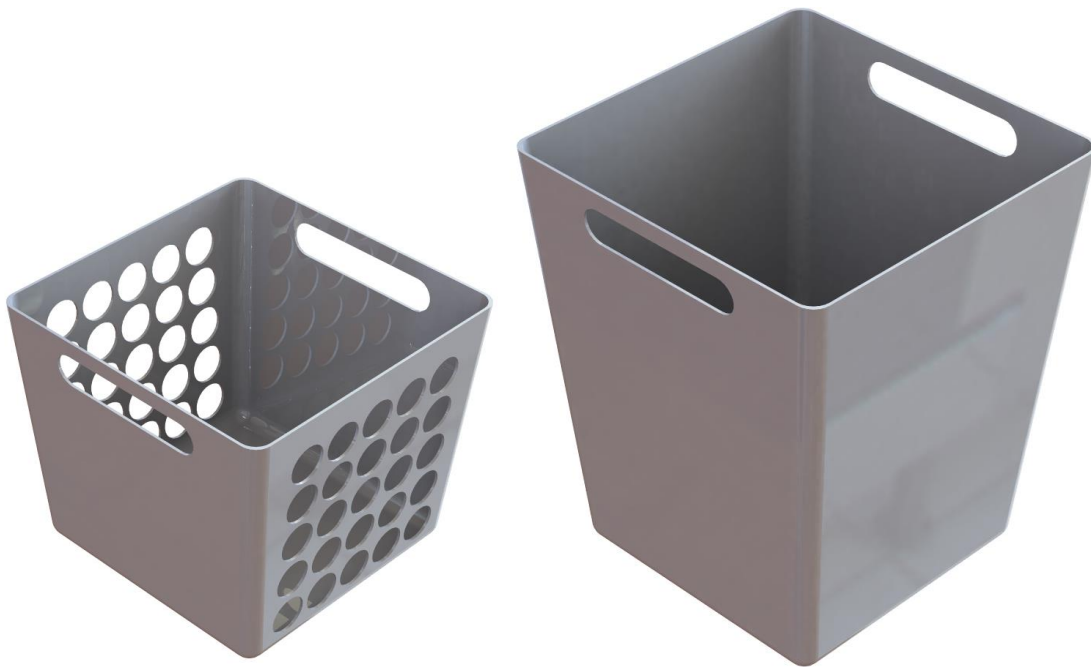


SOLIDWIZE

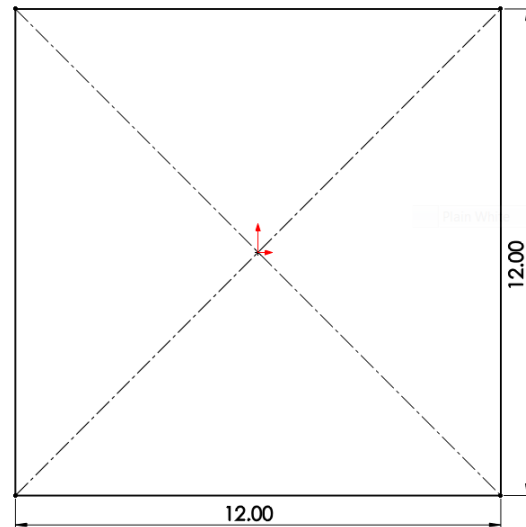
ONLINE SOLIDWORKS TRAINING

Design Table: Basket

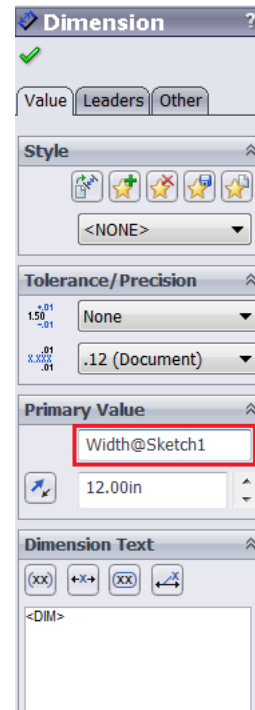
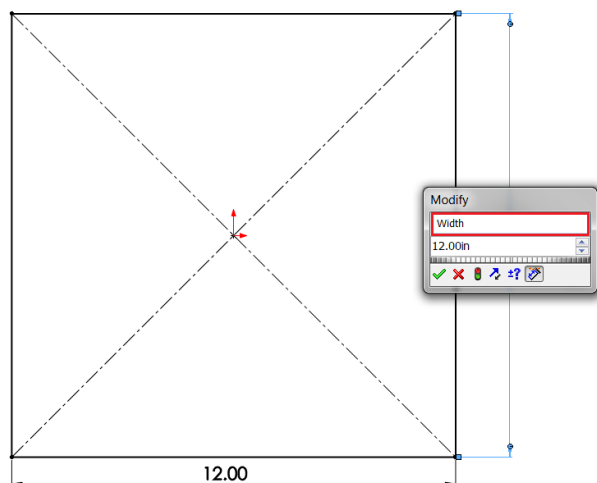


Step 1: Creating the Basket

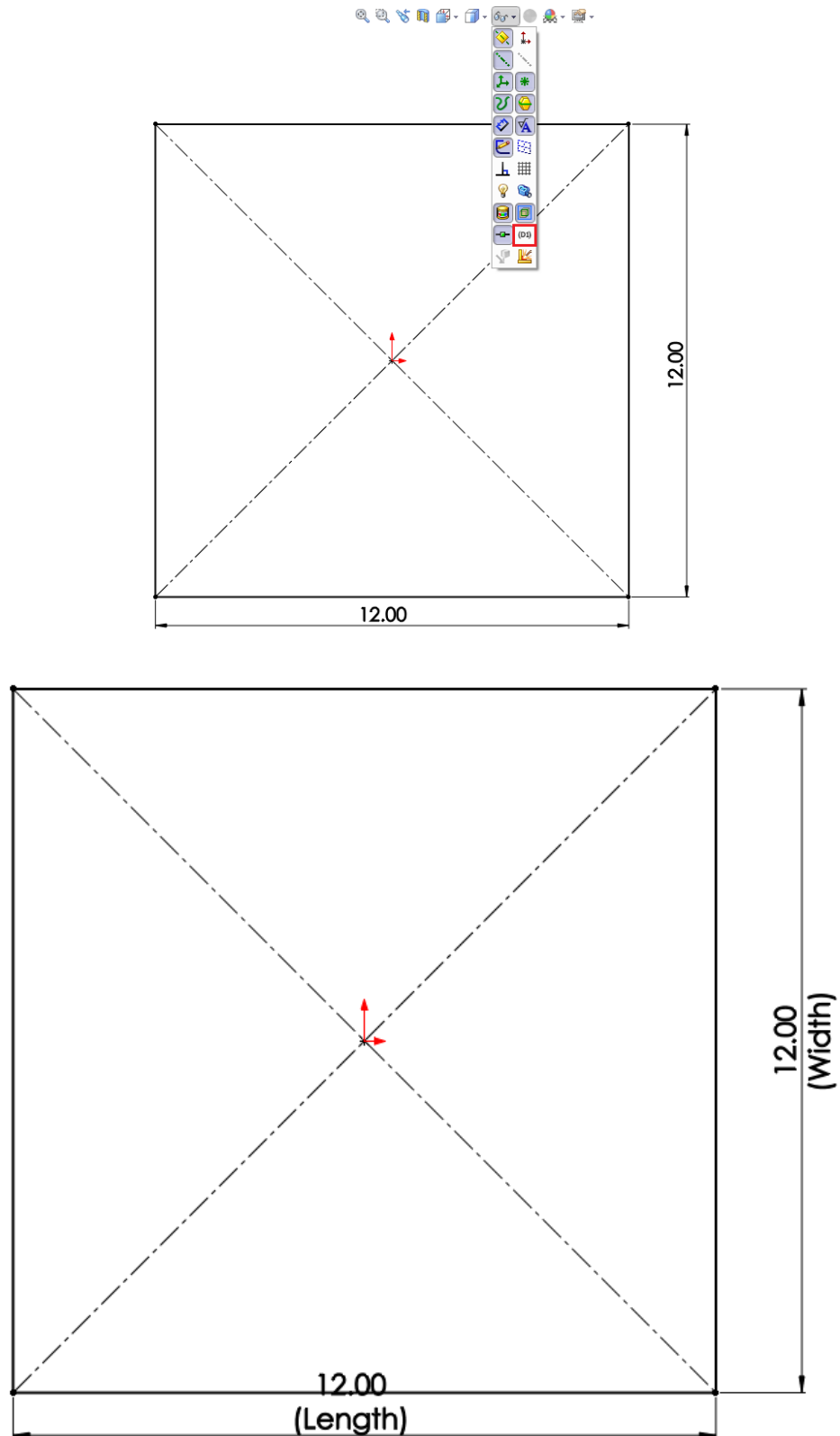
Using inches as the unit, create the following sketch on the top plane.



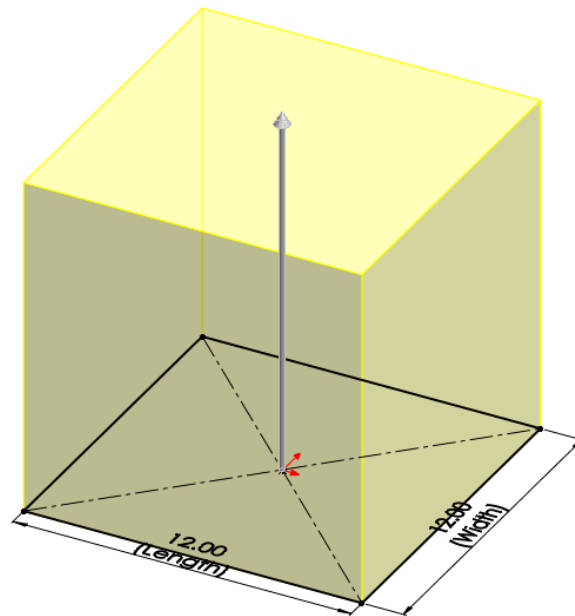
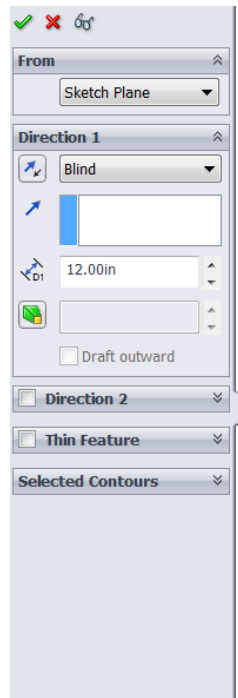
For ease of identification, we want to name the dimensions. You can either change the dimension names when you apply the dimension by changing the name in the pop-up dimensions box, or you can edit the dimension by left-clicking on the dimension and changing its name in the property manager. Name the vertical dimension **“Width”** and the horizontal dimension **“Length”**. The two methods are shown below:



