

PTC Creo® 3.0 Tips and Tricks

PTC Creo Parametric 3.0, the latest version of PTC's 3D parametric CAD solution, uses proven technologies from Pro/ENGINEER® as well as hundreds of new capabilities to unlock design productivity.

New features allow users to import neutral file formats directly into PTC Creo Parametric, create and drive freeform designs parametrically, and improve modeling and reference editing. Extensive hardware libraries, integrated training tools, enhanced graphics, powerful analysis and diagnostic tools, and many other capabilities further enhance the design process.

The PTC Creo Parametric 3.0 user interface itself also incorporates a wide range of new productivity boosters. The following tips introduce you to some of the most useful shortcuts that speed and simplify frequently used commands. You'll find that taking advantage of these new capabilities saves you considerable time and effort as you work over the course of the day.

We've also included several additional tricks that apply to all versions of PTC Creo. Whether you're a rookie or a long-time user with vast experience using PTC Pro/ENGINEER and PTC Creo products, you're likely unaware of these tricks. By having an arsenal of several ways to perform common tasks, you'll be able to select the method that works best for you.

Tip 1—One-Click Switching Between Windows

Previous versions of PTC Creo made it cumbersome to switch between two different open model windows. The only way a user could jump from one window to another was to go to the Microsoft® Windows® icon (shown in Figure 1) access the drop-down list, and then select the desired window. PTC Creo Parametric 3.0 is more intuitive and requires fewer keystrokes

because it behaves just like any other Microsoft Windows program. To activate a window, you simply move your cursor to the window you want to work with and click somewhere inside it.

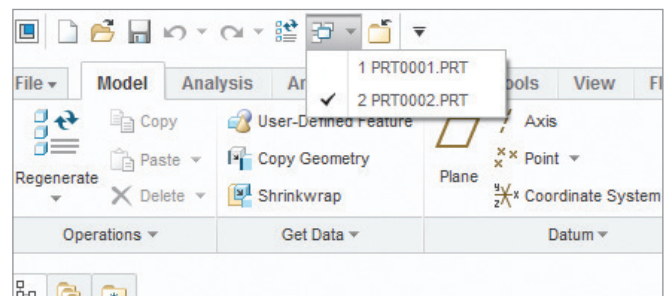


Figure 1: Previous versions of PTC Creo required you to go to a drop-down list on the ribbon menu to switch between open windows. Now you can simply click inside any window to activate it.

Tip 2—More Commands at Your Fingertips

Rather than requiring you to access every command from the ribbon menu at the top of your screen, earlier versions of PTC Creo allowed you to access shortcuts by pressing the right mouse button. This capability eliminated the need to drag the mouse across the screen to use the command. With PTC Creo Parametric 3.0, the right mouse button gives you access to even more shortcut commands, further increasing productivity.

In addition, you can now customize your shortcut menus to better suit the way you work by turning menu items on or off and by re-ordering items in the list. Figure 2 shows the screen used to modify the shortcut menu.

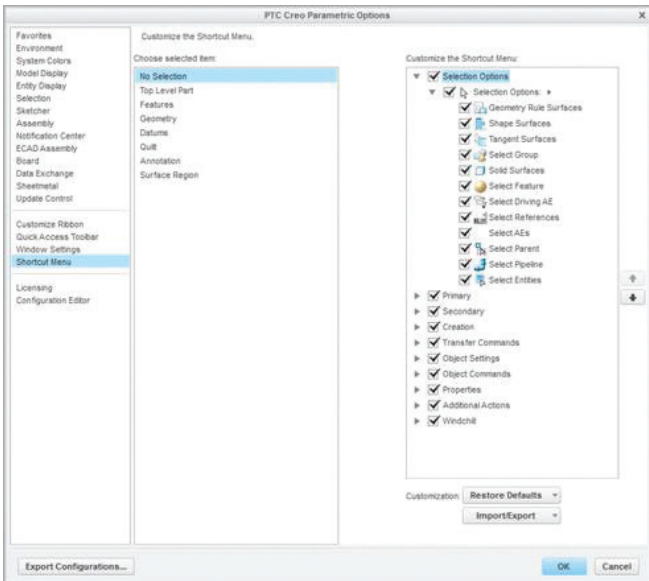


Figure 2: This screen enables you to customize a shortcut menu.

