Aalto University
School of Engineering

EXERCISE 5.1 – CRANK MECHANISM ANALYSIS

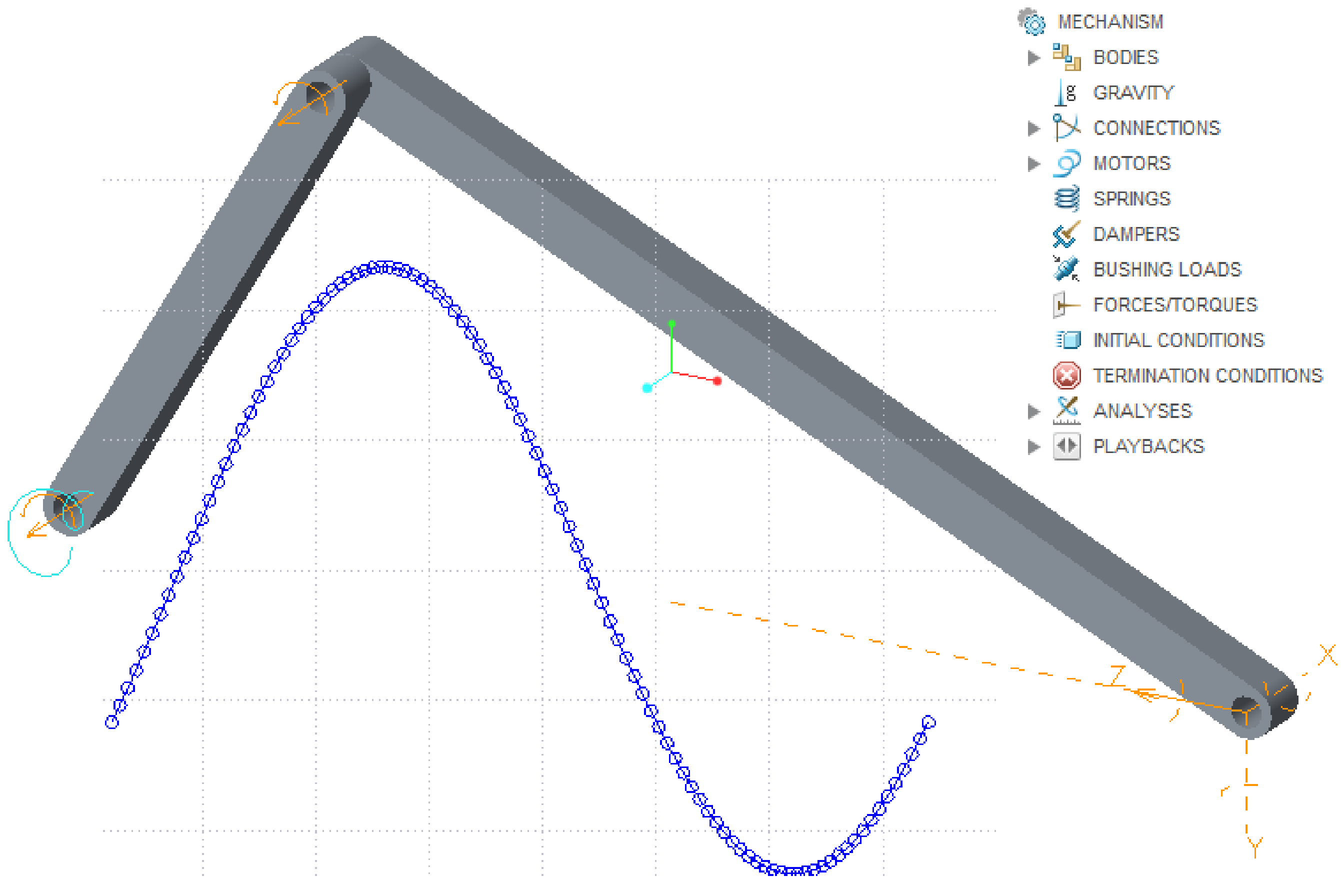


Figure 1: Analyzed mechanism and a graph.

Learning Targets

In this exercise you will learn:

- ✓ to create family table parts
- ✓ to use copy - paste special procedure
- ✓ to create multi-body assemblies
- ✓ to perform mechanism simulation
- ✓ to plot results.

In this exercise, a crank mechanism is created and analyzed. First a simple geometry is created and then Multi-Body Simulation (MBS) tools are used to solve force balance cases and to plot results.

The used program is PTC Creo 6.0.2.0, but the tools presented here work with older versions.

Getting Started

Start **Creo 6.0 & Creolib** from Windows start menu.

The plan is to create a crank mechanism model (Figure 2) and analyze its behavioral.

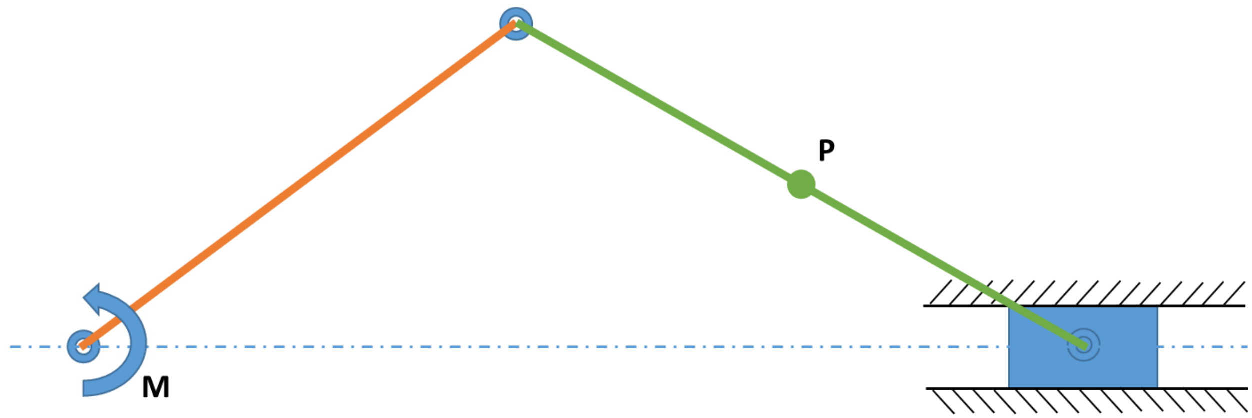


Figure 2: A crank mechanism.

Parts

First we crate orange and green parts in Figure 2 using *Family Tables*. By using this tool, we need to create one parent model, where all the features and dimensions exist, and then we can create several models based on it.

Parent model

Select **New** (, from *Data* group) and check that **Part** is selected as *Type* and **Solid** as *Sub-type*. Give this part a name, for ex. **link** (Figure 3). **OK**.

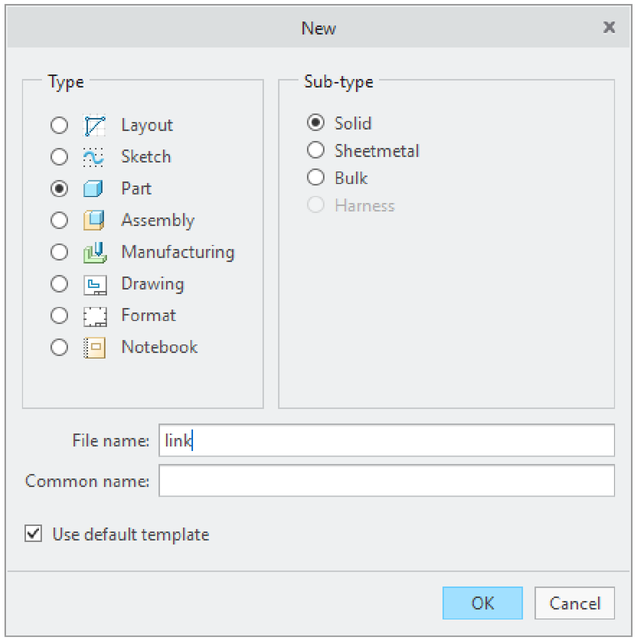


Figure 3: Creating a new part called link.

Extrude

Select **Extrude** (, from *Shapes* group) and select **FRONT** plane from the graphical area (or from the *Model Tree* in the left). A *Sketch* mode appears. Using **Circle** (, from *Sketching* group), create two equal-sized circles, starting from the center of the sketch (dashed lines). When creating the second one, Creo will offer = (*Equal Radii*) constraint when the size of the second circle is about the same as the first one's (Figure 4).

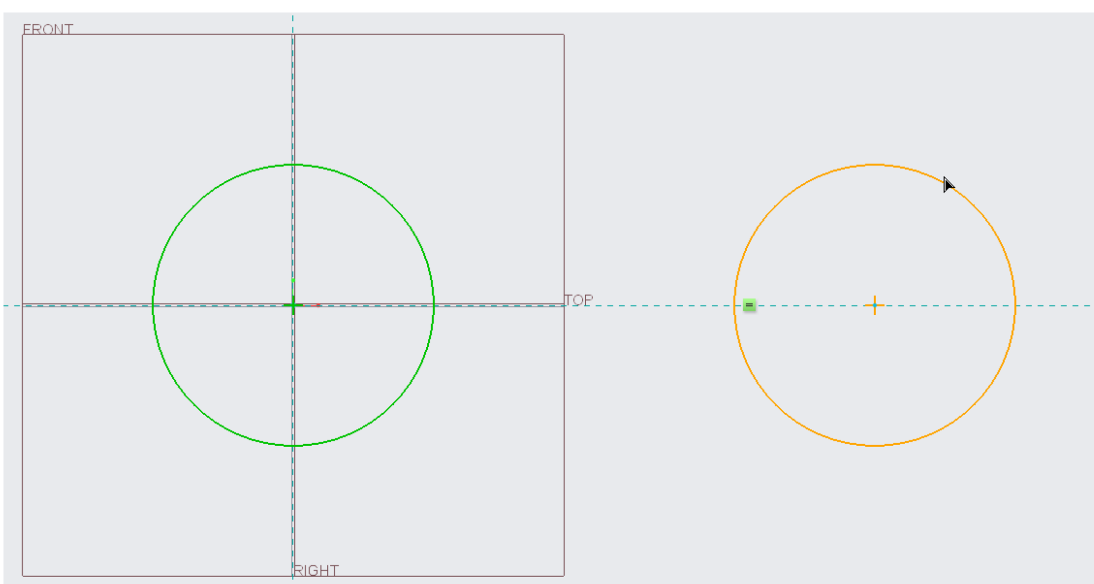


Figure 4: Creating second circle to the right (yellow), program offering = (Equal Radii).