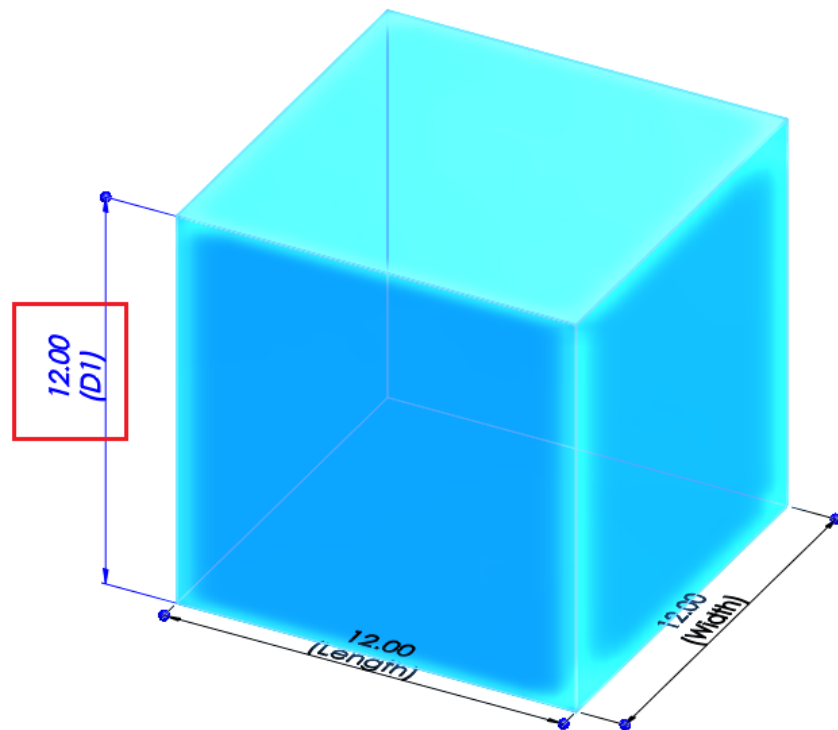
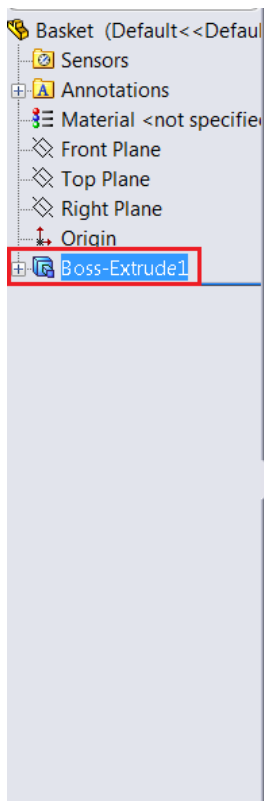
To see the dimension names, you must turn on **“View dimension name.”** To do so, select the **“Eye-glasses”** icon at the middle top of the display pane and select the **“(D1)”** icon as shown below:



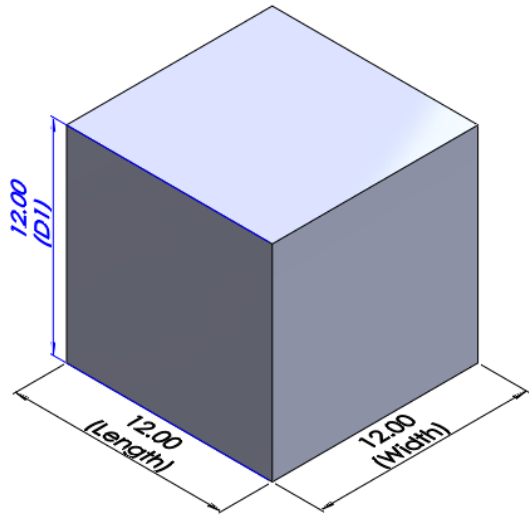
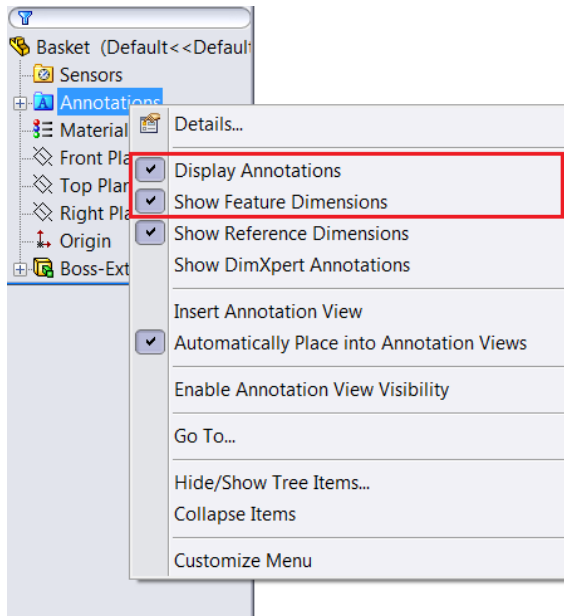
Extrude the basket upwards a distance of **12.00in**.



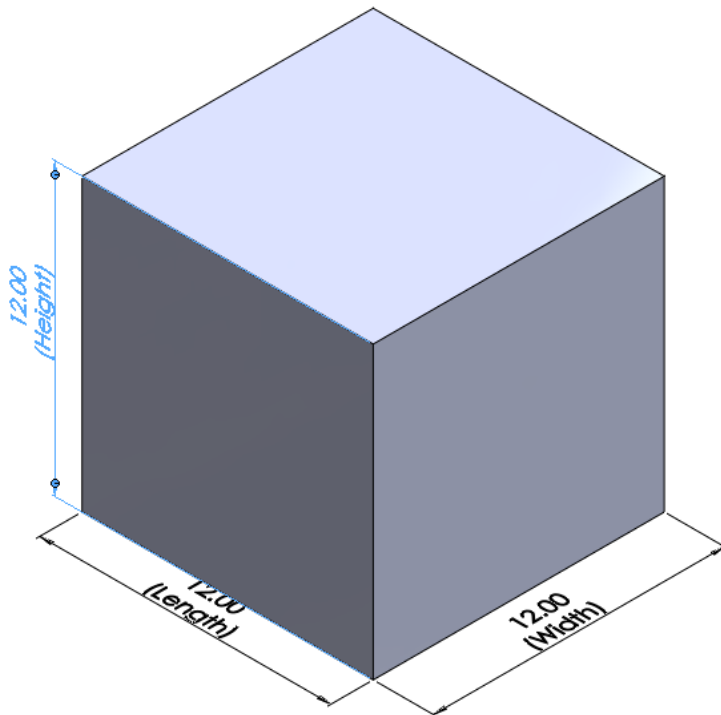
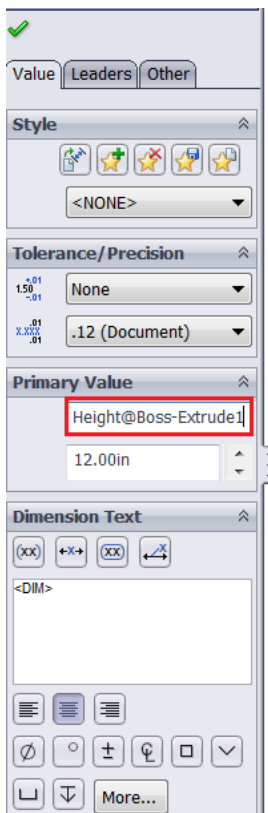
We will also want to name the extrusion distance. This can be done by selecting **Boss-Extrude1** from the feature manager on the left. The dimensions related to this feature will appear.



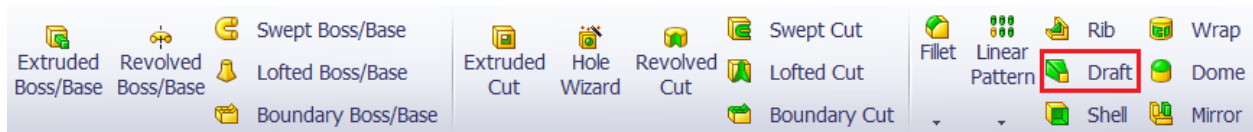
You can also view all dimensions and annotations by right clicking on **“Annotations”** in the feature tree and select both **“Display Annotations”** and **“Show Feature Dimensions.”**



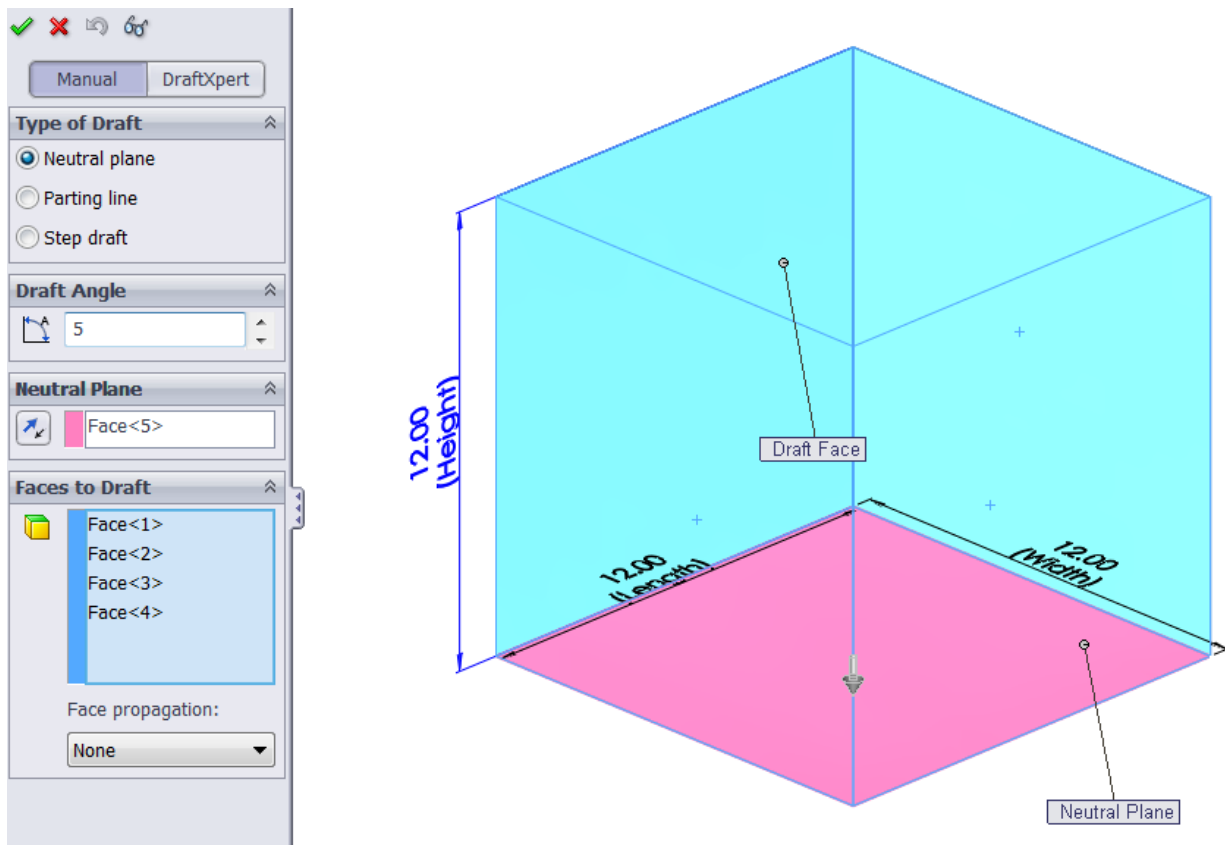
Change the blue extrusion length's name to **“Height.”**



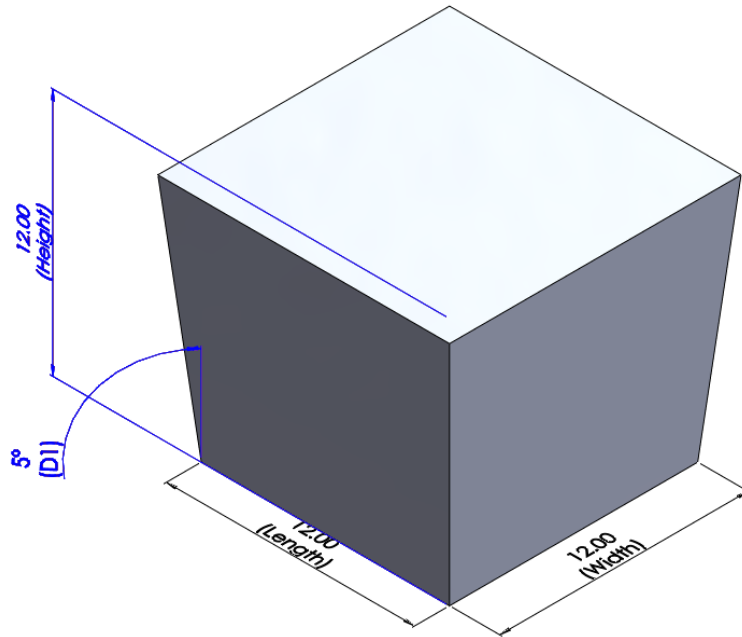
Next, we will add draft to the sides of the box. The draft tool is found under the **Features** tab.



