Tutorial 5

Kinematics

Purpose:

To give the student the basic knowledge related to the animation of a simple mechanism and to perform a spatial analysis of some of its components.

Reference:

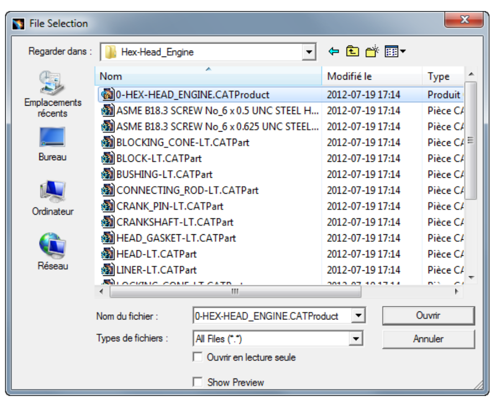
Use the following product file and its related parts: ***05-Hex-Head_Engine.CATProduct.***

1 – Launch CATIA®

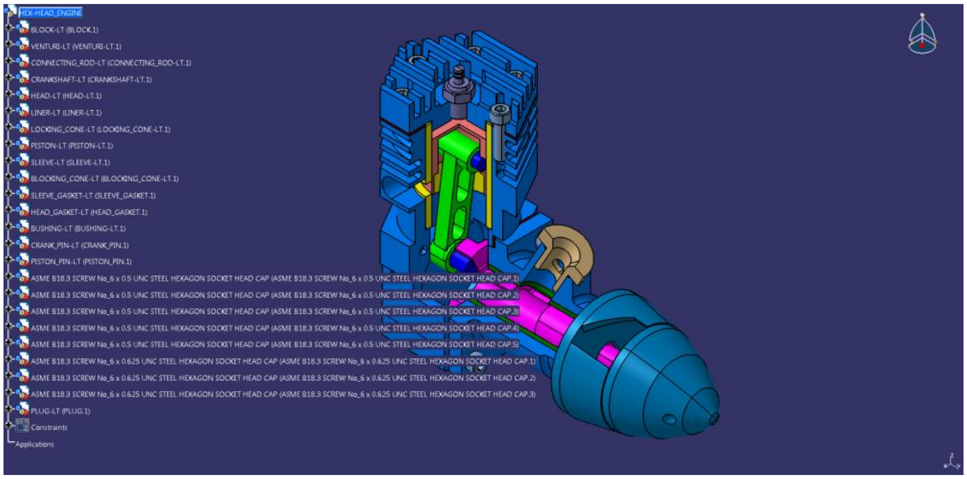
- If a product file is automatically created, close it.

2 – Open the reference product

- Use ***File>Open*** and browse to locate the folder named ***Hex-Head_Engine***.
- Select the file named ***05-Hex-Head_Engine.CATProduct***.



- Click the ***Open*** button. The product will appear on the screen and chances are that the ***Assembly Design*** workbench will be active.

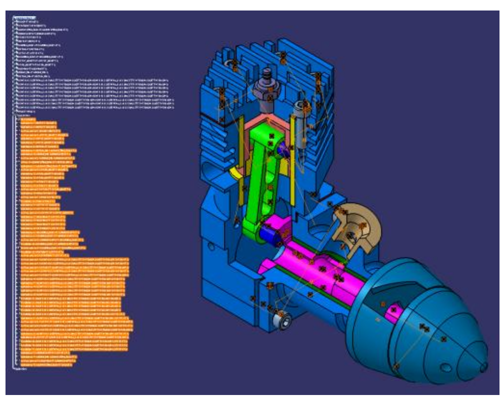


3 – Make a working copy of the product

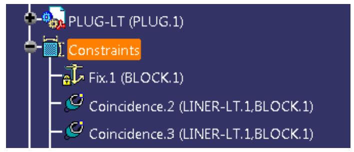
- Use ***File>Save As*** and save the same product under a new name such ***05-Hex-Head_Engine-Kin.CATProduct***. This will keep a copy of the original project as a reference.

4 – Delete the assembly constraints

Kinematics joints create new assembly constraints. In order to avoid any conflict between assembly constraints, it is easier to delete all original assembly constraints at first.



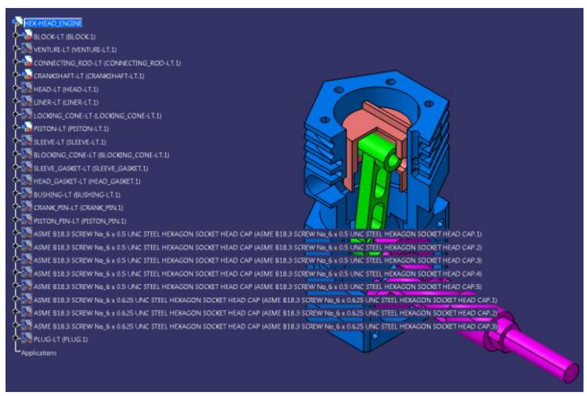
- Select the ***Constraints*** item in the ***Product Specification Tree*** and delete it. All assembly constraints will be deleted simultaneously.



5 – Hide nonrelevant parts

It is usually a good practice to identify key parts at first and build the kinematics mechanism with them only. Then other parts can be added to moving components.

- Keeping the SHIFT key down, select the assembly's first part (BLOCK-LT) and last part (PLUG-LT). Release the SHIFT key.
- Keeping the CTRL key down, select the following parts: BLOCK-LT, CONNECTING_ROD-LT, CRANKSHAFT-LT and PISTON-LT.
- Click the ***Hide/Show*** tool icon  to make other parts disappear.

