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Introduction

CATIA Version 5 Fitting Simulation

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

- Creating tracks
- Editing sequences
- Creating replays and playing them back
- Using pathfinder to determine a path
- Performing clash analysis during a fitting simulation

Fitting Simulation

Fitting simulation often gets confused with Kinematics because it involves motion. The main difference between a fitting simulation and a kinematic simulation is that a fitting simulation moves according to a set path without regard to constraints. A kinematic simulation moves according to defined joints which control the motion of an assembly.

The purpose of fitting simulation is to show how to assemble or disassemble an assembly. This can be useful to create training videos, or just to make sure that it is possible to disassemble or assemble a product in the defined space. The analysis available in Assembly Design only allows for a clash analysis between objects in their static state. In Fitting Simulation, you can perform a clash analysis between objects as they are being disassembled or assembled. Kinematics can perform a clash analysis while an assembly operates.

Since Fitting Simulation does not involve constraints, assembly constraints are unnecessary in order to perform a simulation. However, you would normally want to constrain your assembly in order to have all of the parts located correctly.

Fitting Simulation

DMU Fitting Simulation is used to record manipulations made to assemblies. You can then use the replays generated for various things, such as analyses, instructional purposes, or demonstrations.

Open the Basic Fitting document located in the *Basic Fitting* directory. This model is a pre-defined assembly. All the constraints are in place and there are no clashes. You will use this assembly to get a feel for what fitting simulation can do.

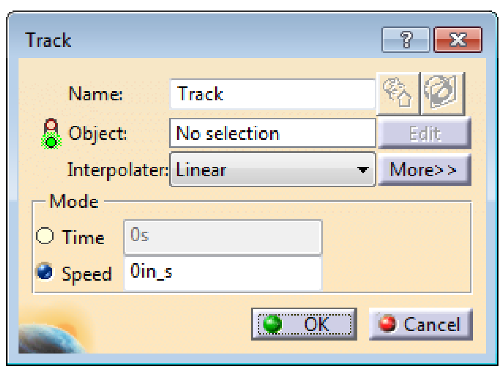
Assemblies that are going to be put into a fitting simulation do not need to be constrained. In fact, fitting simulation ignores all constraints. To make your assembly move according to constraints, you would need to apply kinematic joints.

Creating Tracks

In order to create a simulation, a series of tracks needs to be created. These tracks identify the steps, or movements, of individual parts. The individual track identifies the path for the object, and then a sequence is put together for a collection of tracks and actions.

The first step will be to remove the pulley from the compressor.

Select the Track icon.  The *Recorder* and *Player* toolbars and the *Track* window appear.



Name Allows you to name the track

Object Defines the object that is to follow the track. Only one object can be in the track at a time

Edit Allows you to edit the object

Bind Analysis  Allows you to attach an analysis to the track

Associate Section  Associates a section cut to the track

Interpolater Defines the point interpolation for the track

Time Enables movement to be based on an amount of time for the track

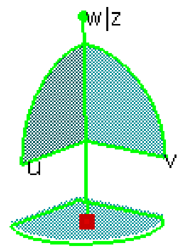
Speed Enables movement to be based on a given speed for the track

Select the *Pulley*. This will snap the compass to the geometric center of the pulley. The compass will be used to manipulate the pulley. The *Manipulation* toolbar appears; this will be discussed later.

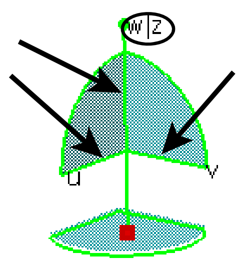
Before utilizing the compass, you will take a closer look at what the compass is, and how it is used.

Compass

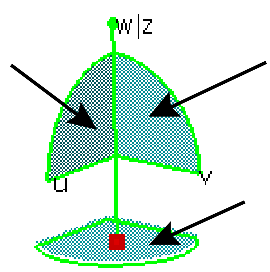
It is important to understand the compass so that it can be used effectively.



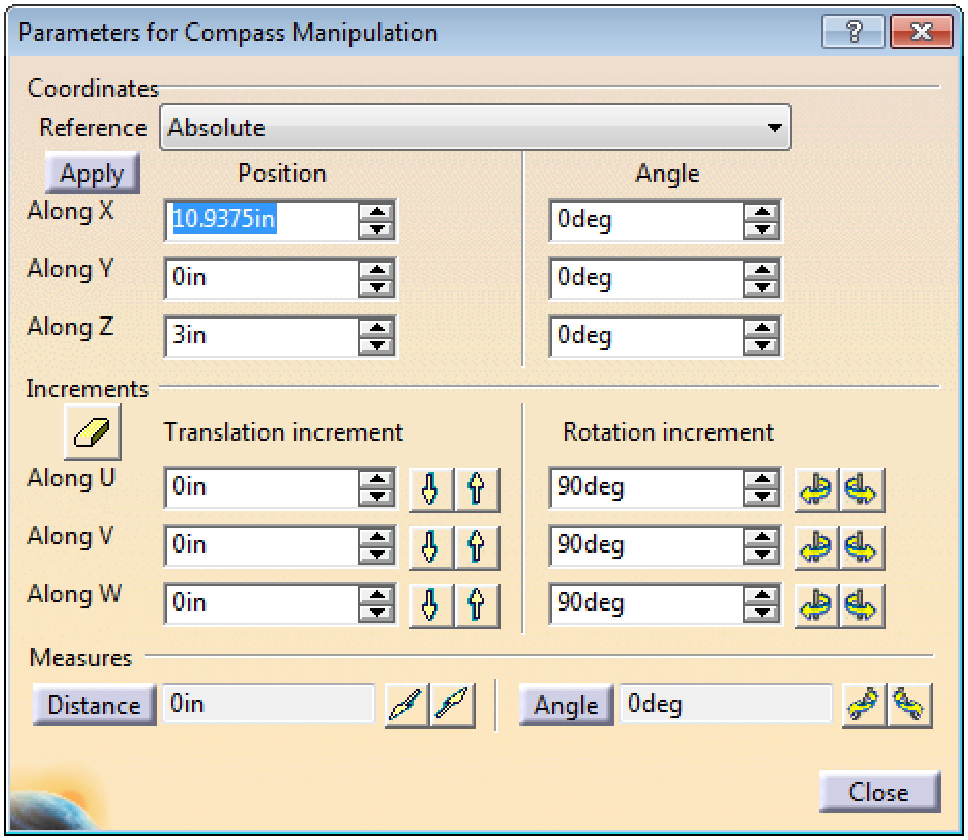
The compass, usually green when attached to geometry, has three main direction vectors: U, V, and W, as shown by the arrows below. Selecting one of the vectors will move the attached object only in the direction of the vector. Normally, the compass will be in the upper right corner of the workspace, labeled X, Y, and Z. When the compass is attached to an object, it is re-labeled to U, V, and W. If one of the vectors is aligned with one of the primary axes, then there will be a pipe (|) symbol and the primary axis is also stated. This can be seen below, circled.



The compass also has the three principle planes, as noted by arrows below. Selecting one of these planes will allow the attached object to move along the selected plane. The rotation arcs, located along the outside of the planes, work the same. Selecting an arc will rotate the object about one of the given axes.



Dragging the compass is only a guess. You do not know how far the compass has moved, and cannot make precise movements. However, if you double select anywhere on the compass, you will get the *Parameters for Compass Manipulation* window. This will allow you to make more exact movements. Take a quick look.



Coordinates

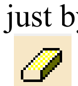
Apply — Any time the position values have been changed, you have to select the *Apply* button in order for the changes to take effect

Reference — Defines the reference element for the coordinates

Position — Gives the actual position of the compass with respect to the *Reference*

Along X, Y, Z — Shows the X, Y, and Z value of the center (red dot) of the compass

Angle — Shows the rotation angles of the compass about the X, Y, and Z axes of the *Reference*

Increments — This allows the compass to be manipulated by increments, not just by set values. The icon will reset all of the increment values to zero. 

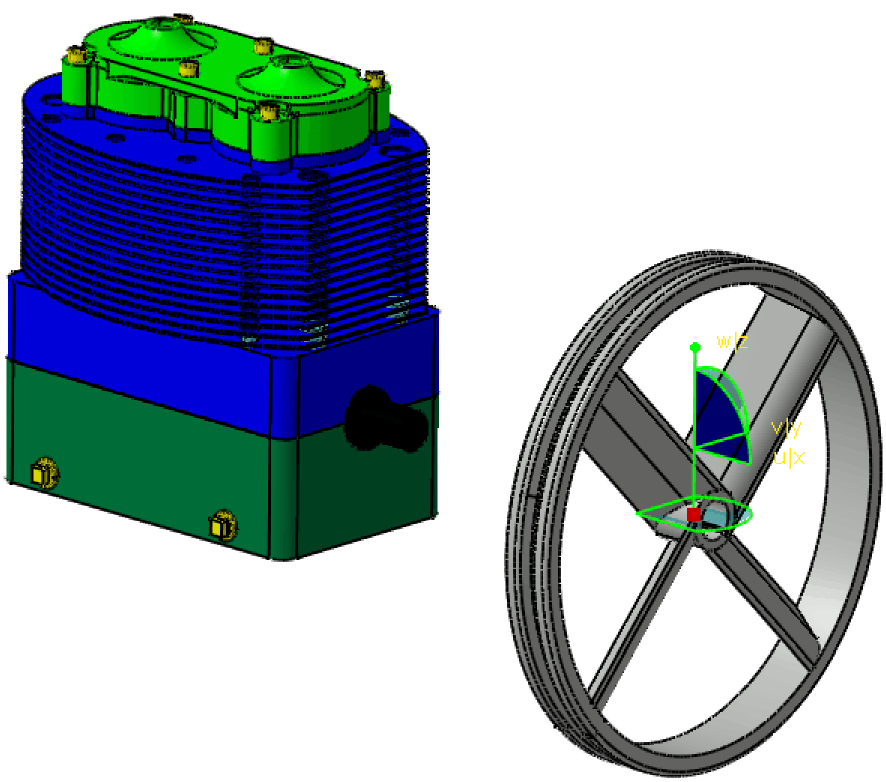
Translation increment	This is an incremental translation along one of the axes. Selecting the down (negative direction) or up (positive direction) arrows next to the value field will move the compass the specified amount in that direction
Rotation increment	This is an incremental angle movement about one of the axes, again with a negative or positive rotation

Measures Allows you to measure a distance or angle

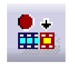
Distance	This will allow you to measure the distance between two elements. You can then move along that direction in either negative or positive increments of the measured distance
Angle	This will allow you to measure the angle between two elements. You can then rotate in either negative or positive increments of the measured angle

With all that in mind, you are going to start manipulating the track.

Select and hold the *U* axis vector and drag the pulley away from the part. The U axis vector is noted as *U|X*, meaning it is parallel to the X axis. Move the cursor to the vector so that it highlights, then just select and hold it while moving the mouse away from the compressor body. Since you selected the U axis, the pulley will only move in the U axis direction. The pulley should look something like the following picture.

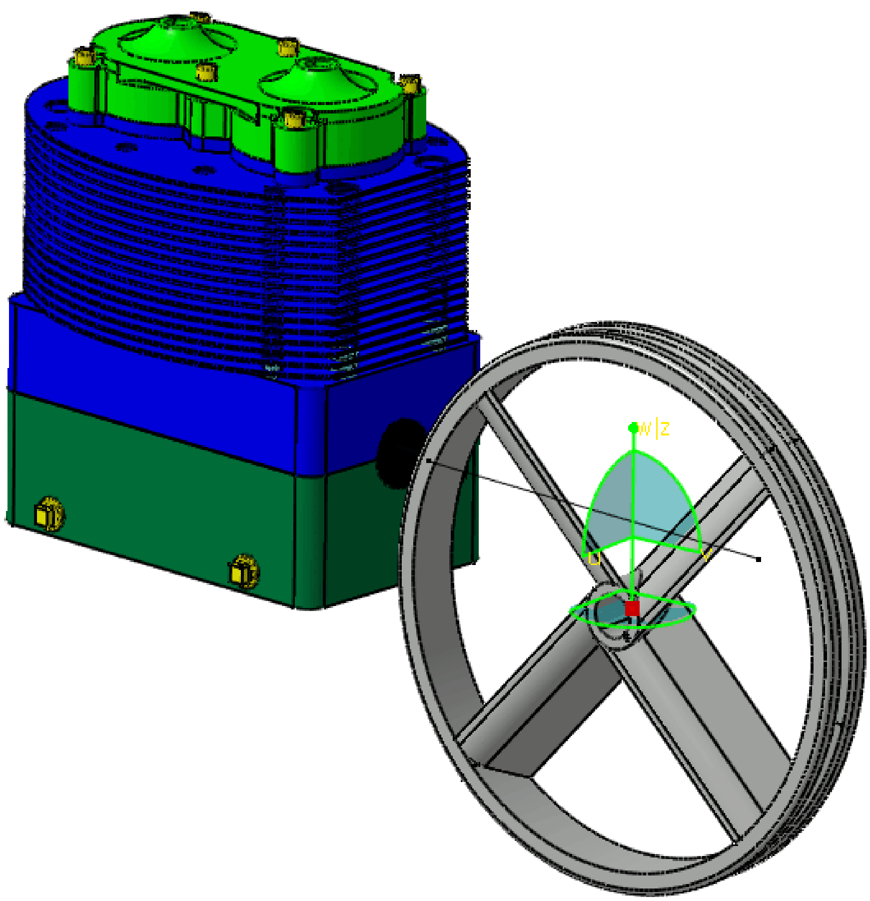


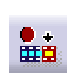
Now the position needs to be inserted into the simulation.

Select the Record icon.  This icon can be found in the *Recorder* toolbar. This will create the first step of the track. Notice the line running from the original position to the new position. You should also note the time has changed. The time may vary, but this is not a concern as to the exact number.

Rotate the pulley about the W axis approximately 90°. This is done by selecting the rotation handle at the bottom of the compass and dragging it around the W axis.

Drag the pulley along the U axis a short distance. The pulley and compressor should appear similar to the following image.



Select the Record icon again.  This will insert the next step. Now view the track that was created.

Select the Skip to Begin icon.  This is located in the *Player* toolbar. This will snap the pulley back to the beginning of the track.

Select the Parameters icon.  This is located in the *Player* toolbar. The *Player Parameters* window appears.

Change the *Sampling Step* to 5s and select the Play Forward icon. The pulley will now move along the track in increments of 5s per step.

