TABLE OF CONTENTS

Introduction

CATIA Version 6 Kinematics

Upon completion of this course the student should have a full understanding of the following topics:

- Creating joints
- Creating simulations and replays
- Performing analysis on a kinematic mechanism
- Using laws to help simulate a mechanism
- Converting assembly constraints to joints

Kinematics

The first item that needs to be understood is the definition of kinematics. Kinematics involves an assembly of parts that are connected together by a series of joints, referred to as a mechanism. These joints define how an assembly can perform motion. When one of the joints move it causes the assembly to move. Kinematics does not involve any type of finite element analysis, which means that there are no associated loads or weights with the parts. You are simply moving the assembly through some range of motion as defined by the joints.

Sometimes kinematics gets confused with animation as well. Although kinematics does perform some actions similar to animation, it is very limited. Kinematics is meant to show the range of motion of a mechanical component. Basically, it will show how the movement of one joint affects all of the other joints defined in the mechanism.

The second concept that needs to be understood are degrees of freedom. Every part has six degrees of freedom. It can move in three directions, and it can also rotate about those three directions. In order for an assembly to be used in kinematics, it must have at least one degree of freedom. The remaining degrees of freedom are controlled with commands. This is what allows the motion of the mechanism to be defined.

The hardest part about kinematics is figuring out what joints need to be defined on an assembly in order for it to operate correctly. The actual definition of the joints is pretty easy. Kinematics requires you to think a little differently without allowing finite element analysis and animation confuse the situation.

Kinematics is used to check for clearances and interferences among moving parts, and analyze the velocities of parts as well. In addition, laws can be applied to the mechanism to force the parts to accelerate.

There are two main kinematic apps: Mechanical Systems Design and Mechanical Systems Experience. Mechanical Systems Design is primarily used to define all the joints in the kinematic mechanism. It will be the focus of the first half of this manual. Mechanical Systems Experience is primarily used after the kinematic mechanism has been defined. This app is primarily used to simulate the mechanism. It will be the focus of the second half of this manual.

Kinematics

Engineering Connection Types

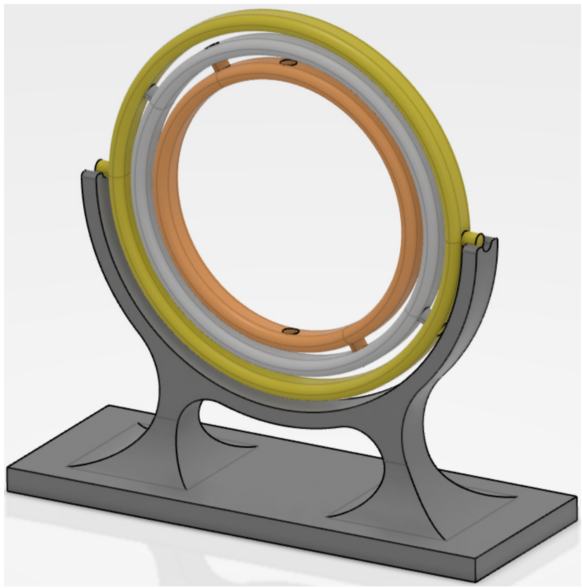
There are many engineering connection types. Each type has degrees of freedom and commands associated with it. A good understanding of these will make performing kinematics much easier. The table below gives an overview of the engineering connections. Some of the connections can be created using axis systems but that will be discussed later.

Engineering Connection	Degrees of Freedom	Commands Available
User-Defined	Depends	None
Rigid	None	None
Spherical	3 Rotations	None
Cylindrical	1 Rotation, 1 Translation	Angle and/or Length
Planar	1 Rotation, 2 Translations	None
Prismatic	1 Translation	Length
Revolute	1 Rotation	Angle
Screw	1 Rotation or 1 Translation	Angle or Length
Point Curve	3 Rotations, 1 Translation	Length
Point Surface	3 Rotations, 2 Translations	None
Gear	1 Rotation	Angle
Rack	1 Rotation or 1 Translation	Angle or Length
Cable	1 Translation	Length
Universal	1 Rotation	None
Roll Curve	1 Rotation, 1 Translation	Length
Slide Curve	2 Rotations, 1 Translation	None
Fix	None	None

Revolute - Null Offset

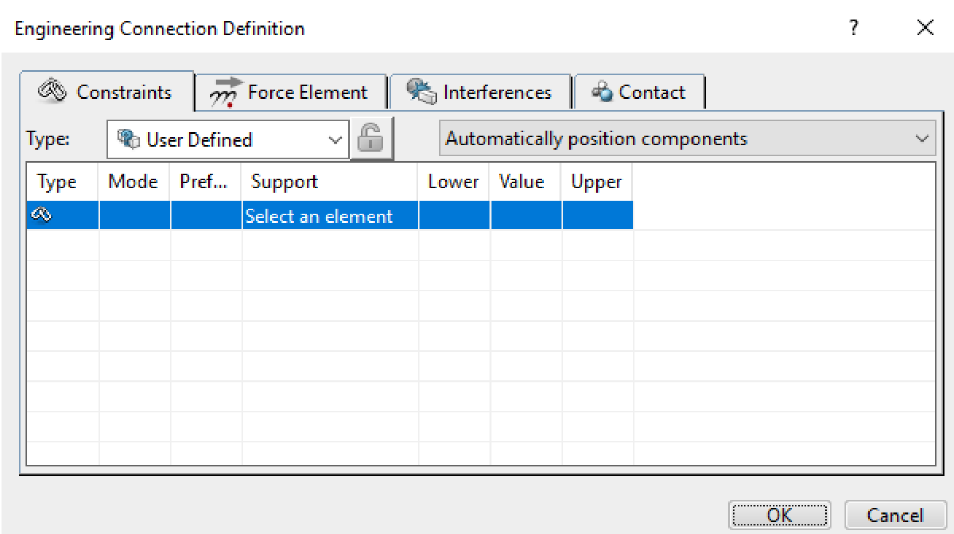
The revolute option allows you to define a joint that represents a rotation. This is useful when you need an object to turn about another object.

Open the KINE110 - Revolute Assembly document. You should see a base with three rings.



If not already there, switch to the Mechanical System Design app.

Select the Engineering Connection icon.  The *Engineering Connection Definition* window appears. This window should be familiar from the Assembly Design workbench.

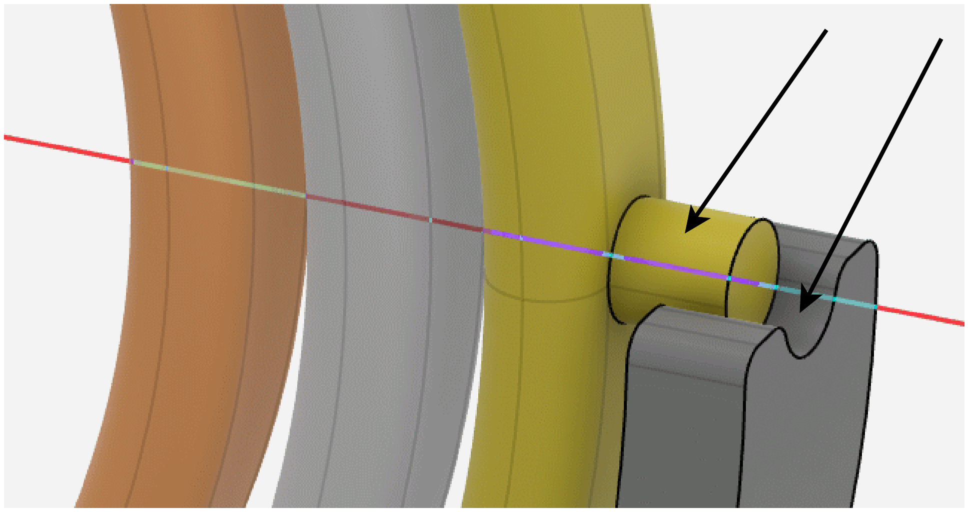


Select the Gyroscope Base from the tree. Notice the *Type* automatically switches to *Fix*. You must always have a fixed object in order to create a kinematic simulation.

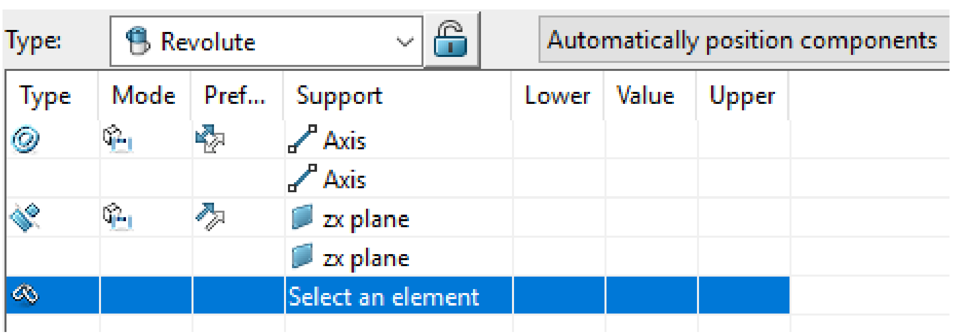
Select *OK*. The engineering connection is created.

Select the Engineering Connection icon.  The *Engineering Connection Definition* window appears.

Select the center line of the cylinder on the outside ring and the center line of the cutout on the base as shown below. This defines the center of rotation and aligns the two lines.



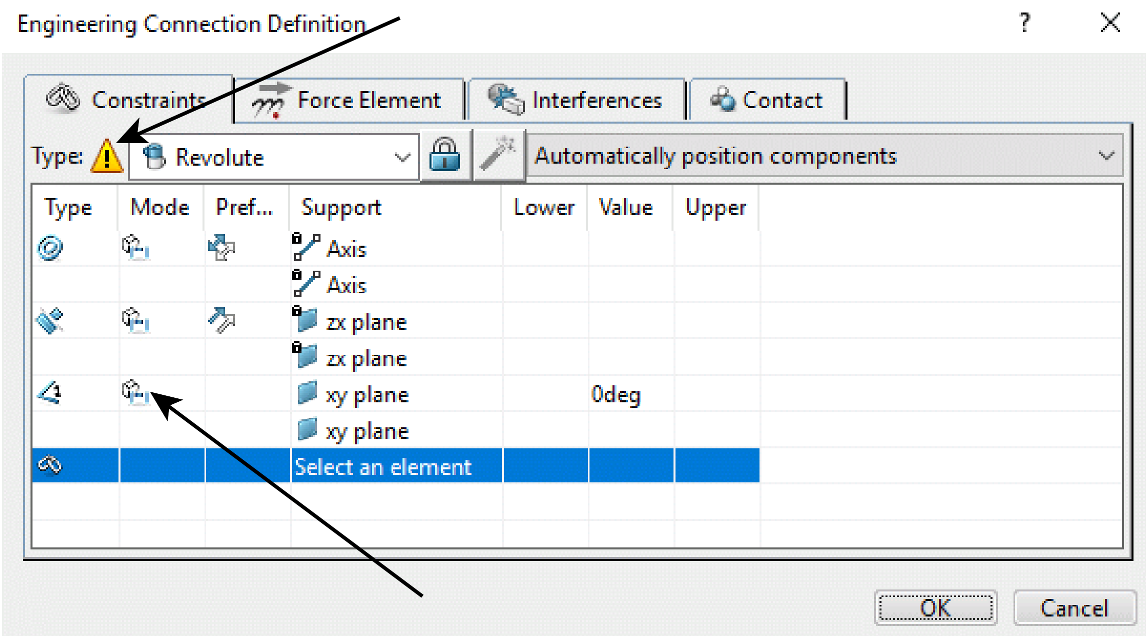
Select the *zx plane* in both parts. This defines the location of each object by aligning the two planes. You will probably have to expand the branches of the Base and 1st Ring in order to select the planes. Notice the *Type* automatically updates to *Revolute*.



The command will need to control the angle of the Cam in relation to the Fixture1 part, so you will need to create an Angle constraint to apply the command to.

Right select on the *Select an element* text in the window and select *Insert, Angle, Angle*. An angle constraint is inserted.

Select the *xy plane* in both parts. This will be the reference for the rotation angle. The window should appear as shown.



Notice the *Type* has an error symbol next to it. The *Mode* of the angle constraint needs to be switched to *Controlled*.

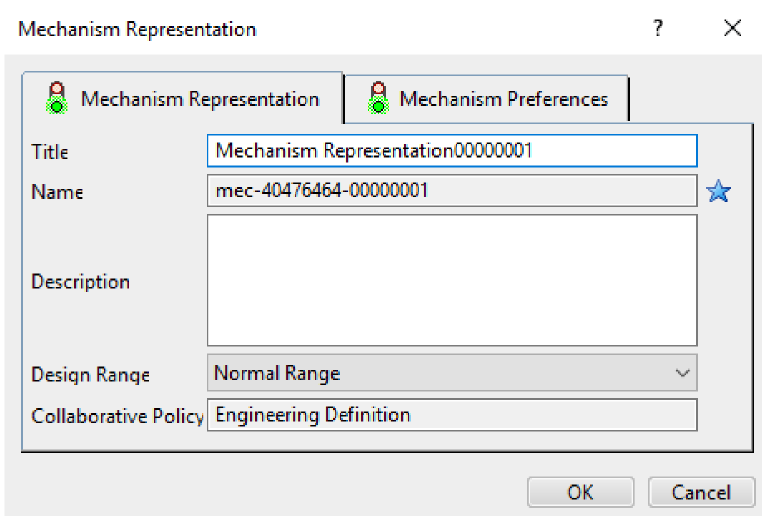
Right select on the symbol under the *Mode* column as shown above and select *Controlled*. The error symbol goes away.

Select *OK*. The engineering connection is created.

Mechanism Representation

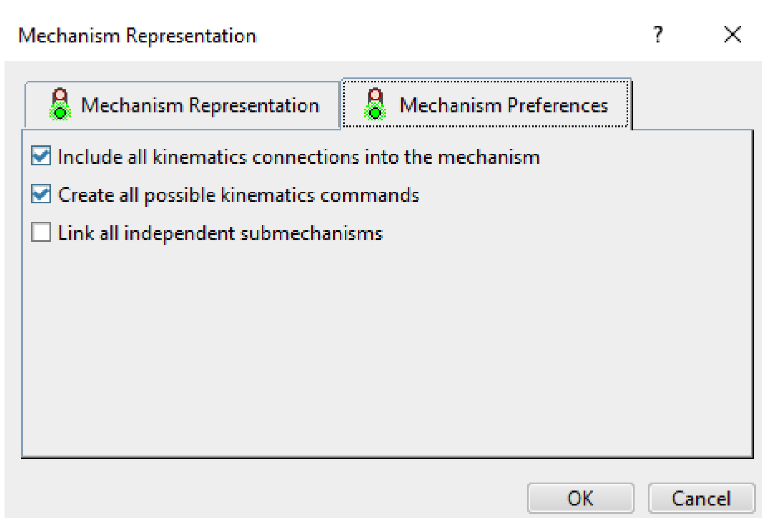
The mechanism representation will contain all of the kinematic information for simulating movement within your assembly. It is a separate representation, similar to a 3d shape representation, that will be stored within the assembly.

Select the Mechanism Representation icon.  The *Mechanism Representation* window appears.



Change the *Title* to <u>KINE110 - Kinematic Mechanism</u>.

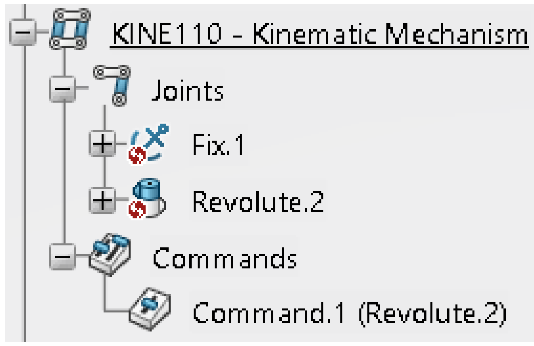
Select the *Mechanism Options* tab.



