

HAIR DRYER TUTORIAL

In this project, you will learn how to create a **hair dryer** in SOLIDWORKS.

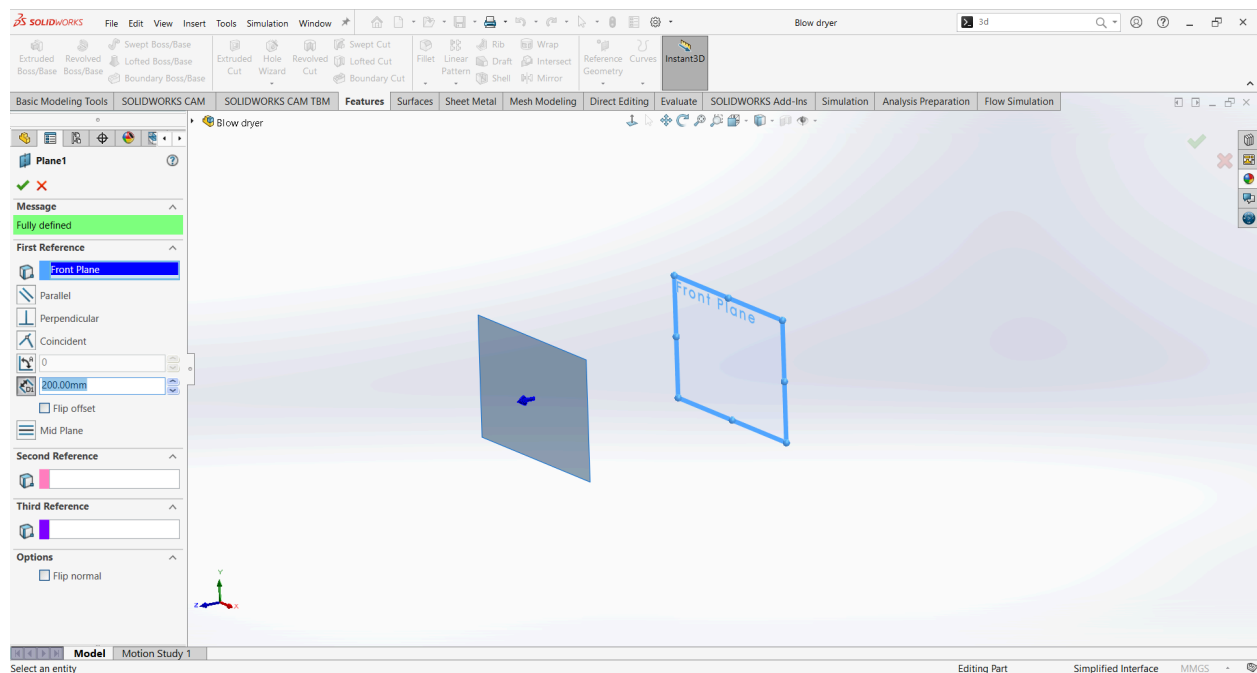
A hair dryer is a good product design example because it combines several different shapes, such as curved bodies, openings, handles, and detailed features.

In this model, you will create the main body, add the handle, shape the front and back sections, and apply finishing details to make the model look more realistic.