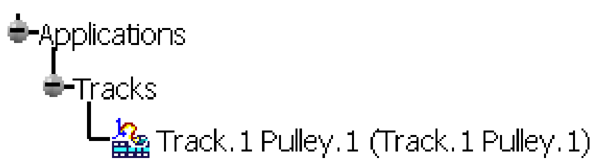
Change the *Sampling Step* to 3s and select the Play Backward icon. Now the pulley moves back to the original position, this time slower.

Here is what is going on in the background. The *Sampling Step* controls how often the display is updated. CATIA knows the exact location of the pulley at each moment in time, but does not display it. By setting the *Sampling Step* to 3 seconds, you are indicating that you want the display to be updated every three seconds (seconds not necessarily being real-time seconds). If you change the *Sampling Step* to be 10 or 100 seconds, CATIA would replay the same simulation, but only update the display every 10 or 100 seconds. It is somewhat similar to having a strobe light in a dark room and then watching the people move. The movement is jerky if the strobe has a long display, however the faster the strobe flashes, the smoother the movement of the people. With the computer however, setting the *Sampling Step* to a very small value (0.001 seconds for example) will cause the simulation to run very slowly.

Change the *Sampling Step* to 10s and the *Temporization* to 0.5s, then select the Play Forward icon when done. Notice what the *Temporization* does for the replay. Every step (ten seconds in this case) the simulation pauses. This pause is the half second *Temporization*.

Set the *Temporization* back to zero and select the Parameters icon. *Temporization* can become annoying when you are not closely analyzing the simulation.

Select *OK* to the *Track* window. This finishes the track. Notice in your tree you now have a *Tracks* branch, with one track, *Track.1*.



Next the drain plugs will be removed. If you remember, tracks can only contain one object at a time, so they will have to have separate tracks.

Select the Track icon and set the object to be the back drain plug (*Drain Plug.1*). This will start the track with the back drain plug being the track object.

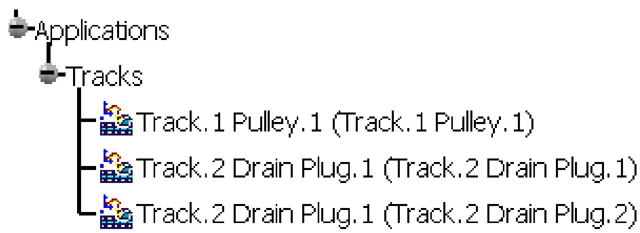
Drag the drain plug straight out (along the V axis) approximately the distance of the pulley.

Select the Record icon, then select *OK* when done. This will create a simple track to move the drain plug straight out.

The front drain plug also needs to be simulated. However, instead of creating an entirely new track, the same track used for the first drain plug will be used for the second.

With the third mouse button, select on the second track (*Track.2 Drain Plug.1*) and select *Copy*. This will copy the track into the clipboard.

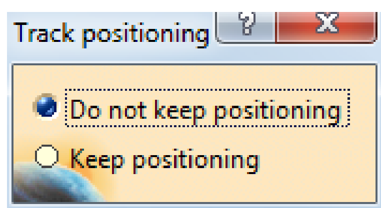
With the third mouse button, select on the main *Tracks* branch, then select *Paste*. This pastes the track back into the assembly.



Now the track needs to be modified to work with the other drain plug.

Double select on the third track (the one you just pasted). This will bring up the *Track* window again.

Select the *Object* field. This is going to bring up the *Track Positioning* window.

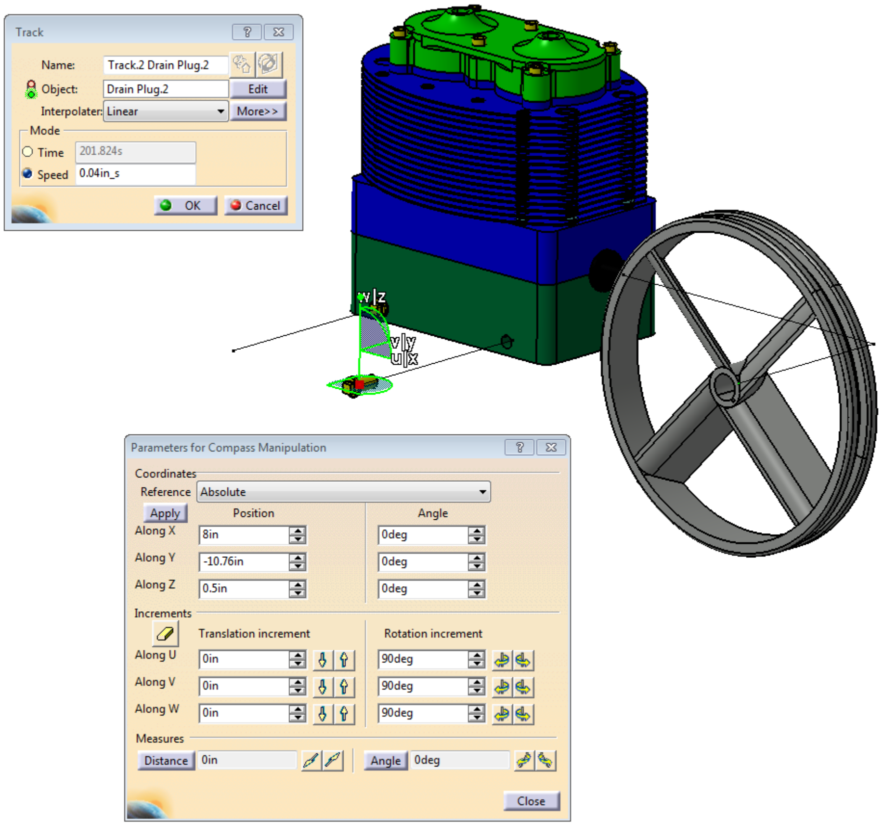


This window is ascertaining where the track will be placed. CATIA knows there is already a track in place, but the object is going to be changed. It wants to know if you want to keep the track in the current position or not. By keeping the track in the current position, the track will act on the new object, but the track guide (the line that denotes the track position) will still be in the current position. This is generally not a good idea. You will want to leave this window on the option of *Do not keep positioning*.

Select the front drain plug. The track moves to the other drain plug.

Select the Skip to Begin icon and play the track forward.  This moves the drain plug out to the end of the track.

With the drain plug at the end of the track, double select on the compass. This is going to display the compass manipulation window.



Change the *Translation increment* for *Along W* to 2.0in. This will make any movement along the W axis to be in two inch increments.

Select the Up Arrow next to the *Along W* translation increment. This moves the drain plug two inches along the positive W axis.

Two inches is not quite far enough.

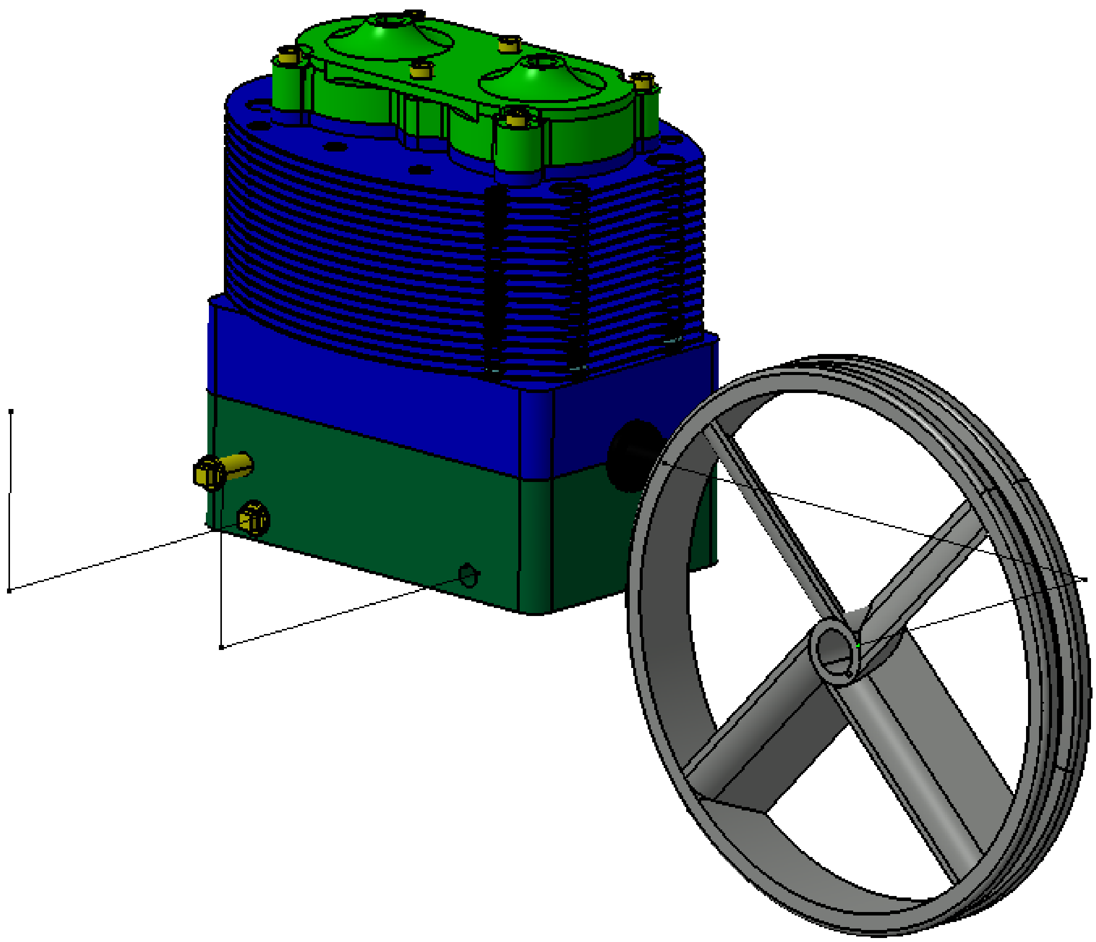
Select the Up Arrow again and select *Close* when done. This will make the total motion to be four inches along the positive W direction.

Select the Record icon. Notice what happened. Not only did the four inch movement get applied to the second track, but also the first. This is due to the fact that the two are copies of each other.

Note: You can also press the Insert key on the keyboard instead of selecting the record icon. This can make recording multiple points much faster.

Copying and pasting a track links the two together. Therefore, whenever one of the two tracks are changed, the other will update with the new changes. If they do not automatically update, select the third mouse button on the un-updated track, select the track object from the contextual menu, and select *Update*.

Select *OK* when done. This will finalize the new track.



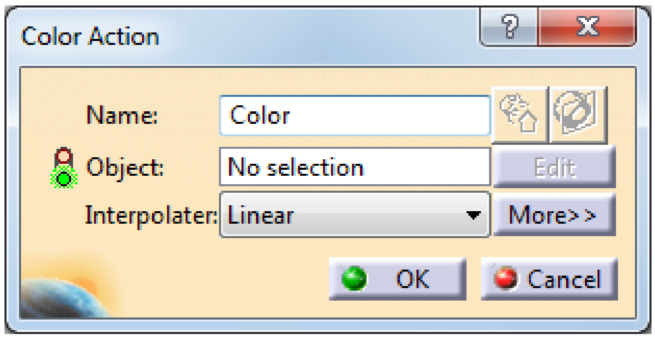
Those are the basic steps for creating tracks. There are a few more advanced features that will be investigated later. Next, the actions will be covered.

This would make a good point to save your fitting simulation.

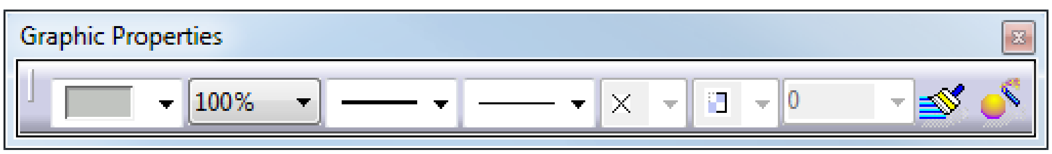
Color Action

Color actions are very simple. They create an action that changes the color and transparency of an object. When creating a fitting simulation, you may want to emphasize one of the parts of the simulation, and changing the color of the object would be a good method to attract attention.

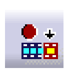
Select the Color Action icon.  The *Color Action* window appears. The options are very similar to the track options.



Select the *Pulley*. This is the *Object* that you are going to have change color. In order to change the color you will use the *Graphic Properties* toolbar.



Change the color to a bright yellow. You can do this by selecting the down arrow next to the color. The color of the pulley changes to bright yellow. You can also change the transparency level using the box next to the color.

Select the Record icon.  This will record the color change.

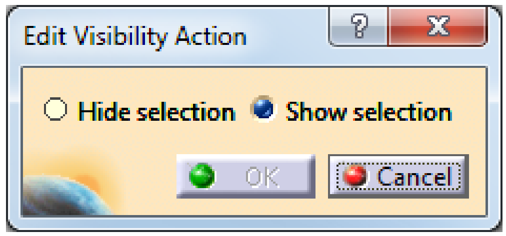
Select the Skip to Begin icon.  This will take you to the beginning before the color change occurs.

Select *OK*. This finalizes the color action. A *Color Actions* branch will appear in the specification tree.

Visibility Action

Visibility actions are just as simple as the color actions. The visibility action will either hide or show objects.

Select the Visibility Action icon.  This will display the *Edit Visibility Action* window. This is located under the Color Action icon.



As you can see, there are only two options that allow an object to be hidden or shown.

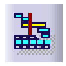
Select the front drain plug. Notice the drain plug looks transparent. This just denotes that it will be hidden when the action is performed.

