How to Change Units in Creo Parametric

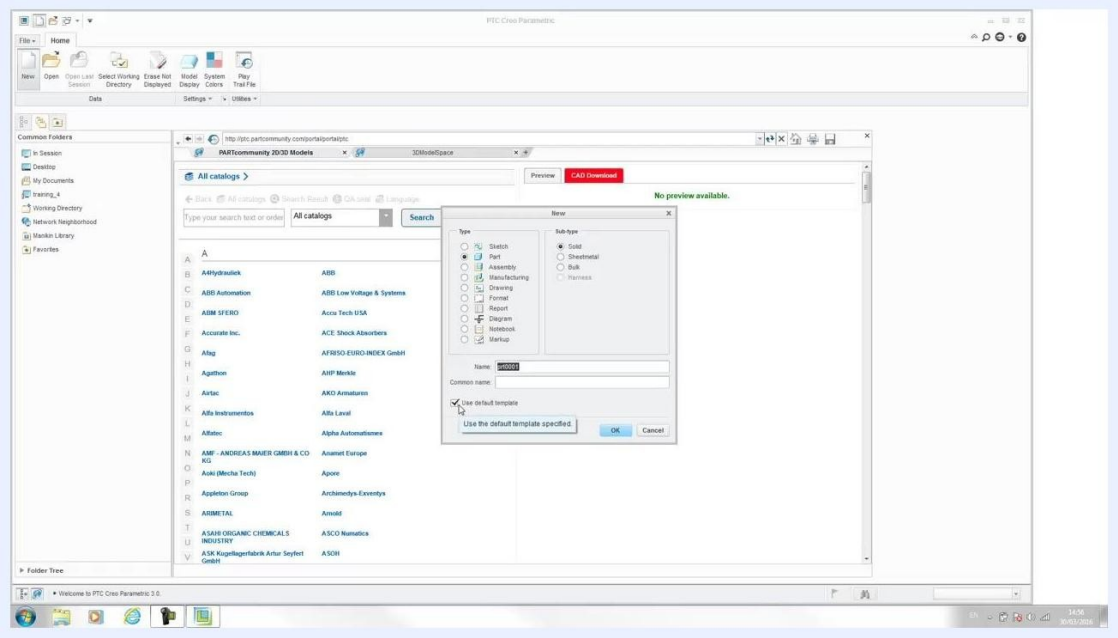
Creo Parametric uses imperial units (inches, pounds, seconds) by default for new parts and assemblies. This guide explains how to change units in Creo Parametric by selecting a metric template for a single part, and how to permanently change the default units for all new files by updating the configuration settings to use millimeter-kilogram-second (mmKs) templates.

Prerequisites

- Creo Parametric installed on your machine.
- Access to the Creo installation directory (the exact path may vary depending on where you installed Creo; adjust navigation steps if your installation folder differs).
- Administrator or write access to update and save the config.pro configuration file.

Understand the default template behavior

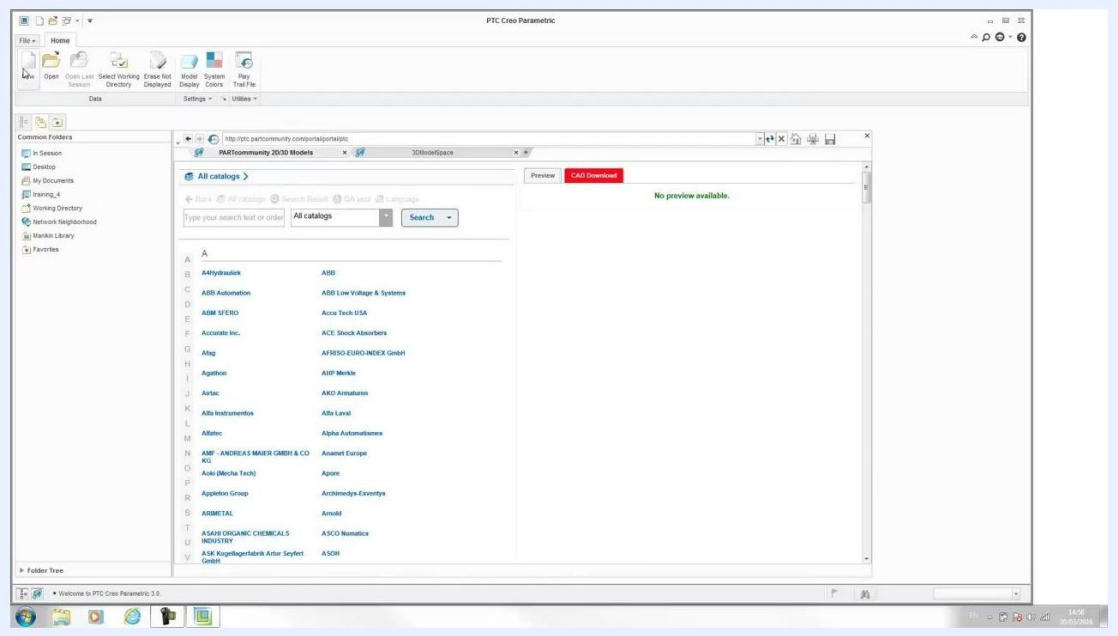
By default, the **Use default template** checkbox is checked when you create a new part, which means Creo applies the default imperial template automatically. The units assigned to a new part or assembly are determined entirely by the template selected at creation time.



