How to open a SolidWorks file in Mastercam, position it, and set up the stock

This guide shows you how to open a SolidWorks file in Mastercam by merging the part into your workspace, moving its top surface to the origin, and setting up the machine and stock. The same steps work for other compatible file types, such as STEP and IGES. When you finish, the part and a wireframe stock outline appear in the workspace, ready for toolpath creation.

Prerequisites

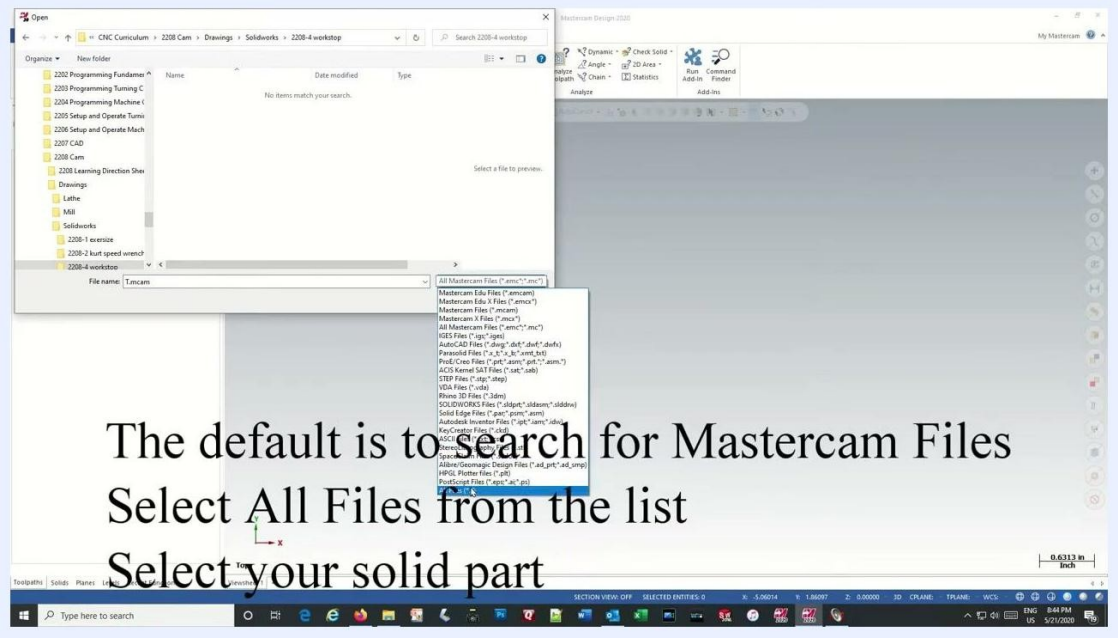
- A solid part file, for example a SolidWorks part file with the .SLDPRT extension.
- The folder location of that file.
- The stock dimensions and material for your job. This example uses a 2.0 x 2.0 x 6.2 stock in ALUMINUM inch - 6061.

