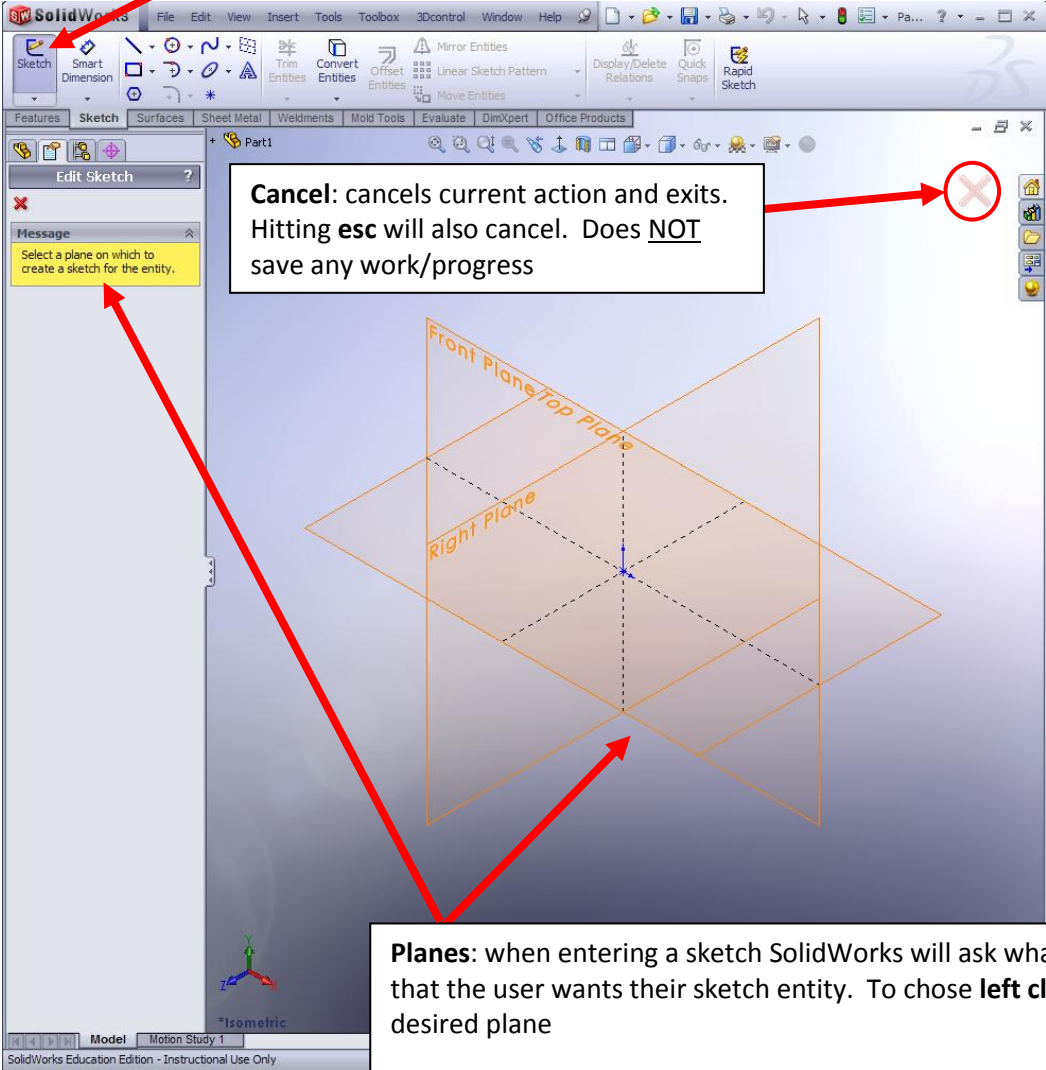


Sketching

Description: The Sketch toolbar is located in every document type (part, assembly, drawing). In a part document it's used for building a single part, in an assembly document it's used when you need the geometry of a part to build/design others, whereas in a drawing it is used to add additional annotations.

Note: Before adding additional tools to the toolbar check each tools drop down menu to ensure that the desired tool is not already in the toolbar. This can avoid clutter.

Entering a Sketch



Sketch: Enters a sketch

Cancel: cancels current action and exits. Hitting **esc** will also cancel. Does **NOT** save any work/progress

Planes: when entering a sketch SolidWorks will ask what plane that the user wants their sketch entity. To chose **left click** on the desired plane

Note: pay careful attention to the orientation of your part, as this selection transfers these views into the standard orthographic views. It is helpful to work off of a design layout sketch to achieve desired results.

The screenshot shows the SolidWorks interface with the Sketch toolbar highlighted. A message box prompts the user to "Select a plane on which to create a sketch for the entity." The 3D model displays the Front Plane, Top Plane, and Right Plane. A red 'X' icon in the bottom right corner indicates the Cancel function.