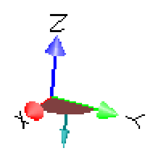
Change Gravity and Unit Settings

1. ***Select** the **Simulation Settings** icon*

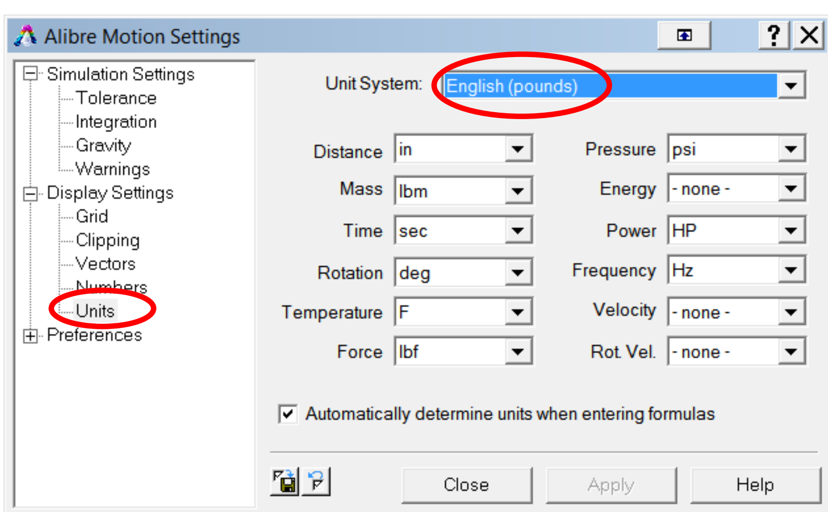
2. ***Select Gravity** from the settings list and **click on –Z.** Leave the **Settings d-box** open.*



***Note:** The cyan colored arrow on the Orientation Indicator represents the direction of gravity*



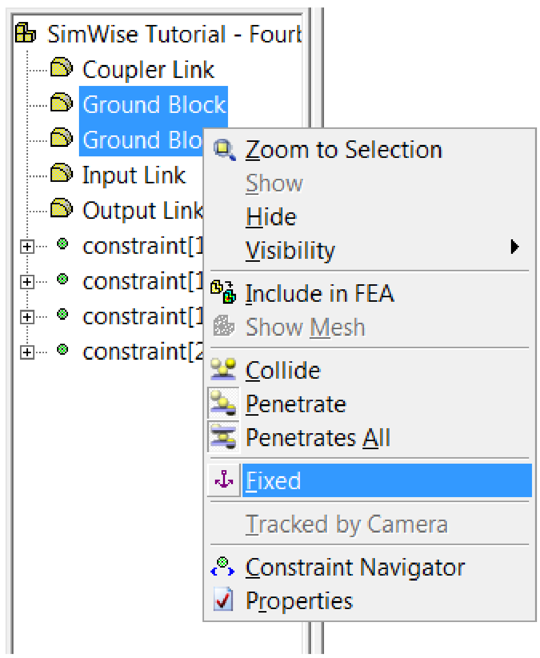
3. ***Select** the **Units** option from the settings list, **select** the **drop down** menu next to **Unit System** and **select English(pounds)**. **Close** the d-box when finished.*



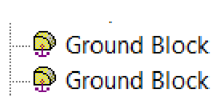
Fix Ground Blocks to Background

1. ***Hold down** the **Ctrl key** and **select** both ground blocks (either in the Object Browser or in the graphics area)*

2. ***Release** the **Ctrl key**, **right-click** on either highlighted part, and **choose** **Fixed***

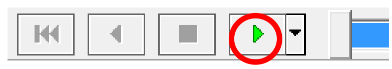


Bodies that are "fixed" will have a small anchor icon next to the body name in the browser



Run the Simulation

1. *To run the simulation,* ***select*** *the* ***Run button*** *located on the* ***Player Control Panel****.*



Run

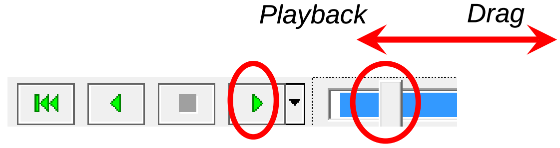
Note that the geometry simply falls with gravity.

2. *The Run button will changeover to a red Stop button.* ***Click*** *on the* ***Stop*** *button at any time to stop the simulation*

Stop

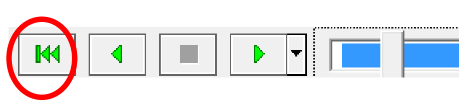
3. ***Playback*** *the results of the simulation by* ***clicking, holding and dragging*** *the* ***slider button*** *next to the* ***Player*** *controls. You can also click on the* ***Play button***

Playback *Drag*



or

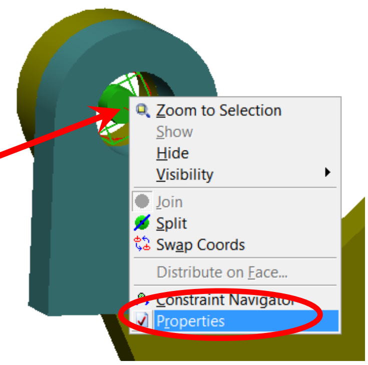
4. ***Select*** *the* ***Reset button*** *to reset the simulation back to the beginning*



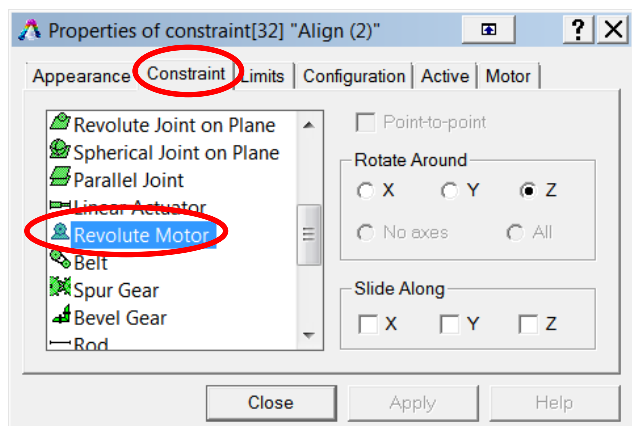
Reset

Add a Motor

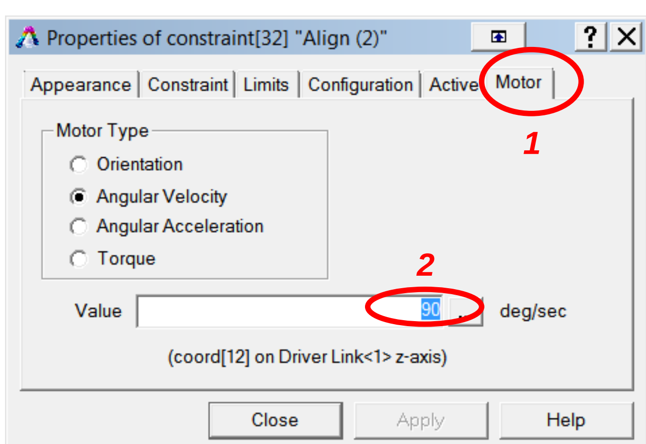
1. *In the graphics window, **right-click** on the constraint connecting the **Driver Link** to the **Ground Block** and **choose** **Properties** from the flyout menu. This will activate the **Properties dialog box**.*



2. *In the **Properties d-box**, **select** the **Constraint** tab and **scroll and select** the **Revolute Motor**. This will change the Revolute joint to a Motor constraint*

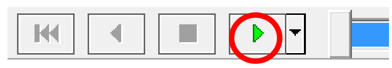


3. *Click on the **Motor tab** and **verify** the motor type is **Angular Velocity** and that the value is set to **90 deg/sec.** **Select** **Apply** and then **Close.***



Run the Simulation

1. ***Select** the **Run button** on the **Player Control Panel** to run the simulation .*

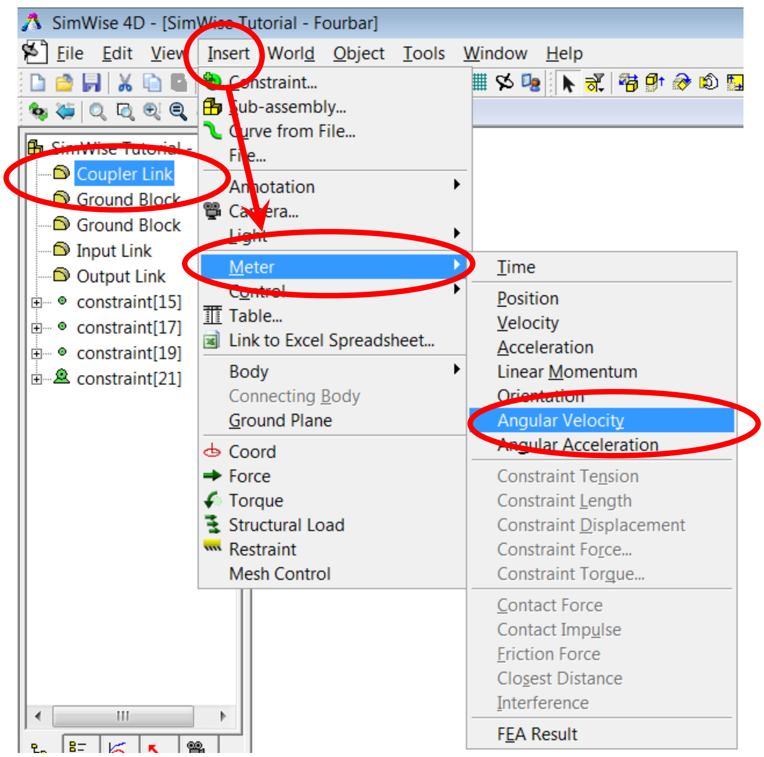


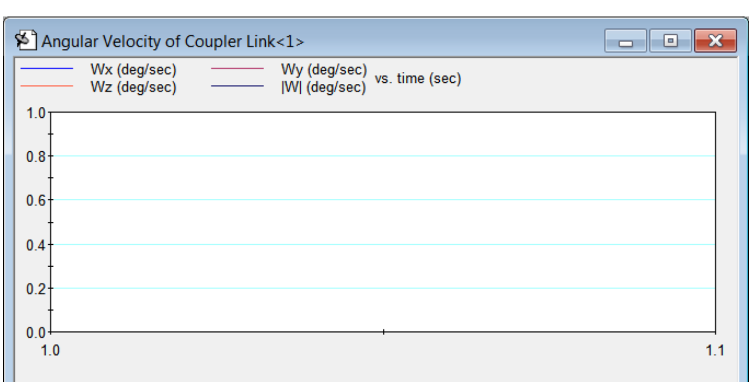
Run

2. ***Stop** and **reset** the simulation*

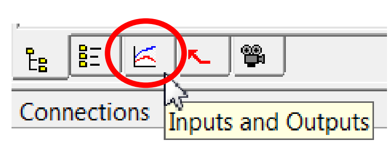
Add an Meter (Angular Velocity)

1. *In the **Object Browser**, **click on Coupler Link**, then **click on Insert**, **Meter**, **Angular Velocity**. A meter will be added below the graphics window.*



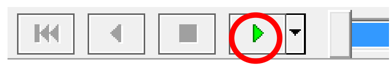


***Note:** You can click on the Inputs and Outputs tab located at the bottom of the Object Browser to see the new meter feature listed*

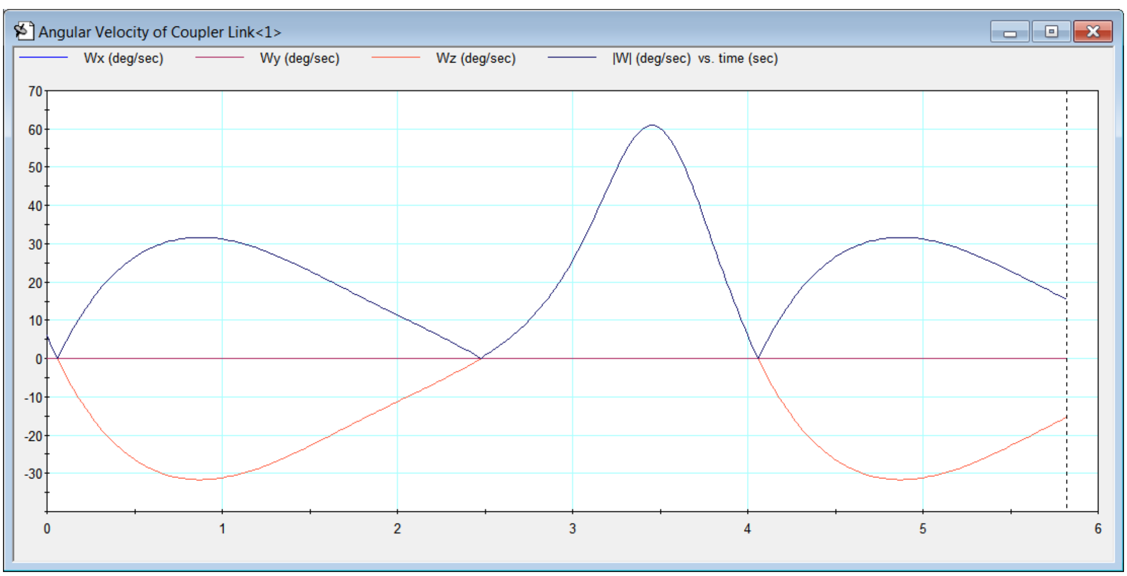


Run the Simulation

1. ***Run** the simulation and stop it any time.*



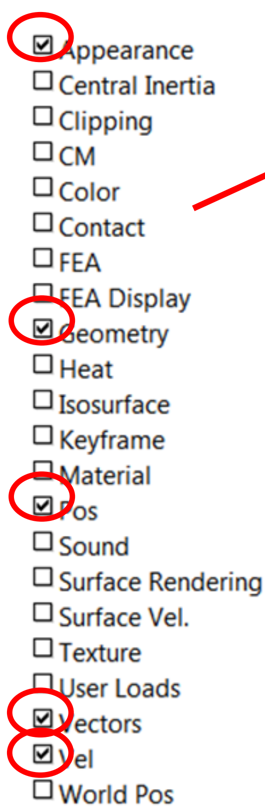
Run

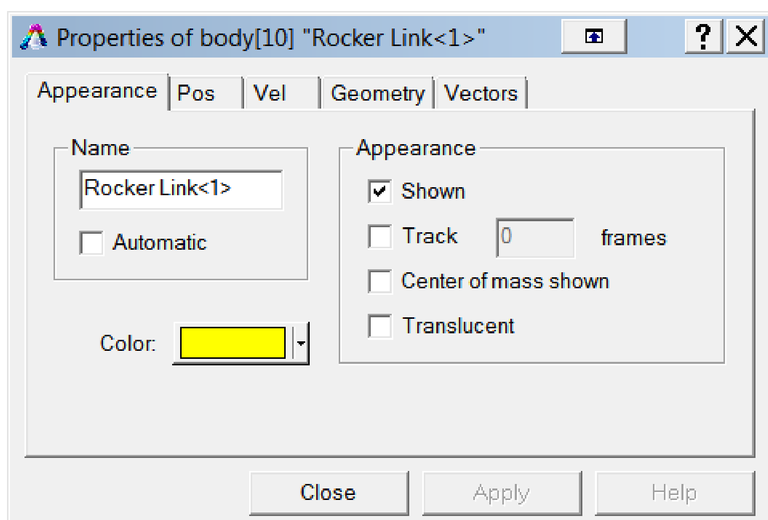


[*End of Exercise*]

The **Properties List** contains all options available for specific model elements. For example, bodies will have certain properties available that are specific to bodies, constraints will have certain properties available specific to constraints, and so on.

When a model element, such as a body or constraint, is selected in the graphics area or in the browser, all available properties for that element will display in the **Properties list**. If an item has a check mark next to it in the **Properties list**, that checked item will have a corresponding tab associated with it in the Properties d-box for the selected element.

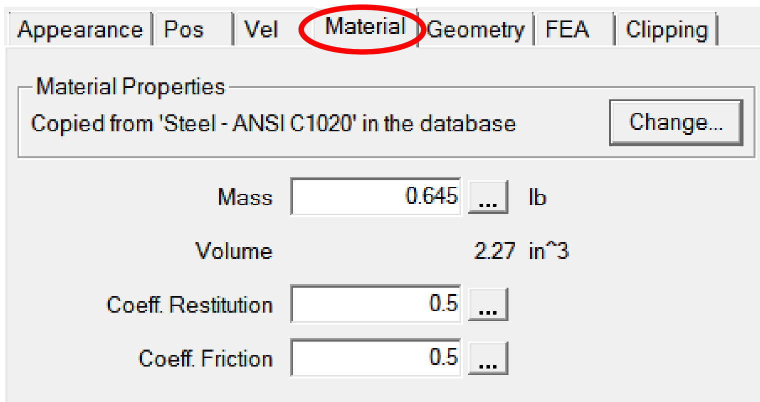




Tabs can be added and removed by simply checking or unchecking the appropriate item in the Properties List.

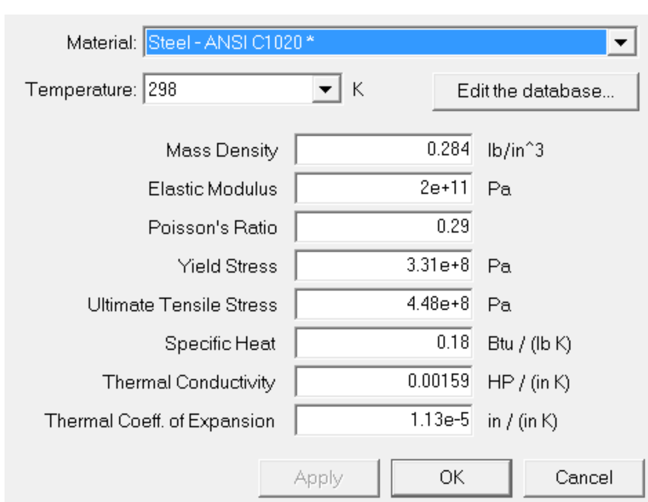
Bodies (Material)

By default, each body is assigned a material of 1020 steel, unless the model is linked to a CAD model via the plug-in. In this case, the CAD system materials will transfer. (



Material Properties d-box

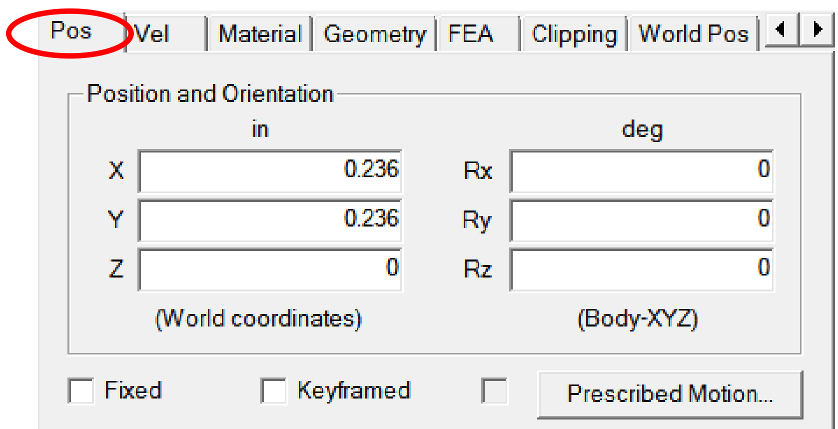
Selecting the **Change** button accesses the material library. Here the user can change a property of a material or use the **Edit the database** option and define a custom material.



Material Parameters d-box

Bodies *(Position and Orientation)*

The position and orientation of a body can be specified with respect to either 1) its own coordinate system (the part origin) or 2) the World coordinate system.



Position Settings d-box

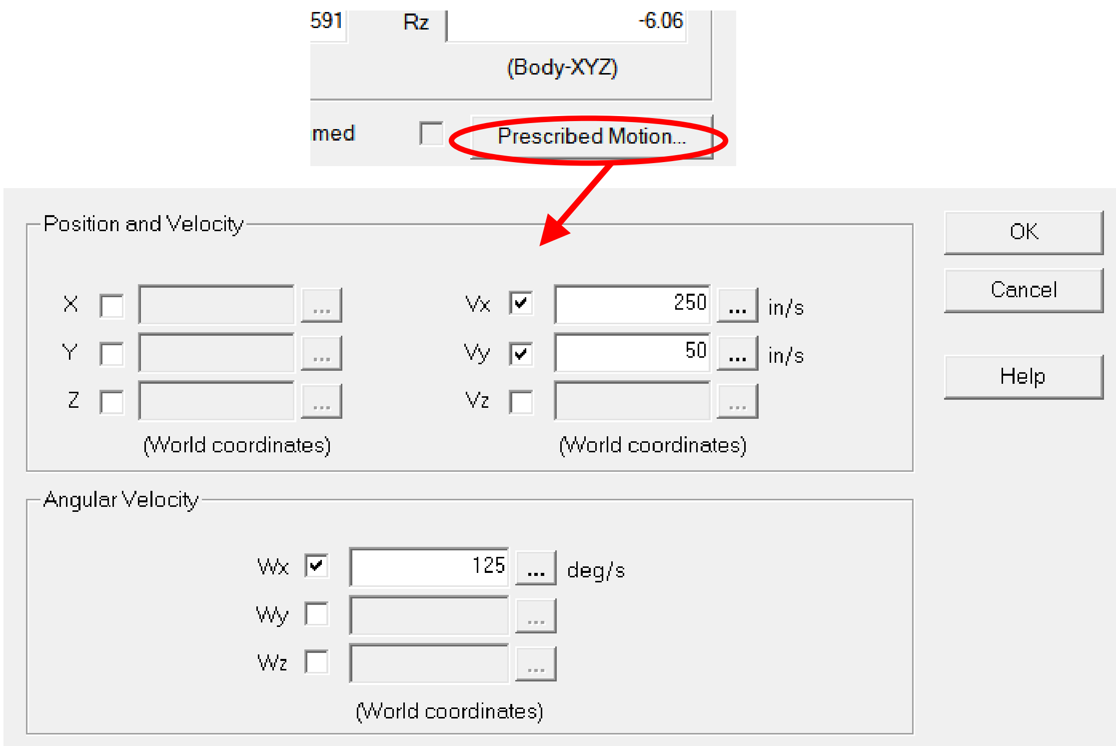
The ***Pos*** tab in the body Properties d-box brigs up the position and orientation in the body's coordinate system. The ***World Pos*** tab brings up the position and orientation in the Global (World) coordinate system

The **fixed** option specifies the part is fixed to the background.

The **Keyframed** option allows the user to control the motion of a body using defined displacements and rotations at specified frames. These specifications will override the simulation time. See Keyframing in the training guide for more information on using this feature.

Bodies *(Prescribed Motion)*

The **Prescribed Motion** feature, found under the Body **Pos** tab, allows the user to define inputs to a body using formulas and/or constants. For example, a body can be specified to move through space with a given translational velocity in two directions and a rotational velocity about one axis.



Prescribed Motion Settings d-box

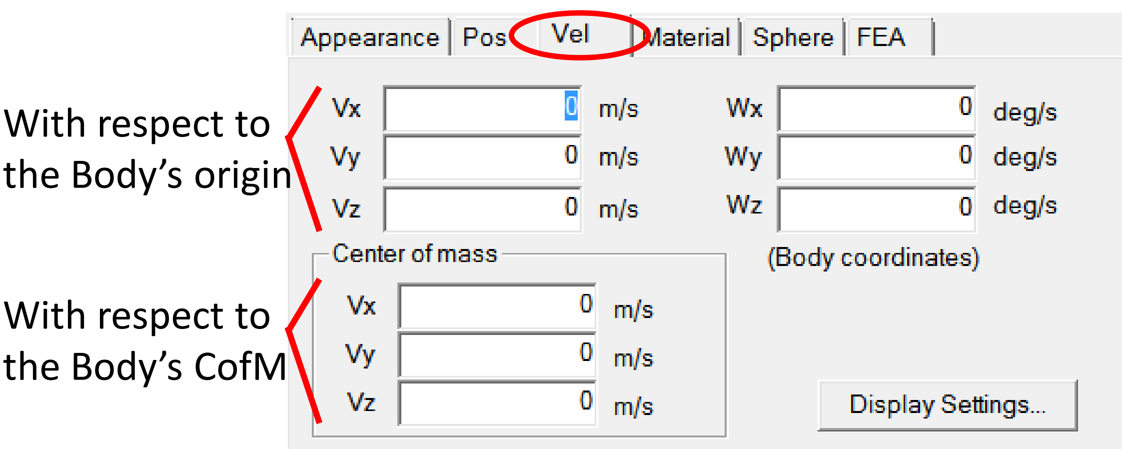
This feature is useful for applying motion to objects that do not have constraints associated with them or where there is no ability to apply a motor or actuator to the body.

The Prescribed motions are not to be confused with initial conditions that can also be applied to bodies, which are defined under the Velocity tab.

Bodies *(Initial Conditions)*

Initial (velocity) conditions can be defined for a body under the **Vel** tab.

Initial conditions affect only the starting conditions of the body at t=0. For example, specifying Vx for a body applies a velocity in the x direction only at time=0. For time >0, the velocity can be overcome if external forces begin acting on the body to slow it down.

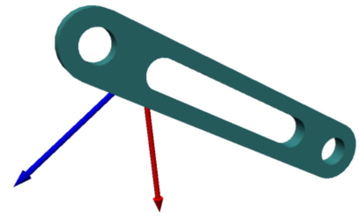


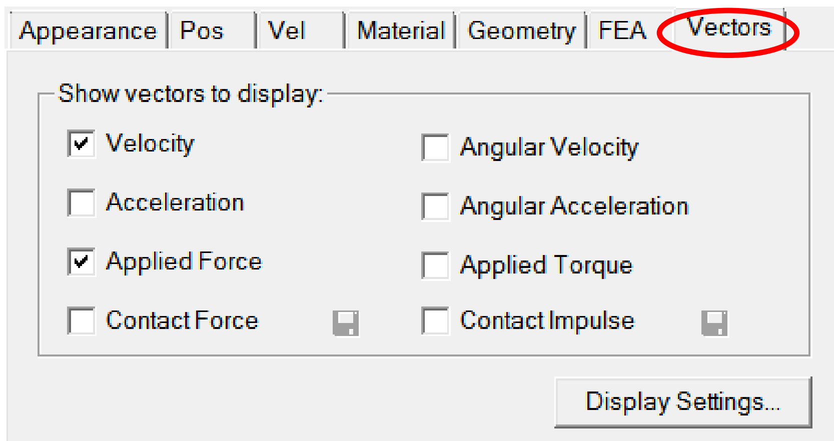
Velocity Settings d-box

The velocities values in the d-box fields will automatically update during a simulation to reflect the velocity of the body at that particular instance I time. If, after running a simulation, the simulation playback control is set to any other point in time besides t=0, and the user selects World, Erase Motion History, then whichever velocity values are showing in the d-box fields will become the new initial velocities for the next simulation

Bodies *(Vectors)*

Vectors allow the user to display graphical arrow representations of different output characteristics of a body.

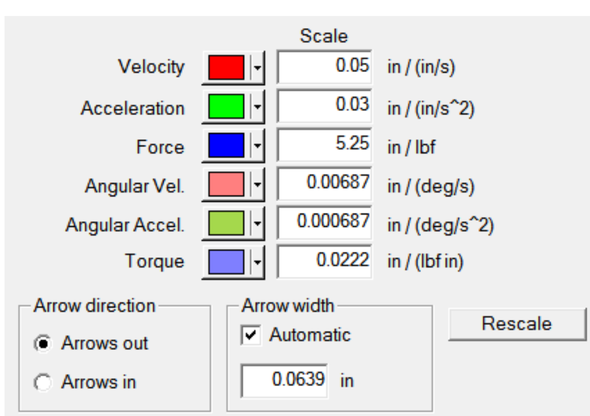




Vector Settings d-box

The length of the arrow graphic will scale accordingly with the change in value for the characteristic the vector represents.

Selecting the **Display Settings** button allows access to change the scaling of the length and width of the vectors

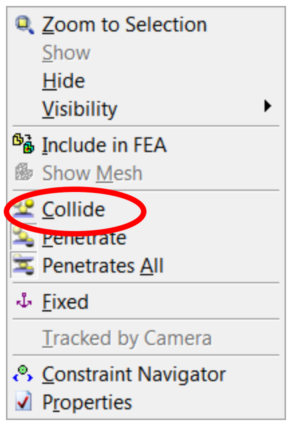


Vector Scale Settings d-box

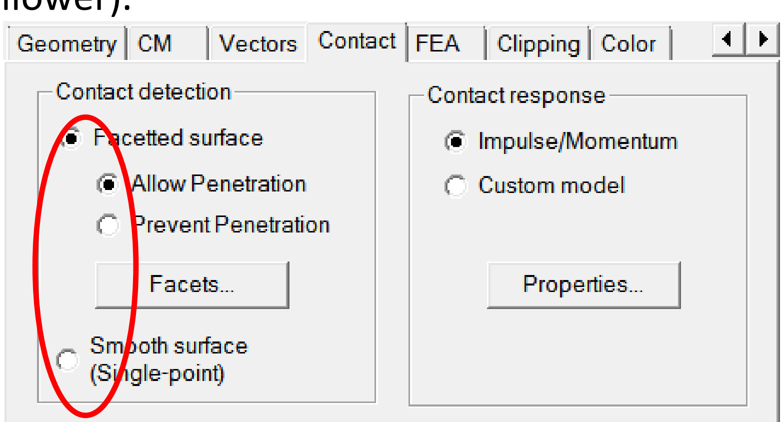
Bodies (Collision)

Bodies can be defined to collide (or make contact) with one another.

To define a collision, select two or more bodies, right click on any one of the selected bodies, and choose **Collide**.

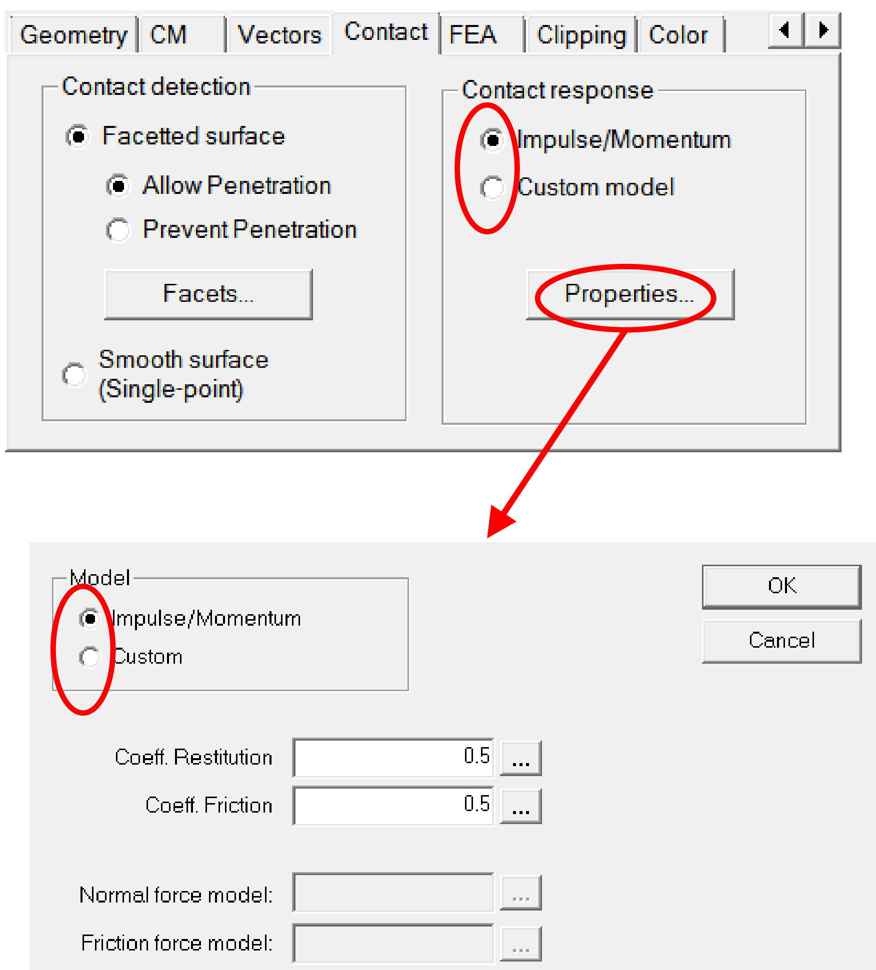


When bodies are set to collide, they will use one of two methods for detecting collision, *facetted* or *smooth surface*. *Facetted* is applicable to situations where two bodies contact each other on more than one surface simultaneously (example: plate on plate). *Smooth surface* is applicable to situations where two bodies contact each other on only one point (e.g. cam/follower).



Bodies (Collision) cont ……..

The **Contact response** method defines how the bodies will respond when they collide. The two options are Impulse/Momentum (i.e. coefficient of restitution) or Custom.



SimWise solves problems using rigid body dynamics solutions. When modeling collisions in problems of rigid body dynamics there needs to be a representation of how the bodies will behave when they collide. Since there is no allowed deformation at the contact interface, as in the physical world, the true stiffness and damping behavior (energy dissipation) must be represented using mathematical formulas. These formulas are estimated representations of the body's true physical-world behavior. There are two different Contact Response methods used in SimWise.

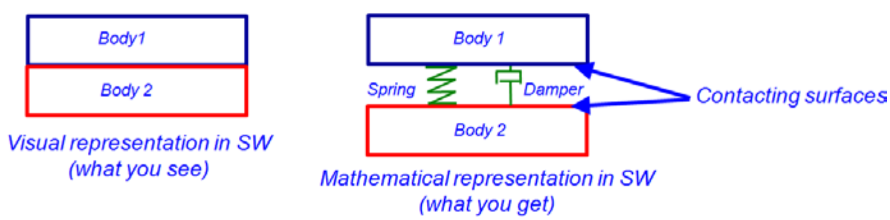
Impulse/Momentum (Coefficient of Restitution)
This method uses a coefficient based on the ratio between the relative velocities after and before a collision.

$$Coeff = \frac{\text{Relative speed after collision}}{\text{Relative speed before collision}}$$

The value returned is between 0 and 1. The lower the coefficient, the less bounce or rebound there is, resulting in an inelastic collision that is damped. This would be typical for harder materials such as concrete or steel. Higher coefficients represent materials that tend to be more elastic, such as rubber, which would be close to 1.

Custom

This method uses an equation that is representative of a linear spring-damper system. You may envision what takes place at the contacting interface if you view the problem as is there were springs and dampers (microscopic) between the surfaces, as illustrated below.



n general form, the equation for a linear spring-damper system is as follows:

$$F = \mathrm{Kx} - \mathrm{Cv}$$

Where:
F = resulting contact force
K = material (spring) stiffness value (not Modulus of Elasticity)
x = amount of penetration between the two surfaces
C = linear damping coefficient
v = relative velocity between surfaces

SimWise uses this same equation in calculating the contact force. All units are metric. :

$$F = (-1\mathrm{e}5\ \mathrm{N/m}) * \mathrm{penetration}() + (-300\ \mathrm{N\ s/m}) * \mathrm{penetrationrate}()$$

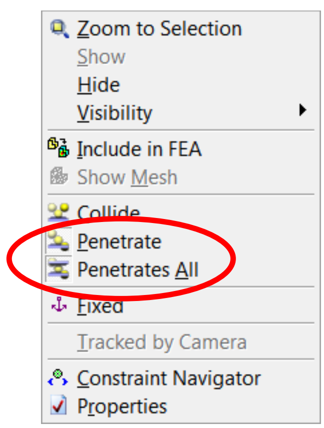
Where:

F = resulting contact force
-1e5 N/m = default material stiffness value
Penetration() = amount of penetration between the two surfaces (automatically calculated)
-300 Ns/m = default linear damping coefficient
penetrationrate() = rate of penetration between the two surfaces (automatically calculated)

The default values used in the SimWise equation are meant as a general numbers characteristic of a moderately-stiff material, such as Aluminum. It is ultimately the final decision of the User to determine if these values are sufficient to best characterize the material and problem-type at hand, or if they may need modification.

Bodies *(Collision)*

To remove a collision, select the two bodies and either select **Penetrate** or **Penetrates All**. Penetrate will remove the collision between only the select pair. Penetrates All will remove the collision between the selected bodies and all other bodies, if there are multiple collisions defined for the body(ies). If there are multiple collisions defined for the body(ies), and it is desired to only remove the collision between the selected pair, use the Penetrate option. If you wish to remove all collision associated with a body, use the Penetrates All option.



Bodies *(Collision, Fine Tuning)*

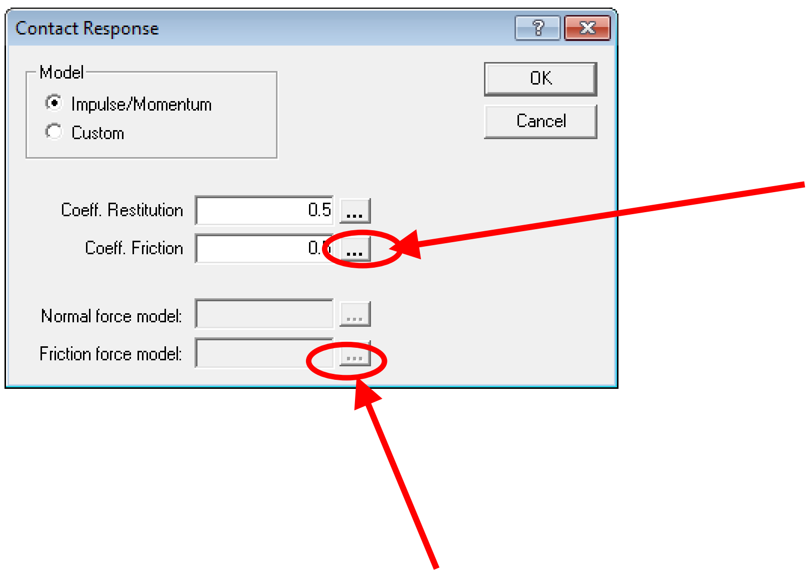
Sometimes it may be necessary to change the simulation tolerance settings (Animation Step and Configuration Tolerance) and/or Contact property settings. There are no exact value settings for specific problems types, as all problems are different. When fine tuning the contact behavior, consider doing the following to help make the proper adjustments:

1. Create a contact force meter for the contact. This will help you see expected and unexpected behavior, such as noise, spikes or discontinuities in the data where there should not be.
2. Add (contact) force vectors for one of the contacting bodies. This helps visualize what is happening at the contact interface.
3. Try using both different Contact Detection and Contact Response methods. Sometimes one option may work better than another. It will depend on the problem itself.
4. Experiment with adjustments to the Position Tolerance, Overlap Tolerance and Animation Frame Rate

Friction in Collisions

Friction can be included when modeling collisions in SimWise.

When using either the **Impulse/Momentum** method for collision, the default friction coefficient is constant. The user may also select the function builder tab next to the coefficient field and define a custom equation instead.



See next page for discussion of the Friction force model when using the Custom collision method

Friction (Custom) in Collisions

The **Custom** collision option in SimWise bases its (default) friction calculations on the relative velocity between the contacting surfaces. Rather than use a static coefficient for 0 velocity conditions and a dynamic coefficient for non-zero conditions, this method generalizes and uses one coefficient for all.

Once there is slip between the surfaces and the relative velocity begins increasing, the friction force begins transitioning from 0 to its maximum value. The maximum friction force is reached when the velocity between the surfaces is non-zero and constant (no acceleration).

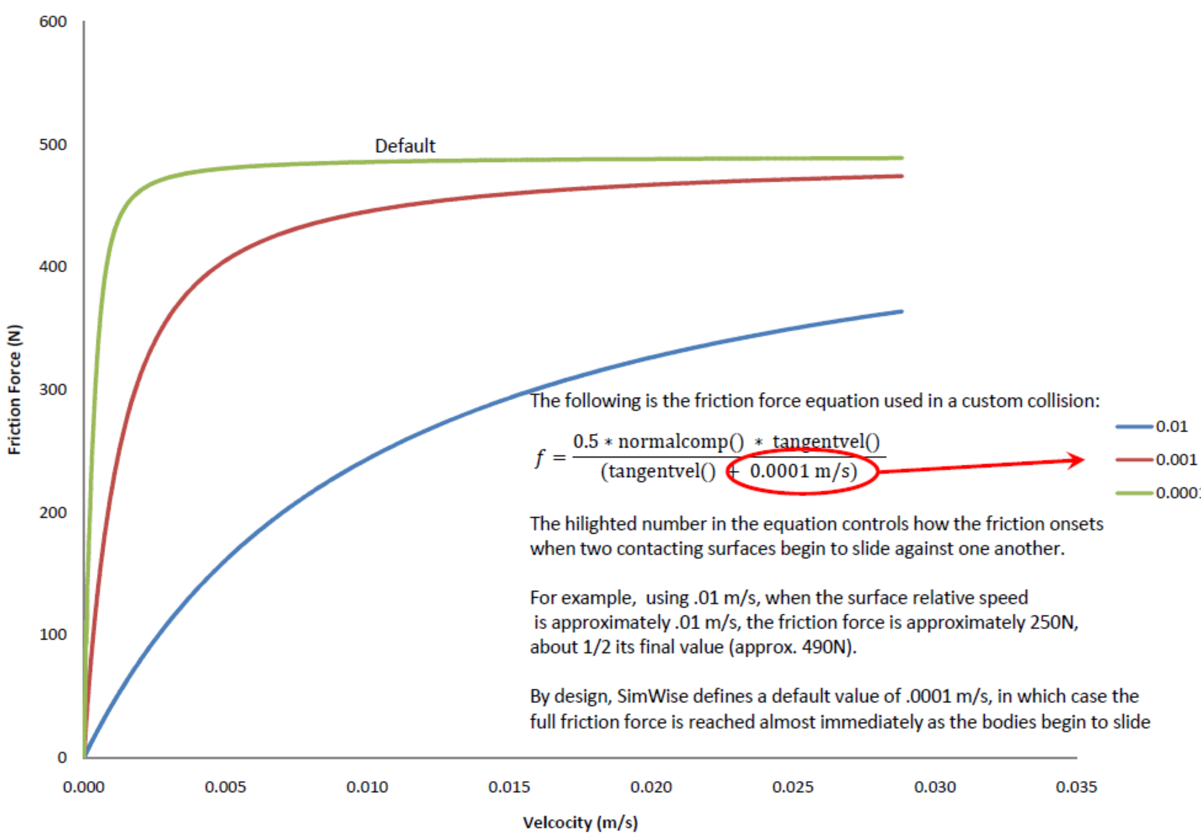
The default friction formula is written as follows:

$$f = \frac{0.5 * \mathrm{normalcomp}() * \mathrm{tangentvel}()}{(\mathrm{tangentvel}() + 0.0001\ \mathrm{m/s})}$$

Where:
f = resulting friction force
0.5 = (default) dynamic friction coefficient
normalcomp() = the normal force magnitude resulting at the contacting interface
tangentvel() = the tangent velocity magnitude between the contacting surfaces
0.0001 m/s = constant used with tangentvel() to characterize the onset of friction

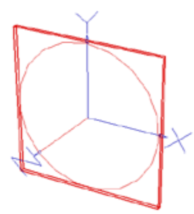
Friction in Collisions

The following graph shows how changing the (circled) parameter in the denominator of the formula affects the onset of friction when using the Custom collision method



Coords (Overview)

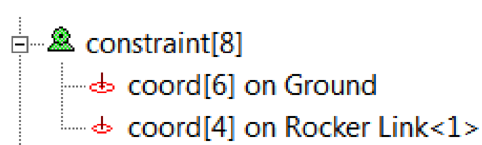
Coords (short for coordinate systems) are the parent basis for almost all elements in SimWise. They establish orientation and position for various elements such as constraints, forces and springs. They also establish reference frames for calculating the various motion data, such as position and velocity.



Note: A coord's orientation and position may be different from the Global orientation and position.

When creating elements such as constraints, the user has the option of creating the coords first and the defining the constraint. Or, the coord can be created automatically while creating the constraint. In the latter case, there is less initial user control over where the constraint is being place and how it is being oriented.

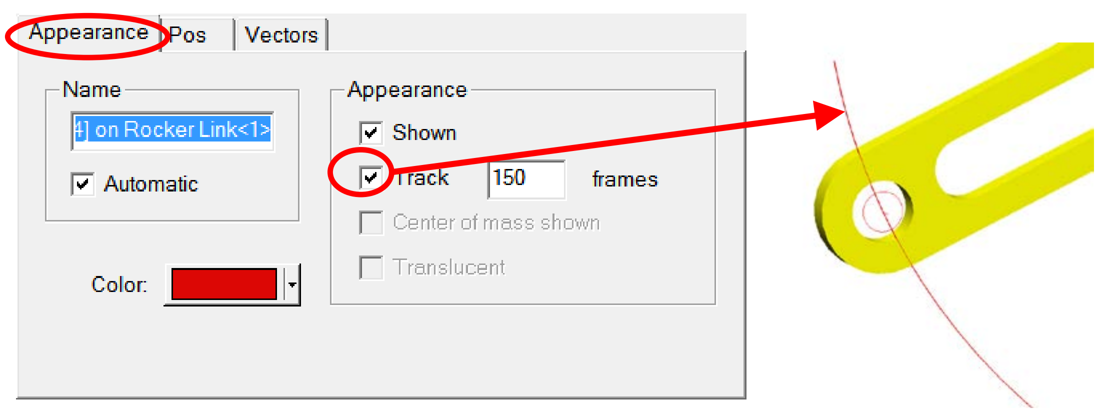
Some model elements, such as forces, only need one coord to define them. Other elements, such as constraints, need two coords. In the case of a constraint, one coord is attached to one body and the other coord is attached to either another body or the background.



Coords *(Overview) cont….*

Certain characteristics of coords can be metered, such as position, velocity and acceleration. Select the coord, then select Insert → meter.

Coords can be “tracked”, where a graphical trace curve will be generated for the position path the coord takes during a simulation. This feature is found under the Appearance d-box in the coord Properties.

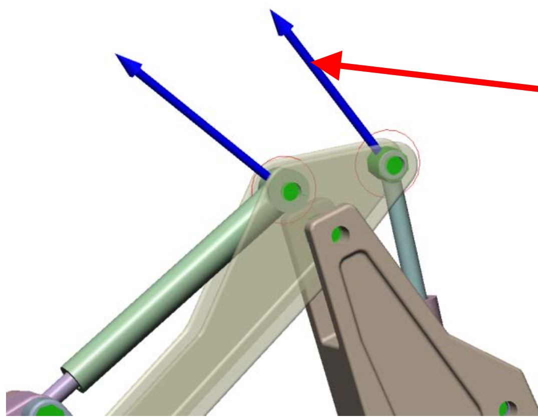


In the hierarchy, when working with constraints, coords are considered as parent features to the element they represent. Deleting the coord (or coords) will automatically delete the related constraint(s). On the other hand, deleting the constraint(s) will not delete the coord(s).

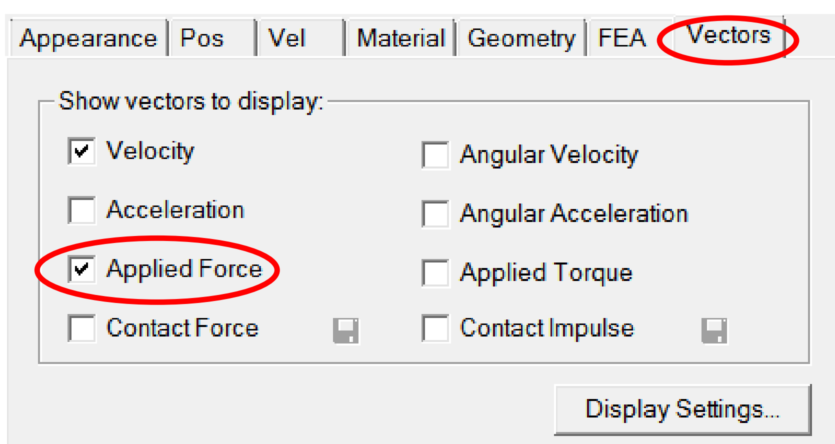
If a coord is attached to a body, and the body is deleted, the coord will not be deleted but will instead be attached the background.

