How to create a thread in Mastercam with the Custom Thread operation

This guide shows you how to create a thread in Mastercam Lathe using the Custom Thread operation. You draw a thread profile, define the raw material, select a tool and chain the profile. You then set the pitch and cutting parameters, and simulate the toolpath to verify the result.

When to use the Custom Thread operation

Custom Thread is a newer feature in the Mastercam Lathe module. You find it among the turning operations. Use it when a component needs a non-standard thread that the regular threading operation cannot produce.

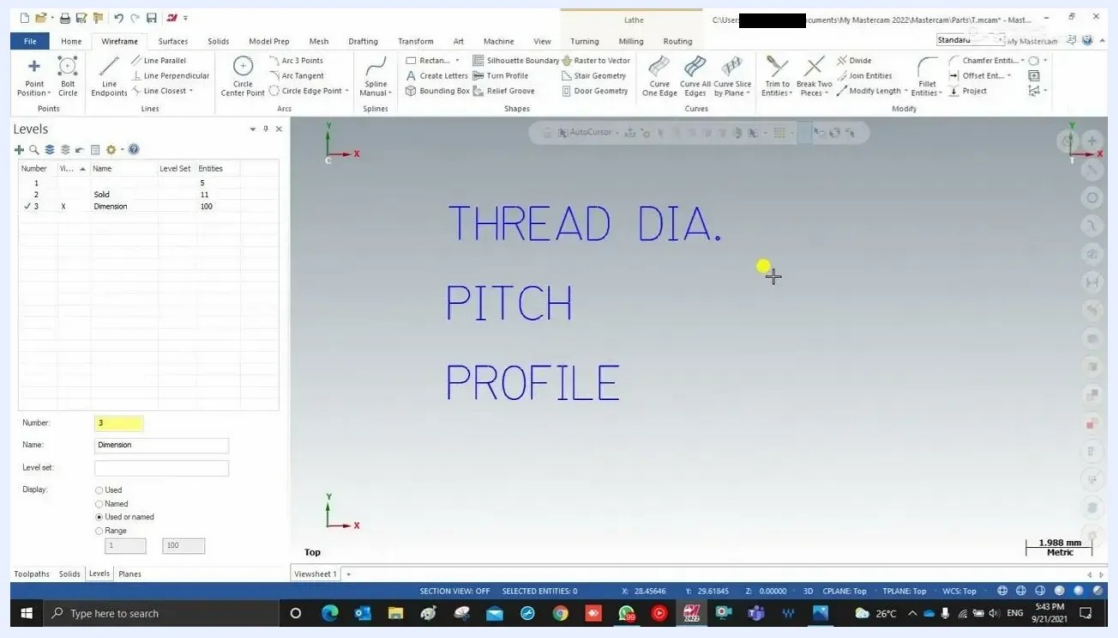
For example, a component may look as if it has a groove but actually has a large, non-standard thread. In that case, the Custom Thread operation is the right choice.

Prerequisites

Before you start, make sure you have these three values for the thread:

- **Diameter:** the thread diameter, either an internal diameter (ID) or an outer diameter (OD).
- **Pitch:** the pitch of the thread.

- **Profile:** the profile of the thread.



Mastercam workspace showing the text "THREAD DIA. PITCH PROFILE" in blue, with the Levels panel on the left and the ribbon at the top.

You also need a detailed sketch or drawing of the thread profile for your component.

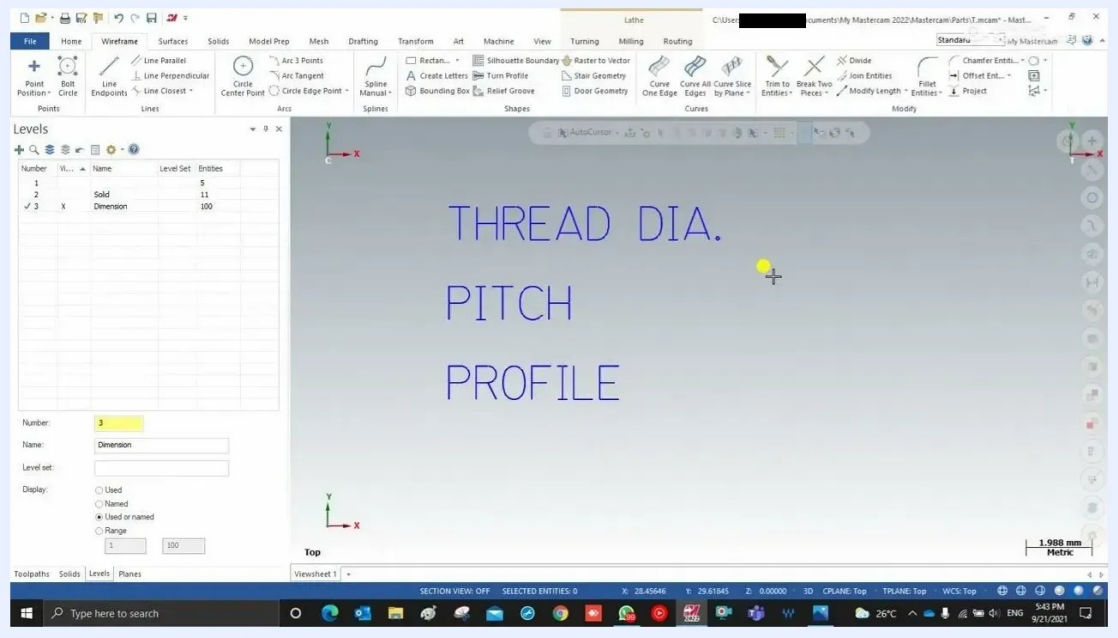
Prepare the thread profile

Step 1: Gather the profile details from the component drawing

Obtain or create a detailed sketch of the thread profile, and make sure all critical dimensions are clearly marked. The example drawing uses these values:

- Total thread length: **12.0 mm**
- Profile segments: **4.0 mm**, **2.0 mm**, **4.0 mm**, **2.0 mm**, as labeled on the drawing
- Profile height: **15.0 mm**

- Profile angle: **20°**

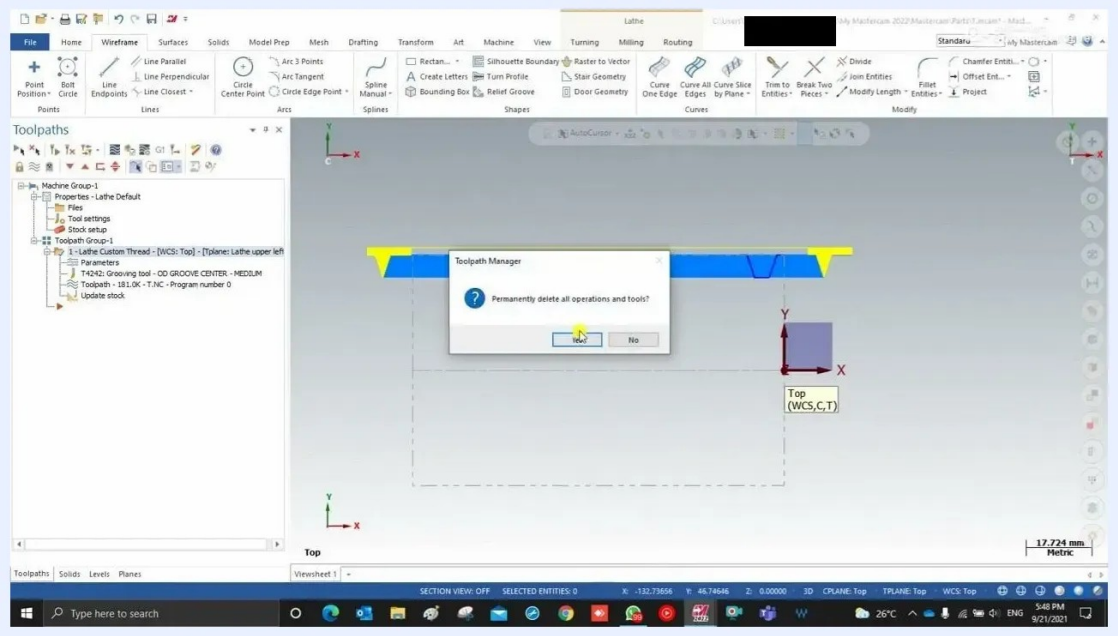


Hand-drawn sketch of the thread profile with labeled dimensions: 4.0, 2.0, 4.0 and 2.0 mm segments, 12.0 mm total length, 15.0 mm height and a 20° angle.

Step 2: Review how the profile sits in the graphics area

In the graphics area, the thread profile appears as a V-shaped groove sketched on the half-side diameter of the part. In Mastercam, you typically create thread profiles on the half-diameter (cross-section) of the workpiece.

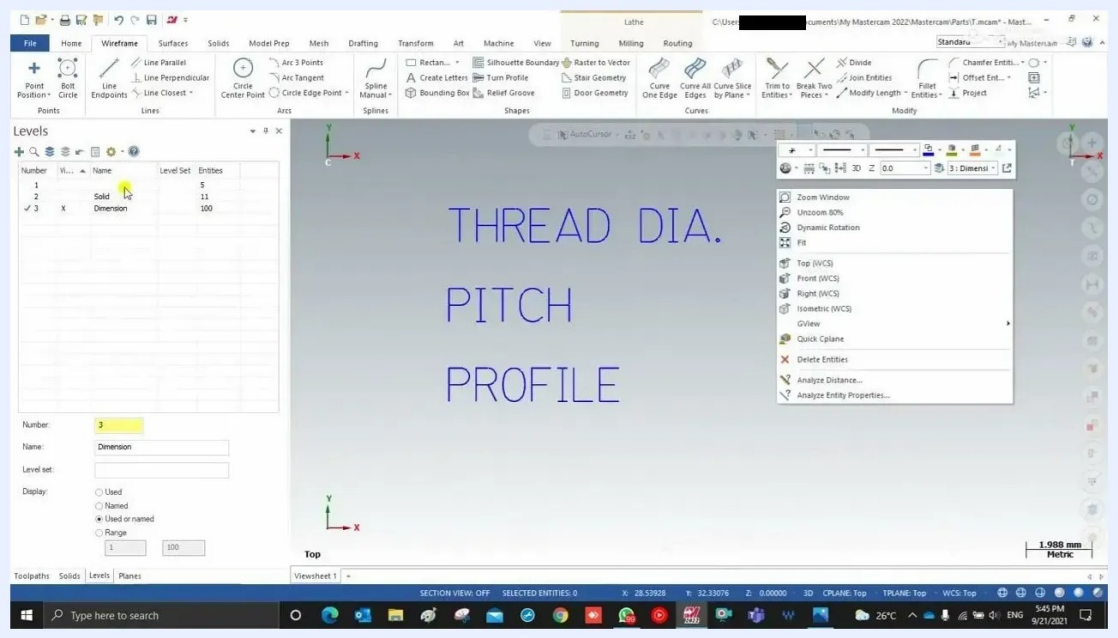
The coordinate system and the reference square (**Top (WCS, C, T)**) are visible for orientation.



Mastercam workspace showing a V-shaped thread profile on the half-side diameter, with coordinate axes and the reference square.

Step 3: Draw the profile in Mastercam

With all profile details at hand, replicate the profile as a 2D sketch in Mastercam. Make sure you understand how each dimension on the drawing translates to your CAD environment.



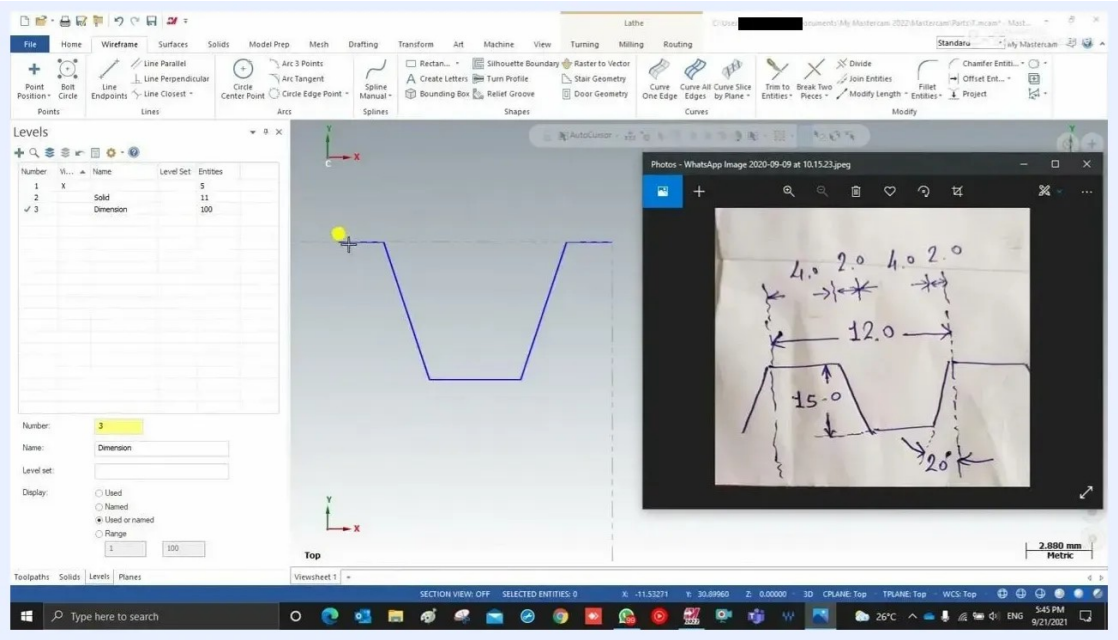
Hand-drawn profile with all dimensions visible, ready to be translated into a 2D CAD drawing.