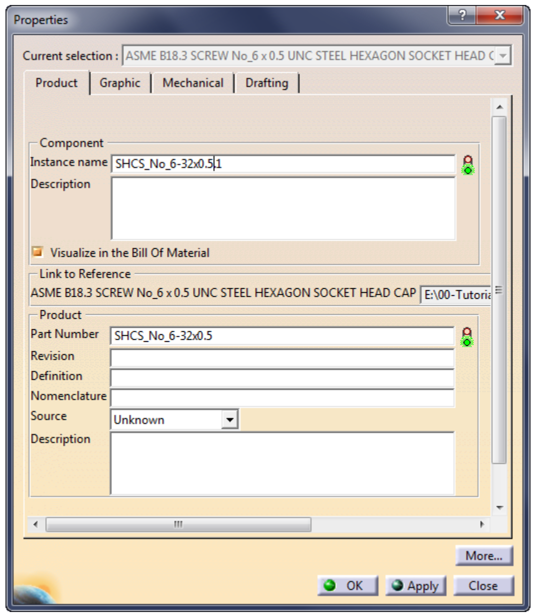


6 – Rename screw instances

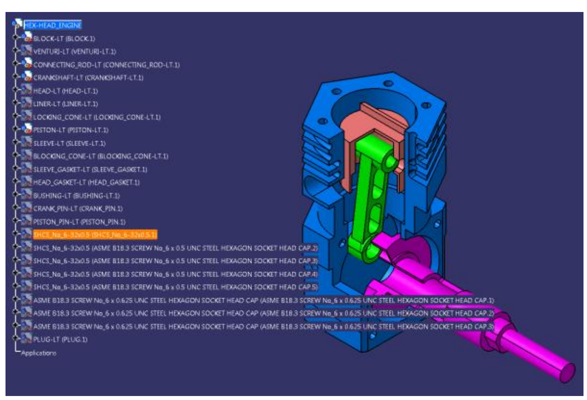
The screws used in the assembly come from the default CATIA® Standard Parts catalog and have names that take a lot of room in the working area. Since the screw files are now standalone in the exercise folder, the link is broken with the catalog and the elements can now be renamed.

- Bring the mouse cursor on the first 0.5 inch long screw item, in the ***Product Specification Tree***, and right click to access the context menu.
- Select the ***Properties*** item in the menu.

- In the ***Part Number*** area, replace the original screw name (ASME B18.3 SCREW No_6 x 0.5 UNC STEEL HEXAGON SOCKET HEAD CAP) by the following one: SHCS_No_6-32x0.5
- Copy the new name (CTRL+C) and paste it (CTRL+V) in the ***Instance Name*** area, taking care to keep the ***.1*** at the end of the name to clearly identify this instance.

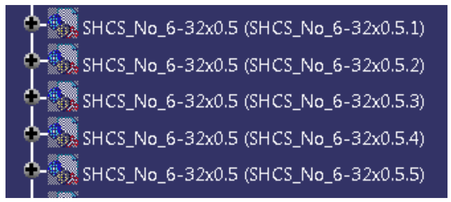


- Click on the ***Apply*** button and look at the ***Product Specification Tree***: All 0.5 inch long screw instances have changed their reference name.

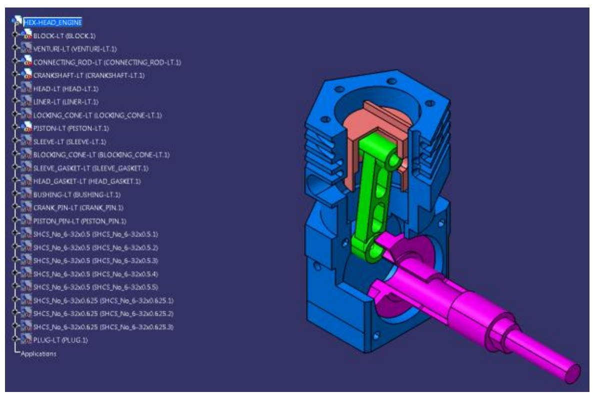


- Click on the ***OK*** button to close the dialog box and edit the properties of the second 0.5 inch long screw.

- Go directly in the ***Instance Name*** area and paste the new name, keeping the ***.2*** this time. The new name is still in memory, from the previous operation.
- Click on the ***OK*** button to close the dialog box, and repeat the preceding operations for the following three screws, always taking care not to delete the instance number when pasting.



- Repeat the complete procedure with the 0.625 inch long screw.



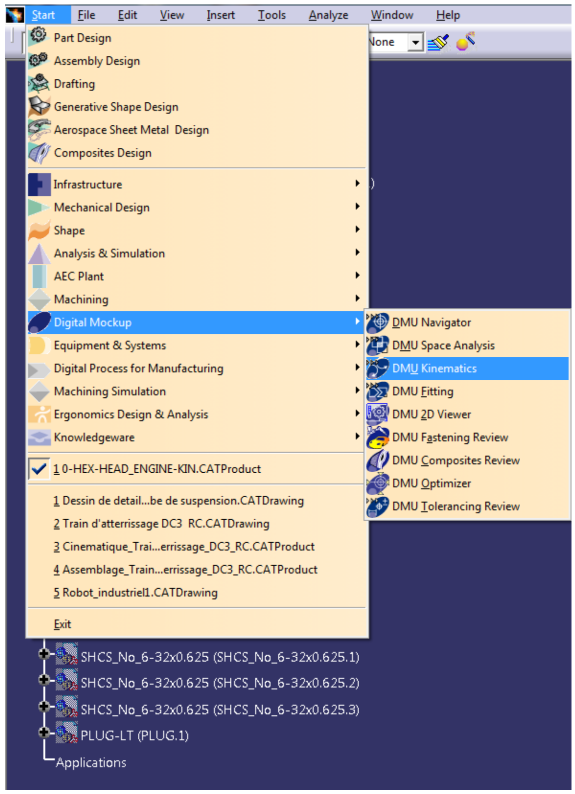
- Save the product.

Now, we can see something!

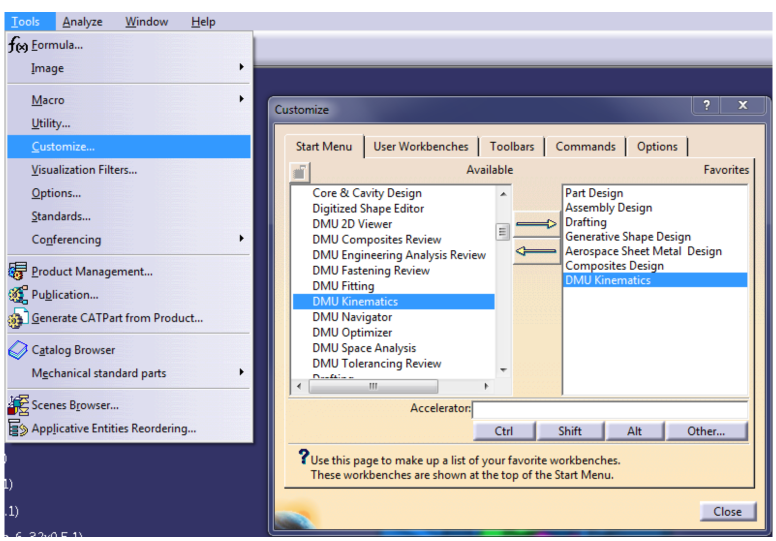
When looking at the assembly, we can note that none of the parts is actually touching another. This makes no difference to the kinematics joints since the relations they will use refer to axis alignment, face coincidence etc.