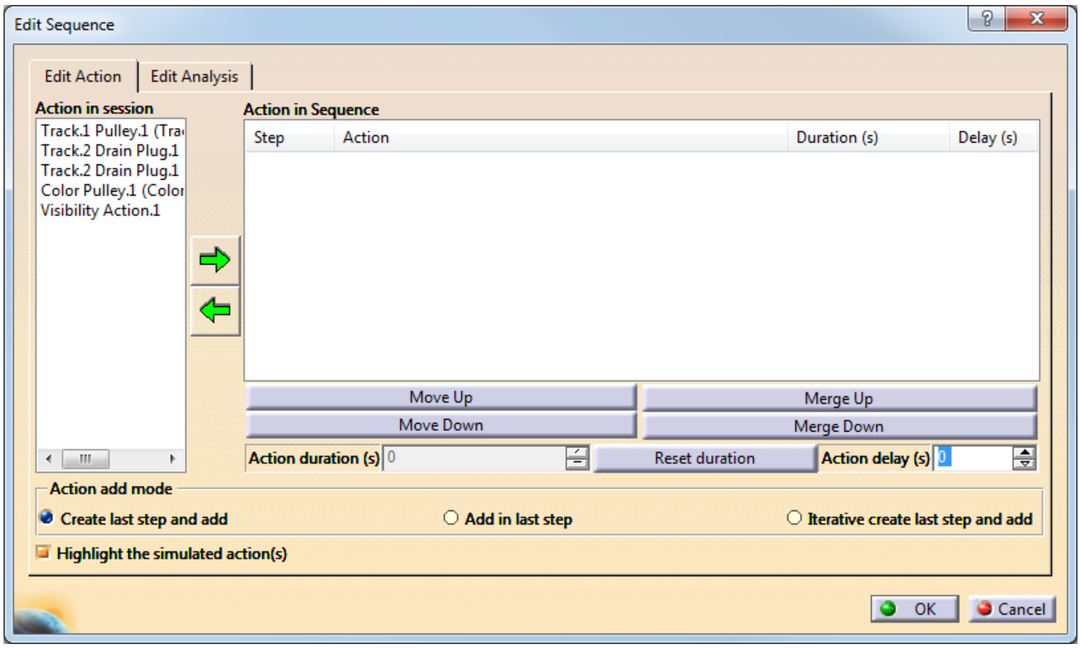
Select *OK* when finished. This is the last type of action to create.

Now that you have all the actions and tracks created, it is time to make a sequence.

Sequences

A sequence is an ordered set of events. All of the tracks and actions that you have created so far can be merged into a sequence. The benefit of a sequence is the ability to control the timing of events. For example, both the drain plugs use the same track. With a sequence, one drain plug track can take three seconds to complete the track, while the other drain plug track can take only one second. The order of events is also controlled.

Select the Edit Sequence icon.  This will display the *Edit Sequence* window. There are a few options and areas in the *Edit Sequence* window, so take a closer look.



Action in session — These are all the actions that can be added to the sequence. All of your tracks, as well as other actions, are in this area

Action in Sequence — This is where your sequence will appear. By selecting one of the actions in the session, then selecting the right arrow, the action will be merged into the sequence

Move Up/Down — Change the order of the actions in the sequence

Merge Up/Down — Merging actions together will make them occur at the same time. For example, merging a track and a color action will make them happen simultaneously

Action Duration — Defines the length of time for each action in seconds

Action delay (s) — Defines how long the action will delay before it occurs

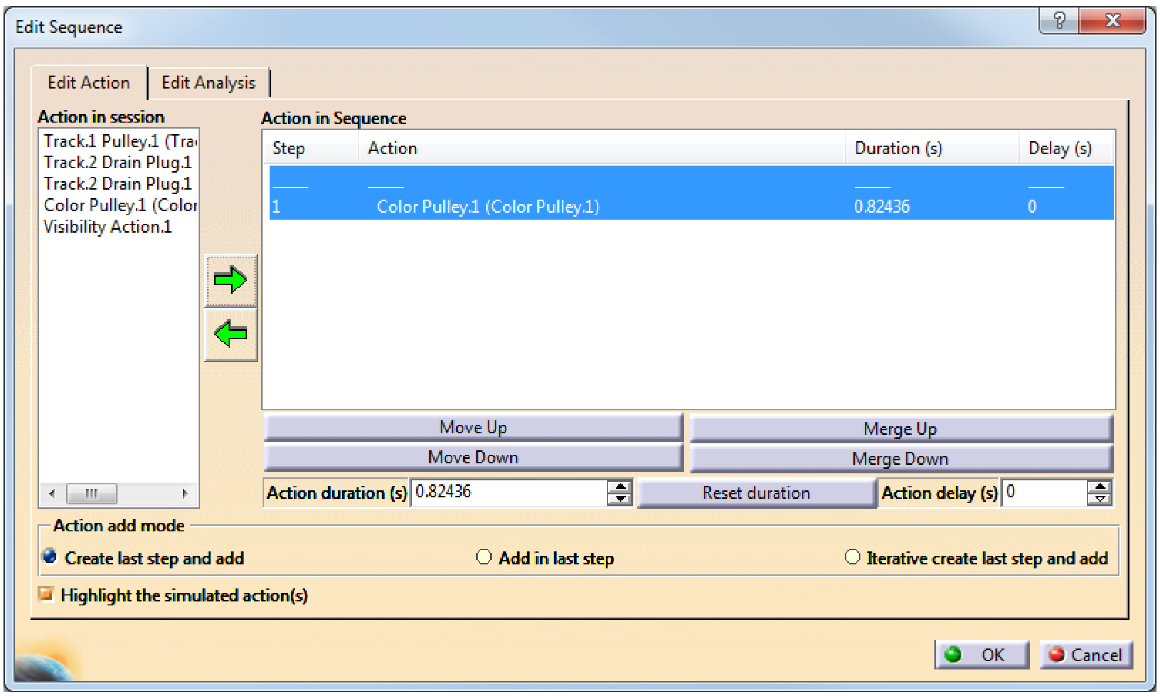
Action add mode — Specifies how the next action gets added to the sequence

Highlight the... — Specifies that the actions will appear highlighted when running a sequence

The first operation that should occur is the color action for the pulley and then the track action of the pulley being removed. This will be followed by both of the drain plugs being removed, and then the first drain plug being hidden.

None of the tracks have unique names. This makes it difficult to determine which track goes with which operation. Fortunately, there are only three tracks, but in a large fitting simulation, there may be hundreds of tracks. Next time, the tracks will get renamed after they are created to ease the process.

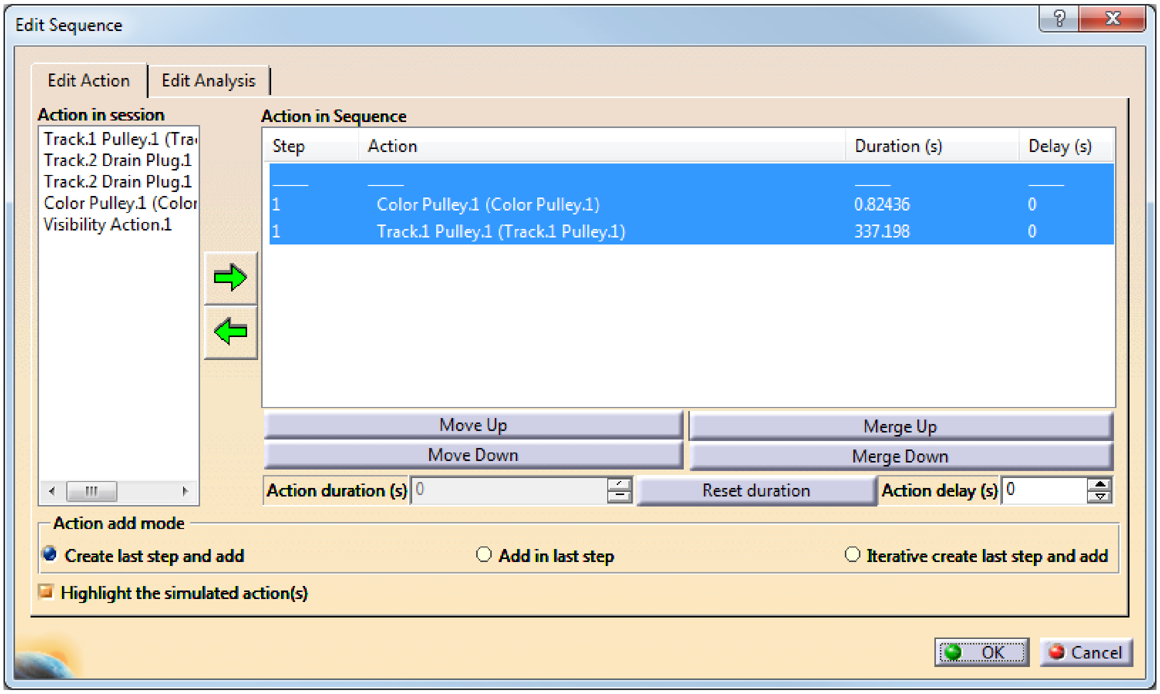
Select *Color Pulley.1* and select the Right Arrow.  This will place the action in the sequence.



Notice the duration is set. The color action will occur over that time, gradually changing colors. A visibility action occurs instantly and will not have a duration.

Select *Track.1Pulley.1(Track.1Pulley.1)* and select the Right Arrow again.  This will place the pulley removal track right after the color change action. In this case it will not matter if these two happen consecutively (two different steps) or simultaneously (all in the same step), but good technique would suggest that since you want both the color to change and the pulley to be removed at the same time, it would be best to put them into simultaneous mode.

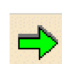
With the second step highlighted, select the *Merge Up* button. This is going to merge the two steps together.



Notice that both the actions now have step 1 beside them. This means that both actions will be performed at the same time. You can have as many actions occur simultaneously as you want.

With *Track.1Pulley.1* in the first step selected, change the *Action Duration* to 10. This will make the first action take ten seconds.

Select *Track.2 Drain Plug.2 (Track.2 Drain Plug.1)*, and then hold down CTRL and select *Track.2 Drain Plug.2 (Track.2 Drain Plug.2)*. This will highlight both of the track actions in the *Action in session* field.

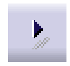
Select the Right Arrow to add them to the sequence.  This makes them part of the sequence. Notice both are added to one step. This is a good method to add two actions that are to happen simultaneously.

Change the duration for *Track.2Drain Plug.2 (Track.2 Drain Plug.1)* to 15, and the duration for *Track.2Drain Plug.2 (Track.2 Drain Plug.2)* to 5. To change the duration, just select *Track.2 Drain Plug.2 (Track.2 Drain Plug.1)* and then change the duration to 15. Do the same for *Track.2 Drain Plug.2 (Track.2 Drain Plug.2)*.

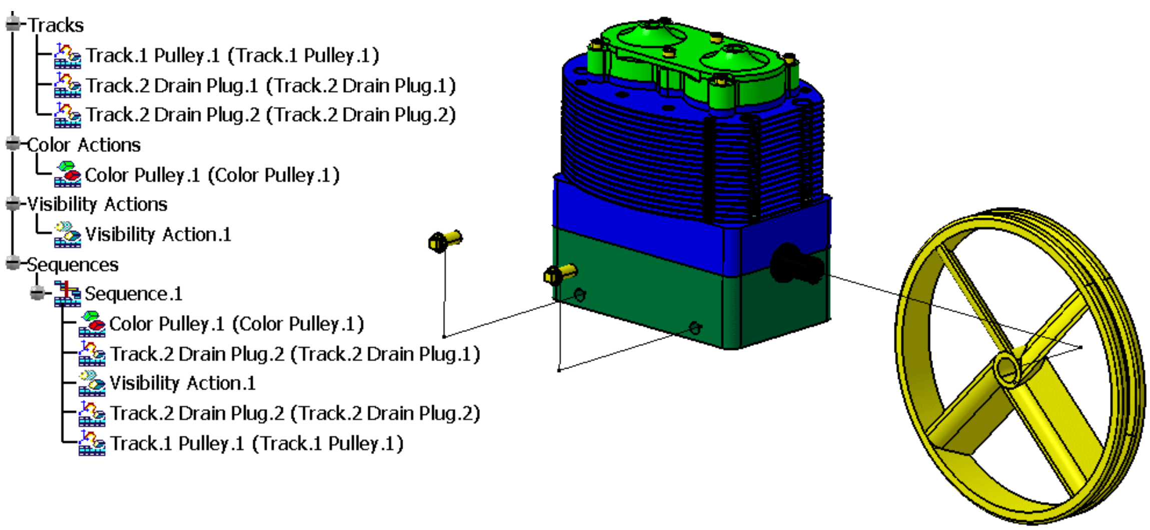
Even though both actions are happening at the same time, it is possible to have one action last longer than the other.

With no *Actions in Sequence* selected, select *Visibility Action.1* and add it to the sequence. Just select in a blank white area in the *Action in Sequence* area, then select *Visibility Action.1*, and then the right arrow. This will insert step three into the sequence. Now replay the sequence.

Select the Skip to Begin icon.  **Make sure the *Sampling Step* is set to 1s.** To check the *Sampling Step* just select the parameters icon.

Select the Play Forward icon.  **Select *OK* to the *Edit Sequence* window when done.** Notice the total time took 25 seconds. This is the ten seconds for the first step, and the 15 seconds of the second step. The five seconds for the one track is contained within the 15 seconds.

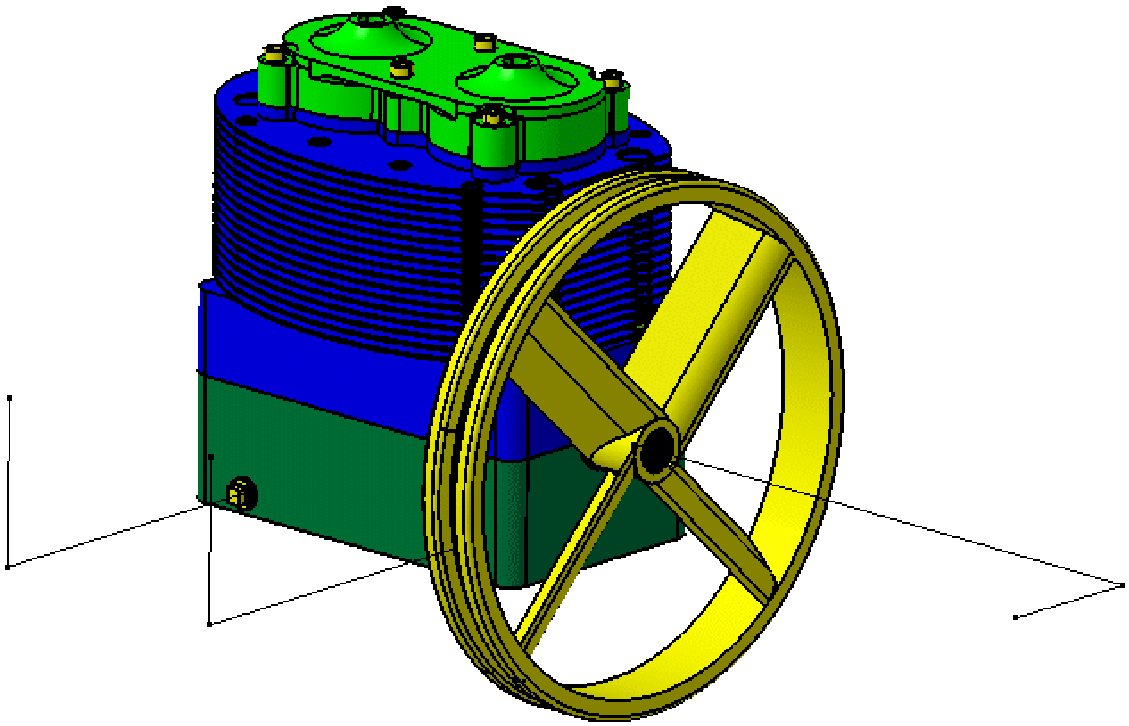
Also note the disappearance of the front drain plug. This is due to the hide command in the visibility action.