Coords (Vectors)

Similar to vectors for bodies, vectors can be displayed for coords also. For example, it may be necessary to show the force reaction vector occurring at a particular constraint.



Force vectors showing for the coords of the constraints



Vector Settings d-box

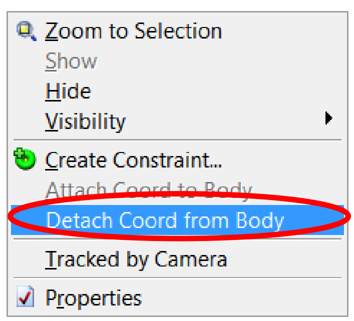
To add a vector for a coord, open the Properties for the coord, display the Vectors tab if not already displayed (from the Properties List), and select the appropriate vector to display

Coords (Detaching/Attaching)

A coord can be moved from one parent element to another. For example, a coord can be detached from one body and attached to another. This prevents having to delete and create a new coord. In addition, it allows constraints to be moved between bodies. For example, the user might want to change the constraint from attaching Part A and Part B to instead attach Part A and Part C.

To move a coord from one body to another:

Step 1: Right-click on the coord and choose **Detach coord from body**. The coord will now be placed on the background.

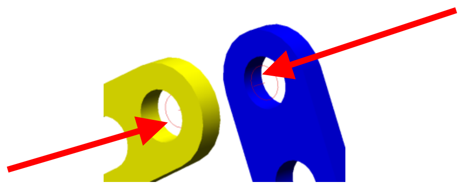


Step 2: Hold down the Ctrl key, select the coord, select the body you wish to attach the coord to, and choose **Attach coord to body.**

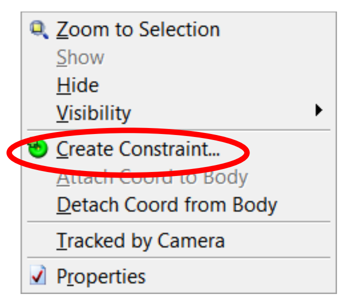
Coords (Assembling bodies)

Coords can be used to assemble bodies, and create constraints at the same time, similar to creating constraints (or mates) in a CAD program.

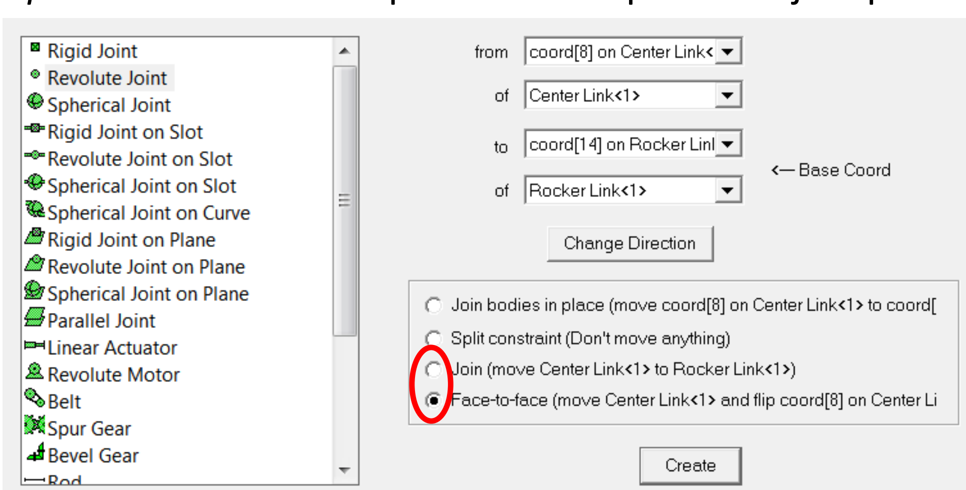
Step 1: **Create** a coord on each body



Step 2: **Hold down** the Ctrl key, **select** both coords, then **right-click** on either coord an **choose** Create Constraint



Step 3: Use one of the options to complete the join process



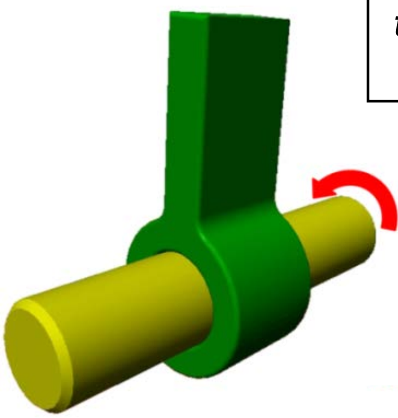
Constraints (Overview)

- The user is not limited to using only CAD constraints, in the case of using a CAD-associative model. SimWise has many different constraint types available to help properly represent the degrees of freedom (DOF) of a system.

Rigid Joint Revolute Joint Spherical Joint Rigid Joint on Slot Revolute Joint on Slot Spherical Joint on Slot Spherical Joint on Curve Rigid Joint on Plane Revolute Joint on Plane Spherical Joint on Plane Parallel Joint	**Kinematic constraints** represent all possible degrees of freedom in physical connections
Linear Actuator Revolute Motor	**Inputs**
Belt Spur Gear Bevel Gear	**Power Transmission**
Rod Rope Separator	**Distance controlling constraints**
Linear Spring/Damper Revolute Spring/Damper	**Springs and Dampers**
Bushing	**Flexible connectors** (Redundant constraint alternatives)
Generic Constraint	**User-defined / customized constraints**

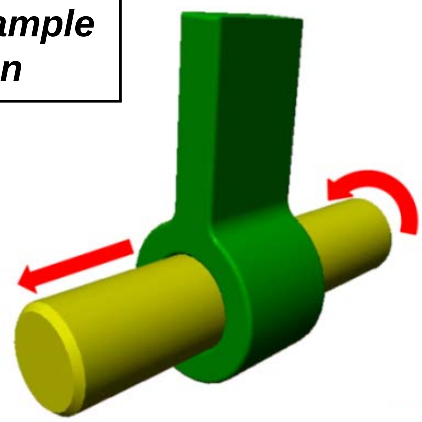
Constraints ***(Kinematic)***

Click on a Constraint picture to see an example animation



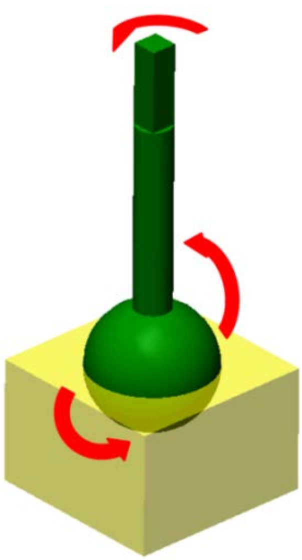
Revolute

(1) Rotations
(0) Translations



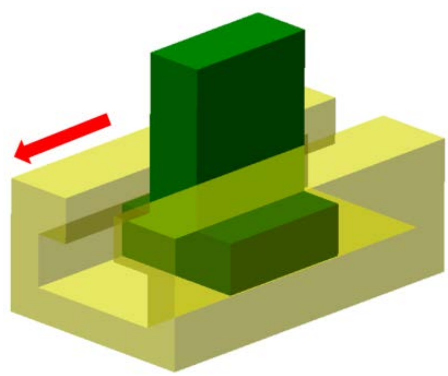
Revolute on Slot

(1) Rotations
(1) Translation



Spherical

(3) Rotations
(0) Translations

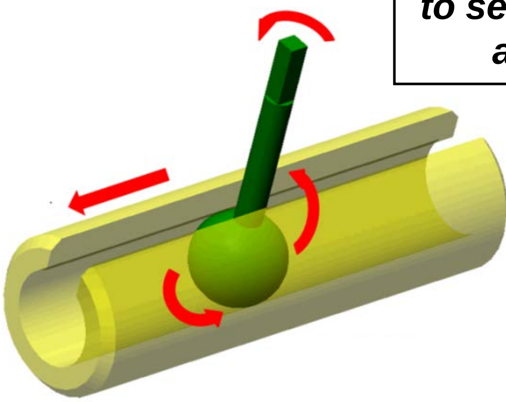


Rigid on Slot

(0) Rotations
(1) Translation

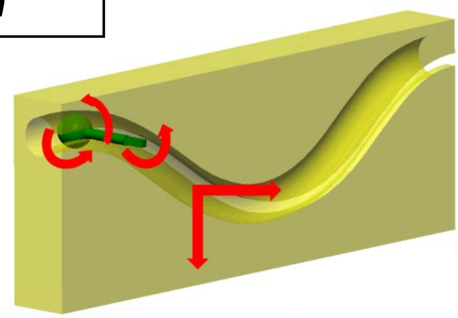
Constraints *(Kinematic)*

Click on a Constraint picture to see an example animation



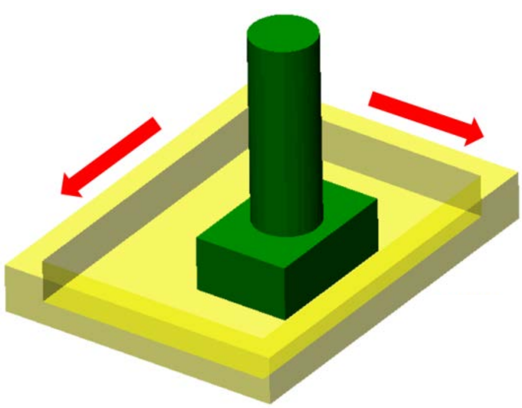
Spherical on Slot

(3) Rotations
(1) Translation



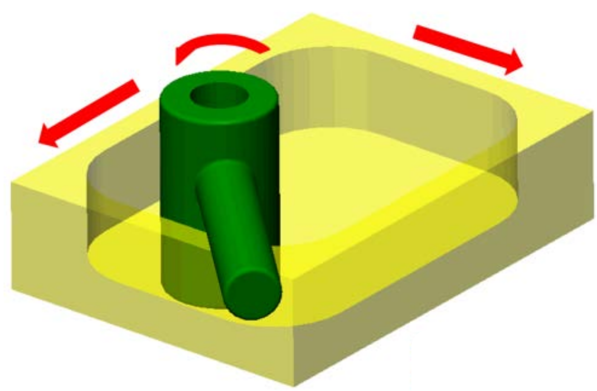
Spherical on Curve

(3) Rotations
(2) Translations



Rigid on Plane

(0) Rotations
(2) Translations

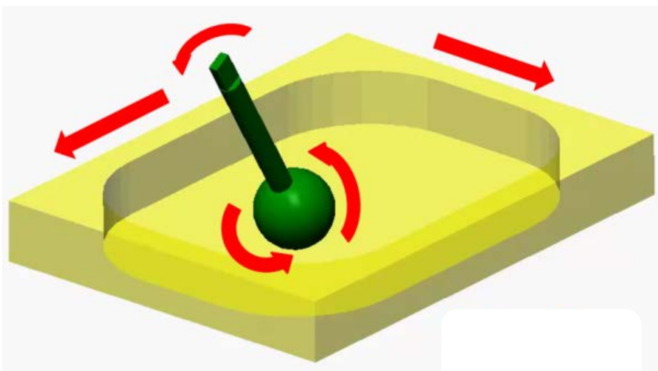


Revolute on Plane

(1) Rotations
(2) Translations

Constraints *(Kinematic)*

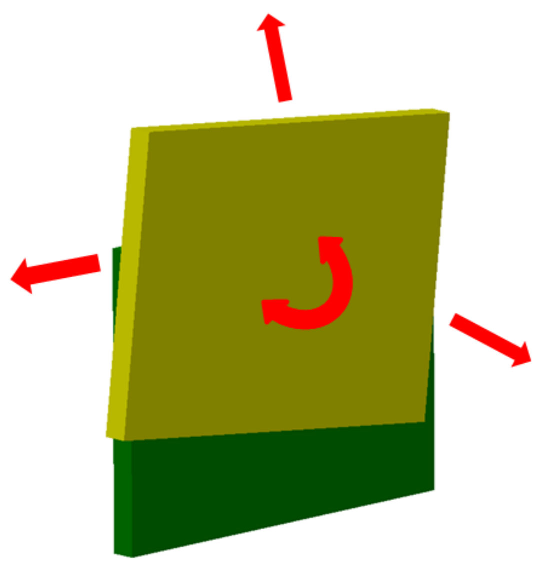
Click on a Constraint picture to see an example animation



Spherical on Plane

(3) Rotations

(2) Translations



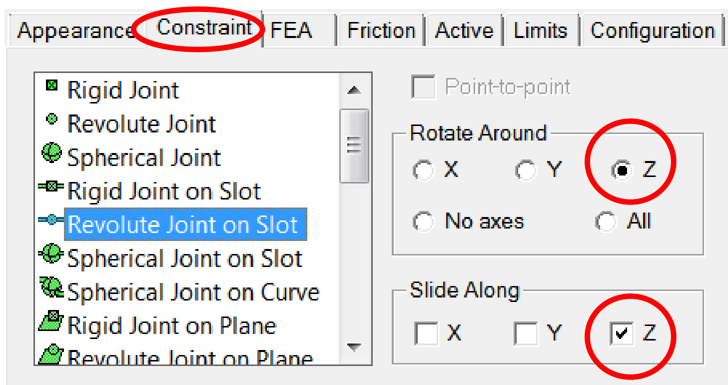
Parallel

(1) Rotations

(3) Translations

Constraints (Degrees of Freedom or DOF)

- By default, each constraint represents a specific DOF of movement. Selecting the Constraint tab in the constraint Properties dialog box will show the assigned DOF for that constraint



Example: Constraint properties for a Revolute Joint on Slot

The user may modify the constraint by either:

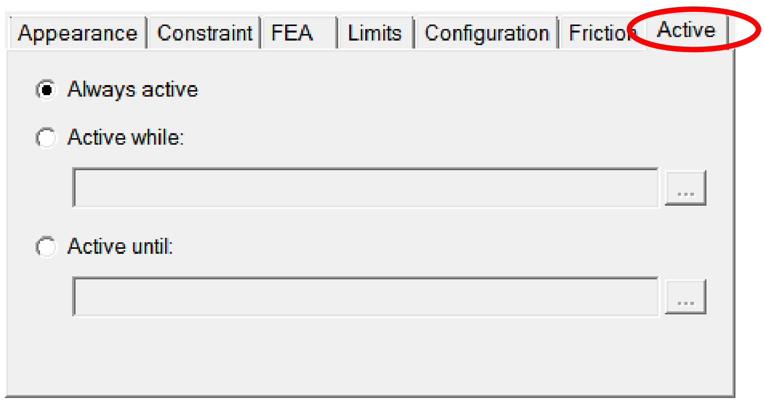
1. Modifying the CAD constraint type(s) and quantity, when using automatically-mapped constraints from a CAD-associative model

2. Selecting a different constraint type from the SimWise constraint list

3. Selecting the different DOF checkboxes from the Rotate Around and Slide Along options. Depending on which DOF are selected, the constraint type will update after applying the changes.

Constraints *(Activating / Deactivating)*

Constraints can be deactivated if necessary. A constraint can be specified as 1) always active (Default), 2) active while certain criteria is true, or 3) active until some criteria has been met



Constraint properties for a
Revolute Joint on Slot constraint

Specifying an **Active While** criteria can be useful when debugging a model. For example, it can be useful for trying to determine which constraint(s) might be causing redundancies. In this case, the user can specify 0 or type the word "False" for the Active While field to deactivate a constraint.

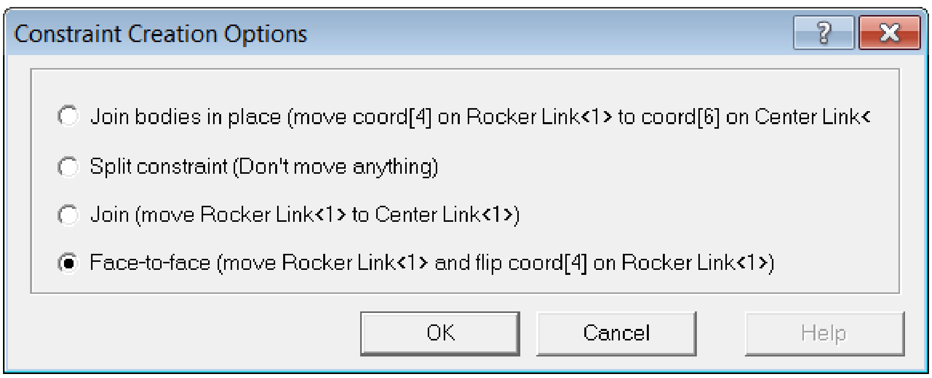
The Active feature can also be useful in simulating the breaking of constraints. For example, a user can specify that a revolute constraint is to be deactivated when the force in that constraint meets or exceeds some user-defined value.

Constraints (Creating)

There are a few different ways to create a constraint

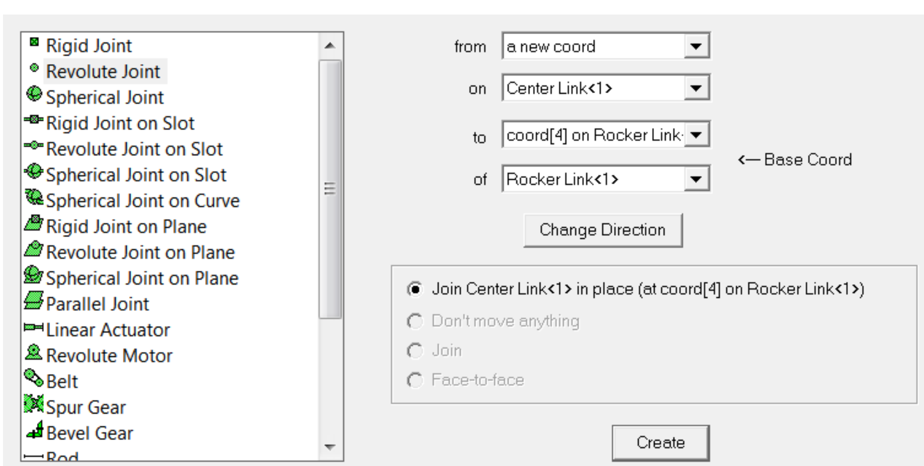
Option 1:

Select a constraint type from the constraint drop down list on the main menu. **Click** on two bodies or a body and the background. The following options will appear:



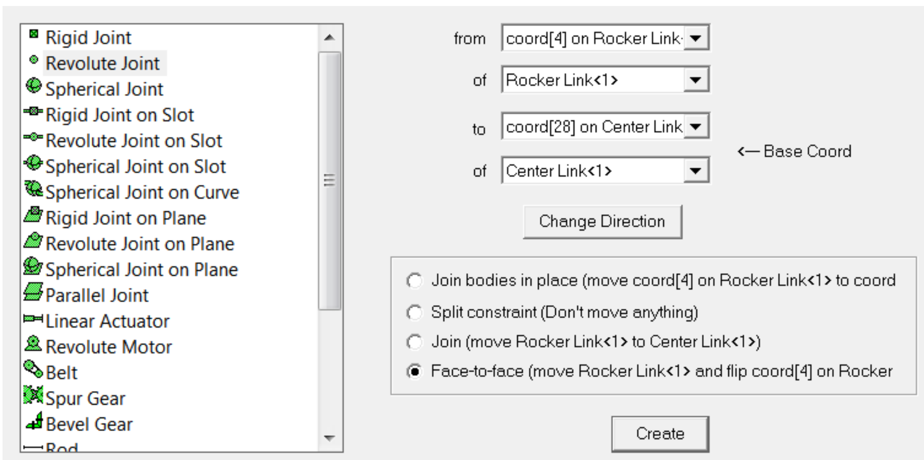
Option 2:

Right-click on a coord and choose **create constraint**



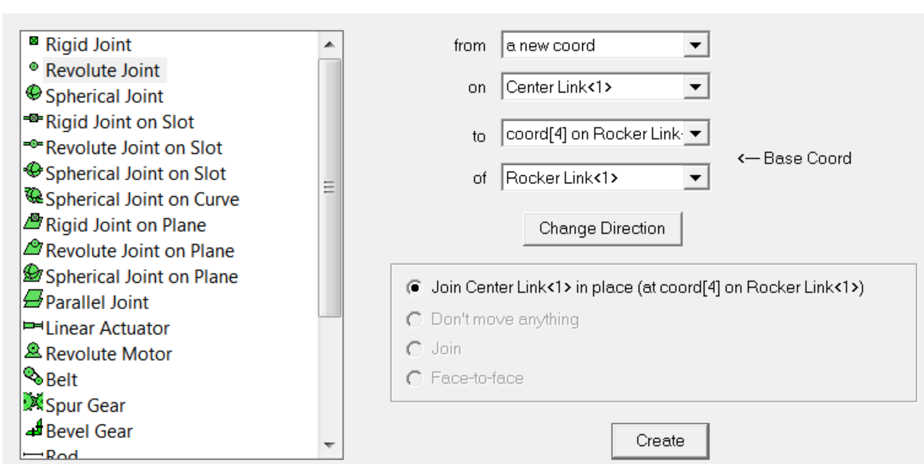
Option 3:

Right-click on two coords (use Ctrl key for multi-select) and choose **create constraint**. Choose one of the options presented to finish the process.



Option 4:

Right-click on a coord and a body and choose **create constraint**



Constraints *(Moving/Splitting)*

When moving elements such as constraints care must be taken that the correct method is used, as there are different ways the constraint can be moved or rotated.

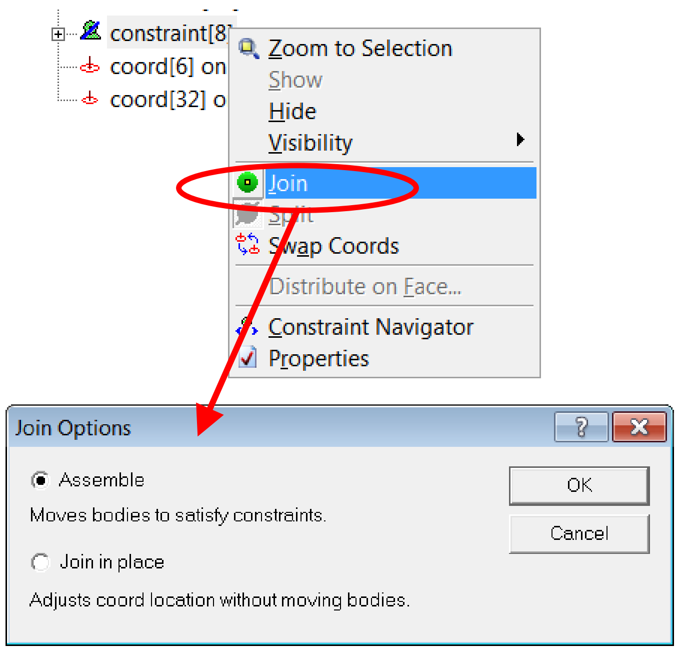
1. By moving or rotating the constraint and then using the **join** feature to reassemble the constraint

2. By moving or rotating one of the coords and then using the **join** feature to reassemble the constraint

3. By moving or rotating both constraints simultaneously.

Constraints *(Moving/Splitting)*

In the case of 1 or 2 when moving a constraint, the constraint icon in the browser will display a blue slash through it indicating it has been “split”. Right-clicking on the icon either in the browser or in the graphics area, and then choosing “join” will produce the following join options:

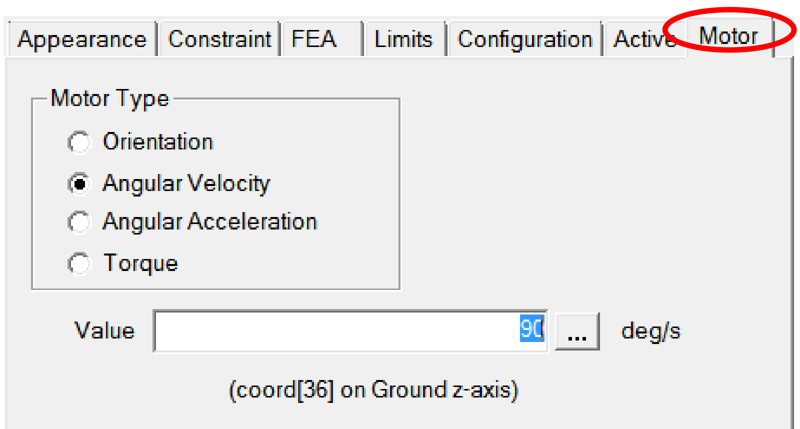


Selecting **Assemble** will move the body to the location of the constraint. Care should be taken when using this method, as other constraints may become invalidated (broken).

Selecting **Join in Place** will move the second coord only, and not the body. Care should be taken when using this method, as a constraint can move to a new location but still be attaching two bodies. For example, a revolute constraint might accidentally be positioned off the parts somewhere, not at the physical pivot point. In this case, the kinematic motion will be correct but a bending moment will be introduced unintentionally.

Inputs (Motors)

Motors are rotational input constraints that apply motion to a mechanism.



Input properties for a Motor

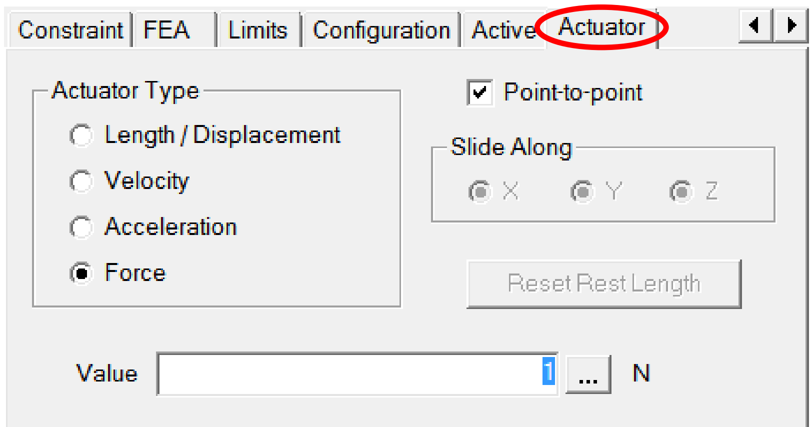
By default, motors also act as a revolute constraint (remove 5 DOF). The user must be careful not to also apply a revolute constraint, for example, in parallel with the motor, as this would result in a redundant setup and produce the potential for inaccurate force readings.

Selecting the constraint tab in the motor properties and changing the Rotate Around and Slide Along options allows the motor to be represented as, for example, a motor on a slot.

A benefit to using a motor input is that user has the ability to create a meter for the motor to determine the amount of torque necessary to achieve a desired output (reverse engineering the input).

Inputs (Actuators)

Actuators are linear input constraints that apply motion to a mechanism.



Input properties for an Actuator

Actuators can be defined as point-to-point constraints (removes no 0 DOF) or they can be defined as, for example, an actuator constrained to a slot. In the latter case, the user must be careful not to also apply a rigid joint on slot, in parallel with the actuator, as this would result in a redundant setup.

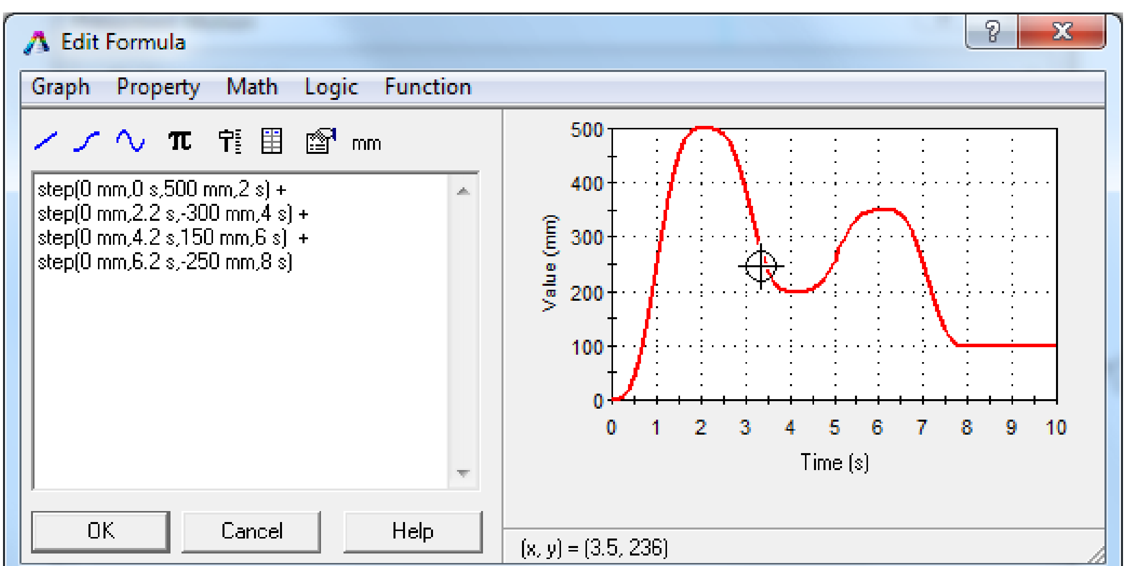
A benefit to using an actuator input is that user has the ability to create a meter for the actuator to determine the amount of force necessary to achieve a desired output (reverse engineering the input).

Inputs (Function Builder)

The Function Builder allows the user to customize different types of inputs to things such as motors, actuators, forces, friction, springs and dampers. It can be accessed by selecting the formula ... button wherever there is a value field for a feature.

Within the function builder, the user has many options for creating input functions. The **Graph, Property**, **Math, Logic** and **Function** menus contain several different functions that can be used as templates to help create and enter formulas directly into the formula window. There are also options for adding in common curve types (STEP, Sinusoidal and Sawtooth), as well as interactive slider controls and tables.

Inside the function builder, a preview window on the right side displays the curve as it is being created so the user can preview what the input will look like.

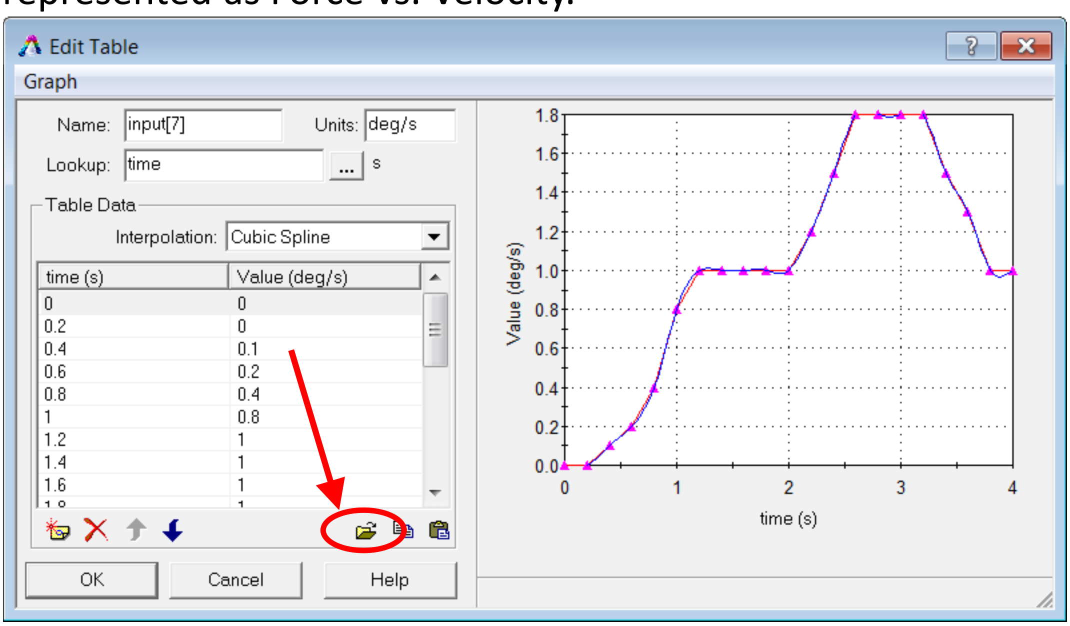


Inputs (Data Tables)

Wherever there is the ability to access the function builder, there is the ability to define a data table. Data tables are used to control the inputs to things such as motors, springs and forces. Using data tables allows data from spreadsheets to be used to drive an input, such as motor velocity or a feature, such as a spring stiffness.

Data can be 1) imported directly from a spreadsheet file (.txt), 2) created by entering values in the table window, or 3) by starting with an existing pre-defined curve, such as a step curve, and then modifying it.

By default, the data is in reference to time. However, the **Lookup** option can be changed so that another variable besides time can be used. For example, a custom damper could be represented as Force vs. Velocity.

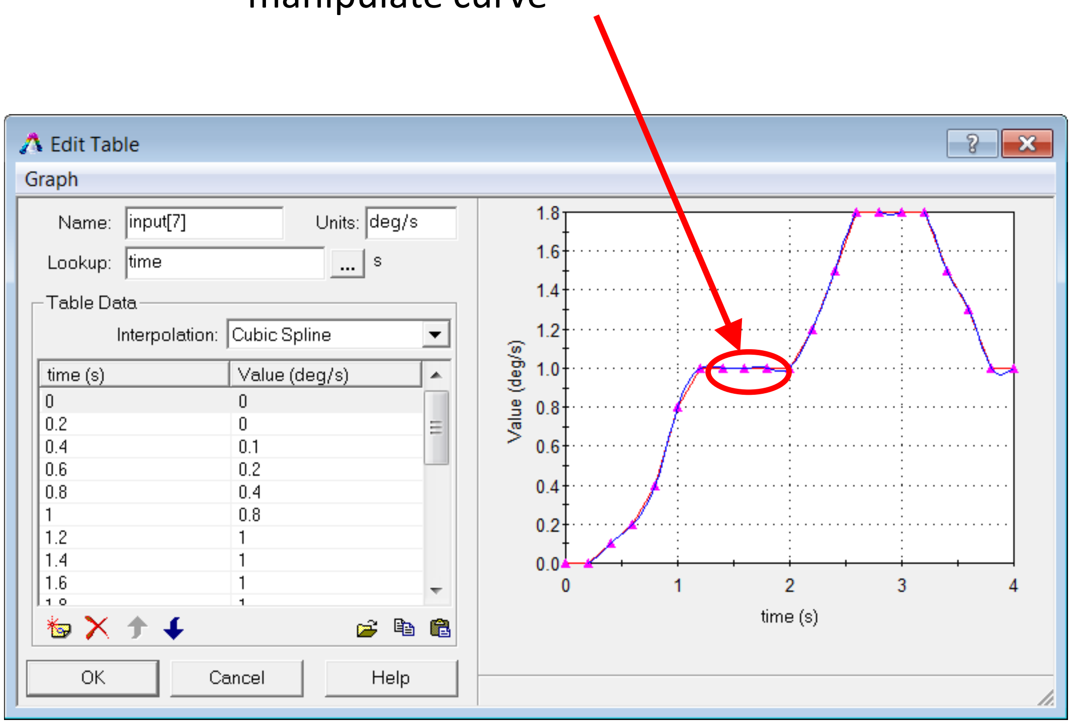


Inputs (Data Tables) cont….

Depending on the data used, the Interpolation method may be changed. Choose a method that shows a good curve fit against the data in the preview window.

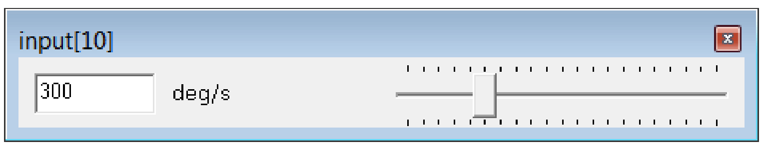
The data curve can be manipulated by clicking and dragging the pink colored data points in the preview window

Click, hold mouse button
and drag pink points to
manipulate curve

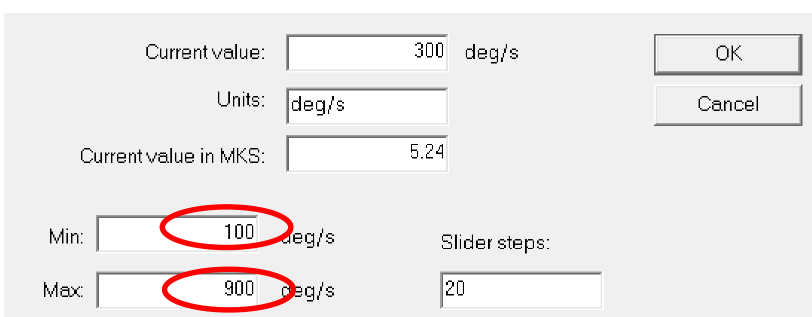


Input (Interactive Controls)

Wherever there is the ability to access the function builder, there is the ability to define an interactive control. Interactive slider controls allow the user to control the values being used in the inputs, in real time. For example, during a simulation, the User can click, hold and drag the slider to modify the input.



Upper and lower bounds can be defined for the controls, as well as an incremental step value for moving in between the upper and lower bounds.

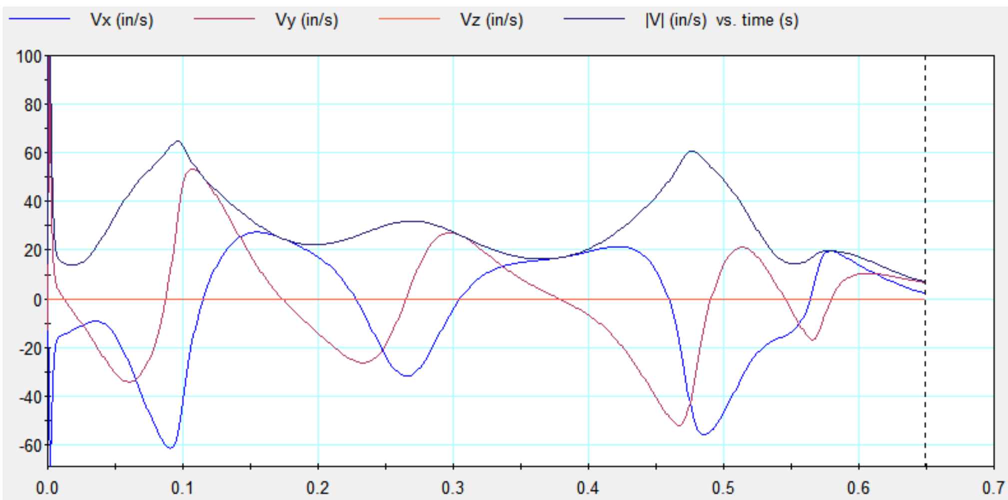


Double-clicking the slider control will open this d-box

Controls can be defined either by 1) clicking on a constraint and choosing Insert, control or 2) by clicking on the insert slider button inside the function builder window for an input.

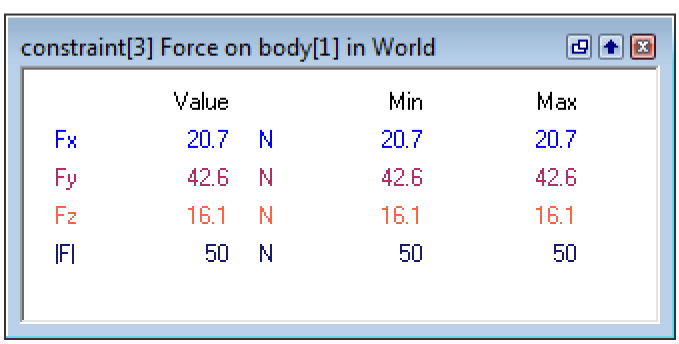
Meters

Meters (plots) are used to display output characteristics of bodies, forces, constraints, or coords.



Graphical Meter

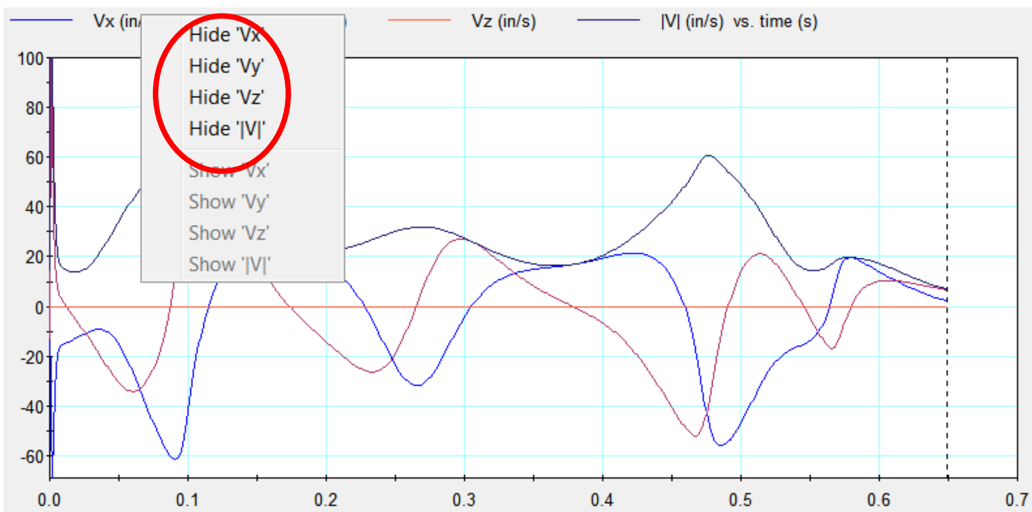
A meter is created by first selecting the body, constraint, force or coord in the graphics area or Object Manager and then choosing Insert, Meter.



Digital Meter

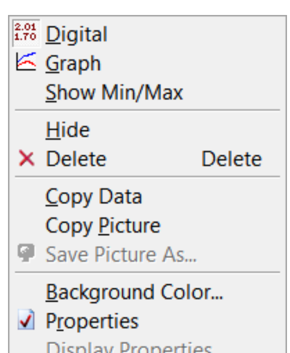
Meters can display data both graphically and numerically. Simply right-click on the boarder of the meter to toggle between graphical and digital.

Meter components can be toggled on/off by selecting (left-click) any one of the x,y, or z component names at the top of the meter and choosing Hide. The same procedure can be done to show a component that has been hidden



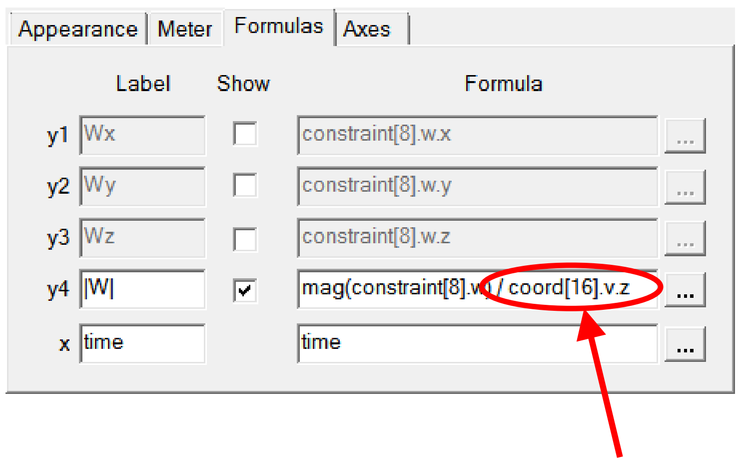
Right-clicking on a meter will open its Properties. Here the user can access other options such as Showing the min and max values, copying data to the clipboard (for pasting into spreadsheet), copying a picture of the meter to the clipboard and modifying many of its properties, such as background color, curve color, curve size and so on.

Right-clicking on a meter to access its properties



Meters cont…..

Meters can be customized to display data that might not be readily available through standard meters. For example, plotting the ratio of an input velocity vs. an output velocity.



Added formula

Meters are also useful for obtaining formula syntax to be used in other features, such as a run control, limits or custom annotations. In such cases, a temporary meter can be created and the user can simply copy & paste the formula into the new feature and then delete the meter. This saves time in having to memorize formulas and syntax, which are found in detail in the program HELP.

A meter must be defined <u>before</u> running a simulation.

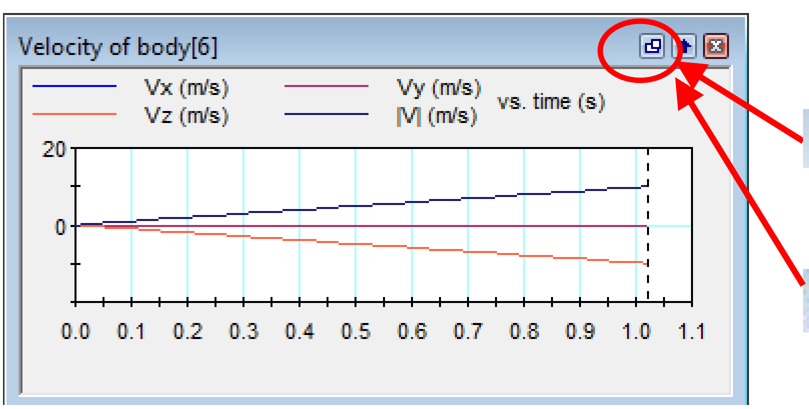
Meters *cont…..*

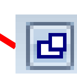
There are various things that can be done to meters within the graphics area.

Meters can be moved by selecting the meter header, holding down the mouse button and dragging the meter to a new position.

Meters can be resized by hovering the mouse over the meter edge or corner, holding down the mouse button, and dragging the meter edge.

Meters can be anchored inside the graphics are by selecting the small tab at the top-right of the meter



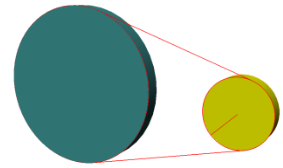


Anchor the meter inside the model graphics window

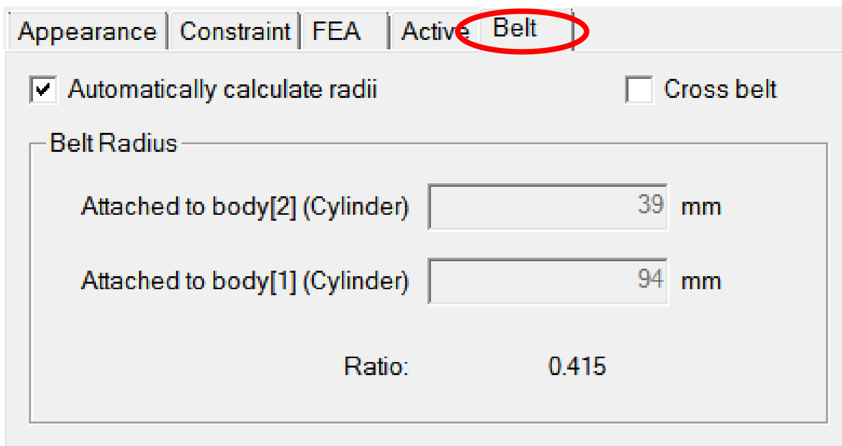
Allow the meter to “float” outside the model graphics window

Right-click the meter and choose Arrange Vertically or Arrange Horizontally to stack the meters together either vertically or horizontally.

Power Transmission (Belts)



Belts are used to transmit rotation from one body to another.



Input properties for a Belt

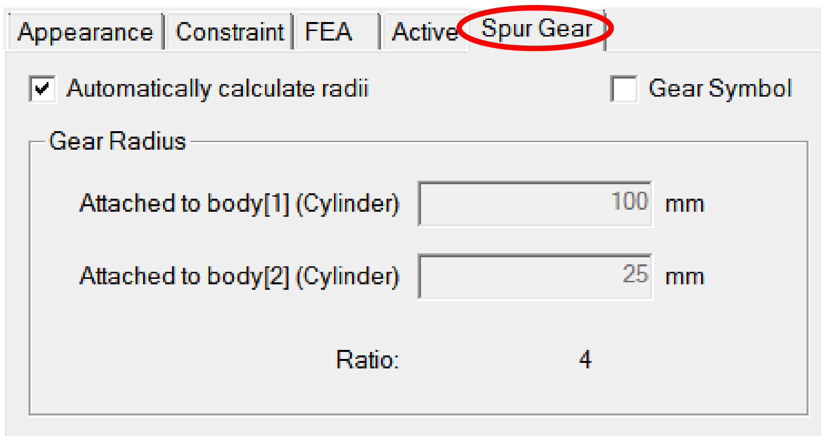
Coords are used to create the belt.