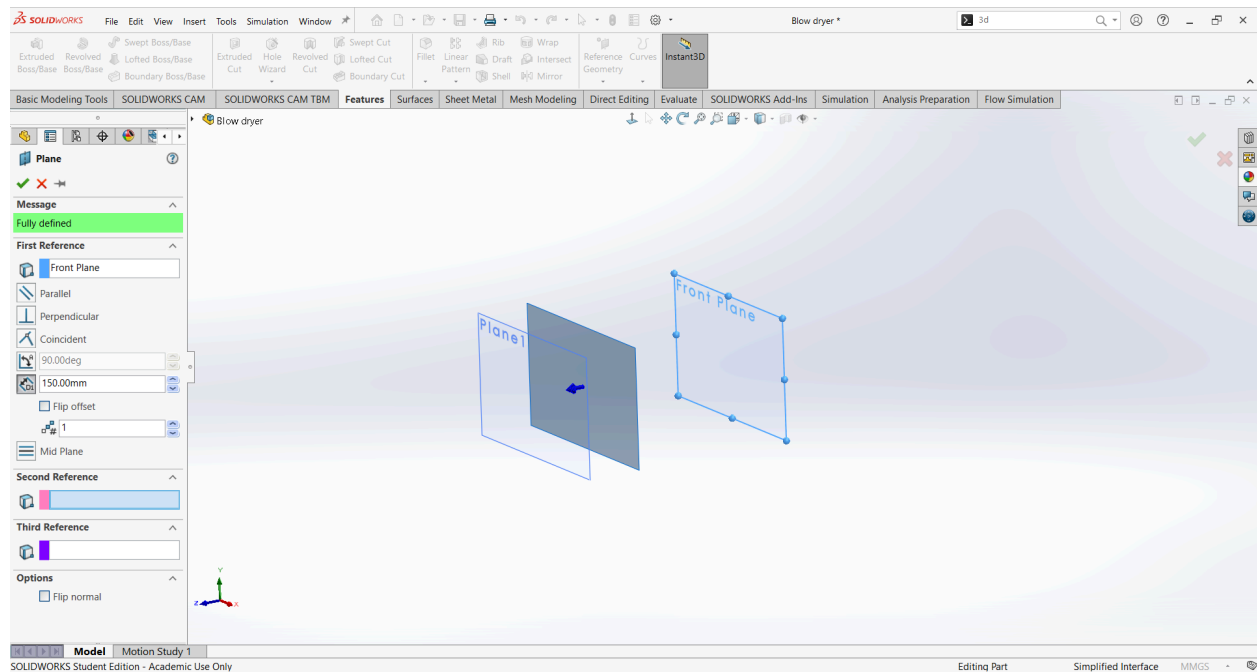
This project will help you practice tools such as **Sketch**, **Extruded Boss/Base**, **Extruded Cut**, **Lofted Boss/Base**, **Fillet**, **Shell**, **Mirror**, and **Pattern**.

The goal is to understand how different SOLIDWORKS features can be combined to create a more realistic consumer product model.

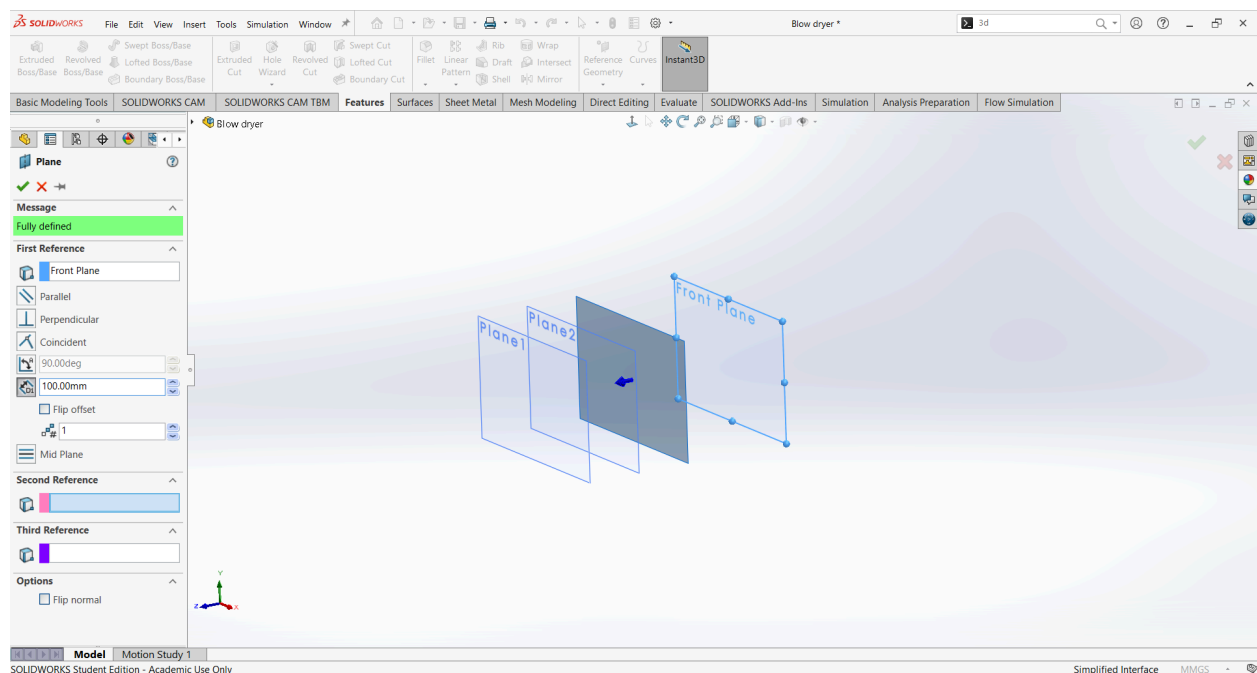
Create **Plane 1** at a distance of **200 mm** from the **Front Plane**. To do this, select the **Front Plane** as the reference. Then set the offset distance to **200 mm**. Make sure **Plane 1** is created in the correct direction, as shown. Click **OK** by selecting the green check mark.



