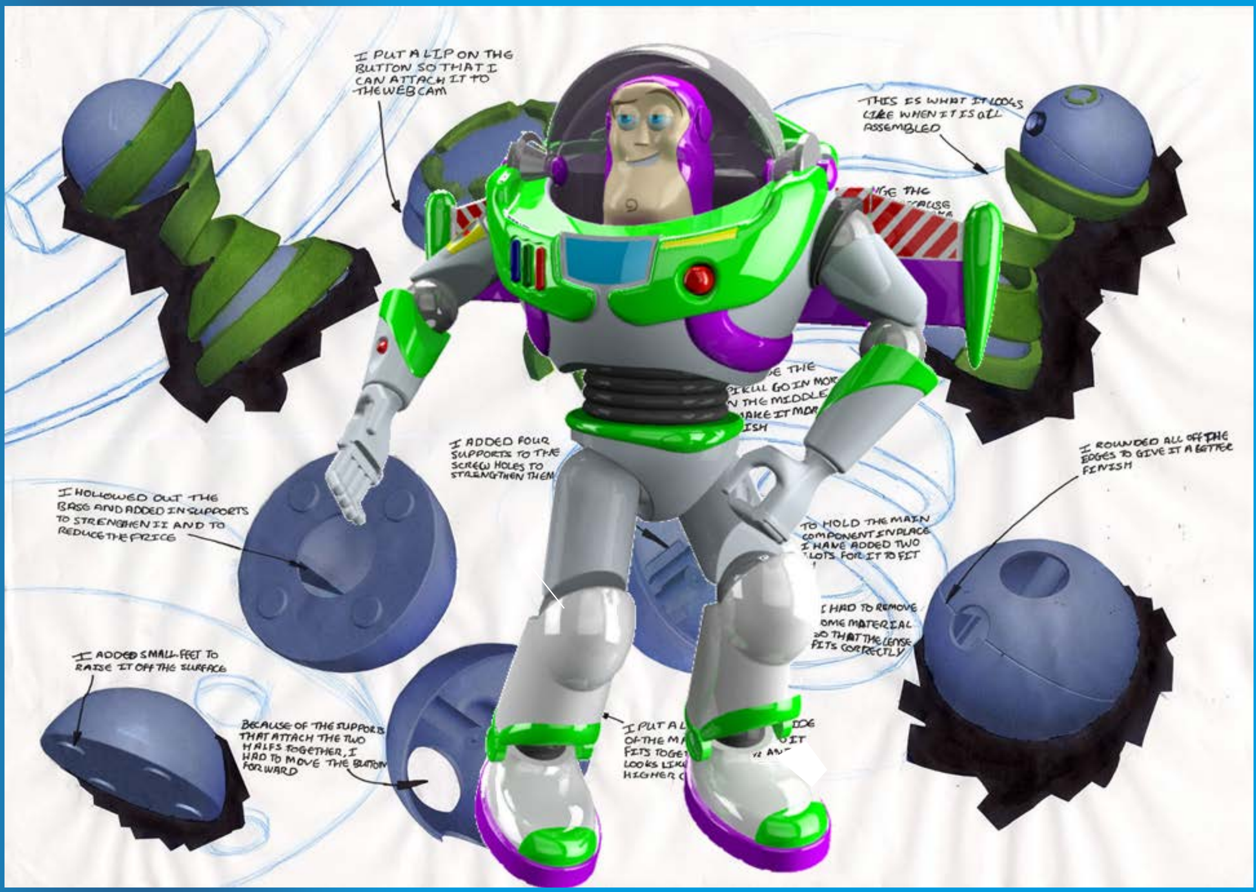
Phase

2

HOW TO DEVELOP A

PRODUCT



Detailed Design

PRODUCT DEVELOPMENT MANUAL

How to Develop a Product

WRITTEN BY JORDAN COX

NOTICE OR RIGHTS

TRADEMARKS

ACKNOWLEDGEMENTS

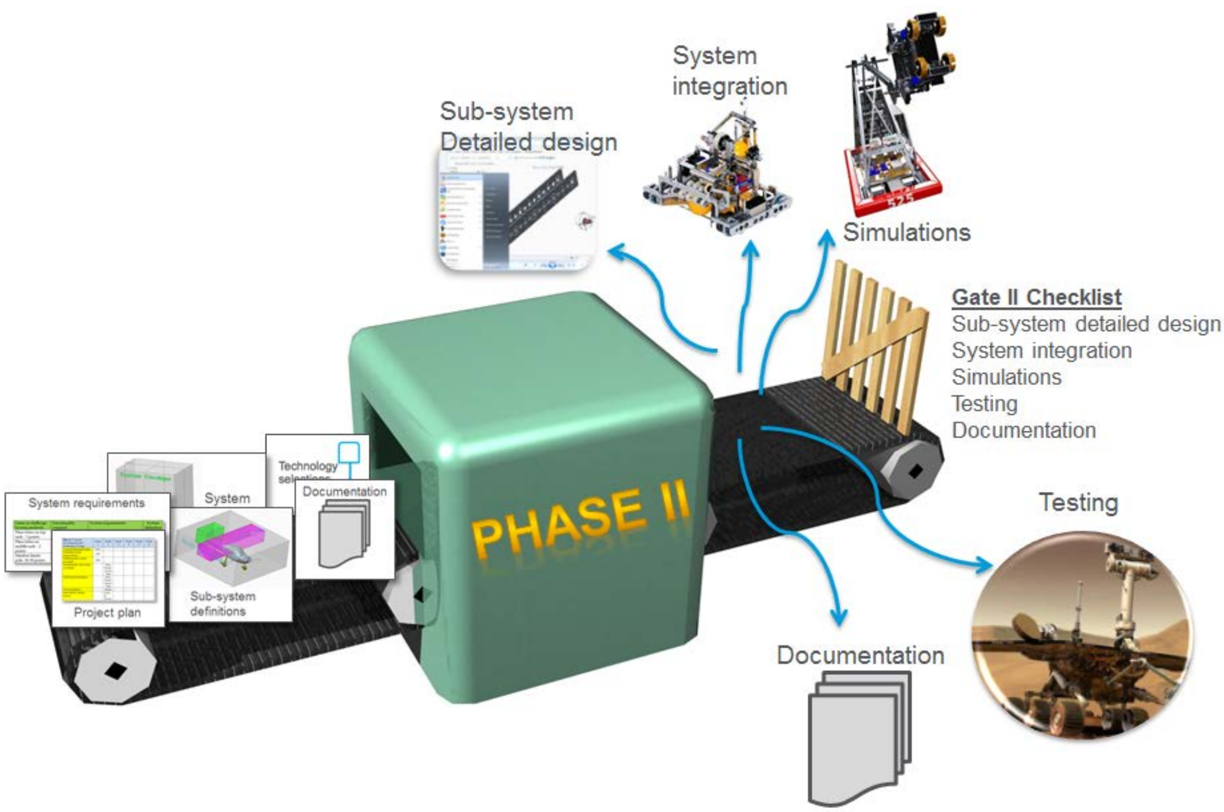
Scott Morris, Mark Fischer, Adam Haas, Ayora Berry

Phase

2

Transforming a Preliminary Design into a Detailed Design

Phase II: Detailed Design, Prototype & Test



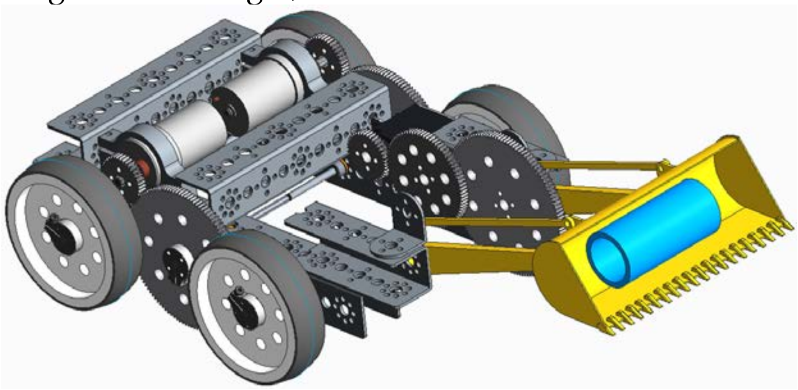
Phase 2 of the product development process focuses on converting the six deliverables of Phase 1 into a detailed design of the product. There can be many artifacts that are produced during this phase. We will focus on five:

1. Sub-system detailed designs
2. An integrated system design
3. Simulations of the product

4. Tests to validate the simulations
5. Documentation

This chapter presents what each of these artifacts are and how they are created. Exercises are recommended which will help with understanding the concepts. At the end of this phase, there will be a design review or gate check, just like in Phase 1, where each of the artifacts will be reviewed.

Each of the exercises in this chapter will be focused on an example product which is a small robot. This example product will provide opportunities to start with a system envelope model and define sub-system models and integrate them into a total product model. It will also provide opportunities to run simulations to insure that the product will operate as designed. So to begin, let's examine the small robot.

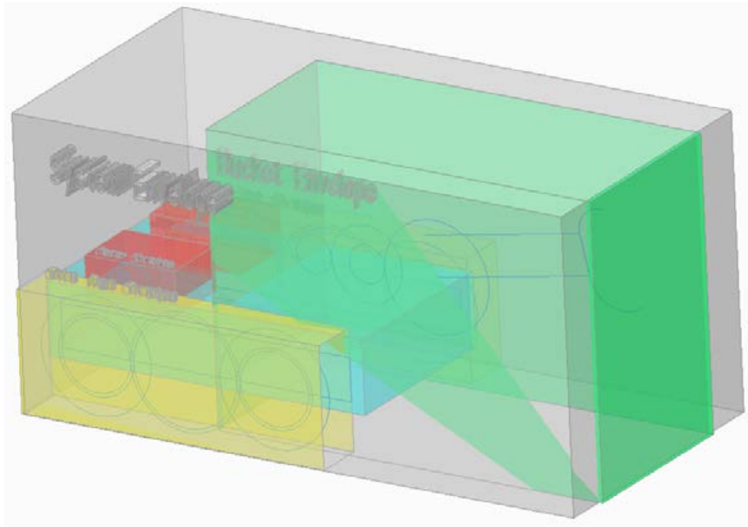


This robot has a drive train powered by two motors and a bucket lift powered by a single motor. Both the drive train and the bucket lift are powered through gear trains thus providing either mechanical advantage or speed.

Exercise 6: If you have not already explored the PTC-Robot, Refer to the Appendix for the file *PTC-Robot Explore* to explore the PTC robot in more detail.

During this section of the workshop, we will build a model of the PTC robot as an example of how to convert a system envelope model into a detailed design model. The system envelope model for the PTC robot was created previously and exists in the workshop folder. It identifies each of the subsystems and their approximate weight and location in the system model. This would be the system envelope model created during the first phase of the product development process. So At this point we want to use it to guide the development of the detailed subsystem models that can be integrated into a total detailed system model. Let's examine the system envelope model since it will be the framework we will use to create the sub-system designs.

Exercise 7: Refer to the Appendix for the file *PTC-Robot Envelope Explore* to explore the system envelope file.

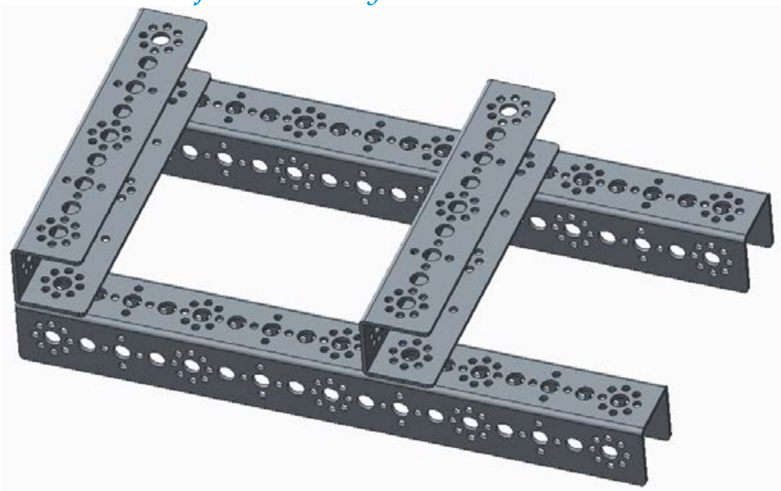


Now that you are familiar with the PTC robot and the system envelope model for the PTC robot, you are ready to begin designing the sub-systems and integrating them into an overall system design. To begin with, it is important to create a system level assembly file that will be the final integrated system detailed design model. It will be empty at the beginning but as the subsystems are designed, they can be added to the system model. In order to insure that all the subsystem designs are designed with respect to the overall system, we create a skeleton file (reference file) in the system model and place all of the envelopes in it. These then become the references that all the subsystem models refer to for orientation. You will see this in the exercises that follow.

Exercise 8: Refer to the Appendix to find the file *Creating the System Model* and complete the exercises.

DESIGNING THE SUB-SYSTEMS

Now that an empty system model has been created with the envelopes as references (Skeleton files), we can begin building each of the subsystem models. Usually, you will want to sub-divide your team into sub-groups to develop the detailed designs of the subsystems in parallel. This is the reason you create a system envelope model so that the team members can sub-divide and work in parallel while still maintaining a system perspective. However, there are some subsystems that may need to be created before other subsystems. This is the case with the PTC robot. The structure sub-system needs to be created first so that all of the other systems can attach to it.

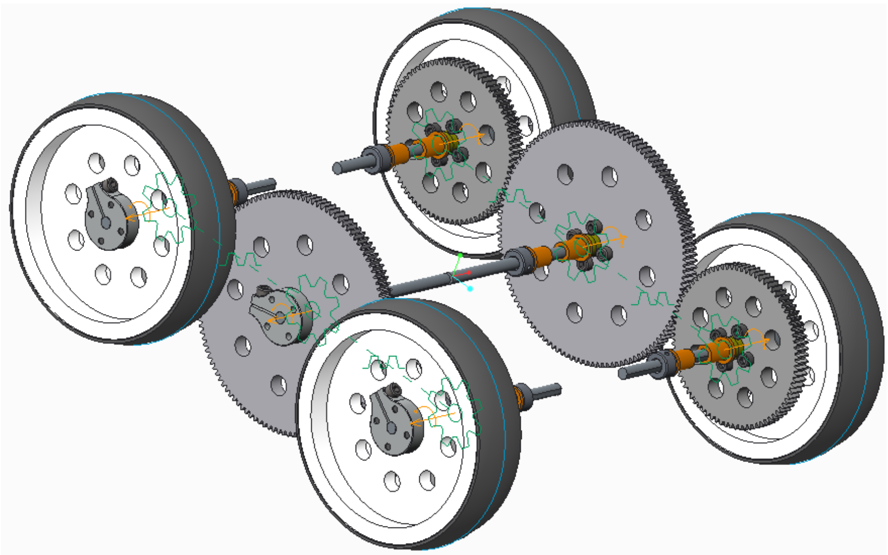


Exercise 9: Refer to the Appendix for the file *Designing the Structure Subsystem* and follow the instructions to design the structure of the PTC robot.

Since it needs to be defined first, we are going to build it within the system model. This means that while the system model is open, we can create a subassembly model and assemble it to the system model. This is one method of creating subassemblies. Another method is to open a separate subassembly model and create the subassembly independent of the system model. This is the method that most designs will follow where independent groups develop their subsystem designs in parallel and bring them into the system model at the end. We will show you both methods.

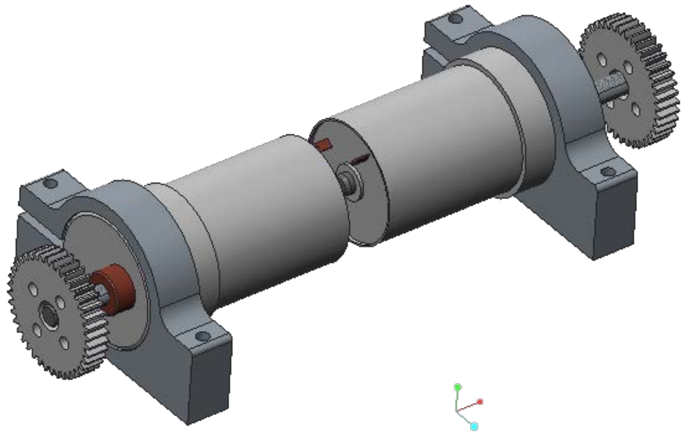
Exercise 10: Refer to the Appendix for the file *Designing the Drivetrain* and follow the instructions.

Now that the structure subassembly has been created let's focused on how to create the other subsystems in parallel. Let's begin by building the drive train subsystem as a separate and independent model.



Once the drive train is designed, we can focus on designing the power subsystem. This subsystem provides the power to the drive train to make the robot move. Normally these subsystems would be designed in parallel so that multiple teams can work at the same time and reduce the overall time to design and build your models.

Exercise 11: Refer to the Appendix for the file *Power Subsystem Design* found in the Phase 2 folder.



Now that the power subsystem has been designed, the only subsystem left to design is the bucket subsystem. Using your skills that you have learned in the past few exercises, define the bucket subsystem on your own.

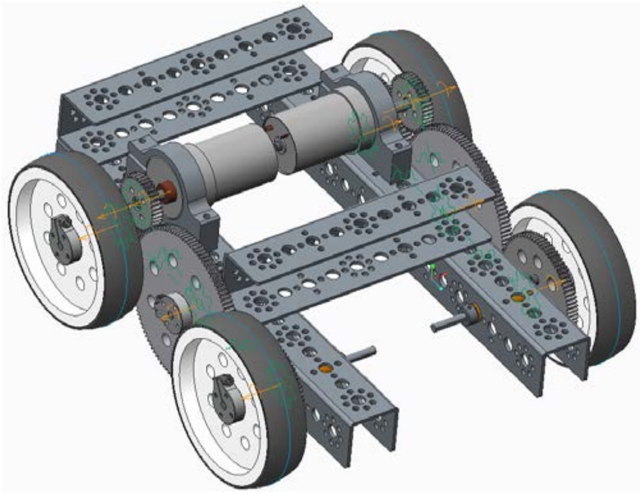
Optional Exercise 12: Create a subassembly file and create a skeleton file with the appropriate reference geometry. Then create the bucket subsystem and make sure all of the kinematics work.

Your bucket design may be different than what is currently on the PTC Robot and that is fine. For the rest of the exercises we will use the PTC robot without the bucket so that you can complete the exercise of creating the bucket on your own.

Integrating the Sub-Systems into a Complete System Design

Once the subsystem designs are completed, they can be integrated into the overall system model all at once or one-by-one as they are completed. We will bring the drive train and power subsystems into the overall system model since the structure subsystem has already been integrated into the system model.

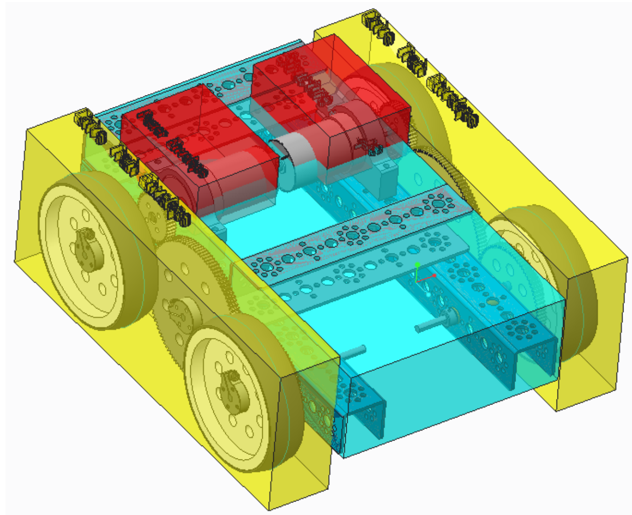
Exercise 13: Refer to the Appendix for the file *Integrating the System* in the phase 2 folder and follow the instructions to create a basic robot.

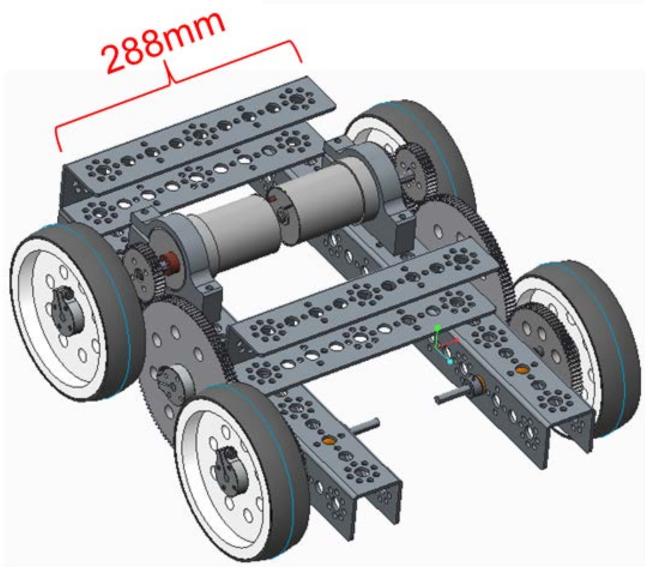


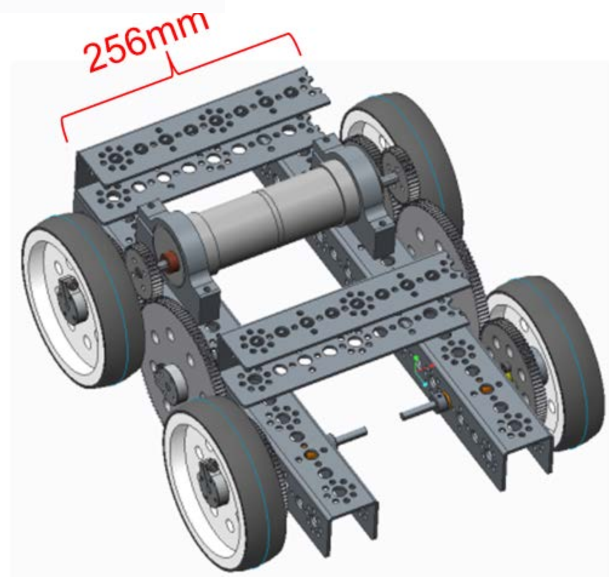
Because each of the subsystems were designed using a top-down strategy that included a system envelope level model that could be used for references and each subsystem assembly file had skeleton files, it is very easy to integrate the subsystems into a complete system model.

This also facilitates changes to the system design. For example, if the robot needed to be thinner, we could make a simple change to the structure subassembly within the system model and then regenerate all of the models. The top-down approach keeps all of the references intact so that changes can be made quickly. In fact let's do it.

Exercise 14: Refer to the Appendix for the file *Changing the System Model* and follow the directions to execute a design change to your robot model.







Top-down design strategies allow rapid changes to your system model while keeping all the relationships intact. Now that we have a basic robot model, let's examine how we can simulate its behavior using Creo Parametric.

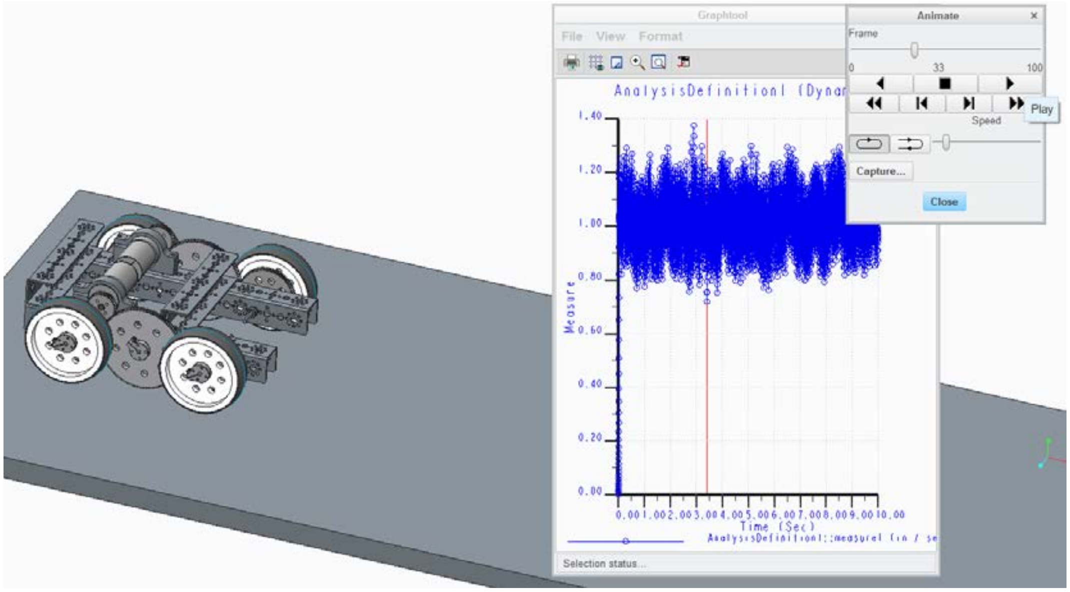
Simulating the Real World

A very important purpose of your system model is for developing predictions of how your designed product will behave in the real world before it is manufactured. Creo Parametric has many tools integrated into it for doing simulations. There are also 3rd party tools that can be connected to Creo Parametric to provide additional simulation capabilities. The simulations you will need to perform will depend upon the function of your product. For FIRST robotics, the most common simulations are kinematic and mechanism simulations. For RWDC the most common simulations are flow analysis

and stress analysis. However, there are many other types of simulations. Electrical power, heat transfer, electromagnetic, and flutter are examples of other types of analyses that can be used to predict product behavior.
There are several analyses that can be done very quickly once you have a system level detailed model of your product. These involve predicting the physical characteristics of your overall product; length, weight, volume, center of gravity, etc.

Exercise 15: Refer to the Appendix for the file *Simulating the real* world and complete the instructions to provide predictions of the robot's behavior.

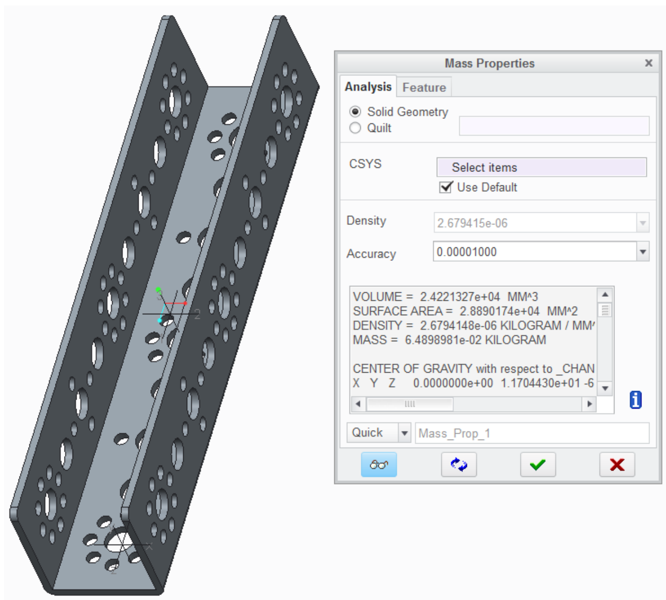
More complex characteristics of your product, such as kinematic motion, mechanical advantage, speed, power, etc. require the use of more sophisticated tools. Flow analysis is a very complex phenomenon and requires the use of a 3rd party tool to calculate.
Since we now have a basic system level model of the PTC robot, let's do some predictions of how it will behave in the real world if we were to manufacture it.



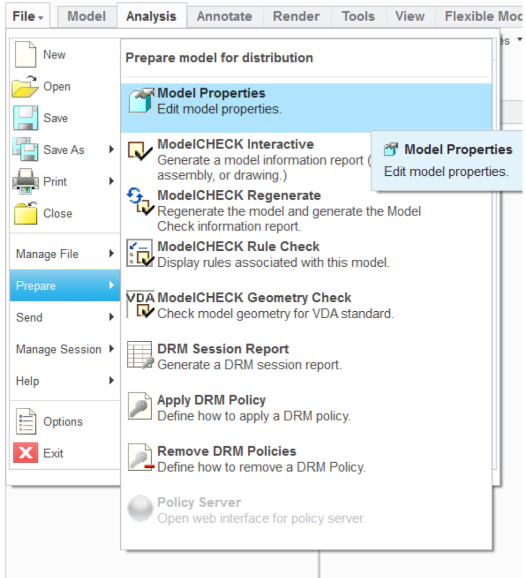
The exercise you just completed allowed you to experience simulating the real world with your robot model. But how do you know whether the calculations were correct? Just because your robot moved in the simulation doesn't mean that it is how it will move when you build it. It is important to test and validate the simulations that you do with your models to make sure that the simulations are correct.

TESTING YOUR DESIGNS

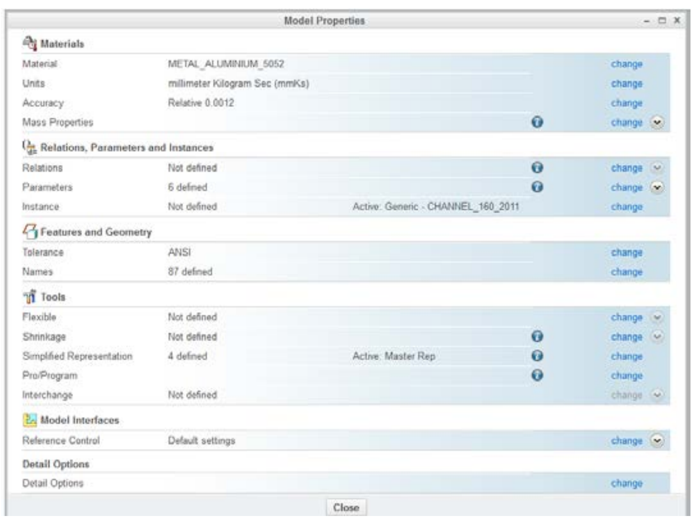
There are many different tests that you can perform to insure that your simulations and calculations are correct. Let's discuss a few. In the previous exercise, you calculated the weight of your robot model. That weight was calculated from the density of the materials and the volume of the parts. How could you check the calculation of the weight of your robot without building it? One possible way would be to check the weight of individual parts to make sure they are accurate. Possibly you could weigh a part that is made out of each of the different materials in your robot model and then check the part's weight against the calculations made by Creo to insure that the densities are all correct.



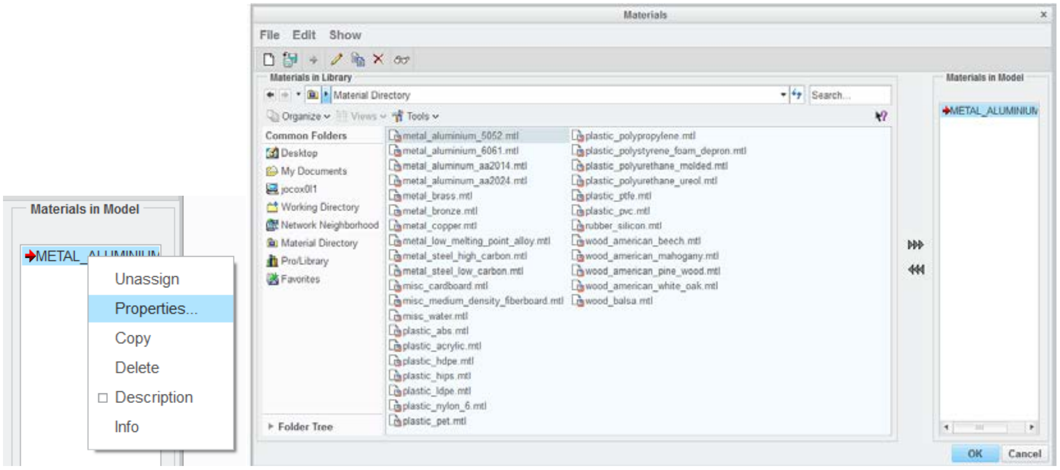
You could weigh the 160mm channel and then check the weight that Creo Calculates. If there are differences, you may need to change the density for the aluminum parts.



You can change the properties of a part by going to the **File** tab in the upper menu and then selecting **Prepare** and **Model Properties**.



You can then select the blue change on the right side of the Material entry. This will allow you to change the material values.



You will notice that there are many selections of predefined materials. If you right click on the selected material and select **Properties** you can see the density and other properties and change the values.

It is very good practice to test all of your material densities to make sure your models are correct.

Another test that is important to validate your simulation is to determine the right coefficient of friction. This is a special number that characterizes the friction between two objects, like between the wheels and the floor. If the coefficient is too high it means there is more friction than really exists. If it is too low, it means it is more slippery than it really is. You can imagine that the coefficients will change whenever the materials change and even sometimes if the humidity changes or there is more dirt or other types of conditions change. The amount of friction between the wheels and the floor will determine how quickly you can accelerate and decelerate.

So how would you test to determine the right coefficient of friction for your robot? Talk with your mentor or coach about ways to validate your calculations and simulations so that they provide useful information about the predicted behavior of your model.

Documenting your Designs

One of the most important parts of your product development is documenting the decisions that are made. Keep a notebook and document each of the system and subsystem design decisions that you make. You can certainly print the tables and pictures of the envelopes and paste them into your notebook. You should also be using PTC Windchill to store all of your models. This repository is part of your documentation.

Conducting a Design Review – Phase II Gate Check

Congratulations, you have completed the first two phases of the product development process. Now you need to do a gate check to make sure everything is ready to move on to the next phase in the product development process. A gate check is best conducted as a design review. Assemble a group of mentors and then go through each of the aspects of your design to date. This is your chance to present the detailed design of your system. Use all of the artifacts you have created especially the detailed models and simulations. Review each of the designs and do a final review on your documentation. Record the design review in your notebook along with recommended changes and have the mentors sign and date it.

Optional Exercises: *Since there are many different models and simulations that can be created for both FIRST and RWDC, here are additional exercises to help in creating those simulations. They can be found in the Phase 2 folder.*

Creating an Airfoil Exercise

Stress Analysis Exercise

Phase

2

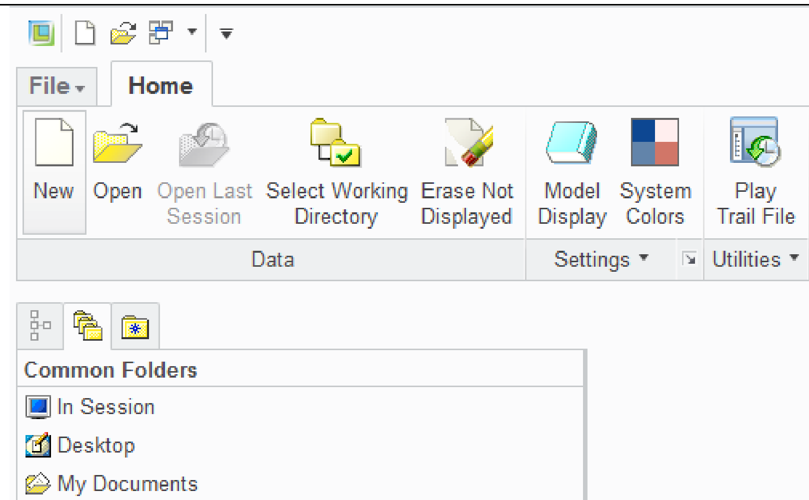
HOW TO DEVELOP A

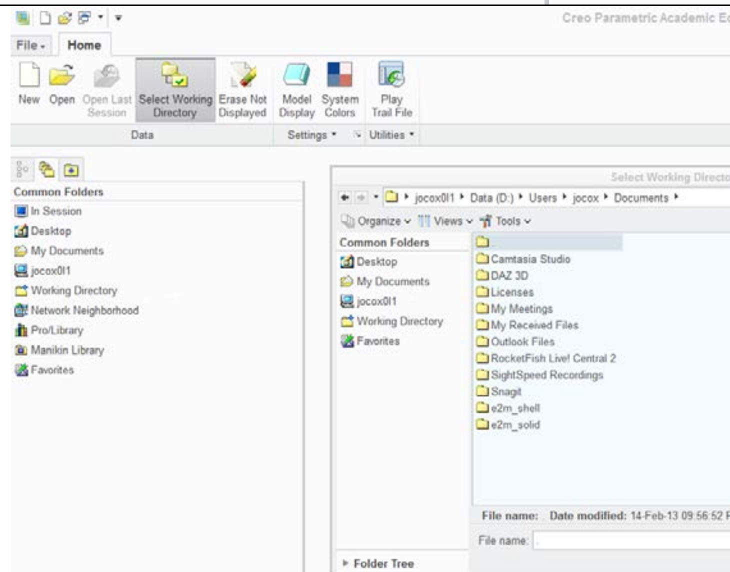
PRODUCT

APPENDIX

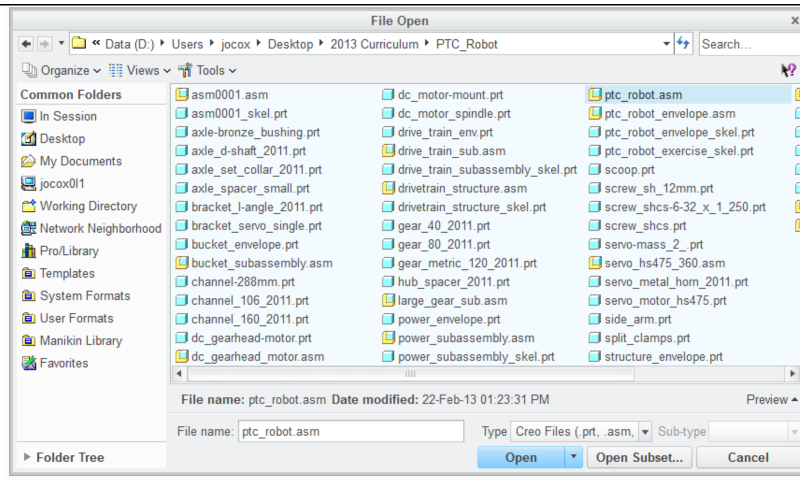
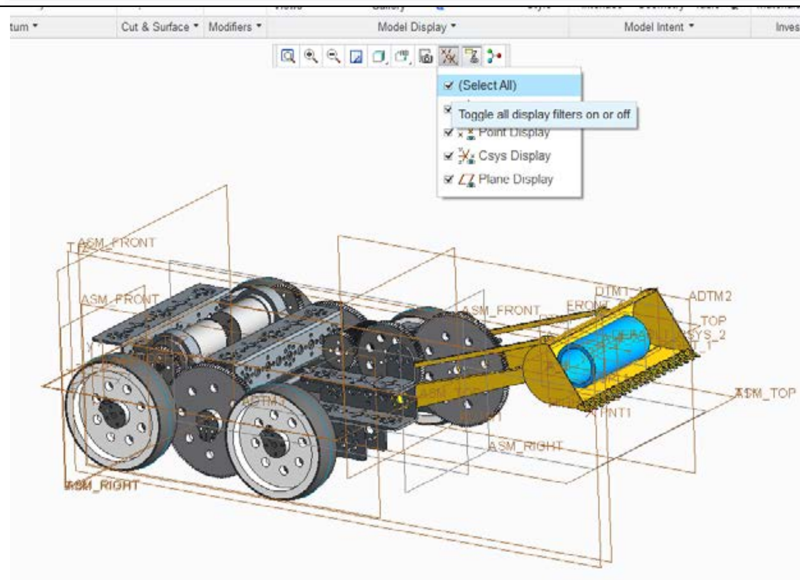
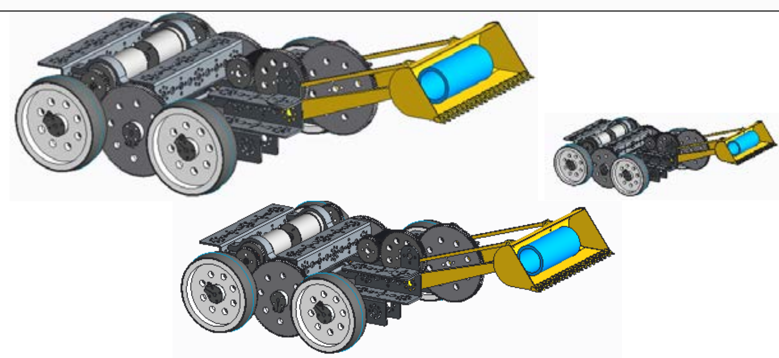
EXERCISE 6.0: PTC ROBOT EXPLORE

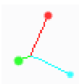
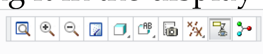
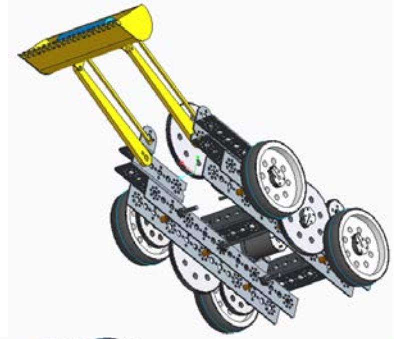
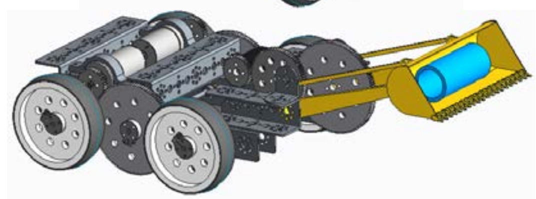
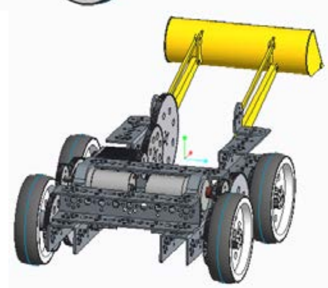
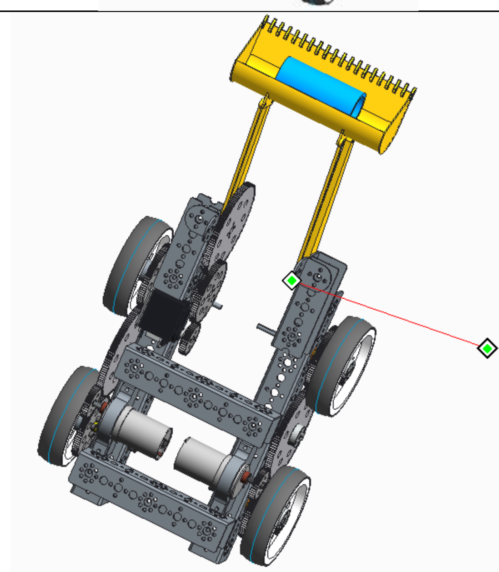
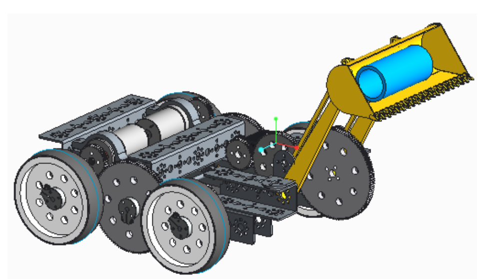
This document provides detailed instructions for exploring the PTC-Robot in preparation for designing the sub-systems.

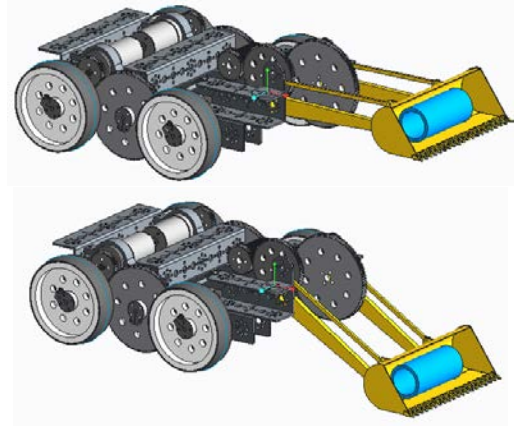
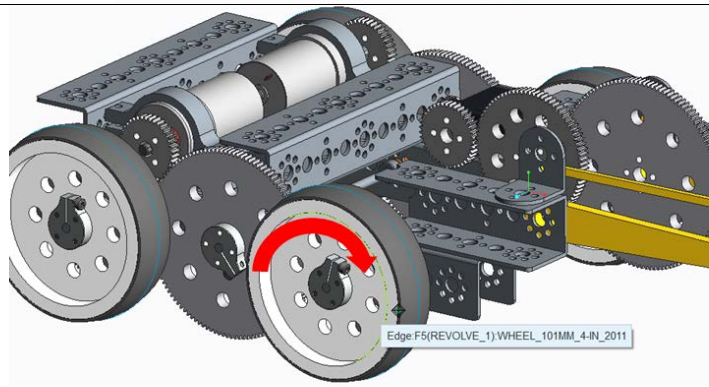
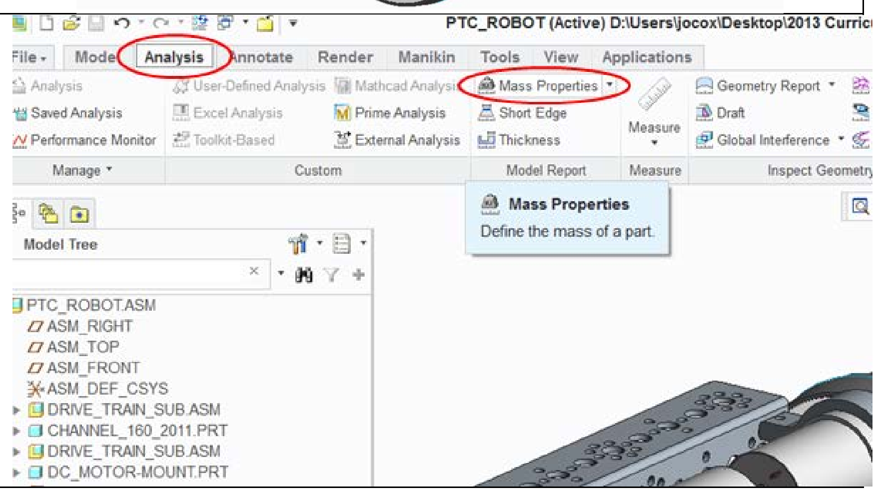
<table>
<tr>
<td>Begin by opening PTC Creo Parametric</td>
<td>


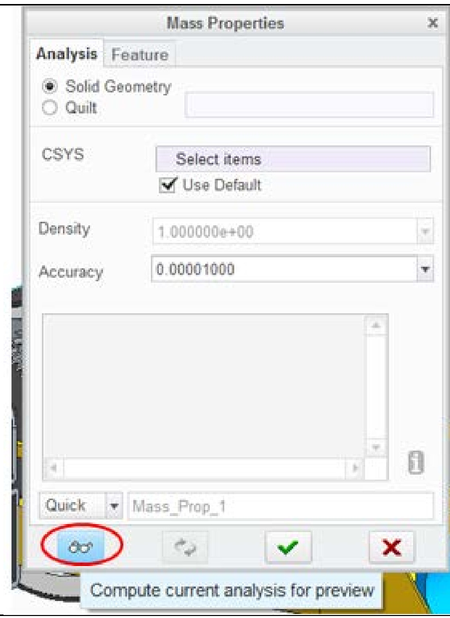
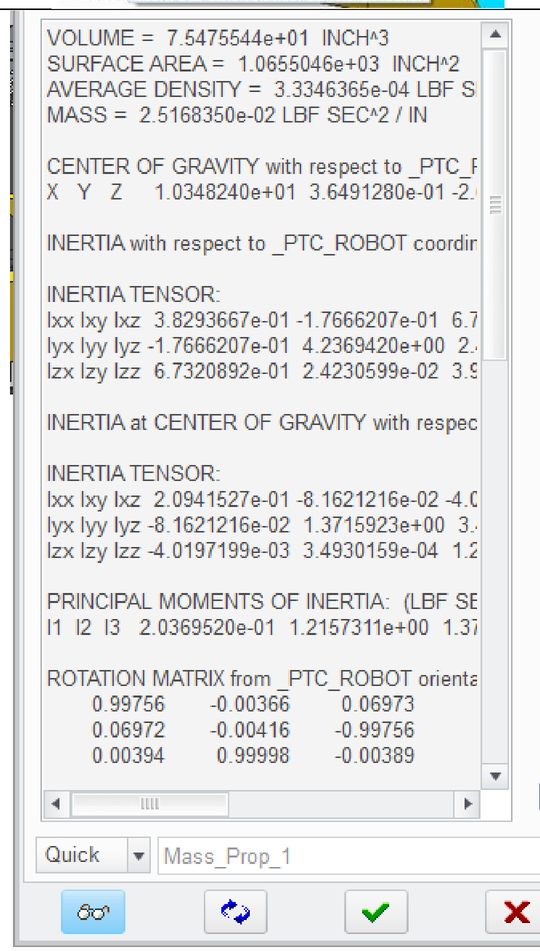
</td>
</tr>
<tr>
<td>Make the working directory be the PTC_Robot folder.</td>
<td>


</td>
</tr>
</table>

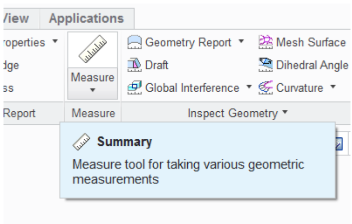
Now open the file **PTC_Robot.asm**	
Toggle off all of the datum planes and axes by selecting the **Datum Display Filters**  And toggling off all of the check marks.	
Now let's review. To zoom simply scroll the middle wheel on your mouse in and out. The zoom center is wherever your cursor is while scrolling.	