By Default, SimWise will automatically size the belt based on the size of the geometry the coords are attached to. The user also has the option of defining the belt size manually.

The Cross belt option allows the belt to crossover the pulleys thereby rotating the pulleys in the opposite direction.

Power Transmission (Spur Gear)

Spur gears transmit rotation from one body to another



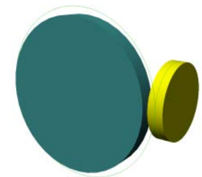
Input properties for a Spur Gear

Two coords can be used to create the gears or the bodies can be selected and the base coords for the gears, and the gears themselves, will be automatically created.

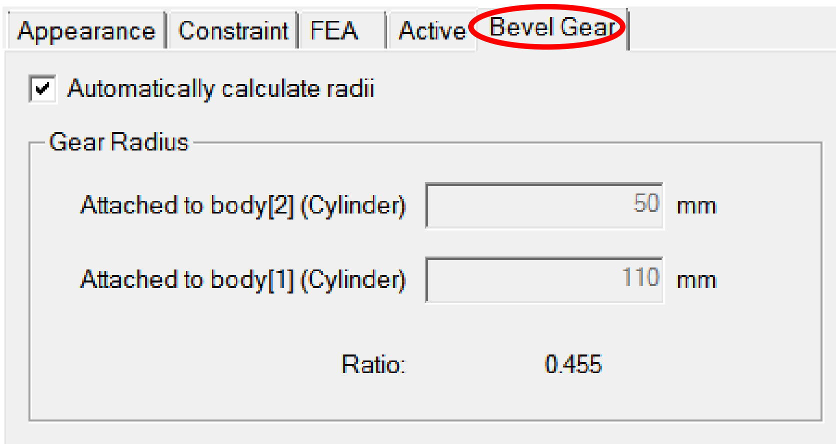
By Default, SimWise will automatically size the gear ratio based on the size of the geometry the coords are attached to. The user also has the option of defining the gear ratio manually.

The Gear Symbol option allows a simple visual representation of the spur gear to be displayed

Power Transmission (Bevel Gear)



Bevel gears transmit rotation from one body to another, at a right angle to one another.

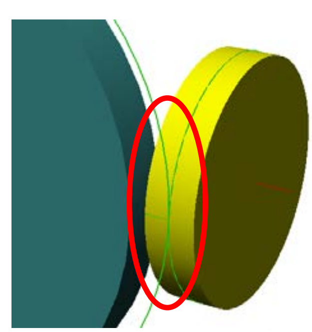


Input properties for a Belt

Two coords can be used to create the gears or the bodies can be selected and the base coords for the gears, and the gears themselves, will be automatically created.

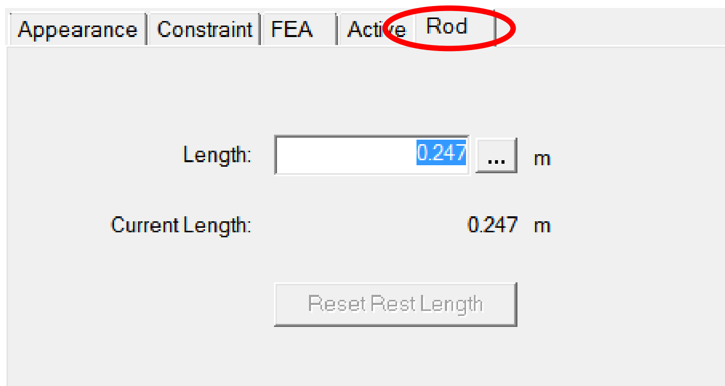
By Default, SimWise automatically size the gear ratio based on the size of the geometry the coords are attached to. The user also has the option of defining the gear ratio manually.

Depending on where the two base coords are positioned, circles representing the gears will extend from the base coord, outward until they are tangent. This may have a tendency to make the ratios slightly different than what was expected. Simply adjust the body positions as necessary.



Distance Controlling Constraints
(Rods, Ropes and Separators)

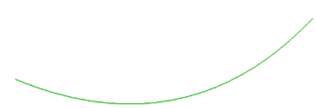
Two coords are used to create Rods, Ropes and Separators



Input properties for a Rod, Rope or Separator

Rods represent a massless, infinitely-stiff point-to-point connection between two coords, that maintains a set distance between the coords.

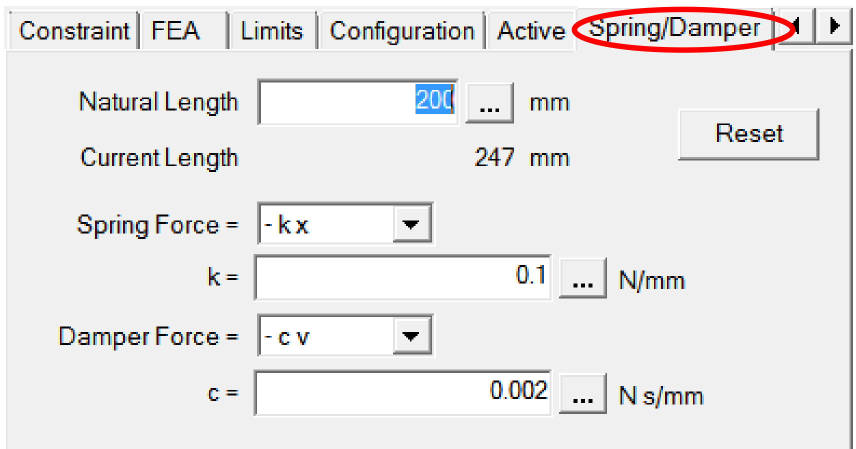
Ropes represent a massless, infinitely-stiff point-to-point connection between two coords, that can be defined with a given amount of slack in the rope. Once there is no slack, the rope does not stretch but instead remains rigid.



Separators represent a massless connection between two coords, whereby the coords can extend farther than the set distance but not closer.

Linear Spring/Damper

Two coords are used to create a linear spring/damper



Input properties for a Linear Spring/Damper

The Natural Length is the free length of the spring. Any difference between the natural length and current length will be used in the Spring Force equation to calculate the force in the spring.

Example: Natural Length = 200mm
Current Length = 247mm
k = .1 N/mm

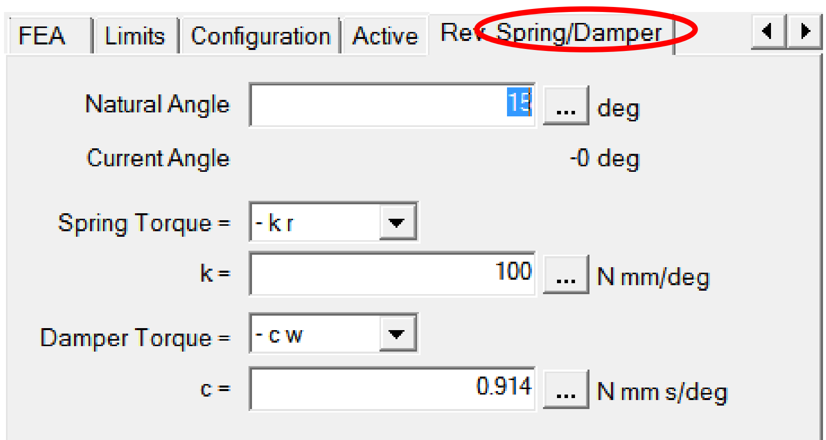
<u>Total Spring Force</u> = -kx
- .1 x (200-247) = **4.7 N**

Positive force = extension
Negative force = compression

The Spring/Damper can be used as a spring-only if the damping coefficient is set to 0. Similarly, it can be used as a damper-only if the spring constraint is set to 0.

Revolute Spring/Damper

One coord is needed to create a revolute spring/damper



Input properties for a Revolute Spring/Damper

The Natural Angle is the free angle of the spring. Any difference between the natural angle and free angle will be used in the Spring Torque equation to calculate the torque in the spring.

Example: Natural Angle = 200mm
Current Angle = 247mm
k = 100 N mm/deg

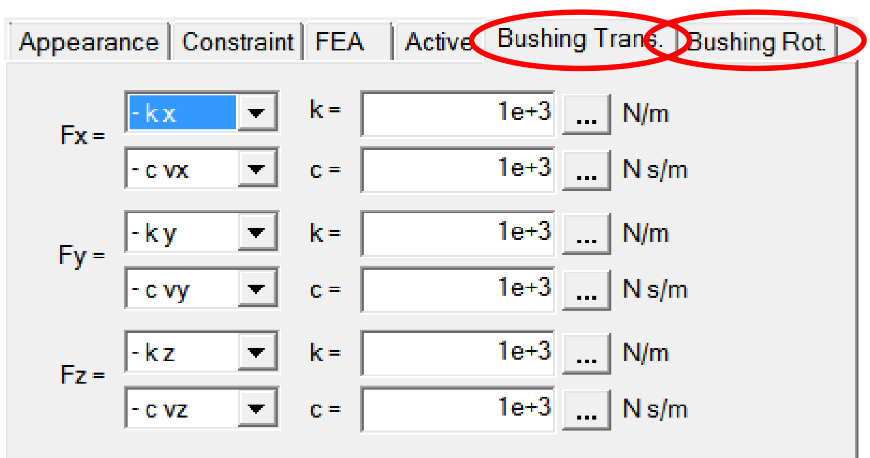
<u>Total Spring Torque</u> = -kr
- 100 x (15-0) = -**4.7 N**

Negative or Positive torque will determine whether the spring is in a CW or CCW wind-up

The Spring/Damper can be used as a spring-only if the damping coefficient is set to 0. Similarly, it can be used as a damper-only if the spring constraint is set to 0.

Bushings

Bushings are 6 DOF flexible connectors. They do not remove any DOF because they are force-based not constraint-based elements. Because of this, bushing are very useful in helping to eliminate redundant constraints. For example, setting a stiffness value to a randomly large value (and damping anywhere between 10% and 100% of the stiffness) simulates that DOF being “locked” down. For example, if it was desired to use a bushing to simulate a revolute constraint, all three Fx, Fy, and Fz translations and two of the rotations would be assigned large K and C values to essentially lock those DOF. The remaining rotational DOF would have its stiffness and damping set to 0



Input properties for a Bushing. One tab controls translational properties. Another tab controls rotational properties.

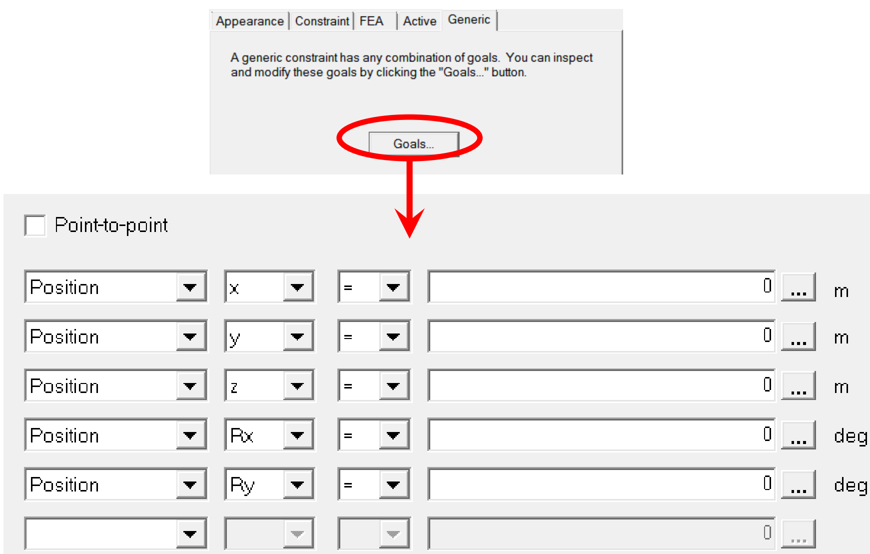
Bushings can add considerable time to a simulation, as they introduce 6 additional DOF into the model.

Bushing parameters may need to be modified if the resulting force and acceleration meter data for the bushings contains “noise”.

Generic Constraint

Generic constraints are free-form constraint types that allow any type of constraint to be represented by establishing goals for each DOF.

The user can specify which DOF are available and which inputs or conditions are applied to those DOF. For example, a generic constraint can also be used to represent a revolute constraint. In that case, all translational and two rotational DOF would be defined as position = 0. This essentially locks those DOF.



Input properties for a Generic constraint. Select the Goals button to activate the value entry fields

First defining any constraint, such as a spherical constraint, rope or bushing, and then changing that constraint type to a generic constraint will automatically populate the value fields with the data that is characteristic of that parent constraint. This is helpful in customizing, for example, a spherical constraint to have velocity applied to one of its rotational DOF.

Exercise – Geneva Wheel

Simulation Objectives:

- Determine contact force on drive pin
- Determine angular velocity output profile

Features Covered:

- Subassemblies
- Coords
- Collisions
- Constraints
- Motor
- Damper
- Stop Control
- Meters
- Running a Simulation
- Results Vectors

Open the SimWise file

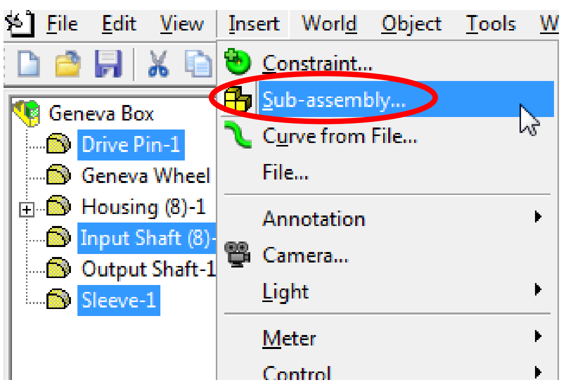
1. ***Start SimWise***

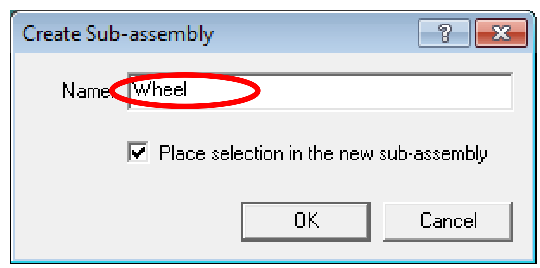
2. *Select **File, Open** and **Browse and locate** the file called "**SimWise Tutorial –Geneva Wheel.wm3**".*

Create Subassemblies

1. *In the **Object Browser**, use the **Ctrl key** to **multi-select select** the **Drive Pin**, **Input Shaft** and **Sleeve***

2. *From the main menu, **choose Insert**, **Sub-assembly** and name it "**Wheel**"*





3. ***Repeat steps 2 and 3** using the **Geneva Wheel** and **Output Shaft** parts. Name the new subassembly "**Geneva**"*

Create Rigid Subassemblies

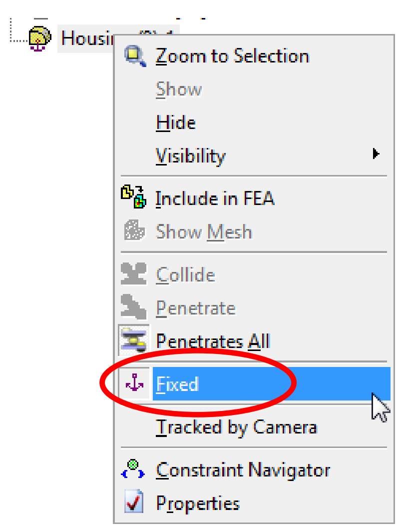
1. *In the **Object Browser**, **right-select** the subassembly named **Wheel** and **choose Rigidly Join Bodies***

The bodies will be joined with Rigid joints automatically.

2. ***Repeat step 1** for the "**Geneva** " assembly*

Fix Body to Background

1. *In the **Object Manager**, **right-select Housing** and **choose Fixed***



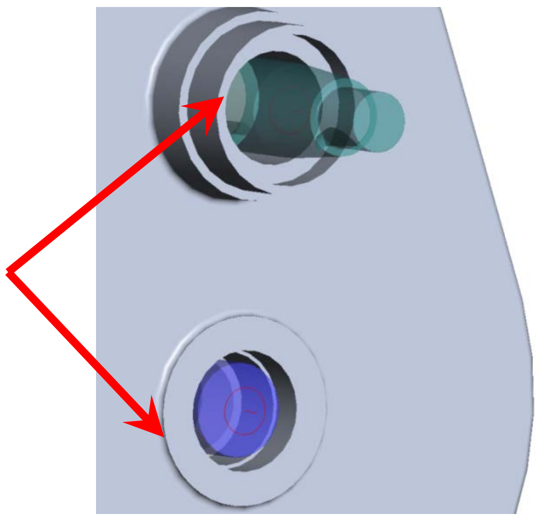
Create Coords (for constraints)

1. ***Rotate*** *the model to view the back side, as shown*

2. *Double-click the **Coord tool** and **select** the **two edges** shown on the **Housing**.*

Select circular edges

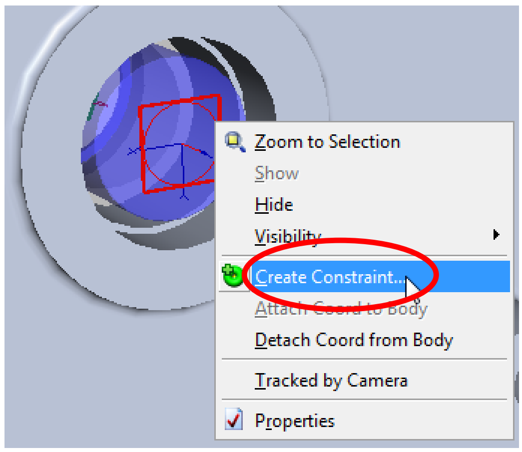
A coord will be created at each center point that defines each edge



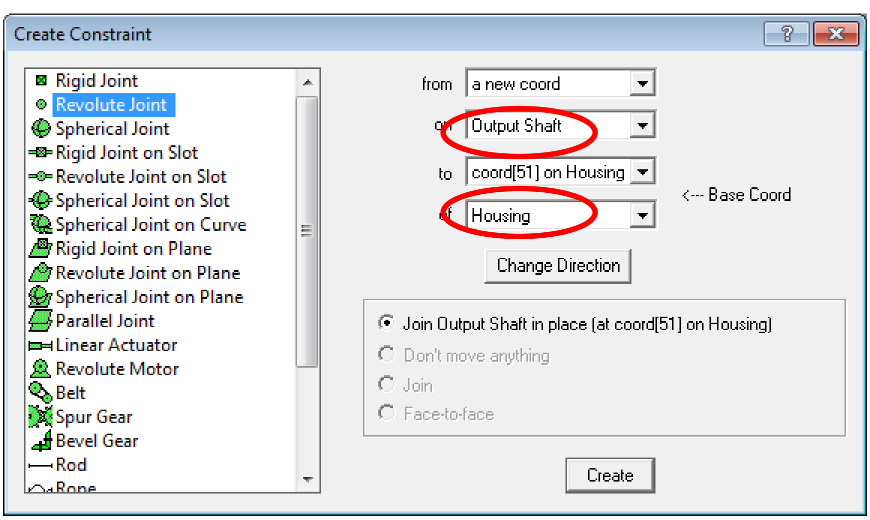
3. ***Click*** *on the **Select icon** to **cancel** the Coord creation*

Create a Revolute Joint

1. ***Right-select** the coord near the **Output Shaft** and choose **Create Constraint**.*



2. *When the Create Constraint window opens, **select Revolute Joint** as the joint type and **define** the constraint between the **Housing** and the **Output Shaft** (use a new coord on Output Shaft)*
   Note: The coord numbers and order of the bodies may be different in your model. This is ok.

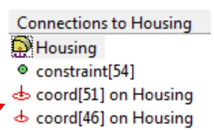


Create a Motor

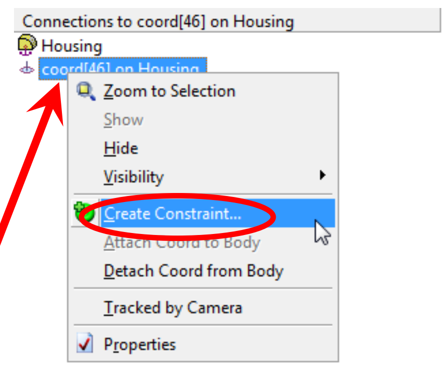
1. ***Select** the **Housing***

2. *In the **Connections List**, **hover the mouse** over each coord and watch the coord graphics highlight on the screen. **Highlight** the coord between the **Input Shaft** and **Housing**, **right-select** it in the list and **choose Create Constraint***

Hover Mouse over coords

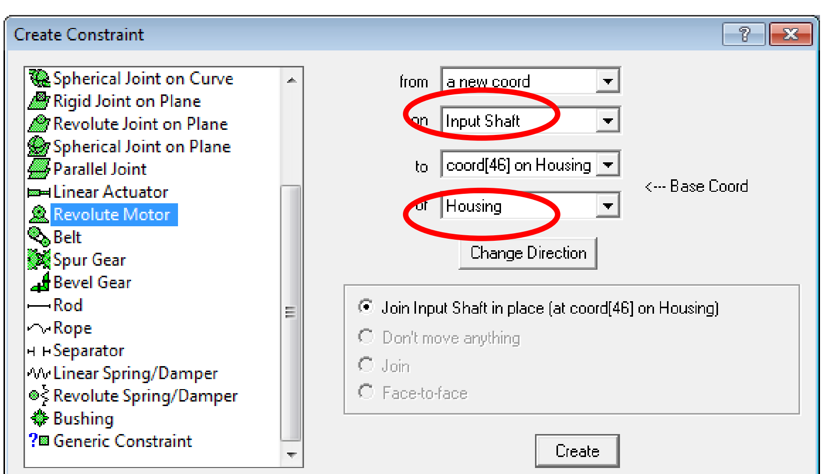


Right-Select highlighted coord



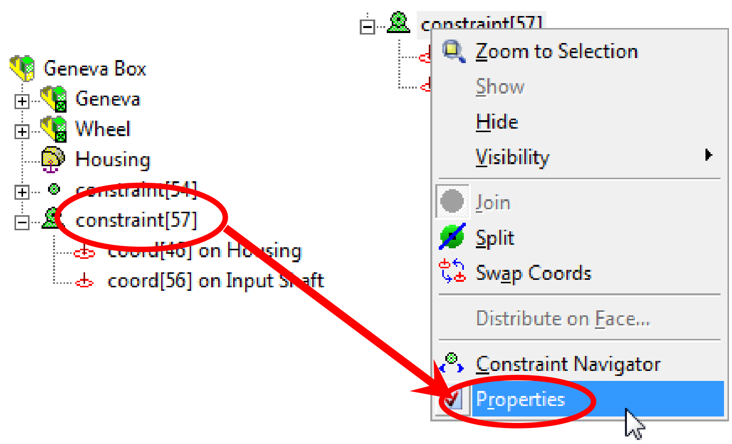
3. *When the **Create Constraint window** opens, **select Revolute Motor** as the joint type and **define** the constraint between the **Housing** and the **Input Shaft** (use a new coord on the Input Shaft)*

Note: The coord numbers and order of the bodies may be different in your model. This is ok.

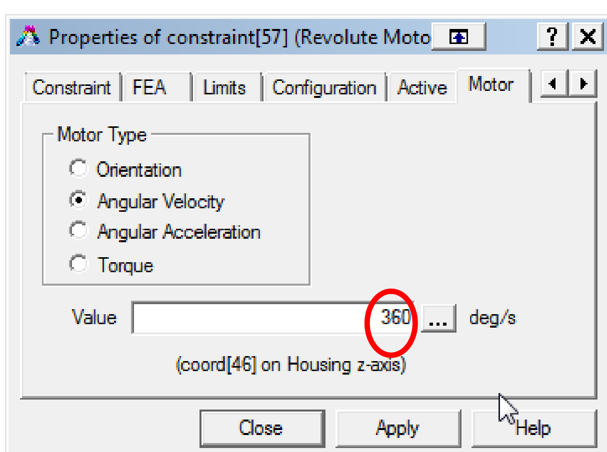


Create a Motor

4. *In the Object Browser, **right-select** the **motor feature** and **choose Properties***

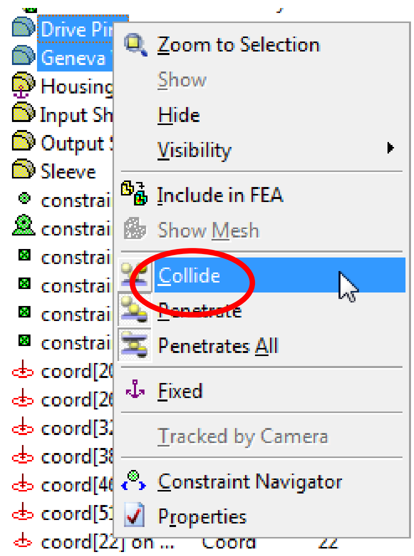


5. ***Scroll** to the "**Motor**" tab, **leave** the default Motor Type and enter **360** for the Value*



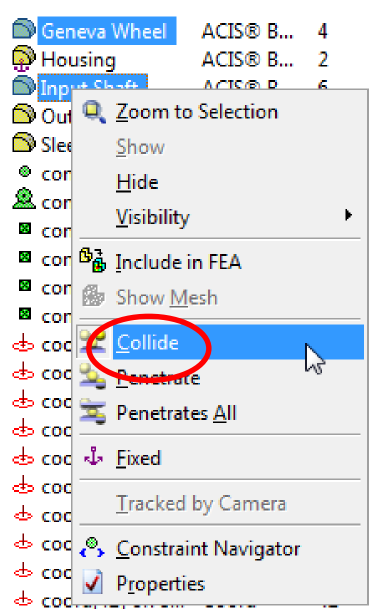
Add a Contact (Collision)

1. *In the* ***Object List****, **select** the* ***Sleeve****,* ***hold down the Ctrl key****, **select** the* ***Geneva Wheel****, **right-select** and **choose*** ***Collide***



Sleeve/ Geneva Wheel

2. *In the* ***Object List****, **select** the* ***Geneva Wheel****,* ***hold down the Ctrl key****, **select** the* ***Input Shaft****, **right-select** and **choose*** ***Collide***

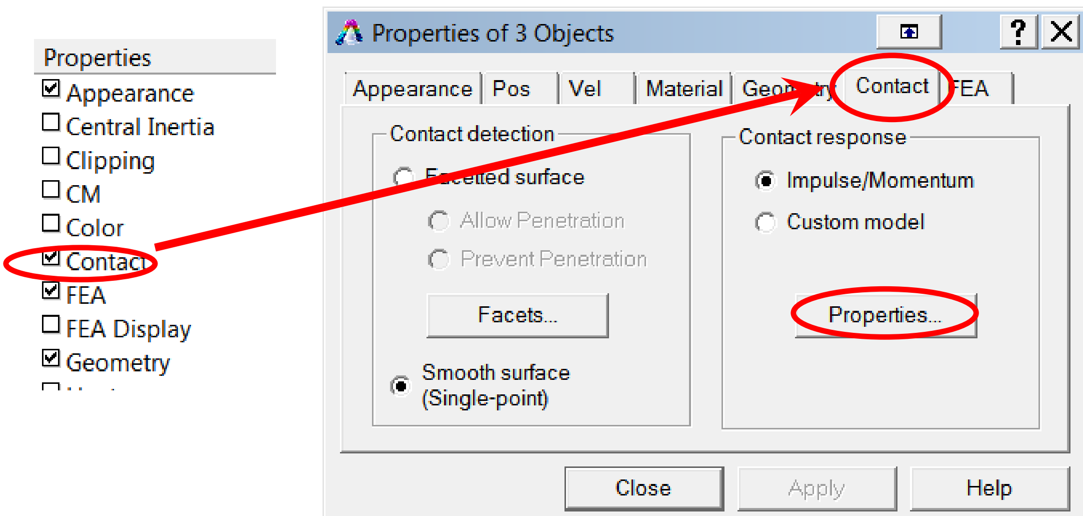


Geneva Wheel / Input Shaft

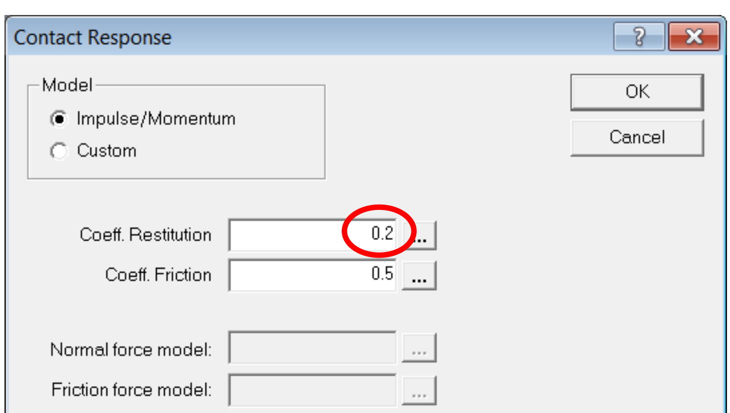
Define Collision Properties

1. ***Hold down the Ctrl key** and **select** the **Geneva Wheel**, **Input Shaft** and **Sleeve***

2. *In the **Properties List**, **select** the checkbox next to **Contact**. This will bring up the Body Properties d-box with the Contact tab activated*

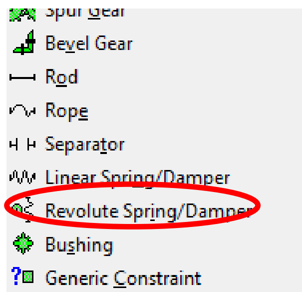


3. ***Select** the **Properties tab** and **change** the **Coeff Restitution** to **0.2**. **Select OK**, **Apply** and then **Close***

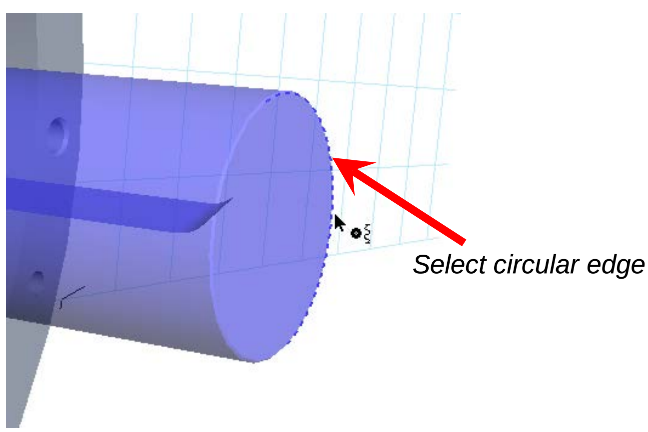


Add a Damper

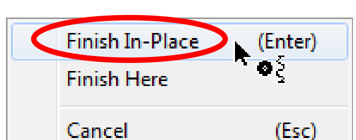
1. ***Click*** *on the* ***constraints tool button*** *and on the drop down list* ***choose Revolute Spring/Damper***



2. ***Select*** *the* ***circular edge*** *on the* ***Output Shaft*** *as shown.*

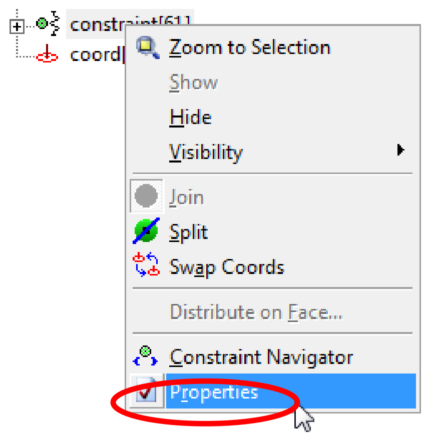


3. *While the cursor is still showing the spring/damper icon,* ***right-select*** *and* ***choose Finish In-Place*** *(or press Enter)*



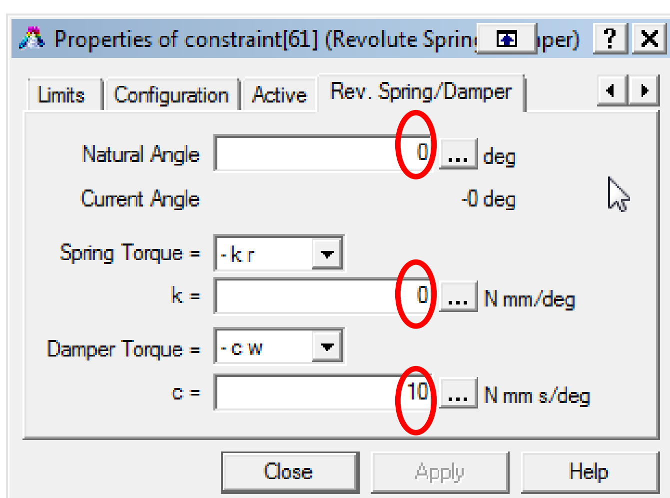
Add a Damper

4. *In the **Object Browser**, **right-select** the **Spring/Damper** feature and **choose Properties***



5. ***Scroll** to the "**Rev. Spring/Damper**" tab and **set the values** as shown*

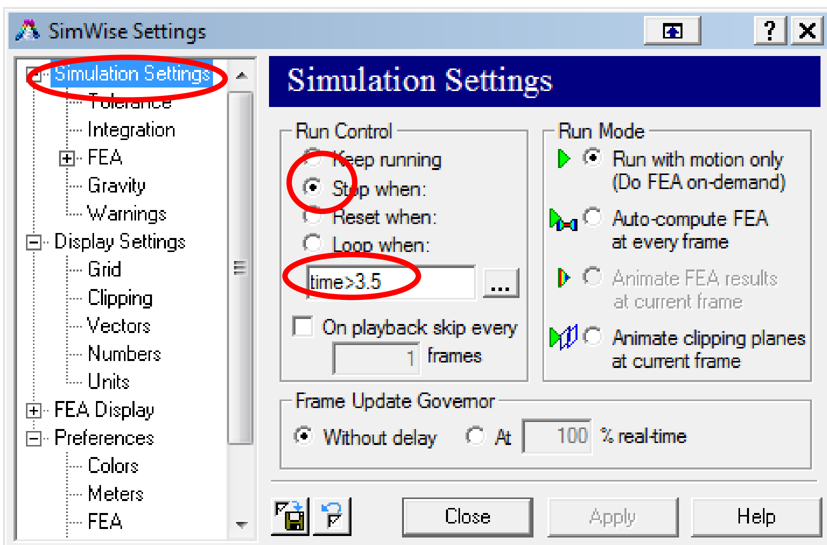
The damper will be used to represent a form of external resistance on the end of the Output Shaft



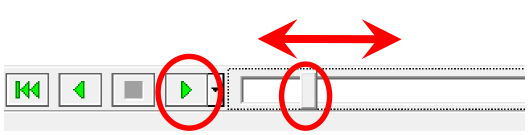
Define a Stop Control and Run the Simulation

1. ***Select** the **Simulation Settings** ico* 

2. ***Select** **Simulation Settings** inside the Simulation Settings window (if not already showing) and **define** a **Run Control** that **Stops** the simulation when **time>3.5**, as shown*



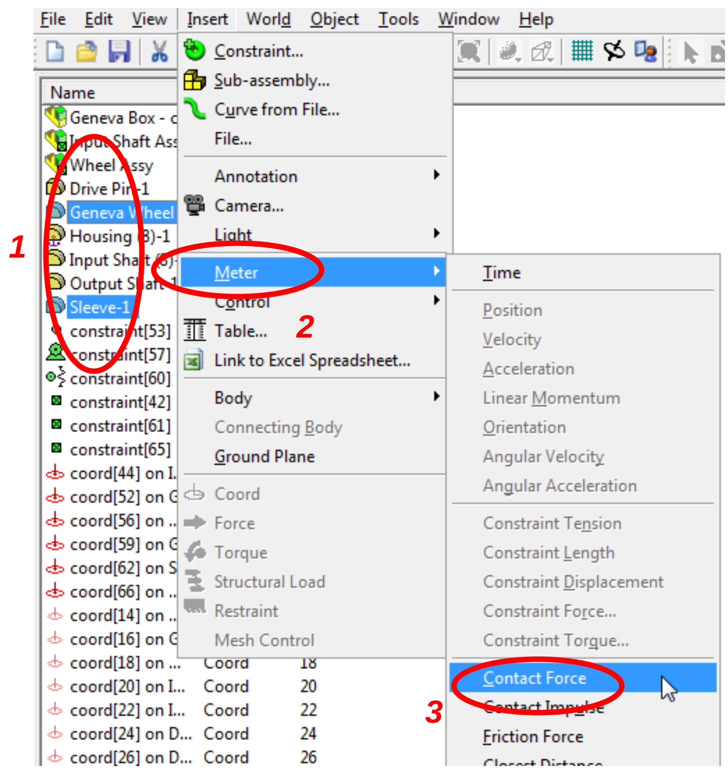
4. ***Select** the **Run** button located on the **Player Control Panel** to run the simulation. Once complete, use the controls on the **Player** , as shown, to replay the animation*



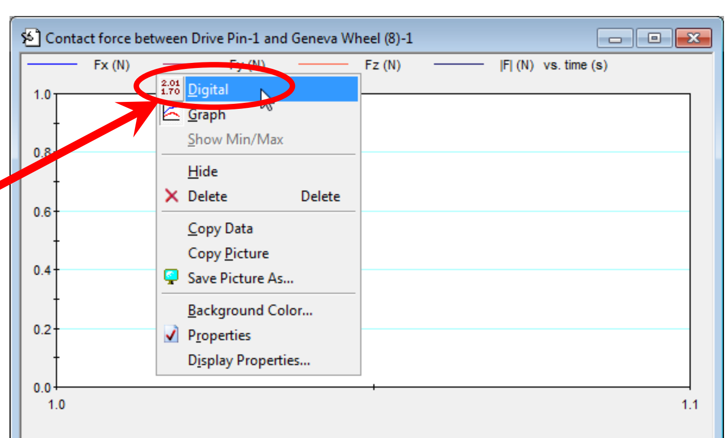
To advance the simulation playback, you can either 1) click, hold & drag the slider button or 2) click on the Play button

Create Meter – *Contact Force*

1. *In the **Object List**, **hold down** the **Ctrl key** and **select** the **Sleeve** and the **Geneva Wheel**, and from the main menu **choose Insert**, **Meter**, **Contact Force***

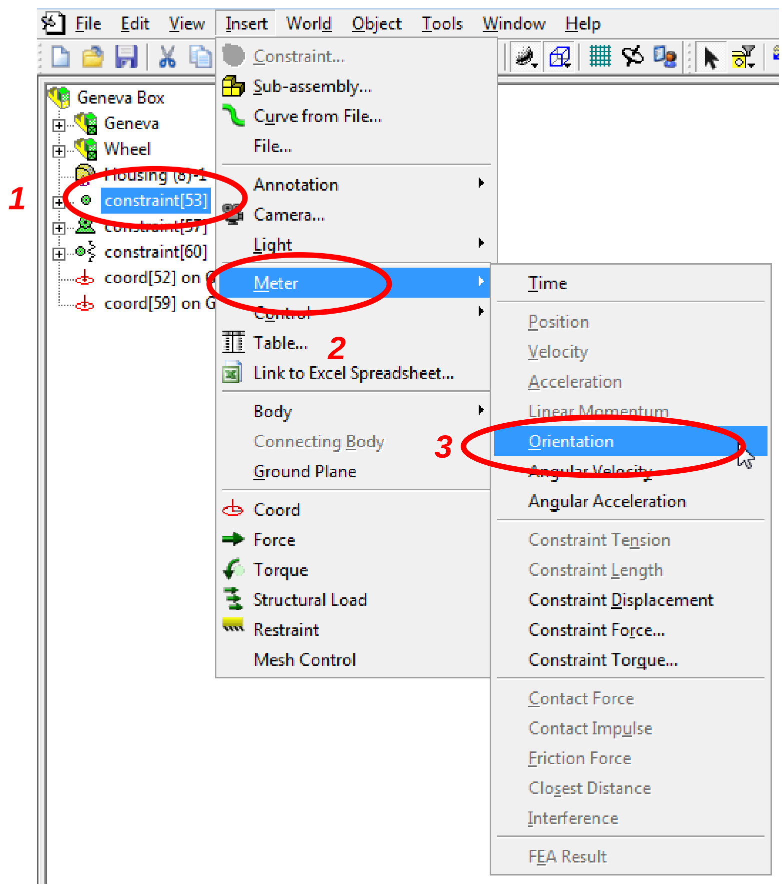


Tip: You can right-select on the meter and choose Digital to view a numerical readout instead of graphical



Create Meter – Geneva Wheel Orientation

1. *In the **Object List**, **select** the Revolute joint connecting the **Output Shaft** to the **Housing** and from the main menu **choose Insert**, **Meter**, **Orientation***

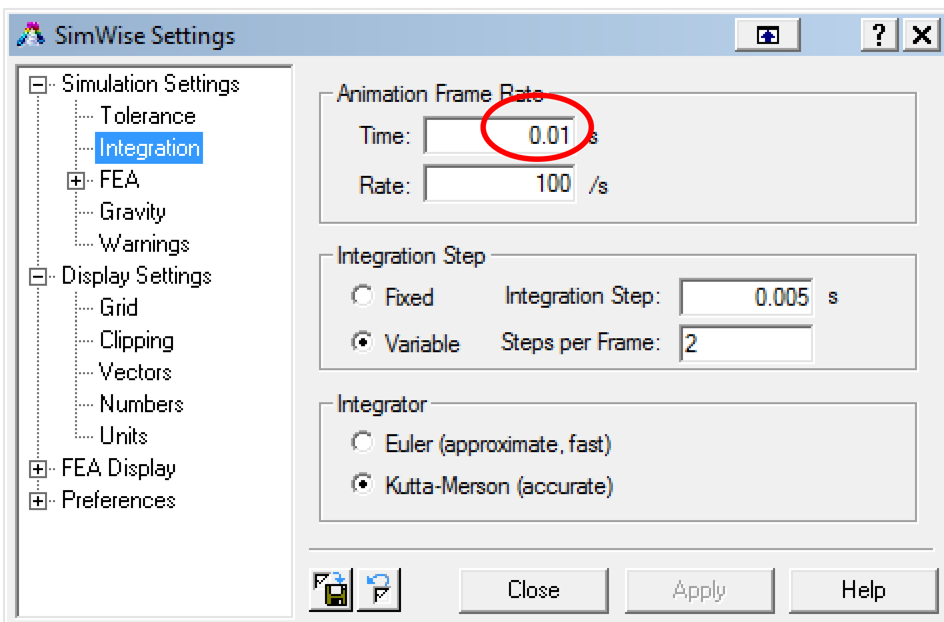


Adjust Simulation Settings

1. ***Select** the **Simulation Settings** **icon*** 

2. ***Select** **Integration** from the **Simulation Settings** submenu*

3. ***Set** the **Time** to **0.01**, then **select** **Apply** and then **Close***



This change will:

1) *Help create more data points on the meter outputs for smoother graphs*

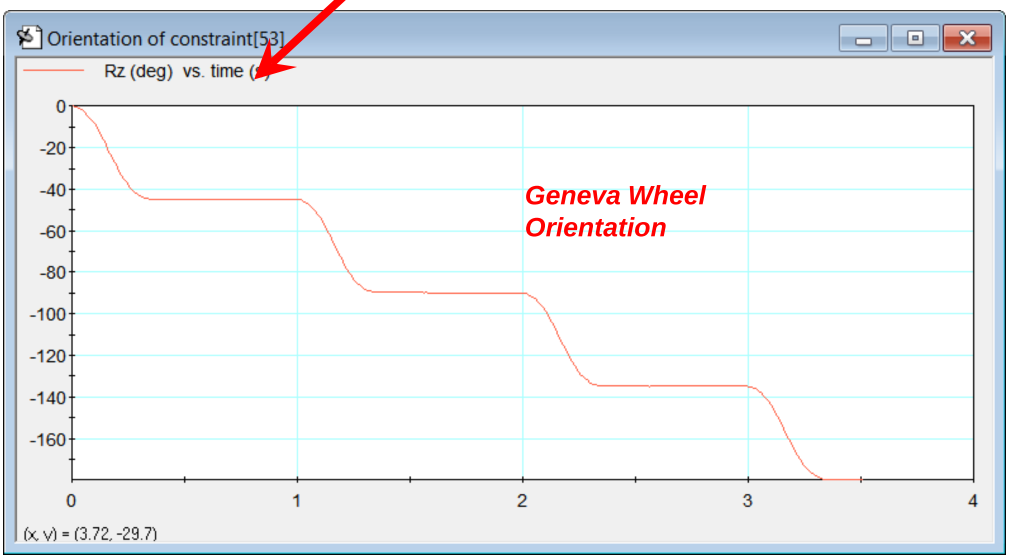
2) *Produce a "smoother" video animation playback*

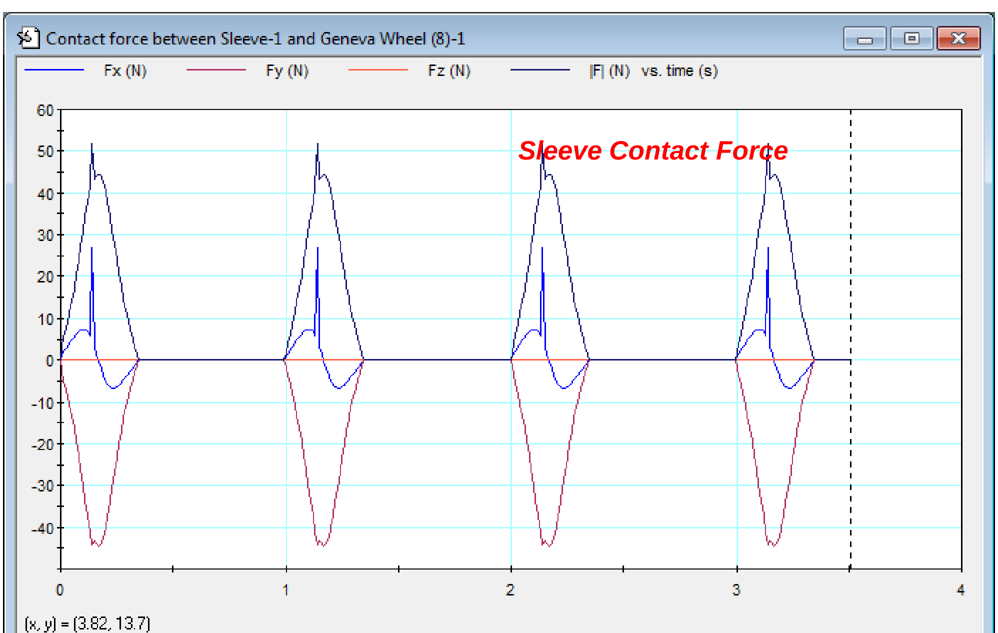
3) *Improve the accuracy of the results*

Observe Results

1. ***Run the simulation*** *and* ***observe*** *the results*

Tip: Click on any of the Rx, Ry or Rz labels to hide or show the corresponding meter data

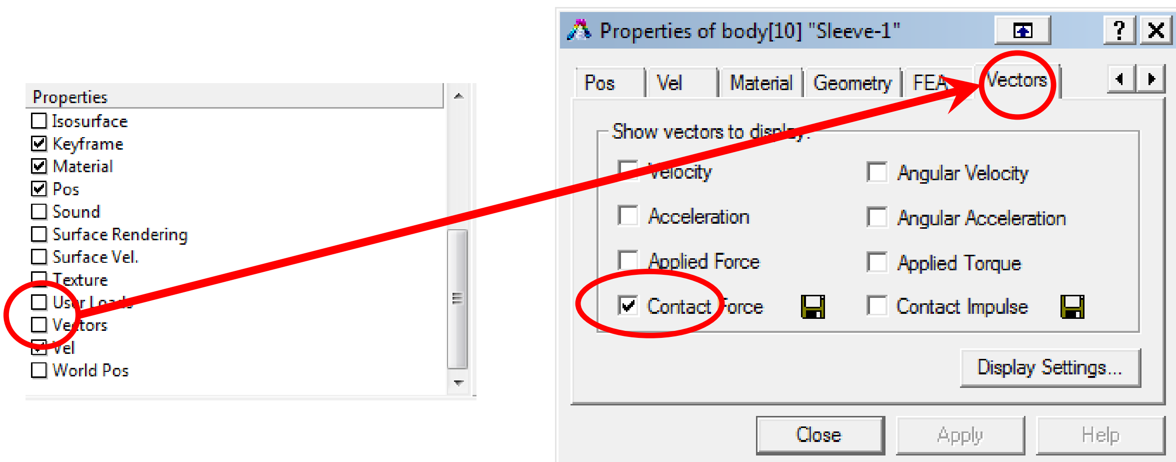




Create a Results Vector *(Contact force on Drive Pin)*

1. *In the* ***Object Browser****,* ***Object List*** *or* ***graphics area****,* ***select*** *the* ***Sleeve***

2. *In the* ***Properties List****,* ***check*** *the box next to* ***Vectors****. This should automatically open the* ***Drive Pin Properties*** *window with the* ***Vectors tab*** *as the active tab*

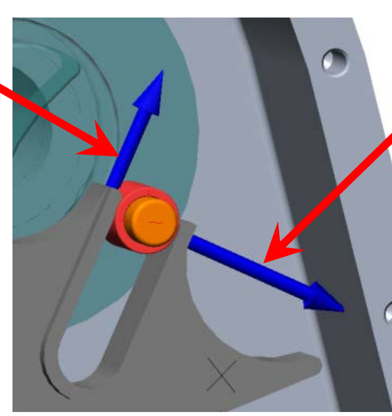


3. ***Check*** *the box next to* ***Contact Force*** *and* ***close*** *the dialog*

Contact normal force

Contact friction force

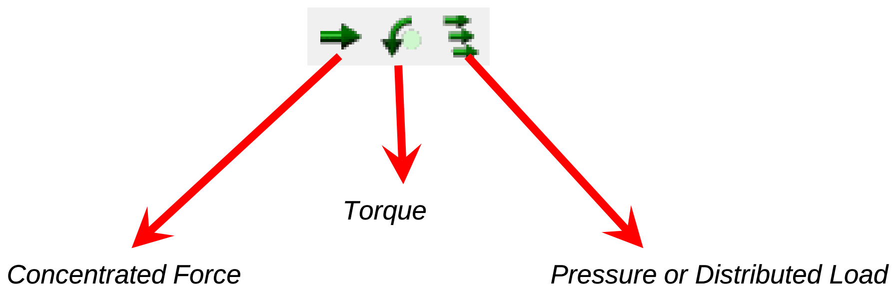
You do not need to re-solve your model for the vectors to appear. Simply use the Player controls to playback the animation already stored

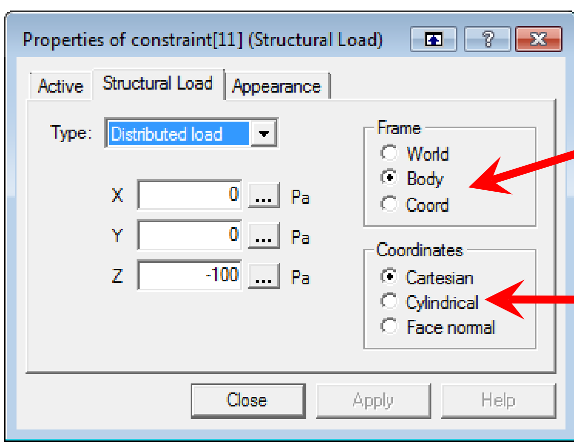


End of Exercise

Forces

SimWise offers the ability to add a force, pressure or torque





Reference Frame: With respect to World, Body or the parent coord

Coordinates: Choose the type of coordinate entry

***Total Force** – Force applied to a face*

***Distributed Load** – Pressure with X, Y and Z components*

***Concentrated Force** – Force applied at single point*

***Pressure** – Pressure applied to face (normal to face)*

***Torque** – Torque applied on a surface (note: the torque icon may not lie exactly on the face. This is strictly visual, it is actually applied to the selected face.*

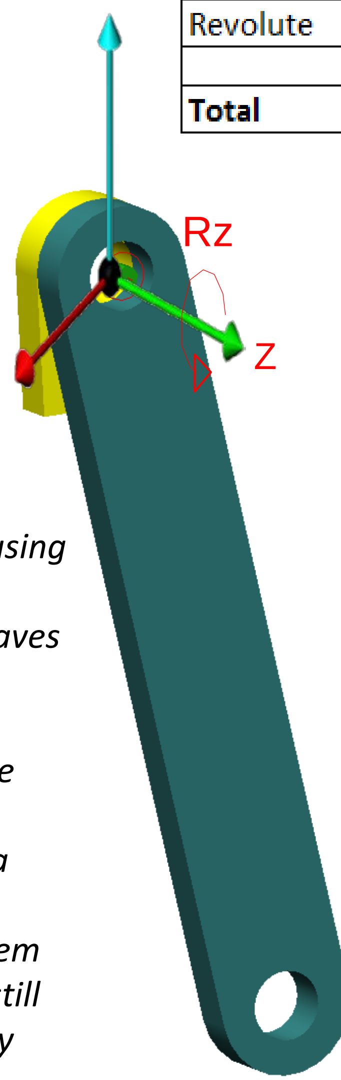
Degrees of Freedom (Background)

- In rigid body kinematic and dynamic modeling, each body has only 6 allowed degrees of freedom (DOF), translation in Global X, Y and Z and rotation about Global X, Y and Z.

