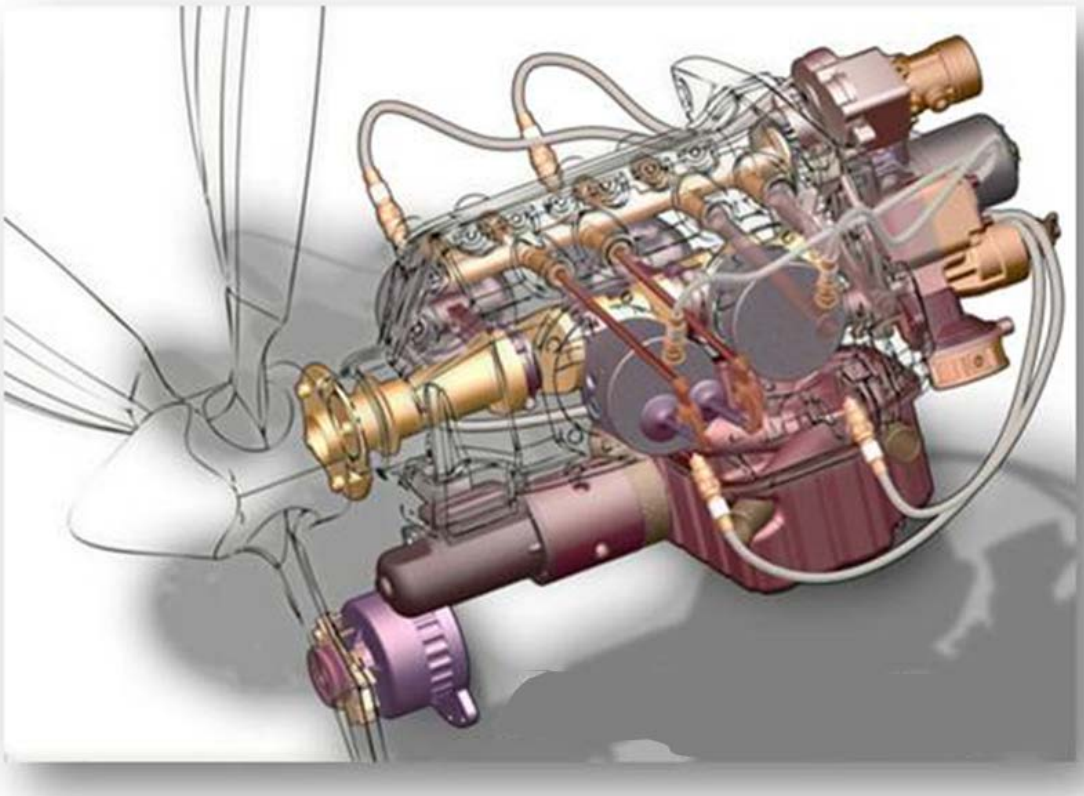
Section

3

HOW TO MODEL ALMOST

ANYTHING



Assembly Modeling

PRODUCT DEVELOPMENT MANUAL

How to Model Almost ANYTHING

WRITTEN BY JORDAN COX

NOTICE OR RIGHTS

TRADEMARKS

ACKNOWLEDGEMENTS

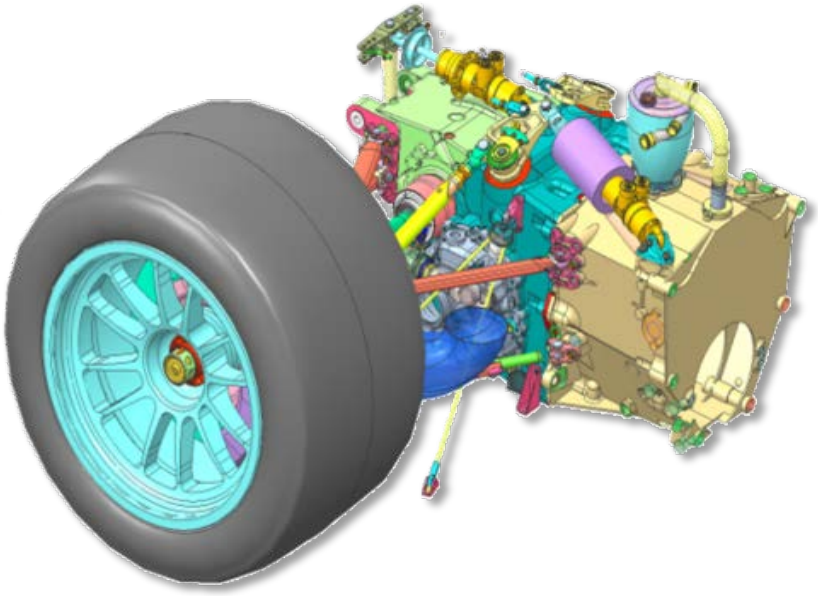
Scott Morris, Mark Fischer, Adam Haas, Ayora Berry

Section

3

CREATING AN ASSEMBLY MODEL

UNDERSTANDING HOW TO PLAN AND CREATE AN ASSEMBLY MODEL



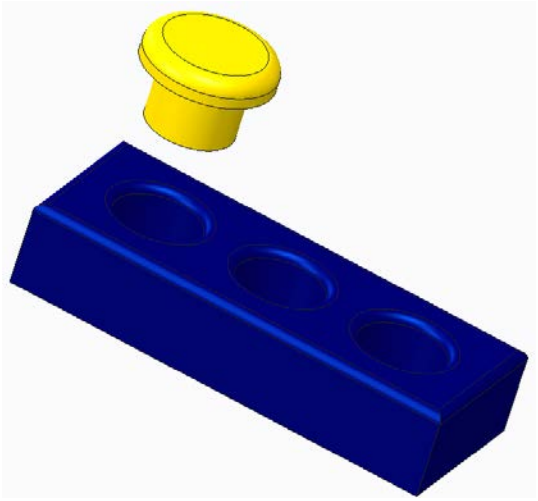
Assembly models are a collection of part models. Each part model must be constrained to have the right relationships with all the other parts in the assembly model.

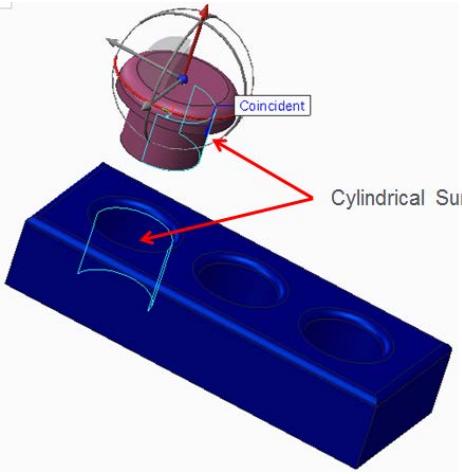
Assembly models are the way we model products. All the parts that make up a product are assembled into the model so that we can determine the overall height, weight, volume, center of gravity and other properties of the final product. To understand assembly models, it is important to understand assembly constraints and how to use them to assemble parts.

There are two types of constraints that are used to build assembly models; static and kinematic.

Static Constraints

When parts are assembled using a static constraint, the parts may no longer move with respect to each other. They are fixed like they are glued together. A simple example will illustrate how these constraint conditions are specified.

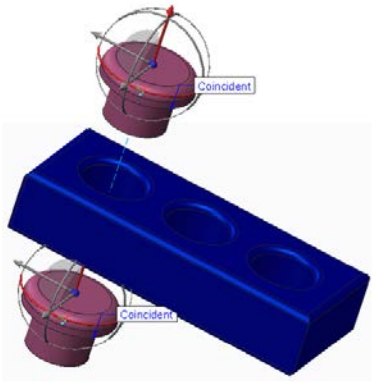


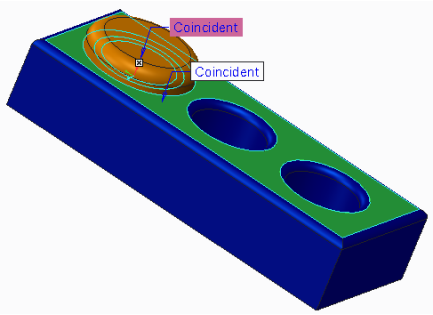


If a yellow peg is to be inserted into a blue block as shown, it requires two constraints; first the axes of the peg and the hole must be aligned and second the flat underside of the head of the peg must be aligned with the top of the blue block.

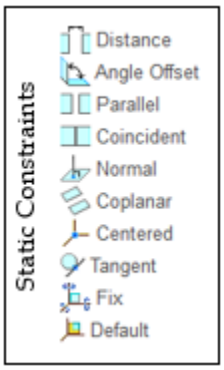
Getting two axes or surfaces to align requires what is called a **Coincident** constraint. The **Coincident** constraint consists of selecting the cylindrical surface of the peg and the cylindrical surface of the hole and telling Creo to **make the axes of these coincident**.

Creo aligns the axes of the two but doesn't know where to place the peg along the axis. So the peg can now move anywhere along the axis of the hole as shown.





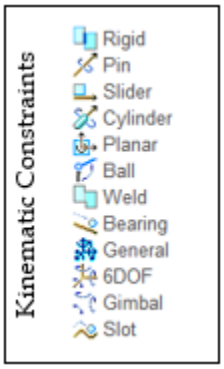
To completely insert the peg into the hole, a second constraint must be specified. This second constraint is also a **Coincident** constraint and consists of selecting the flat underside of the peg and the flat top face of the block since these two faces will be touching when the peg is completely inserted.



Static constraints allow parts to be connected in any orientation needed. The orientation is defined by relating two surfaces. Creo provides a list of static constraints that can be used to place parts with respect to each other and then fix that orientation. The Default constraint places the part so that the part's origin is fixed to the assembly model's origin. It is used for the first part in an assembly since there are no other parts to orient onto.

Kinematic Constraints

The other type of constraint is a kinematic constraint. These are constraints that allow you to orient parts with respect to each other but still allow relative motion. For example if we want to assemble a wheel onto an axle, we want it to stay attached to the axle but we want the wheel to turn with respect to the axle.

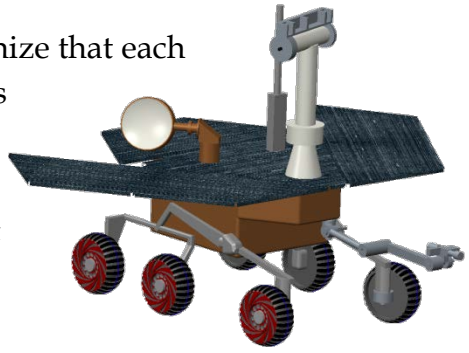


Creo provides a list of these constraints just like for static constraints. These constraints are defined by the motion that they allow.

Defining Constraints

Constraints are defined by selecting a type of constraint and then selecting two surfaces to define the references. If a static coincident constraint is selected, the two surfaces will define what will be coincident; axes if the surfaces are cylindrical or the surfaces themselves if they are flat). Because selecting surfaces is so important in defining constraints, Creo has special tools for selecting surfaces.

When you are defining constraints it is important to recognize that each time you left-click with your mouse, Creo interprets that as selecting a surface. So once you bring a part into an assembly model to place it, you need to be careful about left clicking and make sure that you left-click only to select the appropriate surface.



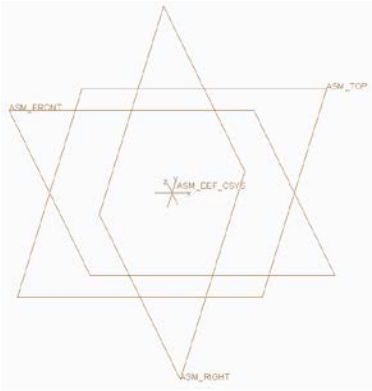
Let's try out what we have learned to this point.

*Find in the Appendix "**Exercise 7: Mars Rover**" and follow the instructions to complete the exercise.*

Planning an Effective Assembly Model

Now that you are familiar with the elements of an assembly model, it is important to learn how to plan an effective model.

Product or Assembly Models



The first part of any effective 3D model plan is to define your datum and references plan. You need to decide where your parts are going to be with respect to the origin. Also you need to define any additional datums and references that you will need in your assembly. To begin with, you will use the default datums and coordinate systems. However, as your models get more sophisticated, you will need to carefully plan your datums.

Assembly Plan

The next part of your plan needs to be the actual assembly plan. This involves identifying the order of the parts as well as the constraints that you will use when assembling them. The first part should be the part most likely to be the base or stationary part. For example, the body is the first part in the wooden man assembly, which makes more sense than making an arm part be the first part.

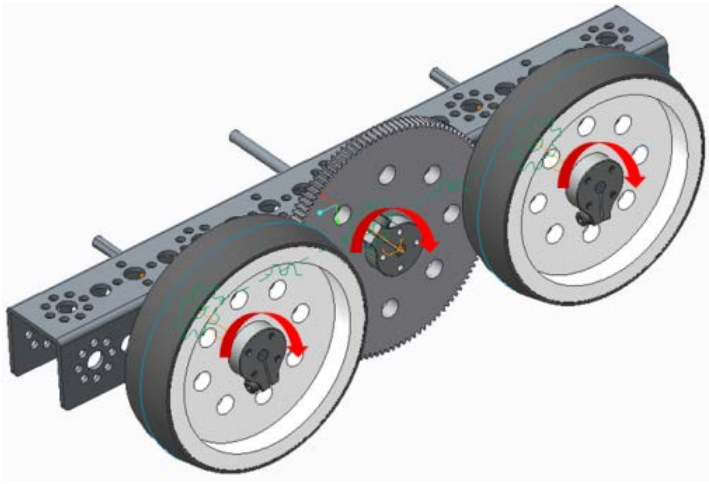
Create a document for your plan. List each part starting with the first one and then identify what types of constraints you will use to place it with respect to the former parts. Here is an example.

Wooden Man Assembly Plan

Part	Constraint Type	Notes
Body 	Default 	Set the body part at the origin with a default constraint.

Neck 	Static Coincident 	Use coincident constraints to place the neck in the hole of the body
Leg 	Kinematic 	Use a kinematic constraint to place the leg on the dowel so that it will move relative to the body.

Now let's practice by doing an exercise.



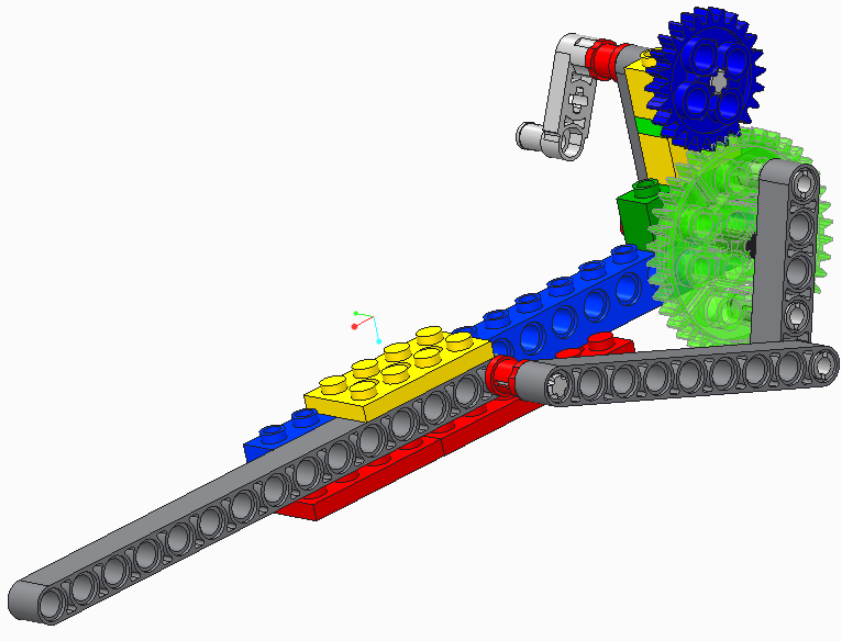
*Find in the Appendix "**Exercise 8: Assembly Modeling**" and follow the instructions to complete the exercise.*

Assembly models provide a method of developing virtual products that can be explored and simulated to determine if they will function as desired before they are

actually created. Now that you have learned about assembly models, exercise 9 will let you plan and create your own.

Kits of Parts

Often companies use a standard set of parts to build their products. These are often referred to as kits of parts. It is easy to create a virtual form of these kits of parts by creating all of the part files and sub-assemblies. In exercise 9 you will use a Lego kit of parts that is made up of a small set of Lego parts. This practice of kits of parts is very helpful when multiple product concepts are being explored.