Then, using **Line** (, from *Sketching* group) create horizontal line starting from the intersection of the leftmost circle and Y-axis (Figure 5) and attach the line to the rightmost circle using (*Tangent Entities*) as seen in Figure 5. Accept with **MMB**.

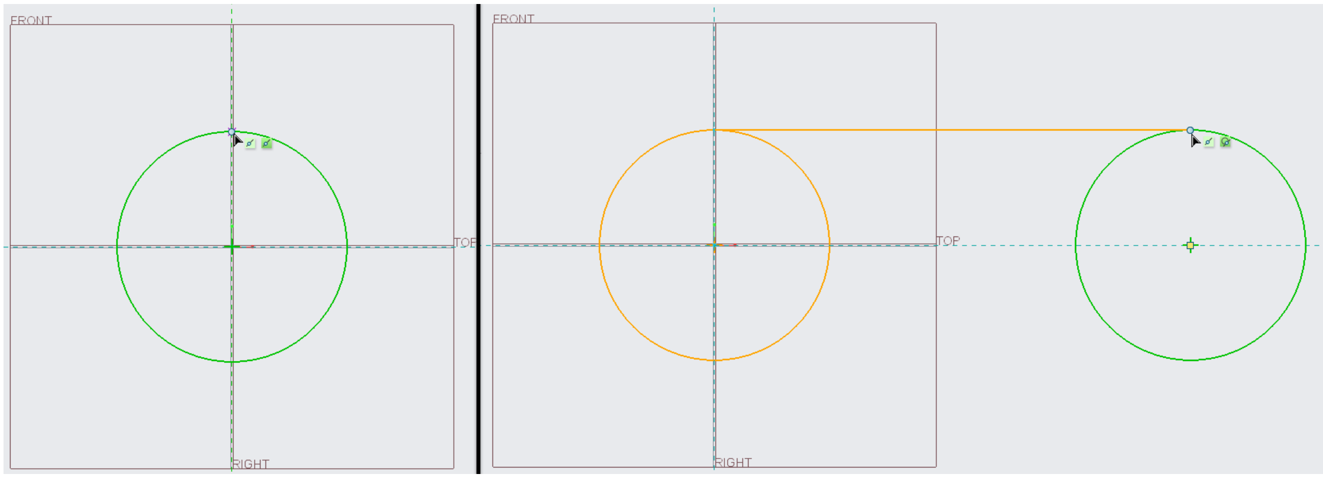


Figure 5: On left: selecting starting point. On right: selecting end point.

Using the same method, create another line on the bottom. You can close the tool by pressing **MMB** second time (the first time ends the current line, the second closes the tool).

Using **Delete Segment** (, from *Editing* group), remove the unnecessary arcs from the sketch by holding **LMB** and moving mouse cursor over removable lines (Figure 6). **MMB** to close the tool.

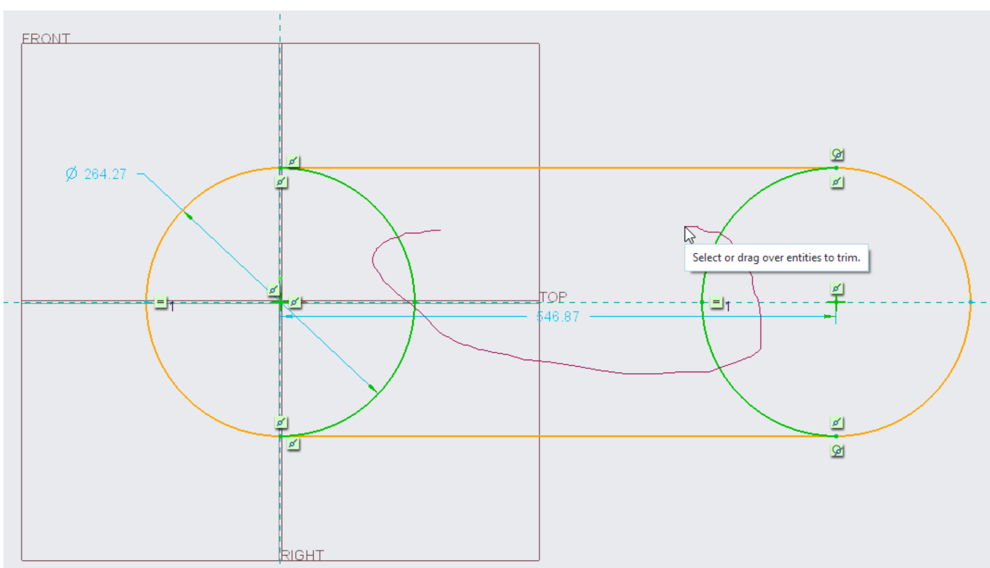


Figure 6: Removing arcs (highlighted in green), the tool path highlighted in red.

Next, using **Dimension** (, from *Dimension* group), create a dimension by first selecting the upper horizontal line, then lower horizontal line and then pressing **MMB** between them (Figure 7). Close the tool with **MMB**.

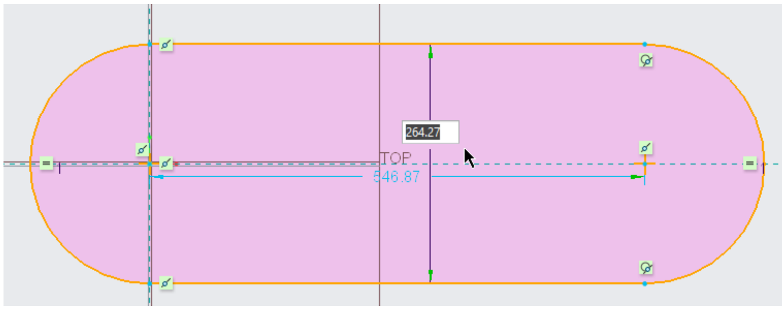


Figure 7: Both horizontal lines selected and MMB pressed.

Give **10** as a height and **100** as the width (double-click the grey dimension to edit it) as seen in Figure 8. You can move the location of the dimension by selecting it and moving around.

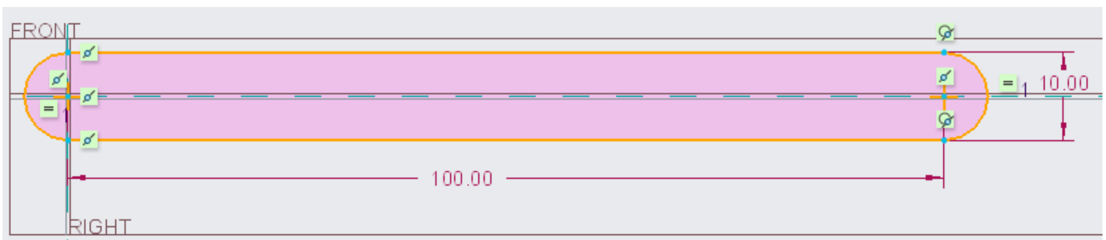


Figure 8: Dimensions created and updated.

When ready, accept the sketch by selecting **OK** (, from *Close* group) or holding **RMB** and selecting .

Give **10** as the length of the extrude and accept the feature (or **MMB**). Select **Extrude 1** from the *Model Tree*, **RMB** and select **Rename**. Give **base** as a name (all user defined names will be in UPPERCASE). It is a good practice to rename features and give them meaningful names. It helps a lot when models are big and some changes are needed.

The First Hole

Next we create two holes. From *Quick Access* toolbar (Figure 9), uncheck **Plane Display** () and check **Axis Display** (). Now we can see axes in the BASE feature.

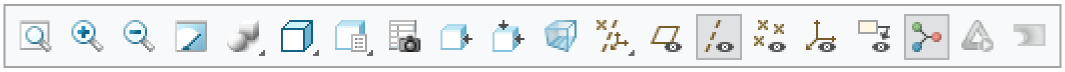


Figure 9: Quick Access toolbar.

Select **Hole** (, from *Engineering* group). Select the starting surface of the hole (green in Figure 10), hold **Ctrl** and select axis seen in Figure 10.

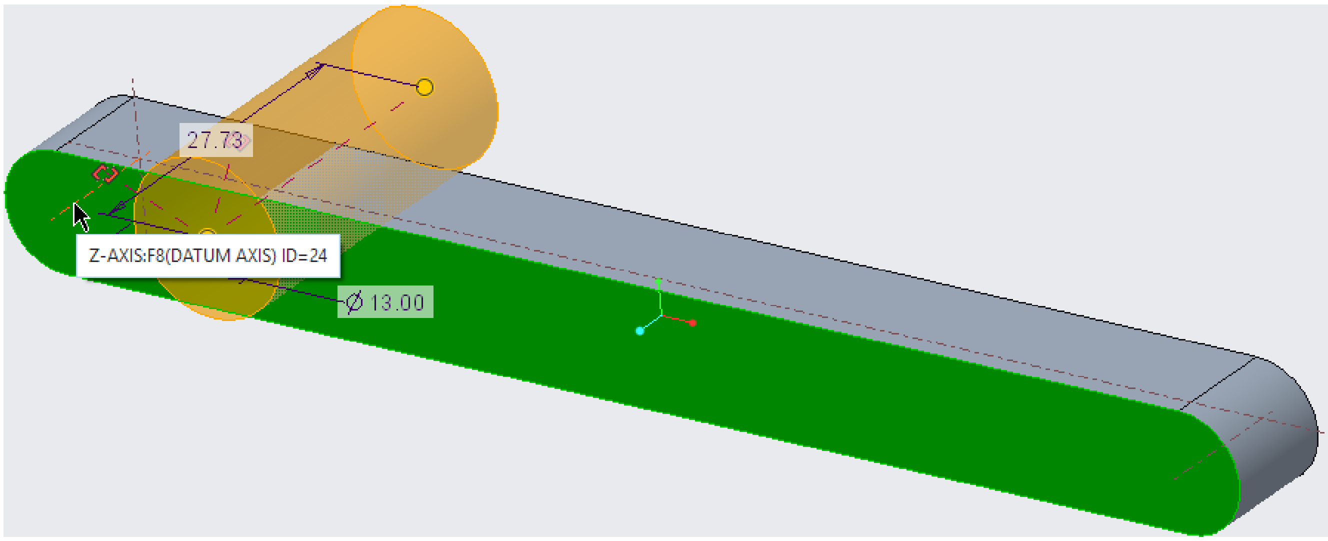


Figure 10: Starting surface selected, selecting axis while holding Ctrl.