Include all kinematics connections...	Creates joints from all of the existing engineering connections in the model
Create all possible kinematic commands	Creates commands from any existing controlled constraints within the existing engineering connections

Link all independent submechanisms	Identifies and assembles existing mechanisms within any subassemblies under the product the new mechanism representation is being created in

Be sure the first two options are turned on and select *OK*. The mechanism representation is created.



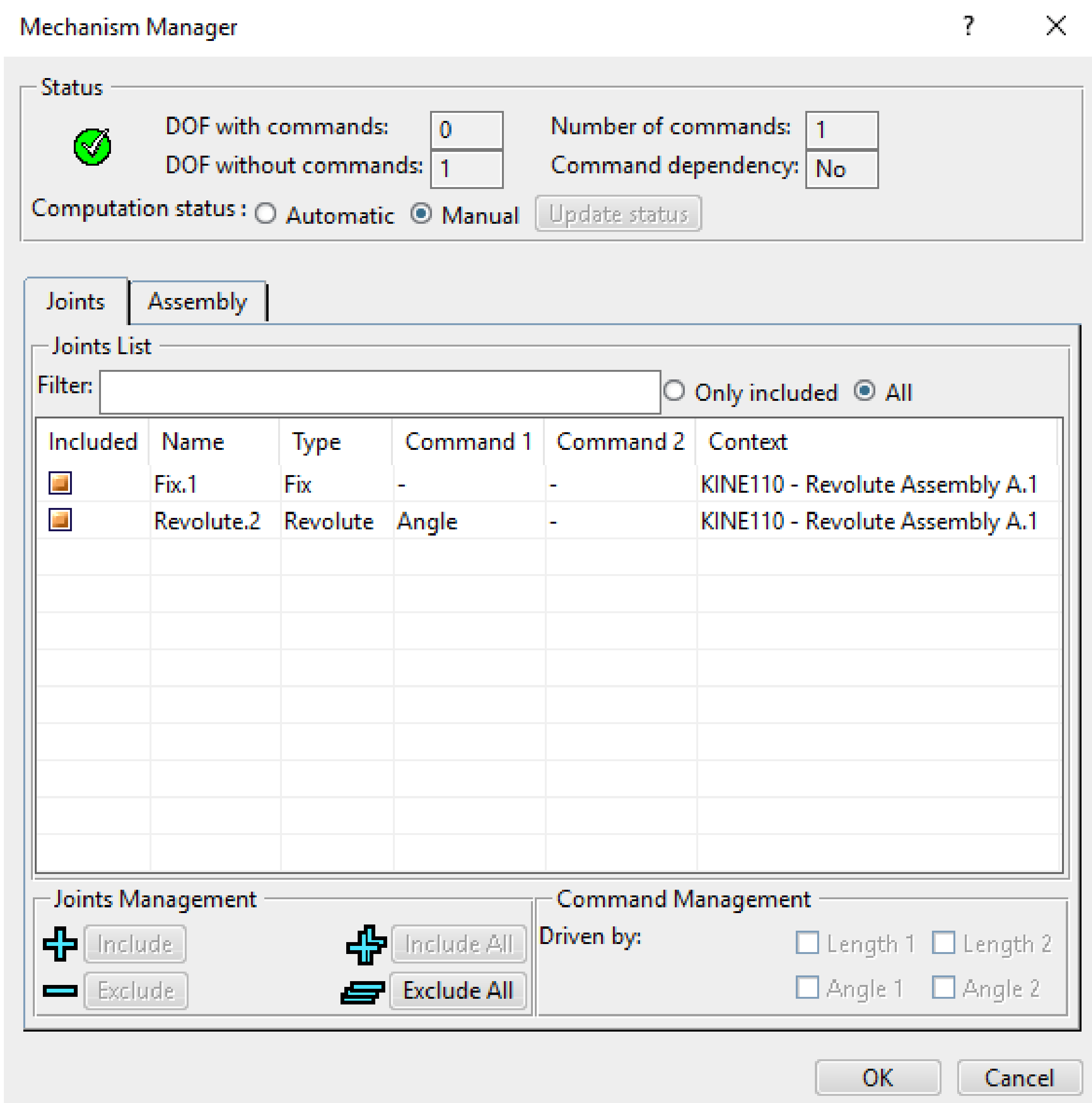
All of the engineering connections were used to create joints within the mechanism. Notice there is one command.

Select the Update icon.  This will update the engineering connections. It is a good idea to make sure things are updated.

Mechanism Manager

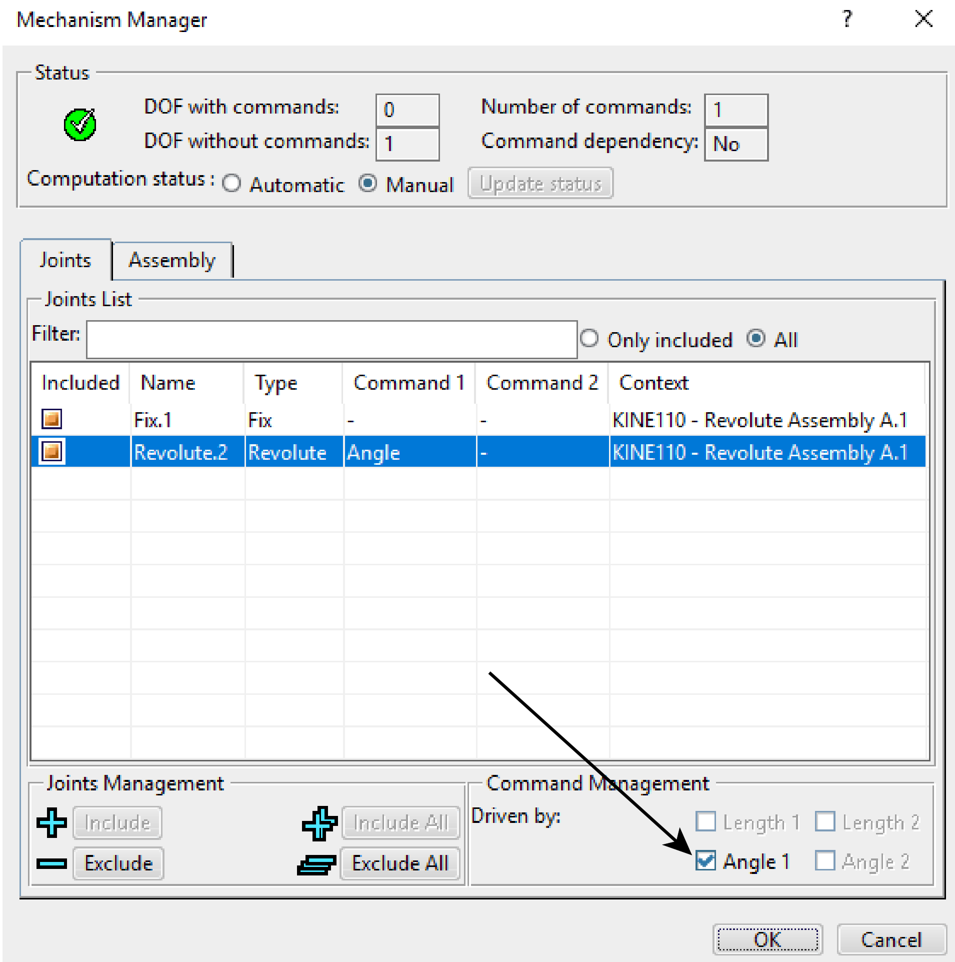
The mechanism manager allows you to get a summary of the existing mechanism, including all of the joints, commands and degrees of freedom. It can be helpful for identifying issues within the mechanism.

Select the Mechanism Manager icon.  The *Mechanism Manager* window appears.



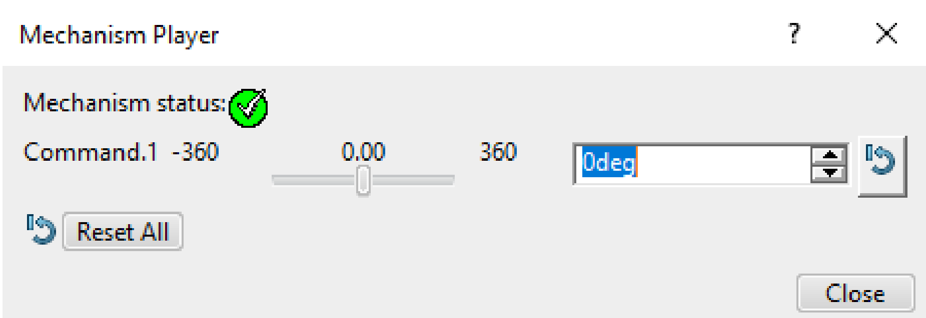
Notice there are no degrees of freedom with the command. This window also identifies which connections could have a command added to them.

Select the revolute in the list. Notice the Angle 1 command is checked. This is the angle command you already added to the revolute engineering connection.



Select *OK*.

Select the Mechanism Player icon.  The *Mechanism Player* window appears.

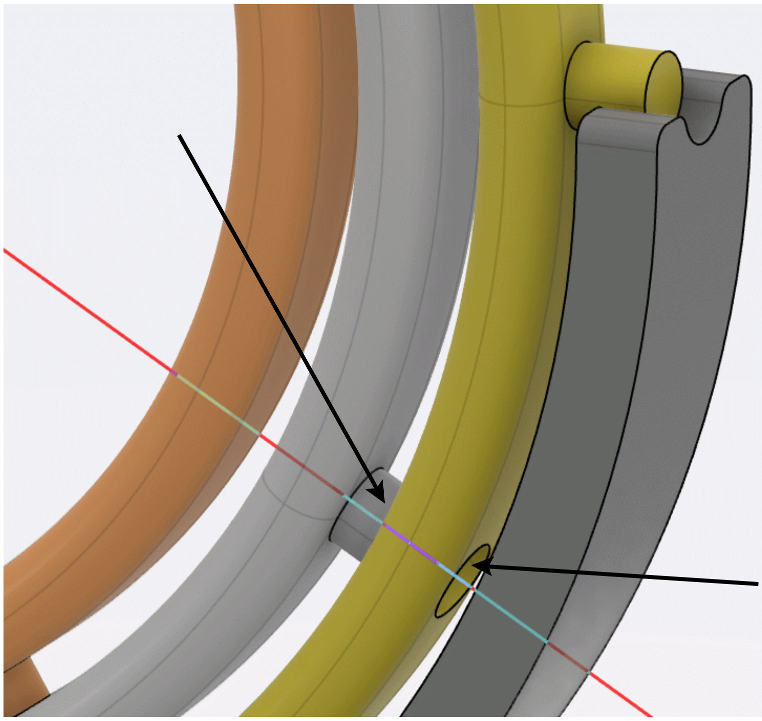


Drag the slider in the window. The ring will rotate.

Select the *Reset All* button and select *Close*. This defines one joint. You will continue to define revolute joints in order for the complete assembly to be defined.

Select the Engineering Connection icon again.  The *Engineering Connection Definition* window appears.

Select the center line of the cylinder on the outside of the middle ring and then the center line of the hole of the outer ring as shown below. This defines the lines.



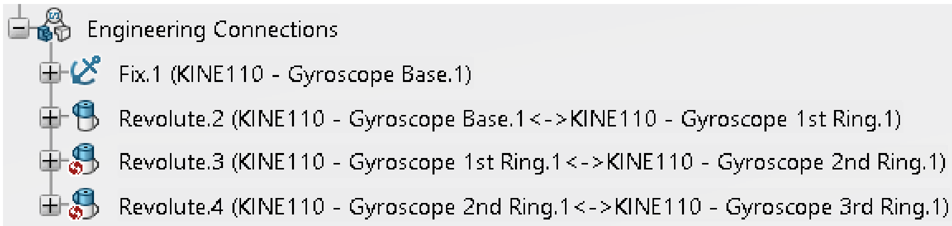
Select the *zx plane* in the 2nd Ring and *Plane.1* from *Geometrical Set.1* in the *1st Ring*.

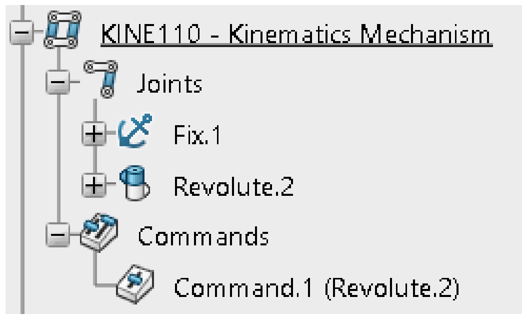
Insert an *Angle* constraint and define the *yz plane* of both rings as the inputs. Switch the *Mode* to *Controlled* in the same manner as before. This will have the revolute defined with a command.

Select *OK.* The window closes.

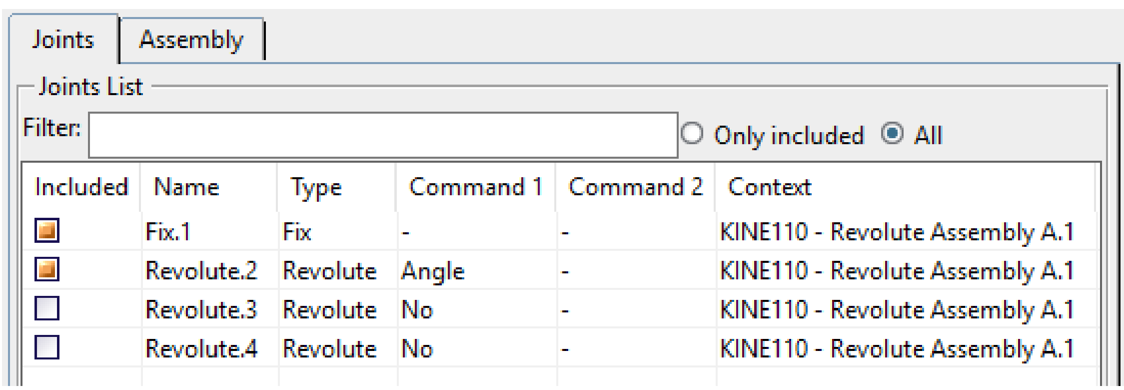
Define the revolute joint between the inner ring and the middle ring. You do this in the same manner as you did the second revolute joint.

Notice the engineering connections are created in the tree, but not populated in the mechanism.



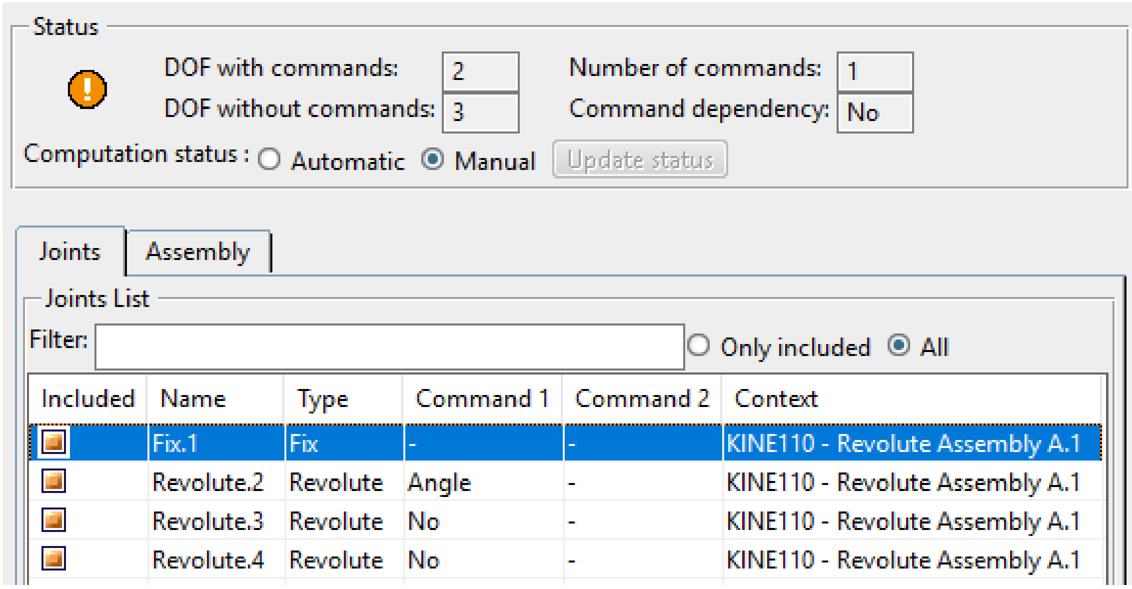


Select the Mechanism Manager icon.  The *Mechanism Manager* window appears.

