Lab 2: Adding a Part to the Product Document

Objective

The objective of this lab is to introduce the procedure for adding a Part Document to an existing Product Document.

Lab 2 Agenda

1. Activating the Product Workbench

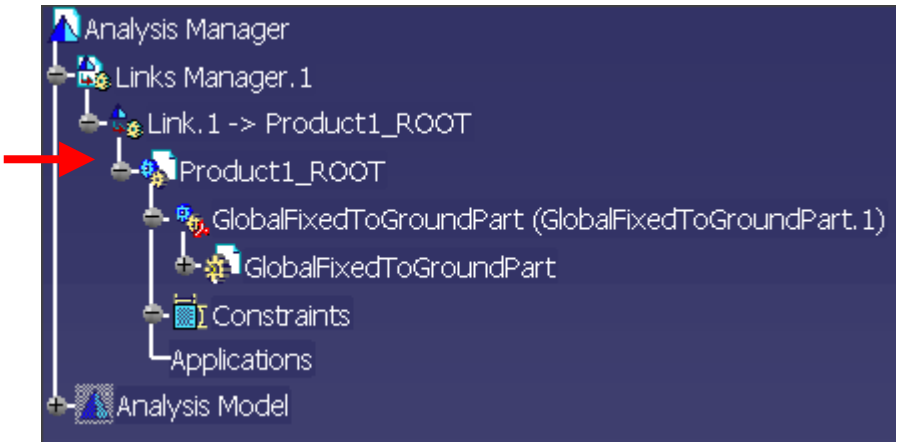
   Selecting the New Part Document Icon

2. Adding Geometry to the New Part Document

Activating the Product Workbench

Double-Click the branch of the Specification Tree labeled Product1_ROOT.

Notice that the Icon shown to the left of this branch has two gears. This icon always identifies a Product document, and can be used to activate the Motion Product Design Workbench.

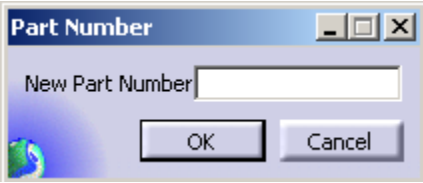


Once this icon has been selected the Motion Product Design Workbench should appear with the following Icon: 

Selecting the New Part Document Icon:

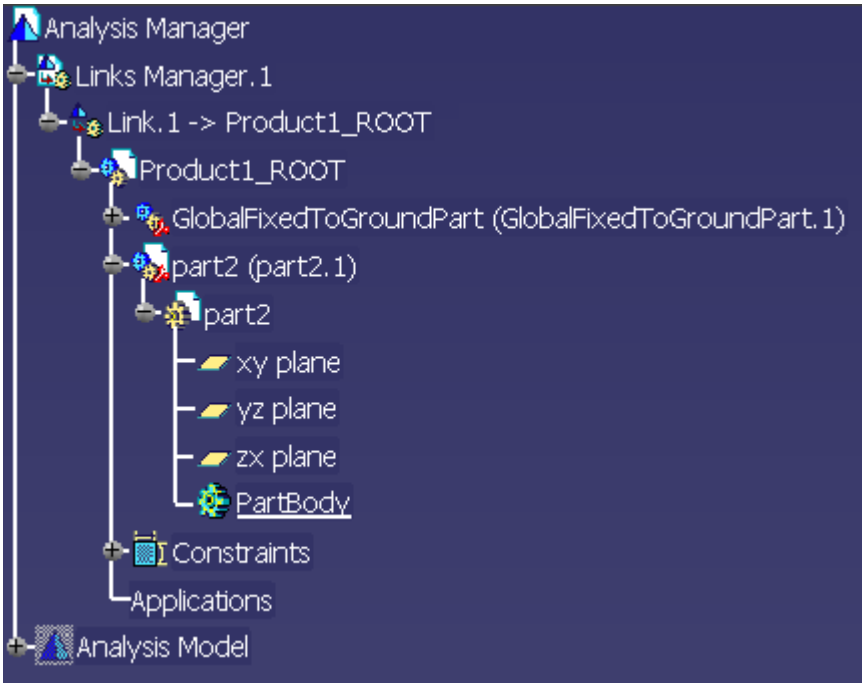
Once the Product Document has been activated, inserting a new Part Document is executed by selecting the New Part Icon:

Select this button once. This will bring up the New Part Number Dialog Field:



Type `Part 2` into the New Part Number Field, select **OK**.

Notice that the added Part Document now has a branch within the Specification Tree beneath the Product1_ROOT Product Document.



Adding Geometry to the Part 2 Part Document.

This section of the lab will focus on defining a cylindrical solid within the Part 2 Part Document.

Activating the Part Workbench

Double-Click the Part 2 Part Document Branch of the Specification Tree indicated by the single gear Icon:

To add solid geometry it will be necessary to use the Part Design Workbench. To switch to this workbench, select **Start → Mechanical Design → Part Design** from the Main Menu. The resulting workbench should show a solid single gear at the top:

Opening the Sketcher window

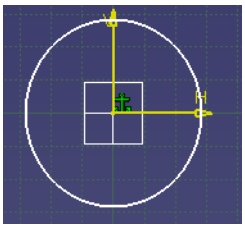
Beneath the Part 2 Part Document, Select the **xy-plane**. This is done by a single click using the left mouse button. It should highlight orange once selected.

From the Part Design Workbench select the Sketcher Button:

Once the Sketcher Button has been selected the Screen will change to show a 2 dimensional Sketching Grid. If the entire grid is not visible, zoom out until it appears. A new Sketcher Workbench will appear to the right, indicated by the Sketcher Icon:

The profile of the cylinder will be a circle. From the Sketcher Workbench, Select the Circle Icon:

To sketch a circle, click once with the left mouse button at the origin of the part indicated by the intersecting yellow horizontal (H) and vertical (V) axes. Drag the mouse outward to a desired circle radius. Click the mouse button again to drop the circle.



Exit the sketcher by selecting the Exit Button from the Sketcher Workbench:

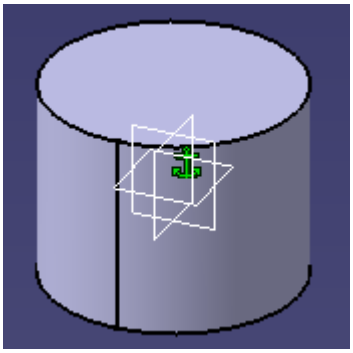
Extruding a Solid from a sketch:

Upon Exiting the Sketcher the Part Document will re-appear to the right of the screen. Select the **Pad Button**:

This will open the Pad Definition Dialog:

Toggle the Mirrored Extent Selection and then Select **OK**.

The result should be a solid cylinder:



This concludes Lab Session 2.