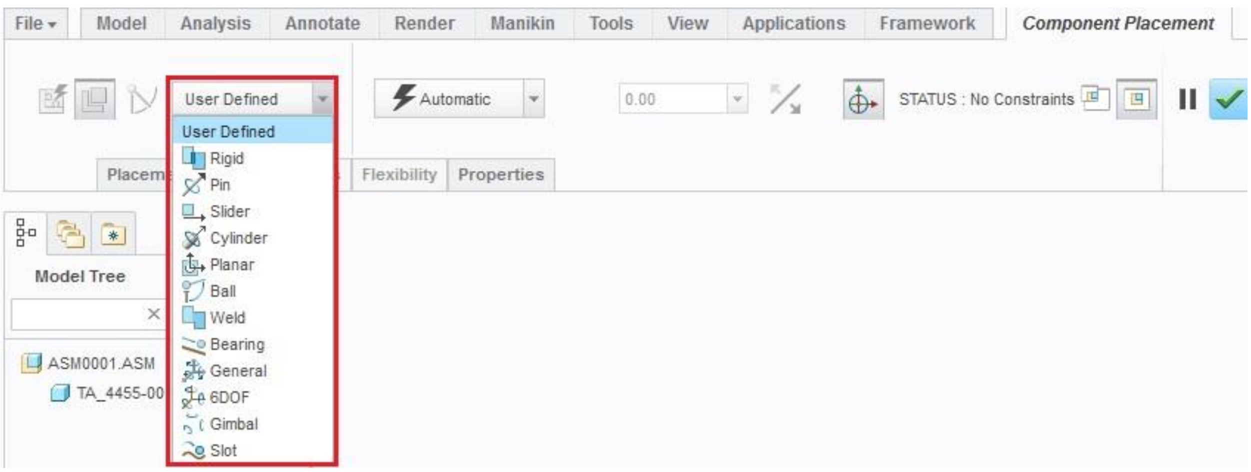
Animations in Creo 3.0 ME170

Part I. Introduction & Outline

Animations provide useful demonstrations and analyses of a mechanism's motion. This document will present two ways to create a motion animation in Creo: one method utilizes snapshots, the second method uses servo motors in Creo's Mechanism package. The snapshot method is arguably the easier of the two. However, the latter method creates a smoother animation.

In order to create and capture a mechanism's motion in an animation, we must first define connections that permit the movement (translation or rotation) of parts. This connection definition occurs in the assembly. The electronic dongle was assembled entirely with rigid constraints, which do not permit any motion. However, Cero can accommodate numerous motion connections. A list of Creo's motion connections is shown below.



The pin connection and slider connection are the two simplest and most common mechanism connections. This document provides instructions for creating each connection. The ME 170 term project instructions state that you must have at least one motion axis in your final project. Your motion axis does not have to be a pin-type or a slider-type. These two connection instructions are provided solely as a reference.

The outline of this document is as follows:

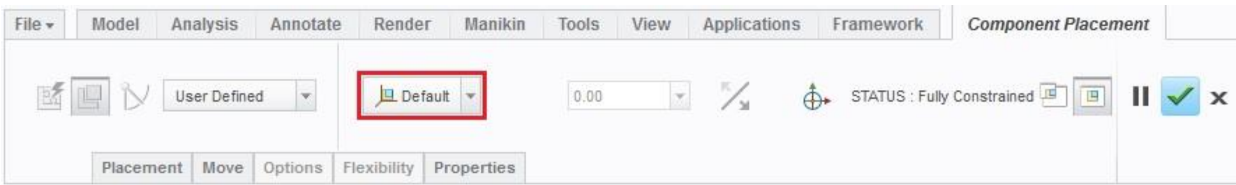
- Pin Connection Definition
- Slider Connection Definition
- Animation with Snapshots
- Animation using a Servo Motor

Additional information on joint connections, Creo's mechanism capabilities, animations, and motion analyses can be found in the "Joint Connection, Mechanism, & Animation Reference" document on the ME 170 course website.

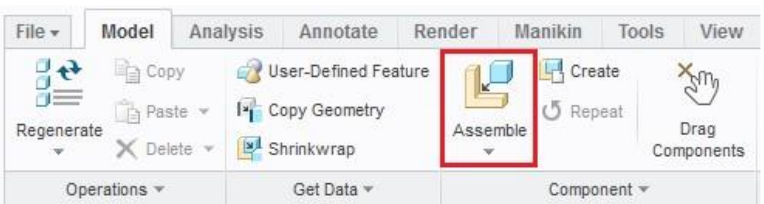
Part II. Pin Connection Definition

1) Assemble and fully constrain the first part(s) of your assembly.

   A "Default" constraint is recommended for the first component of every assembly. This fixes the first part in place. Before defining a joint or motion connection, a fixed, fully-constrained part is needed to serve as a fixed reference.

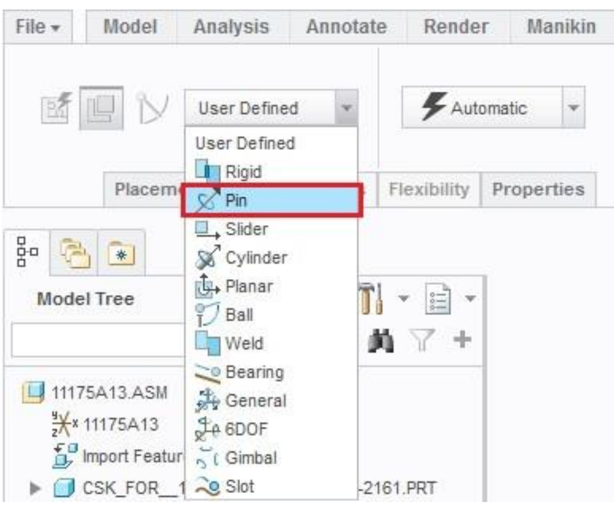


2) Click the "Assemble" button to assemble a new component.

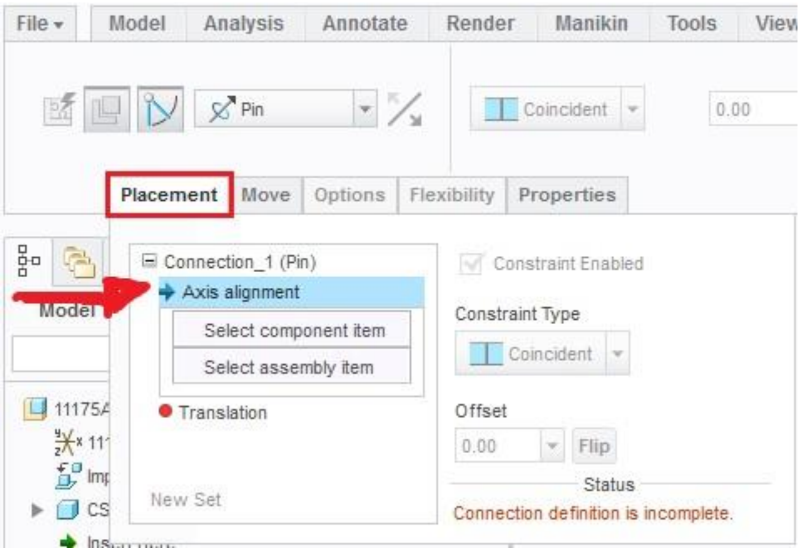


3) Select the component you would like to assemble and click "Open."

4) Select a "Pin" connection from the motion connection drop-down menu.

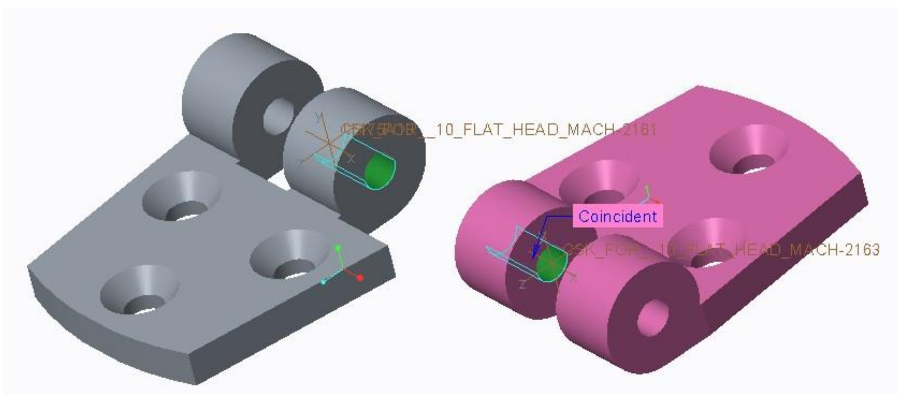


5) Open the "Placement" tab in the assembly ribbon. This will help guide you through the definition of the connection (image below).



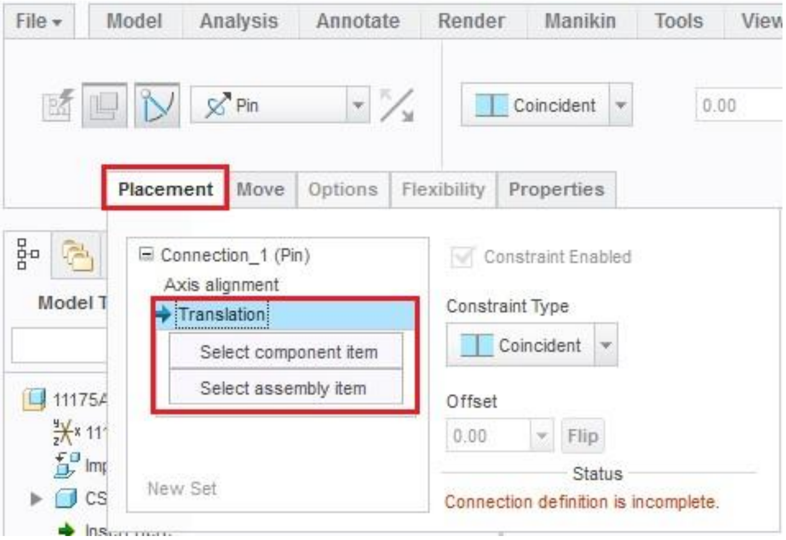
6) Notice that the connection definition is asking for "Axis alignment" references. Consequently, we will first select the axes to align.

7) Select the rotation axis for the pin connection. Note that a cylindrical surface, the axis of a coordinate system, and an actual axis can all be used to define a rotation axis. In the image below, the inner cylindrical surfaces of the hinge have been selected.

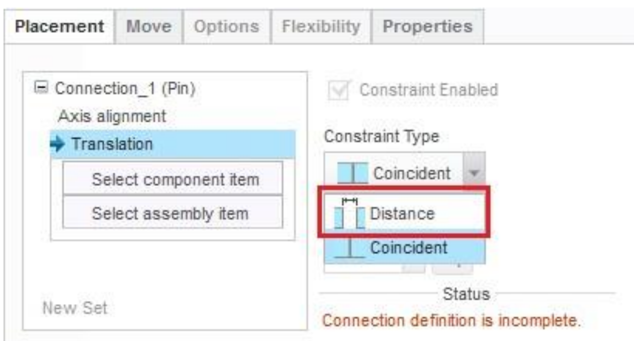


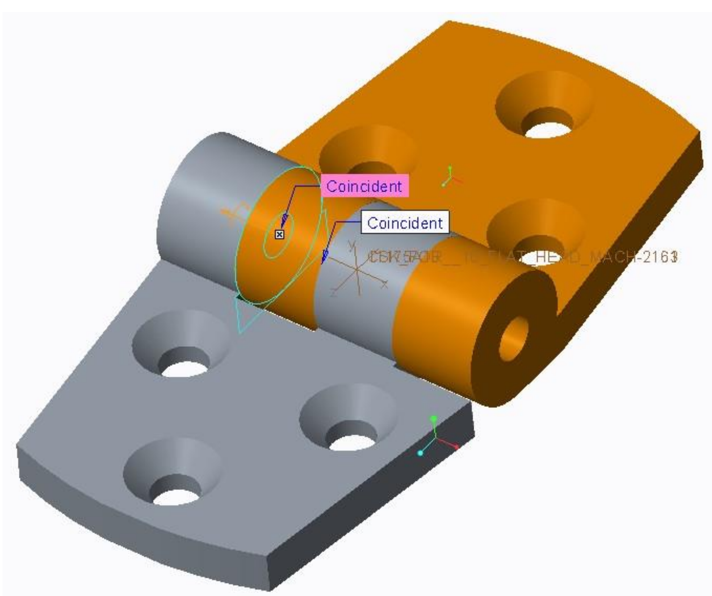
8) After selecting the rotation axes, go back to the "Placement" tab in the assembly ribbon (image below). Notice that Creo now requests that we select "Translation" constraints.

