TABLE OF CONTENTS

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*

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

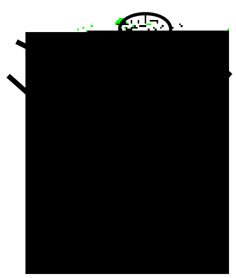
Select the *Pulley*. This will snap the compass to the geometric center of the pulley. The compass will be used to manipulate the pulley. The *Manipulation* toolbar appears; this will be discussed later.

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

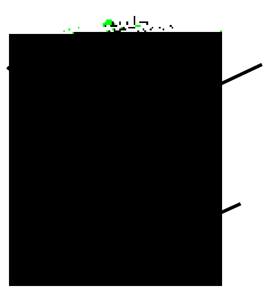
Compass

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,

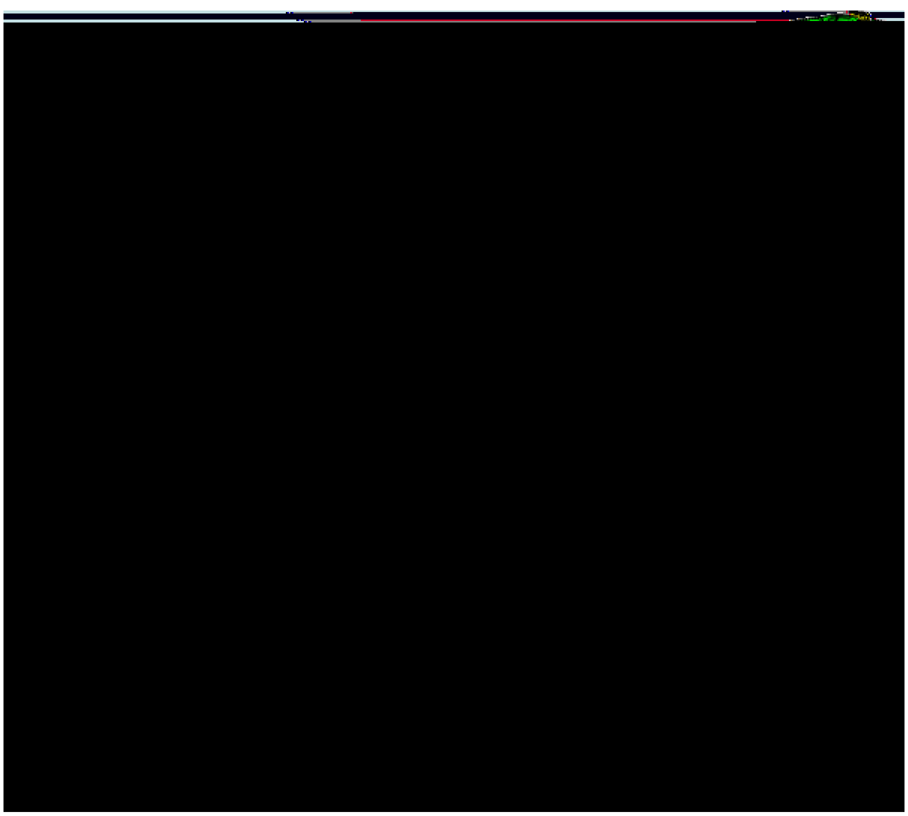
Translation increment	This is an incremental translation along one of the axes. Selecting the down (negative direction) or up (positive direction) arrows next to the value field will move the compass the specified amount in that direction
Rotation increment	This is an incremental angle movement about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

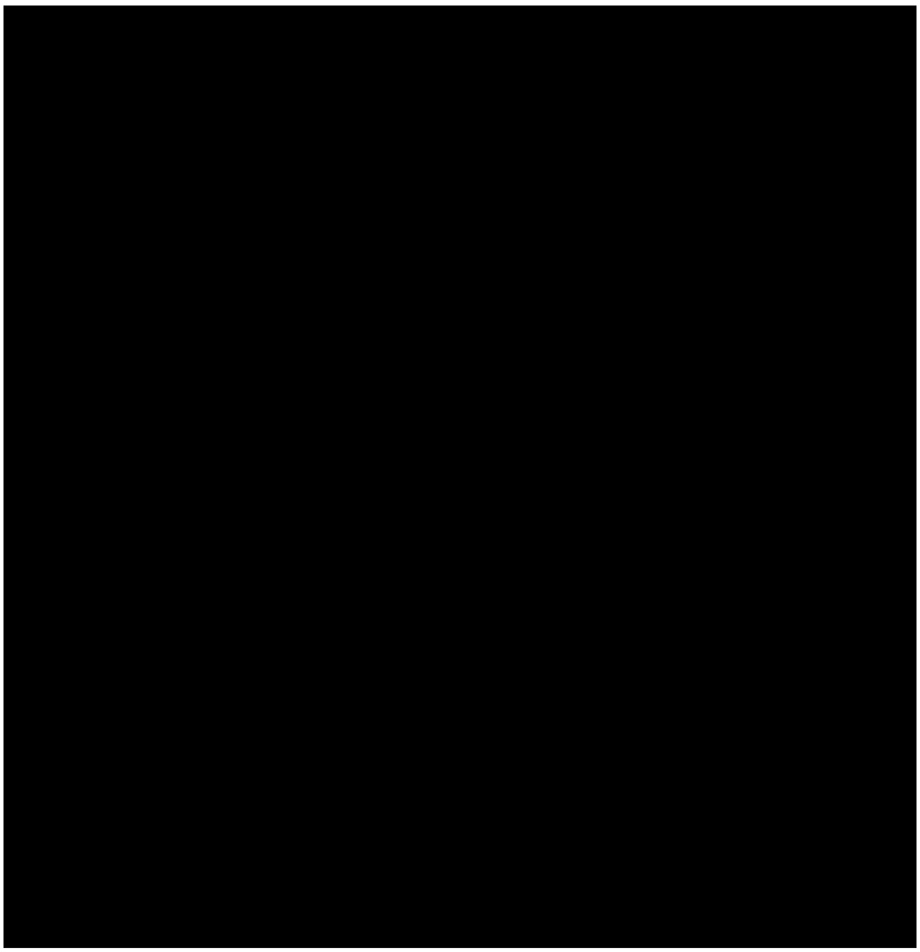


Now the position needs to be inserted into the simulation.

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 90E. 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.

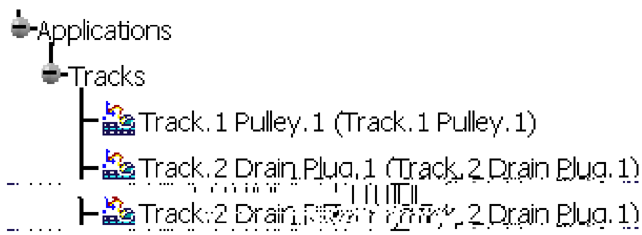
Select the Skip to Begin icon. This is located in the *Player* toolbar. This will snap the pulley back to the beginning of the track.

