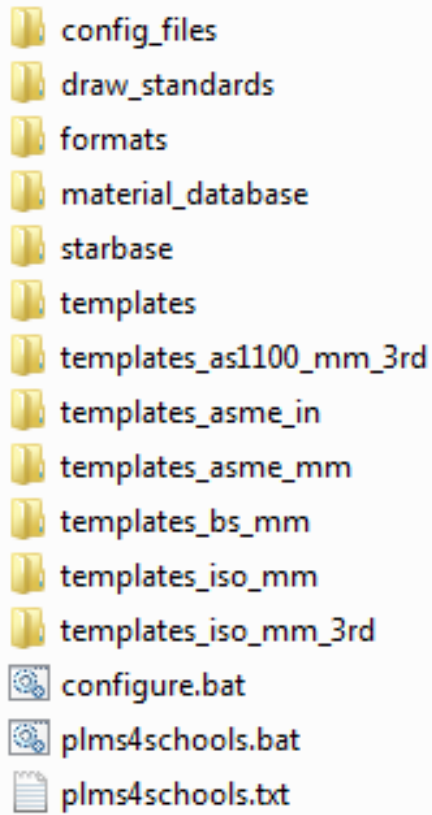


Configuring Creo Parametric 2.0

PTC Academic Program Configuration Files



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Module 1 – The Creo_Standards Folder

To help configure your installation of Creo Parametric, the PTC Academic Program provides a special folder named **creo_standards**.

If you install Creo 2.0 in the default location, the **creo_standards** folder will be located in:

C:\Program Files\PTC\Creo 2.0\Common Files\M050

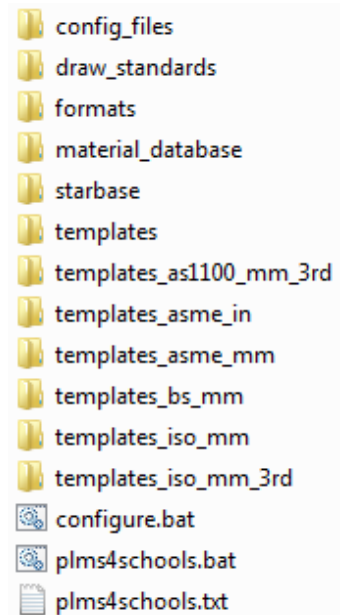


The folder M050 is the *build_code* folder. Its name will change with each new build code you install. Production versions are named using the **M** prefix and a three digit number such as **030**. Pre-production build codes use an F prefix followed by a three digit number starting with **000**.

Inside the **creo_standards** folder, are a set of files and folders that are used to configure Creo Parametric.

You can run the batch file named **configure.bat** to quickly and automatically configure Creo Parametric to use selected drawing standards and unit systems.

The **plsm4schools.bat** file is used to point the embedded Learning Connector away from the commercial PLMS (Precision Learning Management System) server and to the Schools PLMS server.



Module 2 – Enable Full Control of Common Files

To automatically configure your installation of Creo Parametric you must have full control of the **C:\Program Files\PTC\Creo 2.0\Common Files** folder and all of its subfolders.

Full control is required so that the batch file can move and copy files within the **Common Files** folder.

Step 1: Ensure full control of the Common Files folder

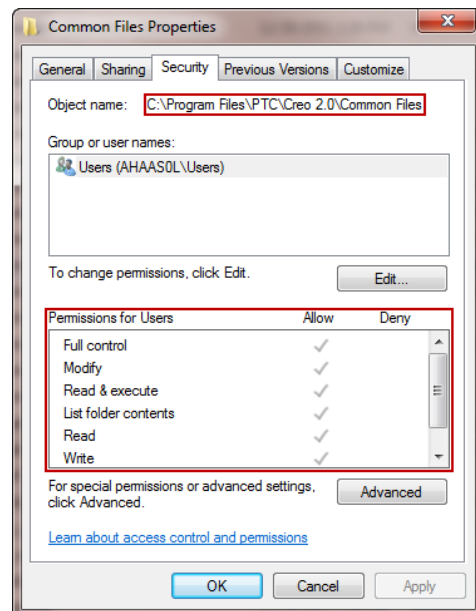


To change access permission on the **Common Files** folder, you will most likely need administrator access to your computer.