*Find in the Appendix "**Exercise 9: Lego Mechanism**" and follow the instructions to complete the exercise.*

Section

3

HOW TO MODEL ALMOST

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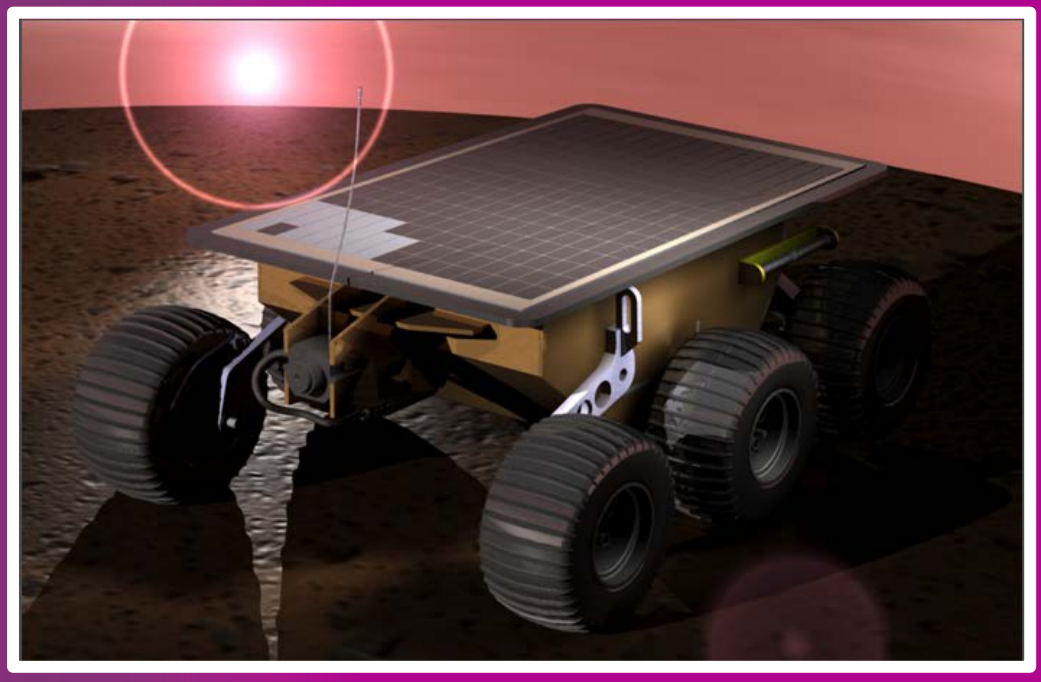
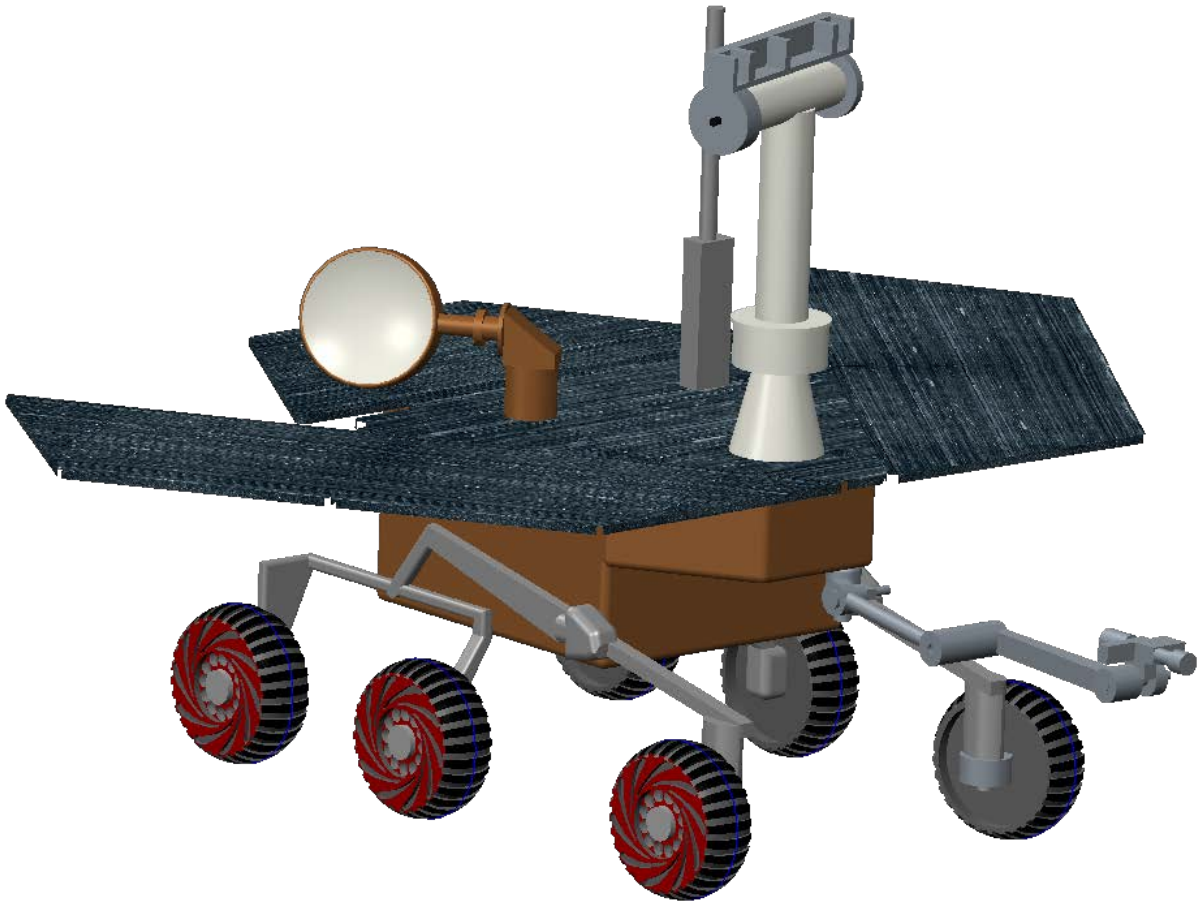


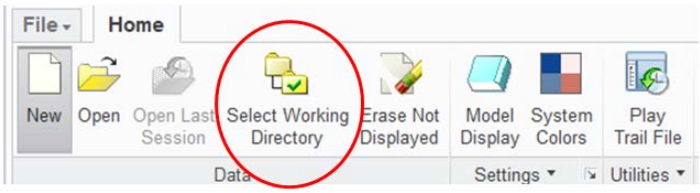
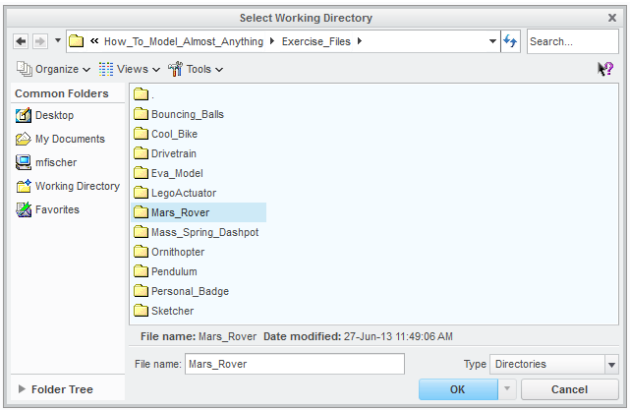
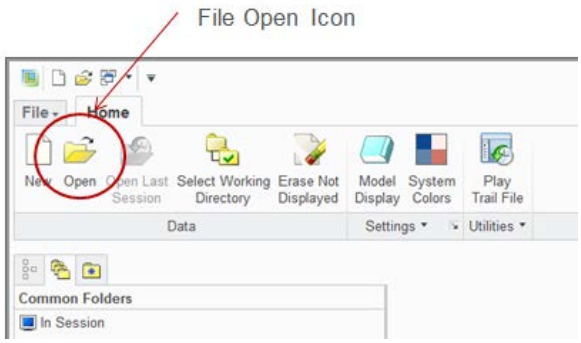
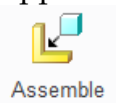
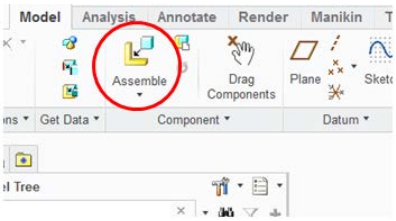
APPENDIX

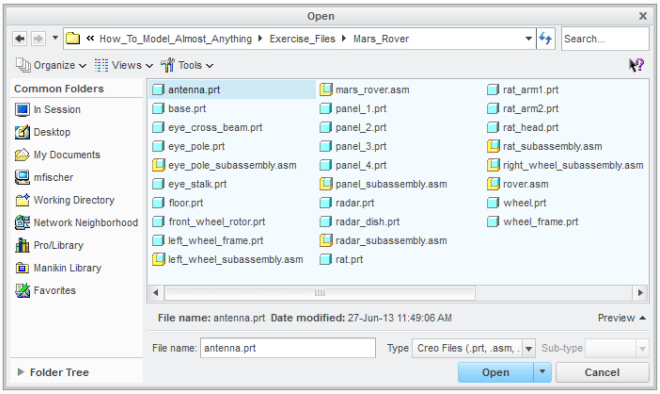
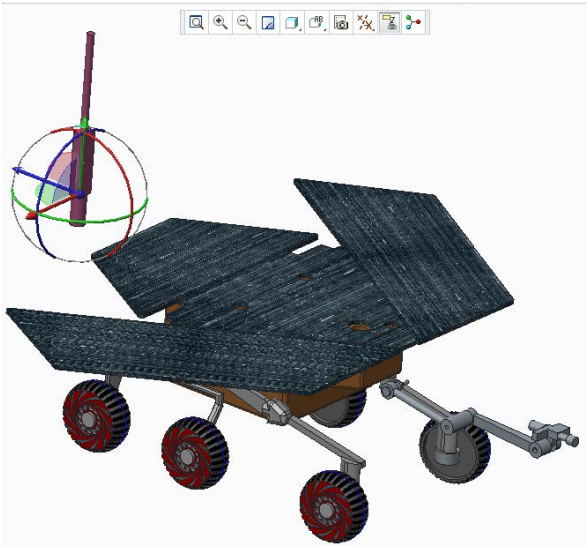
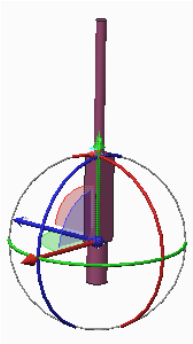
EXERCISE 7: MARS ROVER

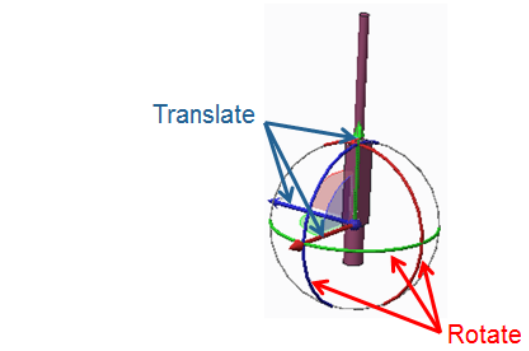
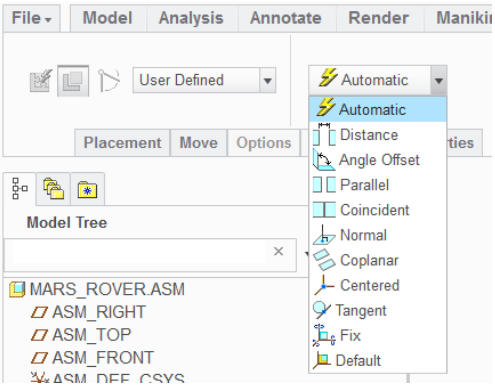
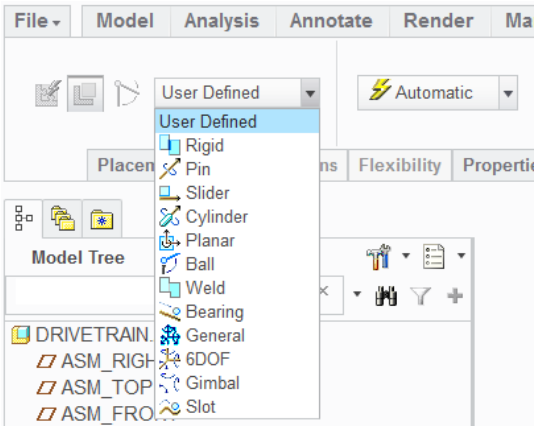
An assembly model of a Mars rover is shown below. This exercise will help you build this assembly model by applying assembly constraints to the parts.

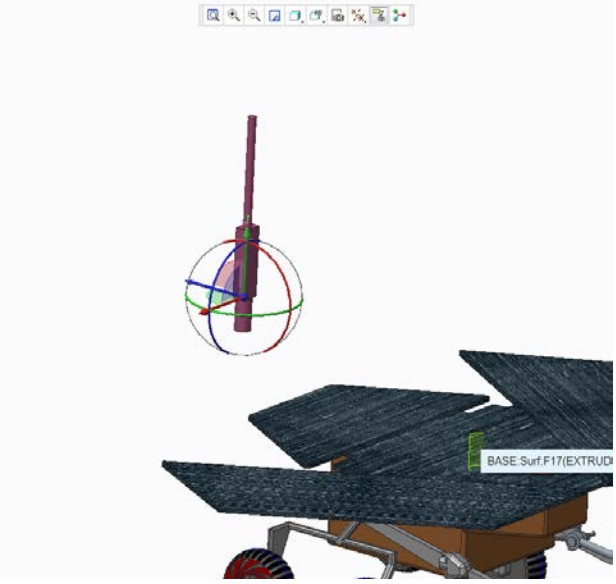
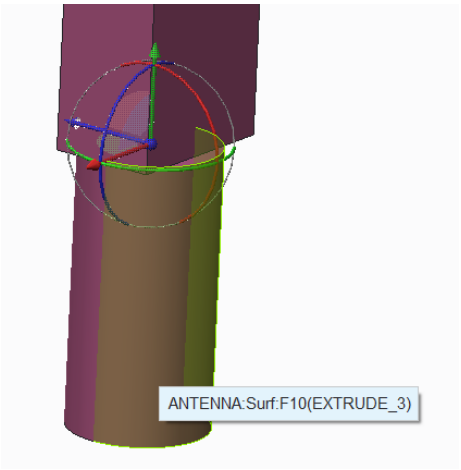
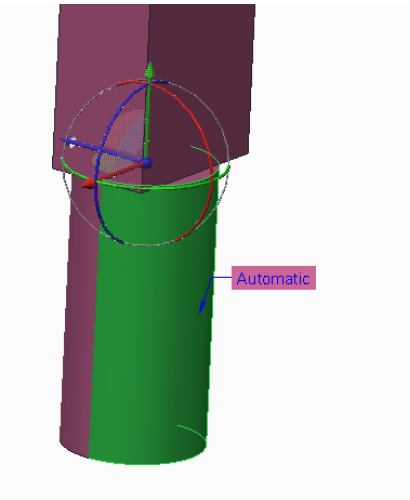


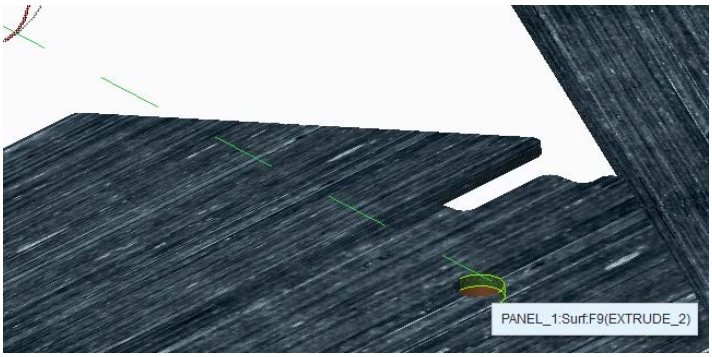
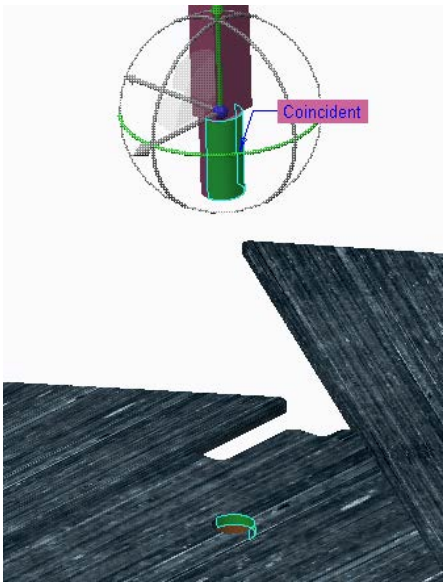
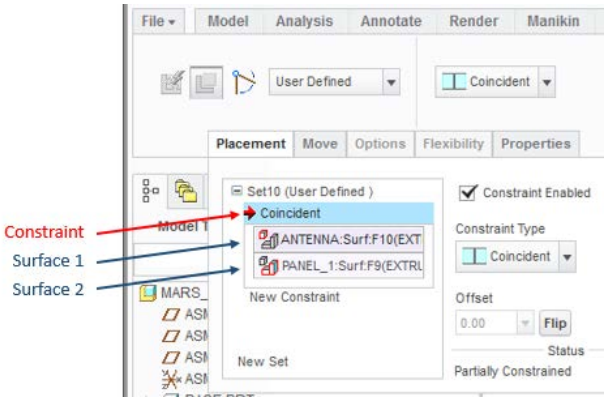
We will begin with the Mars rover partially assembled and then add the remaining parts to complete the assembly.