Notice all of the objects are still moved away from the assembly. This can be easily remedied.

Select the Reset Position icon.  This snaps the assembly back to the position it was when the assembly was brought into the fitting simulation.

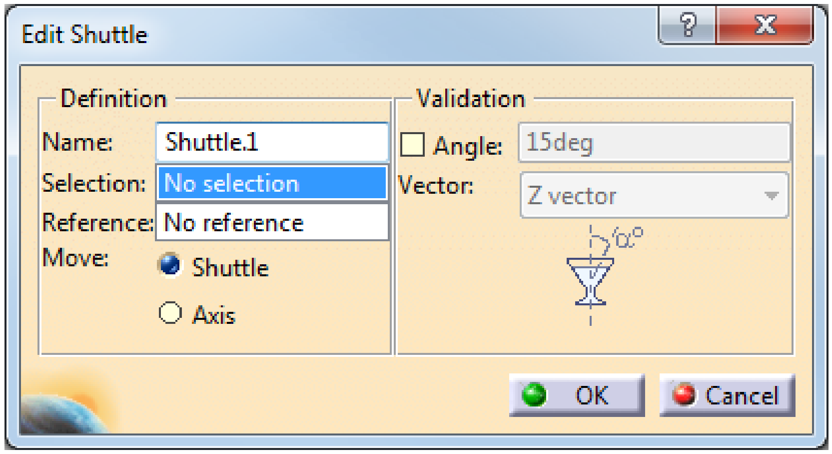
This would make a good point to save your assembly.



Next, the bolts holding the head will be removed. There are two methods to remove all of the bolts. One method would be to create a simulation track for each one, copying and pasting the track, of course. The other method would be to group the bolts together in a shuttle.

Shuttles

Select the Shuttle icon.  This will display two windows. One window will be the *Preview* window. This window shows what is currently in the shuttle. The second window is the *Edit Shuttle* window. This window has a bit more information to it.



Definition

Name This is the name of the shuttle. It is always advisable to name the shuttles with a unique and clear name

Selection Defines what is selected for the shuttle

Reference A shuttle can be referenced back to another shuttle. This will allow it to move with the shuttle that it is referenced to

Move Controls what gets moved with the shuttle

Validation Validation is used with the path finder icon. It will keep the object from rotating beyond a certain angle, about a given axis

Angle Defines the maximum angle the object can rotate

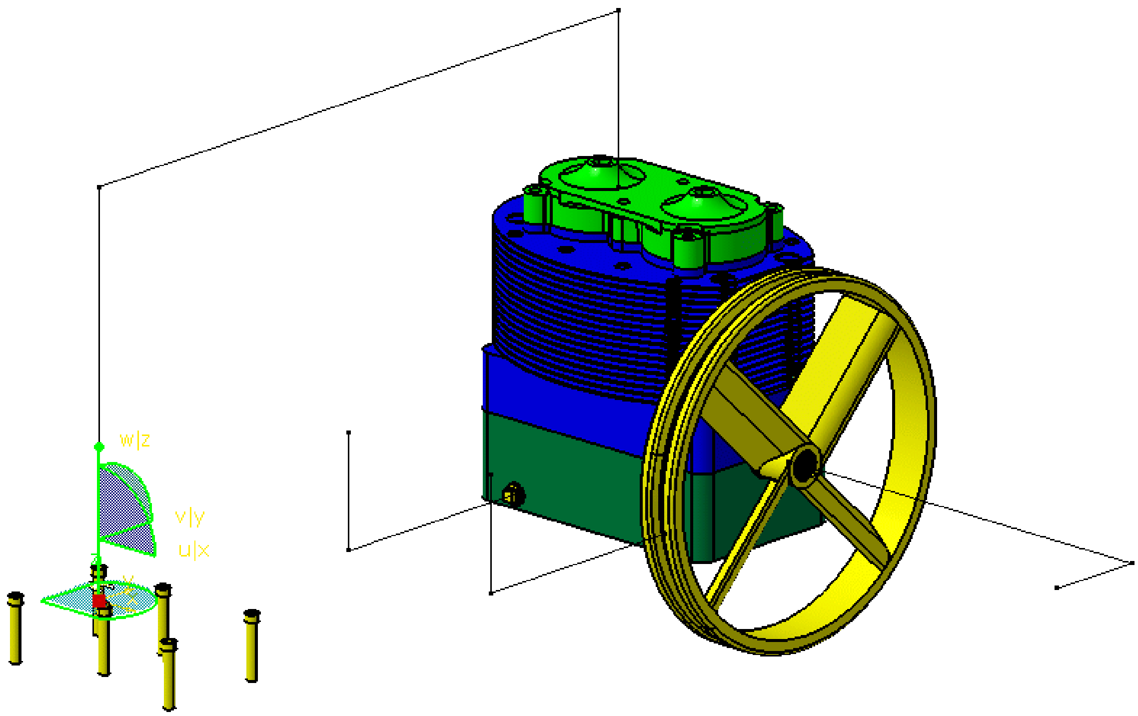
Vector Defines what axis the maximum angle can occur around

Select the six bolts around the top of the compressor. The six bolts get added to the shuttle. Notice the compass and axis (the small hand with a box) is placed on the first bolt you select. The placement of the axis is not important in this case. If all six bolts were selected beforehand, the axis would be placed in the geometric center of the bolts.

Change the name of the shuttle to <u>Head Bolts</u> and select *OK* when finished. This will finish the shuttle. The validation options will be left alone for the moment. This also adds another branch in the tree for the shuttles. The shuttle can now be put into a track, and later, into a sequence.

Select the Track icon and then the *Head Bolts* shuttle.  This will start a track using that shuttle. Since you are only moving the shuttle, you can, at any time, add or remove objects from it.

Move the bolts straight out, over, then down to create the track. Make sure you record each movement in order to create the appropriate track. The bolts should end up similar to the following picture.

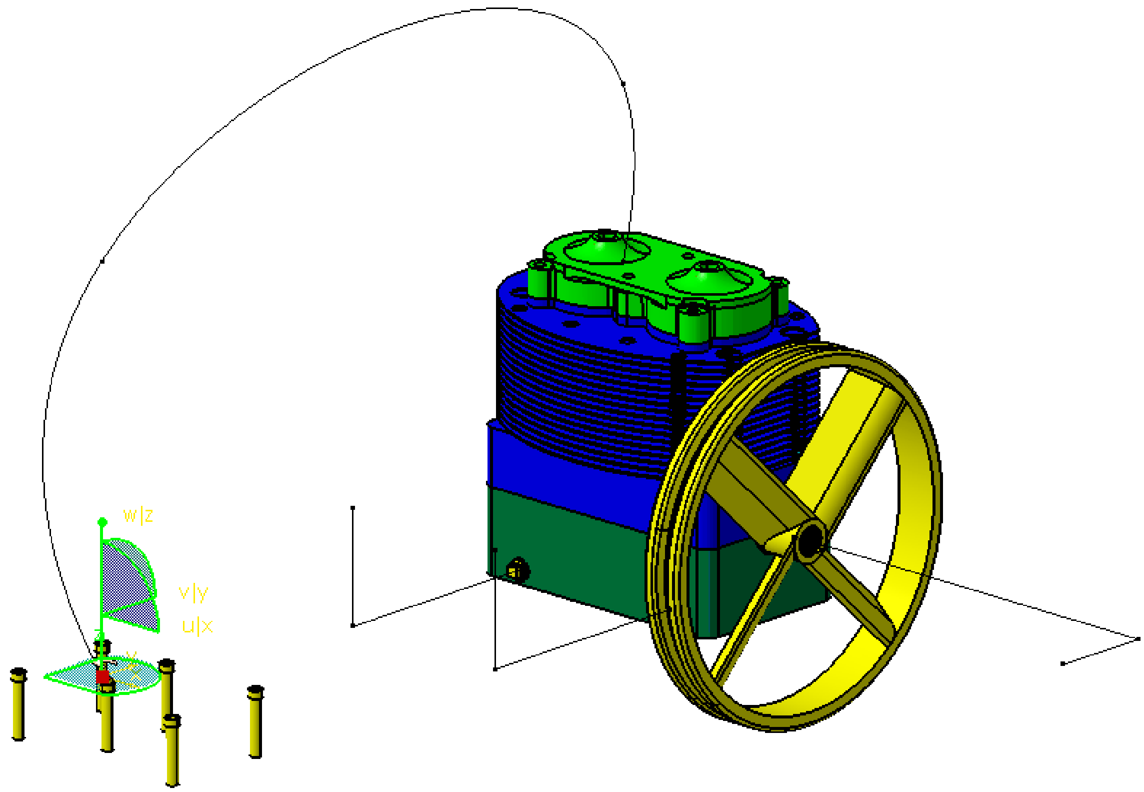


Skip to the beginning of the track, make sure your *Sampling Step* is 10s and play it forward. This is just to make sure the simulation works as you want. Notice the straight corners of the simulation track. This is due to the *Interpolater* being set to linear.

Change the *Interpolater* to *Composite spline*. You will not notice anything different in the track visualization.

Skip to the beginning of the track and play it forward again. This time notice what is happening, the bolts slow down, and then speed back up as they go around a corner.

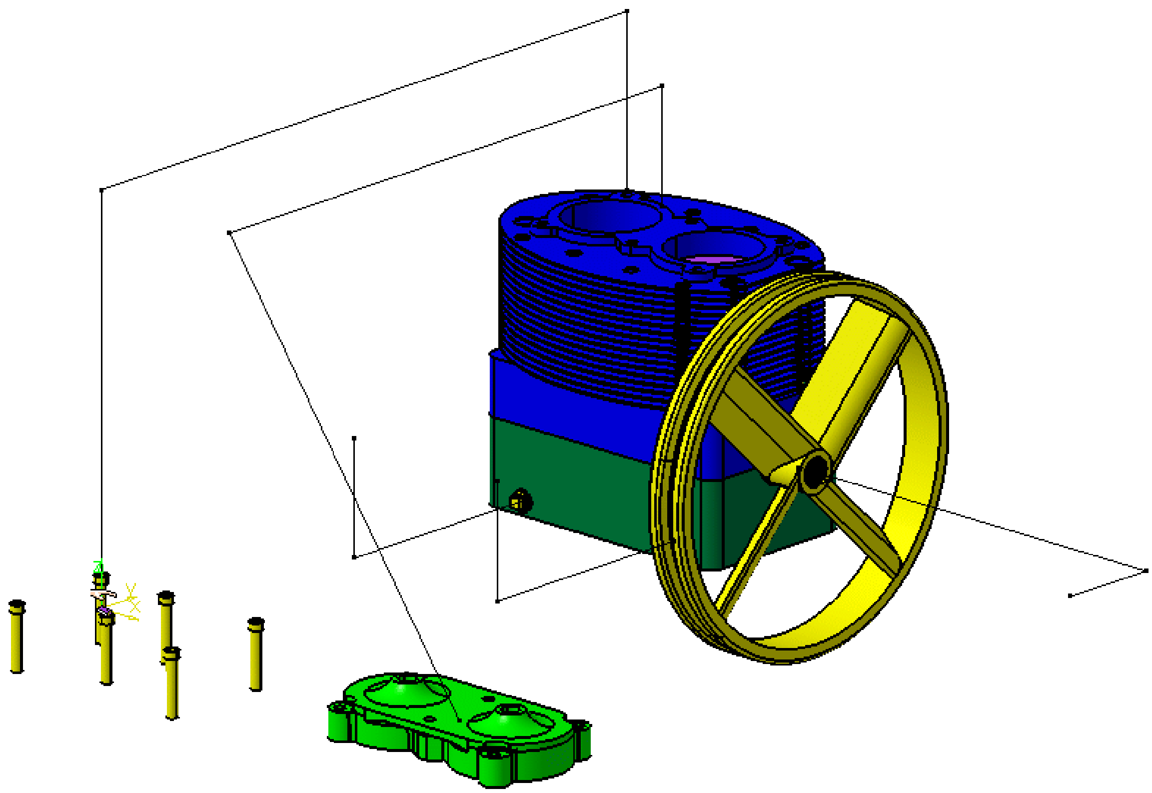
Change the *Interpolater* to *Spline*. This time you will notice a definite change in the track path. In this mode, the recorded points act as anchor points for the spline.



Skip to the beginning of the track and play it forward again. The bolts move in a very nice and fluid motion, but this is not what you are after. The bolts cannot simply move sideways out of the head. Instead, they need to pull straight out.