Tip 3—View Geometry From Another Angle

Imagine you have the model in Figure 3 and you want to orient it to see the highlighted green surface normal to the screen. PTC Creo Parametric 3.0 makes this task much faster and easier, reducing what was once a multi-step process to a single mouse click.

Using older versions of PTC Creo Parametric, you'd first create a new datum plane (which requires at least three mouse clicks) to orient the surface of the triangle.

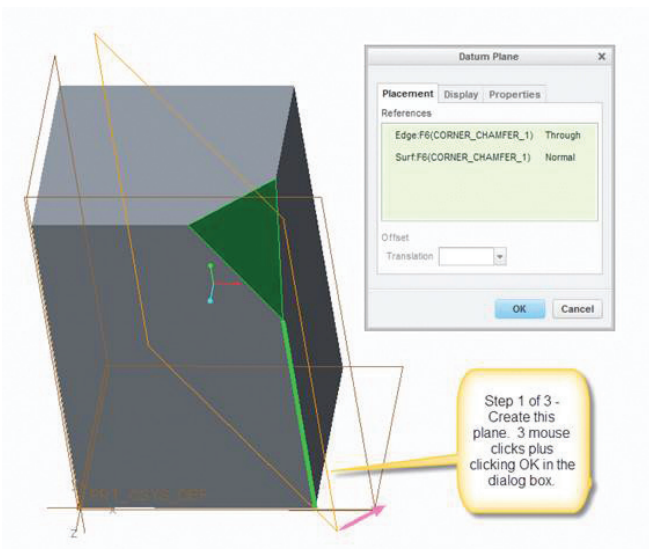


Figure 3

Next, you'd select the triangle that you want to re-orient, as shown in Figure 4.

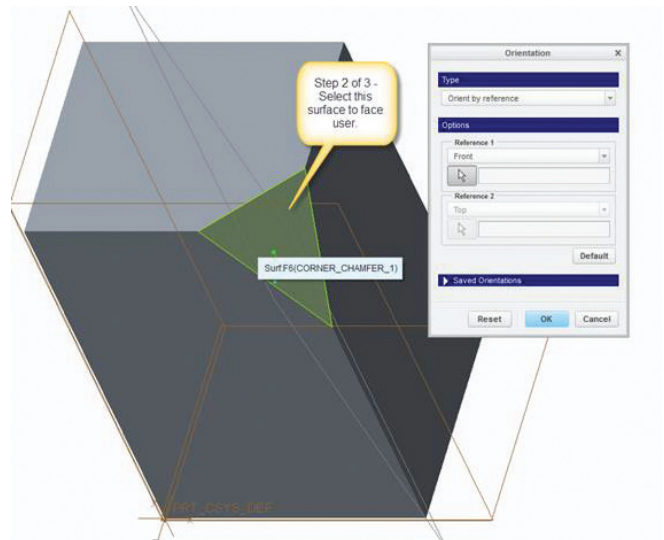


Figure 4

Finally, you'd select the plane that you want to face the top of the screen, as shown in Figure 5.

