How to create threads in Creo Parametric

This guide shows you how to create threads in Creo Parametric using the Helical Sweep tool. You build a base cylinder, sketch a thread path and axis, define a triangular thread profile, and generate the thread. You then review the result and save the part.

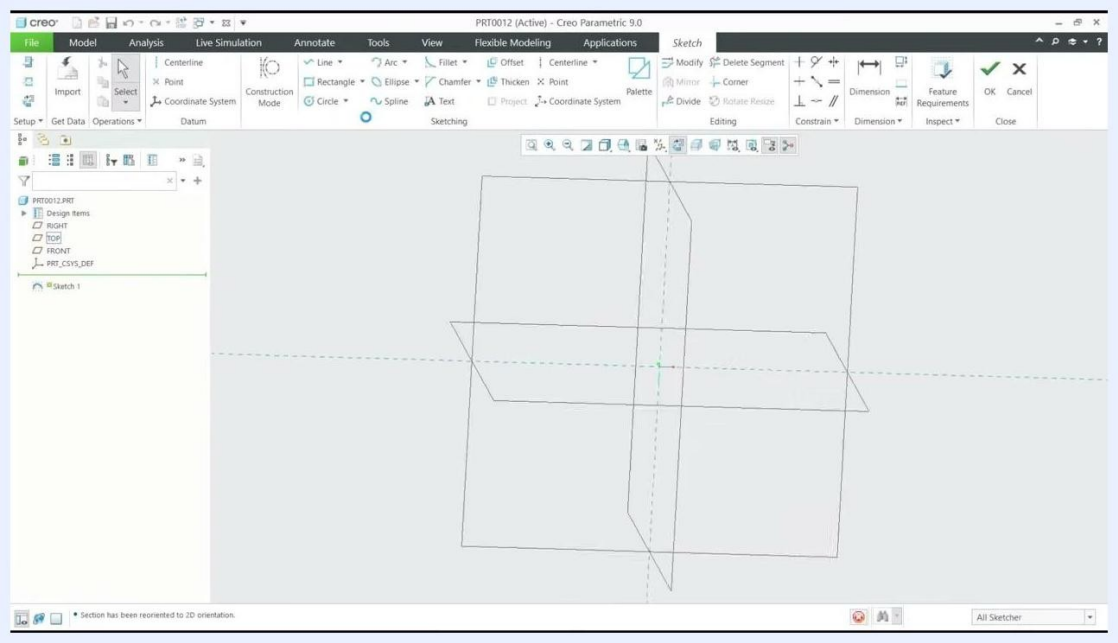
Prerequisites

- PTC Creo installed and ready to use
- Familiarity with the **Model** tab and basic sketching tools

Start a new part and create a base cylinder

Step 1: Start a new part and prepare the workspace

Open PTC Creo and start a new part file. Take a moment to get familiar with the workspace, and make sure you are in the **Model** tab.



