creo™ parametric

PTC CREO WORKSHOP

Manual for Cero Parametric 2.0 Student Edition

Powerstackers FTC 5029 | August 6, 2014

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Contents

Chapter 1: Using PTC Creo

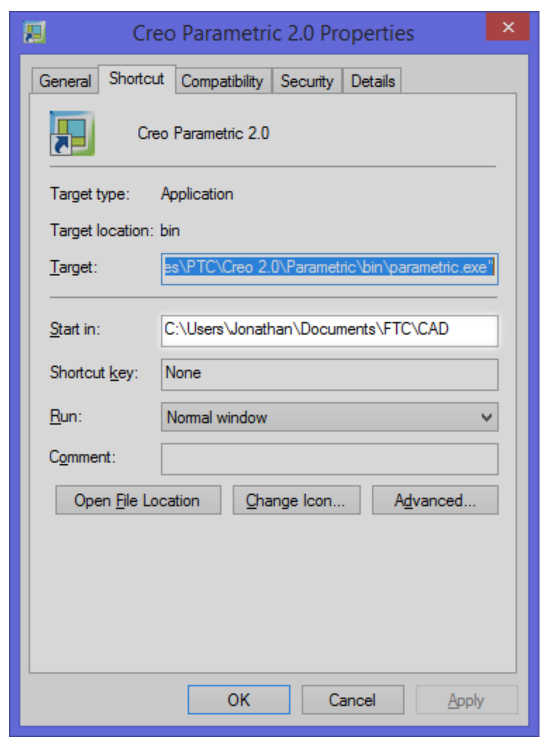
The Working Directory

The Working Directory is simply the folder on your computer where all the model files you will use in Creo are stored. This folder is where all your assembly files are kept, as well as all of their components. It is very important that you keep all your model files in the same folder. When Creo stores an assembly, it does not store all the model information for each individual part. If it stored assemblies that way, file sizes would quickly become too large to handle, and there would be too much redundancy for the method to be efficient. Instead, an assembly file loads model information by referencing part files or other assembly files. For this system to work, all the referenced parts need to be in the same directory on the computer.

Setting the Working Directory

There are two ways that you can set the Working Directory from inside Creo. The first way is to press the "Select Working Directory" button when Creo first opens. The second way is to open the Options menu by going to File->Options. Under the "Environment" tab, find the "Working Directory" line, and click the Browse button. Both of these will open a folder choosing dialog, where you can choose the folder on your computer that you would like to set as the working directory. You can choose any folder on your computer that you would like.

Whenever you open a dialog to open or save a file, you will have an option to go directly to the Working Directory. This makes it simple to keep all your part and assembly files in the same place, and keep them available to use in new assemblies.



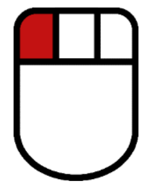
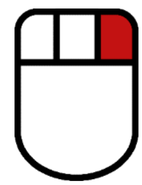
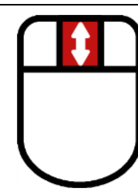
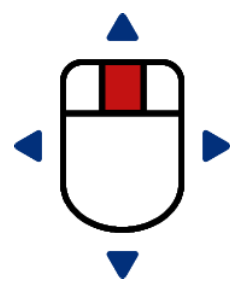
Manually Setting the Working Directory

There is a possibility that your Working Directory will not be saved between sessions. When you close and reopen Creo, your Working Directory will be reset to the default. To fix this, go to your desktop and right click on the Creo desktop shortcut. In the context menu, choose "Properties." In the Properties dialog, change the "Start In" line to the file path of your working directory. Then, click "Apply." Your working directory will now be set to the correct folder, as long as you use this shortcut to run Creo.

Navigating the Interface

To start, open a model by going to File->Open, and choosing a model from your working directory. The part will appear in the window, in the default view.

Mouse functions

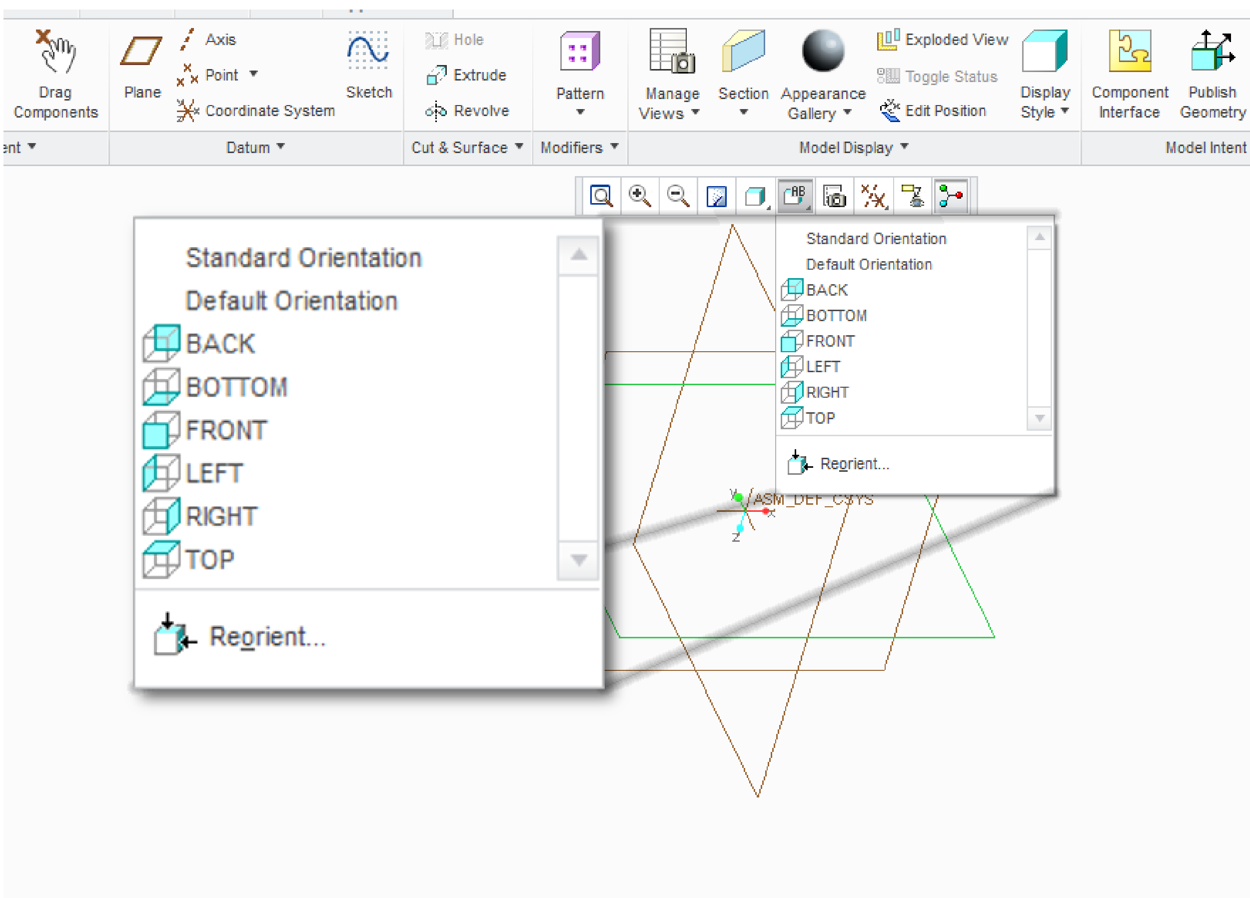
Function	Description	Action
Click	Left click on an object in the window to select it	
Right click and hold	With an object selected, right clicking and holding will bring up a context menu with a list of options	
Scroll	Scrolling will zoom in or out at the point where the cursor is pointing	
Middle click and drag	Middle clicking (pressing down on the scroll wheel) and moving the mouse will rotate the model in the window	
Shift + Middle click	Holding the shift key and the middle mouse button while moving the mouse will pan the view without rotating the model	