	DOF
Fixed Block	0
Pendulum	6
Revolute	-5
Total	**+ 1**

- Kinematic constraints remove DOF from bodies. For example, a revolute joint allows one rotational DOF. In other words, you can say it removes 5 DOF.

- Which of the six DOF are removed from a body depends on the type of constraints and the number of constraints used on that body.



Example:

A pendulum is connected to a fixed block using a revolute joint. The revolute constraint removes 5 DOF from the pendulum and leaves 1 rotational DOF.

If a motor was used in place of the revolute constraint, the motor is also considered a constraint and removes the same DOF as a Revolute but it also removes a DOF for the main rotational axis. In this case, the system would have 0 DOF. Even though it would still rotate about the Z axis, this rotation is fully controlled (or defined).

Redundant Constraints *(Example)*

	DOF
Door	6
Revolute1	-5
Revolute2	-5
Total	**-4**

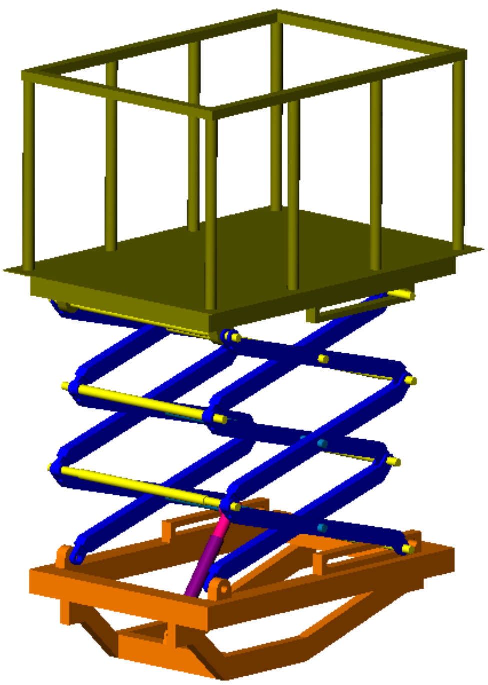
Example:
Consider a door hanging by two hinges. Physical instinct would tell us to connect the door to the frame using two Revolute joints. However, in rigid body motion analysis, this provides more constraining than is necessary.

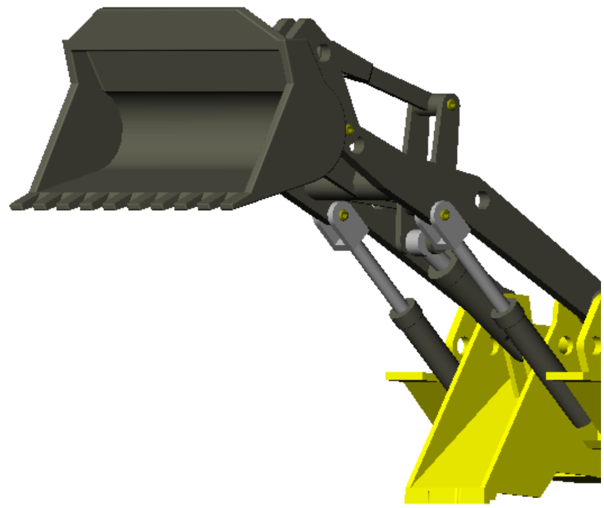
The picture and chart shows the calculation of the degrees of freedom after adding two revolute joints.

The system is over constrained.

Redundant Constraints (Parallel Mechanisms)

- Parallel mechanisms are examples of models that can be easily over constrained.





***Examples – Parallel Mechanisms**:*

One complete side is redundant with the other

Not only can the parallel linkages cause redundant constraints but, in the case of the loader, the two motion-driven actuators acting upon the same linkage are also redundant with one another . In such a model, it would only be possible to determine force requirements on one of the actuators

Redundant Constraints (The Problem)

- The SW constraint equations define the position and orientation for the particular DOFs of the body. Only one set of equations is necessary to define the DOF of a body. When constraint equations are solved for, forces and moments are used to satisfy the solution to these equations. It is these same forces and moments that the user can measure or create meters from.

- Adding in more equations by way of adding more joints creates redundancies. In other words, one constraint becomes redundant with another trying to control the same DOF.

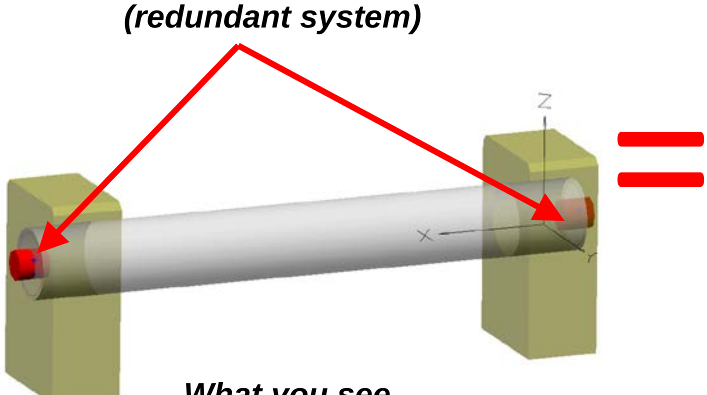
- In order to make the simulation solvable, the SW solver will randomly remove any redundant DOF equations. As a result, there will be no reaction force present in the redundant direction for this constraint.

Redundant Constraints *(What you see vs. What you get)*

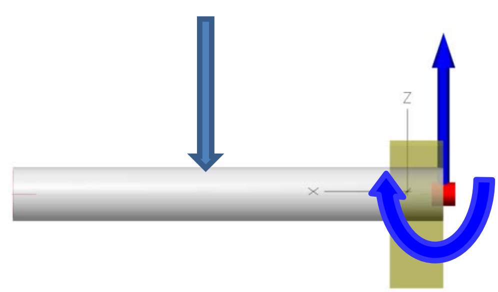
- Redundant constraints can lead to inaccurate force and moment results plotted from certain constraints.

Example:
Consider a shaft supported at each end by a bearing (revolute joint). Each revolute joint removes 5 DOF. The total DOF of the system is therefore -4 DOF (+6-5-5).

Let's look at a specific DOF that is redundant in this model and is in a direction related to force reactions we seek, the vertical Z direction. Each revolute joint attempts to establish the constraint equations to control the DOF of the shaft for this direction (keep it from translating in Z). As a result, the solver allows only one of the joints (right) to govern the equations for this particular DOF and considers the other constraint redundant. As a result, the solver now sees the problem as that of a cantilevered shaft. In other words, all vertical load is absorbed by one joint only, let's say the right side joint. As a result, there will one reaction force and a bending moment, due to the offset of the center of mass. In reality, there should be no moment, as the shaft should instead be supported by two equal vertical reaction forces.



What you see



What you get
(Not desired)

Redundant Constraints *(Preventing by using Constraints)*

- In most cases, redundant constraints can be avoided. It simply takes some familiarity with the different constraint types and an understanding of which constraints to use in certain modeling scenarios. Preventing redundancies may require using a combination of various less-common constraint types to produce the same kinematic result as when using common constraint types.

  Example:
  Consider a shaft supported by two bearings. Using two revolute constraints results in a redundant system and produces only one vertical reaction and also an unwanted reaction moment (Fig A). In contrast, using a spherical joint on one end and a spherical joint on slot on the other yields +1 DOF for the system (Fig B). This is a more desirable approach.

	DOF
Bearing Block	0
Bearing Block	0
Shaft	6
Revolute	-5
Revolute	-5

	DOF
Bearing Block	0
Bearing Block	0
Shaft	6
Spherical	-3
Spherical on Slot	-2
Total	**+ 1**

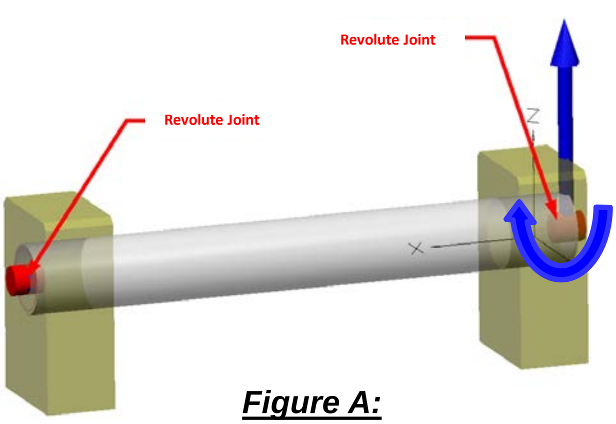


Figure A:
Not Good Practice

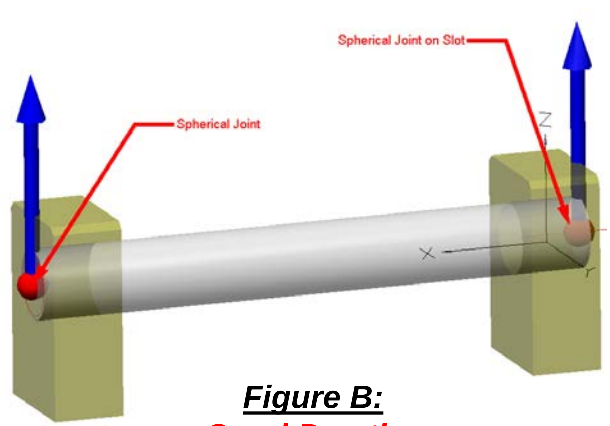


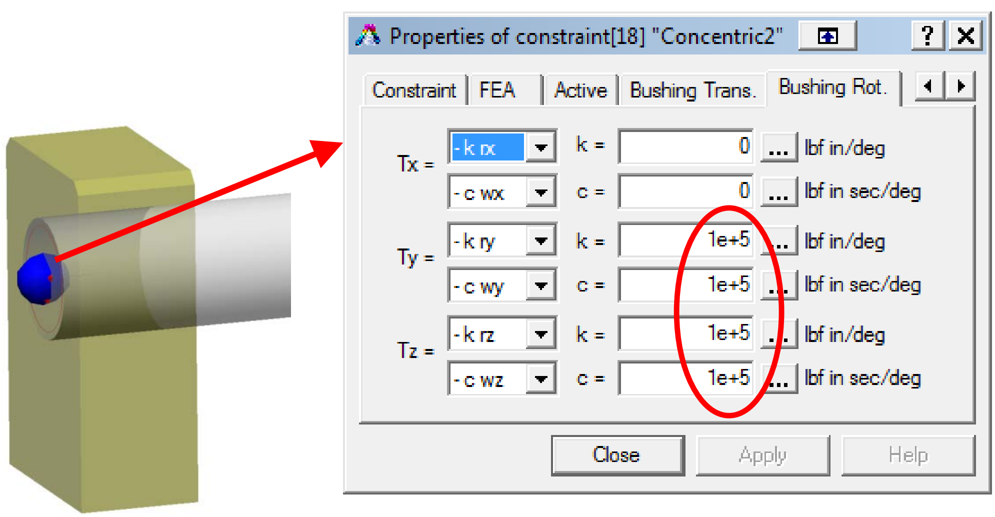
Figure B:
Good Practice

Redundant Constraints ***(Preventing using Bushings)***

- In some cases, it may be difficult or impossible to completely remove redundancies using kinematic constraints only. In those cases, **Bushings** can be used.

- Unlike kinematic constraints, bushings are based off of force equations only. They are represented essentially as 6 DOF linear/non-linear spring/dampers. High Stiffness values represent “locked” DOF. 0 stiffness values represent DOF that are free to move.

<u>Example</u>:
Using a bushing to act as a revolute joint



- *Rotations about Y and Z are set to higher stiffness and damping to prevent displacement*

- *Rotation about X has 0 stiffness and damping. This allows the shaft to spin freely around its axis*

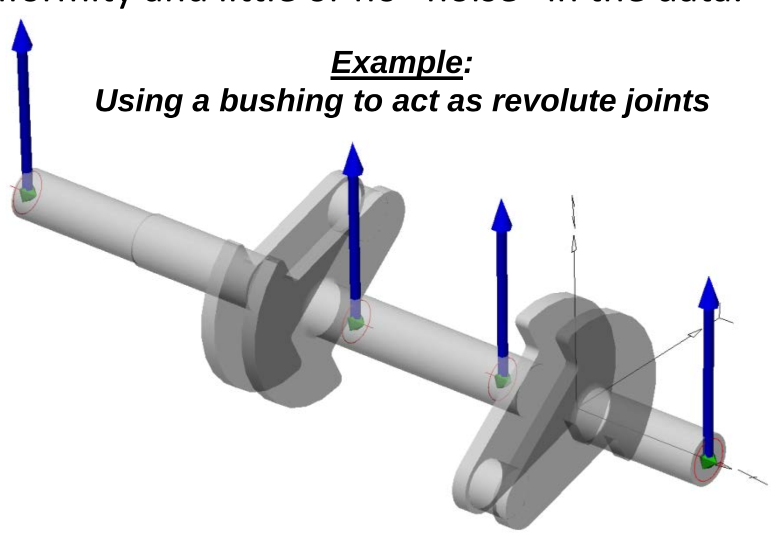
- *In this example, all 3 translations would also be set to higher stiffness and damping values, considering there is no translation allowed.*

Redundant Constraints *(Preventing using Bushings)*

- The advantage of using a bushing is that they do not remove DOF as the kinematic constraints do. The potential disadvantage is that the system now has many DOF and moves from one requiring a faster kinematic solution to one requiring a more involved dynamic solution. Dynamic solutions call for additional solver calculations and simulations can take longer to run.

- In some bushing simulations, making adjustments to the stiffness, damping and solution accuracy settings (smaller time step), can help improve performance time and the accuracy of the results. It is helpful to plot the bushing forces and watch for uniformity and little or no “noise” in the data.

Example:
Using a bushing to act as revolute joints



- *In this example, the bushings shared the same stiffness and damping values, 1e5 and 1e2, respectfully. The Tz rotation direction, however, was set to 0 stiffness and 0 damping for all bushings*

- *The following image shows the reaction force vectors at each of the four bushings*

- *In this case, bushings perform as revolute joints do but they do not remove any DOF*

Redundant Constraints (Summary)

- Redundant constraints are an issue only if the user needs to determine reaction forces or moments on particular constraints OR the user needs to perform an FEA study using motion-calculated load information.

- In a redundant model, a kinematic simulation can still run and calculations for displacements, velocities and accelerations will be accurate.

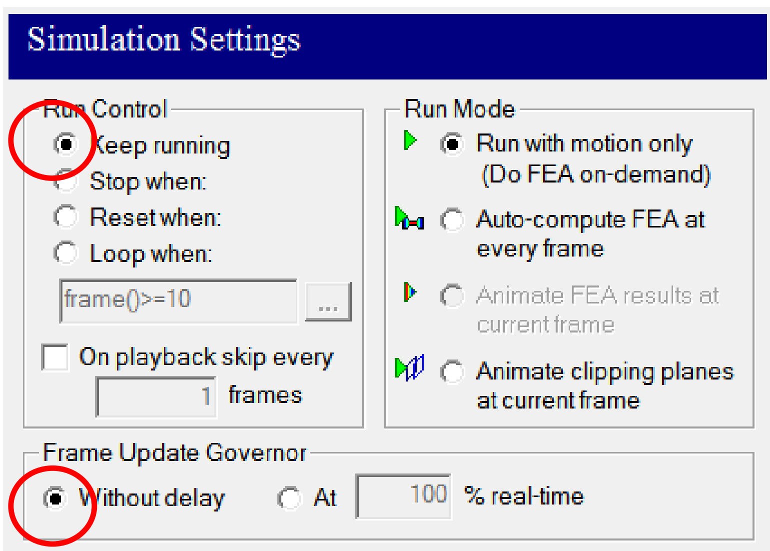
- A redundant model may cause some constraint directions to yield 0 force readings and can induce reaction moments where there should not be any.

- In preventing redundant constraints, it may be necessary to use a combination of less-common constraints that are not as intuitive as the basic more common constraints. For example, it is not intuitive to use a spherical joint and a spherical joint on slot to constrain a door with two pivoting hinges. But it is a viable approach in rigid body simulation.

- Bushings can be used where it becomes difficult to prevent redundant constraints using kinematic joints. For example, there is no way to constrain a shaft to three bearings using kinematic constraints and not have the system be redundant. The system becomes statically indeterminate. Bushings are force-based constraints and do not remove any DOF.

***Simulation Settings** (Run Control & Playback)*

Run control allows the simulation to either stop, reset or loop when a certain user-defined criteria is met. For example, the simulation can be assigned to stop when the x-direction velocity of a body exceeds a certain value. In such an example, the formula defined in the control value would look like: *body[1].v.x>500*

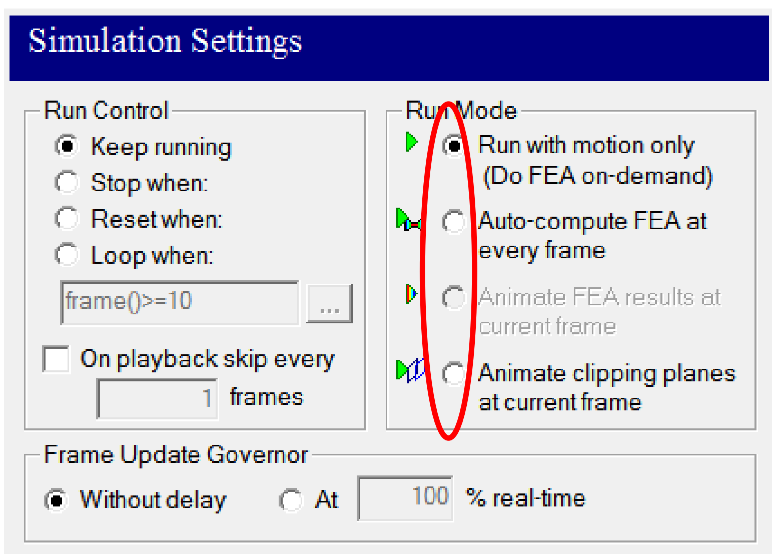


Simulation Settings d-box

The user can speed up the playback of an existing animation by skipping frames. This value only affects the playback of an existing simulation. It does not affect the frames for the simulation solution process.

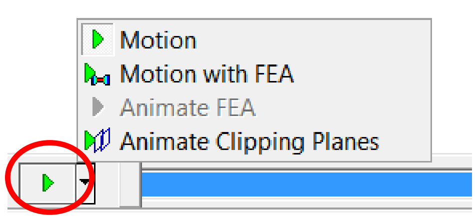
Simulation Settings *(Run Mode)*

Run Mode allows for the specification of performing either 1) a motion-only simulation, 2) a Motion+ FEA simulation. It also has options for animating the FEA results (exaggerated deformation) and clipping (cutting) planes propagating through a body or the assembly



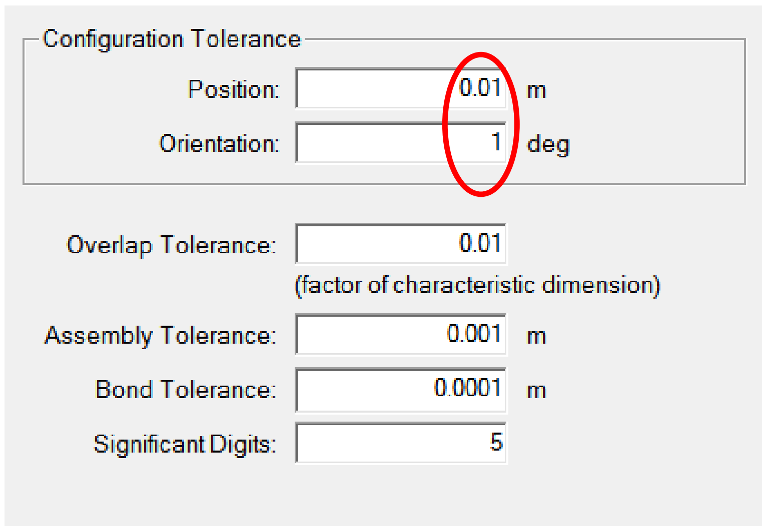
Simulation Settings d-box

The Run Mode settings can also be accessed from the simulation panel



Simulation Settings *(Configuration Tolerances)*

The simulation tolerance settings control to solution accuracy criteria.



Simulation Tolerance d-box

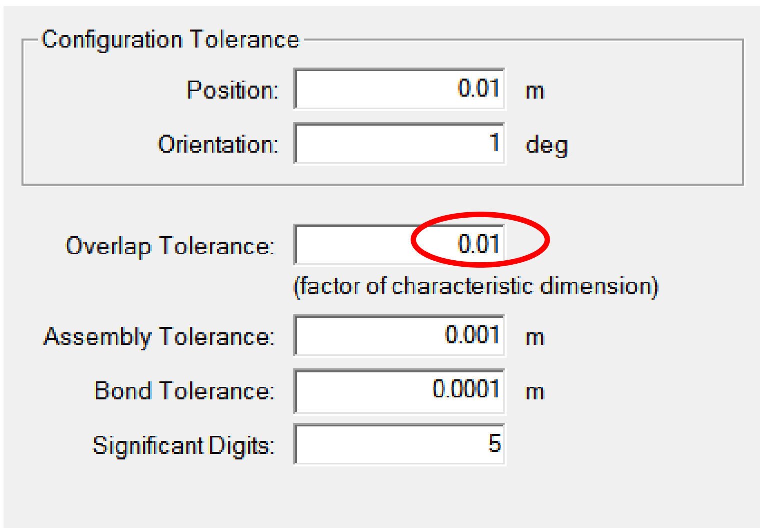
In addition to the Animation Time, the ***Position* and *Orientation*** tolerance settings are the most important settings for controlling accuracy.

If the simulation meter data contains noise, and all other model features, such as dampers, bushings and collisions have been ruled out as possible causes for such data noise, these settings can help improve run time and data output.

Smaller values will tighten the accuracy criteria but can add additional simulation time.

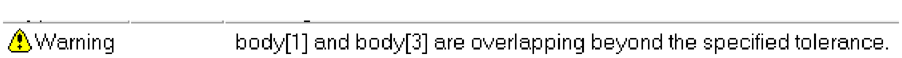
Simulation Settings *(Overlap Tolerance)*

Overlap Tolerance greatly affects the modeling of contacts/collisions. Bodies will not be allowed to penetrate beyond this value.



Simulation Tolerance d-box

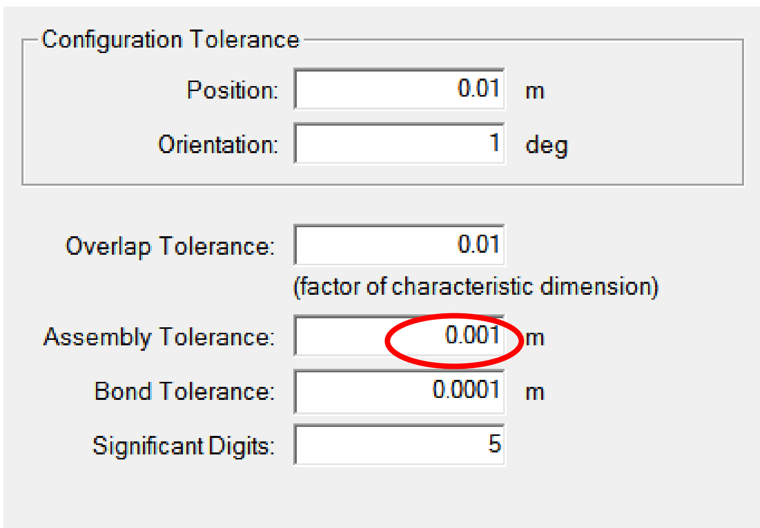
If, at the beginning of a simulation, two bodies are penetrating by more than the overlap tolerance, a warning message will appear:



Smaller values will improve the results accuracy but may also increase simulation time

Simulation Settings *(Assembly Tolerance)*

Assembly Tolerance specifies the accuracy with which to assemble a mechanism together. For example, when the user uses the “join” or “assemble” features in SimWise, this value determines how accurate the assembly process is.

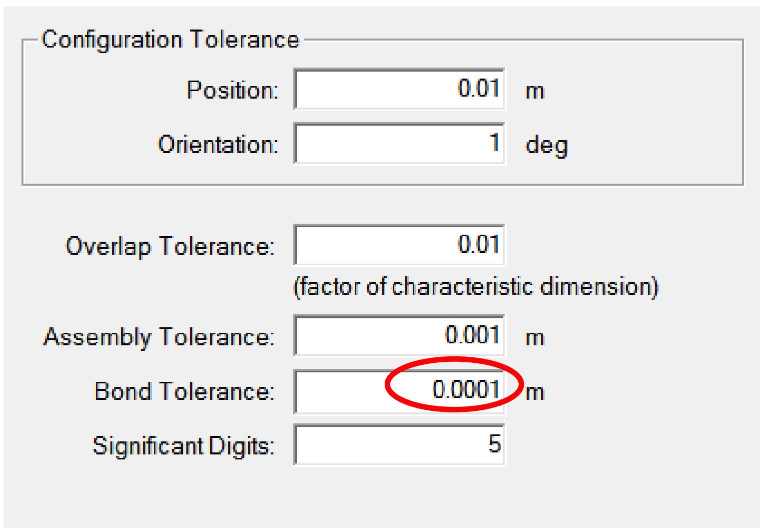


Simulation Tolerance d-box

SimWise may produce an error message if the bodies are not able to be assembled within the specified tolerance

Simulation Settings *(Bond Tolerance)*

The ***Bond Tolerance*** is the maximum amount two bodies can be separated and still be bonded together using the FEA bond option. If the separation is greater than this value, bonding will not take place



Simulation Tolerance d-box

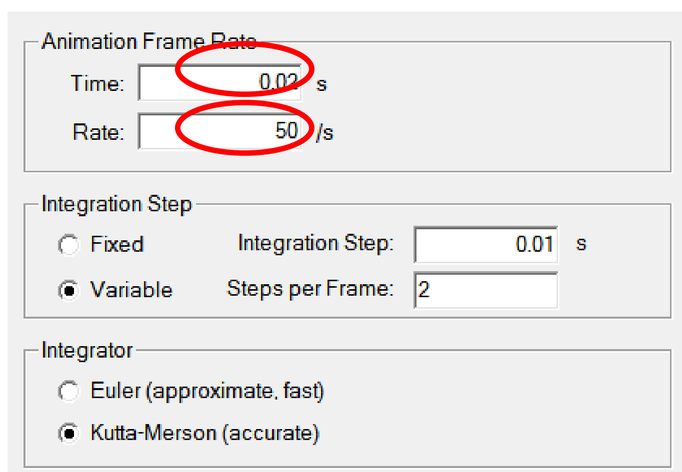
For the bonding feature to work, there must be a constraint defined between the two bodies to be bonded, even if it is a rigid constraint. Also, the bodies must be included in FEA.

Simulation Settings *(Integration)*

The ***Time*** parameter specifies the time between simulation output steps. The value used should be representative of the inputs in the model and the desired quality of the output. In other words, this value should be sufficient enough to capture data when the simulating models with high velocities or very short simulation times.

The ***Rate*** parameter specifies the number of frames to be used for each simulation second. This value is automatically determined when the Time is entered. Or, if the Rate is changed, the Time will automatically be determined.

The Time and Rate values also affect 1) the number of frames used to generate the playback animation and 2) the number of points used to generate the plots in the meters. Decreasing these numbers will increase the quality of both but may also hinder simulation speed.

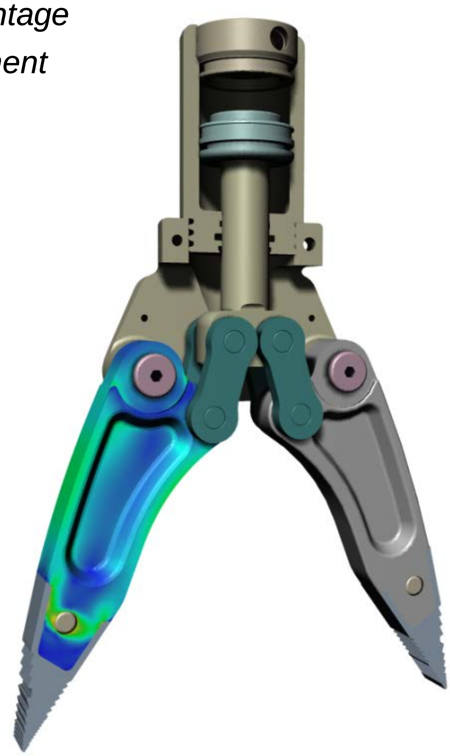


The Integration Step and Integrator settings are rarely adjusted by the user. The variable integration step is preferred over the fixed, especially for models with many DOF. Refer to the program HELP for additional information on these settings.

Exercise - Gripper

Simulation Objectives:

- *Determine force requirements for actuator*
- *Determine clamping mechanical advantage*
- *Determine maximum stress in component*

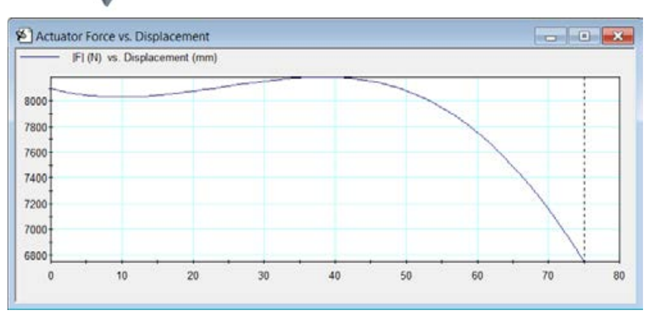


Main Features Covered:

- *Constraints*
- *Actuator Input*
- *Gravity*
- *Unit Settings*
- *Applied Forces*
- *Meters and Meter Customization*
- *Running Simulations*
  - *Motion*
  - *Motion + FEA*

Extra Features:

- *Stop Control*
- *Dimensions*
- *Annotations*
- *Constraint Navigator*
- *Keyframed Animation*
- *Export Video Animation*



Introduction

This tutorial is designed to introduce you to some of the basic capabilities of SimWise and help you get acquainted with how to prepare, run, and explore a basic motion and finite element analysis. Although not mandatory, it is highly recommended that you review the section titled *FEA Modeling Basics*.

You will start the tutorial by opening a SimWise model of a Gripper mechanism, which is based on the principals of a four-bar linkage. You will then define the necessary kinematic constraints, and actuator input and external forces. Finally, you will analyze the model for kinematic and structural characteristics.

Mechanism Background

Whether the Gripper is to be powered hydraulically or pneumatically is irrelevant to the simulation. In either case, there will be an average amount of input force required by the actuator to achieve a specified clamping force. With SimWise you will be able to use a reverse-engineering approach by specifying the desired clamping force and then having the simulation calculate the ***force input requirements*** *at the actuator.*

An important aspect of the Gripper is the mechanical advantage (or disadvantage) it offers. Whether it is a mechanical advantage or disadvantage will depend on the size and arrangement of the linkages. In the case of a mechanical advantage, a small actuator input force will produce a larger output clamping force. And in the case of a disadvantage, the opposite will be true. With SimWise you will be able to easily plot the output vs. input characteristics of the mechanism and determine the overall ***mechanical advantage.***

Finally, the strength of the mechanism is important in determining ***whether or not mechanism components will fail*** *(or deflect too much) while under normal operating conditions. With SimWise you will use the "Motion with FEA" feature to perform a simultaneous kinematic motion and Finite Element Analysis on a critical component, being sure to cover all ranges of possible load conditions and determining whether or not the component will reach the point of yielding (onset of failure).*

Open the SimWise model

1. ***Start SimWise***

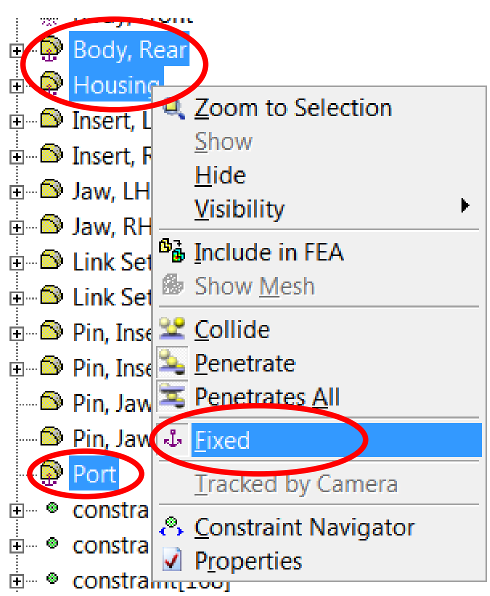
2. *Select **File, Open** and **Browse and locate** the file called “**SimWise Tutorial – Gripper.wm3**”.*

Ground Bodies

When a body is not needed in the movement of the mechanism, it can be grounded (anchored) to the background. This helps reduce model complexity and increase performance in the solution process.

1. ***Hold down** the **Ctrl key** and **select the Body,Rear**, **Housing, Body,Front** and **Port** from the browser*

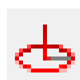
2. ***Right-click** on any of the highlighted bodies and **choose Fixed***



Add Constraints

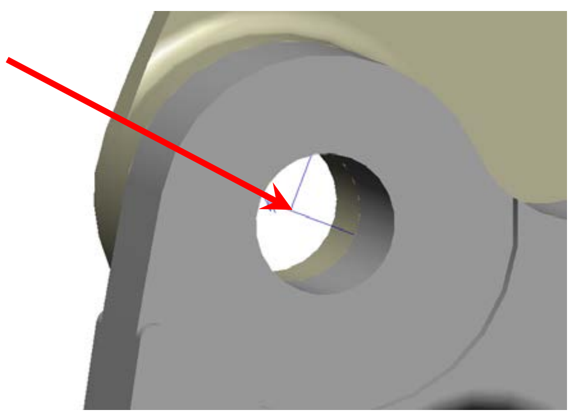
In the following steps (adding constraints), one constraint definition will be outlined in detail so that you may gain familiarity with the procedure to create a constraint. The remaining constraints can be added by using the diagram provided on the next page.

Tip: When adding constraints, it may be helpful to hide/show parts as needed. You may also change the translucency of parts.

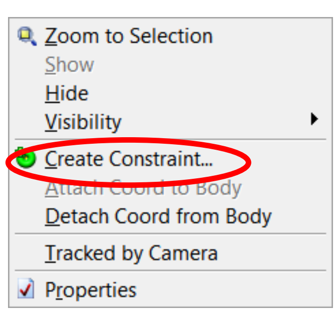
1. *Click on the coord icon*  *on the Edit toolbar*

2. *Hover the mouse over the circular edge inside the hole (Body, Rear and Jaw, LH), as shown, until the cursor changes to a circle with crosshair and then left-click the circular edge to create the coord*

Coord symbol



3. ***Right-click** on the coord and **choose** **Create Constraint***

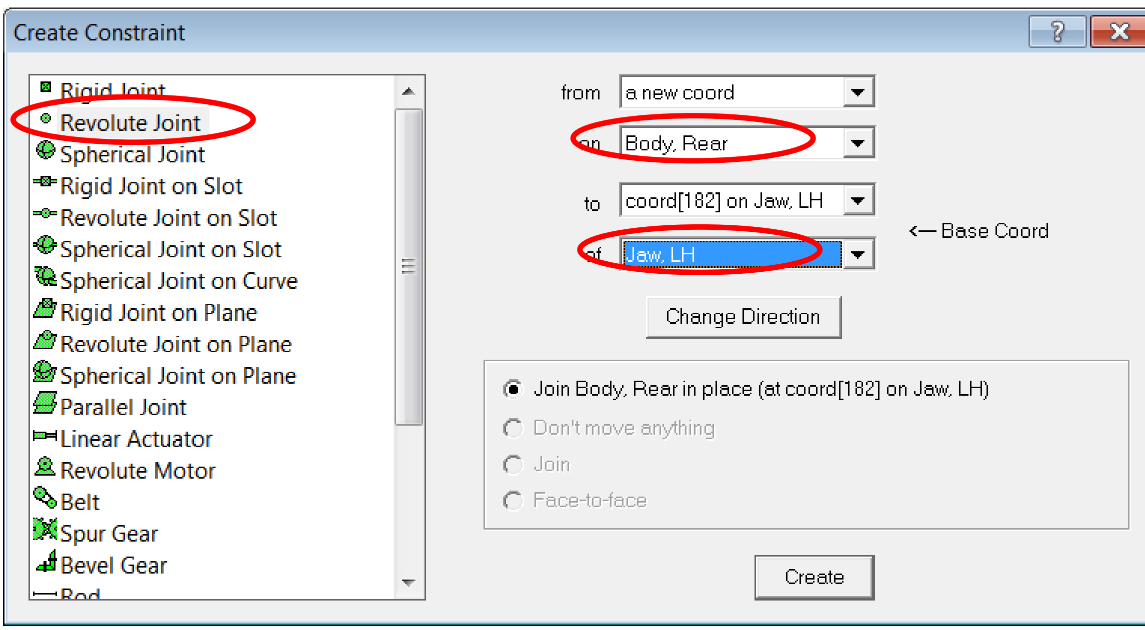


Add Constraints

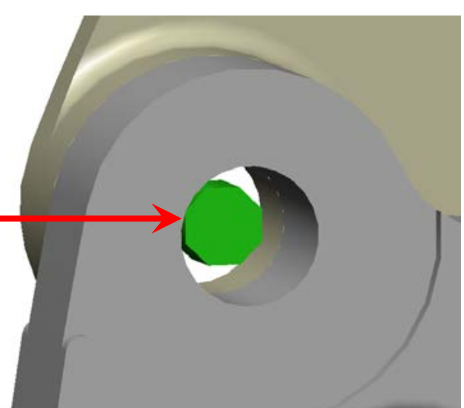
1. *In the **Create Constraint d-box**, make sure the "**on**" and "**of**" fields show **Body, Rear** and **Jaw, LH**. The order does not matter. A new coord will automatically be created coincident with the first coord and it will be attached to second part of the pair.*

2. ***Select Revolute** from the constraint list*

3. ***Select Create***



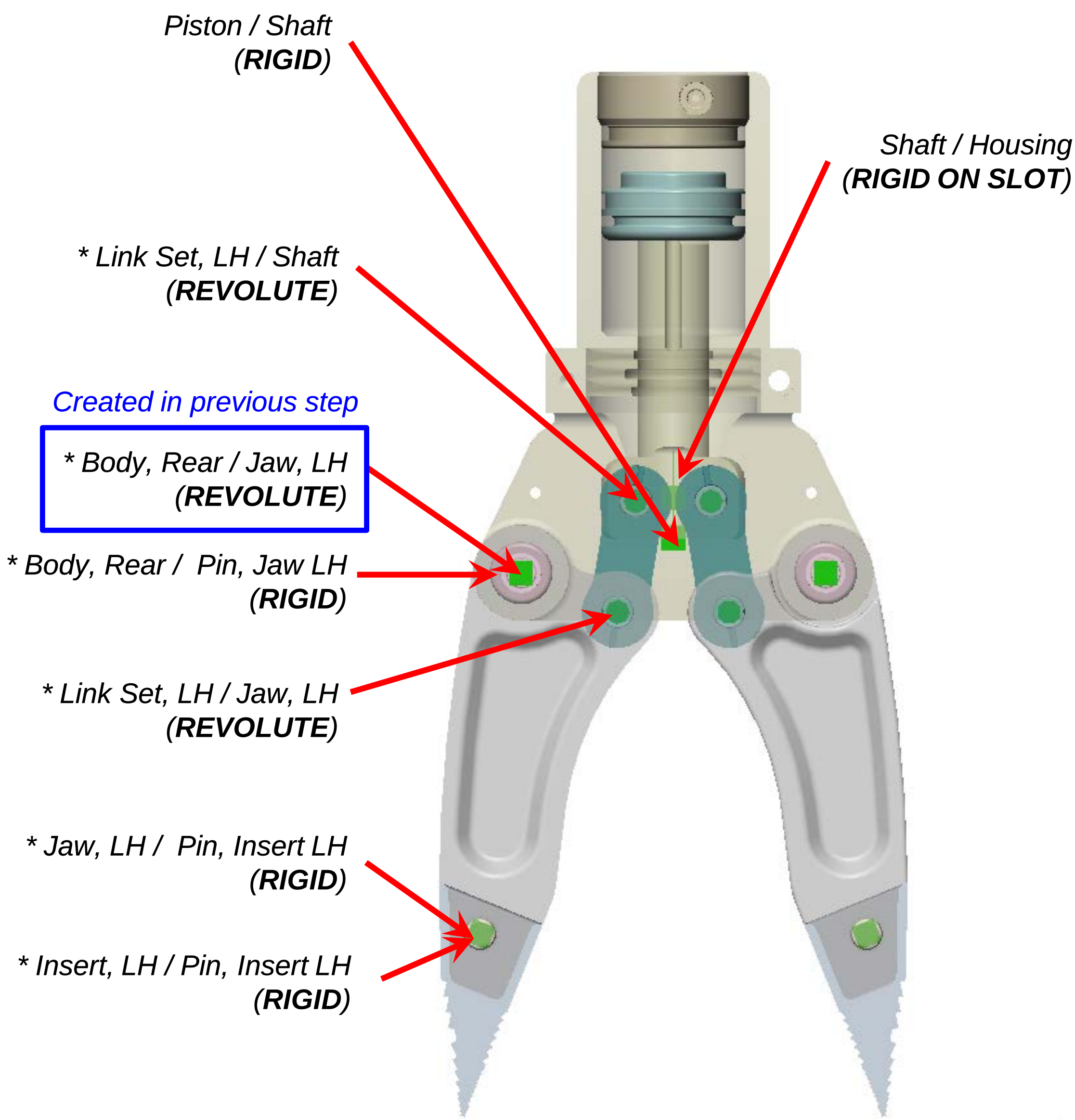
The constraint should appear as follows



Add Constraints

*The assembly is symmetrical. Therefore, only one side of the assembly will be labeled with the necessary constraints. The constraint types used on the other side are identical to the first. Constraints that have a symmetrical counterpart are marked with an *.*

When creating the constraints, choose locations that best represent the physical connection center.