7 – Loading the DMU Kinematics workbench

To load the proper workbench, use the ***Start>Digital Mockup>DMU Kinematics*** option.

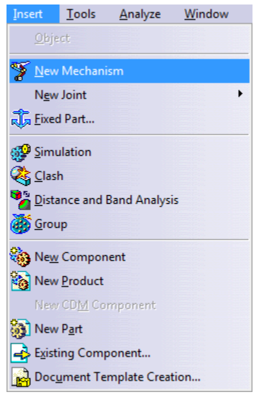


It is also possible to pre-load the workbench by using ***Tools>Customize*** and by adding the ***DMU Kinematics*** workbench in the list of workbenches to be readily available to the user.



8 – Create a new mechanism

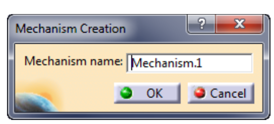
It is possible to create a new mechanism in the product structure by using the ***Insert>New Mechanism*** pull-down option.



However, since the first logical operation to make is to fix a base part in the mechanism and since the ***Fixed Part*** dialog box offers a ***New Mechanism*** button, two actions can basically be done in the same sequence.

- Click the ***Fixed Part*** tool icon . The ***New Fixed Part*** dialog box pops up.

- Click the ***New Mechanism*** button. The ***Mechanism Creation*** dialog box pops up and the name *Mechanism.1* is proposed. Accept this name or enter a new name and then, click the ***OK*** button to get back to the fixed part identification.



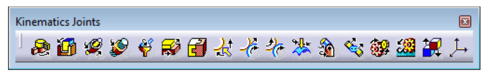
- Select the block.

Automatically, a ***Mechanism.1*** substructure is created at the end of the ***Product Specification Tree***. The fixed part is identified and a ***Fix*** constraint is created in the ***Constraints*** group.

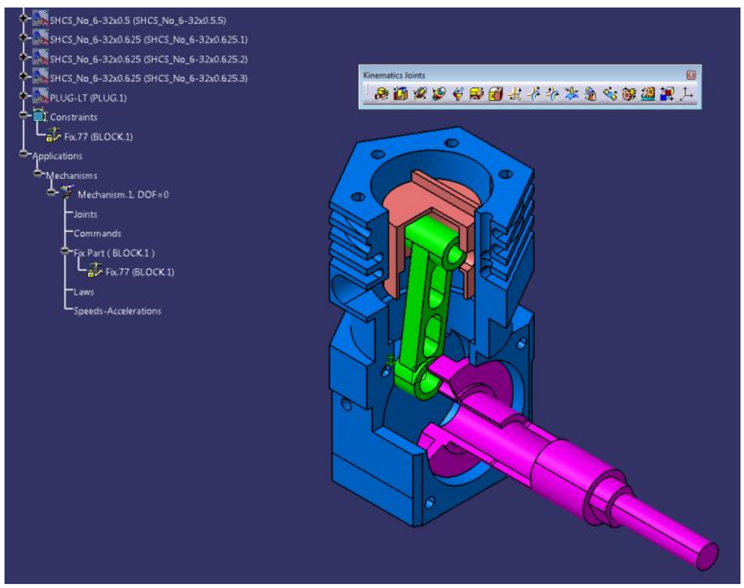
9 – Create the first joint and the first command

This engine mechanism is quite simple and will involve very few different joints to be animated. However, it is enough to teach the basics.

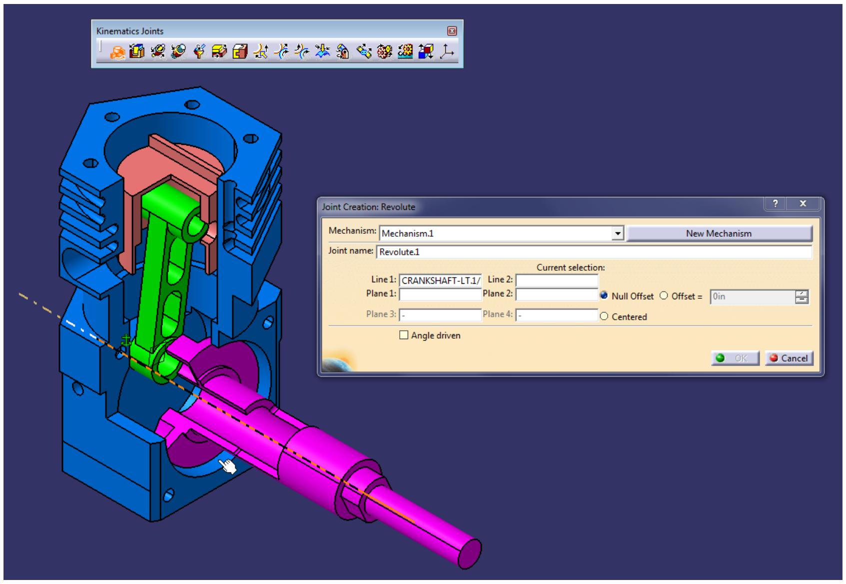
The first joint to be used in this mechanism is a ***Revolute Joint***. It asks for two coincident axes to allow a rotation motion and two parallel faces to stay at a constant distance, if not coincident, to avoid any sliding along the axis. The joint can be converted into a command when necessary.

- Expand and pull the ***Kinematics Joints*** toolbar 

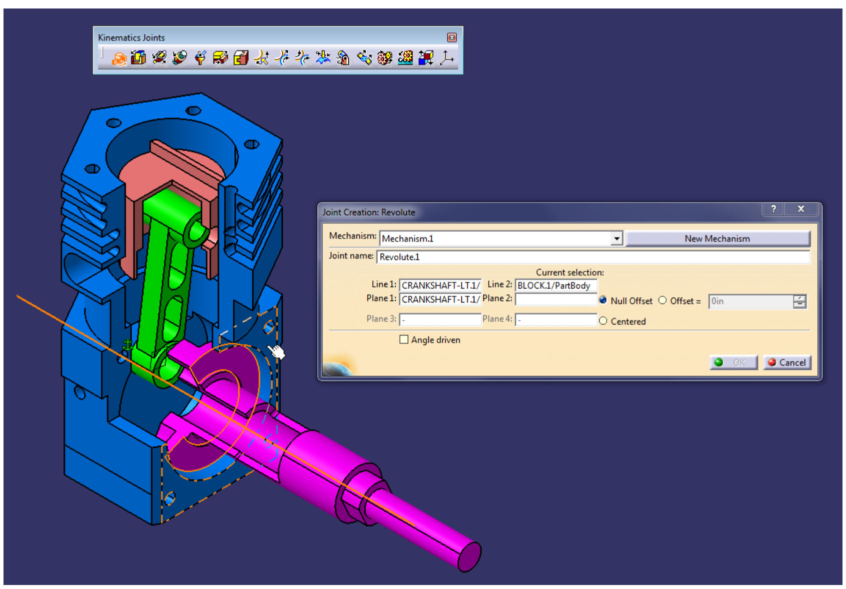
in the working area. It is hidden under the ***Revolute Joint*** tool icon .