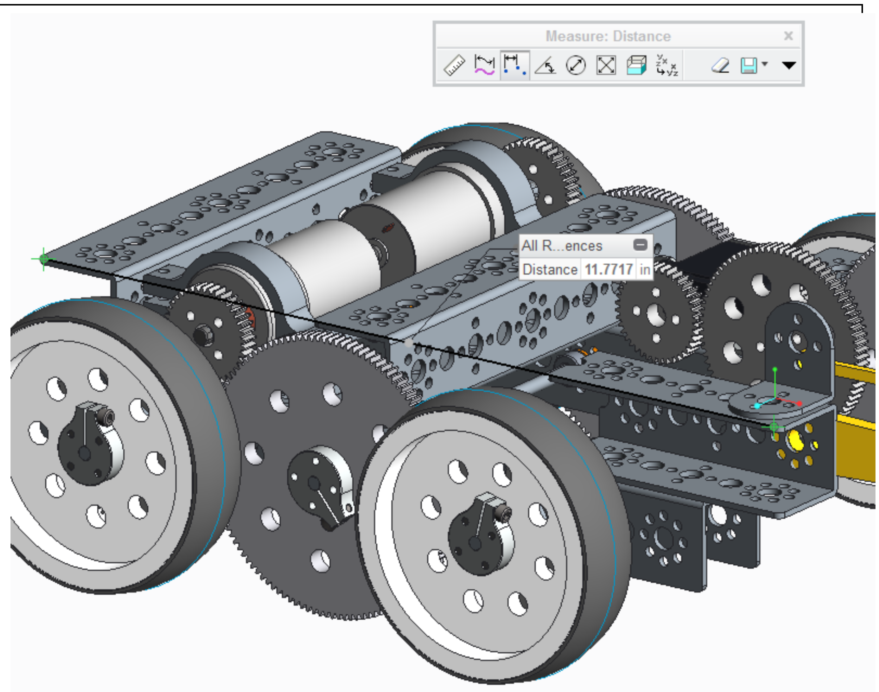
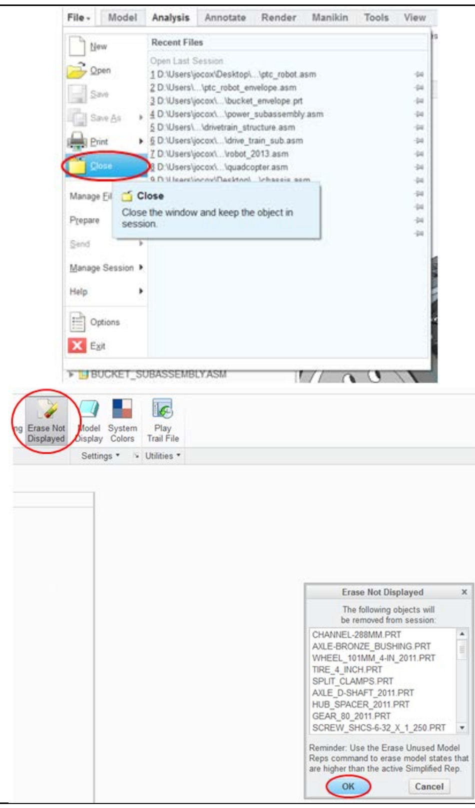
Rotating or spinning the model is accomplished by pushing down the middle scroll wheel on your mouse and holding it down while moving the mouse. The rotation center is indicated by the spin center . If you turn the spin center off by toggling it in the display menu , then the spin center is wherever your cursor is when you spin.	  
You can pan your model on the screen by holding the shift key while pushing and holding the scroll wheel down and moving your mouse.	
Now that you can manipulate the robot on the screen, let's look at some other ways of manipulating the robot. This robot model has been created with kinematic joints. That means the	

appropriate parts can move. Hold down the **CTRL-ALT** keys on the keyboard while left mouse button clicking on the bucket to raise and lower it.	
Try using the **CTRL-ALT** buttons while left clicking on the wheels and turning them.	
Now that we can explore the robot in many different ways, let's find out some important information about the robot. Select **Analysis** in the upper menu and then select **Mass Properties**.	

A dialog box menu will appear. Click on the eye-glasses	
You will see that Creo Parametric has calculated information that is important to this model about its volume, mass, center of gravity, and so forth. This is important information that is available for you when you create your own models.	

<table>
<tr>
<td>You can also measure any aspect of the model. Close the Mass Properties menu and then click on the Measure icon in the upper menu.



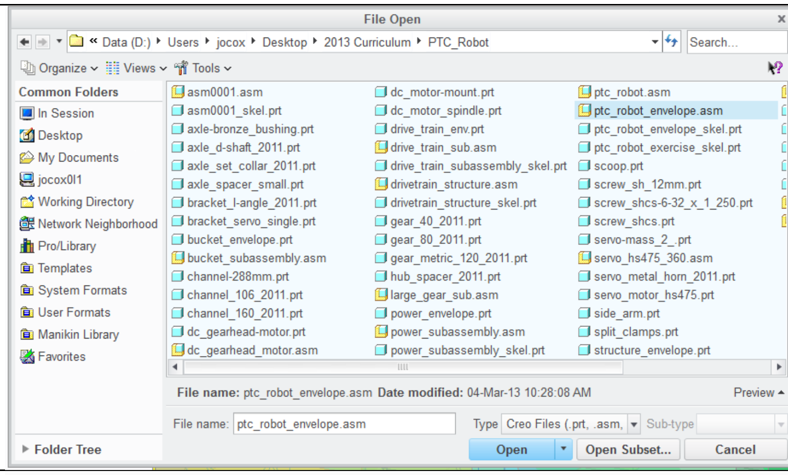
You can measure distances, surface area, diameters, etc. (remember when selecting multiple points to use the CTRL key)</td>
<td>
</td>
</tr>
<tr>
<td>Now that you are familiarized with the PTC Robot, the exercises that follow will teach you how to design and build this model. Close this model by selecting Close in the File menu and then select Erase Not Displayed and select OK to clear the memory.</td>
<td>
</td>
</tr>
</table>

You have finished the exercise.

EXERCISE 7.0: PTC ROBOT ENVELOPE EXPLORE

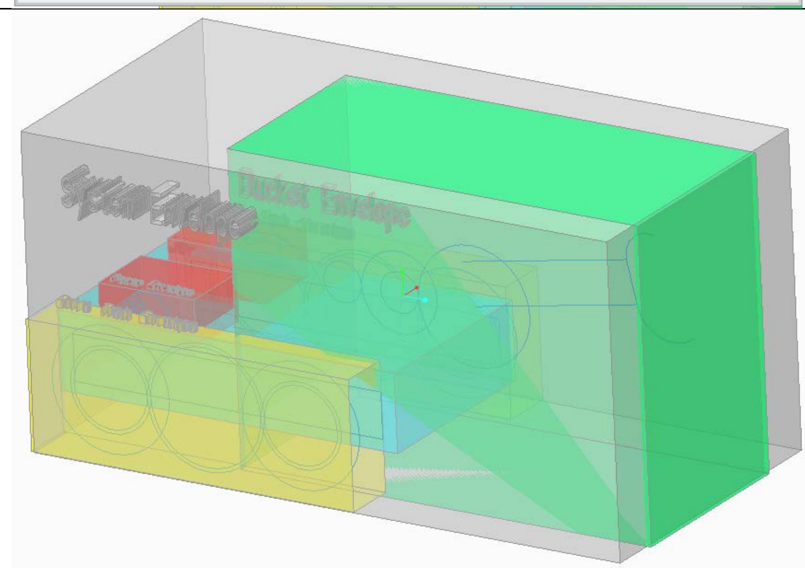
This document provides detailed instructions for exploring the PTC-Robot Envelope model in preparation for designing the sub-systems.

<table>
<tr>
<td>Start Creo Parametric and set your working directory to the PTC Robot folder. Now open the file ptc_robot_envelope.asm</td>
<td>

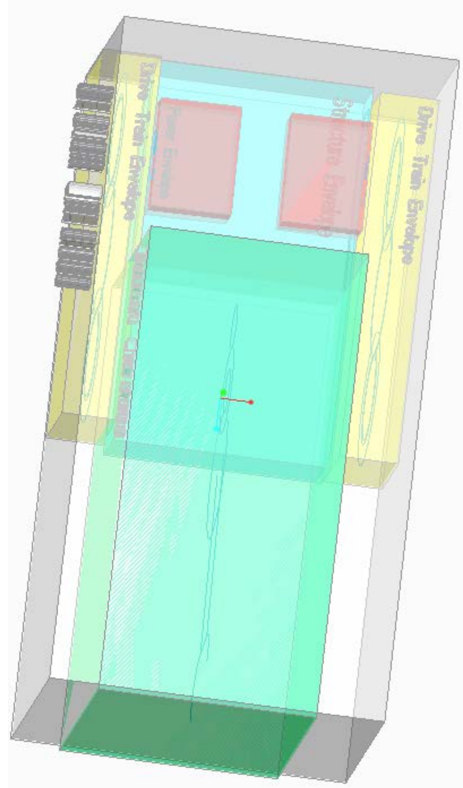
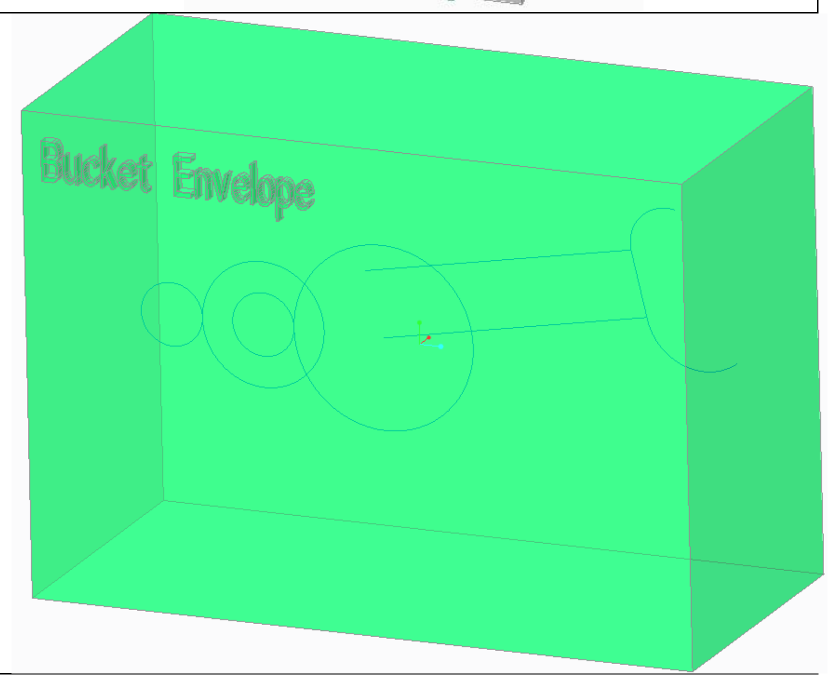


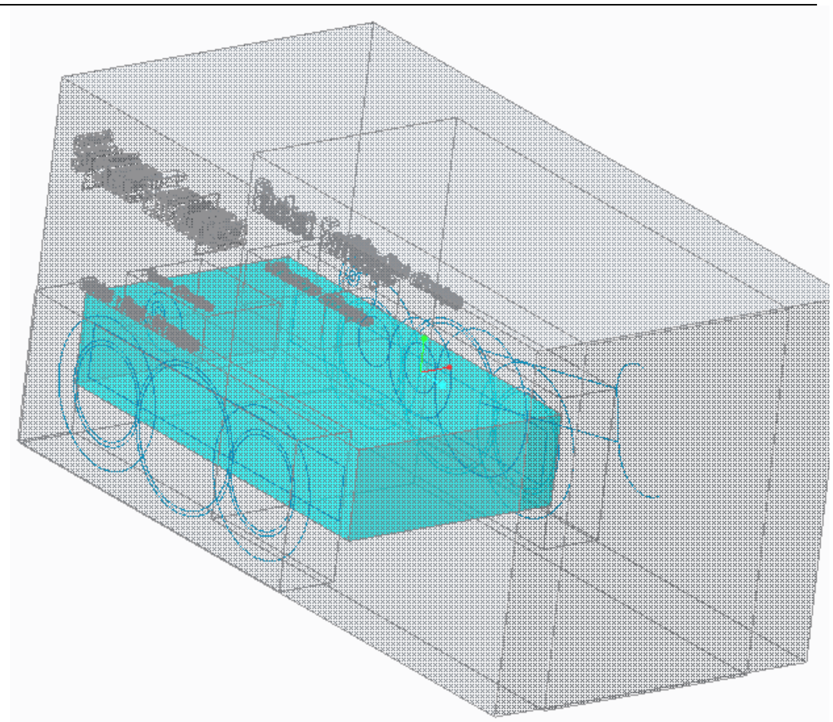
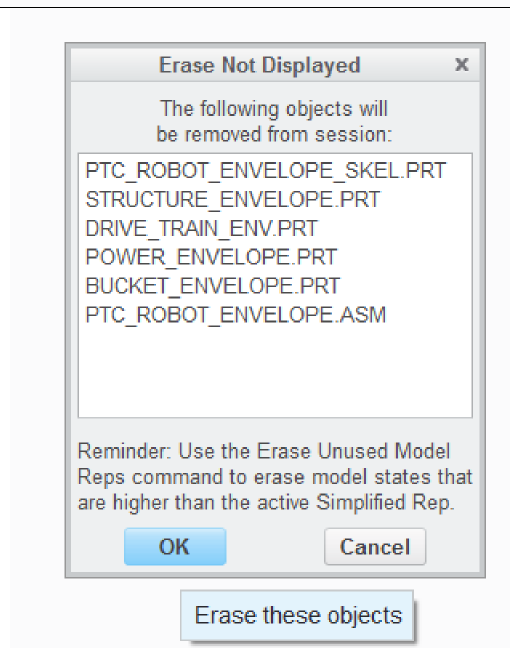
</td>
</tr>
<tr>
<td>Toggle off all of the datum planes and axes by selecting the Datum Display Filters

And toggling off all of the check marks.</td>
<td>



</td>
</tr>
</table>

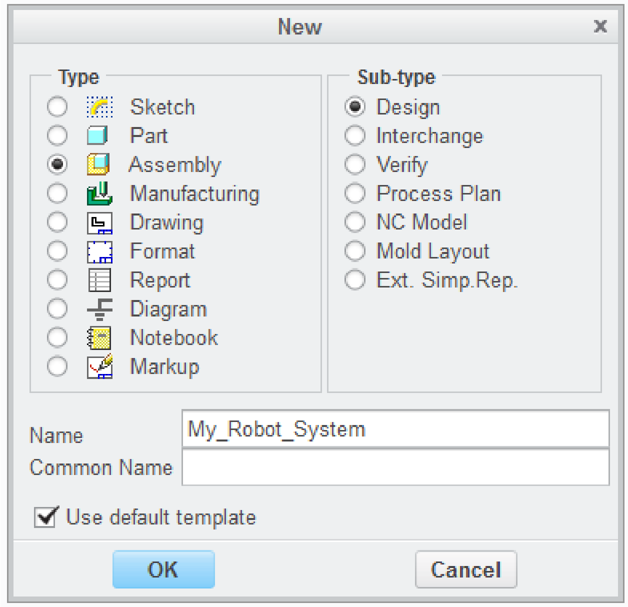
Rotate and zoom to explore the system envelope model for the **PTC-Robot**.	
You can explore each of the sub-system envelopes by right clicking on their names in the model tree and selecting **Open**.	

We will use these envelopes to guide our sub-system models and to make sure that the sub-systems integrate into a whole system model.	
So let's get started building the sub-system detailed models. Close the **PTC_ROBOT_ENVELOPE.asm** file and erase it from memory.	

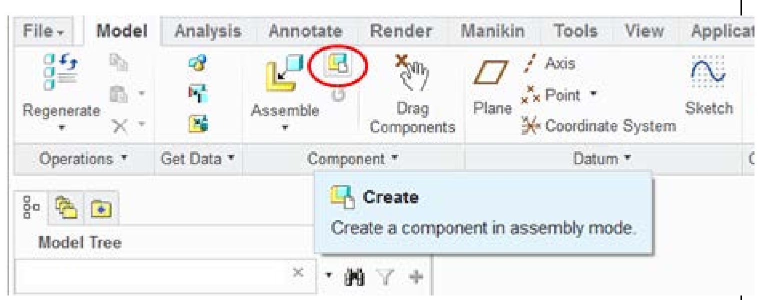
You have finished the exercise.

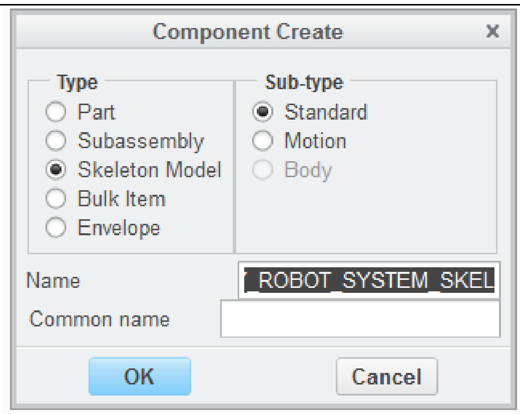
EXERCISE 8.0: CREATING THE SYSTEM MODEL

This document provides detailed instructions for creating the system level model that will be used to integrate all of the subsystem models.

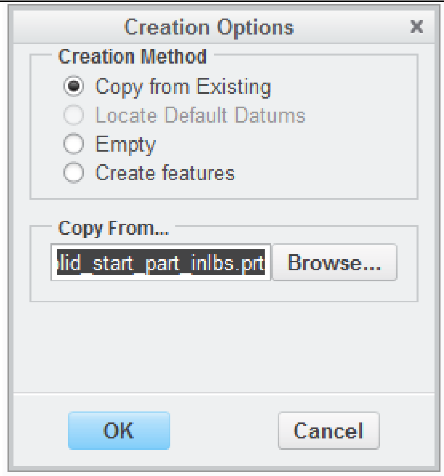
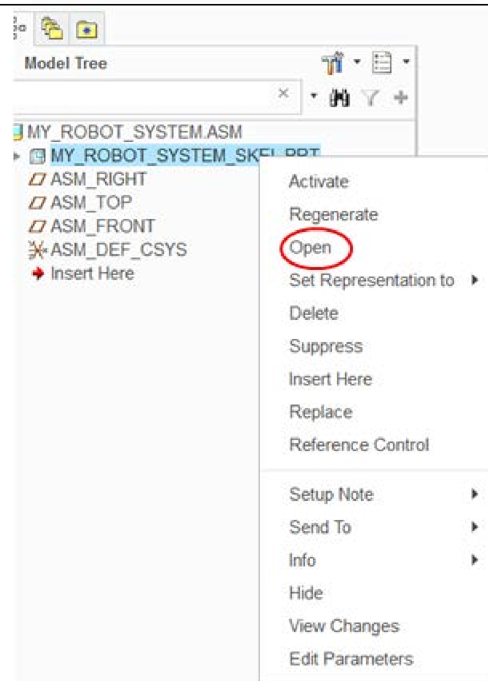
<table>
<tr>
<td>Open Creo Parametric if it is not already open. Select New and create a new Assembly file for the overall robot assembly. Call your robot: “My_Robot_System” and click OK.</td>
<td>


</td>
</tr>
<tr>
<td>We are going to setup the overall system level model so that all of the sub-system models can reference it. This will make sure that they will all integrate together into the final system model.

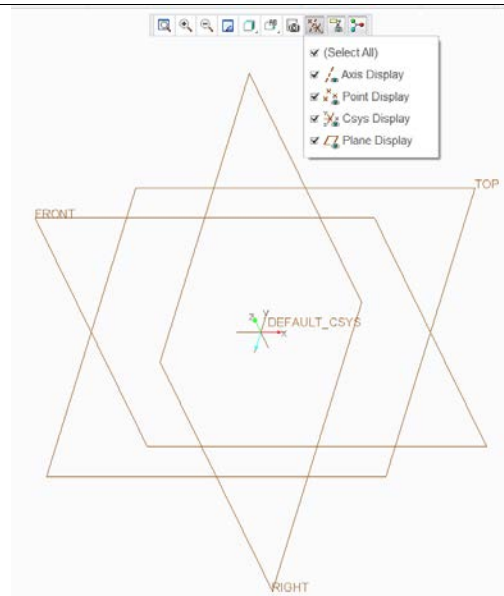
Now select the Create icon to create a Skeleton file.</td>
<td>


</td>
</tr>
<tr>
<td>Select Skeleton Model and click OK.</td>
<td>


</td>
</tr>
</table>

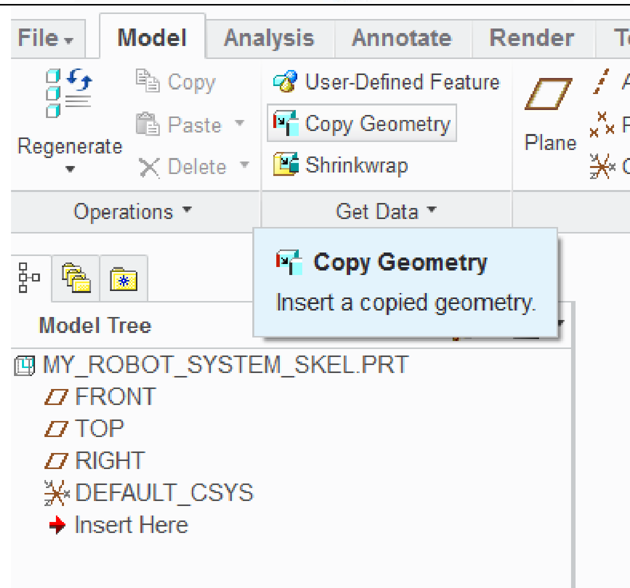
Select “**Copy from Existing**” and click **OK**.	
Now right click on the skeleton model in the model tree and select **Open**.	

The skeleton file will now be open. Toggle all of the datums back on by selecting the **Datum Display Filters**.

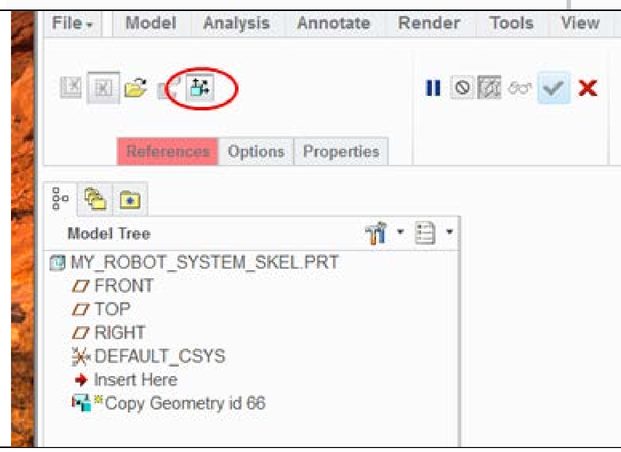


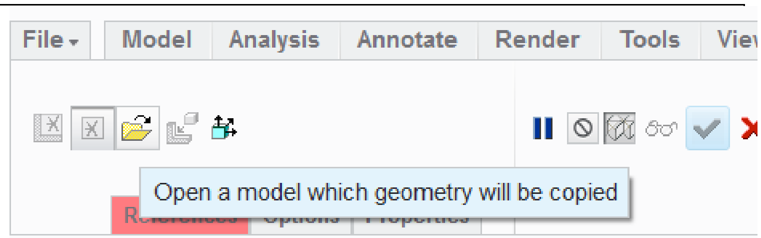
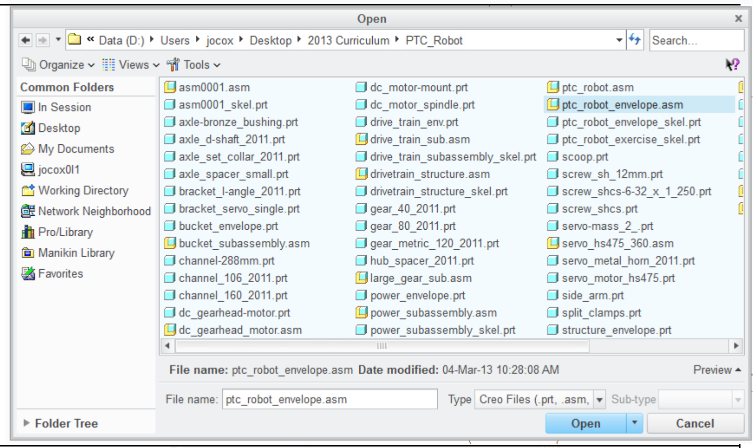
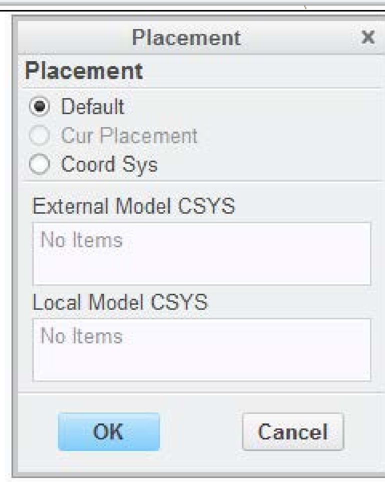
We want to bring all of the pertinent system envelope information into this skeleton file so that as we design each of the sub-systems, the associated models can reference their respective envelopes.

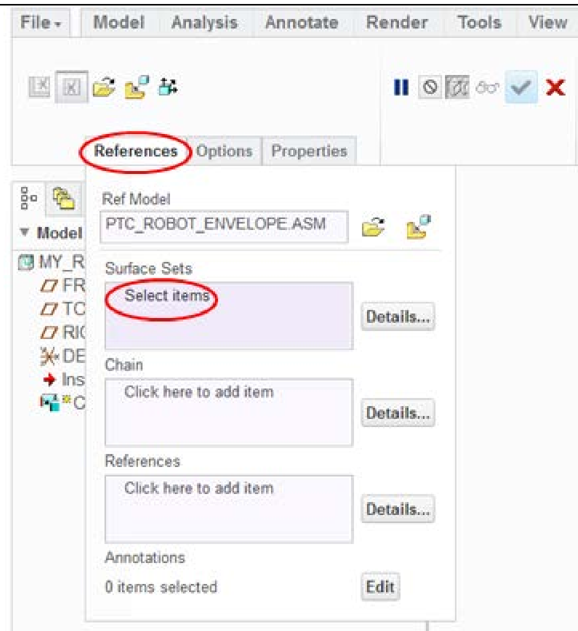
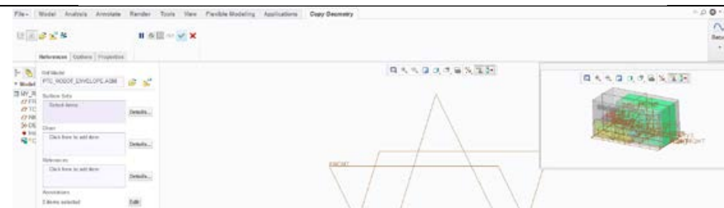
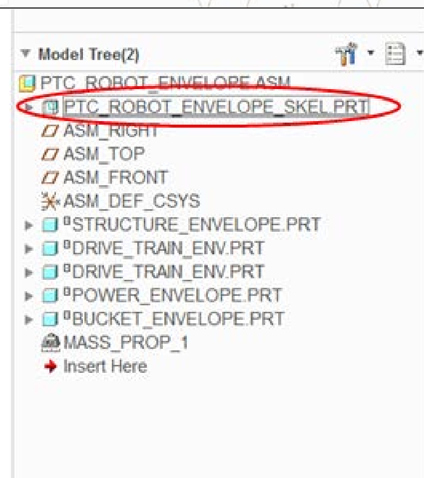
Select **Copy Geometry** in the upper menu.

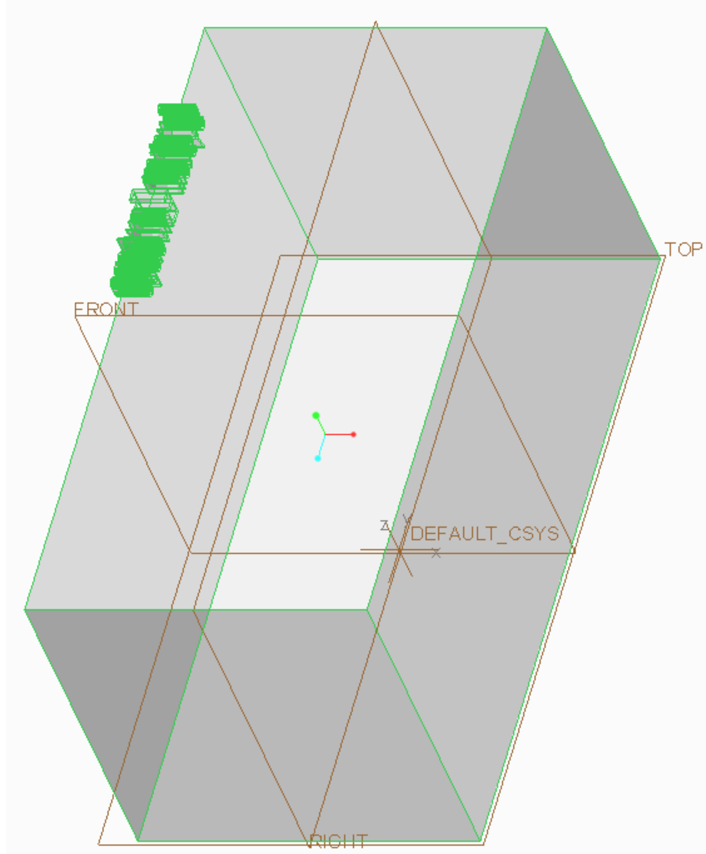
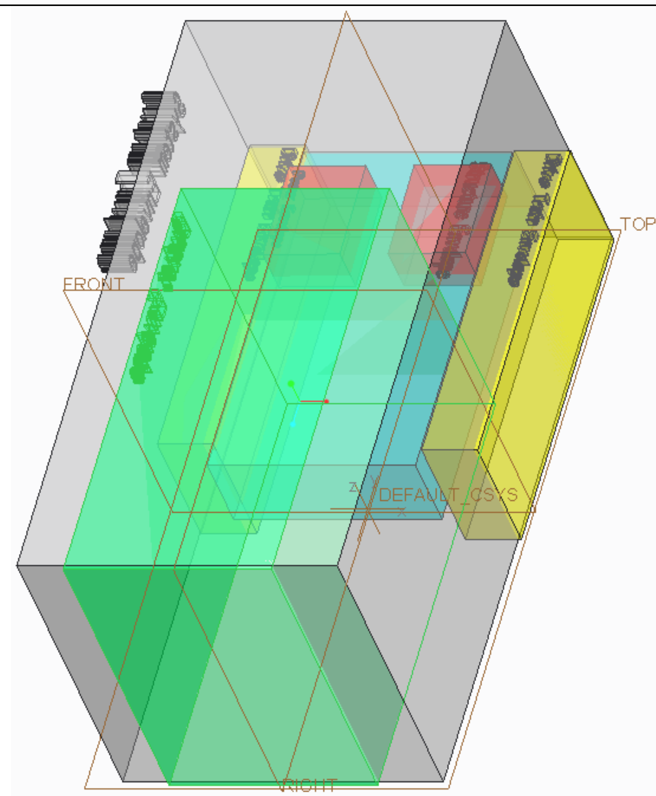


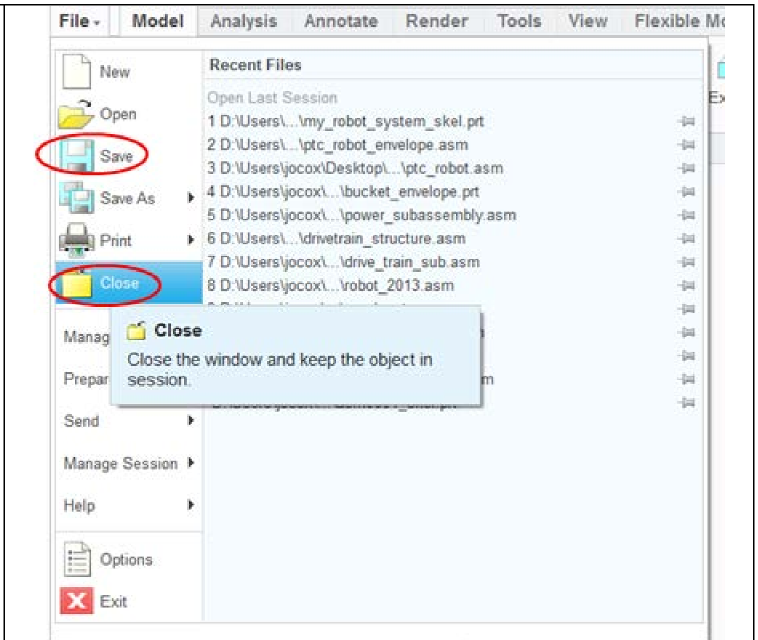
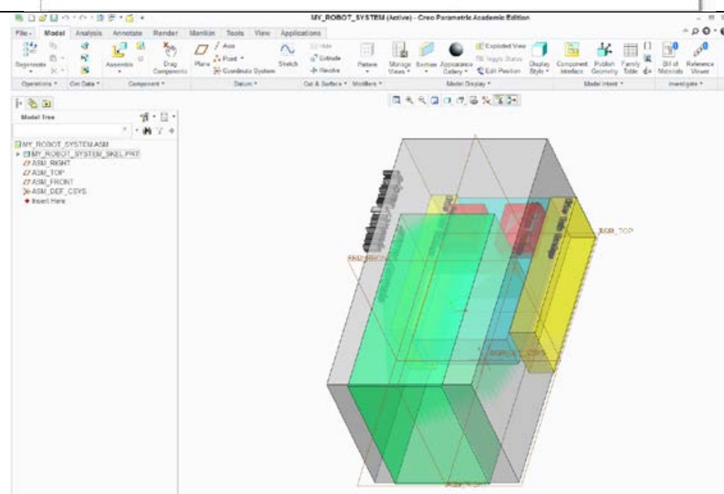
Deselect the publish geometry option by clicking on it to un-highlight it.



<table>
<tr>
<td>Now click on the folder icon to open the system envelope file.</td>
<td>
</td>
</tr>
<tr>
<td>Select ptc_robot_envelope.asm and select Open.</td>
<td>
</td>
</tr>
<tr>
<td>Keep the default placement option. Then click OK.</td>
<td>
</td>
</tr>
</table>

Click on the **References** tab which will be **red**. Then click in the **Surface Sets** area to begin selecting surfaces.	
*You will see a small window with the **ptc_robot_envelope** models. Also the model tree will split and you will see both model trees.*	
The first envelope we want to reference is the overall system envelope. Select the **PTC_ROBOT_ENVELOPE_SKEL.PRT** in the second model tree by left clicking on it. Then click the green check mark in the upper menu.	

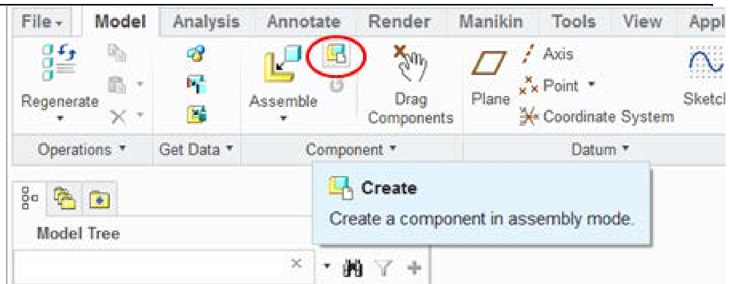
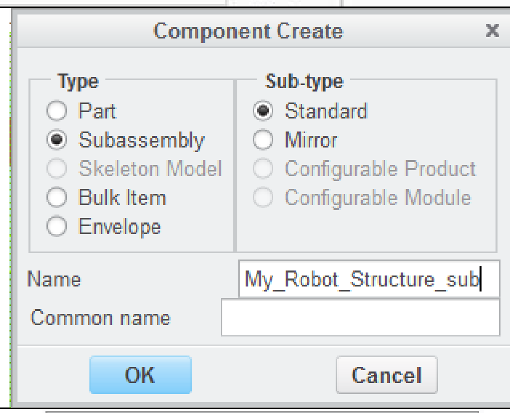
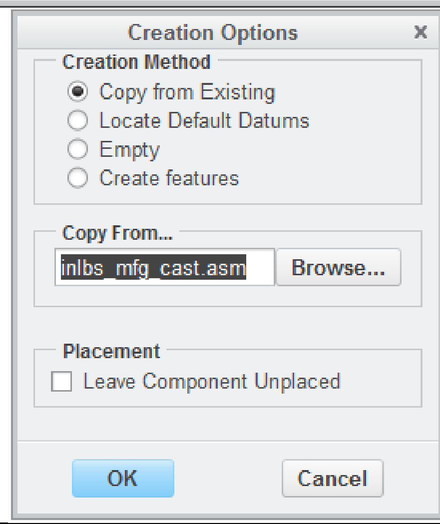
The overall system envelope is now copied into the skeleton file as a reference set. *We need to do this again for each of the sub-system envelopes. Repeat this process to copy all of the sub-system envelopes into your skeleton file.*	
When you are done, your skeleton file should look like this.	

<table>
<tr>
<td>Now save your skeleton file by selecting Save in the File tab and then Close.</td>
<td>
</td>
</tr>
<tr>
<td>All of the envelopes are now in the new robot system model file. So save this file.</td>
<td></td>
</tr>
</table>

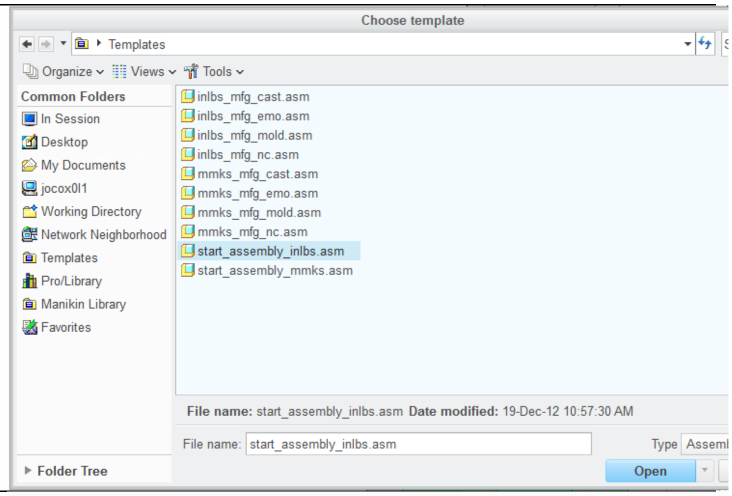
You have finished this exercise.

EXERCISE 9.0: THE STRUCTURE SUB-SYSTEM

This document provides detailed instructions for designing the structure subsystem for the PTC robot.

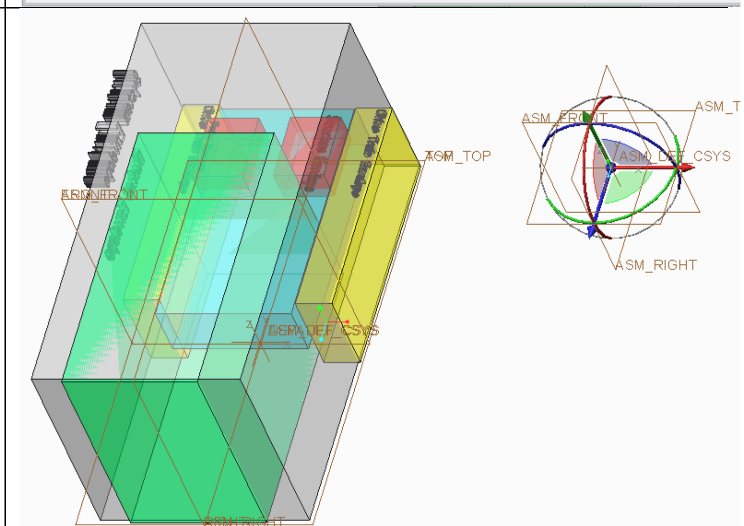
Make sure that you have the system model opened. If not open "**my_robot_system.asm**". Now to begin building the subsystems, we need to select the **Create** icon and select **Subassembly**.	
Type in the name: "**My_Robot_Structure_sub**" and click **OK**.	
Select **Copy from Existing** and click the **Browse** to select the template.	

Select the template file called: "**start_assembly_inlbs.asm**" and click **Open**.

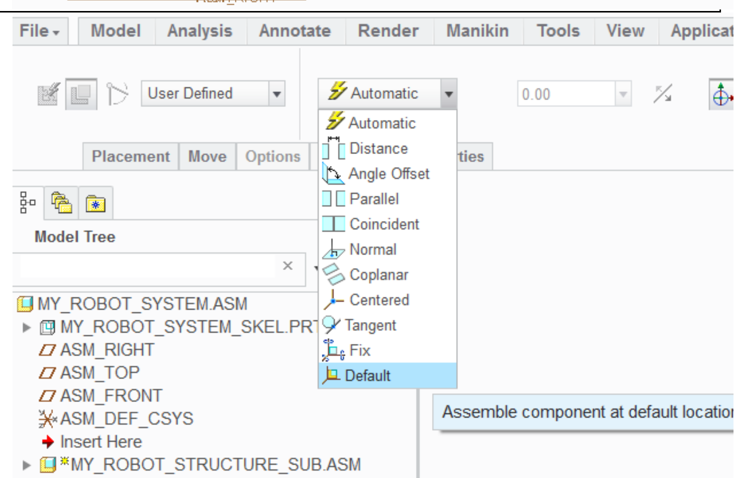


Left click to place the default coordinate system of the new structure subassembly.

This is like assembling a new part into the system assembly but in this case the new part is empty because we haven't designed it yet. Therefore only the origin coordinate system shows up.

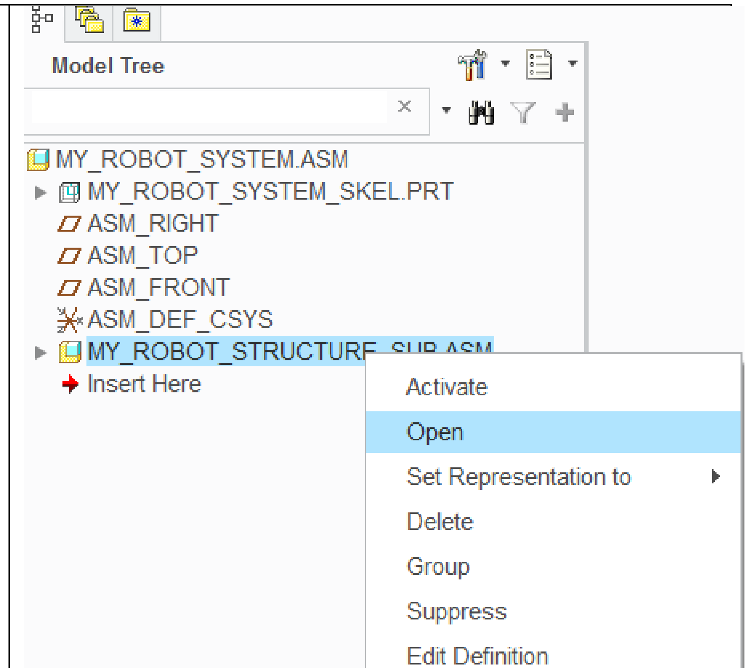


Now select the **Default** constraint in the **Automatic** drop down menu. This assembles the structure subassembly at the default coordinate location. Now click the green checkmark to finish placing the new subassembly.



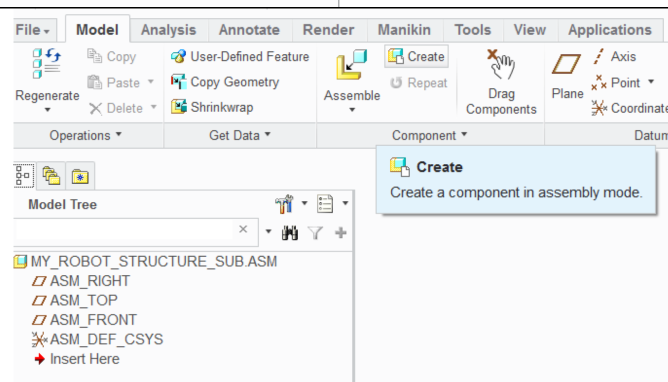
We are now going to open the newly created Structure subassembly model to begin defining this sub-system.

Right click on **MY_ROBOT_STRUCTURE_SUB.ASM** in the model tree and select **Open**.

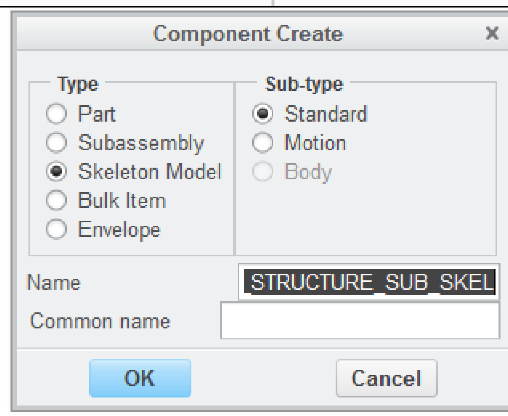


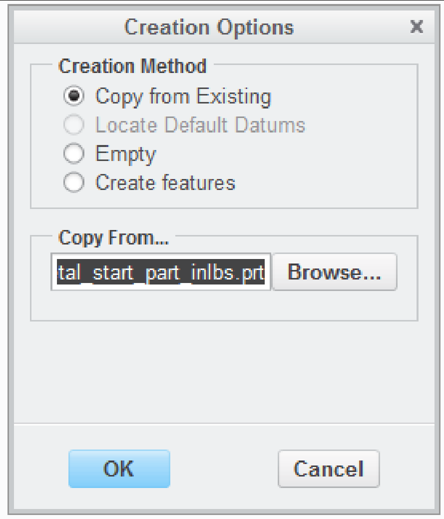
Now before creating the structure, we need to identify references for the structure. We do this by creating a skeleton file with just the structure envelope.

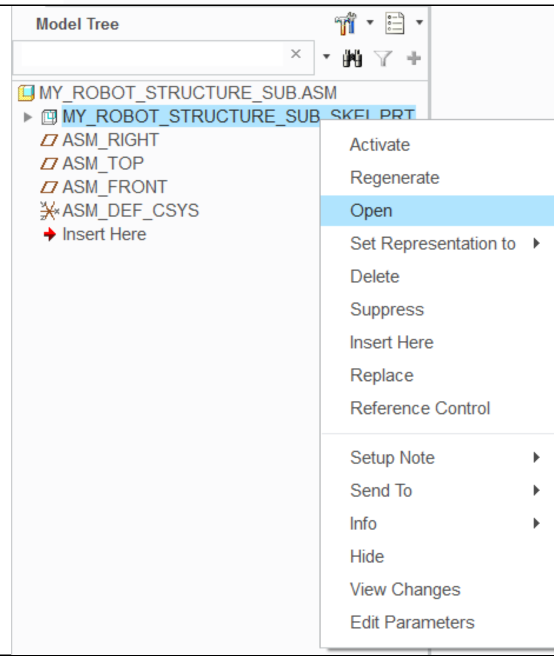
Select **Create** so that we can set the reference geometry in this subassembly.

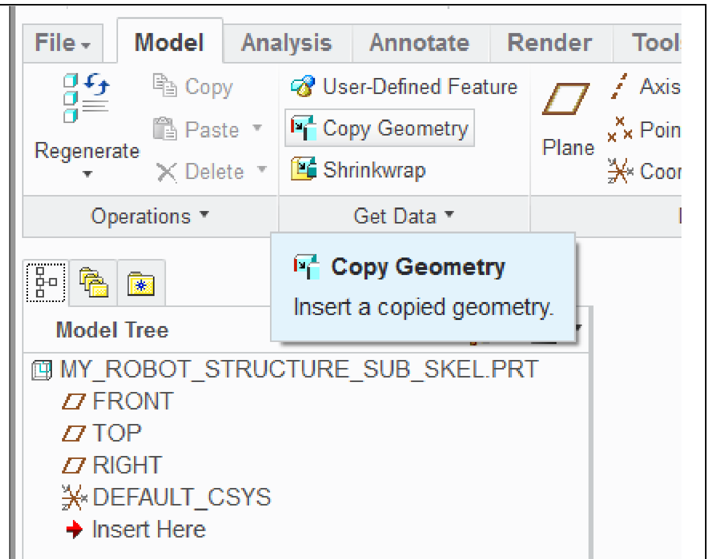
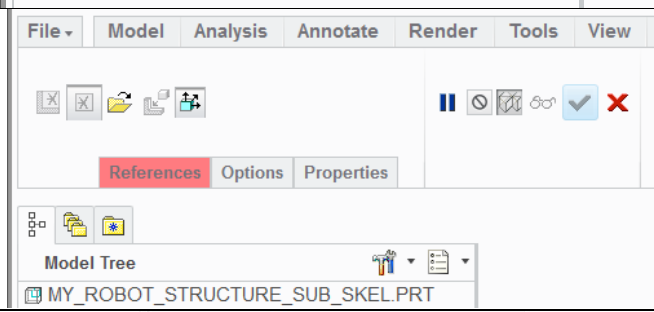
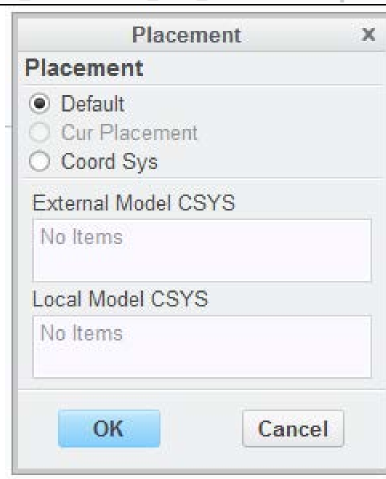


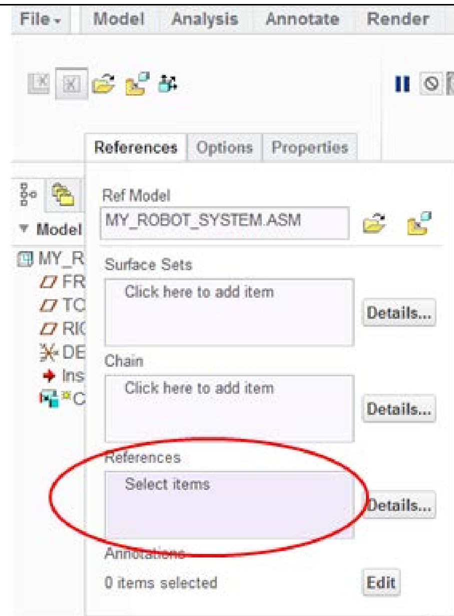
Select **Skeleton Model** and click **OK**.