   The pin connection in Creo does not permit translational motion. It only permits rotation around one axis. It is a single degree-of-freedom (1 DOF) connection. The translation constraint is used to align the rotating components along the rotation axis.



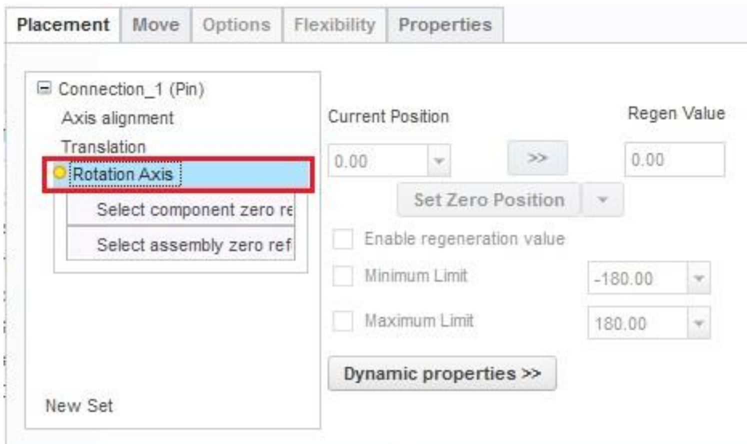
9) Select two planes or surfaces normal to the motion axis to serve as translational alignment constraints. Note that an offset or distance value can be specified for the translation constraint, so the planes or surfaces do not need to be coincident. In the hinge example, two mating surfaces have been selected to be coincident.





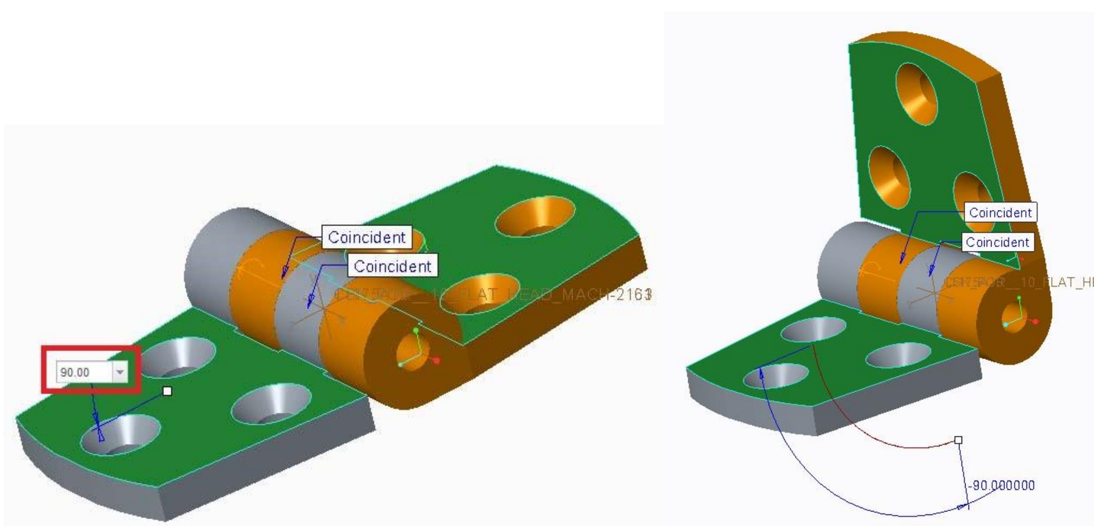
10) Go back to the "Placement" tab on the assembly ribbon. Click on "Rotation Axis" to define additional useful parameters for the hinge.

The parameters under the "Rotation Axis" heading govern the regeneration value and minimum, and maximum limits of motion for the hinge. If these parameters are not defined, Creo will permit unrealistic motion of the component such as that shown in the image below.

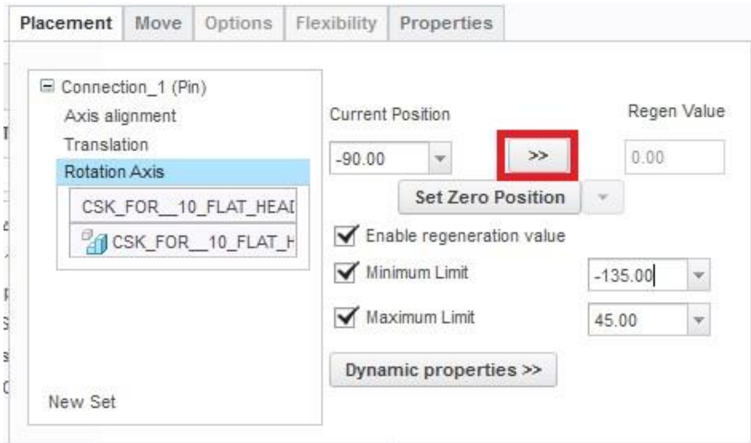


11) Select two planes to serve as angular references for the regen value and max and min limits of motion. The two upper surfaces of the hinge have been selected as shown in the image below.

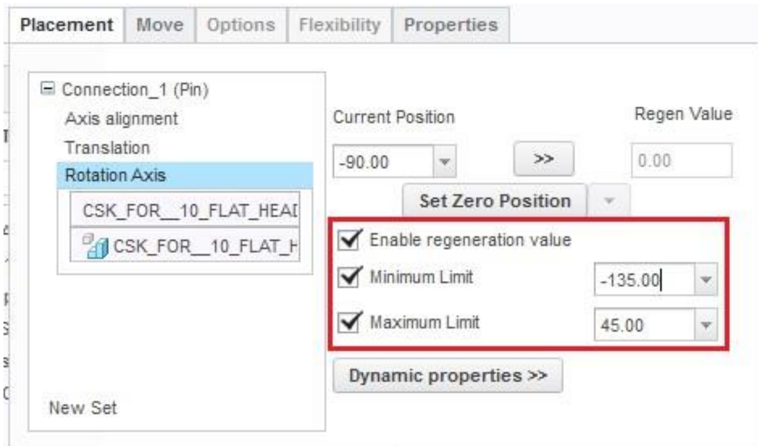
12) Double click the angle in the model window and specify a regeneration value. Click the middle mouse button to confirm the new value. An angle of "90.00" has be specified in the image below. However, "90" was insufficient. The angle had to be adjusted to "-90" to yield a physically-feasible orientation.



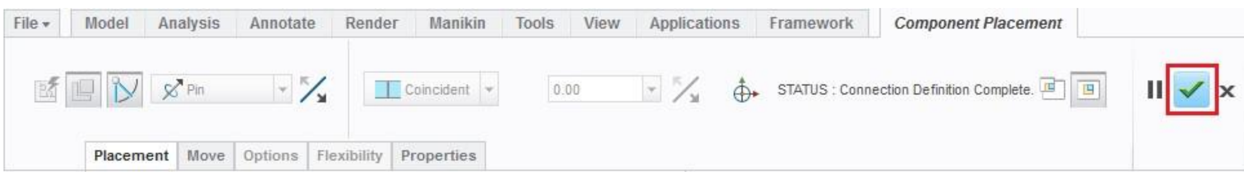
13) Click the ">>" button in the "Placement" tab to set the current position as the regeneration position (image below).



14) Click the "Enable regeneration value" checkbox to enable a regeneration value. If necessary (if required by your design), specify minimum and maximum limits of motion and check the checkboxes to enable these constraints.



15) Click the green checkmark to confirm your joint connection.

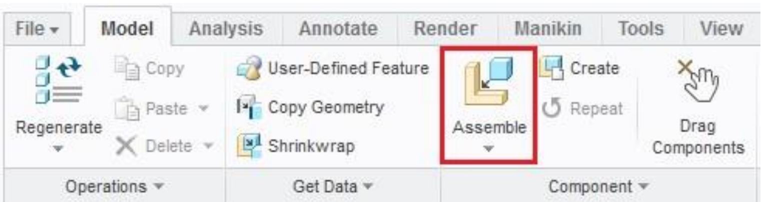


16) Move your newly assembled component by holding CTRL+ALT and clicking and dragging the component. Notice that the motion will stop at the defined minimum and maximum limits. If you have defined a regeneration value, click regenerate or type CTRL+G to regenerate the model to return the model to its base state.

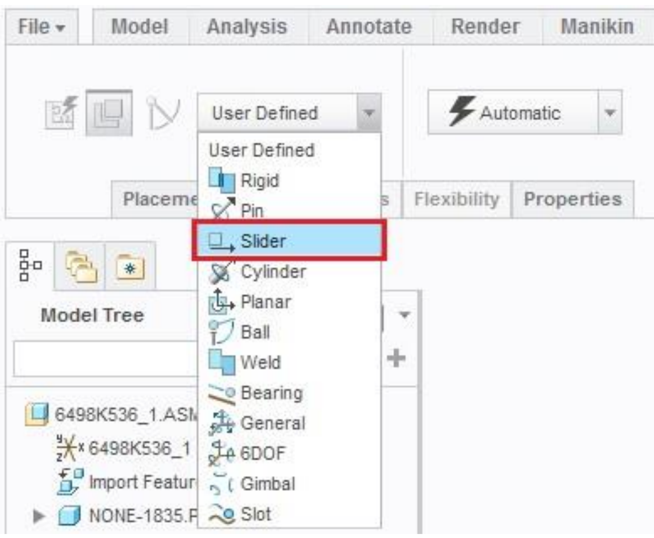
17) **NOTE:** Pay particular attention to your assembly references when assembling components. Additional components which are assembled with references to moving components, may themselves become moving components.

Part III. Slider Connection Definition

1) Click the "Assemble" button to assemble a new component.

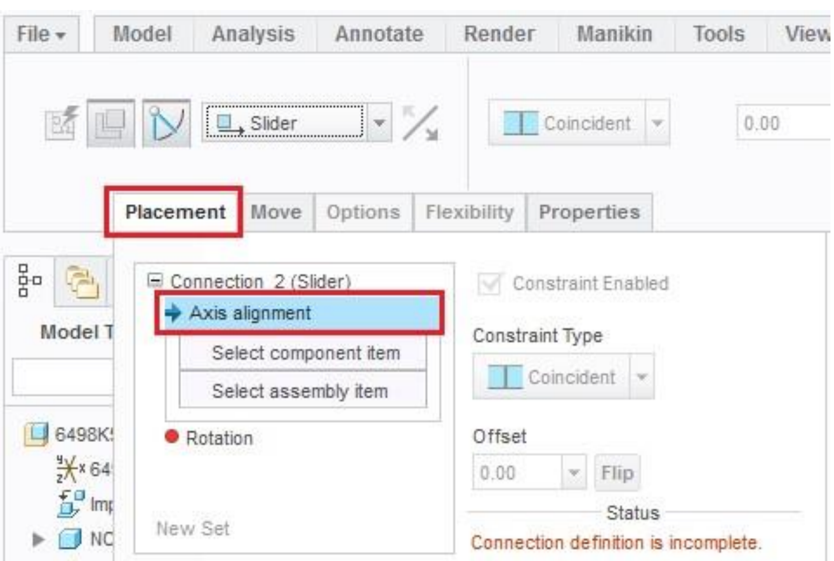


2) Select the component you would like to assemble and click "Open."

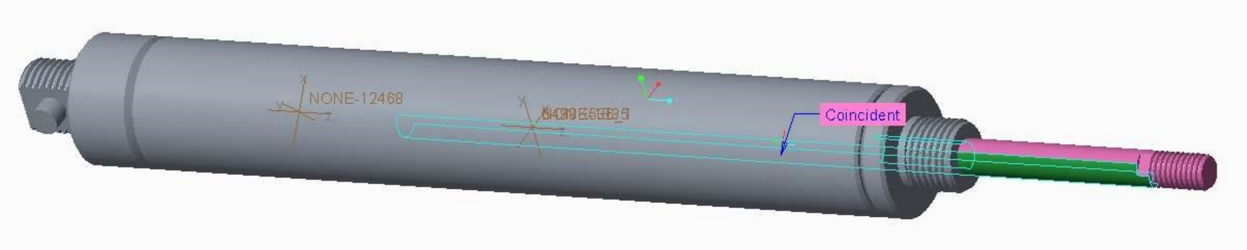


3) Open the "Placement" tab in the assembly ribbon. This will help guide you through the definition of the connection.