Give **6** as a diameter and select **Thru All** () as a depth from the dashboard (Figure 11). or **MMB** to accept the feature.

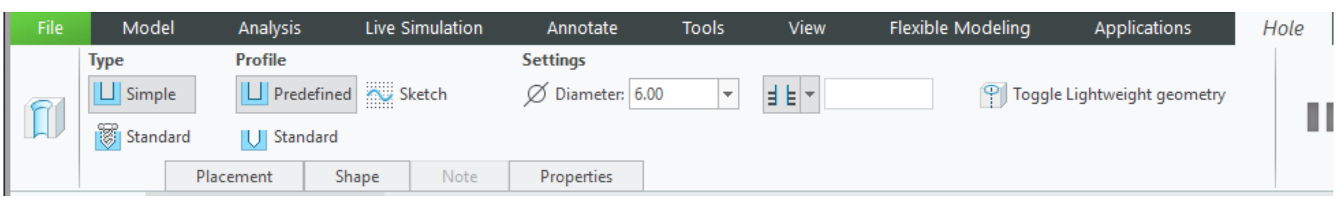


Figure 11: Dashboard and selecting Thru All from the drop-down list.

The second hole

We can use the same method as previously to create a second hole. But we can also use copy-paste to create another hole. Select **Hole 1** from the model tree and press **Ctrl + C**. Then, select **Paste Special** () from *Operations* group (Figure 12).

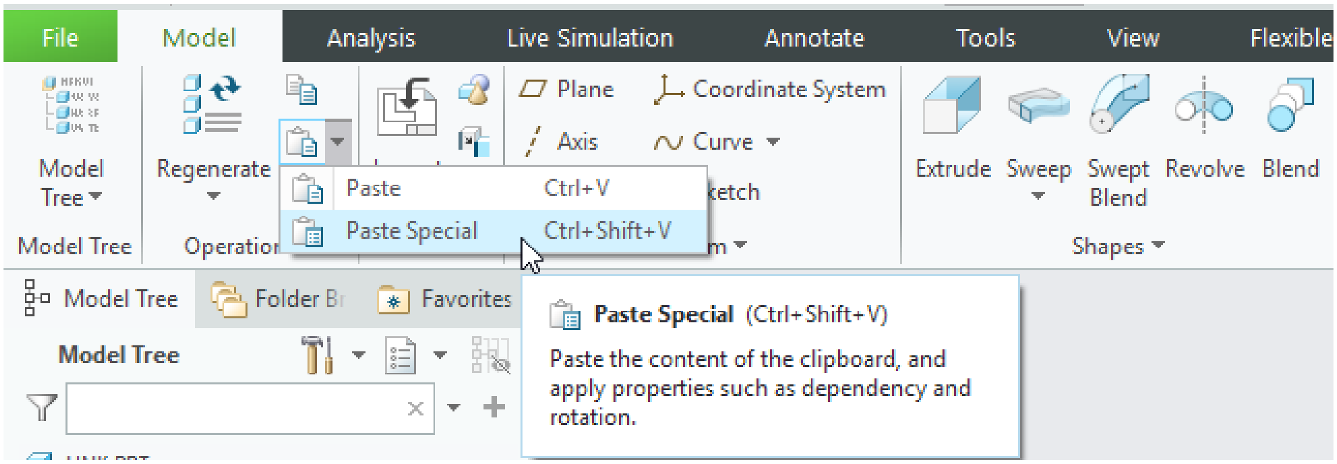


Figure 12: Selecting Paste Special.

Check the option **Advanced reference configuration** and press **OK**. A new window appears. This window lists all references that hole feature has (starting surface and placement axis). Select the AXIS from the list and select <u>the other end's</u> axis from the model. Press ✓ to accept the new copied feature.

Datum points

Next we create several datum points for working as measurement points for mechanical simulation. Select **Point** (, from *Datum* group). Ensure that **Plane Display** () is off and **Axis Display** () is on in the *Quick Access* toolbar. Then, select one of the axes, hold **Ctrl** and select one of the main surfaces in the part (Figure 13). This creates a point in the cross-section of axis and plane.

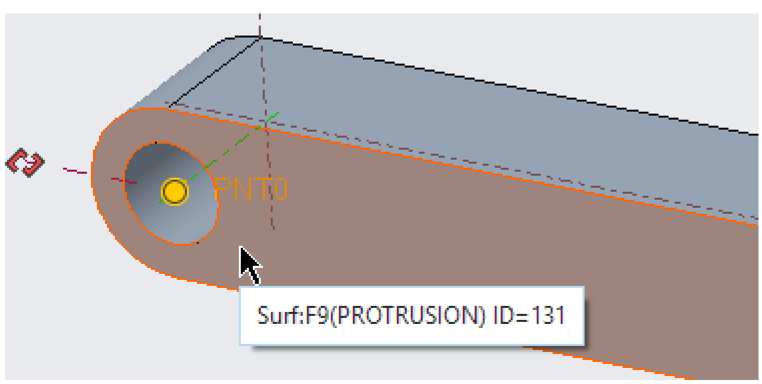


Figure 13: Axis selected (light green), selecting surface (red) while holding Ctrl.

Using the same method, create three other points as seen in Figure 14. **OK** to accept the points and to close the tool (as you can notice, there is only one *Datum Point* feature in the *Model Tree*).

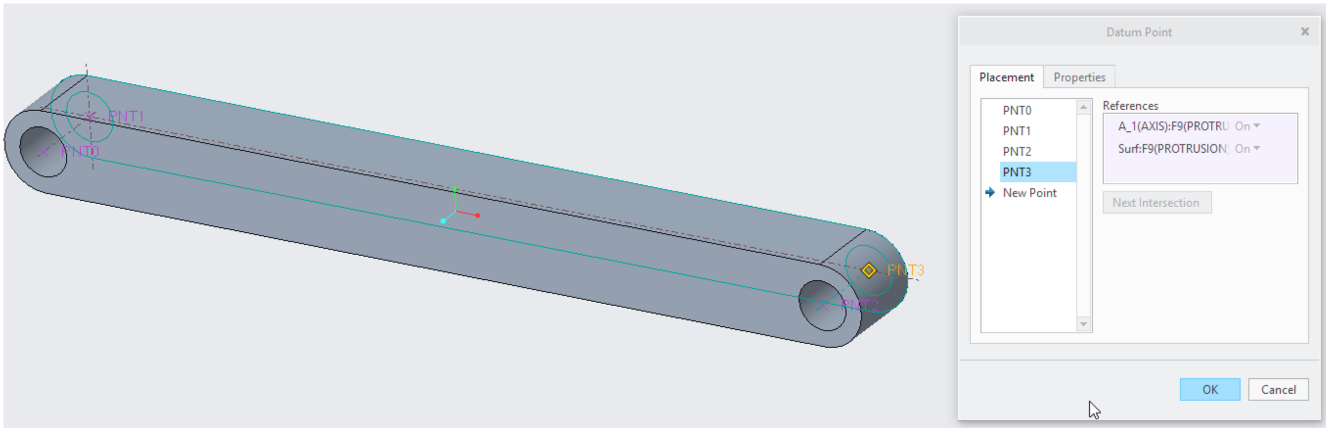


Figure 14: Four points created (cross-sections of the axis and planes).

Sketched points

For measurement reasons, we need to have a point in the middle of the link part. Select **Sketch** (, from *Datum* group). Select a surface as seen in Figure 15 and press **Sketch**.

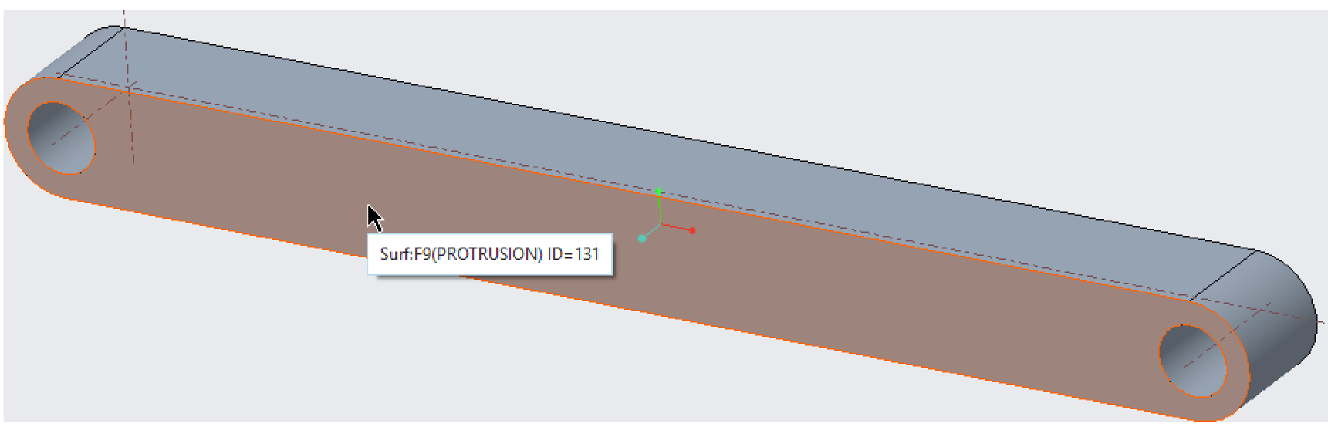


Figure 15: Surface for sketching.

Select **References** (hold **RMB** → **References** or from *Setup* group). Remove all references from the list (select and **Delete**). Them, select the both axes in the model as references (Figure 16) and select **Close**.