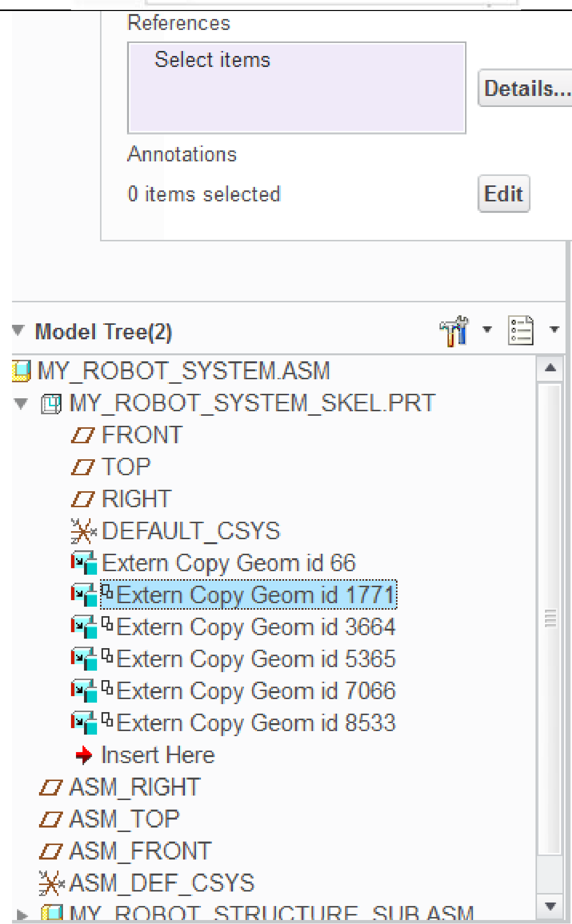


<table>
<tr>
<td>Leave the default setting of "Copy from Existing" and click OK.</td>
<td>
</td>
</tr>
<tr>
<td>Now right click on MY_ROBOT_STRUCTURE_SUB_SKEL.PRT in the model tree and select Open.

This may seem a little confusing but it is necessary to set up these skeleton references in the model to make sure everything integrates together as a system.</td>
<td>
</td>
</tr>
</table>

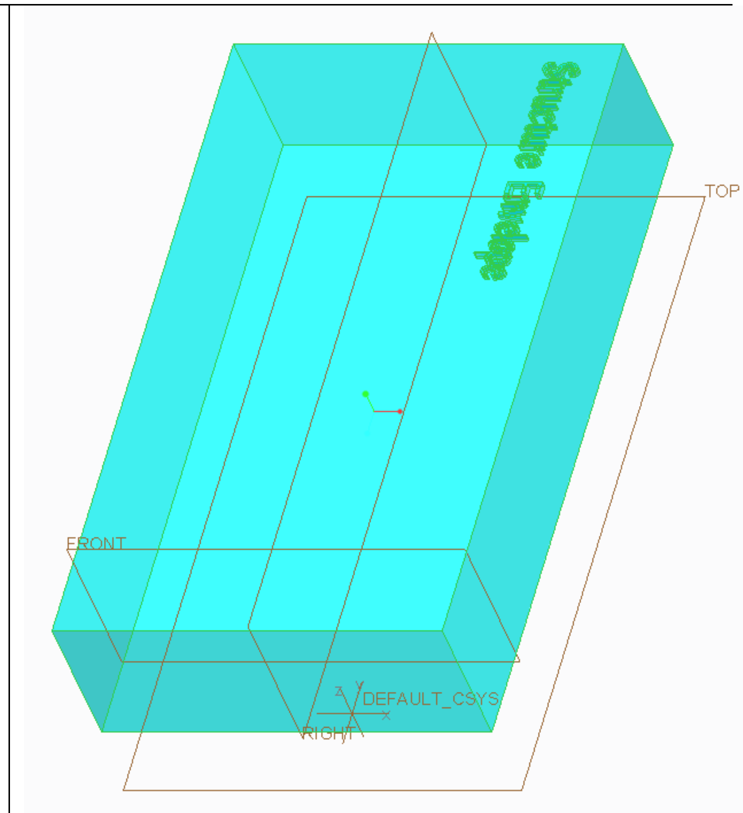
<table>
<tr>
<td>Select Copy Geometry so that we can copy into our structure subassembly the structure envelope as a reference.</td>
<td>
</td>
</tr>
<tr>
<td>Once again unclick the publish geometry icon and then click on the folder icon and select your system assembly file: “my_robot_system.asm” and click Open.</td>
<td>
</td>
</tr>
<tr>
<td>Leave the default option and click OK.</td>
<td>
</td>
</tr>
</table>

<table>
<tr>
<td>This time, click on the References tab and then click in the References area.</td>
<td>
</td>
</tr>
<tr>
<td>Now we want to select the structure envelope. To find this envelope, we need to expand MY_ROBOT_SYSTEM_SKEL.PRT in the second model tree and then click on the second Extern Copy Geom as shown. This is the structure envelope in your system assembly file.

Now click the green check mark to place the envelope.</td>
<td>
</td>
</tr>
</table>

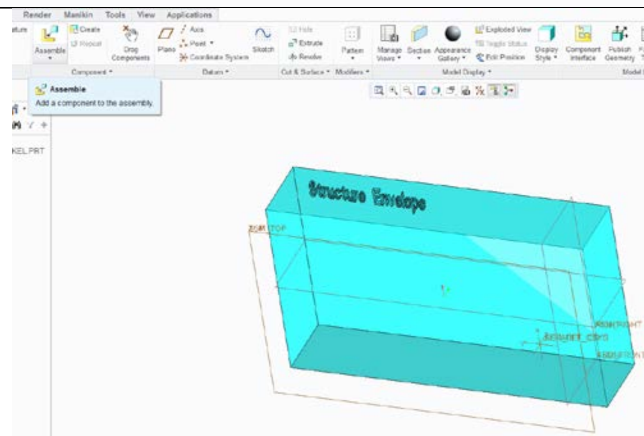
The structure envelope is now placed in the skeleton file of the subassembly. We just need to save and close this skeleton file.

So select **Save** from the **File** menu and then **Close**.

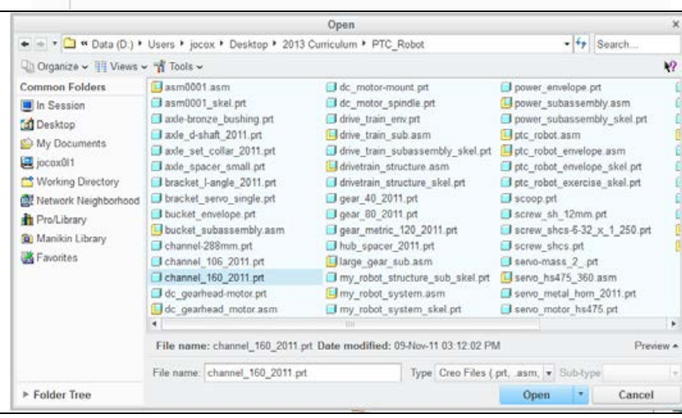


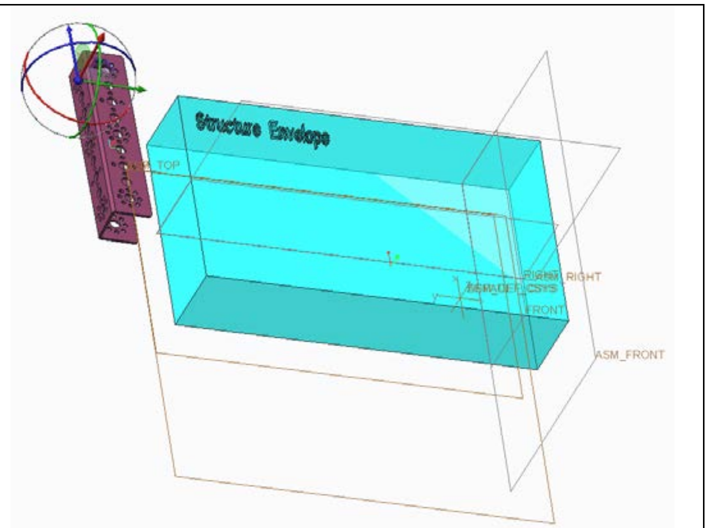
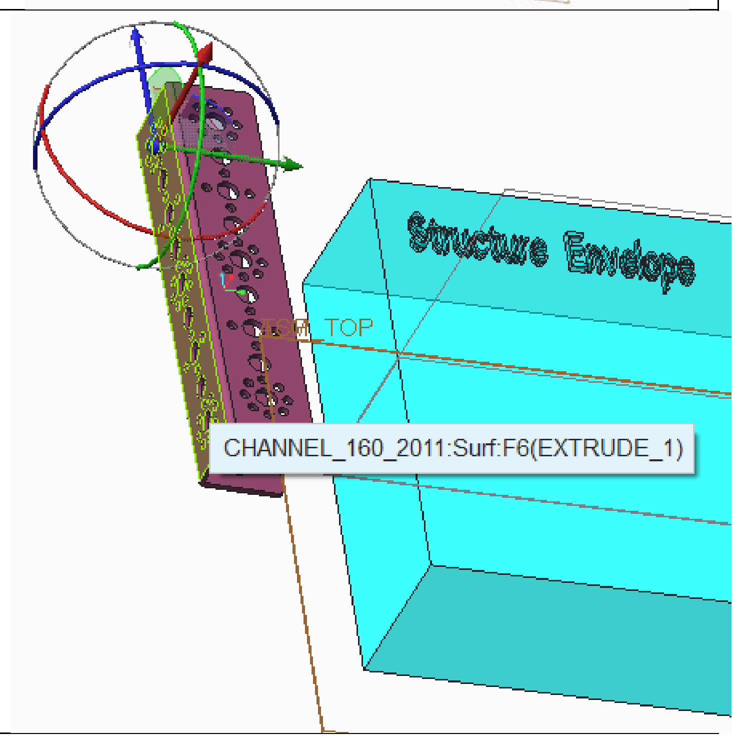
We are now ready to start building the structure subassembly.

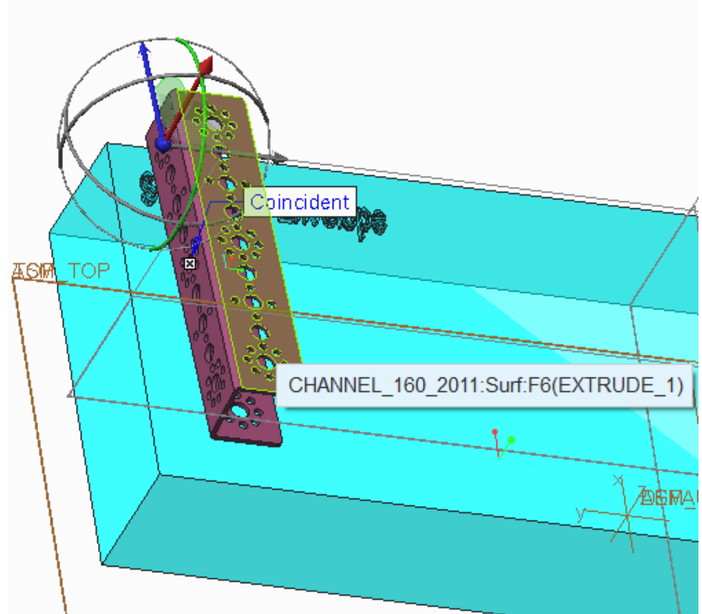
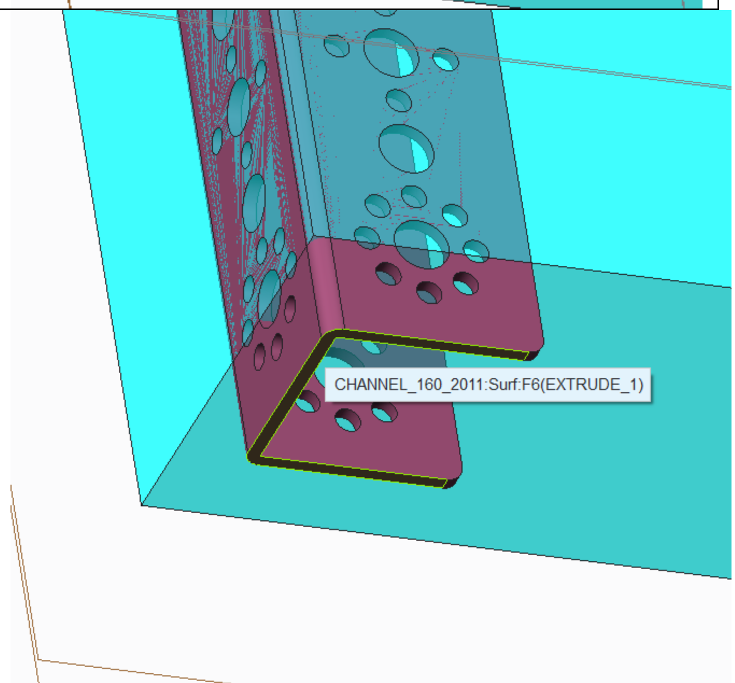
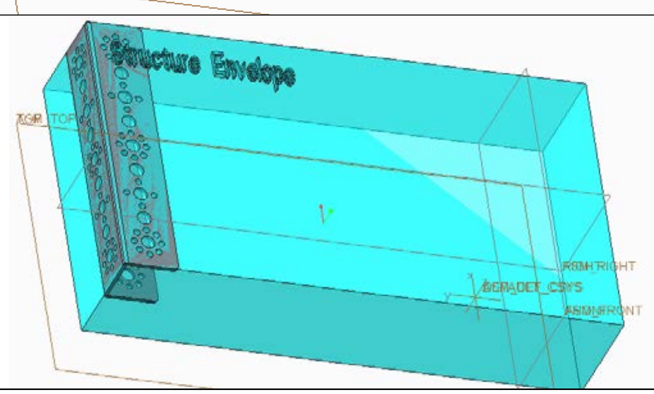
Start by clicking the **Assemble** icon in the upper menu.

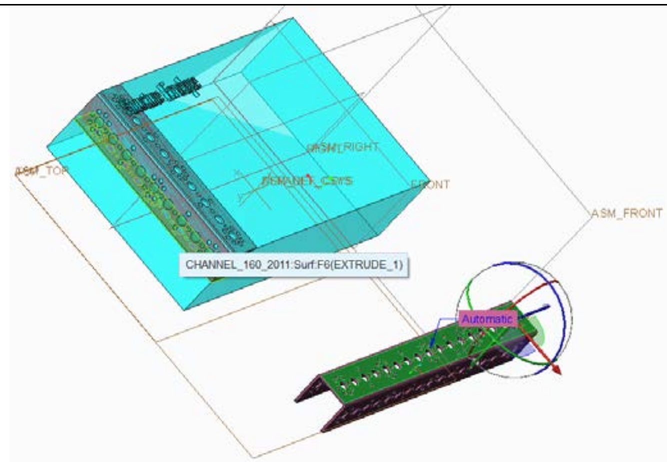
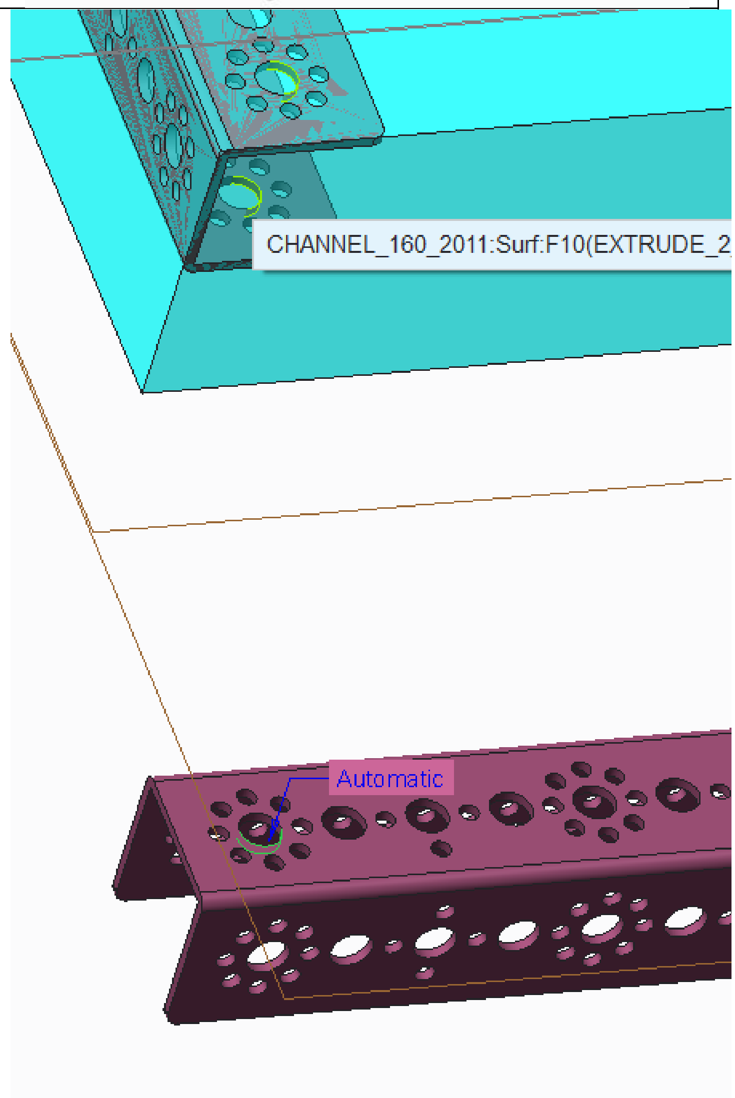


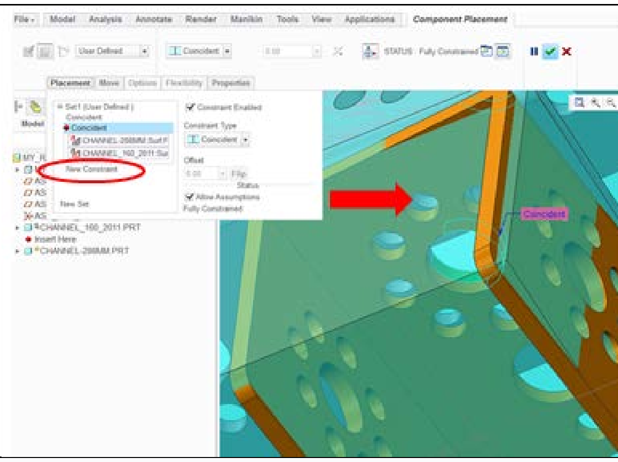
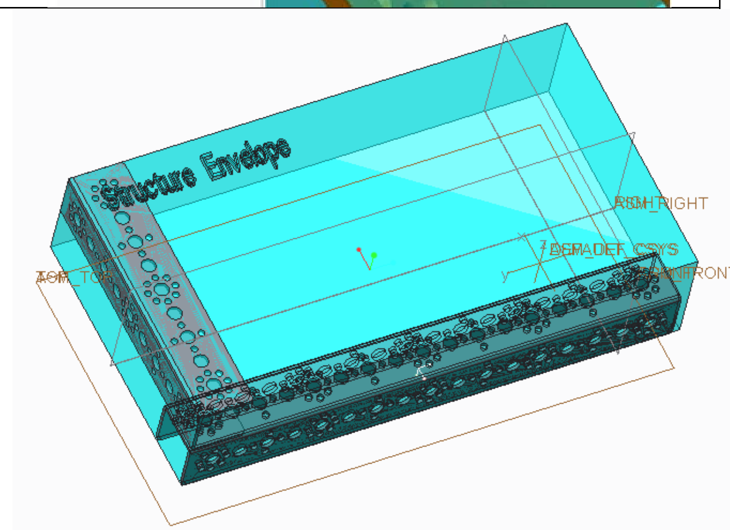
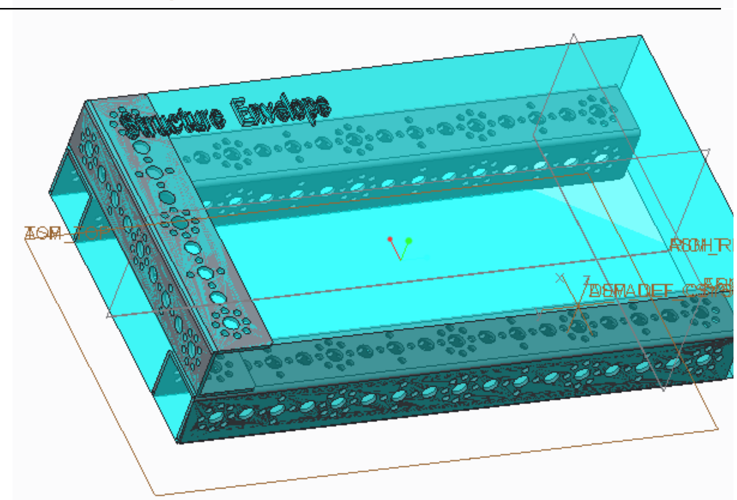
Select **channel_160_2011.prt** and click **Open** to bring in the first channel that will be used in creating the structure subassembly.

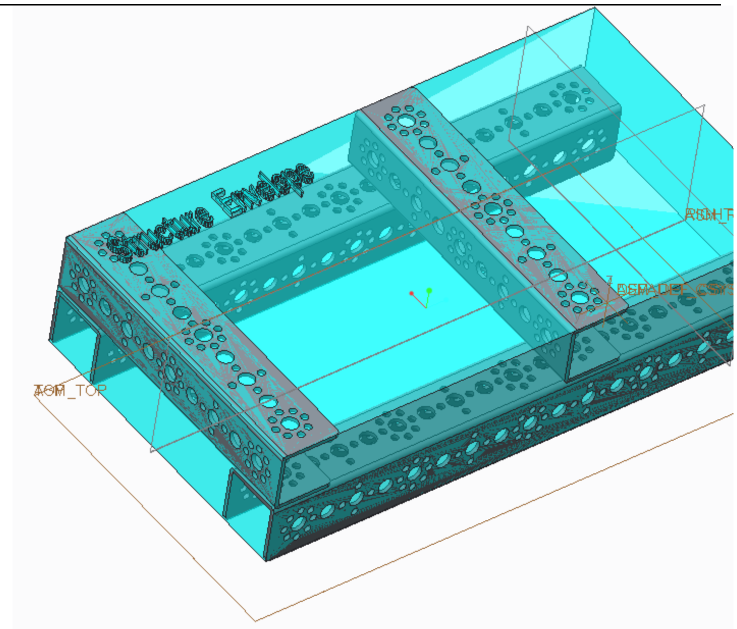


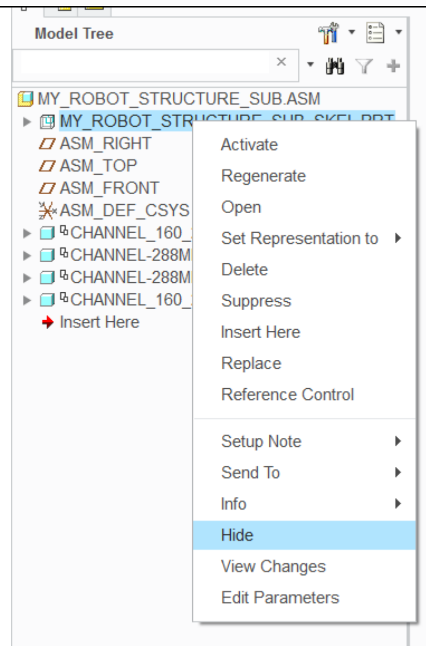
Left click to temporarily place the channel part and then use the orientation sphere to orient the channel as shown.	
Select the back surface of the channel by left clicking on it and then select the back surface of the envelope by left clicking on it.	

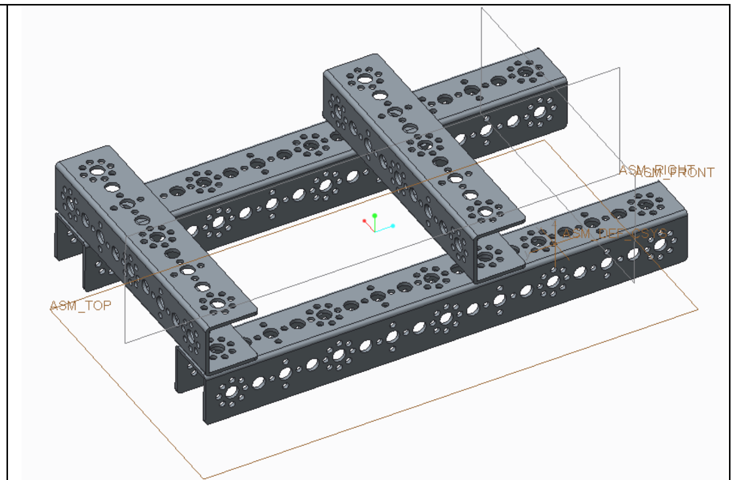
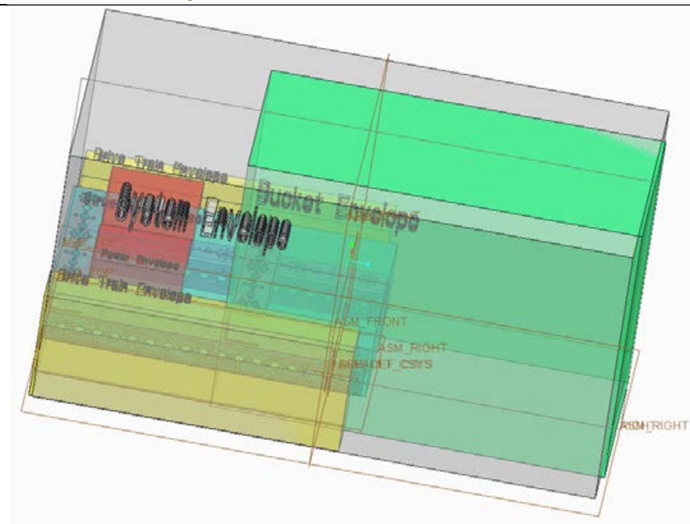
Now select the top surface of the channel by left clicking on it and the top surface of the envelope.	
Finally select the end of the channel and the right side of the envelope to finish placing the channel with respect to the envelope.	
The first part of the structure is in place so now let's place the other channels with respect to this first one. Select **Assemble** in the upper menu and open "**channel-288mm.prt**" and left click to temporarily place it.	

Use the orientation sphere to orient it as shown. Then click on the top surface of the channel and the bottom surface of the first channel.	
Now select the holes at the two ends of the channels as shown.	

Now open the **Placement** tab in the upper menu and click on **New Constraint**. Then select the two smaller holes to align the channels.	
Click the green check mark to finish placing the second channel.	
Use the same procedure to insert and place the second channel as shown.	

<table>
<tr>
<td>Finally use your skills to place the fourth channel in the assembly as shown.</td>
<td></td>
</tr>
<tr>
<td>We can hide the reference envelope at this point to see the final structure subassembly.

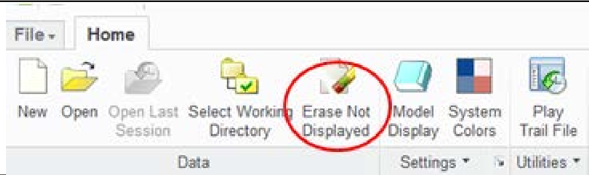
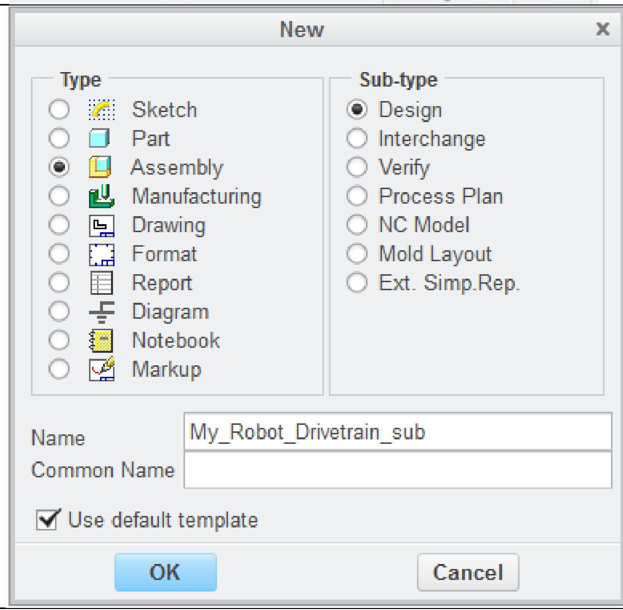
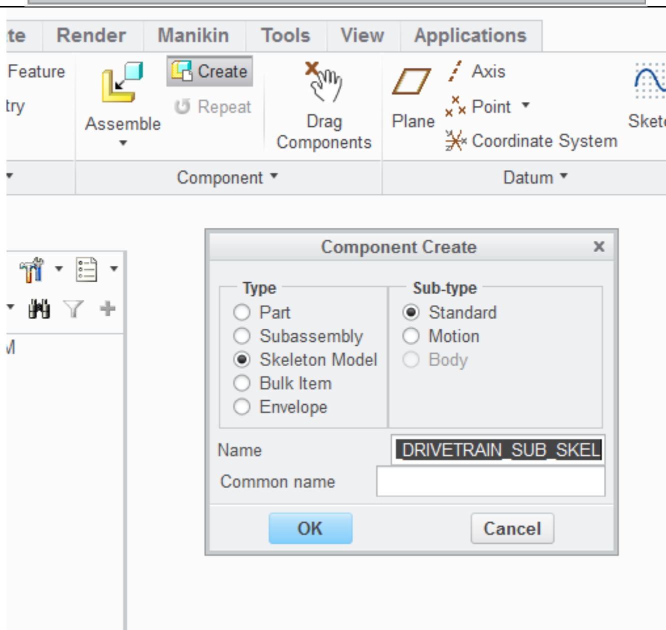
Right click on the skeleton file in the model tree and select Hide.</td>
<td>
</td>
</tr>
</table>

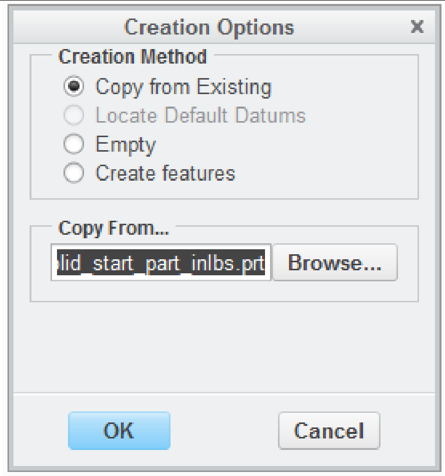
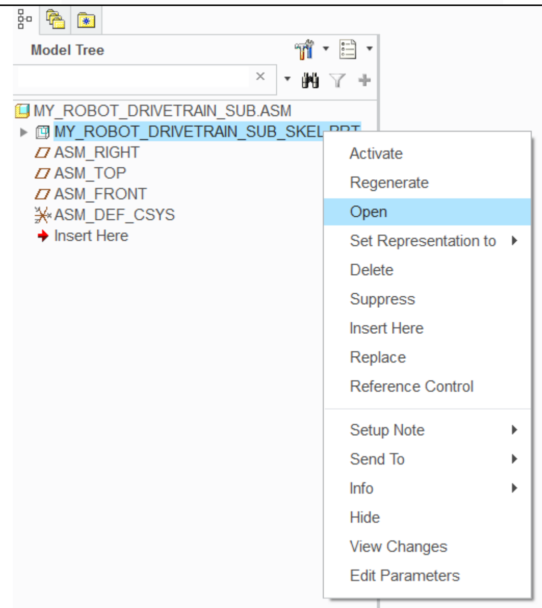
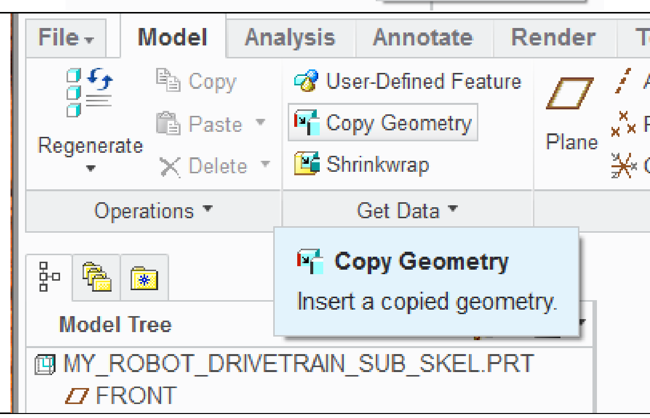
The structure subassembly is now finished. Save the file and close it to see it placed in the overall system model.	
The next step is to create the drive train subassembly.	

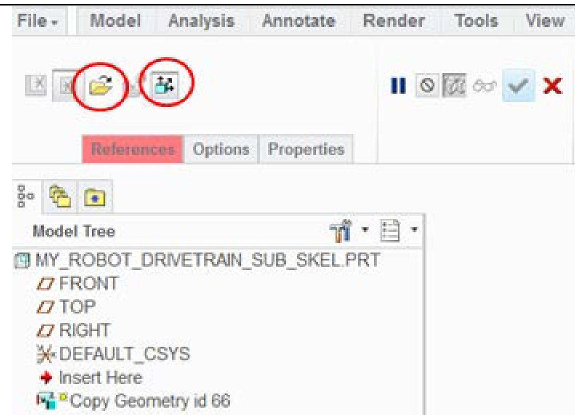
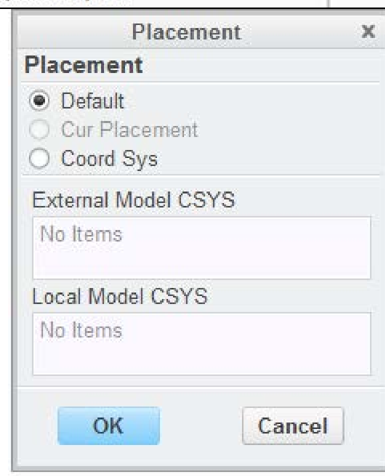
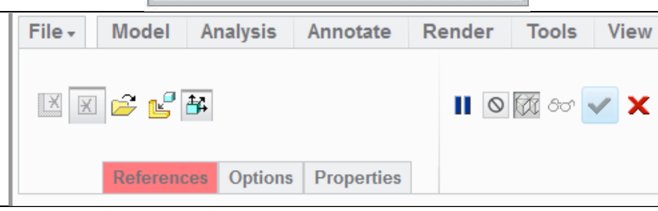
You have finished the exercise.

EXERCISE 10.0: DESIGNING THE DRIVETRAIN

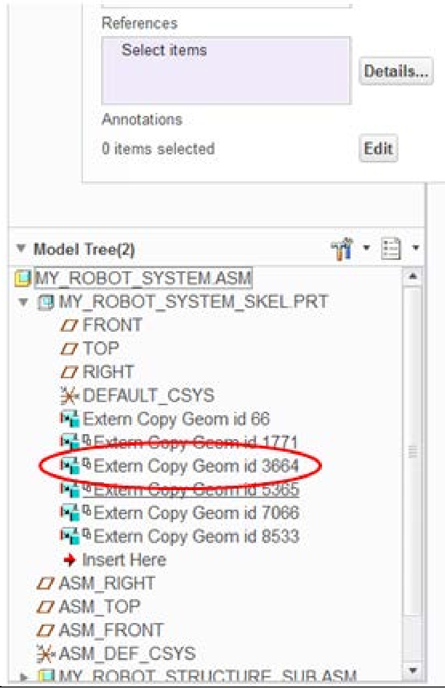
This document provides detailed instructions for designing the drivetrain subsystem for the PTC robot.

Close all of your model files and erase from memory by selecting **Erase Not Displayed** and clicking **OK.**	
Create a new assembly file by selecting **New** and then clicking the **Assembly** button and then name your file **"My_Robot_Drivetrain_sub"** and click **OK.**	
Once again, we want to make sure we have the same references as our system model, so we will create a skeleton file that contains our references. Select **Create** in the upper menu and pick **Skeleton Model** and then click **OK**.	

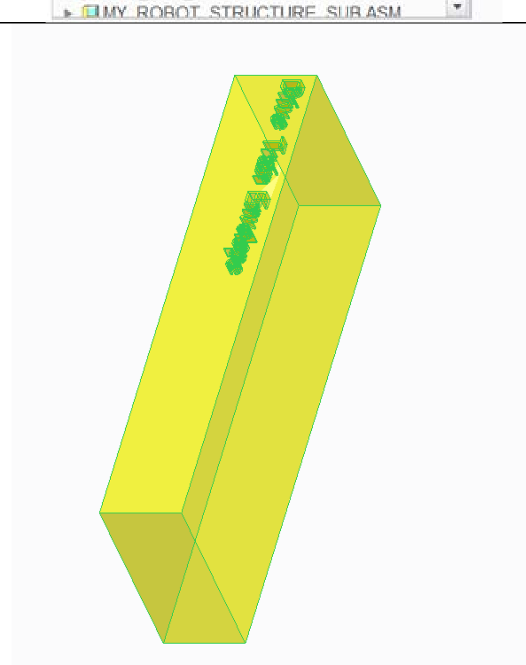
<table>
<tr><td>Select Copy from Existing and click OK.</td><td>
</td></tr>
<tr><td>Right click on the skeleton file in the model tree and then select Open.</td><td>
</td></tr>
<tr><td>Select Copy Geometry in the upper menu.</td><td>
</td></tr>
</table>

Now de-select the publish geometry icon and then click on the folder icon to open the system model. Select "**my_robot_system.asm**" and then click **Open**.	
Keep the **Default** settings and click **OK**.	
Click on the **red** **References** tab.	

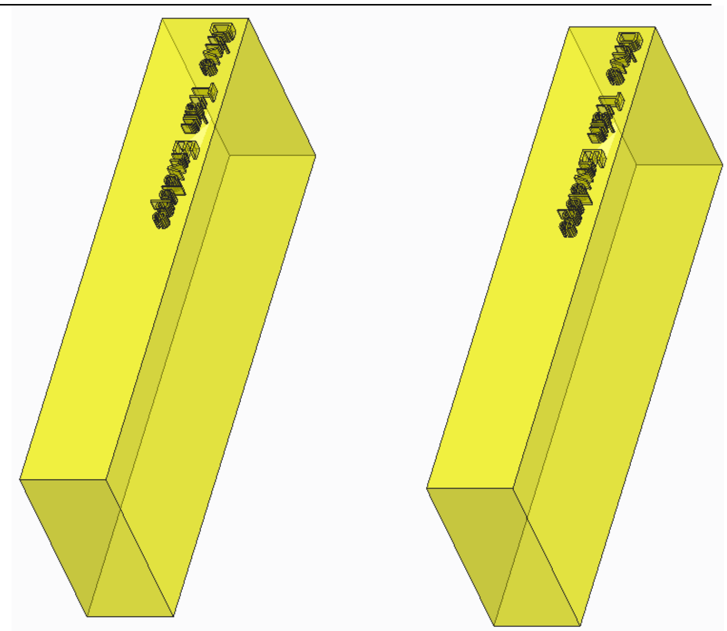
Click in the **References** area and then expand the skeleton file by clicking on the arrow to the left of **MY_ROBOT_SYSTEM_SKEL.PRT** and right click on the third **Extern Copy Geom** item in the tree.



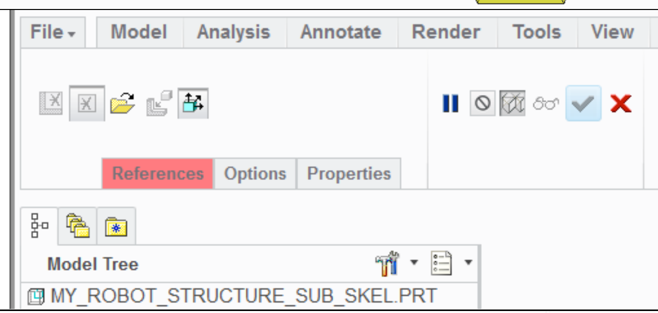
This imports the right drive train envelope into the skeleton file. Now do the same thing to bring the left drive train envelope into the skeleton file, (It is the fourth **Extern Copy Geom** item in the tree).



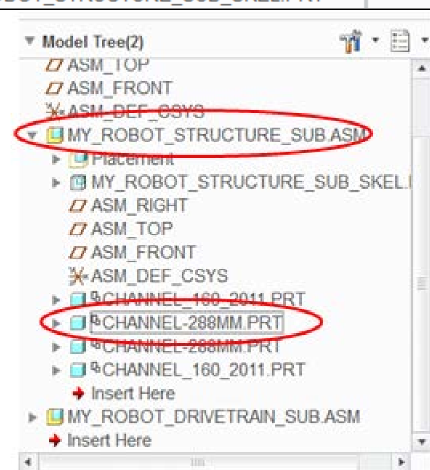
You should now have the two drive train envelopes in your skeleton file. We want to add some other reference geometry into the skeleton file so that we know how to orient our design with respect to the structure subassembly.

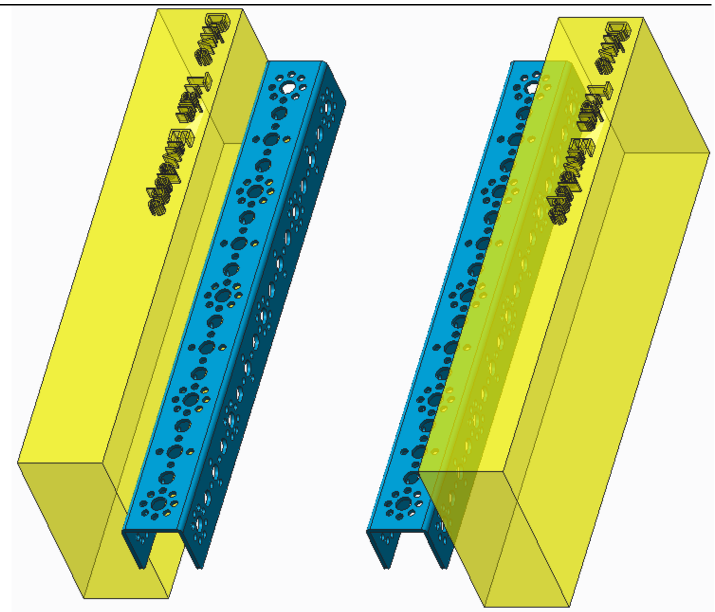


So once again click on the **Copy Geometry** icon in the upper menu and de-select the publish icon and click on the folder icon and select **my_robot_system.asm** and **Open** it. Keep the **Default** orientation.

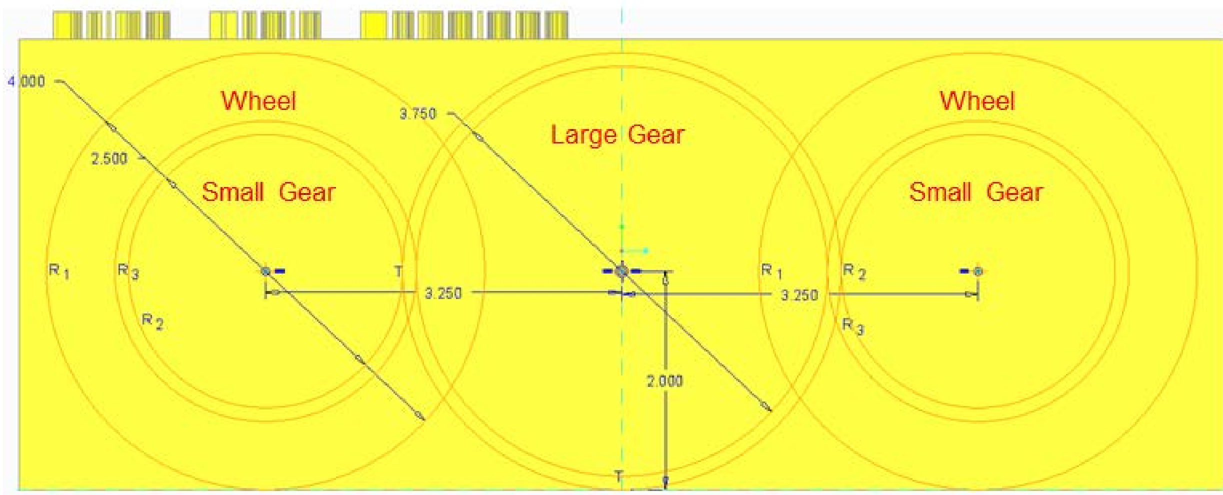


Click on the **red** **References** tab and then click in the **Surface Sets** area. Expand the **MY_ROBOT_STRUCTURE_SUB.ASM** in the model tree and then left click on the first **CHANNEL-288MM.PRT** to import it into the skeleton file. Now click the green check mark to finish.



Do the same procedure to import the second 288mm channel into the skeleton file. *This imports these channels into the skeleton file where they will only be references and not add mass or geometry to the drivetrain subassembly but will allow us to design the drivetrain so that it fits appropriately into the structure subassembly.* Now save the skeleton file and close it. We are now ready to begin creating the drivetrain subassembly.	

To begin, let's look at the technology sketches that were created for this subassembly.



The concept here is to have a large gear in the center that drives two smaller gears connected to the wheels. Notice that the large gear needs to be approximately 3.75 inches in diameter and the small gears 2.5 inches in diameter. The wheels are around 4 inches in diameter as well. So let's figure out how fast the wheels will rotate in comparison to the large center gear.

Every time the large gear rotates once, it travels the length of its circumference $2\pi r$. Using the diameter of the large gear that length would be _________. Every time a small gear turns once, it travels the length of its circumference which would be ___________.

Since the large gear travels farther than the smaller gears, for every turn of the large gear, the smaller gears will turn more than one revolution. But we need to know how much more. This can be calculated by using ratios. Use the following ratio to determine how many revolutions the small gears turn for every one revolution of the large gear:

$$\frac{circumference\ of\ the\ large\ gear}{circumference\ of\ the\ small\ gear} = \frac{2\pi r_{large}}{2\pi r_{small}} = \frac{r_{large}}{r_{small}} =$$

Notice that the revolutions of the smaller gears are greater than 1. That means that the RPMs (revolutions per minute are increasing as you go from the large gear to the smaller gears).

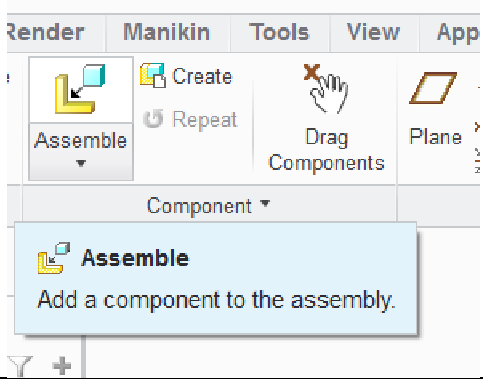
Now, since the wheels are attached to the smaller gears, they have the same RPMs as the smaller gears and yet they are larger than the large gear. Their circumference is _______.

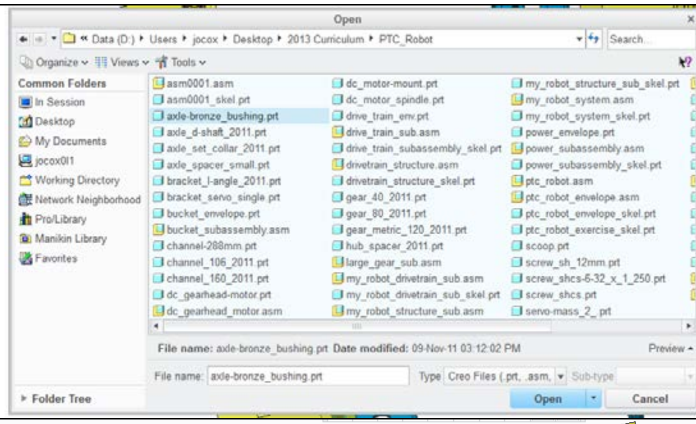
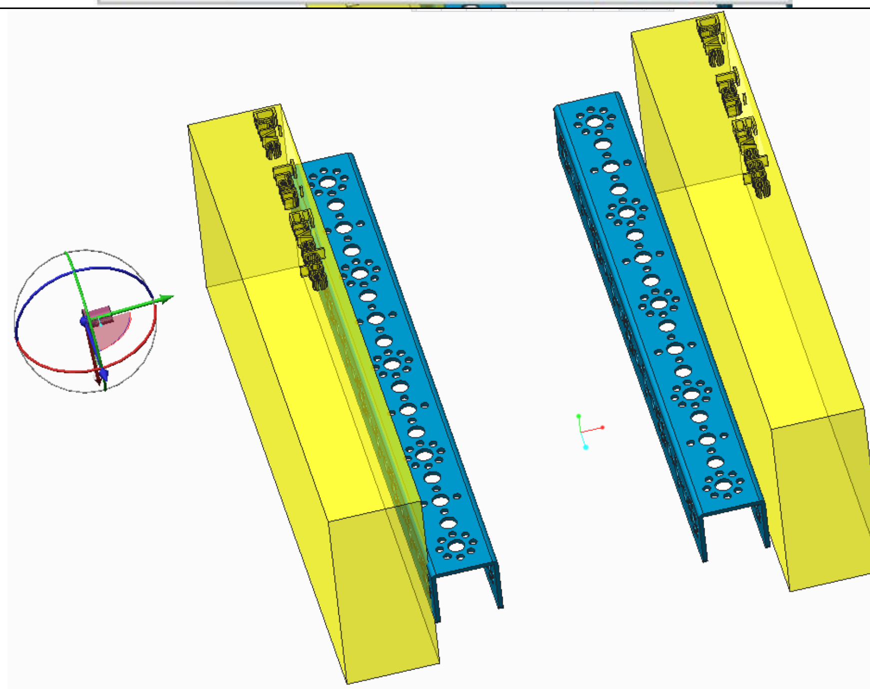
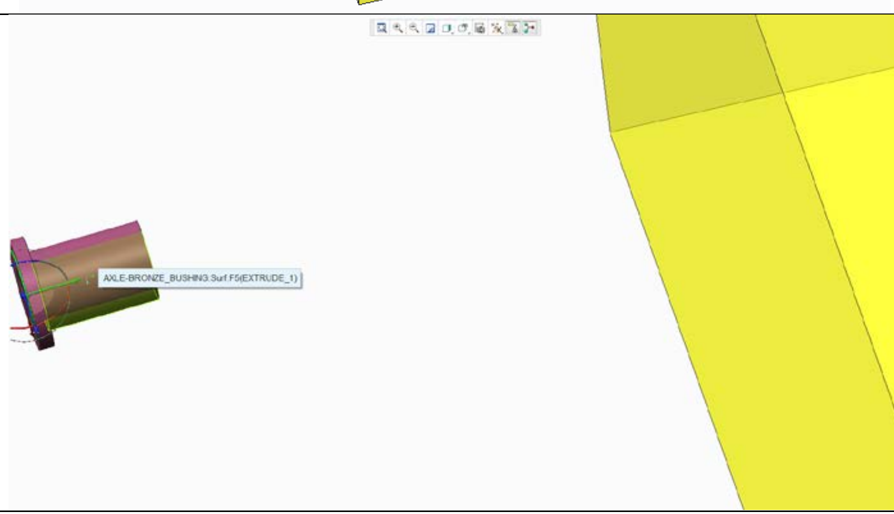
So now let's figure out how fast the wheels will be turning and the robot moving for every one revolution of the large gear.