Named Views

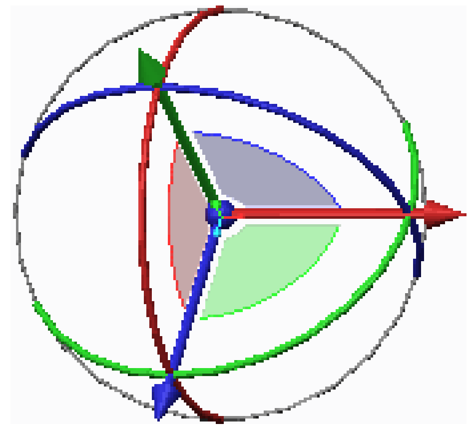
There are a number of pre-programmed view angles that Creo has installed so that you can orient the model in specific configurations. These preprogrammed views can also be used to re-center the model on the screen if you lose yourself by panning or rotating too far.

The pre-programmed views can be accessed by pressing the "Named Views" button on the small banner below the main toolbar. Pressing that button will bring up a drop-down menu with a number of view options.

- Standard orientation – The standard orientation; looking at the model from the top front left corner
- Default orientation – This button is for the default orientation. This is set to the Standard orientation, but can be changed
- Back – Orient to the back of the model
- Bottom – Orient to the bottom of the model
- Front – Orient to the front of the model
- Left – Orient to the left side of the model
- Right – Orient to the right side of the model
- Top – Orient to the top of the model



Moving an Assembly Component



There are a number of ways to move around an assembly component while still in the Component Placement phase. The most useful is the 3-D Dragger. When the part first appears in the window, it is in the default orientation. The part can be translated or rotated using the 3-D Dragger fixed at one corner of the part. The dragger is comprised of four kinds of movement: Axis Rotation, Axis Translation, Planar Translation, and Free Translation.

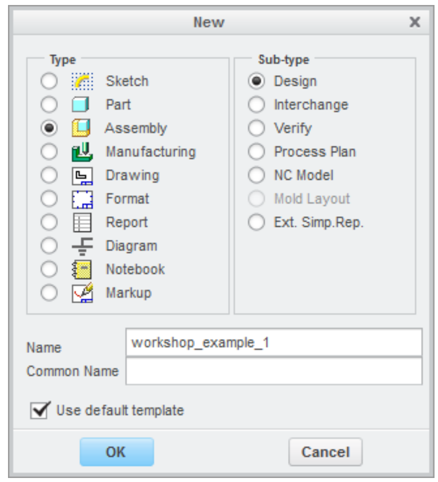
Clicking and dragging on one of the various colored lines and arrows in the 3-D Dragger will move the part around in the model space. The different controls each move the model a certain way.

Control	Type of Movement	Description
Arrows	Axis Translation	Clicking and dragging one of the arrows will move the part in a straight line along a specific axis.
Orbits	Axis Rotation	Clicking and dragging one of the orbits will rotate the part about a specific axis. The part will always rotate about the axis arrow in the 3-D Dragger.
Planes	Planar Translation	Clicking and dragging one of the three planes on the 3-D Dragger will translate the part along that specific plane.
Center Ball	Free Translation	Clicking and dragging on the ball in the center of the 3-D Dragger will free-translate the part, moving it to wherever the mouse points.