1. Open Mastercam and set up your workspace.
2. Draw the thread profile using the dimensions from your sketch.
3. Use the **Levels** panel to organize your drawing layers, for example **Dimension** for profile sketches.

Step 4: Center the main profile and distribute the lengths equally

Compare your CAD drawing with the hand-drawn sketch. Verify that the main profile segment (12.0 mm) is centered. Distribute the remaining straight portions (the 4.0 mm and 2.0 mm segments) equally on both sides of the main profile.

This centering and equal distribution is critical for accurate custom thread creation. Check the **Levels** panel to confirm you are working on the correct layer, such as **Dimension**. Use the sketch as a visual reference for segment placement and symmetry. Double-check that the profile matches the sketch in both shape and dimension.



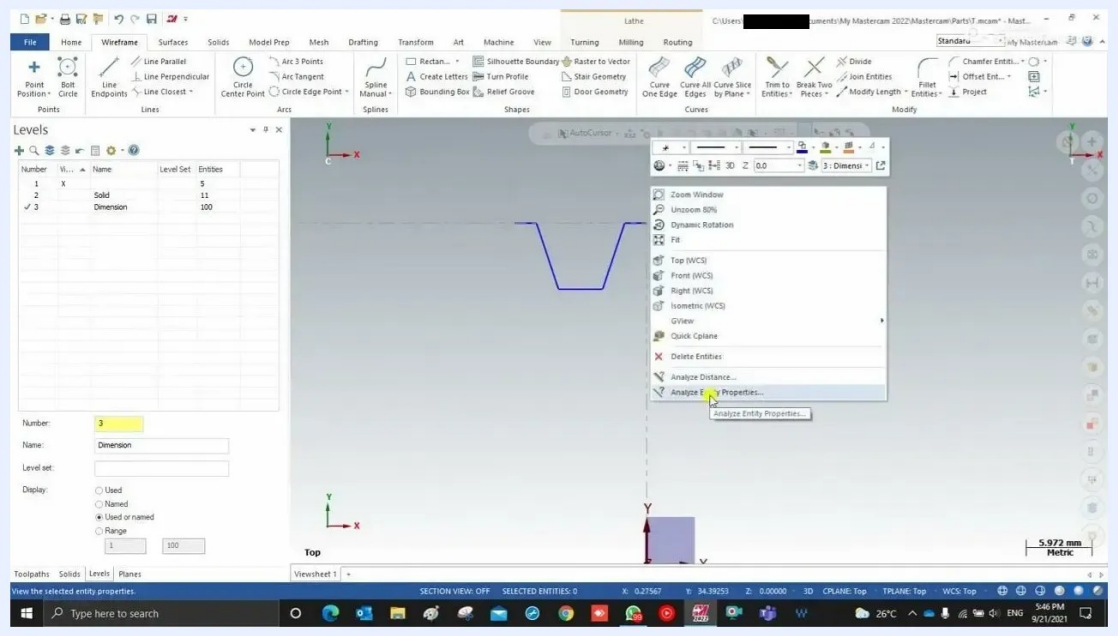
Mastercam workspace with the 2D thread profile and the hand-drawn sketch side by side, showing the centered main profile and equally distributed lengths.

Step 5: Measure the profile with Analyze Distance

Use the **Analyze Distance** tool to measure the total length of the main profile segment.

1. Right-click in the workspace and select **Analyze Distance...** from the context menu.
2. Click the start point and the end point of the main profile segment.
3. Confirm that the measured distance matches the required value of 12.0 mm.

If necessary, adjust the profile so the main segment is exactly 12.0 mm and stays centered.

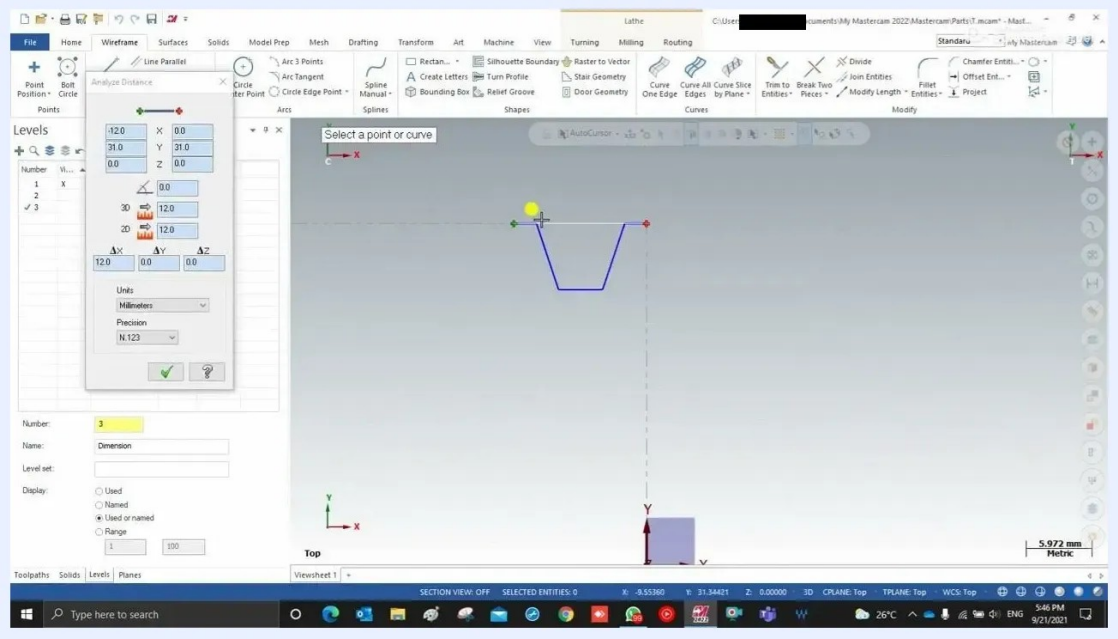


Mastercam workspace showing the right-click context menu with Analyze Distance... highlighted, ready to measure the profile segment.

Step 6: Verify the measured distance and finalize the profile

Check the measurement dialog box after using the tool. Make sure the X-axis distance (ΔX) is exactly **12.0 mm**, as specified in the drawing.

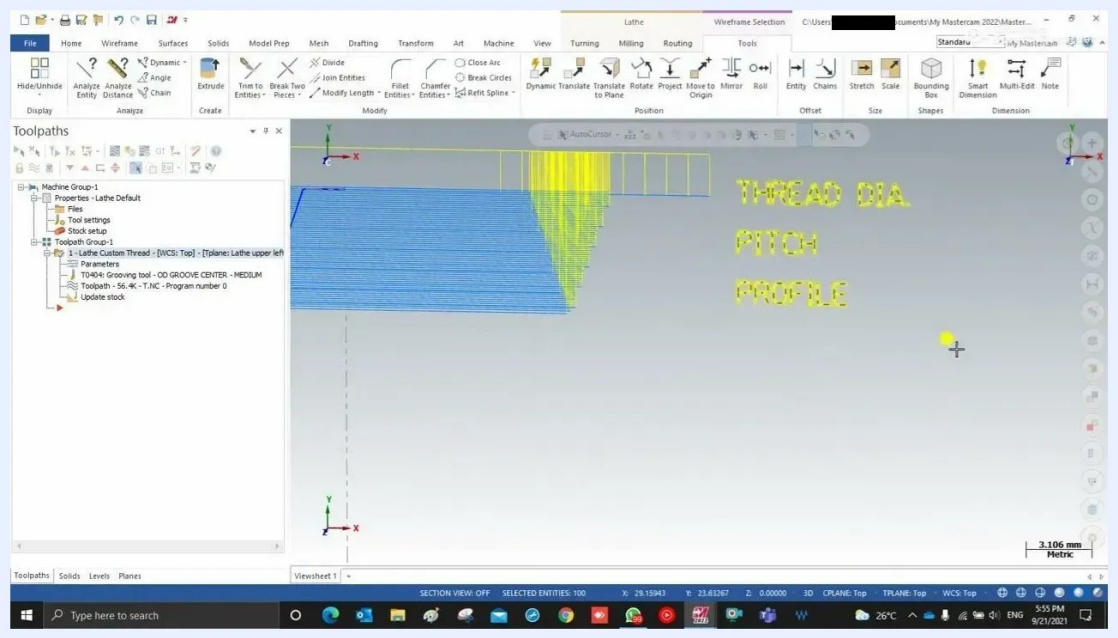
Confirm that the profile endpoints align with the intended positions and that the profile is symmetrical. Make any final adjustments if the measured distance does not match the required value.



Mastercam workspace with the Analyze Distance dialog open, showing ΔX = 12.0 mm and confirming the main profile segment length.

Step 7: Leave equal allowance for extra material

When you define the thread profile, leave extra material (stock allowance) equally on both sides of the profile. This ensures balanced machining and accurate final dimensions.



Close-up of the thread profile and toolpath, with the THREAD DIA., PITCH and PROFILE labels visible.

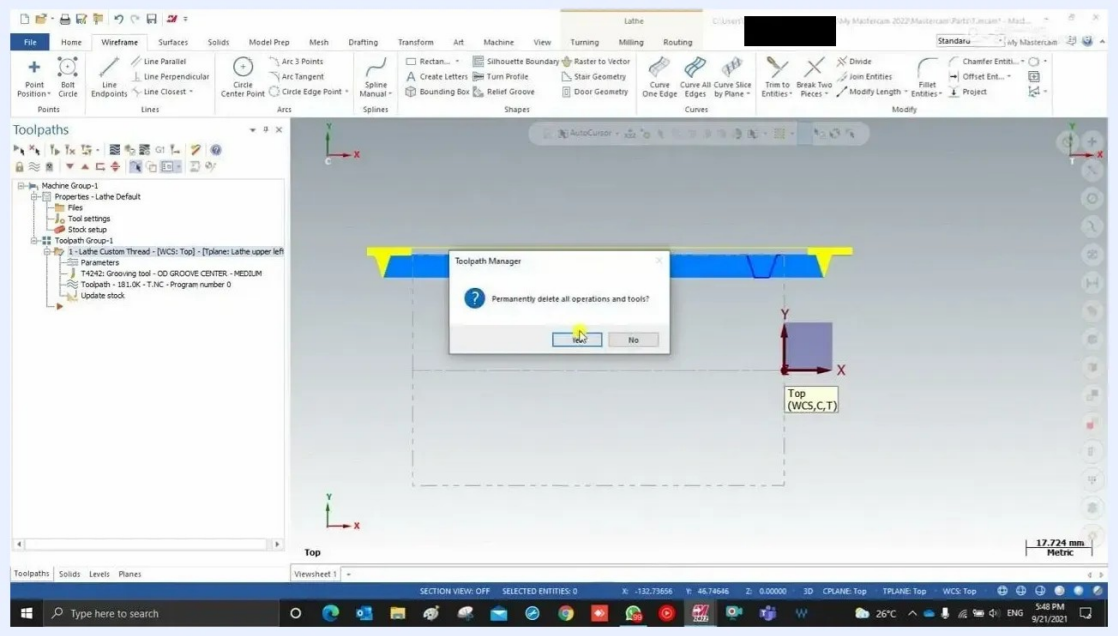
Define the raw material

Step 8: Delete all existing thread operations

Clear any previous thread toolpaths and tools so you can start with a new setup.

1. In the Mastercam workspace, open the **Toolpaths** panel.
2. Select the existing thread operations and tools you want to remove.

3. When the **Toolpath Manager** dialog asks "Permanently delete all operations and tools?", click **Yes**.



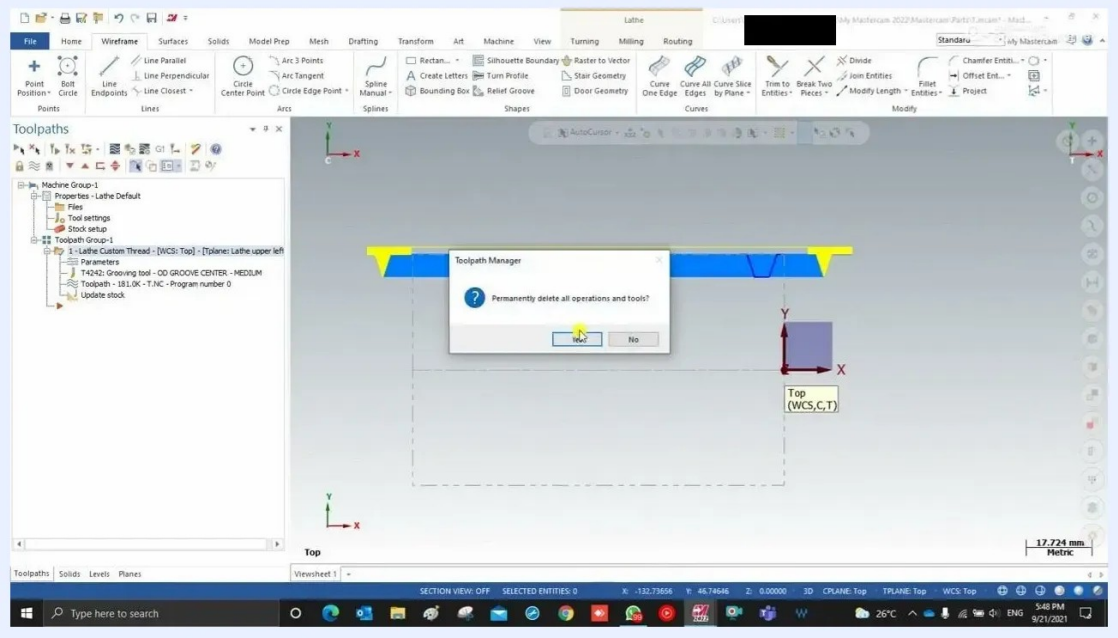
Mastercam interface with the Toolpath Manager dialog asking to permanently delete all operations and tools, with the Yes button highlighted.