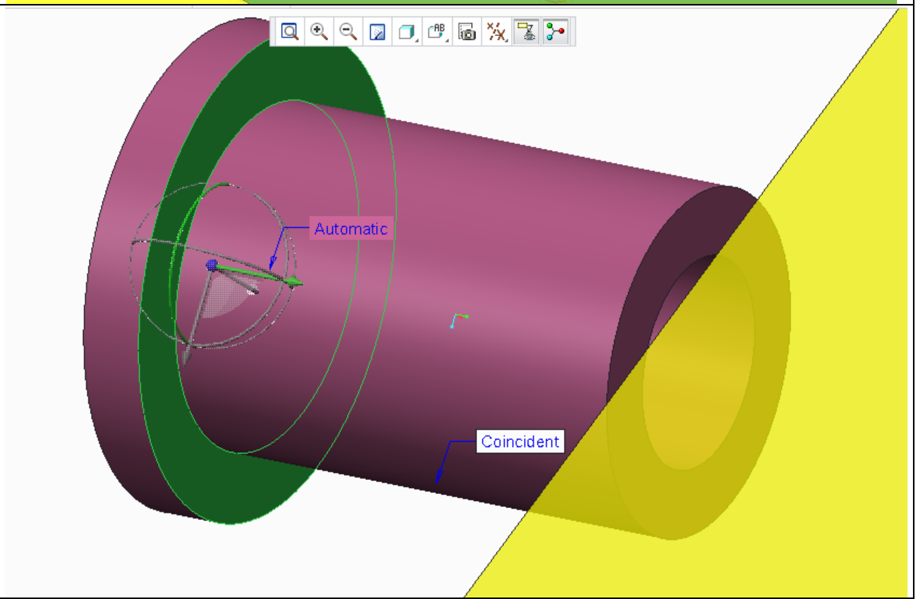
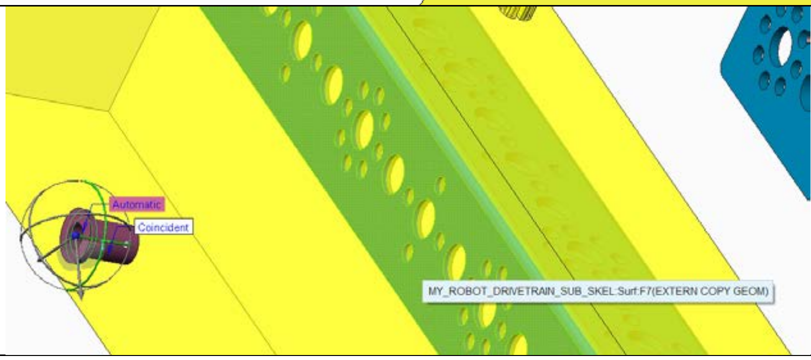
$$\left(\frac{r_{large}}{r_{small}}\right) circumference_{wheel} =$$

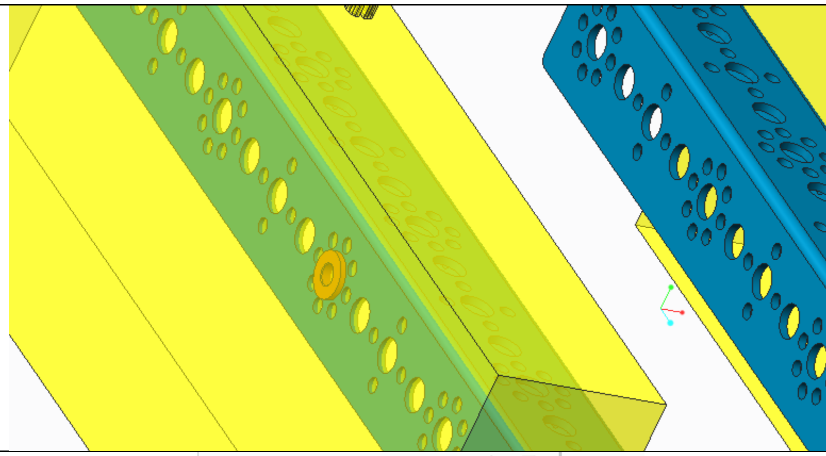
So now if the large gear turns at 30 RPM, how fast would the robot travel?

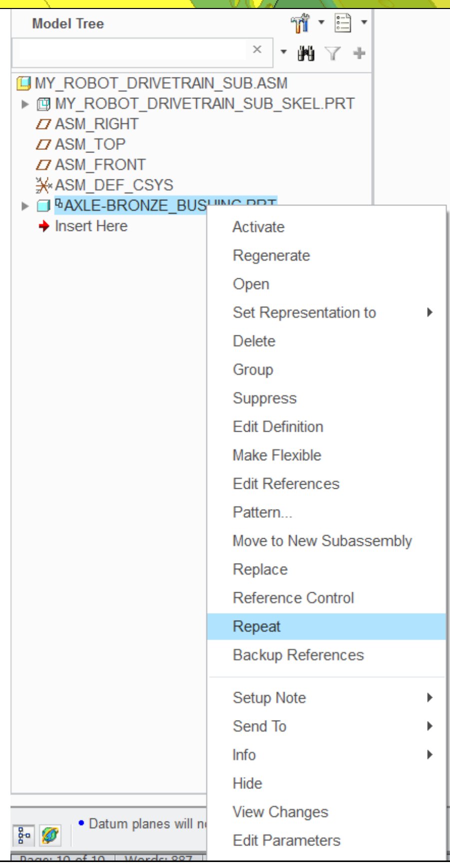
Now that we have an understanding of the gearing, let's create the drive train subassembly.

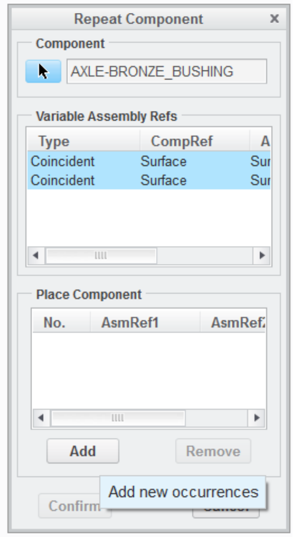
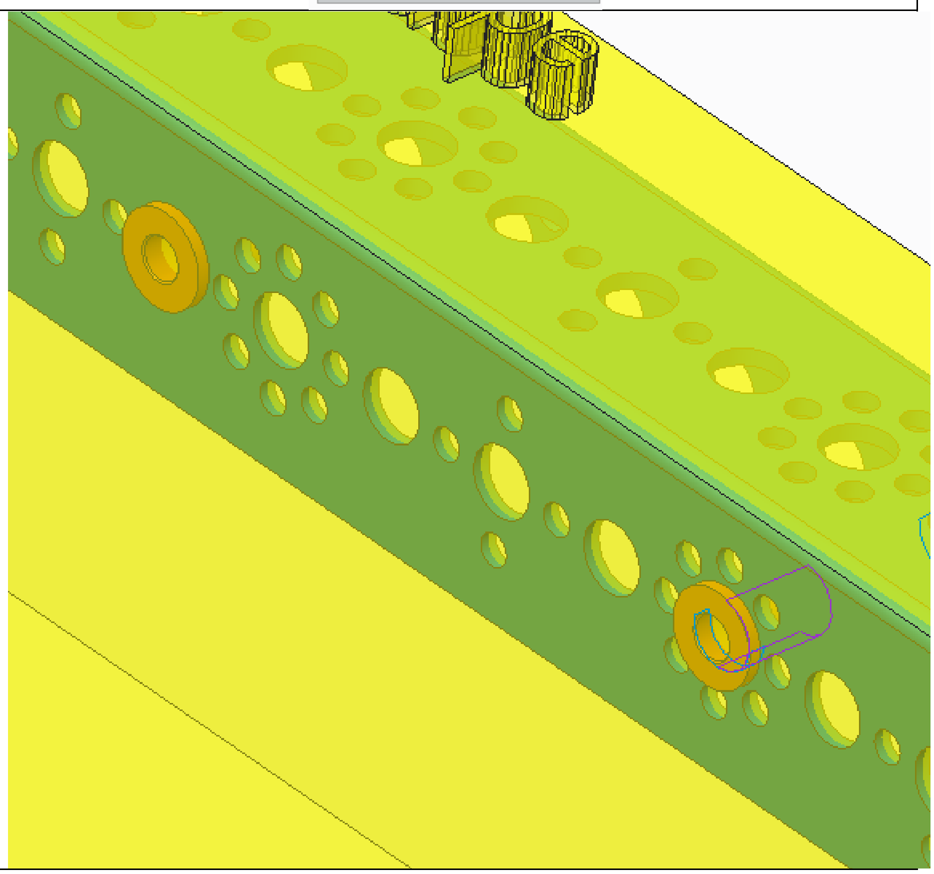
Click on the **Assemble** icon in the upper menu to open the library of parts.	

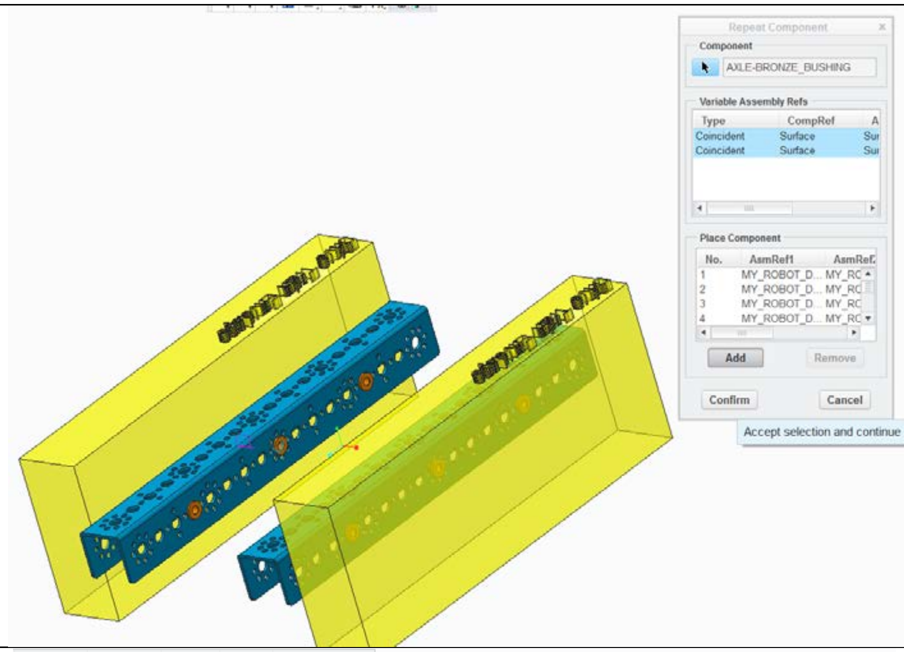
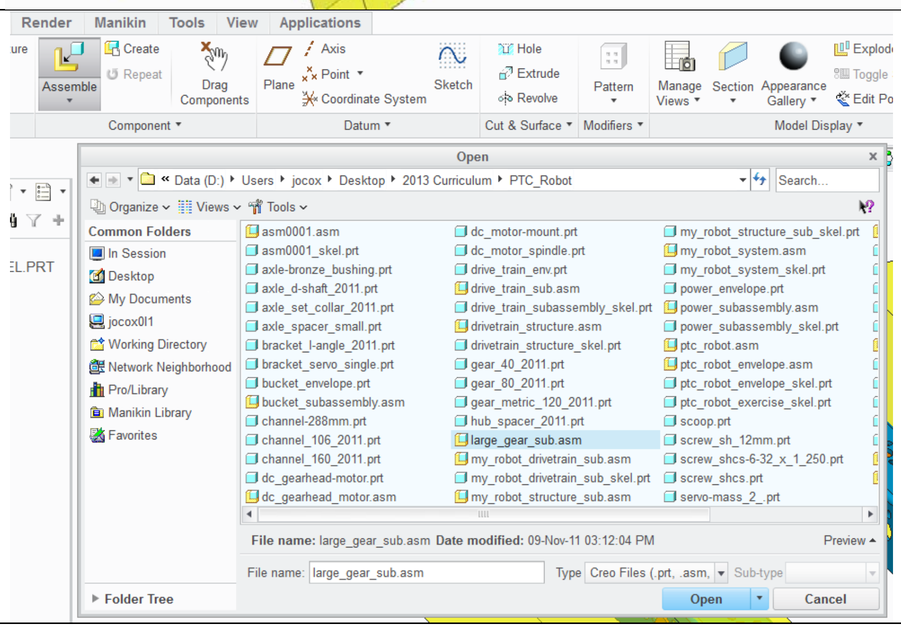
Select **axle-bronze_bushing.prt** and click **Open**.	
Left click to temporarily place the bronze bushing and then use the orientation sphere to orient the bushing as shown.	
Left click to select the cylindrical side of the bushing.	

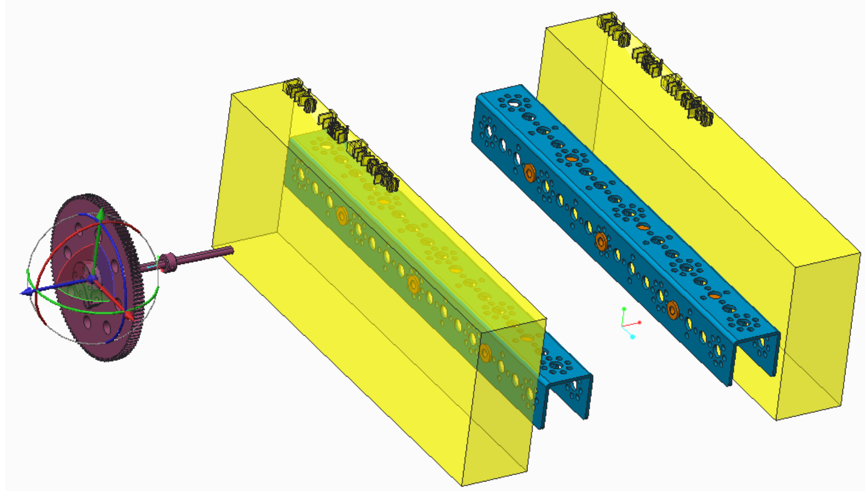
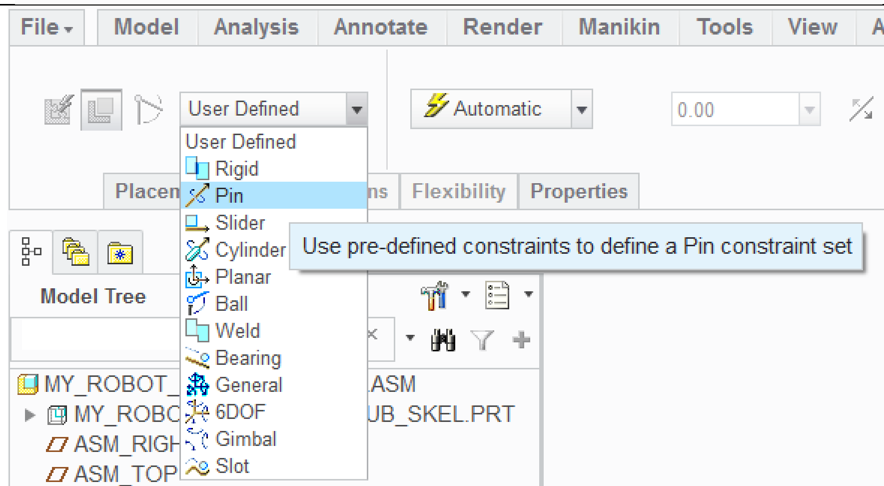
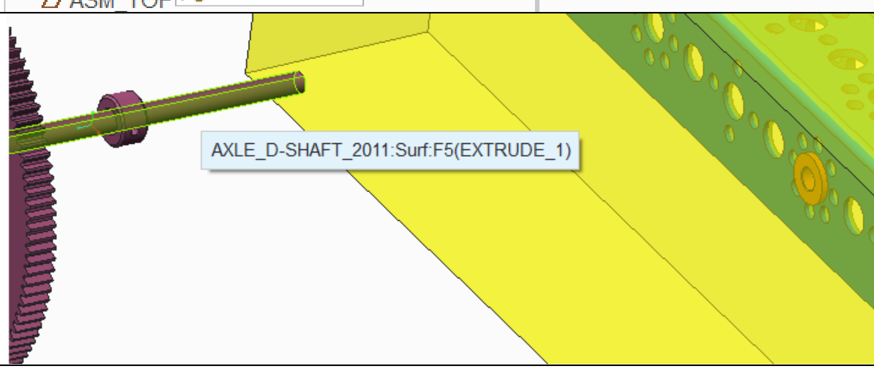
Now select the cylindrical inside of the large hole in the middle of the channel. *Sometimes you may need to right click to toggle through the different surfaces until you get the right one and then left click.*	
Now select the underside of the head of the bushing as shown.	
And then select the side of the channel so that the bushing sits against it.	

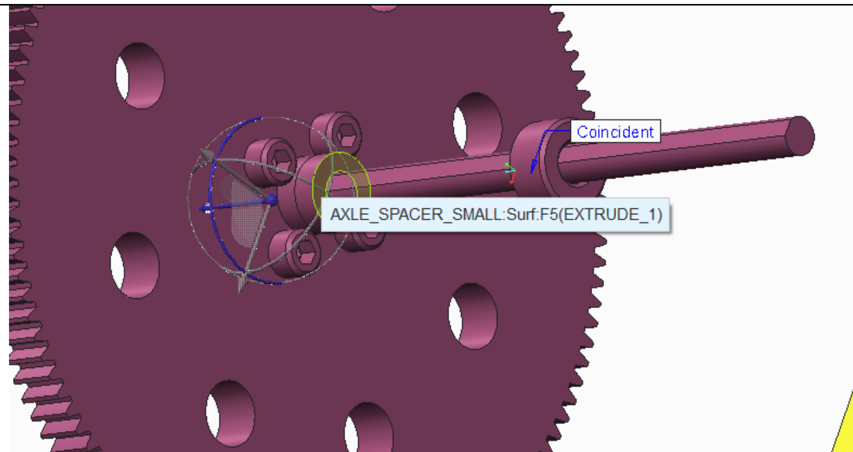
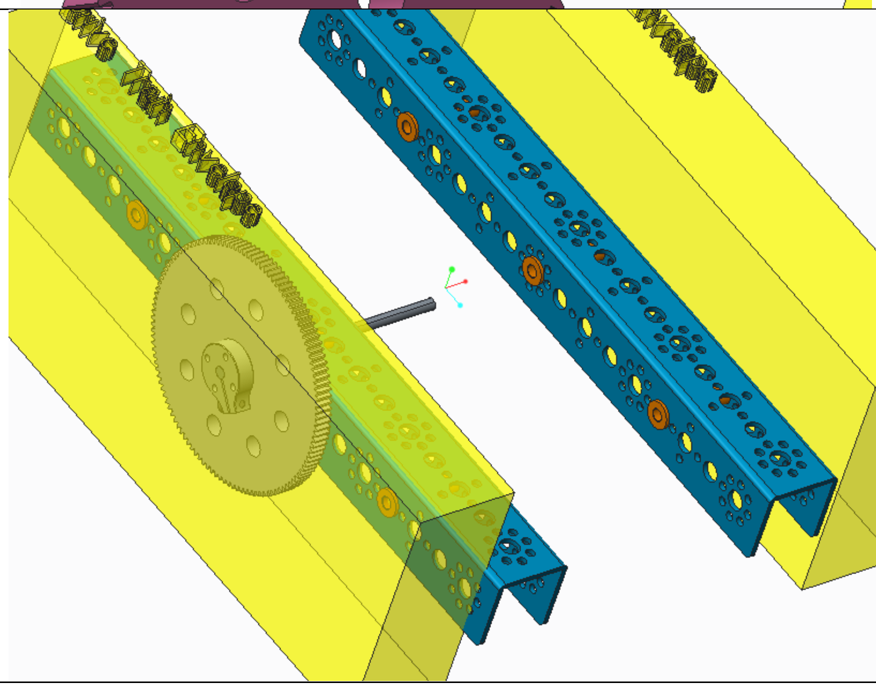
<table>
<tr>
<td>This is the first of 12 bushings that need to be placed in the channels so that the axles of the gears and wheels will spin correctly.</td>
<td></td>
</tr>
<tr>
<td>To place the other 11 bushings, right click on the AXLE-BRONZE_BUSHING.PRT in the model tree and select Repeat.

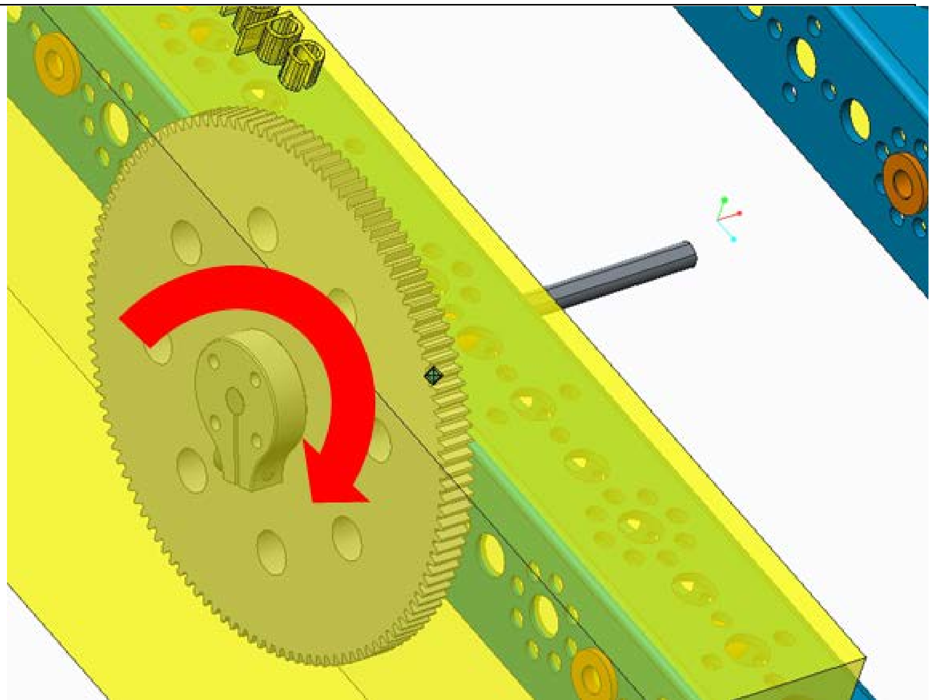
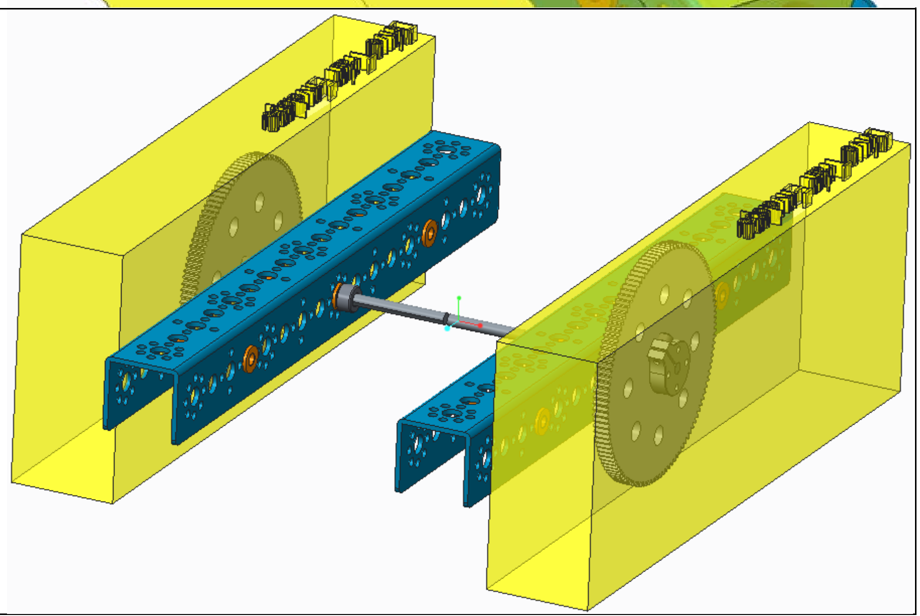
This will allow us to place the other 11 bushings quickly into the assembly.</td>
<td>
</td>
</tr>
</table>

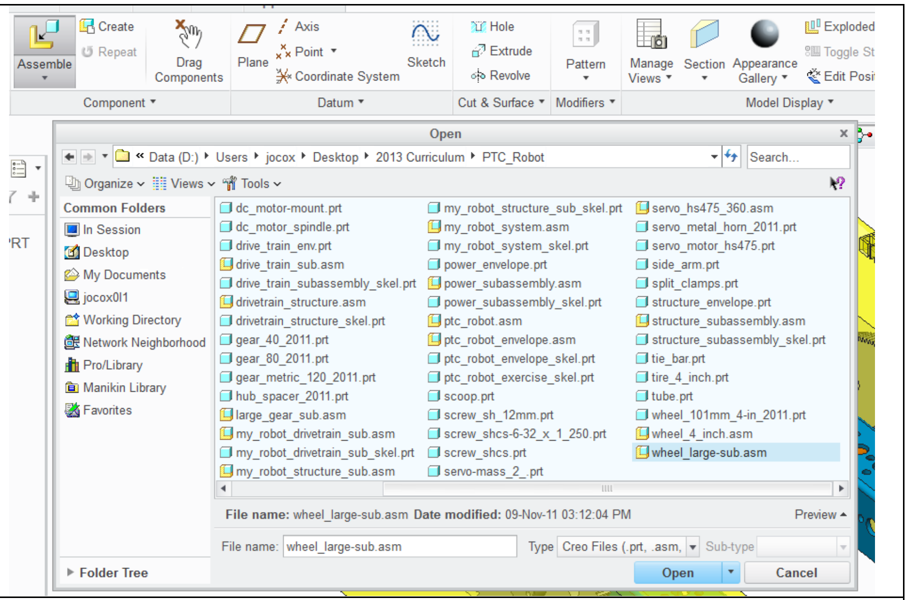
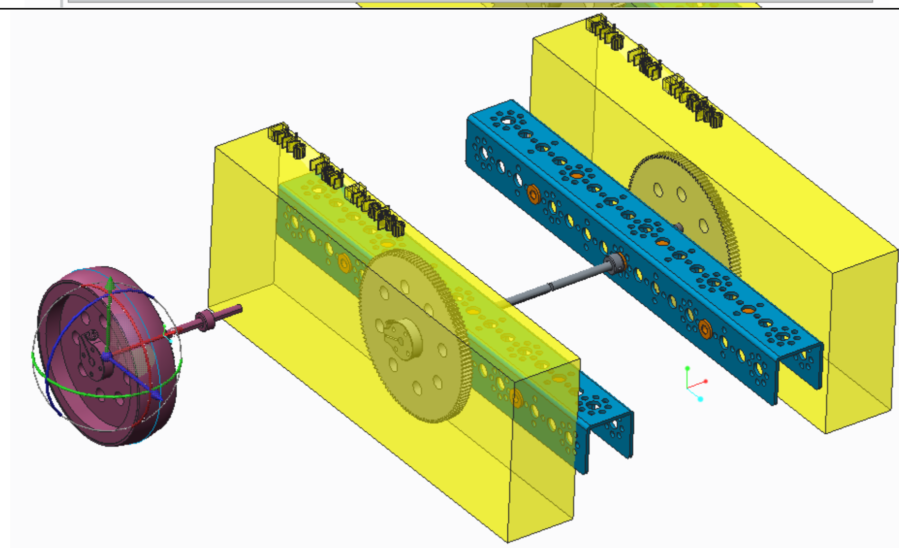
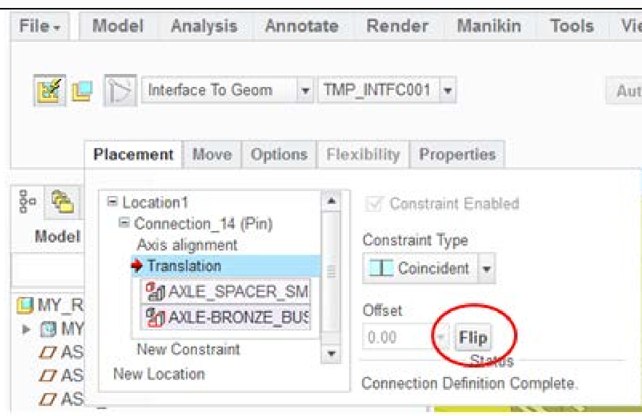
<table>
<tr>
<td>You will see a dialog box showing the two constraints that we used to place the original bushing. Left click to select both of them. Then click Add to start placing the other bushings.</td>
<td>
</td>
</tr>
<tr>
<td>Once you have clicked the Add button, count 5 holes from the original bushing and left click to select the hole and then the face of the channel, (remember you can right click to toggle through the other surfaces until the face of the channel becomes available and then left click to select).</td>
<td></td>
</tr>
</table>

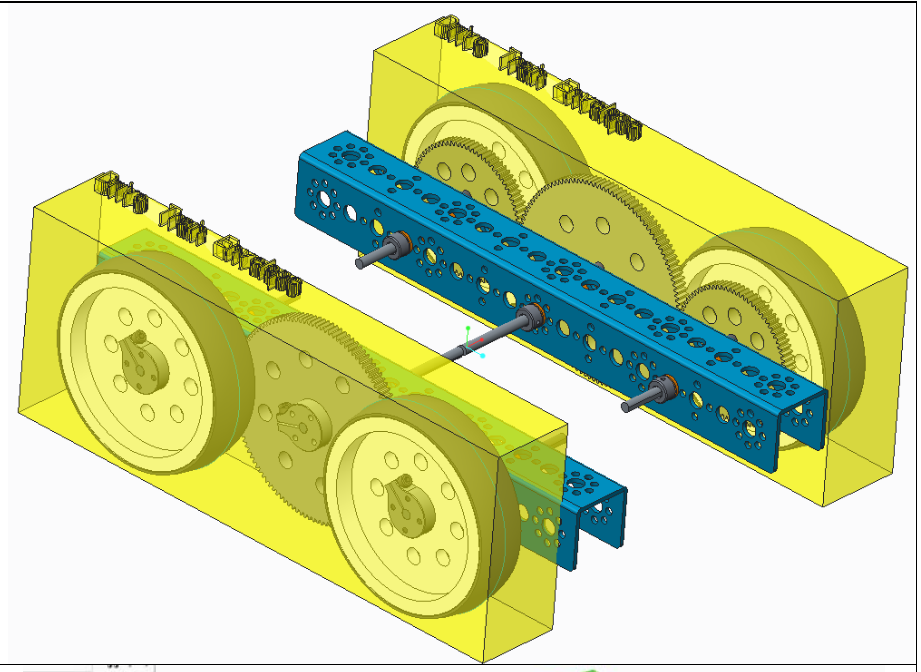
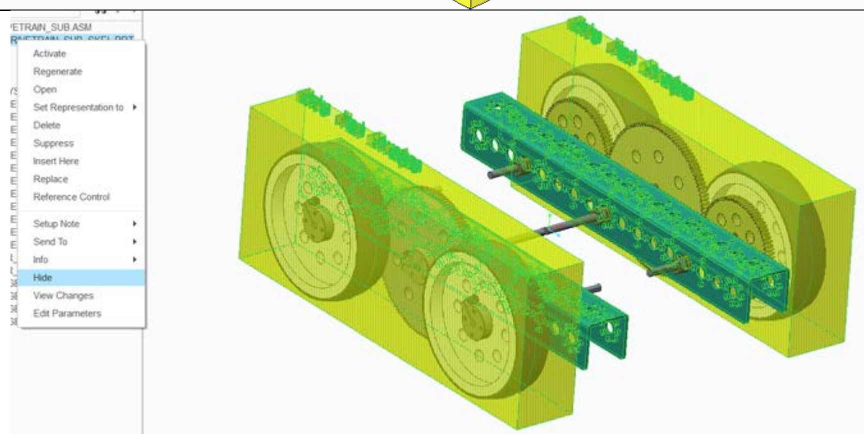
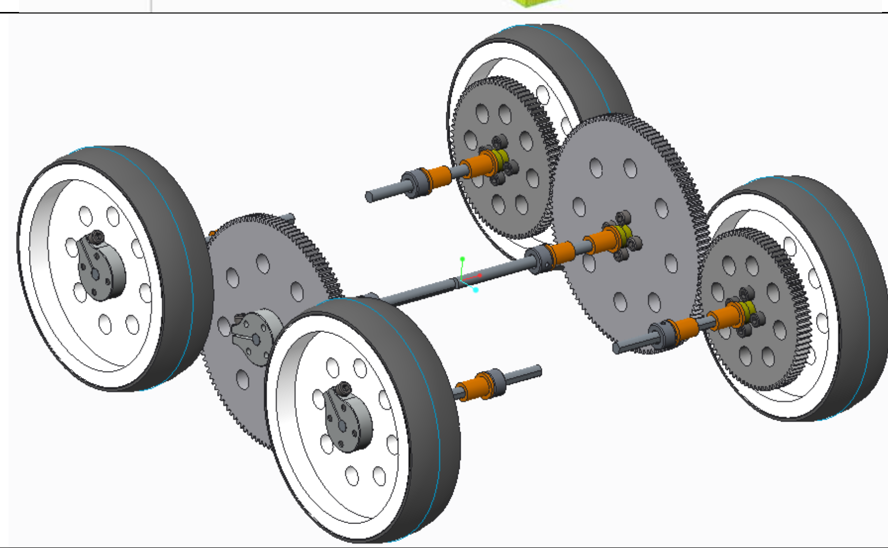
<table>
<tr>
<td>Continue selecting the hole and then the face to place a total of 12 bushings as shown. Then click Confirm to finish the placements.</td>
<td>
</td>
</tr>
<tr>
<td>Now select Assemble again in the upper menu and then select Large_gear_sub.asm from the library of parts and click Open.</td>
<td>
</td>
</tr>
</table>

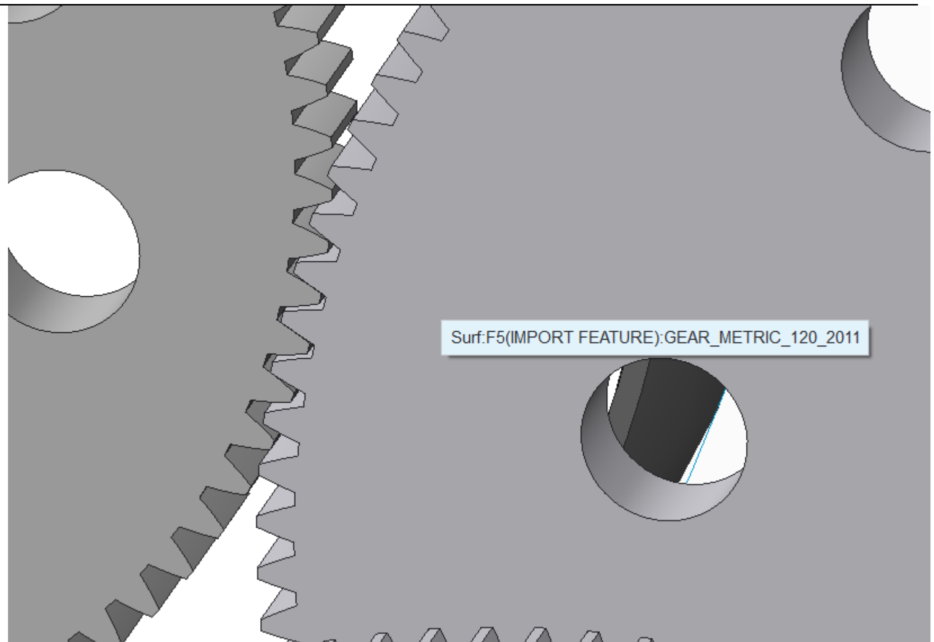
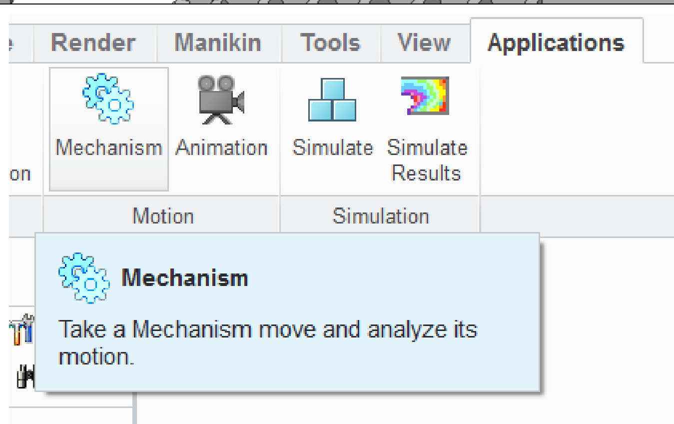
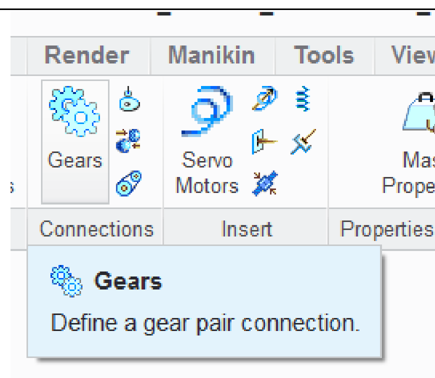
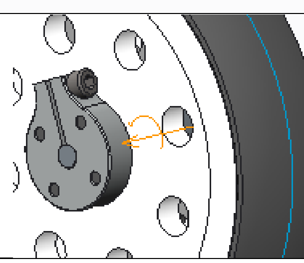
<table>
<tr>
<td>Left click to temporarily place the large gear subassembly and then use the orientation sphere to orient the gear as shown.</td>
<td></td>
</tr>
<tr>
<td>In constraining the gear, we want it to be able to rotate. This requires a kinematic constraint. So open the drop down menu titled “User Defined” in the upper menu and then select a “Pin” constraint.</td>
<td>
</td>
</tr>
<tr>
<td>Left click to select the cylindrical shaft (make sure you select the entire shaft and not simply an edge). Then select the hole of the middle bushing.</td>
<td>
</td>
</tr>
</table>

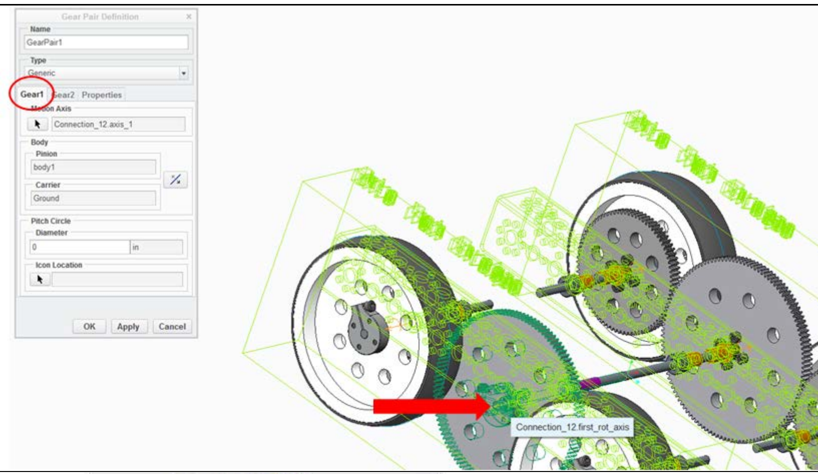
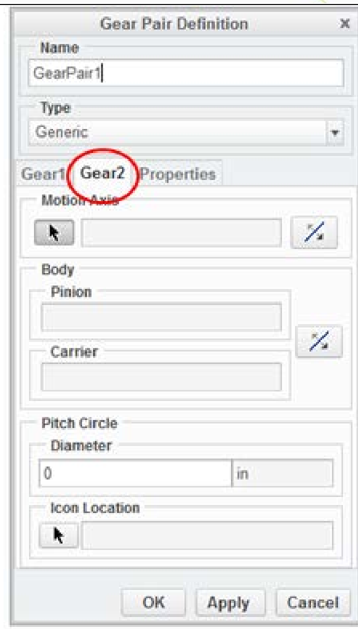
Then select the flat face of the gear bushing and the flat face of the axle bushing to indicate that these faces should touch each other.	
Now click the green check mark in the upper menu to finish placing the gear.	

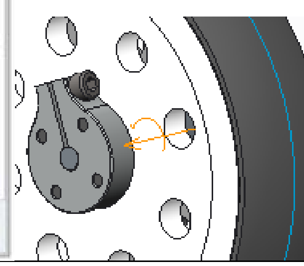
Now check to make sure that the gear rotates by holding down the **CTRL-ALT** keys and left click on the gear to rotate it.	
Now use the same procedure to place a second large gear on the other side as shown.	

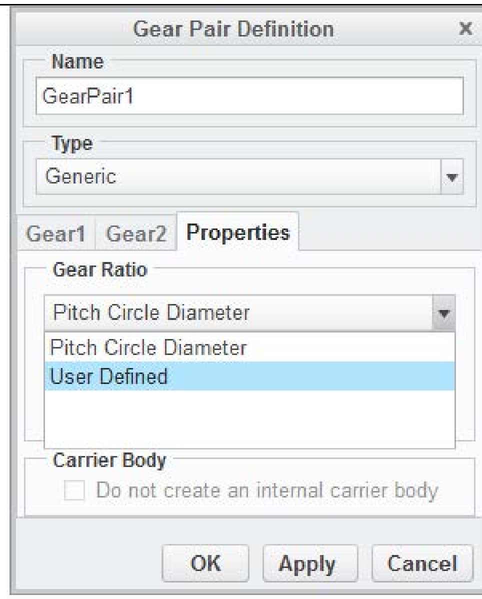
<table>
<tr>
<td>Select Assemble again in the upper menu and then select wheel_large-sub.asm in the library of parts and click Open.</td>
<td>
</td>
</tr>
<tr>
<td>Left click to place the wheel and then use the same Pin constraint to place wheels on both sides of the large gear for both channels.</td>
<td></td>
</tr>
<tr>
<td>Sometimes the wheels may be reversed when they are placed. You can flip them using the flip button under the Placement tab.</td>
<td>
</td>
</tr>
</table>

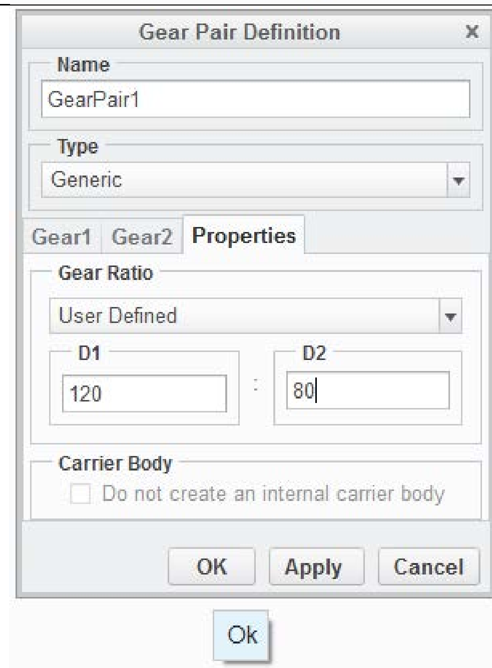
The drive train should look as shown and all the wheels and gears should rotate independently.	
Now that we have the drive train assembled, let's hide all of the reference geometry *(the envelopes and channels)*. Right click on the skeleton file in the model tree and select **Hide**.	
Now you should just see the geometry of the drive train.	

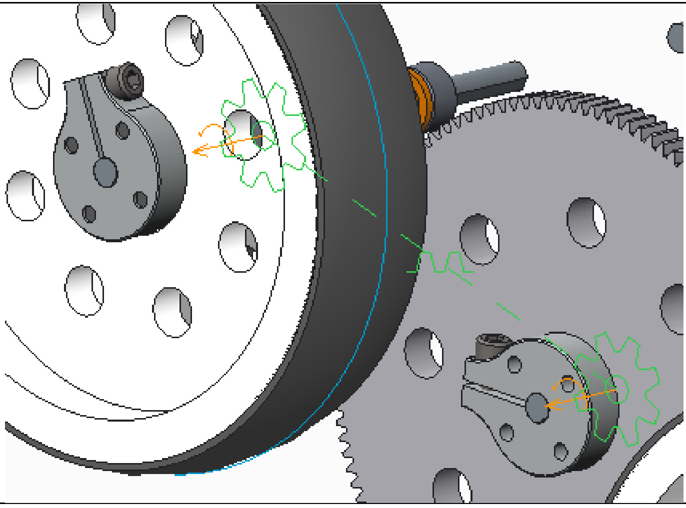
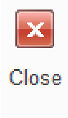
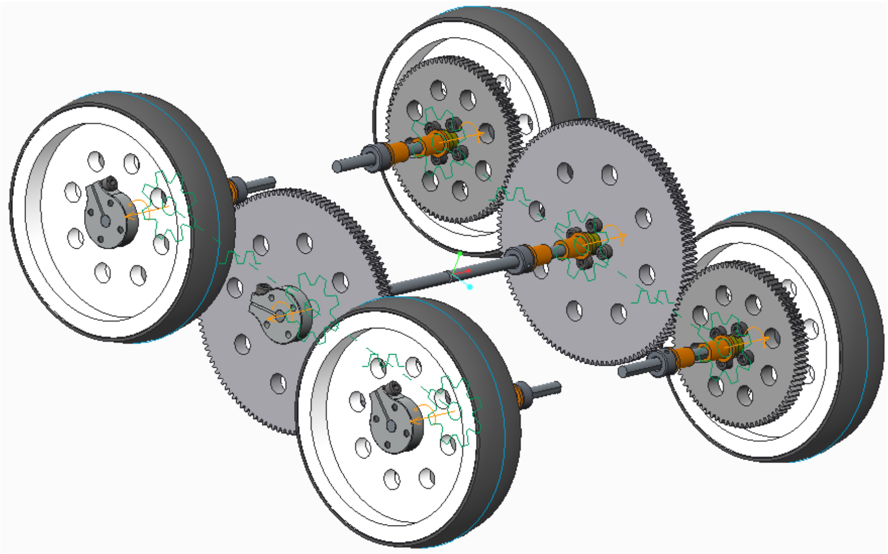
Now use the **CTRL-ALT** feature to rotate the gears so that they are properly meshed.	
In order for the wheels and gears to work as they should, we need to create gear pairs. To do this we need to go into the **Mechanism Application**. Click on the **Application** tab at the top of the screen and then select **Mechanism**.	
Now select the **Gears** icon in the upper menu to define a gear pair connection.	
When you enter the Mechanism application, all of the kinematic connections show up with new icons. You can see each of the pin joints	

<table>
<tr><td>have an axis icon. We will use these axis icons to select the pin joints.</td><td></td></tr>
<tr><td>Select the pin connection icon for the large gear for Gear 1 by left clicking on the icon, (All the graphics change, don’t worry about that).</td><td></td></tr>
<tr><td>Then click on the Gear 2 tab in the dialog box and select one of the wheel pin icons for the second gear in the gear pair.</td><td>

</td></tr>
</table>

<table>
<tr>
<td>Now click on the Properties tab in the dialog box and then select User Defined under the Gear Ratio</td>
<td></td>
<td>
</td>
</tr>
<tr>
<td>We can use the number of teeth on the gears as a way of characterizing the relationship between the gears. The large gear has 120 teeth and the smaller gears have 80 teeth.

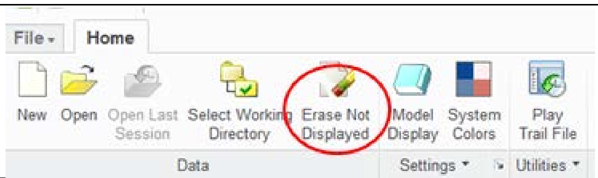
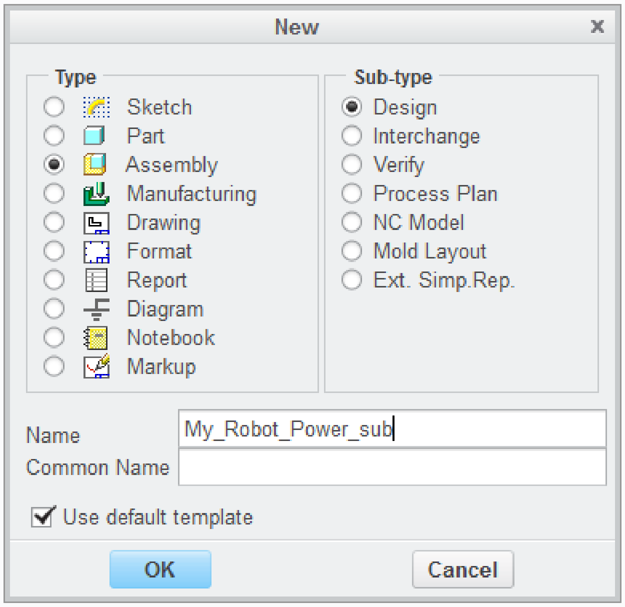
Set those values in the appropriate fields and click OK. (D1 is for the first gear that you selected and D2 is for the second gear)</td>
<td></td>
<td>
</td>
</tr>
</table>

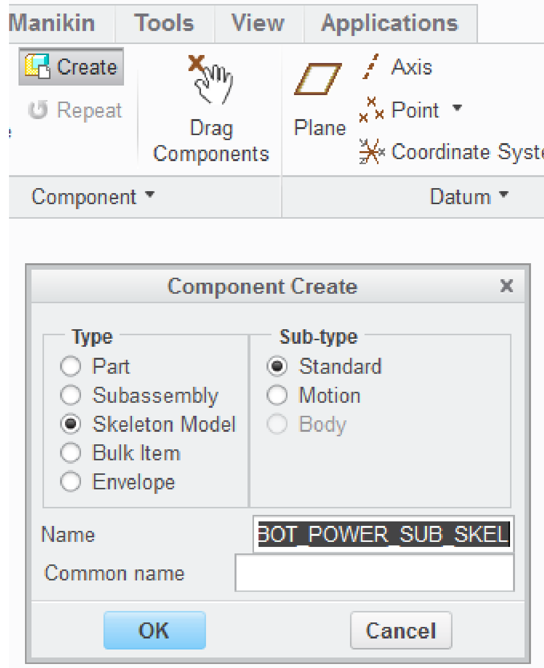
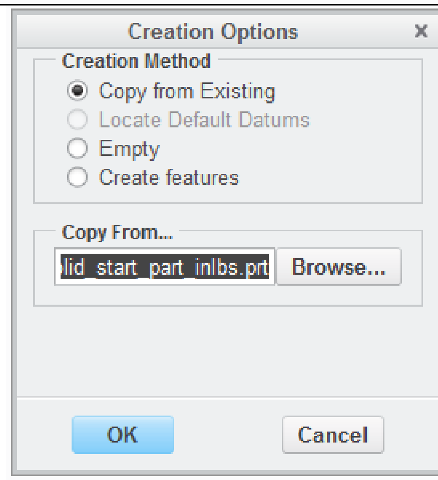
You will now see a green icon with gears and a dashed line connection signifying your gear pair connection. Follow the same procedure to create gear pairs between all of the wheels and the large middle gears.	
When you are done, all your wheels should be driven from the two large gears in the middle. Test them out using the **CTRL-ALT** keys and left clicking to move the middle gears. Now exit the mechanism application by clicking on the close icon. 	
The drive train subassembly is complete so **Save** your model and **Close** the file.	

