Exercise 1 - Working with Files

Goal

The goal of this exercise is to become familiar with the File commands.

Task 1: Start Creo Parametric and change the current working directory

- Double click the appropriate icon on the desktop to start Creo Parametric
- Click Select Working Directory in the Home tab of the ribbon
- Select 'basic' then click OK in the dialog box
- Read the message in the message area at the bottom of the Creo Parametric window
- The working directory is now set to the 'basic' folder

Task 2: Open and change a part model.

- Click File, Open or click the icon
- Select 'lab1-1.prt' then click Open or click the middle mouse button
- Select the hole in the center of the part (select the cylindrical surface inside the hole)
- Press and hold the right mouse button and click Delete in the popup menu
- Click OK to delete the feature

Task 3: Save the part.

- Click File, Save or click the icon
- Click OK in the dialog box or click the middle mouse button
- Read the message in the message area
Task 4: Change the current working directory again.

- Click File, Manage Session, Select Working Directory
- Select 'labs' then click OK in the dialog box
- Read the message in the message area
- The current part was opened from the 'basic' folder
- The working directory is now set to the 'labs' folder

Task 5: Change the part and save it again.

- Select the other hole (select the cylindrical surface inside the hole)
- Press and hold the right mouse button and click Delete in the popup menu
- Click OK to delete the feature
- Click File, Save or click the icon
- Click OK in the dialog box or click the middle mouse button
- Read the message in the message area

Note

The part is saved in the 'basic' folder, not the 'labs' folder.

Task 6: Erase the part from the RAM.

- Click File, Manage Session, Erase Current
- Click Yes in the Erase Confirm dialog box
- Read the message in the message area
Task 7: Open another part, save, and erase from session.

- Click File, Open or click the icon ⭕
- Select 'lab1-2.prt' then click Open or click the middle mouse button
- Select one of the holes (select the cylindrical surface inside the hole)
- Press and hold the right mouse button and click Delete in the popup menu
- Click OK to delete the feature
- Click File, Save or click the icon ⭕
- Click OK in the dialog box or click the middle mouse button
- Click File, Manage Session, Erase Current then click Yes in the dialog box
- Read the messages in the message area

Task 8: Verify the part was saved and a new version was created.

- Click File, Open or click the icon ⭕
- In the File Open dialog box, click Tools, All Versions
- The dialog box lists all versions of the object, notice the part called 'lab1-2.prt' has two versions on the list
- Click Cancel

Task 9: Open another part model.

- Click File, Open or click the icon ⭕
- Select 'lab1-3.prt' then click Open or click the middle mouse button

Task 10: Change the current working directory again.

- Click File, Manage Session, Set Working Directory
- Select 'backup_dir' then click OK or click the middle mouse button
Task 11: Backup the part to the backup folder.

- Click File, Save As, Save a Backup, OK
- Read the message in the message area
- The active object is the backed up part, not the original part
- Click File, Close or click the icon 📄

Task 12: Rename an object.

- Click File, Open or click the icon 📄
- Select 'lab1-4.prt' then click Open or click the middle mouse button
- Click File, Manage File, Rename
- Enter < lab1-5 > for the new name (do not include the < > marks)
- Press 'Enter' or click OK in the dialog box
- Click File, Close or click the icon 📄

Task 13: Use the 'Save a Copy' command.

- Click File, Open or click the icon 📄
- Select 'lab1-6.prt' then click Open or click the middle mouse button
- Click File, Save As, Save a Copy
- Enter < lab1-7 > for the new name then click OK
- Read the message in the message area
- The active object is the old part, not the new copy
- Click File, Close or click the icon 📄
- Click File, Open or click the icon 📄
- Select 'lab1-7.prt' then click Open or click the middle mouse button
- Notice this part is an exact copy of the other part
- Click File, Close or click the icon 📄
Task 14: Create a new part.

- Click **File, New** or click the icon 
- Enter `<lab1-8>` for the name then click **OK**
- Read the message in the message area and:
  - Notice the part was copied from a 'template' part
  - Be sure the template is called 'template_mm.prt'
  - If the template is not called 'template_mm.prt', add the configuration options listed on page 4
- Click **File, Save** or click the icon 
- Click **OK** or click the middle mouse button
- Read the message in the message area
- Click **File, Close** or click the icon 
- Click **File, Manage Session, Erase Not Displayed** then click **OK** in the dialog box
- Read the message in the message area

Task 15: Exit the session.

- Click **File, Exit**
- Click **Yes** in the dialog box to terminate the Creo Parametric session