Import and position the part

1. Open the Merge dialog

1. Go to the **File** menu.
2. Select **Merge**.

3. Browse to the folder that contains your part file.



Mastercam Design 2020 with the File menu open and Merge highlighted

2. Find and select your SolidWorks file

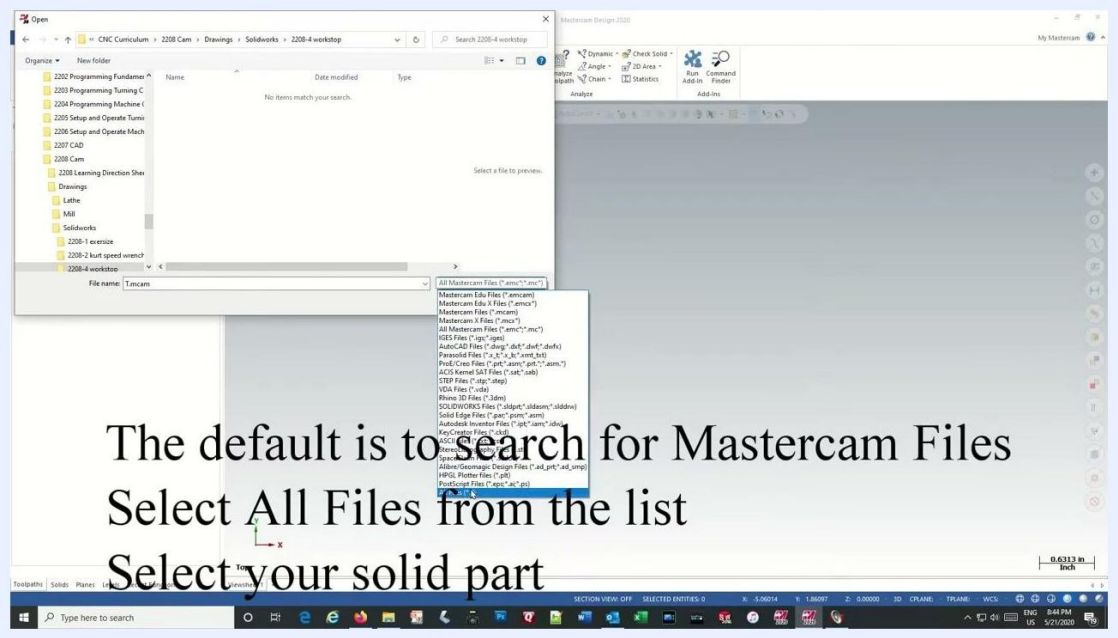
By default, Mastercam searches for Mastercam files only. To see your part, change the file type filter to **All Files (*)**. Then select your solid part file, for example a SolidWorks file with the .SLDPRT extension.

3. Choose the file type and open the part

1. If needed, use the dropdown menu to select the matching file type, such as SolidWorks, STEP, or IGES.
2. Click **Open** to import the part into Mastercam.

4. Check the placement of the imported part

Review where the part sits in the workspace. If you are satisfied with the placement, click **OK** to confirm. To see the origin, press **F9** to toggle the origin axes on and off.



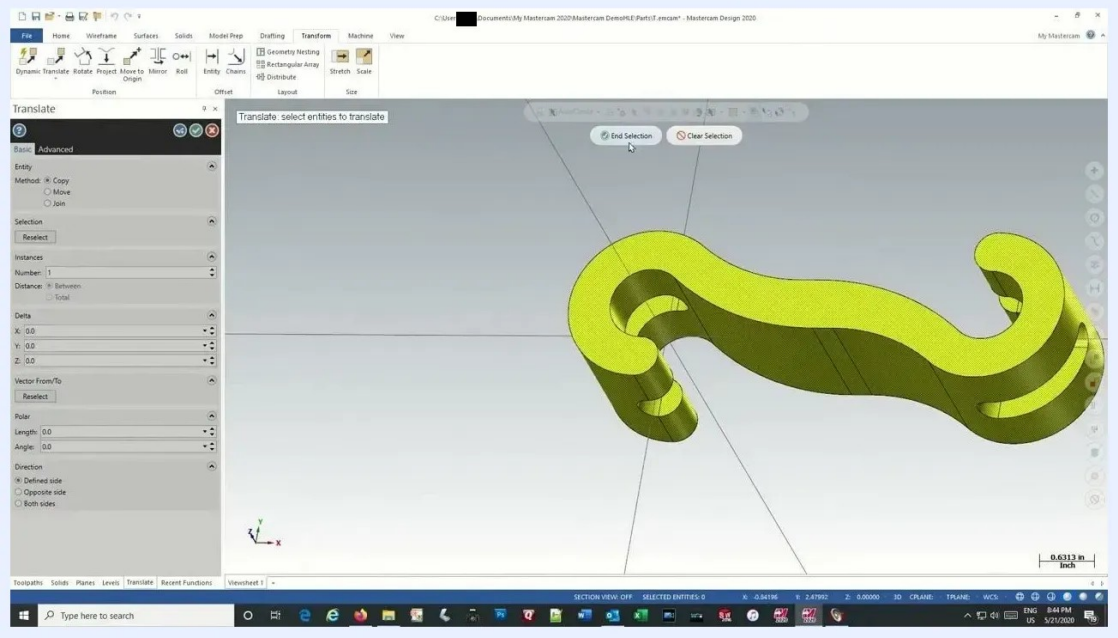
Imported part in the workspace with the origin axes displayed after pressing F9

5. Start translating the part to the origin

If the part's origin is not where you want it, use the Translate function to move it.