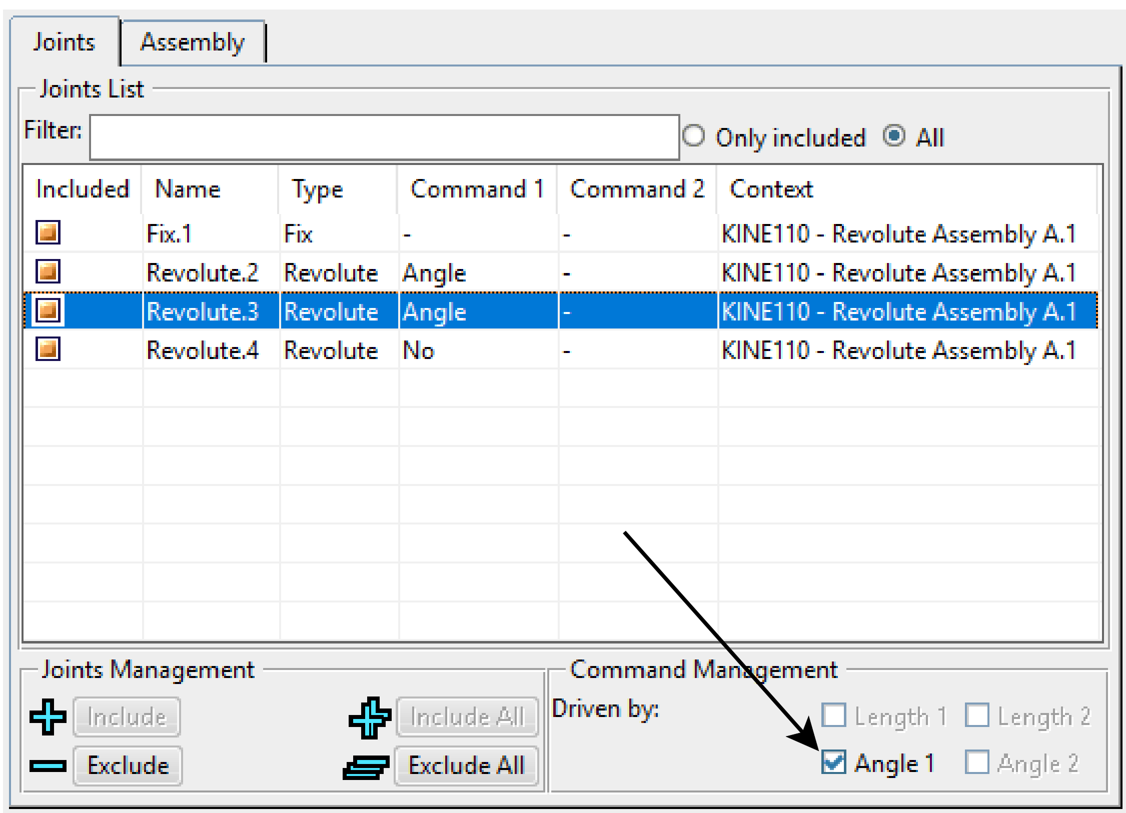


Notice the last two revolutes are not included in the mechanism.

Turn on the checkboxes next to the last two revolutes and then select the *Update status* button. Notice there are now 2 degrees of freedom with commands. This is because only the first command is recognized. You will need to activate the other two commands.

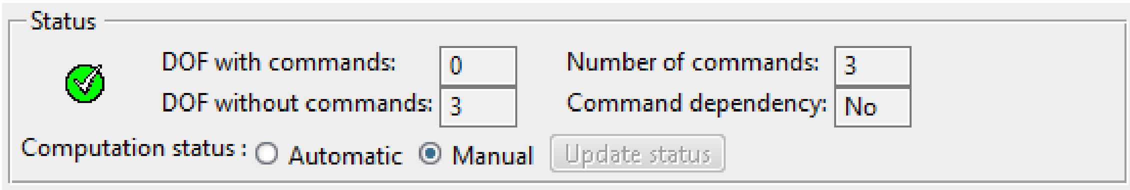


Select the second revolute and turn on the *Angle 1* command. This will need to be done anytime a joint is added to an existing mechanism.



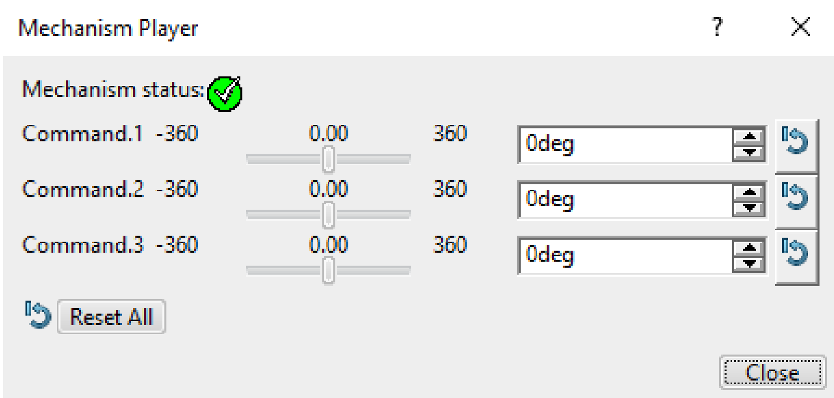
Turn on the *Angle 1* for the last revolute as well.

Select the *Update status* button. The degrees of freedom update.

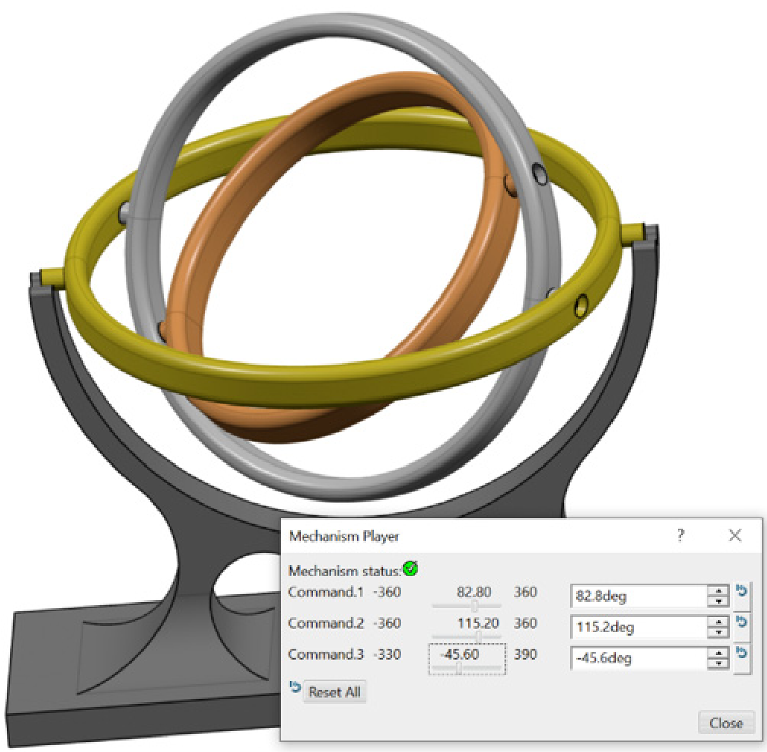


Select *OK*.

Select the Mechanism Player icon.  The *Mechanism Player* window appears.

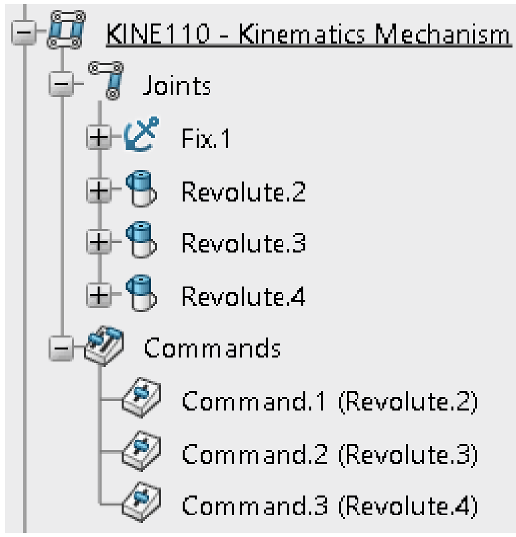


Drag each of the sliders in the window. The rings should all rotate now.



Select the *Reset All* button and select *Close*. This defines one joint. You will continue to define revolute joints in order for the complete assembly to be defined. Your first kinematic mechanism has been defined.

Your final mechanism should appear similar to the one shown below.



Save and close your document.

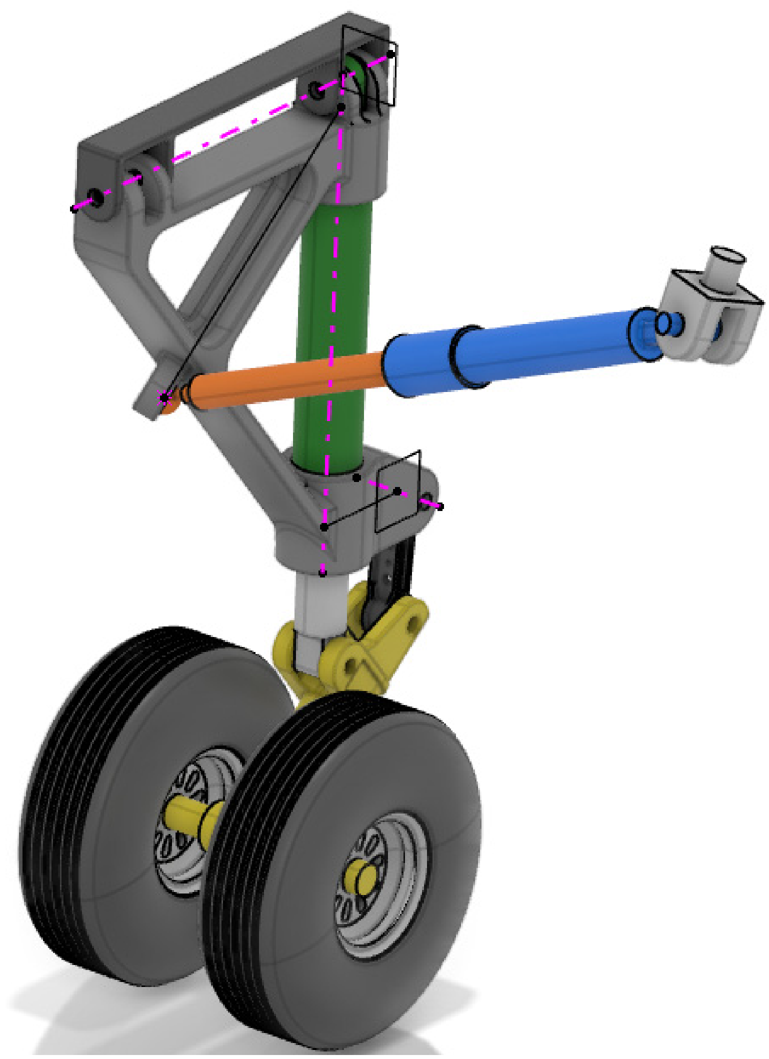
Mechanism Operations

This section will discuss a number of operations with regards mechanism manipulation and analysis.

Dressup

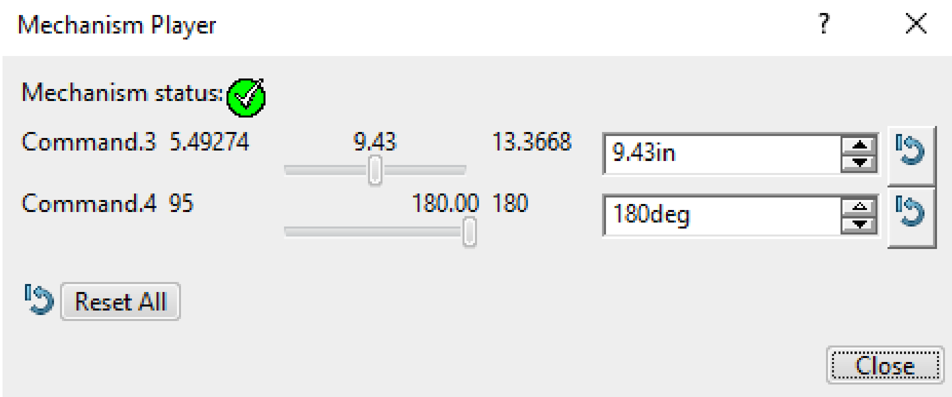
The dressup feature allows you to add another object to your mechanism without having to make it part of the actual mechanism. This is very useful when you already have a mechanism defined in a sub-assembly and you include that into a larger assembly. To be able to use the mechanism contained in the sub-assembly and be able to attach other parts to that mechanism, you must use the dressup feature.

Open the KINE300 - Dressup model. The model should appear as shown.

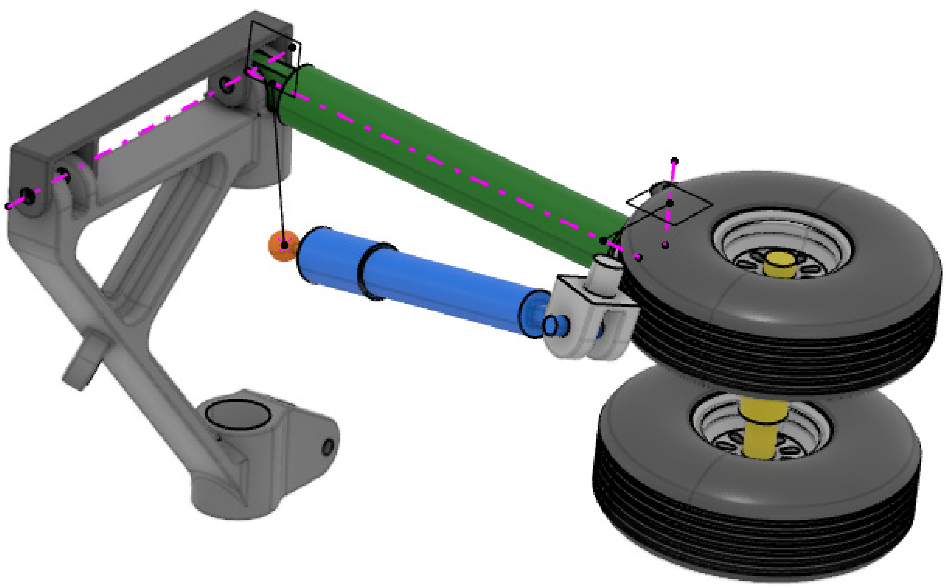


The trunion was initially designed with simple wireframe geometry and is represented by the KINE300 - Turnion Design Geometry model. Now the final trunion part will be added into the kinematic mechansim using a dressup.