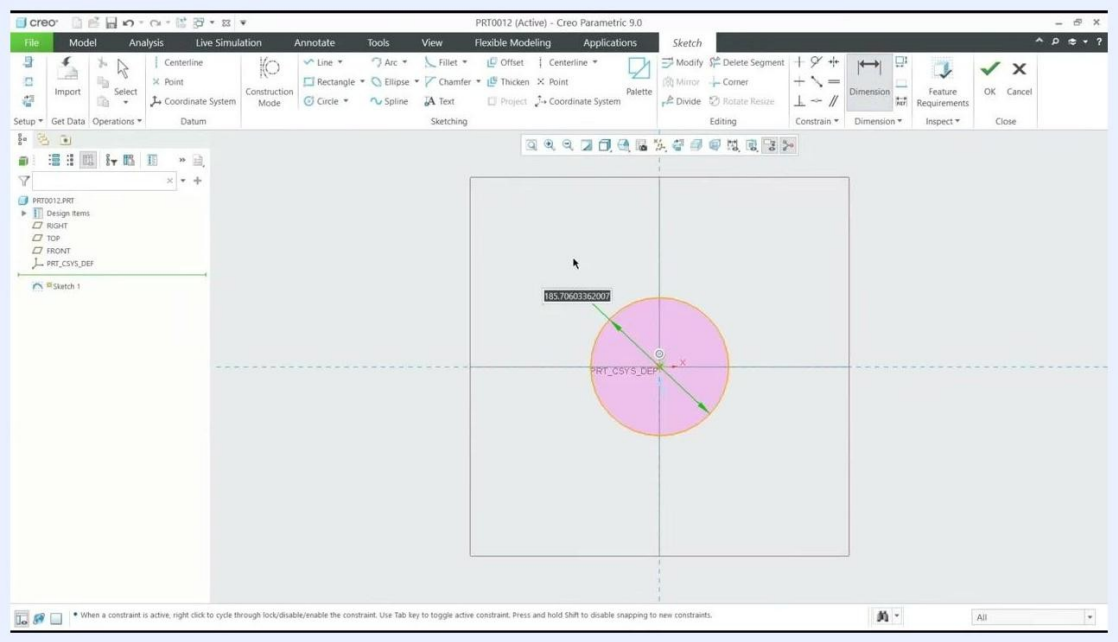
The PTC Creo start screen with the "How to make threads" heading and an illustration of a threaded cylinder.

Step 2: Create a base sketch

Select the **Top** plane from the model tree or the workspace. Then click **Sketch** to begin a new 2D sketch.

Step 3: Draw a circle with a specific radius

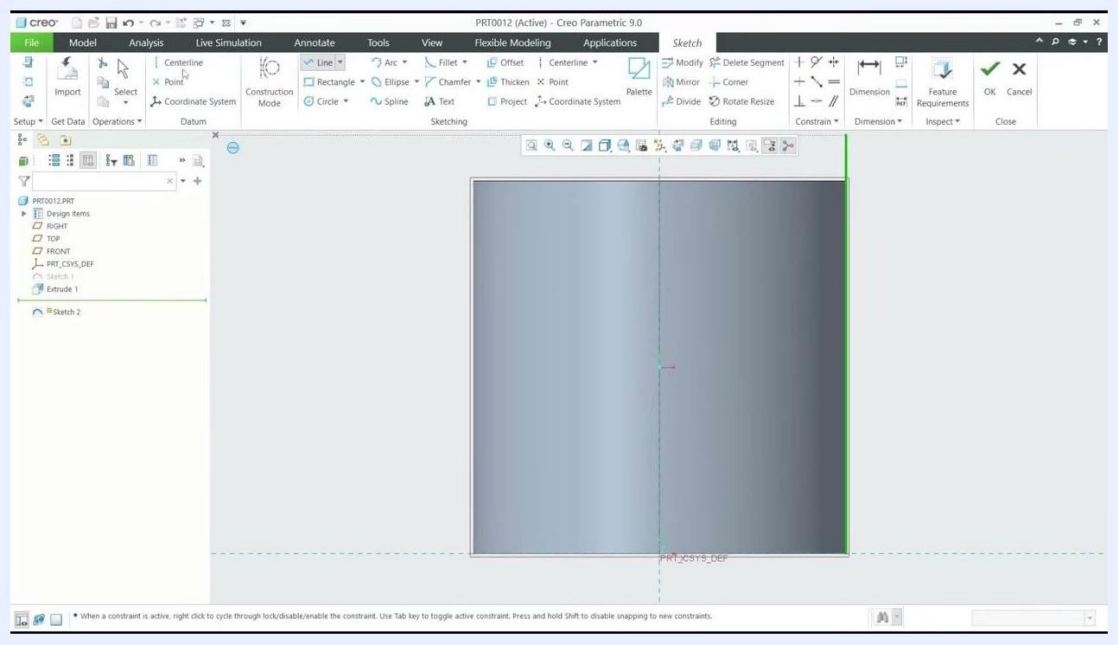
Use the **Circle** tool to draw a circle centered at the origin. Set the radius to **15** units.



The Creo sketch environment with a circle of radius 15 centered at the origin, with construction planes and sketch tools visible.