Chapter 2: Basic Assemblies

Create a New Assembly

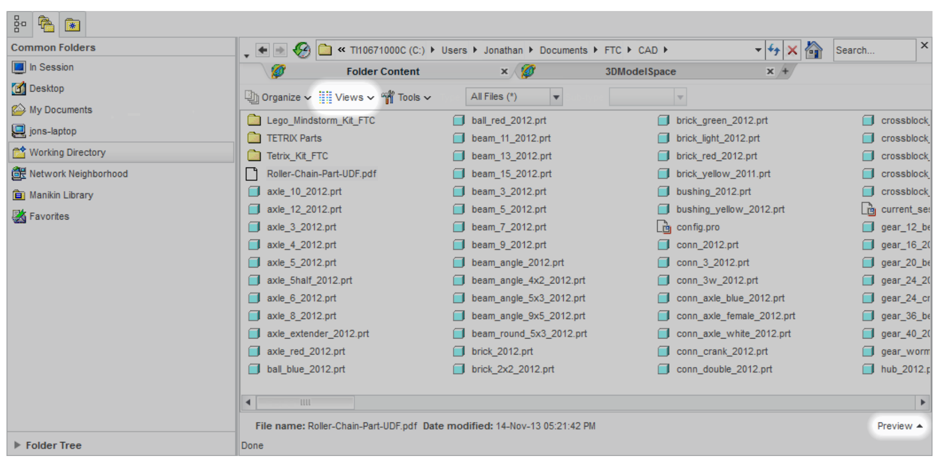
To get started, let's create a new assembly and add our first part. To create a new assembly, go to the File menu and choose "New." In the dialog that appears, choose the "Assembly" option. Then, choose a name for your assembly. It's good to choose a name that is meaningful, and will be easy to recognize. For now, make sure you have the Subset Type set to Design, and that you have the Use Default Template option checked. When you're done, click the OK button. Your new assembly will appear in the window.



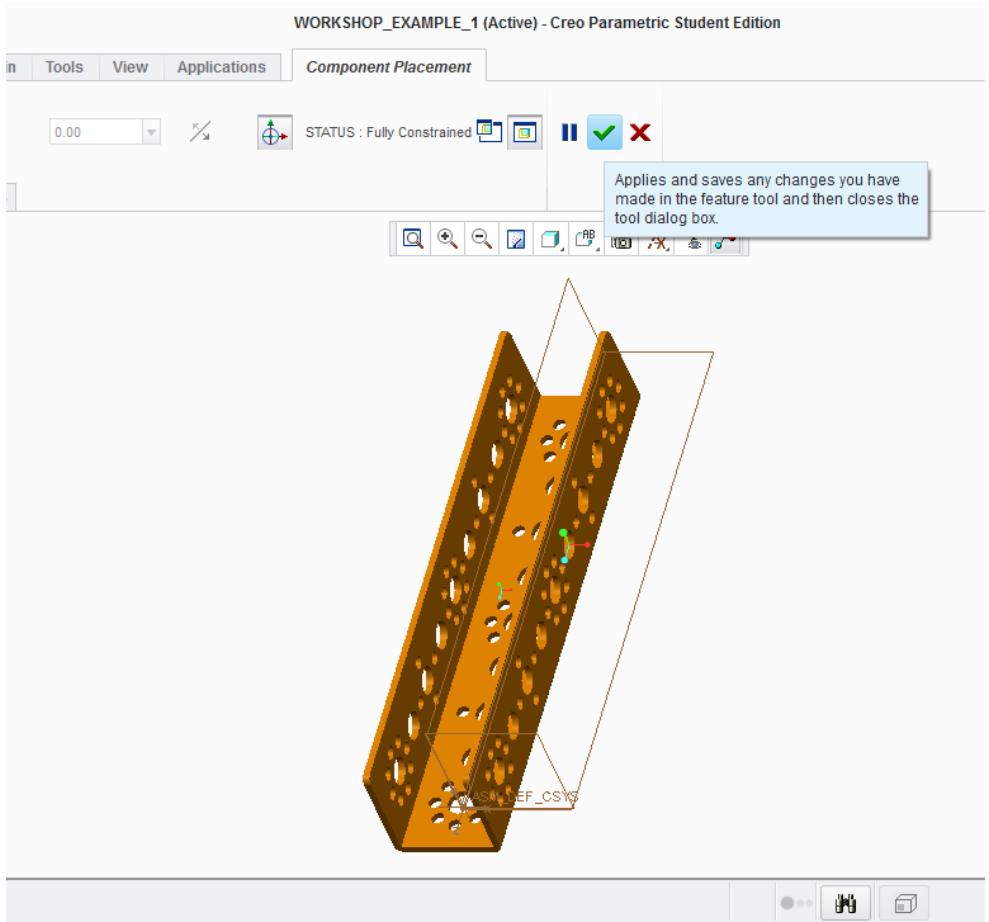
Adding a part

Now, it's time to add your first part. On the left side of the screen, there will be a box with three tabs: Model Tree, Common Folders, and Favorites. Under the Common Folders tab, click the button that says "Working Directory." This will open your working directory inside Creo. You will be able to explore your Working Directory from here, and can drag files directly from this window into your model.

When using this file explorer window, there are some things you can do to make it easier to find the files you are looking for. First, at the top of the window are three drop-down menus: Organize, Views, and Tools. Under the Views menu, change the view mode to thumbnails. Next, on the bottom right of the window, click the button that says "Preview" to open a small window that displays the model currently selected. You can view this small preview of the model in the same way that you would view a model in the main window.



Select a part that you would like to include in your assembly, either by clicking the "Assemble" button on the Model tab and selecting the component from the file selection dialog, or by dragging it in from the Shared Folders tab in the sidebar. In this example, we've used a simple piece of Tetrix channel. Locate the file in the explorer window, and drag & drop it into the model window. The part will appear in the center of the window, highlighted purple. This purple color signals that the part is not fully constrained. Since this is the first part of the assembly, it must be constrained to the origin so that the model is fixed in space. To do this, find the drop down menu in the Component Placement tab containing the constraint types and select "Default." The part will turn gold, indicating that it is fully constrained. Then, to accept the new part, click the green check mark that appears in the Component Placement tab. The part is now locked in place, and will form the base of your whole model.



Constraints

When you add a part to a model, to keep it from being free-floating in space, you must constrain it, or "align it," to another part. The model remembers where in the model part is supposed to be relative to the part you constrain it to. This is how each part of the model is fixed in space, instead of being free-floating. The parts all remember where they are supposed to be relative to each other.