Select the Mechanism Player icon.  The commands should appear as shown.

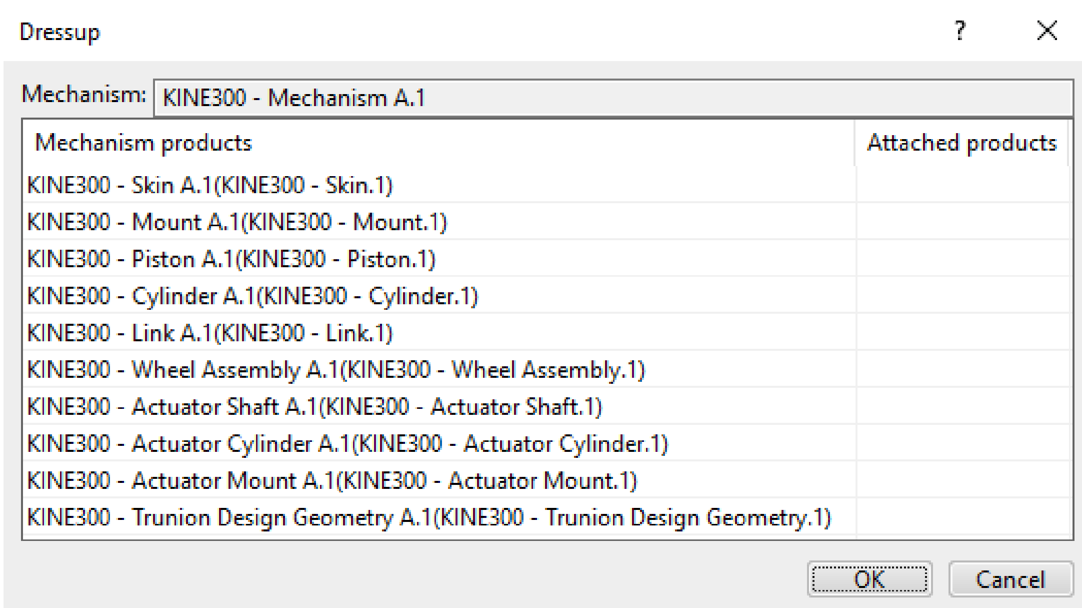


Drag *Command.4* to the left. The assembly should move as shown. Notice the actual trunion part doesn't move because it isn't part of the mechanism yet.



Select the *Reset All* button and close the *Mechanism Player* window.

Select the Dressup icon.  It is in the sub menu of the Mechanism Manager icon. The *Dressup* window will appear.



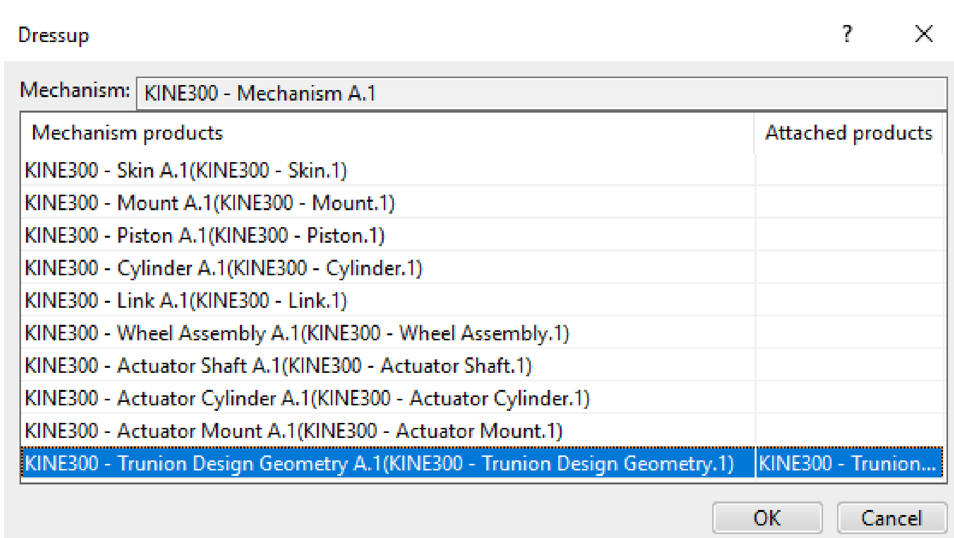
Mechanism Specifies the mechanism that you are dressing up

Mechanism products Shows the products available to attach a dressup

Attached products Defines the product(s) that will be attached with the dressup

Select *KINE300 - Trunion Design Geometry*. This is the part that you wish to attach other objects to.

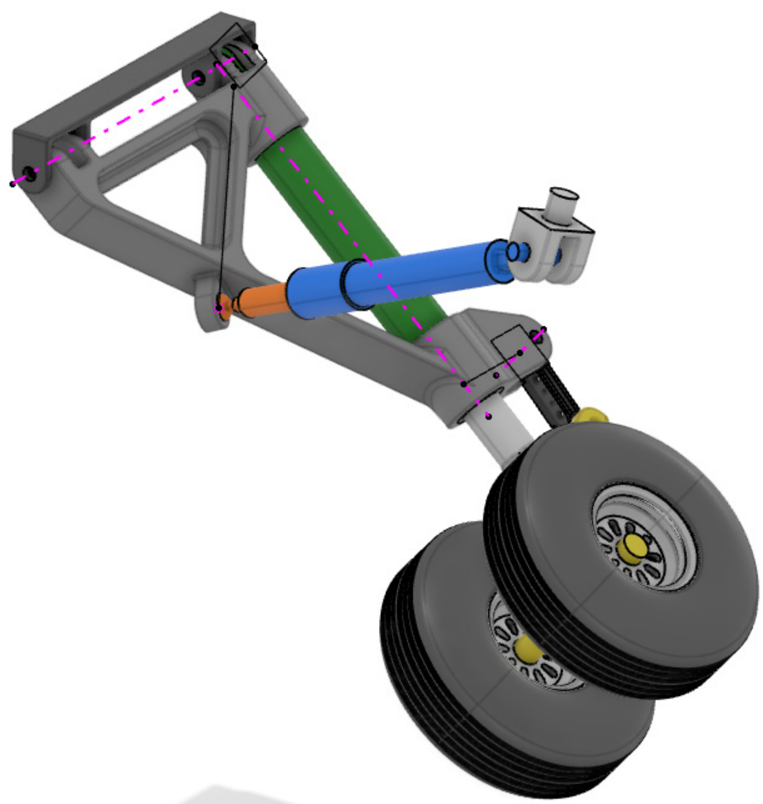
Select the *KINE300 - Trunion* part in the specification tree. It should appear in the *Dressup* window.



Select *OK.* There should now be a *Dressup* branch under the mechanism.

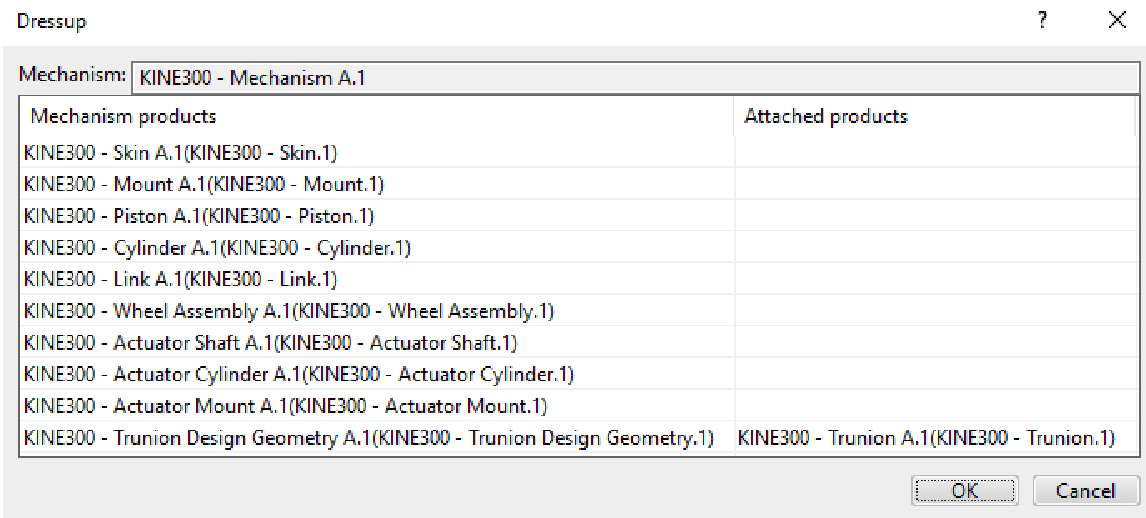
Notice the dressup basically creates a rigid connection between the parts.

Simulate the mechanism. Notice the trunion now moves with the rest of the assembly.

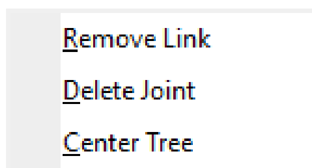


Select the *Reset All* button and select *Close.*

Select the Dressup icon.  The *Dressup* window will appear.



Right select on the trunion in the Attached products column. A contextual menu appears.



Remove Link Deletes the dressup link from the mechanism, but retains the underlying rigid joint

Delete Joint Deletes both the dressup link and the underlying rigid joint.

Select *Cancel*.

The dressup option provides and easy way to attach additional parts or products to objects already in a mechanism. This is especially helpful if a kinematic mechanism was done with simple wireframe geometry for conceptual design.

Save and close your document.

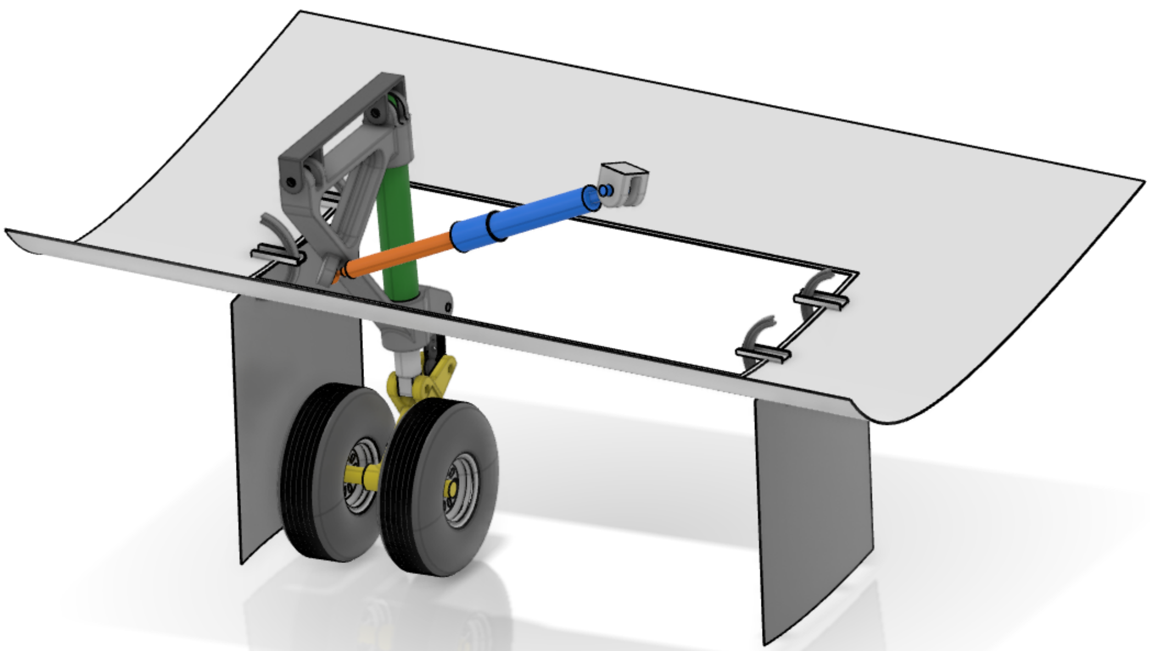
Kinematic Simulations

Up until now you have been simulating your mechanisms using the mechanism player. You can also create a simulation that can be used to better visualize the kinematic movements. A kinematic simulation consists of a Model, Scenario and Result. Each node in the tree has a different function and will be discussed. The model represents the product you are using for the kinematic simulation. The scenario defines the definition of the kinematic movement including excitations and probes. Scenarios can be used in conjunction with assembly sequences that will allow you to put multiple scenarios together and vary their duration. The results node will contain simulation results and any snapshots of specific simulation steps.

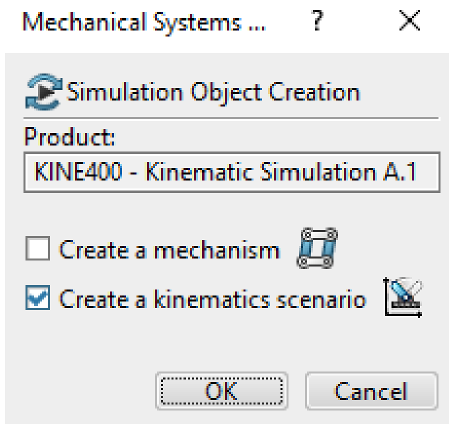
Creating a Kinematic Simulation

This section will discuss creating a kinematic simulation that can be used to better visualize the kinematic movements, as well as analyze them.

Open the KINE400 - Kinematic Simulation model. It should appear as shown.

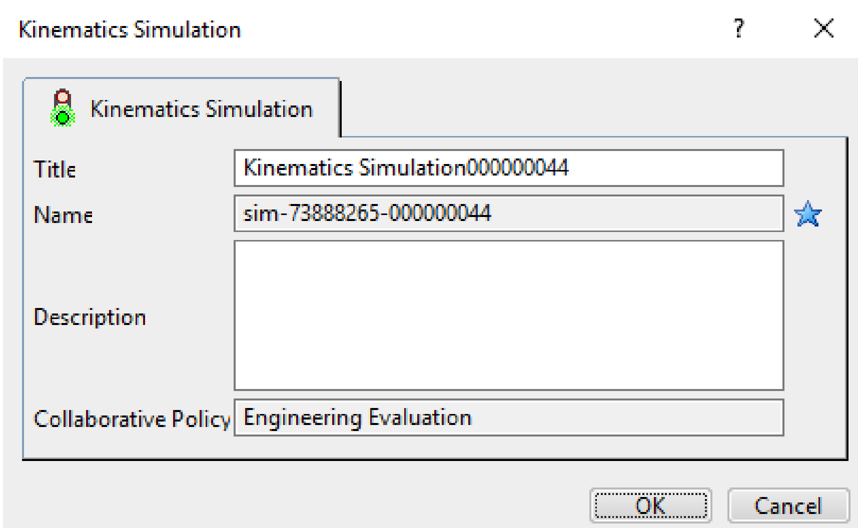


Switch to the Mechanical Systems Experience workbench. The *Mechanical Systems Experience* window appears.

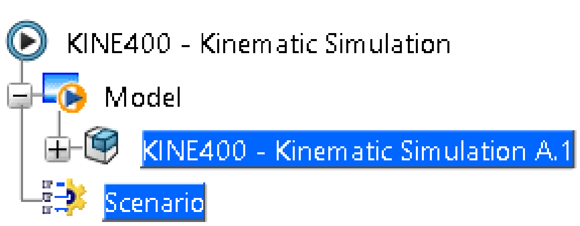


Product	Specifies the product to use for the model node of the simulation
Create a mechanism	Creates a new mechanism if one doesn't already exist
Create a kinematics scenario	Specifies a kinematics scenario will be created

Turn off both check boxes and select *OK*. The *Kinematics Simulation* window appears.