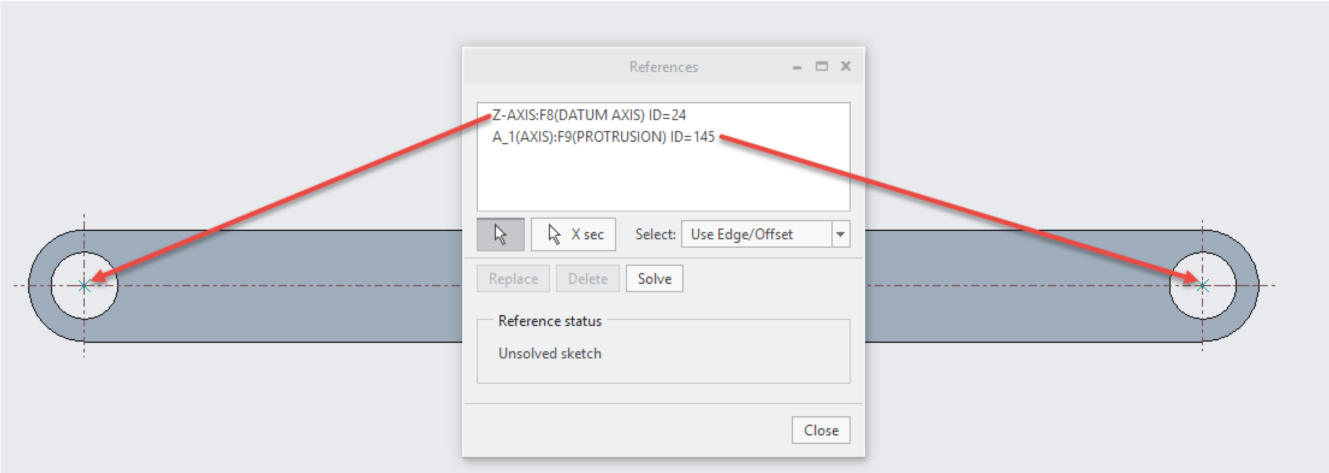


Figure 16: References for a sketch.

Then, using **Line** (, *Sketching* group), create a horizontal line between previously created references (notice, that Creo doesn't snap to datum geometry that is not selected as reference). **MMB** twice to close the tool (first to end the loop, second to close the tool). Select previously created line (if not already highlighted in green), hold **RMB** and select **Construction** (). Now this line is for construction purposes only and doesn't affect sketch. Using **Point** () from *Datum* group, create a point in the middle of the construction line (symbol) as seen in Figure 17. Accept the sketch (**RMB→OK** or ✔ from *Close* group).

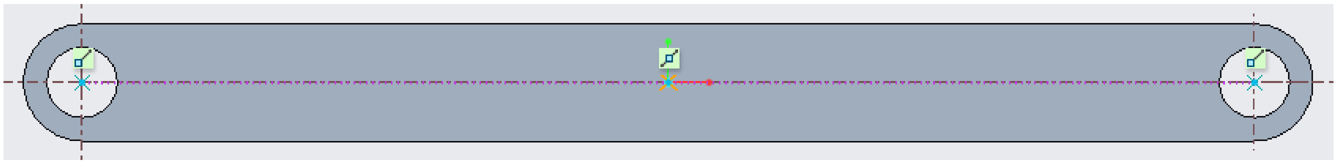


Figure 17: Datum point created in the middle () and construction line in the purple color.

Set the **Point Display** () on from the *Quick Access* toolbar to see all five points in the model. Using **Copy → Paste Special → Advance Reference Configuration** (as with hole), create another point on the other side of the link part. Part geometry is ready, time to save the model (**Ctrl + S**).

Family table

We have one general part created. Using *Family Table*, we can create variations of it (called instances), that may have different dimension values (or features). Select **Family Table** () from *Model Intent* group (Figure 18).

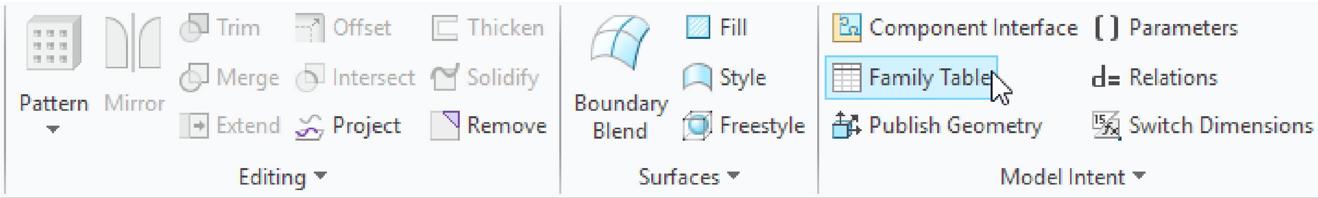


Figure 18: Selecting Family Table.

Select **Insert a New instance** () four times to create four instances (variations) of the current part. Select **Add/delete the table columns** () to add dimensions/features that change. This opens a new window, select **BASE** feature and the **100** dimension from it. A d# is added in the items list (Creo numbers all its dimensions and the naming is related to creation order, so your dimension numbering may not be the same as in this exercise). Click **OK**. Update the table as seen in Figure 19. Accept the table (**OK**) and save the model.

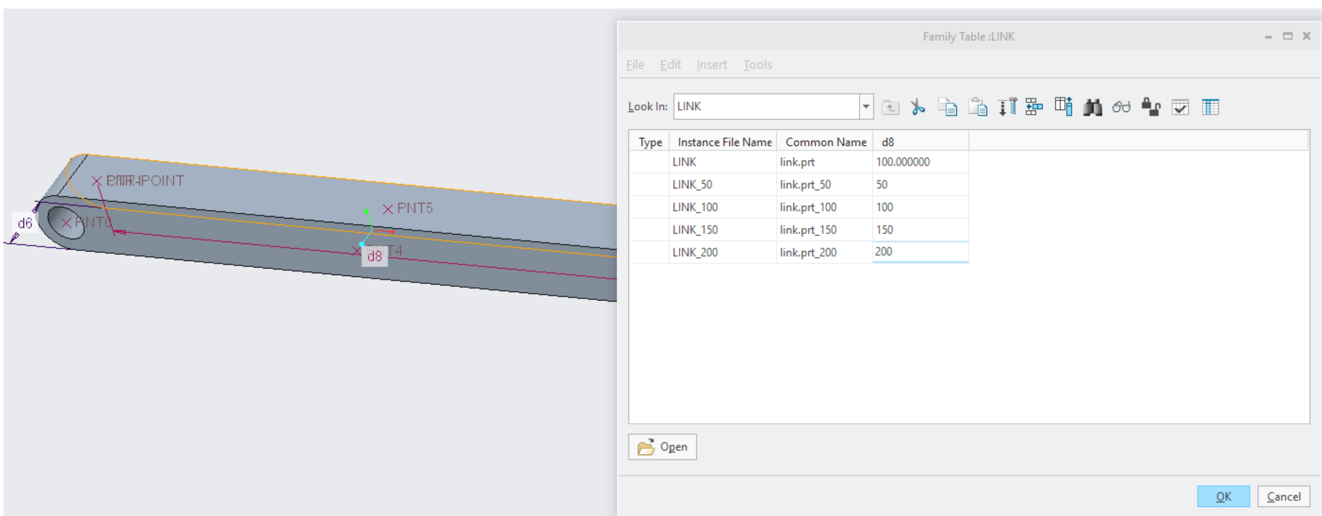


Figure 19: Family table and four instances created.

Assembly

We have different parts created; next task is to create mechanism assembly. Select **New** () and **Assembly** as a *Type*. Name it for ex. crank_mechanism (Figure 20). **OK**, this opens assembly mode.

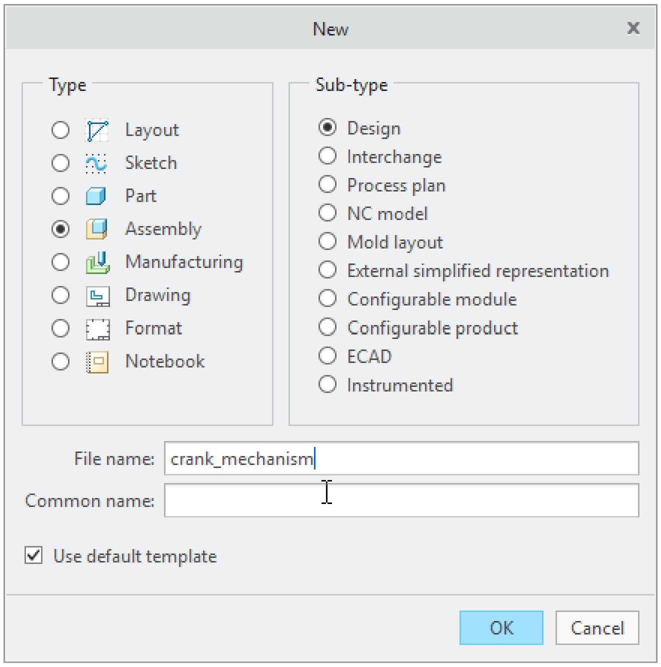


Figure 20: Creating a new assembly.

Datum axes

Notice that assembly has besides three basic datum planes also three axes (Figure 21).

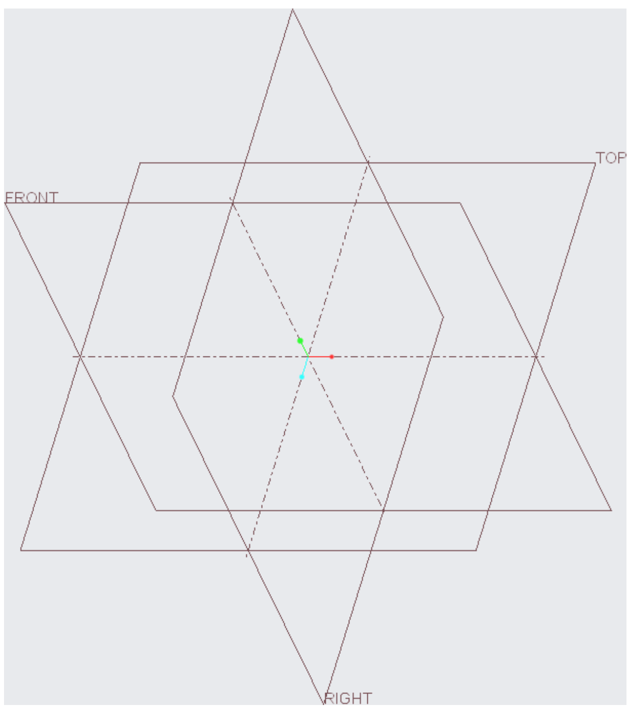


Figure 21: Three axes in the assembly (X, Y and Z direction).