1. Go to the **Transform** tab and select **Translate**.
2. In the Translate dialog, make sure **Move** is selected, not **Copy**.

3. Select the body of the part and click **End Selection**.

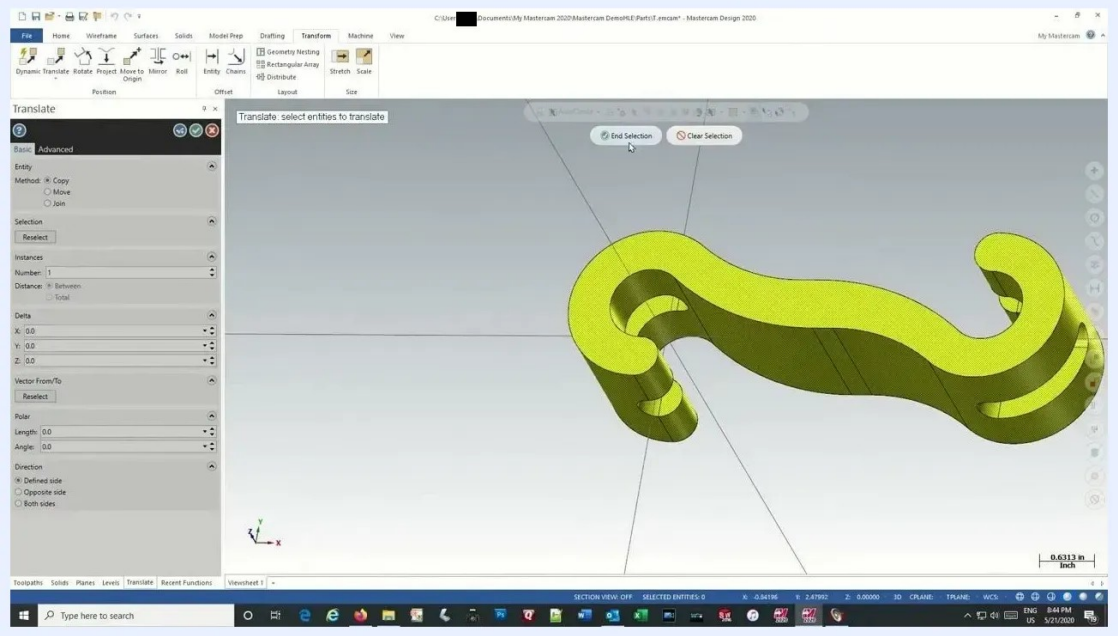


Translate dialog open with the part selected and the Move, Copy, and End Selection controls visible

6. Use the From/To selector to move the part

1. Use the **From** and **To** selector to specify the translation.
2. Select the top edge or surface of the part as the **From** point.
3. Set the **To** point at the origin, where the axes intersect.