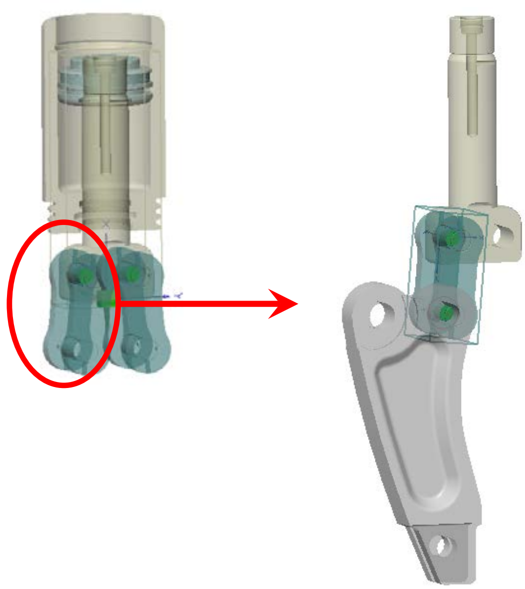


The Constraint Navigator

The Constraint Navigator allows the you to review and verify the connectivity of all the components in the model.

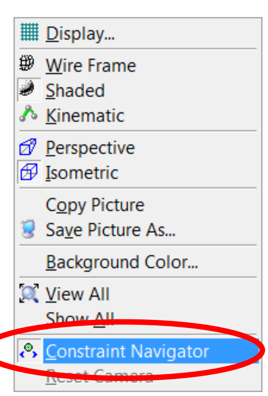
1. ***Right-click*** *anywhere in the* ***graphics area*** *and* ***choose Constraint Navigator***

2. ***Click*** *on the* ***Piston Shaft.*** *Notice that only the parts and constraints connected to this part are displayed. Then click on the link, and so on…*



Tip: *Clicking once in the background of the graphics area will display the entire model*

3. *To close the Constraint Navigator,* ***right-click*** *anywhere on the screen to bring up the settings flyout menu and* ***choose Constraint Navigator***

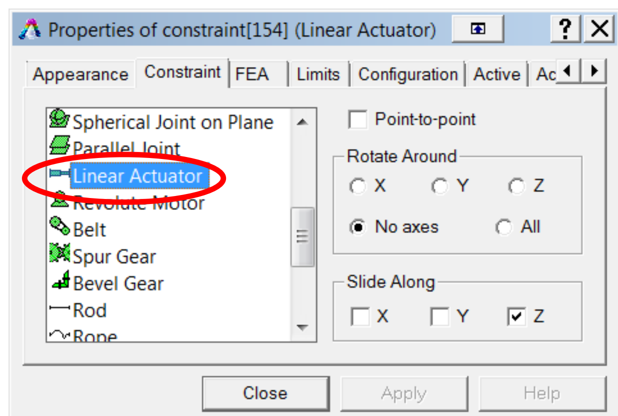


Select to close

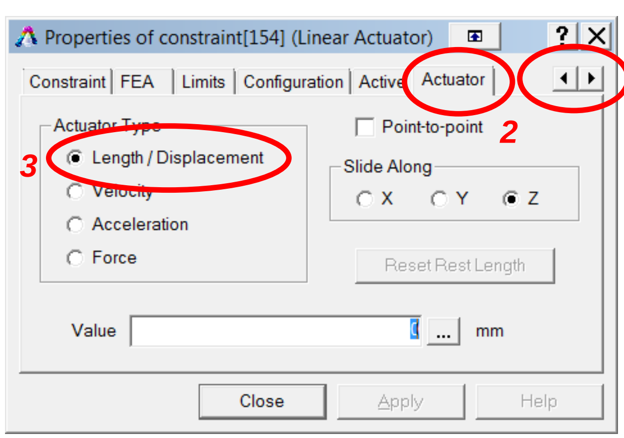
Add an Actuator

1. *In graphics window, **click on the Shaft**. In the Connections List, **right-click** the **Rigid Joint on Slot** constraint), and **choose Properties** in the flyout menu. This will activate the Properties dialog box.*

2. *In the Properties d-box, **select** the **Constraint** tab and **locate and select** the **Linear Actuator**. This will change the Rigid Joint on Slot to an Actuator constraint*

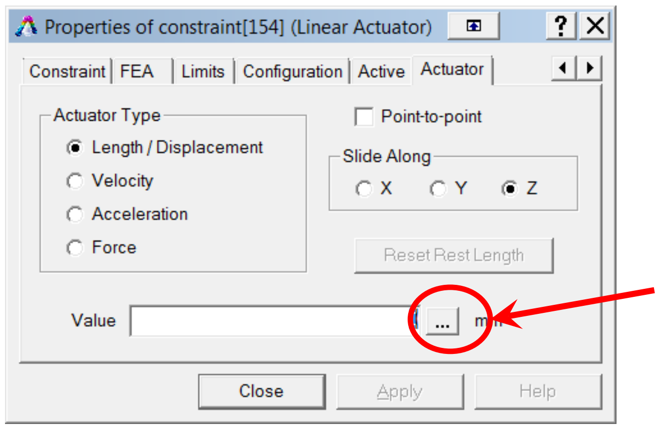


3. *Use the arrow buttons at the upper-right of the d-box to **scroll the tabs** and **select** the **Actuator tab** and **change** the actuator type to **Length/Displacement***



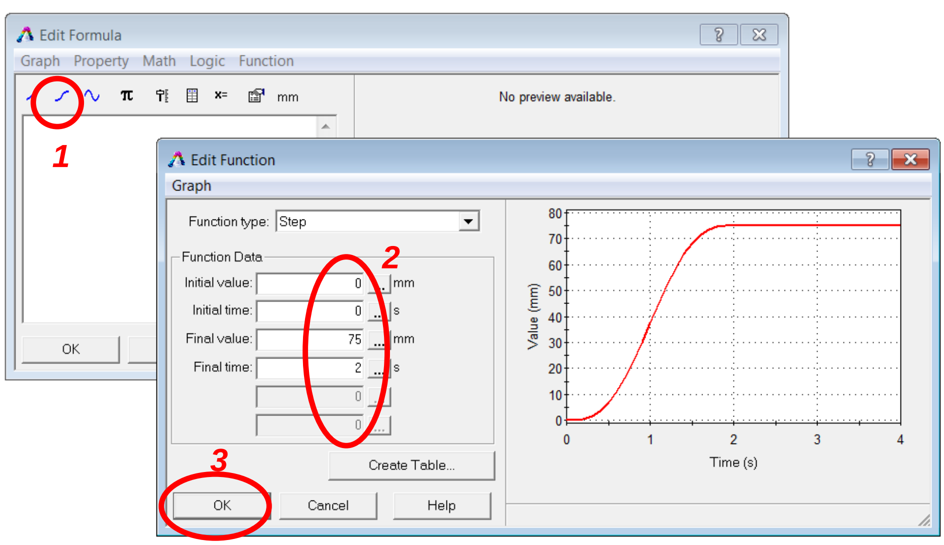
Add an Actuator (Assign Input Function)

1. ***Click** on the **Formula tab** next to the **Value field**, to bring up the **Function Builder***



Select to launch the Function Builder

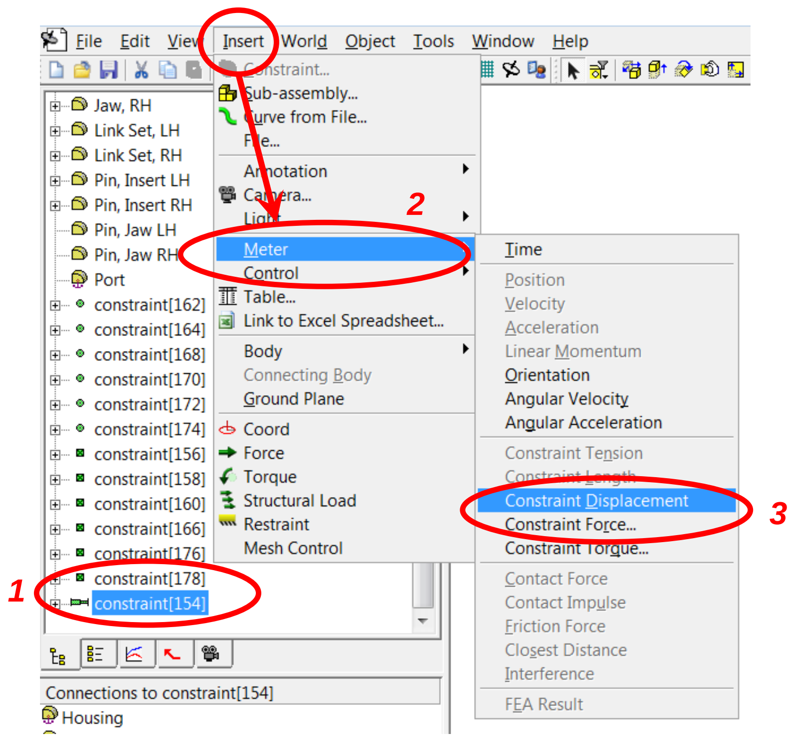
2. *In the **Function Builder**, **select the Insert Step Function** button and when the Step function builder appears, **define the step parameters**, as shown*

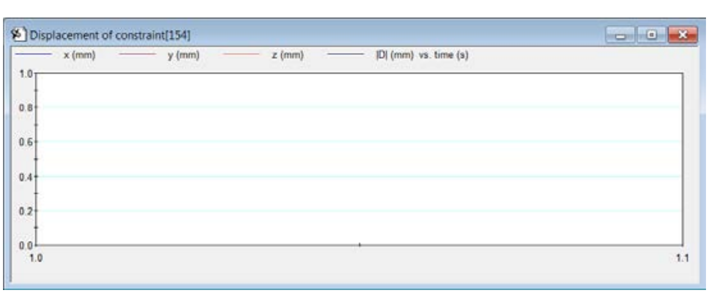


3. ***Select OK , select OK** again, and then **select Close** to exit the actuator Properties*

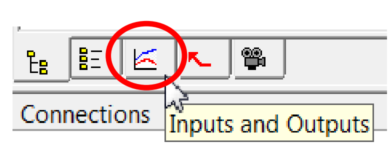
Add a Meter (Actuator displacement)

1. *In the **Object Browser**, **click on constraint[154]** (the actuator constraint), **click on Insert**, **Meter**, **Constraint Displacement**. A meter will be added below the graphics window.*





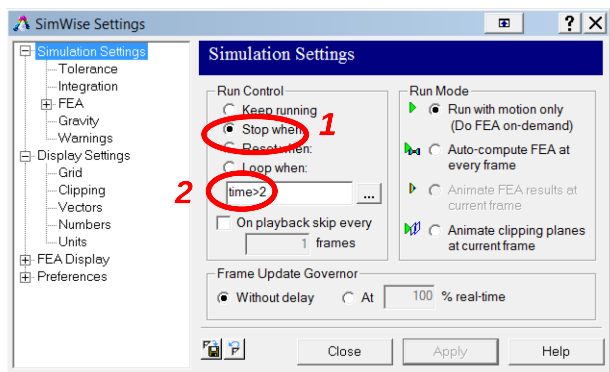
***Note:** You can click on the Inputs and Outputs tab located at the bottom of the Object Browser to see the new meter feature listed*



Define a Stop Control & Run a Simulation

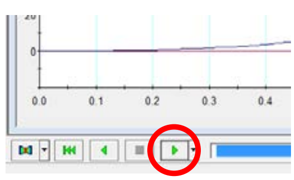
1. ***Select** the **Simulation Settings** icon* 

2. *In the **Simulation Settings** window, **define** a **Run Control** to **Stop when time>2**, as shown, then **select Apply** then **Close***



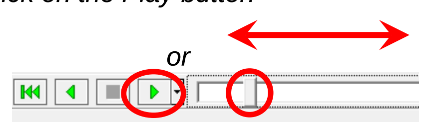
***Tip:** For ease of viewing, you can right-click the Body, Front part and choose Hide*

3. *To run the simulation, **select** the **Run button** located on the **Player Control Panel**. Once complete, use the controls on the Player panel to replay the animation*

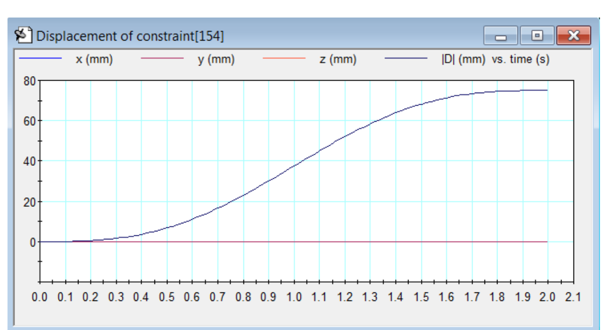


Run

To advance the simulation playback, you can either 1) click, hold & drag the slider button or 2) click on the Play button



Your meter should appear as follows

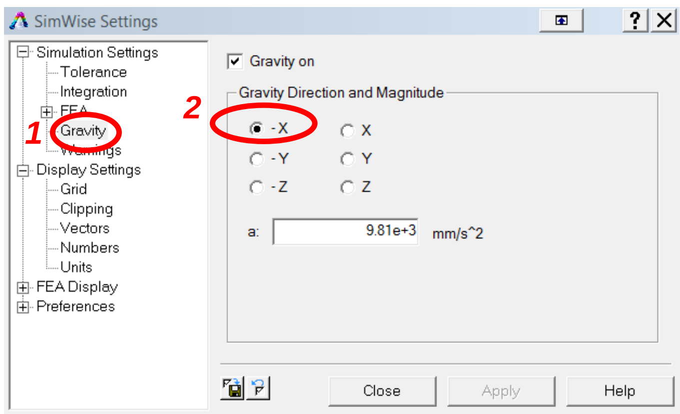


This plot should be identical to the input function applied to the actuator

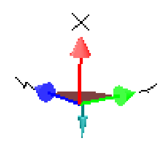
Change Gravity and Unit Settings

1. **Select** the **Simulation Settings** icon 

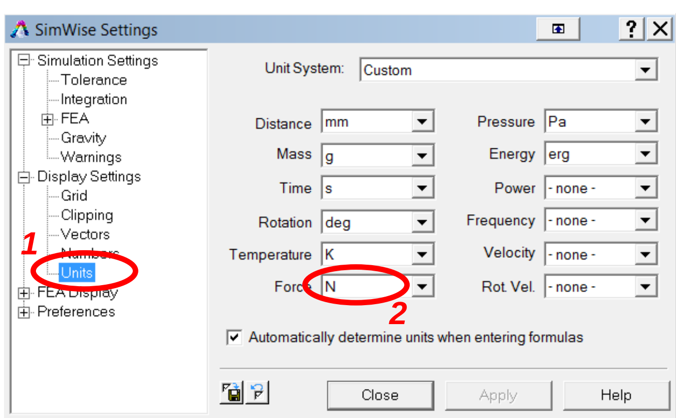
2. **Select Gravity** from the settings list and **select** the **–X** direction option. Leave the **Settings d-box** open.



Note: The cyan colored arrow on the Orientation Indicator represents the direction of gravity

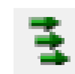


3. **Select** the **Units** option from the settings list, **select** the **drop down** menu next to **Force** and **select N** (Newtons) for the force unit. **Close** the d-box when finished.

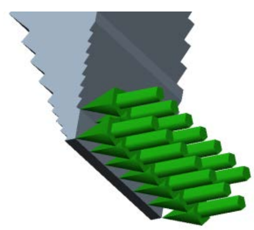


Add a Force

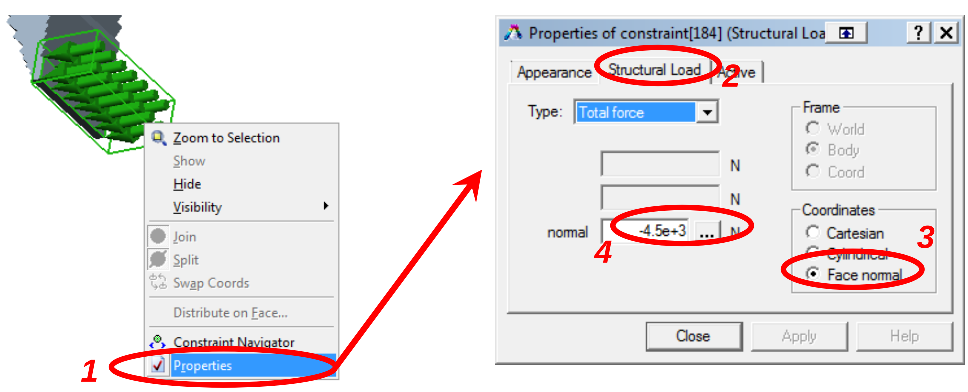
1. *In preparation for this step, **select** the **maximize** button at the top-right of the graphics window. This will enlarge the window over the meter and make viewing easier when working on the model. Then **right-select** one of the inserts and **choose Hide***

2. *On the Sketch Toolbar, **select** the **Structural Load icon*** 

3. ***Click** near the center of the **Insert** part, as shown, to place the force*



4. ***Right-select the force graphic** on the Insert and **choose Properties**. When the Properties d-box opens, change the **Type** to **Total Force**, **select the Structural Load tab**, **select Face Normal** and **define** the force value as **-4500**. **Close** the d-box.*

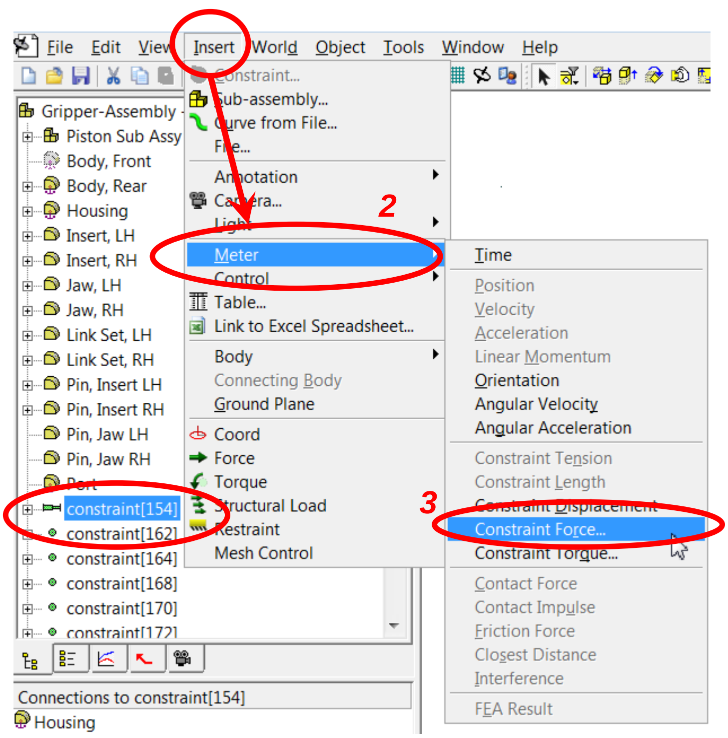


5. ***Hide** the displayed insert. **Right-click** on the other **Insert** part name in the **Object Browser** and **choose** Show.*

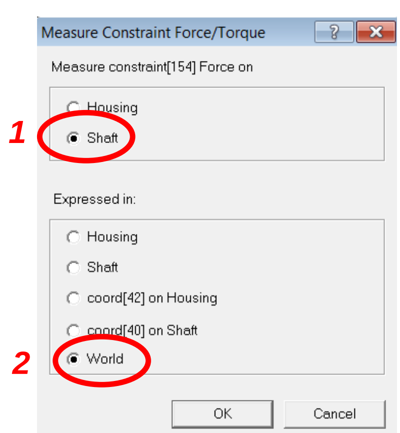
6. ***Repeat** steps **2 through 4 for the other insert** Then, **show** the other hidden Insert, as described in Step 5*

Add a Meter (Actuator input force)

1. In the **Object Browser**, ***click*** on ***constraint[154]*** (the actuator constraint), ***click*** on ***Insert***, ***Meter***, ***Constraint Force***.



2. When the Measure Force on options d-box appears, ***select*** ***Shaft*** and then ***World***. ***Select*** ***Ok***.



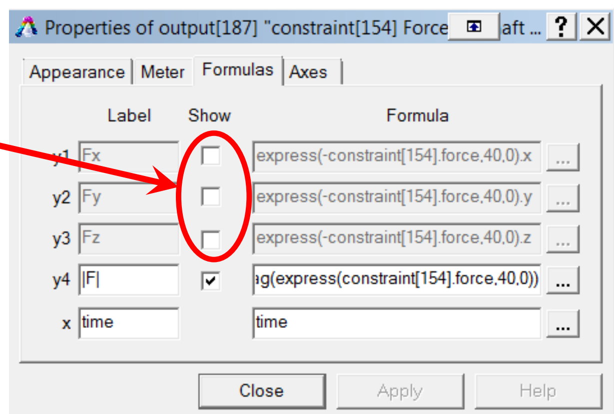
Run the Simulation & Modify the Meter Display

1. Run the Simulation

*2. When the simulation is complete, **double-click either meter** to activate the **Properties** d-box.*

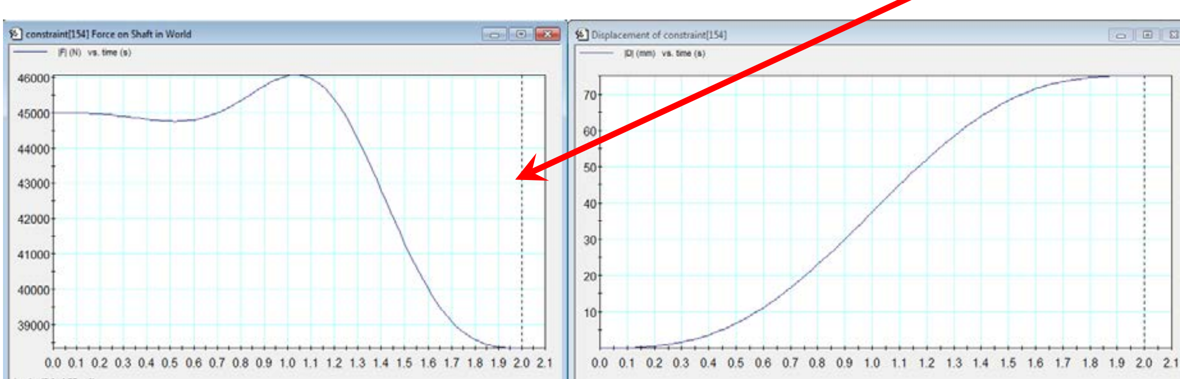
***3. Select** the **Formulas tab** and **uncheck** the **Show buttons** for **y1**, **y2** and **y3**. **Select Close**.*

Uncheck



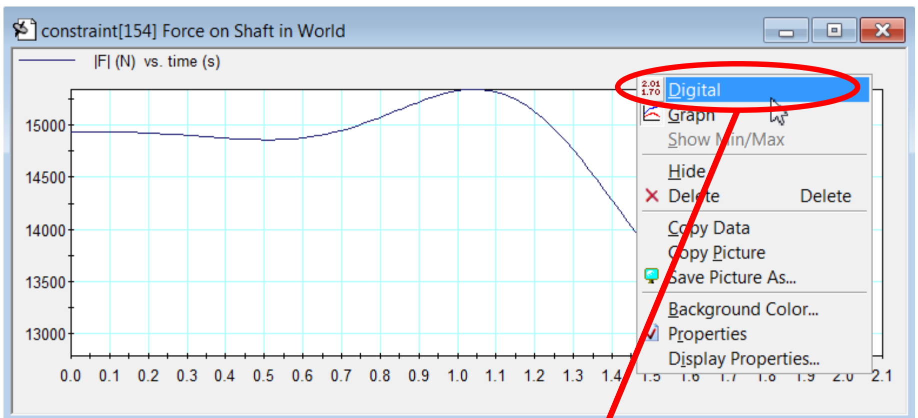
***4. Repeat steps 2 and 3** for the other meter*

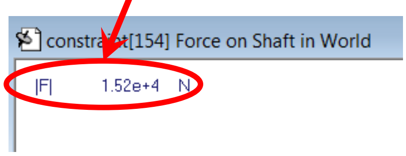
Vertical tracer indicates the current time position of the Animation Playback slider



Your meters should now appear as follows:

1. ***Right-select*** *the* ***force meter*** *and* ***choose Digital***

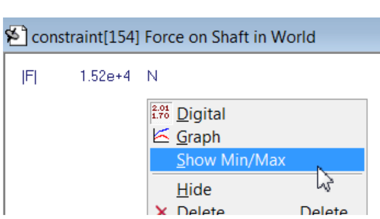


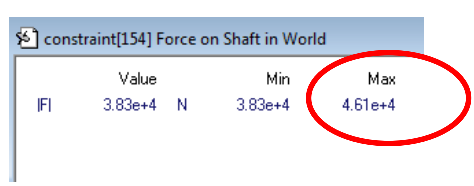


Your meter will display in digital format.

Depending on where the vertical tracer was positioned on your meter before transitioning to the digital readout, the value displayed may be different from what is shown here

2. ***Right-select*** *the force meter again and* ***choose Show Min/Max***





Actuator Force Requirements and Mechanical Advantage

Your digital meter should read an approximate maximum force of 4.61e+4N (46,100 N). This is the amount of actuator input force necessary to achieve a total clamping force of 9,000N (4,500N x 2). Note also there is a minimum input force requirement of approximately 3.83e+4N (38,300 N).

The maximum mechanical advantage of the mechanism is therefore:

Output / Input

or

*9,000 N/ 46,100N = **.19***

It is important to note that because the value is less than 1.0, there is actually a mechanical disadvantage. There is more force required by the actuator than there is being produced at the output of the jaws.

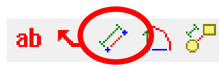
We also recognize that the relationship between the input and output is not entirely linear. It varies depending on the position of the piston.

Add a Dimension

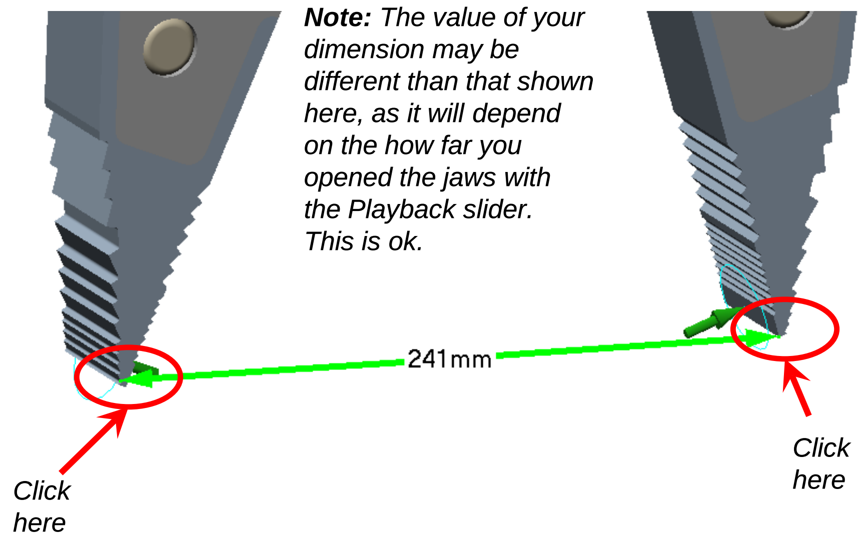
1. *In preparation for this step, **select** the **maximize** button at the top-right of the graphics window to enlarge the window over the meters.*

2. ***Drag** the **simulation playback slider** enough so the jaws are opened*

3. ***Select** the **Distance Dimension** icon on the **Annotation toolbar***

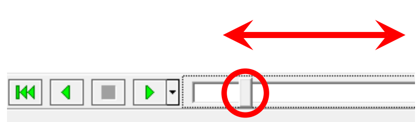


4. ***Click** near the tip of one of the Inserts then **click** on the tip of the other. A dimension will be added.*



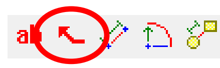
***Note:** The value of your dimension may be different than that shown here, as it will depend on the how far you opened the jaws with the Playback slider. This is ok.*

5. ***Drag** the **Playback slider** back and forth and note the changing dimension value*

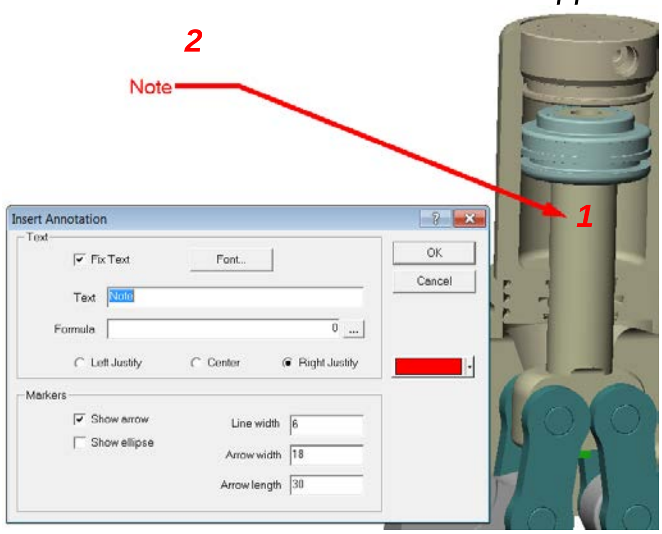


Add A Custom Annotation

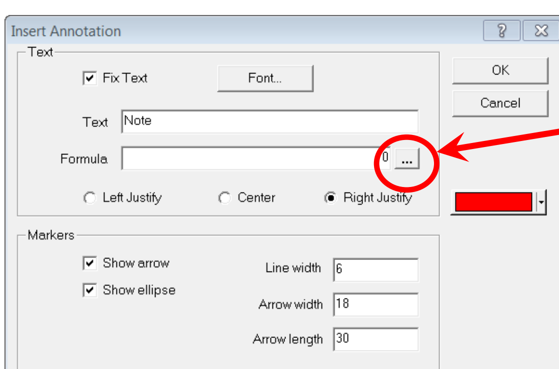
1. ***Select** the **Annotation icon** on the **Annotation toolbar***



2. ***Click** anywhere on the **Piston Sh**aft, **release** the mouse button then **drag** the mouse anywhere to the left and **click again**. An annotation callout will be created and the **Insert Annotation** d-box will appear.*



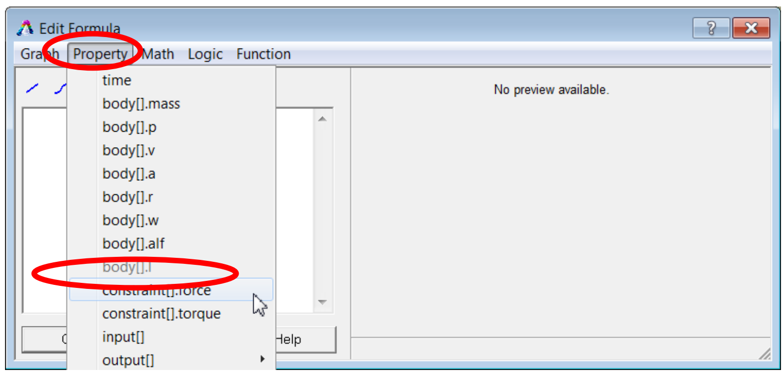
3. ***Click** on the **Formula tab** next to the **Formula field**, to bring up the **Function Builder***



Select to launch the Function Builder

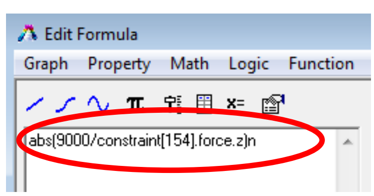
Add A Custom Annotation (continued…)

1. ***Select** the **Property menu** and **choose constraint[ ].force** from the drop down. The syntax constraint[ ].force.x will automatically be placed into the formula field.*



2. ***Modify** the formula to read: **abs(9000/constraint[154].force.z)N** . **Select OK.***

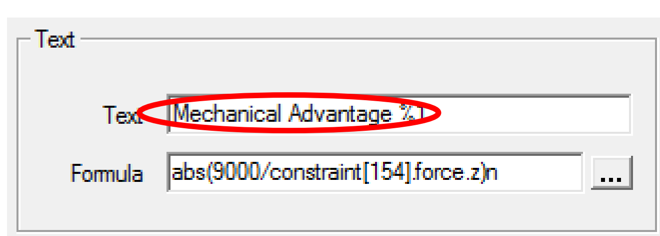
***Note:** the value 9000 represents the total force (4500N x 2) we applied to the Inserts*



***Note:** The default is "force.x". The actuator's coord (orientation basis) has its z axis aligned with the global x direction. We are interested in using the actuator's force along its own axis. Therefore, we need to use "force.z" instead.*

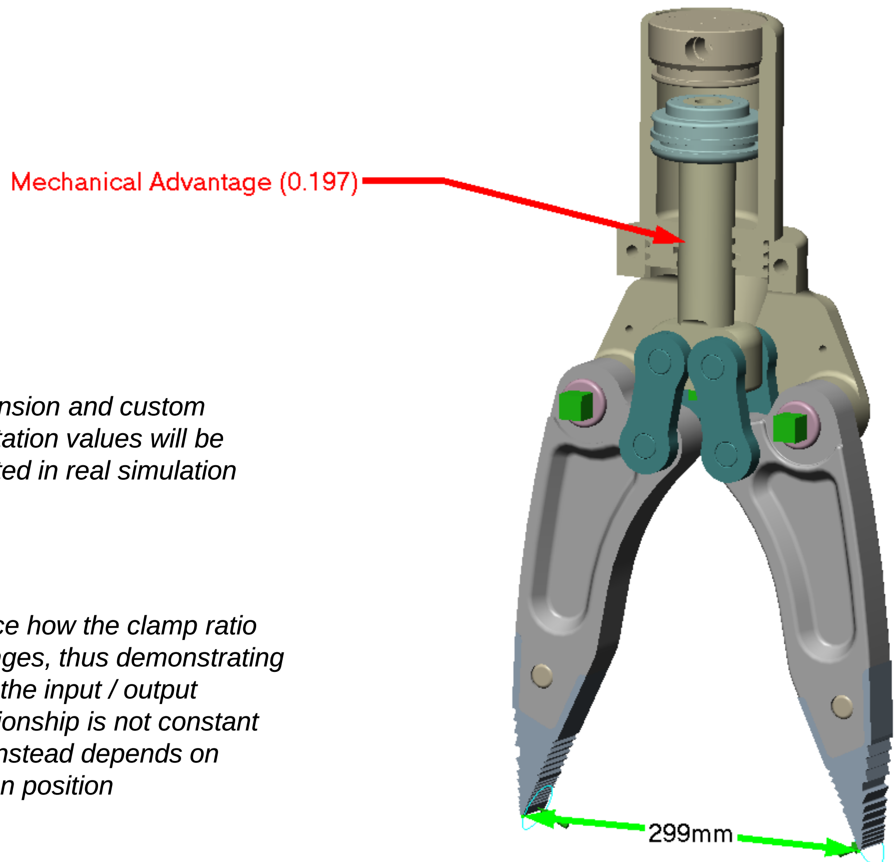
3. *Back in the Property d-box for the annotation, **enter** the following text into the **Text field**: **Mechanical Advantage (%1)** . **Select OK.** The custom callout should now appear in the graphics window.*

***Note:** %1 is the variable syntax for the formula we entered and will allow the value of the formula to be displayed with the text note*



Run the Simulation

1. Run the Simulation



Dimension and custom annotation values will be updated in real simulation time.

Notice how the clamp ratio changes, thus demonstrating how the input / output relationship is not constant but instead depends on piston position

Click on image to see animation

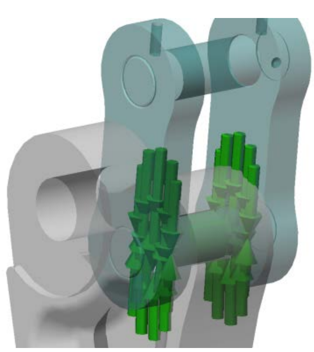
A brief discussion on using motion loads in an FE analysis:

SimWise knows when a constraint is defined between two bodies. However, it does not necessarily know which faces are held together by the constraint in the physical World. Its treatment of bodies in the motion simulation is analogous to using a free body diagram to solve problems in static and dynamics, where only a schematic representation of the model is used for analysis. This approach is primarily concerned with the location of User-applied loads, constraint location, and the center of mass location - a skeleton representation of the model if you will. It does not recognize actual geometric features.

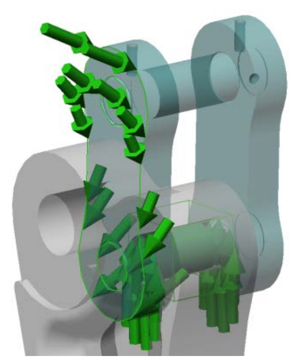
Once a body is set to be included in a combined Motion+FEA study, SimWise bridges the free body diagram approach of the motion simulation with a structural analysis, which is fully dependent on actual geometric part features. In this case, the User will see small (green) graphic arrows appear on the body, near the location of the constraint or possibly even somewhere else unexpected on the body, initially. These graphics represent the resulting motion constraint forces that will be applied to the body for the FE analysis.

In some instances, the placement of the force graphics on the part is determined automatically (initially). As a result, a force from a constraint may inadvertently get applied to a face that is not associated with its true physical connection. Therefore, prior to running an FEA using motion loads, the User must be sure the constraint loads are assigned to the proper faces. For example, if there is a Revolute constraint used to represent a pinned connection between two parts, the force graphics must be applied to the face of the hole and the face of the pin for accurate FEA load distribution (see images below).

In the next section, we will show how to assign the motion forces to the proper faces of a part for accurate load distribution in FEA.



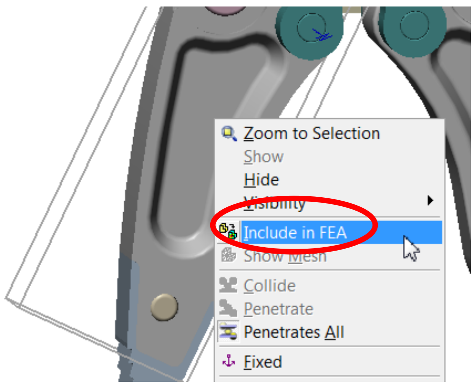
CORRECT:
Force graphics applied to both the hole face and pin face



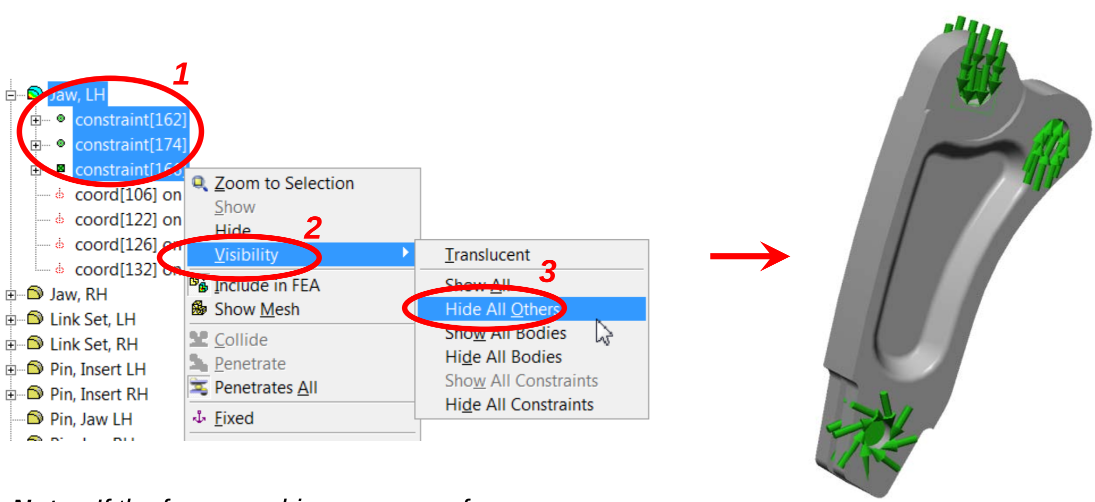
INCORRECT:
One force graphics set applied to hole face and another to side face of link

Prepare a Stress Analysis

1. ***Right-select*** *the* ***Jaw, LH*** *and choose* ***Include in FEA.***



2. *In the* ***Object Browser****,* ***click*** *the* ***+ sign*** *next to* ***Jaw, LH****. Then* ***hold down the Ctrl key****,* ***select*** *the* ***Jaw,LH*** *part name and the names of the three constraints underneath it.* ***Right-click*** *and* ***choose Visibility****,* ***Hide All Others***



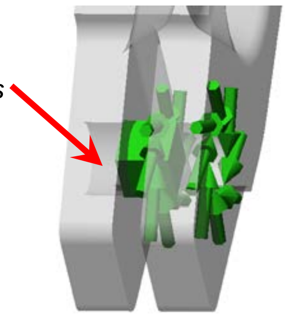
Note: *If the force graphics appear on faces differently from that shown in the image, simply proceeded with the upcoming steps but ALSO view the supplemental vide on this topic*

Only the Jaw part and the three constraints (with force graphics) should be showing

Prepare a Stress Analysis (assign load faces)

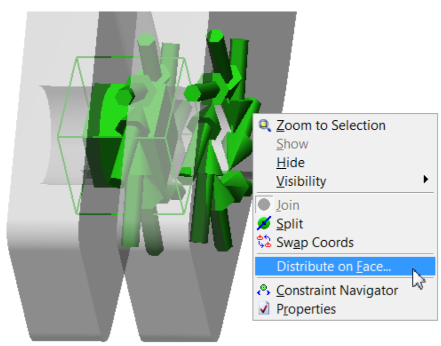
One of the constraints will not have its load-bearing faces assigned correctly. Rather than being applied to the hole face in each ear, only one face has its loads assigned. If your force icons appear on different faces that that shown below, simply review the next two steps but ALSO view [this video](#)

This hole must also be assigned constraint loads

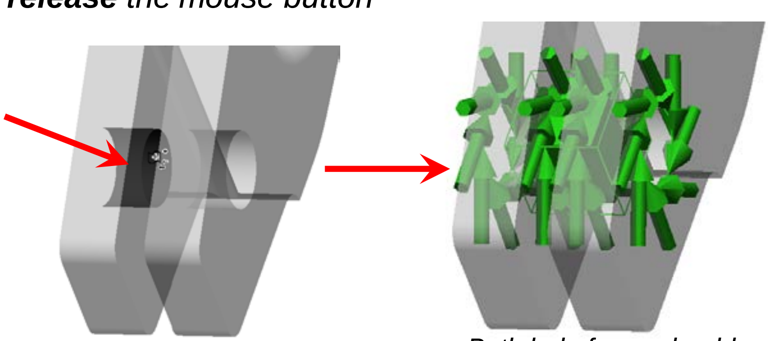


Tip: Change the translucency on the part for easier viewing: Right-select part, choose Visibility, then choose Translucent

1. ***Right-click** on the constraint **force graphics** for the Rigid constraint that connects the Jaw to the Insert (constraint [166]) and **choose Distribute on face***



2. ***Hold down** the **Ctrl key**, **select** the inside face of the hole on the other ear and **release** the mouse button*



***Tip:** Holding down the Ctrl key will allow for the selection of multiple faces for the load*

Both hole faces should now contain force graphics

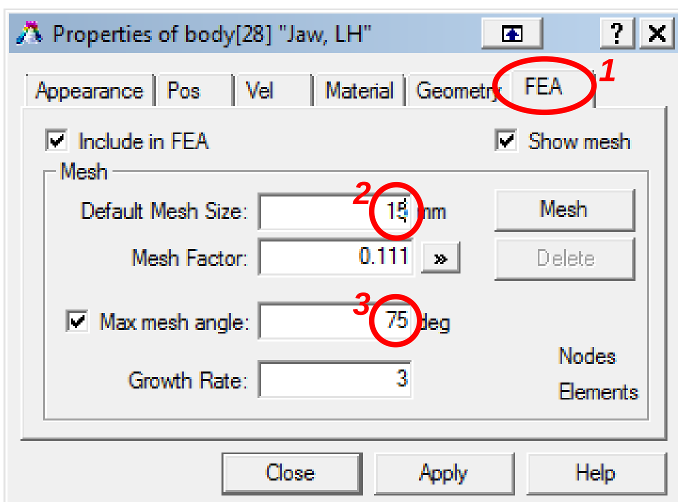
Prepare a Stress Analysis *(create mesh)*

1. ***Double-click*** *the* ***Jaw, LH*** *to open the body* ***Properties*** *then* ***select*** *the* ***FEA tab***

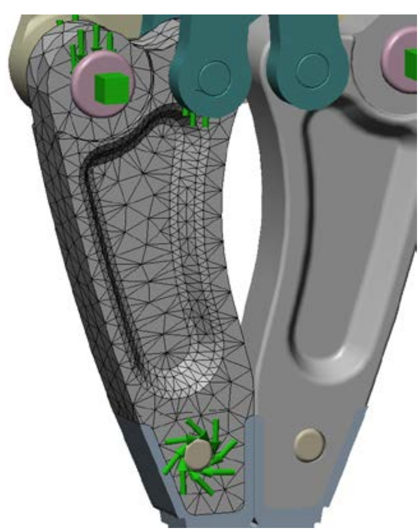
2. ***Change*** ***the Default Mesh Size*** *to* ***15mm****,*

3. ***Select*** ***Max mesh angle*** *and* ***enter*** ***75***

4. ***Select*** ***Show mesh*** *and then* ***select*** *the* ***Mesh*** ***button***



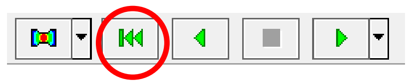
The model shoul now appear as follows



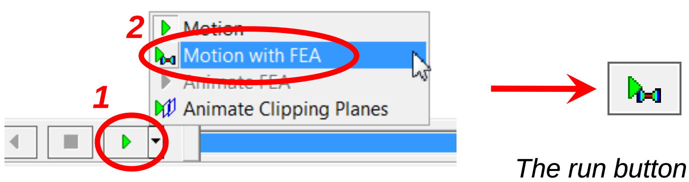
Tip: *To re-display only the bodies and force graphics, open the Object List, click, hold and drag the mouse across all part names (multi-select) and choose Show.*

Run Motion+FEA Simulation

1. ***Rewind** the simulation to the beginning*



2. ***Select** the small arrow next to the **Run** button on the **Player Control Panel** and **choose** the **Motion with FEA** option*



The run button will now appear as follows

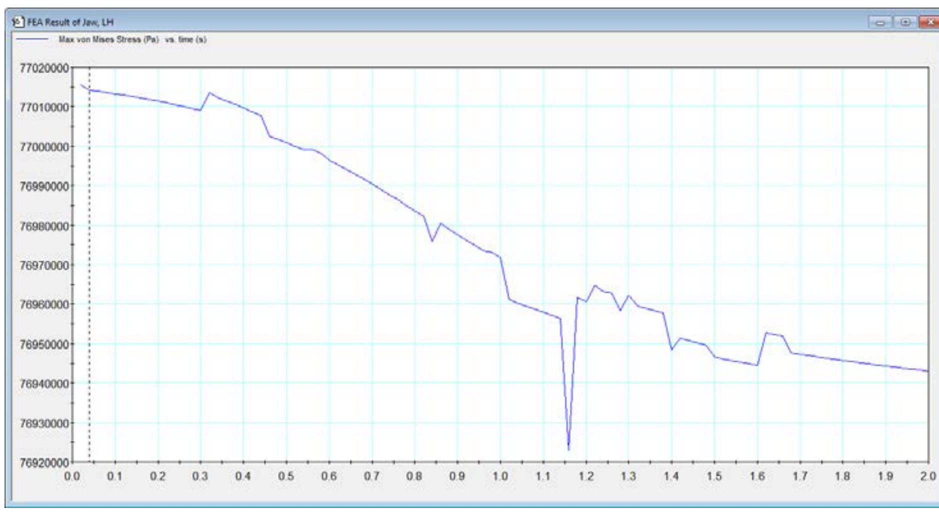
3. ***Select** the **Run** button to begin the combined Motion+FEA simulation*

With each simulation step, SimWise calculates all the motion loads acting on the Jaw and then calculates the stress and deflection resulting from the loads. Once the simulation is complete, we will then generate a meter that will display Stress vs. Time Step and allow us to determine which model position results in the maximum stress in the Jaw. Once that point is identified, we can then perform an h-adaptive analysis which will help us refine the mesh and obtain a more accurate solution for our stress and deflection calculations.