Types of Constraints

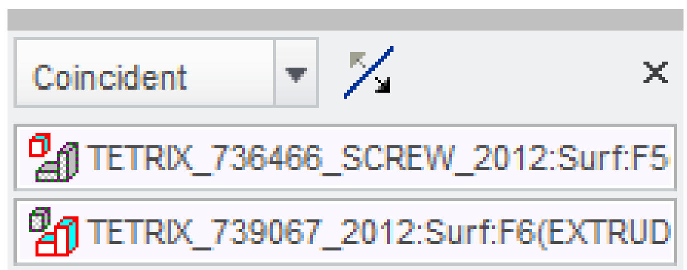
There are a variety of different constraint methods that you can use to lock parts together.

- Distance – The component part remains a specific distance from the assembly part
- Angle Offset – The component part is kept at a certain angle relative to the assembly part
- Parallel – The component part and the assembly part are kept parallel

- Coincident – The component part and the assembly part are directly touching
- Normal – The component part is kept at a 90 angle to the assembly part
- Coplanar – The component part is kept on the same plane as the assembly part
- Centered – The component part is centered on an assembly part
- Tangent – The component part is only touching the curve of the assembly part at one point
- Fix – The component part is locked in a specific point in space
- Default – The component part is fixed to the origin of the assembly [point (0,0,0)].

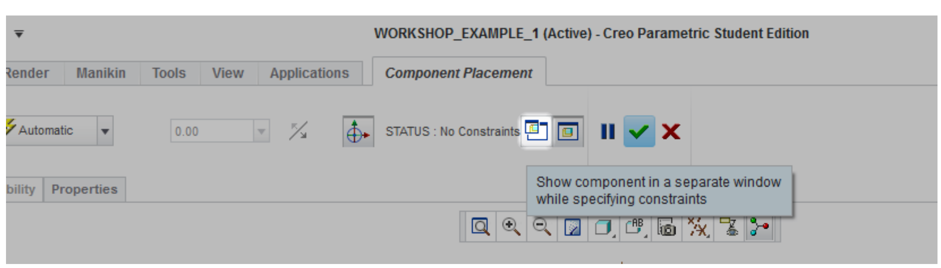
Constraining an Assembly Component

To create a new constraint on a part, click one face of the part, and then click the corresponding face on the assembly. Creo will constrain the part to the assembly component, using the faces you selected as a guide. For example, if you want two flat pieces to be touching each other, select one side of one piece, and one side of another piece, and Creo will match the two faces together. If you would like two holes to be lined up, select the surface inside one hole, then the surface inside another, and Cero will align the two holes using the two faces you selected. When you create a constraint, a small tab will appear on the new part, displaying the type of constraint being used.



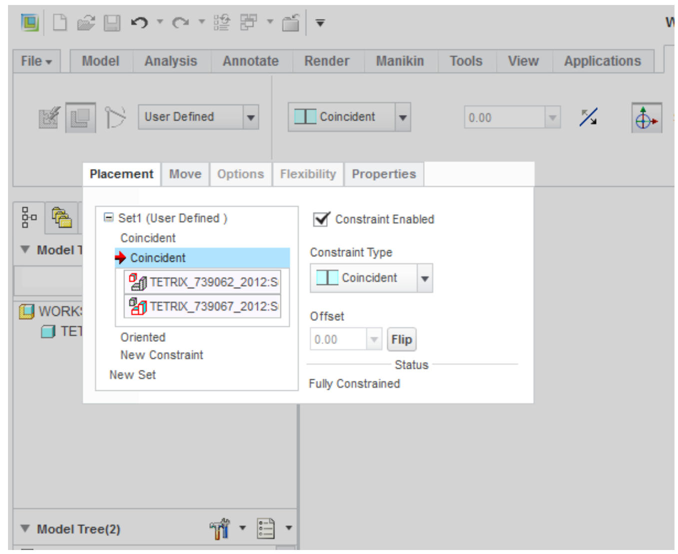
The way that two faces match up is dependent on the type of constraint that you use. When you first drag in a part, it defaults to the "Automatic" constraint mode. This means that Cero will attempt to choose the most appropriate constraint type based on the two surfaces you select. When you create a new constraint using Automatic mode, Cero may not choose the constraint type that you wanted. To change this, double-click on the constraint tab on the part. This will open a small dialog showing the constraint details. This dialog contains a small drop-down menu that will allow you to manually select the constraint type, as well as a button to flip the orientation of the constraint. There are also two lines of text containing the names of the faces that you selected. To change them, click on the name of the face you want to change, and then click on the face on the component that you would like to replace it with.

When you add a part to an assembly, too keep it from floating freely in space, you must constrain it to another part. Constraints are handled by the Component Placement tab in Creo. This tab is only visible when a part is being edited. It has a variety of tools and functions to make placing and aligning the component easier.



When you drag a component into the window, it may be difficult to move it into the right place. It can also be difficult to select faces on the part for constraining, especially if your part is very small and your model is very large. To make this easier to handle, while you are in the Component Placement tab you can click the small button highlighted in the picture, near the green check mark button. This will open a small preview window showing your part. You can navigate this small window in the same way as you would the main window, and can select faces on the component from here, rather than finding it in the main model window.

You can also manually select and edit individual constraints by clicking the “Placement” button on the bottom of the Component Placement tab. This brings up a small menu listing all of the constraints applied to the current piece, as well as a number of options for editing the constraints.



On the left side of the menu is a collapsible list of all the applied constraints. From here, you can add constraints or groups of constraints, as well as edit the faces that the constraint is applied to. Right clicking on a constraint brings up the option to delete or disable it. Disabling a constraint removes its effects on the part, but does not completely delete it.

On the right side of the menu are a number of functions to edit the current constraint. There is a check box to disable or enable the constraint, a drop-down menu to change the constraint type, as well as another text box that contains information that is required for some constraint types (such as the angle used by the Angle Offset constraint type).