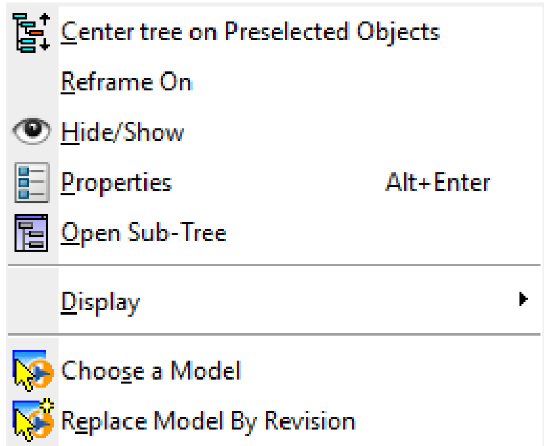


Key in <u>KINE400 - Kinematic Simulation</u> for the *Title* and select *OK*. The kinematic simulation is created.



This is the easiest method for creating a kinematic simulation.

Right select on the *Model* branch in the tree. A contextual menu appears.



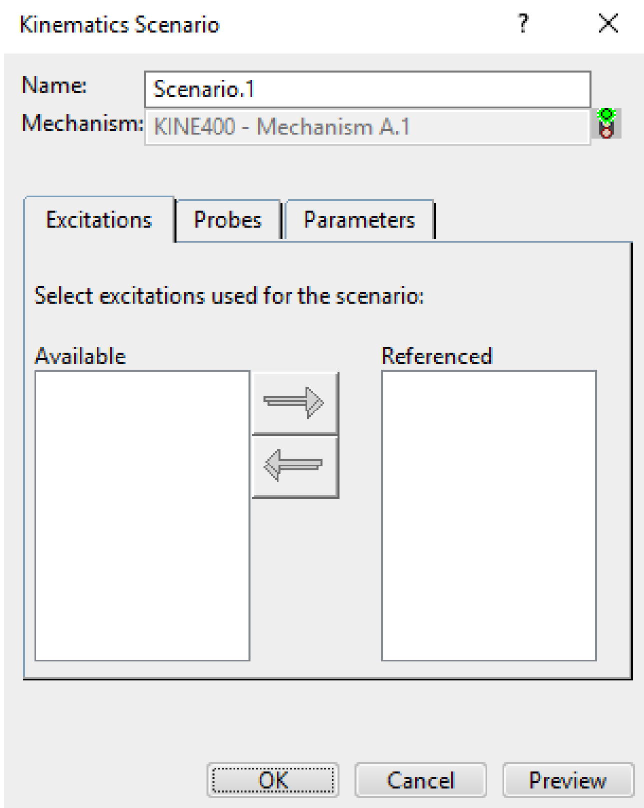
Notice you can use the *Choose a Model* option in order to populate the *Model* branch if you had created the kinematic simulation without having a product open first. You also have the option to *Replace Model By Revision* if you had a new revision of the product.

Select in space. Now you are ready to create a scenario for the simulation.

Kinematic Scenarios

This section will discuss creating a scenario with specific excitations and probes. There are two types of scenarios: kinematic and dynamic. The kinematic type will be discussed here.

Select the Kinematic Scenario icon.  The *Kinematics Scenario* window appears.

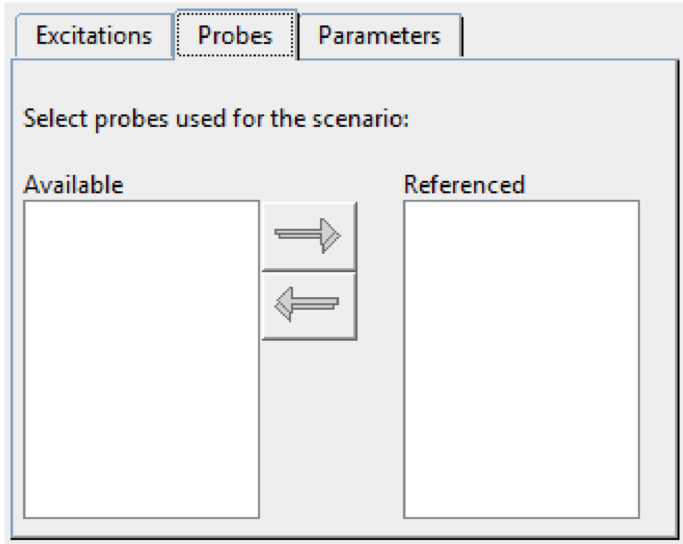


Name Specifies a name for the scenario

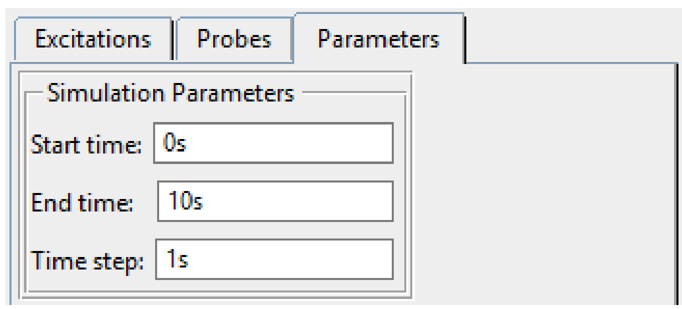
Mechanism Specifies the mechanism to be used for the scenario

Excitations Allows you to specify which excitations will be used for the scenario

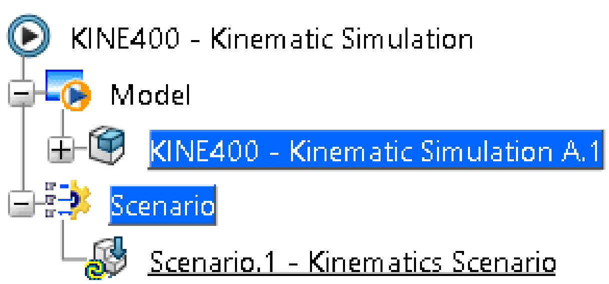
Probes Allows you to specify which probes will be used for the scenario



Parameters Defines the simulation parameters for the scenario



Select *OK*. There are no excitations or probes to select yet. The scenario is created.

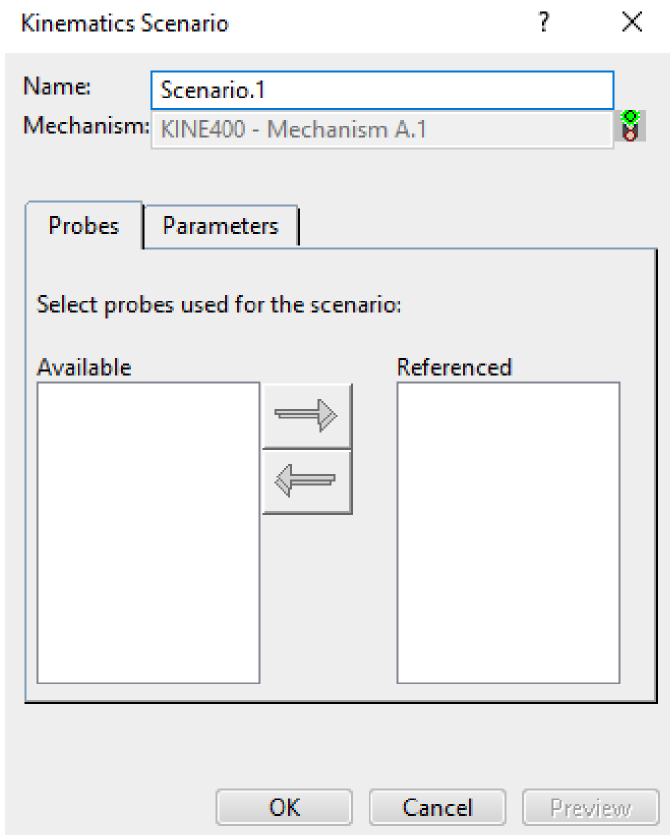


Select Undo until the scenario disappears.  Generally, you won't use the Kinematic Scenario icon unless you already have excitations you can select from. In this case you haven't created any excitations yet, so it will be more efficient to create the scenario while recording the excitation.

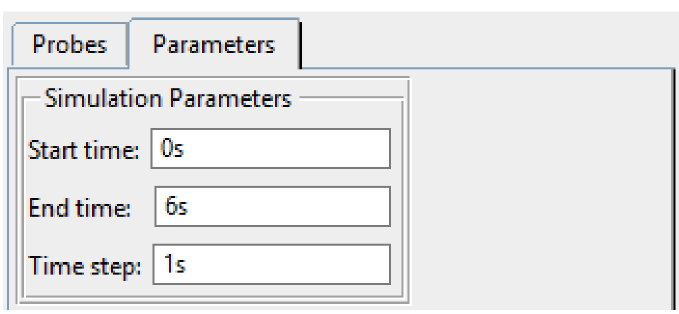
Recording a Scenario

You can manually or automatically record a scenario of your kinematic mechanism moving to create a recorded excitation. If you do not already have a kinematic scenario, this icon will create one in the process.

Select the Kinematic Excitation Recorder icon.  The *Kinematics Scenario* window appears again. Notice it looks the same, except there is no *Excitation* tab.

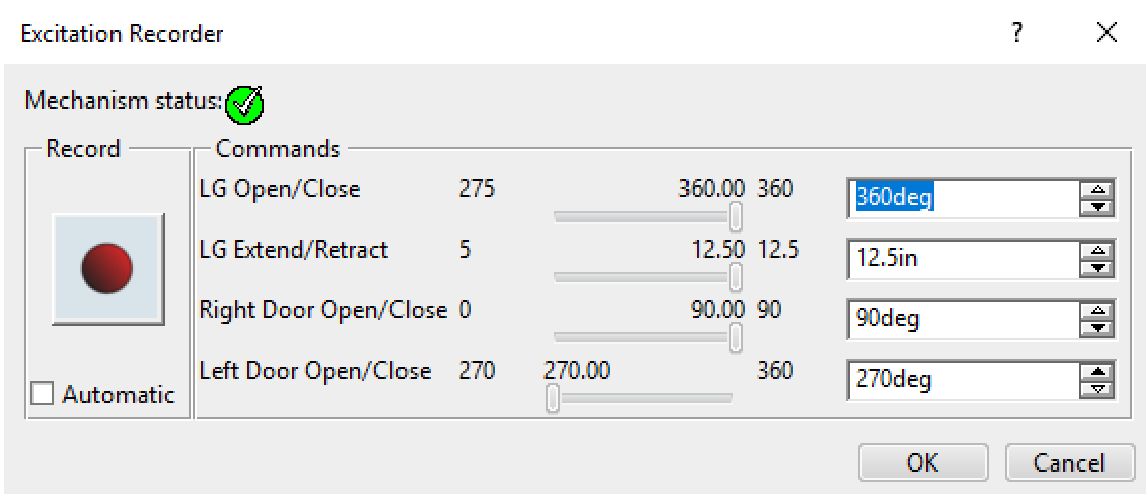


Select the *Parameters* tab and set the values as shown.



In this case, the recording will have a duration of 6 seconds with the position of the mechanism being recorded each second.

Change the *Name* to <u>Recorded Scenario</u> and select *OK*. The *Excitation Recorder* window appears.



This window lists all of the commands available within the selected mechanism. You may manually record the mechanism movements you want or you can turn on the Automatic option and simply drag the sliders for the recording.

Be sure the *Automatic* option is turned off and select the Record icon.  This will record the current position of the commands as the first step.

For the *LG Open/Close* command, key in 345 or drag the slider to approximately 345. This will be the position for the command in the second step.

Key in 11.5 or drag the slider to the approximate value for the *LG Extend/Retract* command.

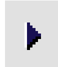
Select the Record icon.  This is the positioning for the second step.

Repeat this process for the pairs of values below. This should have each of the positions recorded for the entire excitation.

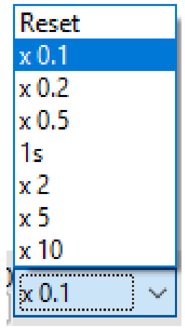
LG Open/Close	**330**	**315**	**300**	**285**	**275**
LG Extend/Retract	**10.5**	**9.5**	**8.5**	**7.5**	**7.0**

Notice the *Experience Player* toolbar is available in the lower left hand corner of the display.

Select the Skip to Beginning icon. 

Select the Play icon.  The simulation plays.

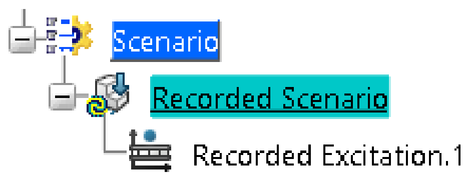
Change the time drop down to be *x 0.1s*.



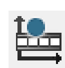
Select the Skip to Beginning icon. 

Select the Play icon.  The simulation plays slower.

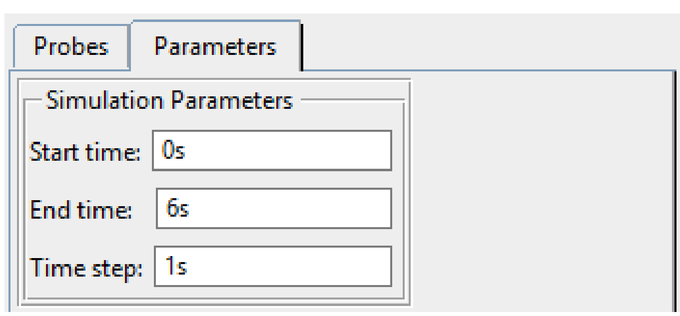
Select *OK* in the *Excitation Recorder* window. The scenario and excitation are created.



Now a second scenario will be created using the automatic record.

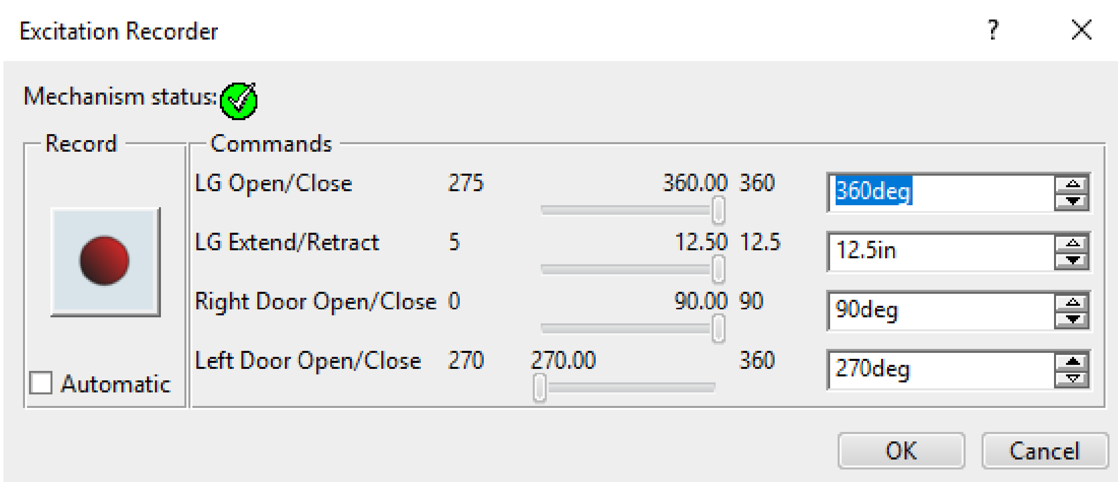
Select the Kinematic Excitation Recorder icon.  The *Kinematics Scenario* window appears.

Select the *Parameters* tab and set the values as shown.



In this case, the recording will have a duration of 6 seconds with the position of the mechanism being recorded each second.