Step 9: Open Machine Group Properties

Open the **Machine Group Properties** dialog by clicking the appropriate icon or right-clicking the machine group.

1. Go to the **Stock Setup** tab. The dialog displays options for stock, chuck jaws, tailstock center and steady rest.
2. Make sure **Stock Plane** is set to **Top**.

3. Under the **Stock** section, click **Properties** to define the raw material.



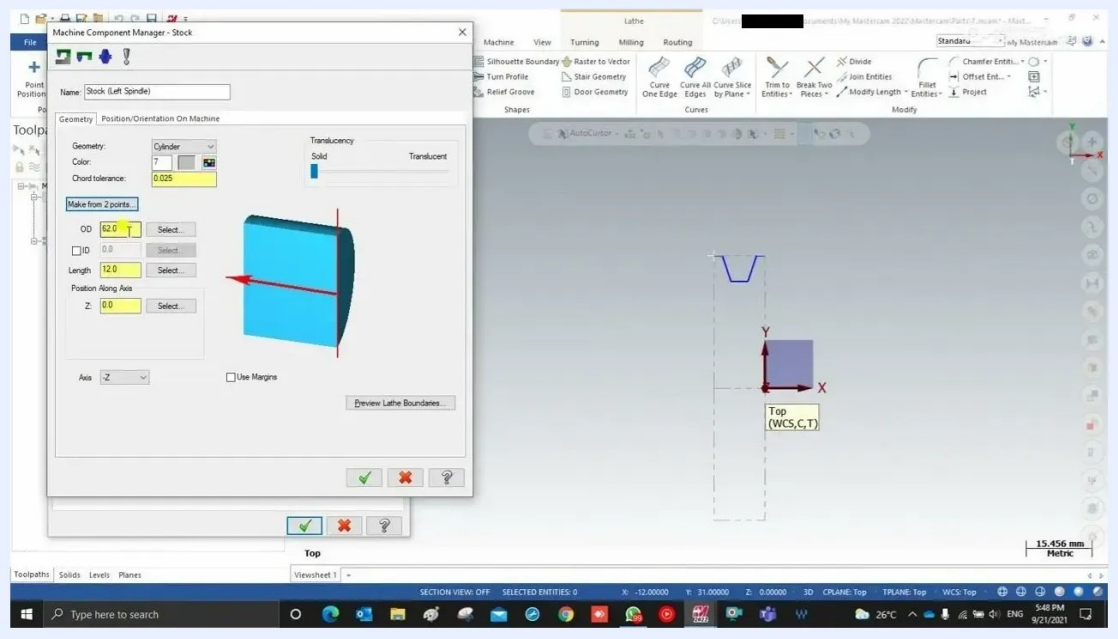
Machine Group Properties dialog showing the stock setup options, with the Properties button highlighted.

Step 10: Define the raw material dimensions

In the **Machine Component Manager - Stock** dialog, set these parameters:

- **Geometry:** Cylinder
- **OD (Outer Diameter):** 62.0 mm
- **Length:** 12.0 mm (you update this in the next step)
- **Chord tolerance:** 0.025
- **Position Along Axis (Z):** 0.0

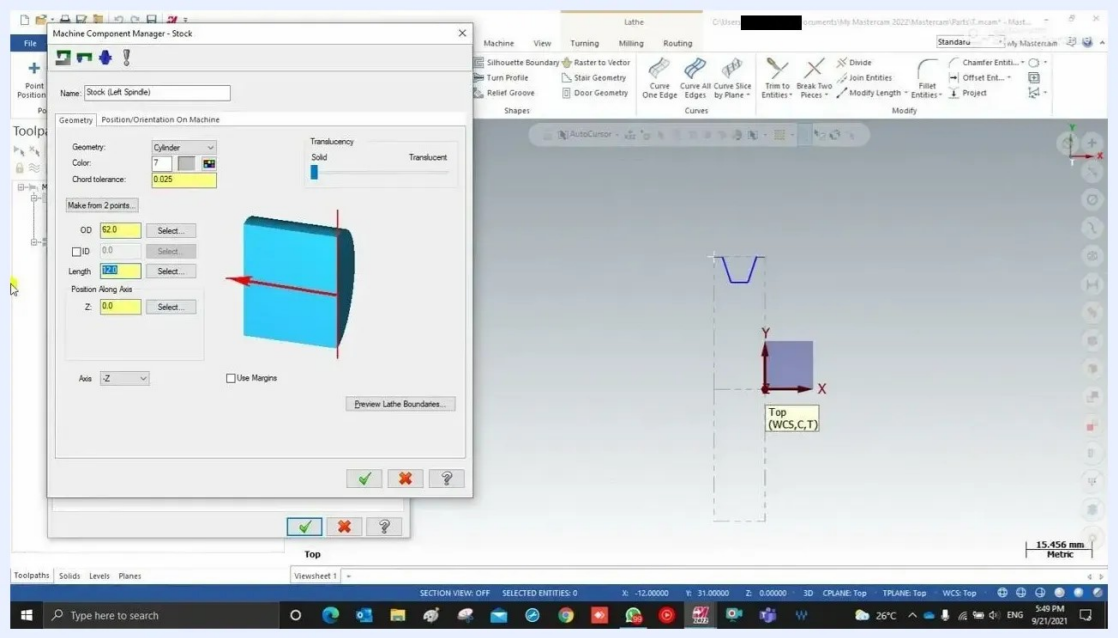
If you prefer to define the stock by selecting two points in the graphics area, click **Make from 2 points...**. The preview window shows a blue cylindrical stock with a red axis arrow.



Machine Component Manager - Stock dialog with OD set to 62.0 mm, Length set to 12.0 mm and a cylinder preview.

Step 11: Update the raw material length

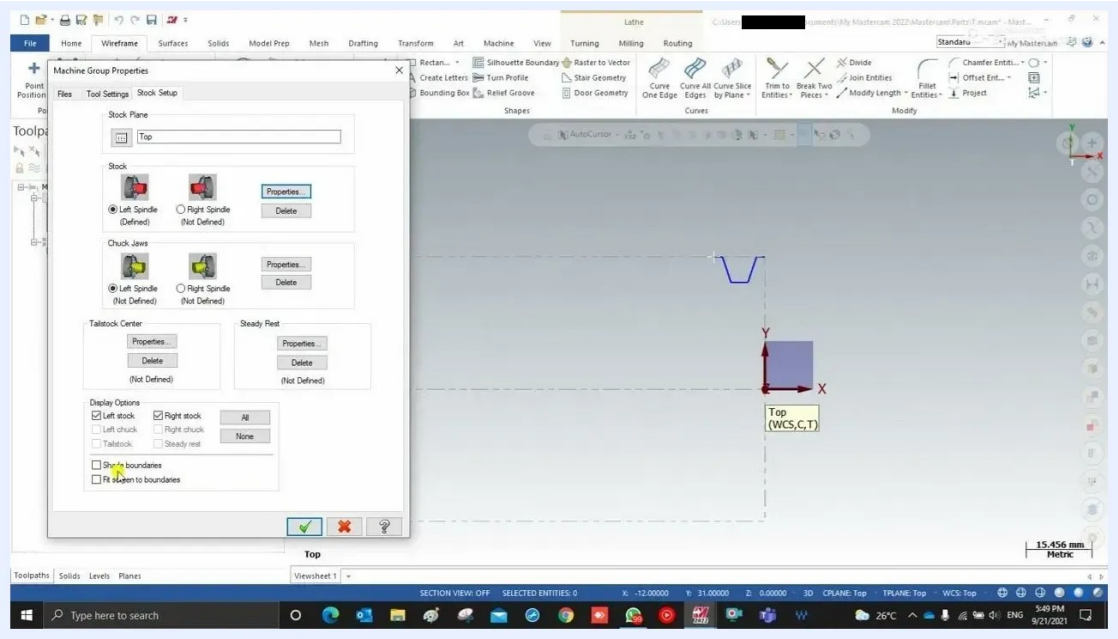
Change the **Length** value to **100.0 mm** to match the total required length of the raw material. Leave all other parameters unchanged. The preview updates to reflect the new stock length.



Machine Component Manager - Stock dialog with OD set to 62.0 mm and Length updated to 100.0 mm.

Step 12: Finalize the stock setup

Return to the **Machine Group Properties** dialog. Make sure the left spindle is defined, and check the display options you need, such as **Left stock** and **Shade boundaries**. Click the green checkmark to confirm and apply the stock setup.

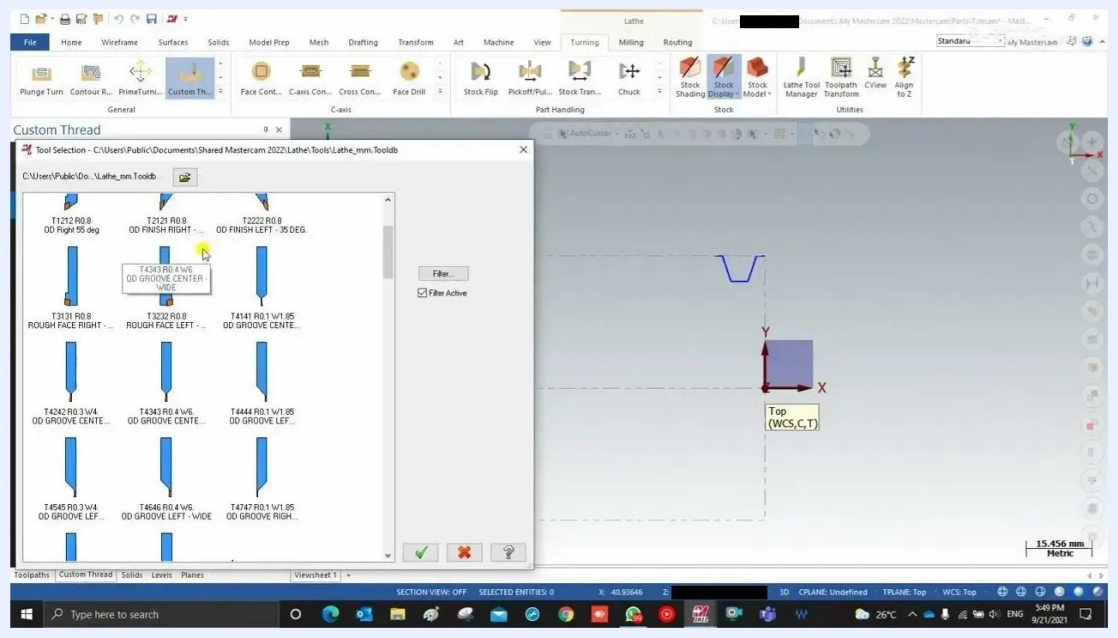


Machine Group Properties dialog with the stock setup finalized and display options selected.

Create the Custom Thread toolpath

Step 13: Start the Custom Thread operation

In the **Lathe** ribbon, click **Custom Thread** under the **General** section. The **Toolpaths** panel now lists a new "Lathe Custom Thread" operation under **Toolpath Group-1**. The graphics area shows the thread profile and the updated stock boundaries.

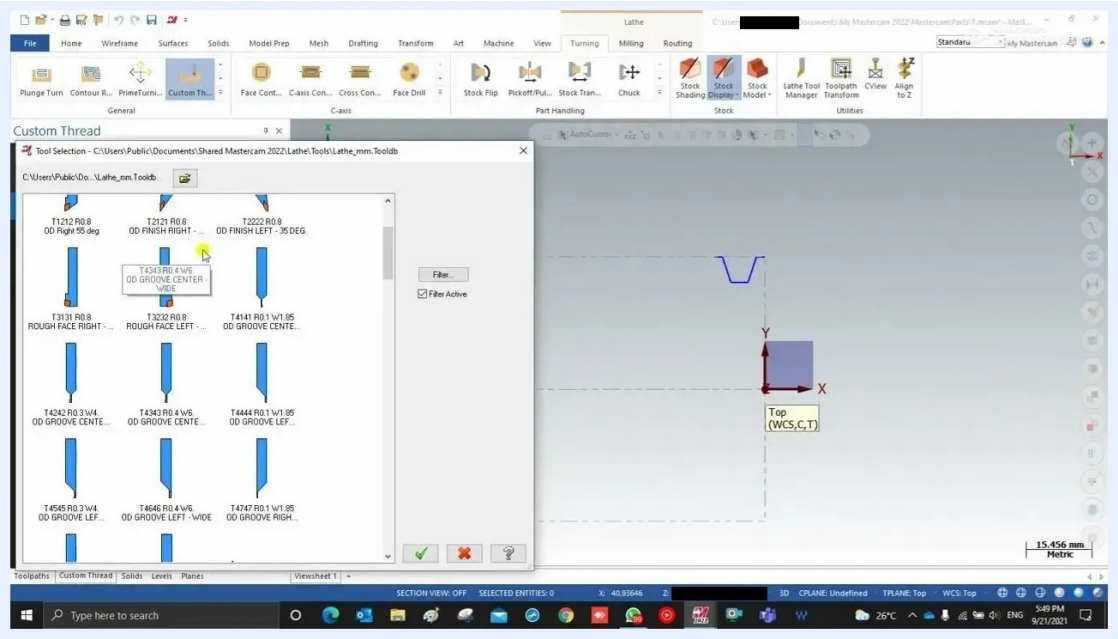


Mastercam workspace with the Custom Thread toolpath started and the thread profile visible.