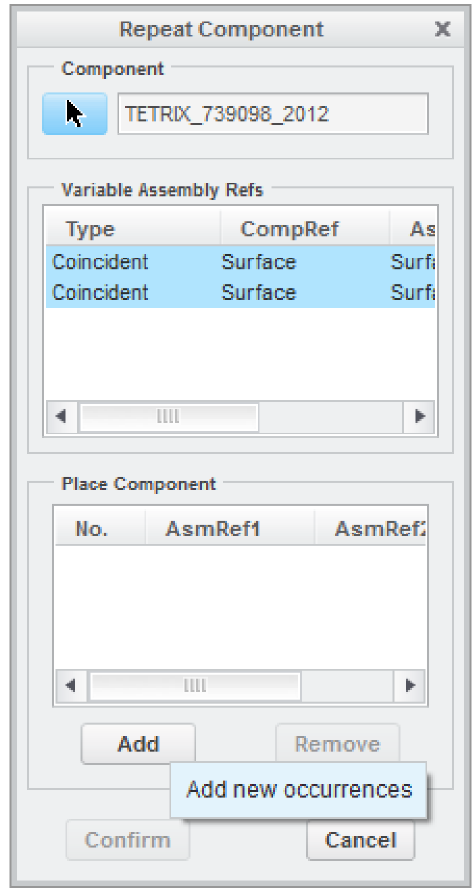
On the bottom of the menu is a small area titled “Status.” This area will display the status of the constraints on the part, which can be one of four: No Constraints, Partially Constrained, Fully Constrained, or Constraints Invalid. Your aim should be to make sure that every part is fully constrained before you accept the new component. A part is fully constrained when it is locked to the assembly in such a way that it cannot move at all. When a part is fully constrained, it will be highlighted gold.

Editing Existing Constraints

In the course of building your assembly, you may need to change some of the constraints of a part you have added already. To do this, first select the part you wish to edit either by left clicking it in the model window, or clicking its name in the model tree. If you are editing a part that is its own subassembly, it’s suggested to click the name in the model tree, so you don’t accidentally select one of the parts of the subassembly, rather than the whole subassembly itself. Then, right click and hold to bring up the context menu, and select “Edit Definitions.” This will take the part back into its editing mode and reopen the Component Placement tab. When you are editing an existing component, you may notice that some parts will disappear. When a component is being edited, only the parts that are above the active part are shown. All parts that are dependent on the component that you are editing will not be shown. You can then edit the constraints of the component the same way you would if you were adding it for the first time, and then click the green check mark to accept your changes.

Repeating a Component

When making a large, complex assembly, parts like fasteners and bushings can be included in large quantities. It would be very time consuming to open all of those parts individually, constrain them, and move on to the next component. Instead, Creo has a fast and simple way to copy existing components into the model. The one requirement is that the part you are repeating has to be fully constrained. To repeat a part, first select it by left clicking on it in the model window, or clicking its name in the model tree. It the component you are repeating is a subassembly, it is suggested to select its name in the model tree instead of selecting it in the model window, in case you accidentally select a component of the subassembly, rather than the subassembly itself. Once the part is selected, right click and hold to bring up the context menu, and select the “Repeat” option. This will bring up a small dialog that we will use to repeat the component.



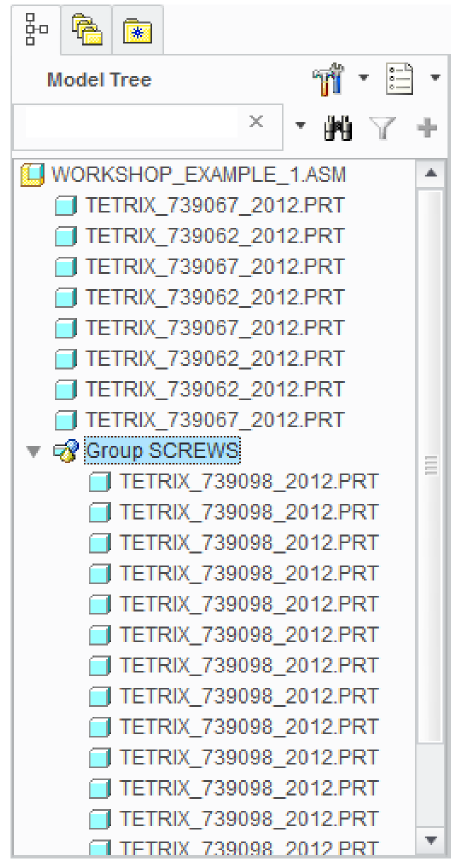
In the “Variable Assembly Refs” section of the dialog is a list of all the constraints applied to the part. Select all the constraints that want the repeated parts to copy. Then, in the “Place Component” section, click the “Add” button. On the part you originally selected, one of the faces will be highlighted purple. This is the constraint you are currently copying. Click a component face on the assembly to correspond to the highlighted face. Once you’ve done that, the next constrained face on the list will be highlighted. Continue selecting faces on the assembly to match the faces on the copied part. Once you’ve selected faces for all the applied constraints, the new copied part will appear. You can continue doing this as many times as you like. When you’ve repeated all the new components that you want, you can click “Confirm,” and all the new components will be added to the model.

The Model Tree

The first tab in the sidebar on the left side of the screen is the Model Tree. The model tree is a list of every part and subassembly included in the model. Whenever you add a part to the assembly, it appears in the model tree. You can select components by clicking their name in the model tree, reorder components, and group them from the model tree. You can select more than one component by holding the Ctrl key and clicking, or by holding the Shift key and clicking.

Groups

Groups are a way of organizing your model tree and making it more readable. A group is similar to a folder in your computer; it has a name, and it contains many other files. To create a group, select all the components in the model tree that you would like to group, right click and hold to bring up the context menu, and select the "Group" option. A new group will be created, and all the components you selected will be moved into the group. You can rename the group by clicking on it and typing in a new name. To open a group and see its contents, click the small triangle button to the left of the group name. Click the triangle button again to collapse the group.



Saving Your Model

Saving your model is a very simple process. Go to File->Save, and choose a name for your model. Make sure that all the components for the model are saved in the same folder as the assembly itself. Once you've chosen a location to save the assembly, click OK and you are done!