New Part dialog with the "Use default template" option checked

Step 1: Start Creo Parametric

Launch Creo Parametric from your desktop or start menu and wait for the application to fully load before proceeding.



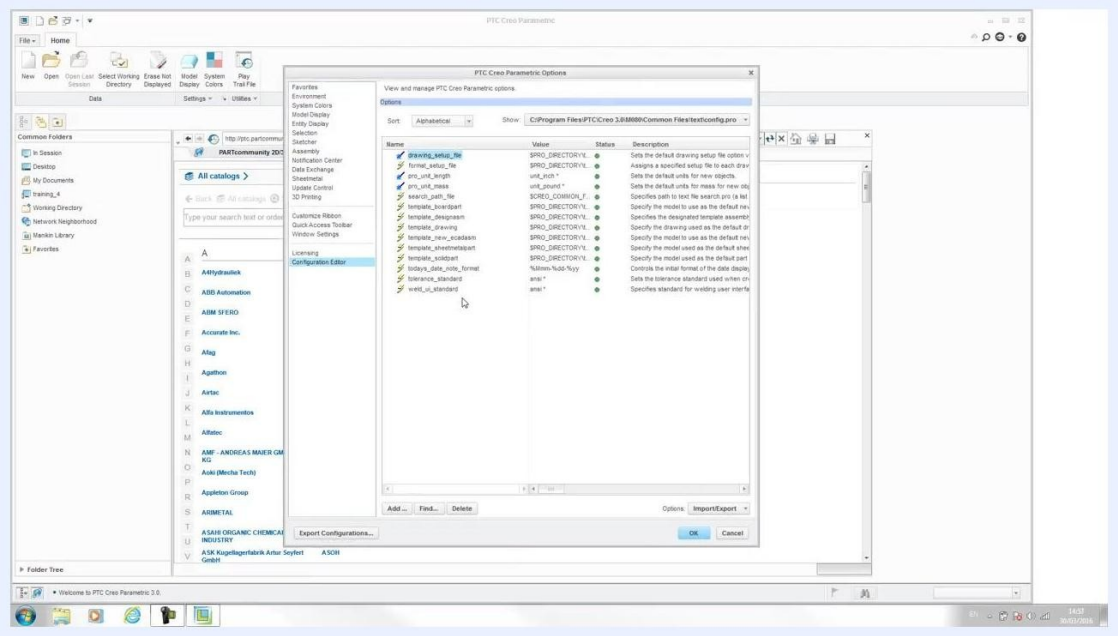
Creo Parametric desktop shortcut visible on the Windows desktop

Step 2: Choose a different template for a single part (optional)

If you only need to change units for one part rather than every new file, you can select a different template manually:

- Click the **File** menu and select **New**.
- In the **New** dialog, choose **Part** as the type and **Solid** as the sub-type.
- Uncheck **Use default template**, then click **OK** to open the template selection dialog.

- In the **New File Options** dialog, select a metric template, such as mmns_part_solid (millimeters, newtons, seconds).