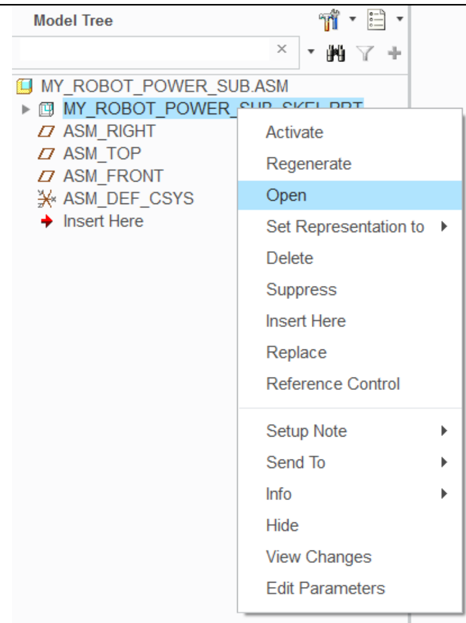
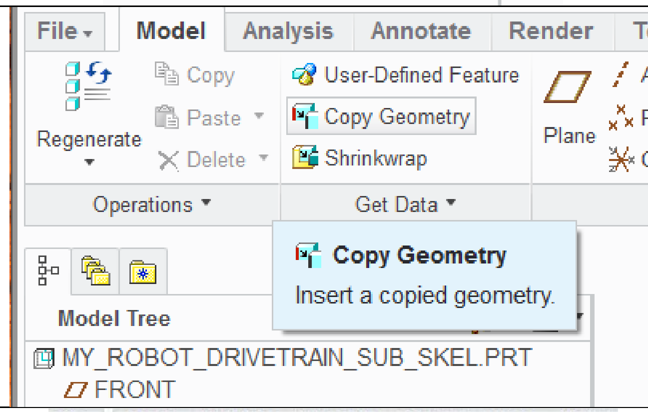
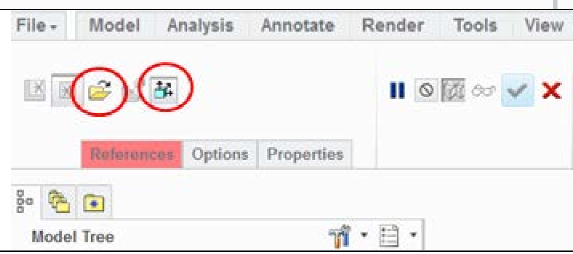
You have finished the exercise.

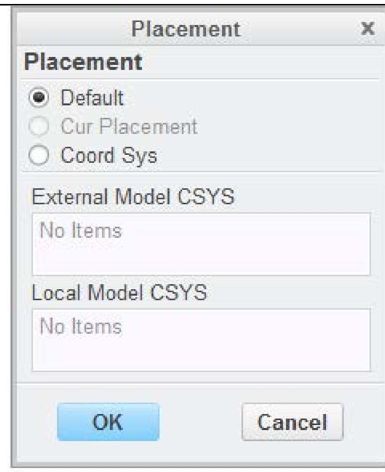
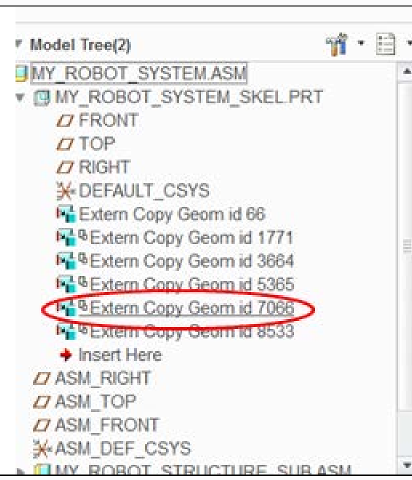
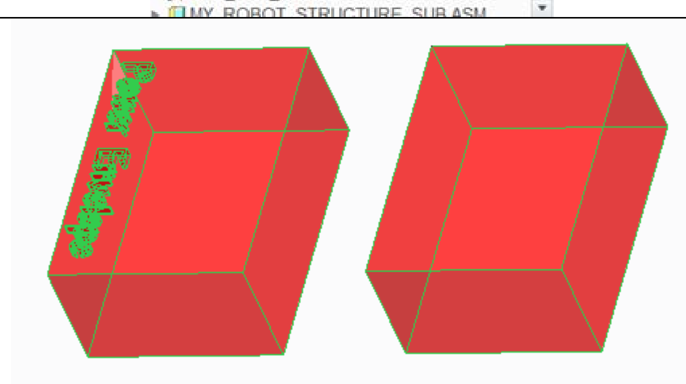
EXERCISE 11.0: THE POWER SUB-SYSTEM

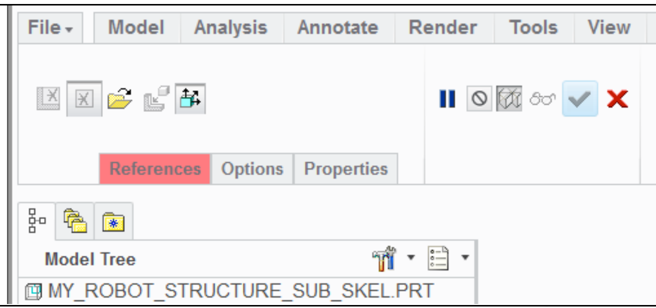
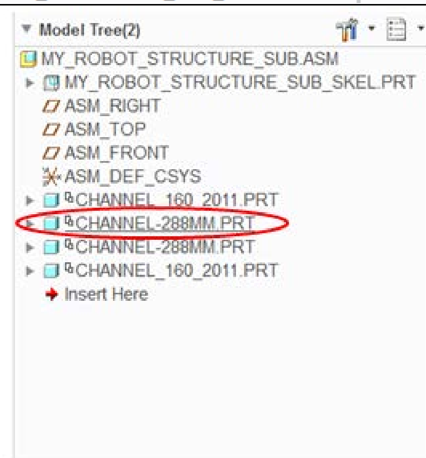
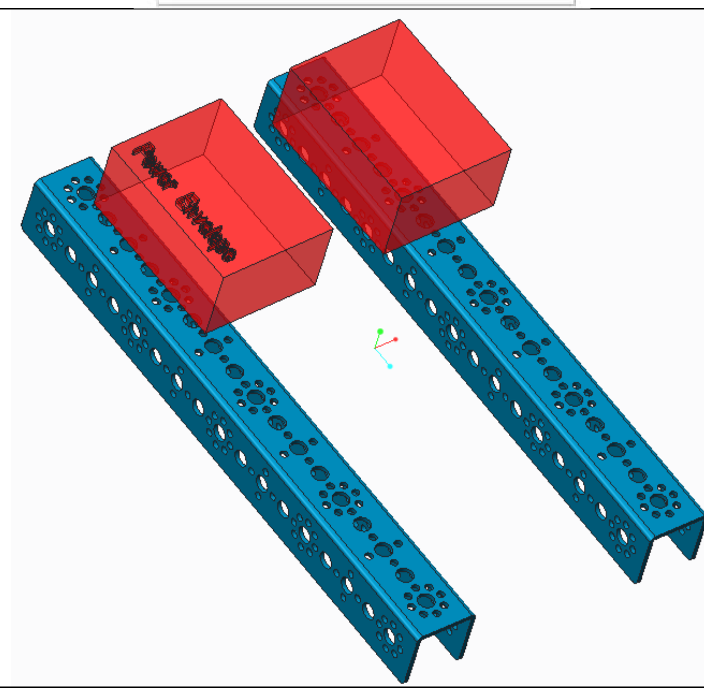
This document provides detailed instructions for designing the power subsystem for the PTC robot.

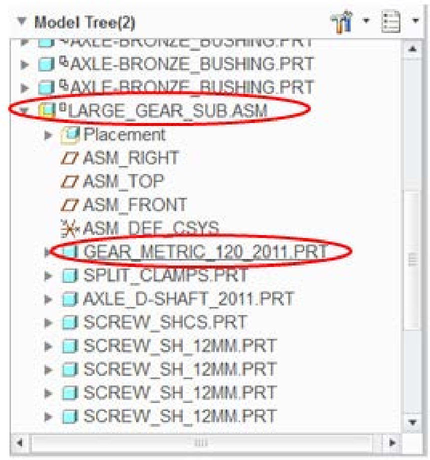
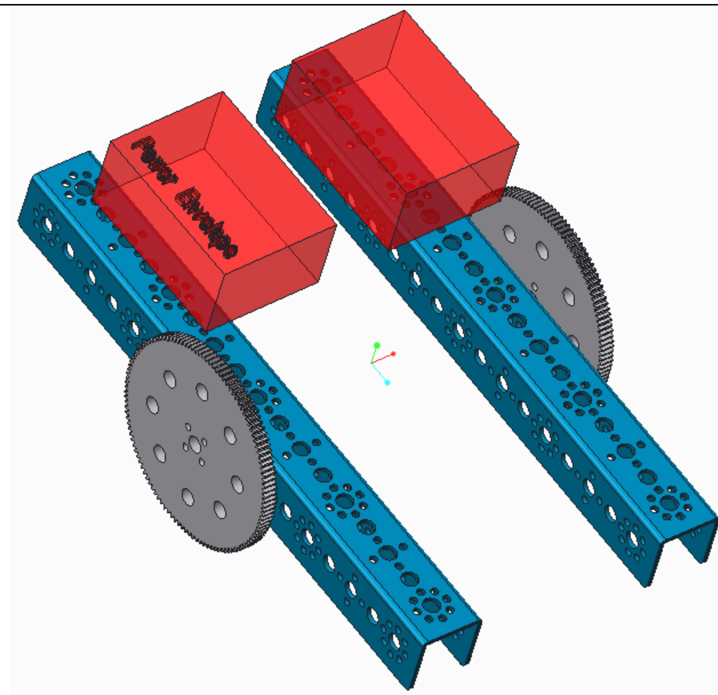
<table>
<tr>
<td>Close all of your model files and erase from memory by selecting Erase Not Displayed and clicking OK.</td>
<td>
</td>
</tr>
<tr>
<td>Create a new assembly file by selecting New and then clicking the Assembly button and then name your file “My_Robot_Power_sub” and click OK.</td>
<td>
</td>
</tr>
</table>

Once again, we want to make sure we have the same references as our system model, so we will create a skeleton file that contains our references. Select **Create** in the upper menu and pick **Skeleton Model** and then click **OK**.	
Select **Copy from Existing** and click **OK**.	

<table>
<tr><td>Right click on the skeleton file in the model tree and then select Open.</td><td>
</td></tr>
<tr><td>Select Copy Geometry in the upper menu.</td><td>
</td></tr>
<tr><td>Now de-select the publish geometry icon and then click on the folder icon to open the system model. Select "my_robot_system.asm" and then click Open.</td><td>
</td></tr>
</table>

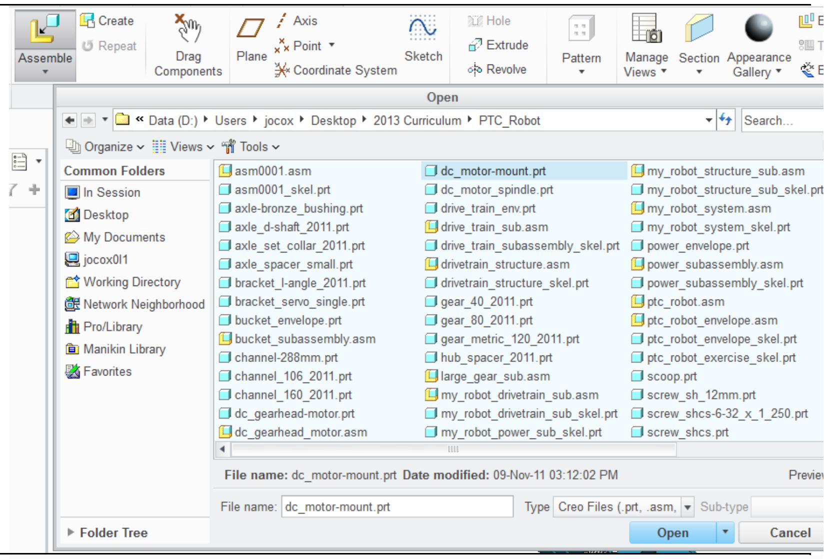
<table>
<tr><td>Keep the Default settings and click OK.</td><td>
</td></tr>
<tr><td>Click on the red References tab.</td><td>
</td></tr>
<tr><td>Click in the References area and then expand the skeleton file by clicking on the arrow to the left of MY_ROBOT_SYSTEM_SKEL.PRT and right click on the fifth Extern Copy Geom item in the tree.</td><td>
</td></tr>
<tr><td>This imports the Power envelopes into the skeleton file.</td><td></td></tr>
<tr><td>We need reference geometry from the two 288mm channels and also the large gears since the power subsystem will connect to both the structure subsystem and the drivetrain.</td><td></td></tr>
</table>

So once again click on the **Copy Geometry** icon in the upper menu and de-select the publish icon and click on the folder and select **my_robot_structure_sub.asm** and **Open** it. Keep the **Default** orientation.	
Click on the **red References** tab and then click in the **Surface Sets** area. Left click on the **CHANNEL-288MM.PRT** to import it into the skeleton file. Now click the green check mark to finish.	
Do the same procedure to import the second 288mm channel into the skeleton file. *This imports these channels into the skeleton file where they will only be references and not add mass or geometry to the power subassembly but will allow us to design the power subsystem so that it fits appropriatelyinto the structure subassembly.*	

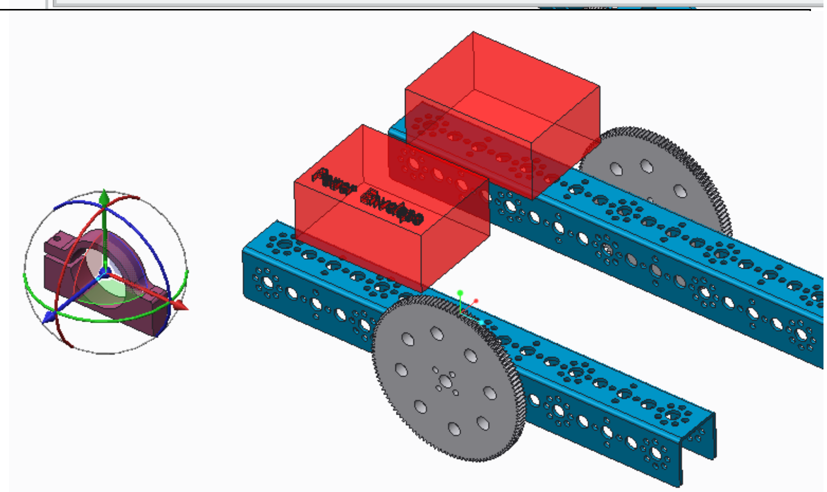
Use the **Copy Geometry** process again and select the Drivetrain subsystem file: "**my_robot_drivetrain_sub.asm**". Then click on the **References** tab and the **Surface Sets** area. Now find the **LARGE_GEAR_SUB.ASM** and expand it then select the **GEAR_METRIC_120_2011.PRT**. This will bring the large gear into the skeleton file as reference geometry.	
Do this procedure again to import the other gear geometry into your skeleton file.	
Now save the skeleton file and close it. We are now ready to begin creating the drivetrain subassembly.	

The technology sketches for the Power subsystem just indicate that a motor will be used. We need to get the motor so that it interfaces with the large gear appropriately and we need to mount it to the structure subsystem.

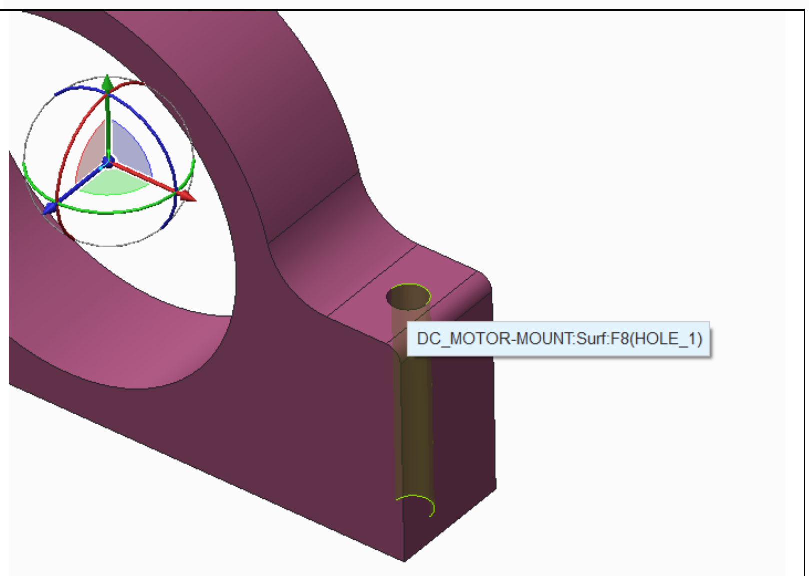
Click on the **Assemble** icon in the upper menu to open the library of parts and select **dc_motor-mount.prt** and **Open** it.

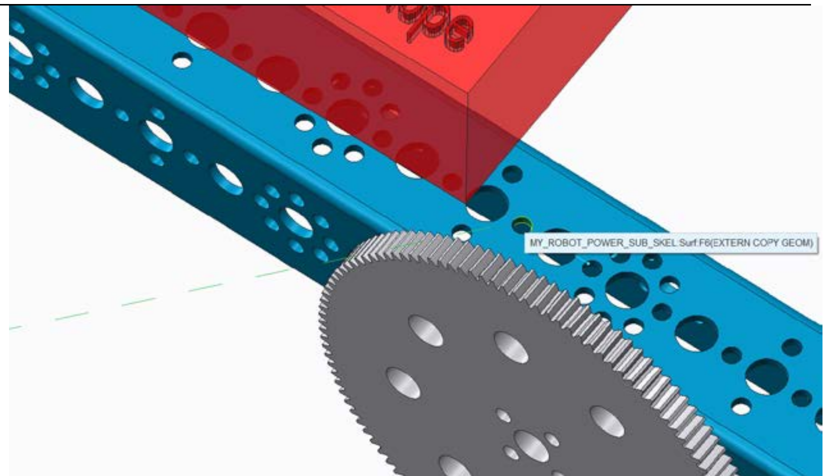
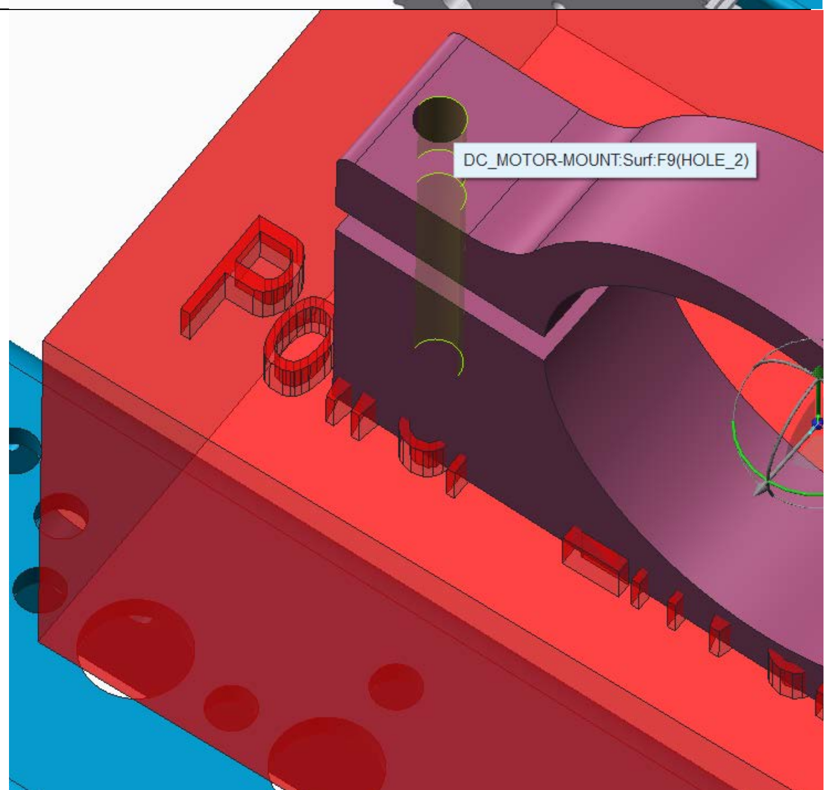
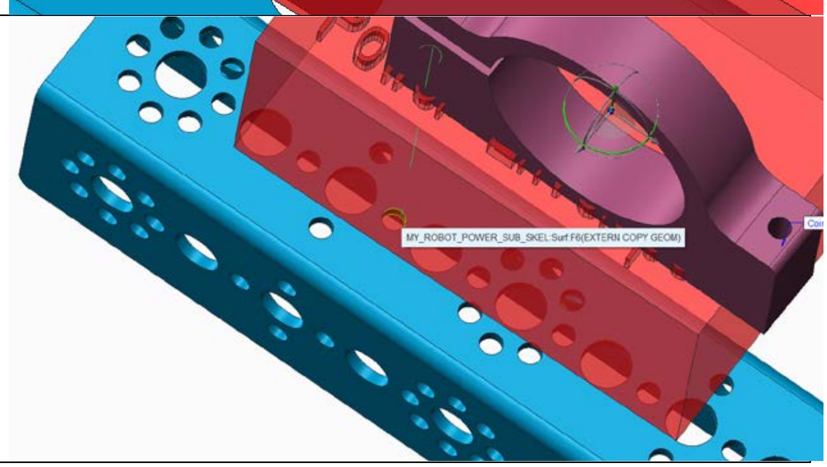


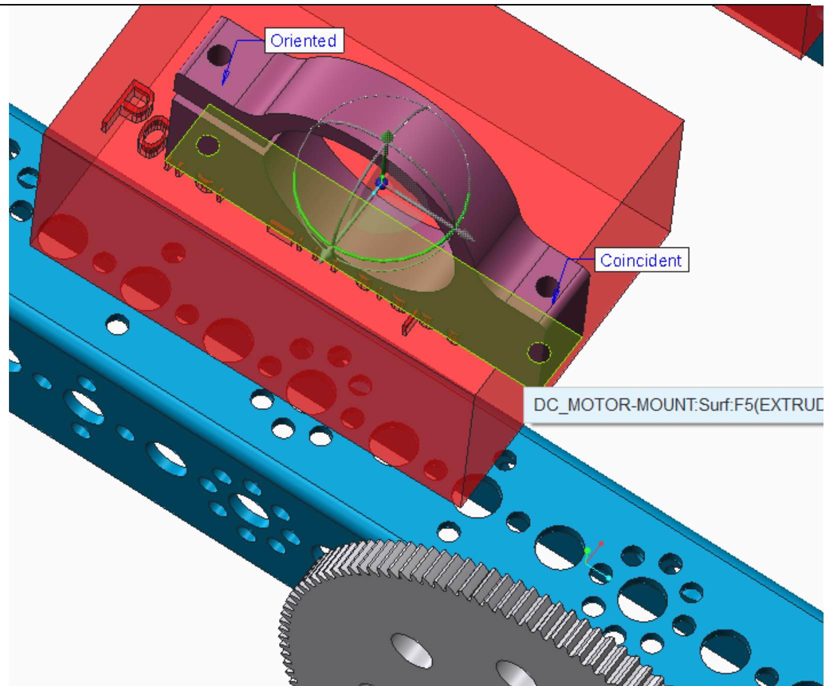
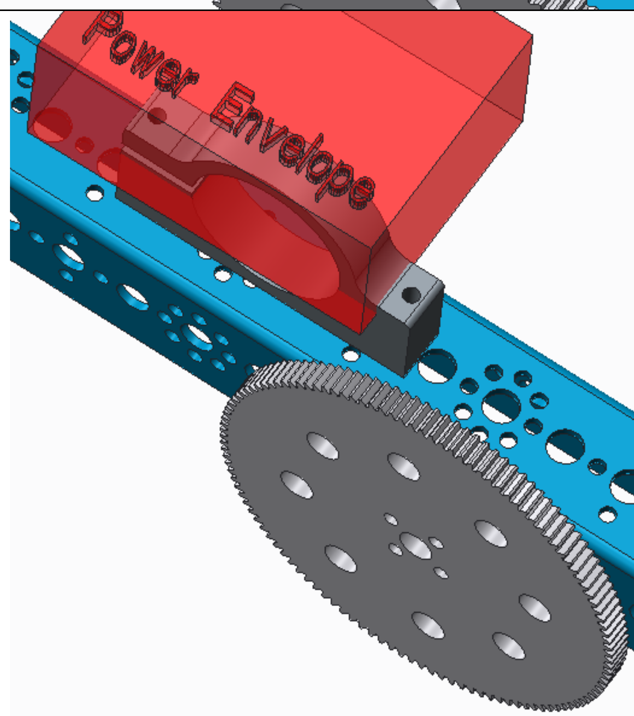
Left click to temporarily place the motor mount and then use the orientation sphere to orient it as shown.

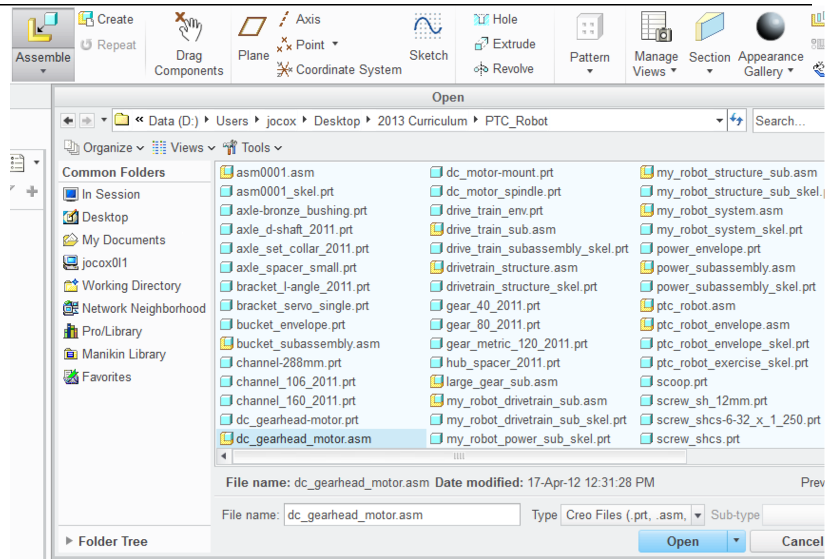
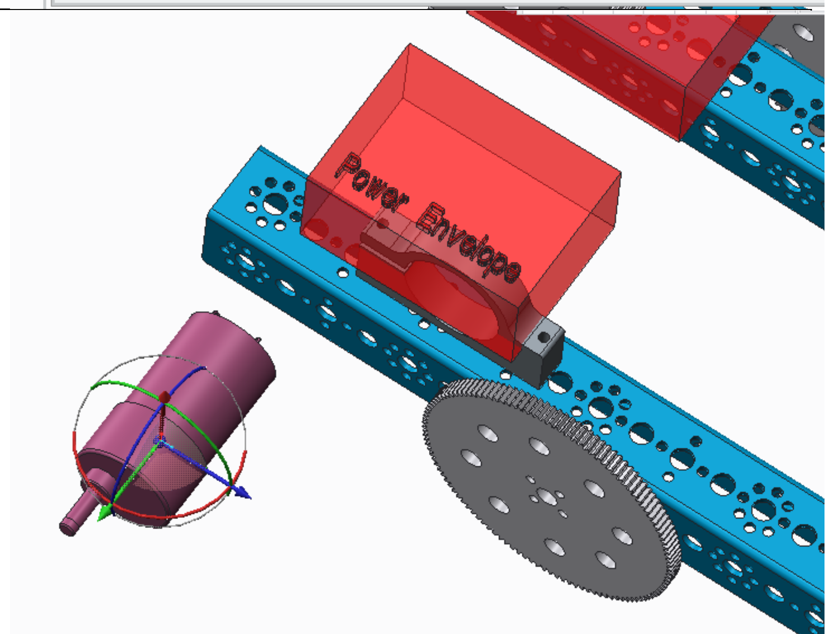
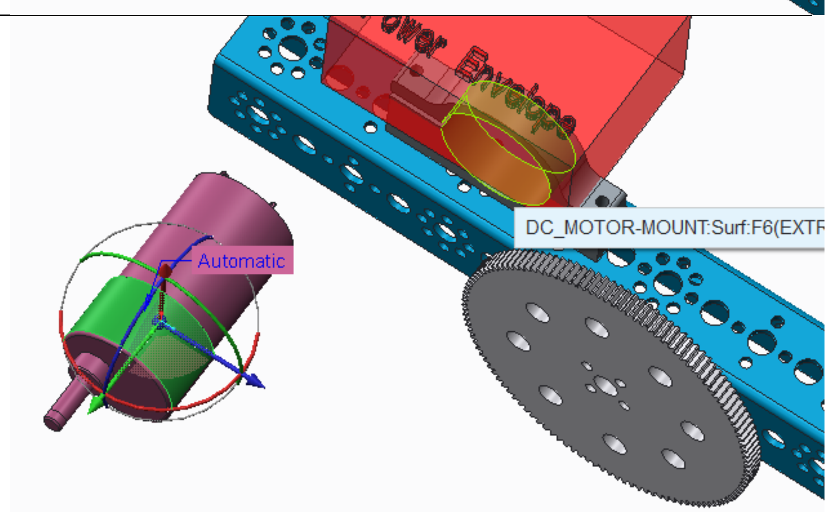


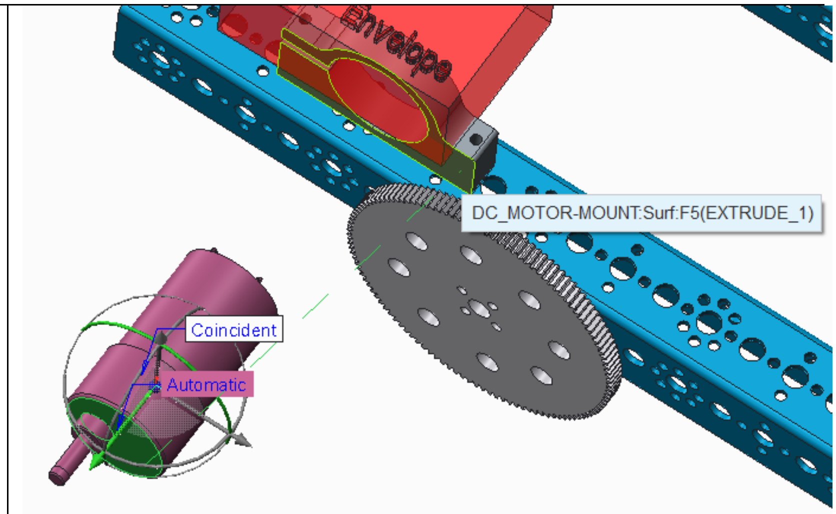
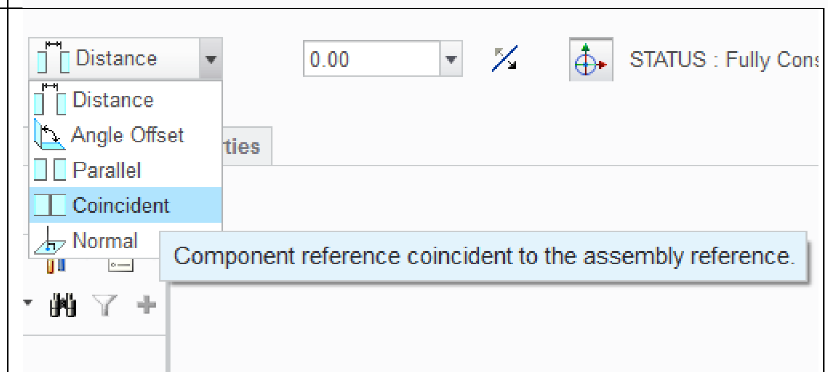
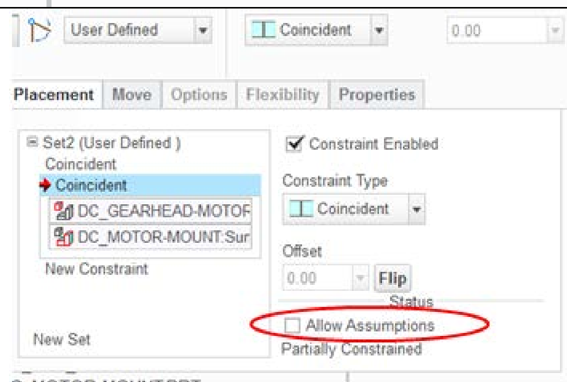
Left click to select the hole in the motor mount.



Left click to select the small hole shown in the picture.	
Now select the other hole in the motor mount.	
And then select the small hole in the channel as shown in the picture.	

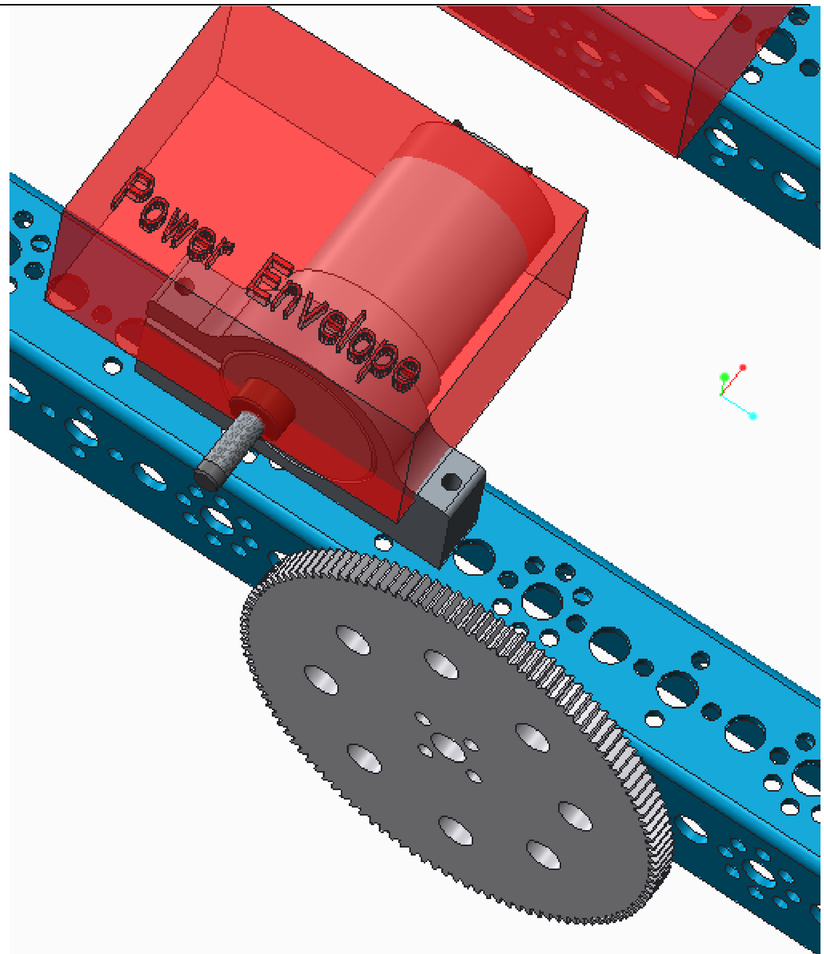
Finally select the bottom surface of the motor mount and the top surface of the channel.	
Then click the green check mark to finish placing the motor mount.	

<table>
<tr>
<td>Now select Assemble again and select dc_gearhead_motor.asm and Open it.</td>
<td>
</td>
</tr>
<tr>
<td>Left click to temporarily place the DC motor and then use the orientation sphere to orient it as shown.</td>
<td>
</td>
</tr>
<tr>
<td>Now select the cylindrical surface of the motor and the motor mount as shown.</td>
<td>
</td>
</tr>
</table>

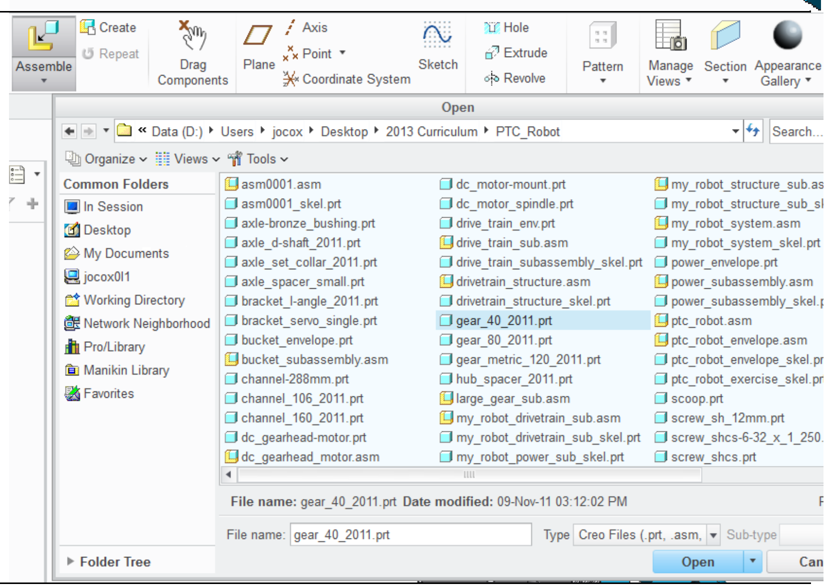
Then select the front face of the motor and the front face of the motor mount as shown.	
If the constraint defaults to a **Distance** constraint then change it to a **Coincident** constraint.	
Now in order for the motor to be rotated to a position where it can engage the large gear it needs to be able to rotate. Open the **Placement** tab and then unclick the **Allow Assumptions** box.	

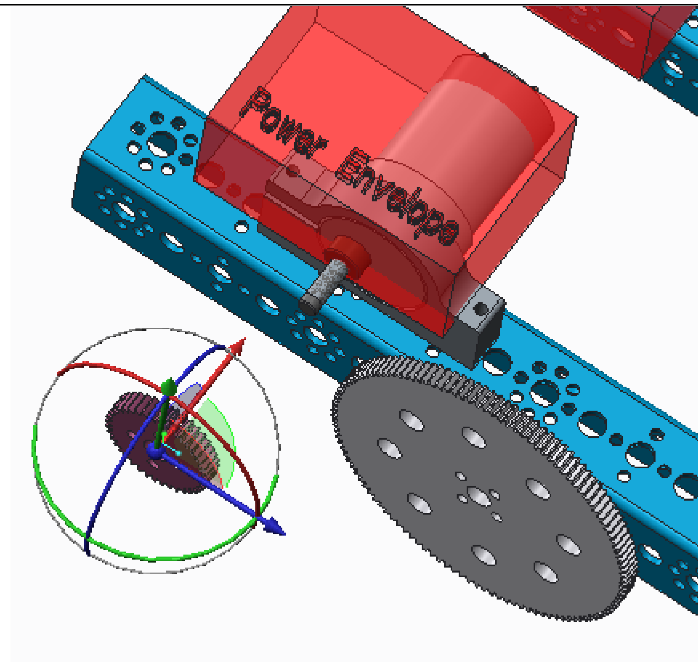
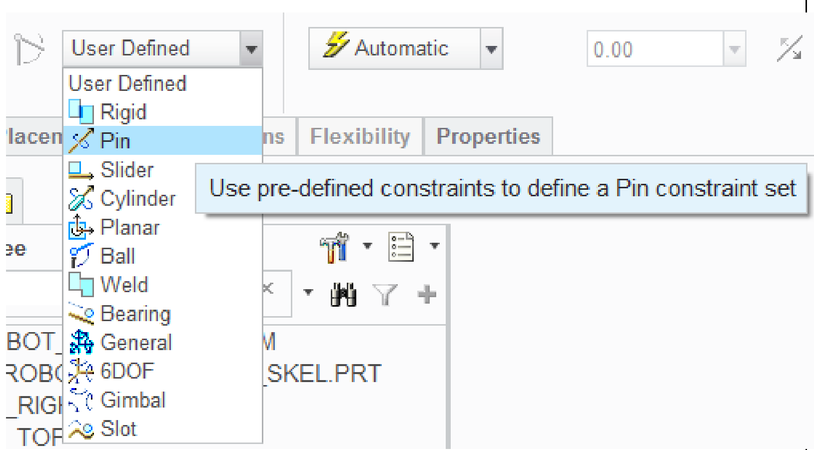
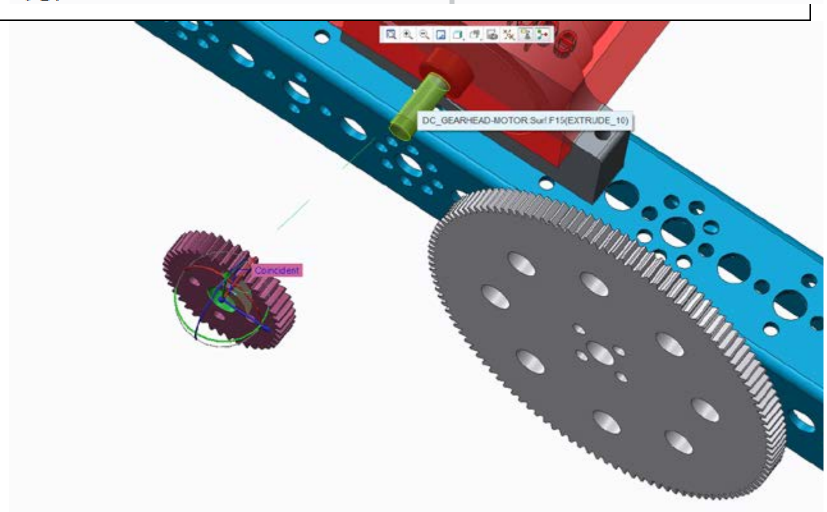
Then click the green check mark at the top to finish placing the motor.

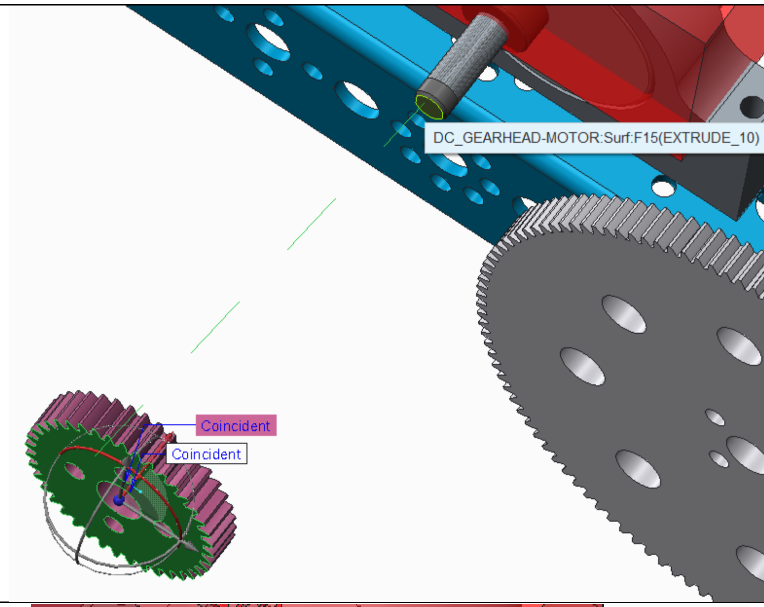
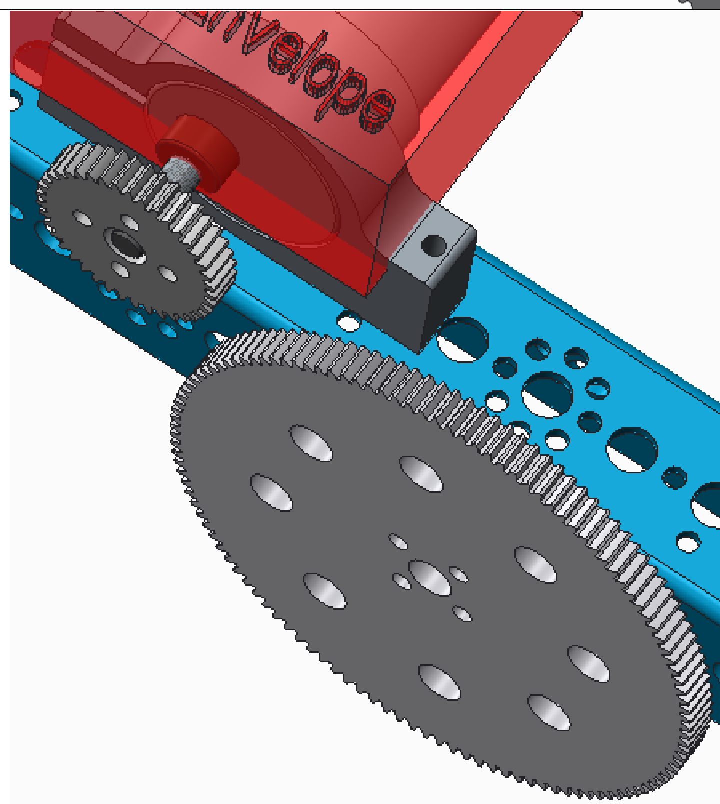
The motor is only partially constrained, but that is OK. We will come back and finish constraining the motor once we have the small gear on the shaft and can see how to engage it to the large gear.

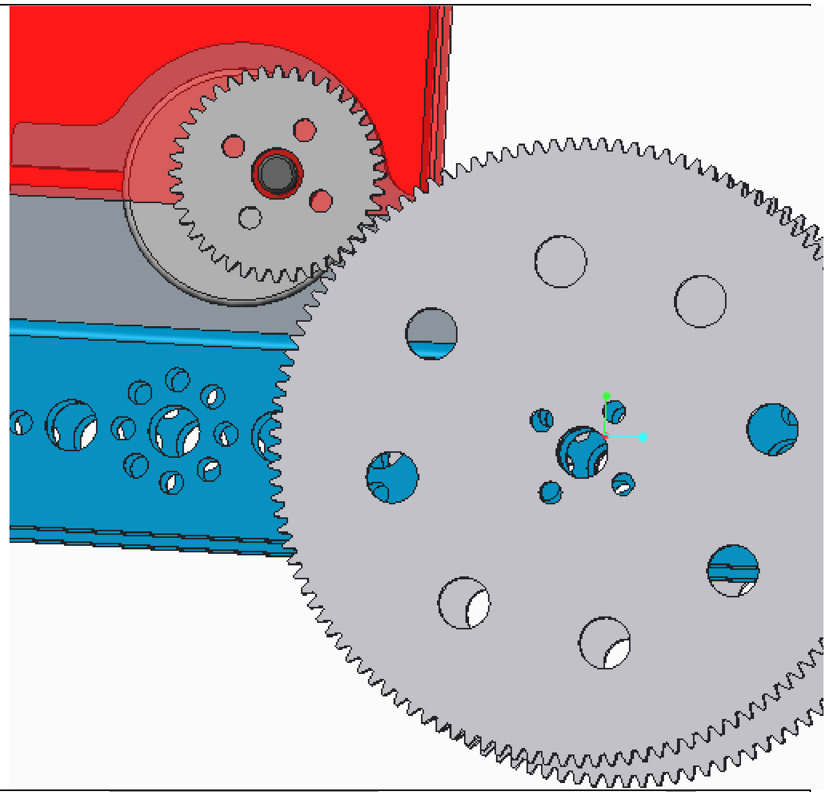
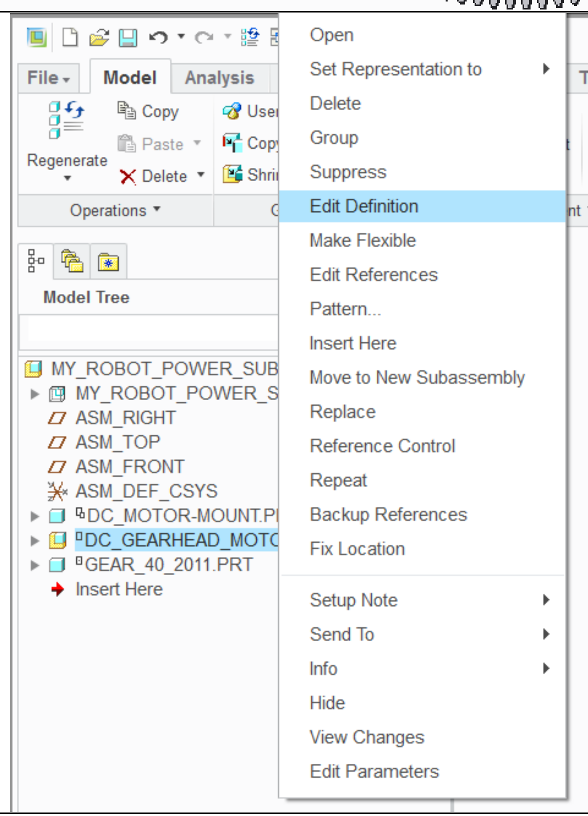


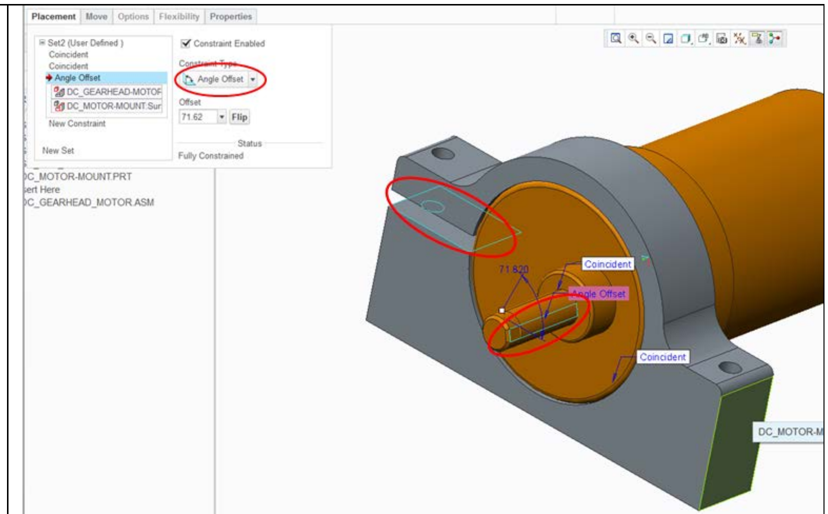
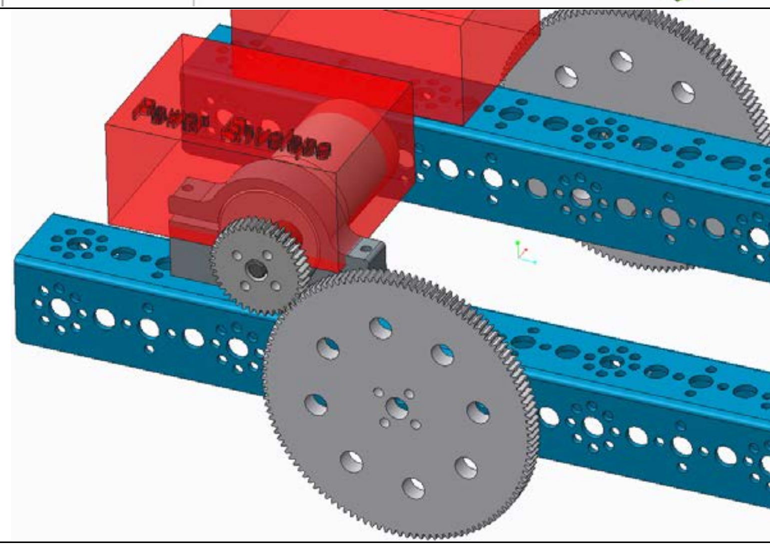
Select **Assemble** again in the upper menu and then select “**gear_40_2011.prt**” and **Open** it.

