CAD and 3D modeling for FTC with PTC Creo - Assemblies

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Alex Averill UT-Dallas Science and Engineering Education Center Introduce PTC Creo Parametric

Demonstrate features most useful to FTC teams

Illustrate use of FTC Kit of Parts library for CAD

Use Creo Parametric to design a simple chassis

Demonstrate how to model 3D parts

Introduction to 3D printing

Especially at advanced championship events, judges expect to see teams use a good design and documentation process when developing their robot.

Using good CAD (computer aided design) tools and techniques sets top teams apart from other teams at competitions.

CAD also enables 3D modeling and the use of 3D printed parts.

For this exercise we will create a simple chassis assembly from TETRIX parts.



To get started, create a folder where the assembly files will be stored, such as *My Documents\simplebot*.

Unzip the FTC TETRIX Kit of Parts file into this directory. This provides us with a library of pre-built TETRIX parts that we can use to assemble the chassis.

## Starting with Creo Parametric

Start PTC Creo Parametric 3.0.

You may see some "Resource Center" windows after starting Creo Parametric; if you do, close them.

First we want to tell Creo the folder we will be working in. Click **Select Working Directory**, navigate to the folder where our model is stored (*My Documents\simplebot*), and click **OK**.





Click the **New** icon at the top left corner to create a new file.

Select **Assembly** in the dialog box, enter a name for your new assembly ("simplebot"), and click **OK**.

In Creo we have parts and assemblies. A *part* is a single component, such as a channel, gear, or wheel. An *assembly* is a collection of parts and other assemblies.



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#### After starting a new assembly, your screen will look like:



## **Placing parts**

Click the **Assemble** tool, then navigate to the *Tetrix\_Kit\_FTC* folder.

Initially Creo displays the available parts only by their filenames, which is hard to navigate. Click **Views** and then **Thumbnails** to also see what each part looks like.



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In the parts window, select the 288mm channel part (#739068) and click **Open**.



The channel part will appear in purple in the graphics area and will follow your pointer.

Left-click anywhere to drop it into the model space.



When you left-click to drop the channel, an orientation sphere appears at one end of the channel.

The sphere allows us to orient the current part by clicking on an axis to rotate or the circles to rotate.

Since the channel is the first part of this assembly, we need to constrain it to the origin. Use the **Automatic** pull-down to select the **Default** constraint.



Now the channel turns orange to indicate it is fully constrained in the model.

Click on the green check mark in the assembly dashboard to finish the placement of the channel.

Now turn off the display of the datums by clicking the datum display tool and unchecking all of the boxes.







## Changing model views

You can use the mouse to change the view orientation of the model at any time. Become familiar with the following:

1. Holding the middle mouse button and dragging will rotate the model view.

2. SHIFT + the middle mouse button will pan the model view.

3. CTRL+D will reset the model view to a "default" orientation and size.

4. The mouse scroll wheel will zoom the view in and out.\*

\*Windows 10: If the mouse scroll wheel isn't working, be sure "Scroll inactive windows" is turned off in Windows' Mouse settings.

# Adding a constrained part

Now let's add a bushing to our model. Click on **Assemble** and open the parts library, then select the bronze bushing (#739091) and click **Open**.

The bushing appears purple and follows the pointer. Left-click to drop it near the third hole of the channel.

The orientation sphere appears.





Use the orientation sphere arcs to orient the bronze bushing in the general direction for insertion in the middle hole.

It's not important to be precise.

Now use the left mouse button to apply constraints by clicking on the cylindrical surface of the bushing and the cylindrical inside of the third hole.



Once you have selected the two cylindrical surfaces, Creo places a **Coincident** constraint on the bushing that forces it to align with the hole.

Parts of the orientation sphere are made inactive, because the bushing can only move along the axis of the hole.



Important: When assembling parts, keep track of every click you make. Creo uses every left-click to determine the types of constraints to be added. Don't click arbitrarily, and use CTRL+Z to undo any extra clicks. The next constraint will define where along the axis the bushing should be placed.

Left-click the flat underside of the bushing's head, and the flat side of the channel.

This will place the bushing flush against the channel.