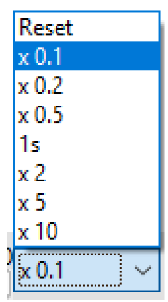
Change the *Name* to <u>Auto Recorded Scenario</u> and select *OK*. The *Excitation Recorder* window appears.



Change the *LG Extend/Retract* to be 8.5 and select the Record icon.  This will be the first step.

Turn on the *Automatic* option and drag the *LG Open/Close* command in approximately 15 degree increments, releasing it each time. Each step will be recorded automatically.

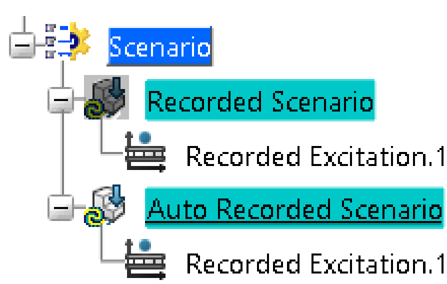
Change the time drop down to be *x 0.1s*.



Select the Skip to Beginning icon. 

Select the Play icon.  The simulation plays.

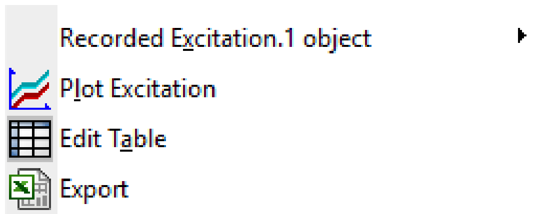
Select *OK* in the *Excitation Recorder* window. The scenario and excitation are created.



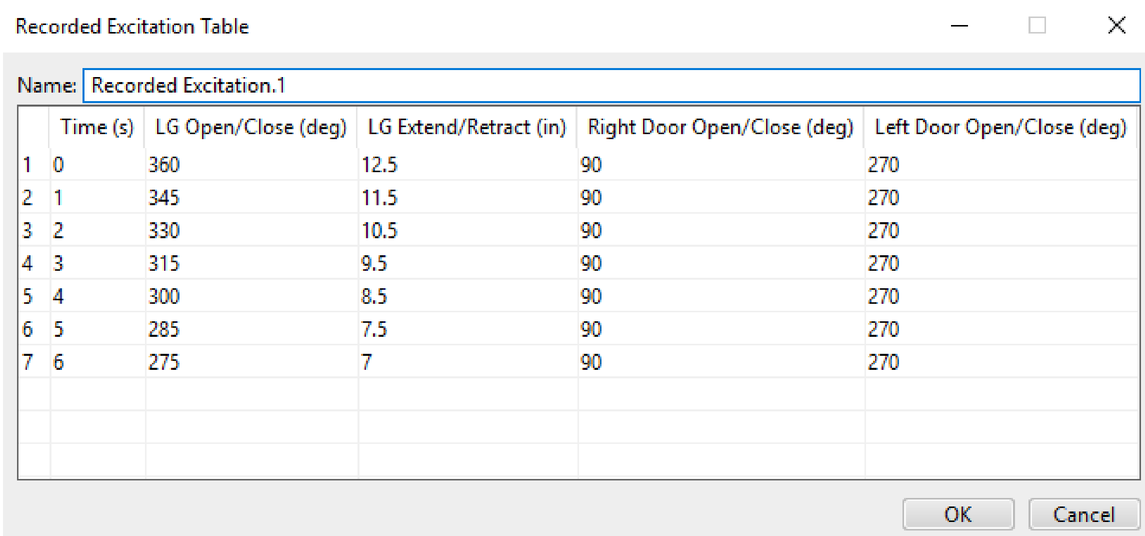
Editing a Recorded Scenario

You can edit a recorded scenario in order to adjust the command values if needed.

Right select on the *Recorded Excitation.1* for the *Recorded Scenario*. A contextual menu appears.



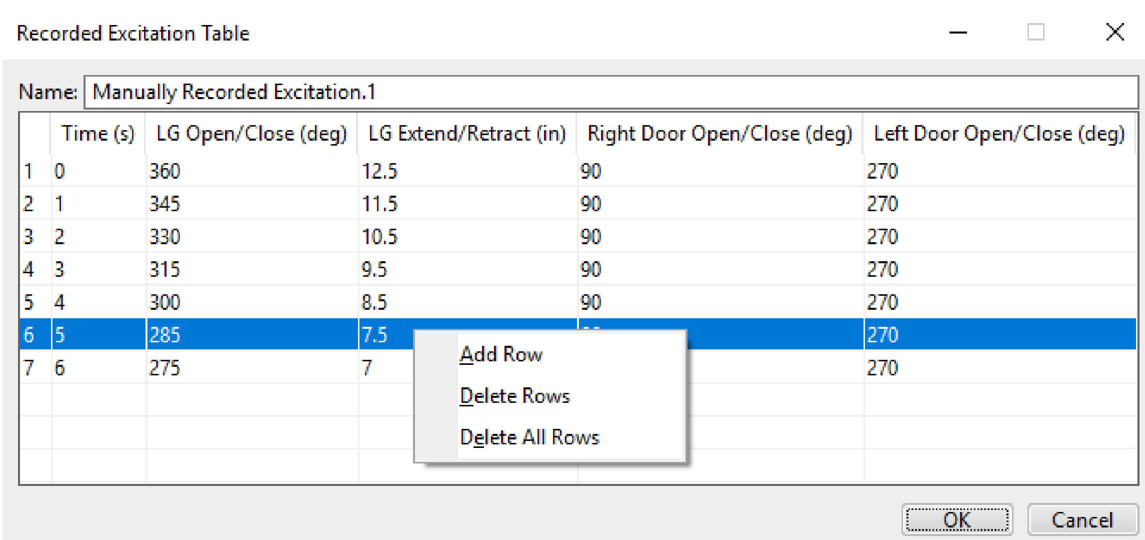
Select the *Edit Table* option. The *Recorded Excitation Table* window appears.



	Time (s)	LG Open/Close (deg)	LG Extend/Retract (in)	Right Door Open/Close (deg)	Left Door Open/Close (deg)
1	0	360	12.5	90	270
2	1	345	11.5	90	270
3	2	330	10.5	90	270
4	3	315	9.5	90	270
5	4	300	8.5	90	270
6	5	285	7.5	90	270
7	6	275	7	90	270

Change the *Name* to <u>Manually Recorded Excitation.1</u>.

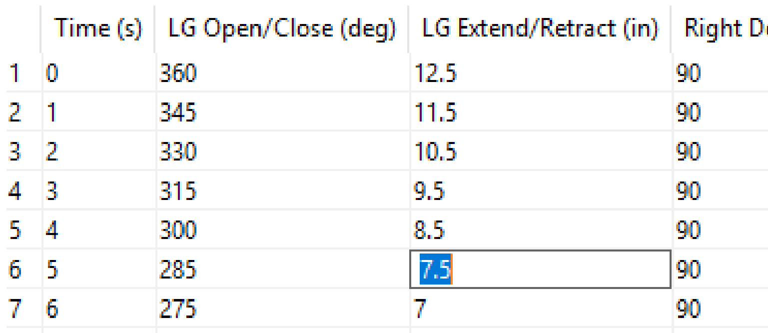
Select row 6, then right select in the table. A contextual menu appears.



	Time (s)	LG Open/Close (deg)	LG Extend/Retract (in)	Right Door Open/Close (deg)	Left Door Open/Close (deg)
1	0	360	12.5	90	270
2	1	345	11.5	90	270
3	2	330	10.5	90	270
4	3	315	9.5	90	270
5	4	300	8.5	90	270
6	5	285	7.5		270
7	6	275	7		270

You can add or delete rows from the table. You can also simply modify the existing values.

Select on the 7.5 value in the *LG Extend/Retract* column. You should be able to edit the value. You may have to select on it twice.

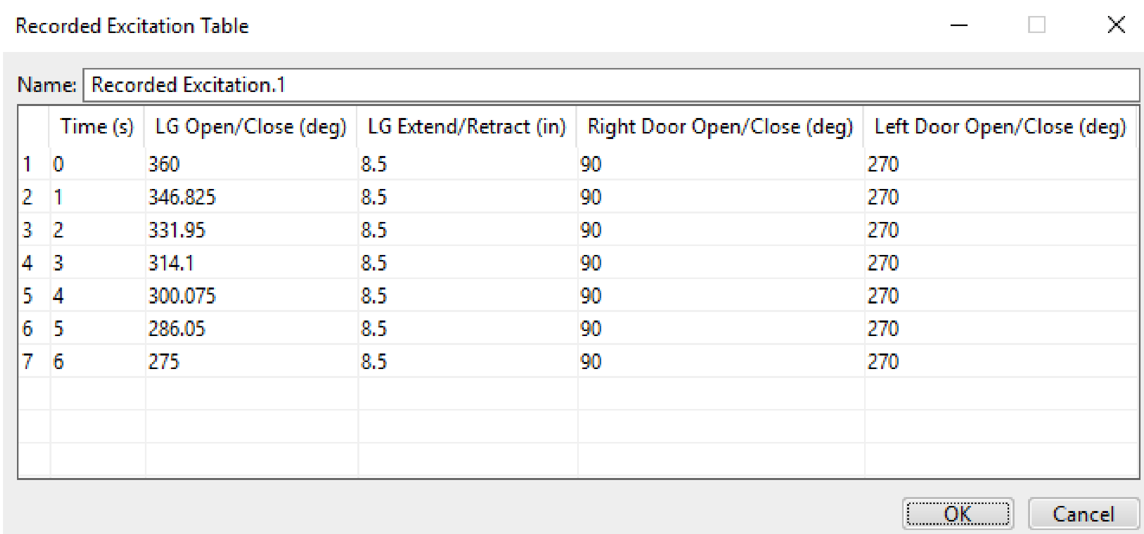


	Time (s)	LG Open/Close (deg)	LG Extend/Retract (in)	Right Dc
1	0	360	12.5	90
2	1	345	11.5	90
3	2	330	10.5	90
4	3	315	9.5	90
5	4	300	8.5	90
6	5	285	7.5	90
7	6	275	7	90

Change it to 8.0 and select off of it to set it. This will update the position of the command for that step.

Select *OK*.

Right select on the *Recorded Excitation.1* for the *Auto Recorded Scenario* and select *Edit Table*. The *Recorded Excitation Table* appears.



Recorded Excitation Table

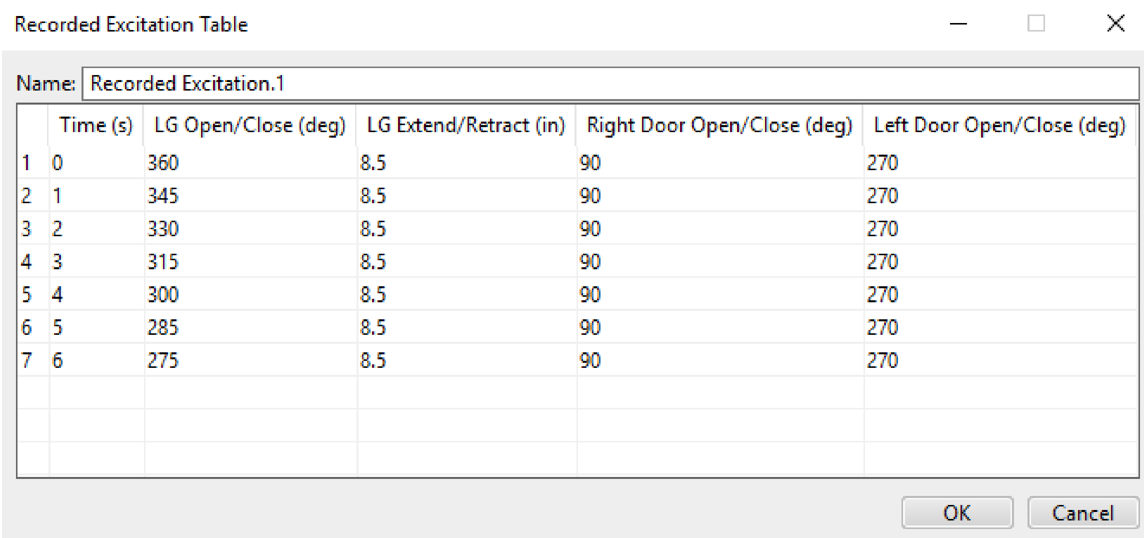
Name: Recorded Excitation.1

	Time (s)	LG Open/Close (deg)	LG Extend/Retract (in)	Right Door Open/Close (deg)	Left Door Open/Close (deg)
1	0	360	8.5	90	270
2	1	346.825	8.5	90	270
3	2	331.95	8.5	90	270
4	3	314.1	8.5	90	270
5	4	300.075	8.5	90	270
6	5	286.05	8.5	90	270
7	6	275	8.5	90	270

OK Cancel

Notice all of the values in the *LG Open/Close* column are random since they were automatically recorded using the slider.

Update the values as shown.


Recorded Excitation Table

Name: Recorded Excitation.1

	Time (s)	LG Open/Close (deg)	LG Extend/Retract (in)	Right Door Open/Close (deg)	Left Door Open/Close (deg)
1	0	360	8.5	90	270
2	1	345	8.5	90	270
3	2	330	8.5	90	270
4	3	315	8.5	90	270
5	4	300	8.5	90	270
6	5	285	8.5	90	270
7	6	275	8.5	90	270

OK Cancel

Change the *Name* to be <u>Auto Recorded Excitation.1</u> and select *OK*.

This illustrates how you can record an excitation for a scenario.

Save and close the document.

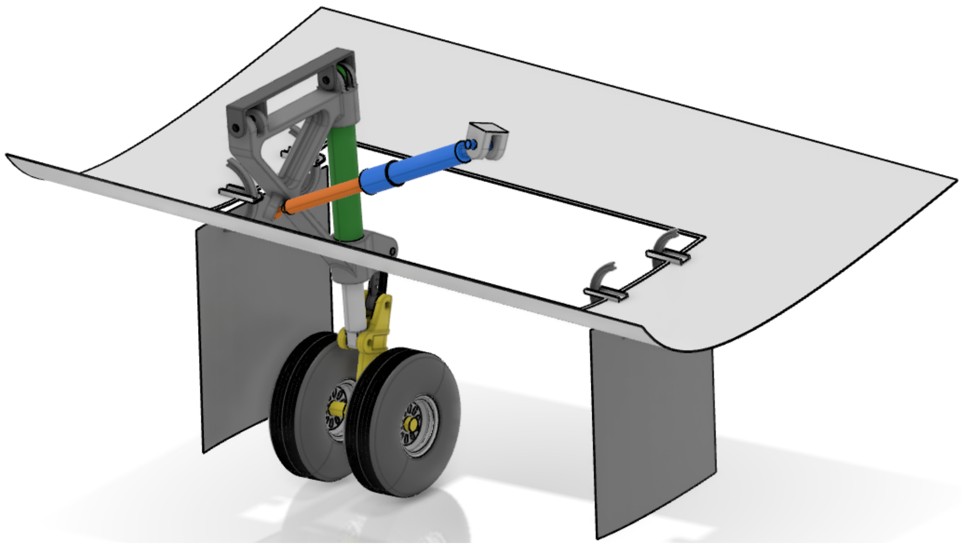
Results

This section will discuss a number of operations with regards mechanism manipulation and analysis.