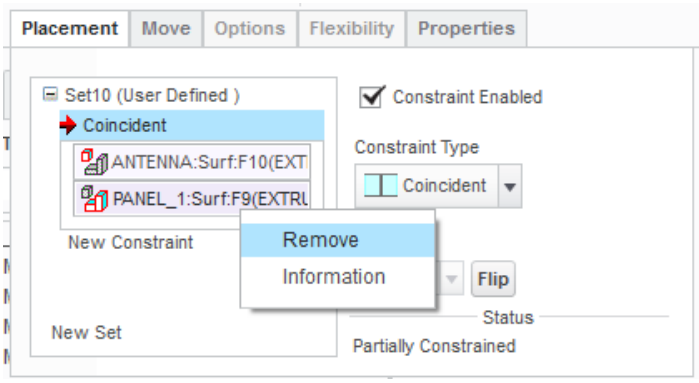
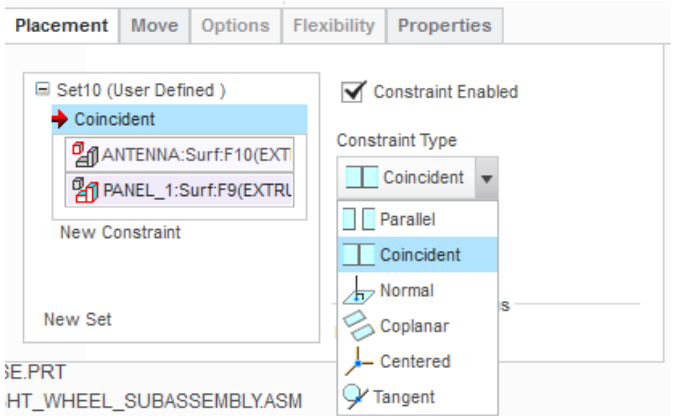
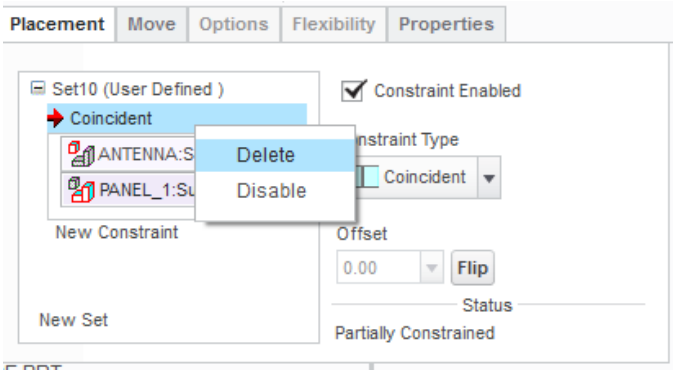
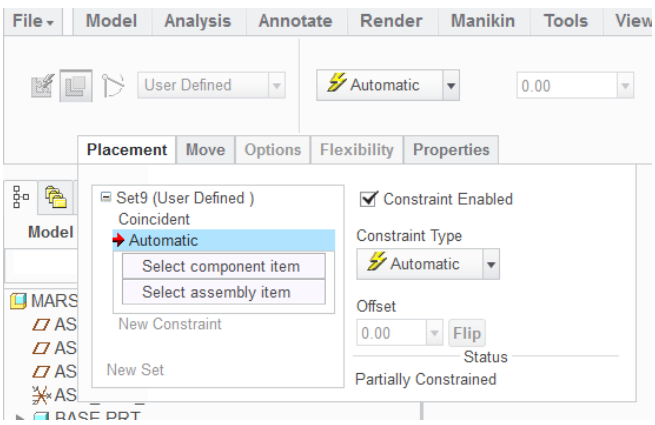
Open **Creo Parametric 2.0** and click the icon called **Select Working Directory** to set the folder in which you will be working.	
Navigate to the folder called "**Mars_Rover**" and click **OK**.	
Now click on the **Open** icon to open the file: "**mars_rover.asm".** Double click on the file or select it and click **Open**.	
The rover is partially assembled. We will add an antenna, a satellite dish and the camera eye stalk. ***Note**: you might want to turn off your datums to see the assembly clearly.*	
Click on the Assemble tool in the upper menu. 	

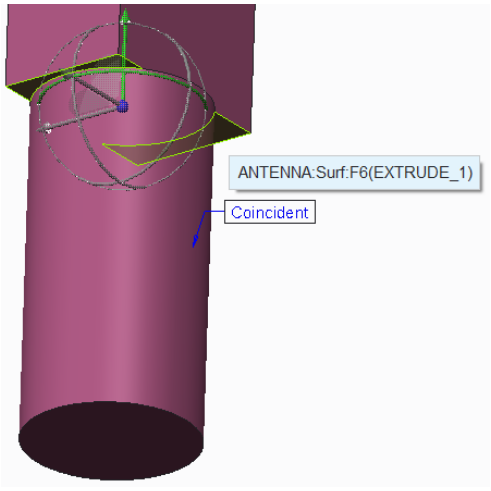
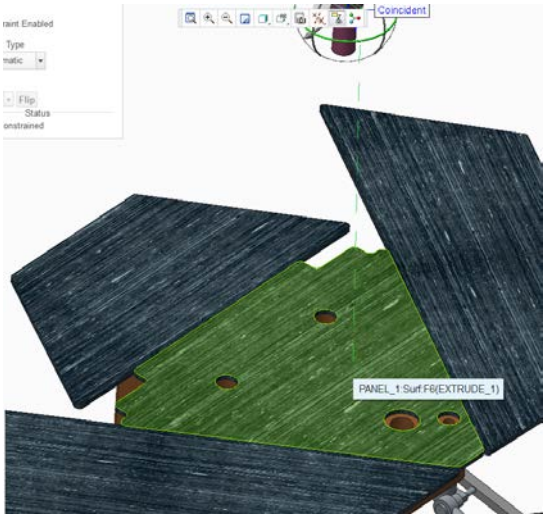
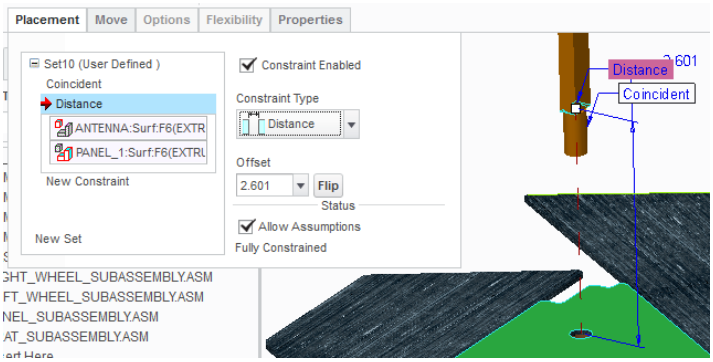
Find the part called "**antenna.prt**" and select it and click **Open**.	
The antenna part will appear in purple and will follow your cursor. Left-click anywhere to drop the part.	
You should also notice that when you left-click to drop the antenna, an orientation sphere appears at one end of the antenna.	

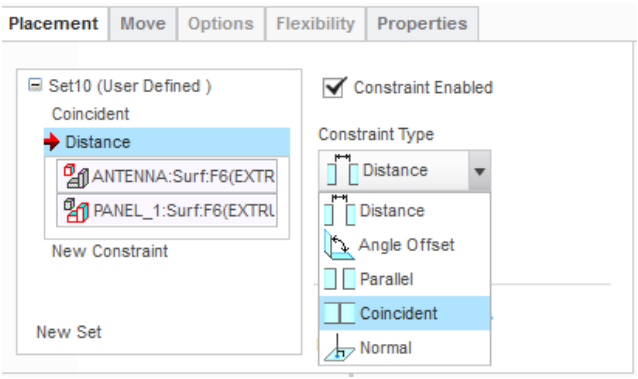
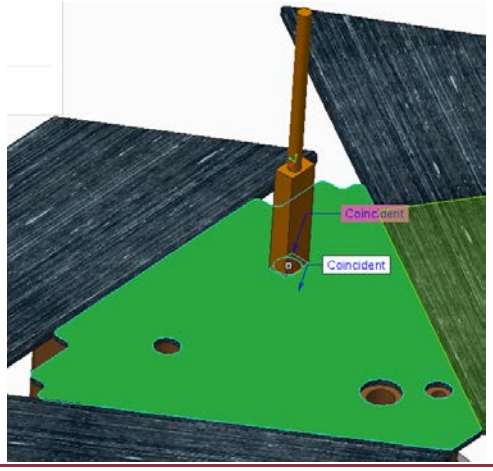
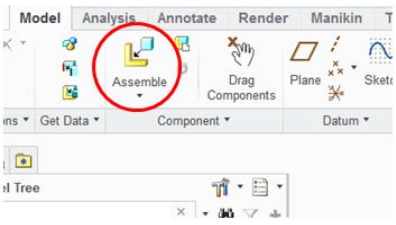
The sphere allows us to orient just the current part by left clicking on the axis to translate or the circles to rotate.	
Also the upper menu changes to the assembly dashboard. The dashboard provides tools specifically for doing assembly.	
The **Automatic** pull-down menu provides a list of the static constraints that can be applied. It is called automatic because these are the constraints that **Creo** uses if you don't specify.	
The **User Defined** pull-down menu contains kinematic constraints. These constraints allow parts to move and must be specified by selecting them before you apply them to parts.	

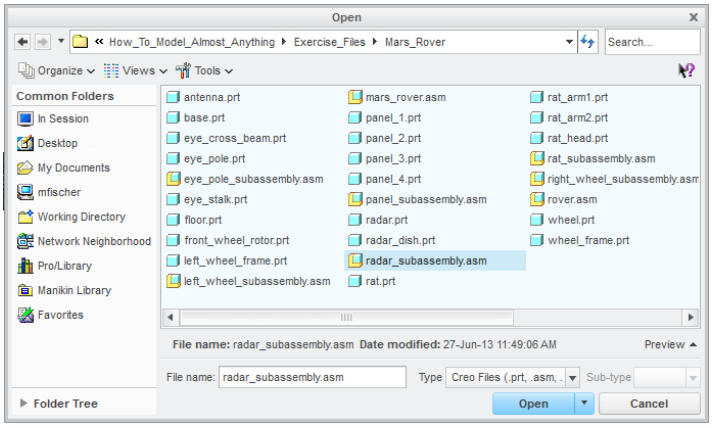
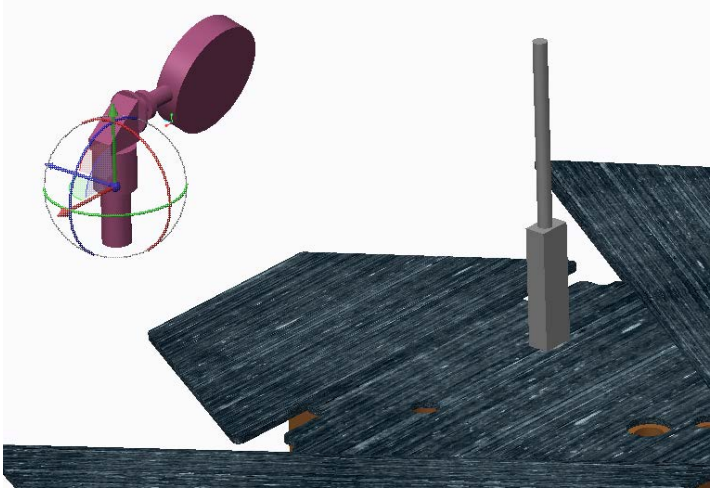
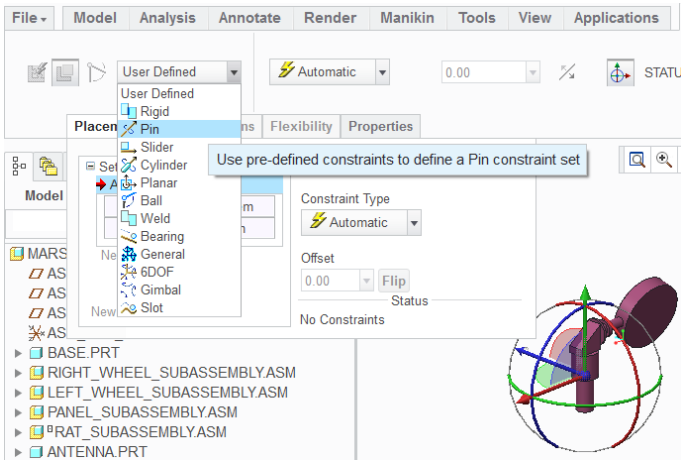
Since the antenna is not going to move with respect to the rover, we can let Creo determine the right static constraint to apply just by selecting surfaces that we want to be coincident.	
Start by zooming into the antenna. This is done by placing your cursor on the bottom of the antenna and then using the scroll wheel to zoom into the antenna. As you hover over the cylindrical surface it will highlight in green. This means that if you left-click, this surface will be selected.	
So left-click on that surface and it will turn dark green to show it has been selected.	

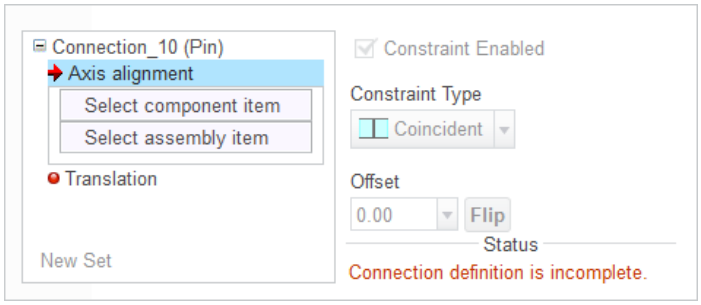
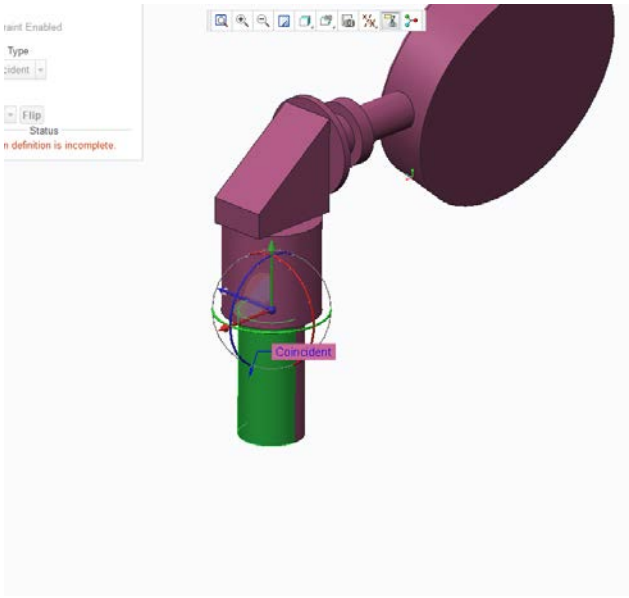
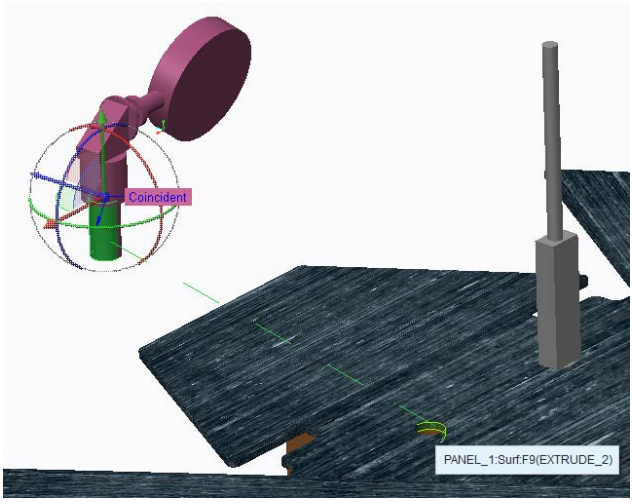
Next, zoom out using your scroll wheel and then place your cursor near the hole shown in the figure. Scroll in and hover until the hole highlights in green.	
Left-click to select the hole and notice that **Creo** aligns the axes of the antenna and the hole.	
Now let's look at the **Placement** tab in the dashboard. It contains the constraint information. Click on the **Placement** tab to open it. A **Coincident** constraint has been added and the two surfaces you selected are identified.	