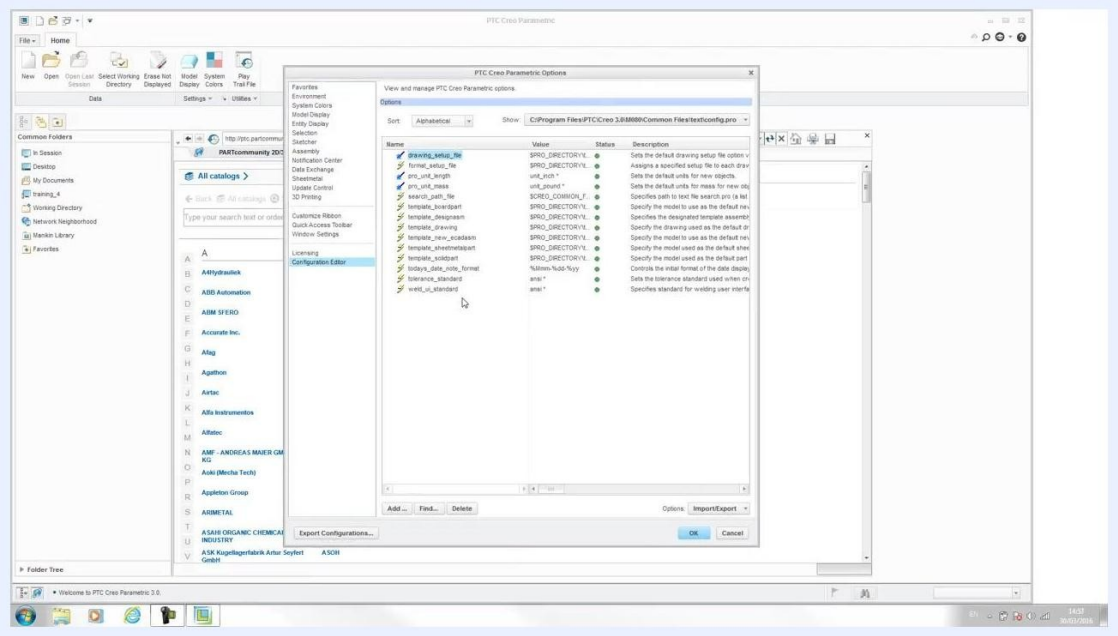
Template selection dialog showing both imperial and metric template options

Step 3: Open the Configuration Editor to change the default template

To make metric units the default for all new parts, update the configuration settings instead of selecting a template manually each time:

- Go to **File** > **Options** > **Configuration Editor**.
- Locate the template setting named template_solidpart.

- You will change this value to the desired metric template, such as mmns_part_solid.



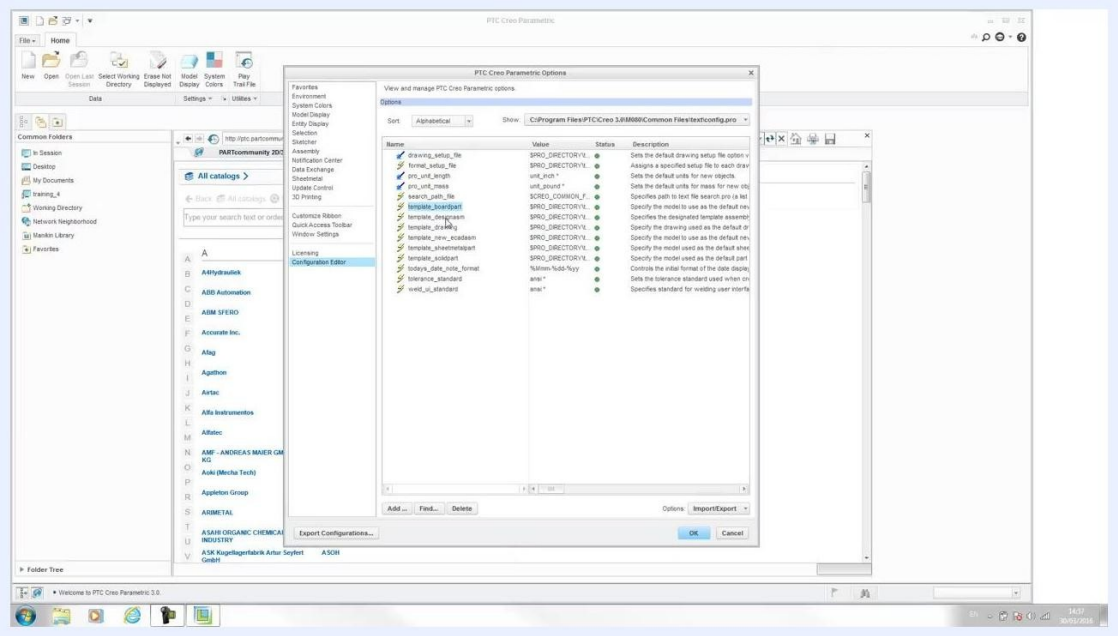
Configuration Editor showing template settings and their current values