Congratulations, you've just completed your first Creo CAD Assembly!

Chapter 3: Basic Part Sketching

When constructing the model of your robot, it is often the case that the provided Tetrix and LEGO parts will not be enough to accomplish the design that you want. You may in fact need to create your own parts.

Create a New Part

To create a new part file, go to File->New, and select Part from the list. Choose a name for your new part, and choose an appropriate sub-type. When you're ready, click OK. The new part will open in an empty window.

Sketch the First Face

First, go to the small banner below the Model tab and select the Front named view. Then, use the zoom tool to zoom in close to the origin. Then, in the Model tab, click the Sketch button to enter Sketch Mode. When the dialog appears, use your cursor to select the Front plane. Then, click the Sketch button in the dialog. You've now entered sketch mode.

Sketching Tools

In the Sketch tab are various drawing tools that you will use to create a sketch of a face of your part. There are three categories of tools: Sketching, Editing, and Constraining tools.

Sketching

- Line – Draw a straight line connecting two points, lines, or faces
- Rectangle – Draw a rectangle by selecting one corner and then the opposite corner
- Circle – Draw a circle by selecting the center point and then dragging out the radius
- Arc – Draw a three-point arc with a center point and two connected points
- Ellipse – Similar to a circle, but with two center points
- Spline – Draw a Bezier curve that travels between many points
- Fillet – Turn a sharp corner into a round edge
- Chamfer – Turn a sharp corner into a flat edge
- Text – Add text to the sketch
- Offset – Draw an edge a specific distance from another edge
- Thicken – Create new edges a certain distance on either side of an existing line
- Project – Project curves or edges onto an existing plane
- Centerline – Create a line at the exact center of two points

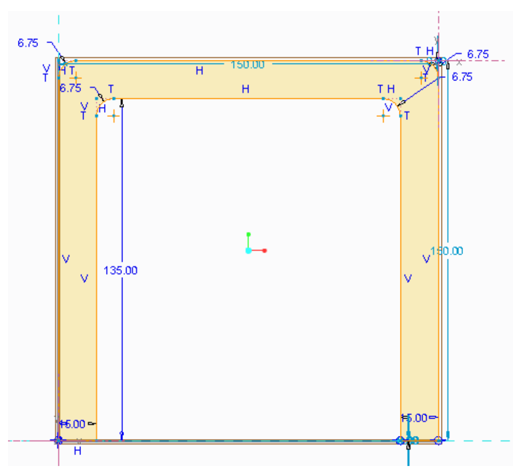
- Point – Add a single point
- Coordinate System – Create a Cartesian Coordinate system on the sketch

Editing

- Modify – Edit the values of an existing line, text object, or spline curve
- Delete Segment – Delete a line. Drag this tool across a segment to delete it.
- Mirror – Create a mirror copy of an entity
- Corner – Trim one segment flush to another
- Divide – Divide an entity into two entities at a certain point
- Rotate Resize – Rotate, move, or resize an entity

Constraining

- Vertical – Force a segment to be vertical
- Horizontal – Force a segment to be horizontal
- Perpendicular – Force one segment to be perpendicular to another
- Tangent – Force a segment to be tangent to a selected curve
- Midpoint – Align an entity so that it is at the midpoint of two other entities
- Coincident – Make two entities touch at a certain point or segment
- Symmetric – Make two points or segments symmetric about a certain point or segment
- Equal – Create equal lengths, angles, or radii
- Parallel - Force two segments or planes to be parallel



Using the various line and shape tools, draw a cross-section of what you want your part to look like. For example, if you were creating a piece of U-channel, you would draw a U-shape. As you are drawing, your cursor will snap to other planes, lines, or vertices that already exist. This makes it easier to create right angles and straight lines. At this stage, you don't have to worry about being precise, as all the dimensions on the part can be fixed later.

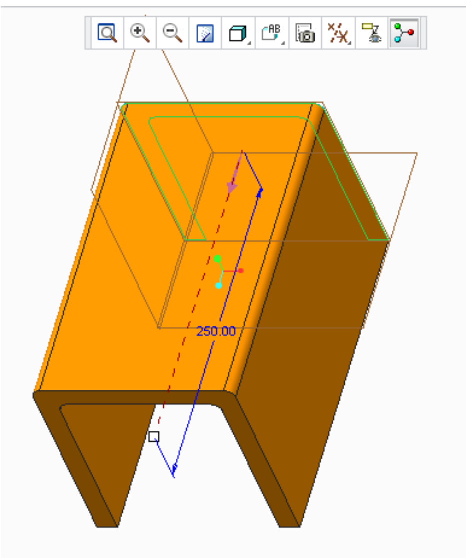
As you are creating your basic sketch, you may have lines that cross empty space or cross other lines. Your final sketch should only be an outline of the part, and should not have any extra lines, so you can use the Delete Segment tool to remove extra lines from your sketch.

You may now want to focus on creating the exact dimensions for your part. To change a dimension, double click on the number and edit it. You can change lengths, angles, and radii this way. Once your sketch's dimensions are the way you need them, click the green check mark to accept the sketch.

Extrusions

Now that you have a two-dimensional cross-section of your part, it is time to create a three-dimensional object out of it. The way to do this is with extrusions. Extrusions take a two dimensional shape, and add a third dimension to it to make a solid object.

First, make sure that your sketch is selected. Then, click the Extrude button in the Shapes section of the Model tab. This will put you into extrusion mode and open the Extrude tab. In this mode, your 2-D sketch will become a 3-D object. You can drag out the small white boxes to tell Creo how far to extrude out the piece. You can also select which extrusion mode Creo uses. There are three extrusion modes:



- Extrude from sketch plane – Extrude the part a specific distance from the front of the sketch plane
- Extrude on both sides of sketch plane – Extrude a certain distance from the front and back of the sketch plane at the same time
- Extrude to selected point – Extrude from the sketch plane out to a selected point, plane, or segment

You also have the option to make the extrusion remove material, rather than adding it. This is useful for creating slots, holes, and grooves. Once your extrusion is where you need it, click the green check mark to accept the extrusion.

