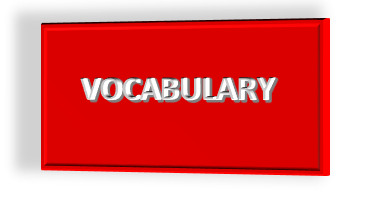
**iCREAT I: Module 9 - Solid Works Vocabulary**



horizontal line

**Vocabulary**

**BOM (Bill of Materials) -** list of parts that comprise an assembly. It includes the name of each part, quantity, material, price etc.

**CAD-** Computer Aided Design-is the use of computer system to the assist the creation of a design

**Cartesian Coordinate systems -** in 3D space, three mutually perpendicular axes, usually referred to as X, Y and Z

**Center lines -** one long dash and one short dash

**Convert Entities -** allows to copy geometry from one sketch to another

**Ctrl + 7** - returns you to Isometric view

**Explode/Collapse -** When the parts of an assembly expand or come back together

**Extruded Boss/Base –** adds material to a sketch to make it a 3D feature

**Extruded Cut -** removes material from a 3D Feature

**F -** Pressing the F key on a keyboard **fits** the object to the space of your screen

**Fillet -** Rounds off the edges

**Fully Defined -** geometry appears black as it is defined to its specifications

**Hold on the mouse wheel-** rotates the image

**Loft-** Creates a feature when making transitions between profiles <http://www.youtube.com/watch?v=3jgJ71wtoxc>

**Mating -** aligning parts in an assembly

**Multiple Parts -->Assembly**

**Ø -** Symbol for diameter (used to dimension circles)

**Origin -** Use the origin to start a sketch

**Orthographic Projection** - The two-dimensional representation of a three-dimensional object.

**Over Defined -** geometry appears red because the program is confused geometry has to many specifications and some need to be deleted

**R -** Symbol for radius (used to dimension arcs)

**Relation -** A way to add definition to a sketch without a dimension, i.e. dimensioning only one circle and making others equal relation.

**Revolve –** Feature that adds material in acircle around a central axis

**Scroll the mouse wheel-** zooms in and out

**Section Lines** - two long dashes and two short dashes

**Section Views-** Cutting an object in half

**Shell -** creates a hollow space inside a 3D feature

**Sketch (2D) --> Features (3D)**

**Smart Dimensions -** provides a way to specify size for a sketch or a feature

**Sweep -** Gives a 3D shape to a profile object ( requires 2 sketches a path and a profile)<http://www.youtube.com/watch?v=VssKlTevseE/>

**Under Defined -** geometry appears blue and needs more specifications to be considered complete (dimensions or relations)