Step 4: Finish the sketch and extrude to 3D

Click **OK** to finish the sketch. Use the **Extrude** tool to turn the 2D circle into a 3D cylinder. You can accept the default extrusion or set the height you want.



The 3D cylinder created from the extruded circle, with axes and planes visible in the Creo workspace.

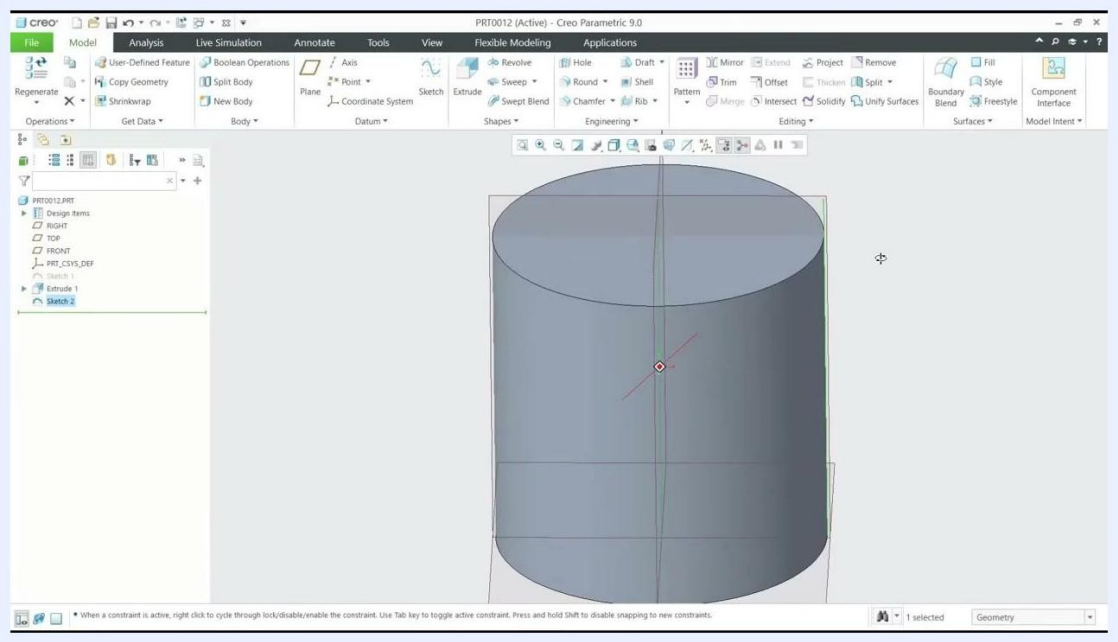
Create the thread path

Step 5: Prepare a sketch for the thread path

Select the **Front** plane. Click **Sketch** to start a new sketch on the front face of the cylinder.

Step 6: Draw the thread path

Draw a vertical line from the bottom to the top of the cylinder. This line defines the thread's path.



The Front plane sketch with a vertical line along the side of the cylinder, ready to define the thread path.