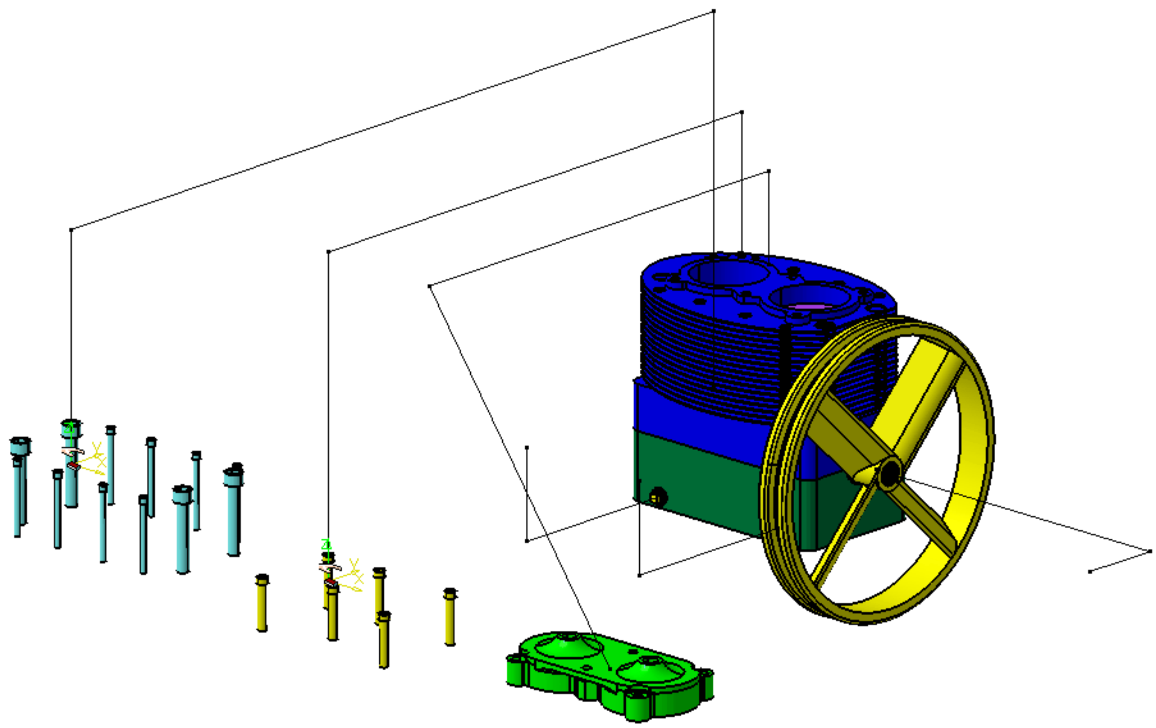
Change the *Interpolater* back to *Composite spline* and select *OK* when done. This will leave the bolts pulling straight out of the block.

Create a track of the green head being pulled off of the compressor. Move the head over by the bolts. The result should look something similar to the image below.



Create a shuttle for all of the aqua bolts at the bottom of the cylinder head and call it <u>Cylinder Bolts</u>. The bolts are the .5x3 and .25x3 bolts. There are twelve bolts total.

Create a track to remove the bolts from the assembly. Move the bolts over to the same area as the rest of the parts. The track should appear something like what is shown.



One more track is to be created. This time, however, you are going to create it on-the-fly in a sequence.

Kinematics

Joints

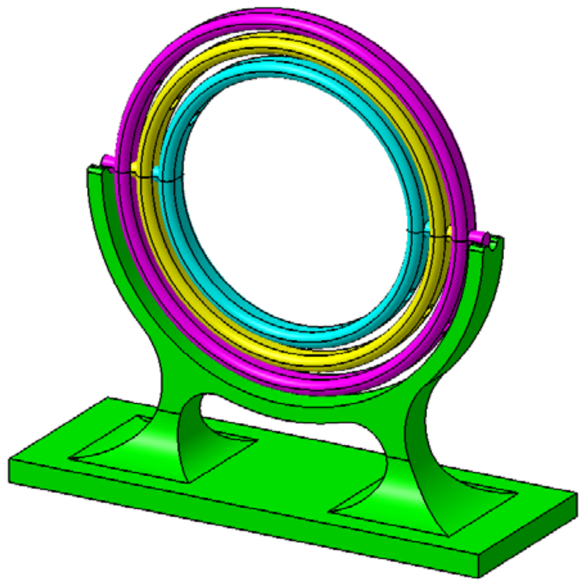
There are many joints available in Kinematics. Each joint has degrees of freedom and commands associated with it. A good understanding of these will make performing kinematics much easier. The table below gives an overview of the joints. Some of the joints can be created using axis systems but that will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

Joint	Degrees of Freedom	Commands Available
Revolute	1 Rotation	Angle
Prismatic	1 Translation	Length
Cylindrical	1 Rotation 1 Translation	Angle and/or Length
Screw	1 Rotation or 1 Translation	Angle or Length
Spherical	3 Rotations	None
Planar	1 Rotation 2 Translations	None
Rigid	None	None
Point Curve	3 Rotations 1 Translation	Length
Slide Curve	2 Rotations 1 Translation	None
Roll Curve	1 Rotation 1 Translation	Length
Point Surface	3 Rotations 2 Translations	None
Universal	1 Rotation	None
CV	None	None
Gear	1 Rotation	Angle
Rack	1 Rotation or 1 Translation	Angle or Length
Cable	1 Translation	Length

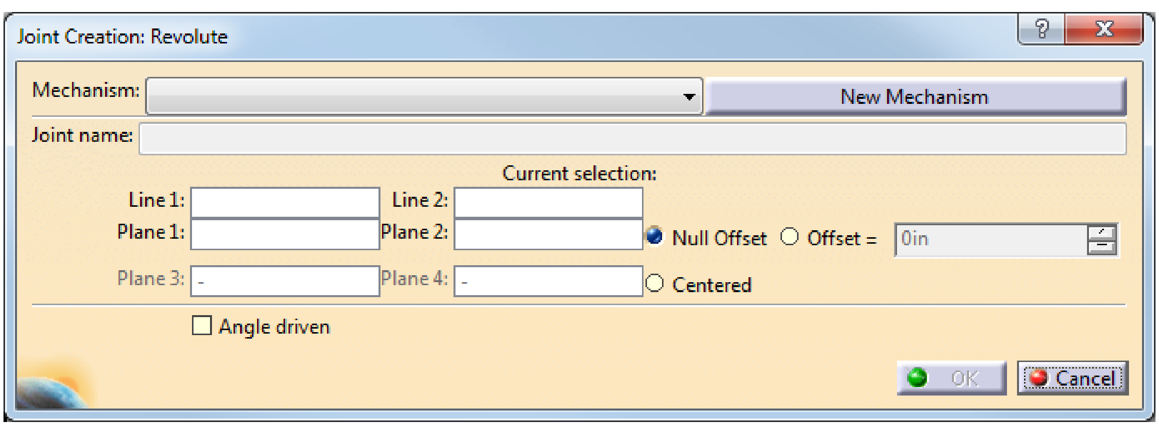
Revolute - Null Offset

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 Revolute Joint document located in the *Revolute* directory. You should see a base with three rings.



Select the Revolute Joint icon.  The *Joint Creation : Revolute* window appears.



Mechanism Specifies the mechanism for the joint to belong to, you have the option of defining a *New Mechanism*

Joint name Specifies a name for the joint

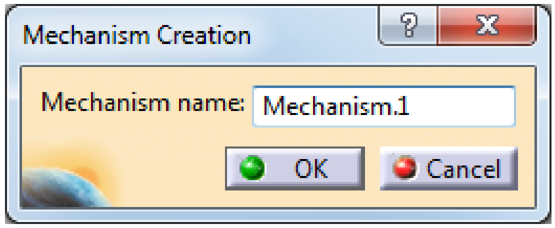
Line 1,2 Defines the two lines that should line up and the mechanism should rotate around

Plane1,2 Defines the two planes that should line up, you have the option of defining an *Offset* value if they do not line up

Plane 3,4 Defines two additional planes in order to make the two objects *Centered*

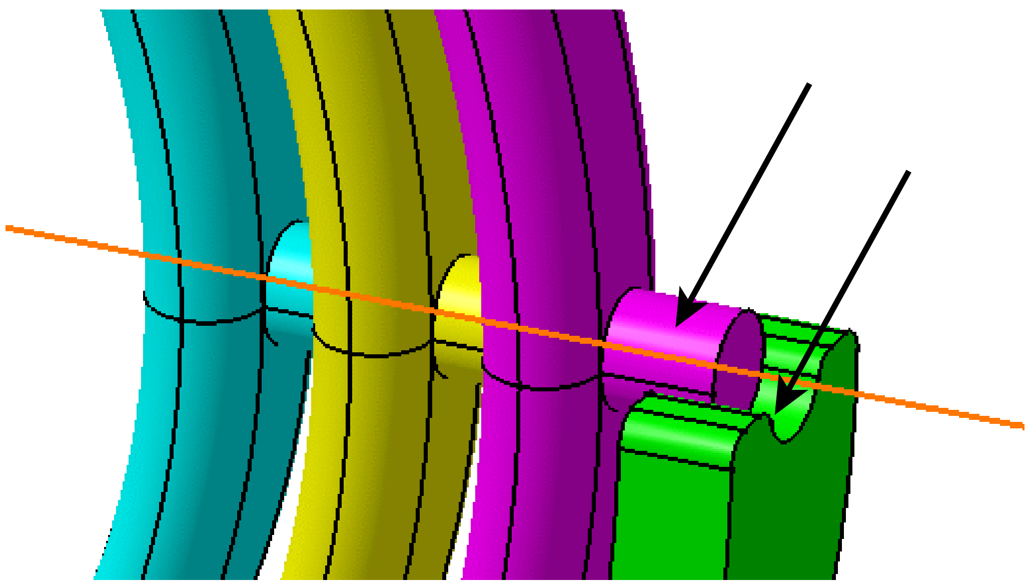
Angle driven Allows you to attach an angle command

Select the *New Mechanism* button. The *Mechanism Creation* window appears. A mechanism needs to exist before a joint can be made. You can do this by using this option within the joint or by selecting pull down menu *Insert, New Mechanism*.



Key Gyro for the *Mechanism name* and select *OK*. You are going to use the default name for the joint.

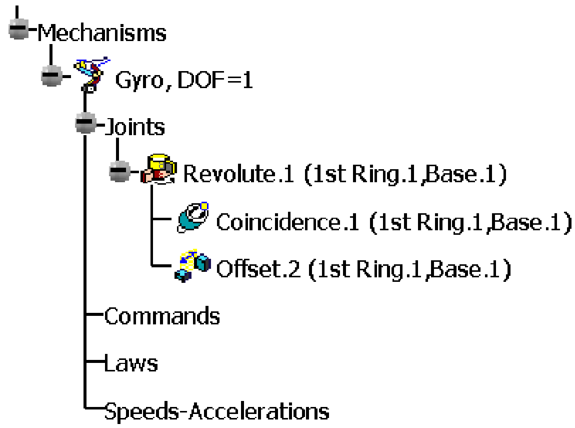
Select the center line of the cylinder on the outside ring and the center line of the cutout on the base as shown below. This defines the center of rotation and aligns the two lines.



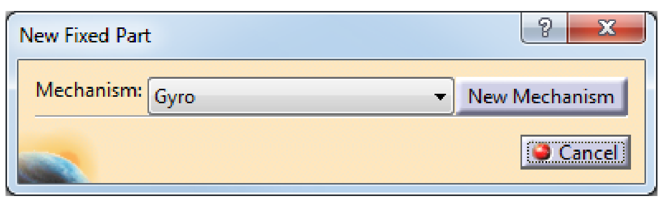
Select the *zx plane* in both parts. This defines the location of each object by aligning the two planes. You will probably have to expand the branches of the *Base* and *1st Ring* in order to select the planes.

Select *OK*. The joint is created. A *Mechanisms* branch appears in your specification tree.

Expand the branch in order to see all the elements below. Notice that the mechanism has one degree of freedom. Remember it was stated that an assembly must have at least one degree of freedom in order to be used in kinematics. However, in order to simulate the mechanism you must have commands assigned to it, giving it 0 degrees of freedom. In addition you must have a fixed part defined.

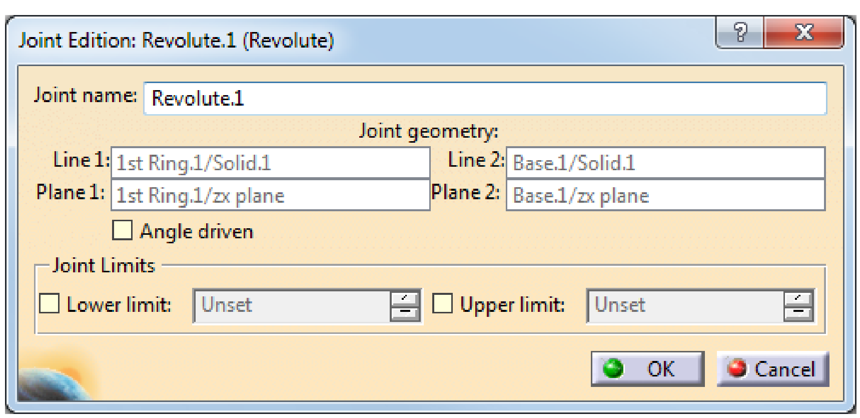


Select the Fixed Part icon.  The *New Fixed Part* window appears. You need to specify which part is fixed. If you had more than one mechanism than you would need to select the appropriate one from the drop down list in order to define the fixed part for that mechanism.



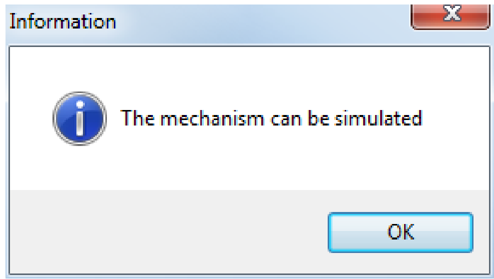
Select the Base. You can select it from the graphical work area or from the specification tree. A *Fix Part* branch appears under the mechanism in the specification tree. Since you did not specify a command when creating the joint you will need to go back and modify the joint to include a command.

Double select on the *Revolute* joint in the specification tree. The *Joint Edition : Revolute* window appears.

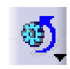


Joint Limits Sets limits for the range of motion if you turn on the *Angle driven* option

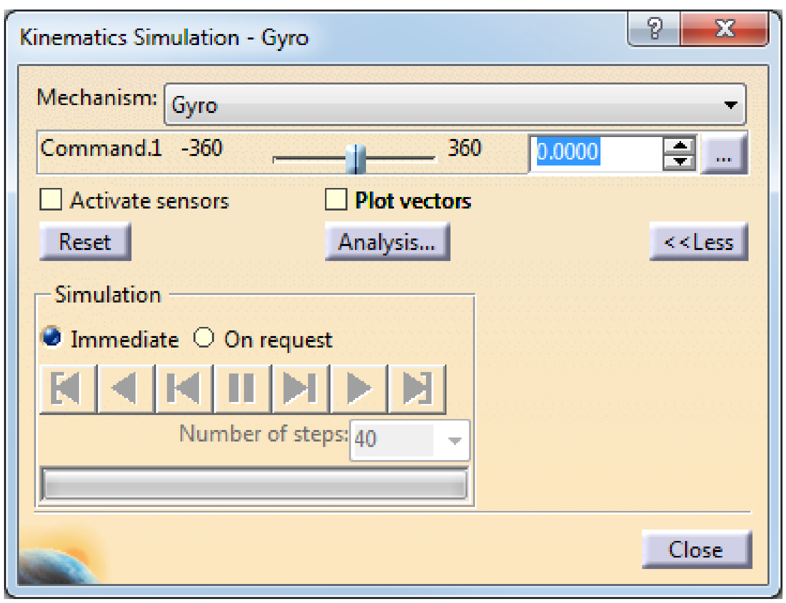
Select the *Angle driven* option and select *OK*. This will add an angle command to the joint. An *Information* window appears telling you that the mechanism can be simulated.