The **Custom Thread** dialog opens with tool selection and operation parameters. In the left panel, you can select or define the tool and set the tool data (Tool number: 4, Offset number: 4, Station number: -1). You can also configure coolant (Flood: On, Mist: Off, Thru-tool: Off). The workspace stays visible in the background for reference.

Step 14: Open the tool selection

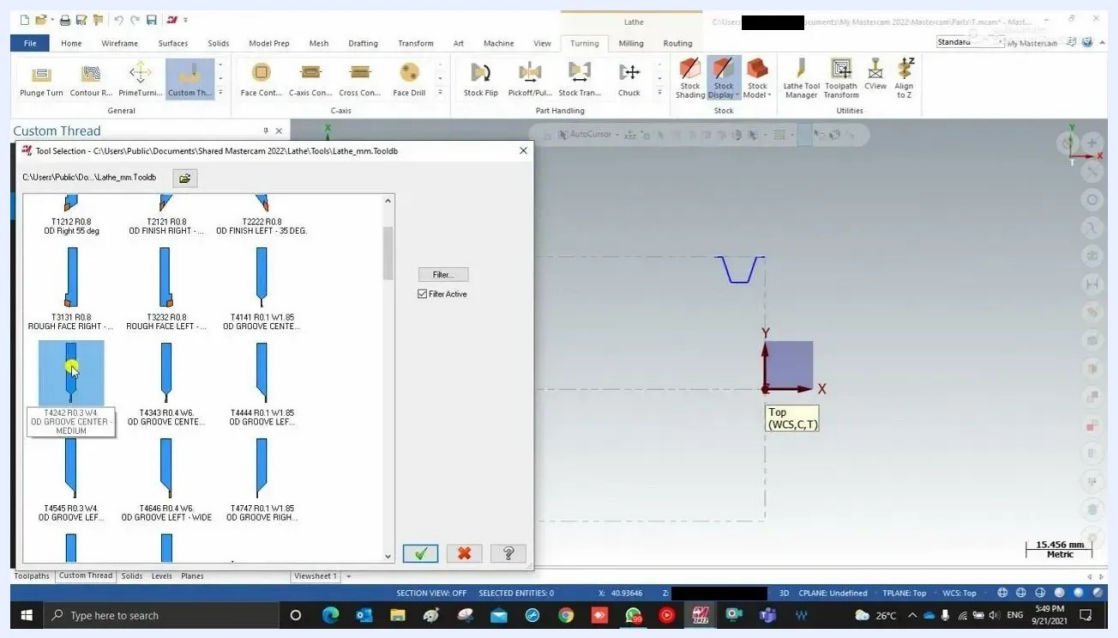
Click the tool selection field to open the **Tool Selection** dialog. Browse the tools in the Mastercam tool library, shown with graphical icons and labels, for example T1212 R0.8 OD Right 55 deg and T4242 R0.3 W4 OD GROOVE CENTER - MEDIUM. Use the filter if you need to narrow down the choices.



Tool Selection dialog showing a list of lathe tools with graphical icons and labels.

Step 15: Choose the neutral grooving tool

Select the tool labeled **T4242 R0.3 W4 OD GROOVE CENTER - MEDIUM** (Neutral Grooving). Click the green checkmark to confirm. The tool is now assigned to the custom thread operation.



Tool Selection dialog with the T4242 R0.3 W4 OD GROOVE CENTER - MEDIUM tool highlighted.

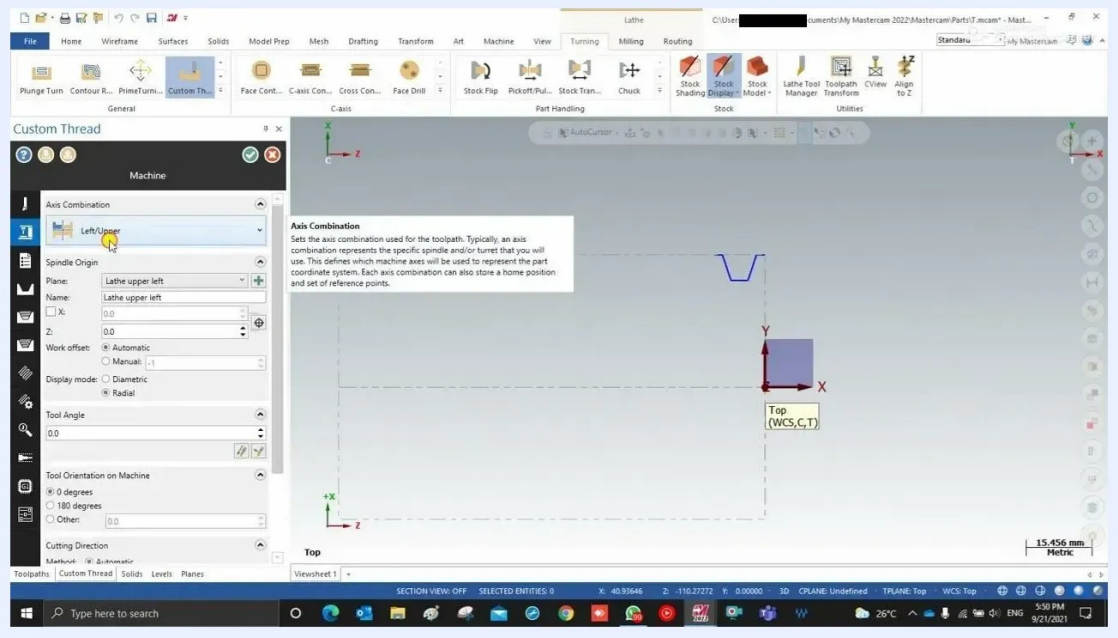
Step 16: Set the axis combination

In the **Custom Thread** dialog, open the **Axis Combination** dropdown and select **Left/Upper**. The tooltip explains that this setting determines which spindle and turret axes the toolpath uses.

Then confirm these settings:

- The spindle origin is **Lathe upper left** with X: 0.0 and Z: 0.0.
- **Work offset** is **Automatic**.

- **Display mode** is **Diametric**.



Custom Thread dialog with Axis Combination set to Left/Upper, the tooltip visible and the spindle origin fields shown.

Configure the thread shape and pitch

Step 17: Open the Shape tab and review the options

In the **Custom Thread** dialog, click the **Shape** tab in the left panel. This tab defines the thread geometry and style.

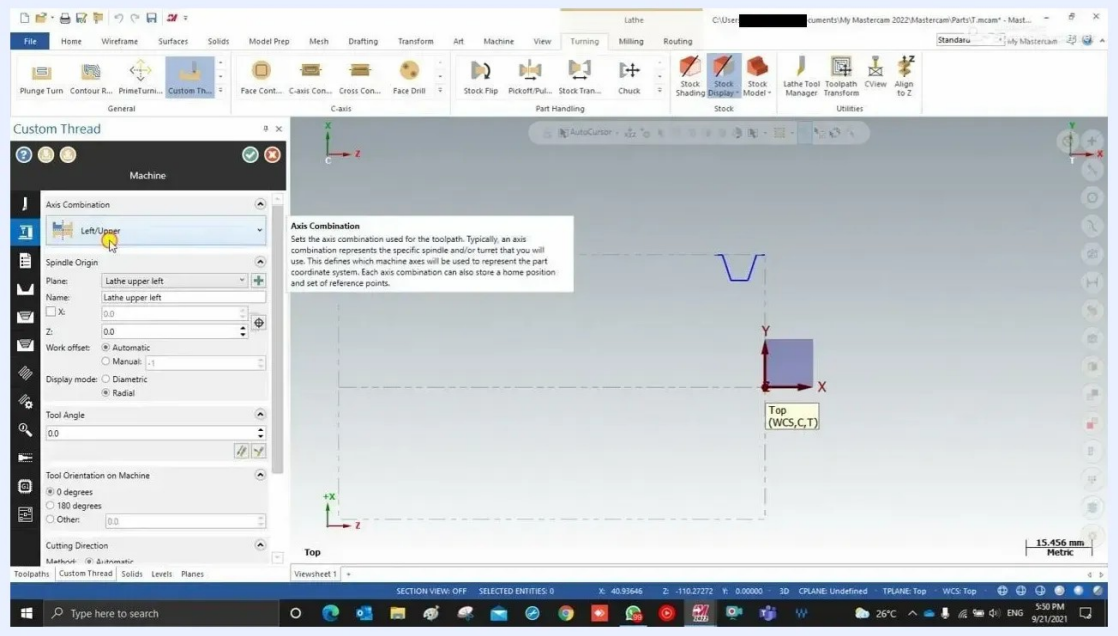
The **Shape Type** section offers two options:

- **Parametric:** Define the thread by entering dimensions and settings.
- **Chain:** Define the thread by selecting a pre-drawn profile (chain) in the graphics area.

The tab also contains **Name**, **Template**, **Thread Orientation** (Outer diameter or Inner diameter) and **Cross Center Line Cut** (Enable/Disable) fields. The **Shape Style** section lets you select the thread type, such as Square, and shows these values:

- **Major diameter:** 50.0
- **Pitch (mm per thread):** 10.0
- **Width:** 10.0

- **Height:** 10.0



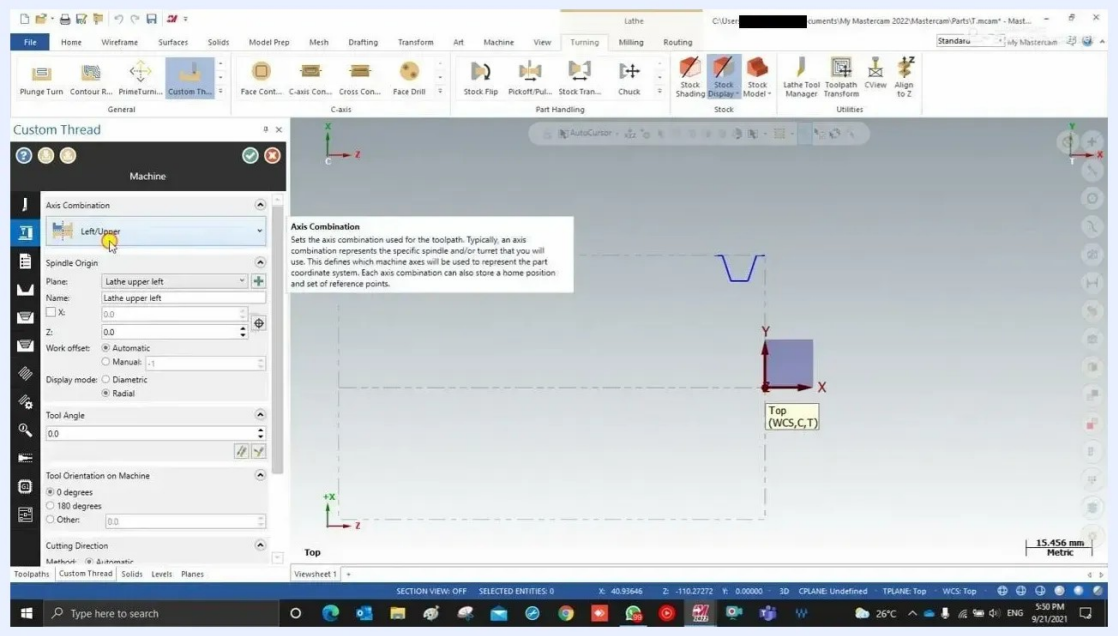
Custom Thread Shape tab showing Shape Type (Parametric and Chain), thread orientation and shape style fields with values.

Step 18: Choose the shape definition method

Decide between the two methods:

- **Parametric:** Specify thread dimensions directly for OD (Outer Diameter) or ID (Inner Diameter) threads.
- **Chain:** Select a custom-drawn thread profile from the graphics area.

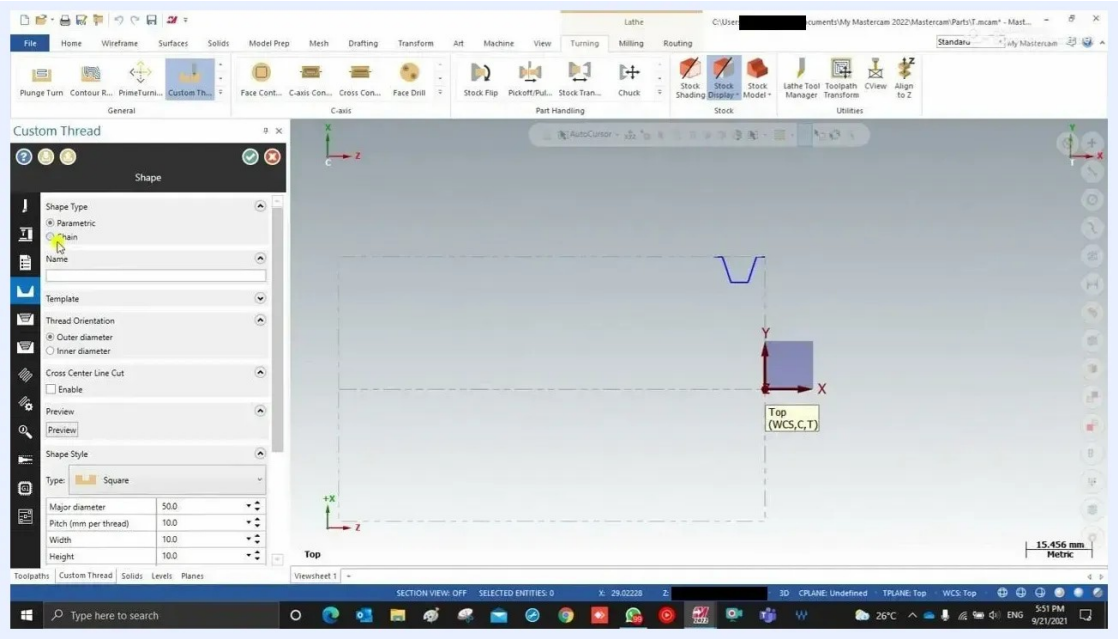
Use **Chain** for the greatest flexibility when you create a thread in Mastercam from a custom profile.