The orange part

Select **Assemble** (, from *Component* group) to add a new component to the assembly. Select previously created **link.prt** and select **Open**. A *Select Instance* window will open, because the link part is a general part of a family table. Choose **LINK_100** from the list and select **Open**. From the dashboard, select **Pin**. Pin is a predefined joint that has one rotational Degree of Freedom (DOF). These joints will be used in the mechanism simulation mode for different purposes.

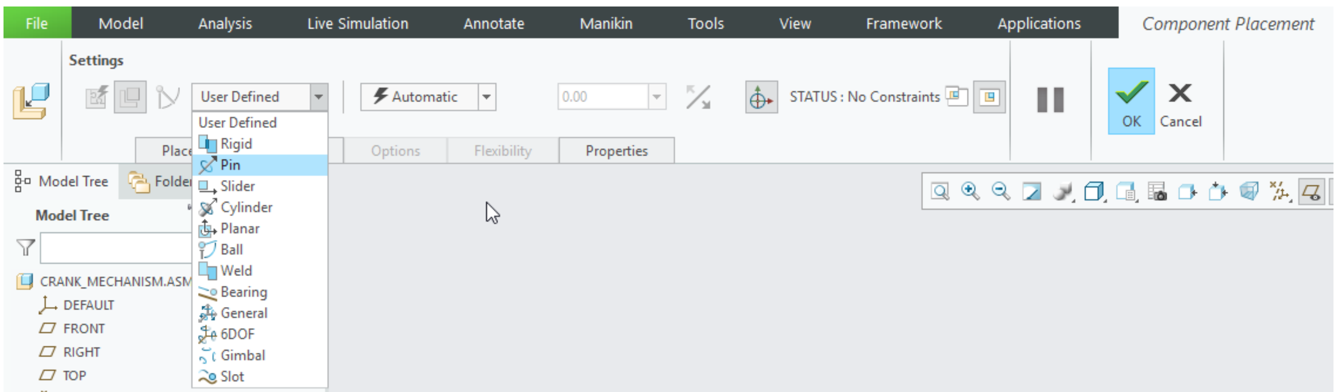


Figure 22: Selecting Pin as attachment type.

Expand **Placement** tab to see what kind of constraint Pin needs:

- Axis alignment; rotational axis of the joint.
- Translation; location of the part in the rotational axis.
- Rotation 1; Not visible yet, rotation angle around rotational axis.

As *Axis alignment*, select from link.prt the axis of the hole, and from the assembly, select the Y-AXIS (you can hide 3D dragger by selecting). As *Translation*, select FRONT from link.prt and FRONT from the assembly. The part is in golden color to show that all needed constraints are defined. Select *Rotation 1*, select TOP from link.prt and TOP from the assembly. Give -**60** as *Current Position* and press » to add this as *Regen Value*. Check the option **Enable regeneration value** (when model regenerates, the part will be in this defined angle). When ready (Figure 23), accept (MMB or)

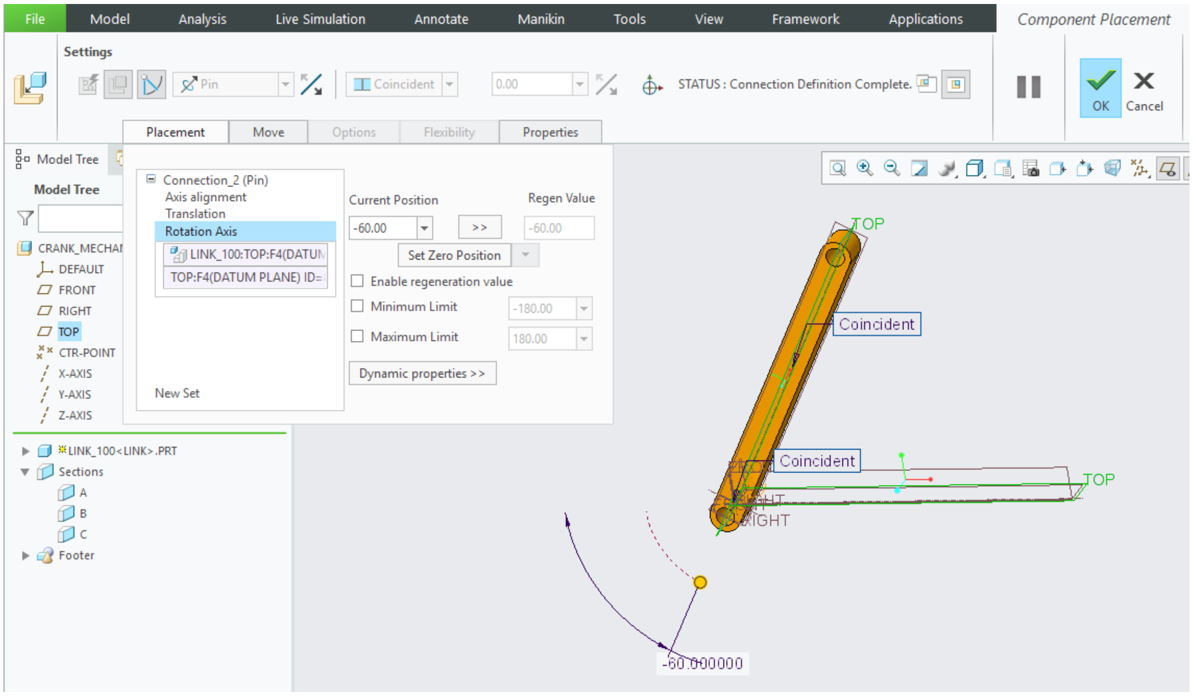


Figure 23: First part added and rotational angle defined.

Green part

Next we add the second part to the assembly (green in Figure 2). Select **Assemble** (, from *Component* group) and select link.prt → LINK_200. Select **Pin** as joint and assemble as seen in Figure 24 (you can hide 3D dragger by selecting).

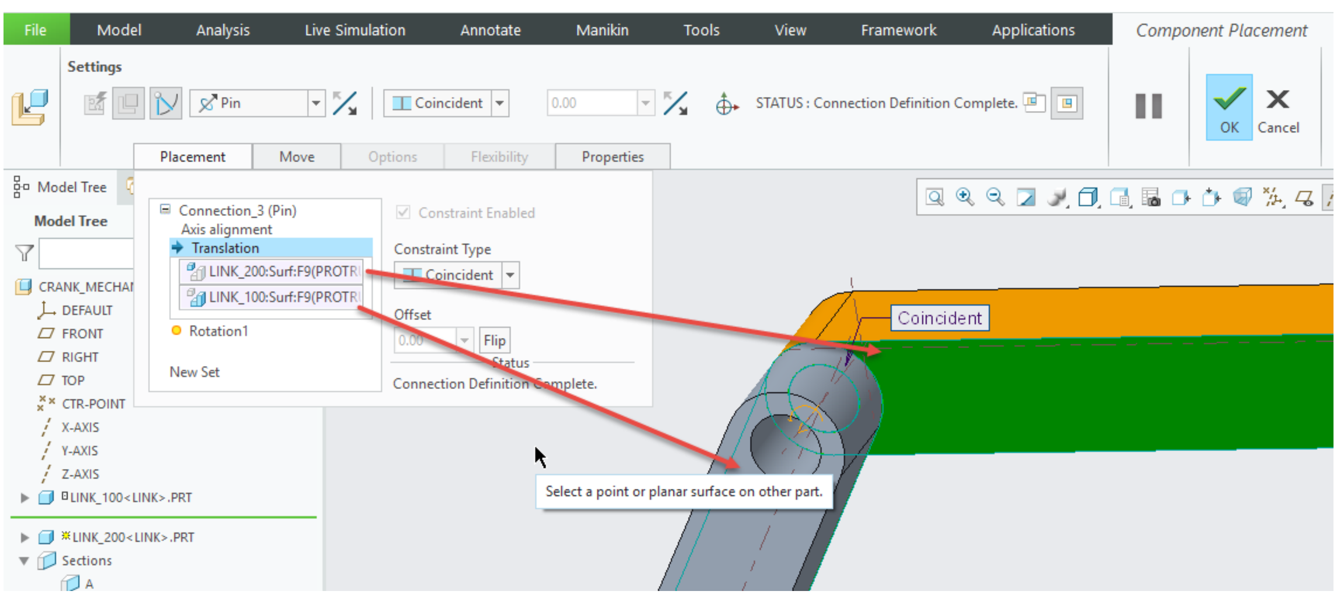


Figure 24: Hole axes constrained and two surfaces added together (arrows).

From Placement tab, select **New Set** to define another joint. Select **General** as a type. From the assembly, select Z_AXIS and from the part, a point (Figure 25).