- Click the ***Revolute Joint*** tool icon .
- Select any crankshaft's cylindrical face to retrieve its axis and then, select the bottom cylindrical face of the block to get a coincident axis.



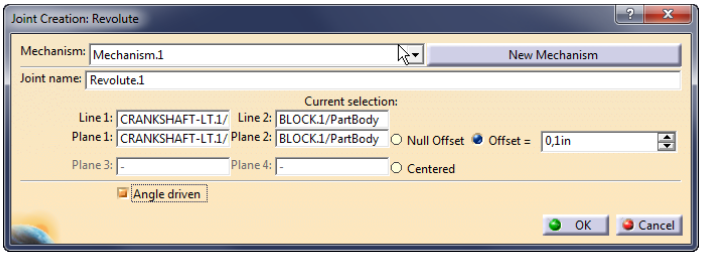
- Select now the crankshaft's flat flange face and the block's front face.



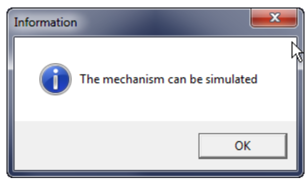
- Since the two faces are not coincident, it is necessary to click the ***Offset*** checkbox in the dialog box. The value that will appear is not important. It is the actual distance between the two parallel faces.

This mechanism will be animated by a command that does not represent reality but that will avoid any problems due to static locks: the rotation of the crankshaft will be used as the main command.

- Click the ***Angle driven*** checkbox in the dialog box and then, click the ***OK*** button to complete simultaneous joint and command creation.



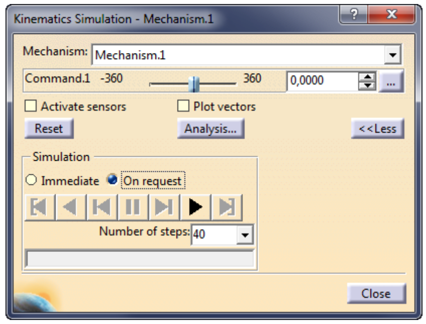
Since all the required elements are present to define a simple mechanism, a confirmation dialog box pops up. Click the ***OK*** button to close it.



10 – Running the basic mechanism

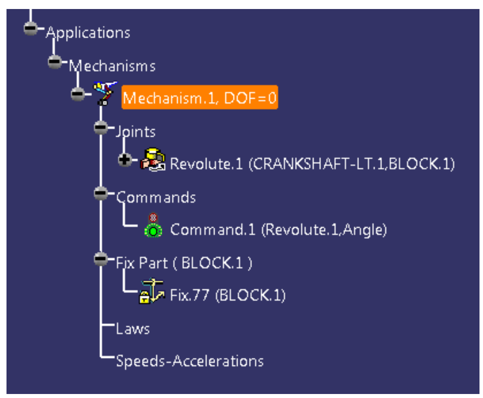
When possible, it is usually a good practice to try to run the mechanism when it is feasible. One thing though: since all parts are free to move with respect to each other, it is important to make such tests by making sure to get back to the original stage in order to be able to pursue with the creation of other joints. In some cases, a 360° rotation will be used, in others, a back and forth testing will be done. It may also be preferred to use the ***Reset*** button, but the main idea is not to forget to get back to the original location when the test is made. It this is forgotten, it will be necessary to bring the parts to their original location manually, using the ***Assembly Design*** workbench tools.

- Click the ***Simulation with Commands*** tool icon . The ***Kinematics Simulation*** dialog box pops up.
- Click the ***On request*** checkbox to avoid non-desired motion of the mechanism when sliding the cursor.
- Select and slide the cursor to the far right end, asking then for a full 360° rotation.
- Click the ***Play forward*** button. The crankshaft makes a complete revolution.



- Click on the ***Close*** button to complete the test.

Note that, in order to run, a mechanism must display a number of DOF – *Degrees of Freedom* – equal to *0*. However, the inverse is not necessary true: for a given mechanism, a number of conflicting joints and constraints could bring the number of DOF to *0* and the mechanism could never be animated…

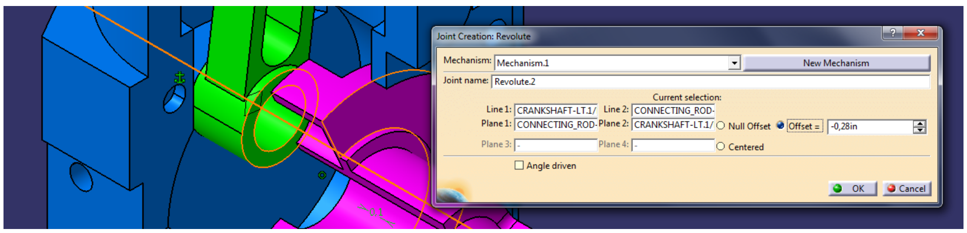


Note also that, instead of using the ***Simulation with Commands*** tool, it is possible to double-click the ***Mechanism.1, DOF=0*** item in the ***Product Specification Tree*** to access the same ***Kinematics Simulation*** dialog box.