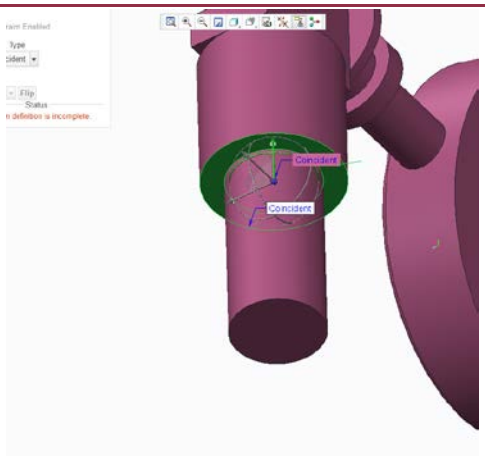
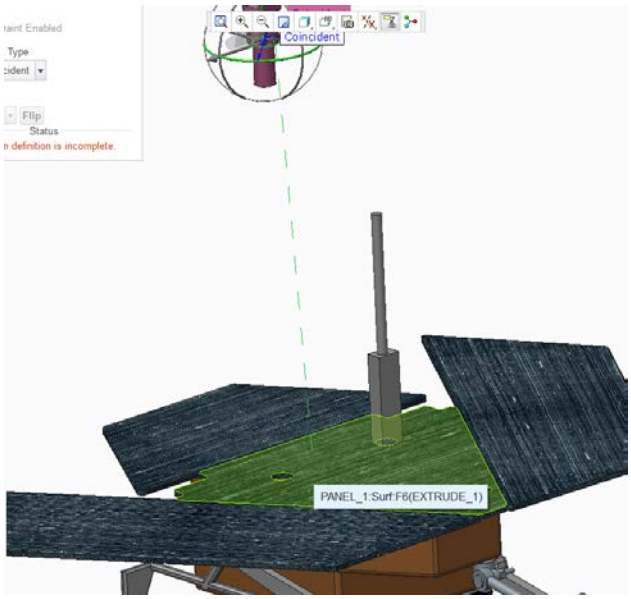
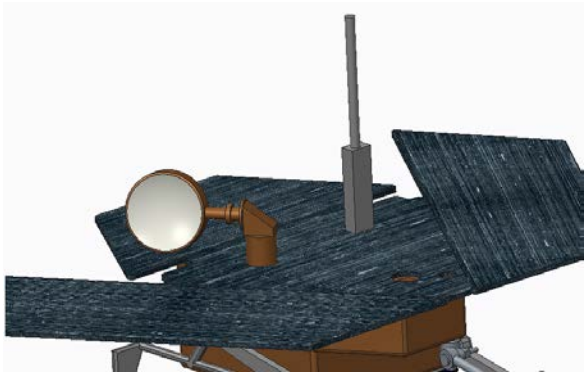
If for some reason you didn't select the right surfaces you can change them by right clicking on the surface and selecting **Remove**.	
You can also change the constraint type by selecting from the pull-down menu at the right side of the **Placement** tab	
You can also delete the constraint and start over by right clicking on the constraint and selecting **Delete**.	
Now let's add a new constraint. You can click on **New Constraint** and just select the next surface. Creo immediately opens an **Automatic** constraint waiting to see what surfaces you will pick to determine the type of constraint that is appropriate.	

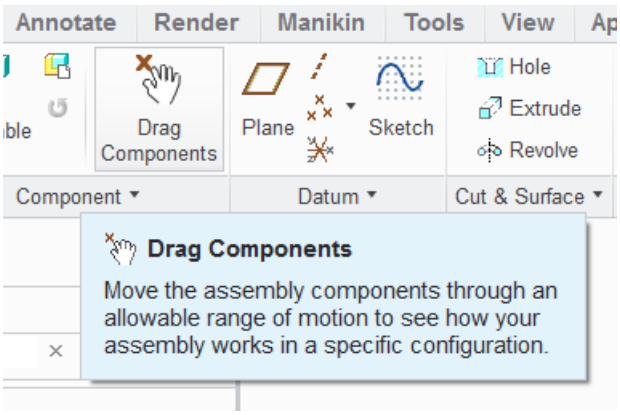
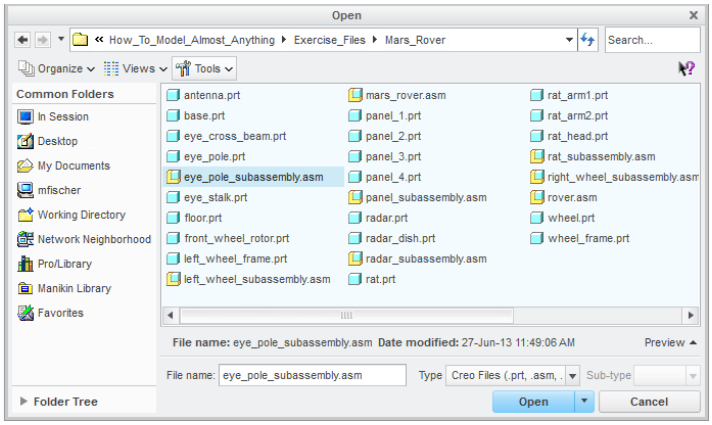
Now select the flat surface at the bottom of the square part of the antenna.	
Now select the top of the panel as the second surface. This tells **Creo** that you want the two surfaces you selected to be coincident.	
Notice that in this case **Creo** chose a **Distance** constraint. We need to change it to a **Coincident** constraint.	

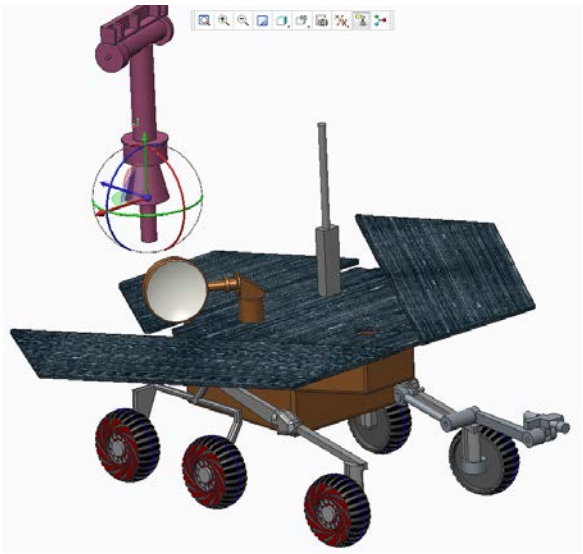
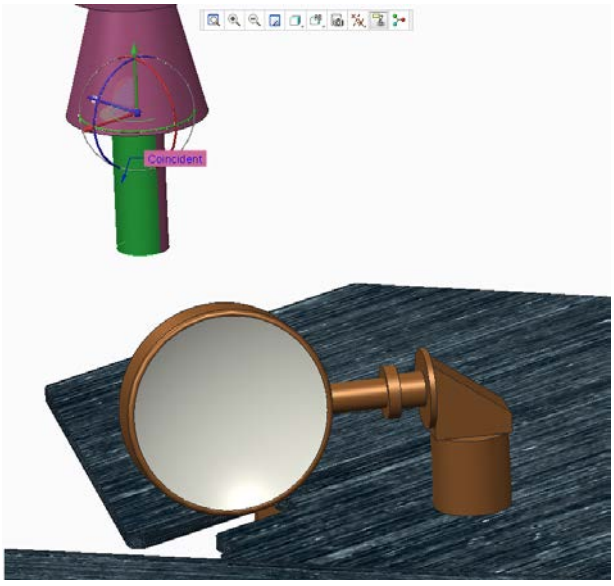
Change it by clicking on the pull down menu and selecting **Coincident**.	
This will change the constraint so that the surfaces are coincident.	
Finish placing the antenna by clicking on the green check mark in the assembly dashboard.  Congratulations!!! You have placed your first part!	
Let's place the satellite dish and use a kinematic constraint so that it can rotate. Click on the Assemble tool in the upper menu.	

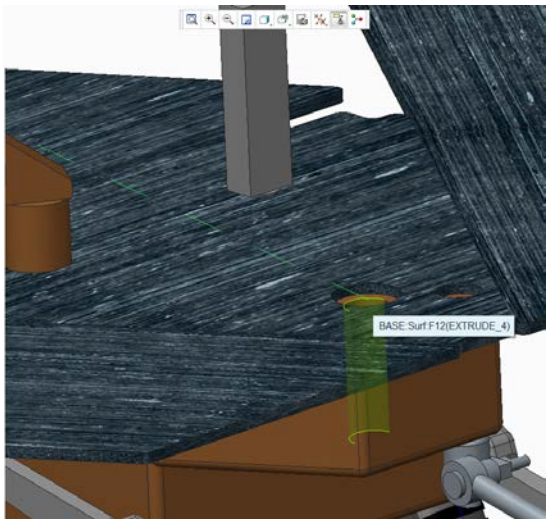
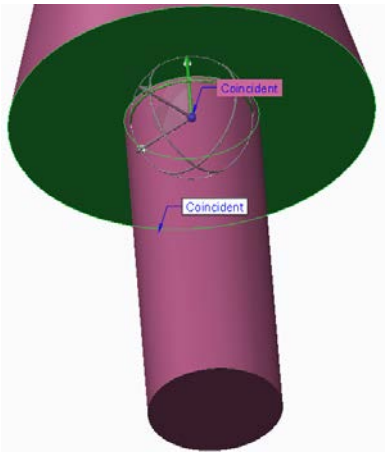
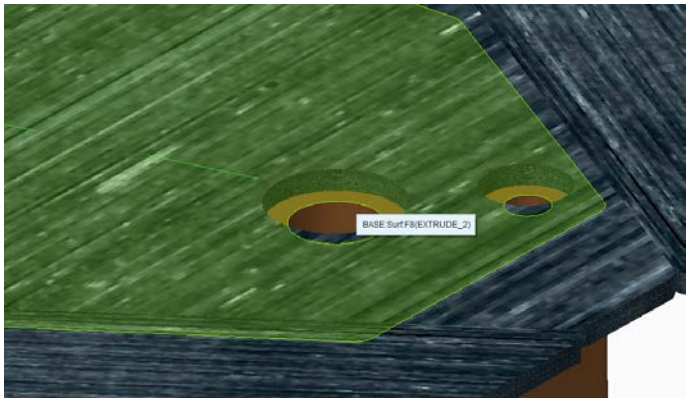
Select the file called "**radar_subassembly.asm**" this is a sub assembly of parts. This means that it is an assembly model that was created previously. Click **Open**.	
Left-click to drop it and then click on the **Placement** tab in the assembly dashboard.	
Now click on the pull down menu called **User Defined** and select a **Pin** constraint.	

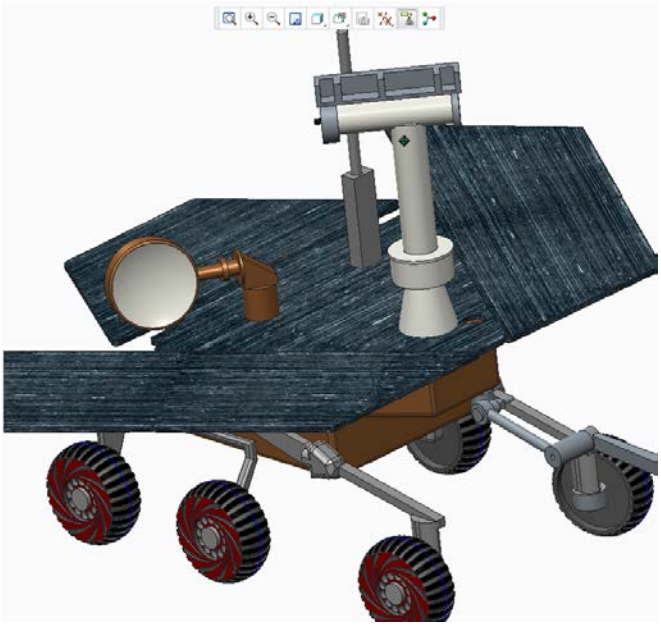
Notice that **Creo** places a **Pin** constraint in the **Placement** tab and identifies that you must complete this constraint by specifying an **Axis alignment** condition and then a **Translation** alignment condition.	
Zoom in and left-click to select the cylindrical surface of the radar dish stem as shown.	
Then zoom out and select the hole on the rover assembly as shown. The axes of the radar and the hole will align.	

Now rotate your assembly model and select the flat face underneath the stem of the radar unit.	
Then zoom back out and select the top of the panel	
Finish the placement of this radar unit by clicking on the green check mark in the assembly dashboard.	

Now use the **Drag Components** tool in the upper menu to spin your radar antenna. Click on the tool and then left-click anywhere on the radar unit and drag it around with your cursor. When you are done, click **Close**.	
Now we just need to place the eye stalk to finish this assembly model.	
Click on the **Assemble** tool in the upper menu and then select the file "**eye_pole_subassembly.asm**" and click **Open**.	

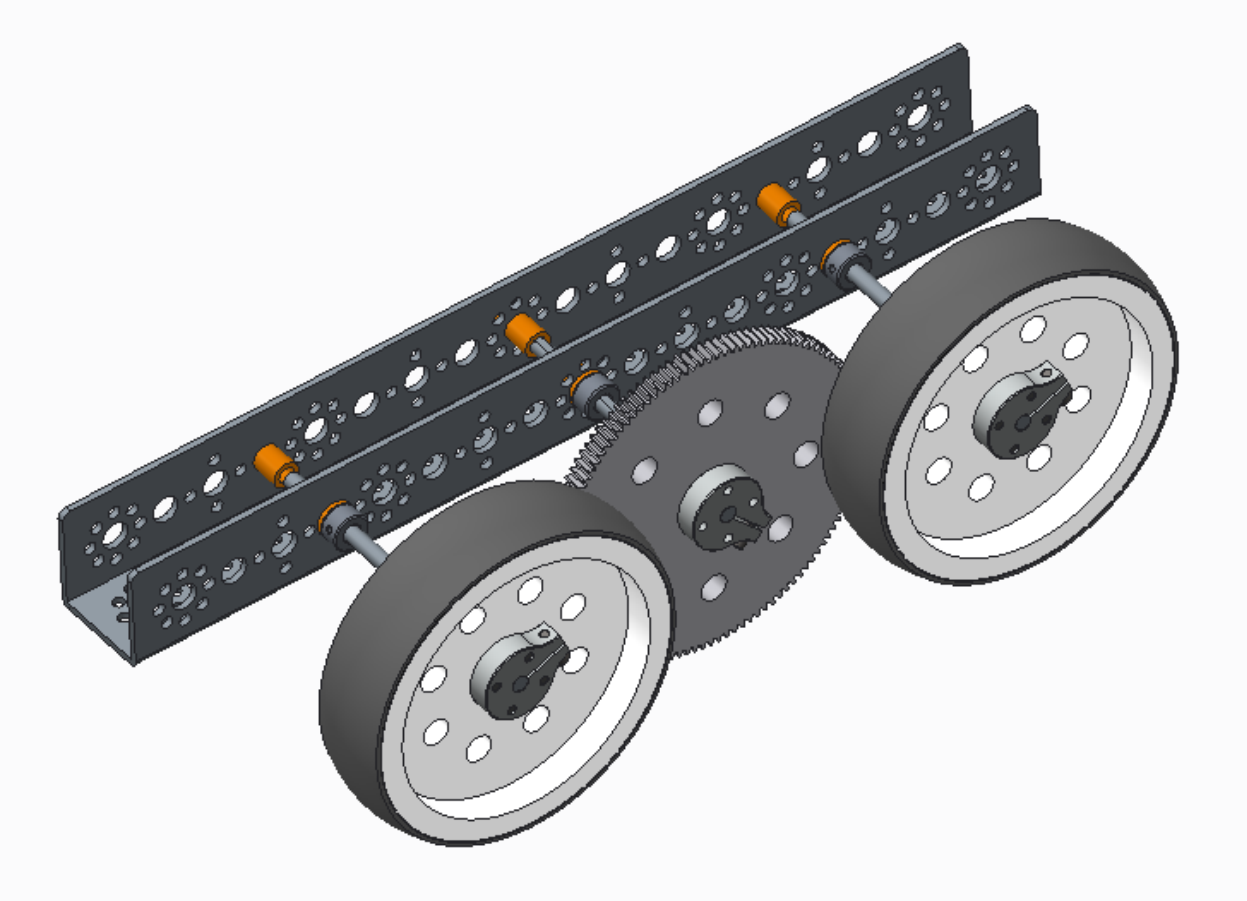
Left-click to drop the eye stalk. Find the **User Defined** pull down menu and once again left click on **Pin**.	
Left-click on the cylindrical surface of the eye stalk pole as shown.	