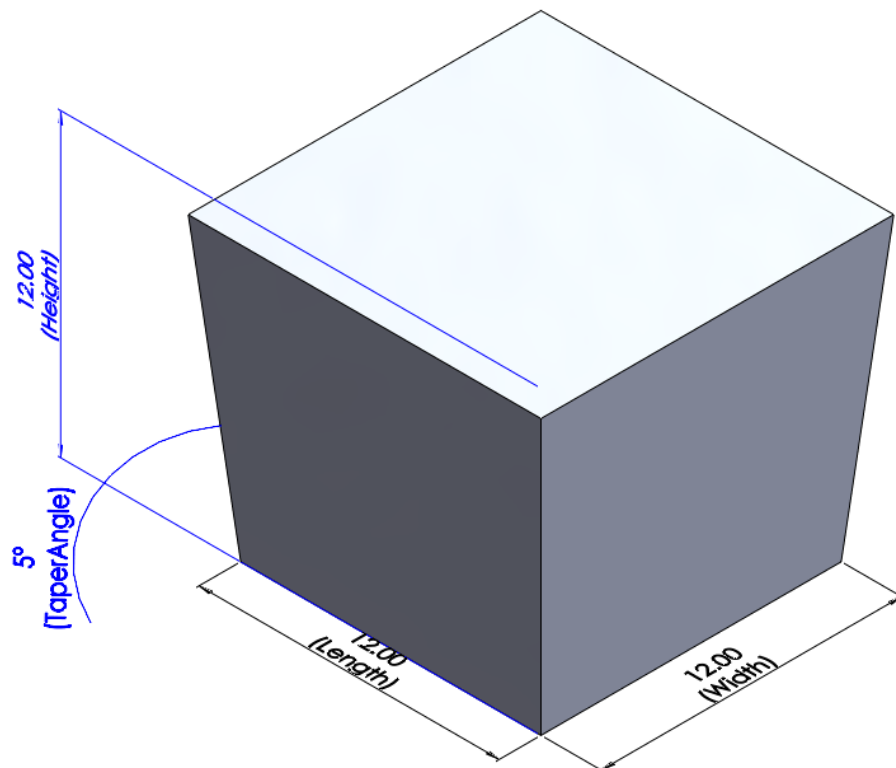
Select the **bottom face** of the cube as the **Neutral Plane**, and select the four sides as the items to draft. Set the draft angle to **5.00 degrees**.



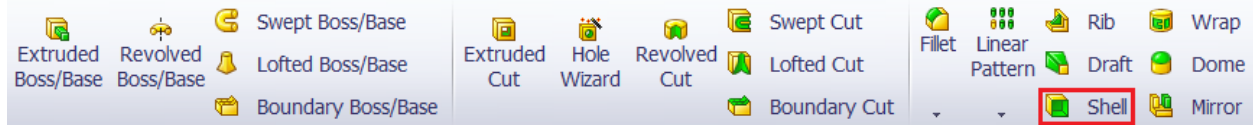
The resulting box should look like the following.



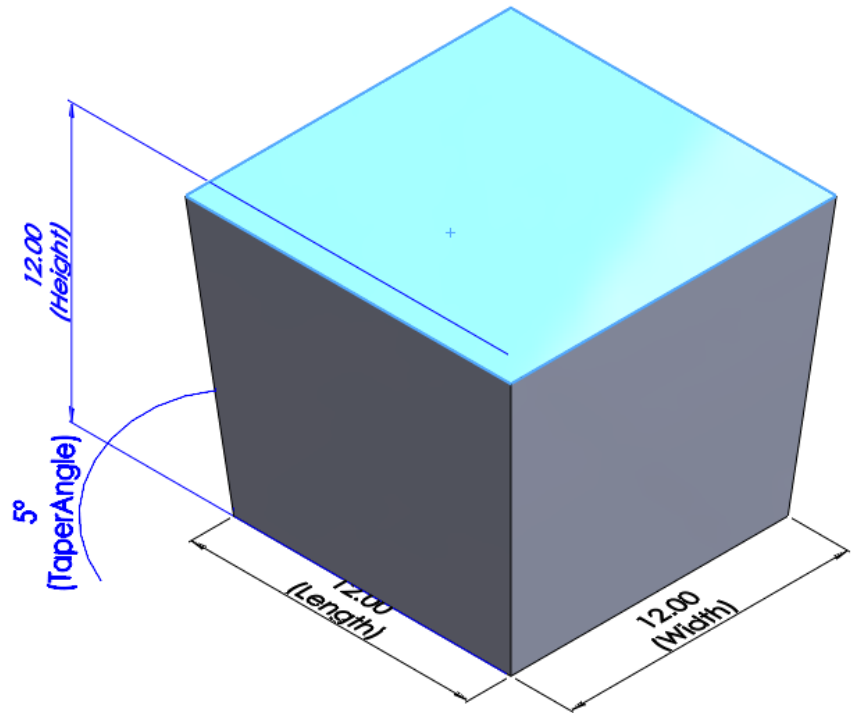
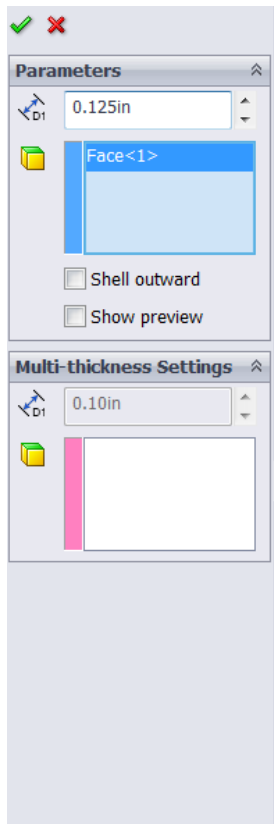
Change the draft angle's name to “**Taper Angle.**”



Next we will shell out the box to create the basket. Select the **Shell** tool from the **Features** tab. The tool can be found under the **Draft** tool by default.

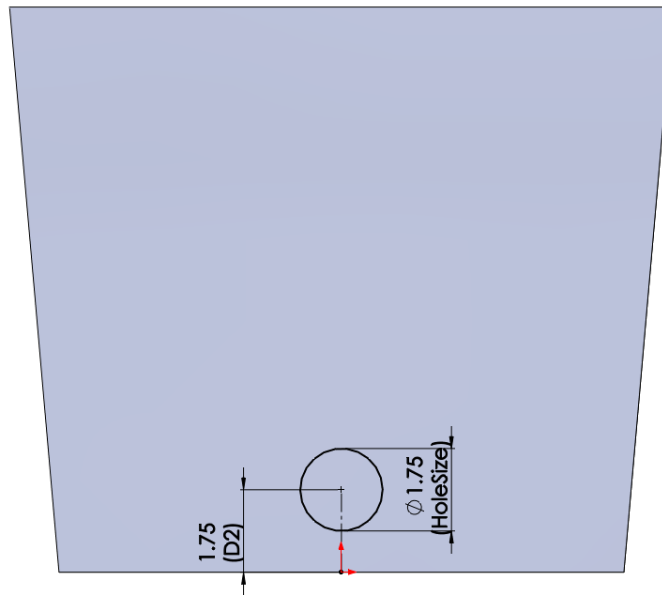


Select the top face. This is the face that will be deleted to shell out the part. Set the thickness to **0.125in**.

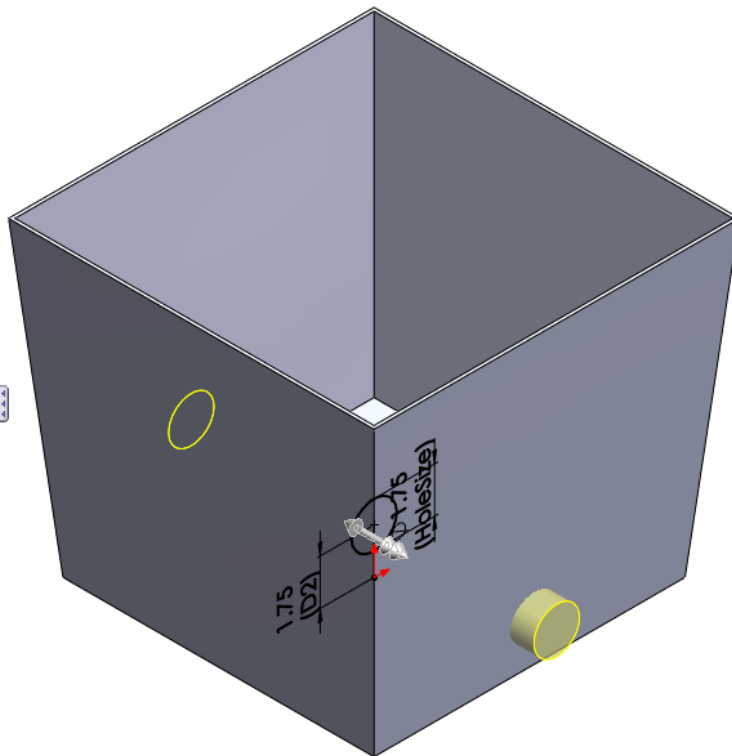
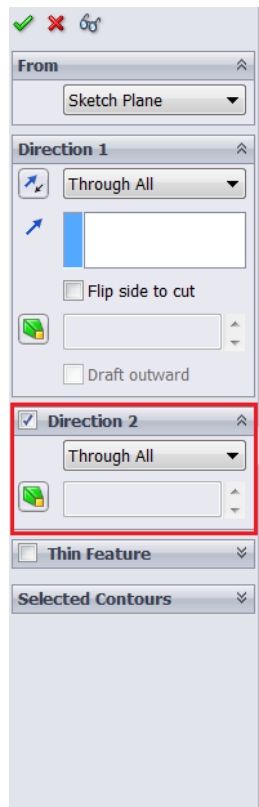


Step 2: Creating Holes

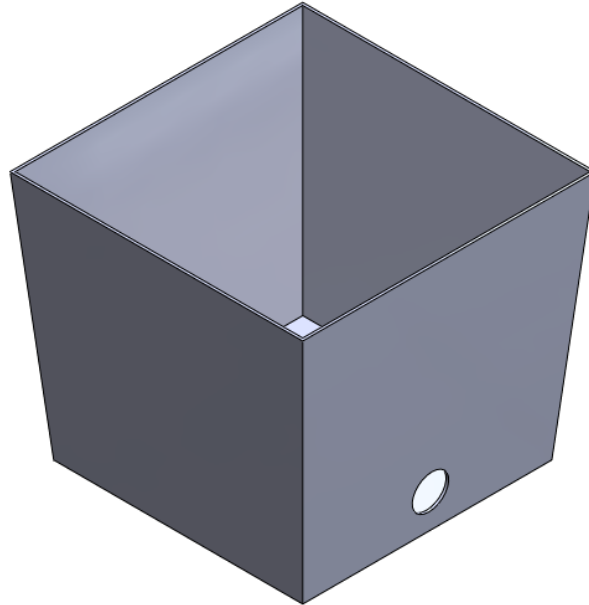
On the **right plane**, create the following sketch. Be sure to name the dimensions accordingly.