Custom Thread Shape tab with the Shape Type options (Parametric and Chain) visible, ready to select Chain.

Step 19: Switch to the Chain method

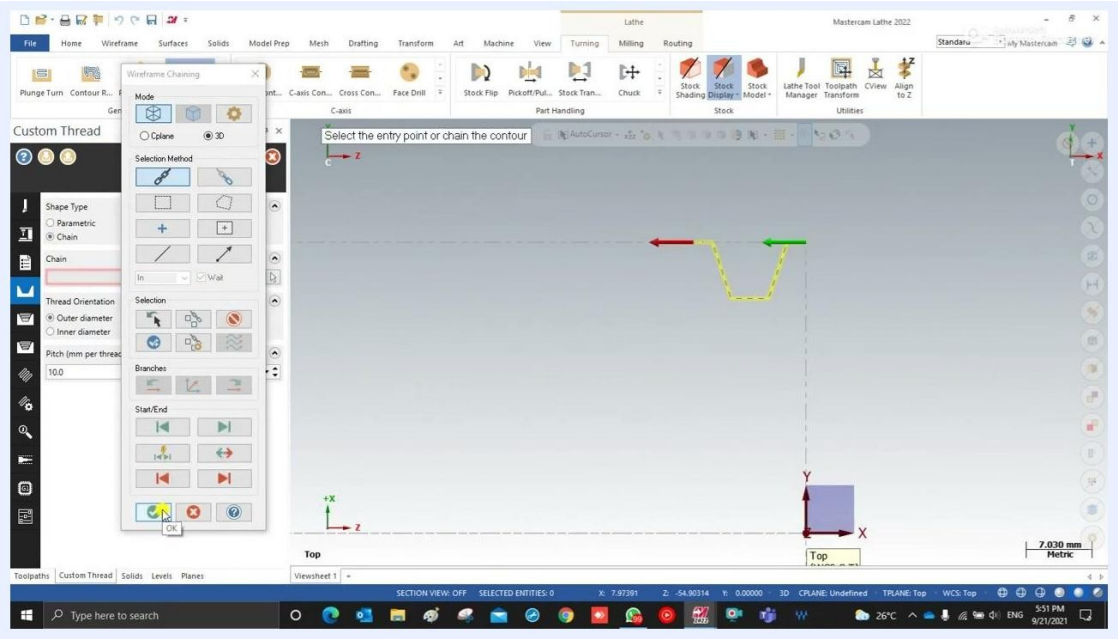
Select **Chain** under **Shape Type**. The **Chain** field becomes active so you can select a pre-drawn profile. Set **Thread Orientation** to **Outer diameter** or **Inner diameter** as required, and enter the **Pitch (mm per thread)** value, for example 10.0.



Custom Thread Shape tab with Chain selected, the Chain field active and thread orientation set to Outer diameter.

Step 20: Select the chain profile

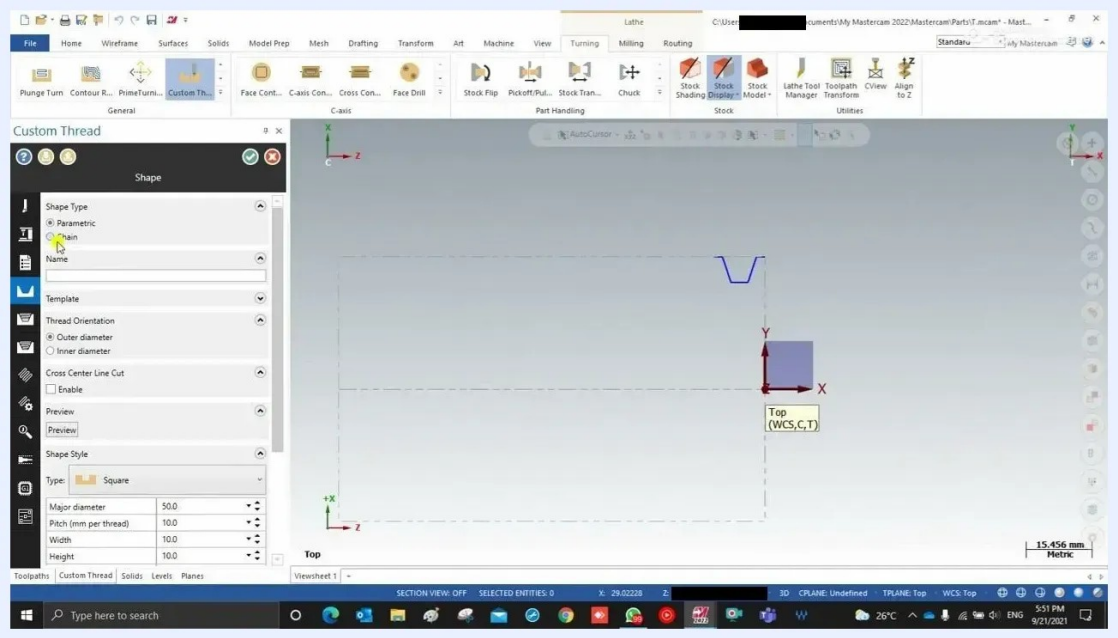
The **Wireframe Chaining** dialog appears and prompts you to select the entry point or chain the contour in the graphics area. The pre-drawn thread profile is highlighted in yellow, with red and green entry and exit arrows. Click **OK** to confirm the selection.



Wireframe Chaining dialog with the thread profile highlighted in yellow, entry and exit arrows visible and the OK button highlighted.

Step 21: Confirm the chain and set the thread orientation

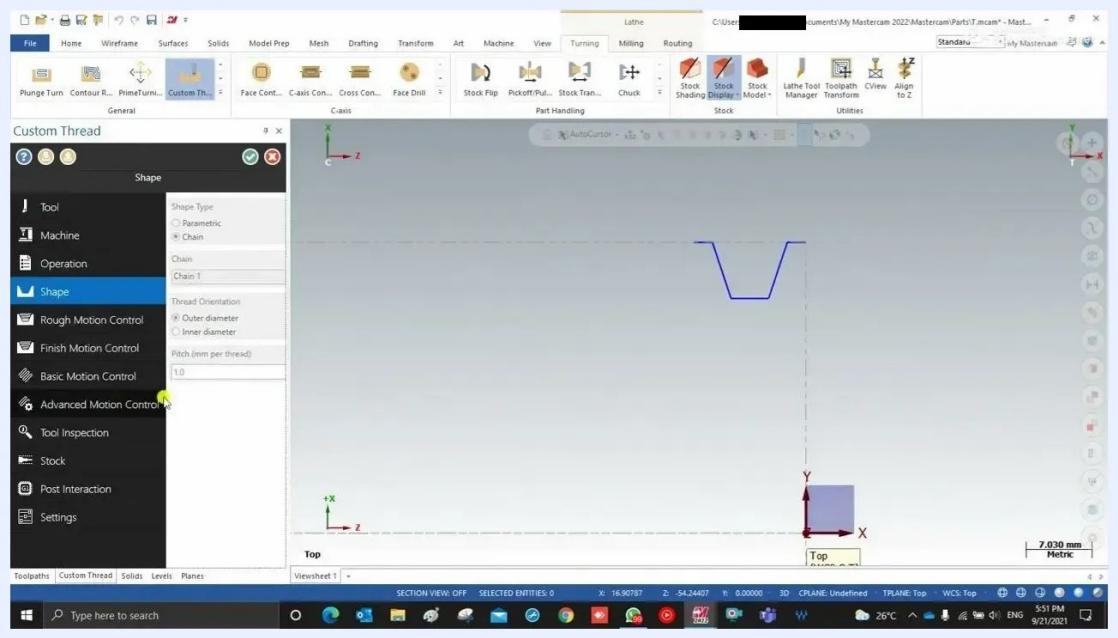
The selected chain is now listed as **Chain 1** in the **Custom Thread** dialog. Set **Thread Orientation** to **Outer diameter**, and make sure **Pitch (mm per thread)** is set to the value you need (10.0 in this stage of the example).



Custom Thread dialog with Chain 1 selected, thread orientation set to Outer diameter and pitch set to 10.0.

Step 22: Enter the pitch value

If you know the pitch of your thread, enter it in the **Pitch (mm per thread)** field. In this example, the required pitch is **15.0 mm**. Confirm all other parameters are correct before you continue.



Custom Thread dialog with the Shape tab active, Chain 1 selected, thread orientation set and the pitch field ready for input.

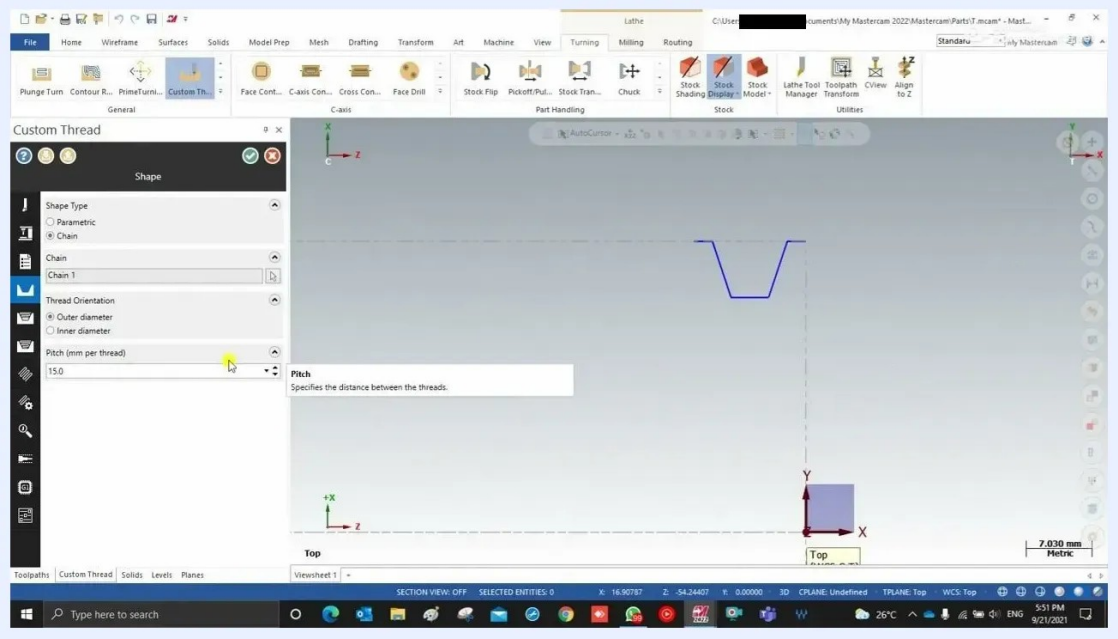
Step 23: Set the thread start position and review the parameters

Make sure the start position of the thread profile is **Z0**. The thread then begins exactly where the part face starts, which aligns its entry point with the workpiece face. Double-check that the selected chain begins at Z0 in the graphics area.

Confirm these settings in the **Custom Thread** dialog:

- **Shape Type:** Chain
- **Chain:** Chain 1 (the selected profile)
- **Thread Orientation:** Outer diameter
- **Pitch (mm per thread):** 15.0

Hover over the **Pitch** field to see the tooltip "Specifies the distance between the threads." Verify that the thread profile is visible in the graphics area with the correct shape and position.



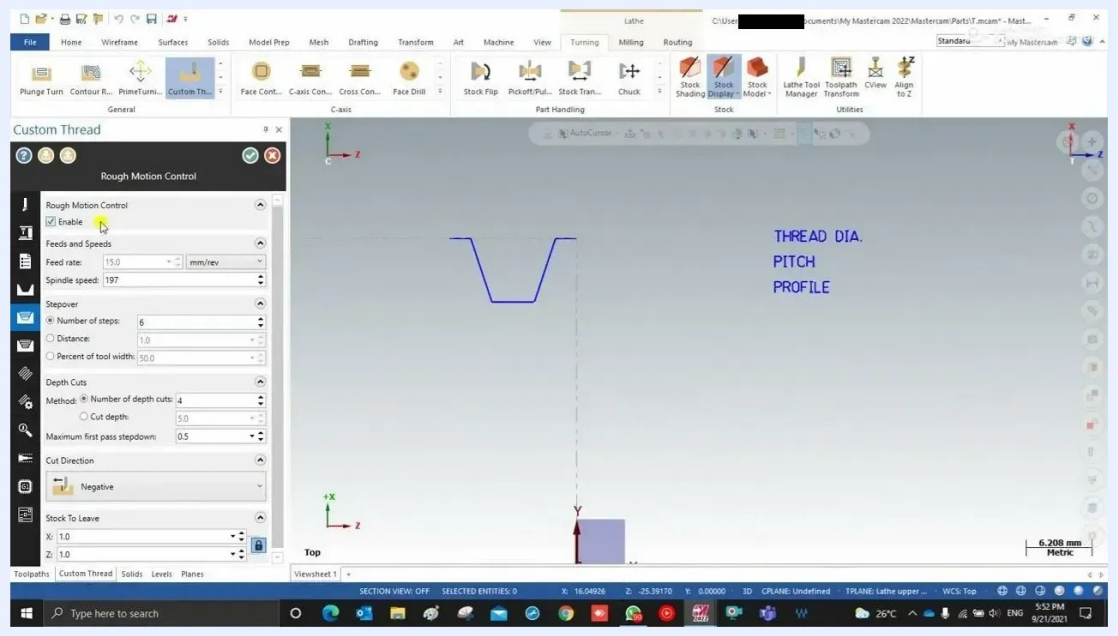
Custom Thread dialog with Chain 1 selected, thread orientation set to Outer diameter, pitch set to 15.0 and the thread profile visible in the graphics area.