Notice that the connection definition is asking for "Axis alignment" definition. Consequently, we will first select the motion axes to align.

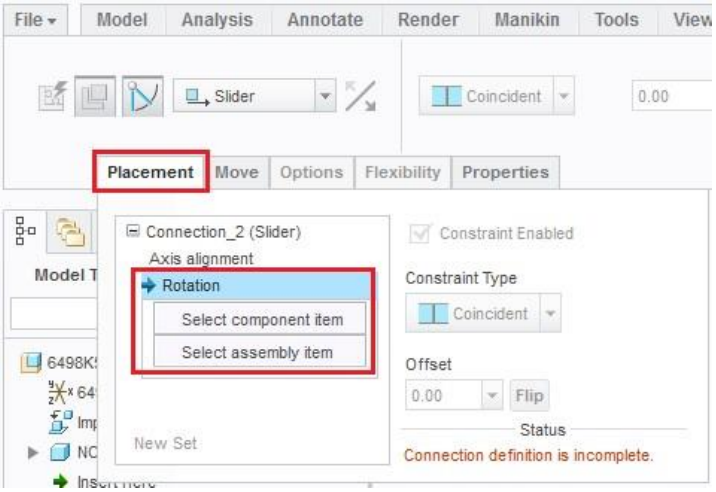


4) Select the motion axis for the slider connection. This is the axis along which the two components will slide. Note that a cylindrical surface, the axis of a coordinate system, and an actual axis can all be used to define a motion axis. In the image below, the inner cylindrical surface of the pneumatic cylinder and the outer cylindrical surface of the shaft have been selected.



5) After selecting the rotation axes, go back to the "Placement" tab in the assembly ribbon. Notice that Creo now requests that we select "Rotation" constraints.

   The slider connection in Creo does not permit rotational motion. It only permits translation along one axis. It is a single degree-of-freedom (1 DOF) connection. The rotation constraint is used to align the translating components along the translation axis.

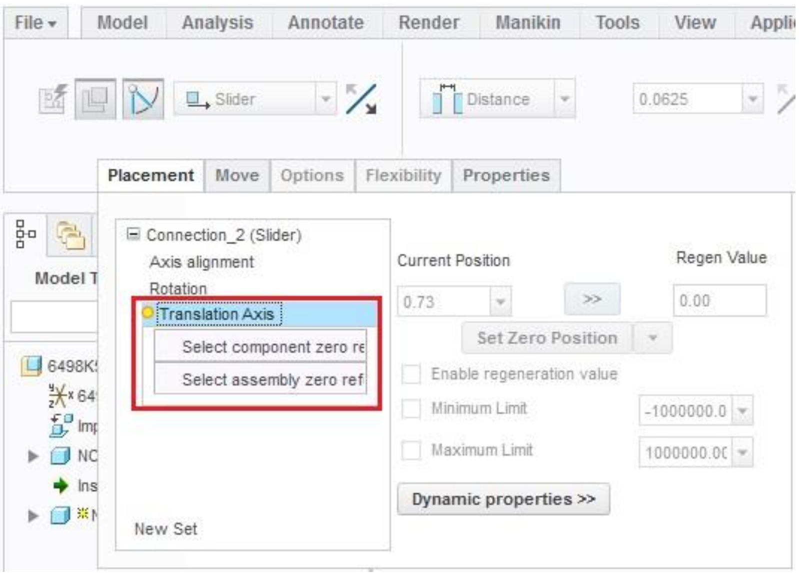


6) Select two planes or surfaces parallel to the translation axis to serve as rotational alignment constraints (image below). These two planes or surfaces will be constrained to be oriented parallel to one another. An offset angle cannot be specified for this rotation constraint. In this example, flat surfaces on the piston and the cylinder have been selected.

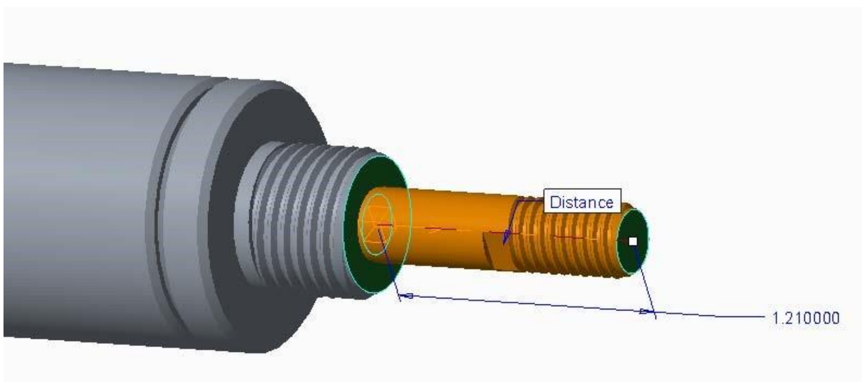
   Note that this is not a particularly realistic constraint for a pneumatic cylinder. The piston of a pneumatic cylinder is typically free to rotate about its translation axis. Consequently, Creo's "Cylinder" motion connection would be more appropriate. However, for the purposes of this example, the pneumatic cylinder will suffice.

7) Go back to the "Placement" tab on the assembly ribbon. Click on "Translation Axis" to define additional useful parameters for the slider connection.

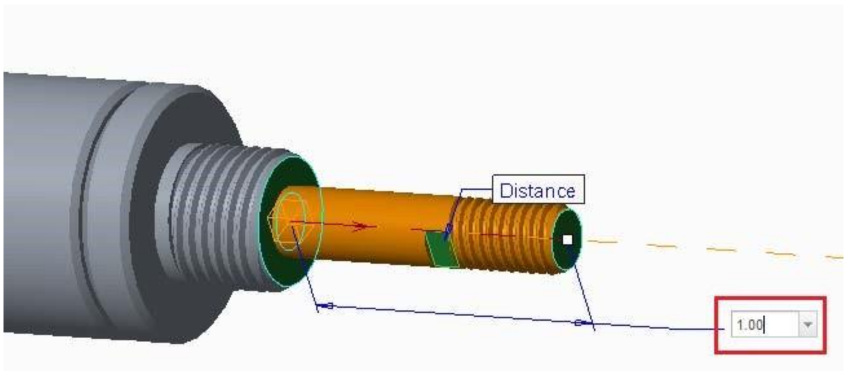
The parameters under the "Translation Axis" heading govern the regeneration value and minimum, and maximum limits of motion for the slider connection. If these parameters are not defined, Creo will permit unrealistic motion of the component such as a piston that moves beyond its mechanical stops in the pneumatic cylinder.



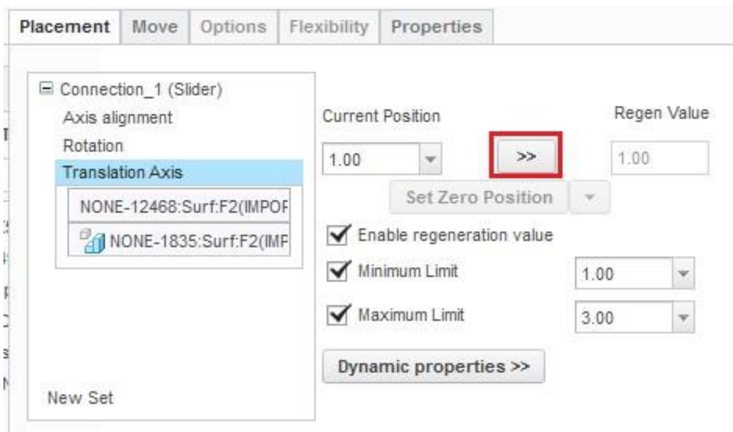
8) Select two planes or surfaces to serve as translation references for the regen value and max and min limits of translational motion. These two planes or surfaces should be <u>normal to the axis of motion</u>. The top surfaces of the piston and cylinder have been selected for this example.



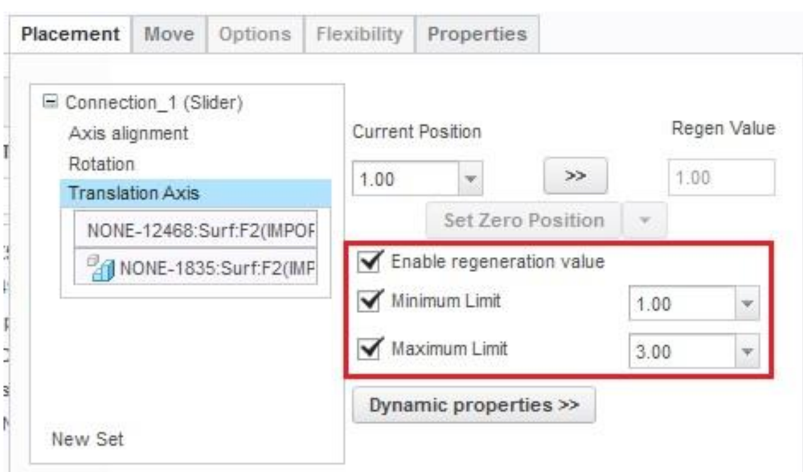
9) Double click the distance in the model window and specify a regeneration value. Click the middle mouse button to confirm the new value. An distance of "1.00" has be specified in the image below.



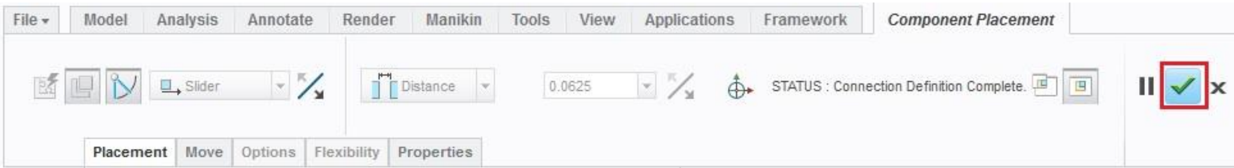
10) Click the ">>" button in the "Placement" tab to set the current position as the regeneration position.



11) Click the "Enable regeneration value" checkbox to enable a regeneration value. If necessary (if required by your design), specify minimum and maximum limits of motion and check the checkboxes to enable these constraints.



12) Click the green checkmark to confirm your joint connection.



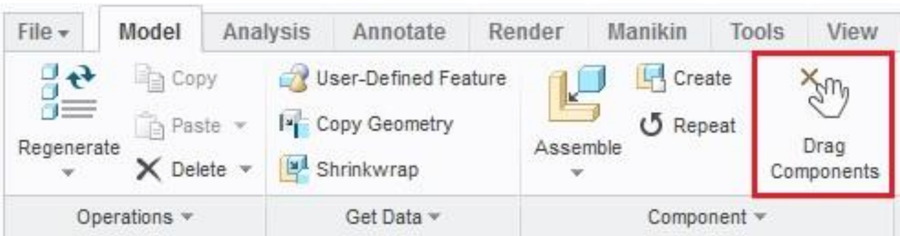
13) Move your newly assembled component by holding CTRL+ALT and clicking and dragging the component. Notice that the motion will stop at the defined minimum and maximum limits. If you have defined a regeneration value, click regenerate or type CTRL+G to regenerate the model to return the model to its base state.

NOTE: Pay particular attention to your assembly references when assembling components. Additional components which are assembled with references to moving components, may themselves become moving components.

Part IV. Animation with Snapshots

1) In order to create an animation with snapshots, we must first create the snapshots that we will use for the animation. Open the assembly that you would like to animate. Your assembly should be fully assembled, appropriately colored, and the appropriate motion connections should be defined.

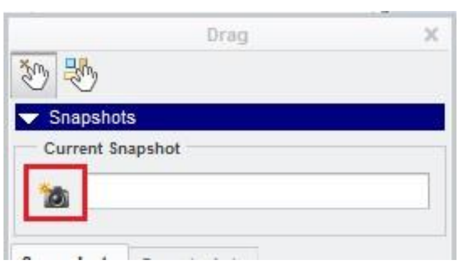
2) Navigate to the "Model" ribbon and click the "Drag Components" icon.