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

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.



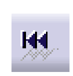
Now the track needs to be modified to work with the other drain plug.

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.

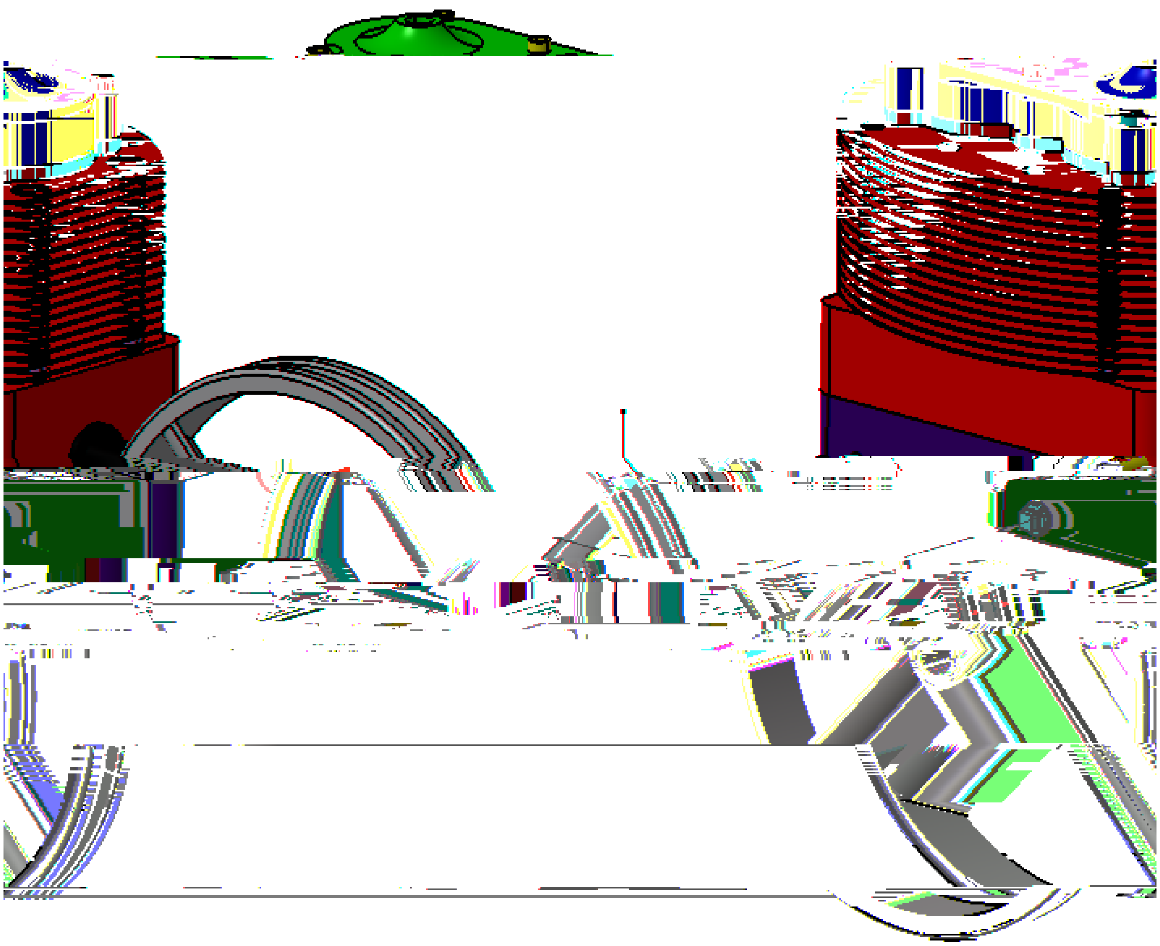
Select the Up Arrow again and select *Close* when done. This will make the total motion to be four inches along the positive W direction.

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.



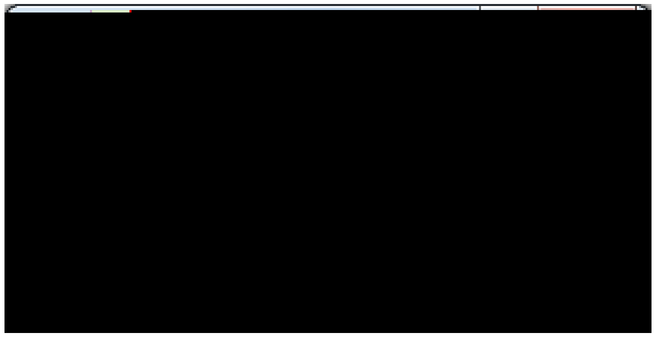
Those are the basic steps for creating tracks. There are a few more advanced features that will be investigated later. Next, the actions will be covered.

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

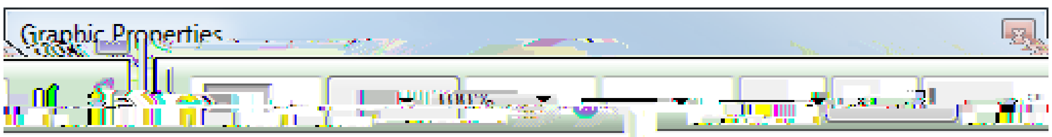
Color Action

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.

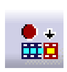
Select the Color Action icon.  The *Color Action* window appears. The options are very similar to the track options.



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.

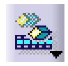
Select the Record icon.  This will record the color change.