Set the cutting parameters

Step 24: Decide whether to enable roughing passes

In the **Rough Motion Control** section, check or uncheck **Enable** depending on whether you want a roughing pass. If you do not want one, leave **Enable** unchecked.

The **Feeds and Speeds** section shows a feed rate of **15.0 mm/rev** and a spindle speed of **197**. Stepover and depth cut parameters are also visible.



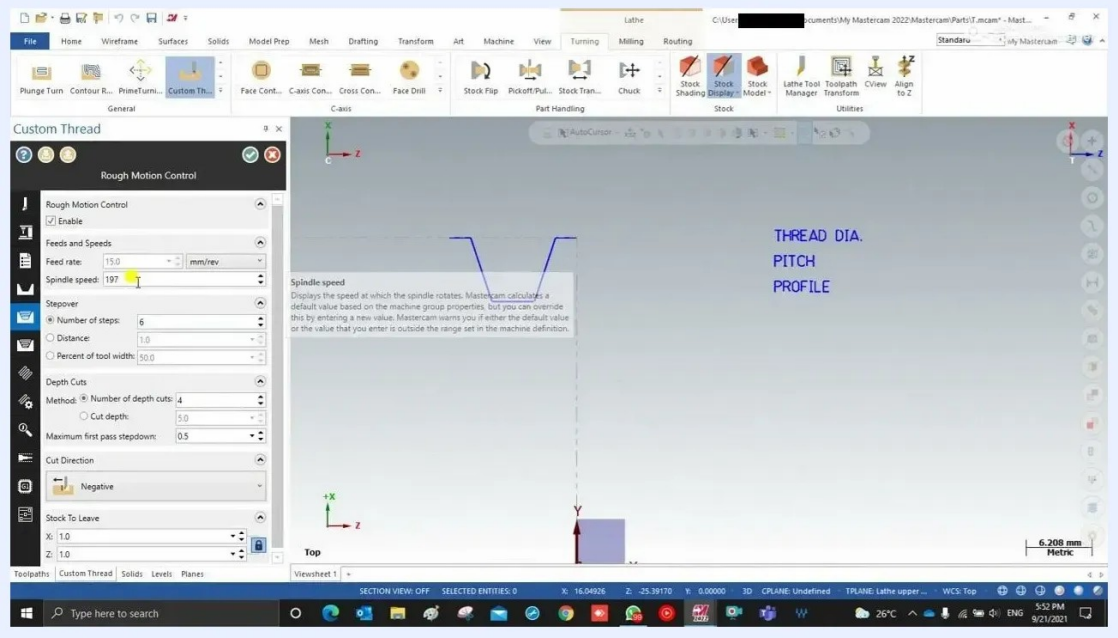
Rough Motion Control section with the roughing pass enabled and the feed rate and spindle speed fields visible.

Step 25: Edit the thread cutting parameters

Adjust these fields as needed in the **Rough Motion Control** section:

- Feed rate (mm/rev): **15.0**
- Spindle speed: **197** (editable)
- Number of steps: **6**
- Distance: **1.0**
- Percent of tool width: **50.0**
- Number of depth cuts: **4**
- Cut depth: **5.0**
- Maximum first pass stepdown: **0.5**

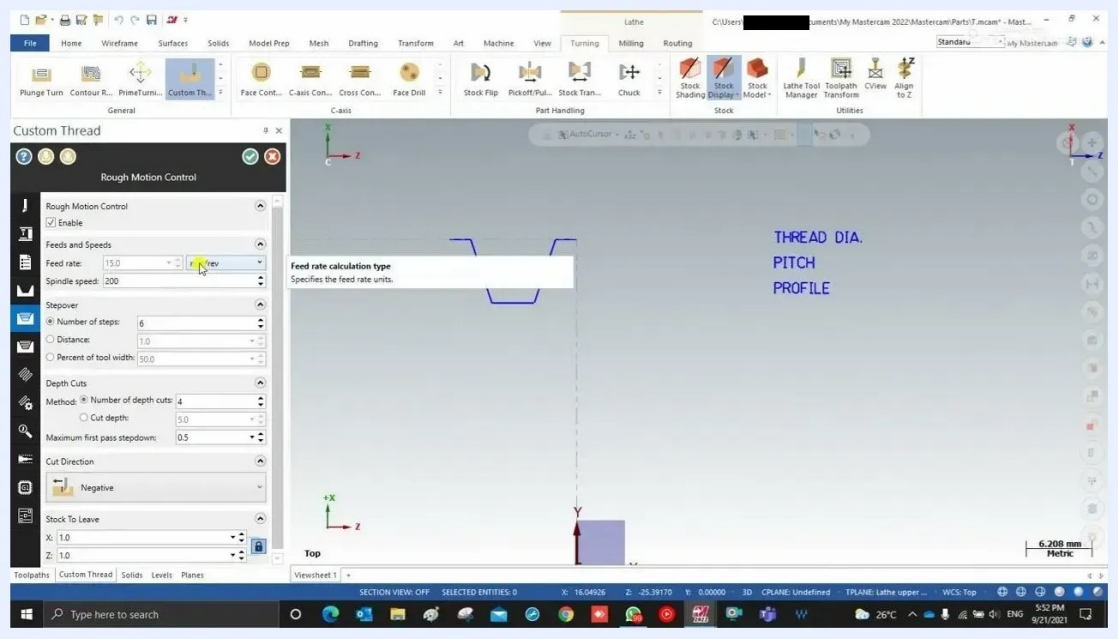
A tooltip may appear to explain the spindle speed parameter.



Custom Thread dialog with editable parameters for feed rate, spindle speed, steps and depth cuts.

Step 26: Adjust the feed rate calculation type

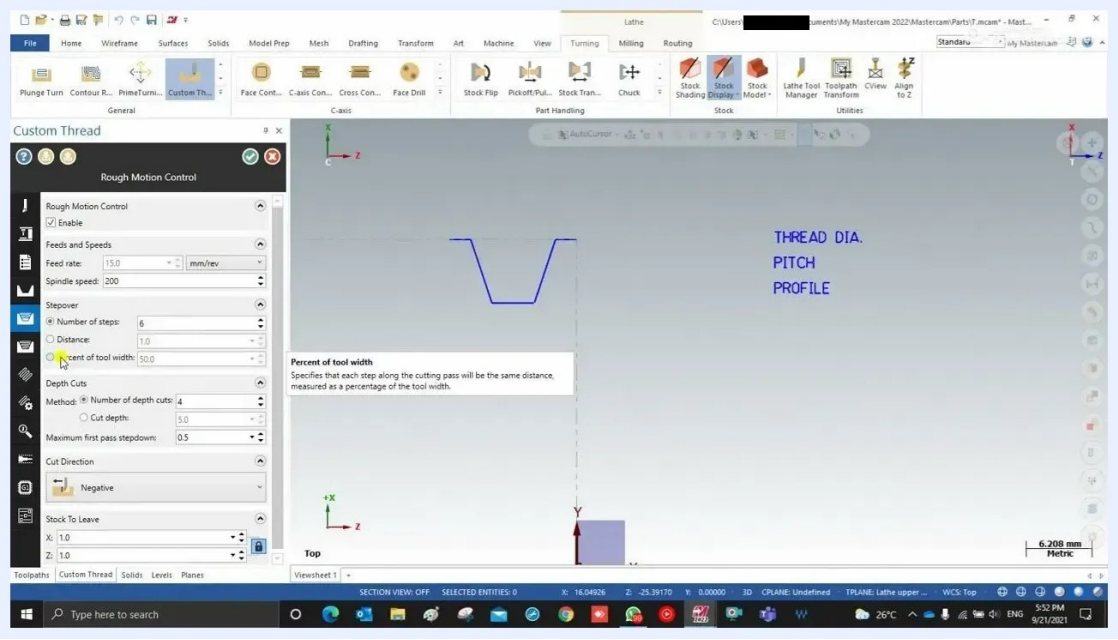
Keep the feed rate at **15.0 mm/rev** and update the spindle speed to **200**. A tooltip may appear to explain the feed rate calculation type.



Feed rate set to 15.0 mm/rev and spindle speed set to 200, with the tooltip for the feed rate calculation type visible.

Step 27: Define the stepover

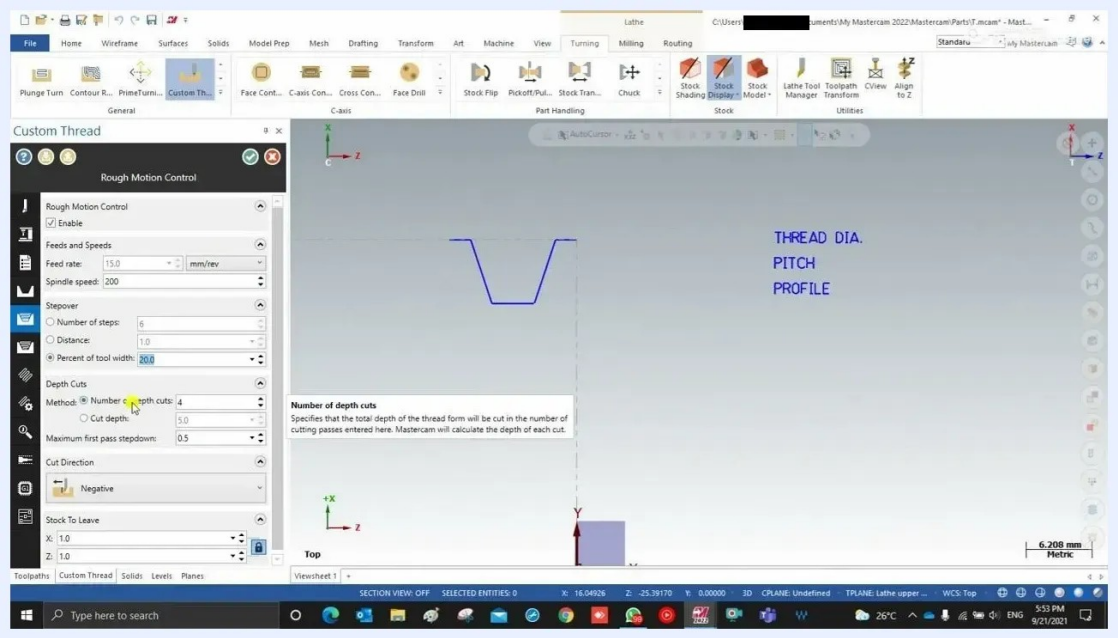
In the **Stepover** section, select **Percent of tool width** and enter **20.0**. Each step along the cutting pass is then 20% of the tool width.



Stepover set to 20% of tool width in the Custom Thread dialog.

Step 28: Define the depth of cut

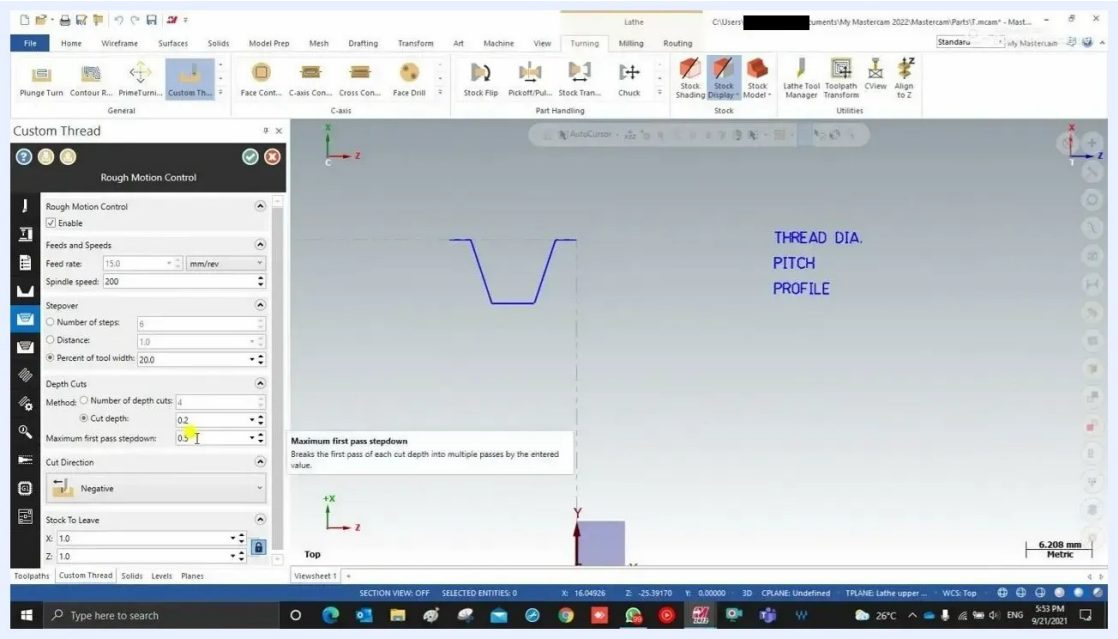
In the **Depth Cuts** section, select **Number of depth cuts** and enter **4**. Alternatively, you can specify a cut depth value. Mastercam calculates the depth of each cut automatically.