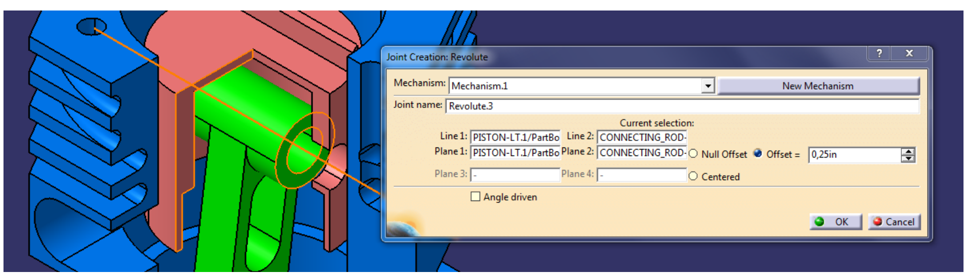
11 – Adding the remaining joints

To work, this basic mechanism still needs three more joints: two to link the connecting rods to the crankshaft and to the piston, and one to align the piston in its cavity. These joints will not be used to create any other command though. At the end, only the command already created will drive the entire mechanism.

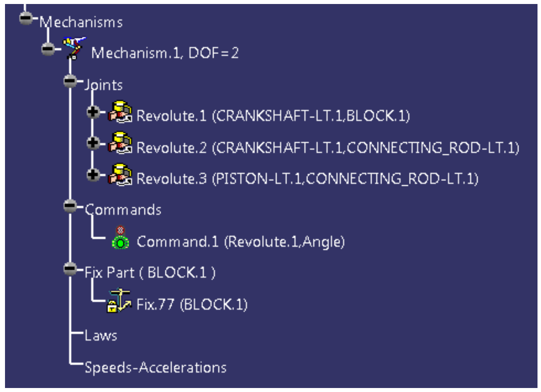
- Double-click the ***Revolute Joint*** tool icon to keep its action continuous.
- Select the crankshaft flange hole's cylindrical face and the connecting rod bottom hole's cylindrical face that have coincident axis and select also parallel planar faces on the two elements. Do not forget to check the ***Offset*** checkbox when the two planar faces are not coincident.
- Do not click the ***Angle Driven*** checkbox. This joint must not be converted in a command.
- Click the ***OK*** button to complete the joint creation.



- Now, select the piston hole's cylindrical face and the connecting rod top hole's cylindrical face that have coincident axis and select also two parallel planar faces on the elements. Do not forget to check the ***Offset***.
- Do not click the ***Angle Driven*** checkbox. This joint must not be converted in a command either.
- Click the ***OK*** button to complete the joint creation.
- Click the ***Cancel*** button to end the continuous behavior.

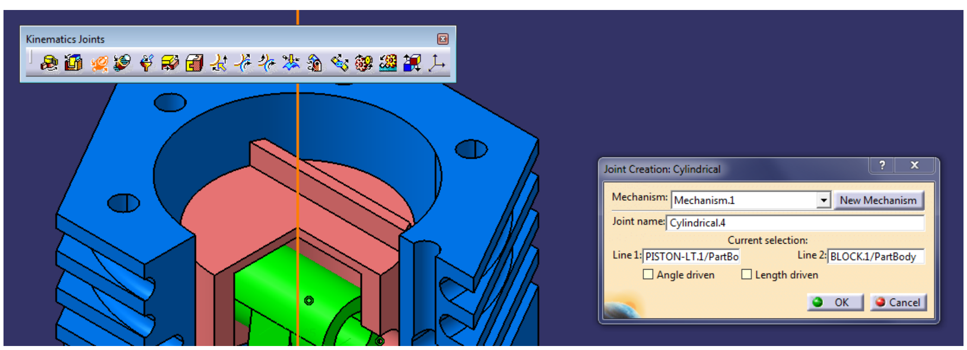


Three joints and a command are now present in the mechanism. Two more DOF must be locked to get a fully constrained mechanism.

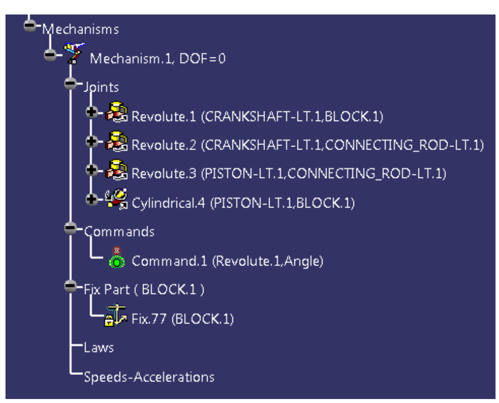


The last joint to create must constrain the piston's displacement in the block. In this case, a cylindrical joint will be used since it allows a single linear displacement on an axis.

- Click the ***Cylindrical Joint*** tool icon .
- Select one of the piston's cylindrical faces and the block's concentric cylindrical face to retrieve both coincident axes.
- Click the ***OK*** button to complete the joint creation.

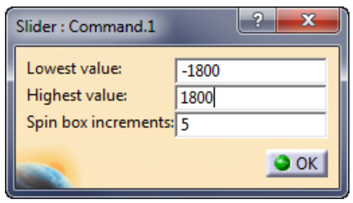


A confirmation message appears and the mechanism can now be simulated.

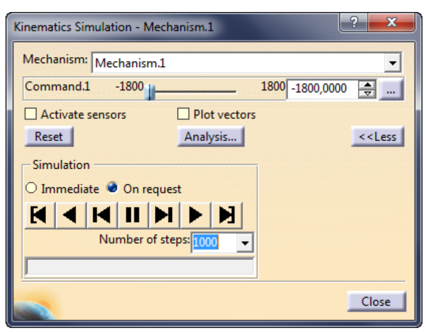


12 – Run the simulation

- Use the ***Simulation with Commands***  tool icon or double-click on the ***Mechanism.1, DOF=0*** item in the ***Product Specification Tree*** to obtain the ***Kinematics Simulation*** dialog box that will allow the simulation of the entire mechanism.
- With the ***On request*** checkbox selected slide the cursor to one of its ends and click the ***Play forward*** button.
- To increase the number of revolutions, click the ellipsis (…) button to modify the range limits. Using *-1800* and *+1800* as values will set the command total range to 3600° - ten turns – which will give enough time to look at the mechanism.



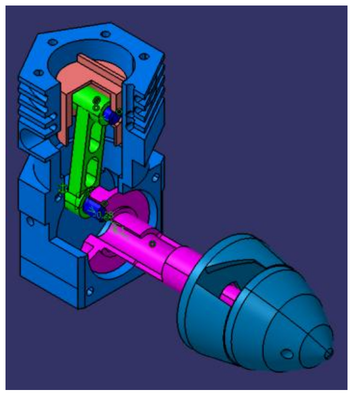
- Slide the cursor to each of the ends and run the simulation in order to get a simulation of the mechanism taking full advantage of the defined range. With so many turns though, the simulation is running too fast to properly analyze the displacement of each part.
- Edit the content of the ***Number of steps*** edition box to make it *1000*. Some alternate values are available by clicking the down pointing arrow located to the right of the box but the maximum default value is 80.