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 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 seque

7KH ILUVW RSHUDWLRQ WKDW VKRXOG RFFXU LQ WKLV VHTXHQFH LV WKH
DFWLRQ RI WKH SXOOH\ EHLQJ UHPRYHG WKHQ WKH GUDLQ SOXJ
UHPRYHG DQG WKHQ WKH ILUVW GUDLQ SOXJ EHLQJ KLG

1RQH RI WKH WUDFNV KDYH XQLTXH QDPHV WR GHWHUPLQH ZKLFK
JRHV ZLWK ZKLFK RSHUDWLRQ)RU FRQYHQLHQFH LQ WKH
VLPXODWLRQ WKHUH PD\ EH KXQGUHGV RI WUDFNV 1DPH
WKH\ DUH FUHDWHG WR HDVH WKH SURFHVV

6HOHFW ColorPulley.1 DQG VHOHFW 5HIUHVK $IWHU 7KLV ZLOO SODFH WKH
VHTXHQFH

1RWLFH WKH GXUDWLRQ LV VHW WR PD\ EH DGGLWLRQDO
FRORUV $ YLVLELOLW\ DFWLRQ DQG FRORU DFWLRQ

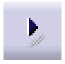
6HOHFW Track.1Pulley.1(Track.1Pulley.1) DQG VHOHFW 5HIUHVK $IWHU 7KLV
ZLOO SODFH WKH SXOOH\ UHPRYDO DFWLRQ
QRW PDWWHU LI WKHVH WZR KDSSHQ
WKH VDPH VWHS EXW JRRG WHFKQLTXH
FKDQJH DQG WKH SXOOH\ WR EH
VLPXOWDQHRXV PRGH

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

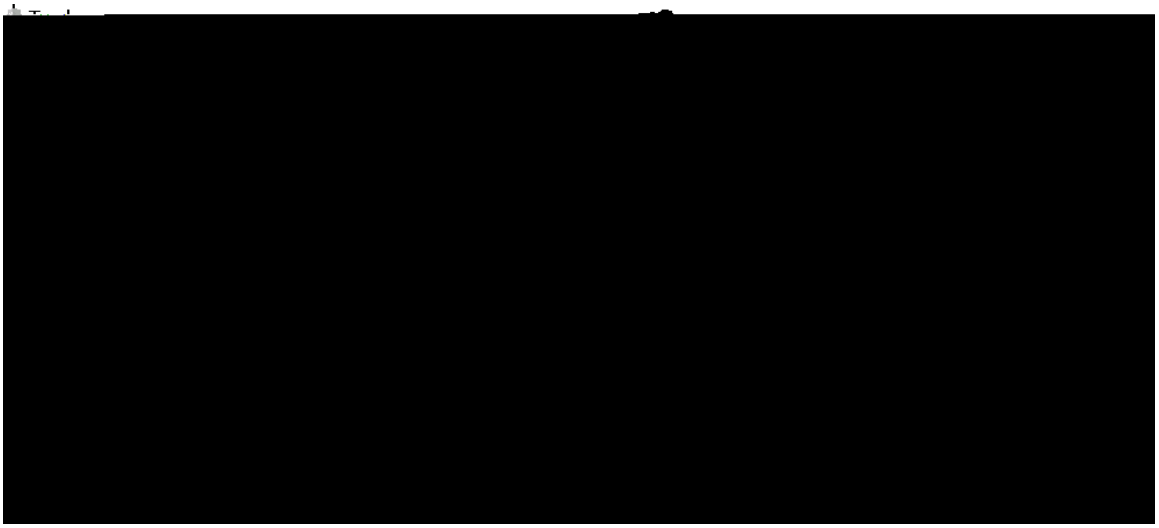
Notice that both the actions now have step 1 beside them. This means that both actions will

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.

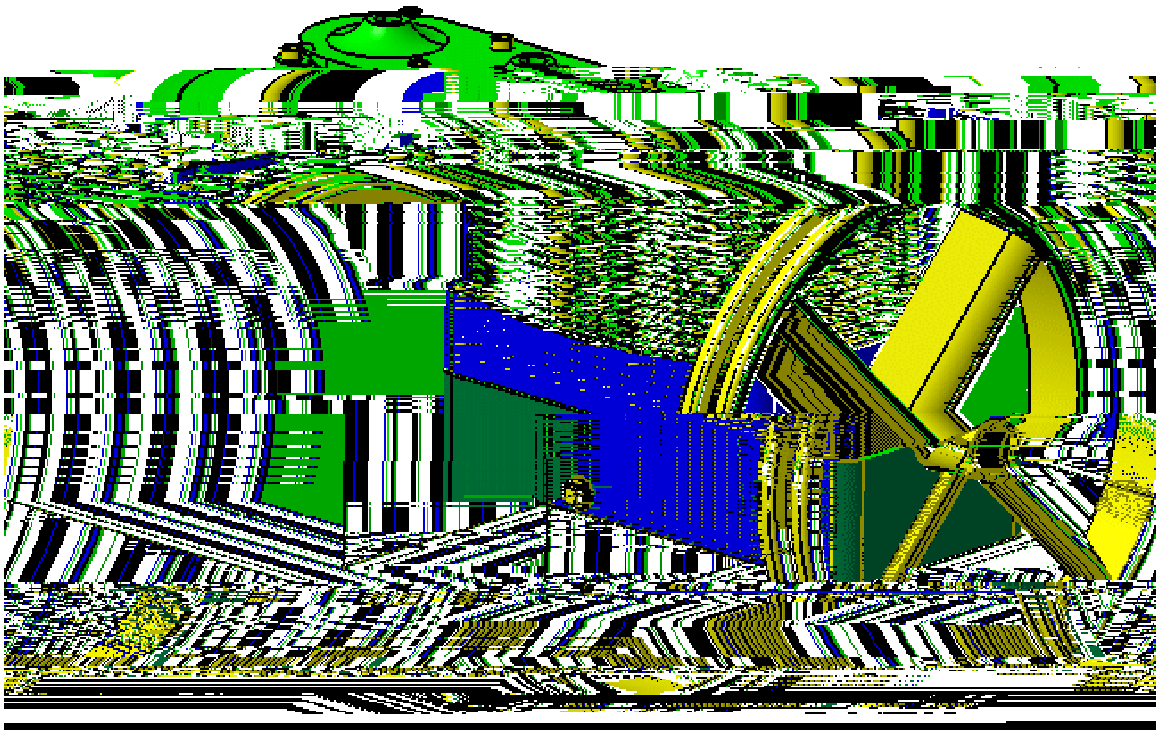
Also note the disappearance of the front drain plug. This is due to the hide command in the visibility action.