Depth Cuts section with the number of depth cuts set to 4 and a tooltip explaining the parameter.

Step 29: Set the maximum first pass stepdown

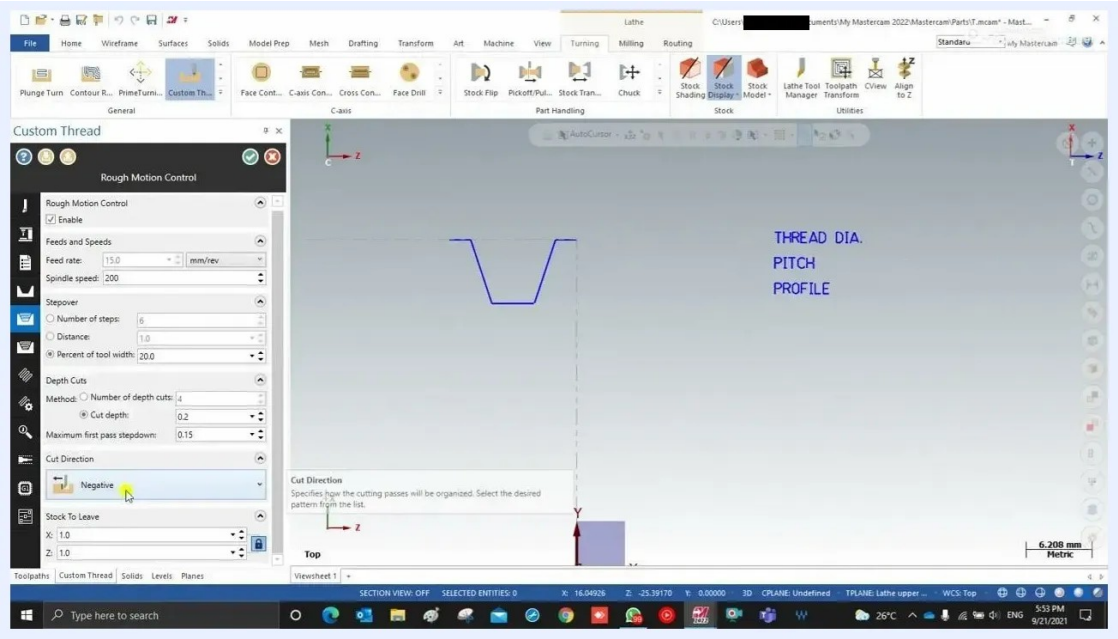
In the **Depth Cuts** section, enter **0.5** for **Maximum first pass stepdown**. This value breaks the first pass of each cut depth into multiple passes. The example values shown are a cut depth of **0.2** and a maximum first pass stepdown of **0.5**.



Maximum first pass stepdown set to 0.5 in the Depth Cuts section with all parameters visible.

Step 30: Set the cut direction

In the **Custom Thread** dialog, find the **Cut Direction** section and select **Negative**. The cutting passes are then organized in the negative direction along the Z-axis. The thread profile and the "THREAD DIA.", "PITCH" and "PROFILE" labels remain visible in the workspace.



Custom Thread dialog with Cut Direction set to Negative and the thread profile and parameter labels visible.

Configure finishing, overcut, end position and stock clearance

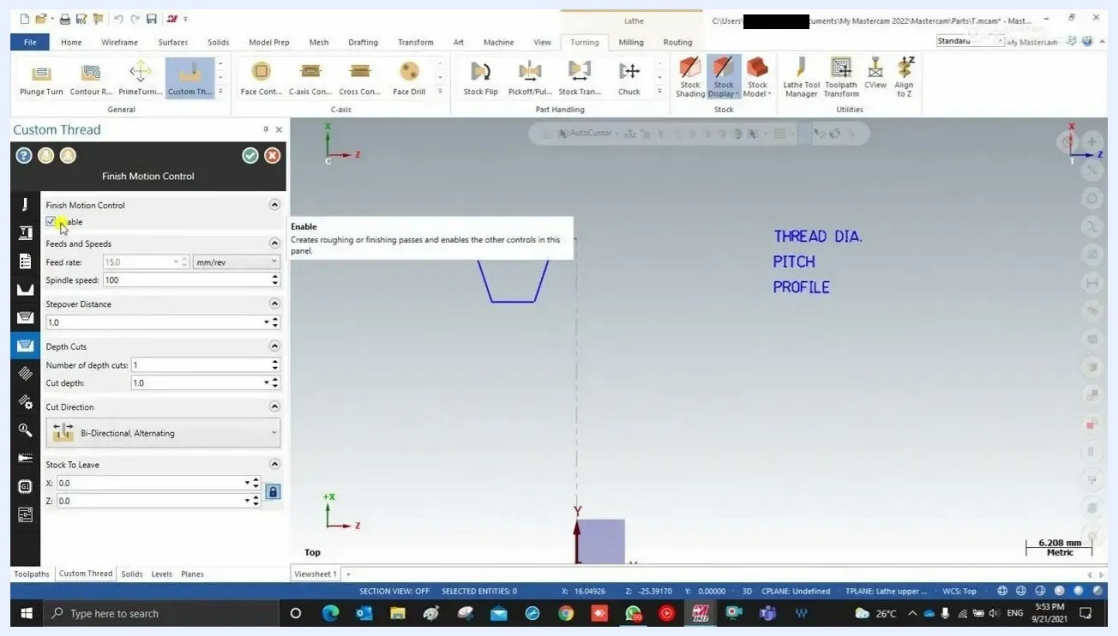
Step 31: Configure the finishing passes

Switch to the **Finish Motion Control** section. To use the current tool for finishing, disable the **Enable** checkbox. If you need finishing passes, enable the checkbox to activate the finishing controls. A tooltip explains the **Enable** option.

The following parameters are visible:

- Feed rate: **15.0 mm/rev**
- Spindle speed: **100**
- Stepover distance: **1.0**
- Number of depth cuts: **1**
- Cut depth: **1.0**

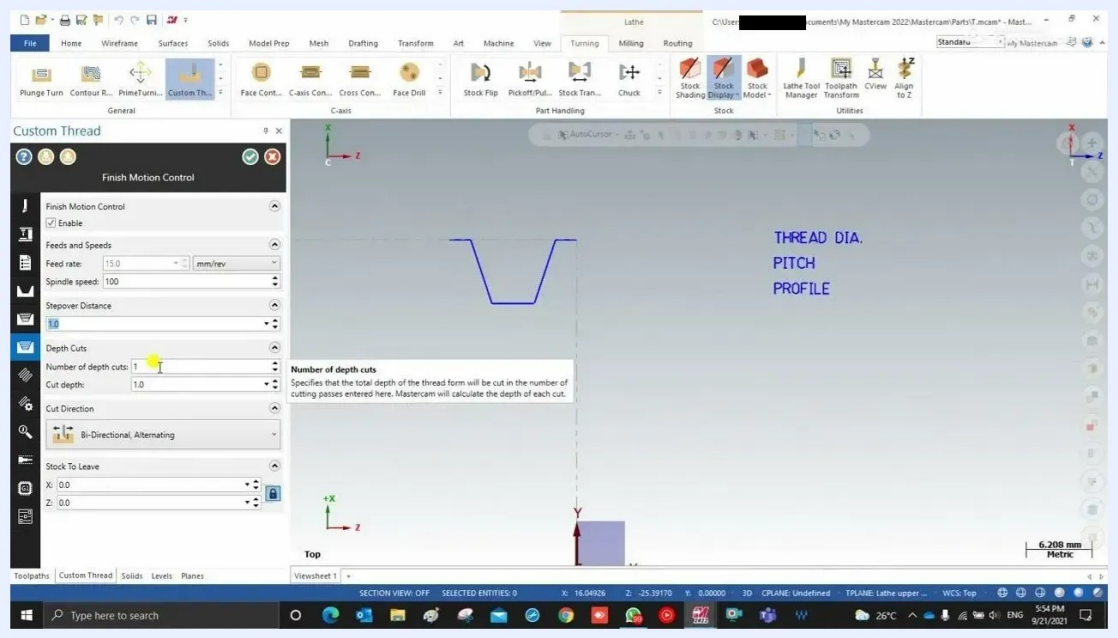
- Cut direction: **Bi-Directional, Alternating**
- Stock to leave: X = **0.0**, Z = **0.0**



Finish Motion Control section with the Enable checkbox highlighted and the finishing parameters visible.

Step 32: Set the overcut value

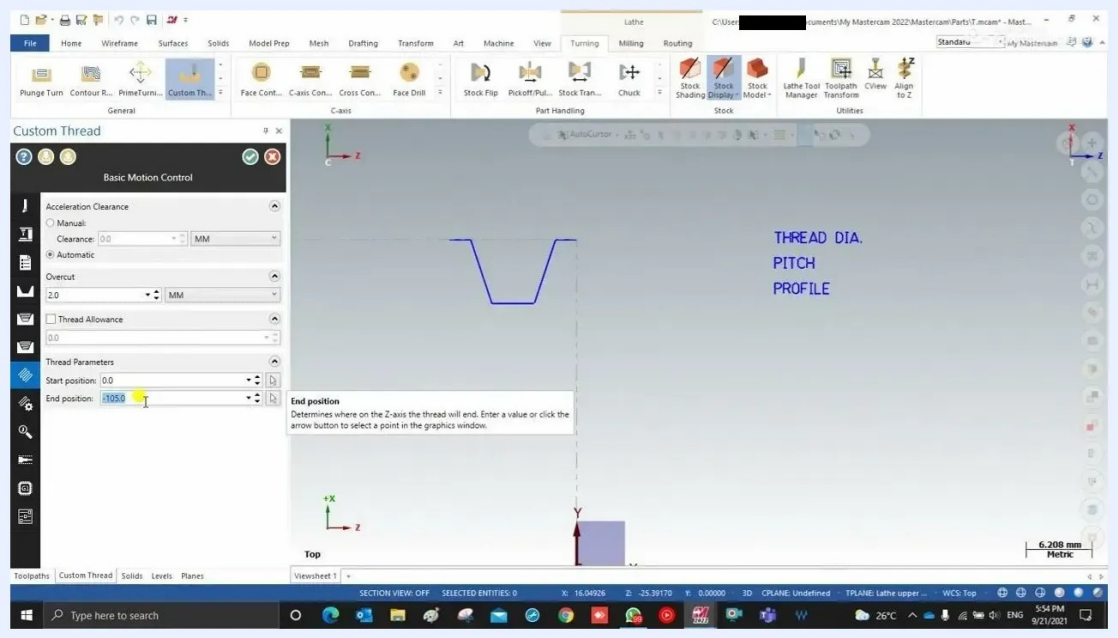
Switch to the **Basic Motion Control** section and enter **2.0 mm** in the **Overcut** field. This value determines how far past the end of the thread the tool travels before it retracts. A tooltip explains the overcut parameter.



Basic Motion Control section with Overcut set to 2.0 mm and a tooltip explaining overcut.

Step 33: Adjust the thread end position

In the **Thread Parameters** section, set **End position** to **-105.0**. This value determines where on the Z-axis the thread ends. **Start position** remains at **0.0**. A tooltip explains the end position parameter.

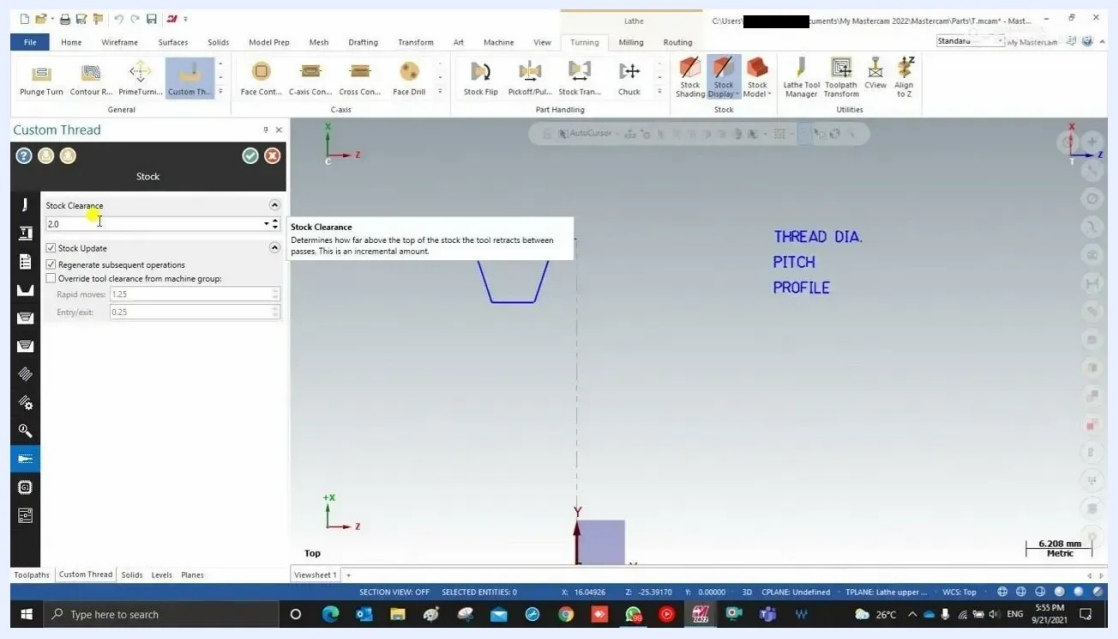


Basic Motion Control section with End position set to -105.0 and a tooltip explaining end position.