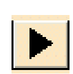
Select *OK*. Notice that the mechanism has 0 degrees of freedom and there is command in the *Commands* branch under the mechanism. You will now simulate the joint.

Select the Simulation with Commands icon.  The *Kinematics Simulation* window appears.

Select the *More*>> button. The window shows more options. These options will be covered in detail later in the course.



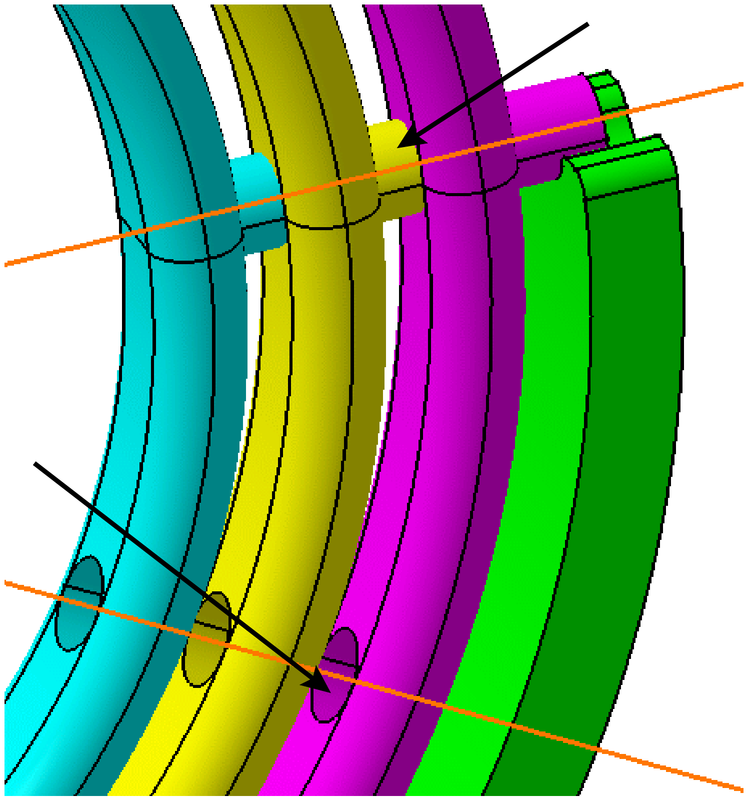
Turn on the *On request* option and drag the slider for *Command.1* to the right to *360*. This will tell it to simulate one revolution.

Change the *Number of steps* to be 40 and select the Play Forward icon.  The outer ring rotates around one complete revolution.

Select the *Reset* 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 Revolute Joint icon again.  The *Joint Creation : Revolute* window appears. This time the mechanism is already specified.

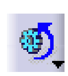
Select the center line of the cylinder on the outside of the middle ring and then the center line of the hole of the outer ring as shown below. This defines the lines.



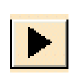
Select the *zx plane* in the *2nd Ring* and *Plane.1* from *Geometrical Set.1* in the *1st Ring*, turn on the *Angle driven* option and select *OK.* The middle ring moves to have the two lines line up. An *Information* window appears.

Select *OK.* The window closes.

Define the revolute joint between the inner ring and the middle ring and select *OK* when the *Information* window appears. You do this in the same manner as you did the second revolute joint. The inner ring should move to align with the hole in the 2nd Ring.

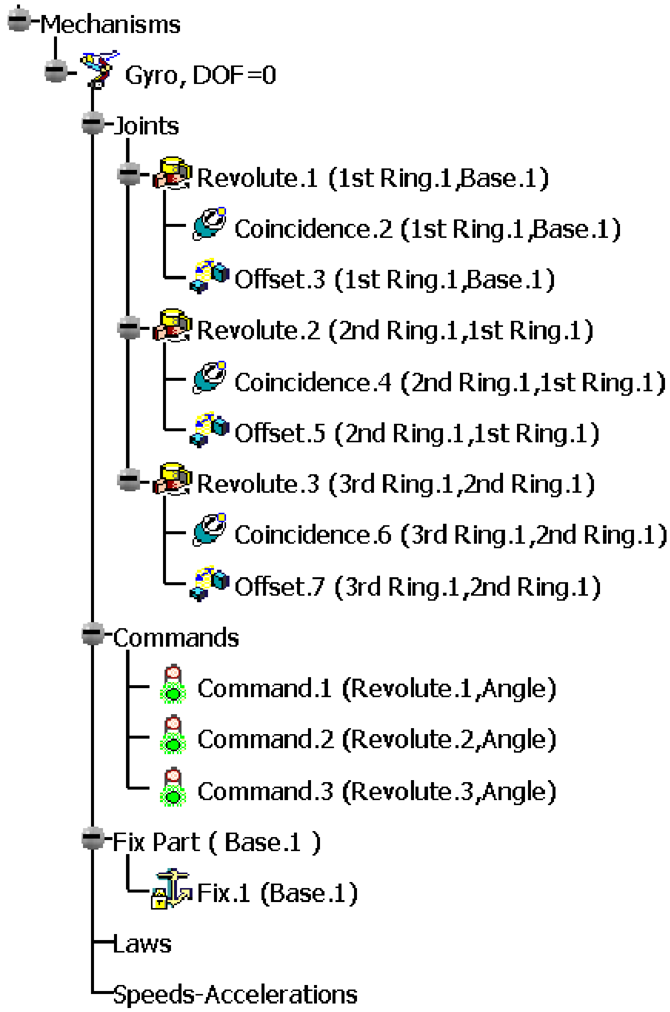
Select the Simulation with Commands icon again.  The *Kinematics Simulation* window appears. This time there are three commands available to simulate.

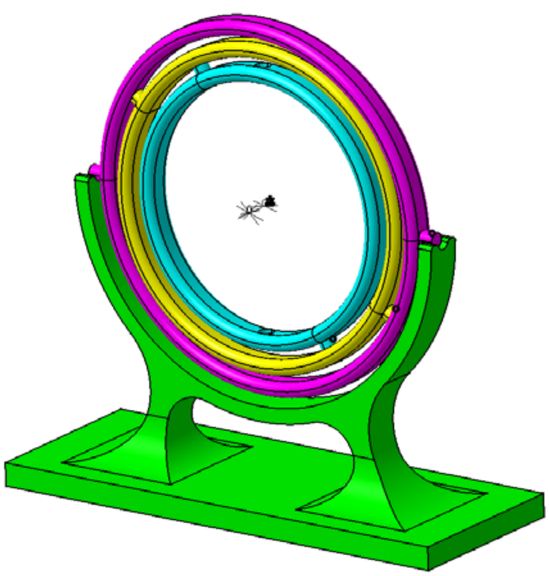
Make sure the *On request* is turned on and drag the slider for all three commands to 360.

Change the *Number of steps* to *80* and select the Play Forward icon.  The three rings rotate one full revolution each.

Select *Close.* The window closes. Your first kinematic mechanism has been defined.

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





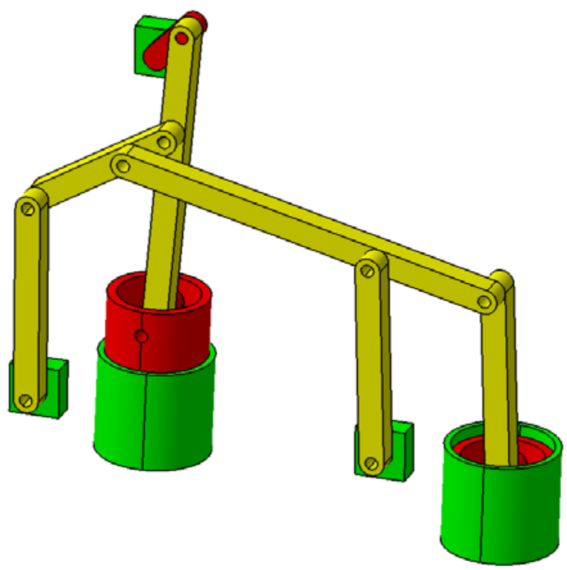
Save and close your document.

Assembly Constraints

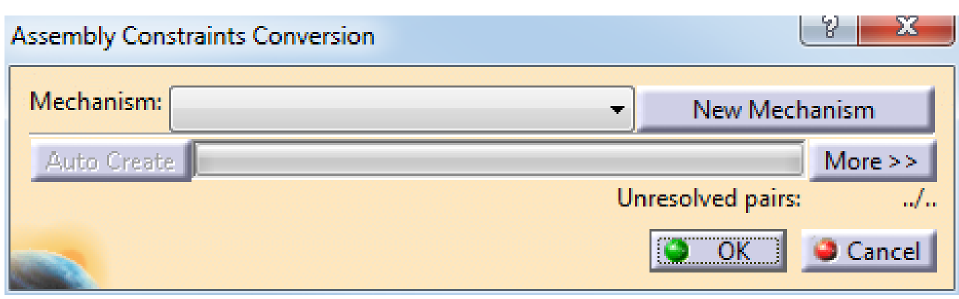
There are many times that you created an assembly and constrained it within Assembly Design. Assembly constraints are not recognized in Kinematics, therefore you have to define joints. However, assembly constraints can be converted into joints.

Auto Create

Open the Assembly Constraints document located in the *Assembly Constraints* directory. You should see a linkage assembly with two pistons. All of the objects have been constrained and the assembly can move with respect to constraints within Assembly Design.



Select the Assembly Constraints Conversion icon.  The *Assembly Constraints Conversion* window appears.



Mechanism Allows you to choose an existing mechanism if one exists or you can create a *New Mechanism*

Auto Create Enables CATIA to automatically create joints from the constraints

More>> Expands the window to show additional options available to manually match constraints to joints

Select the *New Mechanism* button and create a mechanism called <u>Linkage</u>. The *Auto Create* button becomes available.

Select the *Auto Create* button. Joints are automatically defined based on the assembly constraints. There are 17 joints and no *Unresolved Pairs*.

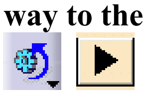
Select *OK*. Notice that you still have 1 degree of freedom. You need to add a command to one of the joints.

Double select on *Revolute.4* from the specification tree. The *Joint Edition* window appears.

Select the *Angle driven* option and select *OK*. An *Information* window appears.

Select *OK*. The window closes.

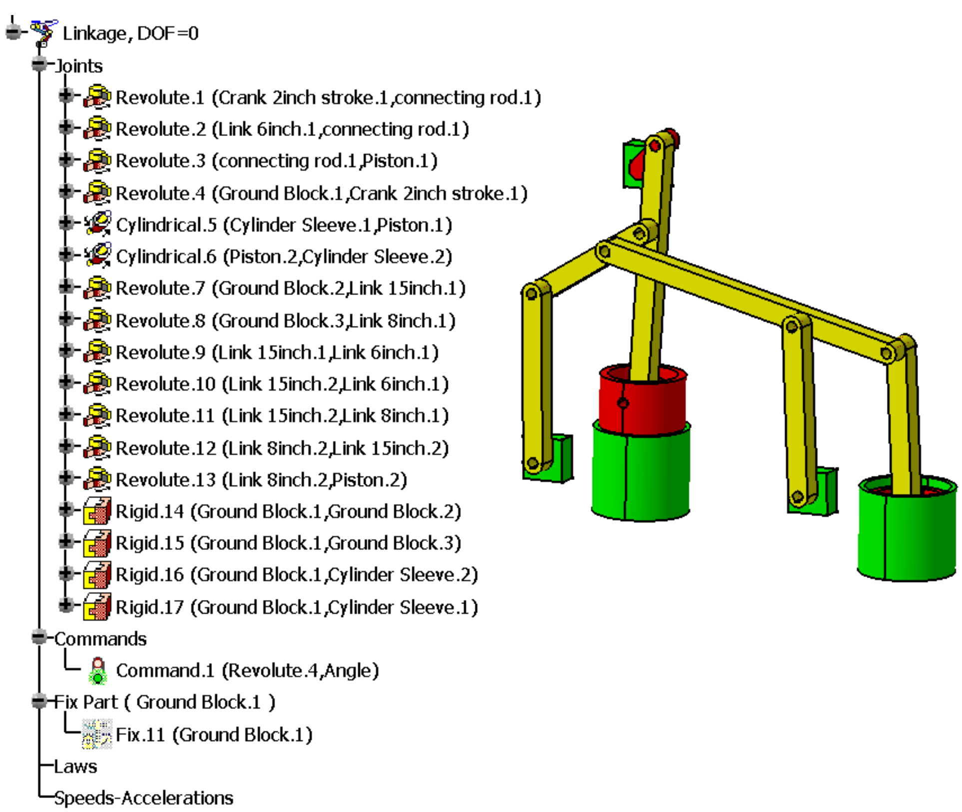
Select the Simulation with Commands icon, select *On request*, drag the slider all the way to the right, change the *Number of steps* to 40 and select the Play Forward icon.



The mechanism goes in motion. As you can see the conversion was very successful and you did not have to manually create any of the constraints.

Select *Close*. The window closes.