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

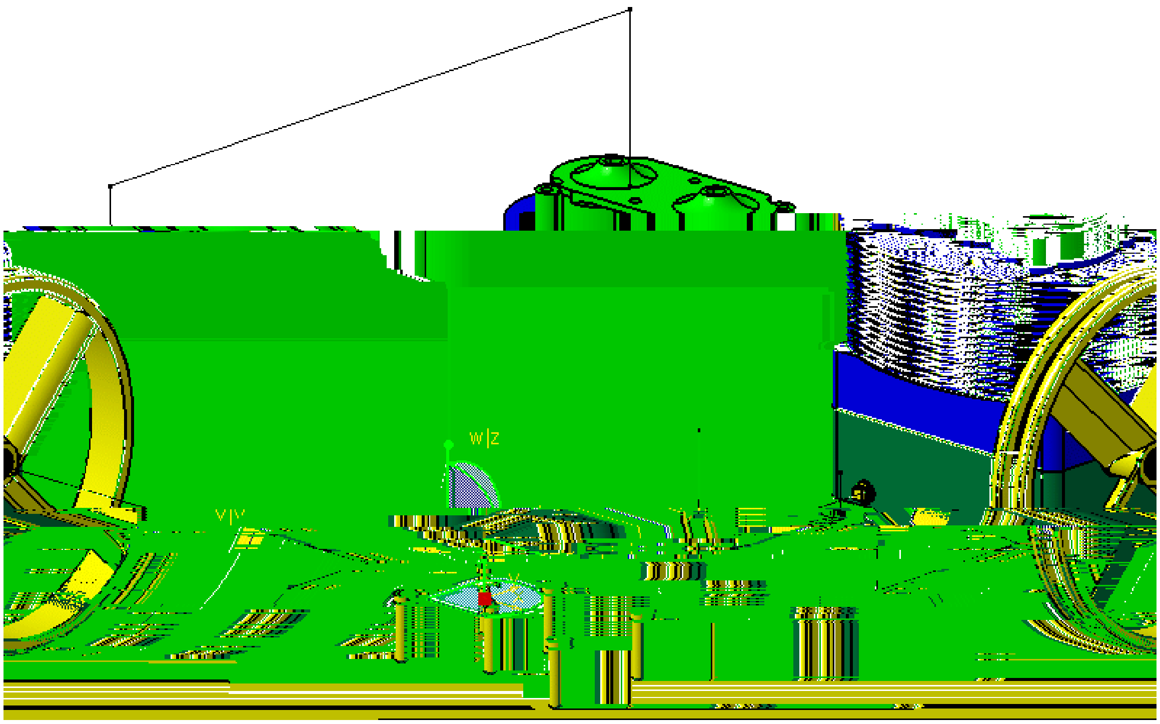
Vector Defines what axis the maximum angle can occur around

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.

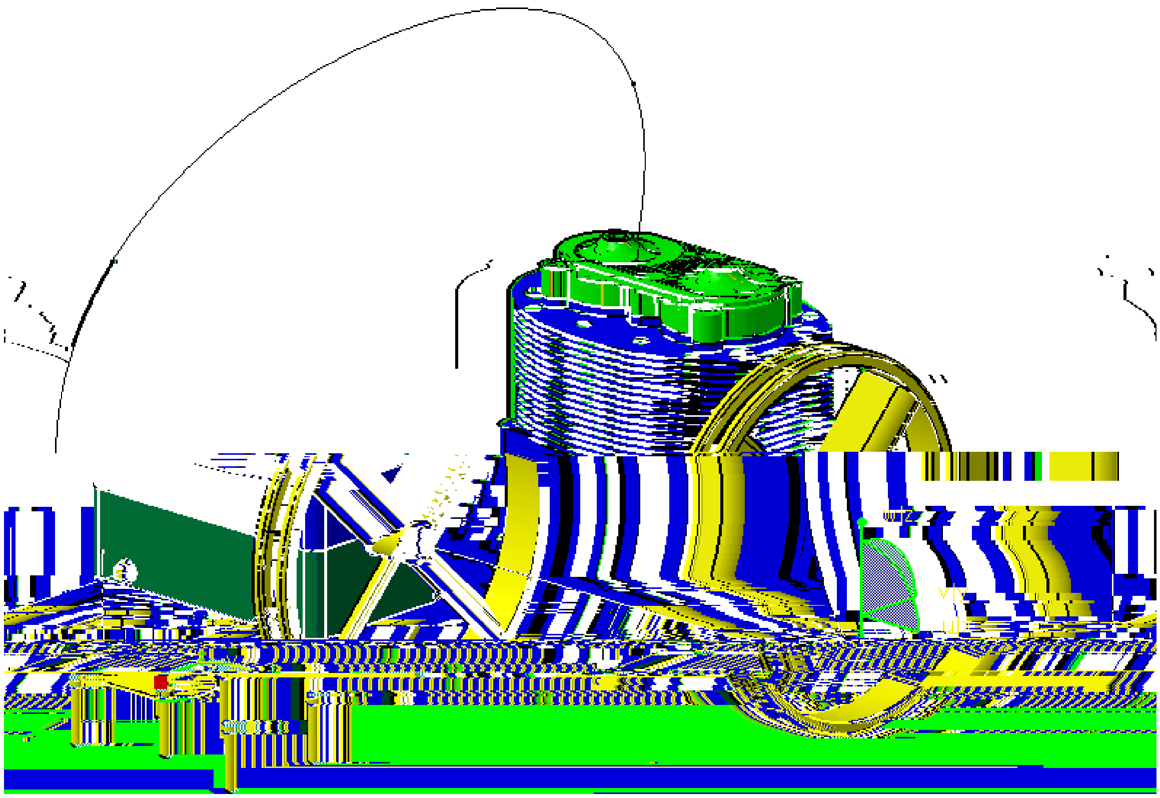


Skip to the beginning of the track, make sure your *Sampling Step* is 10s and play it forward. This is just to make sure the simulation works as you want. Notice the straight corners of the simulation track. This is due to the *Interpolater* being set to linear.

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.



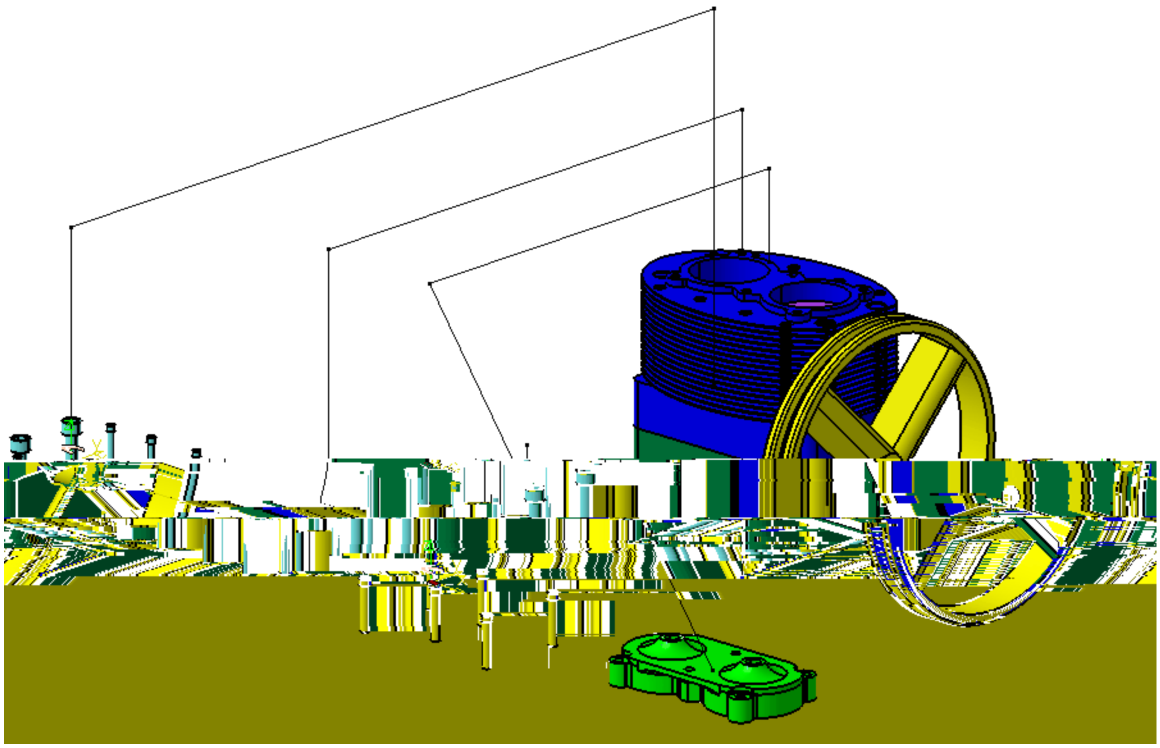
Skip to the beginning of the track and play it forward again. The bolts move in a very nice and fluid motion, but this is not what you are after. The bolts cannot simply move sideways out of the head. Instead, they need to pull straight out.