- Run the animation. It should last for about 15 seconds.

13 – Bringing back moving hidden parts

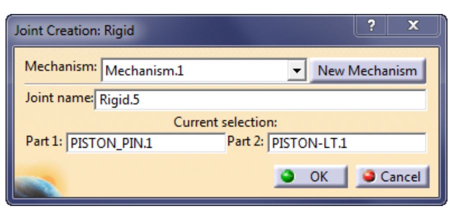
- Keeping the CTRL key down select the following parts in the ***Product Specification Tree***: LOCKING_CONE-LT, BLOCKING_CONE-LT, CRANK_PIN-LT and PISTON_PIN-LT.
- Click the ***Hide/Show*** tool icon to show the selected parts.



14 – Linking part displacements

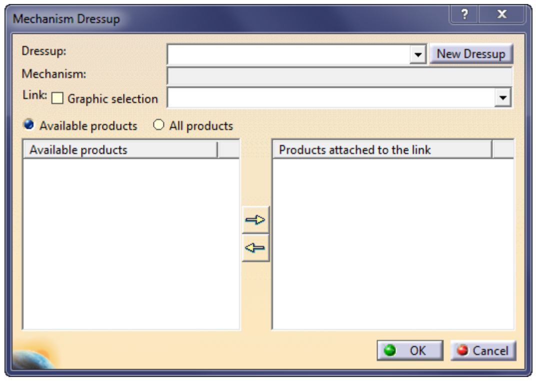
Some of the parts must move the same way in the mechanism. This is the case of the piston pin that moves with the piston and the crank pin, the locking and the blocking cones that move with the crankshaft. Two approaches will be taken to link the parts together: a ***Rigid Joint*** will be used for the piston pin and the piston while a ***Mechanism Dressup*** will be used for the four remaining parts.

- Click the ***Rigid Joint*** tool icon and select the piston pin and the piston either directly in the working area or by clicking their respective item in the ***Product Specification Tree***.

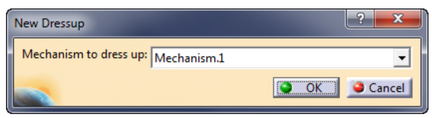


- Click the ***OK*** button to complete the joint creation. A new working confirmation dialog box pops up. Just close it.

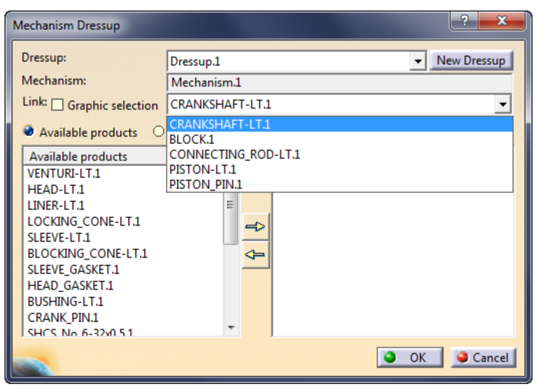
- Click the ***Mechanism Dressup*** too icon . The ***Mechanism Dressup*** dialog box pops up.



- Click the ***New Dressup*** button. Click the ***OK*** button in the ***New Dressup*** dialog box that pops up.

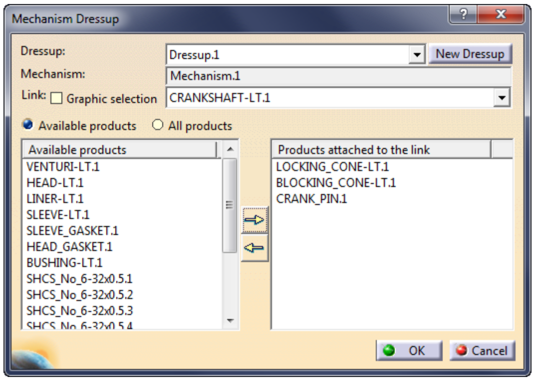


All parts appearing in product are now present in the ***Mechanism Dressup*** dialog box: parts already used for the mechanism are found by clicking the down pointing arrow located to the right of the ***Link*** selection area while all other parts are found in the ***Available product*** area.

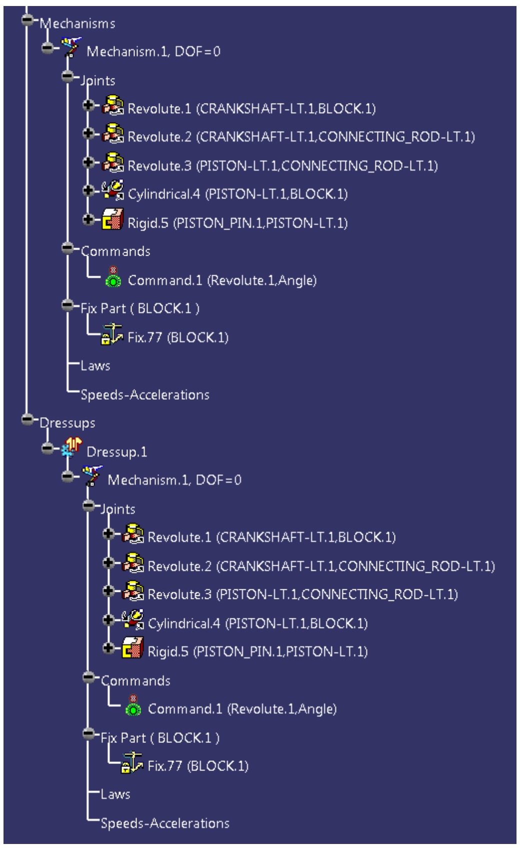


- Select CRANKSHAFT-LT.1 in the ***Link*** selection area.
- Keeping the CTRL key down, select LOCKING_CONE-LT.1, BLOCKING_CONE-LT.1 and CRANK_PIN.1 in the ***Available products*** area.

- Click the right pointing arrow located in the middle of the dialog box to move the selection in the ***Products attached to the link area***.