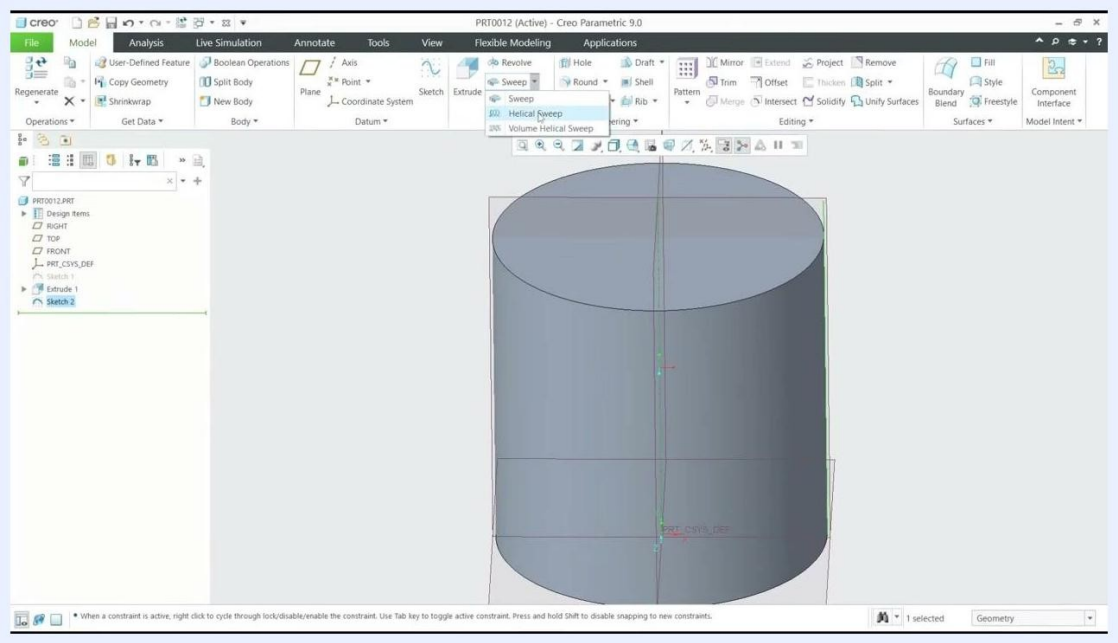
Step 7: Add a centerline for the helix axis

Draw a centerline vertically from the origin point. Then click **OK** to finish the sketch.

Create the helical sweep and thread profile

Step 8: Start the Helical Sweep

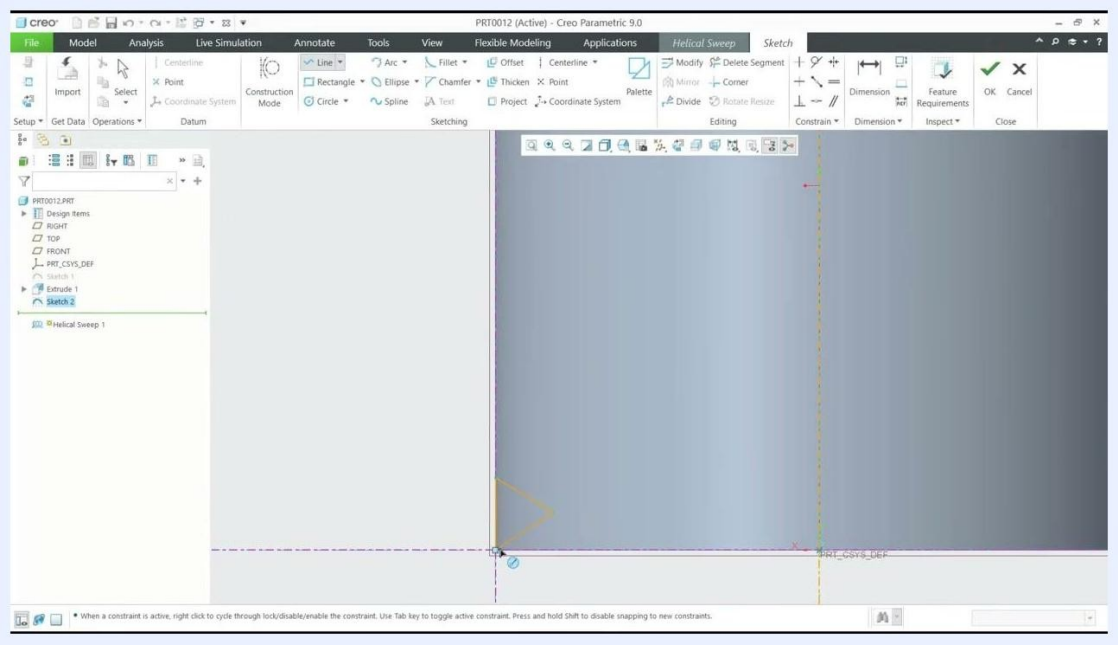
Select the **Helical Sweep** tool from the **Sweep** dropdown menu. The sketch for the thread path is selected automatically. For the axis, select the centerline you just created.