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

&UHDWH D WUDFN RI WKH JUHHQ KHDVGRIEROWV PRYLQJ WKH KHDGRII RYHU E\ WKH EROWV 7KH UHVXOW VKRXOG EH DV VKRZQ 7KH KHDG ORRNV VRPHWKL

&UHDWH D VKXWWOH IRU DOO RI WKH F\OLQGHU EROWV DQG WKH &\OLQGHU 7KH EROWV DUH WKH [DQG V[WEROWV 7KHUH

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

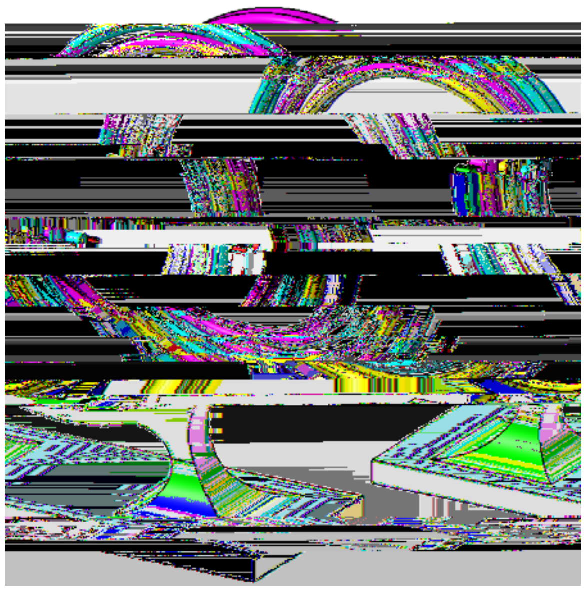
There are many joints available in Kinematics. Each joint has degrees of freedom and commands associated with it. A good understanding of these will make performing kinematics much easier. The table below gives an overview of the joints. Some of the joints can be created using axis systems but that will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

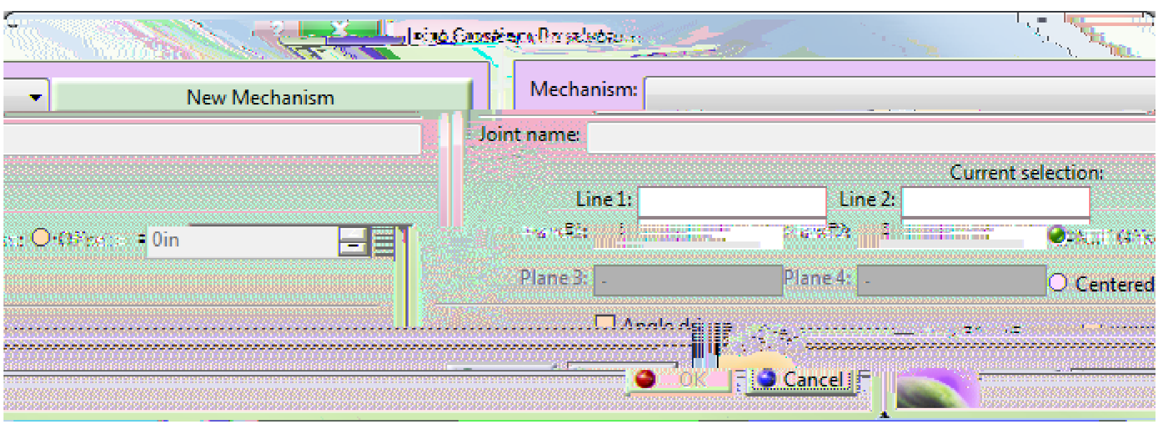
Revolute - Null Offset

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

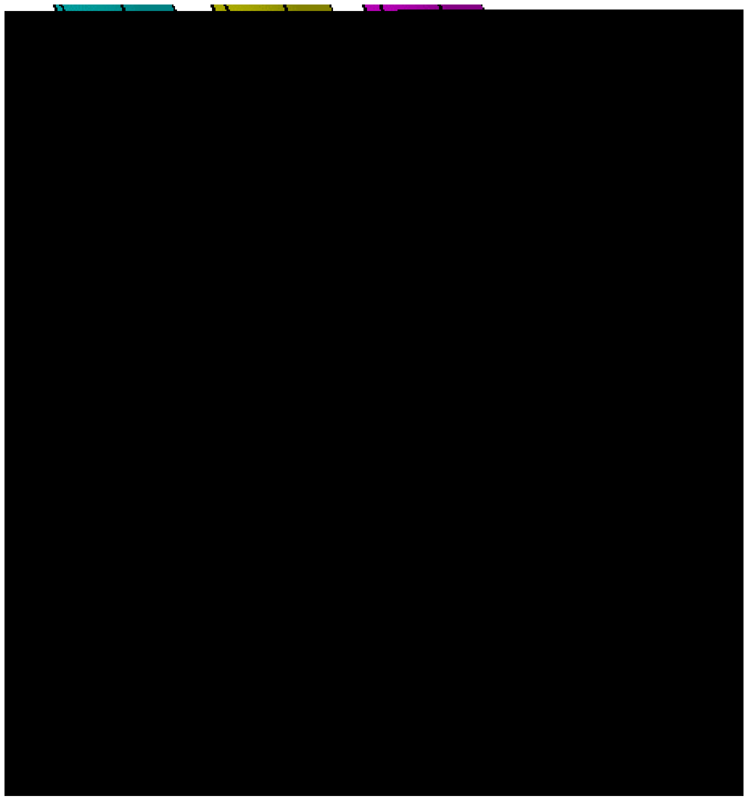
Select the *New Mechanism* button. The *Mechanism Creation* window appears. A mechanism needs to exist before a joint can be made. You can do this by using this option within the joint or by selecting pull down menu *Insert, New Mechanism*.