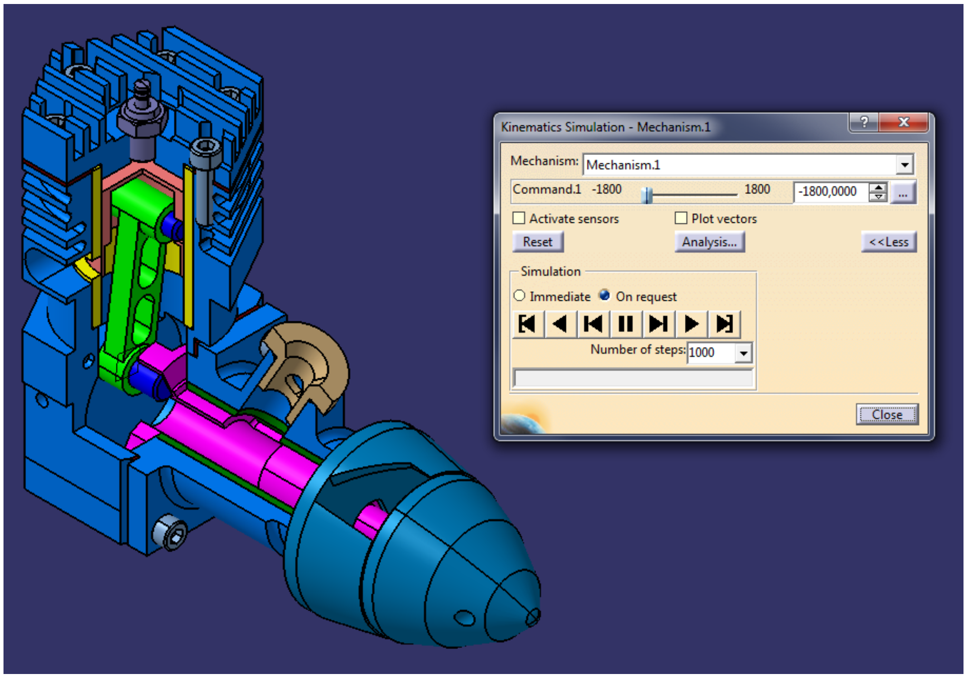


- Click the ***OK*** button to complete the ***Dressup*** creation.
- Expand the ***Dressup.1*** structure in the ***Product Specification Tree***. Note that it duplicates the mechanism structure located just above.



- Run the simulation. All parts are now moving together.

- Bring all static parts from the ***No-Show*** space. Since they are not moving, no constraints really have to be made for any of them.



- Save the product.

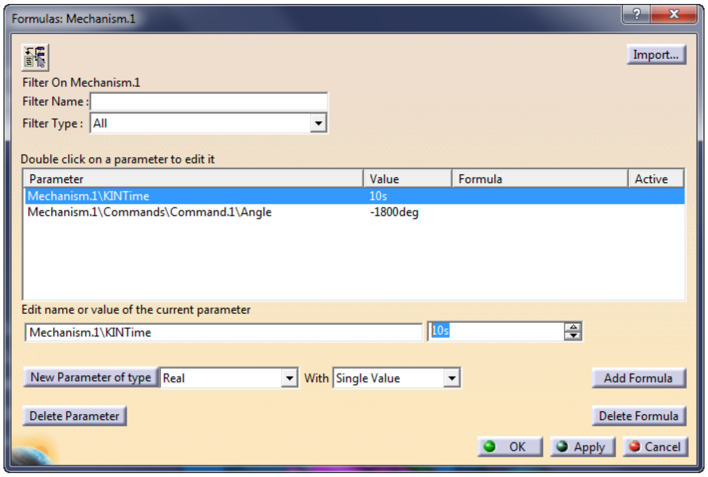
15 – Analyzing part displacement

Since it may be interesting for a designer to analyze the displacement of moving parts in order to optimize the mechanism design, two tools will be used here to achieve this task: the ***Trace*** and the ***Swept Volume*** tools. To use any of them though, it is will be necessary to create a ***Law*** in the mechanism. If the law is absent, error messages will pop up.

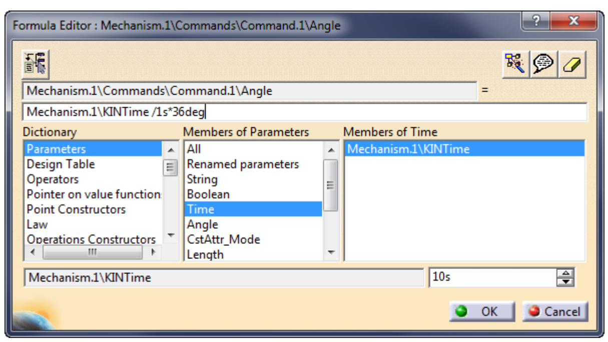
- Select the ***Mechanism.1, DOF=0*** item in the ***Product Specification Tree***.
- Click the ***Formula*** tool icon .

The ***Formulas*** dialog box pops up. Since the ***Mechanism.1*** item was preselected, only two variables appear in the dialog box: ***Kintime***, the time parameter created simultaneously with ***Mechanism.1*** and an angular parameter attached to the ***Revolute Joint*** converted in a ***Command***.

- Edit the ***Kintime*** value to make it *10* seconds.

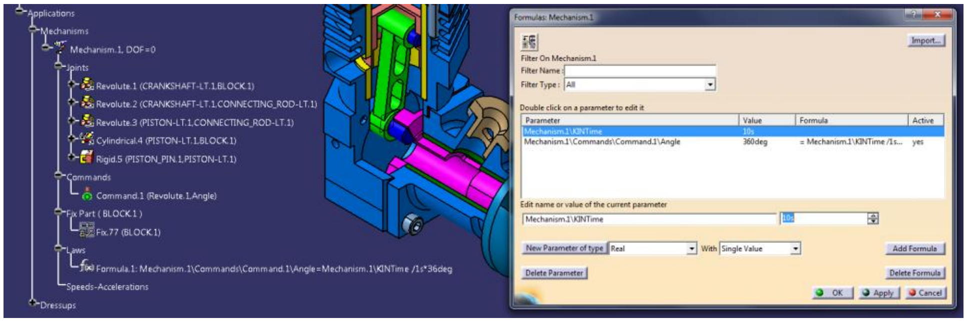


- Double-click the angular parameter or select it and click the ***Add Formula*** button to edit its definition. The ***Formula Editor*** pops up.
- Select the ***Time*** item in the ***Members of Parameters*** area. Only the ***Mechanism.1\Kintime*** parameter will show up in the ***Members of Time*** area.
- Double-click the ***Mechanism.1\Kintime*** item to make it appear in the top edition box.
- Add the following string to the formula: */1s*36deg*



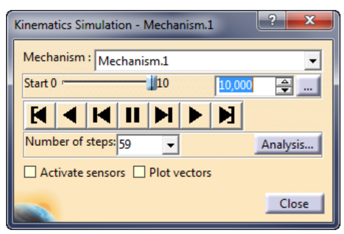
- Click the ***OK*** button.