Then left-click to select the hole in the front of the rover assembly.	
Click on the bottom flat shoulder of the eye stalk.	
And then left click on the top surface of the rover body base *(not the top of the panel)* as shown. To select this surface, zoom into the hole and choose the brown top surface.	

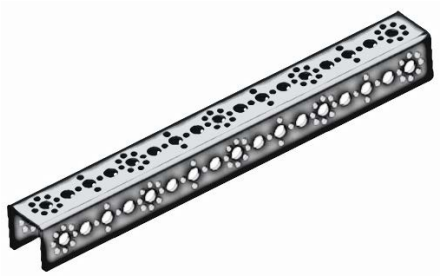
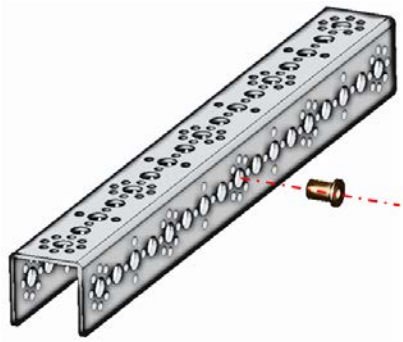
Then click on the green checkmark in the assembly dashboard to finish. Now the eye stalk is in place. Use the **Drag Components** to rotate the eye stalk.	
You have completed the rover assembly. You may use the **Drag Components** tool to move all of the different parts of the assembly. When you are done click **Close** and then save your assembly file using the **File** tab and the **Save** option. Now close **Creo**.	
Congratulations, you have finished this exercise!	

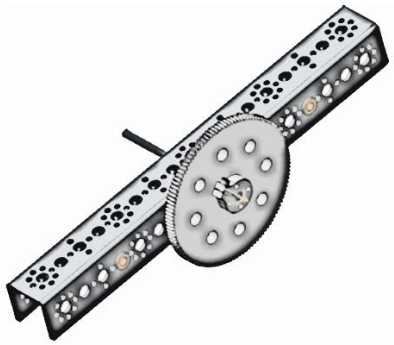
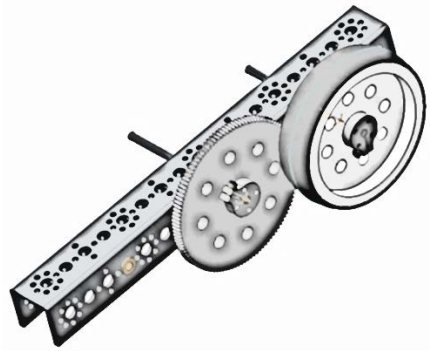
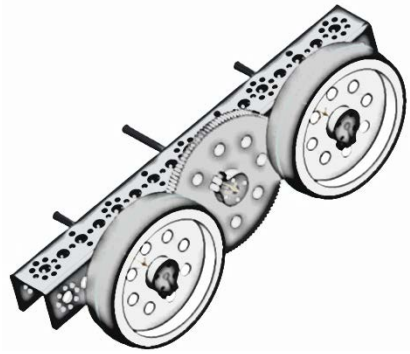
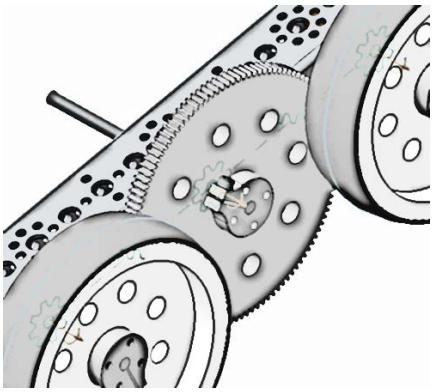
EXERCISE 8: ASSEMBLY MODELING

Using a model plan is always the best practice for creating an assembly model.

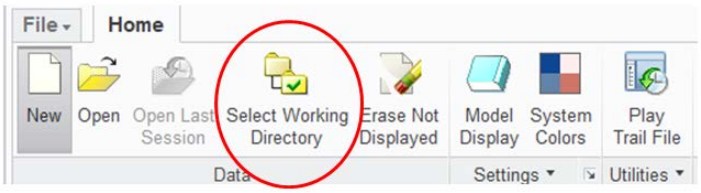
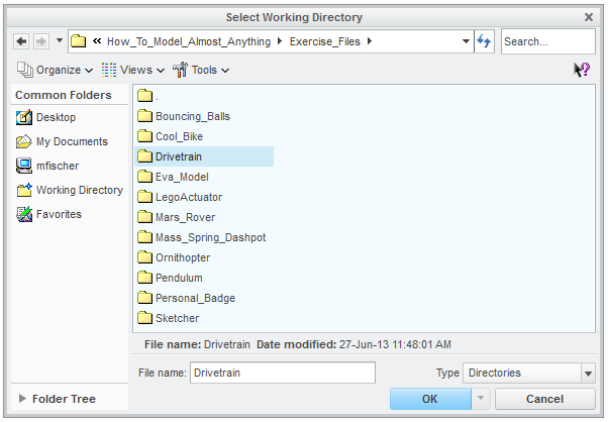
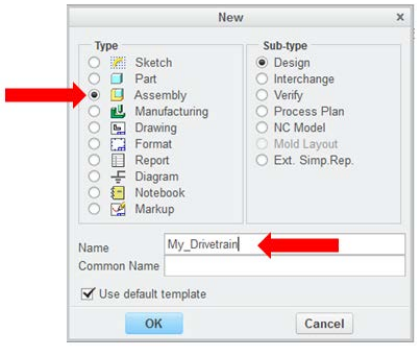
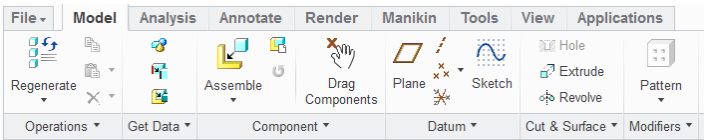


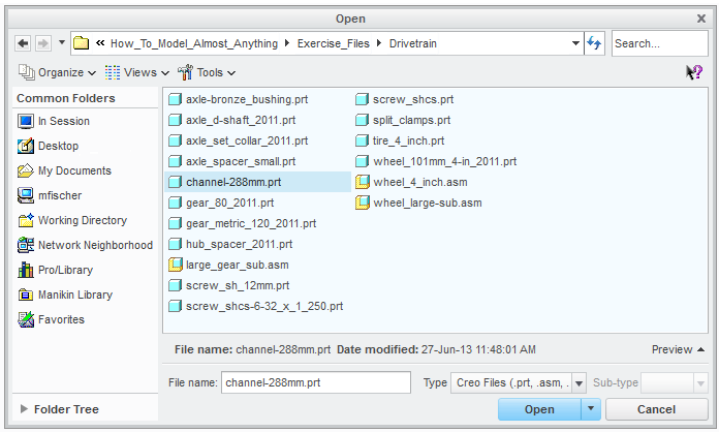
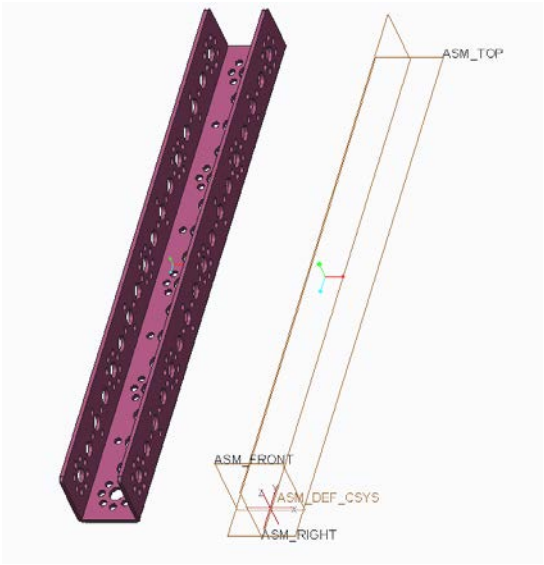
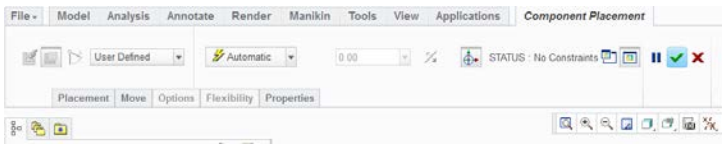
Therefore, let's review a model plan before we get started.

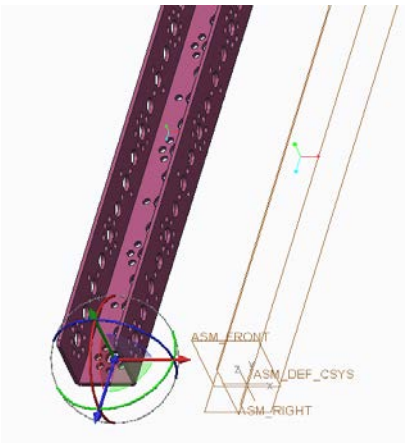
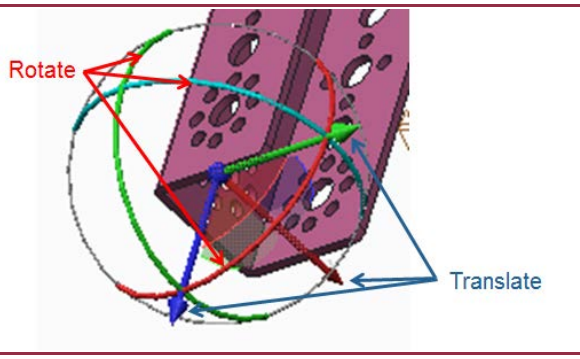
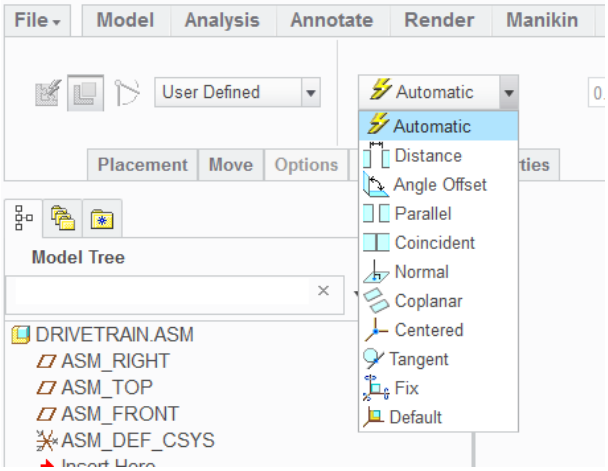
DRIVETRAIN ASSEMBLY MODEL PLAN		
Part	**Constraint**	**Notes**
C-Channel 	Default	Assemble the channel at the origin using a default constraint.
Bushing 	Two coincident constraints Align the axes and then align the flat faces	Add a bronze bushing to the channel using two coincident constraints.
5 more bushings 	Repeat the constraints for all 5 additional bushings.	Add 5 more bushings to the assembly.

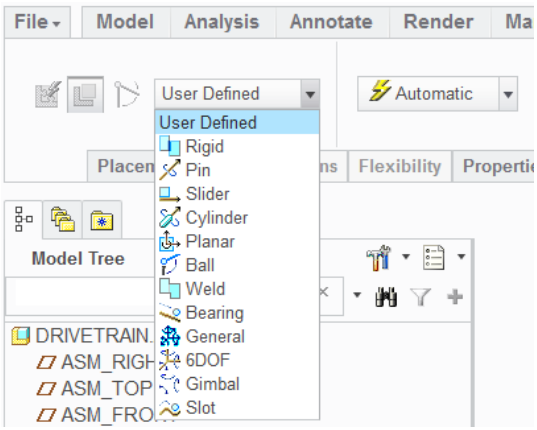
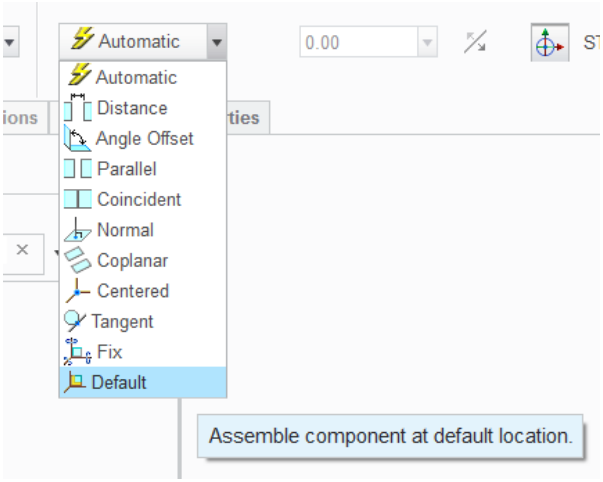
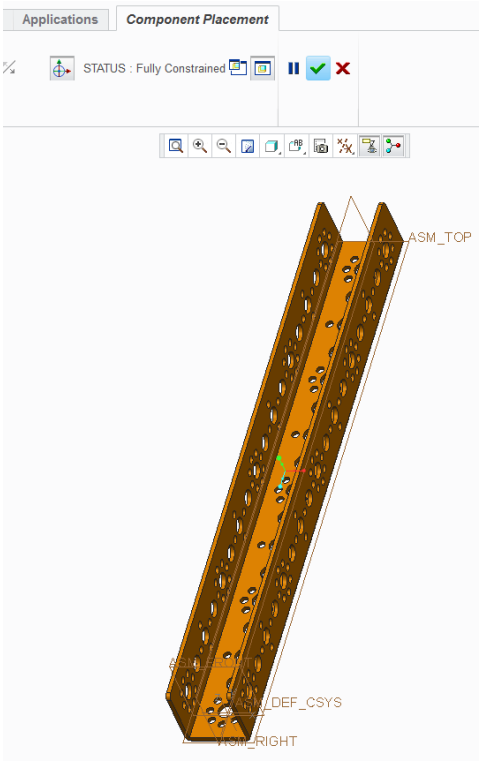
Large gear sub assembly 	Kinematic Pin constraint	Add a large gear sub-assembly using a pin constraint so that it can spin.
Large wheel sub assembly 	Kinematic Pin constraint	Add a large wheel sub-assembly using a pin constraint so that it can spin.
Second large wheel subassembly 	Kinematic Pin constraint	Add another large wheel sub-assembly using a pin constraint so that it can spin.
Gear pair 	Create two gear pairs	Finish the assembly model by creating two gear pairs to link the gears together.