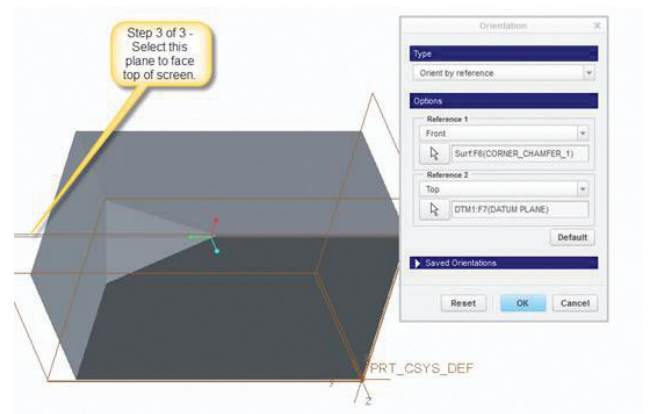


Figure 5

With PTC Creo Parametric 3.0, you can orient the triangle normal to the screen by simply selecting the surface, going to the Graphics toolbar, and choosing the 'View Normal' option, as shown in Figure 6. The View Normal command reduces what was once a multi-step process to a single step.

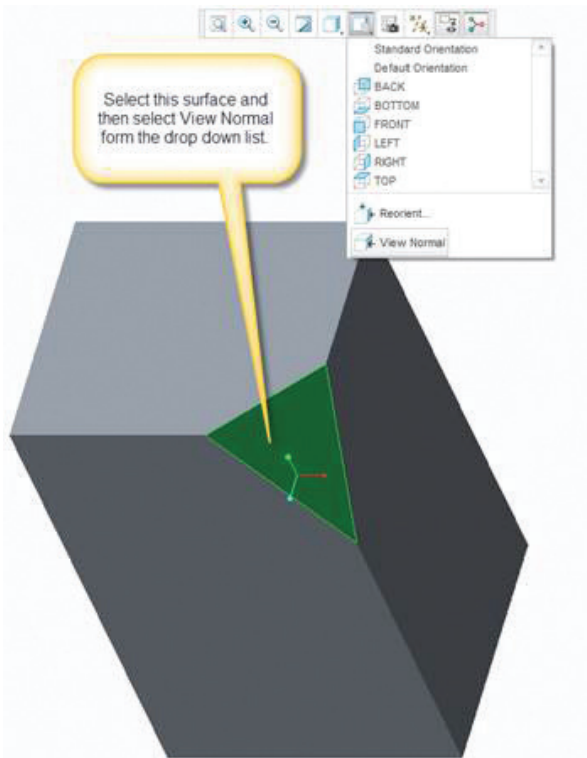


Figure 6: With PTC Creo Parametric 3.0, you orient the green triangle to face the user by simply selecting this surface and using the View Normal command.

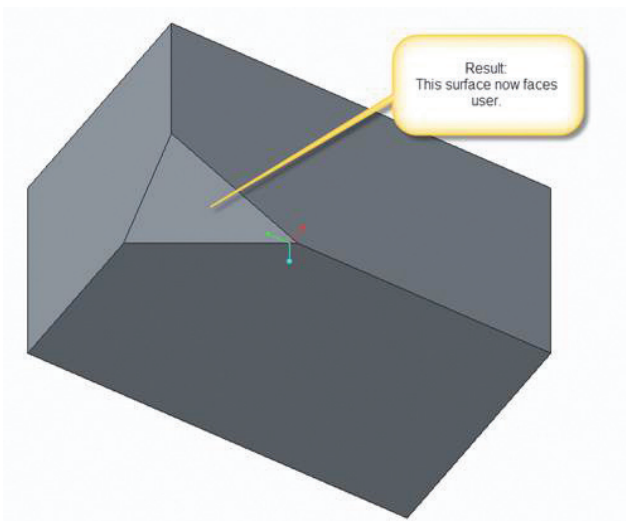


Figure 7: The surface now faces the user.

Tip 4—Simplifying the Smart Filter

PTC Creo Parametric provide various filters, including the Individual Selection filter, Compound Selection filter and Smart filter, to help you select items. PTC Creo Parametric automatically selects the Smart filter, if it's available. The Smart filter enables you to select the most common types of items that are valid for the current geometrical context.

This filter works on two levels. As you begin the selection process, the Smart filter allows you to select general, higher-level items (e.g., features or components), and after you select the higher-level items the Smart filter automatically narrows the selection scope to allow you to select more specific items, such as edges or faces. This process allows you to spend less time changing filters and more time modeling.