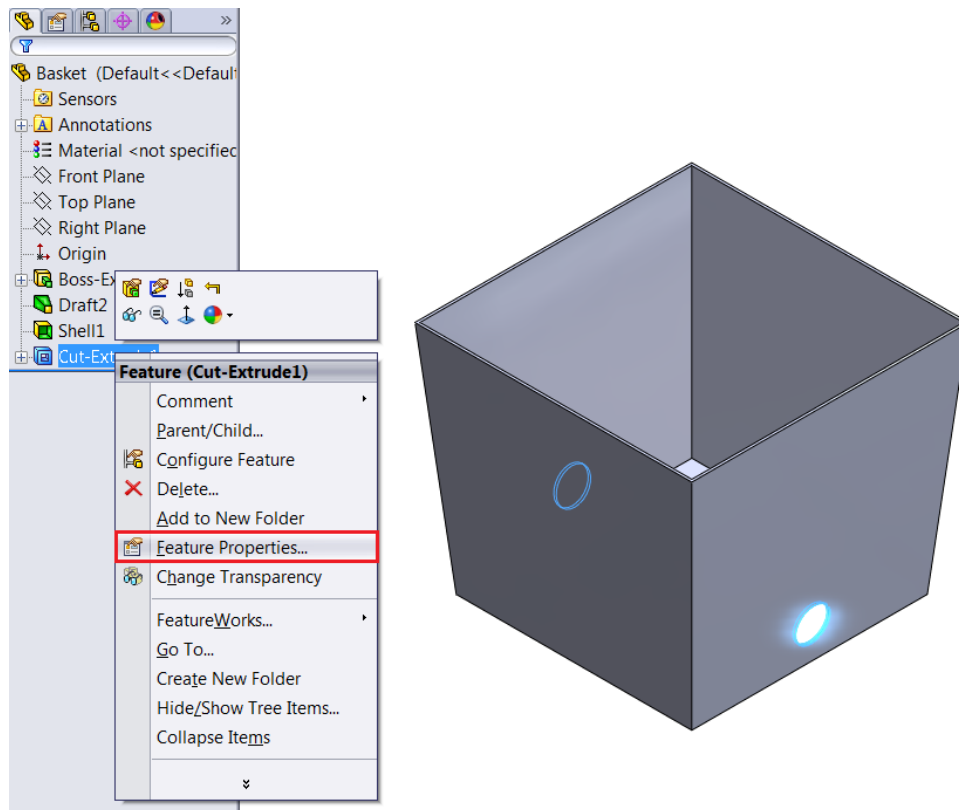
Extrude cut the sketch in **both** direction 1 and direction 1. Use the **Through All** cut type to cut through the entire part.



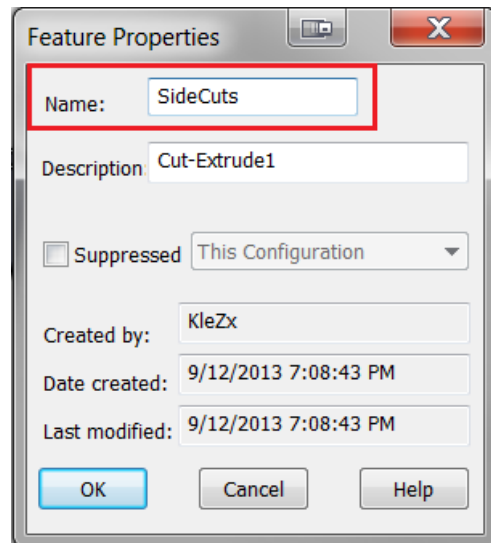
The resulting part should look like the following:



We will now rename this cut feature so that we can identify them later in the design table. To rename the feature, right click on its name and select **Feature Properties**.

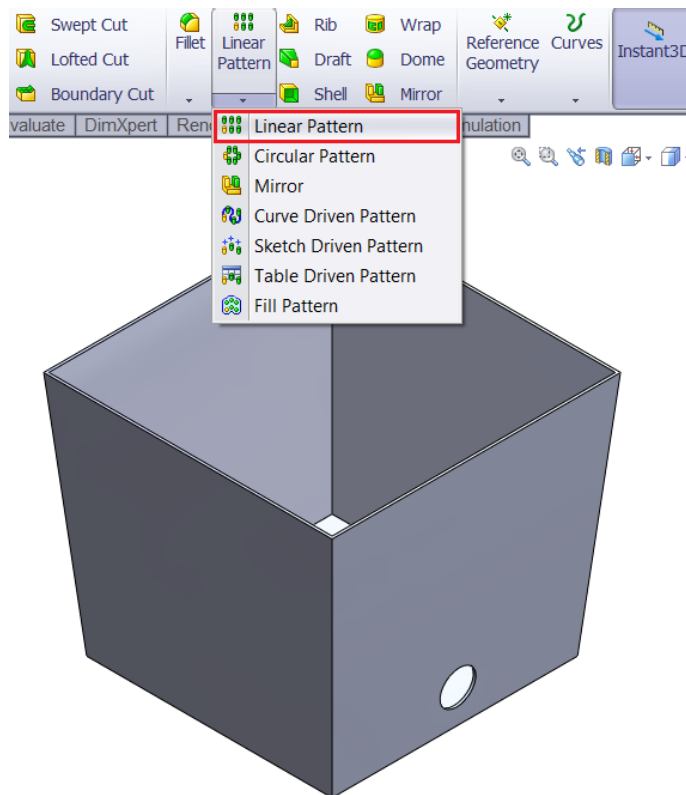


Change the name of the feature to “**SideCuts.**”

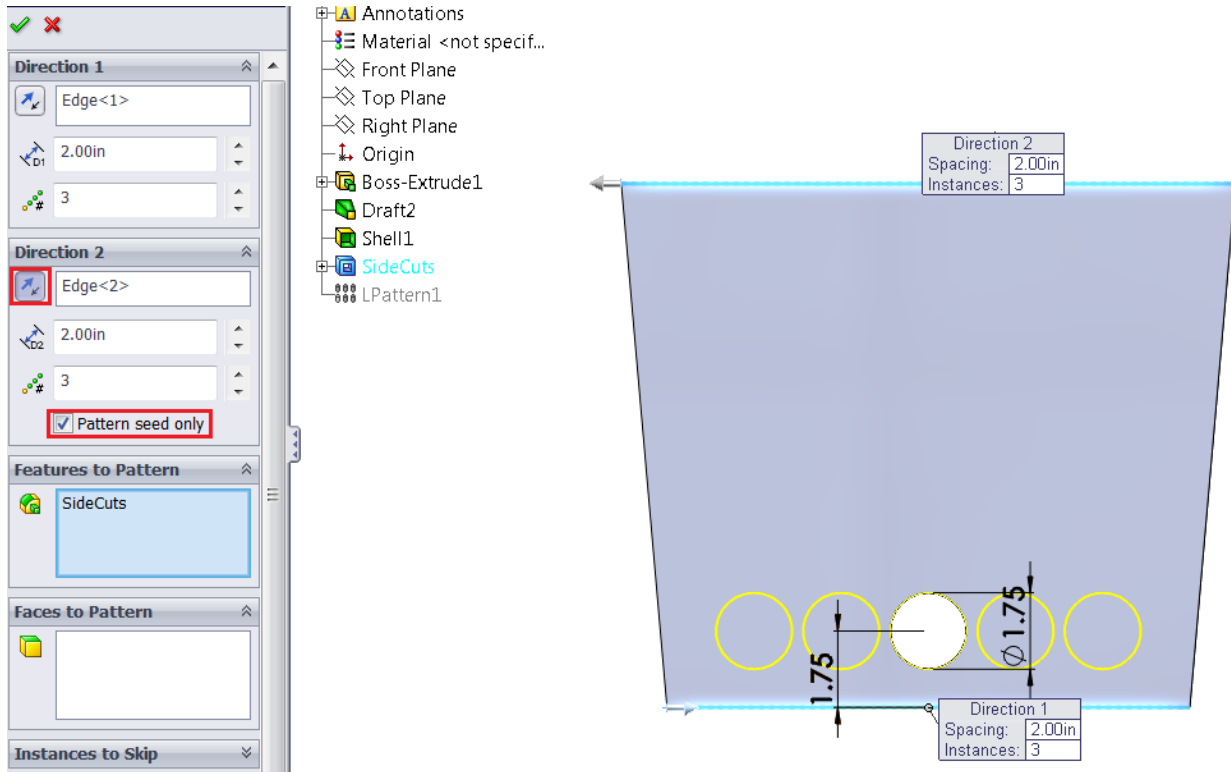


Step 3: Patterning the Holes

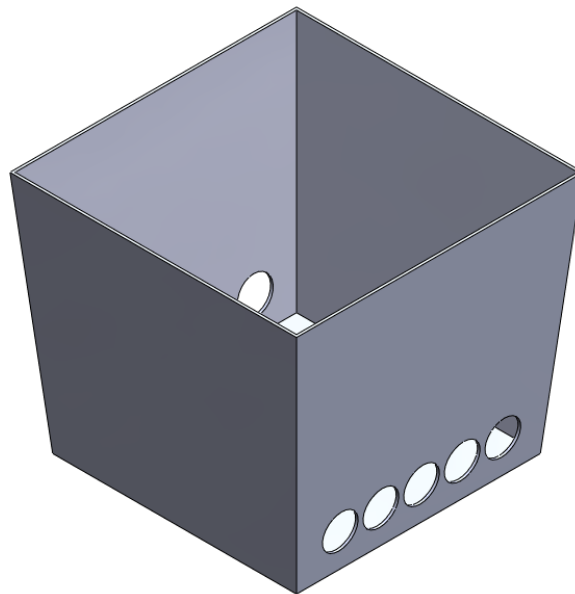
We want to create multiple holes across the side of the box. Select the **Linear Pattern** tool from the **Features** tab; using the pattern tool will save us time, and allow for quick changes in the design table.