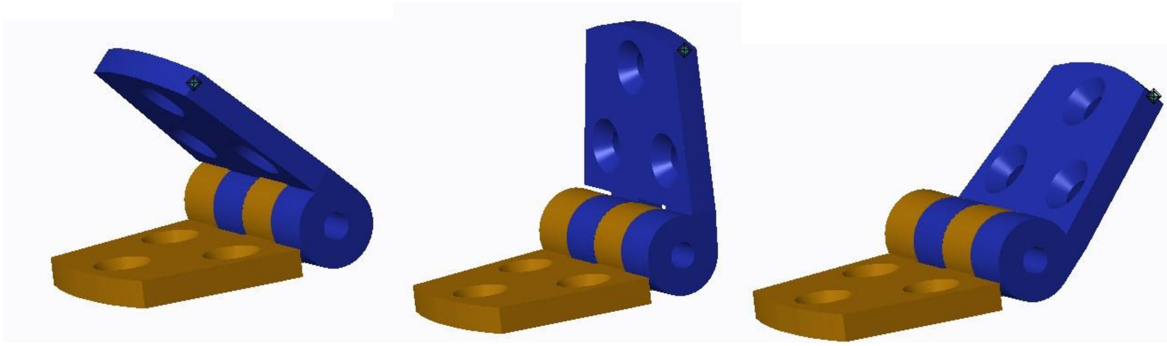
3) In the new window on the right side of the screen, click the arrow next to "Snapshots" to expose the Snapshots window.

4) Click and drag your component to the first position of your animation. Note that you will only be able to record 10 frames for your snapshot animation. It is possible to create multiple animations if one does not adequately demonstrate the functionality of your part.

5) Once your component is in position, click the "Take a Snapshot" button in the "Drag" window (image below).



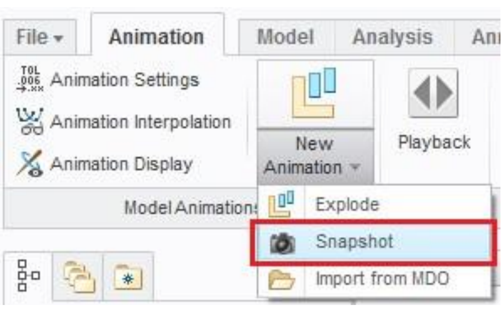
6) Click and drag your assembly to a new orientation. Take multiple snapshots of your component to use for your animation. You can delete snapshots by right clicking on them and selecting "Delete" in the "Drag" window.



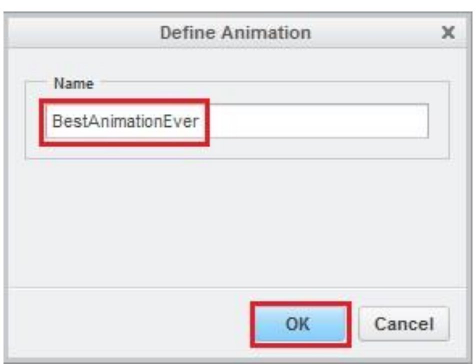
7) Once you have enough snapshots (10 max for the particular animation), close the "Drag" window.

8) To begin an animation, navigate to the "Applications" tab and click the "Animation" icon.

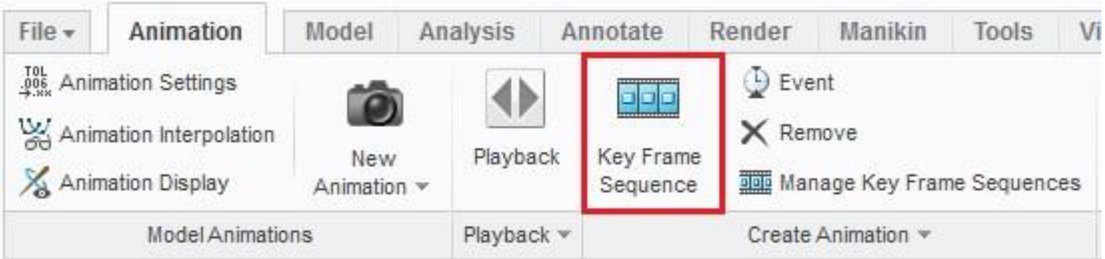
9) Click the "New Animation" drop-down menu and select "Snapshot".



10) Specify a name for your animation and click "OK" (image below).

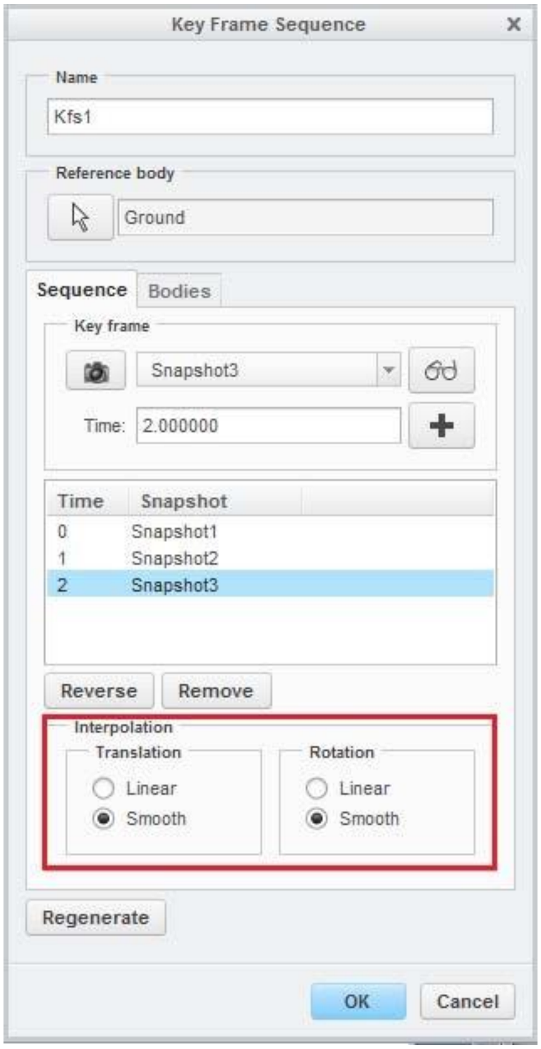


11) Click the "Key Frame Sequence" icon on the top ribbon to create a new frame sequence.



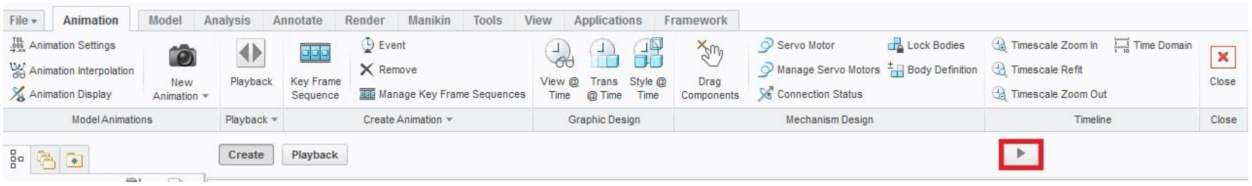
12) Populate your animation with snapshots by selecting a snapshot, specifying the time, and clicking the "+" button. Note that the time will increment automatically by 1 second each time you add a snapshot to the animation. The one-second-per-snapshot baseline is good for timing.

13) Change the interpolation buttons from "Linear" to "Smooth" to increase the production quality of your animation (image below).

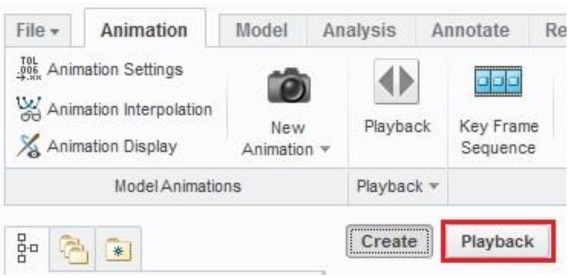


14) Click "OK" to close the "Key Frame Sequence" window.

15) Press the "Generate" button, shown below, to generate your animation.

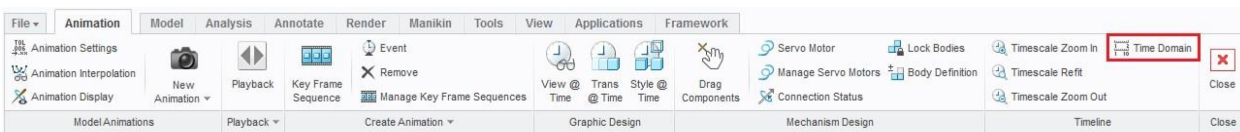


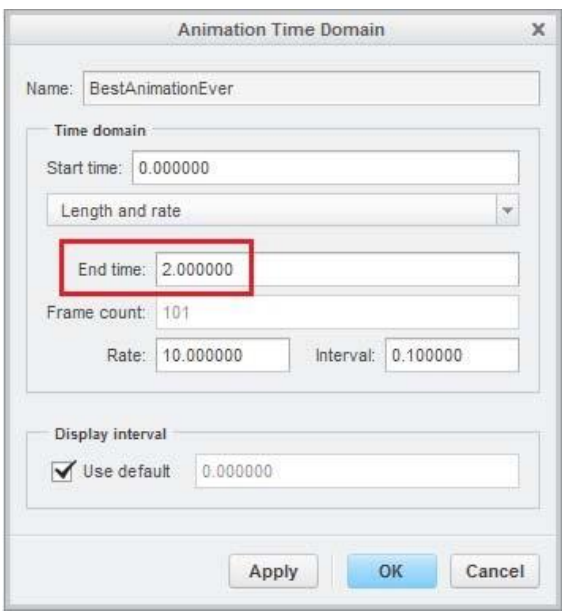
16) Click "Playback" to go to the animation playback controls.



17) Play your animation a few times to confirm it is correct.

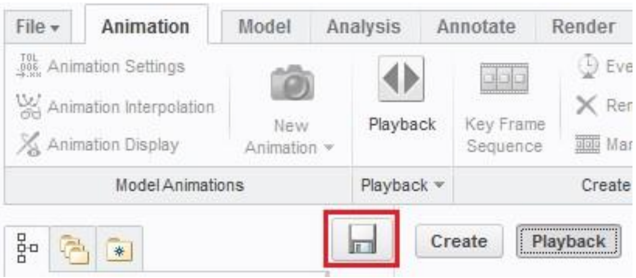
18) If you would like to shorten the length of your animation, navigate back to the "Create" controls. Click the "Time Domain" button on the top ribbon, and specify a end time.



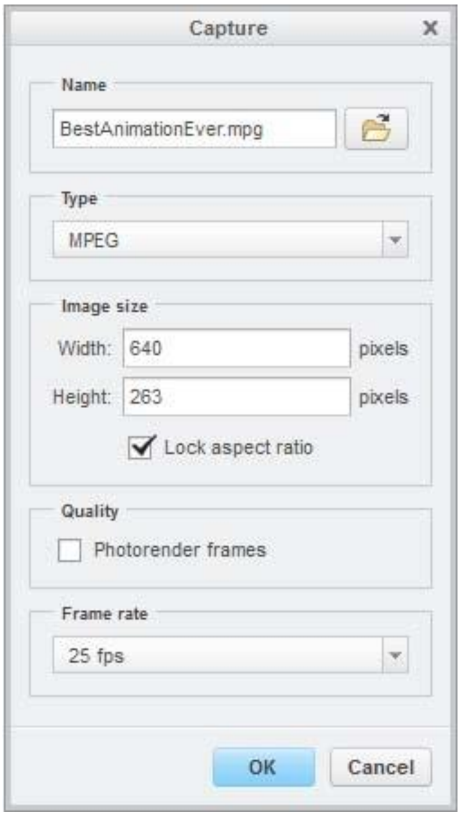


19) Click "OK" to cloes the "Animation Time Domain" box. Click the "Generate" button to rebuild your animation. Navigate back to the "Playback" controls to watch your animation.

20) To save the animation, first save your assembly in Creo (File > Save or CTRL+S). Click the "Capture" buttons in the "Playback" controls area.



21) Specify a name and parameters for your video file (image below).



22) Your video file will be saved to your working directory.