Adding More Sketches

Once you've made an extrusion, all of the segments from your sketch have been made into planes. You can select them with your left mouse button. With a plane selected, click the Sketch button again to create a new sketch on that plane. You can then make another sketch, and extrude that sketch to add or remove material to your part. You can keep doing this to create complex three-dimensional shapes.

Editing Existing Sketches and Extrusions

Just like assemblies, each part has a model tree. This model tree contains each sketch and extrusion used in the part. To edit one of these, simply select its name in the model tree, right click and hold to bring up the context menu, and click Edit Definition. This will take you back into the sketching or extrusion mode, and allow you to edit the sketch or extrusion to fit your needs.

Saving Your Part

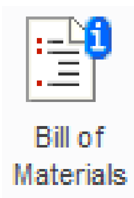
When you've finished your part and are ready to save, go to File->Save. In the dialog, choose a name for your part, and save it in your Working Directory. Unlike assemblies, part files do not reference any other files and can be stored apart from each other. However, any assembly or drawing that needs them will have to be store in the same file.

Congratulations, you've just finished your first Creo CAD part!

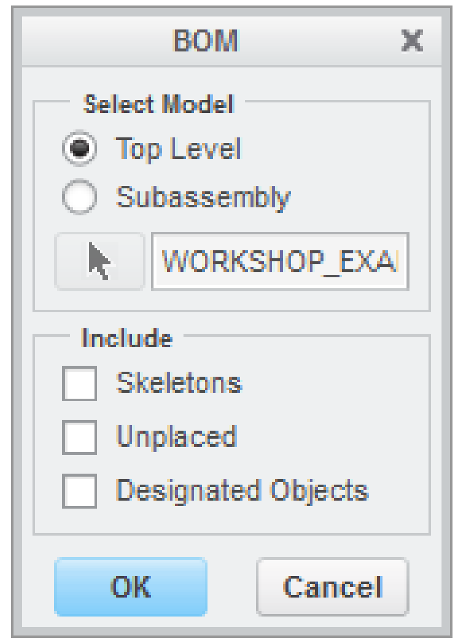
Chapter 4: Using Your Models Outside of Creo

Aside from helping you draft your robot, Creo also has a number of functions that can help you in the real world, outside of the program.

Bill of Materials

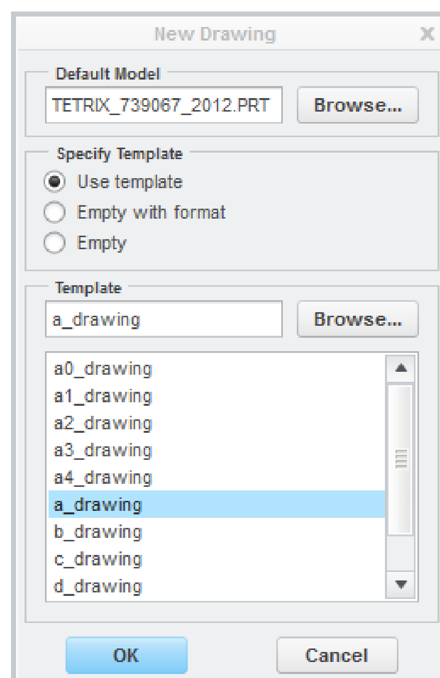


One of the requirements for entry into FRIST Tech Challenge competitions is a Bill of Materials. A Bill of Materials, or BOM, is a list of all the parts included on your robot. This includes things like the amount of sheet metal or plastic used, the types and quantities of parts, and all the fasteners used. It can be difficult to calculate all this information by hand, so Creo has a simple tool to create a list of all the parts of your robot that can be printed out and turned in to the judges.



On the far right edge of the Model tab is the Bill of Materials button. Pressing this button opens a small dialog with a few options. The "Select Model" section has two options: Top Level and Subassembly. This option determines which assembly the BOM is made for. Selecting Top Level creates a Bill of Materials for the whole assembly. Selecting Subassembly allows you to choose a particular subassembly in the model to create a Bill of Materials for.

After you click OK, an HTML document will open in Creo's built-in web browser. The Bill of Materials can then be saved to the computer, or printed off onto paper.



Drawings

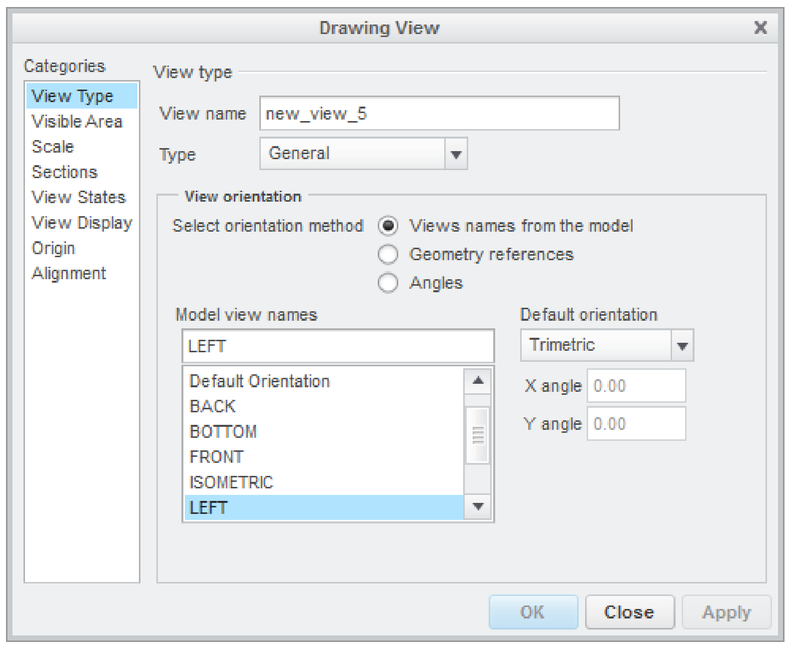
When in the fabrication process of your design, you may want to have on you a set of specific schematics to make the construction of complex pieces easier. Creo's Drawing creator has just this purpose. A drawing is a computer-generated sketch of your model, with measurements for the entire part. Drawings make it much easier for the manufacturers to see how exactly the part should be put together.