Preview Scenario

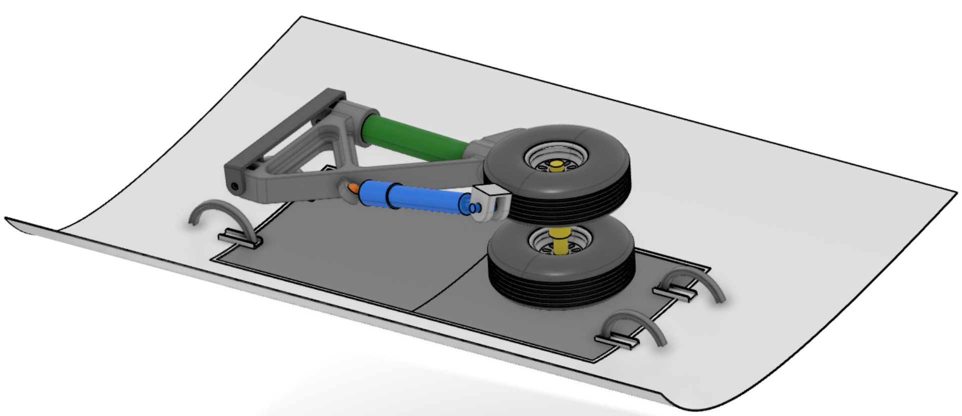
The preview scenario option allows you to review the behavior of a mechanism.

Open the KINE450 - Results model. The model should appear as shown.



Select the Preview Scenario icon.  The player controls should appear as shown.

Change the time step to *x 0.1* and select the Play icon. The assembly should move as shown.

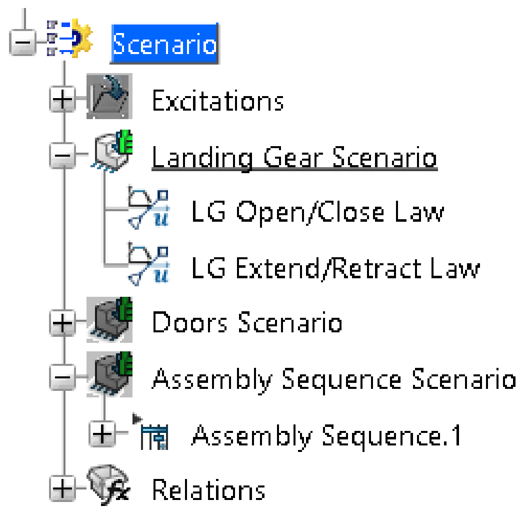


Select the Esc key. This provides a quick easy way to play a scenario.

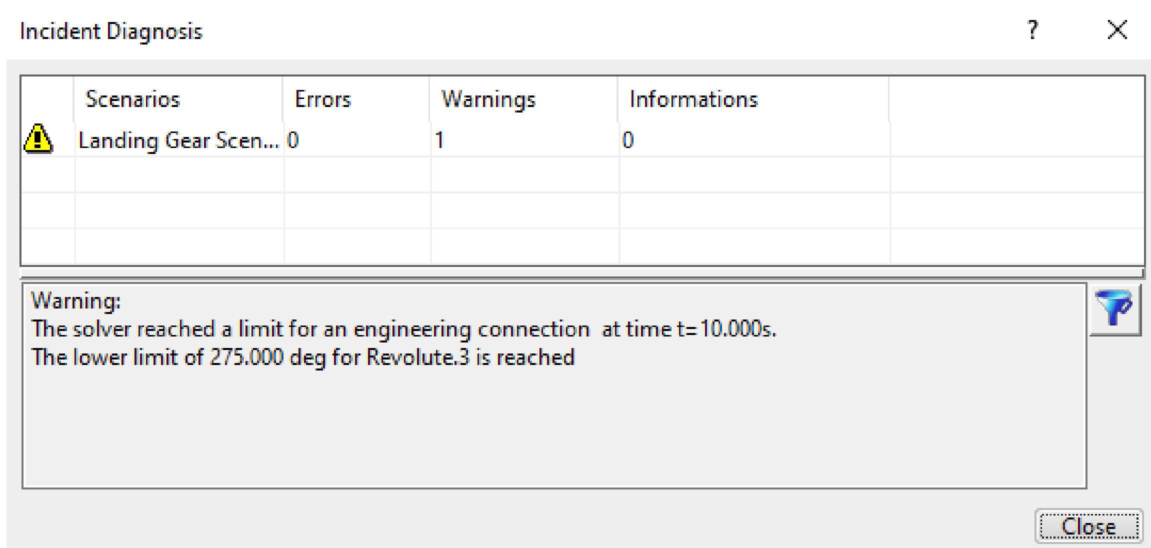
Compute and Generate Results with Local Update

The compute and generate results with local update option allows you create a result computation of the current scenario. If results have already been generated, this option can also be used to update them if any local modifications have been made to the scenarios involved.

Right select on the *Landing Gear Scenario* and select *Set as Current*. This scenario will be used for this exercise.

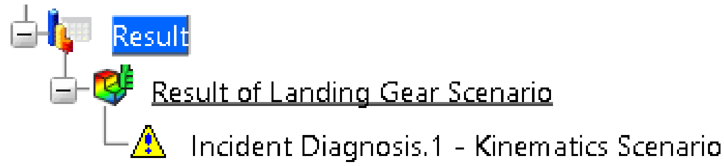


Select the Compute and Generate Results with Local update icon.  You should see the scenario play and you may receive a warning.

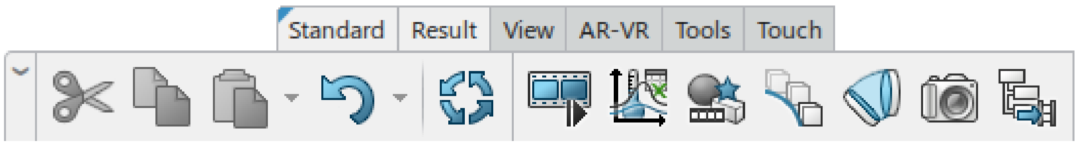


Close the *Incident Diagnosis* window. This is just stating a limit was reached during the simulation.

You now have a *Result* branch in the tree as well.



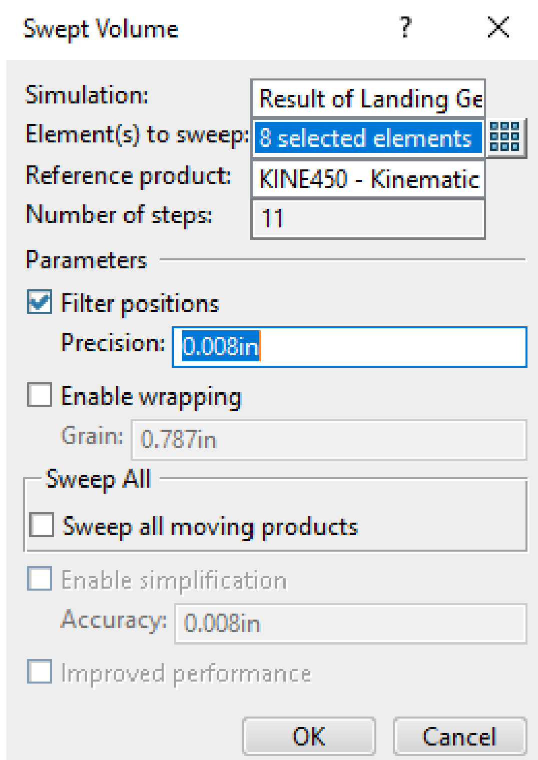
The *Result* branch is active and you should see that you have a new toolbar.



To illustrate a change being made locally, a swept volume will be created.

Select the Swept Volume icon.  The *3D Shape* window appears.

Select *OK* to create the new 3d shape. The name doesn't matter as this won't be saved in this exercise. The *Swept Volume* window appears.



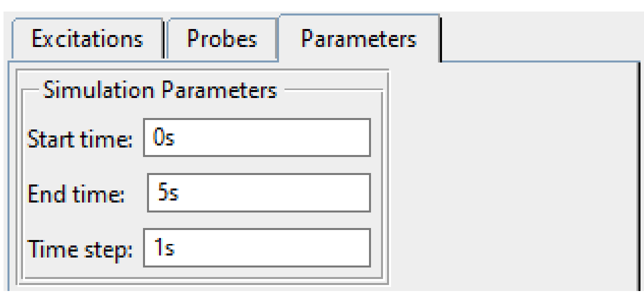
This window will be discussed in detail later.

Select *OK*. The swept volume is generated and should appear in the new 3d shape model.

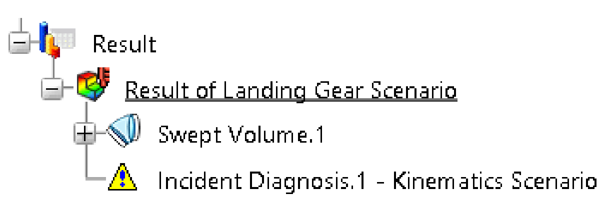
Switch back to the *KINE450 - Results* model.

Double select on the *Landing Gear Scenario* in the tree. The *Kinematics Scenario* window appears. You may have to double select twice.

In the *Parameters* tab, change the *End time* to 5s and select *OK*. This will limit the scenario to only the first half of the simulation.



Notice the result is no longer up to date.



Select the Compute and Generate Results with Local update icon.  You should see the scenario play, but only half way this time.

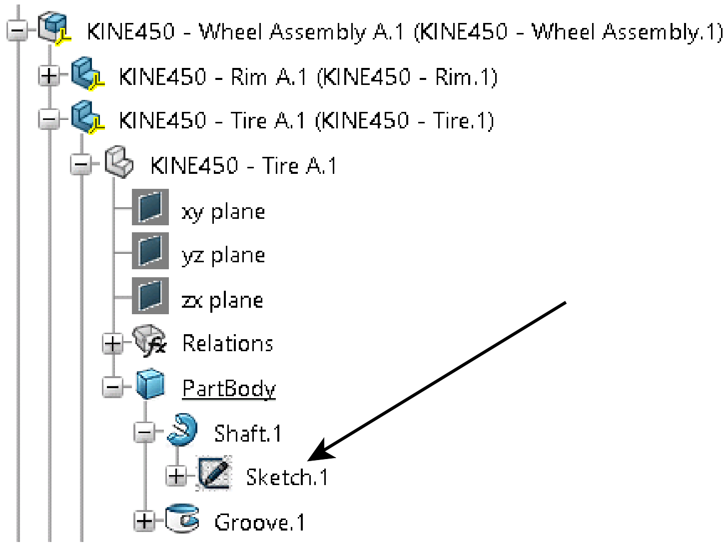
Switch to the 3d shape tab with the swept volume. Notice it has been updated.

Now you will make a change at the model level and see how the global update option works.

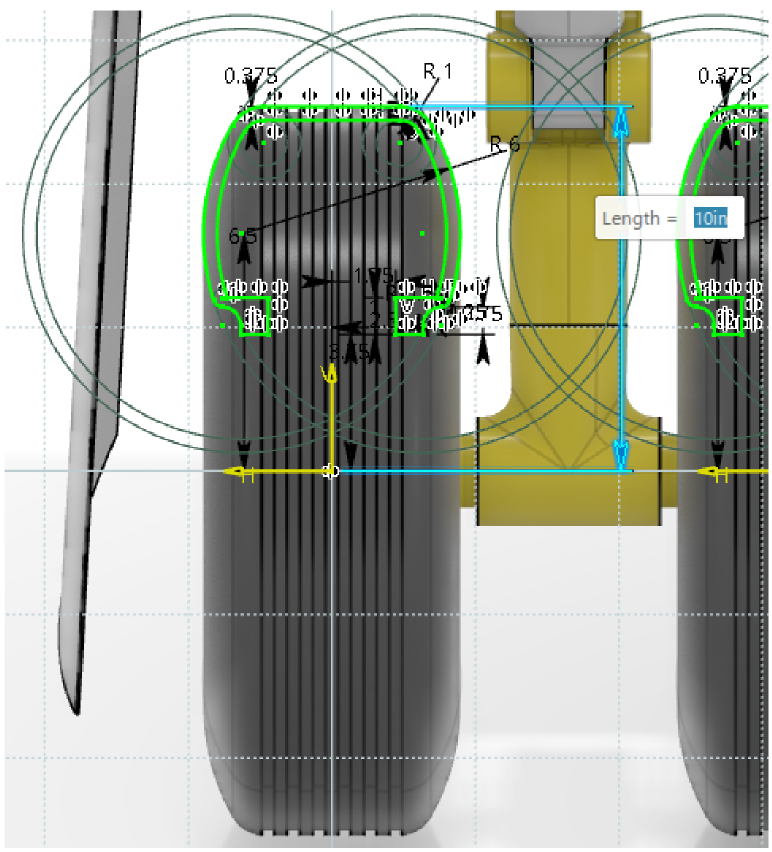
Compute and Generate Results with Global Update

The compute and generate results with local update option allows you create a result computation of the current scenario. If results have already been generated, this option can also be used to update them if any local modifications have been made to the scenarios involved.

Edit *Sketch.1* as shown in the *KINE450 - Tire* model.

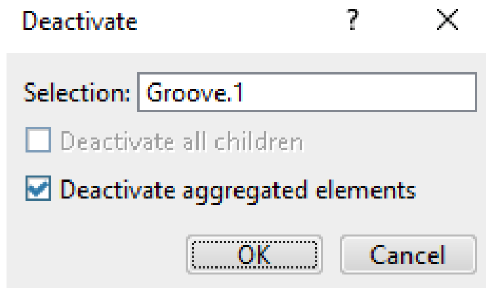


Change the 10in length constraint to be 9 inches.



Exit the sketch.

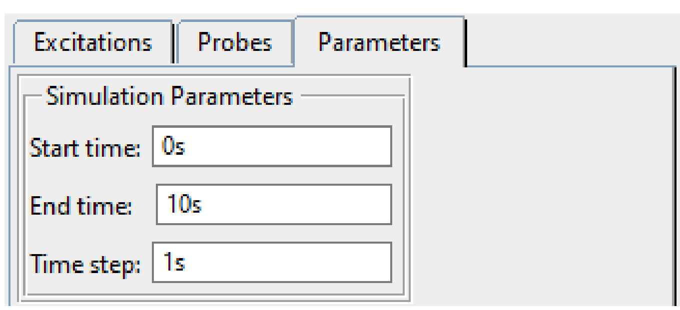
Right select on *Groove.1* in the tree and select *Groove.1 object, Deactivate*. The *Deactivate* window appears.



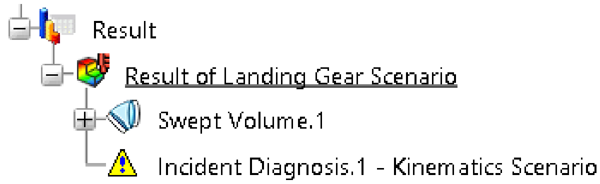
Select *OK*. The groove operation is deactivated and the grooves are removed from the tries.

Double select on the *Landing Gear Scenario* in the tree. The *Kinematics Scenario* window appears. You may have to double select twice.

In the *Parameters* tab, change the *End time* to 10s and select *OK*. This will set the scenario back to play the full simulation.



Select the Update icon. Notice the result is still not up to date.



Select the Compute and Generate Results with Global update icon. You should see the scenario play and you may receive a warning.

Close the *Incident Diagnosis* window. This is just stating a limit was reached during the simulation.

Switch to the 3d shape tab with the swept volume. Notice it has been updated, including the change the tire design.

View Scenario Results

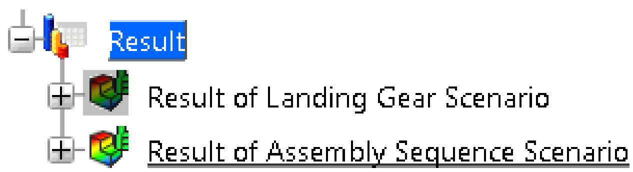
The view scenario results option allows you to review the results of a kinematic scenario including probe and excitation parameters. The results can be viewed in plot or table format and can also be exported.

Double select on the *Scenario* branch in the tree. You should be returned to the Mechanical Systems Experience app.