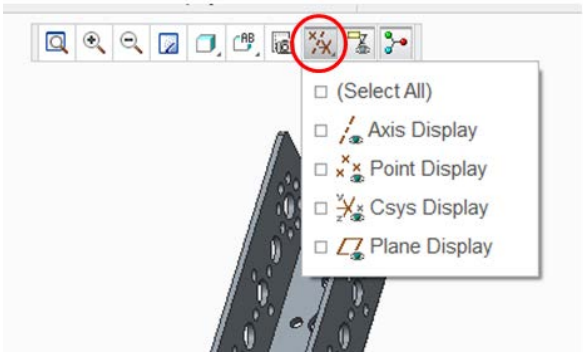
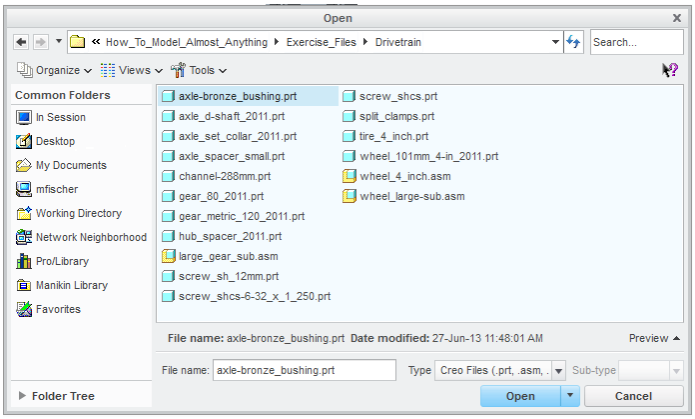
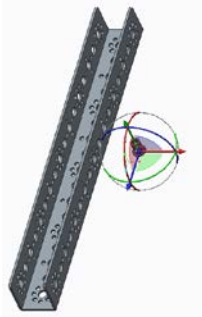
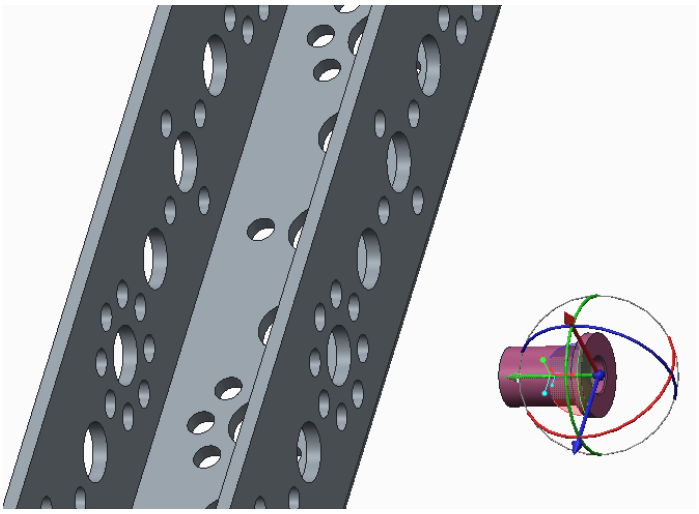
So now we have a good idea of how we are going to make this assembly model, let's get started.

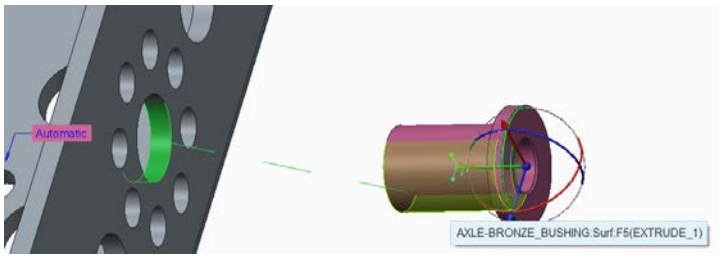
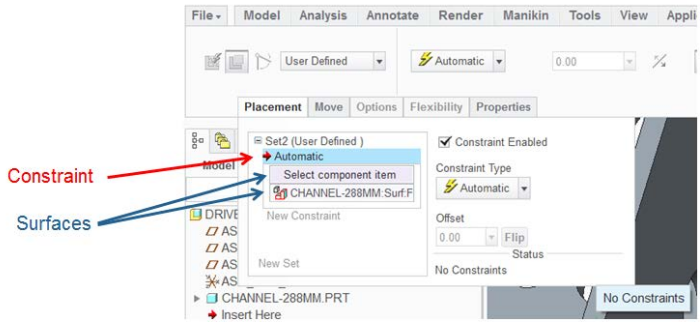
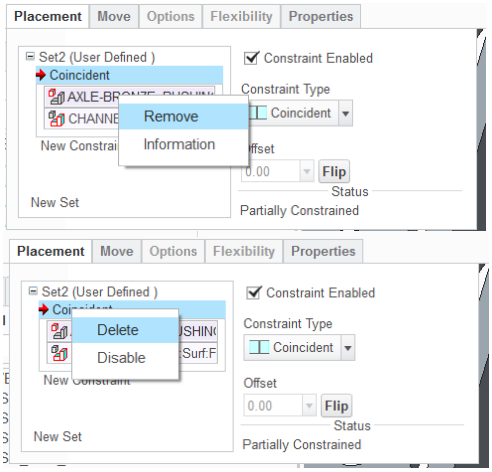
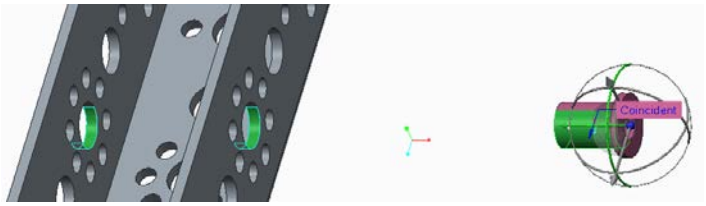
Open **Creo Parametric 2.0** and click the icon called **Select Working Directory** to set the folder in which you will be working.	
Navigate to the folder called "**Drivetrain**" and click **OK**.	
Now click on the **New** icon at the top left corner of the screen to create a new file. Make sure that you pick **Assembly** and then type in the name of the assembly as "**My_Drivetrain**" and click **OK**.	
You will notice that the tool bar at the top of the screen is different because we are in the 3D assembly room of Creo.	

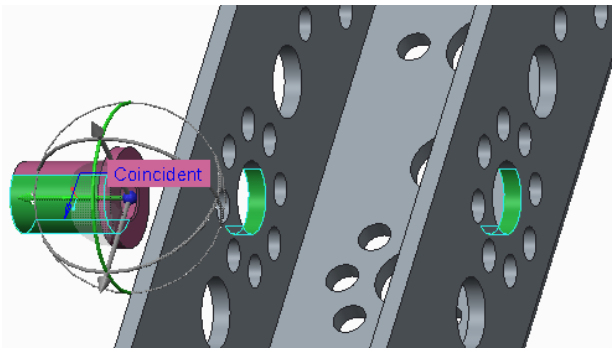
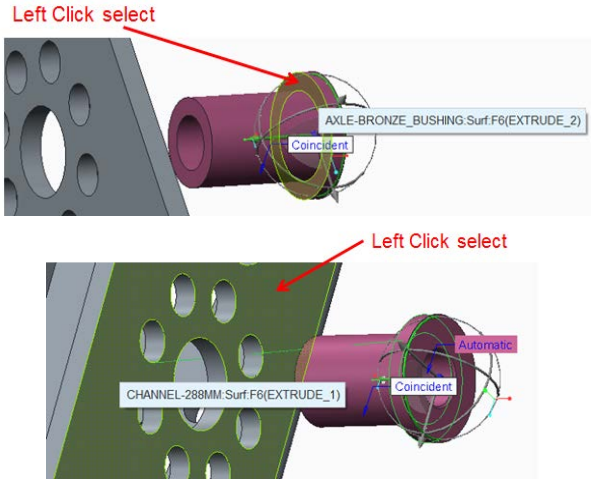
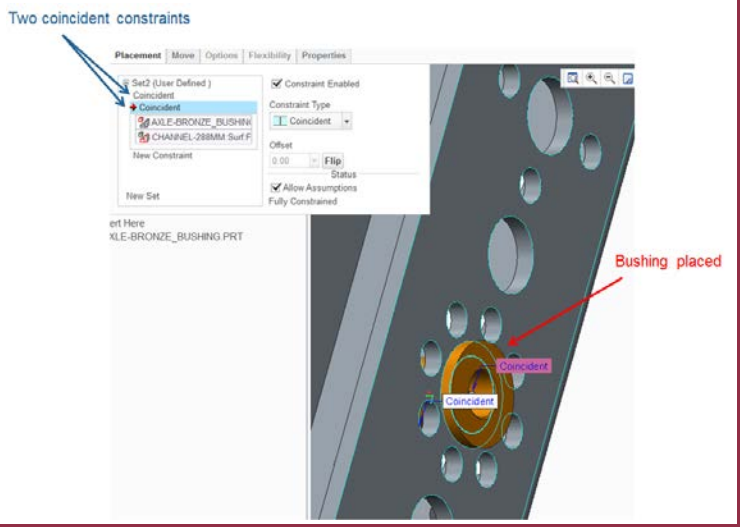
We will start by using the **Assemble** tool to bring in our first part. Click the **Assemble** tool and then select the part called: "**channel-288mm.prt**" and click **Open**.	
The channel part will appear in purple and will follow your cursor. Left-click anywhere to drop it on the screen.	
You will notice that there is a new dashboard in the upper part of your screen. It is the assembly dashboard and it has tools for helping you assemble your models.	

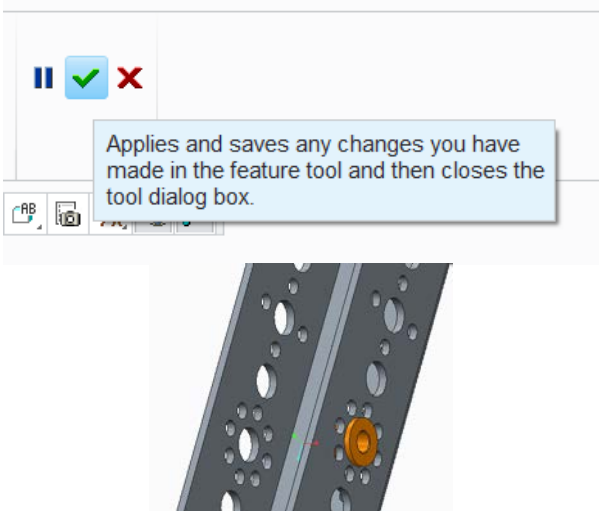
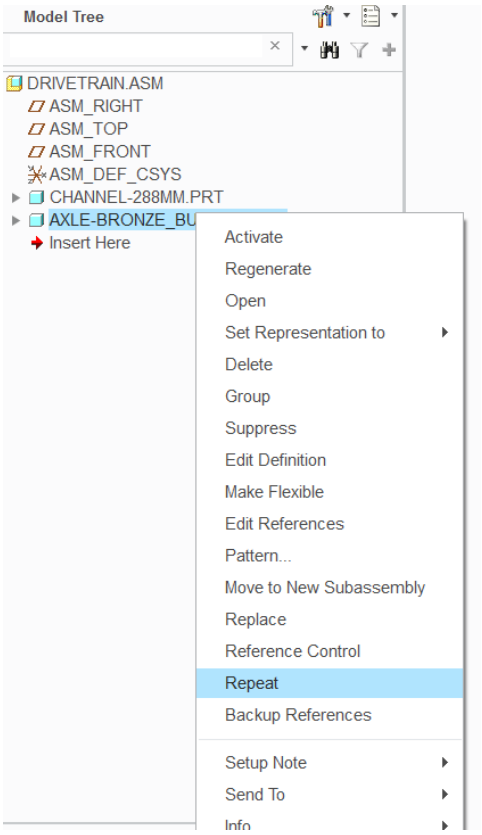
You should also notice that when you left-click to drop the channel, an orientation sphere appears at one end of the channel.	
The sphere allows us to orient just the current part by left clicking on the axis to translate or the circles to rotate.	
The assembly dashboard has two pull-down menus that allow us to pick the types of constraints we want to use to place our parts. The **Automatic** pull-down contains static constraints and is the default.	

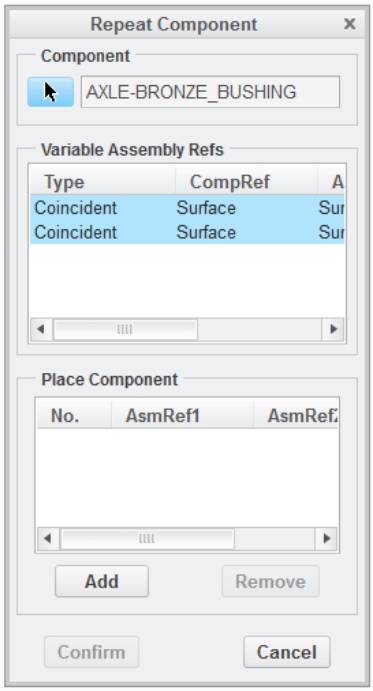
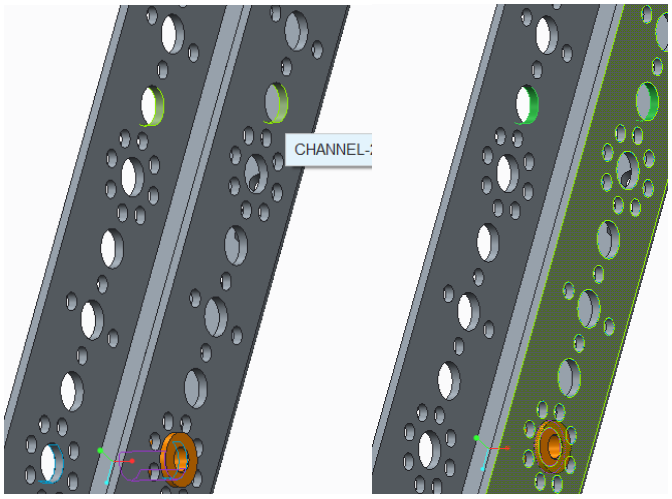
The **User Defined** pull-down contains kinematic constraints.	
Since the channel is the first part in this assembly model, we need to give it a constraint that fixes it to the origin. So use the **Automatic** pull-down to find the **Default** constraint and select it.	
Once the part has been constrained using the default constraint it is fully constrained and turns orange to indicate that it is fully constrained. Now click on the green check mark in the assembly dashboard to finish this part placement.	

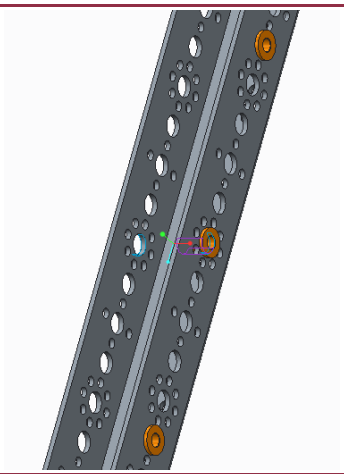
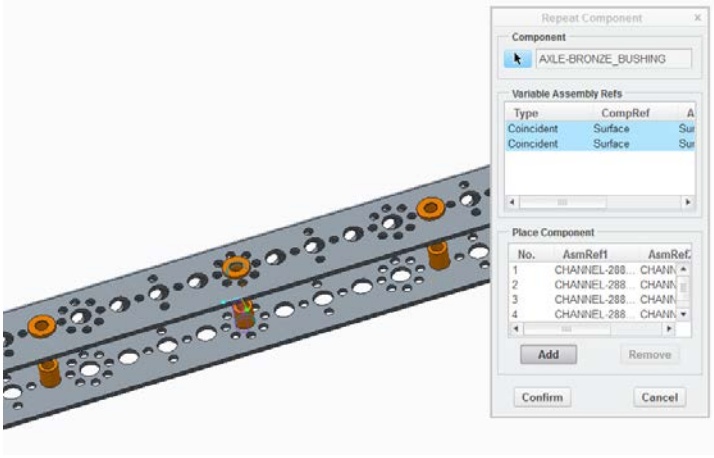
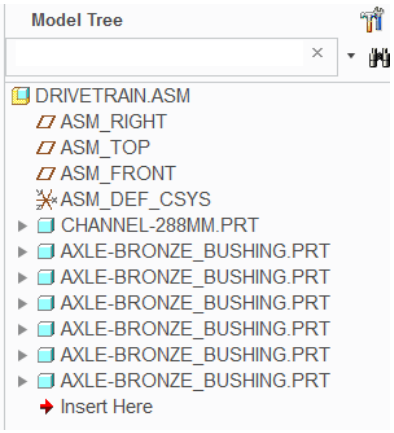
Now turn off the display of the datums by clicking on the datum display tool and making sure all of the boxes are unchecked.	
Click on the **Assemble** tool again to bring in the next part. Find the part called: "**Axle-Bronze_Bushing.prt**" and select it and then click **Open**.	
Once again it appears purple and follows your cursor. Left-click to drop it near the middle hole in the channel as shown.	
Now let's use the orientation sphere to orient the bronze bushing in the orientation that matches how we want to insert it into the middle hole.	