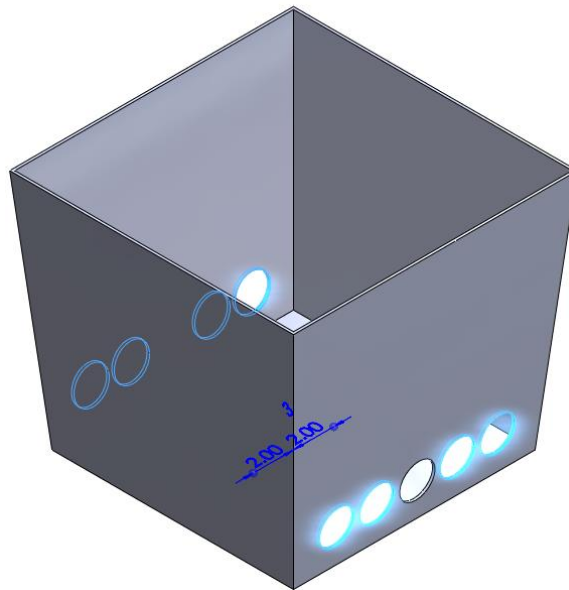
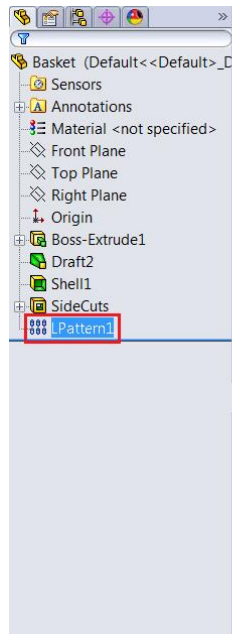
Select the horizontal top and bottom lines as the pattern direction. Set the **SideCuts** as the feature to be pattern. Set both pattern distances to **2.00in** and pattern instances to **3**. Select the **“arrows”** icon to pattern in the reverse direction. Make sure to select the **Pattern Seed Only** box, this will allow of patterning in parallel directions.



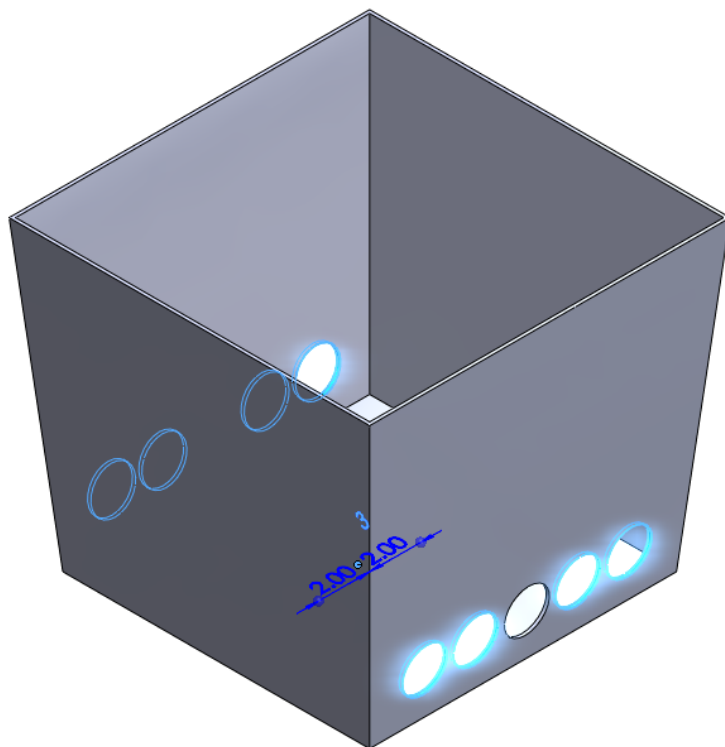
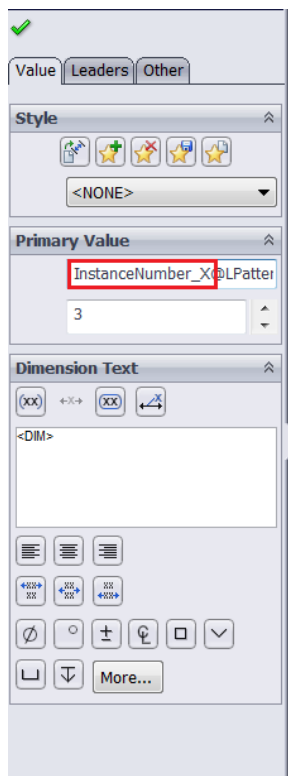
The resulting pattern should look like the following:



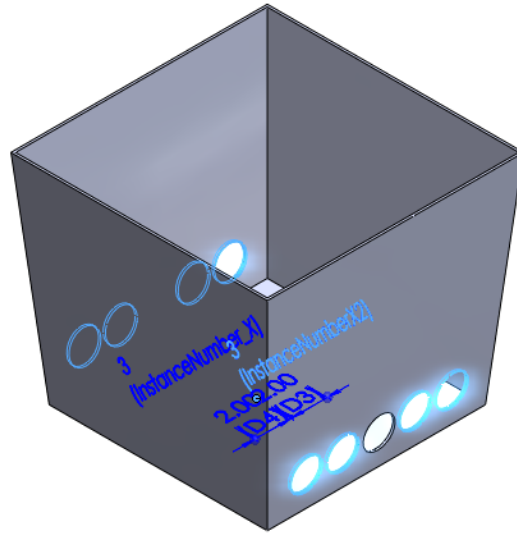
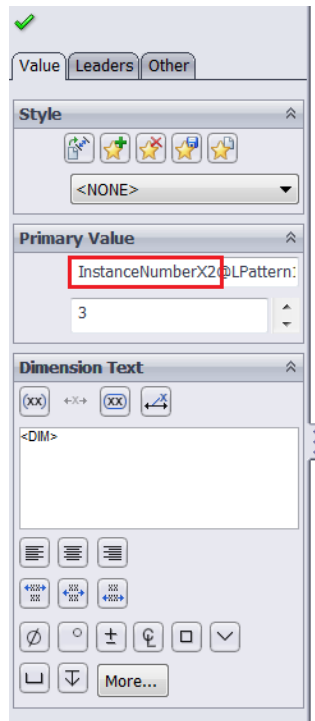
We will also want to name the **instance number** so that we can find it later for the design table. To do so, select the pattern feature. Notice the number “3,” that is the pattern instance. However, since we patterned in two different directions, there is in fact two numbers overlapping; each one signifies the instance number for each pattern.



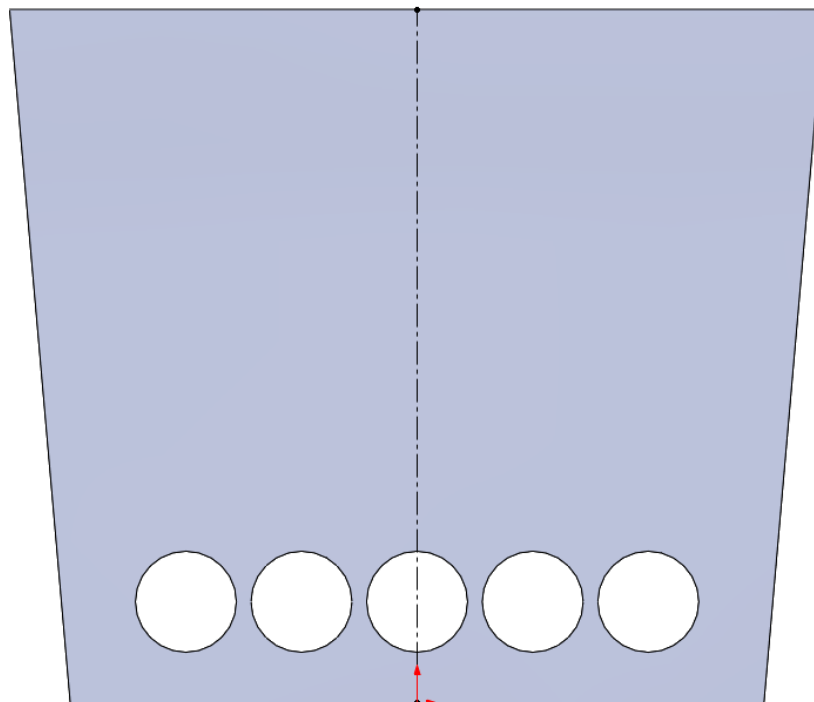
Rename one of the instance number to “**InstanceNumber_X**” :



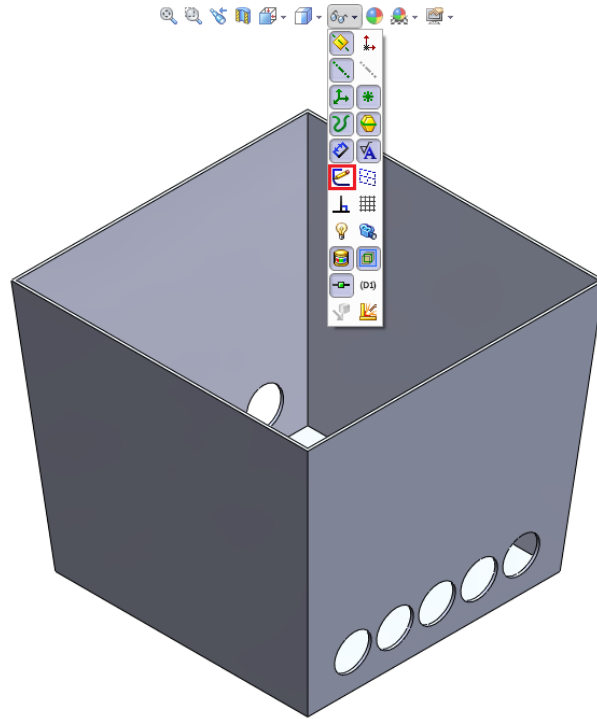
And rename the other instance dimension as **"InstanceNumber_X2"**:



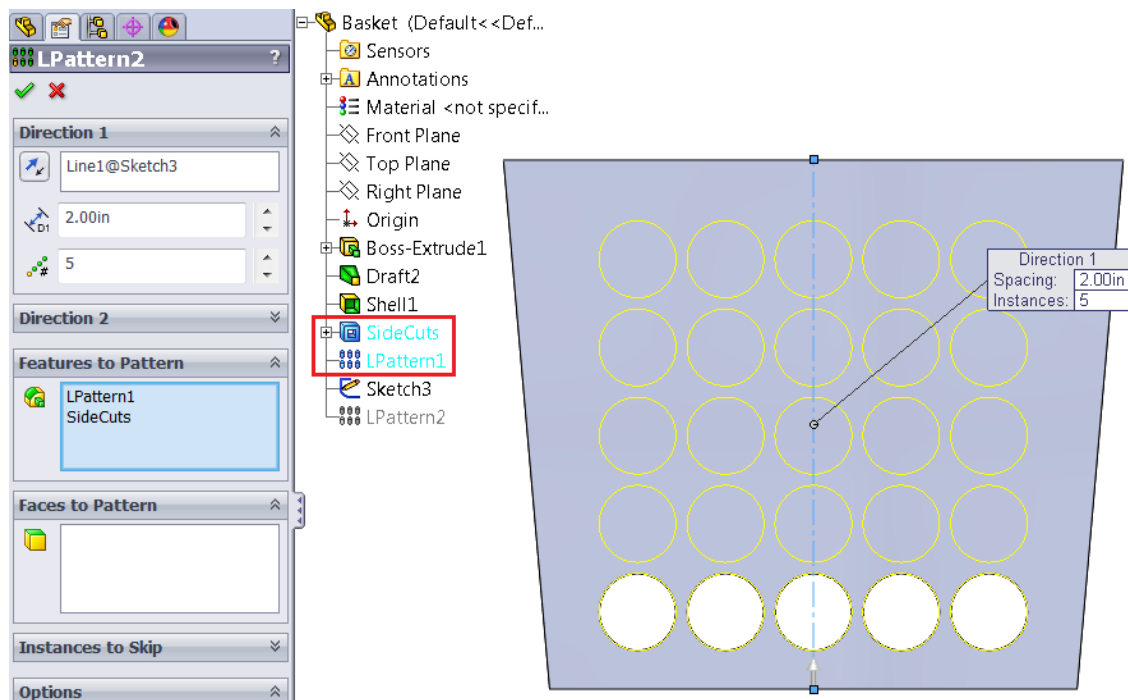
We now want to pattern the holes vertically. To do so, we will need a reference edge or line that is vertical. Draw a vertical construction line on the right plane as shown below:



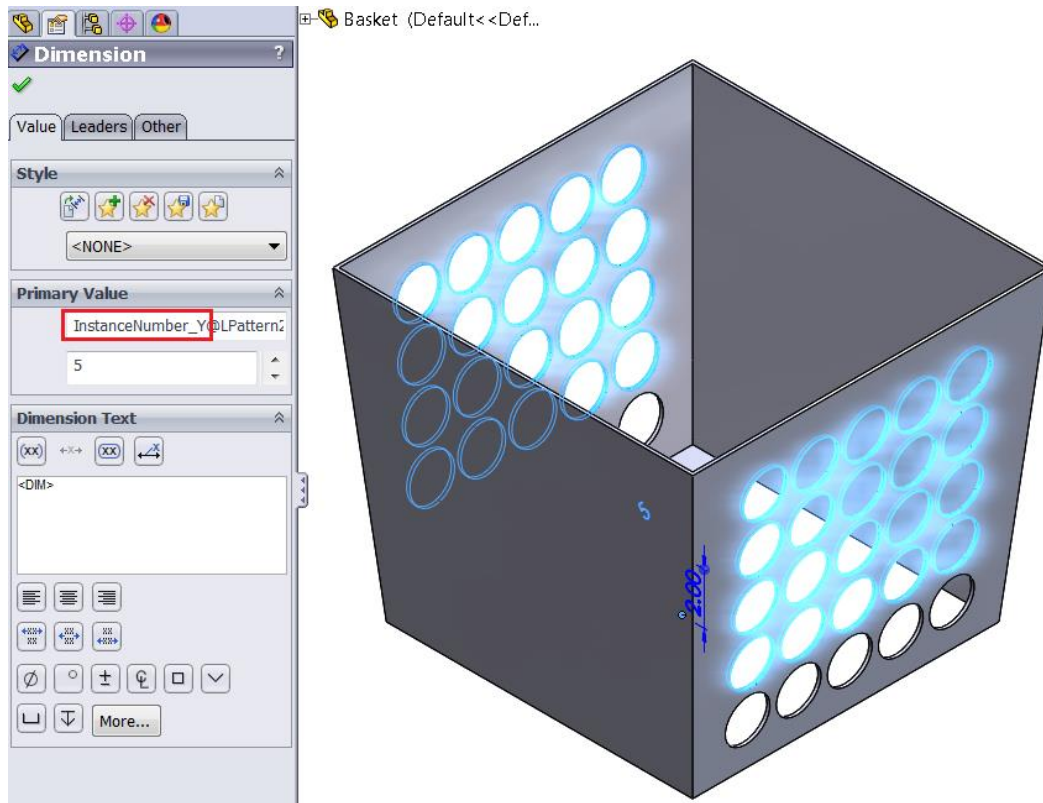
Once you have exited the sketch, if the sketch is not in view, turn on view sketches by selecting the sketch icon from the drop-down “**Eye-glasses**” menu.



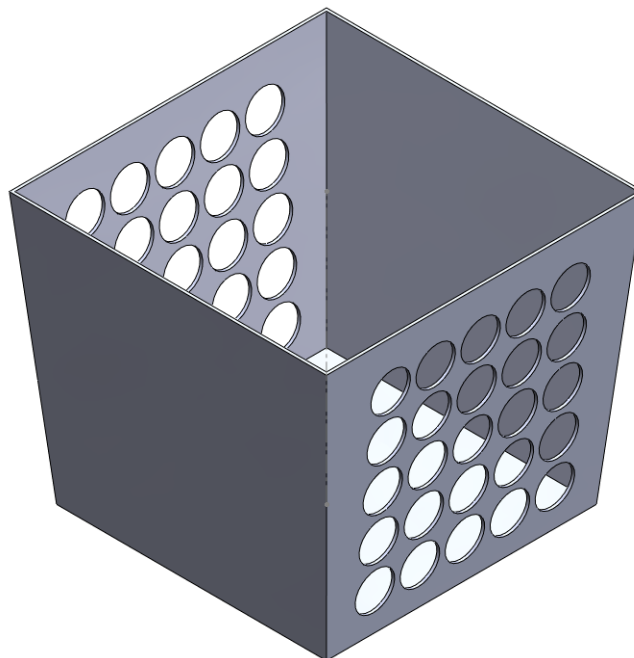
Again, use the linear pattern tool to pattern the holes. This time, select both the previous linear pattern and the SideCuts as features to pattern. Select the construction line as the direction of pattern. Keep the pattern distance the same and set the instance number to 5.



Once you have patterned the holes. Change the pattern instance name to **“InstanceNumber_Y”**.

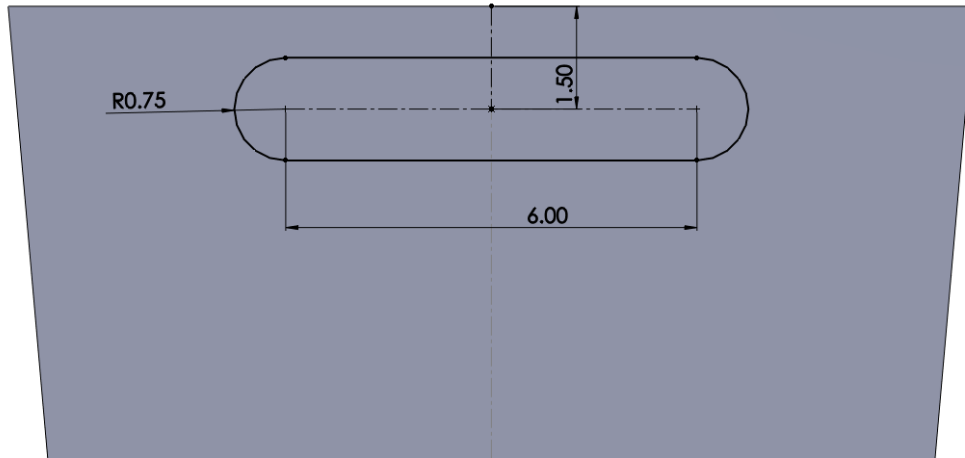


The basket should now look like this:

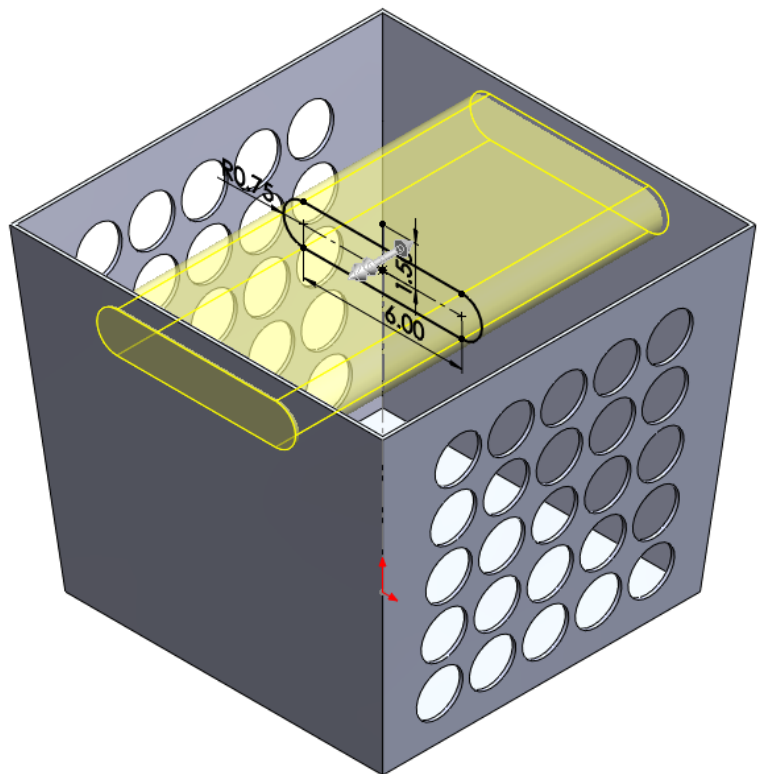
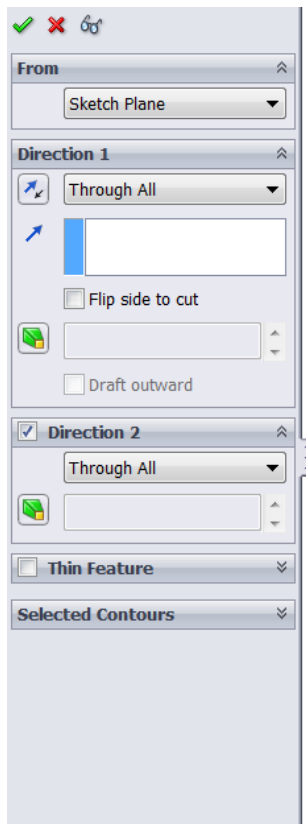


Step 4: Creating the handle

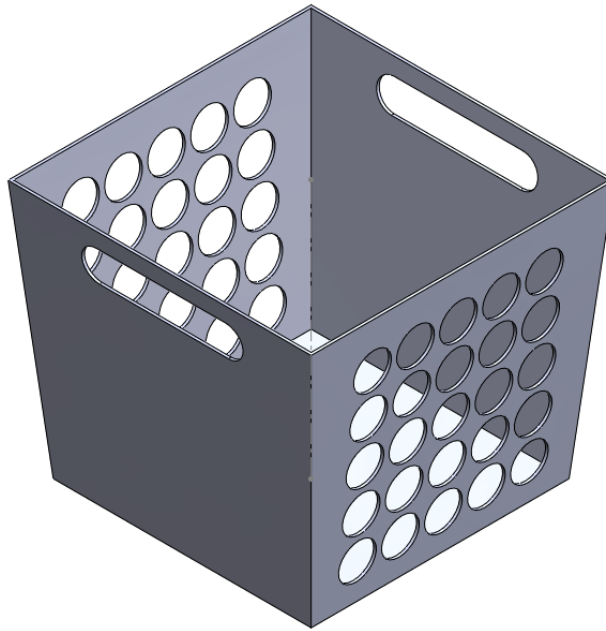
Create the following sketch on the front plane.