Step 4: Review the available template settings

On a fresh installation with a new configuration file, the Configuration Editor lists several template settings, each pointing to a file path on your local machine:

- template_boardpart
- template_designasm
- template_drawing
- template_new_ecad.asm
- template_sheetmetalpart
- template_solidpart

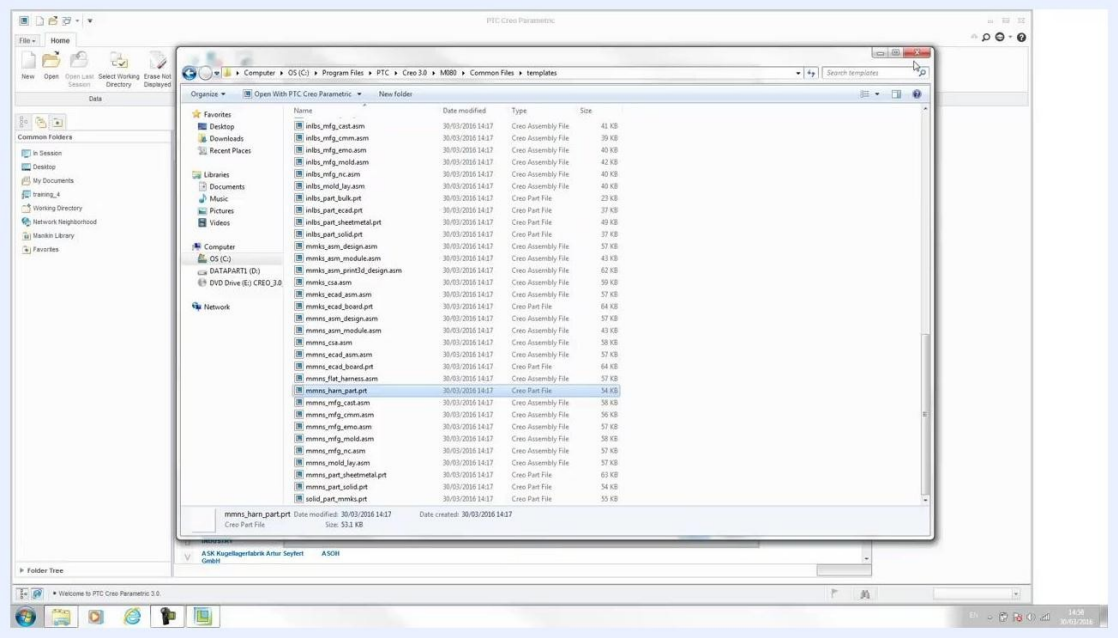
These templates are copied into new parts or assemblies when you create them; they are not modified directly by editing this list.



PTC Creo Parametric Options dialog showing template settings and their file paths

Step 5: Locate the templates folder

Open the file selection dialog and navigate toward your Creo installation directory to find the templates folder. The exact path depends on where Creo was installed on your system, so adjust the navigation if your installation location differs.



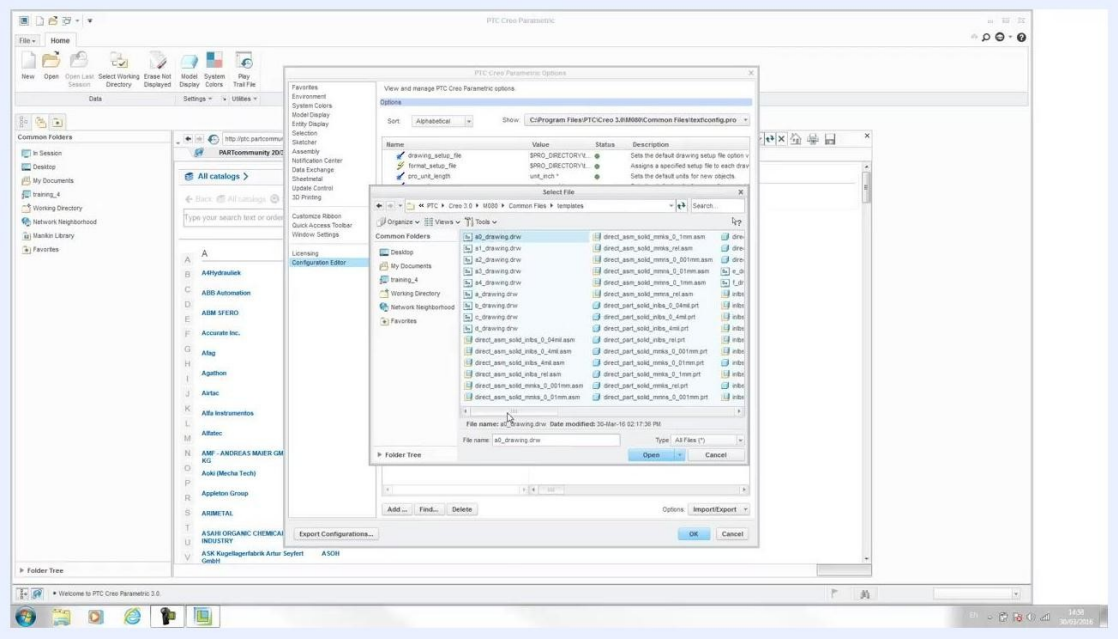
File dialog open to Program Files, ready to select the PTC folder