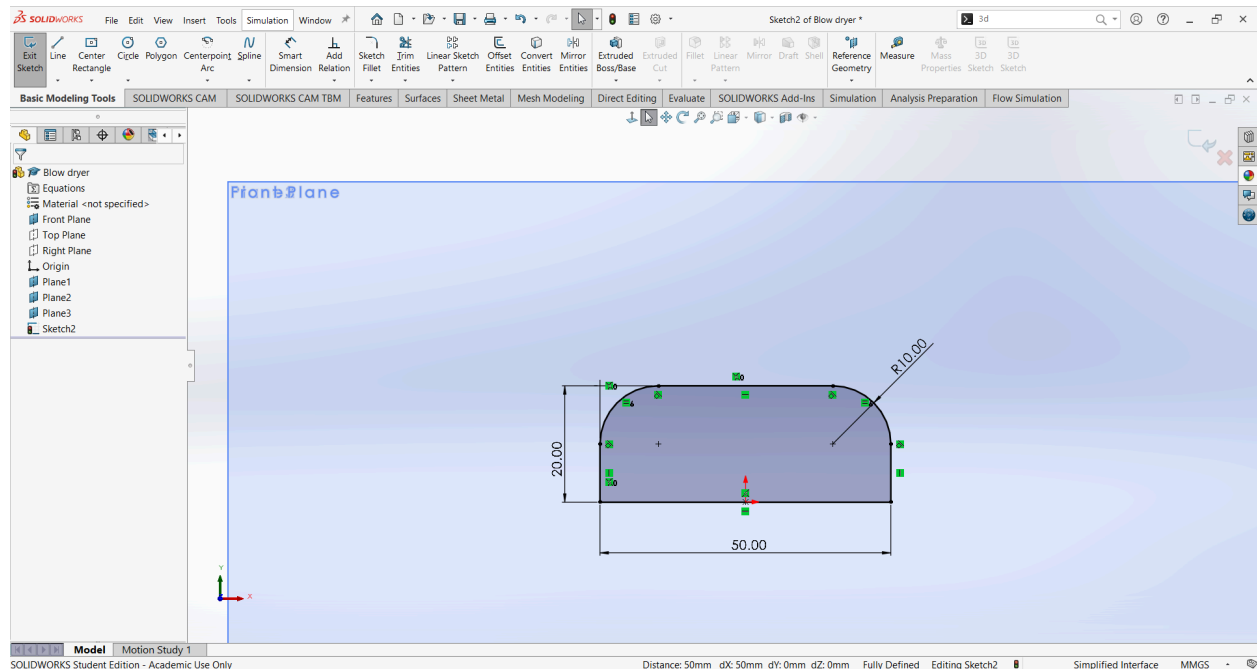
Create **Plane 2** at a distance of **150 mm** from the **Front Plane**. Select the **Front Plane** as the reference. Then set the offset distance to **150 mm**. Make sure **Plane 2** is created in the correct direction, as shown. Click **OK** by selecting the green check mark.