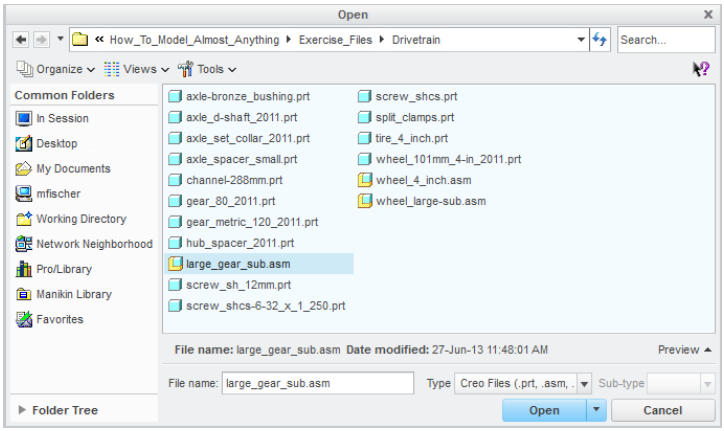
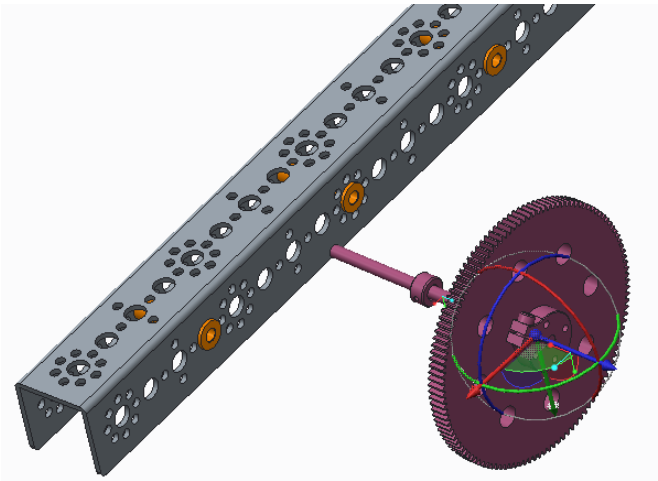
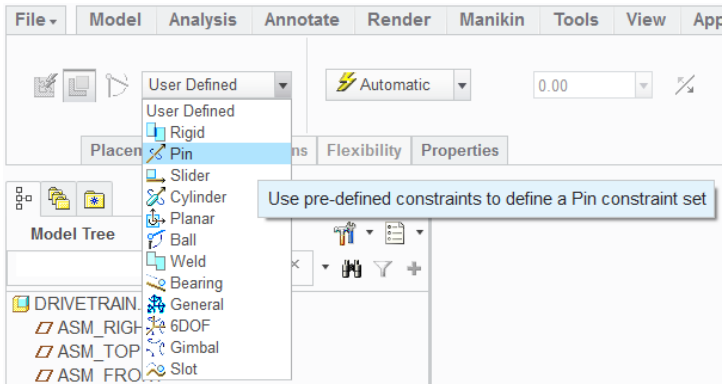
Now this bronze bushing should move and so we will use static constraints which are the default type of constraints in **Creo**. Start applying constraints by left clicking on the cylindrical surface of the bushing and then the cylindrical inside of the middle hole.	
Note: It is important during assembly to keep track of every click you make. Creo uses these left-click to determine what types of constraints you want to use. The **Placement** tab in the dashboard shows you what surfaces you have selected and what constraints you are placing on your parts.	
You can always change the constraints and surfaces selected by right clicking and choosing either **Remove** to change a surface selection or **Delete** to change a constraint type.	
So once you have selected the two cylindrical surfaces, **Creo** automatically places a coincident constraint on the bushing that makes its axis and the holes axis coincident.	

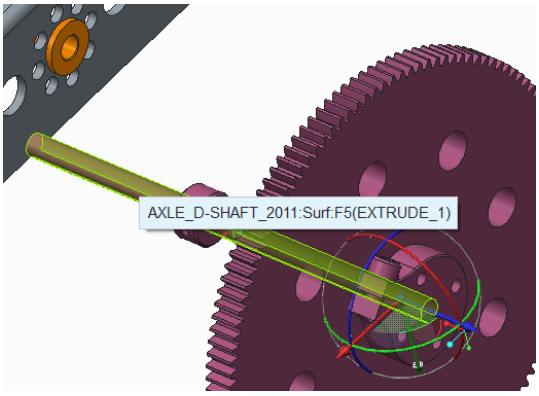
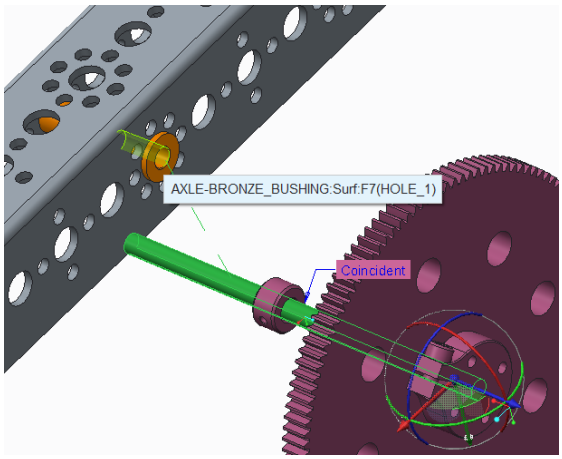
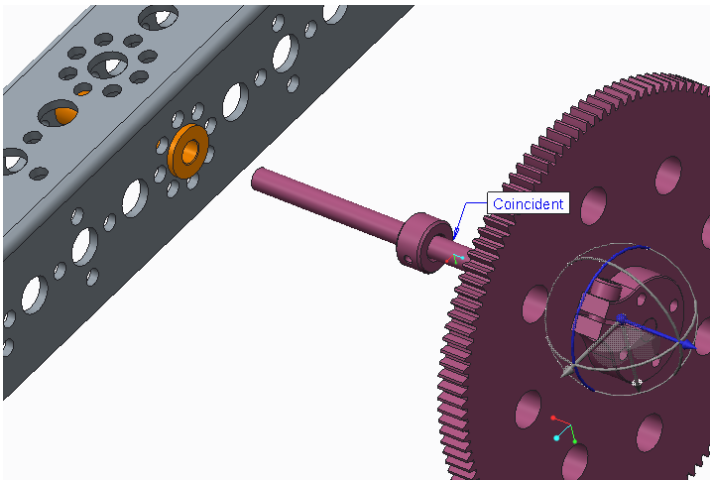
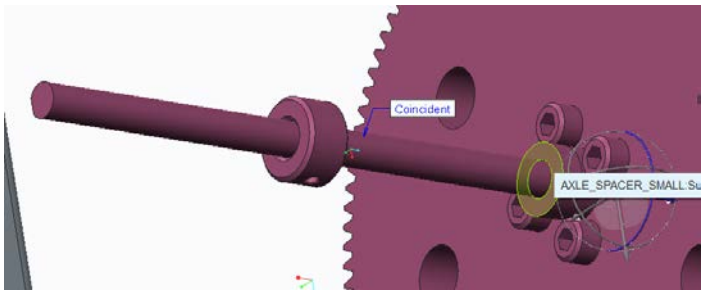
This means that the bushing can now only move along the axis of the hole.	
The next constraint we need defines where along the axis the bushing should be placed. So left-click on the flat underside of the bushing's head and the flat side of the channel.	
This will place the bushing where it should be placed.	

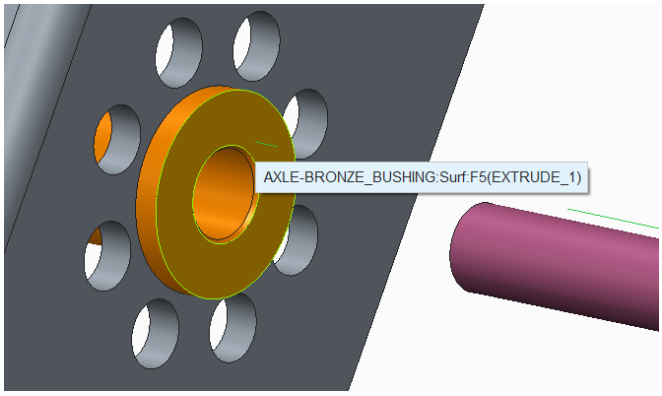
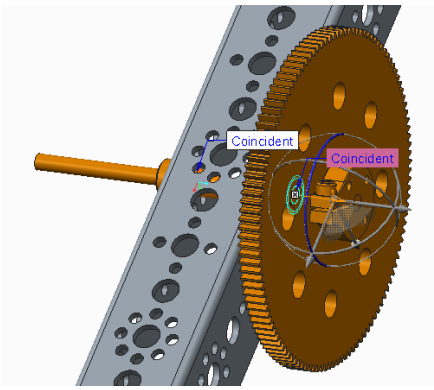
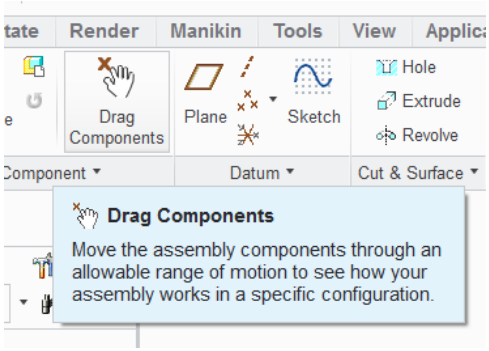
Click the green check mark in the assembly dashboard to finish this parts placement.	
We want to place 5 more of these bushings in the channel so we will use the **Repeat** Tool. Right-click on the **Bushing** in the model tree and select **Repeat**.	

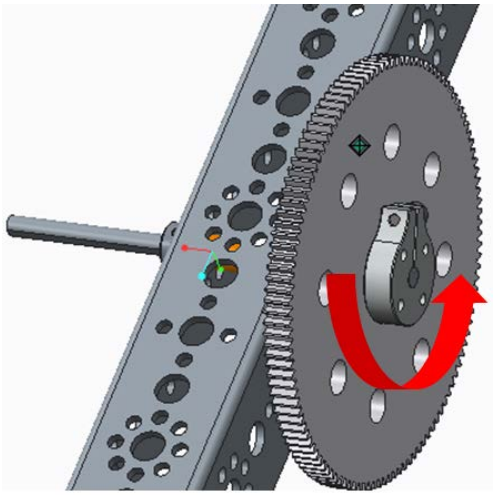
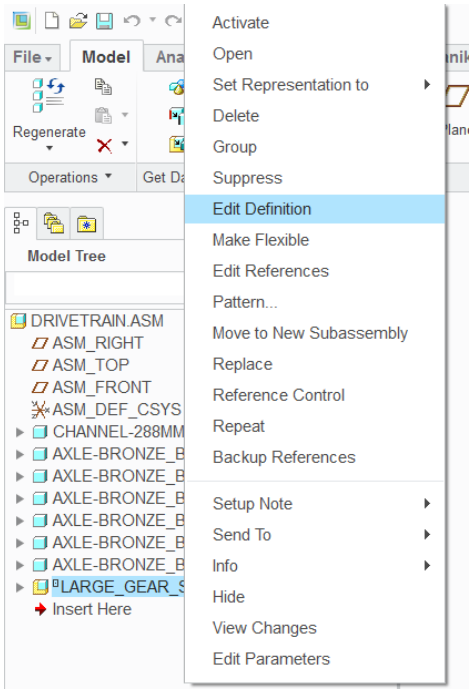
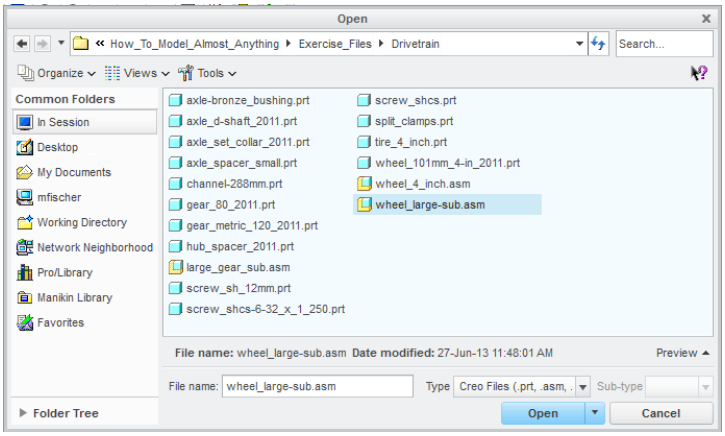
A dialog box will appear. Select the two coincident constraints by left clicking on them and then click **Add**. You will now need to click only the surfaces of the channel where you want to place the bushings.	
So to start, left-click on the hole inner surface five holes to the right of the existing bushing. Then click on the face of the channel and a bushing will appear.	

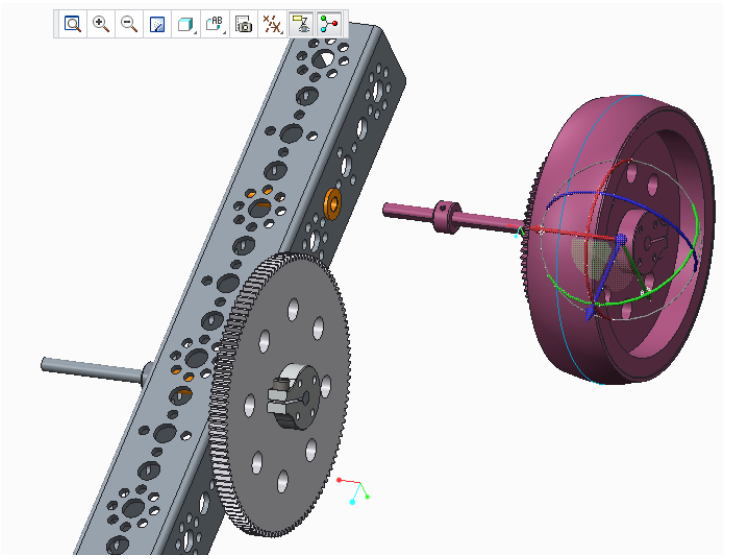
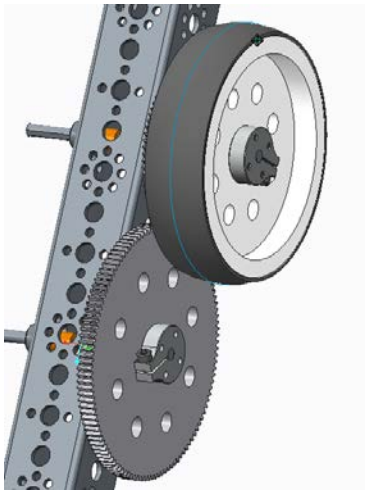
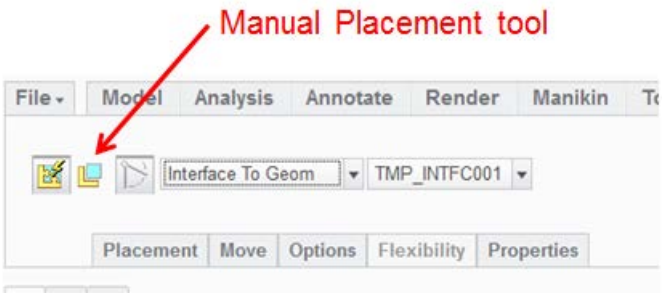
Do the same thing on the other side of the first bushing.	
Now turn the channel over and insert bushing in the holes that correspond to the inserted bushings and then click **Confirm**.	
Notice that you can track your progress in the **Model Tree** at the left of the screen.	

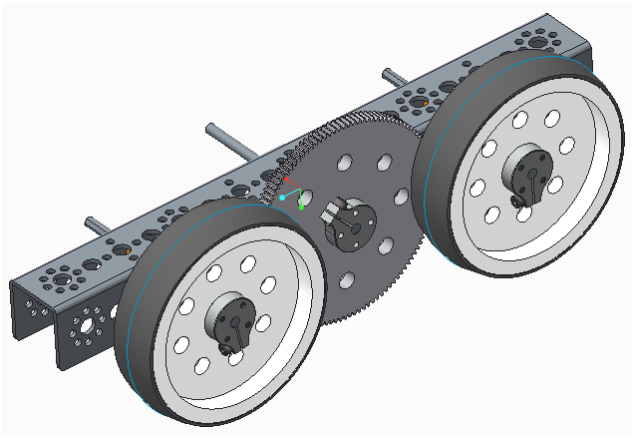
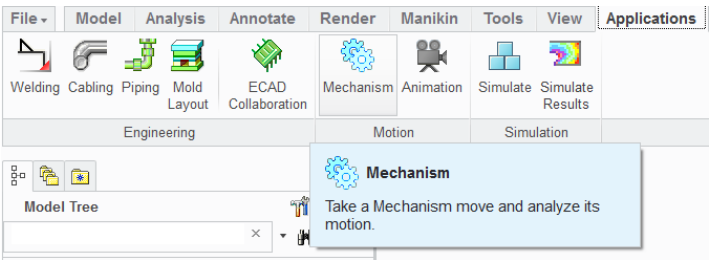
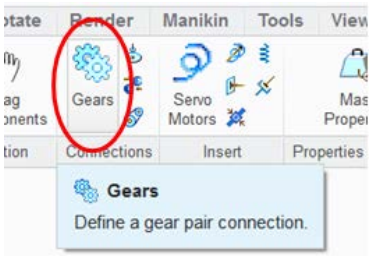
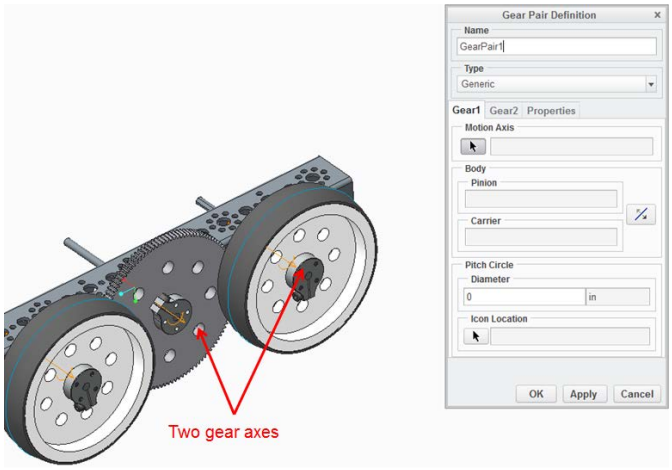
Click the **Assemble** tool again and select the part called: "**Large_gear_sub.asm**" and then click **Open**.	
Left-click to drop the gear and then use the orientation sphere to orient it as shown.	
This time we want the gear to rotate so pull-down the **User Defined** menu and select the constraint called "**Pin**".	