If you are an administrator setting the computer up for other users, you do not have to open up permissions for all users, only yourself.

The procedure for changing permissions on a folder will vary depending on the operating system you are using. The following instructions are based on the Windows 7 operating system.

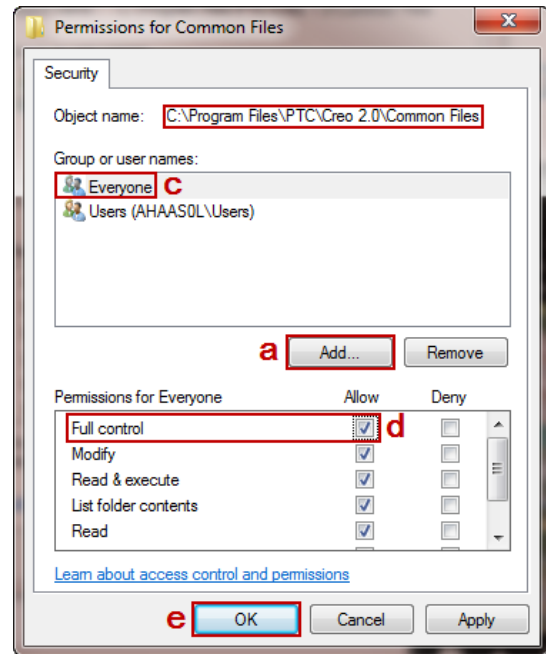
1. In Windows Explorer, browse to the **C:\Program Files\PTC\Creo 2.0** folder.
2. Right-click the **Common Files** folder and select **Properties** from the pop-up menu.
3. Open the **Security** tab and click to select your user name.
4. If **Allow** is checked for **Full control**, then you have full control over this folder and you can continue to **Module 3**.
5. If **Allow** is not checked for **Full control**, you will need to change permissions so that you have Full control of the Common Files folder and all of its subfolders.



Give **Full control** of the Common Files folder to everyone:

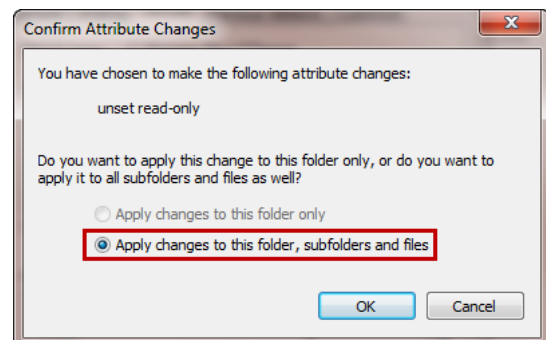
6. Right-click the **Common Files** folder and select **Properties** from the pop-up menu.
7. In the Common Files Properties dialog box:
 - In the **General** tab, clear the **Read-only** check box.
 - Click to open the **Security** tab and click **Edit**.

8. In the Permissions for Common Files dialog box:
 - a. Click **Add**.
 - b. In the new dialog box, type the word **Everyone** and then click **OK**.
 - c. Select **Everyone** from the **Group or user names** list.
 - d. Enable the check box to Allow **Full control** of the folder.
 - e. Click **OK** to close the dialog box.



9. Click **OK** from the Common Files Properties dialog box.
10. If necessary, enable the **Apply changes to this folder, subfolders and files** and click **OK**.

The new access permissions will now be applied to the Common Files folder and all of its contents.



Module 3 - Configuring Your Installation

The batch file named **configure.bat** contained in the **creo_standards** folder enables you to easily configure your installation of Creo Parametric to use different drawing standards and unit systems.

Step 1: Run the **configure.bat** file

1. If necessary, exit Creo Parametric.
2. In Windows Explorer, browse to the folder **C:\Program Files\PTC\Creo 2.0\M050\creo_standards**.