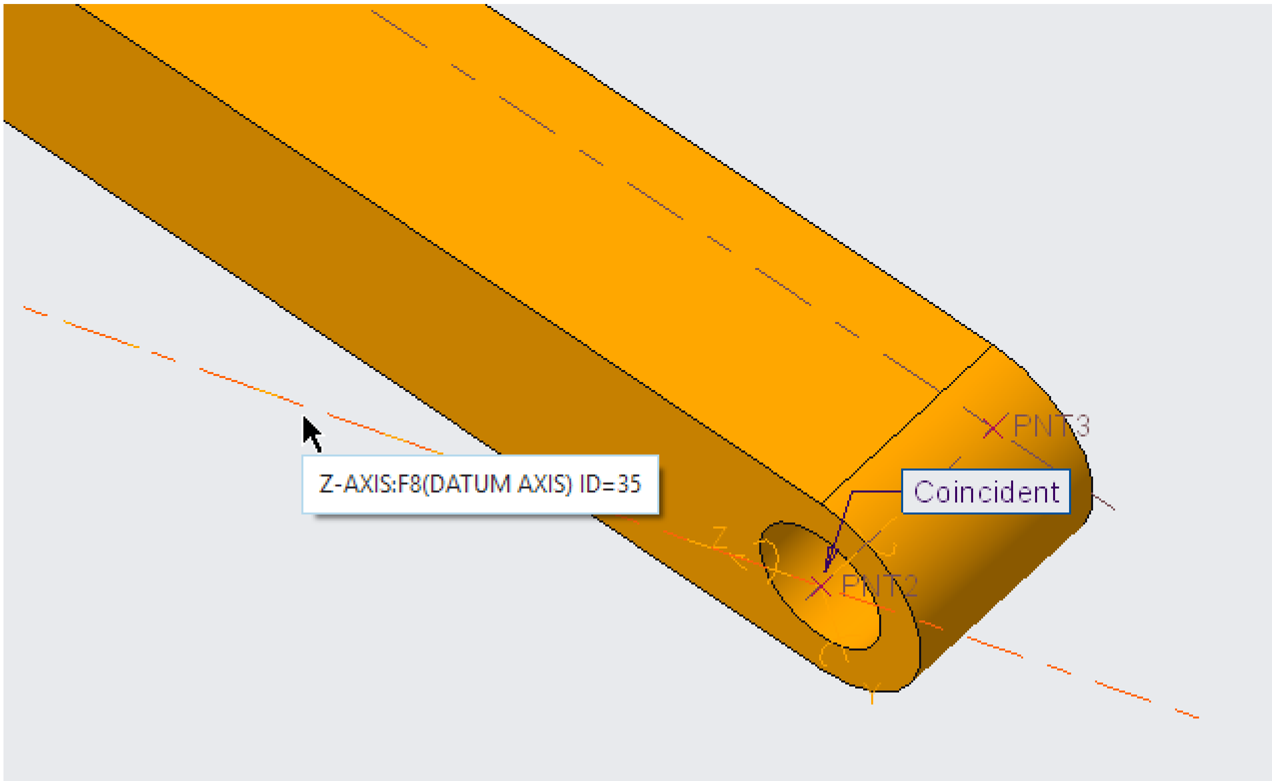


Figure 25: An axis (Z_AXIS) and a point (PNT2 in this case) added together.

Now all joints are defined, accept (**MMB**).

You can use **Drag Components** (, from *Component* group) to select an entity and drag mechanism around to see how it works. When ready, press **Ctrl + G** to regenerate model to the default position. Save the model.

Multi-body Simulation

Select **Applications** tab and **Mechanism** (, from *Motion* group) to start using mechanism analyses mode. Here you can see which kind of joint your model has (Figure 26). In our case, two rotational joints (an arrow and arc over it) and one translation general joint (arrow towards Z). Click **Highlight Bodies** (, *Bodies* group) to see different moving bodies in the assembly (there should be two bodies).

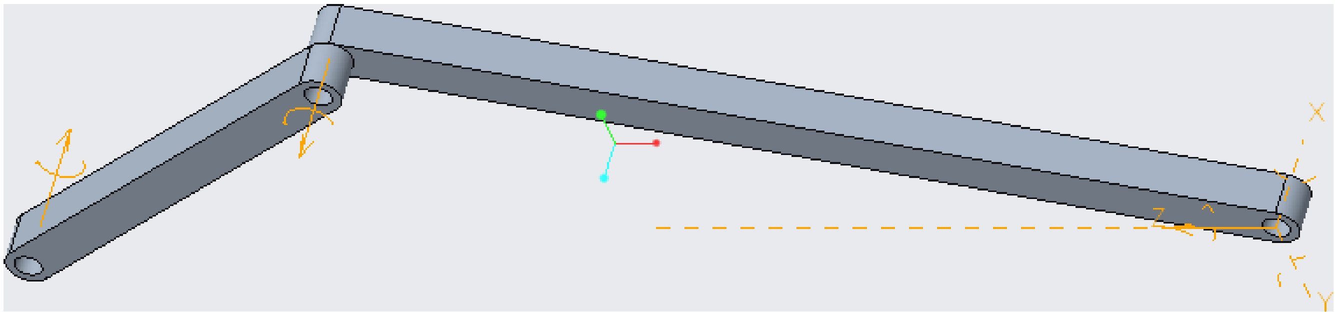


Figure 26: A crank mechanism. Joints highlighted in golden.

Needed torque

First we calculate the needed torque to hold the mechanism still in 60 deg angle.

(Manually calculated torque is 58.84 Nmm).

The simulation process includes:

- Define a servo motor that stays in the right angle.
- Define measurement.
- Define dynamic analysis including gravity.
- Run simulation.
- Plot result.

Servo motor

Select **Servo Motors** (, from *Insert* group) and select the rotational DOF of the first part (over Y_AXIS). Select Profile tab and select **Position** as *Specification* and -**60** as a value. Name it as static_motor (Figure 27).**OK** to accept.

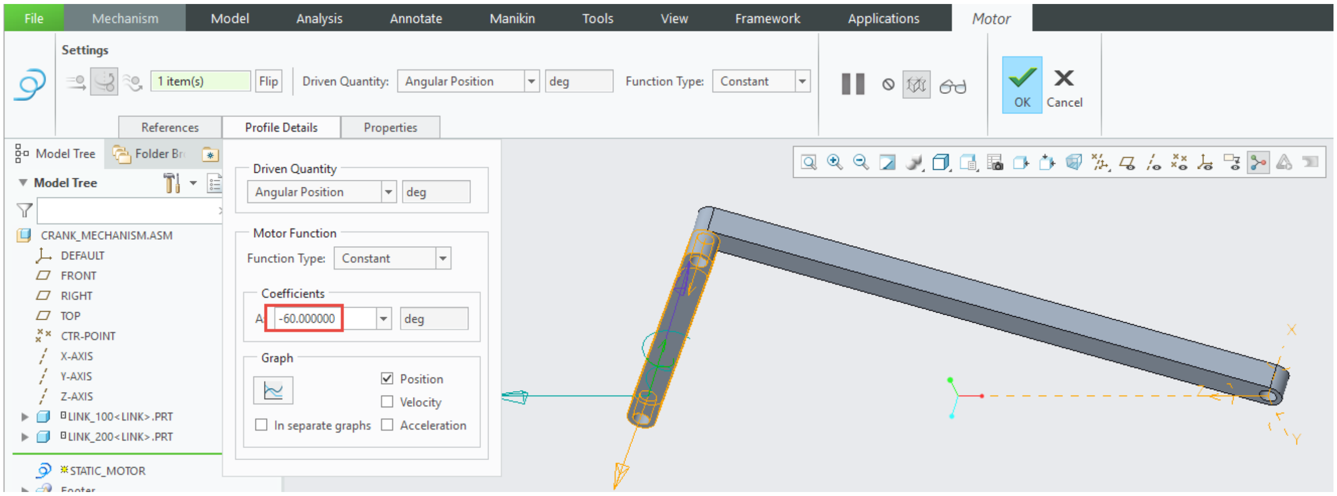


Figure 27: Servo motor created on the joint (purple arrow).

Measurement

Select **Measures** (, from *Analysis* group) and select **New** (). Give **torque** as a name and select **Connection reaction** as a *Type*. As *Connection*, select the connection where servo motor is defined and select **Axial moment** as *Component* (Figure 28). When ready, **OK** and **Close**.

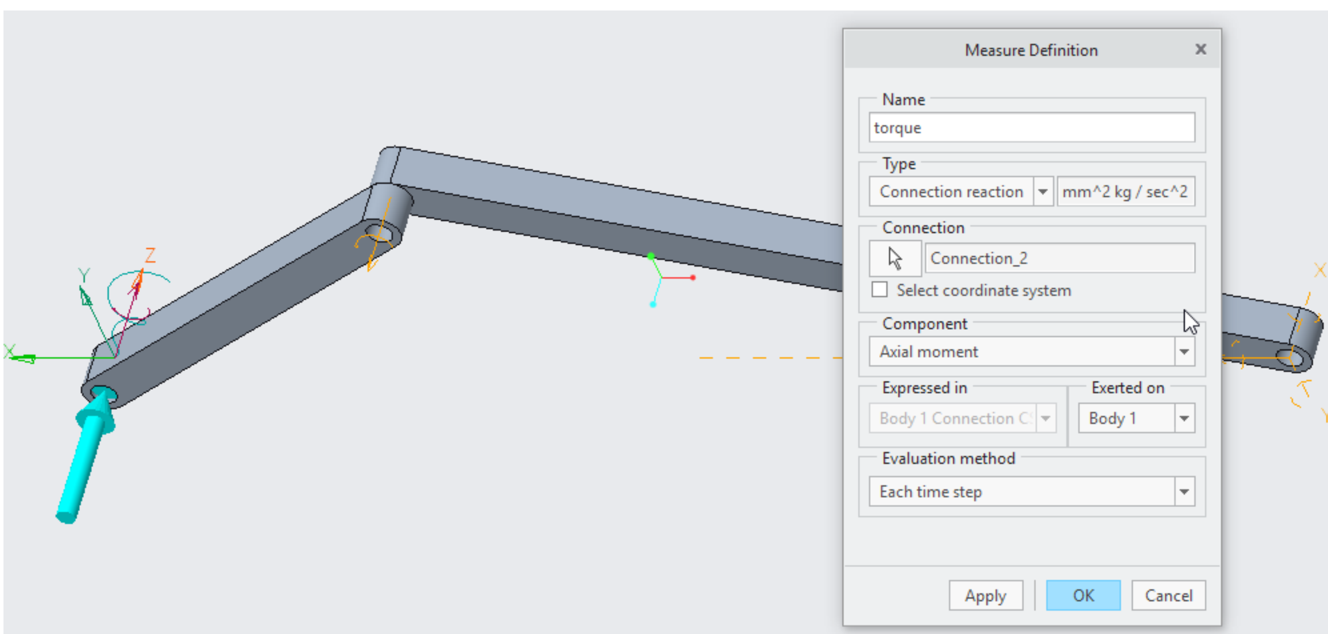


Figure 28: Torque measurement. Thick arrow points the measurement axis.

Dynamic analysis

Select **Mechanism Analysis** (, from Analysis group). *Name* it as **M_torque** and select **Dynamic** as a *Type*. Change *Duration* to **1**. Select **Motors** tab to check that *static_motor* is listed. Select **External loads** tab and check option **Enable gravity**. Select **OK** to accept the simulation.

Running a simulation

In the bottom-left you can see *Mechanism Tree*, where all mechanism related features (motors, connections, loads etc.) are listed. Select **BODIES** and expand it to see, that this model has three bodies (one ground and two bodies). Select **ANALYSES** and expand it. Select **M_torque**, hold **RMB** and select **Run** ().