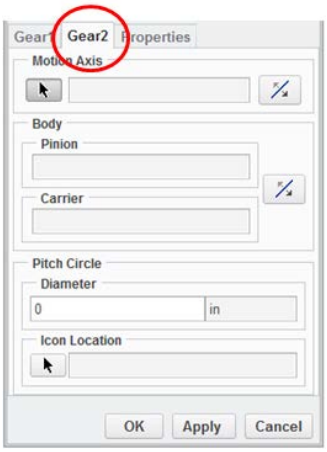
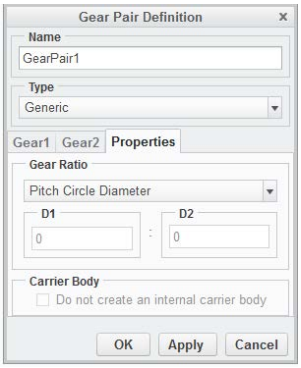
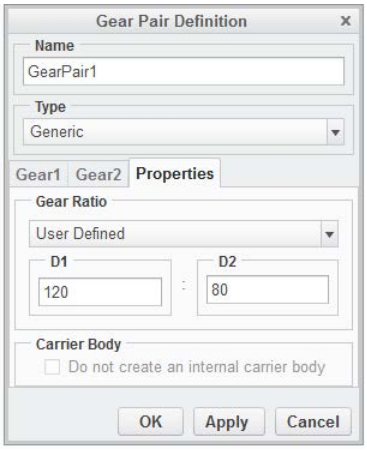
Then left-click on the shaft of the gear as shown. Make sure you get the whole shaft and not just an edge.	
Next left-click to select the inside surface of the bushing hole as shown.	
Once again the axes are aligned but we need to identify where the gear must be placed along that axis.	
Rotate the entire model using the middle mouse button (hold it down and move the mouse) and then left-click on the flat shoulder of the gear as shown.	

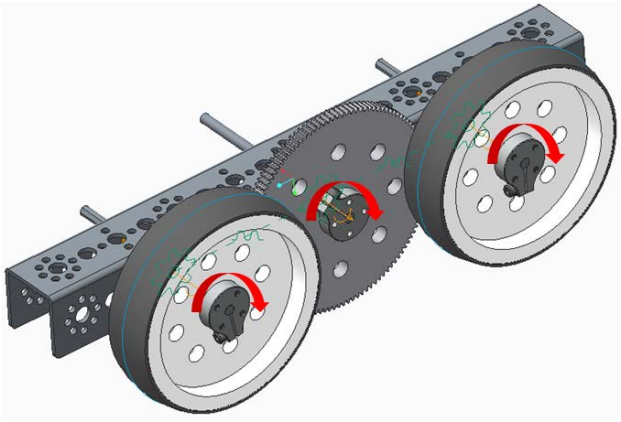
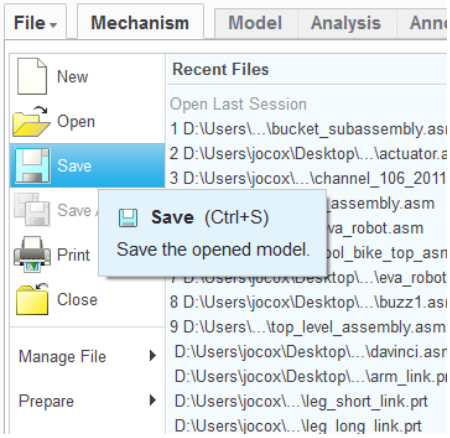
Now left-click to select the top flat face of the bushing. This will bring these two flat surfaces together.	
The gear is now placed so select the green check mark in the assembly dashboard to complete the placement of the gear.	
Now check to make sure your gear turns by clicking on the **Drag Components** tool in the upper menu and left clicking somewhere on the gear.	

The gear should now turn as you move your cursor. Finish testing the gear movement by clicking **Close**.	
Sometimes you make mistakes in setting up the constraints for each of your parts and you need to go back and edit the constraints. To edit constraints, find the part that needs to be edited in the **Model Tree** on the left. Right-click and select **Edit Definition**. You can then go to the **Placement** tab to edit the constraints.	
Click on the **Assemble** tool again and select the part called: "**Wheel_Large-Sub.asm**" and **Open** it.	

Left-click to place it as shown and then apply a pin constraint to it just like you did with the large gear.	
Test it to make sure it rotates correctly once you have finished using the **Drag Components** tool.	
When you assemble the same part a second time, **Creo** assumes you want to do it exactly the same way and so it defaults to an automatic placement that allows you to just select the surfaces on the channel and bushings without having to select the surfaces on the wheel. You can do that if you like or you can turn the automatic placement tool off and do it the same way you did it before by clicking on the manual tool	

Once you are done, check to make sure the two wheels and the gear all turn. You will notice that the wheels will not turn together. To make that work we need to define gear pairs.	
To create gear pairs we need to go into the **Mechanism Application**. So click on the **Application** tab and then select **Mechanism**.	
Now click on the **Gears** tool to define a gear pair.	
To define a gear pair we need to identify two gears by selecting their axes.	

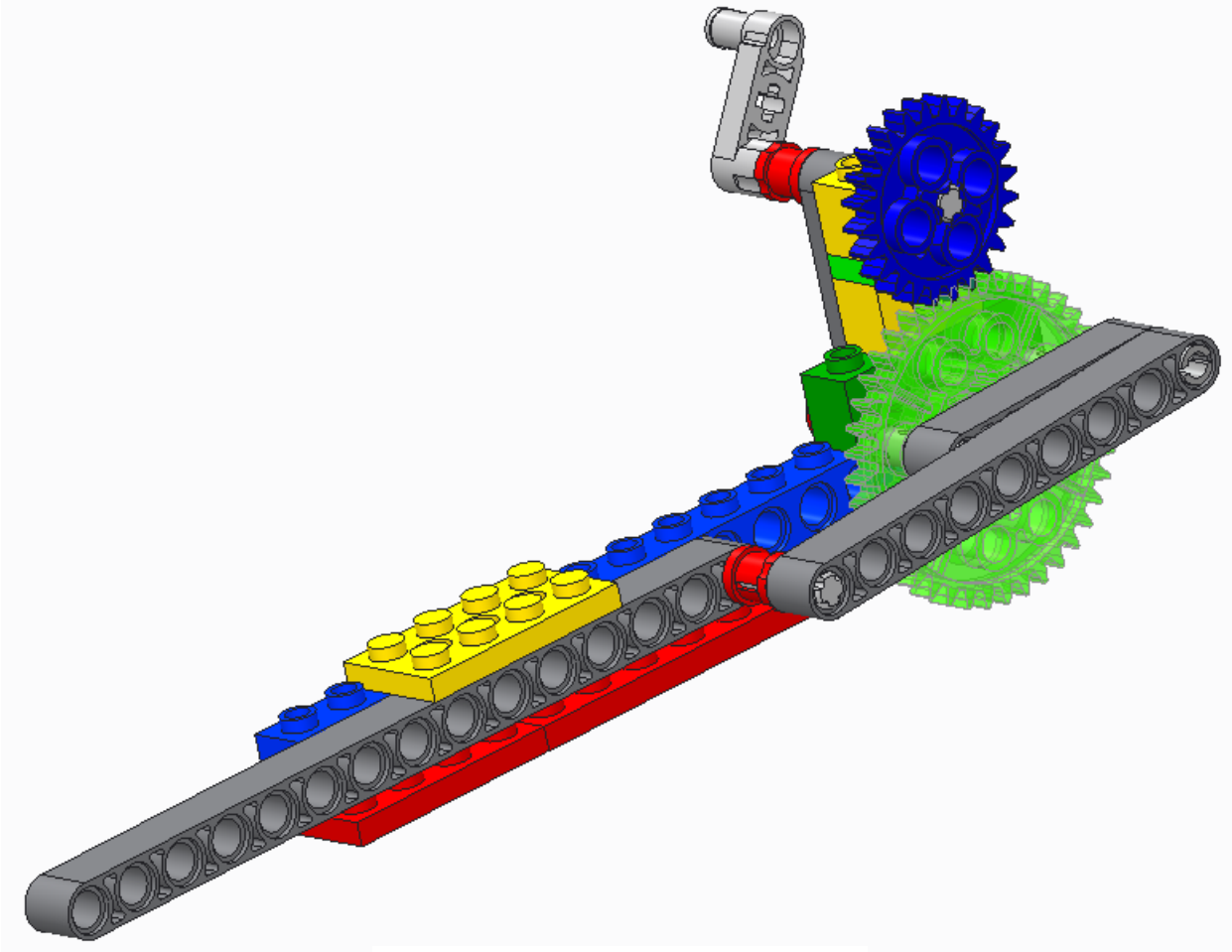
Left-click on the middle gear axis first and then click on the **Gear2** tab and then click on the right gear's axis	
Now click on the **Properties** tab in the dialog box.	
Click on the pull-down menu called "**Pitch Circle Diameter**" and select **User Defined**. Then set the gear ratio based upon the number of teeth each gear has as shown. Click **OK**.	

Do the same thing to set up a gear pair between the large gear and the left gear. Then make sure that all the wheels and gears move together using the **Drag Components** tool.	
Now save your assembly model by clicking on the **File** and then **Save** tabs. Exit Creo.	

Congratulations, you have finished this exercise!

EXERCISE 9: LEGO MECHANISM

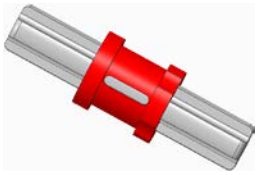
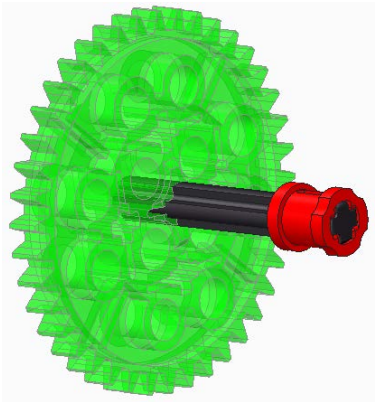
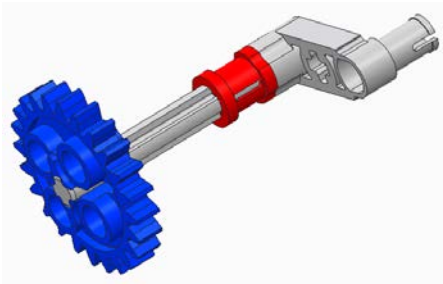
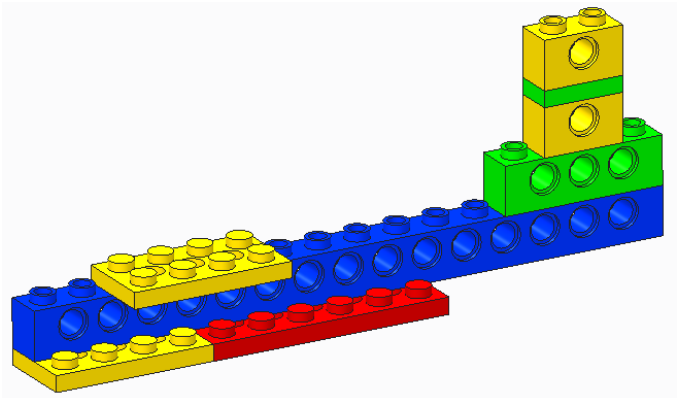
A slider-crank mechanism created from Lego parts is shown below. A kit of Lego parts was used to build this mechanism. Study this mechanism and the parts in the kit of parts and then design your own assembly model using parts from the Lego kit of parts.

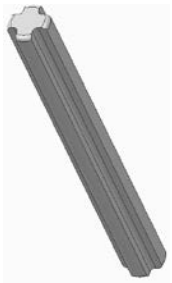
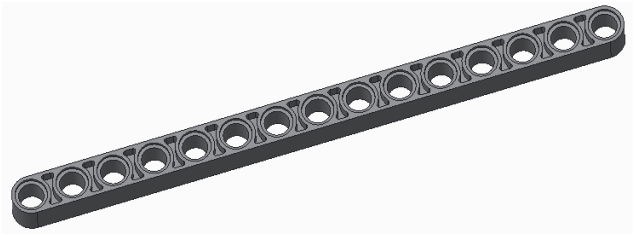


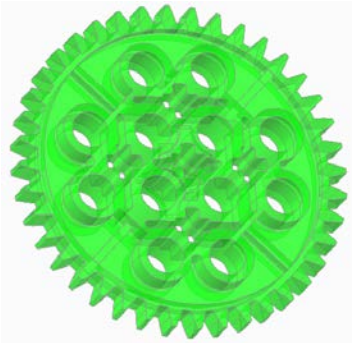
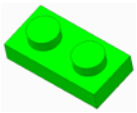
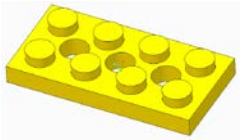
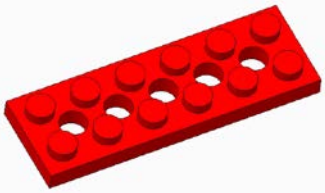
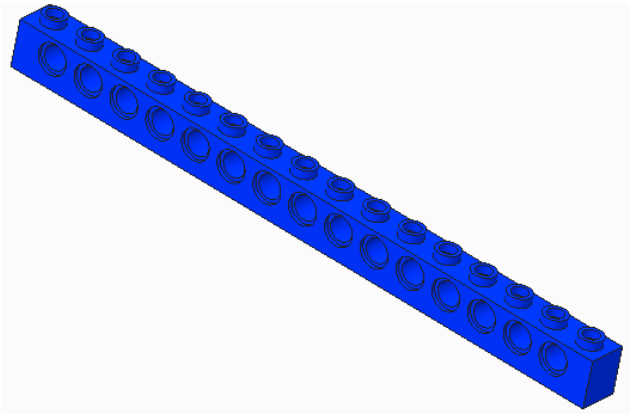
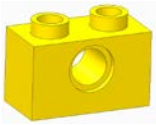
Actuator.asm

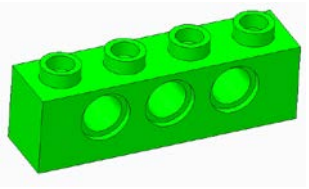
Create an assembly model plan and then build the mechanism using Creo. You can just rebuild the slider-crank mechanism shown here or you can design your own. Make it as simple or complex as you feel your skills will allow. One possibility is to add parts to this assembly building on what already exists.

You will find this model and all of the Lego parts in the Lego Actuator folder.

LEGO KIT OF PARTS	
axle.asm	
40t_axle.asm	
24t_axle.asm	
actuator_base.asm	
axle_3_2012.prt	

axle_4_2012.prt	
axle_5_2012.prt	
beam_15_2012.prt	
beam_5_2012.prt	
beam_9_2012.prt	
bushing_2012.prt	
conn_2012.prt	
conn_crank_2012.prt	

gear_24_2012.prt	
gear_40_2012.prt	
plate_2x1_2012.prt	
plate_4x2_2012.prt	
plate_6x2_2012.prt	
sbeam_16_2012.prt	
sbeam_2_2012.prt	

sbeam_4_2012.prt	

YOUR ASSEMBLY MODEL PLAN		
Part	**Constraint**	**Notes**

So now you have a good idea of how you are going to make this assembly model, you can get started.