The folder M050 is the *build_code* folder. Its name will change with each new build code you install. Production versions are named using the **M** prefix and a three digit number such as **030**.

Pre-production build codes use an F prefix followed by a three digit number starting with **000**.

3. Double-click **configure.bat** and then select the drawing standard and unit system you want to use by entering the number that coincides with that standard:

1 - ASME drawings with INLBS unit system
2 - ASME drawings with MMKS unit system
3 - AS1100 drawings, 3rd Angle Projection, MMKS unit system
4 - BS8888 drawings and an MMKS unit system
5 - ISO drawings and an MMKS unit system (<i>1st Angle Projection</i>)
6 - ISO drawings and an MMKS unit system (<i>3rd Angle Projection</i>)
7 - Default ISO drawings and MMNS unit system. <i>Use this configuration when you are taking a PTC University training course.</i>
8 - Default Creo ASME drawings and INLBS unit system.

4. Restart Creo Parametric to verify that your default unit system and drawing standard has changed.
 - Create a new part and then click **File> Prepare> Model Properties** to check its units.
 - Create a new drawing using one of the templates, then zoom into the lower-right corner of the drawing format. A small note will show the standard and model units used in the creation of the drawing.

Module 4 – Setup Learning Connector

The ***plms4schools.bat*** file is used to point the embedded Learning Connector away from the commercial PLMS (Precision Learning Management System) server and to the Schools PLMS server.

Step 1: Run the ***plms4schools.bat*** file

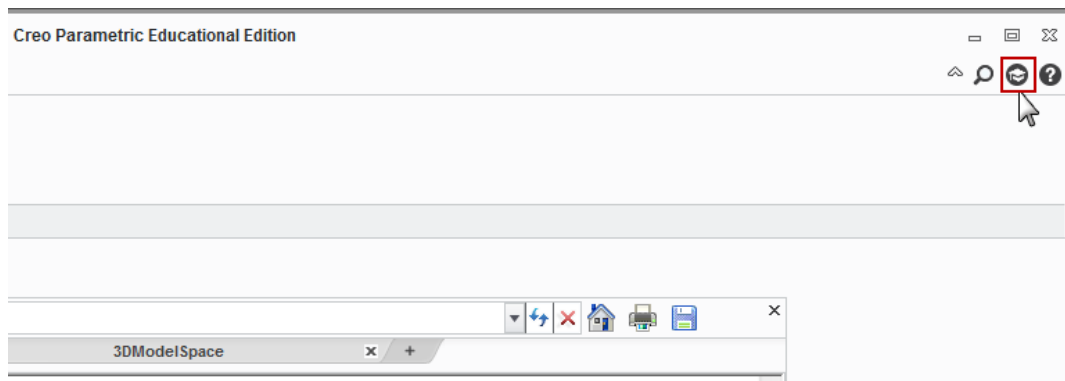
1. If necessary, exit Creo Parametric.
2. In Windows Explorer, browse to the folder ***C:\Program Files\PTC\Creo 2.0\Common Files\M050\creo_standards***.



The folder M050 is the *build_code* folder. Its name will change with each new build code you install. Production versions are named using the **M** prefix and a three digit number such as **030**.

Pre-production build codes use an F prefix followed by a three digit number starting with **000**.

3. Double-click ***plms4schools.bat***.
4. Restart Creo Parametric. Now when you start the **Learning Connector** from the upper-right corner of the Creo window, it will point to the Schools PLMS server.



To successfully run the *plms4schools.bat* file, write access is required on the folder ***C:\Program Files\PTC\Creo 2.0\Parametric\bin*** and the files contained in it.

To manually point your installation to the Schools PLMS server, add the following line of text to the end of the ***parametric.psf*** file:

```
ENV=CONNECTOR_PLMS_BASE_URL=https://plms4schools.ptc.com/precisionlms/
```

For access to Schools PLMS content, register at www.ptc.com/go/plms4schools.