Refine Mesh with H-adaptivity

1. ***Select** the **Jaw** and **choose** **Insert**, **Meter**, **FEA Result**. A meter will display the Von Mises stress vs. simulation time*

***Note:** The maximum stress occurs near the beginning of the simulation when the jaws are in their closed position. However, there will not actually be any loads on the jaws until the spread distance between the inserts is approximately 105m (approximately .58 sec)*



***Note:** The noise (spikes) in the data is the result of a mesh that is inadequate for the geometry at-hand. Regenerating this curve after H-adaptive mesh refinement will produce smoother, more accurate results*

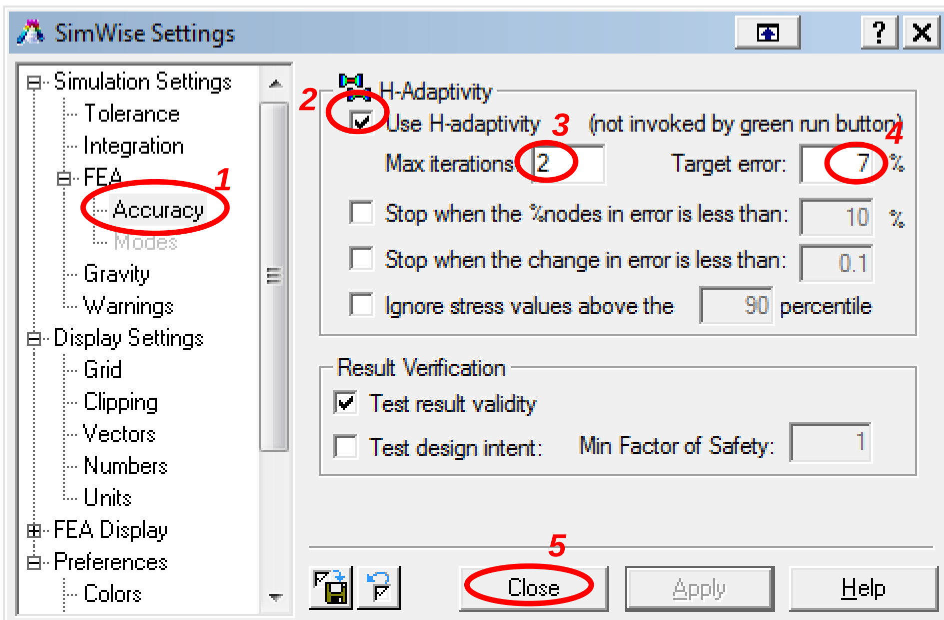
2. ***Drag** the **Playback slider** until the distance dimension between the inserts reads approximately **105mm**.*

3. ***Select** the **Simulation Settings icon***  *to open the **Simulation Settings d-box**,.*

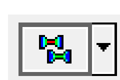
***Tip:** You can also right-click in the background of the graphics window and choose Display to access the Simulation Settings*

Refine Mesh with H-adaptivity (cont…)

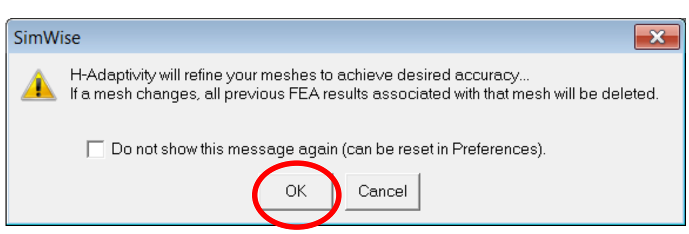
1. ***Click*** *on the "**+**" sign next to **FEA** and select **Accuracy.** Click on **Use H-adaptivity, change Max iterations to 2 and set** the default **Target Error to 7%**. **Select Close.***



2. ***Click*** *on the **Solve H-adaptive FEA** button*

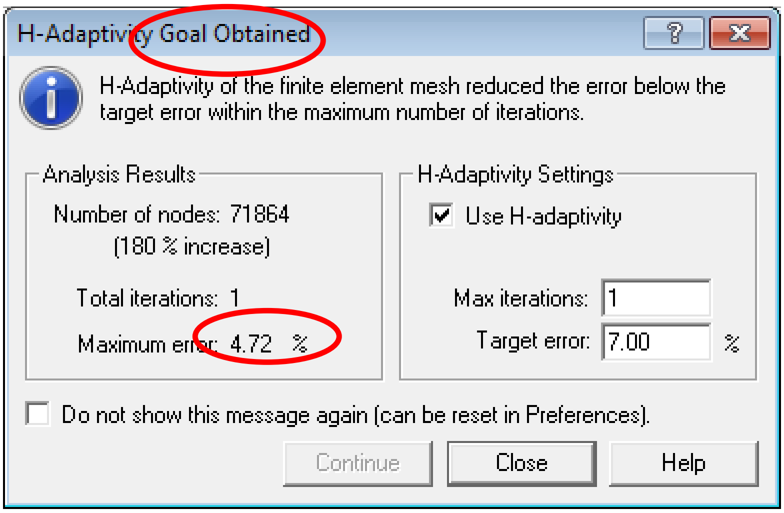


3. *The following message will appear. **Select OK***

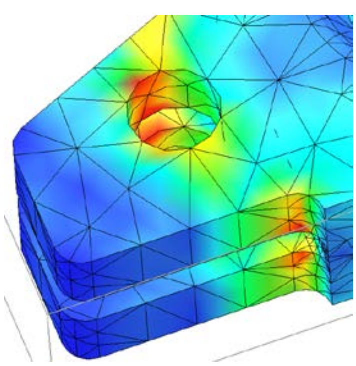


Maximum Stress Results

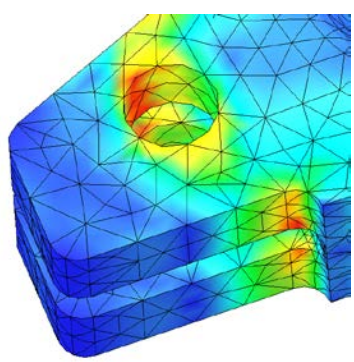
After two iterations, the H-adaptivity Goal window should appear showing that convergence was achieved. In addition the convergence criteria was exceed.



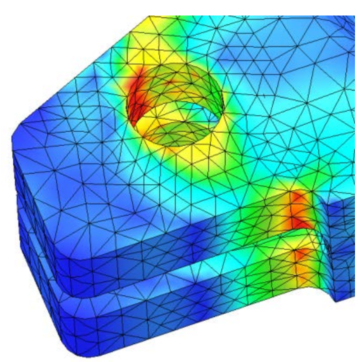
Below are the results of running the analysis one h-adaptive iteration at a time. The initial mesh (far left) is gradually improved in the necessary areas until the stress error in the elements in below the specified value (final mesh at far right). The initial mesh and solve resulted in a FOS of 4.6 and the final resulted in a FOS of 4.4 Note that additional refinement beyond this point will most likely result in negligible differences in the overall maximum stress. These negligible differences indicate convergence.



Stress
7.2e7 Pa



Stress
7.35e7 Pa

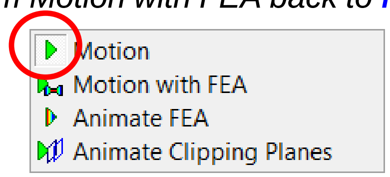


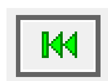
Stress
7.5e7 Pa

Create a Keyframed Animation

1. ***Zoom out*** *from the Model such that the model takes up approximately ¼ of the screen.*

2. ***Change*** *the simulation run type from Motion with FEA back to* ***Motion***

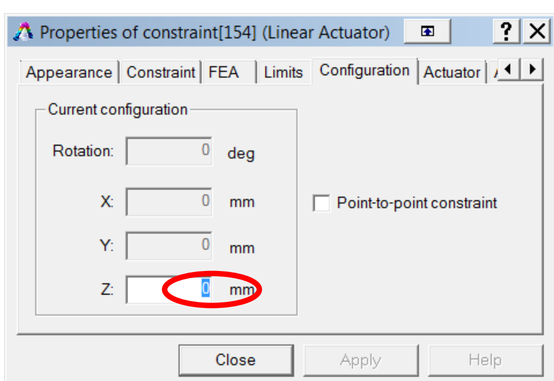


3. ***Reset*** *the simulation* 

4. ***Click*** *on* ***World***, ***Erase FEA history***

5. *In the* ***Object List***, ***double-click*** *the* ***Actuator constraint*** *to open the* ***Actuator Properties***.

6. ***Click*** *on the* ***Configuration tab*** *and enter* ***0mm*** *for the* ***z direction*** *then* ***select Close***

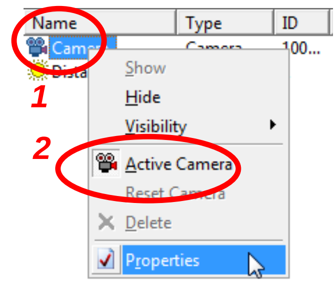


Note: *The reason for this step is because after we solved for the H-adaptive meshing, this open jaw position became the new starting configuration for the model. Running the simulation again would invalidate the input function used for the Actuator (i.e. it would no longer start at 0mm)*

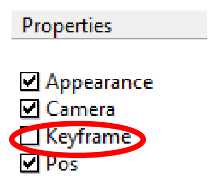
7. ***Select*** *the* ***Camera tab***  *located on the bottom of the Object Browser, to view the Cameras List*

Create a Keyframed Animation (cont…)

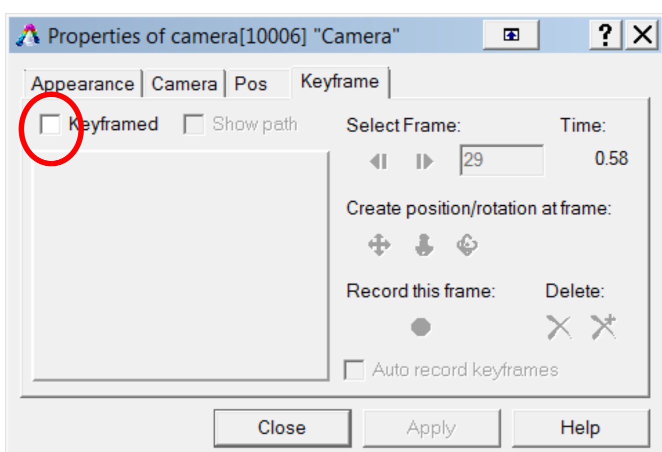
1. *In the Camera List, **right-select** the **default camera** and **choose** **Properties***



2. *In the **Properties List**, **check** the box next to **Keyframe**. This will activate the Keyframed tab in the camera Properties dialog box*



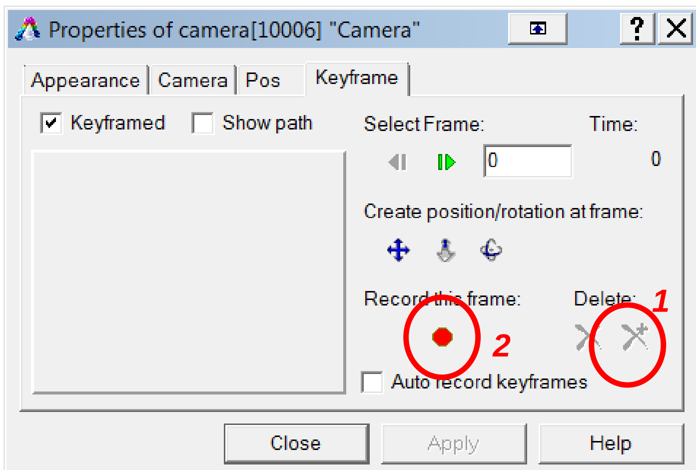
3. *In the Properties dialog for the camera, **check** the box next to **Keyframed***



***Note:** If the model is not already in Perspective view mode a message will appear indicating you need to switch to perspective view. If this message does appear, prompting you to switch view mode, select Yes.*

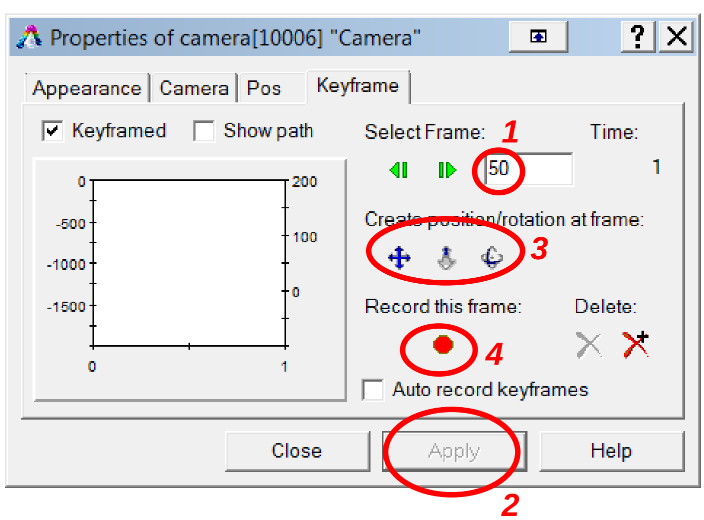
Create a Keyframed Animation (cont…)

1. *The Record button will turn red. If it doesn't, **select** the **Delete X+** button first. Then, **rewind the simulation** to the beginning and **select** the **Record** button*



2. *In the **Frame value field**, **enter 50** and **select Apply**, use the **Pan**, **Zoom** and **Rotate** buttons to position and orient the model to any new view representation and then **select** the **Record** button again*

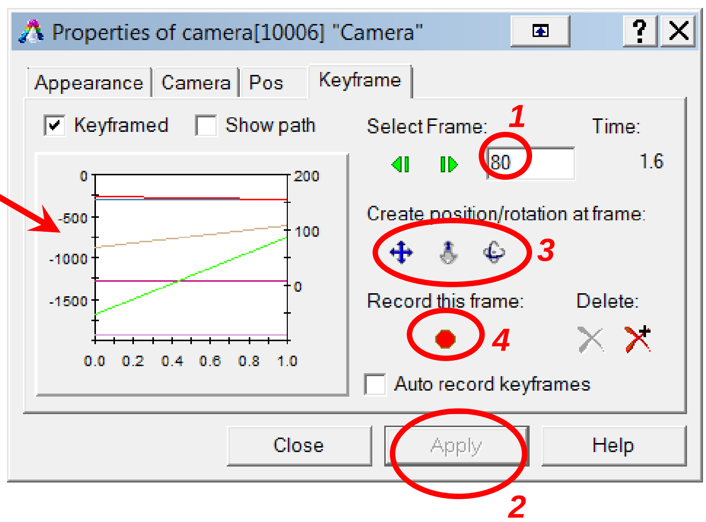
Note: *With keyframing, you are specifying how long the model will take, between keyframed points, to transition between the different view orientations /positions you specify. The greater the difference between keyframes, the longer it will take for the model to transition to the new view representation*



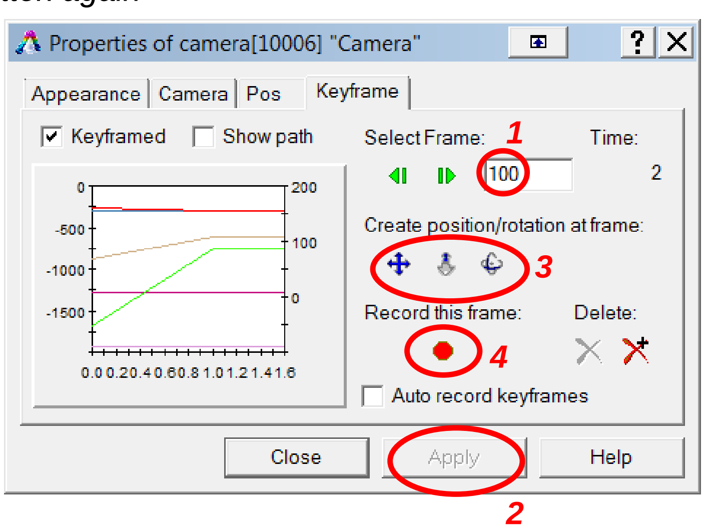
Create a Keyframed Animation *(cont…)*

1. *Repeat the previous step – In the* ***Frame value field****,* ***enter 80 and select Apply****, use the* ***Pan****,* ***Zoom*** *and* ***Rotate*** *buttons to position and orient the model to any new view representation and then* ***select*** *the* ***Record*** *button again*

Note: *Your keyframed data chart may be different from that shown here. It will depend on how you choose to manipulate your views for the keyframing.*



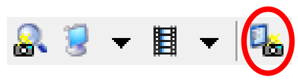
2. ***Repeat*** *the previous step - In the* ***Frame value field****,* ***enter 100 and select Apply****, use the* ***Pan****,* ***Zoom*** *and* ***Rotate*** *buttons to position and orient the model to any new view representation and then* ***select*** *the* ***Record*** *button again*



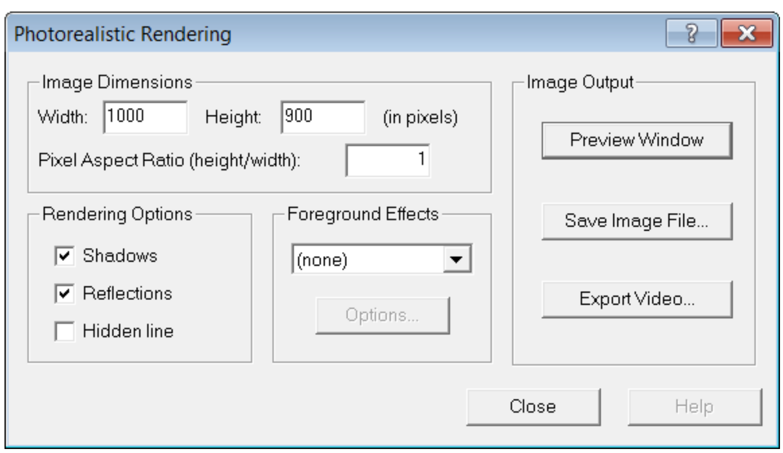
Export Video Animation

Use the Playback slider to review the saved keyframed animation

1. ***Click*** *on the* ***Render Settings*** *button located on the* ***Render toolbar***



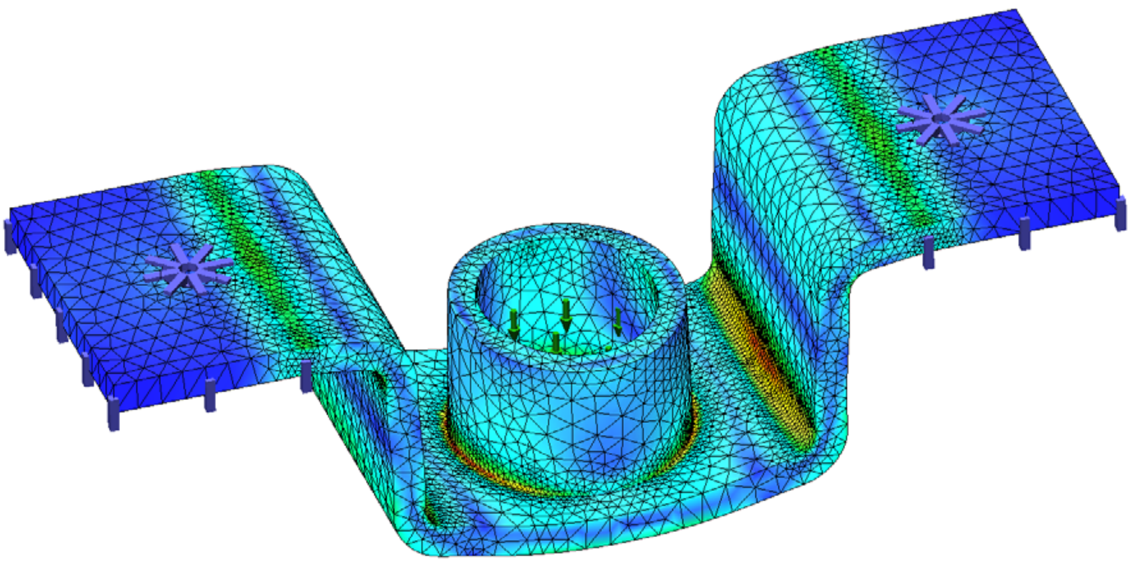
2. *In the Rendering d-box,* ***click*** *on the* ***Preview Window*** *button to preview the size of the animation area that will be saved to video. The default (256 x 256) will most likely be too small. Use a size that best fits your model view area. Simply close the preview window, type in a new Width and Height and click on Preview again. Do this until you have an acceptable view size and then select* ***Export Video****.*



<u>End of Exercise</u>

FEA Modeling Basics

A background guide to performing a stress analysis with SimWise



Introduction

Design Simulation Technologies, Inc. will not be held liable for any loss or damages incurred in connection with the use and/or implementation of the material contained in this document.

SimWise solves problems in linear Finite Element Analysis (FEA), in which the following assumptions are made:

- *The relationship to stress over strain is linear and elastic*
- *The deformations are small enough such that they may be approximated as linear*
- *The magnitude, orientation and distribution of the loads does not change during deformation*
- *Any strain-hardening of the material due to deformation can be neglected*

The information presented in this document serves only as guidelines to choosing the best FEA modeling approach using SimWise. The following sections explain things such as the basic operating principals behind SimWise, how to build and refine a mesh, and also where and when the various FEA features found in SimWise become applicable.

It is assumed the user already has a basic understanding of mechanics of materials, principals of stress, strain and the various failure modes. If necessary, there is a wide array of material available on the market that goes into more depth on these subjects.

With any FEA program, accurate results require the geometry, material properties, boundary conditions and mesh be as close as possible to the physical conditions. The mesh process can arguably be considered as the most important of the four criteria and therefore is where the majority of the FEA setup work is typically required.

Geometry & Restraints

Geometry:

Unlike kinematic motion simulation where small geometric features such as fillets , rounds and sharp edges do not affect the results of the simulation, the opposite is true when it comes to structural analysis. Certain geometric features can introduce large and undesirable stress concentrations and others can alleviate them. Doing a first-pass analysis on a design without all final design features is acceptable. However, it may become necessary to include such features in subsequent runs if the first pass run shows high stresses in areas where the geometry was left out.

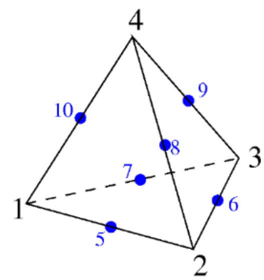
Assemblies can distribute loads entirely different from how one might manually apply loads to a single part. Load values and load directions are often overlooked. Bending moments are often introduced but overlooked in single part analysis. Modeling a problem as a single part or an assembly could have a great impact on the ability to achieve good results and one must be sure of the assumptions used to justify either approach.

Restraints:

Restraints can be applied to vertices, edges and faces. Care should be taken when applying restraints to the model in that the restraint type and location should be representative of the physical setup. The restraints should define the proper degrees of freedom of the physical connection they represent. If you are modeling a single part that is actually part of an assembly, be sure the interaction with the missing part is properly represented with loads or restraints. Over restraining a part or assembly can actually prohibit deformation and act to increase stress. *See the section titled Splitting Faces for Restraint Application for information on designating specific areas to apply restraints.*

Mesh Elements

SimWise uses 10-noded linear tetrahedral elements in the mesh process. There are 4 vertex nodes and 6 mid side nodes, as shown below.



When SimWise solves the FEA, it uses the material properties (Elastic Modulus, Poisson's Ratio) along with the model forces to determine the nodal displacements and in turn, the strains and stresses, at each node. Using the stresses for all nodes of an element, the overall average stress in that element is calculated.

The quality of an element is defined by the aspect ratio. The aspect ratio is defined as the ratio between the longest edge and the shortest normal dropped from a vertex to the opposite face. An ideal mesh element will have an aspect ratio of 1. The larger the aspect ratio, the more distorted an element is. The more distorted an element is, the greater the stress differences between nodes and therefore the greater the overall stress error in the element. The following image shows an ideal element compared to a less-desired , distorted element.

Aspect Ratio = 1

<u>Ideal</u>

Aspect Ratio = >1

<u>Distorted</u>

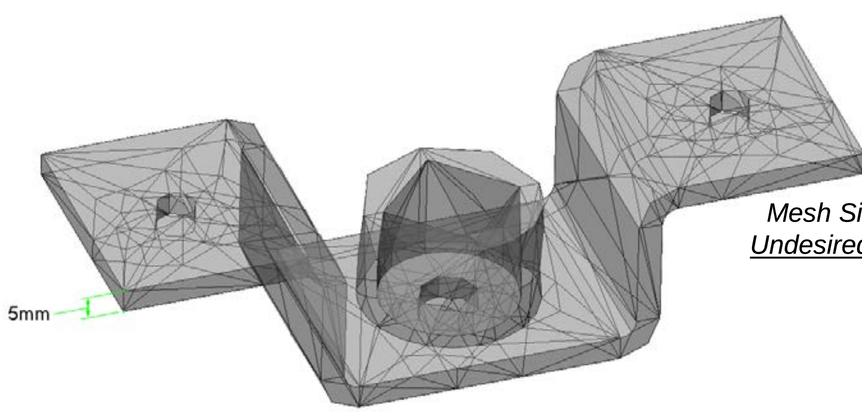
SimWise has various tools for checking , improving and controlling the mesh quality.

Mesh Element Quality

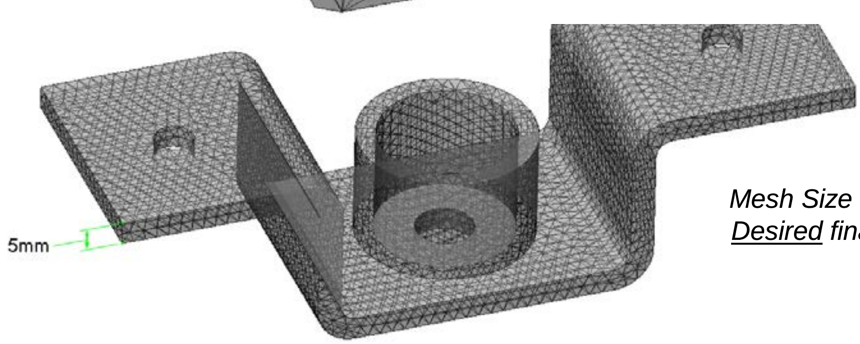
Mesh element quality is affected by different things such as the size of the geometry, the arrangement of the geometry and the transitioning between features. SimWise will do its best to create a mesh. However, sometimes the resulting mesh may contain distorted elements that do not conform well to the geometry being meshed, thus posing the potential for large stress errors that affect overall stress results. This is where the User must determine whether or not these roughly-meshed areas will impact the results. If they do The mesh must be refined, using the various mesh improvement tools offered by SimWise, until the adjustments help produce accurate and acceptable simulation results.

A general rule of thumb in generating a good initial quality mesh is to use an element size that is at least ½ of the thickness of the smallest geometric feature. If the overall size of the model is large compared to the thickness of the smallest feature (long thin pipe, for example), using the ½ rule may result in a very large number of elements and the mesh will take longer to solve. The User should put forth their best judgment in deciding whether or not it is necessary to mesh all areas with the smallest determined mesh size. If it is not necessary, then other methods , such as mesh control , decreasing the mesh maximum angle, etc. an be used. See the section titled *Mesh Control**.*

The example below shows a part meshed using an element size of 50mm and one using 2.5mm. The thickness of the flat sections is 5mm. To capture any potential deflection across the thin sections of the part, a good starting mesh size is 2.5mm. It is easy to see how the large element size does not conform well to the round surface. In addition, the flat areas contain highly distorted elements (large aspect ratio).



Mesh Size = 50mm
Undesired final quality

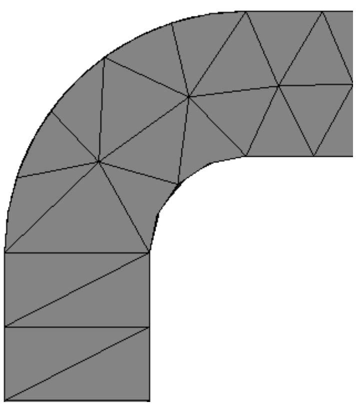


Mesh Size = 2.5mm
Desired final quality

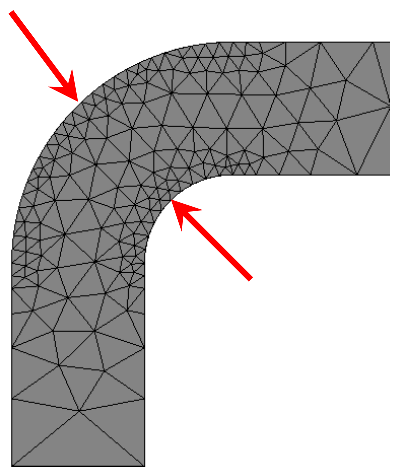
Mesh Control

Mesh Control is a feature that allows the User to specify the initial mesh size of a geometric feature, regardless of the body's default mesh size. It can be defined on features such as vertices, edges and faces. From the starting feature on which the mesh control is applied, the mesh size then gradually transitions (or grows) to a larger (or smaller) element size until it meets up with the default mesh size. This allows small features and problematic edges to be forced to start with a smaller element size than the size specified for the overall part., thus moving toward a final mesh faster. Additionally, it can make the h-adaptive convergence process run faster because the initial size of the elements in the high stress areas was small enough to begin with o not introduce large stress error.

The example below shows a bend where both the inside and outside radii had mesh control applied. The default mesh size for the body is 2.5mm. The mesh control initial size is 25% of the default mesh size (or .625). Notice the gradual size transition to the larger mesh.



No mesh control



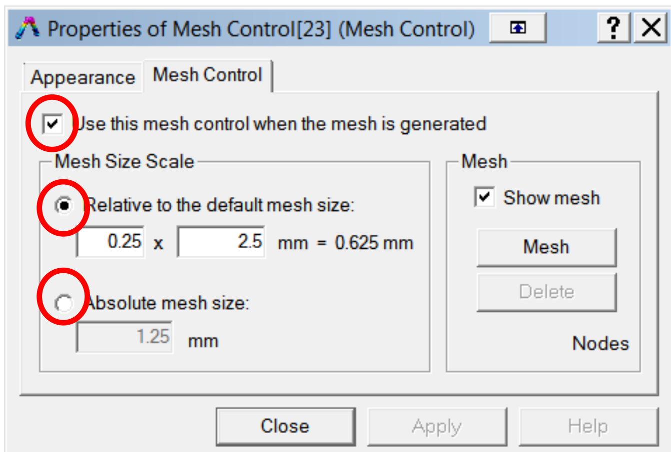
Mesh control applied to inside and outside bend radii

In general, mesh control is primarily useful where there are only a few small features that exhibit large stress. This allows the User to apply a smaller mesh only in the significant stress areas and let s SimWise automatically handle the remaining areas. This can help make meshing and solving faster and more efficient.

SimWise also has a feature called h-adaptivity, which automatically refines a mesh in specific areas flagged by User-defined accuracy criteria. You may read more about this in the section titled [*H-Adaptivity Overview (automatic mesh refinement).*]

Using Mesh Control

The following section explains the use of the mesh control properties d-box.

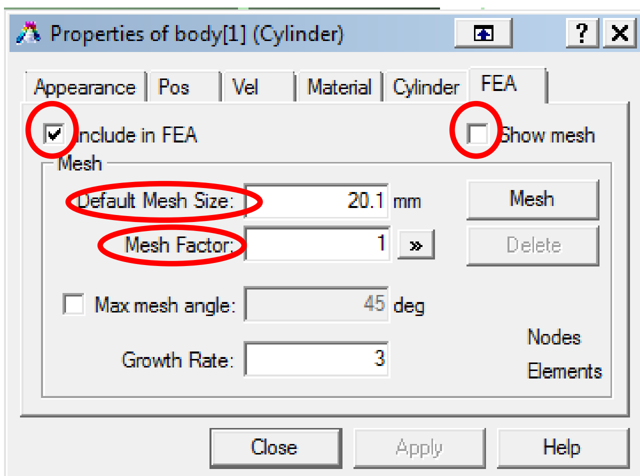


- ***Use this mesh control when the mesh is generated** - activates or deactivates the mesh control after it is defined*
- ***Relative to the default mesh size** – specify a value for the initial mesh control, relative to the default mesh size*
- ***Absolute mesh size** – Alternative to specifying the mesh control size as a relative value of the default mesh size. Uses the defined value as the starting size.*

Even though a User might plan on using h-adaptive mesh refinement, mesh control can still help get closer to a better quality mesh so that the refinement process is faster and more efficient. H-adaptive refinement can override the mesh control size, if it sees a need, to reduce stress errors further in those areas.

FEA Properties *(initial mesh)*

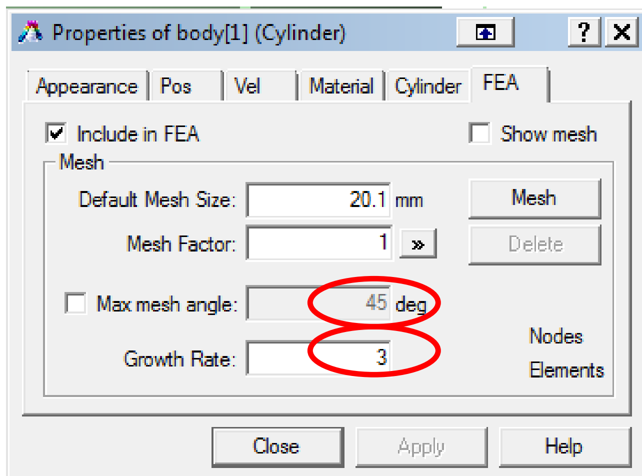
In this section, we will cover some of the features of the FEA Properties d-box, which control the initial default mesh.



- ***Include in FEA*** *- activates a part in the FEA (or deactivates it). You can also access this feature by right-clicking on the part name in the browser or right-clicking on the body itself in the graphics area.*
- ***Show Mesh*** *– Display or hide the mesh for that body. This can also be done by right-clicking on a body and toggling Show Mesh on or off.*
- ***Default Mesh Size*** *– Initial mesh size recommended by SimWise. This value is based on the overall length, width and height of the object.*
- ***Mesh factor*** *– A number that is related to the Mesh Size. When using the Mesh Factor to establish the mesh (element) size, the value is based off of an equation relating the the length of the diagonal of the bounding region of the body to the mesh factor value. It allows bodies to be meshed with a size that is relative to their own size, rather than relative to another body.*

FEA Properties *(initial mesh)*

In this section, we will cover the remaining features of the FEA Properties d-box which mainly address means of controlling the mesh quality.



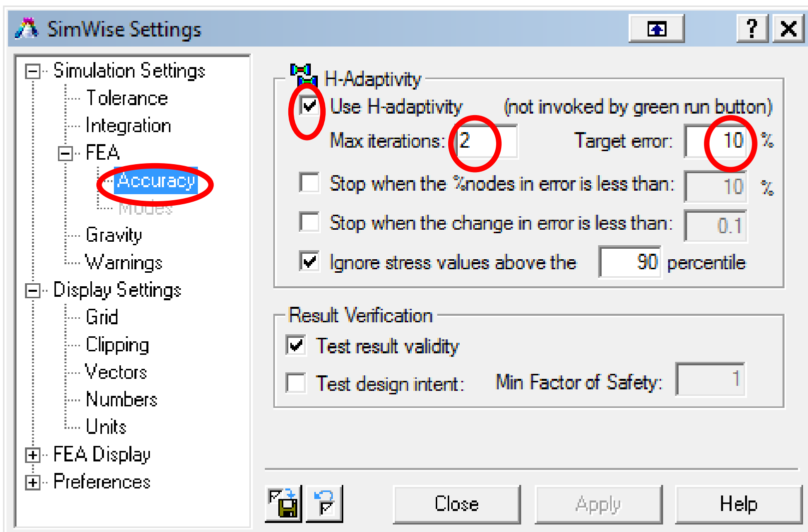
- ***Max Mesh Angle*** *– Maximum spanning angle, across curved surfaces, for the generated elements. Default is 30 deg. A smaller angle will increase the number of elements across all curved surfaces in the model. A large angle will decrease the number of elements across curved surfaces, thus resulting in a coarser curved surface mesh.*

    Note: this feature can help improve solve times by intentionally defining larger angle values and creating a coarser mesh, to roughly represent curved surfaces. Not all curved surfaces in the model will necessarily exhibit high stress. Therefore, it may not be necessary to mesh all curved surfaces using the default angle. As with any FEA mesh input parameter, proper judgment must be used when determining when to use this feature.

- ***Growth Rate****– Governs the rate at which the size of adjacent elements can grow. The ratio of the edge lengths of adjacent elements will not exceed the specified growth rate. Value cannot be less than 1.5. Default is 3.*

FEA Accuracy Settings

The following d-box is used for setting various FEA results accuracy criteria. A few of the settings will be discussed below. You may select the context-sensitive help button ? at the top-right of the d-box and then click on any of the fields to display information about the additional features not discussed here.



- ***Use H-adaptivity*** *– forces SimWise to use the H-adaptive mesh refinement process to achieve the specified error criteria*

- ***Max iterations*** *– Number of times the mesh will be refined to attempt to meet the Target error criteria*

- ***Target error*** *– percentage of overall model stress error allowed. SimWise will consider a solution converged when it can refine the mesh to meet or exceed this criteria.*

Note: SimWise does not model contact between bodies in FEA. Bodies that are to have interaction (make contact) may have to be treated as bonded. If this is the case, the User must be confident that modeling the interaction using bonding is not over constraining the model or misrepresenting the structure. See Assembly Bonding for more information.

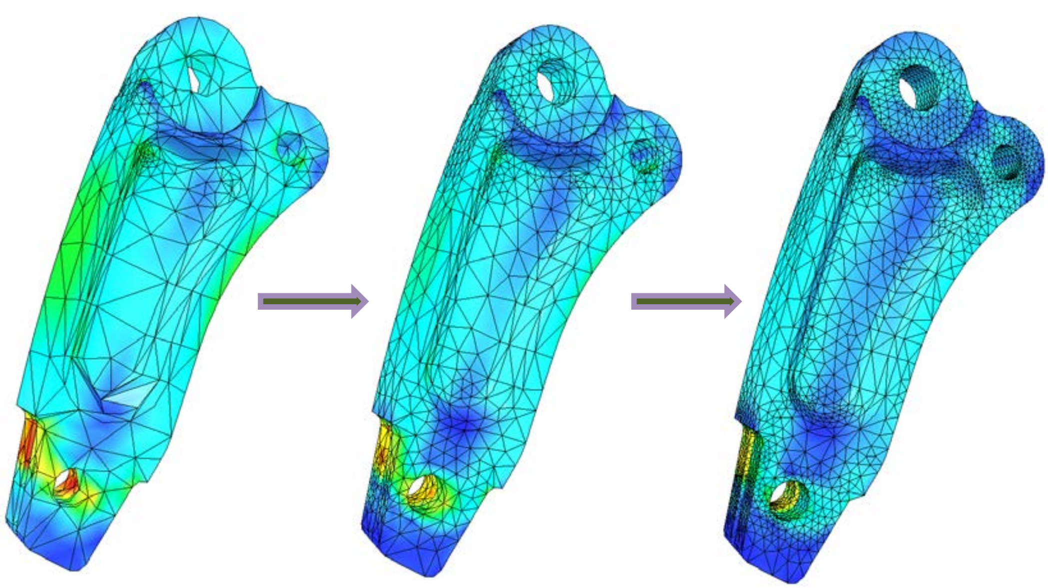
H-Adaptivity Overview (automatic mesh refinement)

In FEA, the process of successively refining a mesh to achieve optimal results is called convergence. When the mesh is continually refined in areas of high stress error, such that successive runs begin yielding negligible result differences, one can say the solution has converged within the desired tolerance.

Faster and more efficient solutions in FEA can be achieved using the largest element size possible while still retaining an accurate mesh. SimWise has a feature called H-adaptivity, which uses an automatic mesh refinement procedure to selectively modify the element size and mesh density in areas that experience high stress error. These errors can be caused by things such as distorted elements , elements that are too large, or elements whose size does not properly fit the geometric feature being analyzed, such as a small fillet.

SimWise allows the User to specify convergence criteria for the H-adaptive process. These criteria can be things such as the percentage error across elements, the percentage of nodes that are in error and the overall change in stress error between successive mesh refinements. See *[FEA Accuracy Settings](#)**.*

The following example shows a part that has had 2 levels of automatic refinement .You can see how the mesh has been refined in selective areas. Mesh control can also be used to produce similar results but requires more work in setting up the mesh. In addition, when using mesh control, one does not easily know whether too much or too little control has been applied without running each iteration separately.

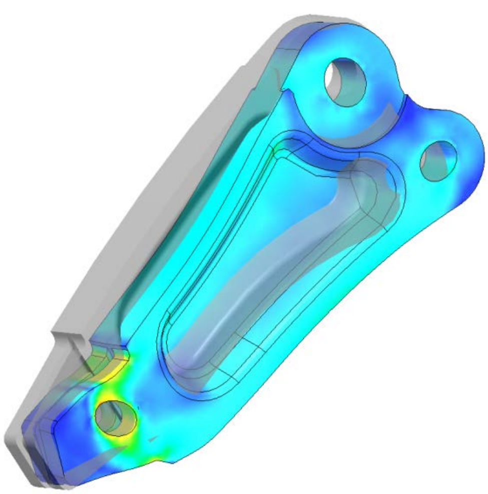


H-adaptive refinement

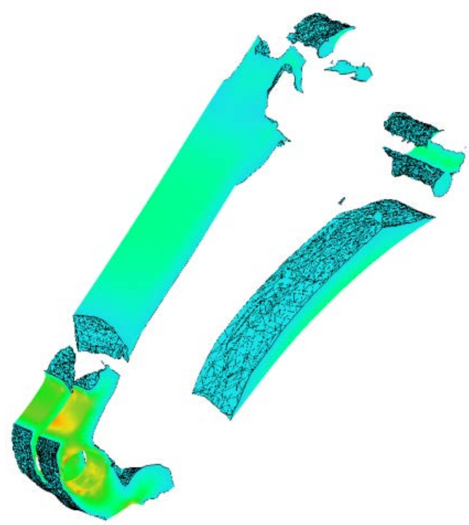
FEA Post Processing Tools

*SimWise offers several additional tools for post processing FEA results and gaining a better understanding of the analysis results: Several plot type options give you the ability to investigate other components such as **plane**, **shear** and **principal stresses**. In addition to **strain** there are also **Factor of Safety** plots and **stress error** plots*

*There are visual tools too. **Clipping** allows you to cut a section plane through the part. **Isosurface** allows you to isolate areas inside the part either above or below a defined stress or displacement value. **Surface probing** allows you to hover the cursor over an area and have a flag appear near the mouse that indicates the stress or displacement value at that node location. And **Show deformed** allows you to show an exaggerated scale of the deformation of the part*



Deformed shape (scaled by 1,000) with original geometry overlay

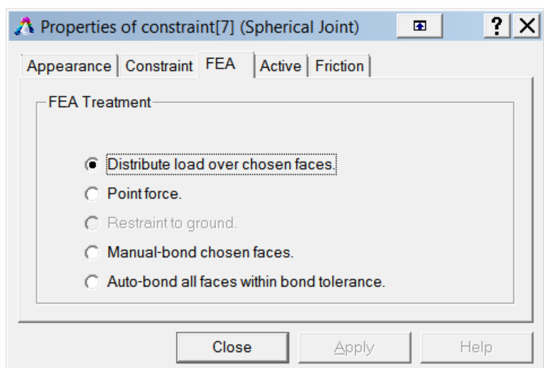


Isosurface with mesh display

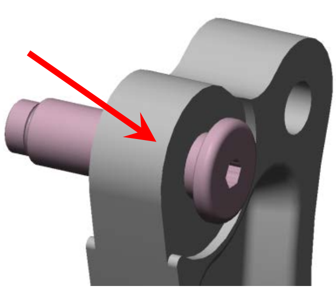
Distributing Loads From Constraints

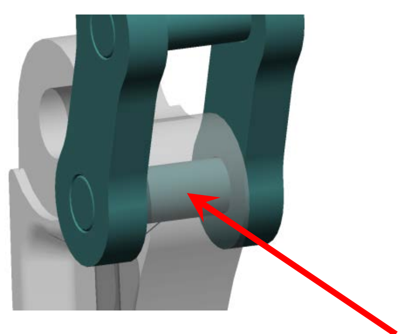
When there are constraints defined between bodies, the User must tell SimWise how and where to distribute the loads resulting on these constraints. For example, a pin and hole are connected using a revolute constraint. The User needs to specify that any loads for the constraint should be distributed on both the outside face of the pin and inside face of the hole. Another example is when two bodies are to be welded together. The User needs to specify that the face areas where the weld occurs are to be bonded.

The FEA tab in the constraint Properties d-box has options for choosing how to distribute the loads.



The following images show examples of when you might use the Distribute loads over chosen faces option. Holding down the Ctrl key will allow for the selection of multiple faces for the load.





<u>Revolute joint examples</u>:
Loads should be distributed on ID of hole and section of pin surface in contact with the hole. It may be necessary to apply a split face region on the pin shoulder so that the loads can be defined as only contacting this mating region

Distributing Loads From Constraints

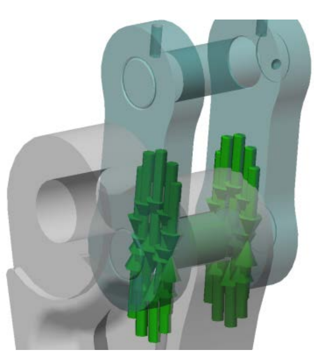
A brief discussion on using motion loads in an FE analysis:

SimWise knows when a constraint is defined between two bodies. However, it does not necessarily know which faces are held together by the constraint in the physical World. Its treatment of bodies in the motion simulation is analogous to using a free body diagram to solve problems in static and dynamics, where only a schematic representation of the model is used for analysis. This approach is primarily concerned with the location of User-applied loads, constraint location, and the center of mass location - a skeleton representation of the model if you will. It does not recognize actual geometric features.

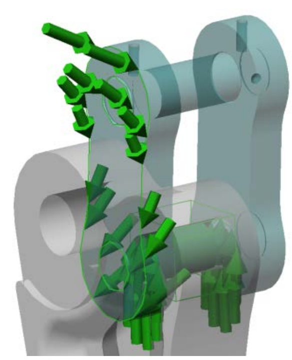
Once a body is set to be included in a combined Motion+FEA study, SimWise bridges the free body diagram approach of the motion simulation with a structural analysis, which is fully dependent on actual geometric part features. In this case, the User will see small (green) graphic arrows appear on the body, near the location of the constraint or possibly even somewhere else unexpected on the body, initially. These graphics represent the resulting motion constraint forces that will be applied to the body for the FE analysis.

In some instances, the placement of the force graphics on the part is determined automatically (initially). As a result, a force from a constraint may inadvertently get applied to a face that is not associated with its true physical connection. Therefore, prior to running an FEA using motion loads, the User must be sure the constraint loads are assigned to the proper faces. For example, if there is a Revolute constraint used to represent a pinned connection between two parts, the force graphics must be applied to the face of the hole and the face of the pin for accurate FEA load distribution (see images below).