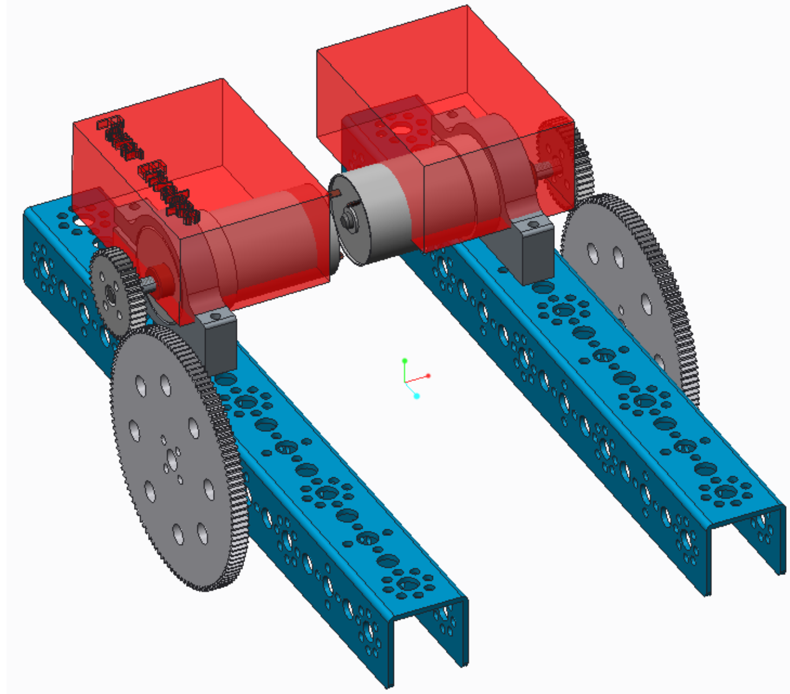
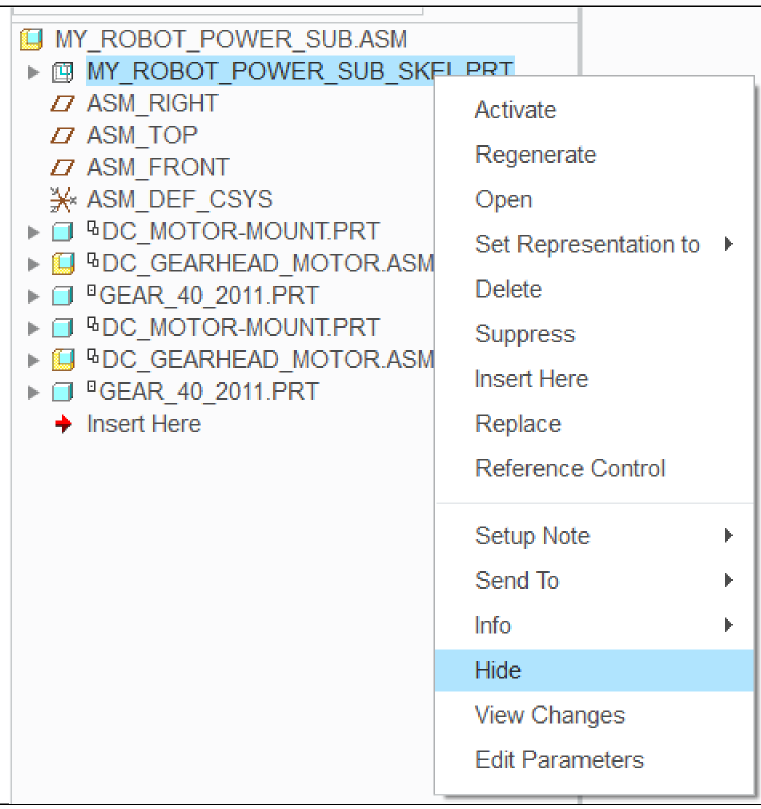


Left click to place the small gear as shown.	
We will use a **Pin** constraint again because we want the gear to be able to spin. So click on the **User Defined** pull down menu and select Pin. *(By the way, the shaft of the motor would normally spin but the motor models in the kit of parts do not function. So we make the gears spin on the shafts instead.)*	
Select the inner hole of the gear and the shaft of the motor.	

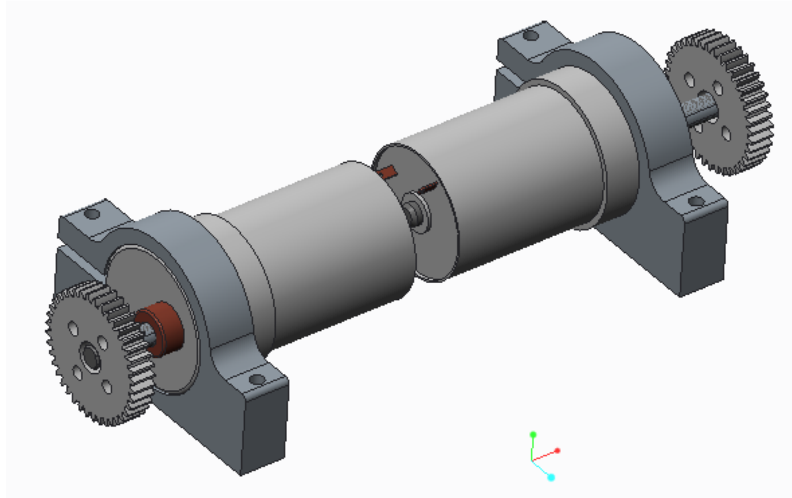
Then select the face of the gear and the front end of the motor shaft as shown.	
Click the green check mark to finish applying the **Pin** constraint to the small gear.	

Now use the **CTRL-ALT** keys and left click on the motor to rotate it so that it engages with the large gear appropriately.	
Now right click on the "**DC_GEARHEAD_MOTOR.ASM**" and select **Edit Definition** to fix the location of the motor in the mount.	

Open the **Placement** tab and select **New Constraint**. Then select from the **Automatic** pull down menu the **Angle Offset** constraint. Then select the flat part of the motor shaft and the flat slot of the motor mount to fix the motor in this orientation.	
Now click the green check mark to finish the placement of the motor.	

<table>
<tr>
<td>
Use the same procedure to place a motor assembly on the other channel and engage it with the other large gear.
Notice that our initial guess for the power envelope was not accurate but in this case it was in the general area and about the right size.
</td>
<td></td>
</tr>
<tr>
<td>
We can once again hide the reference geometry in the skeleton file by right clicking on the skeleton file in the model tree and selecting Hide.
</td>
<td>

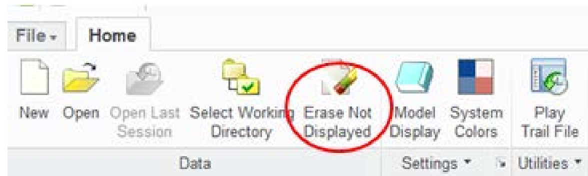
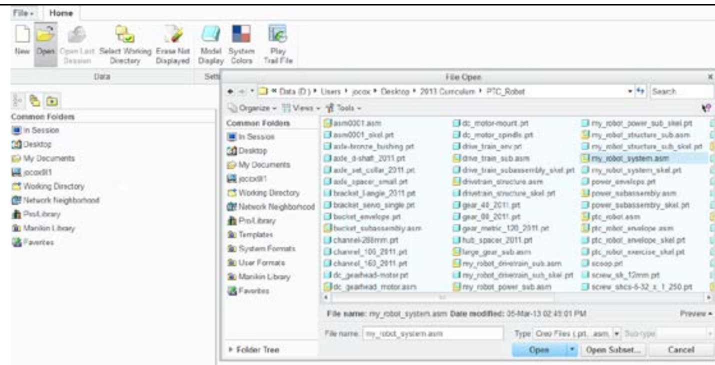
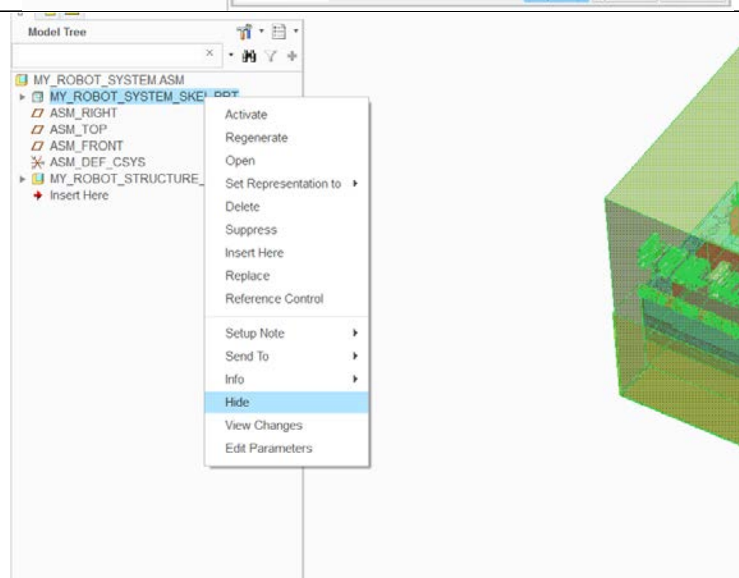
</td>
</tr>
</table>

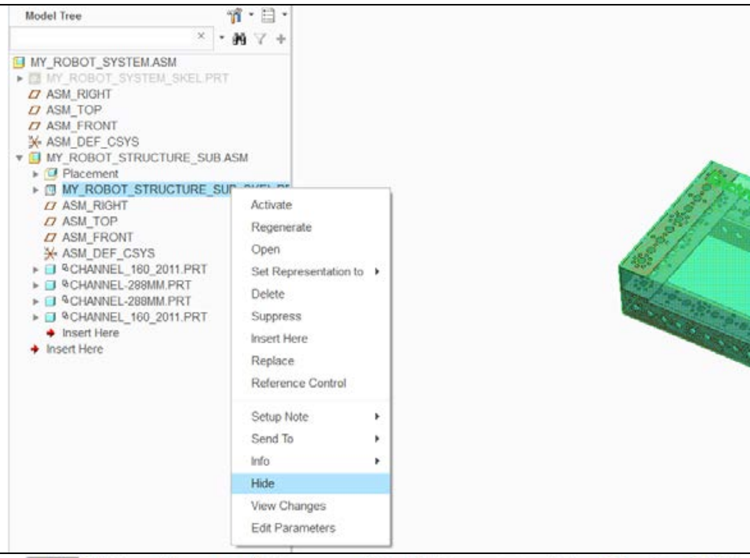
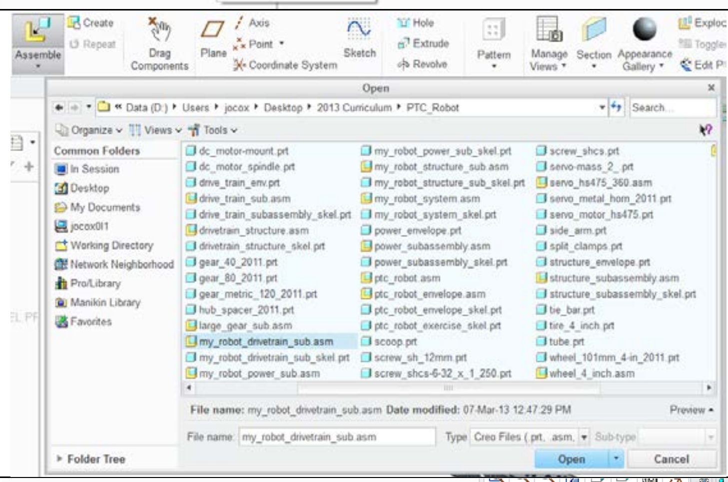
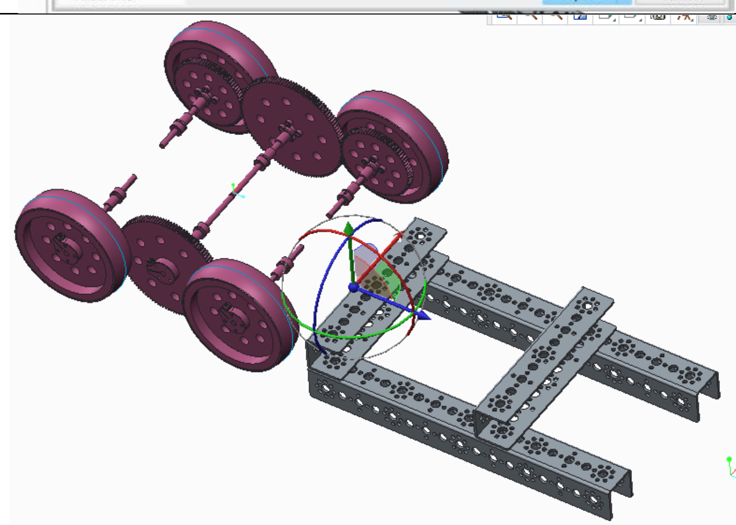
The power subsystem is now complete. Save your model and close the file.	

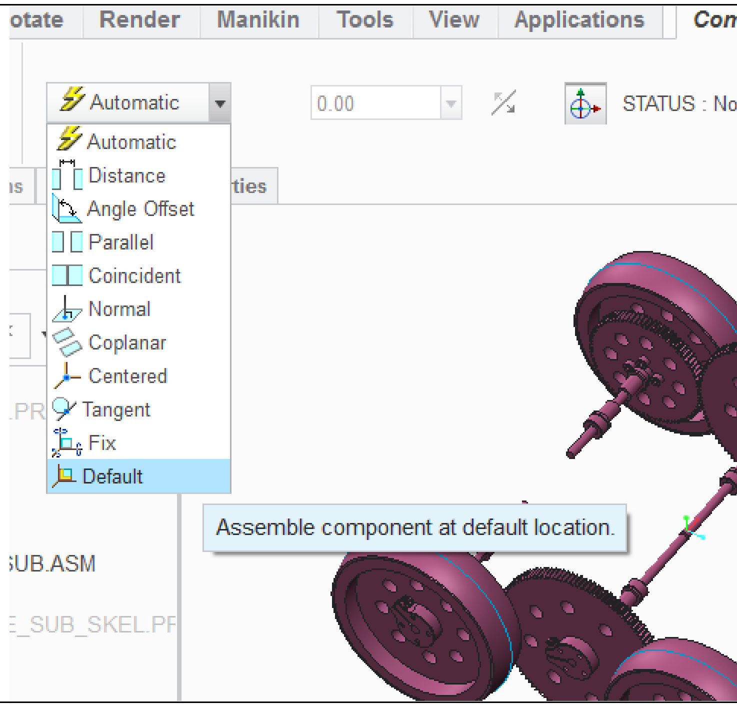
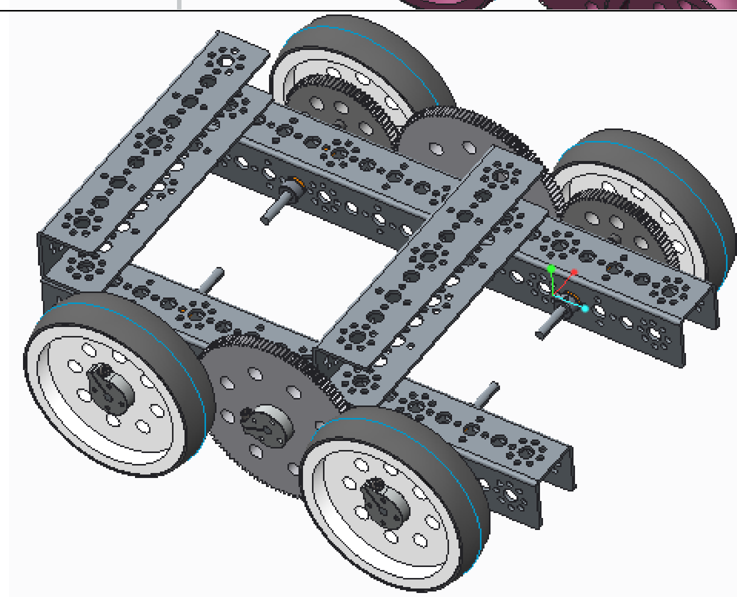
You have finished the exercise.

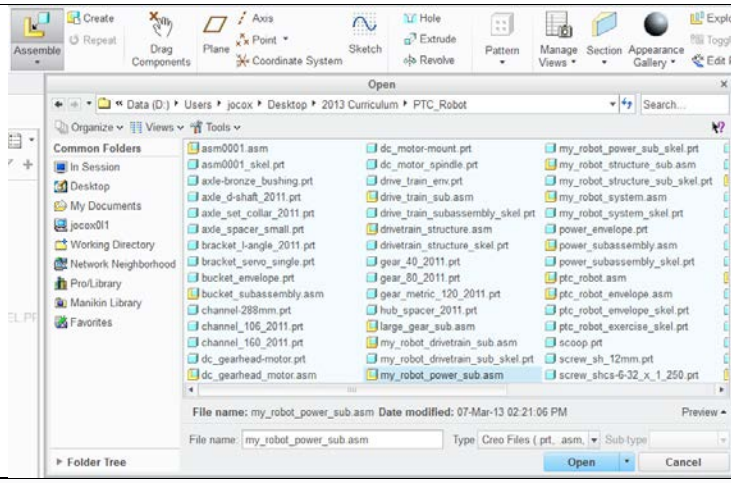
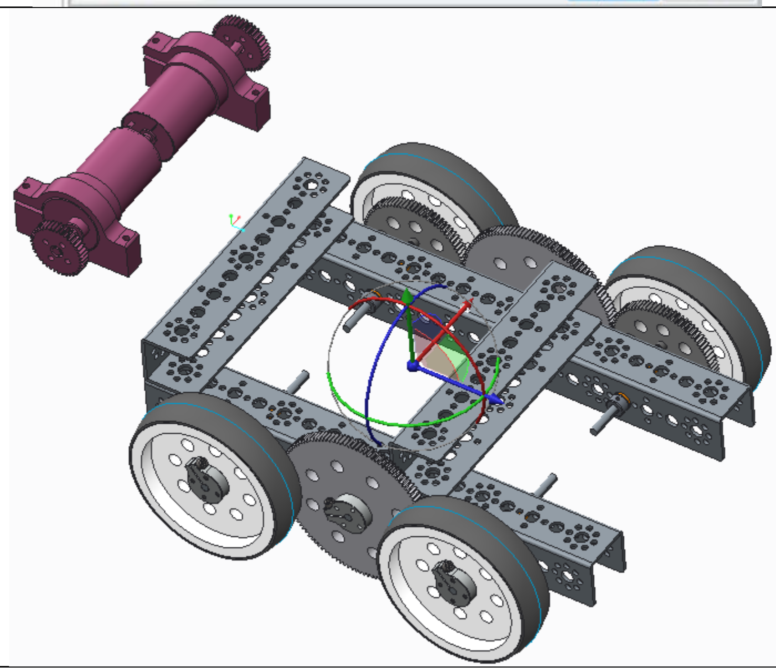
EXERCISE 13.0: INTEGRATING THE SYSTEM

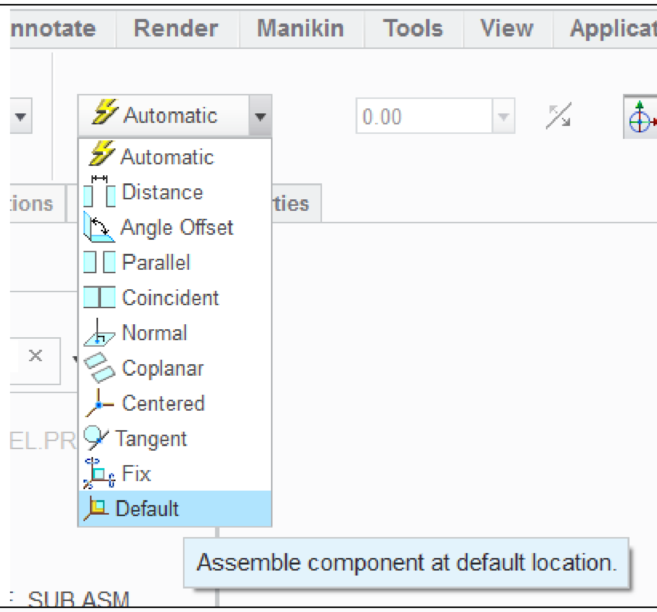
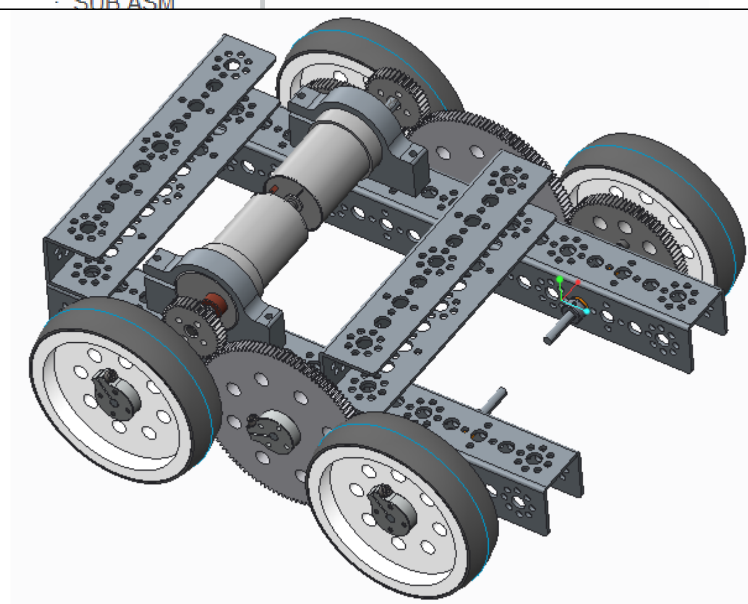
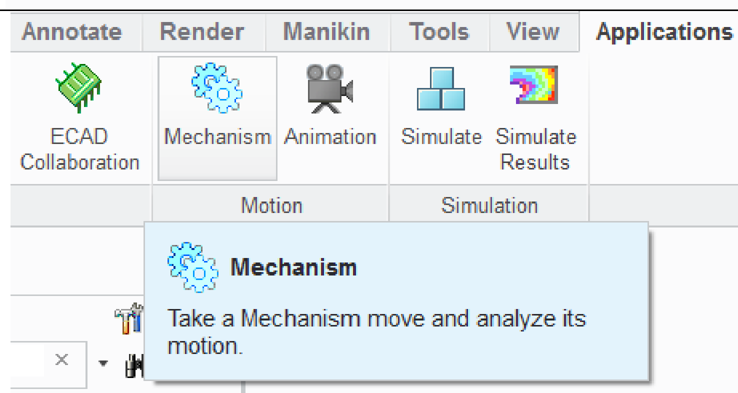
This document provides detailed instructions for integrating the entire PTC robot system model.

Close all of your model files and erase from memory by selecting **Erase Not Displayed** and clicking **OK.**	
Open the system model "**my_robot_system.asm**".	
Right click on the skeleton file in the model tree and select **Hide** so that all of the reference geometry is hidden.	

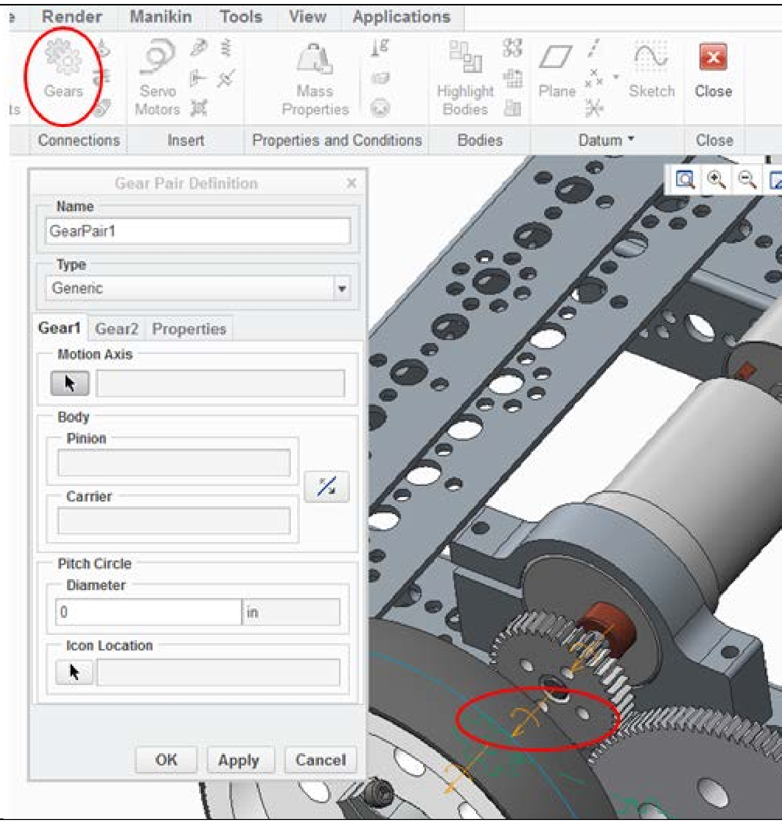
That will only hide the system reference geometry so you will need to hide the reference geometry in the structure subassembly as well.	
Now select **Assemble** and select "**my_robot_drivetrain_sub.asm**" and **Open** it.	
Left click to temporarily place the subassembly.	

Now select the **Default** constraint to locate the subsystem and click the green check mark to finish.	
Now you can check to make sure the wheels and gears rotate.	

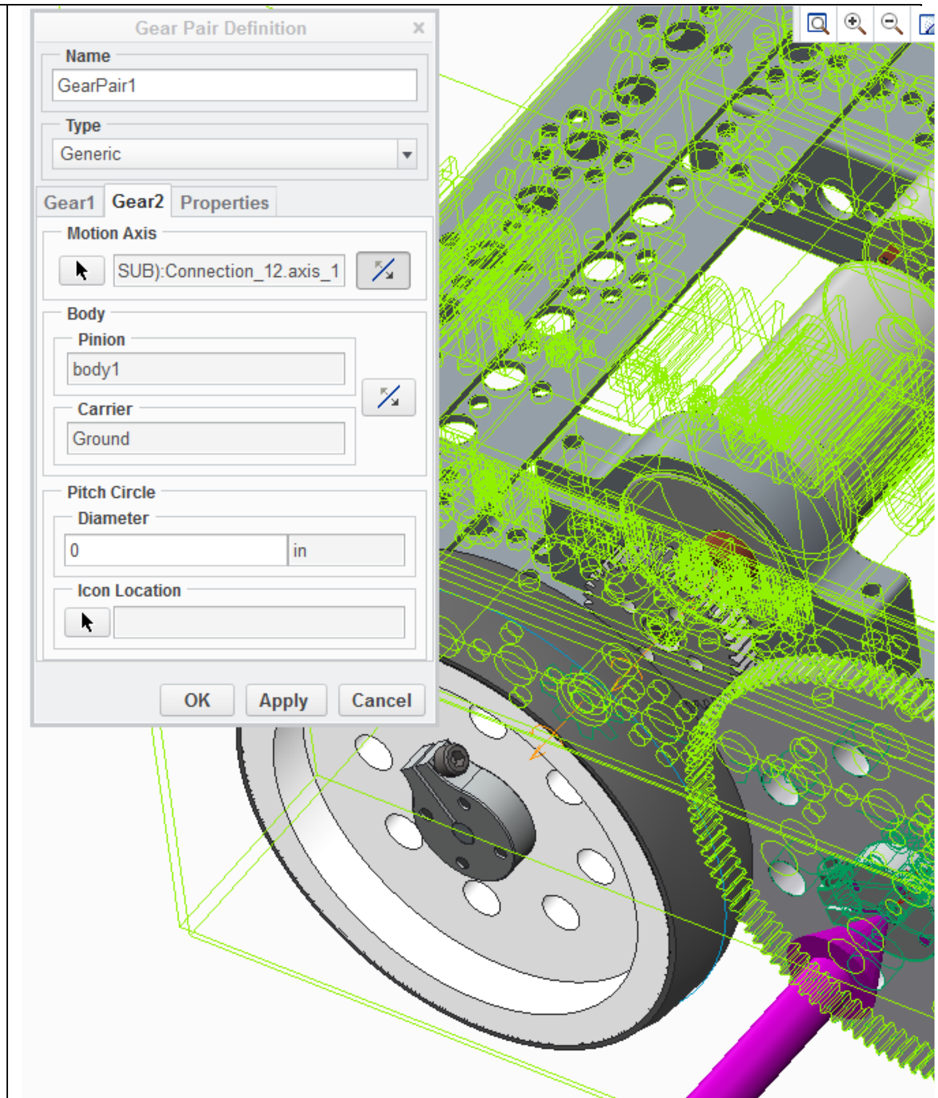
Select **Assemble** and select the file "**my_robot_power_sub.asm**" and **Open** it.	
Left click to temporarily place the power subsystem.	

Once again select the **Default** constraint to locate the power system in the right position. Then click the green check mark to finish.	
Now we need to set up gear pair connections between the power subsystem and the drive train subsystem.	
Once again select the **Applications** tab in the upper menu and then select **Mechanism**.	

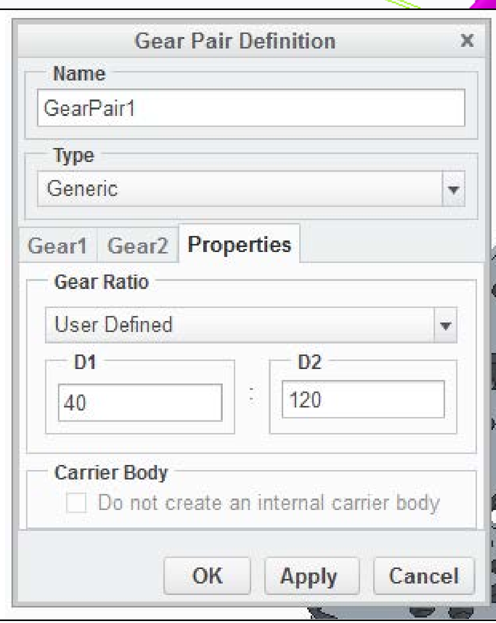
Now select the **Gears** icon to create a gear pair connection. Select the axis icon associated with the small gear for **Gear 1**.

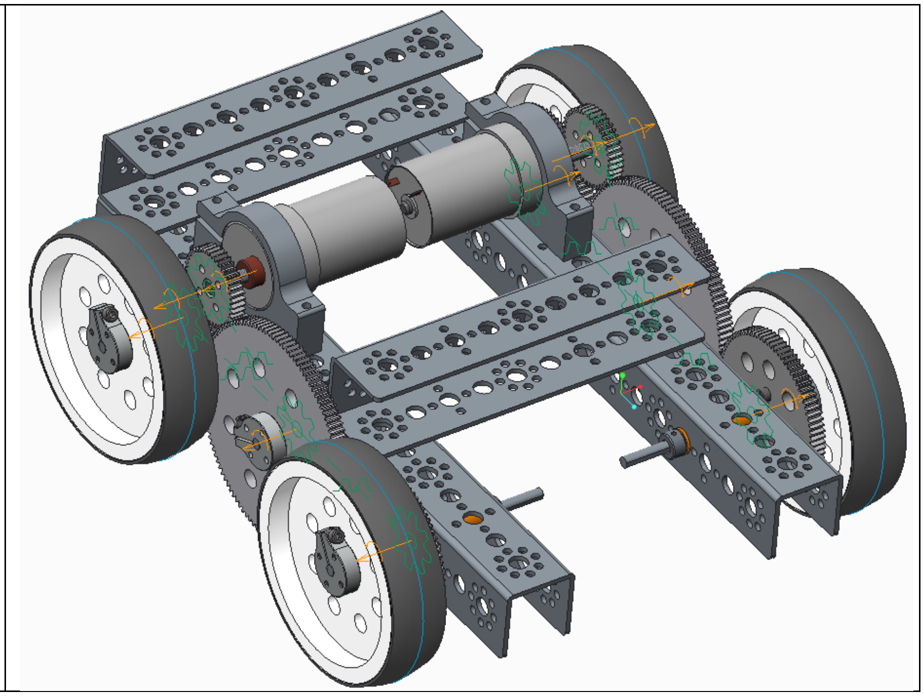


Select the axis icon for the large gear for **Gear 2** and then select the **Properties** tab.



Set the ratio to **40** by **120** and click **OK**.



Now do the same procedure on the other side of the robot to set up the gear pair with the second motor and gear. Now you will have a complete basic robot system. Save your model.	

The drive train gearing allowed the large gear to drive the smaller gears that were attached to the wheels. Since the small gear diameters were smaller than the large gear diameter, we calculated that the RPMs would increase and since the wheel's diameter was greater than the large gear, it would cause the robot to travel faster. Now we need to complete our gearing calculations since we are now driving the drive train gearing with a small gear from the power subsystem. Calculate the overall gear ratio. An easy way to do this is to use the number of teeth.

$$\left(\frac{40}{120}\right)\left(\frac{120}{80}\right) = \frac{40}{80} = \frac{1}{2}$$

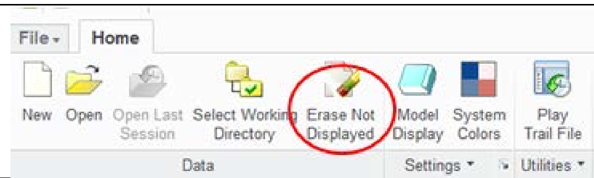
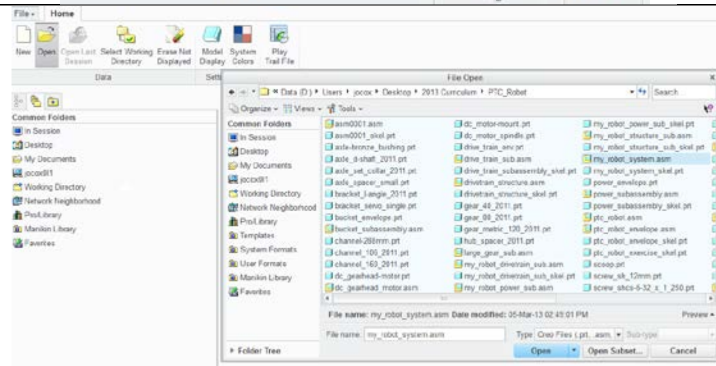
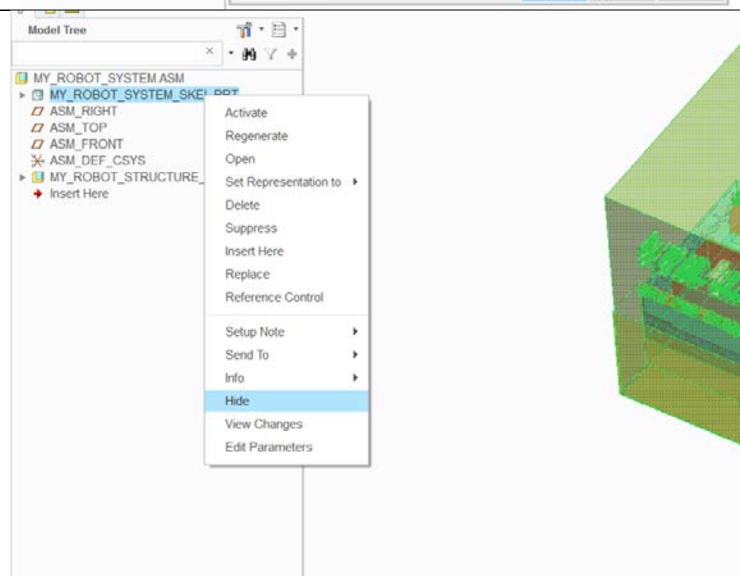
Which means that we should see a reduction in speed of ½ or a 2 to 1 mechanical advantage because of the gearing we have chosen. This means the robot should have power to climb but won't move very fast.

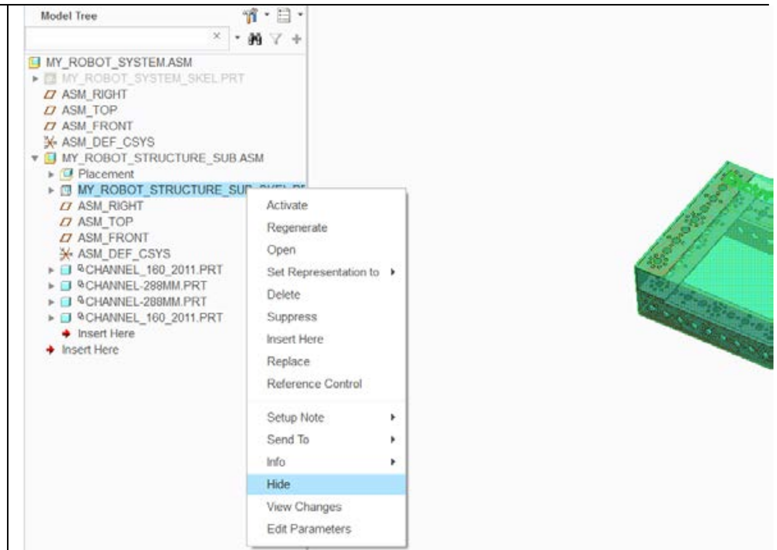
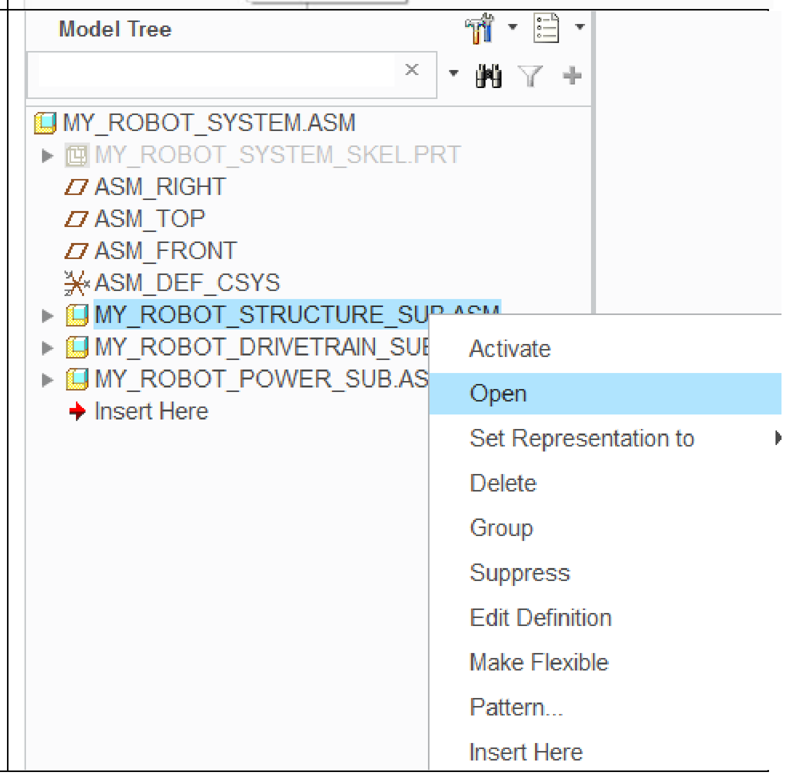
Now calculate the speed with which your robot will move based upon a single revolution of the small driving gear.

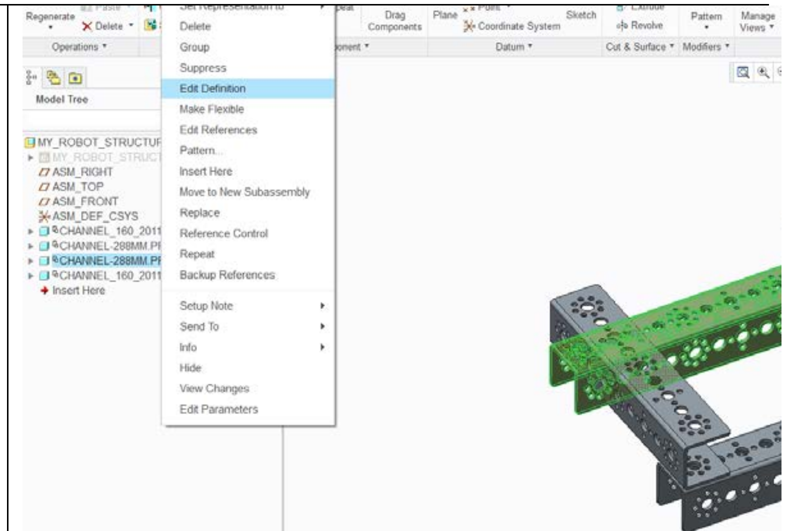
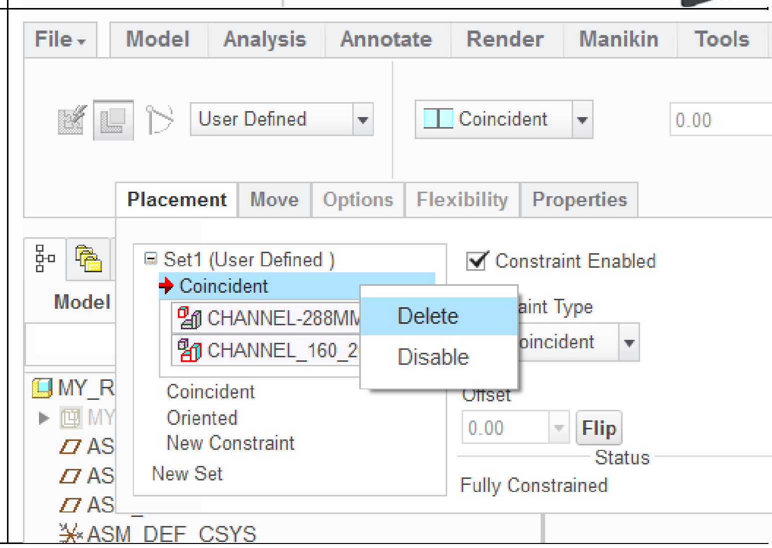
You have finished the exercise.

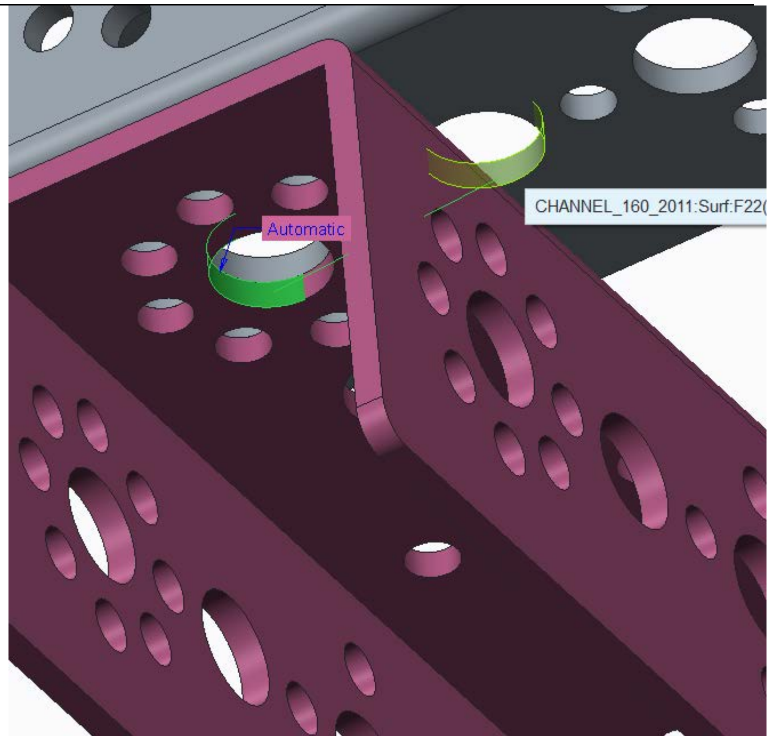
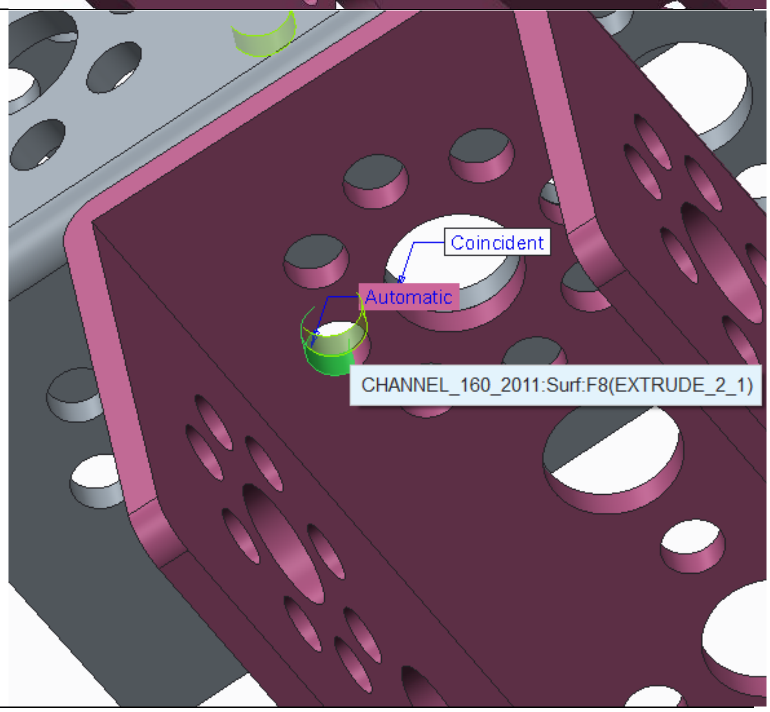
EXERCISE 14.0: CHANGING THE SYSTEM

This document provides detailed instructions for making changes to the entire PTC robot system model.

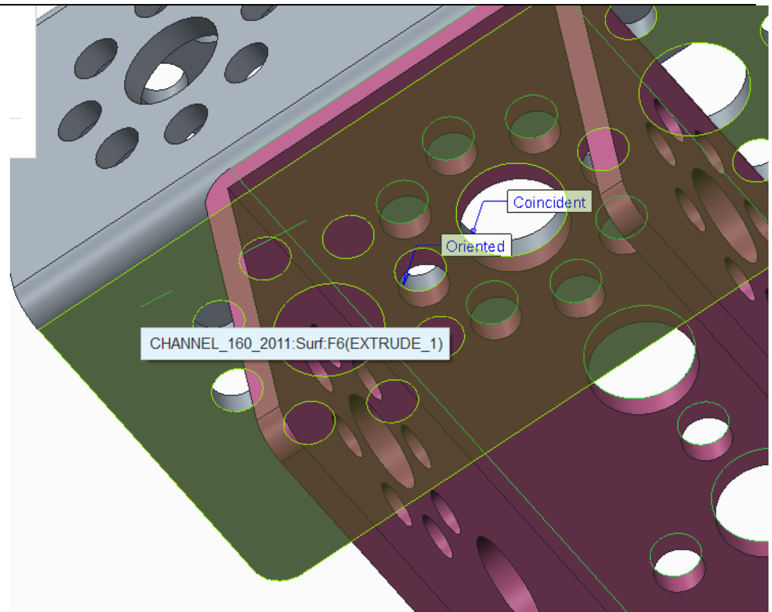
Close all of your model files and erase from memory by selecting **Erase Not Displayed** and clicking **OK.**	
Open the system model "**my_robot_system.asm**".	
Right click on the skeleton file in the model tree and select **Hide** so that all the reference geometry is hidden.	

<table>
<tr>
<td>That will only hide the system reference geometry so you will need to hide the reference geometry in all of the other subassemblies as well.</td>
<td>
</td>
</tr>
<tr>
<td>Now right click “MY_ROBOT_STRUCTURE_SUB.ASM” in the model tree and Open it.</td>
<td>
</td>
</tr>
</table>

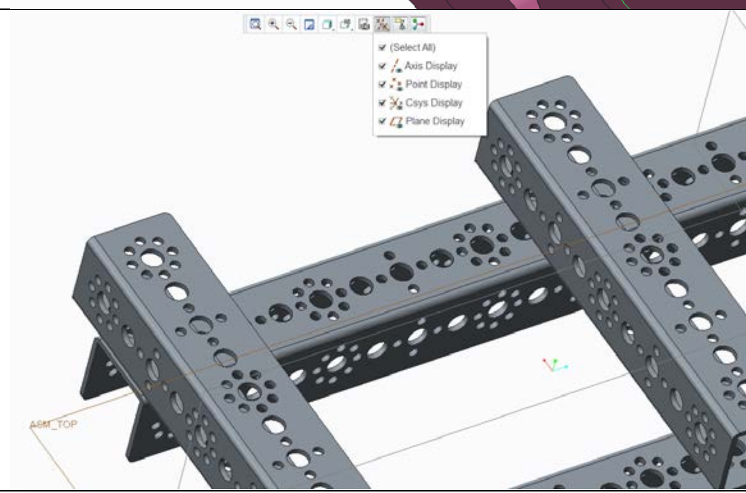
Now find the second **channel_288mm.prt** in the model tree and right click on it and select **Edit Definition**.	
Open the **Placement** tab and delete all of the constraints by right clicking on each one and selecting **Delete**.	