4. Confirm the move.

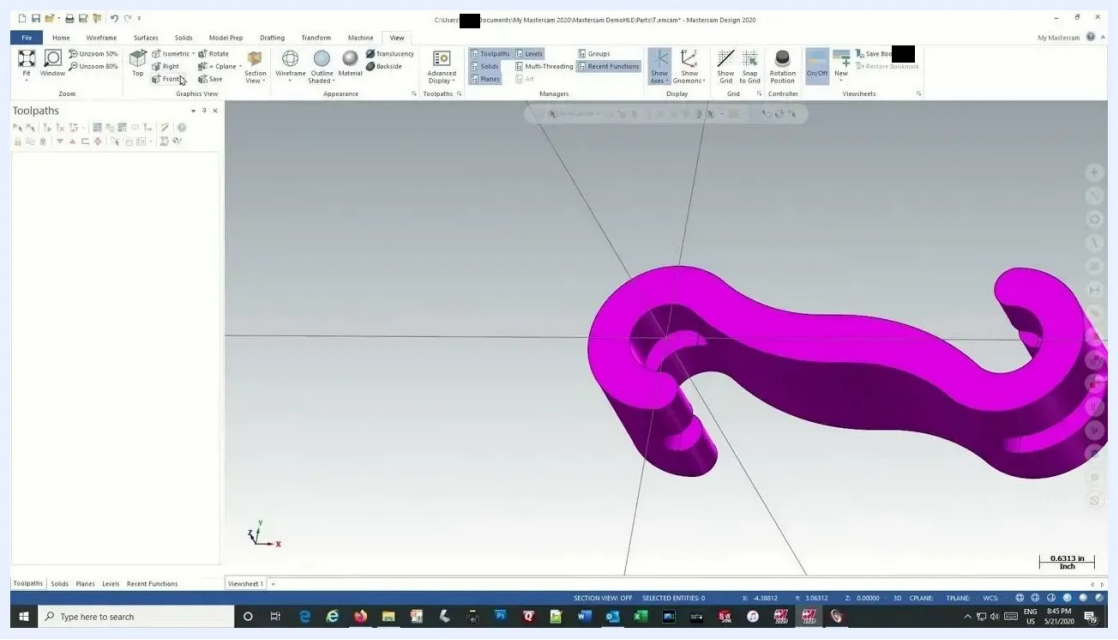


Translate dialog prompting you to select the point to translate from, a line or solid edge

Check that the part now sits with its top surface at the origin. If you are satisfied, click **OK** to finalize the placement.

7. Verify the origin in the front view

Switch to the front view to check the part's orientation. Confirm that the origin, where the axes intersect, is now on the top surface of the part.

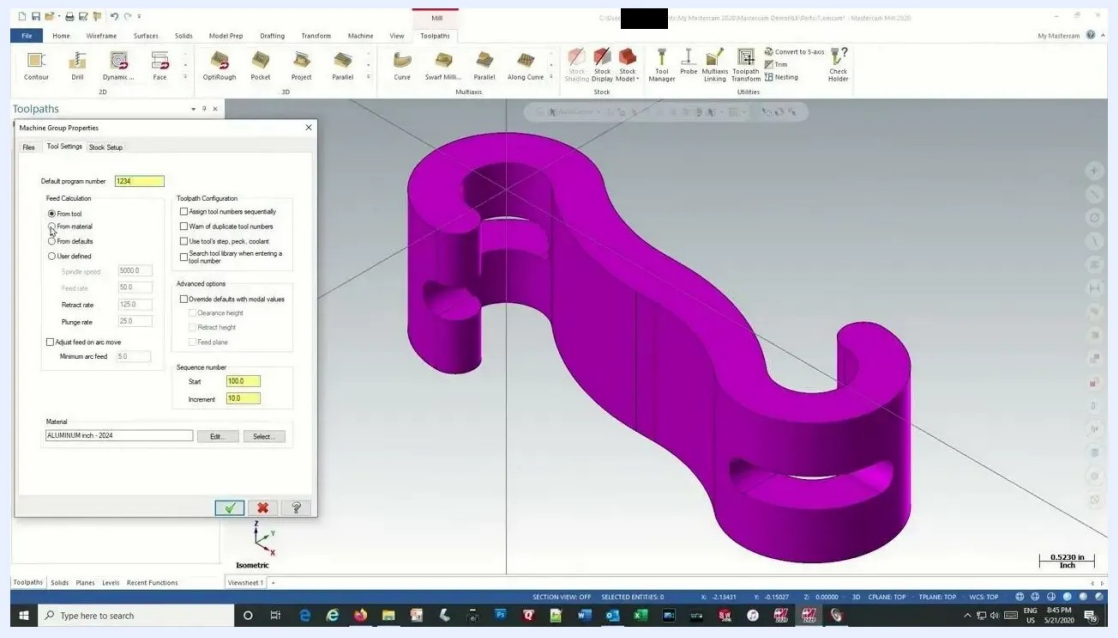


Front view of the part in magenta with the origin on its top surface and the axes visible

8. Switch to the isometric view and select the machine type

1. Click the **Isometric** button in the Graphics View section for a clear 3D view of the part.
2. Select the **Mill** machine type.

