**Section Conclusion: Let us Start Sketching:**

This section covered sketching in SOLIDWORKS

Before your leave this section make sure you know the following:

* How to select a plane to sketch on.
* How to link your sketches to the origin
* How to sketch basic shapes like lines, rectangles, circles, arcs, and ellipses.
* How to use different methods to create shapes.
* How to use dimensions to define your sketches.
* How to use relations to define your sketches.
* What are under-defined, fully-defined, and over-defined sketches.
* How does SOLIDWORKS apply relations automatically.
* How to know and delete active relations.
* What are construction and center lines and how to use them.
* How to use the commands: mirror, offset, patterns and trim.

If you do not know or unsure of any of the listed aspect above, we recommend reviewing the lectures in this section again before moving to the next!!

Also, you can always post any questions, comments, suggestions or whatever in the discussion board or directly contact the instructor :)