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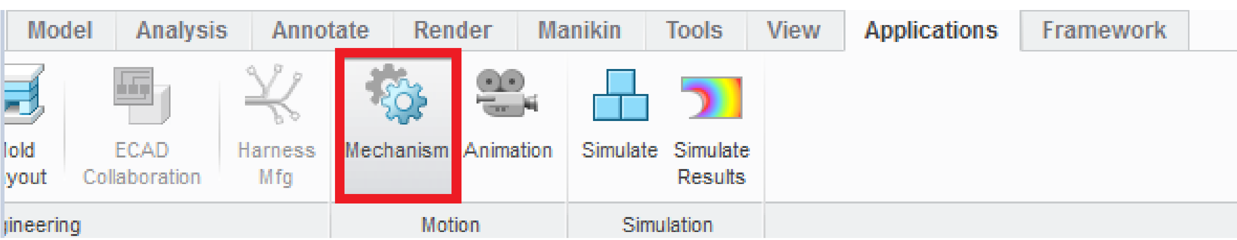
Part V. Creo Mechanism

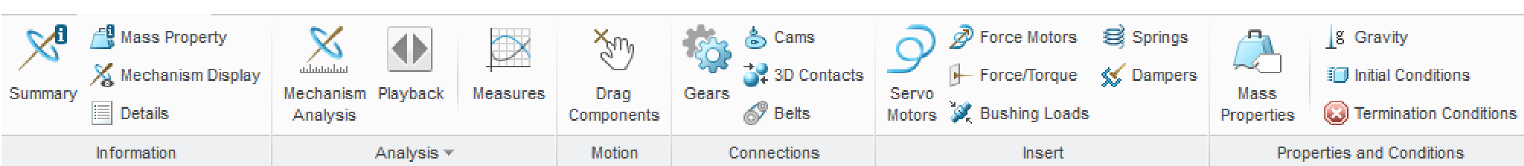
This plugin allows the user to run motion analysis on assemblies. Instead of using 3rd party software, an assembly can be directly imported into Mechanism and analyzed. Features like springs and dampers can have real life properties assigned to make the model as accurate as possible. Along with showing you the actual animation of your assembly, Creo Mechanism can output position, velocity, acceleration and even force data for any point in the assembly.

The following pages will go over the basics of Creo Mechanism to help you constrain and successfully animate your model. Parts must be constrained together with mechanism connections before a full analysis can be run.

Start by opening up the Mechanism plugin.



You will see that there are a lot of options to change when setting up a mechanism. The Mechanism ribbon, shown below, is where all of the setup and analysis is configured.



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Cams

Cam connections can be made by selecting the **Cams** button located on the Mechanism ribbon. The **Cam Follower Connection Definition** dialog box will pop up with the Cam 1 page selected.

I. The **Cam 1** page is used to setup the cam and the **Cam 2** page is used to setup the follower.

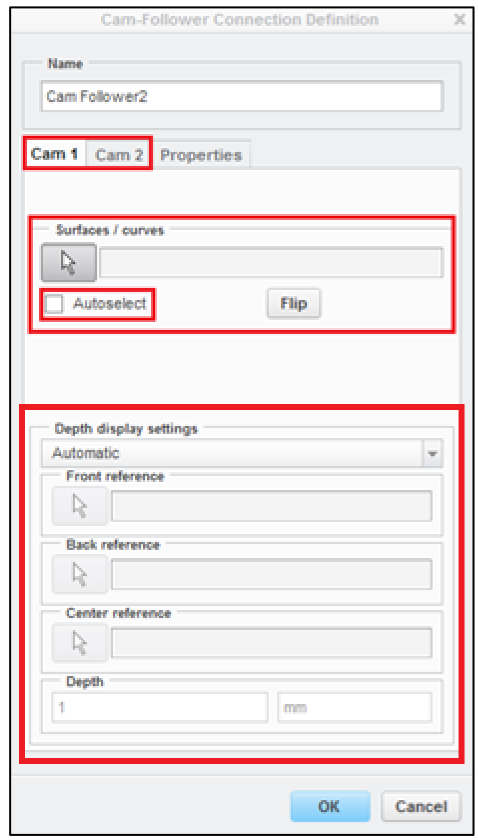
II. The **Surfaces/Curves** selection box is used to select the contact surfaces between the cam and the follower.

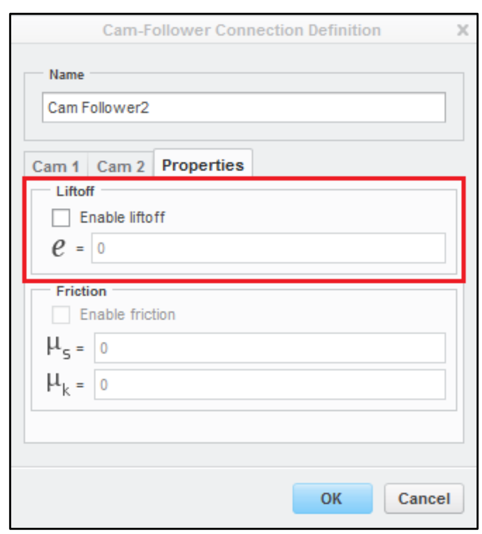
   a. Creo will automatically find tangent surfaces to the selection when **Autoselect** is used.

III. Creo will require extra references for flat surfaces. These references will need to be set using the **Depth Display Settings** located below the initial surface selection.

   a. A flat surface will require a front and back reference point or vertex.

IV. The **Properties** tab allows friction and liftoff to be enabled. If liftoff is selected, the follower can lift off from the cam during its range of motion.





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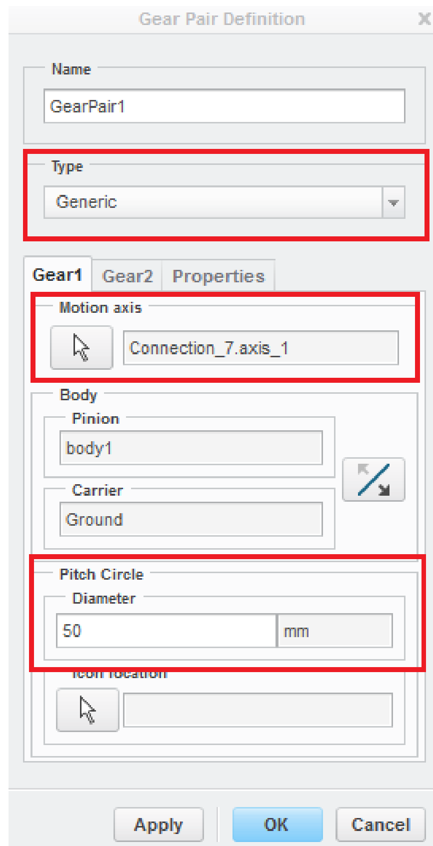
Gears – Generic

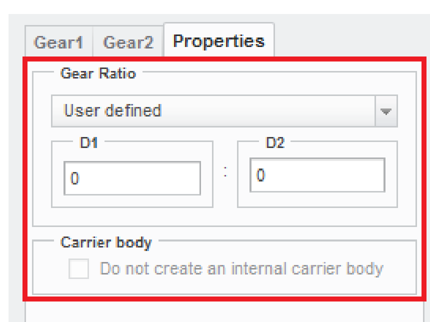
Gear connections can be made by selecting the **Gears** button located on the Mechanism ribbon. The **Gear Pair Definition** dialog box will pop up with the Gear1 page selected.



I. The <u>**Type**</u> dropdown box is used to select the gear pair type. The options are a generic gear pair, spur gears, bevel gears, worm gears and rack and pinion set.

II. The **Generic** setting can be used for straight gear sets along with the following three pairs of joint axes
   a. Rotation-Rotation
   b. Rotation-Translation
   c. Translation-Translation

III. The **Motion Axis** selection box is used to select the axis of rotation for each gear.
   a. Enter the **Pitch Circle Diameter** for if the joint axis is rotation
   b. Enter the **Gear Ratio** under the **Properties** tab if the motion is translational



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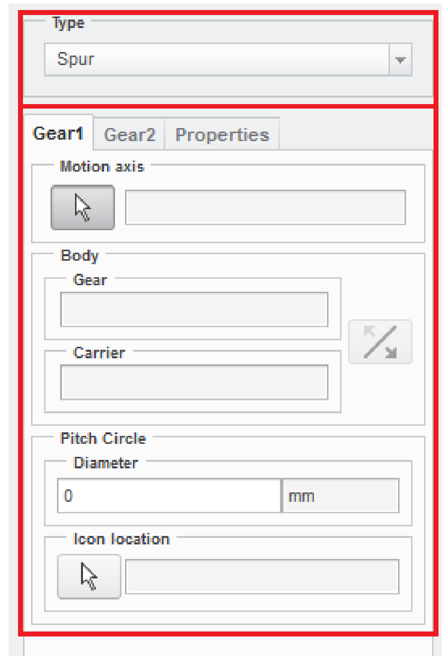
Gears – Spur

Gear connections can be made by selecting the **Gears** button located on the Mechanism ribbon. The **Gear Pair Definition** dialog box will pop up with the Gear1 page selected.



I. The **Type** dropdown box is used to select the gear pair type. The options are a generic gear pair, spur gears, bevel gears, worm gears and rack and pinion set.

II. The **Spur** setting is used when the pressure angle and helix angle are required. The joint axes must also be parallel.

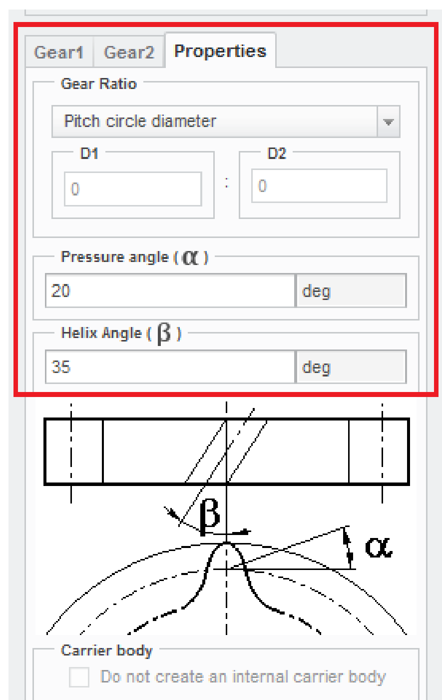
III. The **Motion Axis** selection box is used to select the axis of rotation for each gear.

   a. Enter the **Pitch Circle Diameter** for Gear 1

   b. The Pitch Circle Diameter will automatically calculate for gear 2

IV. The **Helix Angle** and **Pressure Angle** are set using the properties page

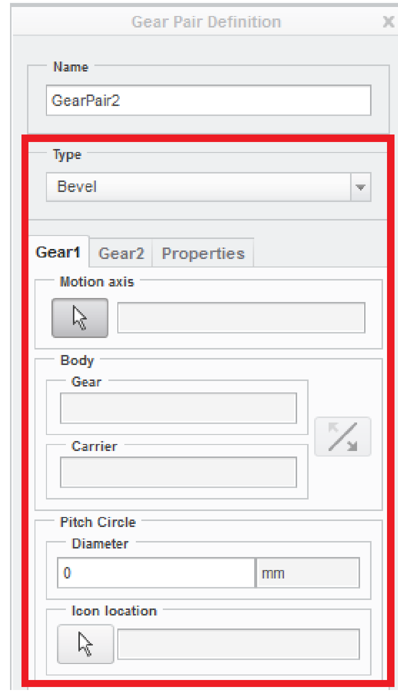
V. A preview of the gear is created



Animations in Creo 3.0 ME170

Gears – Bevel

Gear connections can be made by selecting the **Gears** button located on the Mechanism ribbon. The **Gear Pair Definition** dialog box will pop up with the Gear1 page selected.



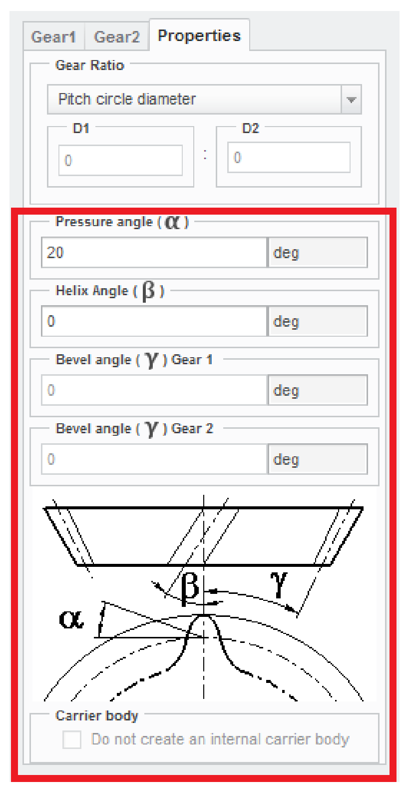
I. The **Type** dropdown box is used to select the gear pair type. The options are a generic gear pair, spur gears, bevel gears, worm gears and rack and pinion set.

II. The **Bevel** setting is used when the center axes of both gears intersect. Gear teeth for bevel gears are conically shaped.