3. Choose the default mill configuration for this example.



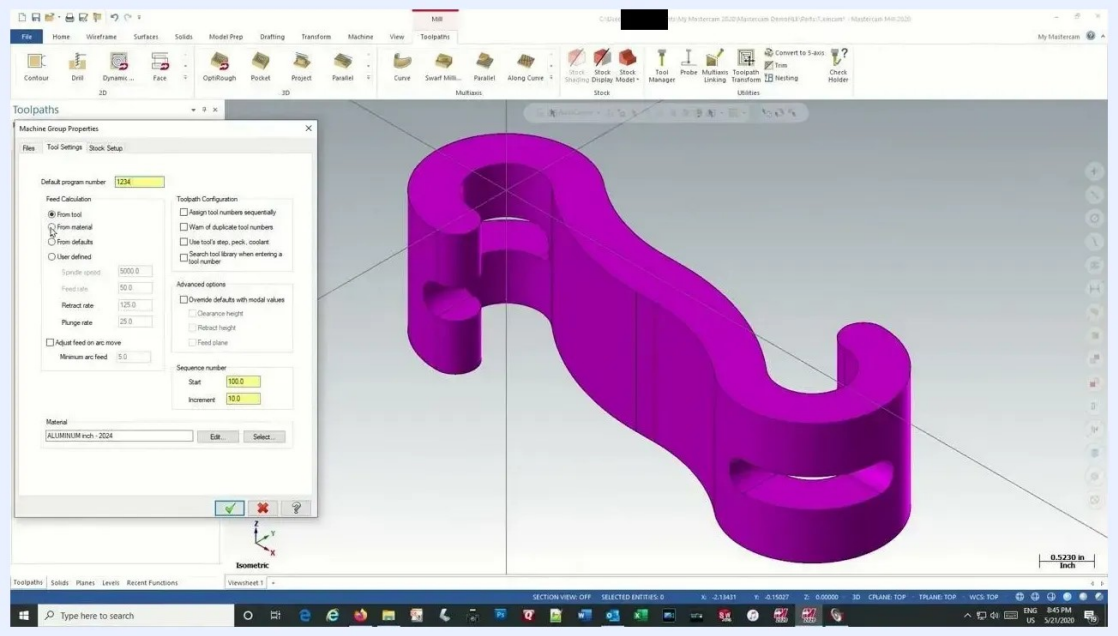
Isometric view of the part, ready for machine and stock setup

Set up the machine and stock

9. Open the tool settings

1. In the Toolpaths Manager in the left panel, expand **Machine Group 1**.

2. Double-click **Tool settings** to open the Machine Group Properties dialog.



Toolpaths Manager with Machine Group 1 expanded and Tool settings highlighted

10. Set the program number and feed calculation

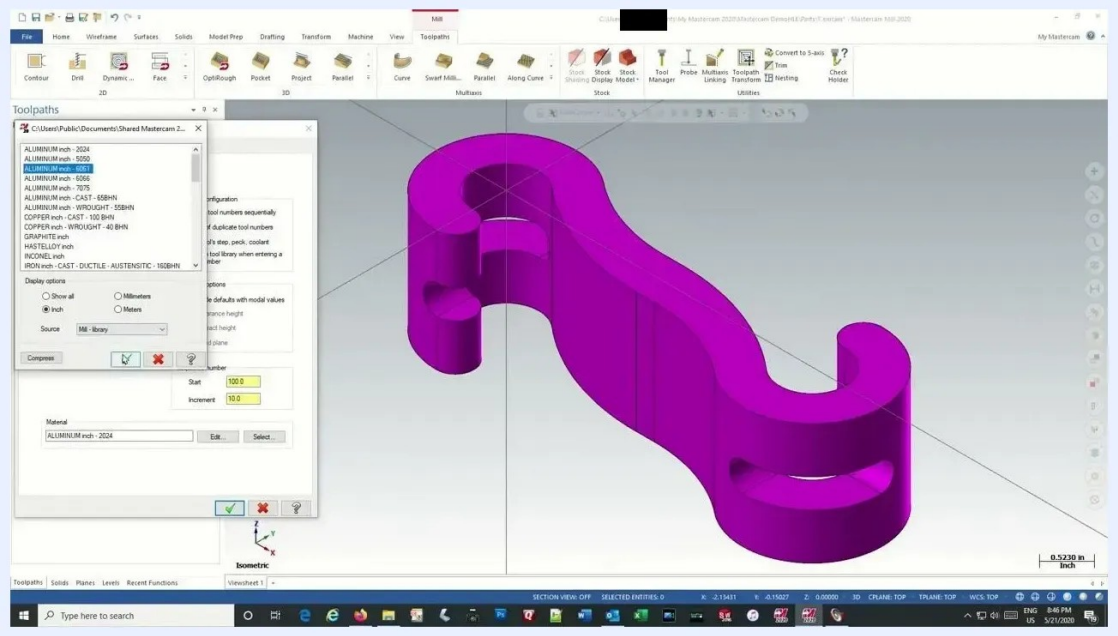
In the Machine Group Properties dialog:

- Enter the program number you want, for example 1234. If you leave it at 0, the program defaults to zero.
- Under **Feed Calculation**, select **From material**.
- Select the speed calculator based on the material.
- Set the minimum arc feed to 5.0.

For easier setup, make sure these options are checked:

- Assign tool numbers sequentially
- Warn of duplicate tool numbers
- Use tool's step, peck, coolant
- Search tool library when entering a tool number

- Override defaults with modal values (clearance height, retract height, feed plane)

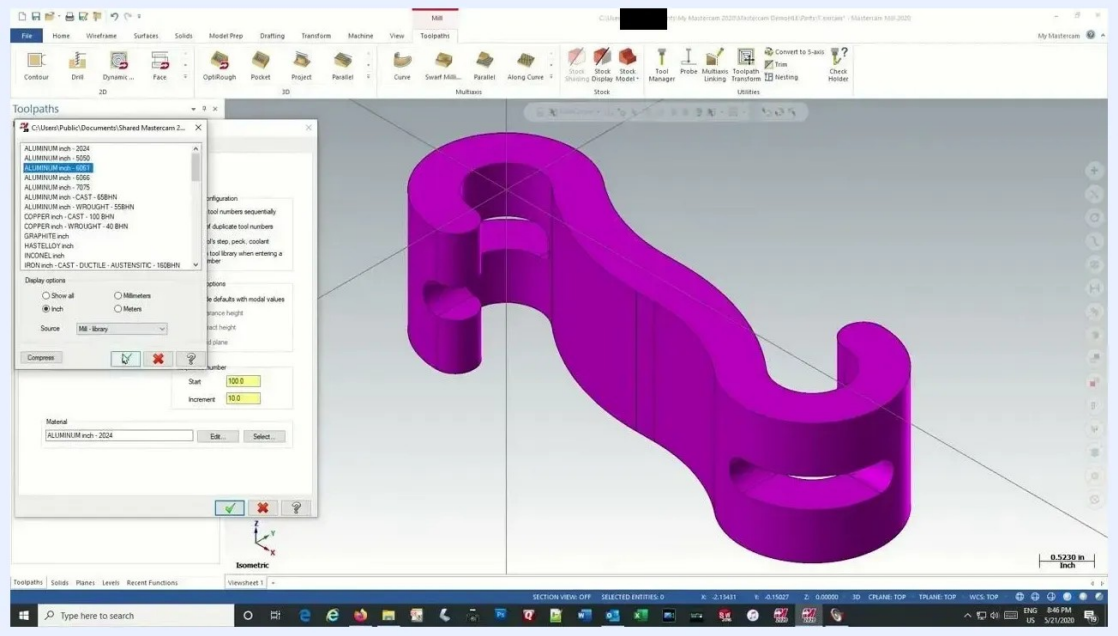


Machine Group Properties with program number 1234, Feed Calculation set to From material, and minimum arc feed 5.0

11. Select the material

1. Click the **Select** button next to the Material field.
2. In the material selection dialog, choose **ALUMINUM inch - 6061** from the Mill library.