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

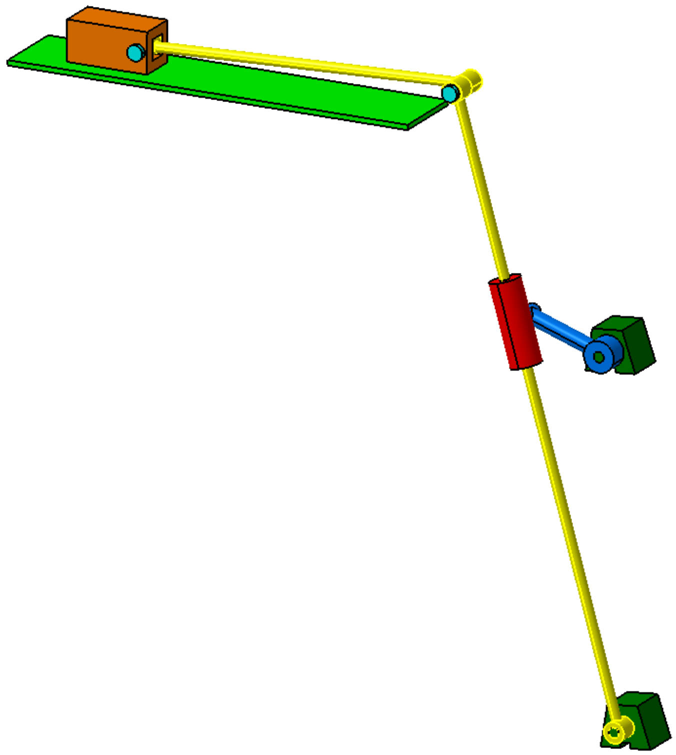


Save and close your document.

The previous exercise had you create the kinematic joints from assembly constraints using the *Auto Create* option. This works great for many cases, but you are leaving a lot of decisions up to CATIA. You may want to control which joints get created. By using the additional options in the assembly constraints conversion option, you can pick and choose which joints are created.

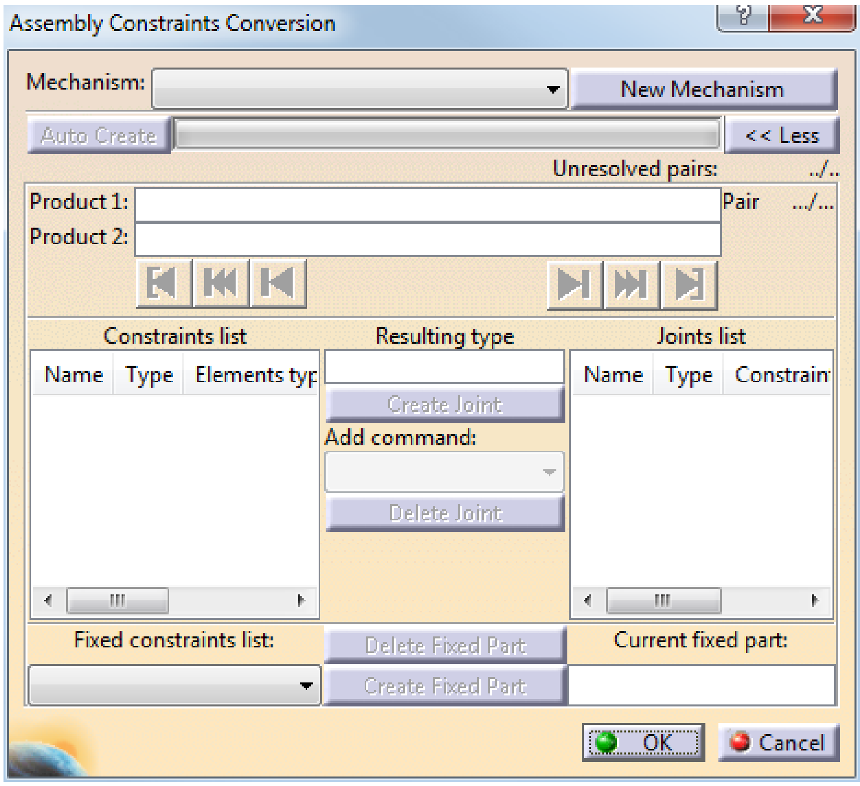
Advanced

Open the Assembly Constraints Advanced document in the *Assembly Constraints Advanced* directory. You should see a slider assembly. All of the objects have been constrained and the assembly can move with respect to constraints within Assembly Design.



Select the Assembly Constraints Conversion icon.  The *Assembly Constraints Conversion* window appears.

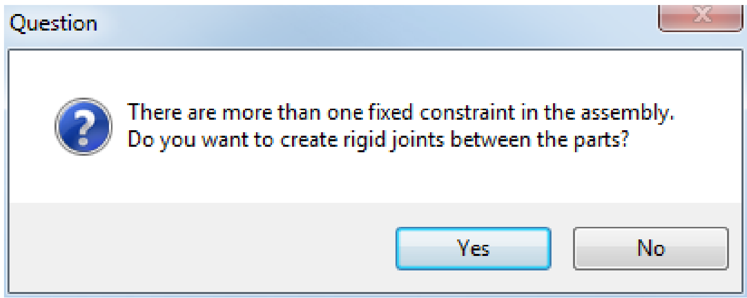
Select the *More* >> button. The window expands.



Product 1,2	Shows the two products involved
Constraints List	Shows the constraints that are associated to the pair of products shown
Resulting type	Shows the resulting joint that will be created based on the constraints that are selected in the *Constraints List*
Create Joint	Creates the joint shown in the *Resulting type*
Add Command	Adds a command to the joint
Delete Joint	Deletes the joint selected in the *Joints List*
Joints List	Shows the joints that are associated to the pair of products shown
Fix Constraints List	Shows all of the objects that have a fixed constraint
Delete Fix	Removes the fixed part shown in the *Current Fixed Part* box
Create Fix	Makes the object shown in the *Fix Constraints List* the fixed part for the mechanism
Current Fixed Part	Shows the fixed part for the mechanism

Select the *New Mechanism* button and create a mechanism called <u>Slider</u>. Items appear in the boxes. The first pair is shown. In the *Fix Constraints List* you should see the first fix constraint which is on the *Ground Block.2* component.

Select the *Create Fixed Part* button. The *Ground Block.2* component becomes the *Current Fixed Part*. A *Question* window appears stating that there are more than one fix constraints in your assembly. You have the option of creating rigid joints between those parts..



Select *Yes*. Two rigid joints were created and added to your mechanism. You should notice that the two components that are in the first pair are highlighted in the graphic area.

Select the *Coincidence* constraint in the *Constraints List*. The *Resulting type* is *Cylindrical* when only that constraint is considered.

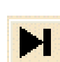
Select the *Surface Contact* constraint in the *Constraints List*. The *Resulting type* is *Planar* when only that constraint is considered.

Select both constraints in the *Constraints List*. You may need to use the Ctrl key to select both at the same time. The *Resulting type* is *Revolute*. This is the joint that you want and you want it to drive the mechanism.

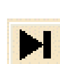
Select the drop down for the *Add Command*. The available commands for this joint appears.

Select the *Angle* command. You are now ready to create the joint.

Select the *Create Joint* button. The joint is created and there are no more constraints available for this pair of components. You have 0 degrees of freedom in your mechanism.

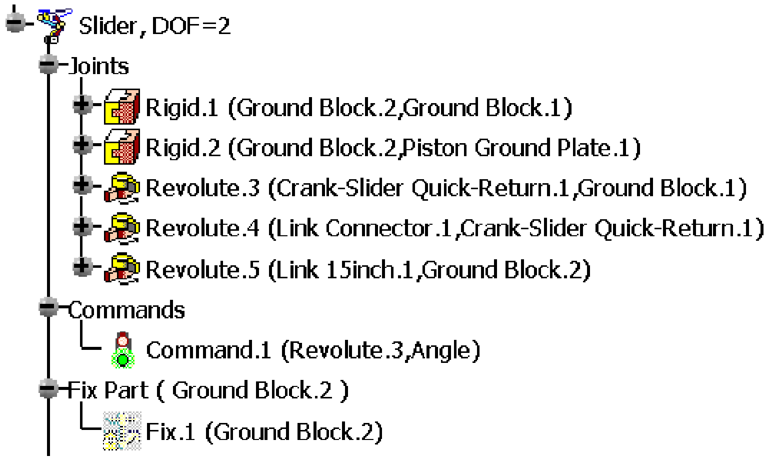
Select the Step Forward icon.  This moves to the next pair of components. The two components highlight. You will want a revolute joint between these two as well.

Select both of the constraints, the *Resulting type* should show *Revolute* and select the *Create Joint* button. You have 1 degree of freedom since you did not assign a command to the joint. This will be taken care of by creating additional joints.

Select the Step Forward icon.  This moves to the next pair of components. The two components highlight. This is an offset constraint between the two ground blocks. There is no need to create a joint for this since they already have a rigid joint between them.

Select the Step Forward icon. This moves to the next pair of components without creating a joint. The two components highlight. You will want a revolute joint between these two components.

Select both of the constraints, the *Resulting type* should show *Revolute* and select the *Create Joint* button. You now have 2 degrees of freedom. Your mechanism should appear similiar to the one shown below at this moment.



Select the Step Forward icon. This moves to the next pair of components and shows that they already have a rigid joint defined between them.

Select the Step Forward icon. This moves to the next pair of components without creating a new joint. The two components highlight. You will want a revolute joint between these two components.

Select both of the constraints, the *Resulting type* should show *Revolute* and select the *Create Joint* button. You now have 3 degrees of freedom.

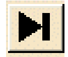
Select the Step Forward icon. This moves to the next pair of components. The two components highlight. This time you are going to create a cylindrical joint.

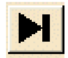
Select the *Coincidence* constraint, the *Resulting type* should show *Cylindrical* and select the *Create Joint* button. The degrees of freedom reduces to 1.

Select the Step Forward icon. This moves to the next pair of components. These constraints are between the pin and the link. You do not need a joint between them since the pin should be rigid to the link.

Select the Step Forward icon. This moves to the next pair of components. You will want a revolute joint between these two components.

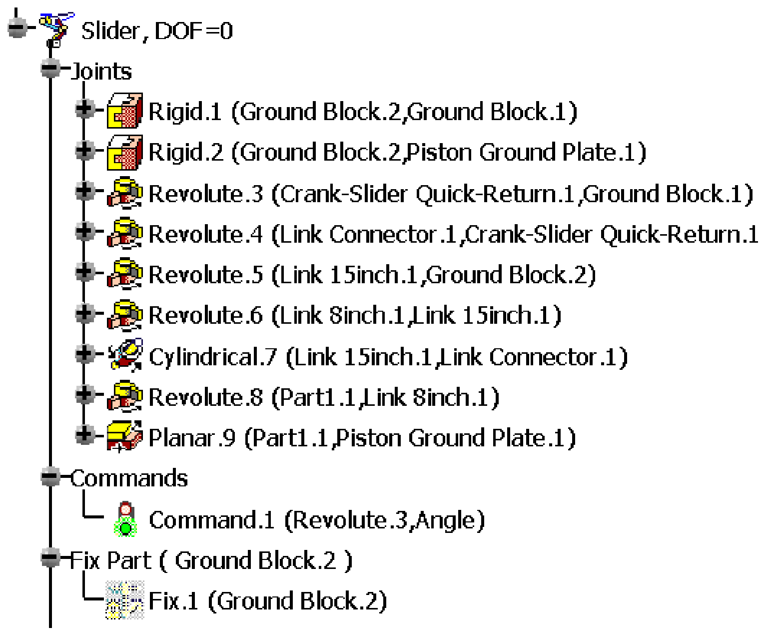
Select both of the constraints, the *Resulting type* should show *Revolute* and select the *Create Joint* button. You now have 2 degrees of freedom again.

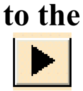
Select the Step Forward icon.  This moves to the next pair of components. These constraints are between the pin and the sliding block. Once again, you do not need a link between them since the pin should be rigid to the block.

Select the Step Forward icon.  This moves to the next pair of components. You will want a planar joint between these two components.

Select the constraint, the *Resulting type* should show *Planar* and select the *Create Joint* button. You now have 0 degrees of freedom. You are also at the end of the conversion.

Select *OK.* Your mechanism should appear similar to the one shown below.



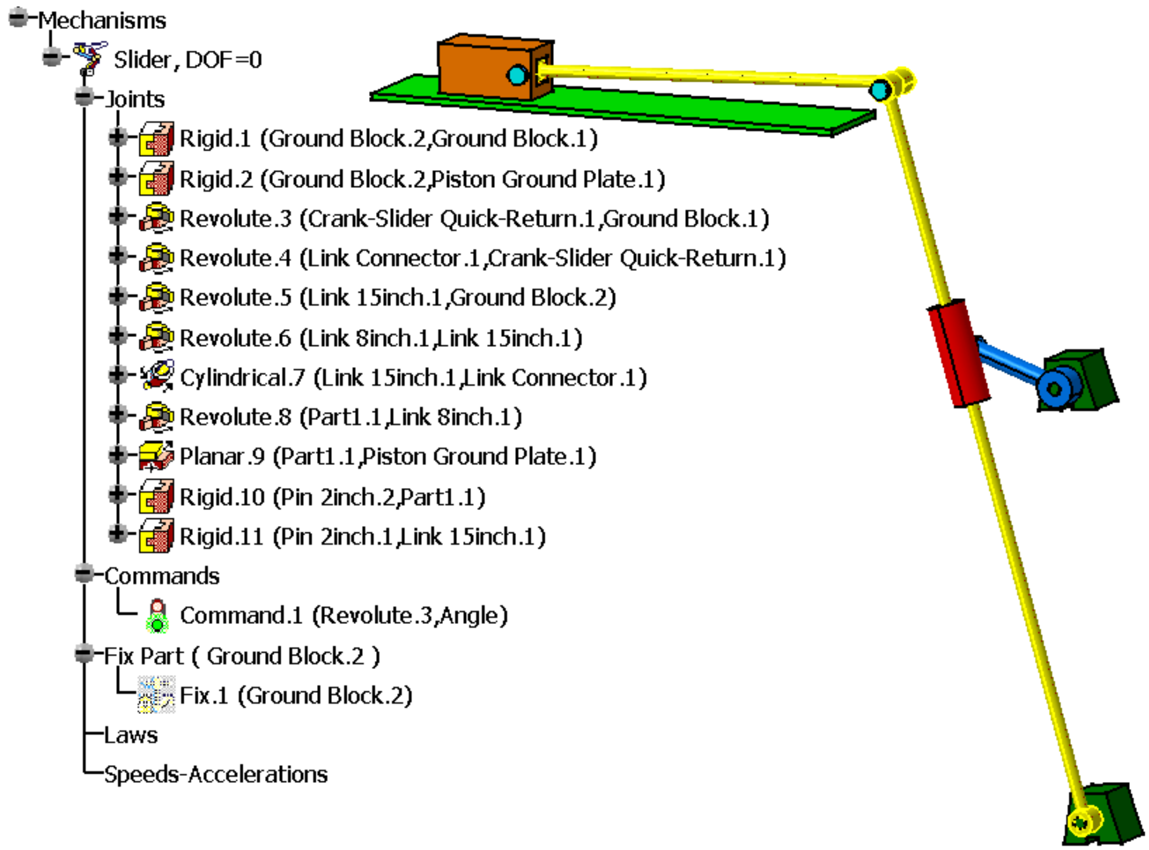
Select the Simulation with Commands icon, select *On request*, drag the slider all the way to the right, change the *Number of steps* to 40 and select the Play Forward icon.   The mechanism goes in motion. Notice that the two pins did not move. You never defined the rigid constraints with the pins.

