At this point, we can understand **features** as **our way into going into the third dimension in SOLIDWORKS!!**

Starting with building models, we will be going through the three steps:

1. Select a plane (covered in the section “Lets Start Sketching")
2. Drawing a sketch (covered in the section “Lets Start Sketching")
3. Apply a feature (we'll cover in THIS SECTION!)

More on features in the next lecture!