The problem with using the Smart filter is that it can take several mouse clicks to select geometry when the filter is engaged. For example, to select an edge to round on a model with the Smart filter on, you'd first select the feature, choose the surface that contains the edge, and then select Edge—for a total of three clicks.

You can reduce the number of mouse clicks from three to one if you temporarily disable the Smart filter. To do so, simply hold down the Alt key. Now you can select the Edge by mouse clicking directly on it. This tip can be used in any version of PTC Creo Parametric.

Tip 5—A Faster Way to Select References

As you create geometry using the Sketcher tool, you may want your cursor to automatically snap to existing geometry (e.g., a reference) to more easily align components within your drawing. However, if you're in the middle of using the Sketcher command, it's

too late to select a reference. You have to create your references before you start sketching. Alternatively, you can cancel the Sketcher tool, go to the upper left hand corner of your screen, use the Reference tool to specify the references, and then go back to using the Sketcher tool.

The Alt key suspends the Sketcher tool, allowing you select your references as you work. As you sketch, simply hold down the Alt key and select your references. Releasing the Alt key instantly returns you to the Sketcher tool. You can now snap your cursor to the references you just selected. This tip applies to any version of PTC Creo Parametric.

Tip 6—An Easier Way to Extrude Geometry

To transform a 2D sketch to a 3D extrusion using any version of PTC Creo Parametric, you sketch the shape and then key in the desired depth. Sometimes, however, you'll want to extrude the sketch to a reference, such as another cube or plane. In PTC Creo Parametric, the Shift key is a shortcut that makes it easy to extrude sketches to a reference.

Say you wanted to extrude the rectangle shown in Figure 8 to the plane. Simply select the drag handle, press the Shift key, and then drag the rectangle to the plane. The rectangle will automatically snap to the plane.

Most people are unfamiliar with this Shift key shortcut. Instead, they will right click on the drag handle (which produces a popup menu), select the "To Surface" option, and then choose the surface they want to extrude to—a three-click process. The Shift key shortcut saves mouse travel and clicks.

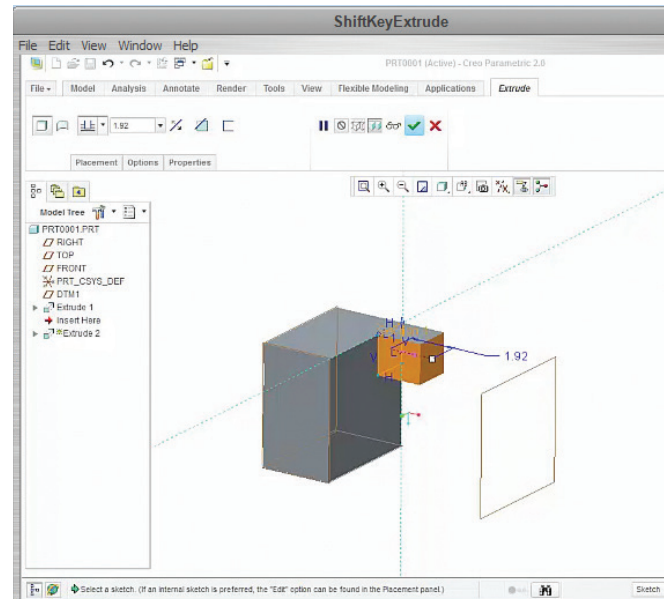


Figure 8: Pressing the Shift key allows users to extrude the orange rectangle to the plane with fewer mouse clicks.

Conclusion

Some of these tips are brand new capabilities found only in PTC Creo Parametric 3.0 that are designed specifically to improve productivity throughout the solution. Others are commonly unknown tips and tricks for all versions of PTC Creo Parametric that are easily overlooked in the [help files](#), so are widely unknown. These are all representative of the types of productivity improvements that interacting with instructors during PTC University training sessions will bring to light.

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