Plot result

Select **Measures** (, from *Analysis* group). Select **torque** from the *Measures* list, **M_torque** from the *Result set* list. Next to the torque you can see the result (~-58412). If you press **Graph**, you can see the used unit system (mm^{2}kg/sec^2) .

Full cycle

We want to calculate the needed torque during one cycle. Select **Measures** () and create a **New** () measure named **deg**. *Type* can be **Position**. Select the joint in the Y_AXIS, **OK**. Select **deg** and previous result set (M_torque) to see, that the value is -60. **Close** to close the tool.

Create a new **Servo Motor** () on the Y_AXIS using **full_cycle** as a name, **Angular Velocity** as a *Specification* and **36** as a value (full cycle in 10 s).

Create a new **Mechanism Analysis** () using full_cycle as name, **Dynamic** as a *Type* and **10** as a *Duration*. In **Motors**, unselect **static_motor** by selecting it and selecting **Delete highlighted rows** (). In **External loads**, check **Enable gravity**. **OK**. Run the created simulation.

In **Measures** (), select *Graph type* as **measure vs. measure** and select **deg** as *Measure for X-axis*. Then select **torque** as *Measures* and **full_cycle** as *Result set* (Figure 29). Click

Graph () to see the needed torque during one cycle (Figure 29). Notice that the deg starts with -60. This is because the defined starting angle for the mechanism. It can be changed by changing the regeneration value of the joint to 0, if needed. Notice also, that this graph can be exported to MS Excel (File → Export Excel).

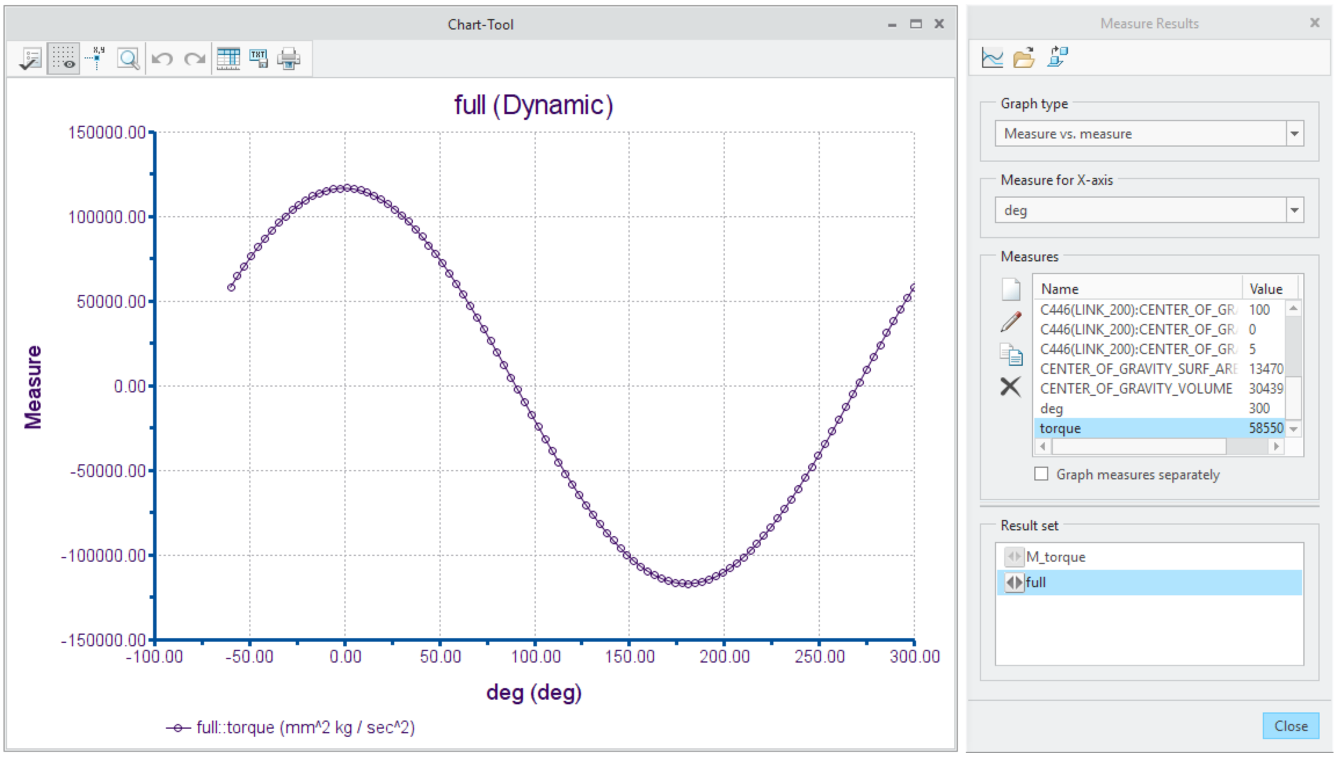


Figure 29: On left: setting for graph. On right: graphed torque.

Location of a point P

To plot the location of point P (as seen in Figure 2), use **measures** and make two position measures to one of the points in the middle of link part. Then graph them using one as X and other as Y-axis (*Component* field X and Y). The result should be like in Figure 30. In the *Ghart-Tool*, you can change the texts in the graph by selecting **Show Panel** ().

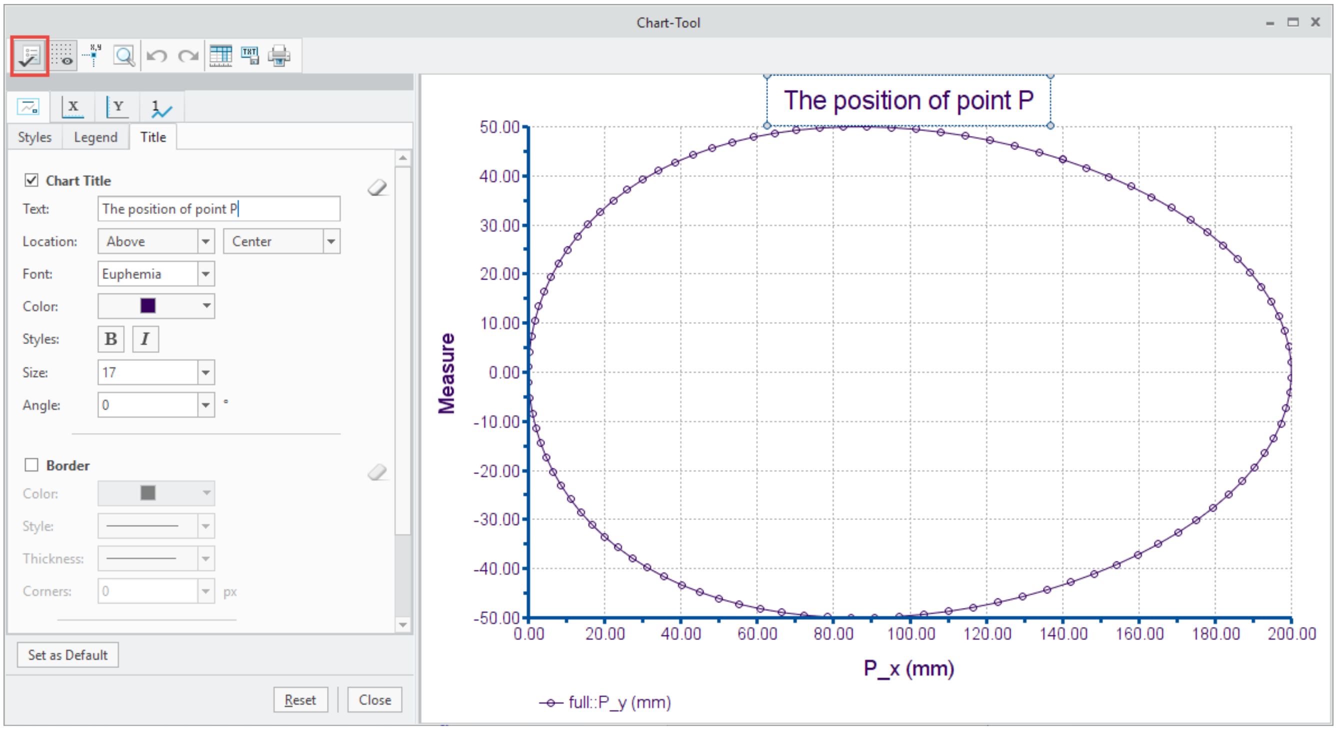
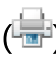


Figure 30: Location of the point P.

To export the graph as a picture, select **Print** () from the Chart-Tool window and select JPEG as a Output Format, define Paper Size (A4), print to file and define the file (Figure 31). OK to save the graph to the defined file. You can check that the file has correct size and plot thru operation system's file manager.