3. Confirm your selection.

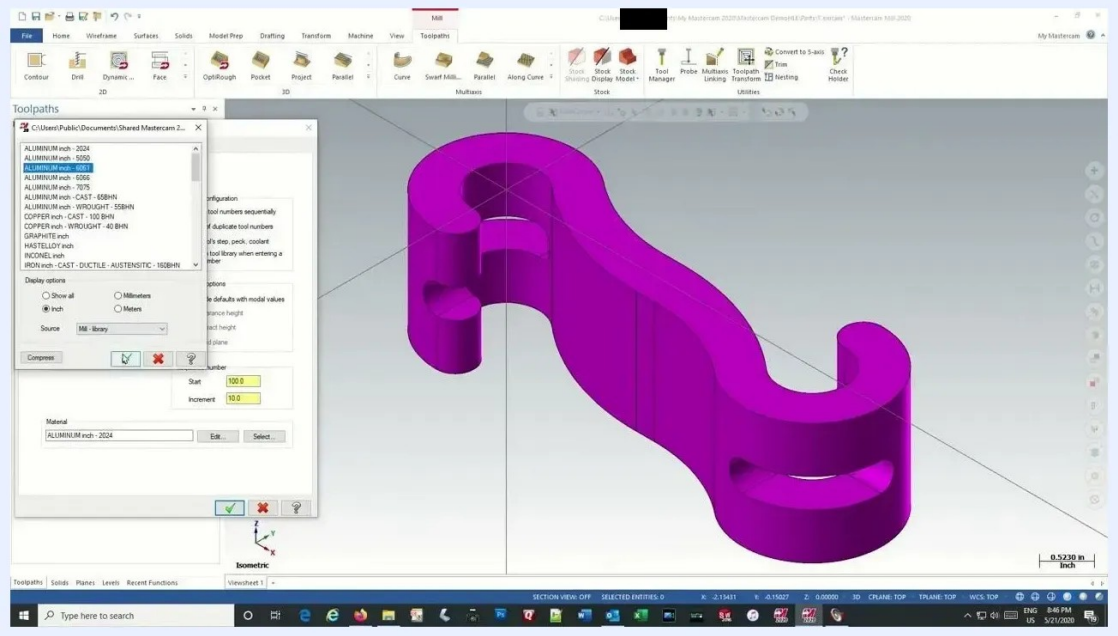


Material selection dialog with ALUMINUM inch - 6061 highlighted in the Mill library

12. Configure the stock

1. Go to the **Stock Setup** tab in the Machine Group Properties dialog.
2. Set the stock shape to **Rectangular**.
3. Click **All Solids** to use all solid bodies as the stock reference.
4. Make sure the **Display** checkbox is checked so the stock is visible.
5. Set the stock dimensions to X: 2.0, Y: 2.0, and Z: 6.2.

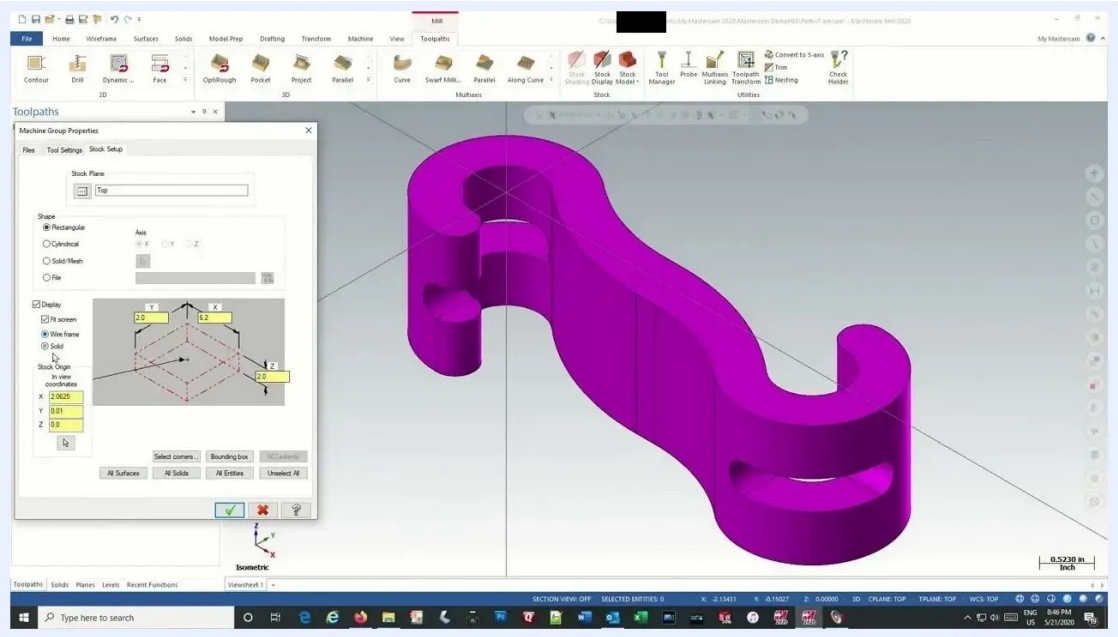
6. Set the stock origin X offset to 0.625.



