

Start-up & Navigating through SolidWorks

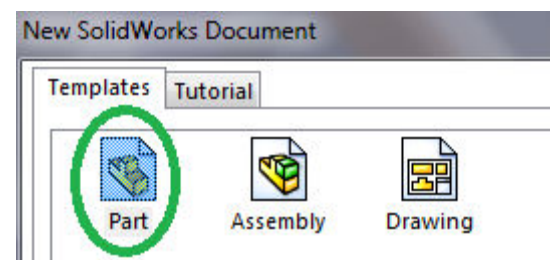
- Double-click on the SolidWorks icon on the desktop, which should be similar to the one on the right.
- Give the programme a few minutes to load up, it's usually slow!
- In the top left corner, you will see a tool bar, click on the blank white sheet.



- Click on the Part Document in templates and click OK.

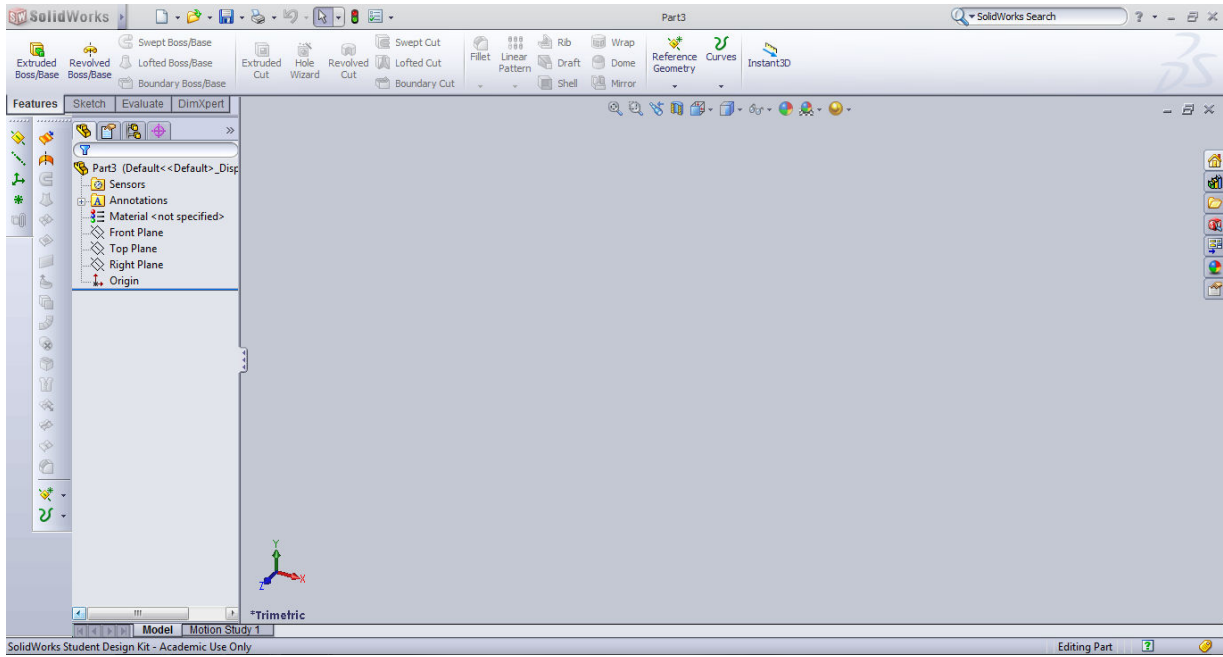
NOTE:

1. The part file is the standard file for creating a sketch or part.
2. The Assembly file allows you to import parts into a single file and assemble these parts together to represent a scene or an object with motion etc.
3. The Drawing file allows you to import a single part or assembly file and convert it to a 2 Dimensional drawing incorporating the standard orthographic views and many more.