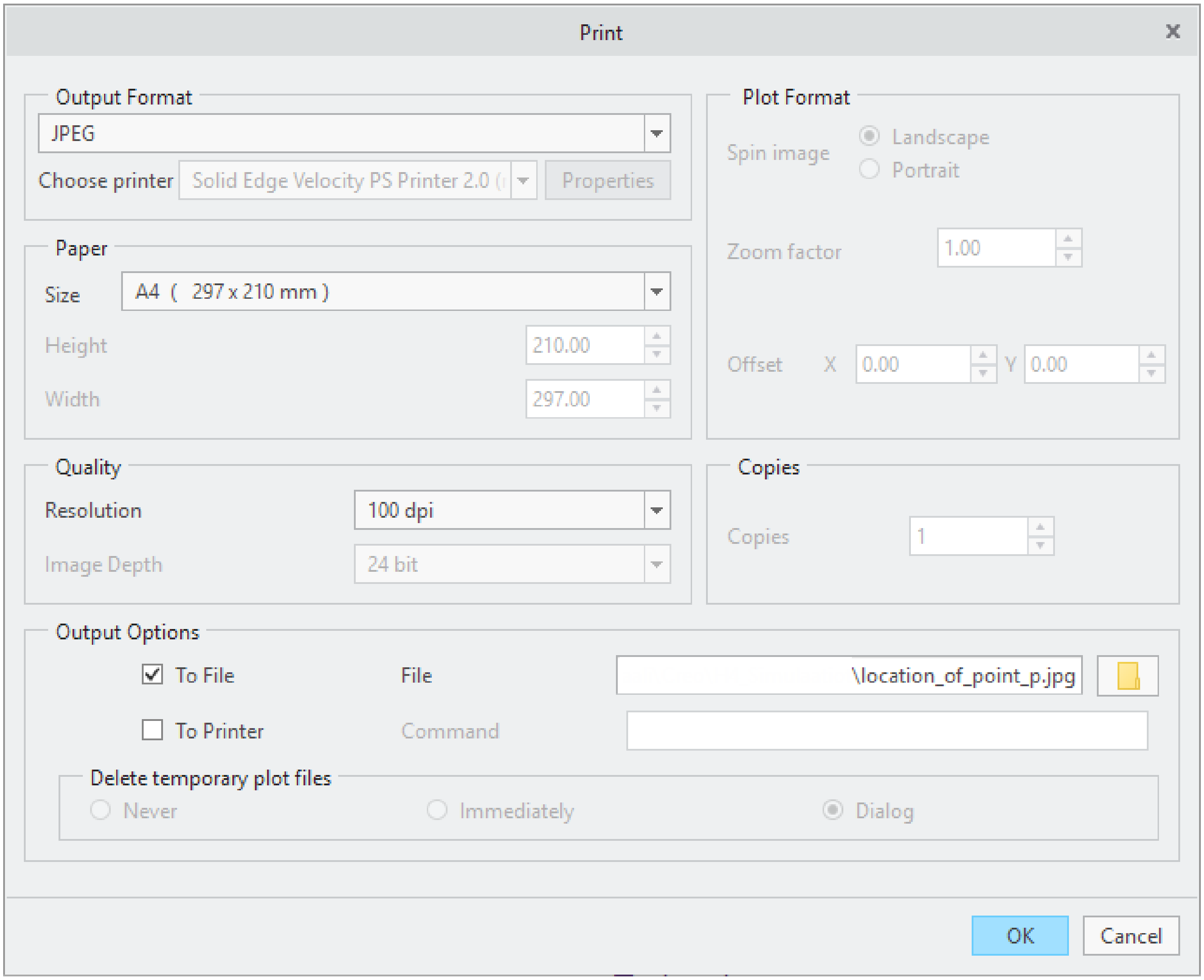


Figure 31: Settings to print the graph.

Calculating Needed Volume

We can use our mechanism model to estimate how much space it needs when operating. Select **Playback** (, *Analysis* group). In Playback window, ensure that **full_cycle** is *Result Set* and select **Create a Motion Envelope** (, Figure 32).

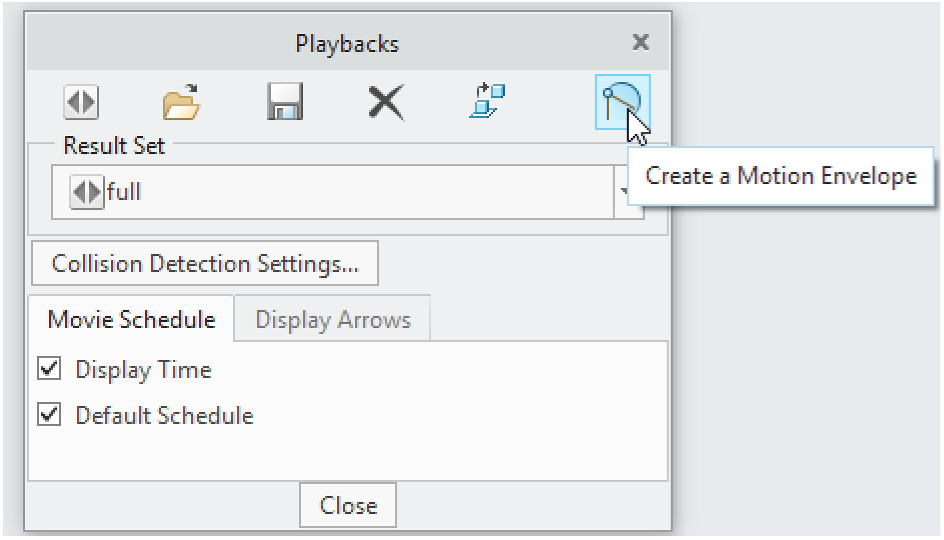


Figure 32: Selectin Create a Motion Envelope.

From the new window, set *Quality* to **2** and click **Preview** (Figure 33).

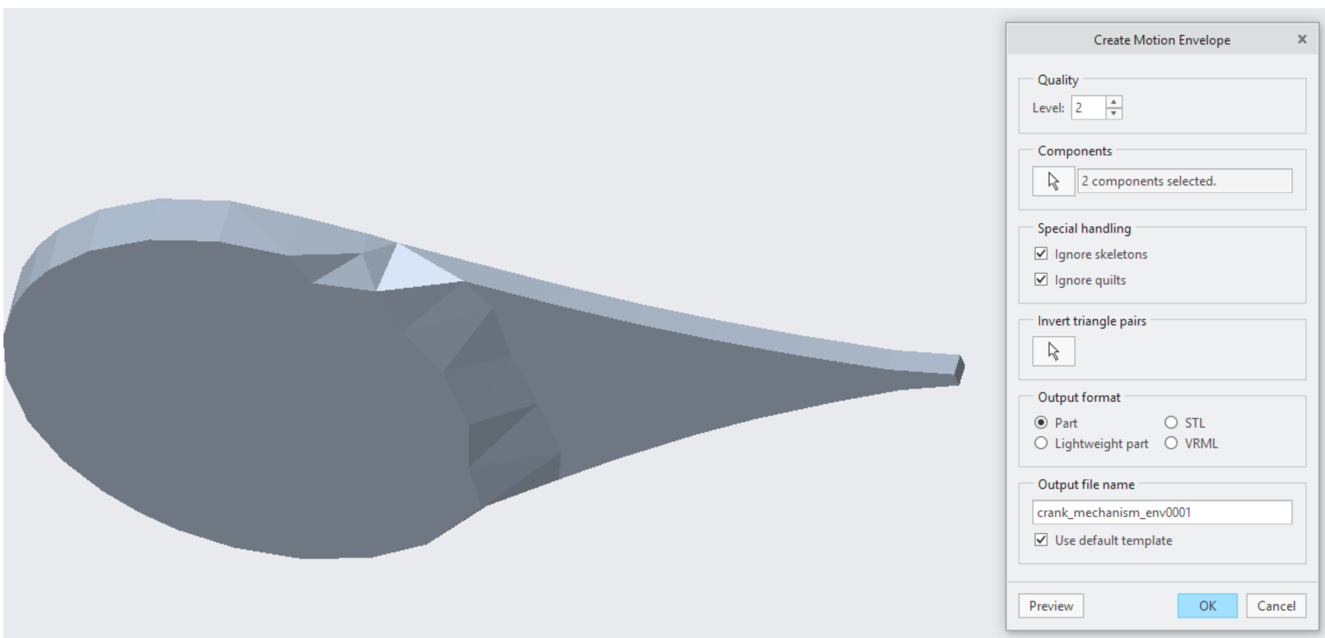
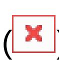


Figure 33: Motion envelope created.

Change quality to **6** to see the differences. Select **Part** as *Output format* and click **OK**. This will save the motion envelope geometry to your working directory. Click **Cancel** to close the tool and **Close**. Save your model and close **Close** () the mechanism model. When prompted, save results from mechanism analyses to your working directory (will create .pdk files).

Click **Open** (**Ctrl + O**) and select the previously created envelope part (default name crank_mechanism_env0001.prt). Select **Analysis** tab and **Mass Properties** (, *Model Report* group). Click **Preview** to calculate mass properties. If needed, you can save *Mass Properties* as a feature by replacing **Quick** from the drop-down list and giving a name.

To Return

Return to the MyCourses returning box:

1) JPEG-picture of *the Location of the point P* graph
2) The envelope part (.prt).

Bonus – Replacing Components

You can check how different dimensions affect the needed torque or the location of a point. This can be done in the assembly mode. Select the part to be replaced (for ex. LINK_200), hold **RMB** and select **Replace**. Click the folder icon () and select a family table member (for ex. LINK_150), **OK** and **OK**. If needed, regenerate the model (**Ctrl + G**). You can run the simulation again and see how this change affect the results.

Conclusions

Now that you have a working mechanism and besides the cool animation you can perform a different kind of measurements to better understand its performance. In the exercise you created a measurement of the crankshaft position in degrees, other types of measurements available are:

- ✓ **Position** — Measures the location of a point, vertex, or motion axis during the analysis.
- ✓ **Velocity** — Measures the velocity of a point, vertex, or motion axis during the analysis.
- ✓ **Acceleration** — Measures the acceleration of a point, vertex, or motion axis during the analysis.
- ✓ **Connection Reaction** — Measure the reaction forces and moments at connections.

- ✓ **Net Load** — Measures the magnitude of a force load on a spring, damper, servo motor, force, torque, or motion axis. You can also confirm the force load on a force motor.
- ✓ **Loadcell Reaction** — Measures the load on a loadcell lock during a force balance analysis.
- ✓ **Impact** — Determines whether impact occurred during an analysis at a connection limit, slot end, or between two cams.
- ✓ **Impulse** — Measures the change in momentum resulting from an impact event. You can measure impulses for connections with limits, for cam-follower connections with liftoff, or for slot-follower connections.
- ✓ **System** — Measures several properties that describe the behavior of the entire system. Properties that can be measured are Degrees of Freedom, Redundancies, Time, Kinetic Energy, Linear Momentum, Angular Momentum, Total Mass, Center of Mass, and Total Centroidal Inertia.
- ✓ **Body** — Measures several that describe the behavior of a selected body.
- ✓ **Separation** — Measures the separation distance, separation speed, and change in separation speed between two selected points.
- ✓ **Cam** — Measures the curvature, pressure angle, and slip velocity for either of the cams in a cam-follower connection.
- ✓ **User Defined** — Defines a measure as a mathematical expression that includes measures, constants, arithmetical operators, Creo parameters and algebraic functions.
- ✓ **Belt** — Measures the belt tension or slip.
- ✓ **3D Contact** — Measures the Contact area, pressure angle, or slip velocity during contact.

As you can see the list is quite long. It is clear that for time reason we cannot propose exercises that explore all the possible functionality. We hope that you understood how CAD models can be much more than just a collection of geometries. Thus, in the future we expect that you will be eager to try out the analysis feature that Creo offers.