Step 6: Set the solid part template to MMKS

Within the templates folder of your Creo installation directory, locate and select the file named solid_part_mmks.prt. This template uses millimeters, kilograms, and seconds as the default units.

- Click **Open** to confirm your selection.

- The template_solidpart setting in the Configuration Editor now shows the new file path, with an asterisk (*) indicating the change is pending save.



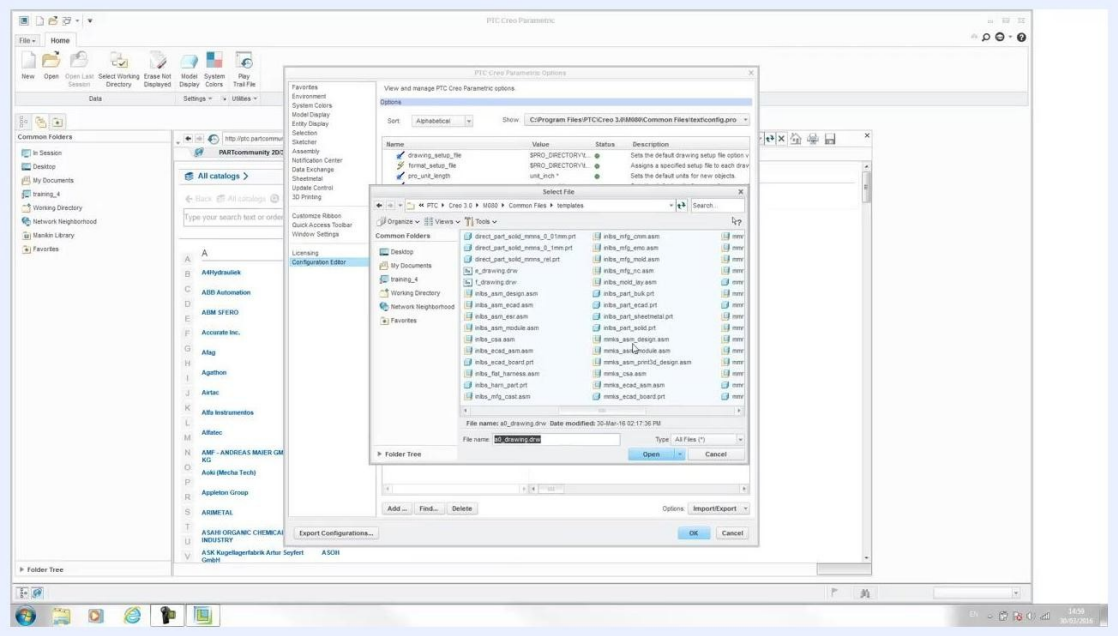
File dialog open to the templates folder, with "solid_part_mmks.prt" visible among the other template files

Step 7: Set the assembly template to MMKS

Repeat the process for assemblies:

- In the Configuration Editor, locate the template_designasm setting.
- Click **Browse** next to the template path.
- In the file dialog, select mmks_asm_design.asm, the assembly template using millimeters, kilograms, and seconds.
- Click **Open** to confirm your selection.

Creo remembers the last template folder you used, which makes it easier to locate the correct file for the assembly template.