Now select the end hole in the 288mm channel and the next hole over in the shorter channel.	
Select **New Constraint** in the **Placement** tab and then select the two small holes to align the channels.	

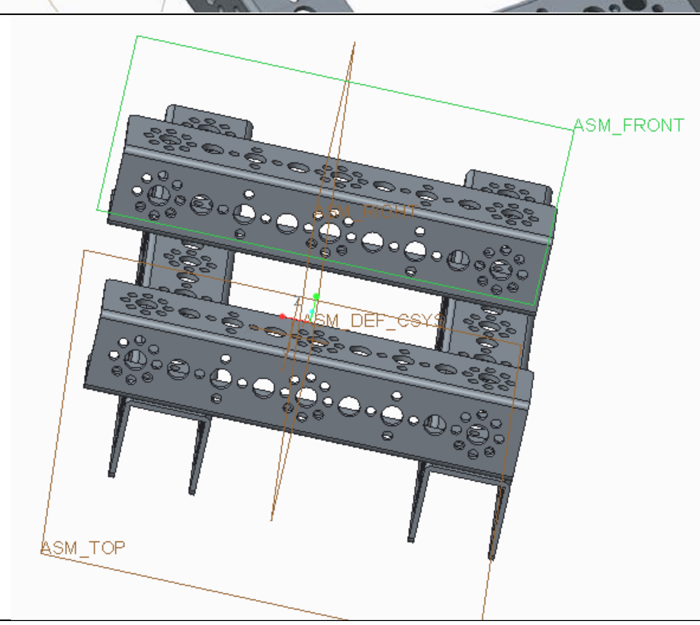
Finally, **select New Constraint** in the **Placement** tab and then select the top surface of the 288mm channel and the bottom surface of the shorter channel to finish placing the parts. Now click on the green check mark.

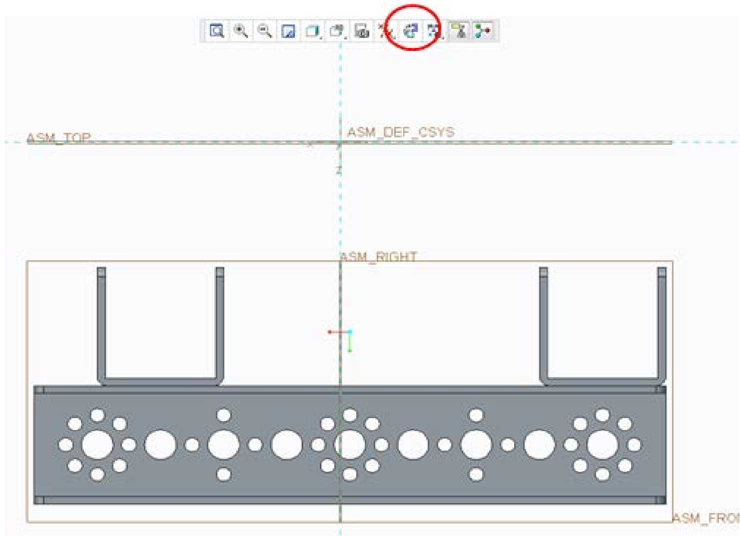
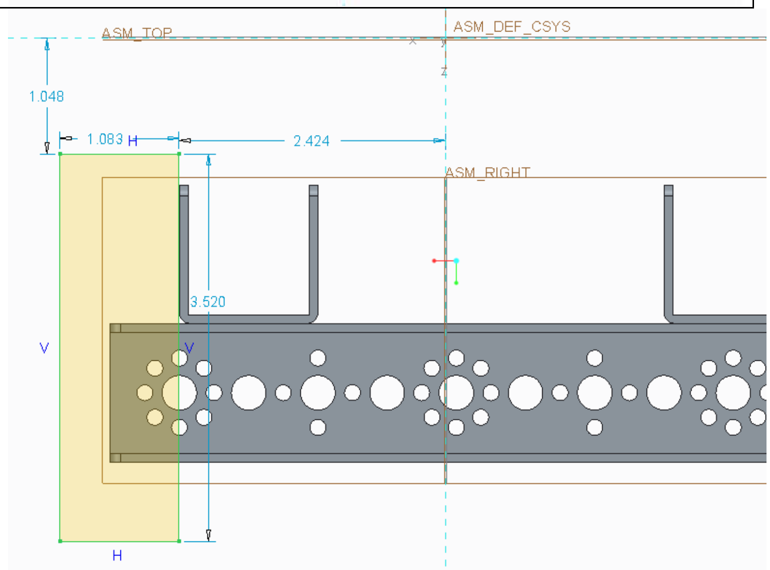


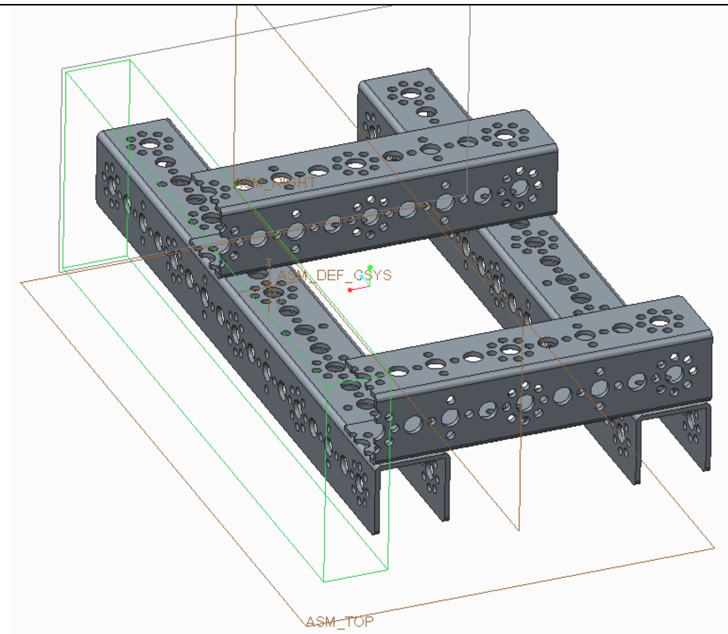
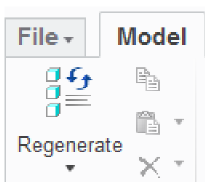
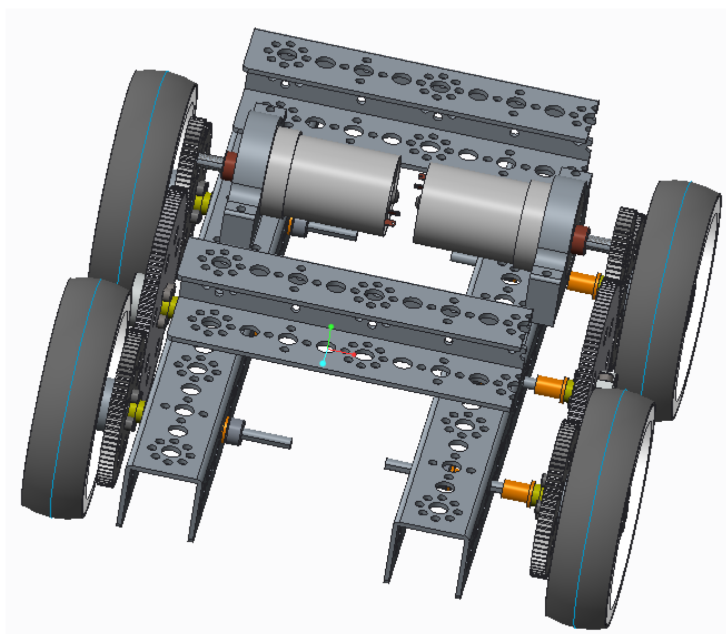
Turn on the datum planes by selecting the datum filter tool and checking **(Select All)**.

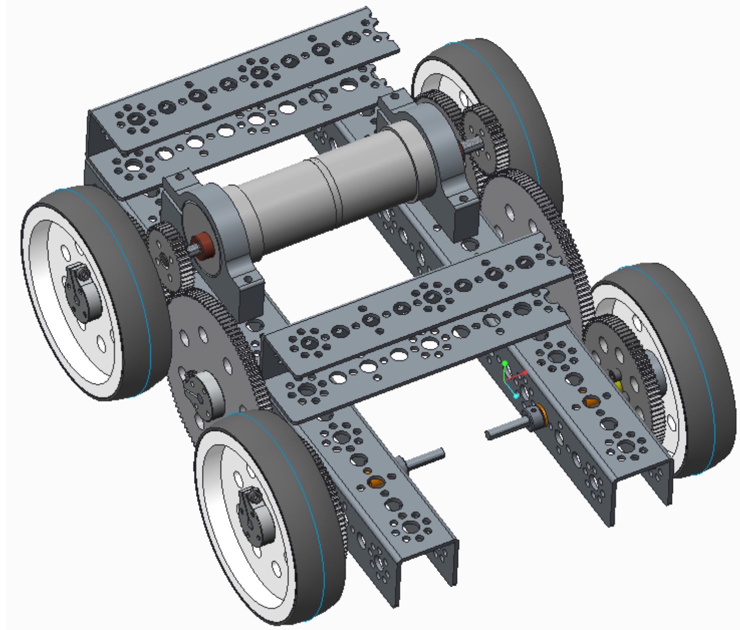
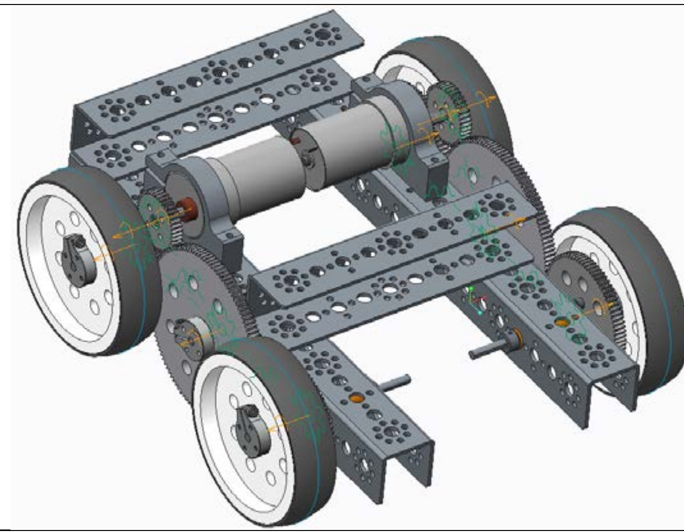


We need to cut the channels that stick out on the left side of the robot, so select the **FRONT** plane and then click on the **Sketch** icon in the upper menu.



Orient the sketching plane so that you can look straight on it by clicking on the **Orient View** icon	
Now select the **Rectangle** tool and sketch a rectangle as shown. The dimensions don't have to be exact. The rectangle just needs to cover the parts of the channel that need to be cut off.	

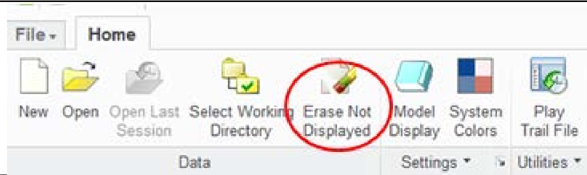
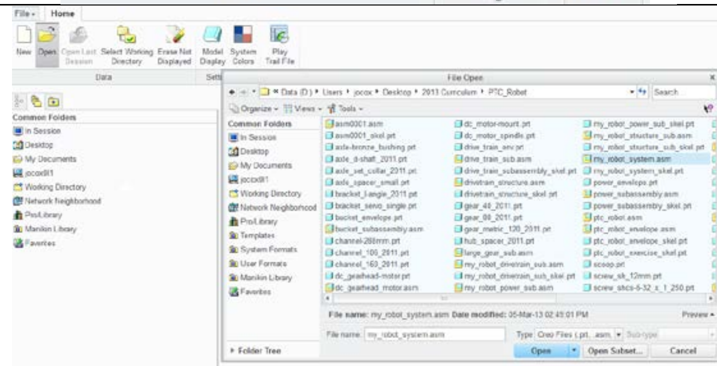
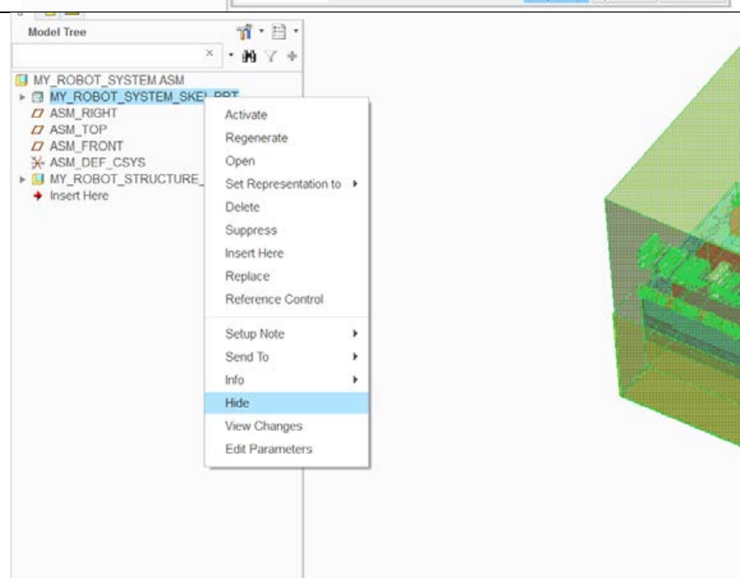
Click the green check mark to exit the sketcher and then select **Extrude** and make the length of the extrude long enough to cut off both channel pieces. Then click the green check mark to finish the extrude. *(The extrude will be displayed as wireframe since you are creating it in an assembly file. BTW, If you do this for real, watch out for the sharp edges.)*	
Now since we are just exploring this change, don't save your file. Simply select **Close**. You will notice that the robot system has some problems due to the shortened width. Don't worry, because of our top-down design strategy all we need to do is click on **Regenerate** and everything will be resolved. You can find the **Regenerate** icon in the upper left of the **Model** menu. 	

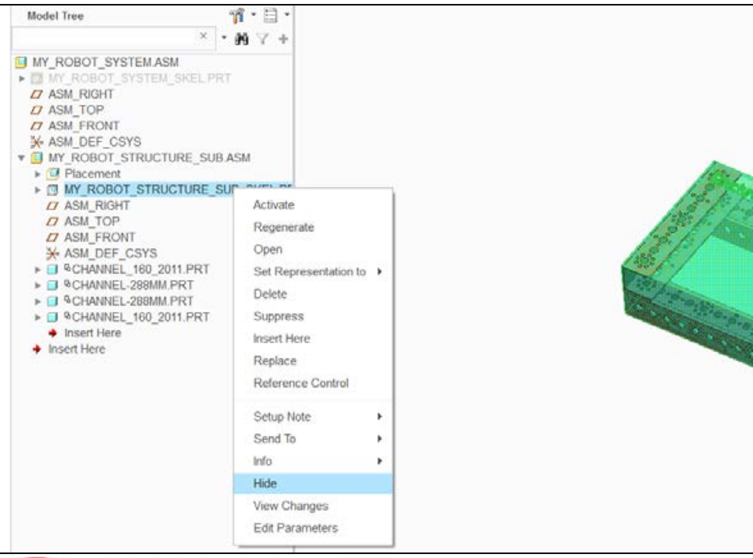
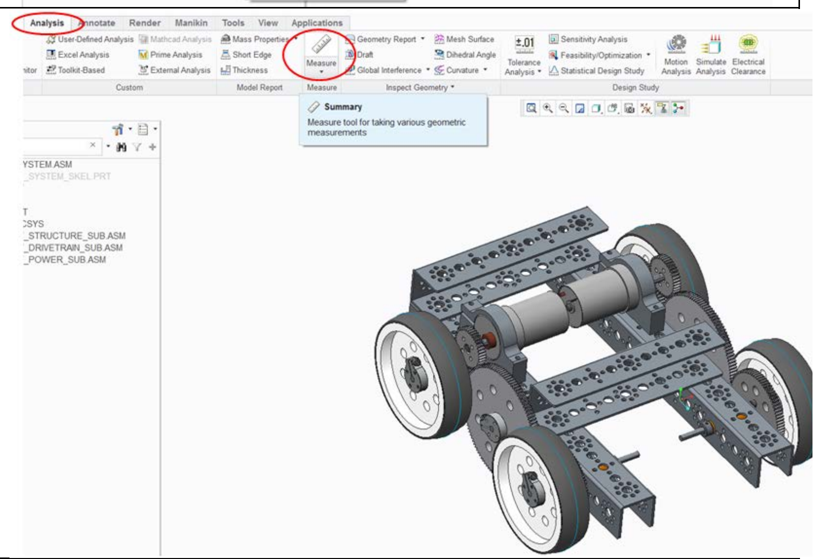
Once the system model has been regenerated, all the parts fit correctly.	
Top down design strategies allow you to explore changes to your design very quickly. Your overall system model doesn't break down. Now select **Close** again because we don't want to save this change and then select **Erase Not Displayed** and click **OK**.	

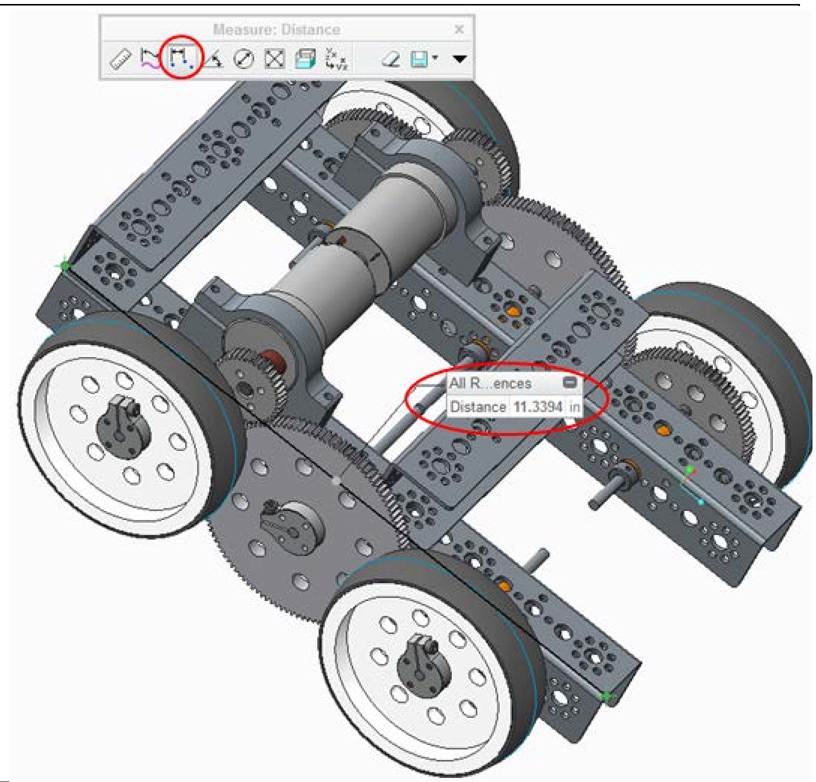
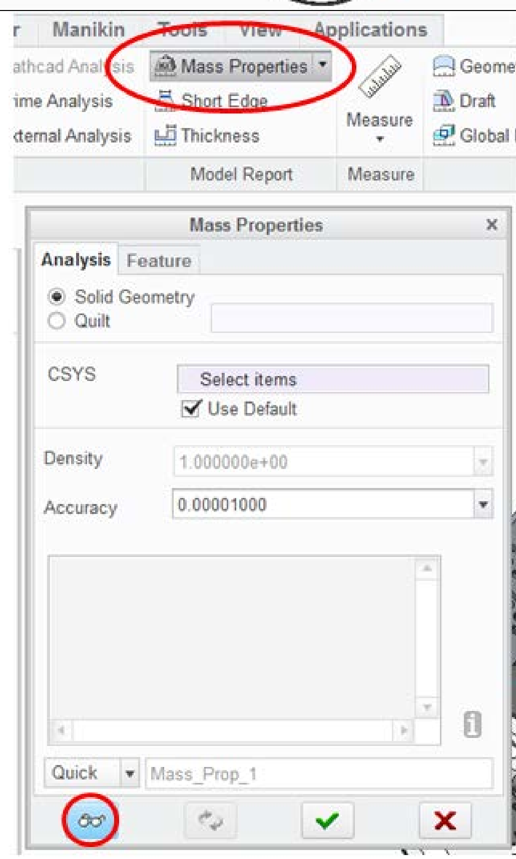
You have finished the exercise.

EXERCISE 15.0: SIMULATING THE REAL WORLD

To begin with, let's explore some of the physical characteristics of the PTC robot model you have completed.

Close all of your model files and erase from memory by selecting **Erase Not Displayed** and clicking **OK.**	
Open the system model "**my_robot_system.asm**".	
Right click on the skeleton file in the model tree and select **Hide** so that all the reference geometry is hidden.	

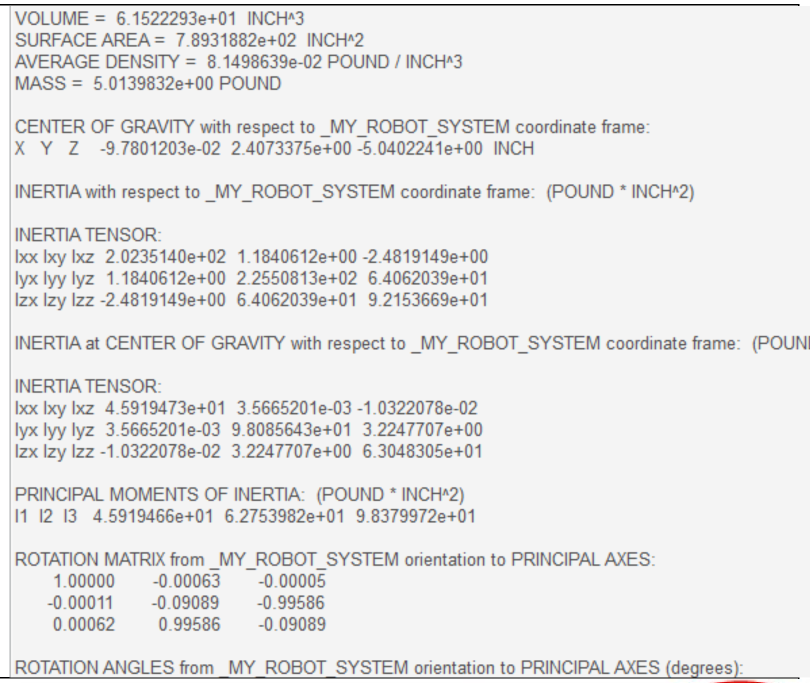
That will only hide the system reference geometry so you will need to hide the reference geometry in all of the subassemblies as well.	
Now select the **Analysis** tab in the upper menu and then click on the **Measure** tool.	

Click on the **Measure Distance** tool and then select two points, (remember to use the **CTRL** key to select multiple points). *There are several measurement tools available. You can measure length, angle, diameter, surface area, volume, etc.*	
Now close the measurement tools and select **Mass Properties** in the upper menu. Then click on the eye glasses to calculate the properties.	

Notice that there are many properties that are calculated.

These properties are calculated using the material information that has been provided or defaulted for each of the parts used in the assembly.

We will talk about how to validate or test these calculations in the next exercises.


VOLUME = 6.1522293e+01 INCH^3
SURFACE AREA = 7.8931882e+02 INCH^2
AVERAGE DENSITY = 8.1498639e-02 POUND / INCH^3
MASS = 5.0139832e+00 POUND

CENTER OF GRAVITY with respect to _MY_ROBOT_SYSTEM coordinate frame:
X Y Z -9.7801203e-02 2.4073375e+00 -5.0402241e+00 INCH

INERTIA with respect to _MY_ROBOT_SYSTEM coordinate frame: (POUND * INCH^2)

INERTIA TENSOR:
Ixx Ixy Ixz 2.0235140e+02 1.1840612e+00 -2.4819149e+00
Iyx Iyy Iyz 1.1840612e+00 2.2550813e+02 6.4062039e+01
Izx Izy Izz -2.4819149e+00 6.4062039e+01 9.2153669e+01

INERTIA at CENTER OF GRAVITY with respect to _MY_ROBOT_SYSTEM coordinate frame: (POUN

INERTIA TENSOR:
Ixx Ixy Ixz 4.5919473e+01 3.5665201e-03 -1.0322078e-02
Iyx Iyy Iyz 3.5665201e-03 9.8085643e+01 3.2247707e+00
Izx Izy Izz -1.0322078e-02 3.2247707e+00 6.3048305e+01

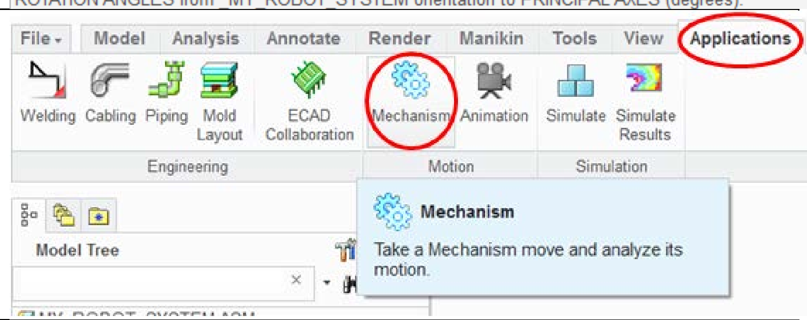
PRINCIPAL MOMENTS OF INERTIA: (POUND * INCH^2)
I1 I2 I3 4.5919466e+01 6.2753982e+01 9.8379972e+01

ROTATION MATRIX from _MY_ROBOT_SYSTEM orientation to PRINCIPAL AXES:

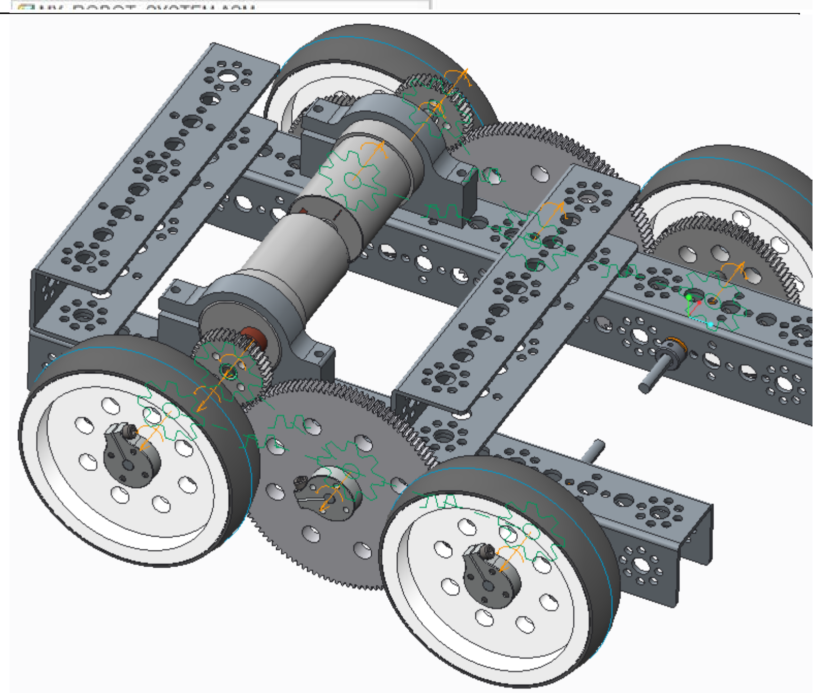
1.00000	-0.00063	-0.00005
-0.00011	-0.09089	-0.99586
0.00062	0.99586	-0.09089

ROTATION ANGLES from _MY_ROBOT_SYSTEM orientation to PRINCIPAL AXES (degrees):

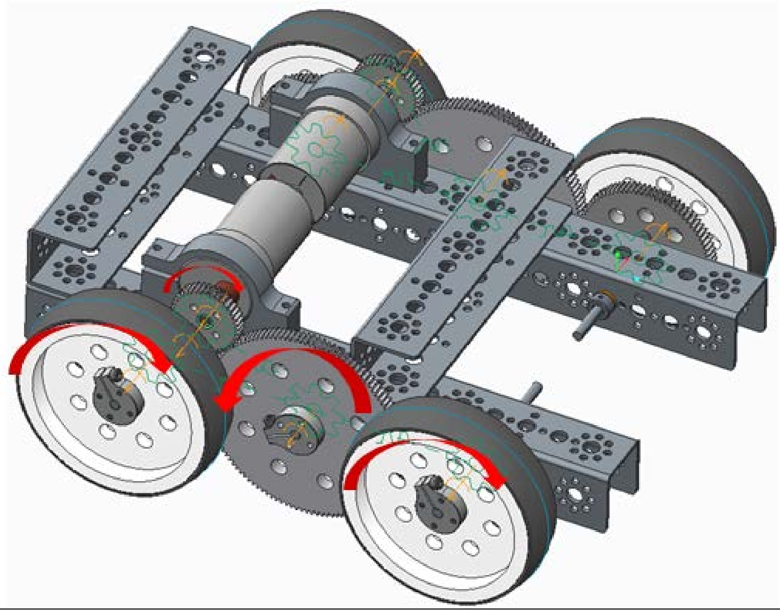
Close the **Mass Properties** dialog box and select the **Applications** tab in the upper menu. Then select **Mechanism**.



There are several different types of analyses that we can perform in the ***Mechanism*** *application. We will do two different types with the PTC robot; a kinematic analysis and a dynamic analysis.*

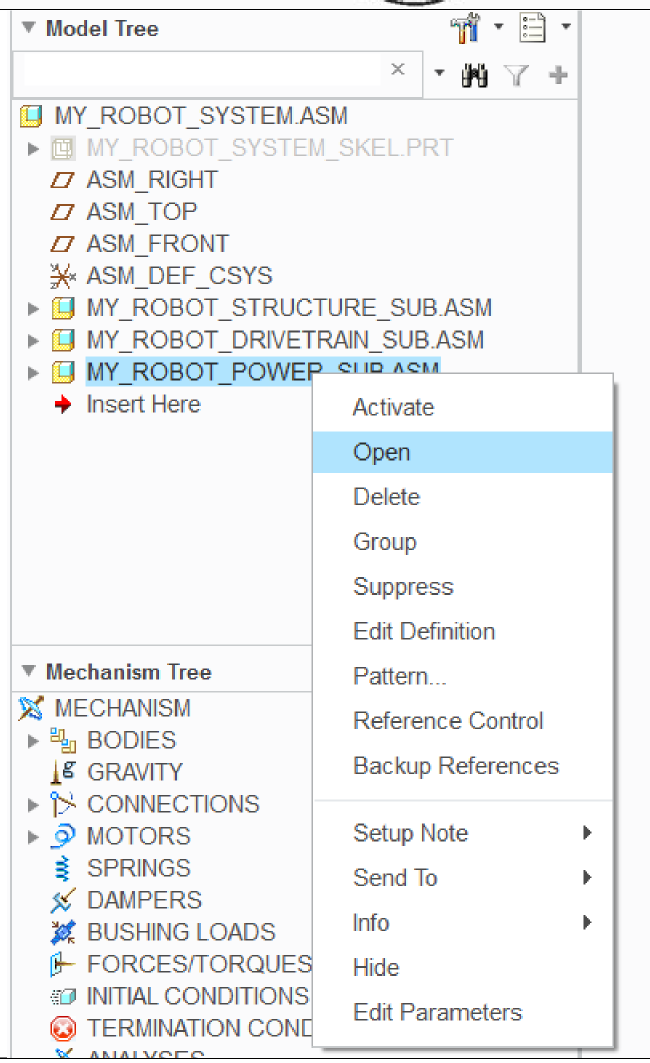


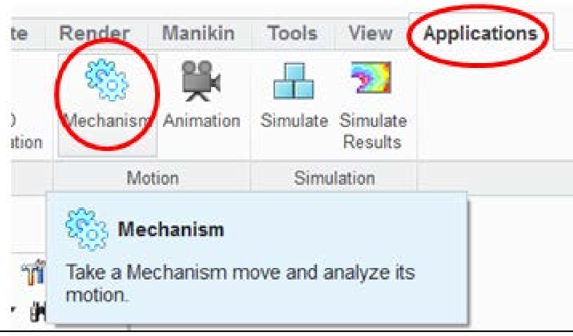
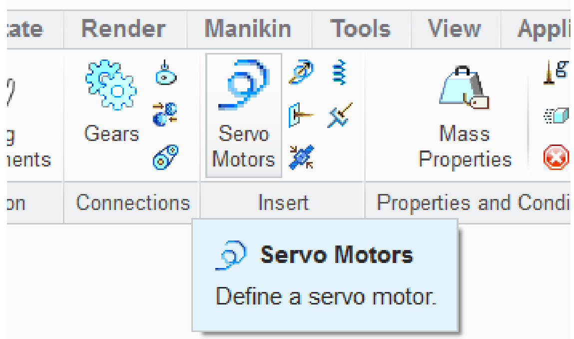
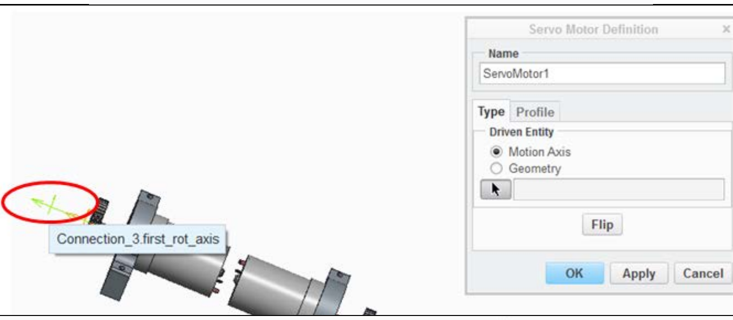
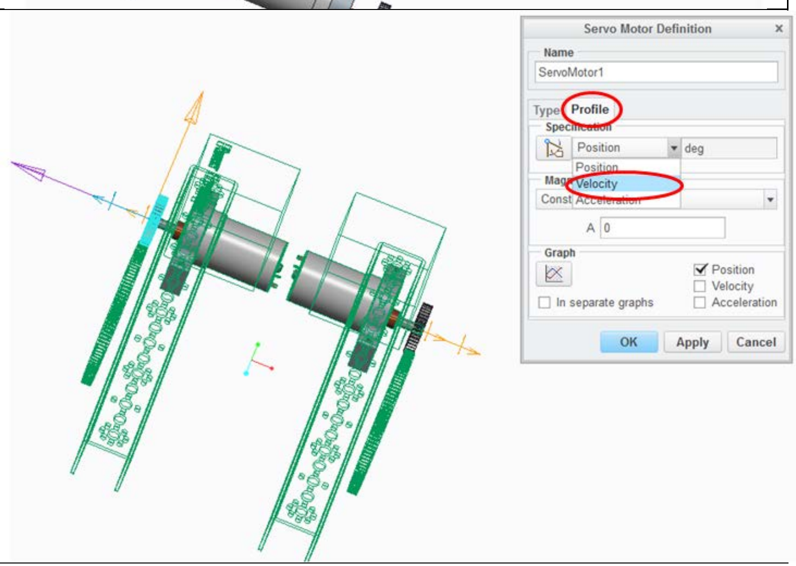
A kinematic analysis is a simulation of how all the moving parts will work to make sure everything is moving correctly. There are no forces involved only motion.

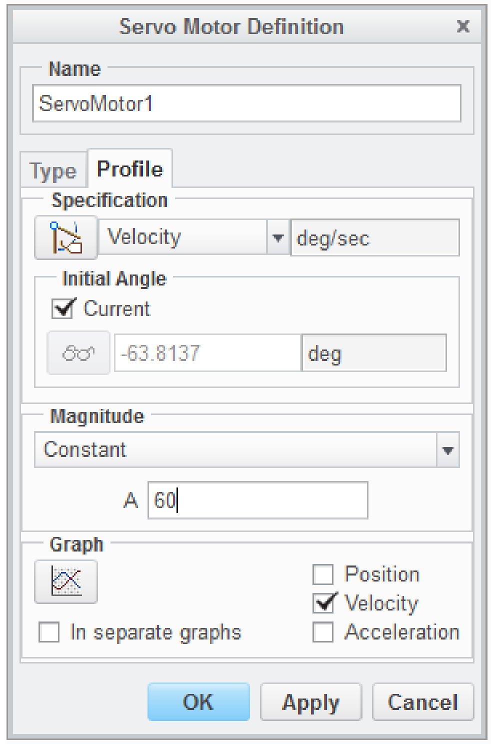
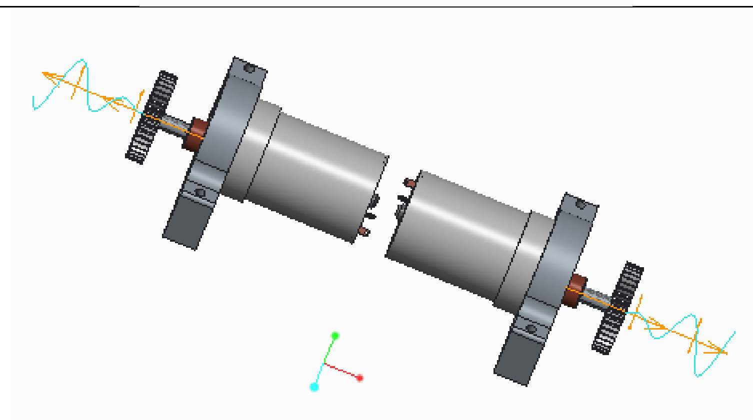
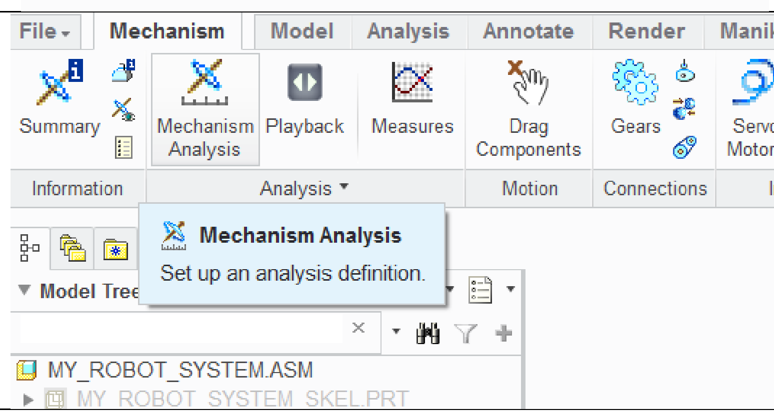


In order to conduct a kinematic analysis, we need to define a servo motor that will drive the motion. Left click on **MY_ROBOT_POWER_SUB.ASM** in the model tree and select **Open**.

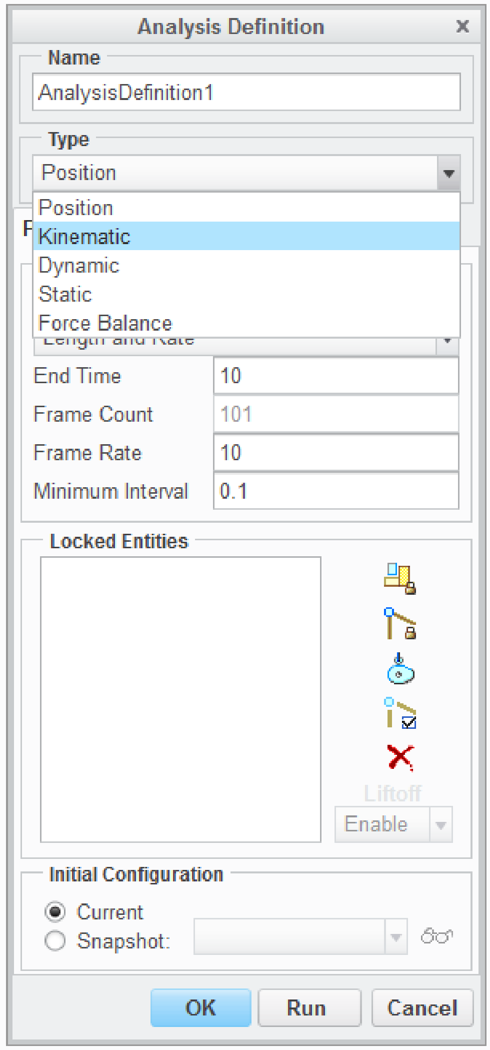
We will define the servo motors within the power subassembly model.



Now that the subassembly model is open, go to **Applications** and **Mechanism** so that we can define the servo motor.	
Select **Servo Motors** in the upper menu.	
Then select the axis icon associated with the gear as shown, by left clicking on it.	
Now select the **Profile** tab and then select **Velocity** as shown.	

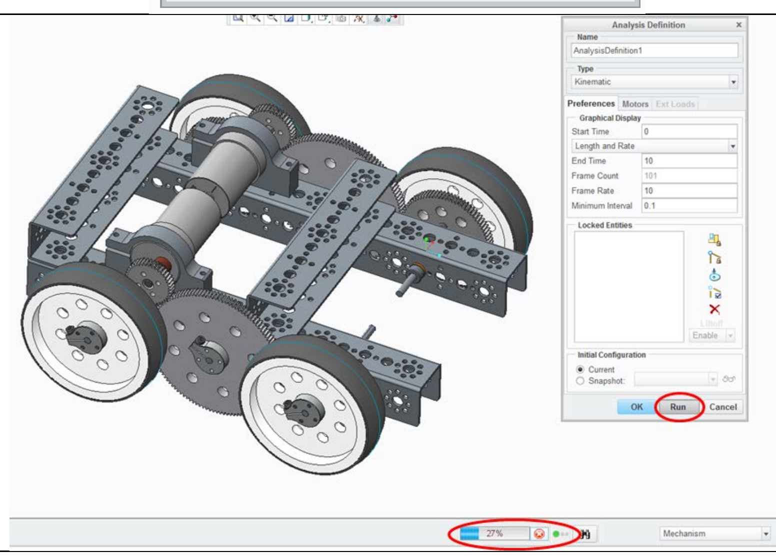
Now we need to set the velocity of the servo motor in degrees per second. For this exercise we will set it to **60** degrees per second. (How many RPMs is this?). Now click **OK**.	
Now do the same procedure on the second motor and gear. Your two motors should have servo motor icons as shown. Now save your model and close it.	
Now let's do a kinematic analysis. Select **Mechanism Analysis** in the upper menu.	

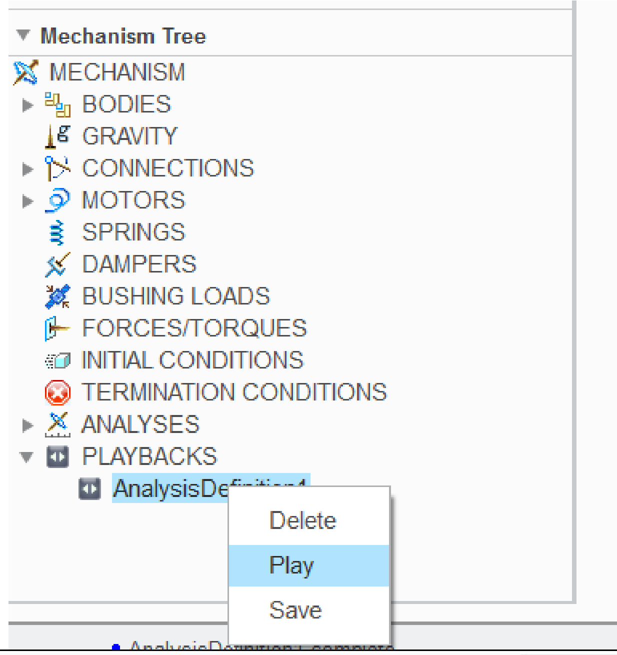
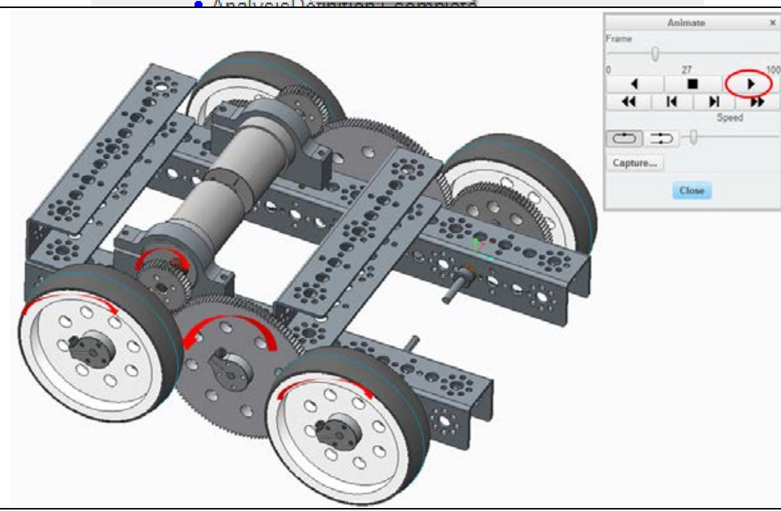
In the **Mechanism Analysis** dialog box, set the Type of analysis to **Kinematic**.

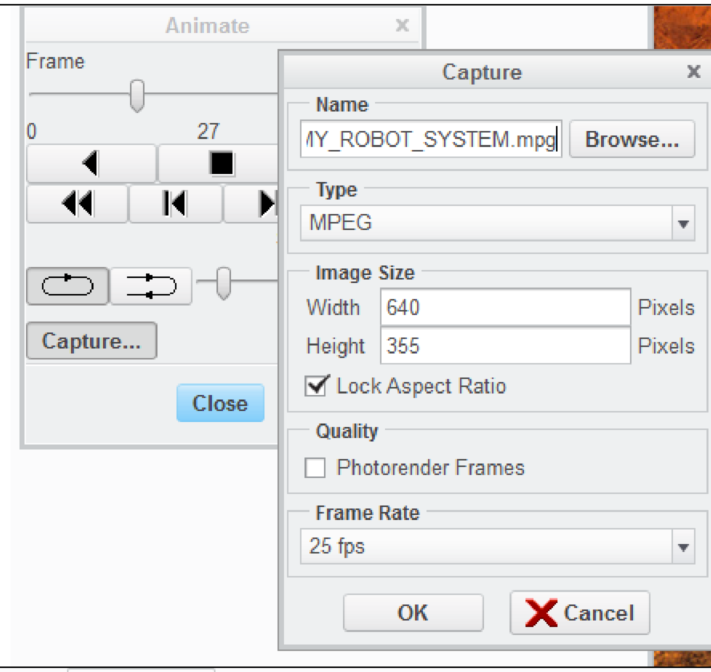
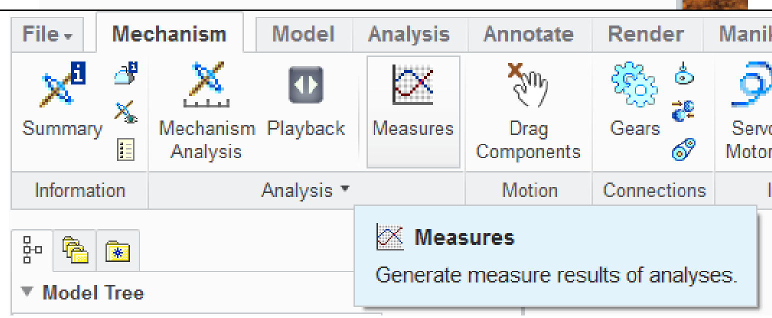


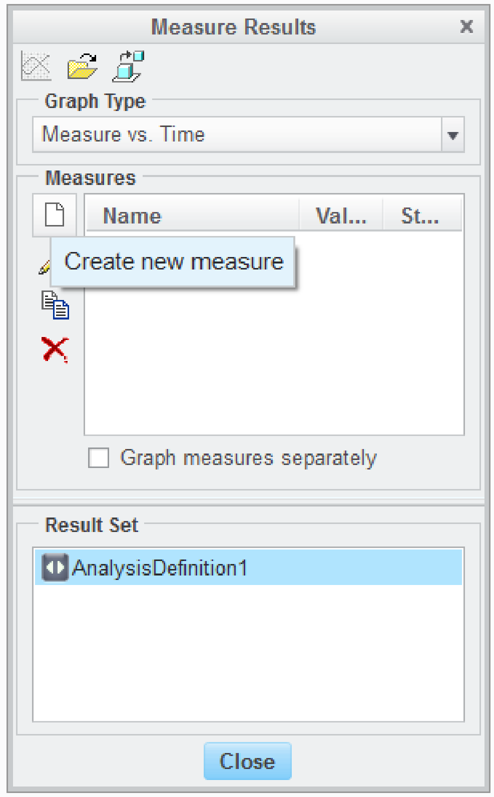
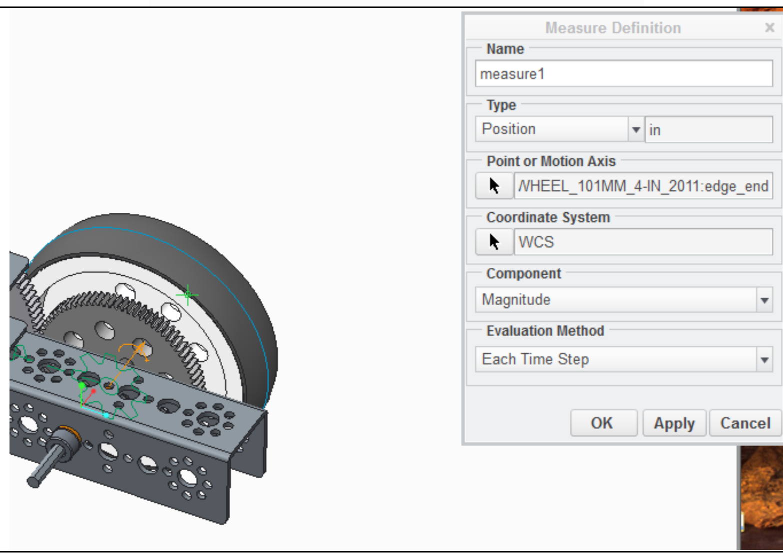
Now click **Run** and notice that as Creo calculates each of the positions in the analysis, there is a percentage shown at the bottom of the screen which indicates how far along in the calculations Creo is as well as a red stop sign for canceling the calculations.