Select the *Reset* button and select *Close*. The window closes.

Create a rigid joint between the pin and the block and one between the other pin and the long link.

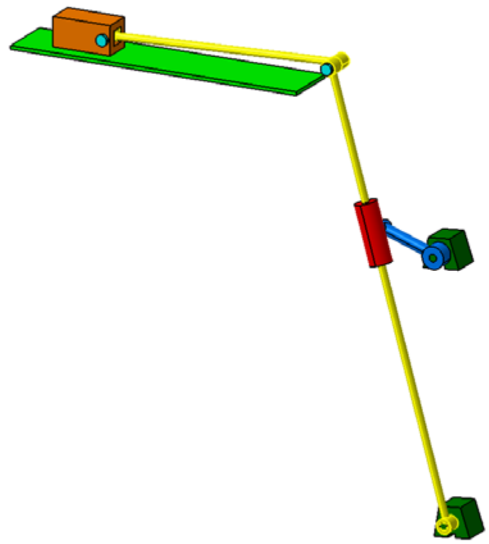
Simulate the mechanism again. This time the pins move with mechanism.

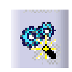
Close your simulation. Your final mechanism should appear similar to the one shown below.



Save and close your document. You are going to see what happens if you used the *Auto Create* option instead of manually converting the constraints to joints.

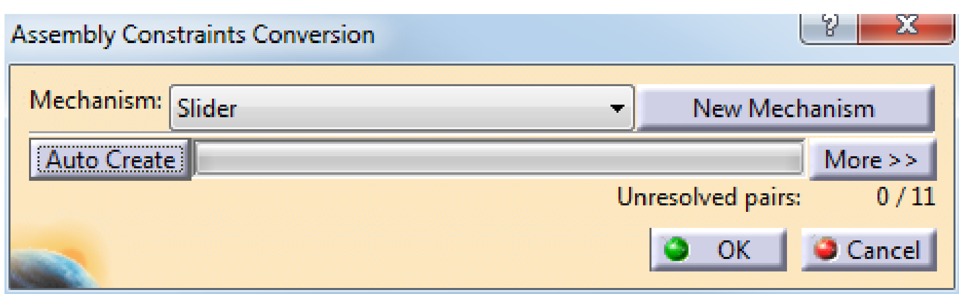
Open the original Assembly Constraints Advanced document again.



Select the Assembly Constraints Conversion icon.  The *Assembly Constraints Conversion* window appears.

Select the *New Mechanism* button and create a mechanism called <u>Slider</u>. The *Auto Create* button becomes available.

Select the *Auto Create* button. Joints are automatically defined based on the assembly constraints and the window now shows there are 0 unresolved pairs.



Select *OK*. Notice that you still have 3 degrees of freedom. You need to add a command to one of the joints.

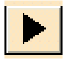
Double select on *Revolute.1* from the specification tree. The *Joint Edition* window appears.

Select the *Angle driven* option and select *OK*. You still have 2 degrees of freedom. This is because it generated constraints with the pins.

Delete the *Revolute.7* and *Revolute.9* joints. You now have 0 degrees of freedom.

Delete the *Planar.3* joint as well since it is unnecessary. The degrees of freedom do not change since this joint had no effect on the mechanism. The reason this joint is unnecessary is there is already a rigid joint defined between them.

Select the Simulation with Commands icon, select *On request*, drag the slider all the way to the right, change the *Number of steps* to 40 and select the Play Forward icon.

  The mechanism goes in motion. Notice that the two pins did not move. You still need to define rigid joints between the pins and respective parts.

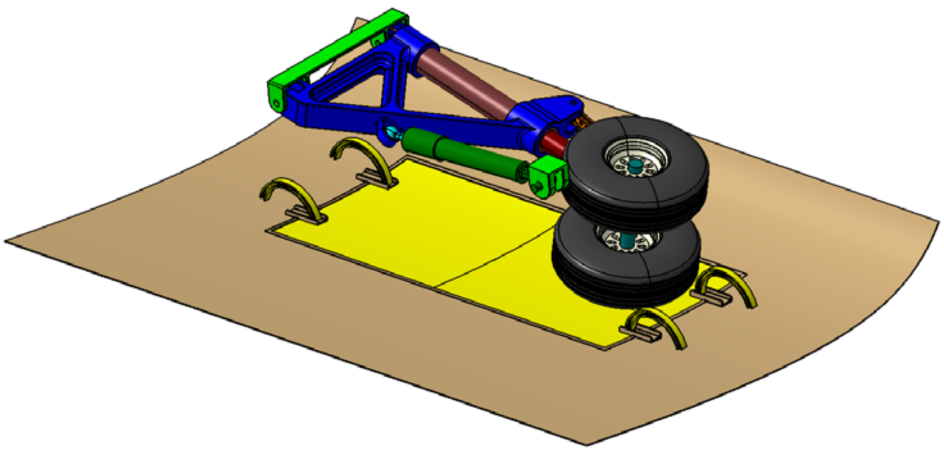
Select *Close*. The window closes. You can see that when you use *Auto Create* it may make mistakes on the joints that it creates. This requires that you check the mechanism out before calling it good. It is up to you to determine which method works best for the assembly that you are working on.

Close your document. There is no reason to save.

Law Review - Rules

In this review, you will be applying the different types of laws to a landing gear assembly. This will show how laws can simulate multiple commands simultaneously, some with more control and ease than others.

Open the Law Review document from the *Law Review* directory. The Landing Gear assembly should appear. All of the joints, limits, and commands have already been created for you.



Expand the *Mechanisms* branch. Notice that there are several commands to control this mechanism. You will be first creating a Knowledgeware Rule to control these commands.

Create a new Rule with a name and description of your choosing. Remember, you must switch to the Knowledge Advisor workbench.

Select *OK*. The *Rule Editor : Knowledgeware Rule Active* window will appear. You will be writing a rule that makes both the doors lower at the same rate, after they are partially open the landing gear will begin to lower as well. The two doors will be fully open before the landing gear is completely down.

Create the first If-Then statement for KINTime to be less than or equal to 6 seconds. This will be very similar to the process you used earlier. Make sure you select the mechanism from the tree first, then double select the appropriate member from the *Members of All* section within the *Rule Editor* window in order to add it to the If-Then statement.

The *Then* statement is going to be a little different. Anytime you are controlling multiple commands you will need to enclose them all in braces { }.

Set the *Left Door Angle* command equal to 5 degrees per second.

Set the *Right Door Angle* command equal to the Left Door Angle command.

Set the *Lifting Cylinder Length* equal to 0 inches. This will keep the landing gear stationary for the first six seconds. Be sure to end this statement with a brace.

Your rule should now look like the following.

```
/*Rule created by nathan 3/29/2012*/
if `Landing Gear\KINTime` <=6s
{
            `Landing Gear\Commands\Left Door Angle\Angle` = (`Landing Gear\KINTime` /1s)*5deg
            `Landing Gear\Commands\Right Door Angle\Angle` = `Landing Gear\Commands\Left Door Angle\Angle`
            `Landing Gear\Commands\Lifting Cylinder Length\Length` = 0in
}
```

Start another If-Then statement for the time greater then 6 seconds and less then or equal to 26 seconds.

Set the *Left Door Angle* command equal to 5 degrees per second. Be sure to subtract the initial time from the KINTime and then add the ending position of the previous step to the equation.

Set the *Right Door Angle* command equal to the Left Door Angle command.

Set the *Lifting Cylinder Length* equal to 0.5 inches per second. Your rule should now look like the following.

```
/*Rule created by nathan 3/29/2012*/
if `Landing Gear\KINTime` <=6s
{
            `Landing Gear\Commands\Left Door Angle\Angle` = (`Landing Gear\KINTime` /1s)*5deg
            `Landing Gear\Commands\Right Door Angle\Angle` = `Landing Gear\Commands\Left Door Angle\Angle`
            `Landing Gear\Commands\Lifting Cylinder Length\Length` = 0in
}
if `Landing Gear\KINTime` >6s and `Landing Gear\KINTime` <=26s
{
            `Landing Gear\Commands\Left Door Angle\Angle` = (`Landing Gear\KINTime`-6s) /1s*5deg+30deg
            `Landing Gear\Commands\Right Door Angle\Angle` = `Landing Gear\Commands\Left Door Angle\Angle`
            `Landing Gear\Commands\Lifting Cylinder Length\Length` = (`Landing Gear\KINTime` -6s)/1s*0.5in
}
```

Now you want the doors to stop moving and the landing gear to continue lowering.

Start another If-Then statement for the time greater then 26 seconds and less then or equal to 38 seconds.

Set the *Left Door Angle* command equal to 130 degrees. This is the ending value after the doors have traveled 5 degrees per second for 26 seconds. This will keep the doors at a constant 130 degrees for the duration of this step.

Set the *Right Door Angle* command equal to the Left Door Angle command.

Set the *Lifting Cylinder Length* equal to 0.5 inches per second. Be sure to subtract the initial time from the KINTime and then add the ending position of the previous step to the equation.

Your rule should now look like the following.

```
/*Rule created by nathan 3/29/2012*/
if `Landing Gear\KINTime` <=6s
{
		`Landing Gear\Commands\Left Door Angle\Angle` = (`Landing Gear\KINTime` /1s)*5deg
		`Landing Gear\Commands\Right Door Angle\Angle` = `Landing Gear\Commands\Left Door Angle\Angle`
		`Landing Gear\Commands\Lifting Cylinder Length\Length` = 0in
}
if `Landing Gear\KINTime` >6s and `Landing Gear\KINTime` <=26s
{
		`Landing Gear\Commands\Left Door Angle\Angle` = (`Landing Gear\KINTime`-6s) /1s*5deg+30deg
		`Landing Gear\Commands\Right Door Angle\Angle` = `Landing Gear\Commands\Left Door Angle\Angle`
		`Landing Gear\Commands\Lifting Cylinder Length\Length` = (`Landing Gear\KINTime` -6s)/1s*0.5in
}
if `Landing Gear\KINTime` >26s and `Landing Gear\KINTime` <=38s
{
		`Landing Gear\Commands\Left Door Angle\Angle` = 130deg
		`Landing Gear\Commands\Right Door Angle\Angle` = `Landing Gear\Commands\Left Door Angle\Angle`
		`Landing Gear\Commands\Lifting Cylinder Length\Length` = (`Landing Gear\KINTime` -26s)/1s*0.5in+10in
}
```

Your rule should now be ready to simulate your mechanism.

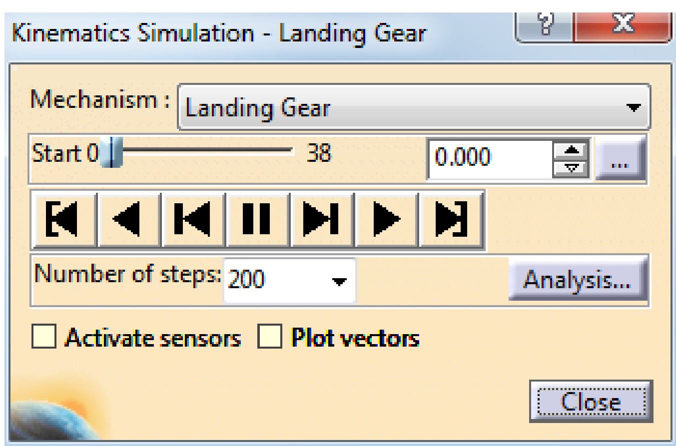
Select *OK* to the *Rule Editor* window.

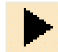
Switch back to the Kinematics workbench.

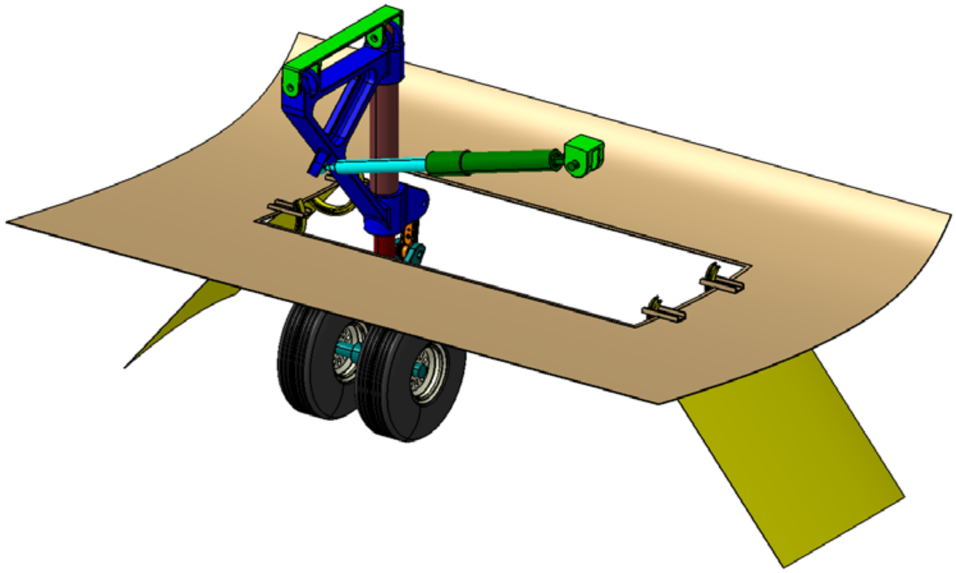
Select the Simulate with Laws icon.  The simulation window appears. An information window may appear stating the *Update all external references* option will be temporarily deactivated for the simulation. Select *OK* if necessary.

Change the *Maximum Time Bound* to be 38s, the length of your rule.

Change the *Number of steps* to 200.



Select the Play forward icon.  The doors should begin to open then the landing gear should lower.



Select the Start icon. 

Close the simulation window. This is how you go about creating a rule for a mechanism that has multiple commands and sequencing of events.

Save your document but do not close.