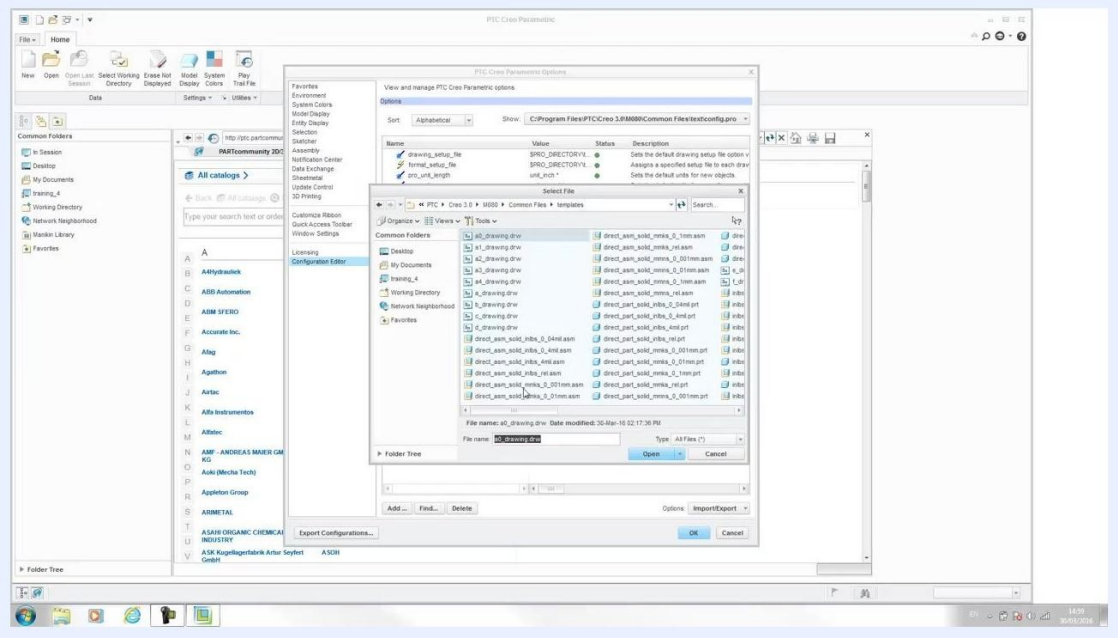
File dialog open to the templates folder, with "mmks_asm_design.asm" visible and ready for selection

Step 8: Save the updated configuration

To apply your changes permanently:

- In the Configuration Editor, click the **Import/Export** dropdown at the bottom right.
- Select **Export per current filter** to save your changes.
- When prompted, overwrite the existing config.pro file to apply the new template settings.

- Click **OK** to close the Configuration Editor.



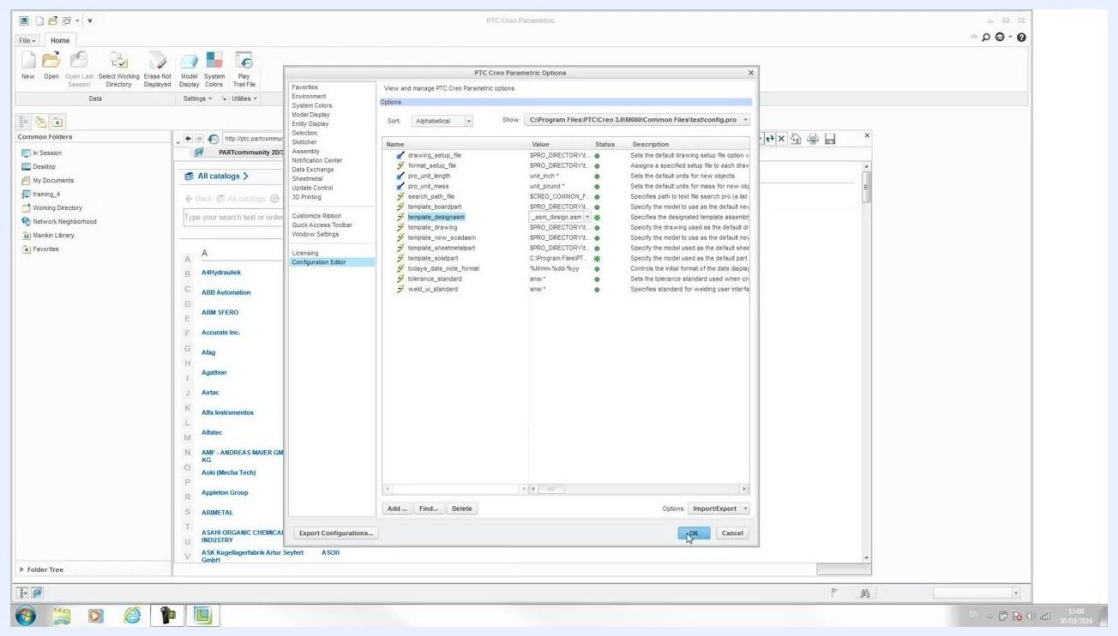
Configuration Editor with the Import/Export dropdown open, ready to export the configuration