Stock Setup tab with rectangular stock, dimensions entered, and display enabled

13. Enter the machining allowance in the Z field

Enter 0.01 in the Z origin offset field. This leaves a machining allowance on the top face. If you entered 0.01 in the Y field by mistake, move it to the Z field.

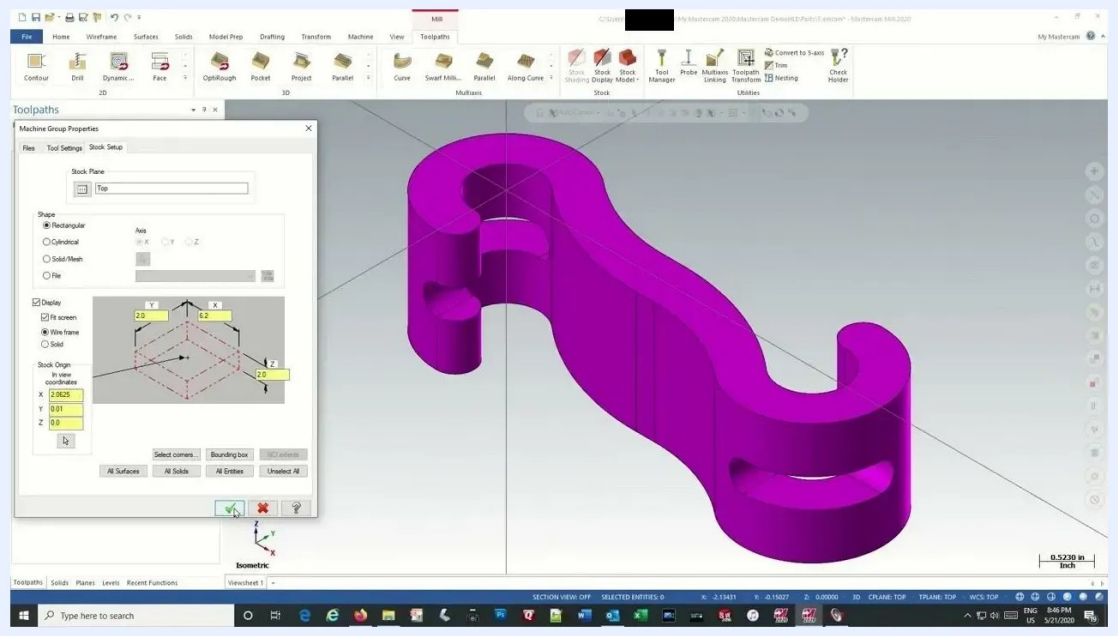


Stock Setup tab with the 0.01 machining allowance entered in the Z offset field

14. Display the stock as a wireframe

1. In the **Stock Setup** tab, select **Wire frame** under the Display section.
2. Confirm that the stock dimensions are still X: 2.0, Y: 2.0, and Z: 6.2.
3. Confirm that the 0.625 X offset and the 0.01 Z machining allowance are still set.
4. Click the green checkmark to apply the settings and close the dialog.

Wireframe mode shows the stock as an outline around your part, so the part geometry stays visible. It also helps you verify clearances and check that the stock fully encloses the part.



Stock Setup tab with Wire frame selected and the magenta part visible in the workspace

Summary and what's next

You have opened a SolidWorks part in Mastercam, moved its top surface to the origin, and set up the machine group, material, and stock. The part and a wireframe stock outline are now visible in the workspace. Next, you can create toolpaths and run a machining simulation.