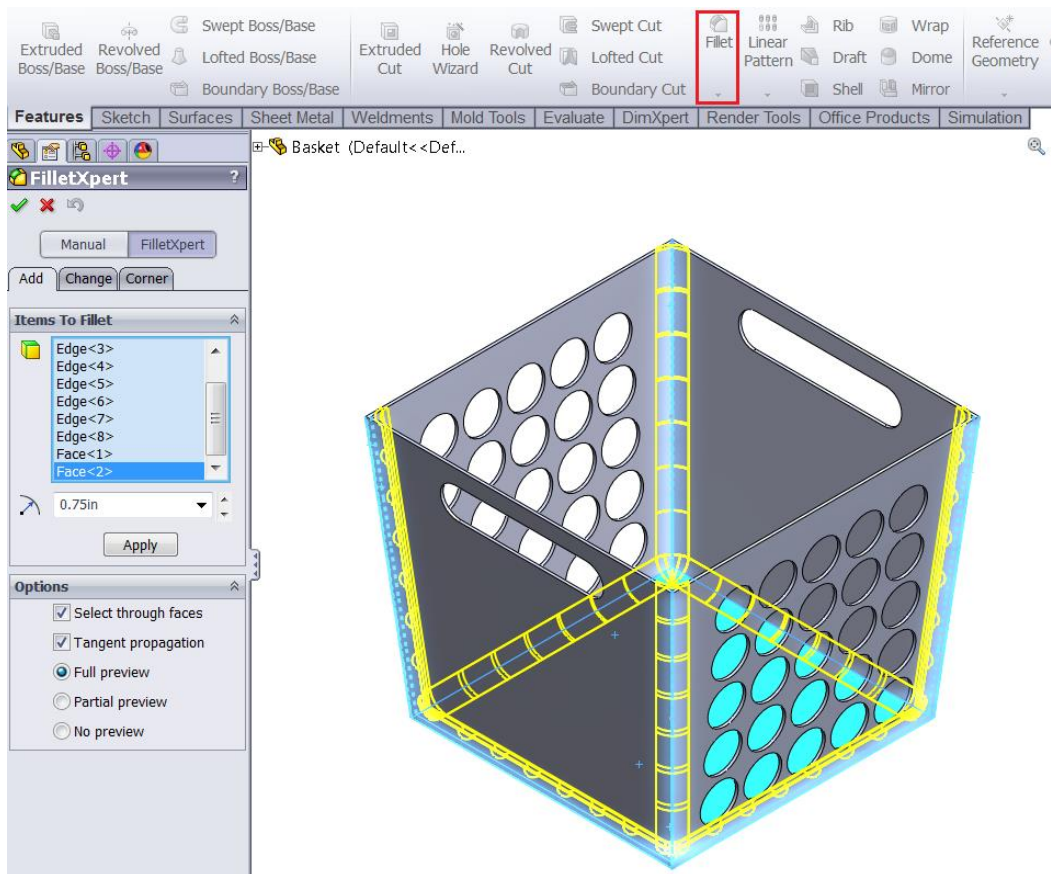
Extrude cut the sketch in both directions using the **Through All** cut type.



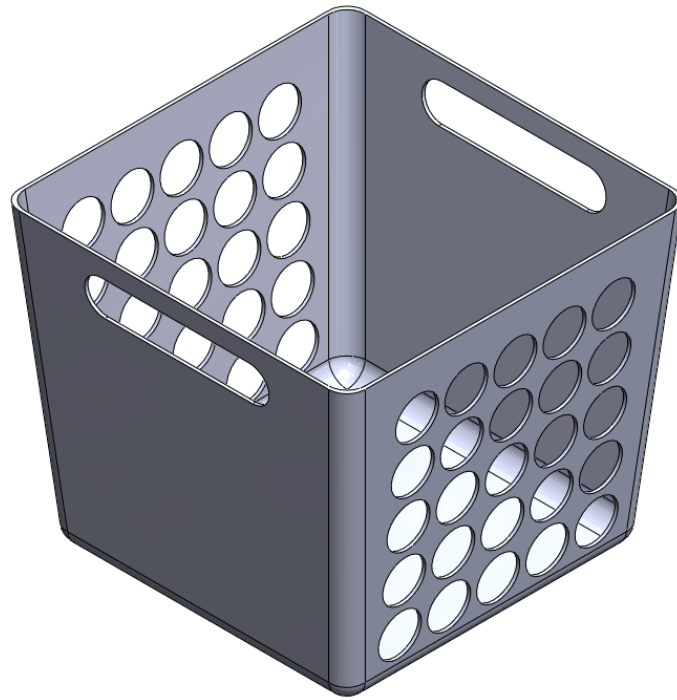
The resulting part is the following:



Add fillets to all side and bottom corners. Use a fillet radius of **0.75in**.

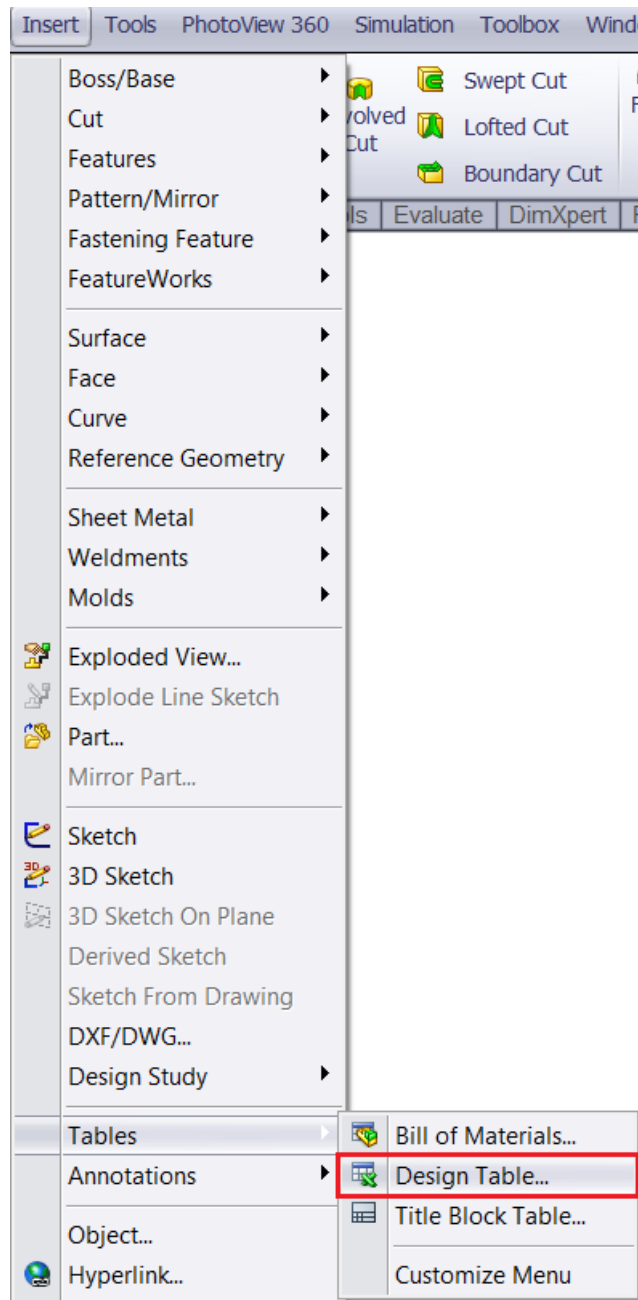


The finished basket looks like the following:

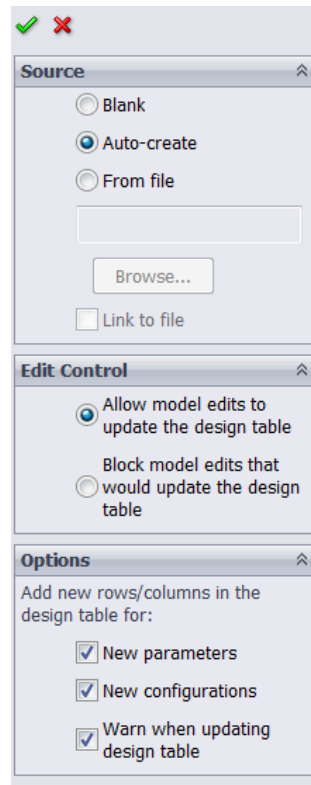


Step 5: Creating a Design Table

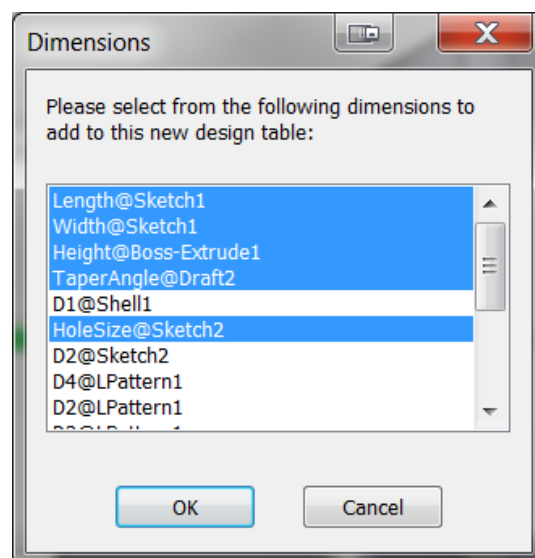
To create a design table, select **Insert>Tables>Design Tables** as shown below:



The following feature manager will appear. For our purposes, the default setting is fine and we can accept the table by selecting the green check mark.



After accepting the table, the following window will appear. This window prompts us to select which dimensions you want to add to the design table. Do not worry, we can add or delete dimensions later if need be. Hold **CTRL** and select the following dimensions: **Length**, **Width**, **Height**, **TaperAngle**, **HoleSize**, **InstanceNumber_X**, **InstanceNumber_X2**, and **InstanceNumber_Y**.



An Excel spreadsheet will appear in the top left of the display pane. This is your design table. Notice the dimensions selected have been added in the first row of the design table. The configurations are listed in the first column. Change the name of the **“Default”** configuration to **“12 x 12 x 12.”**

[illegible]

Select the cell under **InstanceNumber_X2** and rewrite it as “=G3” so that the pattern instances are equal.

The screenshot shows a SolidWorks spreadsheet interface. The formula bar at the top displays f_x and $=G3$. The spreadsheet grid has columns A through L and rows 1 through 16. Row 1 is titled "Design Table for: Basket". The data in the grid is as follows:

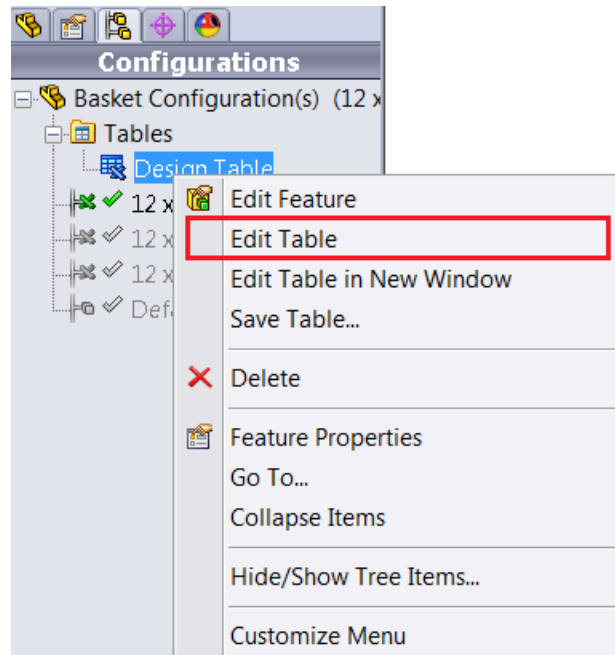
	A	B	C	D	E	F	G	H	I	J	K	L
1	Design Table for: Basket											
2		Length@Sketch1	Width@Sketch1	Height@Boss-Extrude1	TaperAngle@Draft2	HoleSize@Sketch2	InstanceNumber_X@LPatten1	InstanceNumber_X2@LPatten1	InstanceNumber_Y@LPatten1			
3	12 x 12 x 12	12	12	12	5	1.75	3	=G3	5			
4												
5												
6												
7												
8												
9												
10												
11												
12												
13												
14												
15												
16												

A yellow tooltip message is displayed over cell H3, stating: "InstanceNumber_X2@LPatten1 Enter a valid value for this parameter."