In the next section, we will show how to assign the motion forces to the proper faces of a part for accurate load distribution in FEA.



CORRECT:
Force graphics applied to both the hole face and pin face



INCORRECT:
One force graphics set applied to hole face and another to side face of link

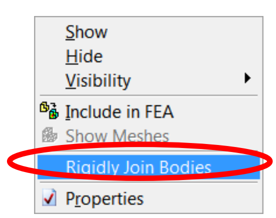
Assembly Bonding

SimWise does not model intermittent contact between bodies that may occur during deformation. This is different from modeling collisions in a motion analysis.   Bodies that may undergo such interaction may have to be treated as bonded. If this is the case, be sure you are confident that modeling the interaction using bonding is not over constraining the model or misrepresenting the structure. Many times even though physical parts might be allowed to separate and then make contact (assuming small deformation theory), the contribution this has to overall model stress may be minimal. In that case, whether they are bonded or not might not make a difference.

Criteria for bonding:

1. *Applies only to those parts that have been included in the FEA.*
2. *The **Bond Tolerance** used by SimWise is the maximum separation that can exist between two features and still allow them to be bonded at their adjacent nodes.   If the separation is greater than this value, bonding will not take place. This setting can be accessed under the FEA Properties*
3. *The bodies owning the feature(s) to be bonded must be **joined** in some way with a constraint (rigid, spherical, revolute, etc.). The constraint location can be anywhere on the bodies. As we discussed in the section on distributing loads, depending on the type of constraint used there will be different options for the bonding.*

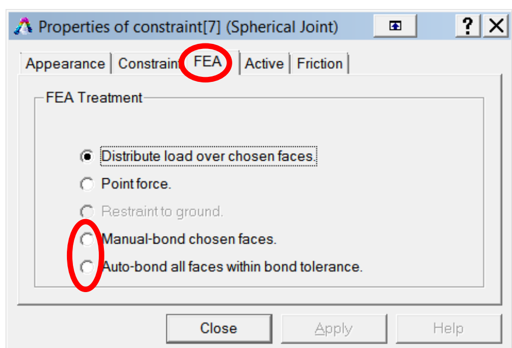
If there are no constraints already joining the bodies then the easiest and fastest way to bond two bodies is to use a rigid joint. This is common when doing an FEA only and no motion analysis.   To rigidly join two bodies together, you have the option of either 1) manually defining a rigid constraint between them or 2) using the "Rigidly join bodies" option found when right-clicking on a sub assembly name or the main assembly name in the browser.



Assembly Bonding (manual vs. automatic)

*Bonding is defined either **manually** or **automatically**.*

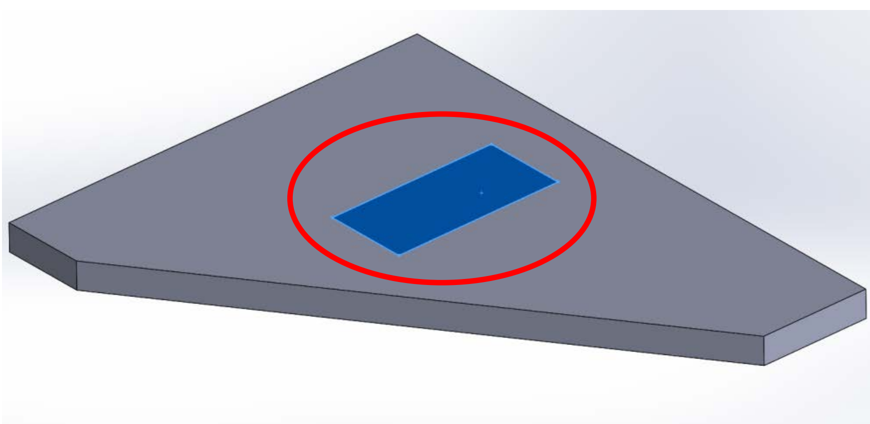
- *Manual bonding is applied by selecting the FEA tab under the constraint Properties and choosing Manual bond chosen faces. In this case, the user selects which faces to bond. Remember that the features must be within the bond tolerance in order to have their nodes joined.*
- *Auto bonding automatically bonds any faces between parts that are separated by no more than the bond tolerance*



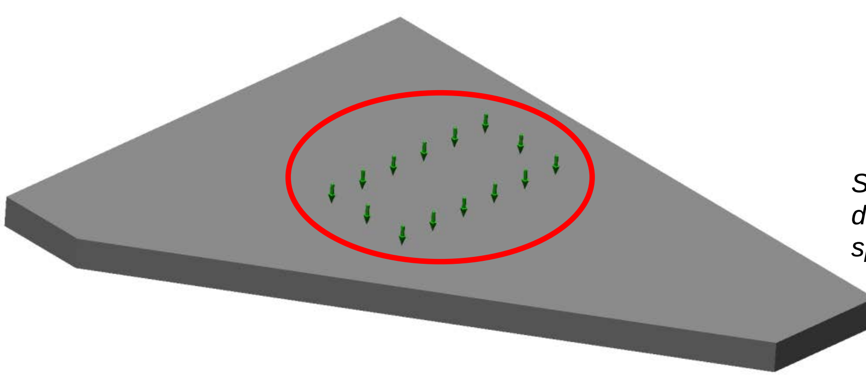
Splitting faces for restraint application

Sometimes it becomes necessary to apply a load or restraint only in a specific region on a body. This can be impossible unless a r projection or silhouette region is created inside the CAD model first.

Using a feature in the CAD program , such as "split line", "split face", silhouette projection", etc. , faces can be segmented into regions (or split) so that applied loads or restraints are applied only to the specific regions on the faces where they need to be.



CAD model with split region

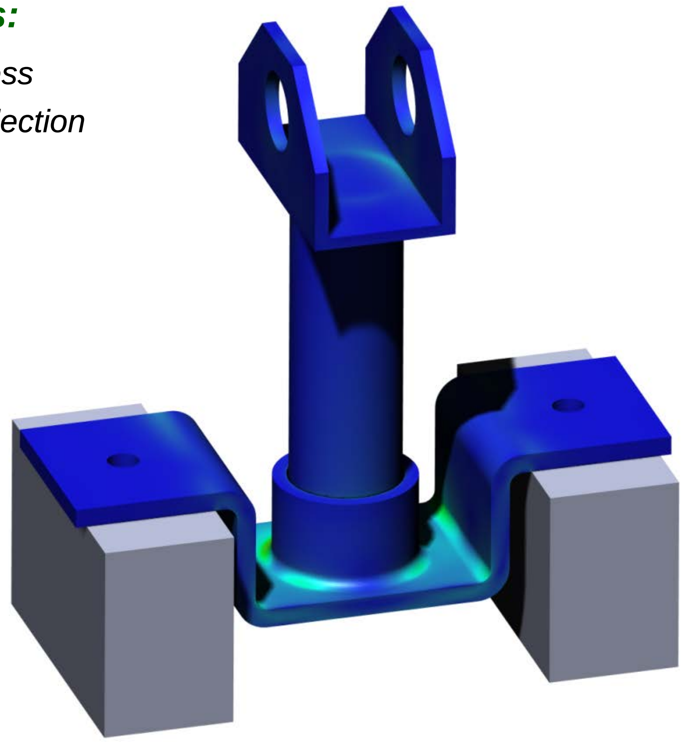


SimWise model with distributed load applied to split region

Exercise - Bracket Assembly

Simulation Objectives:

- Determine maximum stress
- Determine maximum deflection



Features Covered:

- Material property assignment
- Restraints
- Applied loads
- Bonding
- Meshing
- Determining stresses
- Determining displacement (deflection)
- H-Adaptive mesh refinement
- Overlaying original geometry
- Isosurface Plotting
- Animating FEA Results
- Photorealistic Images

Introduction

This tutorial is designed to introduce you to some of the basic capabilities of SimWise as applied to a structural analysis and help you get acquainted with how to prepare, run, and analyze the results of a finite element analysis.

In this exercise, you will perform a basic stress and deflection analysis on an assembly of parts.

Open the SimWise file

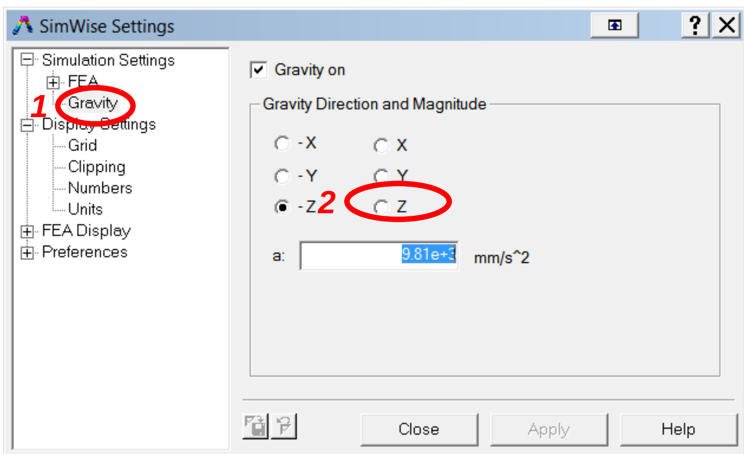
1. ***Start SimWise***

2. *Select **File, Open** and **Browse and locate** the file called "**SimWise Tutorial – Bracket Assembly.wm3**".*

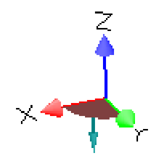
Change Gravity and Unit Settings

1. ***Select*** *the* ***Simulation Settings*** *icon* 

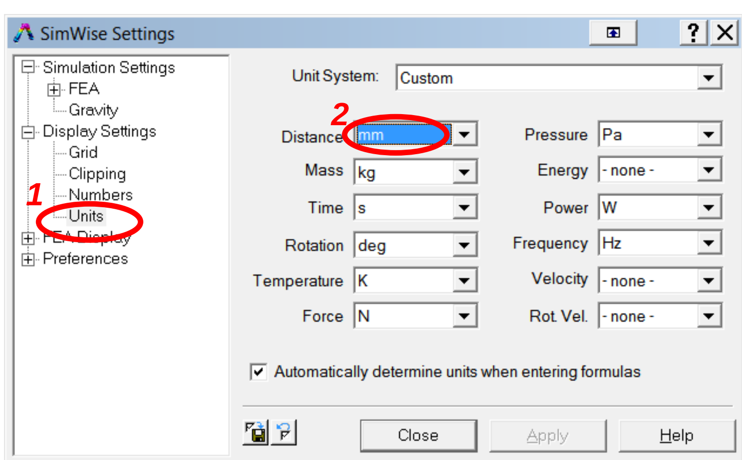
2. ***Select Gravity*** *from the settings list and* ***select*** *the* ***–Z*** *direction option. Leave the* ***Settings d-box*** *open.*



Note: *The cyan colored arrow on the Orientation Indicator represents the direction of gravity*



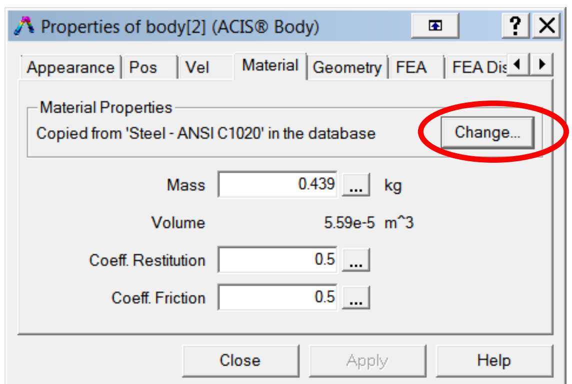
3. ***Select*** *the* ***Units*** *option from the settings list,* ***select*** *the drop down menu next to* ***Distance*** *and* ***select mm****.* ***Close*** *the d-box when finished.*



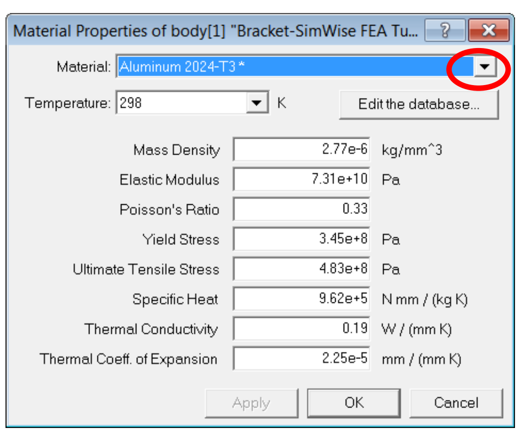
Assign Material Properties

1. ***Right-select** the **Bracket** (or double-click) and **choose Properties***

2. *In the **Properties window**, **select** the **Material tab** and then **select** the **Change button**.*



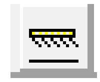
3. ***Select** the **drop-down arrow** next to the Material field and **choose Aluminum 2024-T3** as the material.*



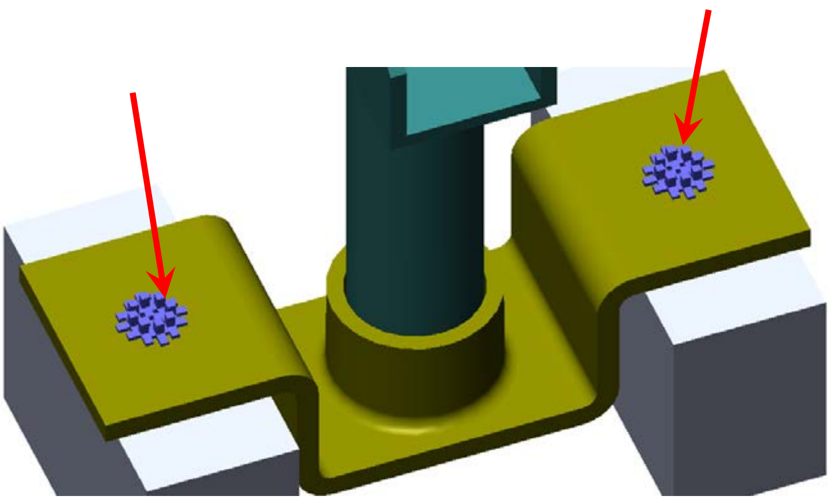
***Tip:** You may enter your own materials using the "Edit the Database" option and then selecting the New Material button*

4. ***Repeat steps 1 through 3** for the **Post***

Add Hole Restraints

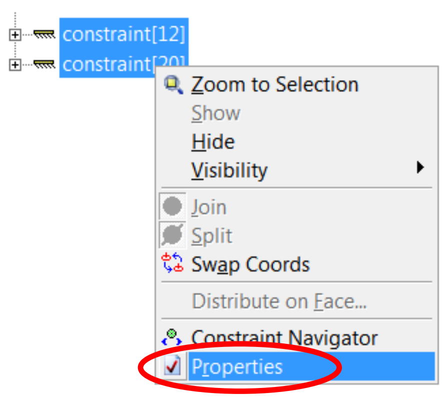
1. ***Double-click*** *the* ***Restraint icon*** *on the* ***toolbar*** 

2. ***Select*** *the two holes as shown*



Tip: *Clicking an icon once, such as the restraint tool, will allow that feature to be used once, then the cursor will revert back to the arrow tool. To use a feature multiple times, double click its icon. To exit the mode, either select the arrow tool or right-click in the graphics area an choose Select*

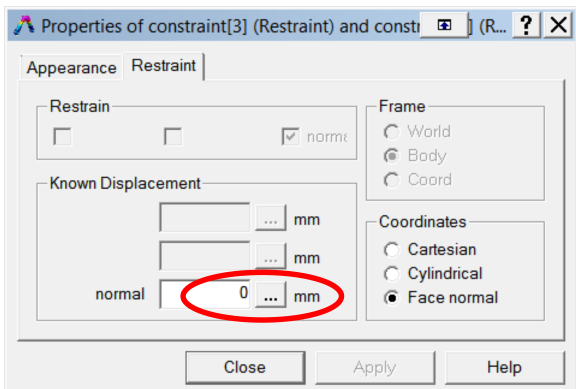
3. *In the* ***Object Browser****,* ***hold down*** *the Ctrl key and* ***select*** *both restraints. Then* ***Right-click*** *on either of the highlighted restraint names and* ***choose Properties***



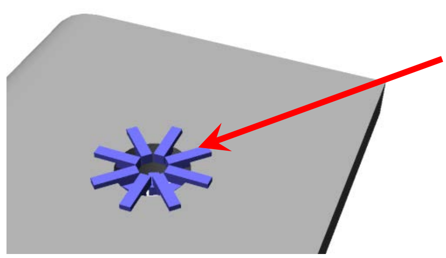
Tip: *You can also double-click (or right-click) the restraint graphic in the viewing area and choose Properties*

Add Hole Restraints

1. *In the **Properties D-box**, select the **Restraint tab** and **select** the **Face Normal** option under **Coordinates**. Leave the default value of **0mm**. **Select Close**.*



***Tip:** A restraint may also be given a pre-defined displacement value. This is useful for determining loads required to achieve a given deflection*



The restraint icon on the hole should now appear as follows

Coordinates	Frame	Description
Cartesian	World	Restrained in XYZ directions, aligned with respect to World origin
	Body	Restrained in XYZ directions, aligned with respect to Body origin
	Coord *	Restrained in XYZ directions, aligned with respect to the coord
Cylindrical	World	Restrained in cylindrical coordinate directions (radial/tangential/longitudinal), aligned with respect to World origin
	Body	Restrained in cylindrical coordinate directions (radial/tangential/longitudinal), aligned with respect to Body origin
	Coord *	Restrained in cylindrical coordinate directions (radial/tangential/longitudinal), aligned with respect to the coord
Face Normal	N/A	Restraint normal (perpendicular) to geometric face

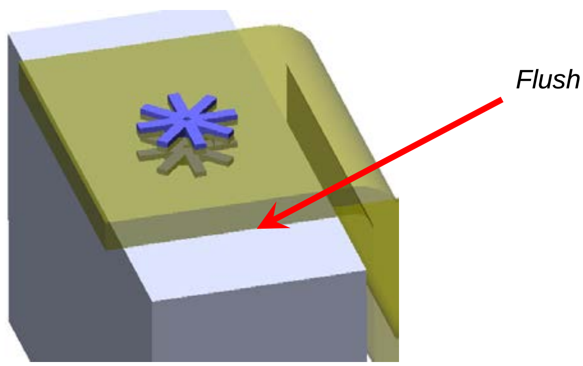
** If the restraint is applied to a surface, the coord is created at the location of the cursor selection (mouse click) on the surface*

Bonding Criteria

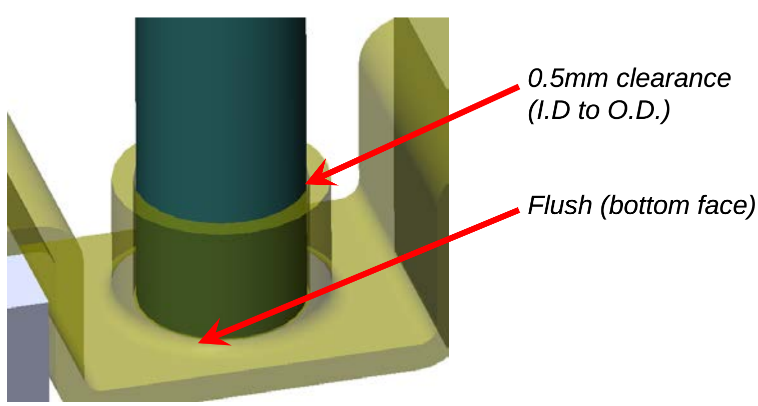
To model the assembly conditions, the bracket will be bonded to the two blocks (ground) and the post will be bonded to the bracket.

*The **Bond Tolerance** used by SimWise is the maximum separation that can exist between two features and still allow them to be bonded at their adjacent nodes. If the separation is greater than this value, bonding will not take place.*

Each flange of the bracket is in flush contact with the top of each block.



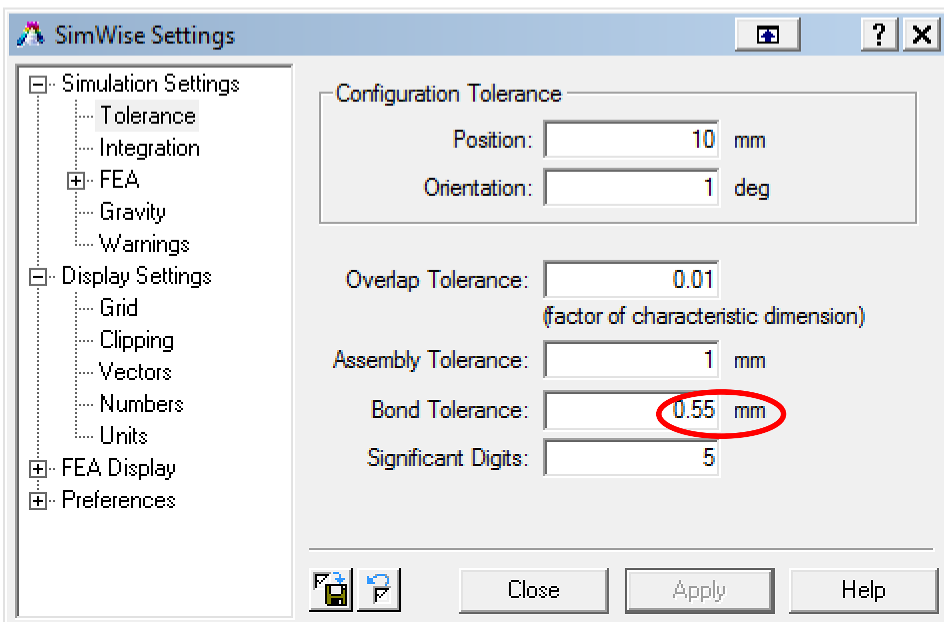
The outside diameter of the post is 28mm and the inside diameter of the bracket hole is 29mm. Therefore, there is a total of 0.5 mm between the cylindrical faces of each part. The bottom of the post is in flush contact with the bracket



Set Bond Tolerance

1. ***Select** the **Simulation Settings icon** to open the **Simulation Settings d-box**.*

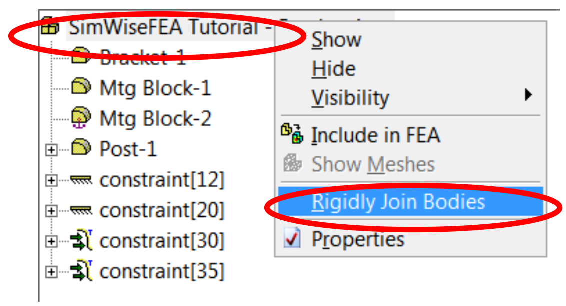
2. ***Click** on **Tolerance** and **enter .55** in the **Bond Tolerance** value field*



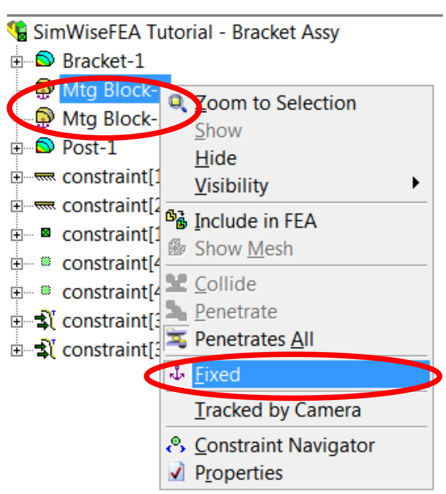
Add Constraints

The bodies that are to be bonded must have a constraint defined between them, typically a rigid constraint.

1. ***Right-click** on the assembly name in the browser and **choose Rigidly Joint Bodies**. Rigid constraints will be added between the two Block/Bracket pairs and the Bracket/Post pair.*

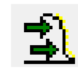


2. ***Hold down** the **Ctrl key**, **select** both **Mtg Block** names in the browser, **right-click** on either name and **choose Fixed**.*

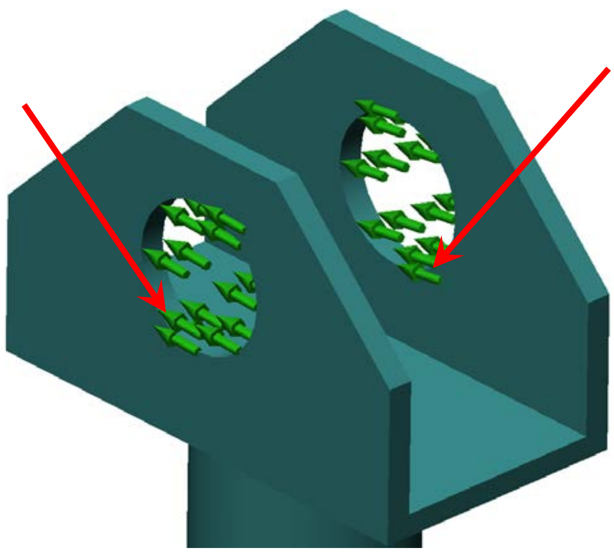


Note: Since we are not interested in analyzing the stress in the blocks, they will be left out of the analysis

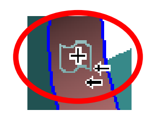
Add Load

1. ***Double-click** the **Structural Load** icon on the toolbar* 

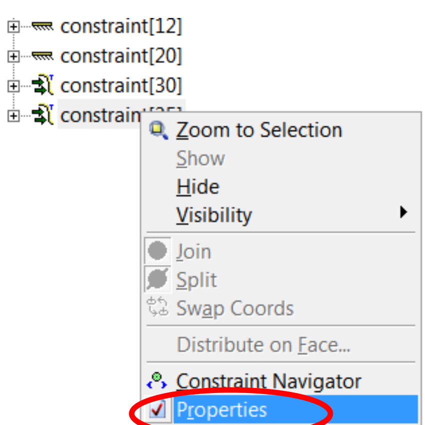
2. ***Select** the two faces **face** on the Post, as shown*



***Note:** The cursor selection icon will change depending on which geometric entity you are moving the mouse across. When selecting a face, for example, the cursor icon will change to a flag, as shown here*



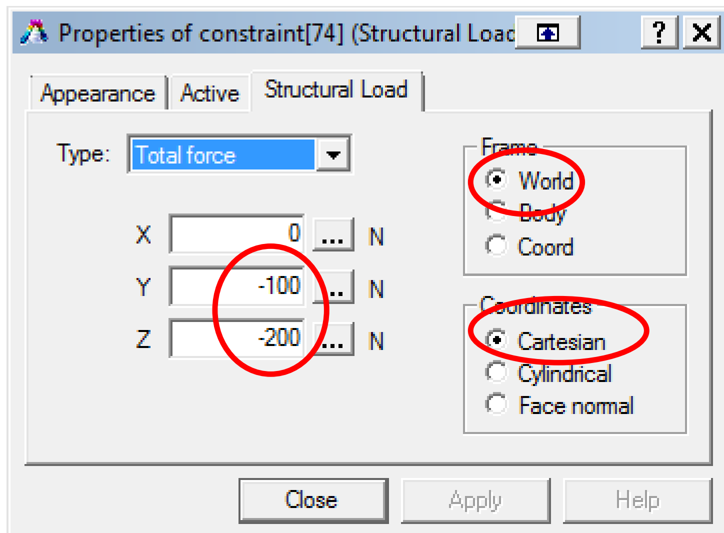
3. *In the **Object Browser**, **select** the force, then **Right-click** on the force name and **choose Properties***



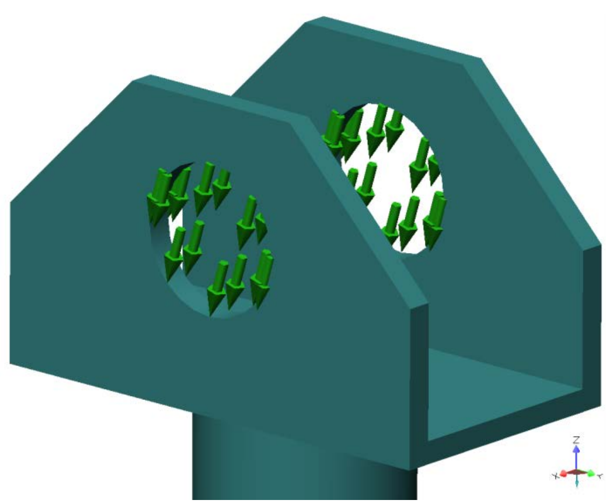
Modify Load

1. *In the Properties D-box, **select** the **Structural Load** tab, **set** the force **Type** **to Total Force**, **set Coordinates** to **Cartesian, Frame** to **World,** and **enter** **-100** in the **Y field** and **-200** in the **X field**.. **Select Apply** and then **Close**.*

Note: We are defining two components of force on the Post, which results in a vector acting at a slight angle to the vertical (z) direction. This will results not only in a reaction force where the Bracket contacts the Post but also a bending moment



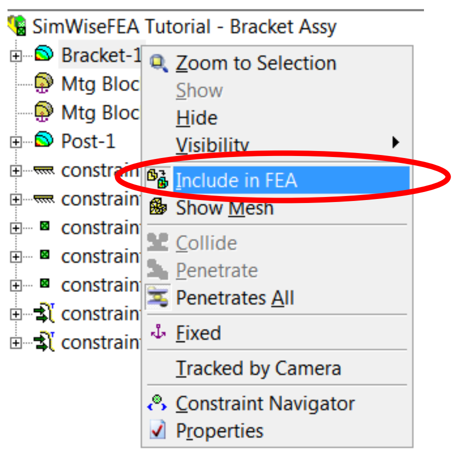
The structural load graphics should appear as follows



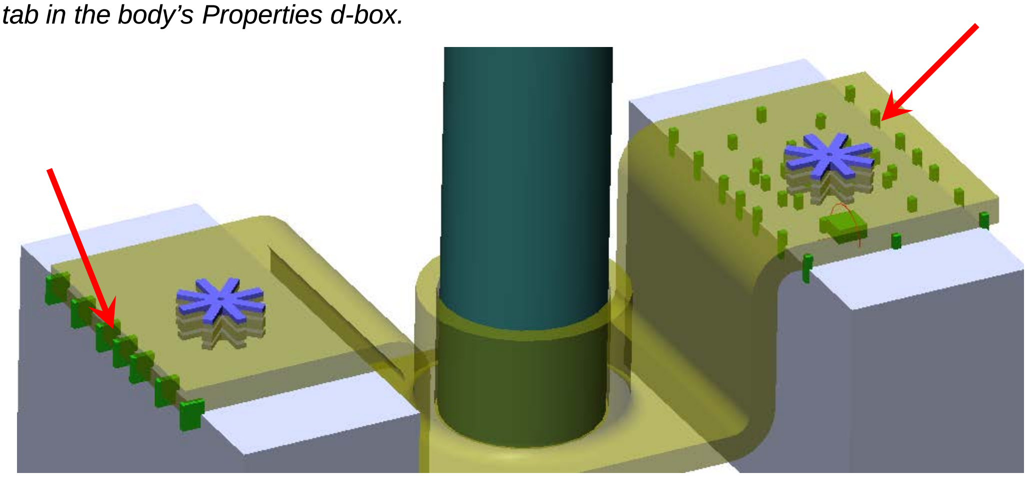
Include Bodies in FEA

1. ***Hold down** the **Ctrl key** and **select** the **Bracket** and **Post** part names in the Browser*

2. ***Right-Click** on either of the highlighted part names and **choose Include in FEA***



Once the bodies are included in the FEA study, new graphics will appear on various faces. These graphics represent where bonding and /or load assignments will be placed. Whether parts are bonded automatically or manually, or loads (instead of bonds) are used in place of bonds, depends on which settings are used under the FEA tab in the body's Properties d-box.

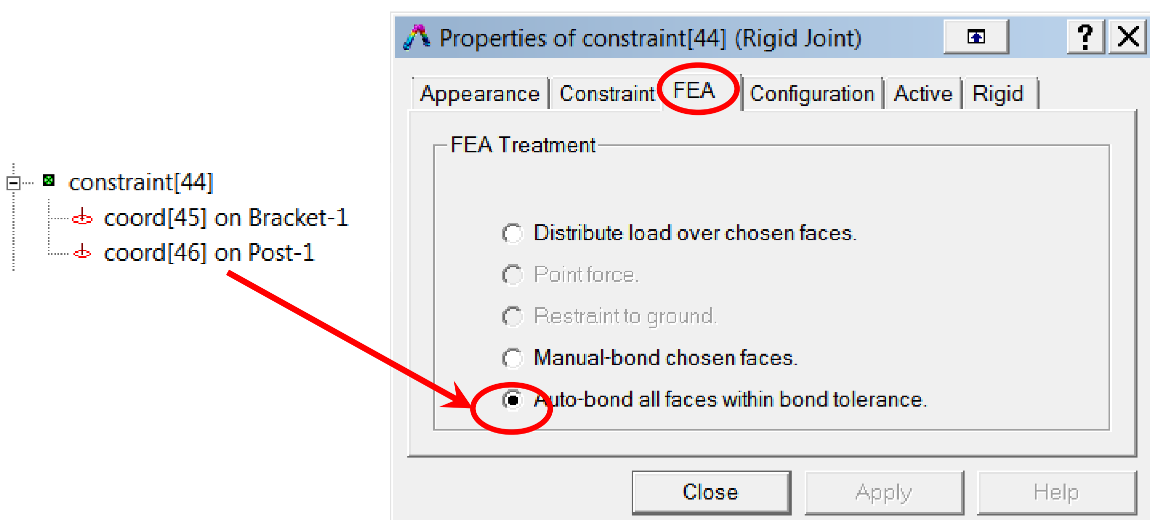


Specifying Bonding Criteria

In assigning the graphics to their correct faces, it is helpful to hide all other constraint except the one you are assigning the bonds for. Many times, constraint graphics for more than one constraint will overlap. Once the assignment is complete, hide that constraint and show the next.

1. ***Right click on rigid constraint that connects the Bracket and Post parts and choose Properties***

2. ***Click** on the **FEA tab** and, if not already selected, **select** the option to **Auto-bond all faces within bond tolerance***

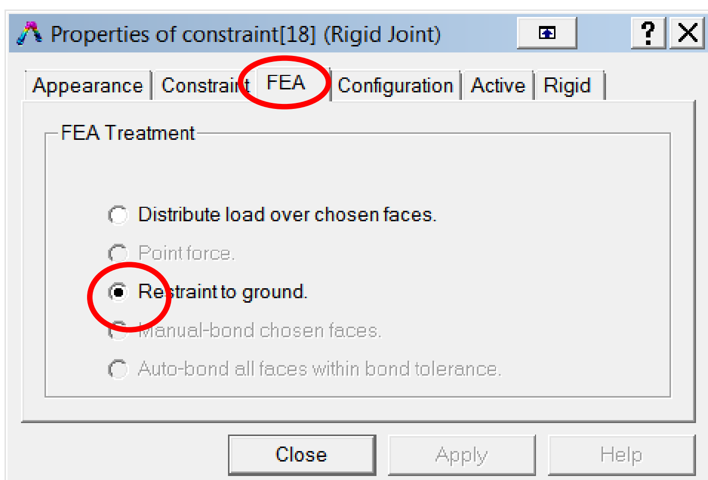


3. ***Select Close***

Specifying Bonding Criteria cont….

Since the bottom of each of the bracket ears are connected to blocks that are fixed (or rigid), the bonding type can be set to mimic the ears being attached to ground.

1. ***Hold down** the **Ctrl key** and **select** both **Block/Bracket** rigid constraints in the browser , **choose Properties**, and **click** on the **FEA tab***
2. ***Right-click** on either of the highlighted **constraint names, choose Properties**, **click** on the **FEA tab and select** the **Restrain to ground** option*



Notes on meshing

At this point, it is possible to solve the FEA. This means that before the solve begins, the bodies will be automatically meshed using the default mesh parameters. As was explained previously, the default mesh may be sufficient for first-pass approximate results. However, if the User desires more accurate results initially, the bodies can be pre-meshed individually using the various controls in SimWise, such as mesh control, Growth Rate, Maximum Angle, etc.

In this example, we will mesh each part individually and some mesh control applied. In this particular example, it will allow for faster convergence on a solution. In addition, in most modeling instances, it is more common for the User to mesh each part to a desired element size and pre-check the mesh quality, as opposed to using the default mesh.

***Create Mesh** (Initial)*

1. ***Hold down** the **Ctrl key**, **select** both the **Bracket** and the **Post**, **right-click** on either highlighted body and **choose Properties** and **select** the **FEA tab** in the **Properties d-box***

2. ***Click** on **Include in FEA***

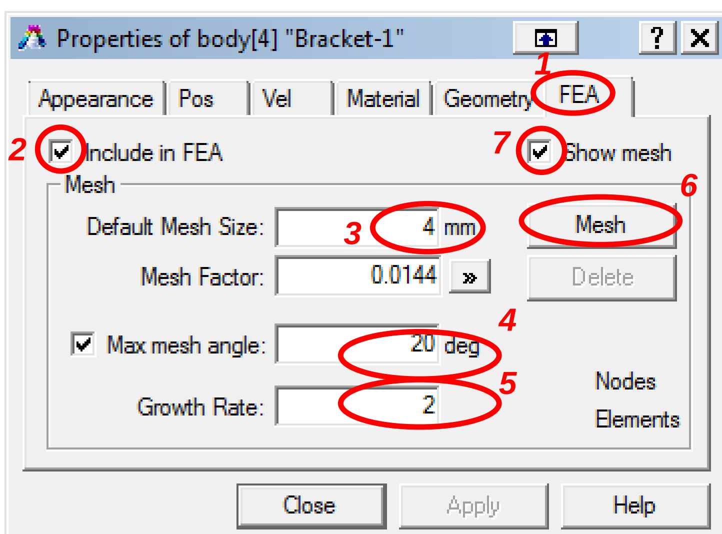
3. ***Change** the **Default Mesh Size** to **4***

4. ***Select Max Mesh Angle** and **enter 20 deg** as the value*

5. ***Enter 2** for **Growth Rate***

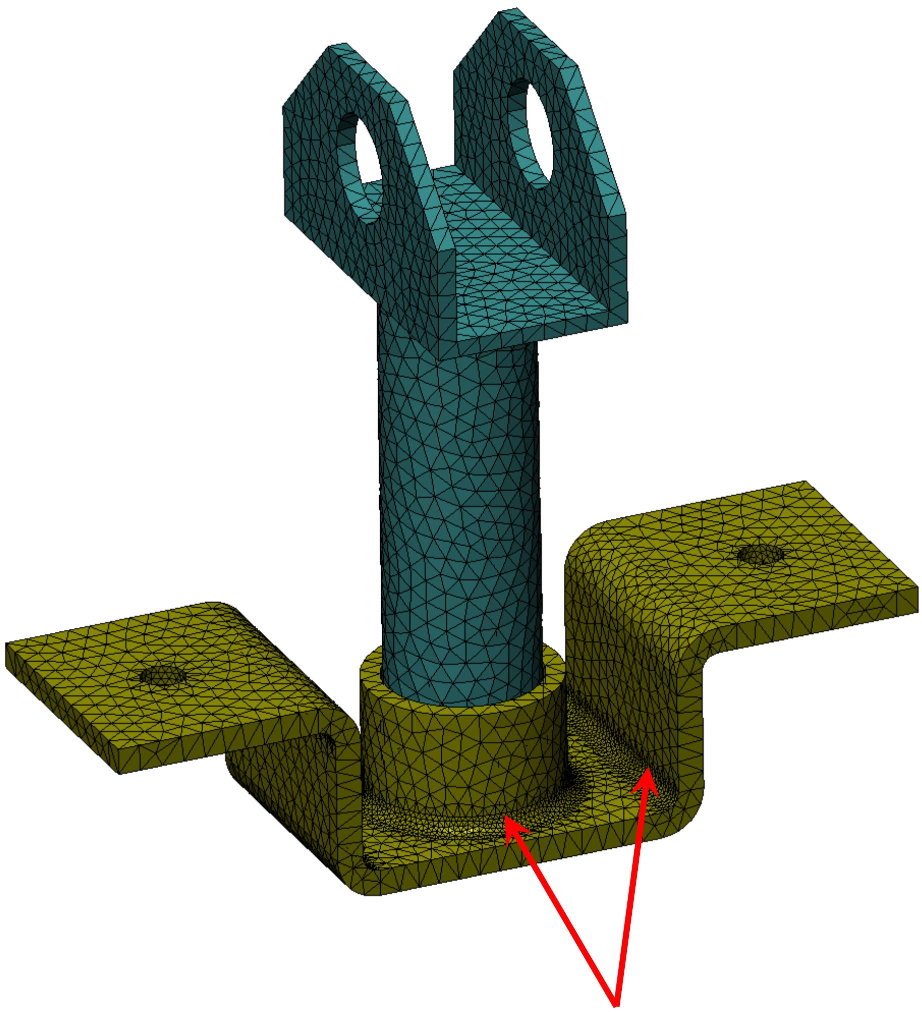
6. ***Click** on **Mesh***

7. *After the mesh process is complete (1 or 2 seconds) **click** on **Show Mesh***



8. ***Select Close***

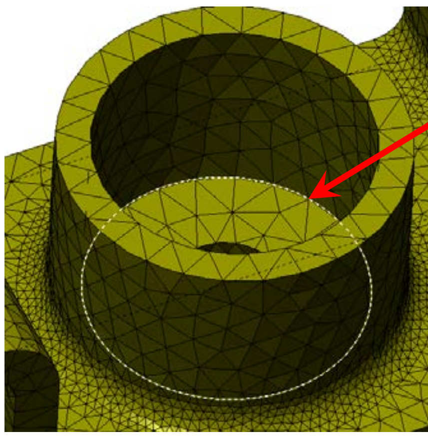
Create Mesh *(Initial)*



Notice the higher mesh density in the fillet areas. The Mesh Angle (20 deg) is the primary reason for this reduction in element size

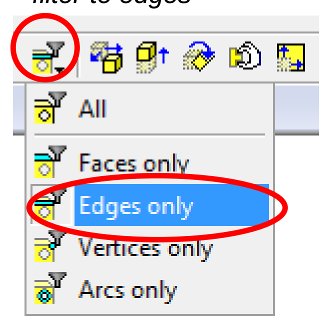
Apply Mesh Control

1. **Right-select** the **Post** and **choose Hide**
2. **Click** on **Insert** in menu and **choose Insert Mesh Control**
3. **Select** the inside edge as shown

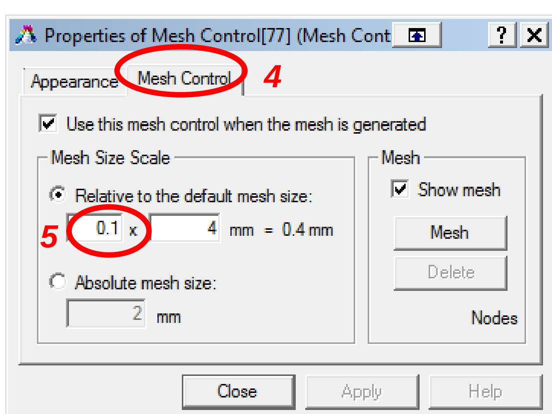


Select inside edge.

Tip: It may be necessary to change the selection filter to edges



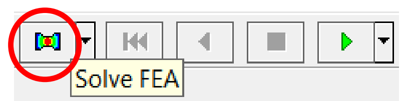
3. **Double-Click** the Mesh Control graphic icon to open its Properties
4. **Select the Mesh Control tab**
5. **Enter .1** in the **Relative To** field, as shown



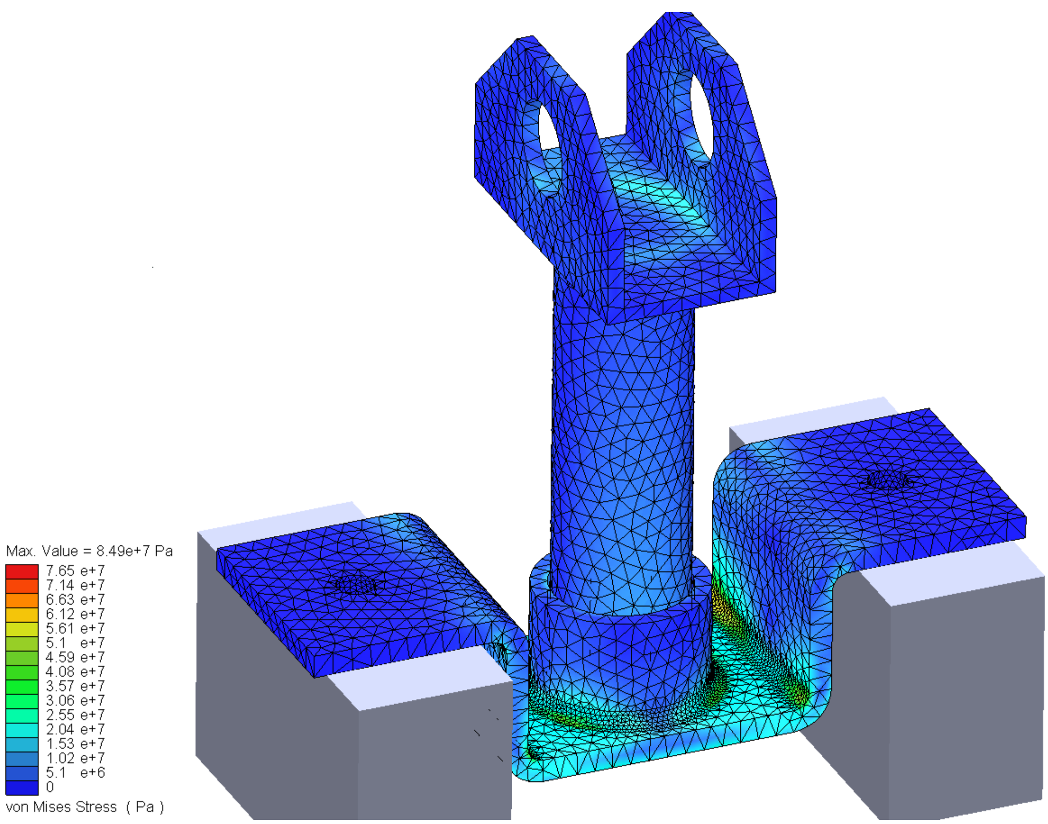
6. **Select Close**

Run FEA Simulation

1. *On the* ***Player control panel****,* ***click*** *on the* ***Solve FEA*** *button, as shown*



When the analysis is complete, a contourVon-Mises stress plot will appear and the model will be colored with respect to areas of high and low stresses



Mesh Refinement (H-adaptivity)

1. ***Select** the **Simulation Settings icon** to open the **Simulation Settings d-box**.*

   Tip: *You can also right-click in the background of the graphics window and choose Display to access the Simulation Settings*

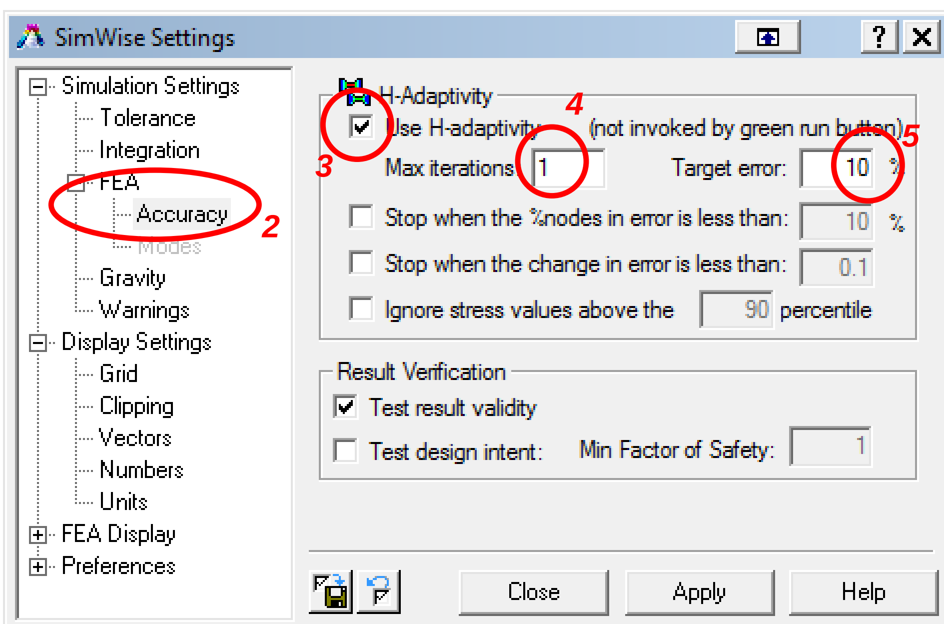
2. *In the simulation settings, **click** on **FEA**, **Accuracy***

3. ***Select Use H-Adaptivity***

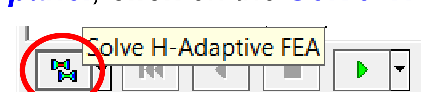
4. ***Enter 1** for the **Max iterations***

5. ***Enter 10** for **Target error***

6. ***Select Apply** and then **Close***



7. *On the **Player control panel**, **click** on the **Solve H-Adaptive FEA** button, as shown*

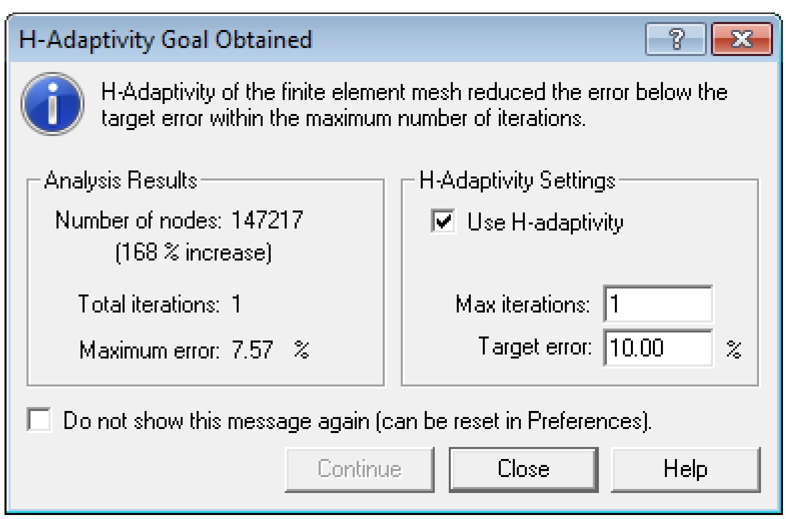


Results (Stress)

The H-adaptive analysis has allowed the analysis to reach a maximum error of 7.57%. This exceeds our goal of 10%.