The law is now created. It appears in the angular parameter description as well as in the mechanism structure in the ***Product Specification Tree***. Since a complete cycle of the engine mechanism is made in 360°, it is not necessary to ask for more rotation. With the ***Kintime*** parameter set to *10* seconds, the law converts the time parameter to force an angular output: 10 s / 1s * 36 deg = 360°

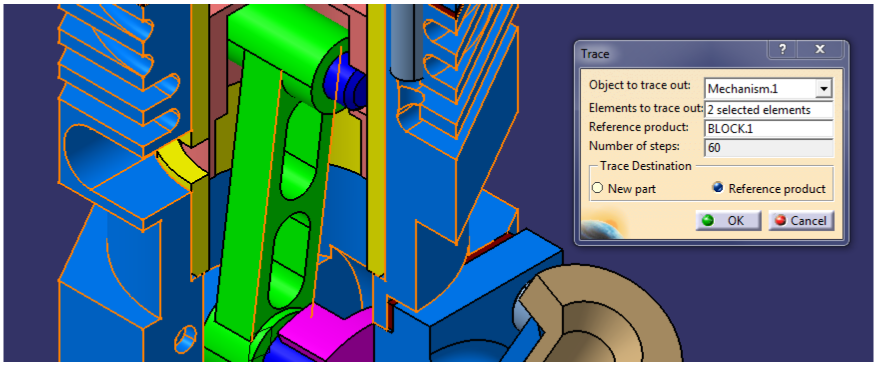


- Click the ***OK*** button to close the ***Formulas*** dialog box.
- Save the product.

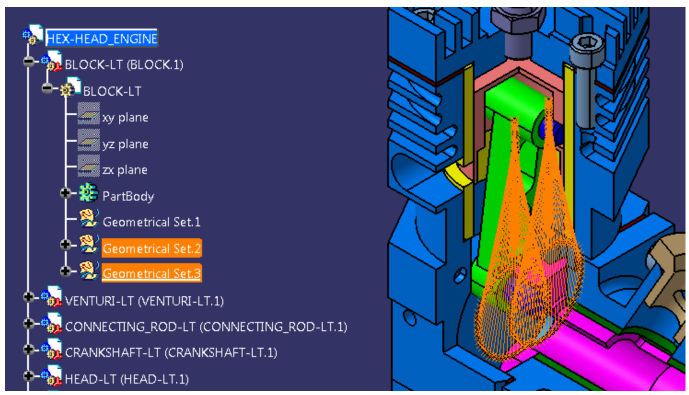
The first analysis to be made will be a trace of the two connecting rod edges. The ***Trace*** tool will always generate one more item than the number of steps used when running the animation. In this case, if a trace of the edges is required every 6°, 60 outputs will be necessary. To get them, it is then necessary to run the animation one more time, asking for *59* steps, prior to generate the trace. Note that the cursor range is now set by the ***Kintime*** value.



- Click the ***Trace*** tool icon .
- Select the connecting rod's two front edges. The two selected elements appear in the ***Elements to trace out*** area.
- To avoid the generation of the outputs in a new part that would need to be added to the product to be analyzed, click in the ***Reference Product*** edition area to activate it and then select the block to force the generation of the wireframe elements in its file.



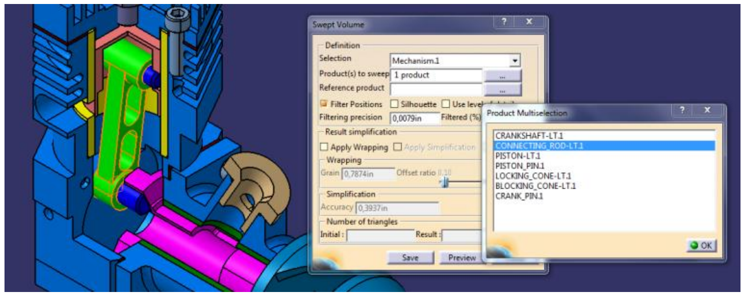
- Click the ***OK*** button to launch the trace generation. The result should appear within seconds. The generated elements are found in two separate block's ***Geometrical Sets***.



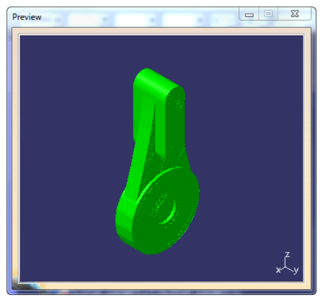
- Hide the two geometrical sets.
- Use ***File>Save All*** to save the product and the block file that has been modified.

A generation of the moving connecting rod's working envelope will now be generated. This envelope can be saved under a ***.CGR*** format which can be inserted in a product but cannot be opened as a single file. If some geometry have to be extracted from a ***.CGR*** file, it is necessary to convert it into ***.Model*** and ***.CATPart*** files prior to be able to work in it.

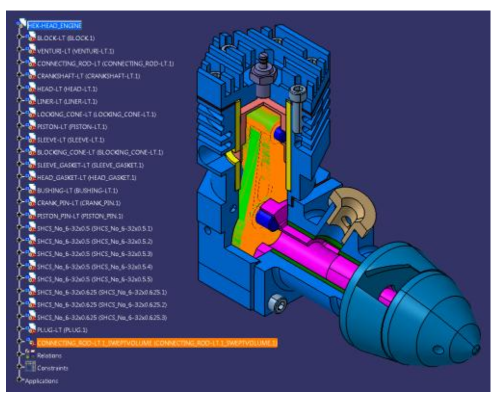
- Click the ***Swept Volume*** tool icon . When launching the tool, all moving parts are selected and would create a complex swept envelope.
- Click on the ellipsis (…) button next to the ***Product(s) to sweep*** area.
- In the ***Product Multiselection*** dialog box, select the CONNECTING_ROD-LT.1 element only.



- Click the ***OK*** button to close the dialog box.
- Click the ***Preview*** button to see the proposed swept envelope.



- Close the ***Preview*** window and click the save button to create a ***.CGR*** file of the generated envelope.
- Use ***Insert>Existing Component*** to add the ***.CGR*** file to the product.



- Save the product.

Thanks to Alice Michaud for revising this text.