III. The **Motion Axis** selection box is used to select the axis of rotation for each gear.
   a. Enter the **Pitch Circle Diameter** for Gear 1
   b. The **Pitch Circle Diameter** will automatically calculate for gear 2

IV. The **Helix Angle** and **Pressure Angle** are set using the properties page

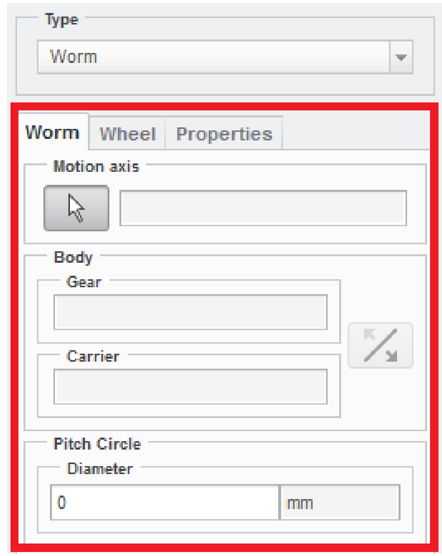
V. A preview of the gear is created



Animations in Creo 3.0 ME170

Gears – Worm

Gear connections can be made by selecting the Gears button located on the Mechanism ribbon. The **Gear Pair Definition** dialog box will pop up with the Gear1 page selected.



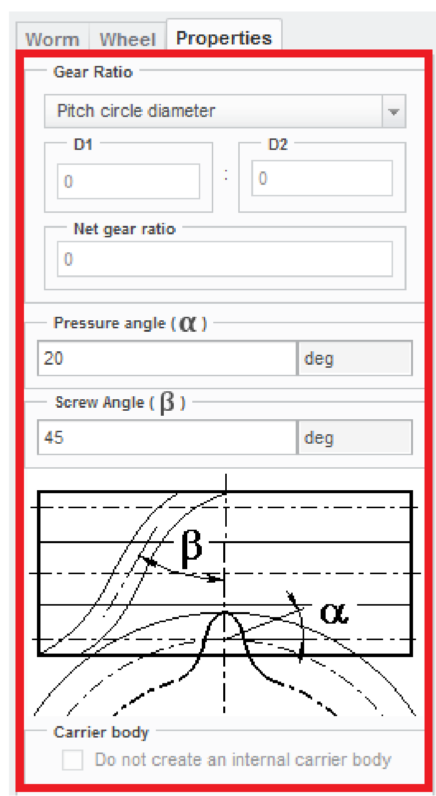
I. The **Type** dropdown box is used to select the gear pair type. The options are a generic gear pair, spur gears, bevel gears, worm gears and rack and pinion set.

II. The **Worm** setting is used when an assembly contains a worm screw gear driving a worm wheel gear.

III. The **Motion Axis** selection box is used to select the axis of rotation for the worm and wheel.
   a. Enter the **Pitch Circle Diameter** for the worm.
   b. The **Pitch Circle Diameter** will automatically calculate for the wheel

IV. The **Helix Angle** and **Pressure Angle** are set using the properties page

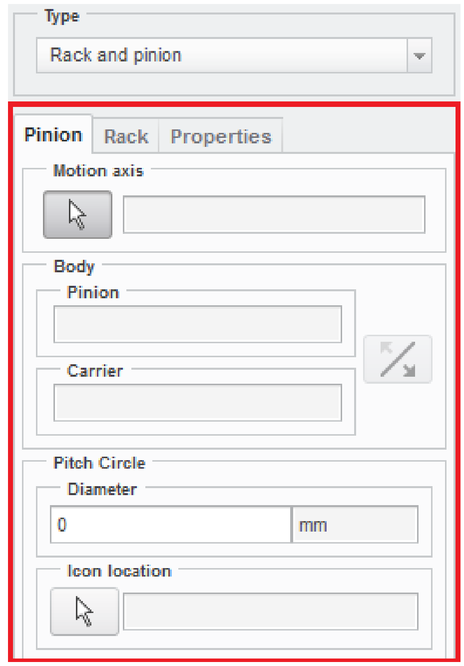
V. A preview of the gear is created



Animations in Creo 3.0 **ME170**

Gears – Rack and Pinion

Gear connections can be made by selecting the Gears button located on the Mechanism ribbon. The **Gear Pair Definition** dialog box will pop up with the Gear1 page selected.



I. The **Type** dropdown box is used to select the gear pair type. The options are a generic gear pair, spur gears, bevel gears, worm gears and rack and pinion set.

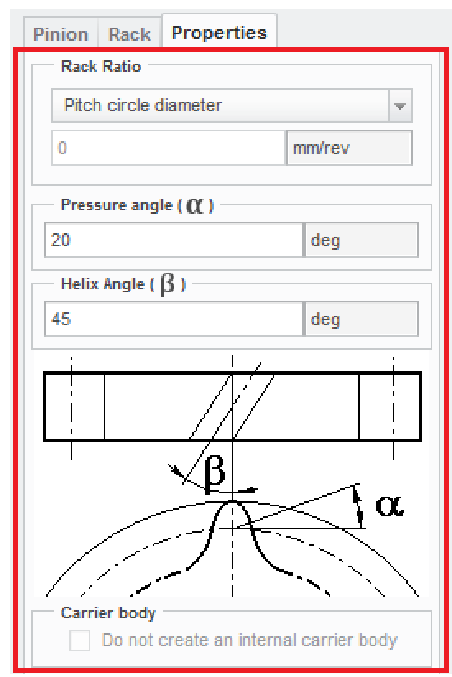
II. The **Rack and Pinion** option is chosen when a normal gear is meshed together with a gear constrained as a slider.

III. The **Motion Axis** selection box is used to select the axis of rotation for the pinion and the translation axis for the rack.

   a. Enter the **Pitch Circle Diameter** for the pinion.

IV. The **Helix Angle** and **Pressure Angle** are set using the properties page

V. A preview of the gear is created



Animations in Creo 3.0 ME170

Springs – Extension/Compression

Spring can be added by selecting the **Springs** button located on the Mechanism ribbon.

I. To add a spring, click on the **Springs** button on the **Insert** tab. Make sure you turn on **Point Display** to show datum points. A spring requires a top and bottom datum point or a translational axis to constrain it.

II. For the point method, select the point corresponding to the top of the spring and then hold CTRL and select the bottom point of the spring to complete it

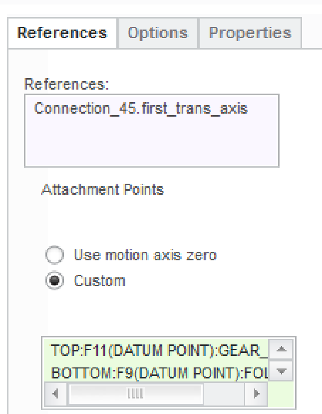
III. When creating a spring along a translation axis, the spring length must be set using the references tab.

   a. Points or vertices can be used to set the spring length by selecting the **Custom** option in references.

   b. The spring length can also be constrained to start at the initial, or zero, position of the translational axis by selecting the **Use motion axis zero** option

IV. The spring definition ribbon allows the user to change the spring stiffness, the stiffness units and the free length.

V. The visual diameter of the spring can be changed by using the **Adjust Icon Diameter** under the options tab in the spring definition ribbon. This has no physical effects.



Animations in Creo 3.0 ME170

Springs – Torsion

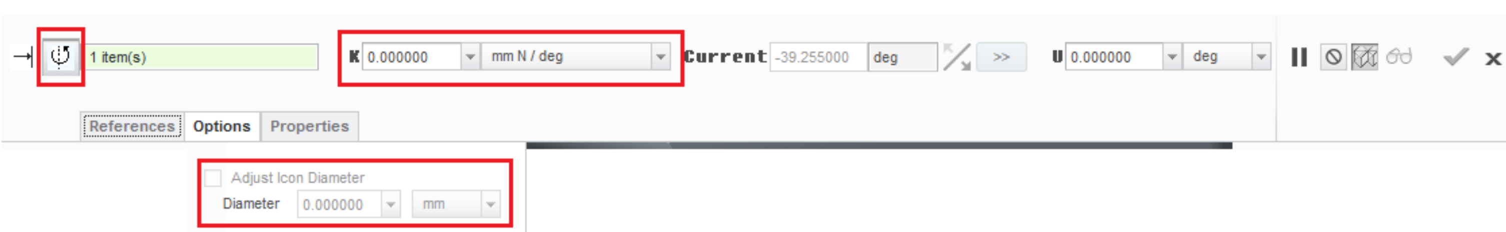
Springs can be added by selecting the **Springs** button located on the Mechanism ribbon.

I. To add a torsion spring, click on the **Springs** icon on the **Insert** tab. Then click on the **Torsion Spring** button to the left of the green reference box. A torsion spring requires a rotation axis to constrain it.

II. When creating a spring along a rotational axis, the spring angle must be set using the references tab.
   a. Planar surfaces can be used to set the spring angle by selecting the **Custom** option in references.
   b. The spring angle can also be constrained to start at the initial, or zero, position of the rotational axis by selecting the **Use motion axis zero** option

III. The spring definition ribbon allows the user to change the spring stiffness and the stiffness units.

IV. The visual diameter of the spring can be changed by using the **Adjust Icon Diameter** under the options tab in the spring definition ribbon. This has no physical effects.

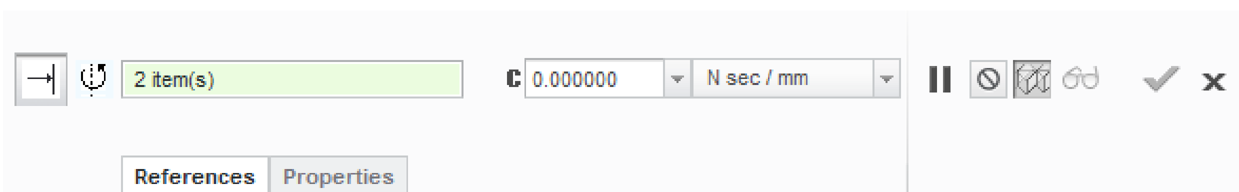


Animations in Creo 3.0 ME170

Damper – Extension and Compression

Dampers can be added by selecting the **Dampers** button located on the Mechanism ribbon.

I. To add a damper, click on the **Damper** icon on the **Insert** tab. Make sure you turn on **Point Display** to show datum points. A damper requires a top and bottom datum point or a translational axis to constrain it.
II. For the point method, select the point corresponding to the top of the damper and then hold CTRL and select the bottom point of the damper to complete it
III. When creating a damper along a translation axis, simply select the axis.
IV. The damper definition ribbon allows the user to change the damping rate and its units.



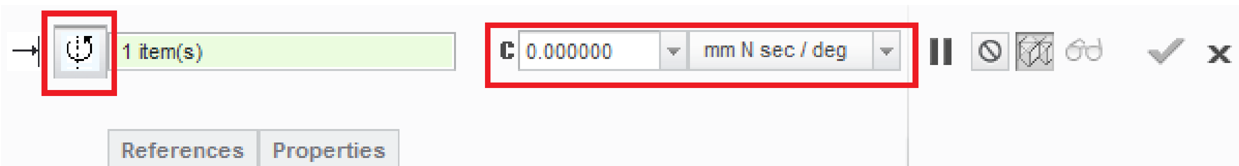
Animations in Creo 3.0 ME170

Damper – Torsion

Dampers can be added by selecting the **Dampers** button located on the Mechanism ribbon.

I. To add a torsion damper, click on the **Damper** icon on the **Insert** tab. Then click on the **Torsion Damper** button to the left of the green reference box. A torsion damper requires a rotation axis to constrain it.

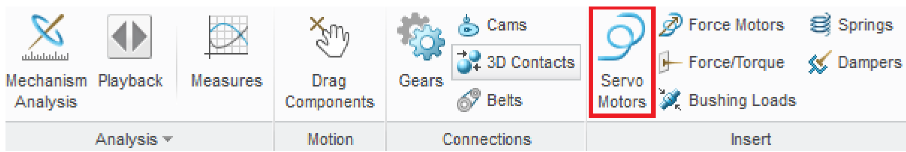
II. The damper definition ribbon allows the user to change the damping rate and the damping rate units.



Animations in Creo 3.0 **ME170**

Servo Motors

Servo motors move the mechanism according to a user specified profile. Servo motors do not take friction, mass or other parameters into account, they just move the model.