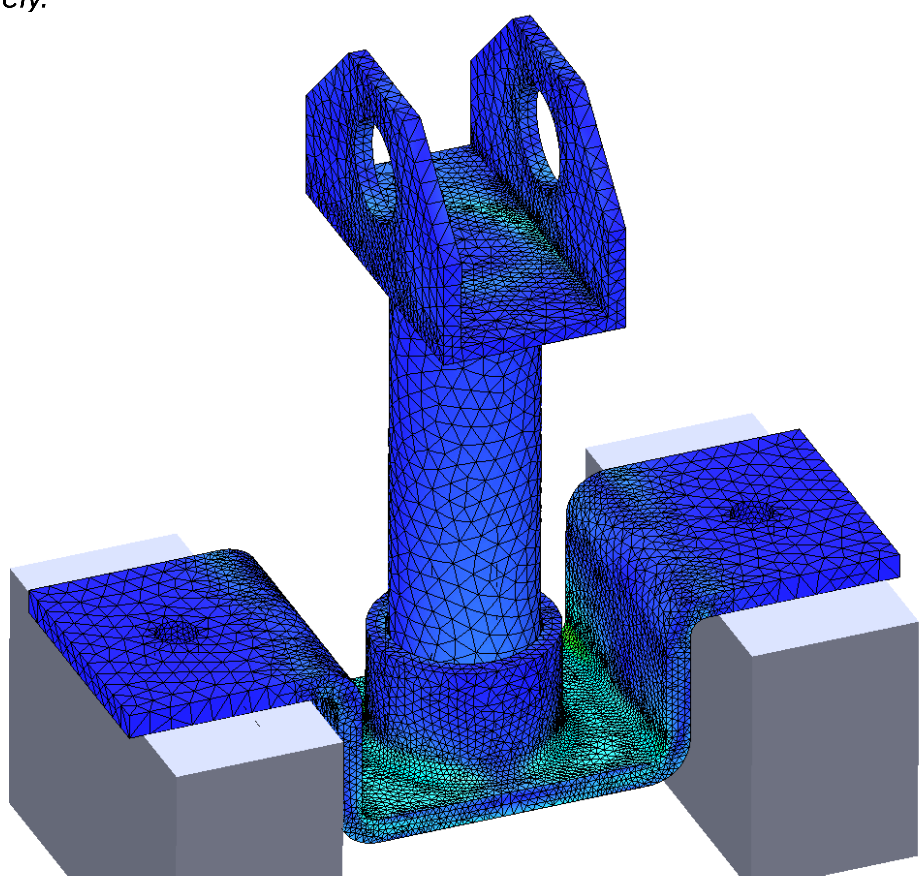
Setting the Target Error allowsTo check the difference in convergence, two successive H-adaptive refinements were performed. The first used a 10% error criteria and actually obtained 7.5%, and achieved a maximum stress of 7.95e7 MPa. The second used 7% , actually obtained 4.5% ,and achieved convergence with a maximum stress of 1.14e8 MPa.

Both results do not yield a significant difference between them considering the respective resulting factor of safety values are 1.6 and 1.5. Additional refinement beyond this point in attempt to achieve even lower stress error will most likely result in negligible difference and lengthy solve times. Depending on the application, such small differences may or may not be important.



Results (Stress)

The following image shows the resulting mesh after the convergence to 4.5% error. Notice the high element density in areas of high stress error. Note also that if you are to change the plot type to show "Error Values", and then use the Isosurface plotting tolol, you will see that the highest error is mainly due to a few singularity points near the bonding interface of the two bodies. This residual error is typical and acceptable so long as the proper tools and assessments are being made with regard to which result characteristics can be disregarded and which need to be considered more closely.



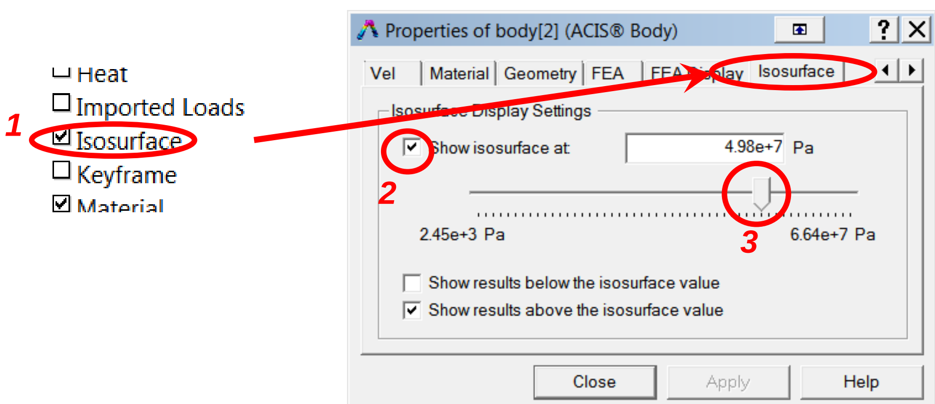
Tip: The resulting mesh from the H-adaptive mesh refinement may be used as the new initial starting mesh for subsequent analyses.

Create Isoplot

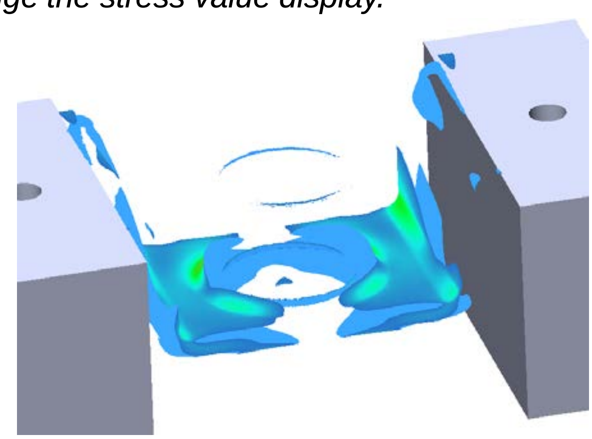
The Isosurface feature allows for areas either above or below a defined value of stress to de displayed. This is useful feature for isolating stress values in critical areas. Tip: The performance of the isolplotting can be enhanced by hiding loads and restraints graphics.

1. ***Hold down** the **Ctrl key** and **select** the **Bracket** and the **Post**.*

*2. In the **Properties List** window, **choose Isosurface** to activate the **FEA Display** tab in the body **Properties d-box**.*



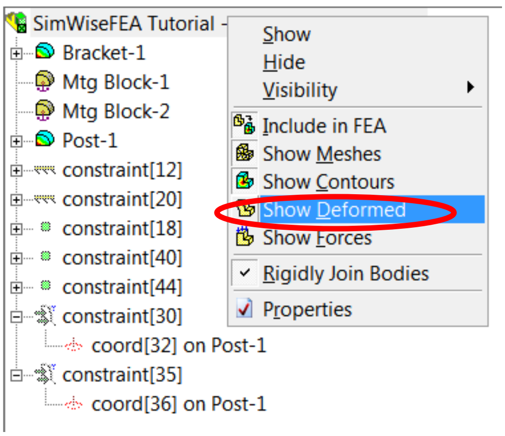
3. ***Click** on **Show isosurface** at and then slowly **click**, **hold** and **drag** the slider to change the stress value display.*

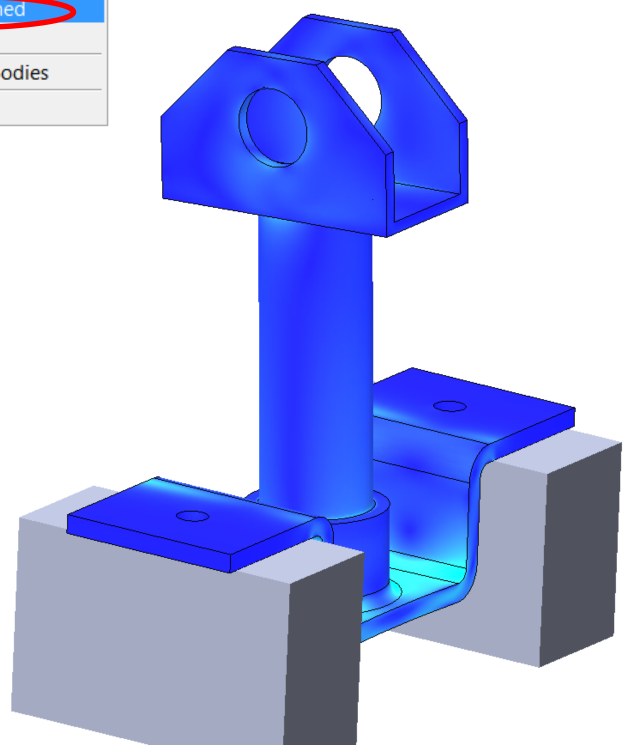


4. ***Uncheck** the **Show Isosurface** at checkbox when finished to return the model to the previous display state.*

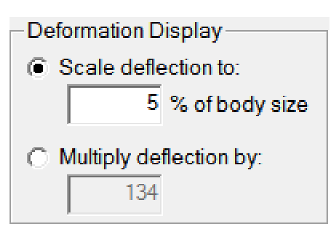
Show Deformed Shape

1. ***Right-Click** on the assembly name in the browser and **choose** **Show Deformed**. In the upper-left corner of the graphics window, the exaggerated deformation value is shown, 13.1 times, in this case.*





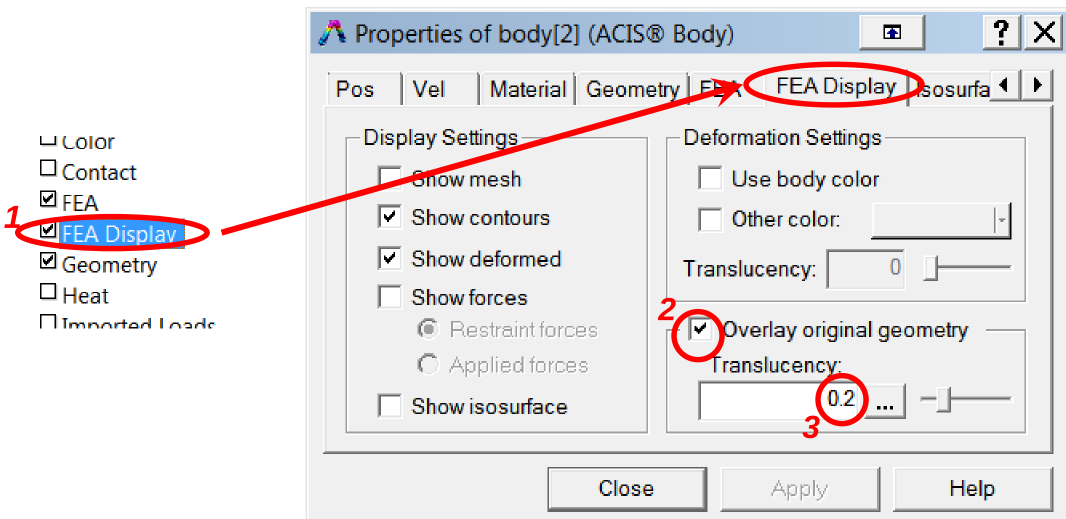
Note: The exaggerated deformation size can be changed under the FEA Display settings in the main SimWise Settings d-box



Overlay Original Shape

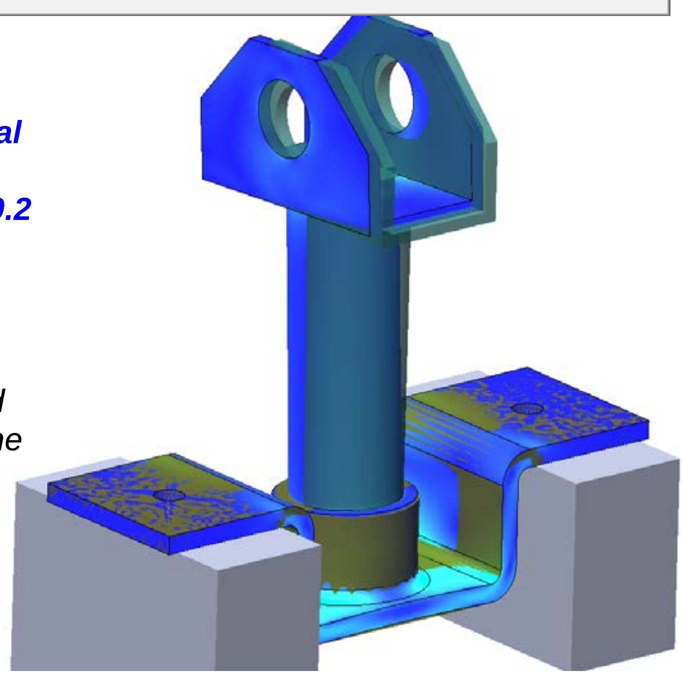
1. ***Hold down** the **Ctrl key** and **select** the **Bracket** and **Post.***

2. *In the **Properties List** window, choose **FEA Display** to activate the **FEA Display tab** in the body **Properties d-box**.*



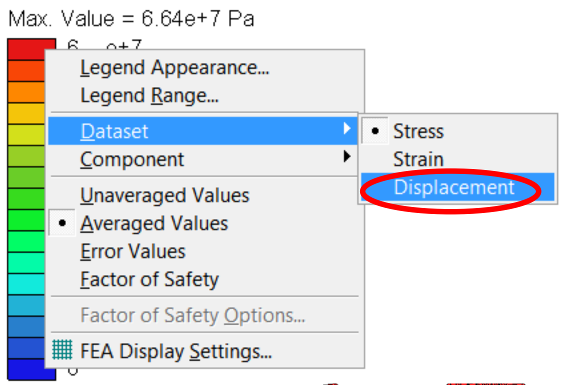
3. ***Click** in **Overlay original geometry** and **set** the **translucency** value to **0.2***

4. ***Uncheck** the **Overlay original geometry** checkbox when finished to return the model to the previous display state.*

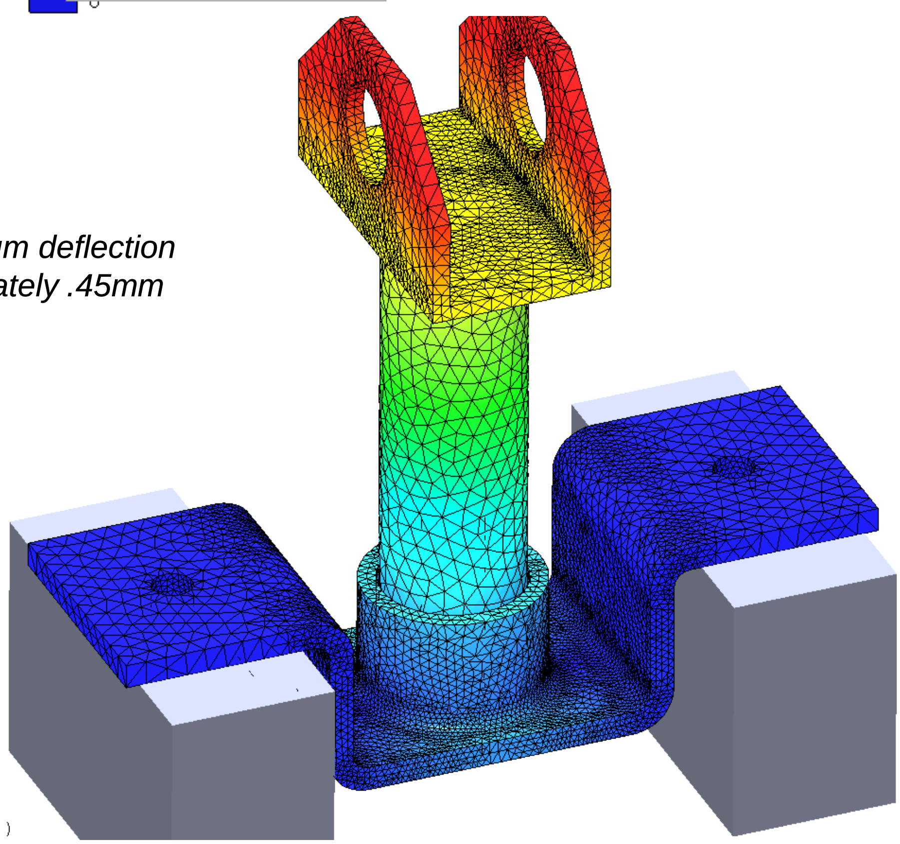


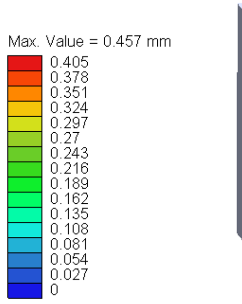
Results *(Displacement)*

1. ***Right-Click*** *on the plot legend and* ***choose Dataset****,* ***Displacement***



The maximum deflection is approximately .45mm

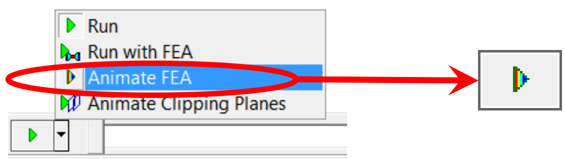




Animate FEA (Exaggerated Deformation)

Use the Playback slider to review the saved keyframed animation

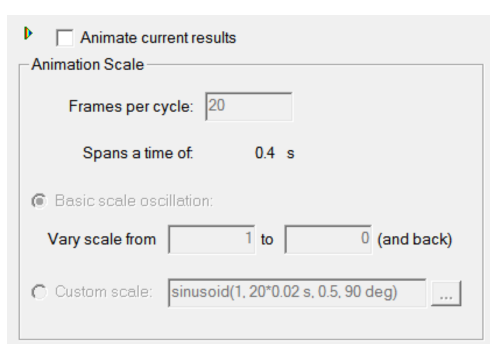
1. *On the* ***Player control panel****,* ***click*** *on* ***drop-down arrow*** *next to the Play symbol and* ***choose Animate FEA****. The playback button will change to the colored FEA animation button*



2. ***Click*** *on the* ***Animate FEA button*** *to create the FEA animation.*

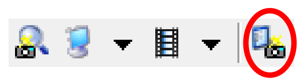
   Note: The animation may take some time to develop, as it create a deformation display for each frame of the animation. After it complete all frames, the animation should playback smoothly and quickly.

Tip: The playback rate for the animation can be changed under FEA Display, Animation in the SimWise main settings

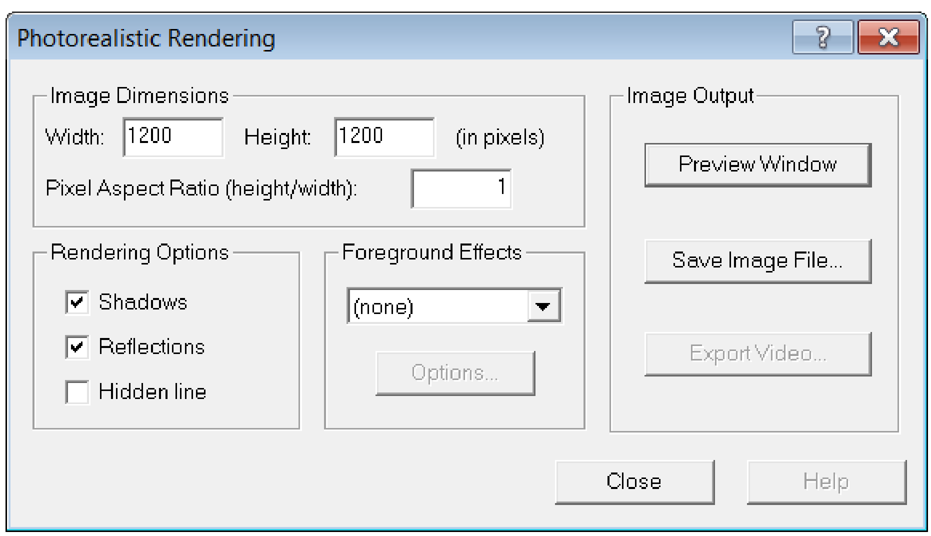


Export Photorealistic Image

1. ***Click** on the **Render Settings** button located on the **Render toolbar***



2. *In the Rendering d-box, **click** on the **Preview Window** button to preview the size of the animation area that will be saved to video. The default (256 x 256) will most likely be too small. Use a size that best fits your model view area. Simply close the preview window, type in a new Width and Height and click on Preview again. Do this until you have an acceptable view size and then select **Save Image File**.*

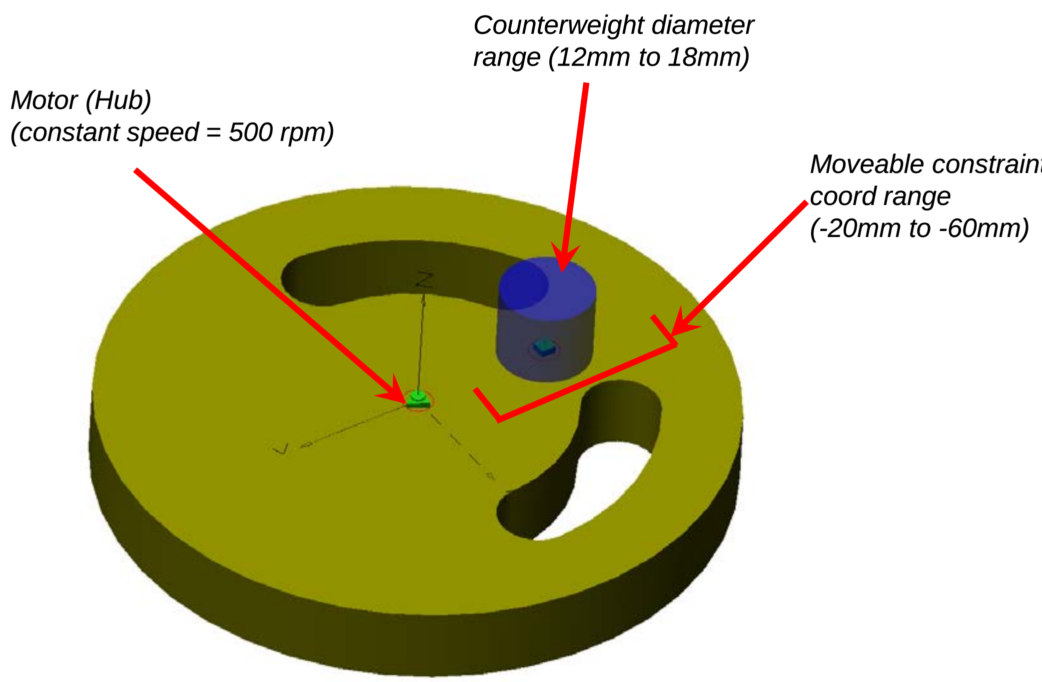


End of Exercise

Exercise – Vibration Reduction

Simulation Objectives:

- Optimize counterweight diameter and radial location
- Minimize the vibration



Features Covered:

- Paramaterizing variables
- Setting up and running an Optimization study
- Applied loads
- Bonding
- Meshing

Table of Contents

- *Topics are hyperlinked. Click on a topic name to link directly to that topic page*

- *Click on the icon at the bottom of any page to return here to the Table of Contents (TOC)*

Open the Example Model

1. ***Start SimWise***

2. *Select **File, Open** and **locate** the file called "SWOpt Tutorial-Disc Balancing.wm3D".*

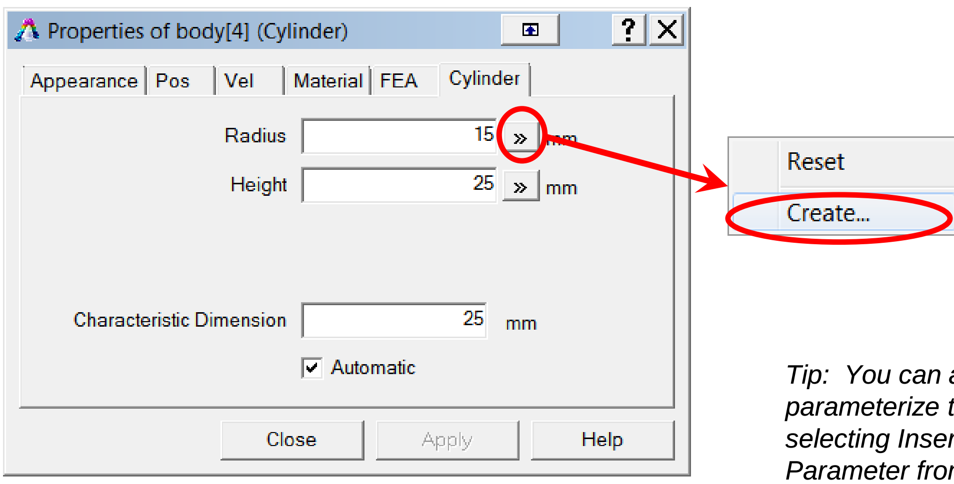
The model consists of a disc with a motor at the center hub and a counterweight attached to the disc with a rigid joint. The motor speed has already been defined as a constant 500 rpm.

Parameterize the Diameter

1. ***Double-click** the **blue counterweight** to **open** the body Properties*

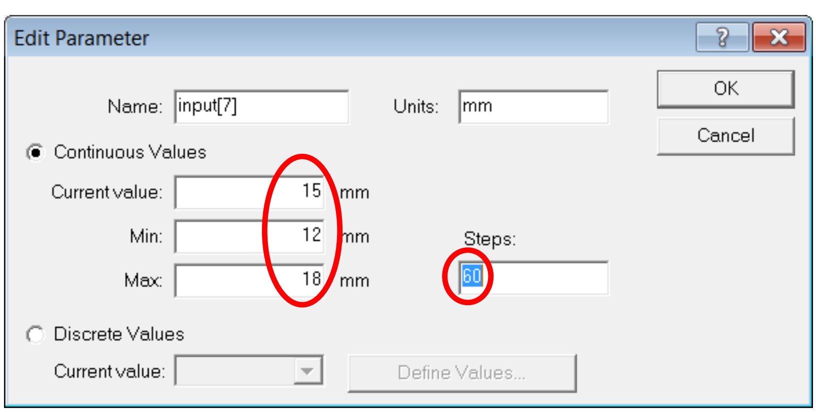
2. ***Select** the **Cylinder tab***

3. ***Select** the **formula button** next to the **Radius field** and **choose** **Create***



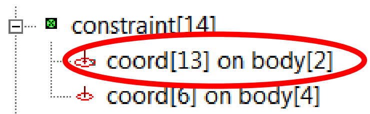
Tip: You can also parameterize this value by selecting Insert → Control → Parameter from the main menu.

4. *In the **Parameter d-box**, **enter the values** as shown*



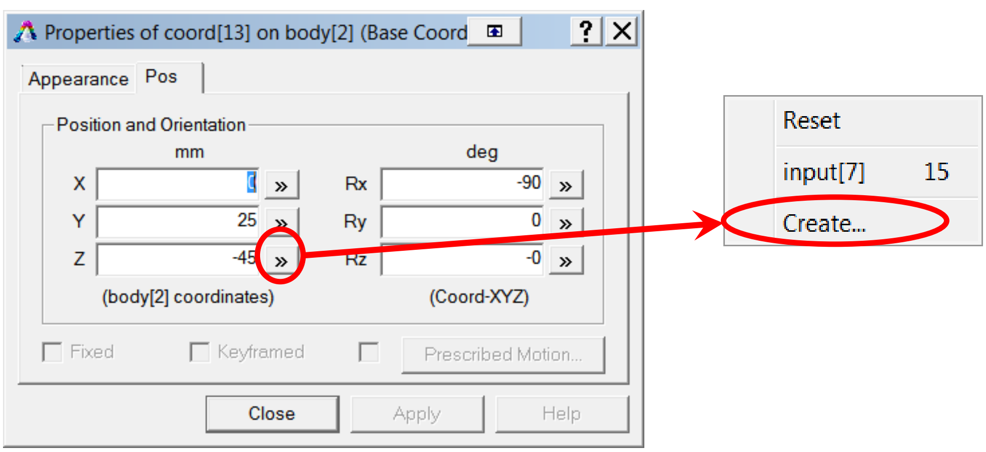
Parameterize the Location

1. ***Expand*** ***Constraint[14]*** *and* ***double-click*** *on* ***Coord[13]*** *to open its* ***Properties d-box***

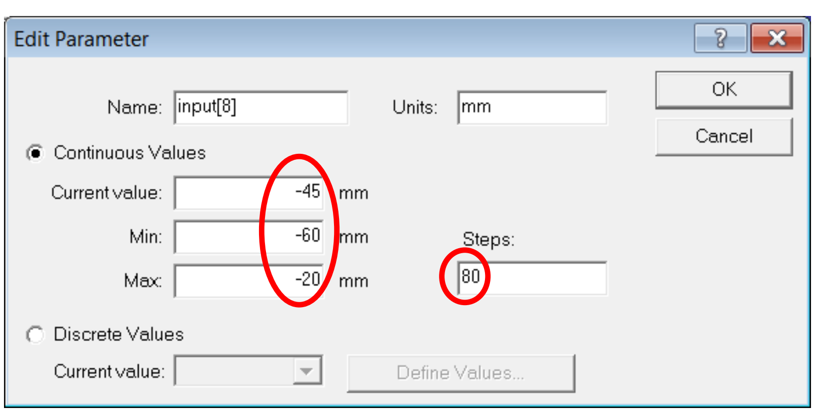


2. ***Select*** *the* ***Pos tab***

3. ***Select*** *the* ***formula button*** *next to the* ***Z field*** *and* ***choose Create***

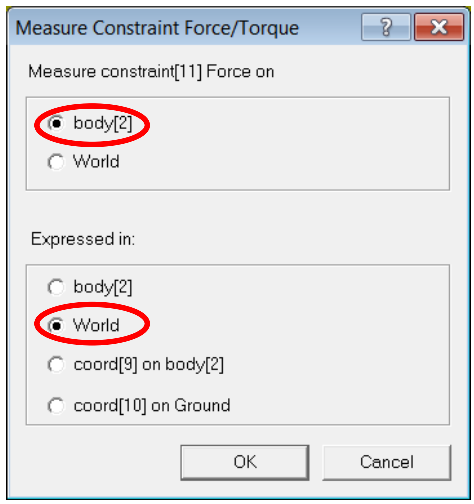


4. *In the* ***Parameter d-box***, ***enter the values*** *as shown, then select* ***OK*** *and then* ***Close***



Create a Meter

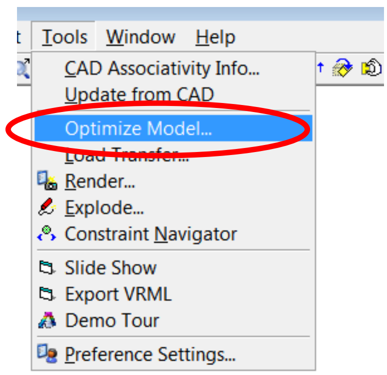
1. ***Click** on **Constraint[11]**, **Insert**, **Meter**, **Constraint Force** and **select** **body[2]** and **World**, as shown*



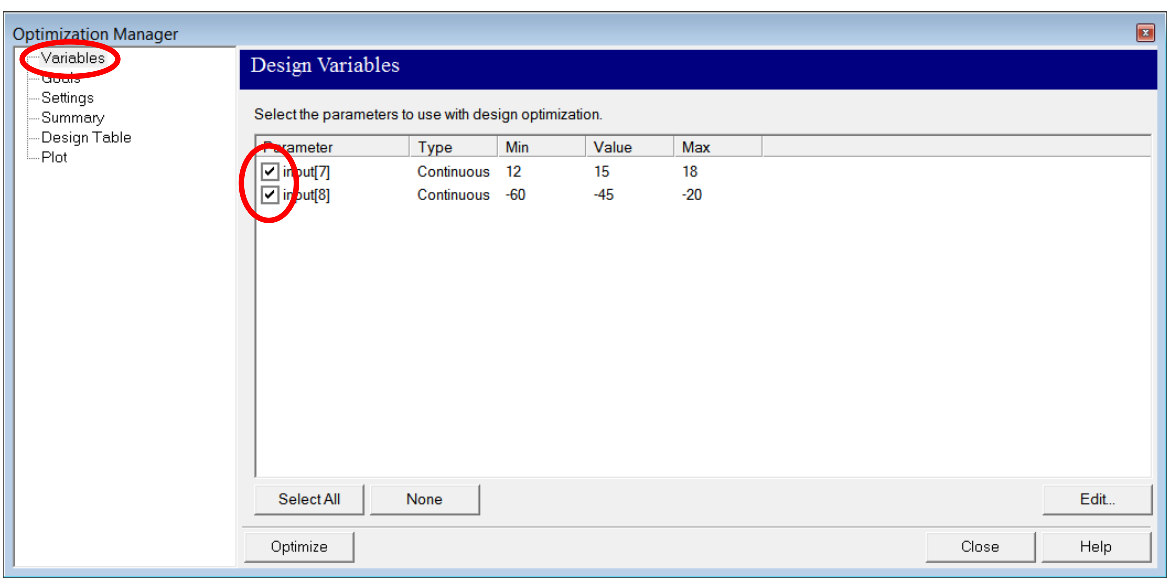
This meter will be used in defining both the Goal and constraints for the optimization

The Optimization Manager ***(Specify Input Parameters)***

1. *On the main menu, **click** on **Tools**, **Optimize Model***



2. ***Click** on the **checkboxes** next to each input parameter in the **Design Variables** window.*



The Optimization Manager ***(Add Objectives)***

1. ***Click*** *on* ***Goals***

We will define the following two objectives:

Objective 1:
Meter = Constraint[11] force meter
Formula = Fx
Value = Maximum
Type = Minimize

Objective 2:
Meter = Constraint[11] force meter
Formula = Fy
Value = Maximum
Type = Minimize

Note: The optimization will work to minimize the maximum Fx and Fy forces, relative to our constraints (defined next)

2. *Under* ***Object Type****,* ***select Meters***

3. *Under* ***Object****,* ***select Constraint[11]***

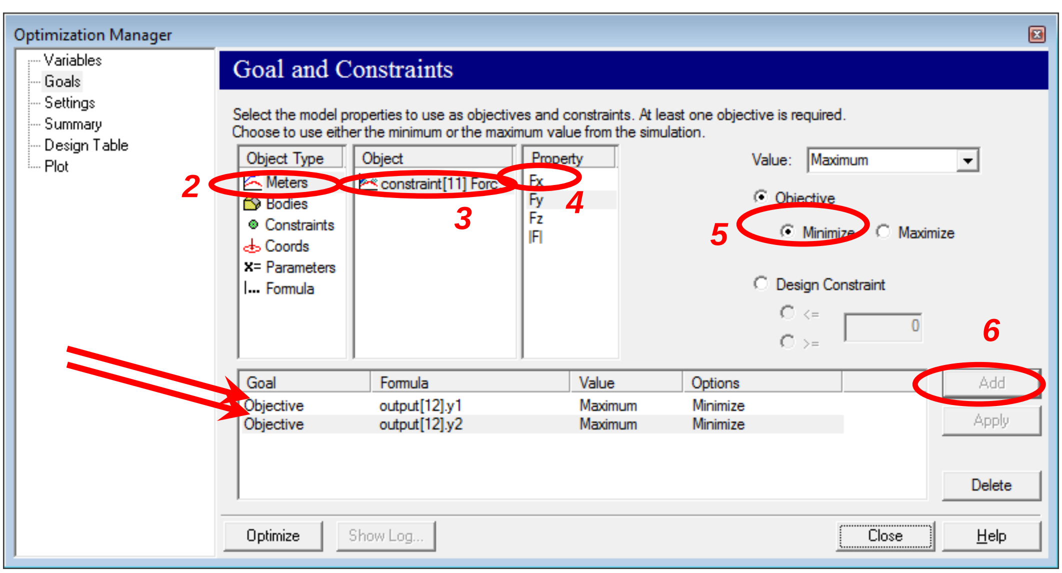
4. *Under* ***Property****,* ***select Fx***

Note: We do not want to use |F| (magnitude) because the magnitude will also take into account the vertical force due to the weight of the disc, which we are not interested in since this loading is perpendicular to the direction of vibration

5. ***Click*** *on* ***Objective,*** *accept defaults of* ***Maximum*** *and* ***Minimize***

6. ***Select Add***

7. ***Repeat steps 2-6,*** *except in step 4****, select Fy***



The Optimization Manager *(Add Constraints)*

1. Click** on **Goals

We will define the following two constraints:

Constraint 1:
Meter = Constraint[11] force meter
Formula = Fx
Value = Maximum
Type = <=
Limit = 1

Constraint 2:
Meter = Constraint[11] force meter
Formula = Fy
Value = Maximum
Type = <=
Limit = 1

Note: The optimization will work to find all feasible solutions that result in vibration loads less than or equal to 1N

*2. Under **Object Type**, **select Meters***

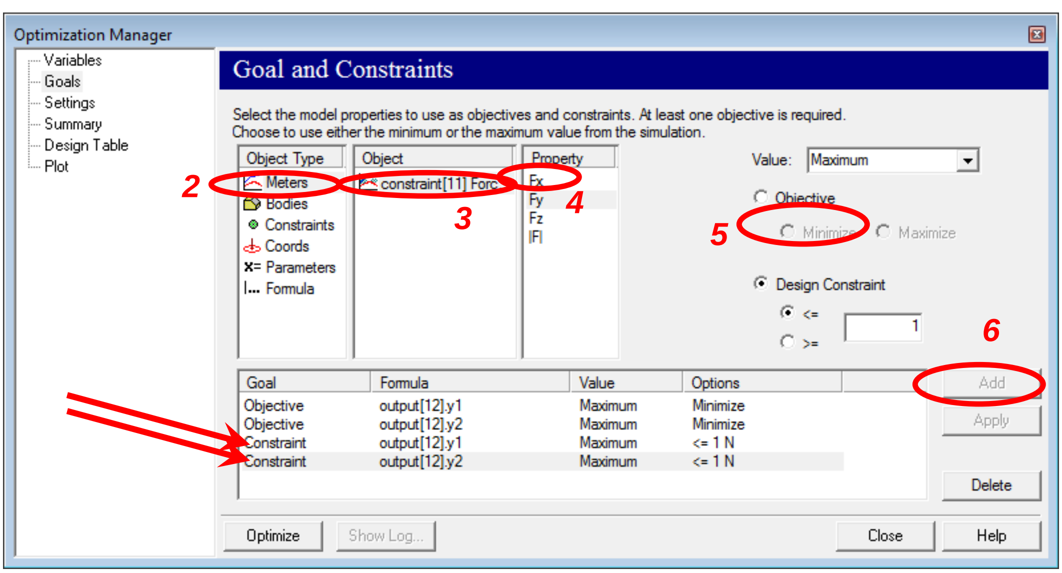
*3. Under **Object**, **select Constraint[11]***

*4. Under **Property**, **select Fx***

***5. Click** on **Design Constraint, click** on **<=** and **enter** 1 in the field*

6. Select Add

7. Repeat steps 2-6,** except in step 4**, select Fy

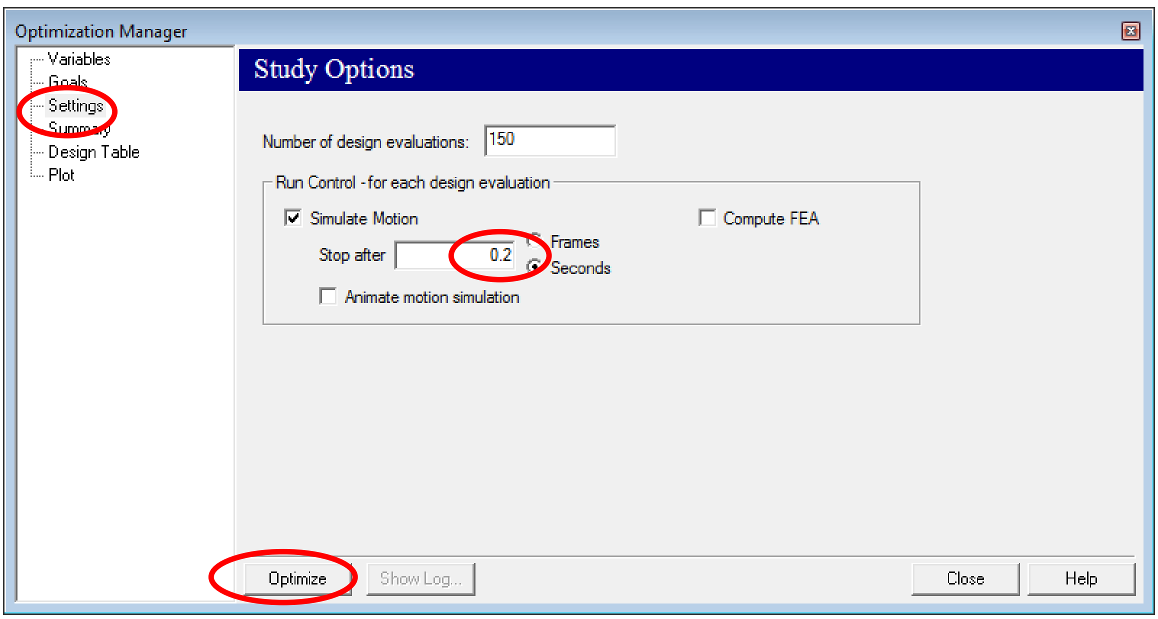


The Optimization Manager ***(Settings)***

1. ***Click*** *on* ***Settings***

2. *In the* ***Stop after*** *field,* ***click*** *on* ***Seconds*** *and* ***enter 0.2*** *as the stop criteria*

Note: Since the disc is rotating at a constant 500 rpm, it will make approximately 1-1/2 revolutions in 0.2 seconds. This is enough to capture the forces needed to evaluate one iteration (or evaluation) of the optimization study

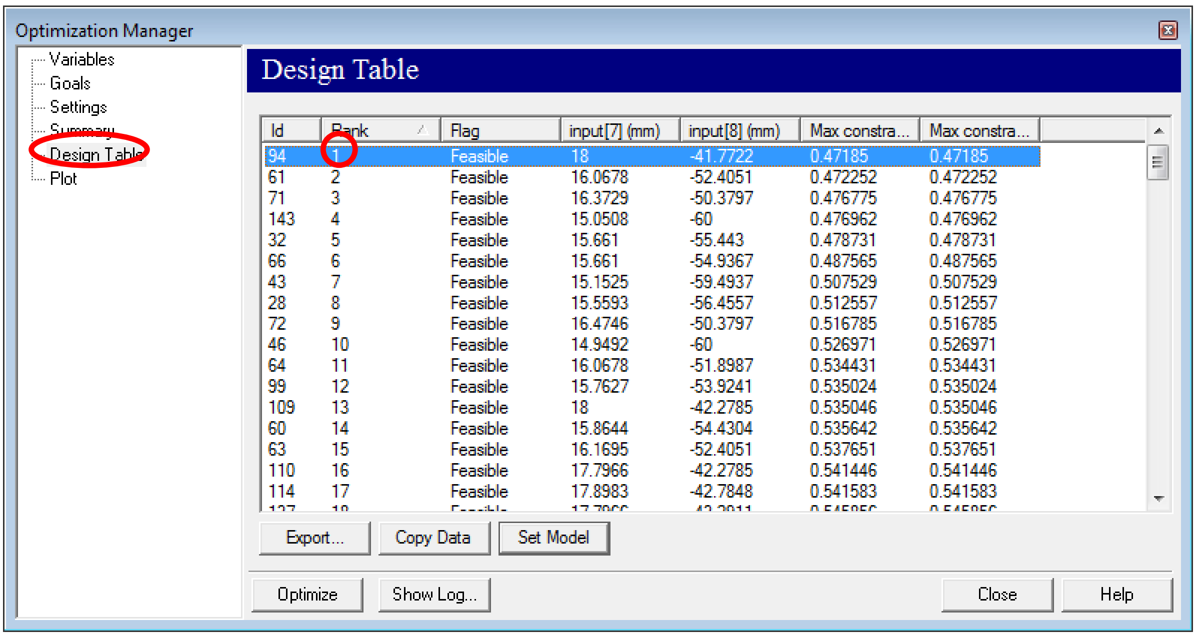


3. ***Select*** *the* ***Optimize*** *button to begin the optimization study*

Note: While the optimization is running, all features associated with parameterized values will move rapidly and change position in the graphics window, as the optimization cycles through all possibilities set forth in the parameter ranges

Results

1. ***Click** on **Design Table** in the Optimization Manager to see the list of each evaluation and their results*



2. ***Select** any **Rank** listing and **choose Set Model** to display the suggested design configuration. All parameter values will be permanently set to the values used for that iteration.*

Summary

Although the highest rankings are listed as feasible in the sense of achieving the goal, the actual parameter values used to achieve those designs may not be entirely feasible from a manufacturing standpoint, considering they are listed out to 4 decimal places. Once the best design is identified, it is a good idea to round the values of the parameters to placeholders feasible for manufacturing, apply those values directly to each parameter and then perform a single run of the simulation using those values to obtain exact results. For example, in looking at Rank 5, one might use 15.6 mm for the counterweight diameter, -55.4 mm for the mounting position of the counterweight and do a single normal run using those values.

Another alternative to finding feasible designs to attainable manufacturing tolerances is to use the "Discrete" option when defining parameters. Discrete options can work toward optimization using a list of pre-assigned variable values.

End of Exercise