Step 34: Set the stock clearance

Switch to the **Stock** section and enter **2.0 mm** in the **Stock Clearance** field. This value determines how far above the top of the stock the tool retracts between passes.

The **Stock Update** option is enabled, and additional rapid move and entry/exit values are visible. A tooltip explains the stock clearance parameter.



Stock section with Stock Clearance set to 2.0 mm and a tooltip explaining stock clearance.

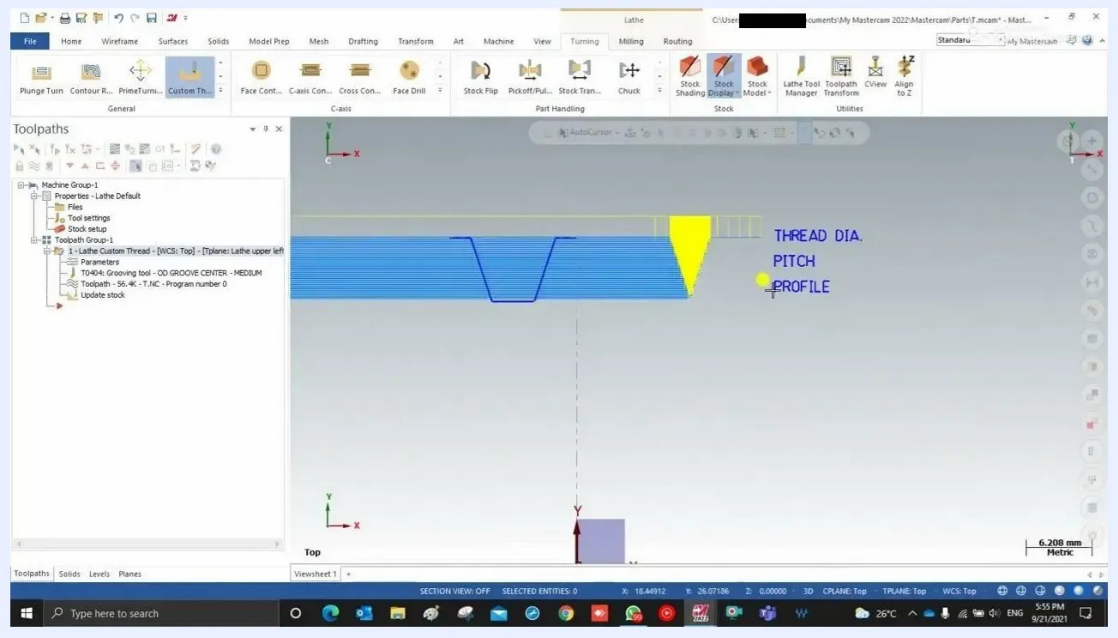
Generate and simulate the threading toolpath

Step 35: Confirm creation of the threading program

Click **OK** in the **Custom Thread** dialog to finalize your settings. The threading toolpath is generated and displayed in the workspace.

- The **Toolpaths** panel lists "Lathe Custom Thread" under **Toolpath Group-1**.
- The workspace shows the thread profile with blue toolpath lines and yellow rapid moves.

- The "THREAD DIA.", "PITCH" and "PROFILE" labels remain visible.

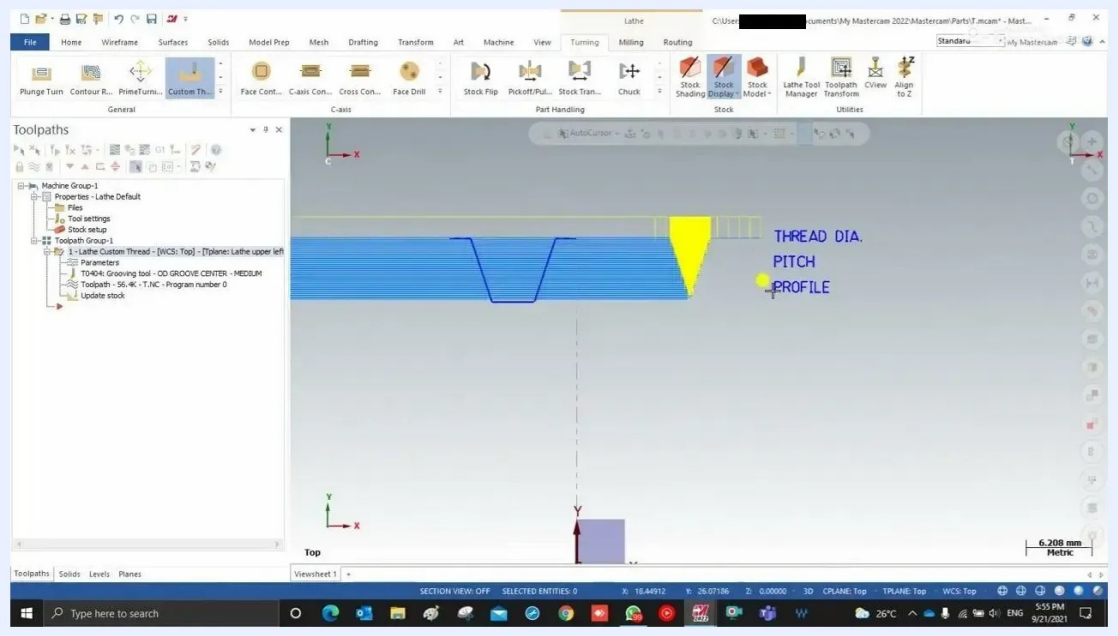


Threading toolpath generated, with the Toolpaths panel listing Lathe Custom Thread and the thread profile and parameters visible.

Step 36: Start the simulation

Select the new "Lathe Custom Thread" toolpath in the **Toolpaths** panel. Start the simulation by clicking the appropriate simulate or verify button.

A "Processing NCI data..." dialog appears and shows progress up to 100%. Wait for the simulation data to finish processing before you continue.



Processing NCI data dialog at 100% with the toolpath and workspace visible.

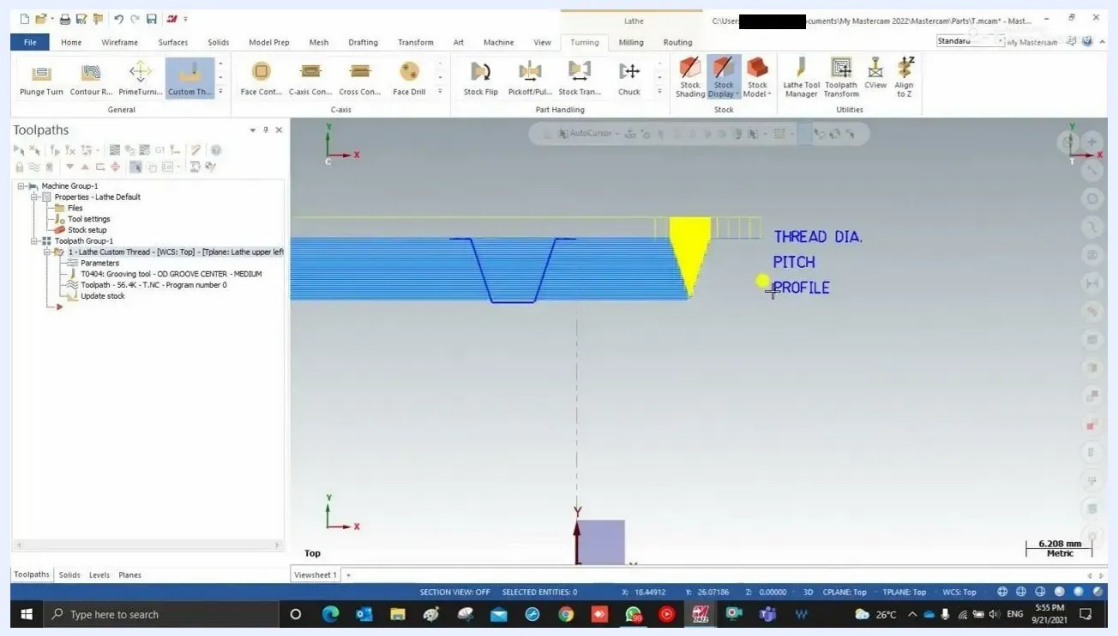
Step 37: Review the simulation results

Run the simulation for the custom thread program. It performs two operations:

- **Roughing operation:** The tool removes the bulk of the material along the thread path.
- **Finishing operation:** The tool refines the thread profile to the final required shape and size.

Watch the simulation to make sure the thread starts at Z0, the tool follows the correct profile and pitch, and the final thread matches the desired geometry.

The Mastercam Simulator window opens with a 3D view of the part. The tool appears above the part, and the thread path is highlighted in yellow on the cylindrical workpiece. The right panel shows move and toolpath information, including feed length, feed time, rapid time and total length. Use the playback controls at the bottom to review the simulation step by step or in real time.

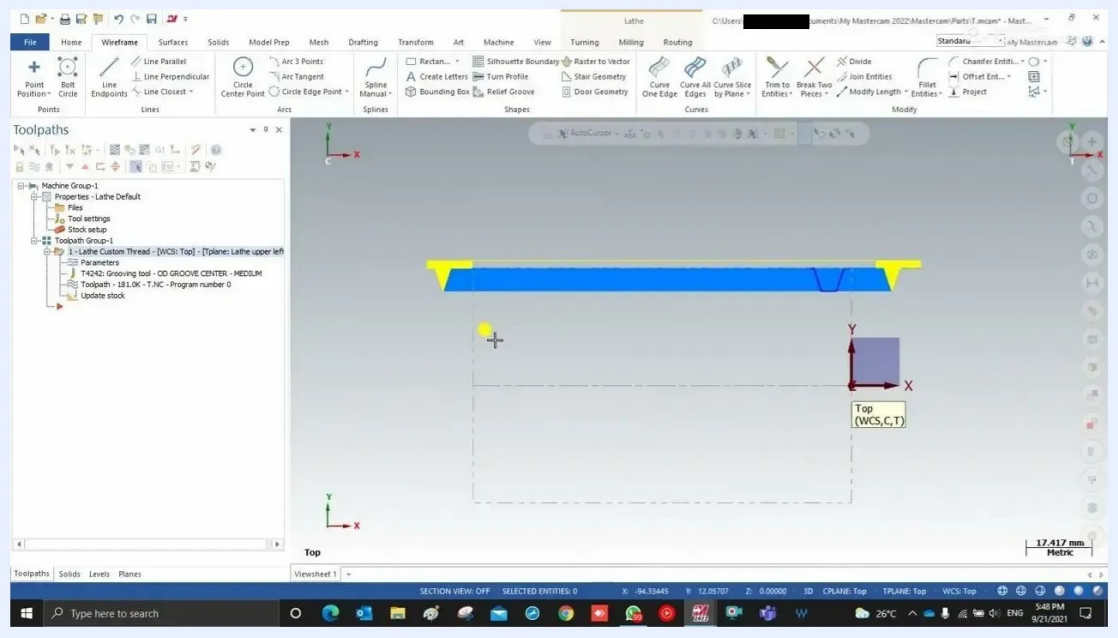


Mastercam Simulator window with a 3D part, the yellow thread path, the tool above the part and the toolpath information panel visible.

Step 38: Inspect the completed thread profile

Right-click in the simulation window to open the context menu and adjust the view, for example Fit, Top, Front, Right or Isometric. Rotate and zoom to inspect the thread from different angles. Confirm that the groove spacing and depth are consistent.

After the simulation, inspect the resulting thread in the graphics area. Confirm that the thread shape, size, and start and end positions are correct. Also confirm that the roughing and finishing operations produced the required profile safely and accurately.



Simulation window with the context menu open for view adjustment, showing the finished thread from an angled perspective.

Summary

You now know how to create a thread in Mastercam using the Custom Thread operation. You prepared a centered, verified profile and defined a raw material of OD 62.0 mm and length 100.0 mm. You then assigned the neutral grooving tool, chained the profile, set the pitch and configured the cutting parameters. Finally, you simulated the toolpath.

Key values used in this example:

- Thread pitch: **15.0 mm**
- Feed rate: **15.0 mm/rev**
- Spindle speed: **200**
- Stepover: **20% of tool width**
- Number of depth cuts: **4**
- Maximum first pass stepdown: **0.5**
- Cut direction: **Negative**

- Finishing pass: enabled or disabled as needed, with feed rate **15.0 mm/rev**, spindle speed **100**, stepover **1.0**, depth cuts **1**, cut depth **1.0**, and cut direction **Bi-Directional, Alternating**
- Overcut: **2.0 mm**
- Thread end position: **-105.0**
- Stock clearance: **2.0 mm**

These steps ensure all critical, advanced and finishing parameters for custom thread cutting are set accurately in Mastercam Lathe 2022.

What's next

The thread is now ready for final review or export for machining. Before you export, compare the simulated result with your drawing one more time. Make any final adjustments to the profile or parameters if needed.