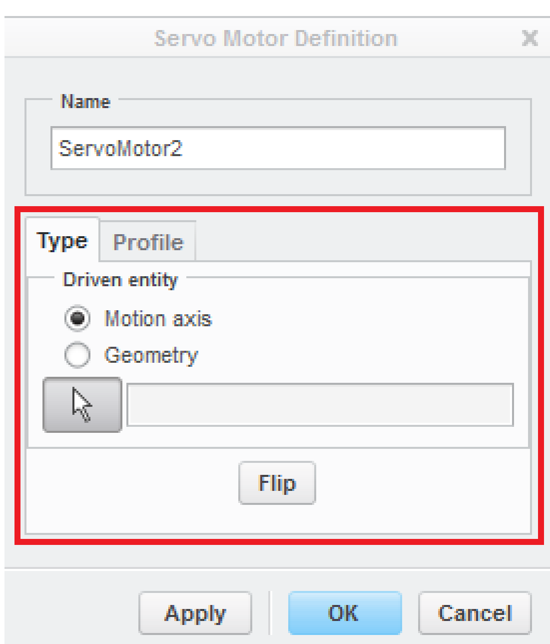


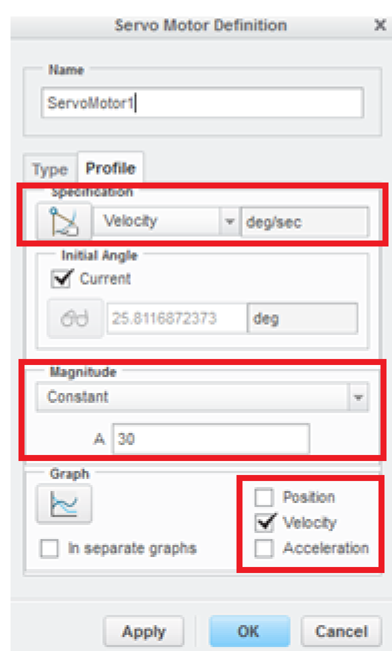
I. To add a servo motor click on the **Servo Motors** icon on the Insert tab.

II. A definition window will open up and ask you to select a motion axis. Select the motion axis that you want to animate.

III. The **Profile** tab in the definition window can be used to specify the desired motion profile. A constant velocity profile is arguably the simplest motion profile for an animation. However, acceleration and position profiles can also be specified with constant, linear, parabolic, sinusoidal and user define functions.

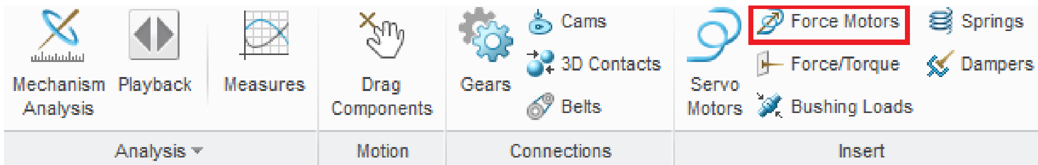
IV. You can graph the position, velocity, and acceleration profiles for your components by selecting the appropriate check boxes and clicking the **Graph** icon in the lower part of the dialog box. These graphs are quite useful when working with and troubleshooting more complex motion profiles.





Force Motors

Force motors move the mechanism according to a user specified profile. Force motors take mass into account. A dynamic or force analysis will only run if mass properties are defined for every part in the mechanism.

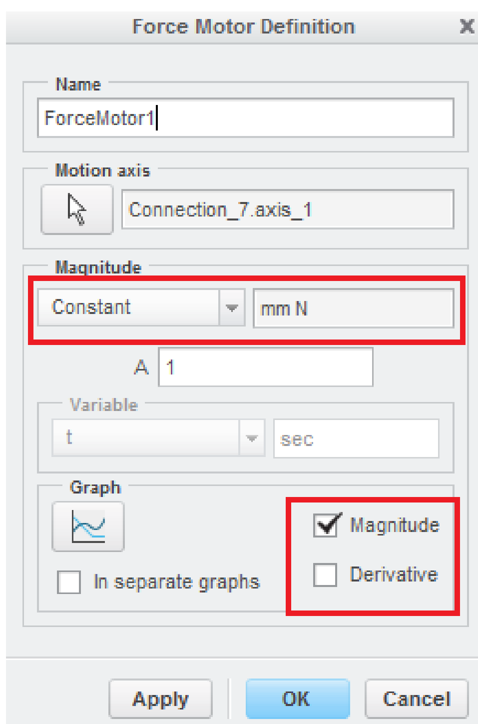


I. To add a force motor click on the **Force Motors** icon on the Insert tab.

II. A definition window will open up and ask you to select a motion axis. Select the motion axis that you want to animate.

III. The same definition window can be used to specify the desired motion profile. A constant profile is arguably the simplest motion profile for an animation. However, profiles can also be specified with constant, linear, parabolic, sinusoidal and user define functions.

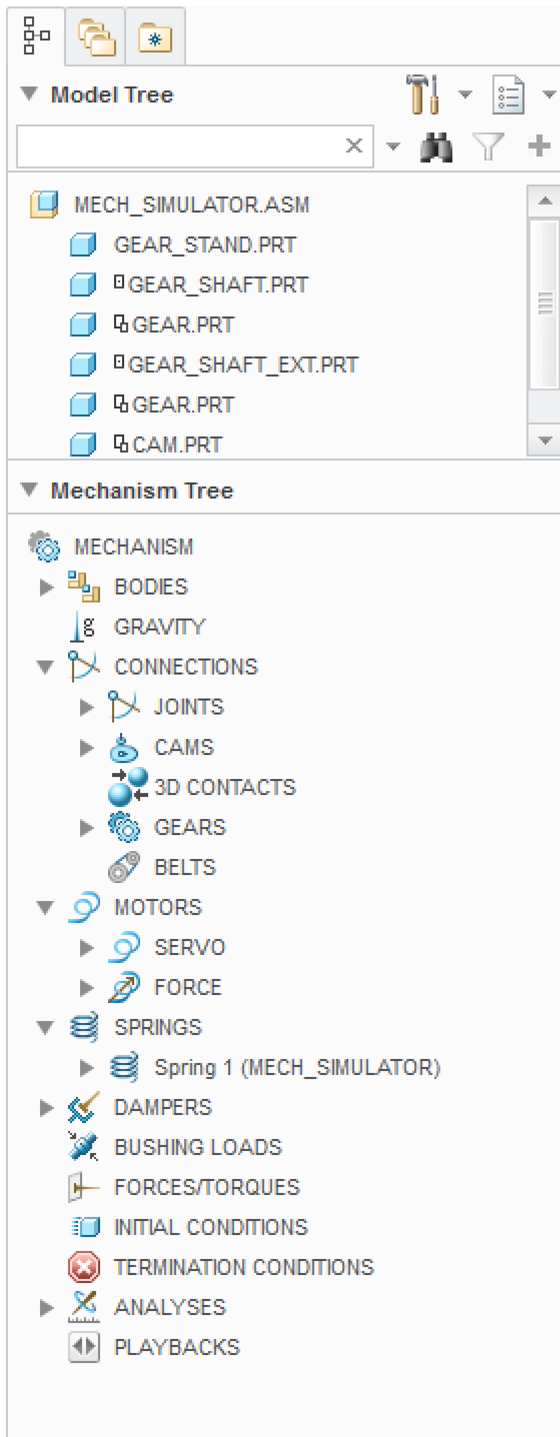
IV. You can graph the magnitude and derivative profiles for your components by selecting the appropriate check boxes and clicking the **Graph** icon in the lower part of the dialog box. These graphs are quite useful when working with and troubleshooting more complex motion profiles.



Animations in Creo 3.0 ME170

Mechanism Tree

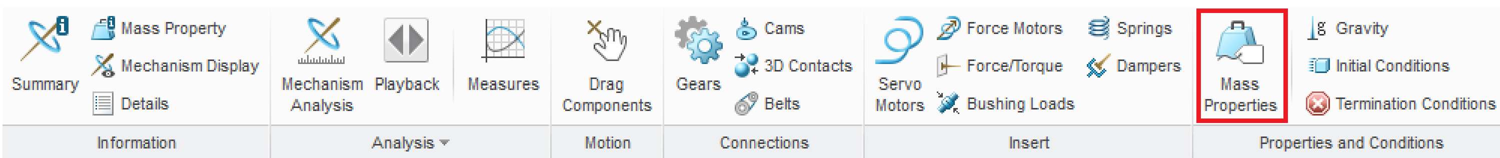
Similar to the Model Tree, the Mechanism Tree is used to edit the mechanism's features. Motors, connections, spring and other parameters can be accessed and modified from this tree.



Animations in Creo 3.0 ME170

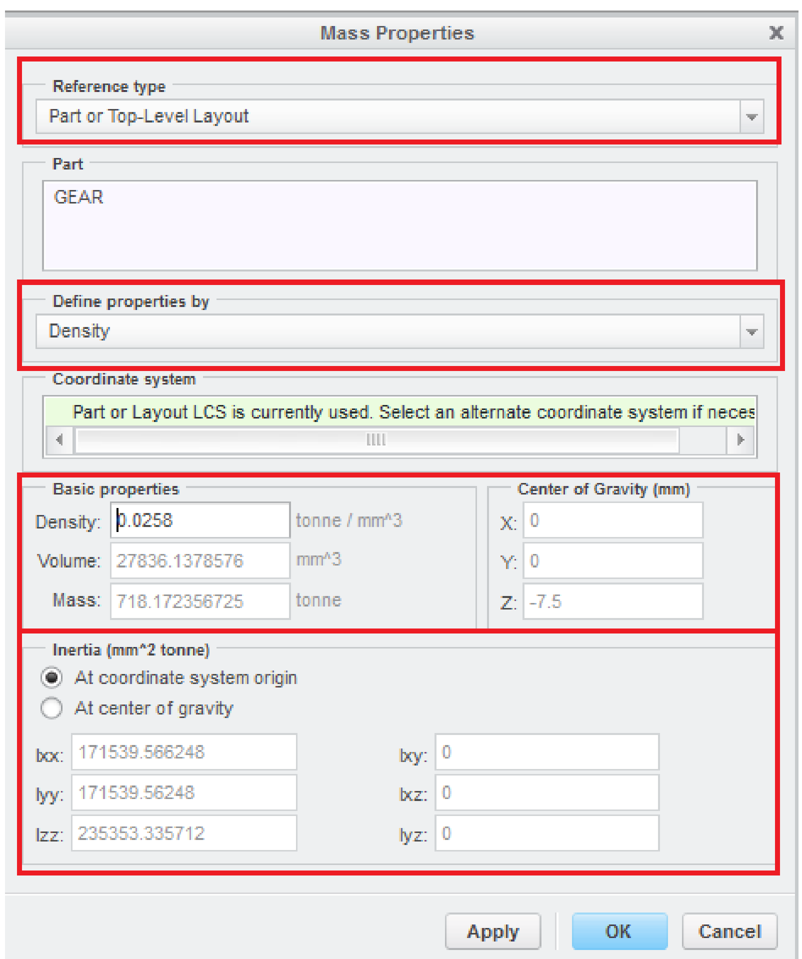
Mass Properties

Mass properties are important for applications that require a dynamic or static analysis. For these analyses to work, all bodies need to have a mass associated with them. The user can set the mass properties in multiple ways when using the mechanism application.



I. Mass properties can be setup when creating a part in Creo Parametric. It is good practice to set your mass properties when creating each part.

II. Mass properties can also be setup while in the mechanism application. Any mass properties setup while in the mechanism application will take priority over mass properties setup elsewhere in Creo.



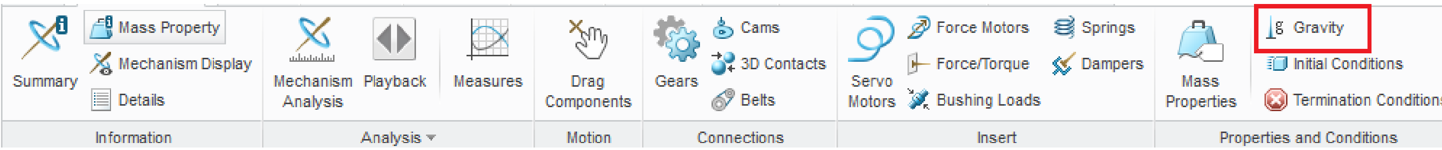
Animations in Creo 3.0 ME170

I. Click on the **Mass Properties** button located on the Properties and Conditions tab
II. Select the reference type. This menu allows you to set the mass properties of a specific **par**t or an **assembly**. The body option only allows you to view the mass properties of a body, not edit them.
III. The **Define properties by** menu selects the properties to be defined.
   a. The **Default** option uses the mass properties set in Creo Parametric
   b. The **Density** option allows the user to set a density that will override the Creo Parametric density
   c. The **Mass Properties** option allows the user to manually set the mass, center of gravity and inertia values.