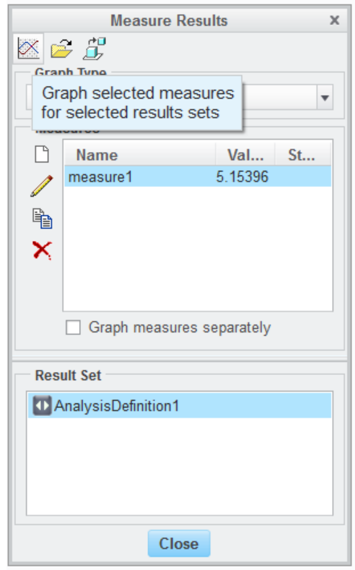
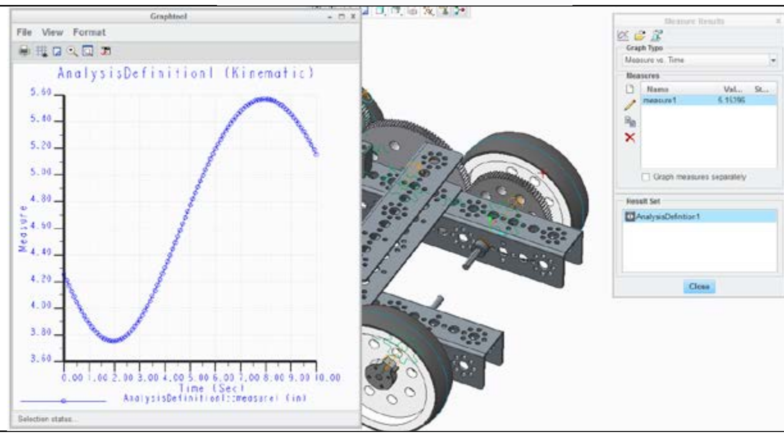
Once the calculations are done, click **OK** to finish.

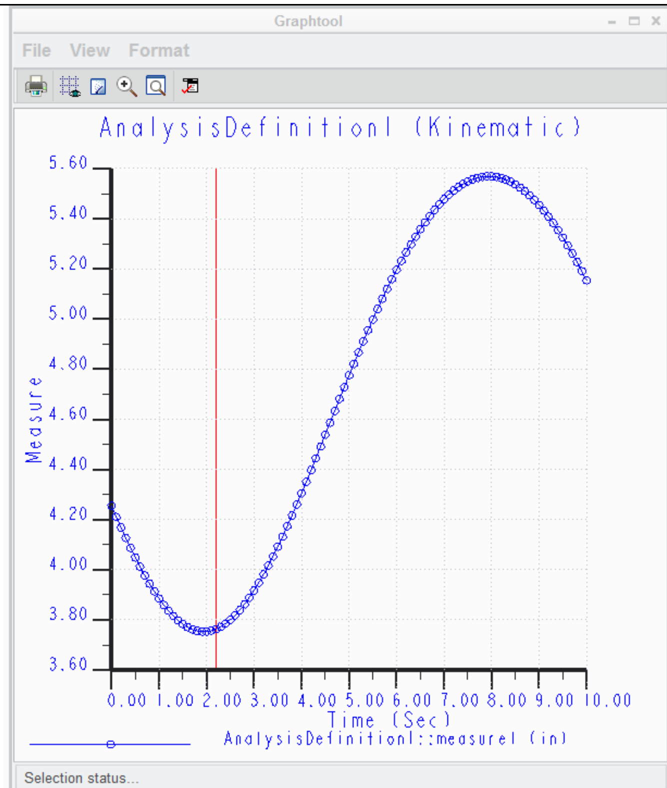
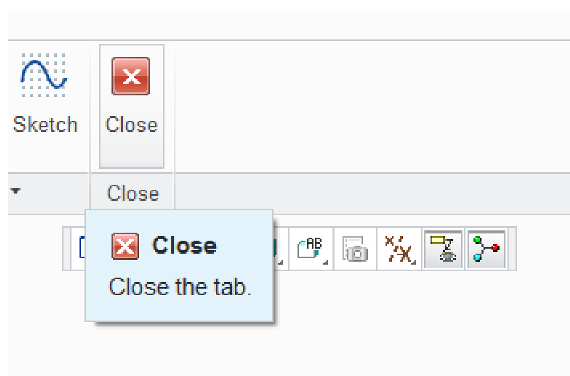


Expand the **PLAYBACKS** entry in the **Mechanism Tree** and right click and select **Play** to play back the analysis.	
You can now select the play button to watch the analysis again. Notice that you can rotate, zoom, or pan during the playback.	

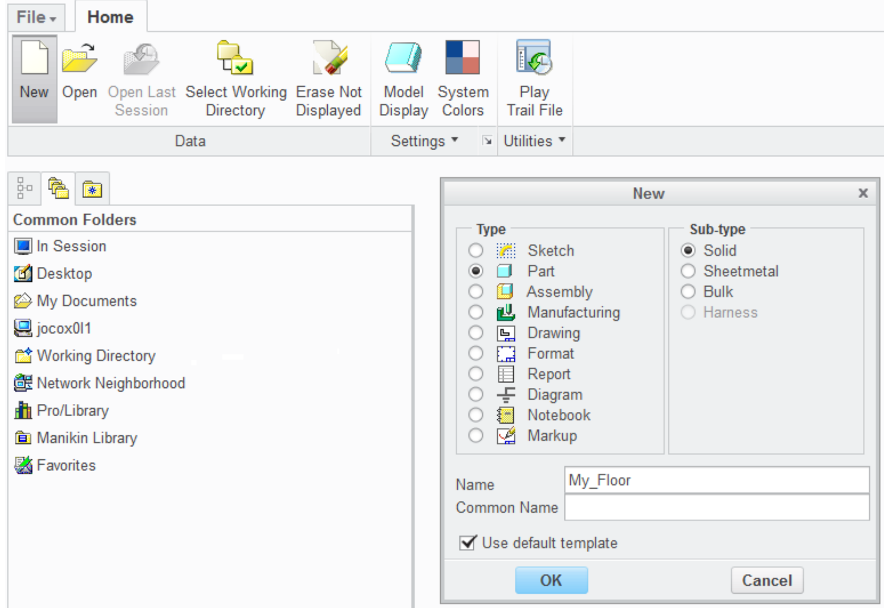
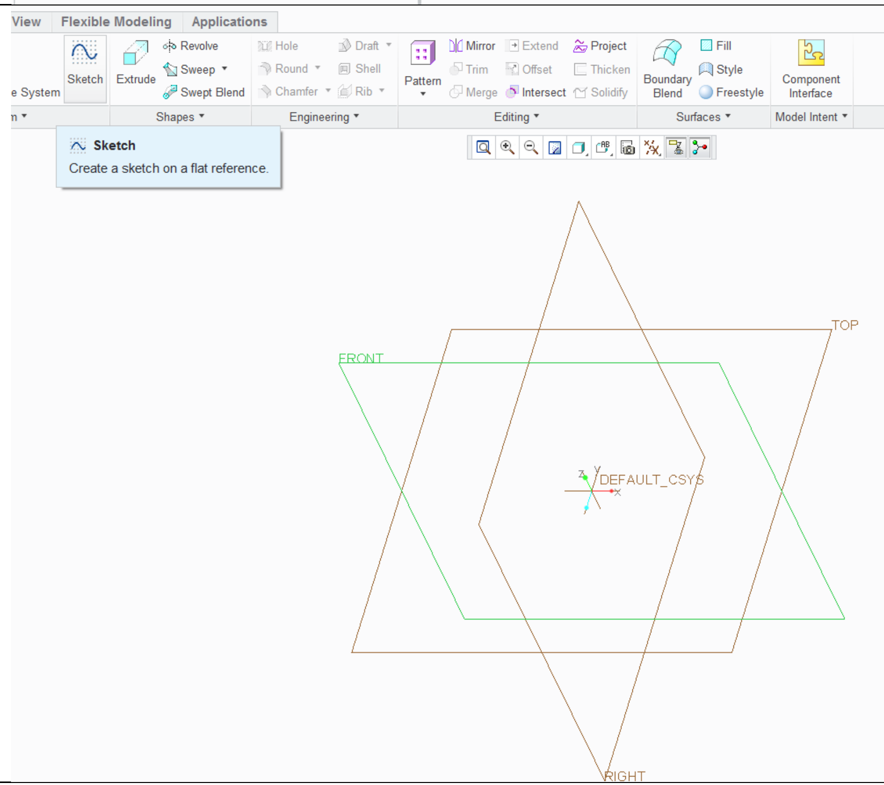
<table>
<tr>
<td>If you click on the Capture button, you can create a video of the analysis.</td>
<td>
</td>
</tr>
<tr>
<td>Another helpful tool is to measure values during the simulation like position, velocity, and acceleration. To do this close all the playback dialogs and click on Measures in the upper menu.</td>
<td>
</td>
</tr>
</table>

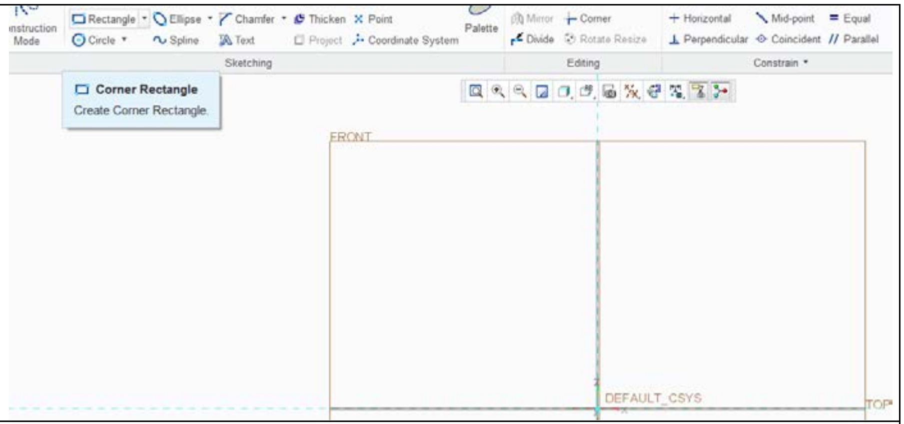
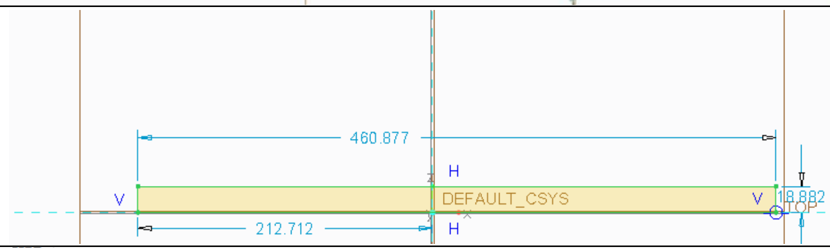
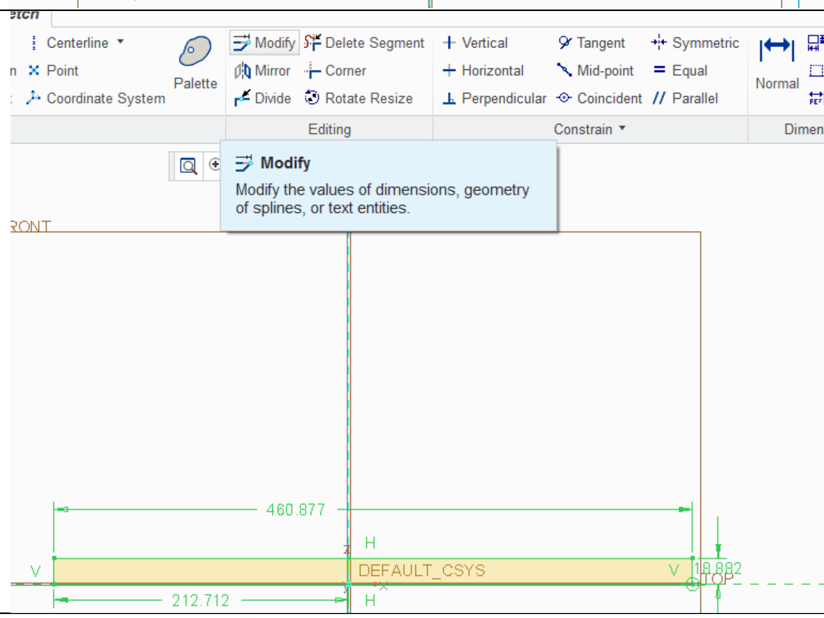
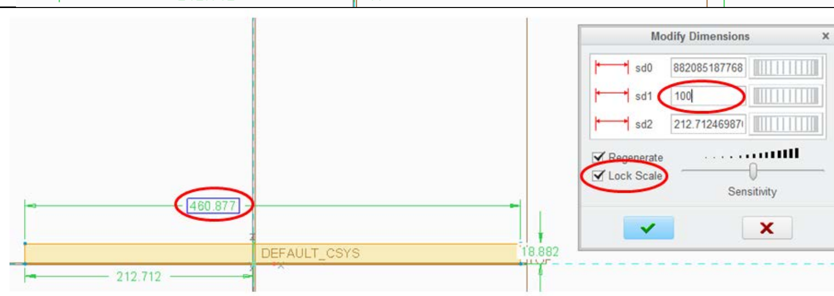
Click on the **AnalysisDefinition1** in the bottom of the dialog box and then select **Create new measure**.	
Select a point on one of the wheels by left clicking and click **OK**.	

<table>
<tr>
<td>Then select the Graph icon to see the new measure.</td>
<td>
</td>
</tr>
<tr>
<td>The position of that point on the wheel is plotted throughout the entire analysis.</td>
<td>
</td>
</tr>
</table>

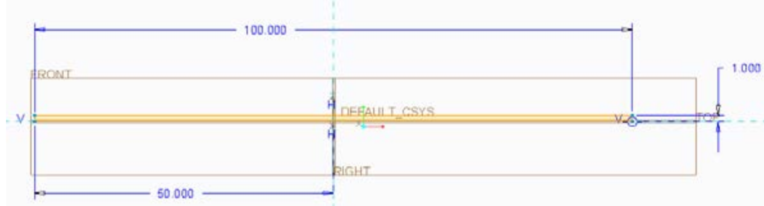
If you close the dialog box but leave the graph open and then play the analysis playback again, a red line will indicate the position on the graph as the wheel turns.	
This analysis allows you to see what motion will occur when the gear pair connections and all other moving parts are driven by the appropriate motors. However, it will not provide any information about forces or accelerations. Close the **Mechanism** application by clicking on the red **Close** icon and choose exit without saving.	

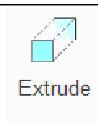
We have completed a kinematic analysis. If all of the parts move correctly, then we can run another simulation that deals with the kinematic motion under the influence of forces such as gravity, friction, etc. To do this simulation, we need to create a floor that we can place the PTC robot on and apply gravity and friction forces.

Close the robot model file and create a **New** part file called: "**My_Floor.prt**"	
Select the **Front** plane and then click on the **Sketch** icon to enter Sketch mode. (You may have to turn on the datum planes using the datum filter )	

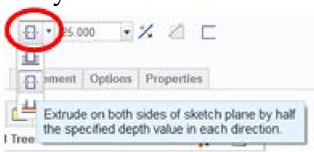
Orient your view so you are looking straight on the sketching plane, (use the **Sketch View**  tool). Then select **Corner Rectangle**.	
Create a corner rectangle by left clicking on the two ends and then middle clicking to exit the rectangle tool.	
Drag select the entire rectangle and select the **Modify** tool in the upper menu.	
Check the box to **Lock Scale** and then set the length of the rectangle to **100** and click the green check mark.	

Now set the thickness to **1** and center it. Then click the green check mark to exit Sketch mode.

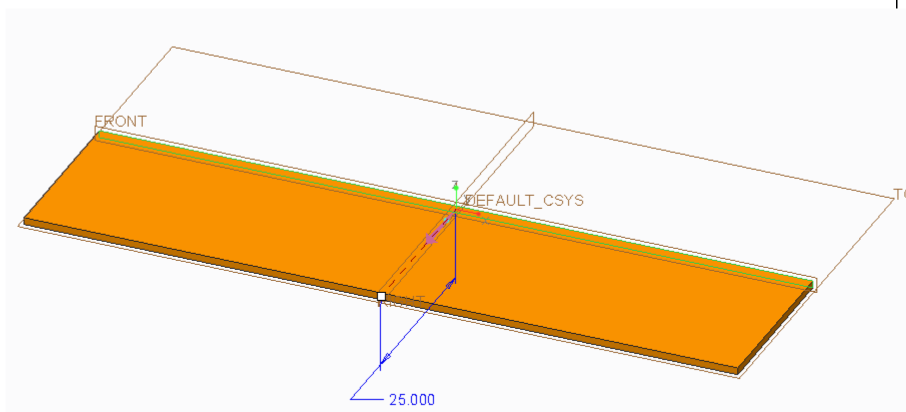




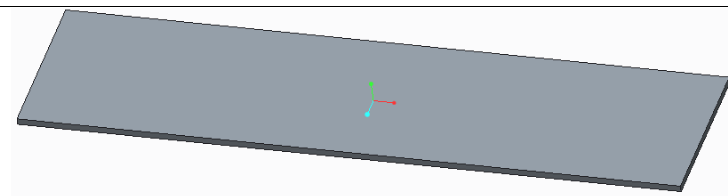
Select **Extrude** and then set the width of the floor to **25** and also set the extrude to be symmetric



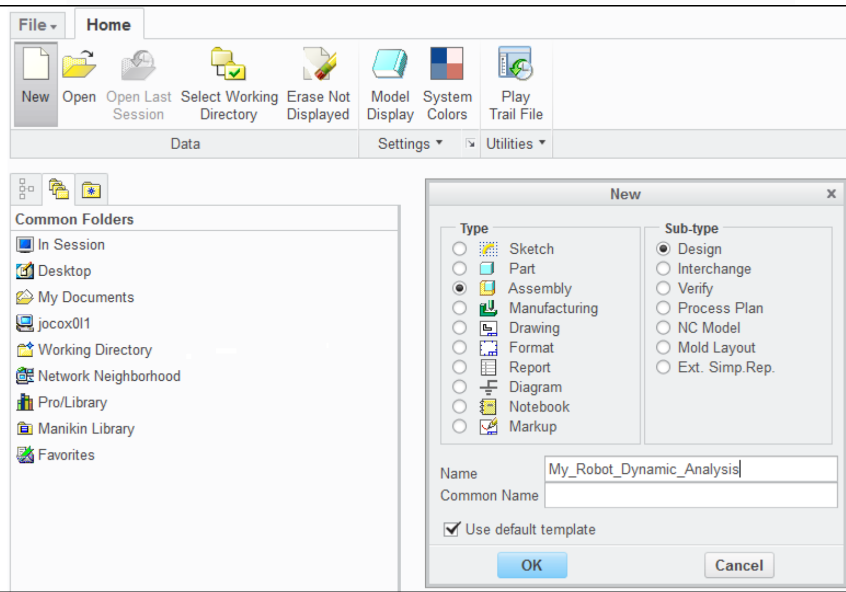
and then click the green check mark.



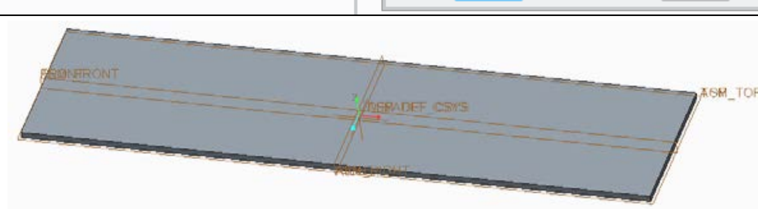
Now **Save** the model of the floor and **Close**.

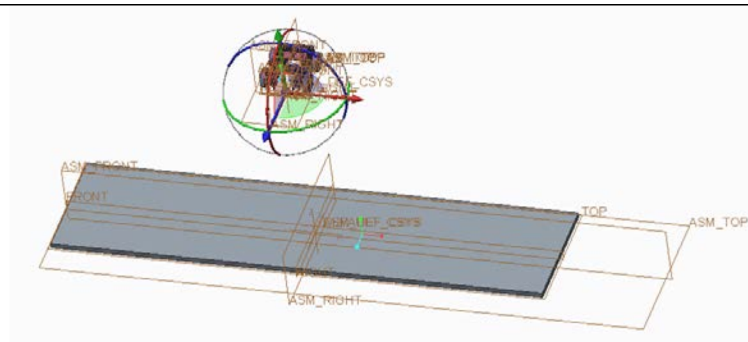
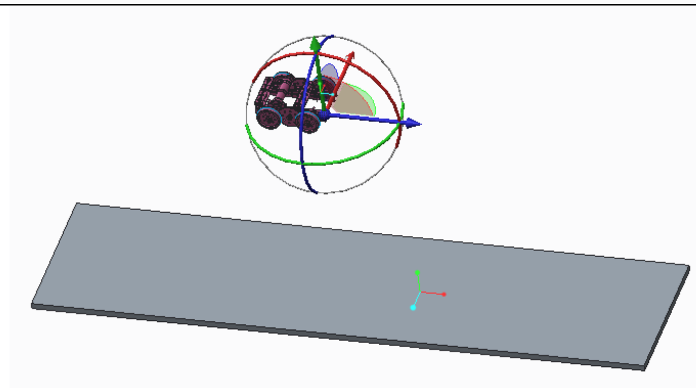
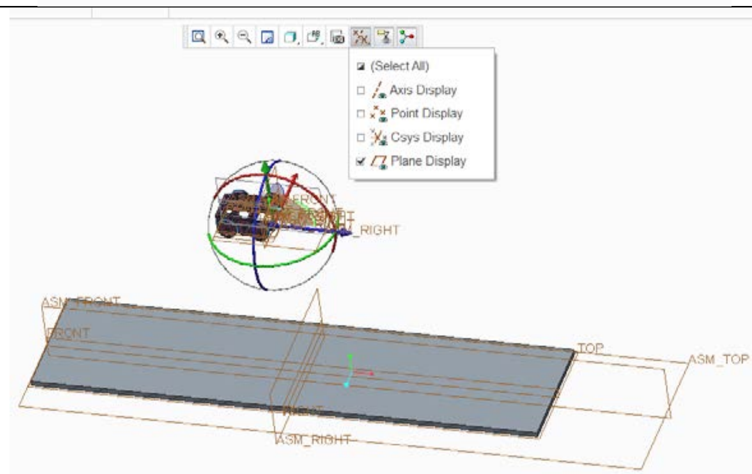
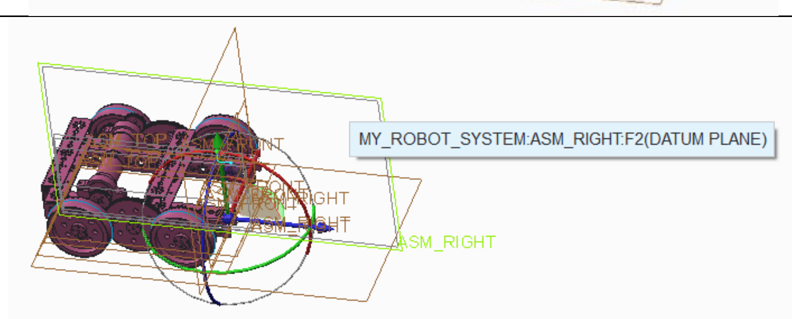


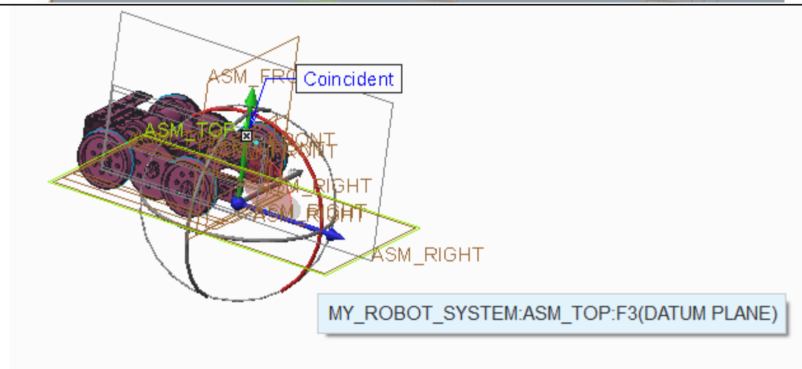
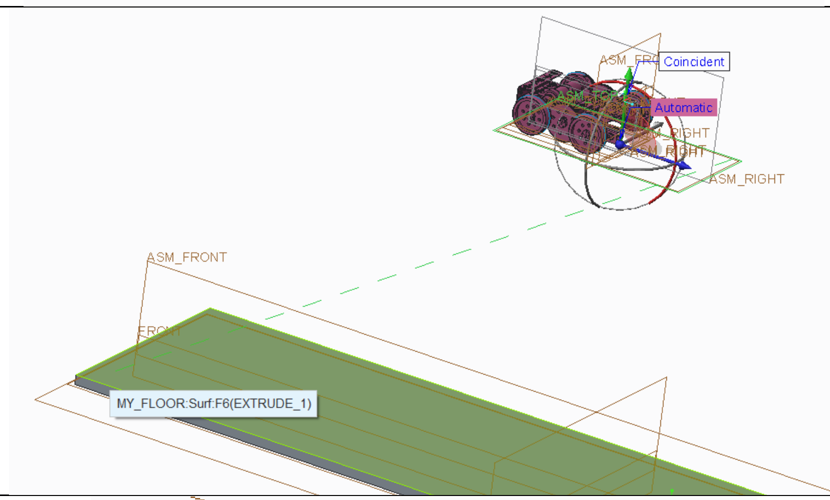
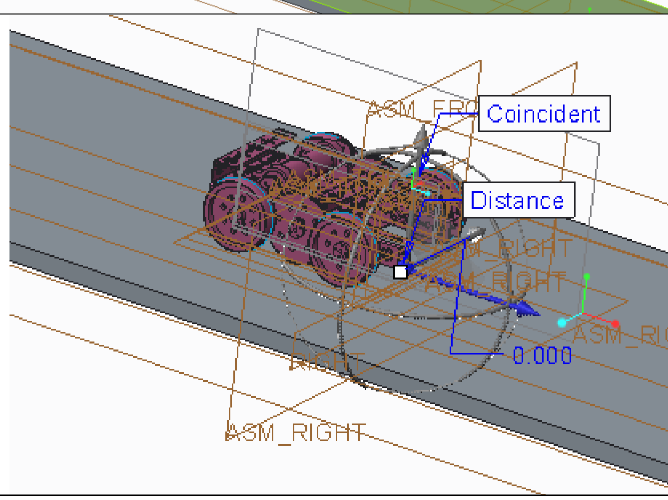
Now create a new assembly model called: “**My_Robot_Dynamic_Analysis**”.

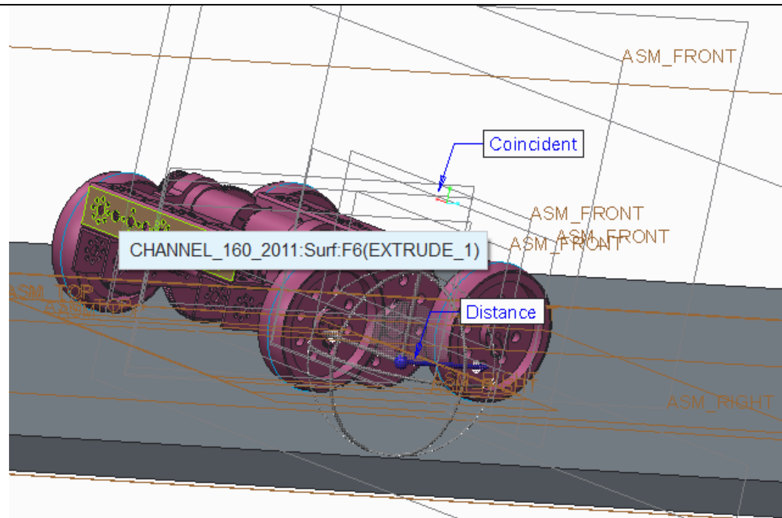
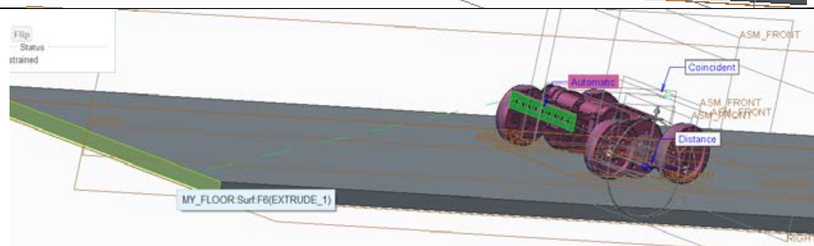
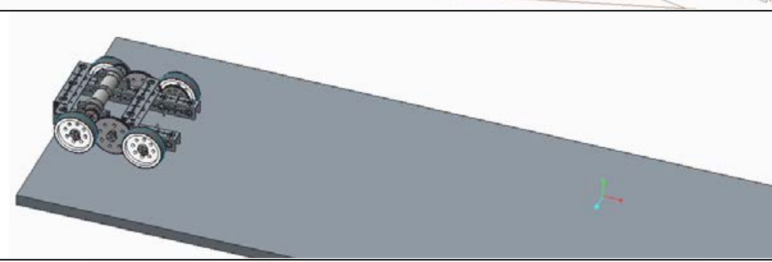
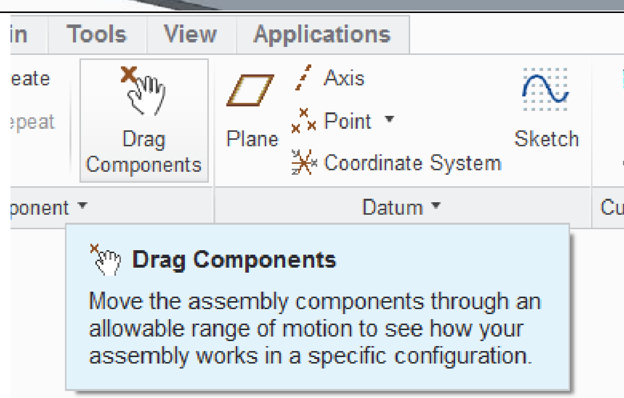


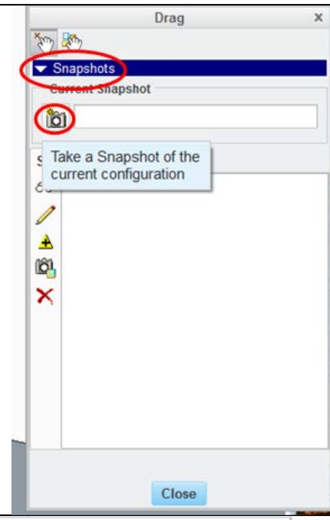
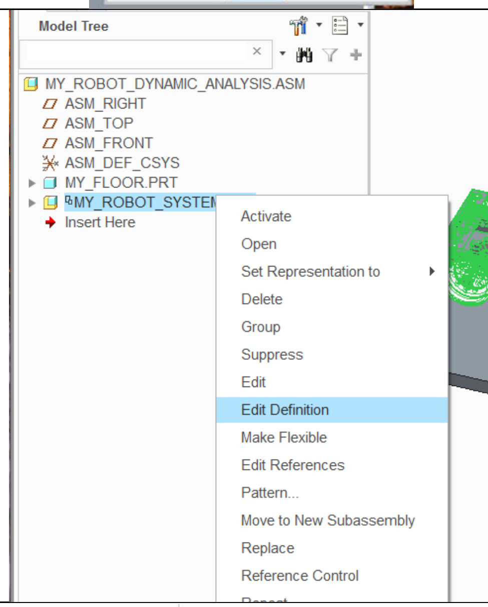
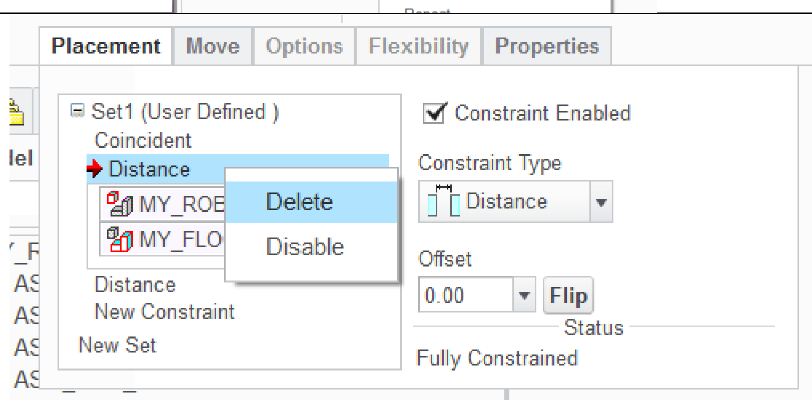
Click on the **Assemble** icon and choose “**My_Floor.asm**”. Left click to place it and then select **Default**

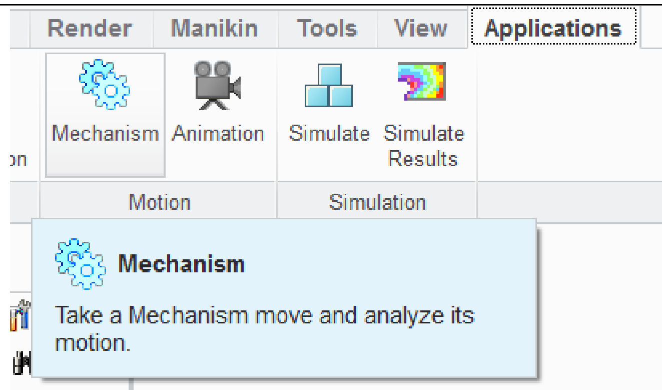
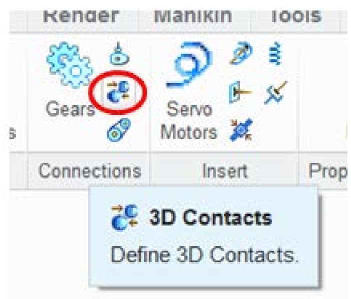
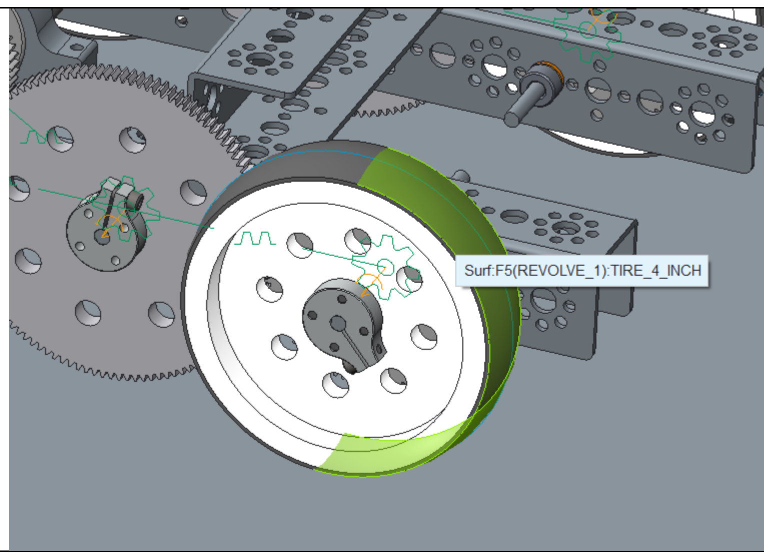


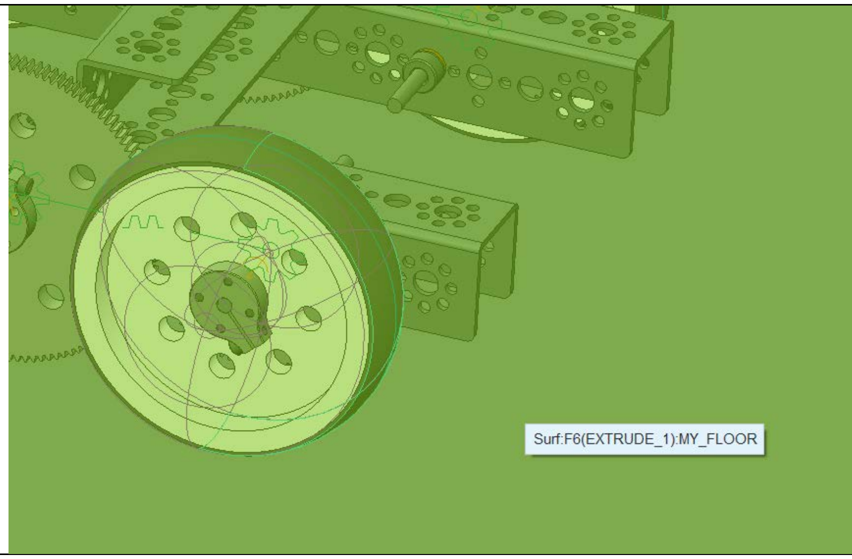
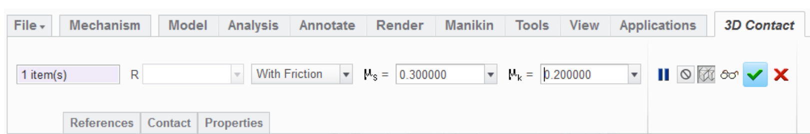
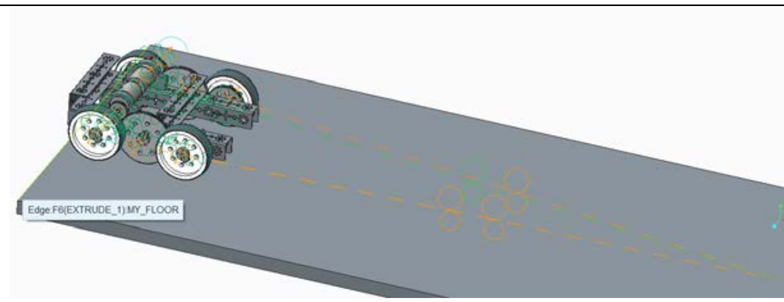
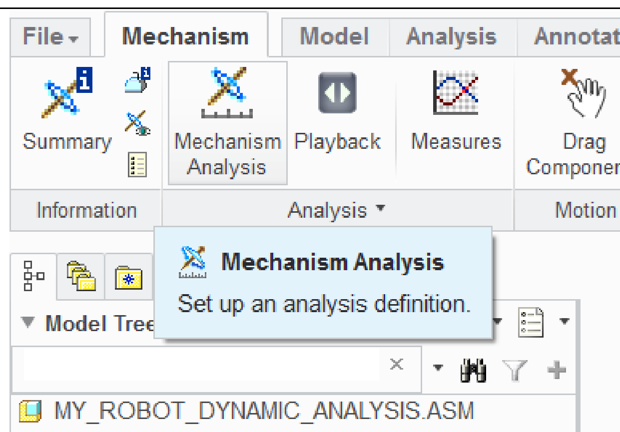
<table>
<tr><td>from the Automatic menu. Then click the green check mark to finish placing the floor.</td><td></td></tr>
<tr><td>Select the Assemble icon again and choose “my_robot_system.asm” to bring your robot model into the assembly. Left click to place it and then turn off the datum planes.</td><td>
</td></tr>
<tr><td>Use the orientation sphere to orient your robot as shown.</td><td></td></tr>
<tr><td>Then turn only the datum planes back on.</td><td>
</td></tr>
<tr><td>Select the Right plane in the robot assembly.</td><td>
</td></tr>
</table>

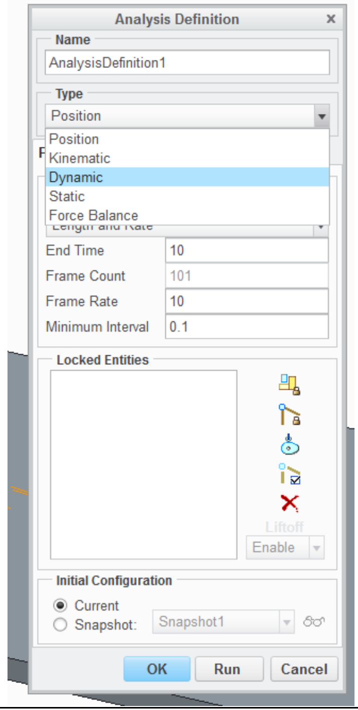
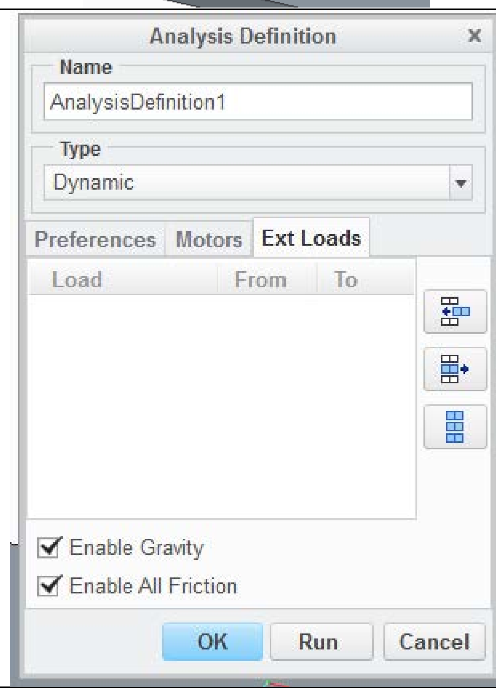
<table>
<tr><td>And the Front plane in the assembly. This will align the robot to the center of the floor.</td><td>
</td></tr>
<tr><td>Now select the TOP plane in the robot assembly.</td><td>
</td></tr>
<tr><td>Select the top of the floor.</td><td>
</td></tr>
<tr><td>Set the distance to 0. This will place the robot on the floor.</td><td>
</td></tr>
</table>

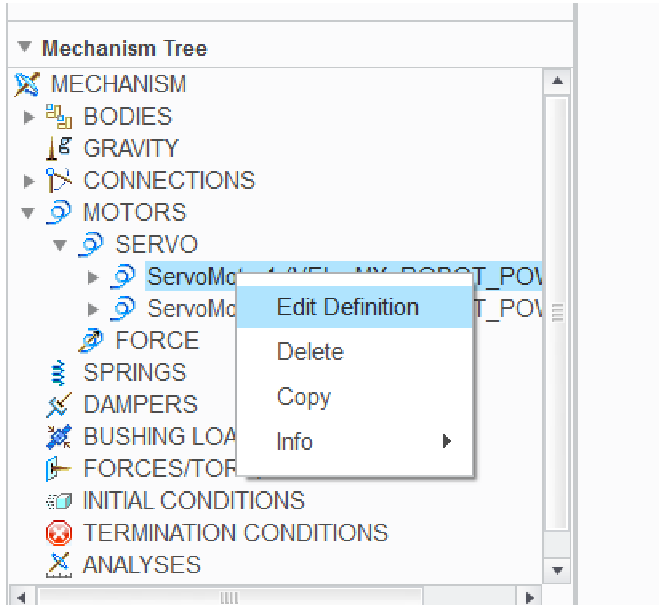
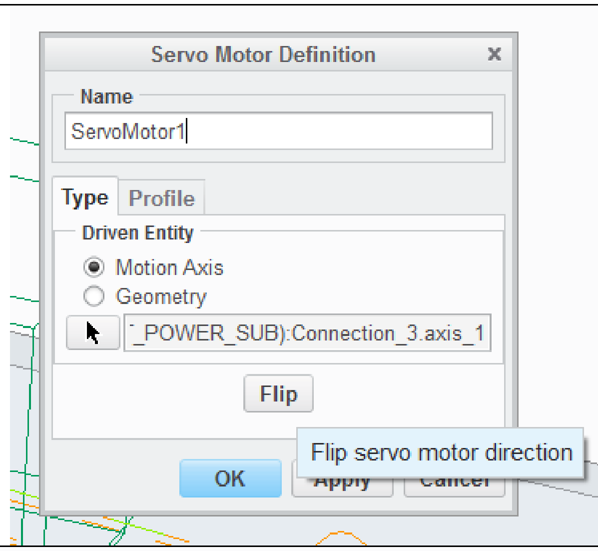
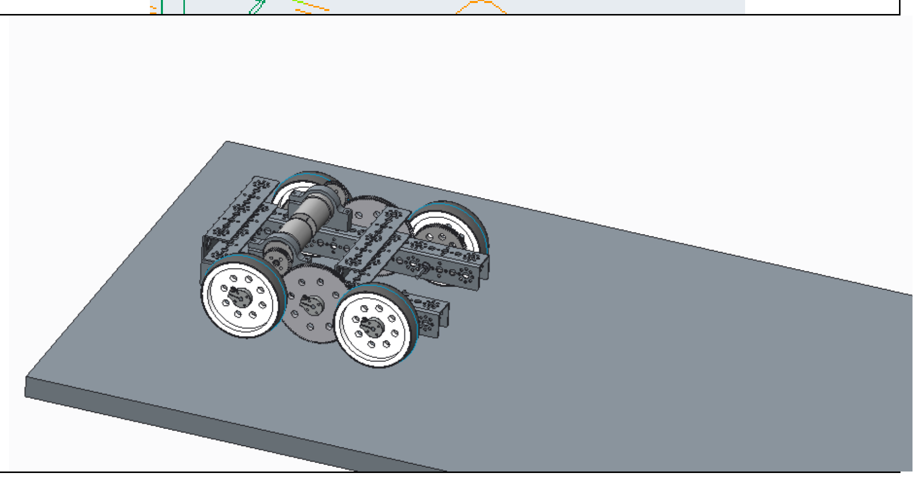
Select the back of the channel on the robot.	
And select the back of the floor to place the robot at the beginning point.	
Now select the green check mark to finish placing the robot at the beginning of the floor.	
Click on the **Drag Components** icon in the upper menu to set up an initial starting point. This will allow us to run the analysis and then with a simple click, set the robot back to its starting position.	

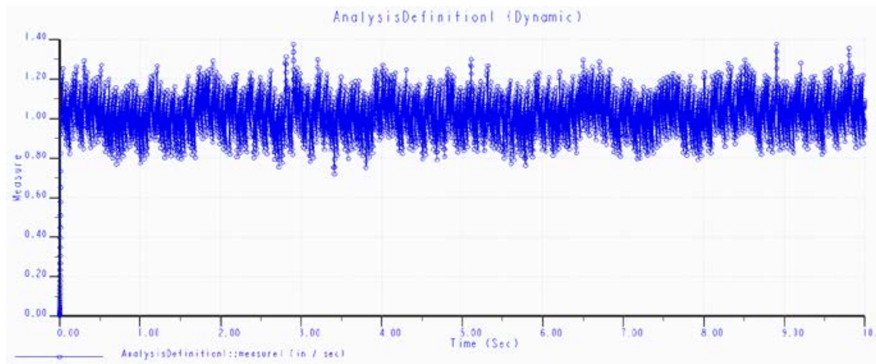
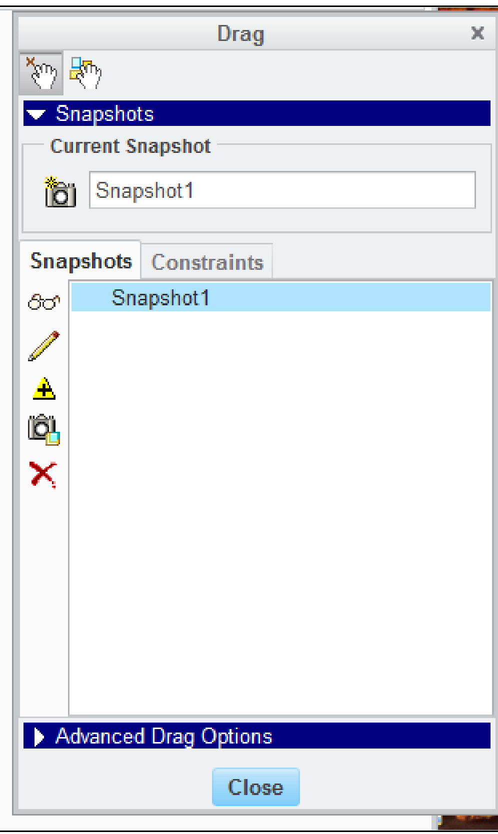
Expand the **Snapshots** arrow and then click on the camera icon to record the current location. Then close the dialog box.	
We are now going to release the robot from two of the constraints so that it can move when we turn on the motors. Find **MY_ROBOT_SYSTEM** in the model tree and right click and select **Edit Definition**.	
Open the **Placement** tab and then delete the two **Distance** constraints by right clicking on them and selecting **Delete**. Then click the green check mark to finish.	

Now go into the **Mechanism** application by selecting **Applications** in the upper menu and then **Mechanism**.	
We need to define a connection or relationship between the wheels and the floor. To do this we will use a **3D Contact** connection. Select the **3D Contact** icon in the upper menu.	
Then select the front of one of the robot wheels.	

<table>
<tr>
<td>And then the floor.</td>
<td>
</td>
</tr>
<tr>
<td>Then change the pull down menu in the upper dashboard to With Friction and set the value of the static friction to .3 and the value of the kinematic friction to .2 and then click the green check mark.</td>
<td>
</td>
</tr>
<tr>
<td>You have now established a 3D connection with the selected wheel and the floor. You need to do the same for each of the wheels.</td>
<td>
</td>
</tr>
<tr>
<td>Now we are ready to run an analysis. Select Mechanism Analysis in the upper menu.</td>
<td>
</td>
</tr>
</table>

<table>
<tr>
<td>Change the type of analysis to Dynamic.</td>
<td>
</td>
</tr>
<tr>
<td>Click on the Ext Loads tab and select to Enable Gravity and Enable All Friction. Then select Run.</td>
<td>
</td>
</tr>
</table>

Watch while the analysis is being calculated to make sure the wheels are turning in the right direction. If not, stop the analysis by clicking on the red stop sign and then click **Close**. Expand the **MOTORS** entry in the **Mechanism Tree** and then expand the **SERVO** entry and right click on the servo motors and select **Edit Definition**.	
Then select **Flip** to flip the direction of the motors. Then select **OK**. Make sure both motors are turning in the right direction.	
Once you have both motors oriented correctly, go back to the **Mechanism Analysis** and select a **Dynamic** analysis, set the **Ext Loads** so that gravity and friction are enabled and then select **Run**.	

Once it has finished, you can play it back and look at measures. Here is the velocity of the robot during the analysis. Can you figure out why the graph looks like this? Do the gears have any impact on the velocity? Why isn't it more smooth? Could there be slippage from there not being enough friction between the wheels and the floor?	
Now let's go back and change the speed of the servo motors and try the analysis again. Find the servo motors in the **Mechanism Tree** and change their rate to **360** degrees/sec. Then click on the **Drag Components** icon in the upper menu and double click on **Snapshot1** to place the robot back at the beginning. Run the analysis again.	

You have finished the exercise.