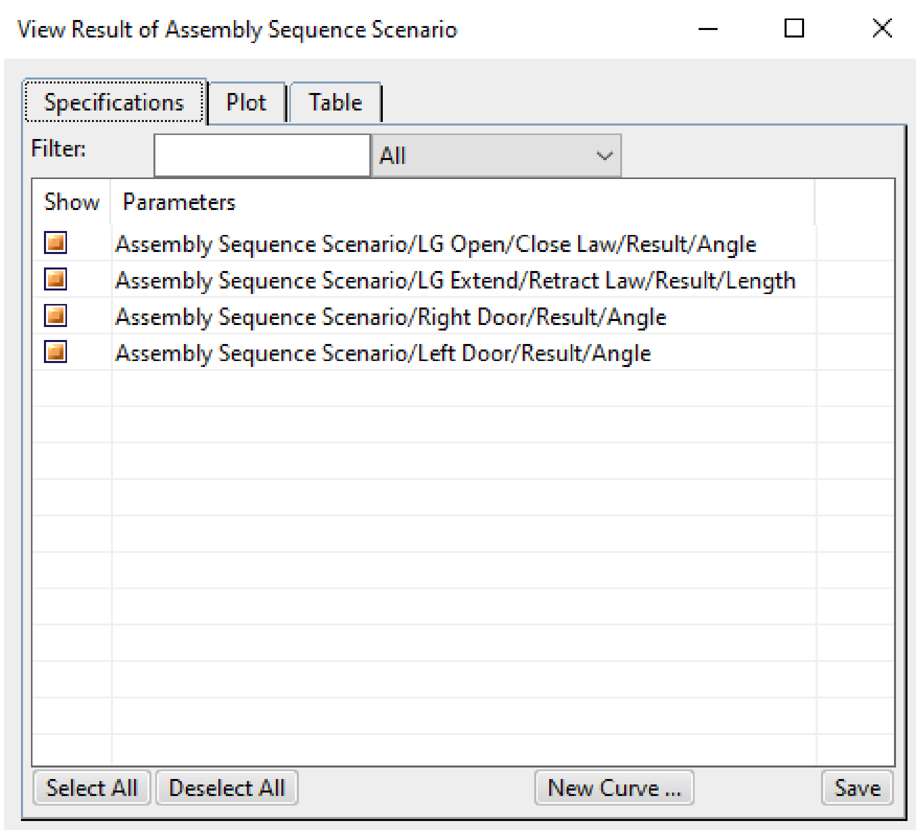
Right select on the *Assembly Sequence Scenario* and select *Set as Current*.

Select the Compute and Generate Results with Global update icon.  You should see the scenario play and you may receive a warning. You could have used either the local or global update option in this case, since it is creating a new result for the scenario.

Close the *Incident Diagnosis* window. This is just stating limits were reached during the simulation. Another result appears in the *Result* branch.



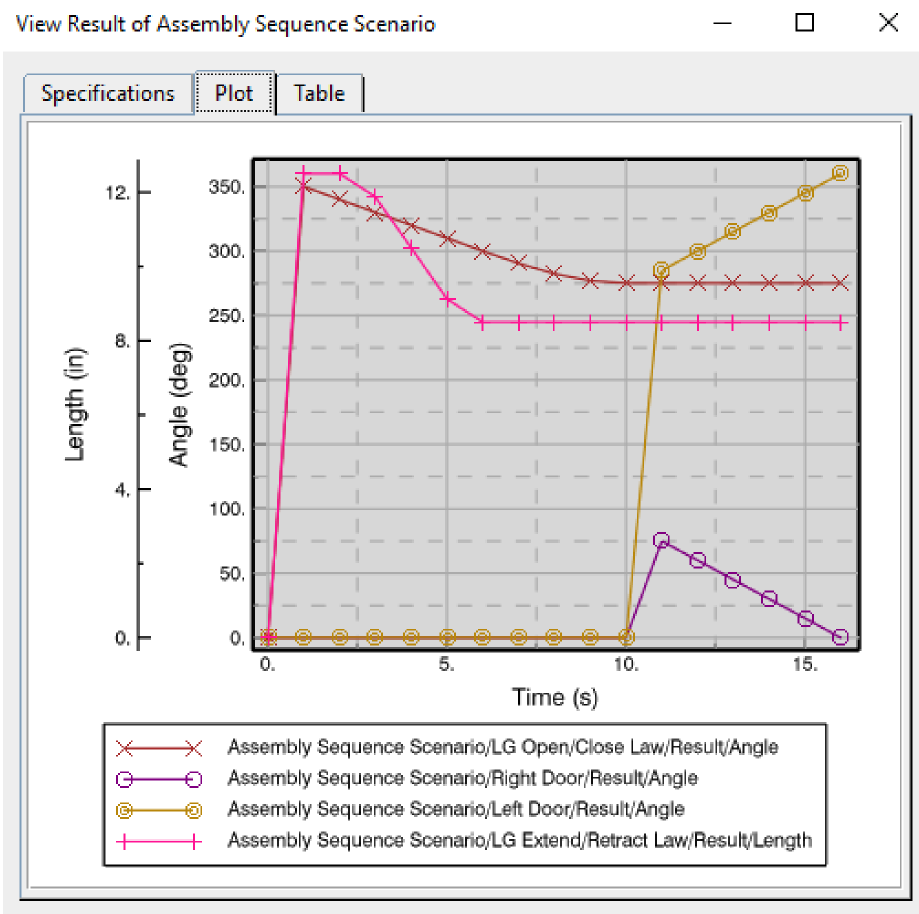
Select the View Scenario Results icon.  The *View Result* window appears.



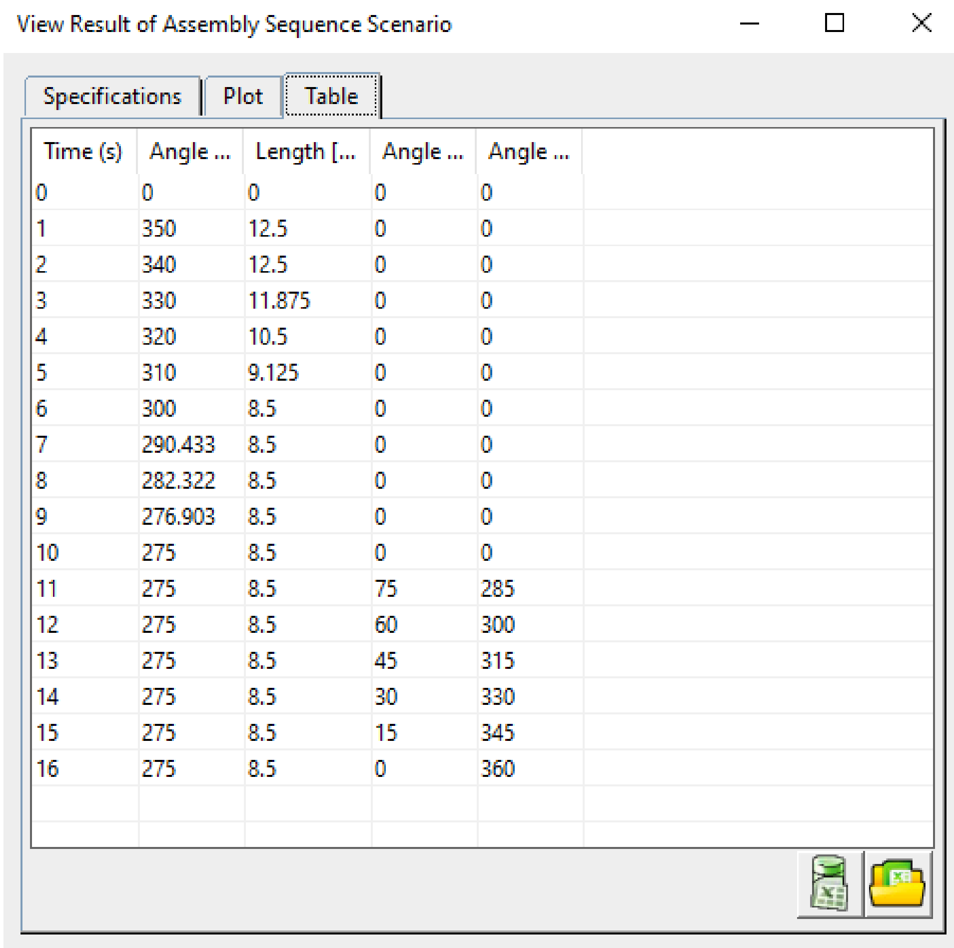
Specifications Displays the available probe and excitation parameters for the scenario

New Curve Allows additional plots to be created between the existing parameters

Save Saves the plot results in the tree



Plot Displays the plots of the selected parameters



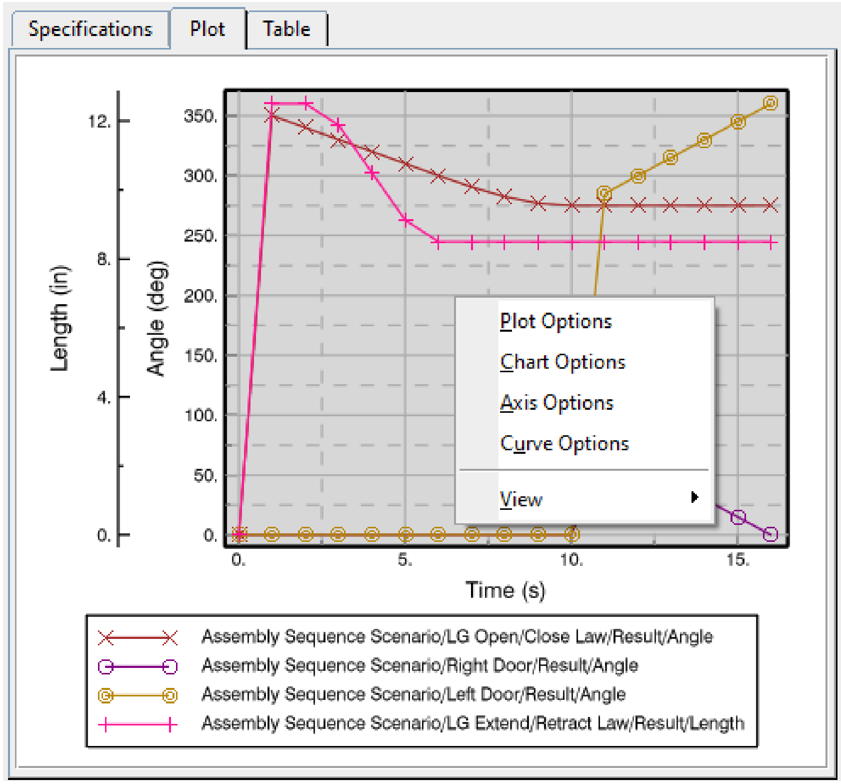
Time (s)	Angle ...	Length [...	Angle ...	Angle ...
0	0	0	0	0
1	350	12.5	0	0
2	340	12.5	0	0
3	330	11.875	0	0
4	320	10.5	0	0
5	310	9.125	0	0
6	300	8.5	0	0
7	290.433	8.5	0	0
8	282.322	8.5	0	0
9	276.903	8.5	0	0
10	275	8.5	0	0
11	275	8.5	75	285
12	275	8.5	60	300
13	275	8.5	45	315
14	275	8.5	30	330
15	275	8.5	15	345
16	275	8.5	0	360

Table Displays the values of the parameters in table format

 Export to Database — Exports the table to a spreadsheet saved in the database

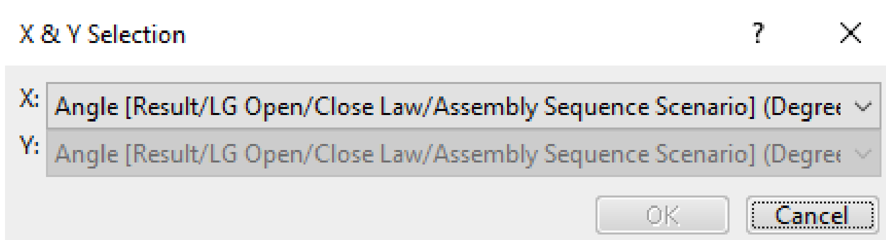
 Export to File — Exports the table to a spreadsheet saved locally

Select the *Plot* tab, then right select in the plot. A contextual menu appears.



These options will allow you to modify what the plot looks like.

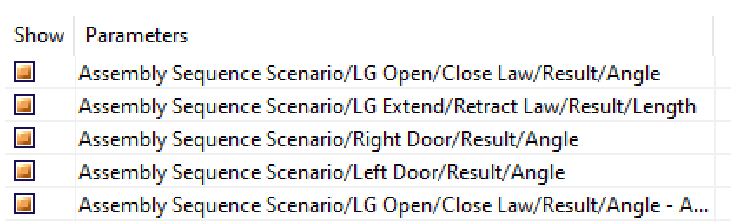
Select the *Specifications* tab and then select the *New Curve* button. The *X & Y Selection* window appears.



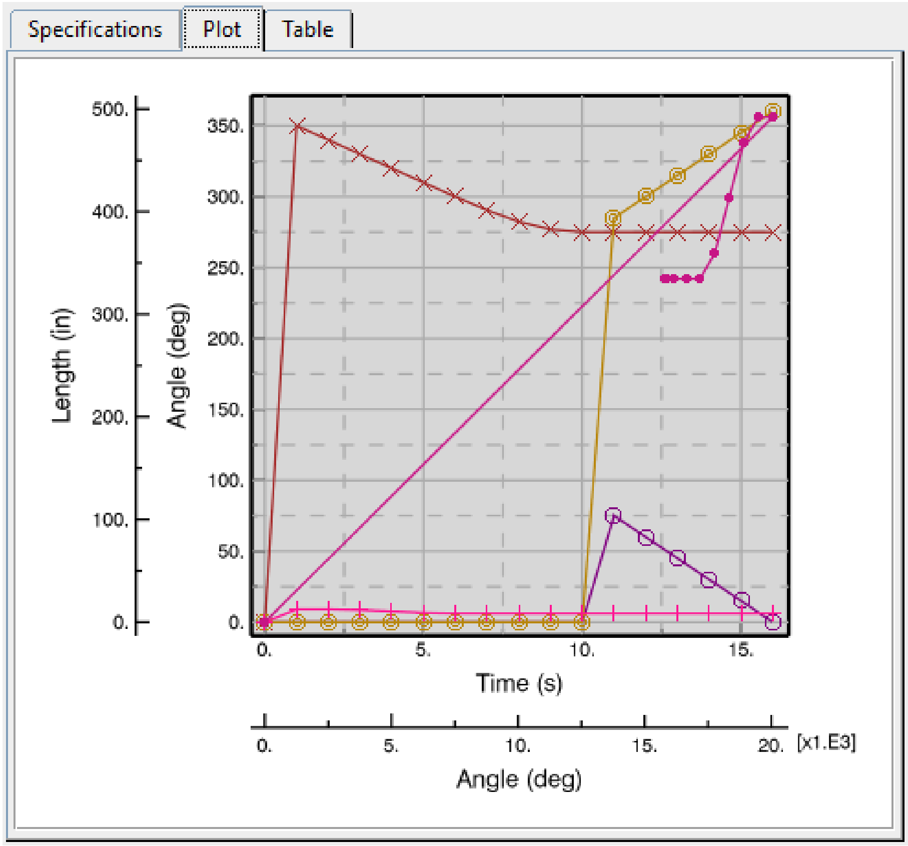
Select the drop down for the *X* and select the first *Angle* parameter.

Select the drop down for the *Y* and select the *Length* parameter.

Select *OK*. The new option for the two parameters vs one another appears in the list.



Select the *Plot* tab. Notice the new curve.



Close the window.

Close all documents without saving.