Key <u>Gyro</u> 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

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.

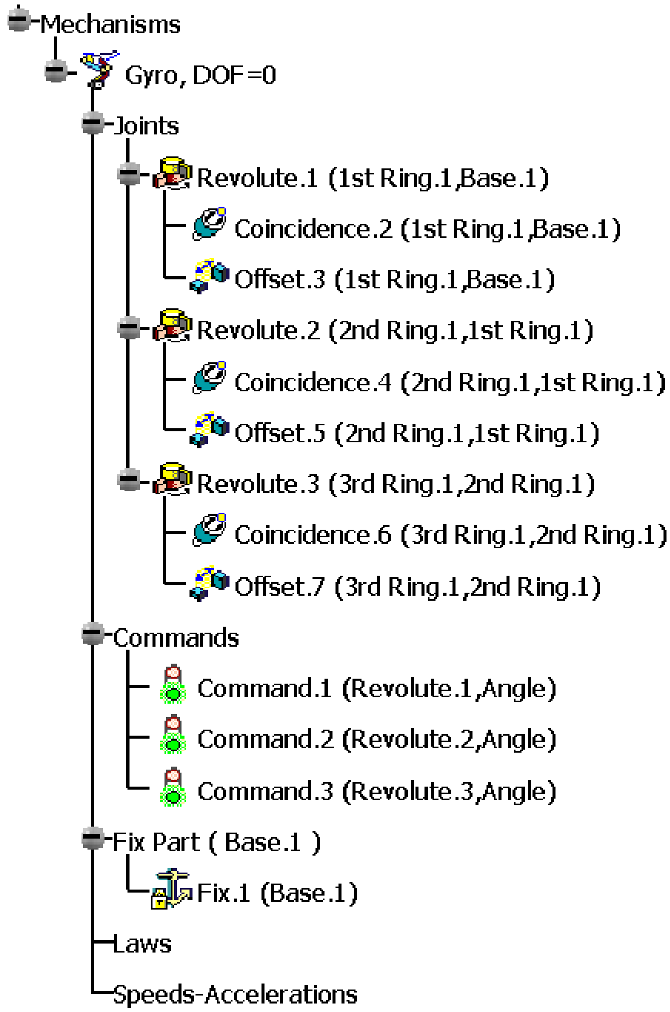
Define the revolute joint between the inner ring and the middle ring and select *OK* when the *Information* window appears. You do this in the same manner as you did the second revolute joint. The inner ring should move to align with the hole in the 2nd Ring.

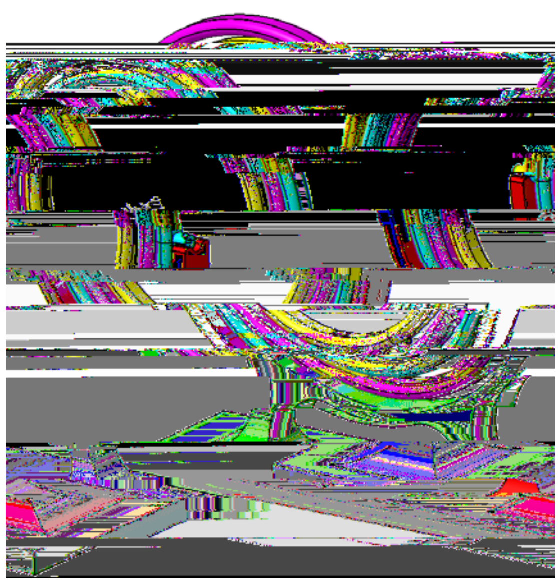
Select the Simulation with Commands icon again. The *Kinematics Simulation* window appears. This time there are three commands available to simulate.

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 Forwardid l M Y A l ¾ ה a

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





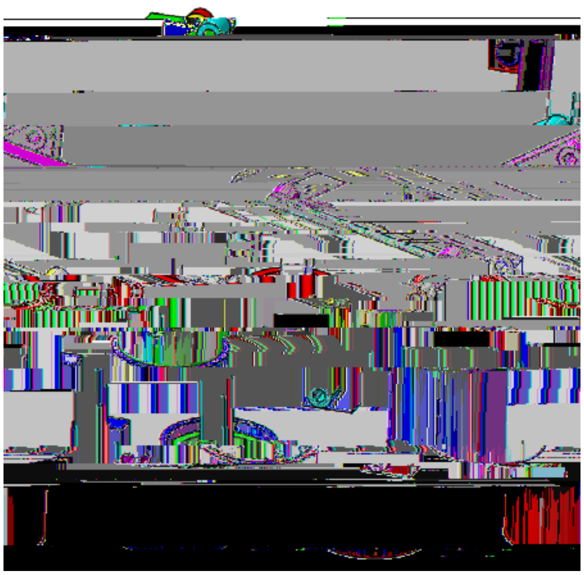
Save and close your document.

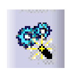
Assembly Constraints

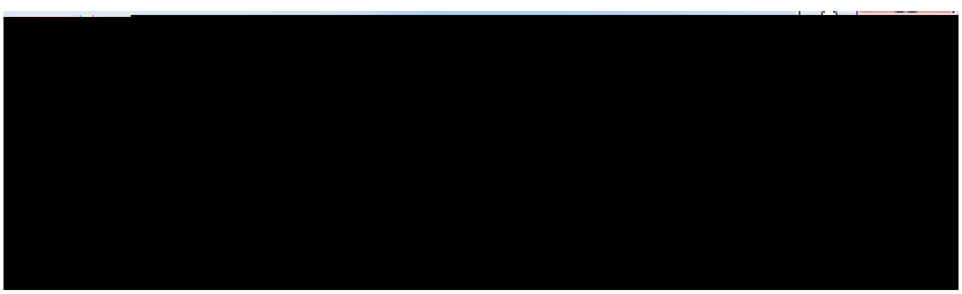
There are many times that you created an assembly and constrained it within Assembly Design. Assembly constraints are not recognized in Kinematics, therefore you have to define joints. However, assembly constraints can be converted into joints.

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*.