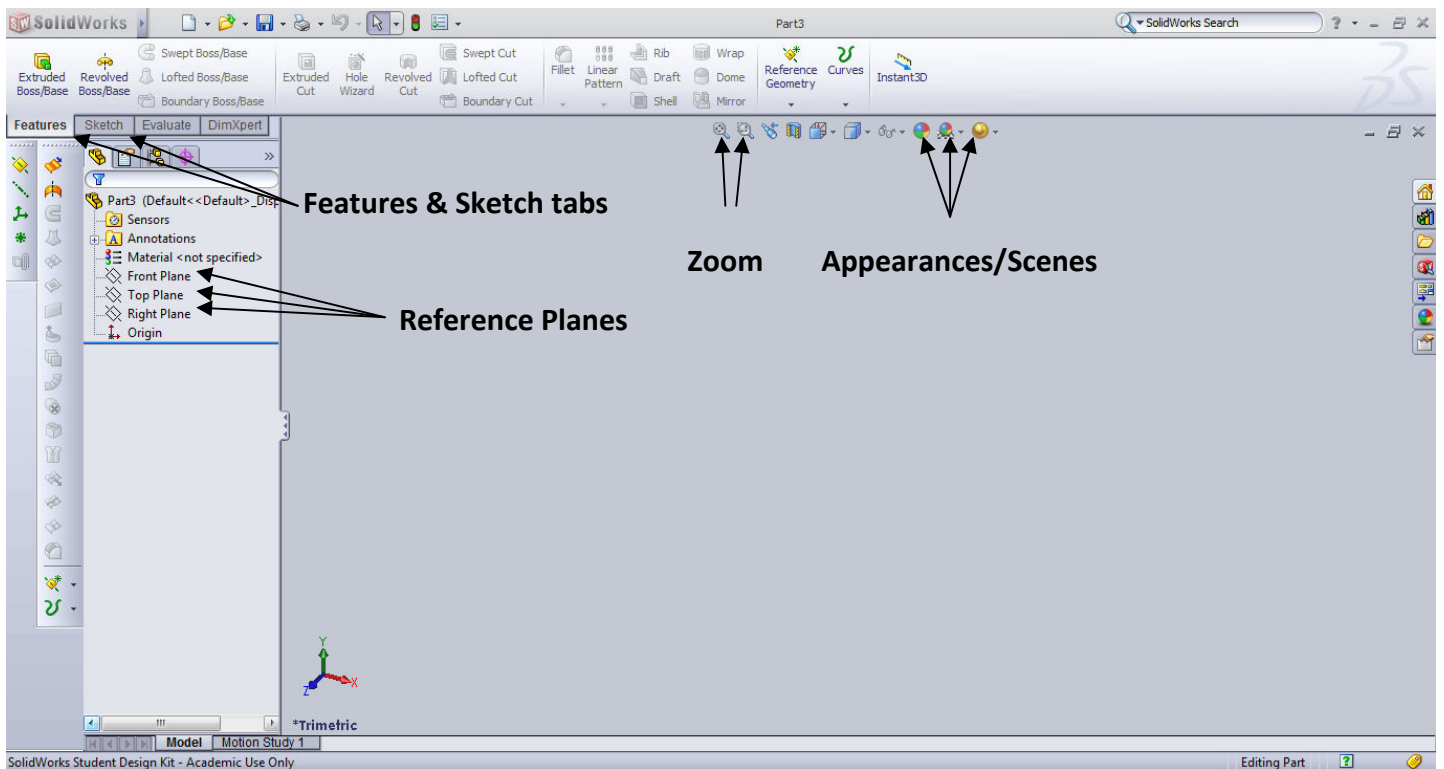
- You are now in the work area of SolidWorks.

Transition Year DCG Module



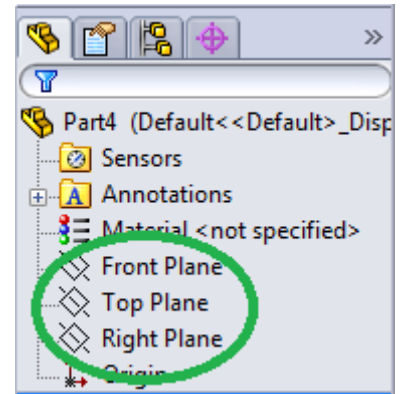
SolidWorks work area

Understanding the Layout and Key Features

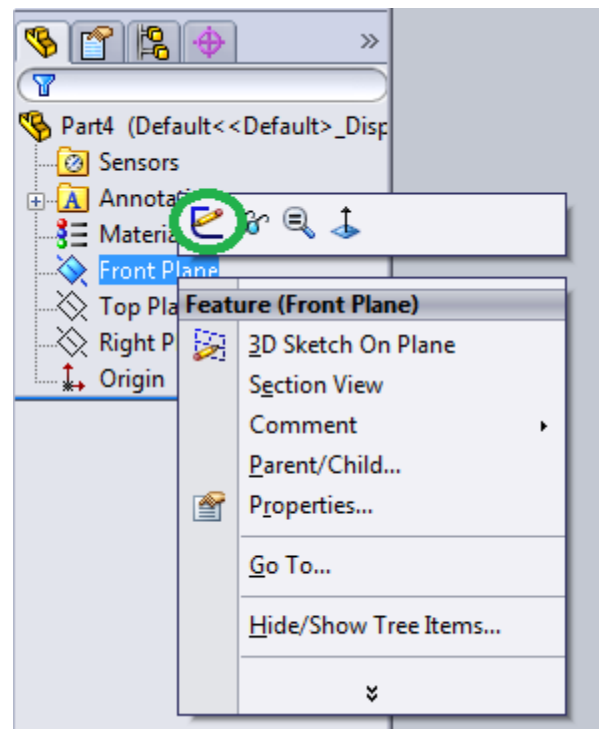


- The features pointed out above are the key and most commonly used features in SolidWorks.
- The **Reference Planes** are our first port of call as we need to select the correct plane to draw our sketch and hence create our object upon. If we chose the wrong plane to create our sketch on, it would lead to the different views being incorrectly positioned when it comes to importing the sketch/part to a drawing file.

1. The Elevation is always positioned on the Front Plane.
2. The plan is always positioned on the Top Plane.
3. The End Elevation is always positioned on the End/Right Plane.



- To open a sketch plane, we right click on the plane that we want to sketch on and click the sketch tab.



- The **Features and Sketch tabs** incorporate a series of icons which carry out specific operations in part files.
- The **Zoom icon** on the left is used to enlarge your sketch/part to the full size of the screen with the whole sketch/part remaining clearly visible.
- The icon on the right hand-side, is used to hi-light a particular area and zoom in on it as much as you wish.
- The Appearance and Scene's icons allow you to change the colour of your work and the background scene to make your model look more realistic.

Saving your Work and Exiting

- To correctly save your work, click the save icon in the top left-hand corner.



- 'Double click' the folder that you wish to save your work in, and click **save** again.
- To exit, click the bottom X first which exits out of the sketch or part that you were working on.



- Then click the top X which exits out of SolidWorks altogether.