The Helical Sweep tool selected from the Sweep dropdown, with the cylinder and centerline visible.

Step 9: Sketch the thread section profile

Click **Section**, and then click **Sketch** to define the thread's cross-sectional profile. Use the **Line** tool to draw a triangle at the base of the cylinder, starting from the intersection point.



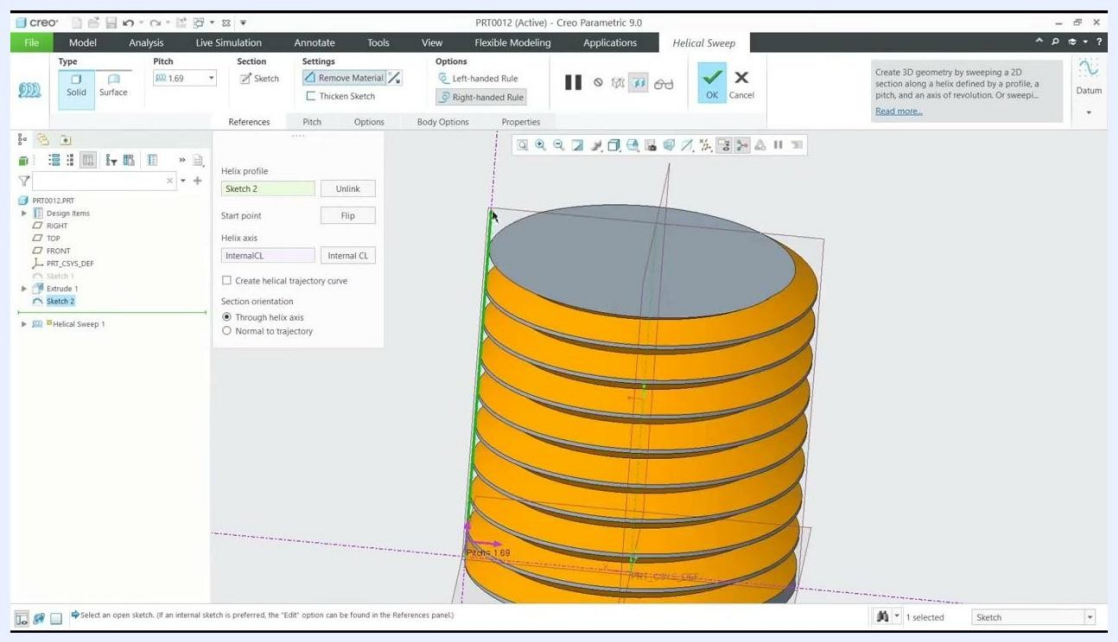
The sketch environment with a triangle being drawn at the base of the cylinder as the thread profile.

Step 10: Apply constraints to the thread profile

Use the constraint tools to make the sides of the triangle equal. This gives the thread a symmetric profile.

Step 11: Set the thread pitch dimension

In the thread profile sketch, set the pitch dimension to **1.5** units by entering "1.5" in the dimension field. Click **OK** to confirm and exit the sketch.