Click the green check mark to finish placing the bushing. (Clicking the middle mouse button is also a shortcut for this.)



Now let's add a bushing on the opposite side of the channel. Click **Assemble**, and again select the bronze bushing (#739091). Left click to drop it in the model space, and orient it towards the channel.

Creo remembers the two constraints we placed on the previous bushing. Therefore only need to left-click the hole and surface of the channel for the placement of this bushing.



Once the bushing is in the proper location, middle-click to finish its placement.

Note that the model tree panel is now showing three parts in our model – a channel and two bushings.





## Add axles, spacers, hubs

Now assemble an axle (#739088) to go inside the bushings. As before, drop and orient the axle in the general vicinity of the bushing.

Then use left-clicks to constrain the curved part of the axle to the inside of the bushing. Be sure to select the axle's curved surface and not the flat part.

Once the axle is constrained to the axis of the bushing, slide it into the bushings and middleclick to finish placement.





Add a nylon spacer (#739101) onto the axle. Use left-clicks to constrain the inside cylinder of the spacer to the curve of the axle, and the flat end of the spacer to the top face of the bushing.

You may notice the axle move; if so, we'll fix this in a moment.

Add a motor hub (#739172\_collar) to the assembly, aligning the inside cylinder of the hub with the axle and the flat face with the spacer.







Add a wheel assembly (#739025) to the axle hub. The inside cylinder of the wheel should be constrained to the outside cylinder of the hub flange.

After the wheel is added, press CTRL+D to see the full assembly. Drag with the middle mouse button to view from different angles.





## Modifying a part's placement

Because the axle has only one constraint, it may have moved when we attached other parts to it. We can adjust parts after they've been placed.



Right-click on the axle in the **Model Tree**, then click on the **Edit Definition** icon. We can then re-place the component to a new location.



#### Attaching a plate or channel

In the real world, we would use screws and nuts to attach plates and channels to our model. This can be tedious.

For many purposes we can just constrain mounting holes and faces instead.

Click **Assemble**, then select a mounting plate (#739073) from the TETRIX kit of parts.

Drop and orient the plate so that it can be attached to the flat surface of the channel.





Use left-clicks to constrain a pair of mounting holes you will use to attach the plate to the channel – exactly as if you were using a screw.

Constrain another pair of mounting holes. Be sure to select holes that correspond to the correct distance and placement. Mounting holes Coind Automatic

Left-click the bottom face of the plate with the top face of the channel to attach them together.

Middle-click to accept the placement.

Press CTRL+D to see the resulting model.





#### Constraints and the model tree

If we change the placement of a part in our model, then all of the other parts that depend on that part will move with it.

Suppose we decide to move our wheel to the end hole. The wheel, hub, and spacer are constrained to the axle, which is constrained to a bronze bushing.

Changing the placement of the bronze bushing will move the entire wheel assembly.



# Changing a part's placement

Right-click on the bronze bushing (#739091) in the **Model Tree**, then select **Edit Definition**.

The **Graphics Area** will revert to the model as of the point where the part is added, and display the constraints on the part.





Left-clicking on a new location will change the target of the highlighted constraint.

The bushing moves to its new location.

Middle-click to accept the new placement, and all of the other parts constrained to the bushing have also been moved.



Constraints can also be added, adjusted, and removed using the **Placement** tab while placing a part.

# Use **Edit Definition** to adjust the location of the other bronze bushing.





# **Deleting parts**

If you delete a part from the **Model Tree**, then all of the other parts that depend on it for placement will also be

deleted.





## Adding parts with new constraints

When adding a later instance of a part that has already been placed in the model, Creo remembers the constraints used for placing the earlier part.

To start over with new constraints for the new part, use the **Place Manually** button in the placement ribbon.





### **Further Resources**

#### STEP files

- Widely used interchange format for 3D objects
- Companies such as AndyMark and ServoCity publish STEP files for parts they sell
- Download and install like FTC Kit of Parts files

#### How to Model Almost Anything guide

- https://www.ptcusercommunity.com/docs/DOC-4618
- Section 2, Exercise 5: Geared drive train with moving parts