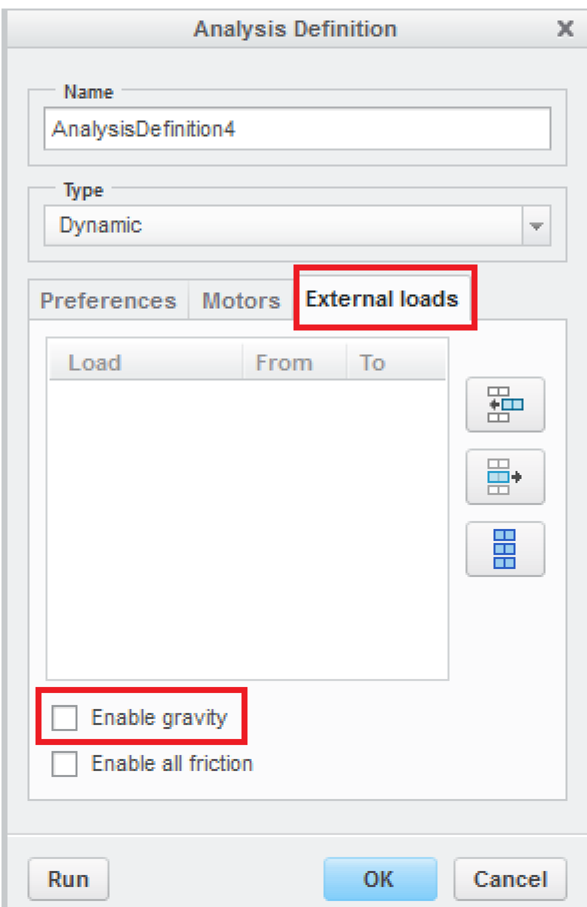
Animations in Creo 3.0 ME170

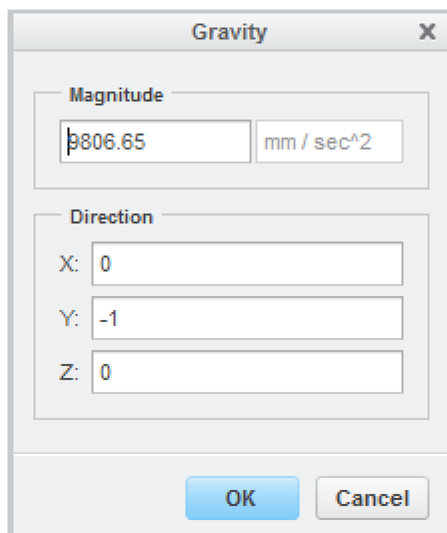
Gravity

Gravity can be setup for use in dynamic and static analyses.



I. Click on the **Gravity** button to open up the gravity definition window.
II. The gravity definition window allows the user to setup the magnitude and direction.
III. The X, Y and Z options follow the default coordinate system of the mechanism.
IV. Gravity is turned on and off in the analysis setup. It is located on the **External Loads** page of a Dynamic, Static or Force Balance analysis.

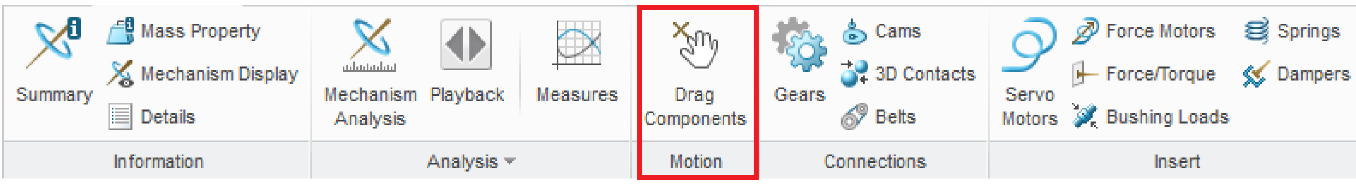




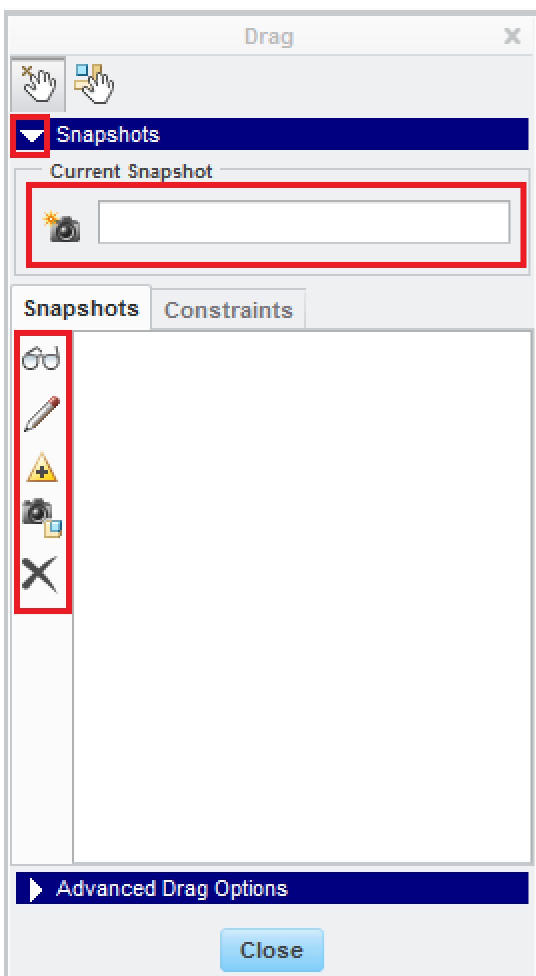
Animations in Creo 3.0 ME170

Snapshot

Snapshots of the mechanism can be taken and used as the initial configuration for an analysis. This is a useful tool to start at the same initial position for multiple analyses.



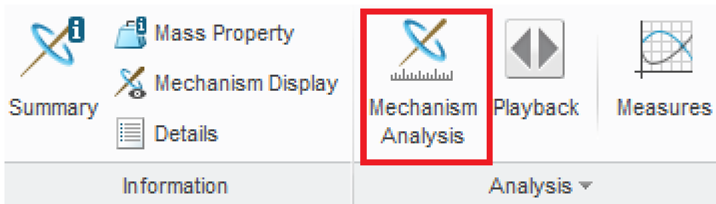
I. Start by clicking the **Drag Components** button.
II. Click on the white arrow to open the snapshot options.
III. Clicking the camera icon under **Current Snapshot** will record a snapshot of the current mechanism position.
IV. Use the options on the left side of the Drag Components definition window to modify the snapshot.



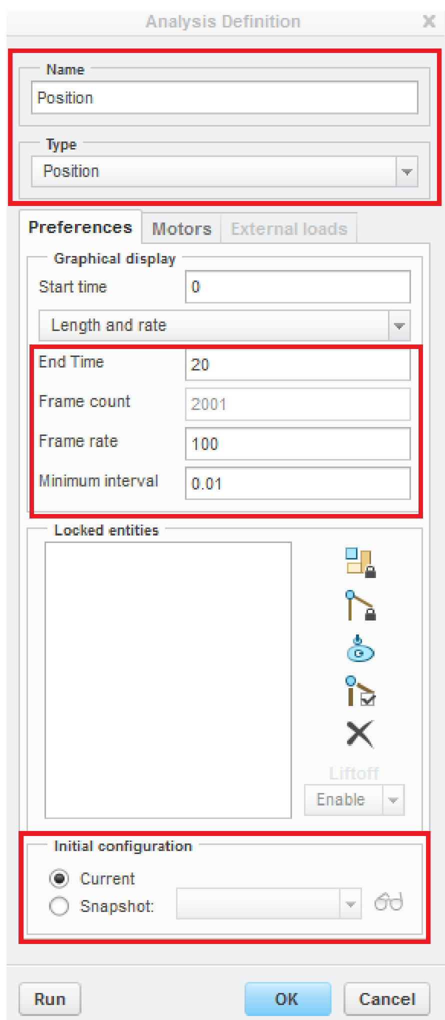
Animations in Creo 3.0 ME170

Mechanism Analysis

The actual animation of the mechanism is done through a Mechanism Analysis.



I. Click on the **Mechanism Analysis** button located on the Analysis tab.
II. The analysis definition window will open up. This definition window is used to select the type of analysis, along with the start time, end time, frame rate and initial configuration.
III. The **Initial Configuration** can be setup to use the mechanism's current position, or setup to use a premade snapshot of the mechanism.



Animations in Creo 3.0 ME170

Mechanism Analysis

To create a basic animation, a position or kinematic analysis is recommended. Static, dynamic and force balance analysis require mass properties to run, while a position or kinematic analysis will run without those setup.

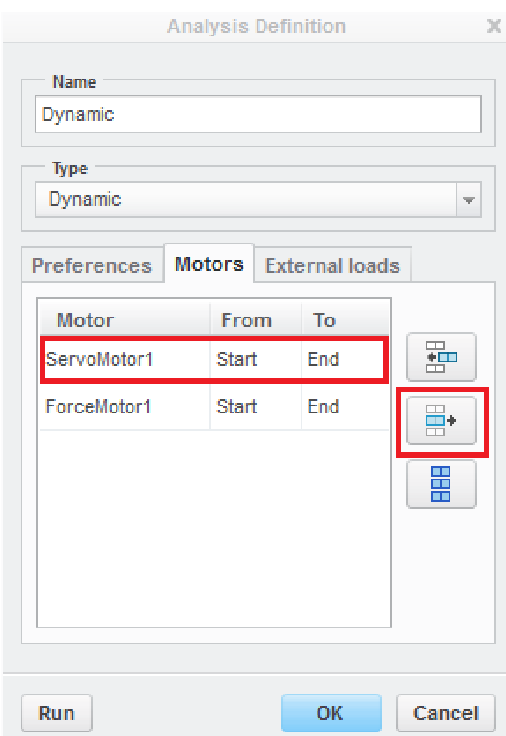
Once the type of analysis is selected, the motors to move the model must be setup.

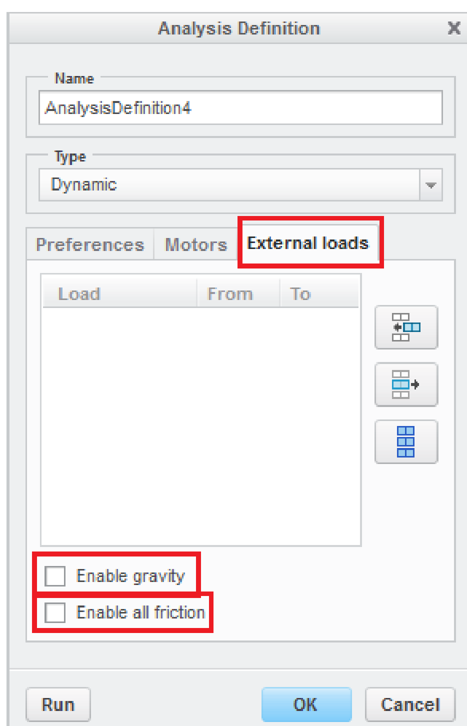
I. The **Motors** tab is used to specify which motors will run the analysis. For the position and kinematic analysis, only the servo motor will be listed.

II. Dynamic, Static and Force Balance analysis can also use force motors. When running these analyses, both force and servo motors will be listed in the motors tab. It is important to remove motors if they are not required as they can “fight” each other if setup incorrectly.

III. Use the buttons to the right of the motors to add or delete motors.

IV. The **External Load** tab can be used to setup external user defined loads and can also be used to turn on friction and gravity.

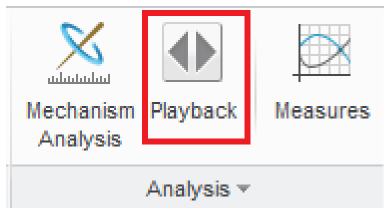




Animations in Creo 3.0 ME170

Playback

The Playback button can be used to playback the animation of any analysis and also export it as a video.



I. Click on the **Playback** button located on the Analysis tab.
II. Select your desired **Result Set** and click the **Play Current Result Set** button located to the left of the folder icon.
III. Click the **Capture** button to export the animation as a video.
IV. Clicking capture opens up the **Capture Definition** window. This window can be used to rename your video and change its resolution and frame rate. Click **OK** when finished to start the export process.