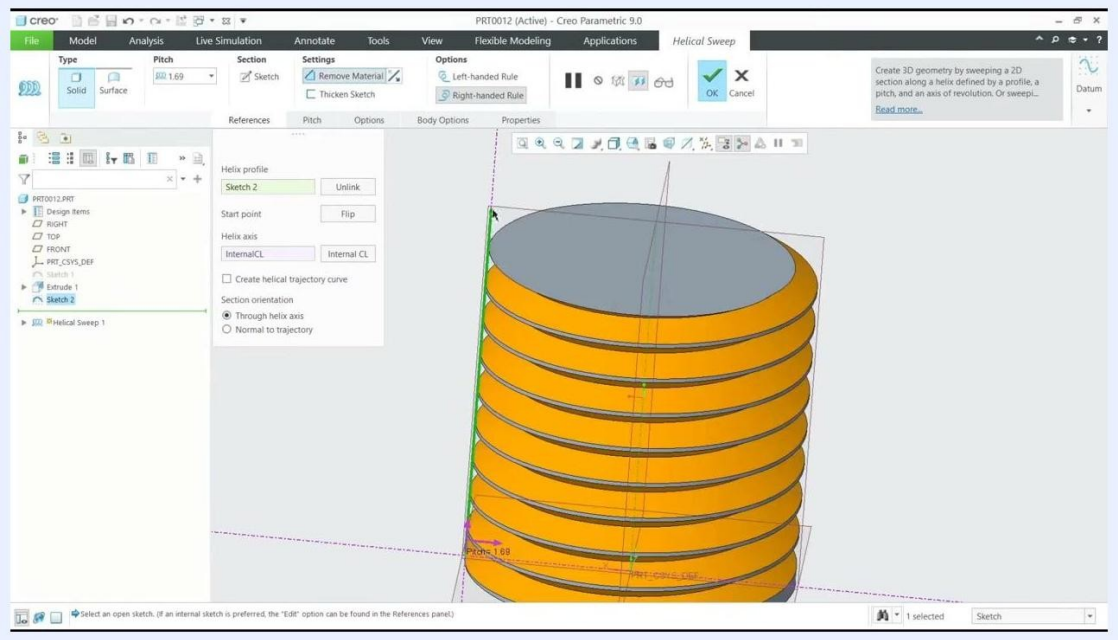
The triangular thread profile in the sketch environment with the pitch dimension being set to 1.5 and the OK button visible.

Step 12: Configure the Helical Sweep options

Review the Helical Sweep dialog and check these settings:

- Set **Helix profile** to "Sketch 2".
- Set **Helix axis** to "Internal CL".
- Set **Pitch** to **1.69**, as shown in the dialog.
- Select **Remove Material** if you want to create an internal thread.
- Set **Section orientation** to "Through helix axis".

Click **OK** to apply the helical sweep and generate the thread.

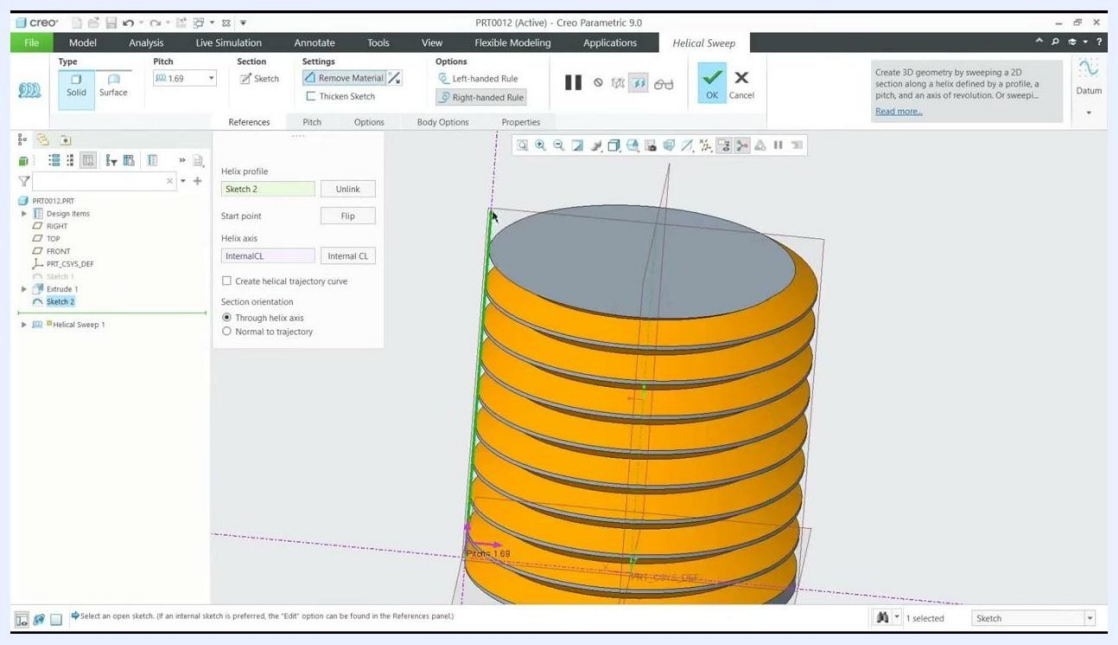


The Helical Sweep dialog with the options set, the cylinder highlighted in orange, a thread preview, Pitch set to 1.69, Remove Material checked, and the OK button highlighted.