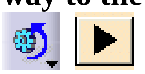
Select *OK.* Notice that you still have 1 degree of freedom. You need to add a command to one of the joints.

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.

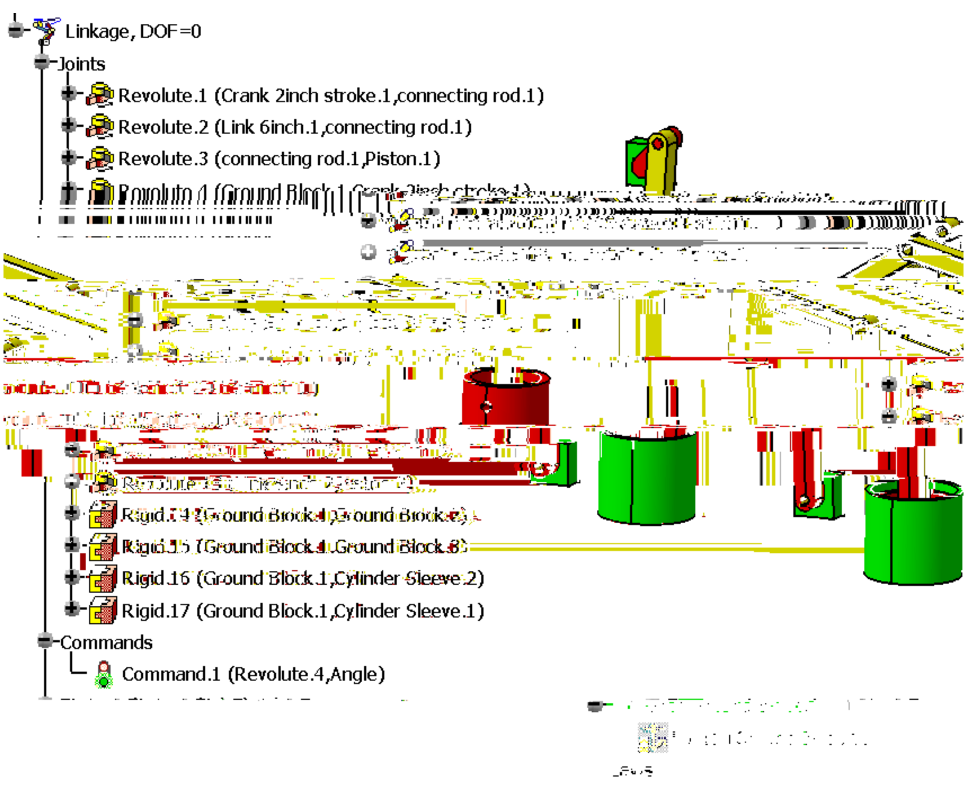
Select the Simulation with Commands icon, select *On request,* drag the slider all the way to the right, change the *Number of steps* to 40 and select the Play Forward icon.



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.

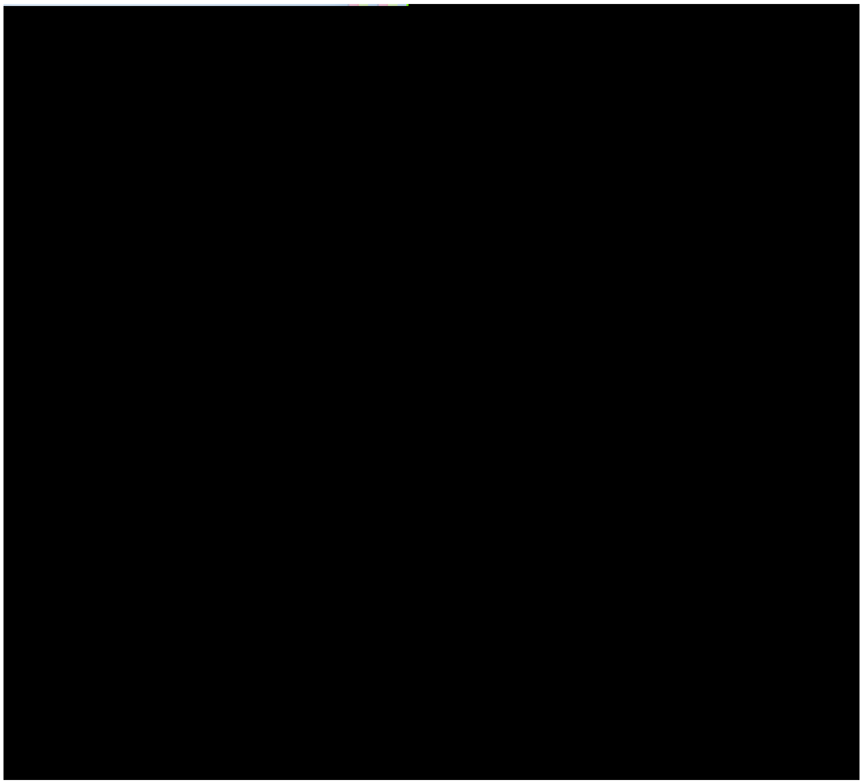
Your final mechanism should appear similar to the one shown below.



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

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

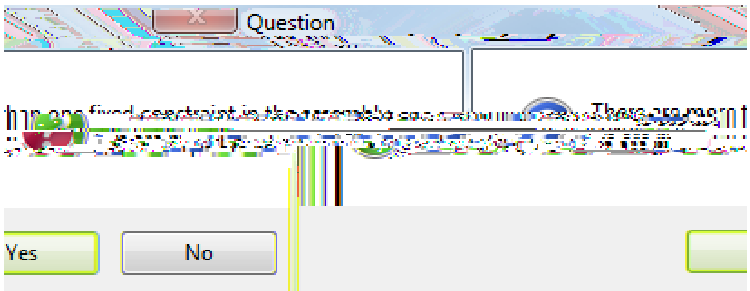
Delete Fix Removes the fixed part shown in the *Current Fixed Part* box

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

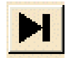
Select the *New Mechanism* button and create a mechanism called <u>Slider</u>. Items appear in the boxes. The first pair is shown. In the *Fix Constraints List* you should see the first fix constraint which is on the *Ground Block.2* component.

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..

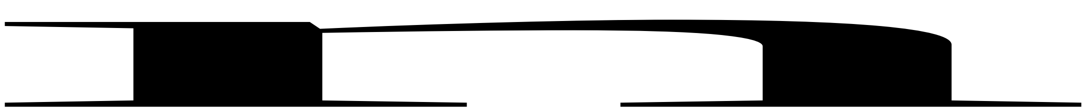


Select *Yes*. Two rigid joints were created and added to your mechanism. You should notice that the two components that are in the first pair are highlighted in the graphic area.