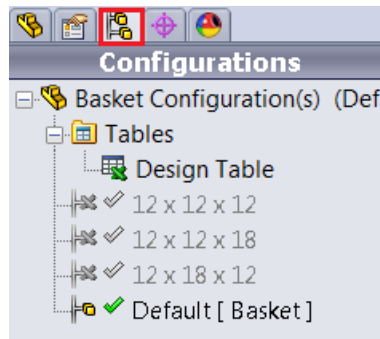
Add the following configurations to the design table as shown below:

	A	B	C	D	E	F	G	H	I	J	K	L
1	Design Table for: Basket											
2		Length@Sketch1	Width@Sketch1	Height@Boss-Extrude1	TaperAngle@Draft2	HoleSize@Sketch2	InstanceNumber_X@LPat	InstanceNumber_X2@LPat	InstanceNumber_Y@LPat			
3	12 x 12 x 12	12	12	12	5	1.75	3	3	5			
4	12 x 18 x 12	12	18	12	5	1.75	4	4	5			
5	12 x 12 x 18	12	12	18	5	1.75	3	3	8			
6												
7												
8												
9												
10												
11												
12												
13												
14												
15												
16												

To exit the design table simply click anywhere in the display pane. If you have exited the design table by accident, you can edit the table by going to the configurations tab in the feature manager, expanding the **Tables** drop-down, right click **Design Table**, and select **"Edit Table"**.

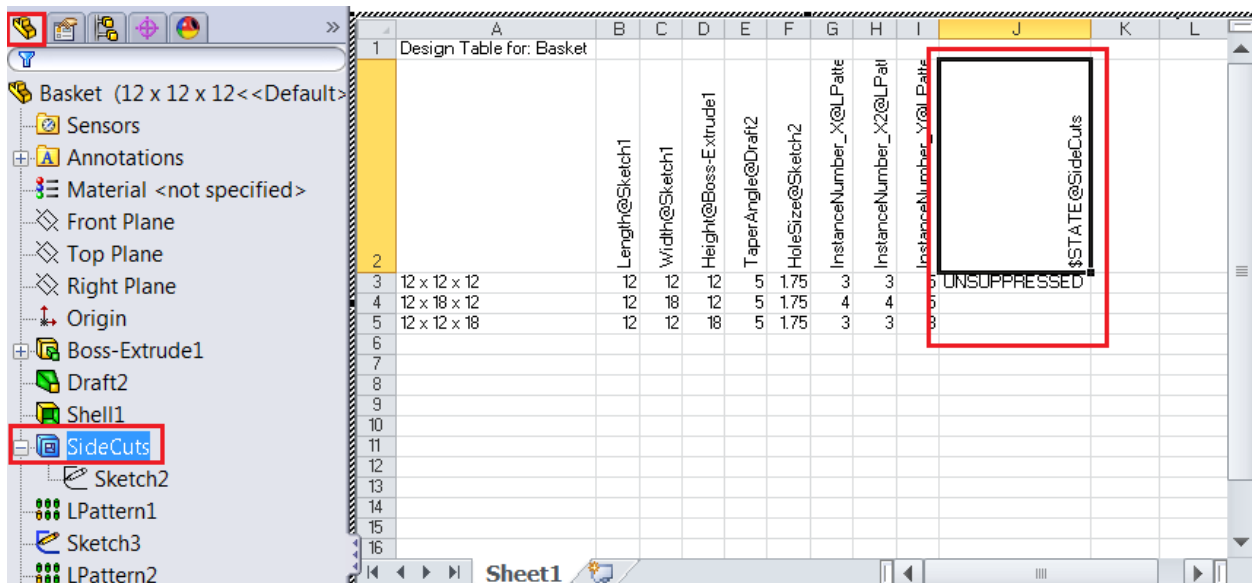


Notice that after you have exited the design table, the configurations you added in the Excel sheet has been automatically added as new configurations.



Step 6: Adding Features and Dimensions to the Design Table

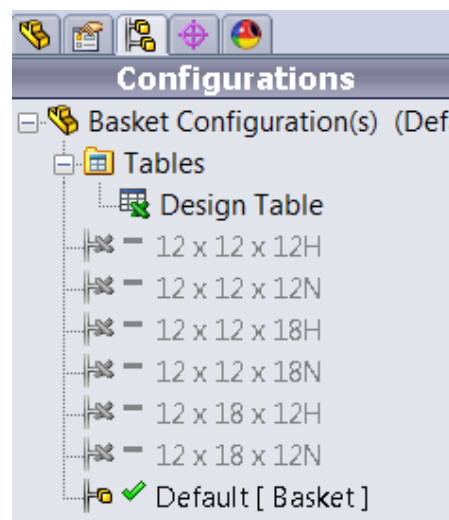
Enter the design table again by selecting the “Edit Table” command. We want to create configurations where there are no holes in the basket. To do so, select cell “**J2,**” or the empty cell next to the last dimension added. With the design table open, go to the feature tree and **double-click** on the **SideCut** feature. Double-clicking on a feature or dimension will add that feature or dimension to the design table. Since the **SideCut** is a cut extrude feature, the only options allowed for it are either “**Suppressed**” or “**Unsuppressed**”.



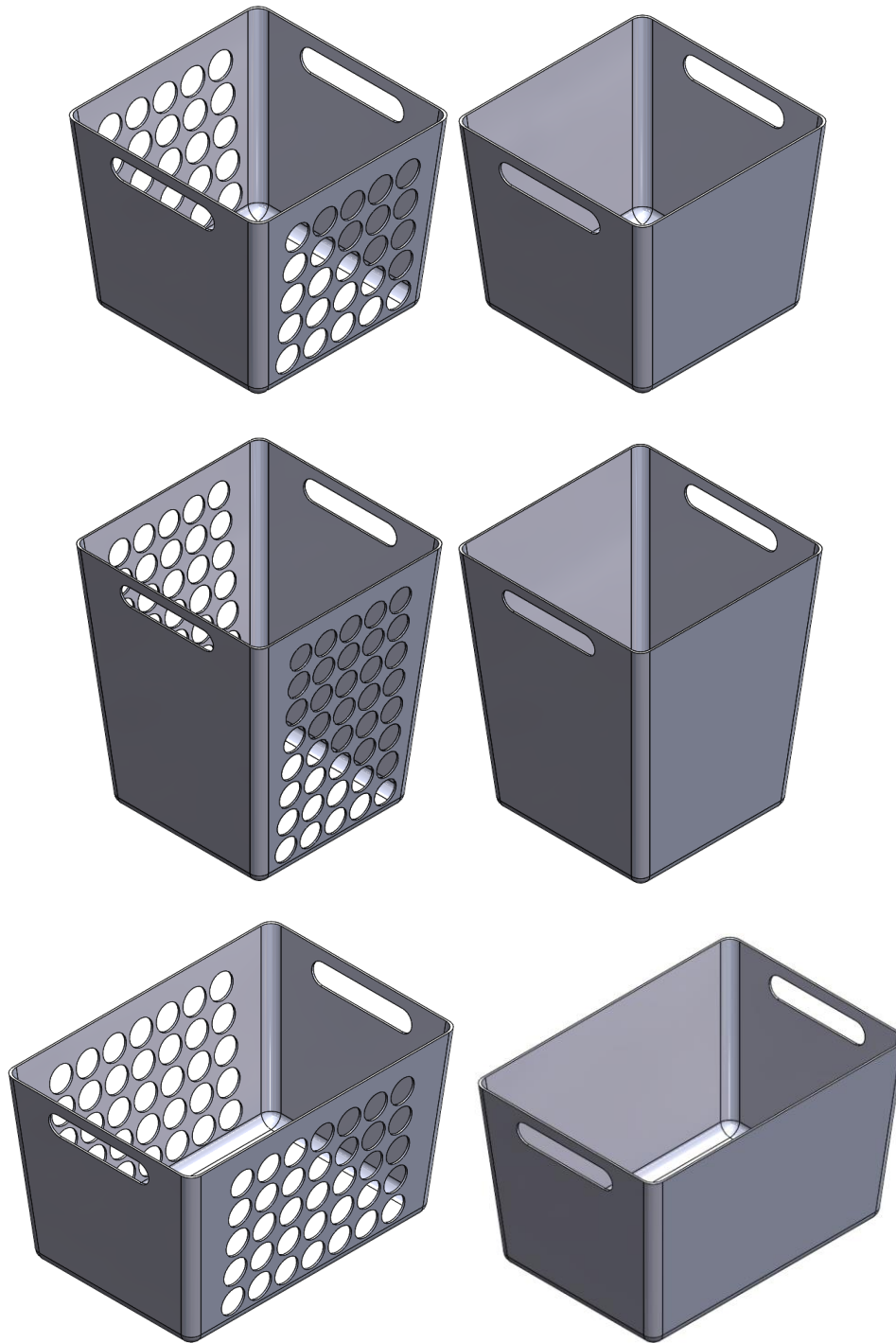
Add the following configurations to the design table. Notice that the previous configuration names have also been changed. Configurations with holes are now marked with a succeeding “H” and configurations without holes have a succeeding “N” to its name. You can either use the keyword “**Suppressed**” and “**Unsuppressed**” or you can just use the letters “U” and “S” to let SolidWorks know if the feature is suppressed or not.

Design Table for: Basket									
	Length@Sketch1	Width@Sketch1	Height@Boss-Extrude1	TaperAngle@Draft2	HoleSize@Sketch2	InstanceNumber_X@LPatte	InstanceNumber_X2@LPatte	InstanceNumber_Y@LPatte	\$STATE@SideCuts
12 x 12 x 12H	12	12	12	5	1.75	3	3	5	U
12 x 18 x 12H	12	18	12	5	1.75	4	4	5	U
12 x 12 x 18H	12	12	18	5	1.75	3	3	8	U
12 x 12 x 12N	12	12	12	5	1.75	3	3	5	SUPPRESSED
12 x 18 x 12N	12	18	12	5	1.75	4	4	5	SUPPRESSED
12 x 12 x 18N	12	12	18	5	1.75	3	3	8	SUPPRESSED

Once you exit the design table, you will notice the new configurations have been added. SolidWorks will prompt you if you would like to delete the old configuration names. You can also manually delete these names.



All the configurations have now been added using the design table. The design table allows for quick changes to any feature or dimension of the basket. Below are the results of the six configurations created in this exercise.



Step 6: Save and Exit

Save the part as **DesignTable_Basket.sldprt** and exit the part.