Creating a New Drawing

You create a new drawing the same way that you would any other file in Creo: go to File->New, and select the Drawing option. Enter a name for your drawing, and click OK. This will bring up a small dialog of options for your drawing. The first section of options is titled "Default Model." This section has a drop-down menu where you can choose which model you'd like to make a drawing of. This defaults to the active model inside Creo, but you can browse for any other model that you would like. The next section, "Specify Template," allows you to choose whether Creo will set up the views of the drawing for you, or whether you would like to do that yourself. For the sake of this example, we won't select a template, and will create all the views ourselves. The final section, "Template," allows you to choose what overall dimensions to make your drawing. The templates listed all correspond to different standard paper sizes, such as A4, A2, C, etc. Choosing a template will set the overall size and aspect ratio of your drawing. If you are planning to print your drawing onto standard 8½x11 inch printer paper, the A4 template would be the closest match. When you've chosen all the options that you want, click OK to generate your new Drawing. In the View tab, make sure that the Display Style in the Model Display tab is set to Hidden Line.

Adding Views to Your Drawing

If you selected the default template or no template, the drawing generated by Creo may not have all the view angles that you will need to accurately represent the part. To add a new view, click the "General" button on the layout tab to create a new general view. In the dialog that appears, make sure "No combined state" is selected, and click OK. Then, click the place on the drawing where you want the view to appear. A dialog will appear with a number of options for your model. The most important option that we will focus on is the view angle option. In the View Orientation section of the View Type tab, select one of the named views discussed in chapter 1. When you are ready, click Apply, and then click Close.



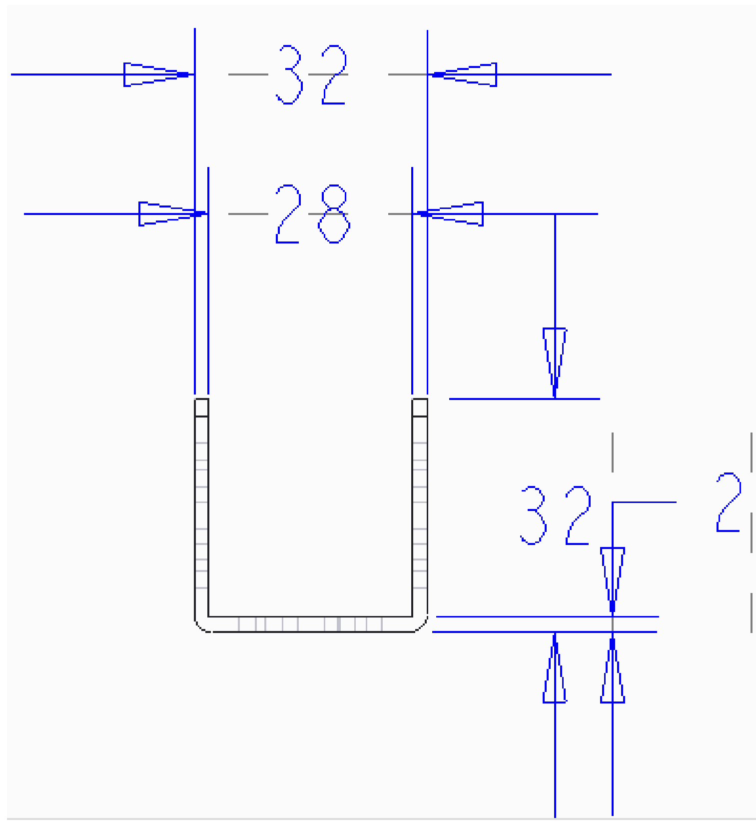
Once the first view is created, it is very simple to create more views of your part. Simply select the existing view, right click and hold to bring up the context menu, and select "Insert Projection View." Then, simply drag in any direction to insert a view of that corresponding side of the part. For example, if you had originally create a Front view of the part, and dragged the inserted view upwards, the inserted view would be a Top view of the part.

Adding Annotations to Your Drawing

Once all the necessary views of your part have been added to the drawing, it's time to add the annotations. Annotations are small notes about dimensions and distances between different parts of the component. Annotations allow the manufacturers to see the exact size and position of all the faces and extrusions on the component. The Annotations tab controls all the dimensions and notes applied to your drawing.

Dimensions

In the Annotations section of the Annotate tab, select "Dimension." Then, select two edges on the drawing that you would like to add a dimension for. Once you've selected the two edges you want, middle click somewhere on the drawing to add the dimension marker at that spot. You can continue adding dimensions until you click "Return" in the Menu Manager dialog.



While adding dimensions, they may become somewhat messy and difficult to read. You can rearrange dimensions any time by clicking on them and dragging them to a different area. Or, you could click and drag to create a large selection of dimension markers, and then click the "Cleanup Dimensions" button in the Annotate tab. This will open a dialog with a number of options that you can play with to achieve the kind of organization that you want. When you're finished, click "Apply," and then click "Close."

Notes

You may also want to add some text notes to your drawing to help the manufacturers or to provide some other information about the part. To add a text note, click the Note button in the Annotate tab. In the Menu Manager dialog that opens, click Make Note. Creo will then ask you to select a free point on the drawing where you would like to insert the note. The note can be moved later the same way that a dimension can. In the small dialog that appears, type out your note, and click the green check mark. The dialog will reappear clean, and you can continue adding more lines of text until you have everything you need. When you're done, click the red X to close the dialog, and click Done/Return on the Menu Manager Dialog.

Saving Your Drawing

Similar to assemblies, drawings reference other files to create their content, and thus must be stored in your working directory. To save the drawing, go to File->Save, choose a name for your drawing, and click OK.

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