Select the *Coincidence* constraint in the *Constraints List*. The *Resulting type*

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.



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.

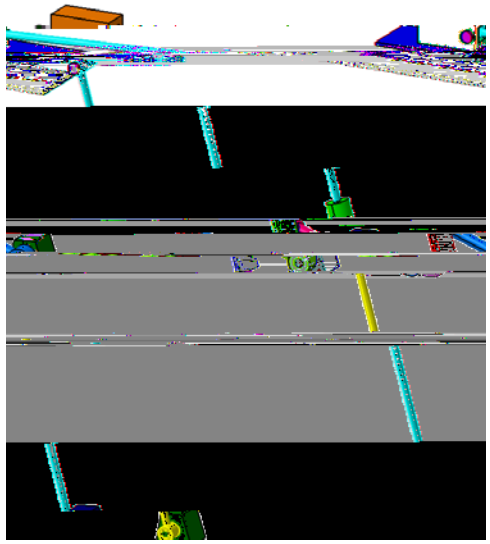
Create a rigid joint between the pin and the block and one between the other pin and the long link.

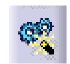
Simulate the mechanism again. This time the pins move with mechanism.

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.

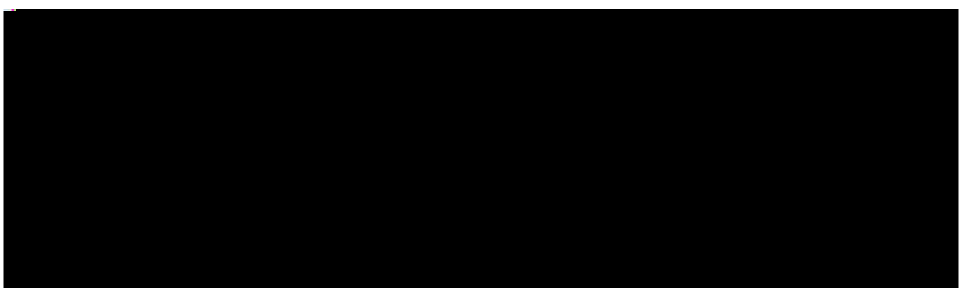
Open the original Assembly Constraints Advanced document again.



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.

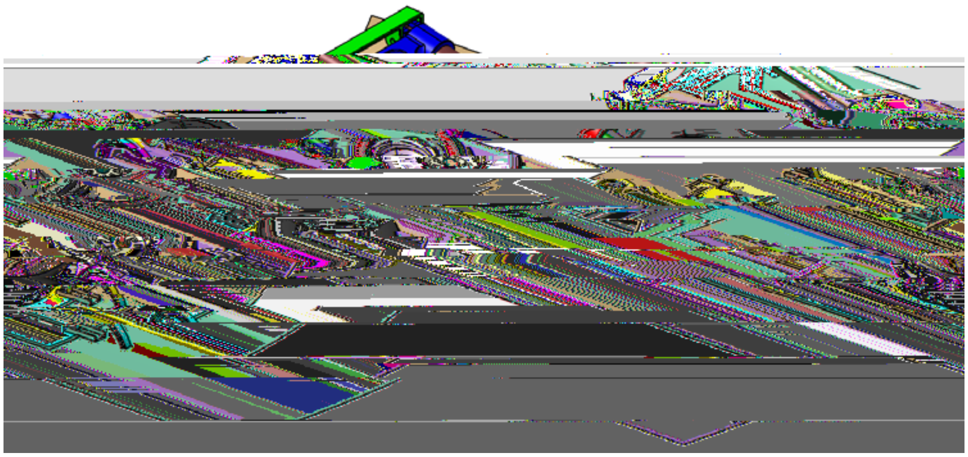
Delete the *Planar.3* joint as well since it is unnecessary. The degrees of freedom do not change since this joint had no effect on the mechanism. The reason this joint is unnecessary is there is already a rigid joint defined between them.

Select the

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.

<RXU UXOH VKRXOG QRZ ORRN OLNH WKH IROORZLQJ

6WDUW DQRWKHU W,I 7KHQ WKH VWDWHPHQW... HTXDO WR VHFRQGV

6HWLeft Door Angle FRPPDQG HTXDO WR GHJUHHV... LQLWLDO WLPH IURP WKH .,17LPH... HTXDWLRQ

6HWRight Door Angle FRPPDQG HTXDO WR WKH /HIW 'RRU $QJOH F...

6HWLifting Cylinder Length HTXDO WR LQFK... ORRN OLNH WKH IROORZLQJ

1RZ \RX ZDQW WKH GRRUV WR VWRS...

6WDUW DQRWKHU ,I 7KHQ VWDWHPHQW... HTXDO WR VHFRQGV

6HWLeft Door Angle FRPPDQG HTXDO WR... WKH GRRUV KDYH WUDYHOHG GHJUHHV... FRQVWDQW GHJUHHV IRU WKH GXUDWLRQ RI WKLV VWHS

6HWRight Door Angle FRPPDQG HTXDO WR WKH /HIW 'RRU $QJOH F...

6HWLifting Cylinder Length HTXDO WR LQFK... LQLWLDO WLPH IURP WKH .,17LPH... HTXDWLRQ

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
}
```

```
t Door Angle\Angle` = (`Landing Gear\KINTime`-6s) /1s*5deg+30deg
ght Door Angle\Angle` = `Landing Gear\Commands\Left Door Angle\Angle`
```

Your rule should now be ready to simulate your mechanism.

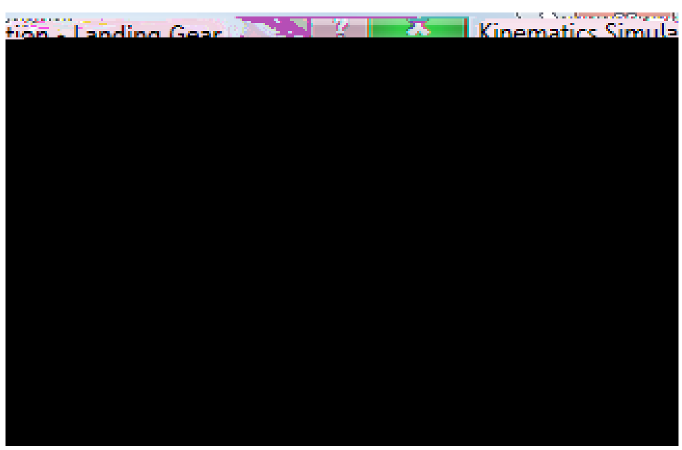
Select *OK* to the *Rule Editor* window.

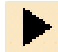
Switch back to the Kinematics workbench.

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.