Review and save your work

Step 13: Review the generated thread

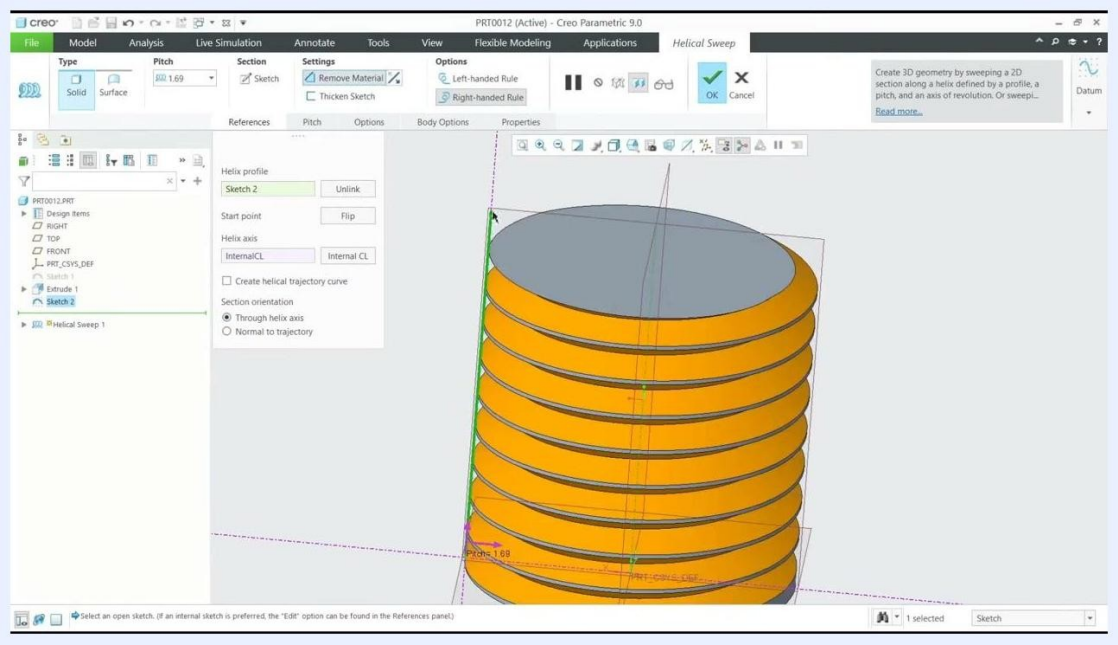
Inspect the 3D model to confirm the thread runs along the cylinder. Depending on your settings, the thread appears as a helical cut or protrusion around the cylinder.



The cylinder with the completed helical thread feature highlighted in orange, showing multiple thread turns, with the Helical Sweep dialog still visible.

Step 14: Finalize and save your work

Exit the Helical Sweep tool and return to the main modeling workspace. Review the final threaded part, and then save your part file to preserve your work.



The final Creo workspace with the completed threaded cylinder in gray and all features listed in the model tree.

Tips

- Adjust the pitch, profile, and sweep settings to match different thread standards or custom requirements.
- Choose **Remove Material** for an internal thread, or leave it cleared if you want the thread to protrude from the cylinder.

Summary

You now know how to create threads in Creo Parametric. You built a cylinder, sketched a thread path and helix axis, drew and constrained a triangular profile, and generated the thread with the Helical Sweep tool.

What's next

Continue refining the thread parameters and dimensions to suit your design requirements. Try changing the pitch, the profile shape, or the sweep options to see how they affect the thread.