Step 9: Verify the new default units in a new part

Confirm that the configuration change took effect:

- Create a new part in Creo by selecting **New Part** and using the default part option.
- Go to **File** > **Prepare** > **Model Properties**.

- Confirm that the units are now set to millimeter, kilogram, second (mmKs) instead of inches and pounds.



Model Properties dialog open, showing units set to millimeter kilogram second (mmKs)

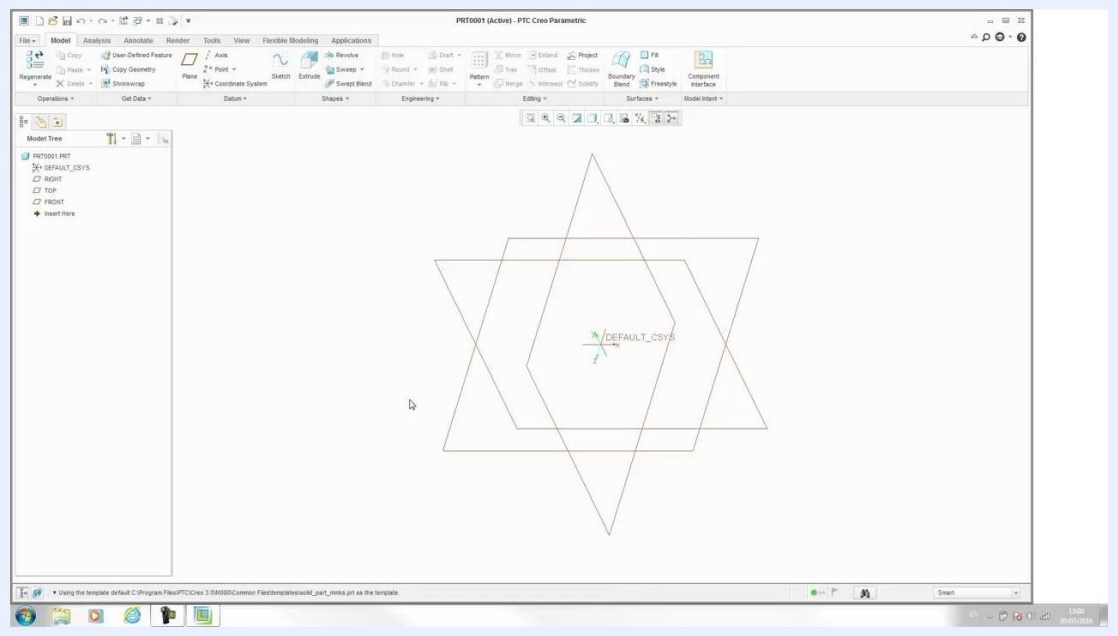
Step 10: Confirm the new template is active in the workspace

In the main modeling window, check that the new part file (PRT0001.PRT) displays the default coordinate system (DEFAULT_CSYS) and standard reference planes (RIGHT, TOP, FRONT) in the Model Tree on the left. The geometric center and reference planes should be visible in the main view area, confirming successful template application.

Check the status bar at the bottom of the window for a confirmation message similar to:

Using the template default C:\Program Files\PTC\Creo 3.0\M080\Common Files\templates\solid_part_mmks.prt as the template.

This confirms that the interface is ready for further modeling and that all previous configuration steps were applied correctly.



Creo Parametric workspace showing a new part with the MMKS template active, Model Tree on the left, reference planes and coordinate system visible, and status bar confirming template usage

Summary

You have now set up Creo Parametric to use metric (mmKs) templates for new parts and assemblies by updating the template_solidpart and template_designasm settings in the Configuration Editor and saving them to config.pro. All new models created going forward will start with the correct millimeter-kilogram-second units and standard reference geometry already in place.