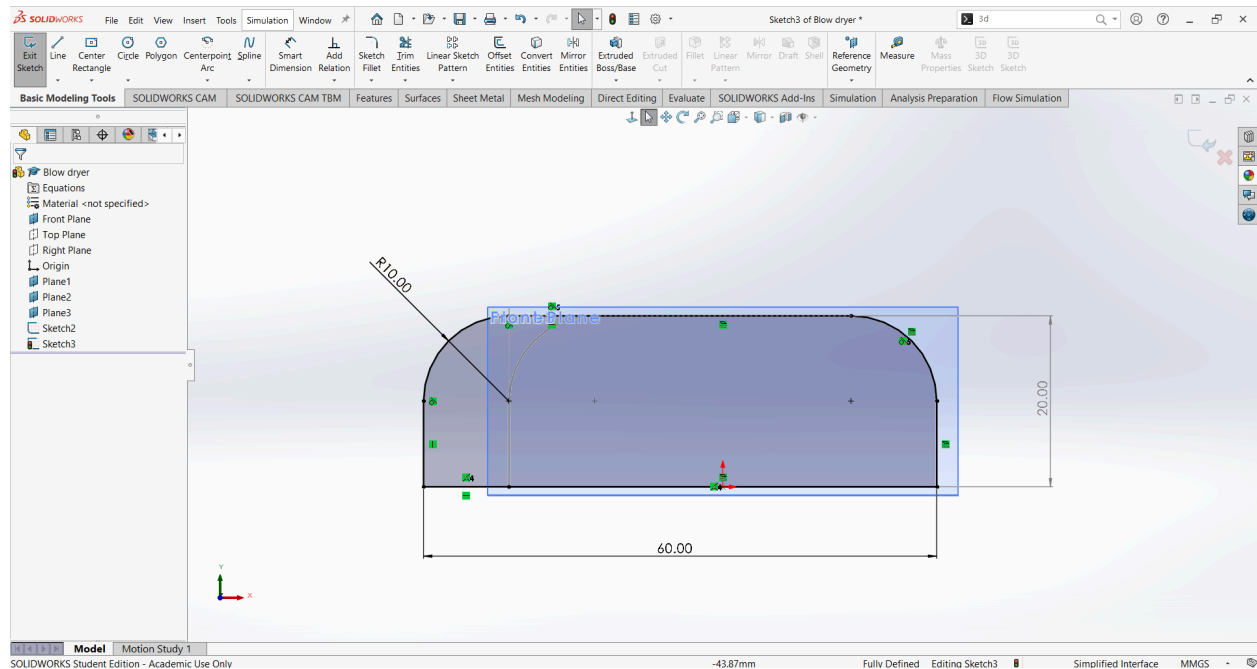
Create **Plane 3** at a distance of **100 mm** from the **Front Plane**. Select the **Front Plane** as the reference. Then set the offset distance to **100 mm**. Make sure **Plane 3** is created in the correct direction, as shown. Click **OK** by selecting the green check mark.



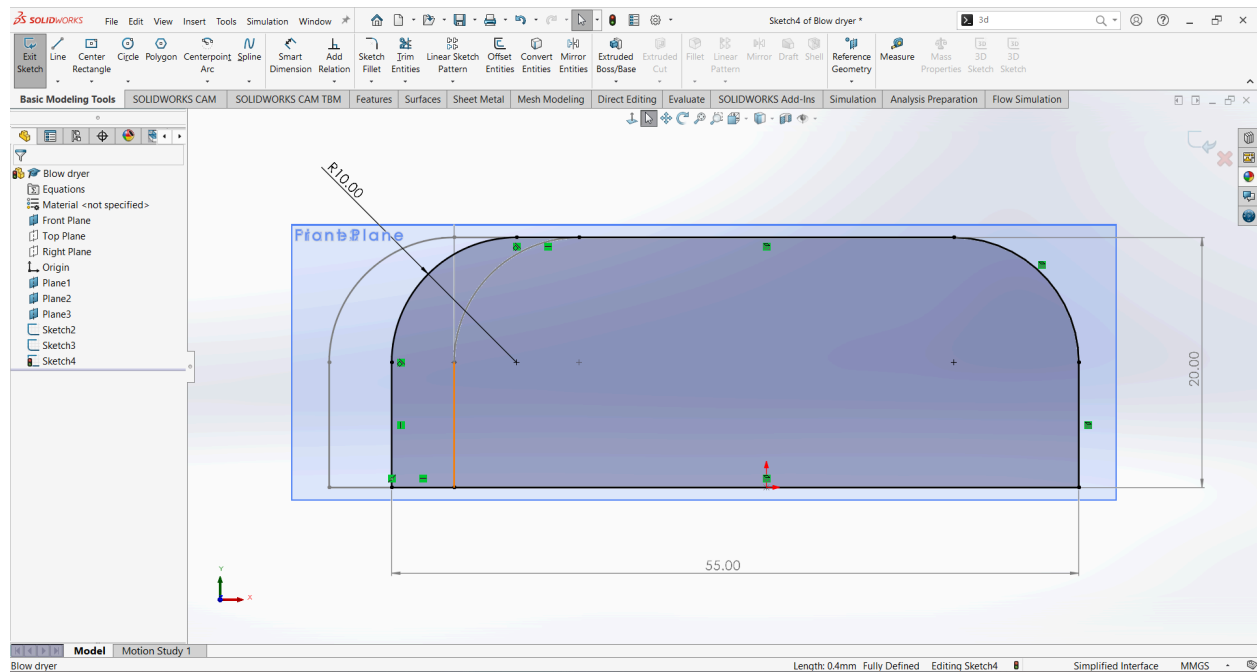
On **Plane 1**, start a new sketch. Create the sketch shown below. Use the required sketch tools, dimensions, and relations to fully define the profile. Make sure the sketch is positioned correctly on **Plane 1**, as shown. This sketch will be used as one of the main profiles for creating the body of the blow dryer.



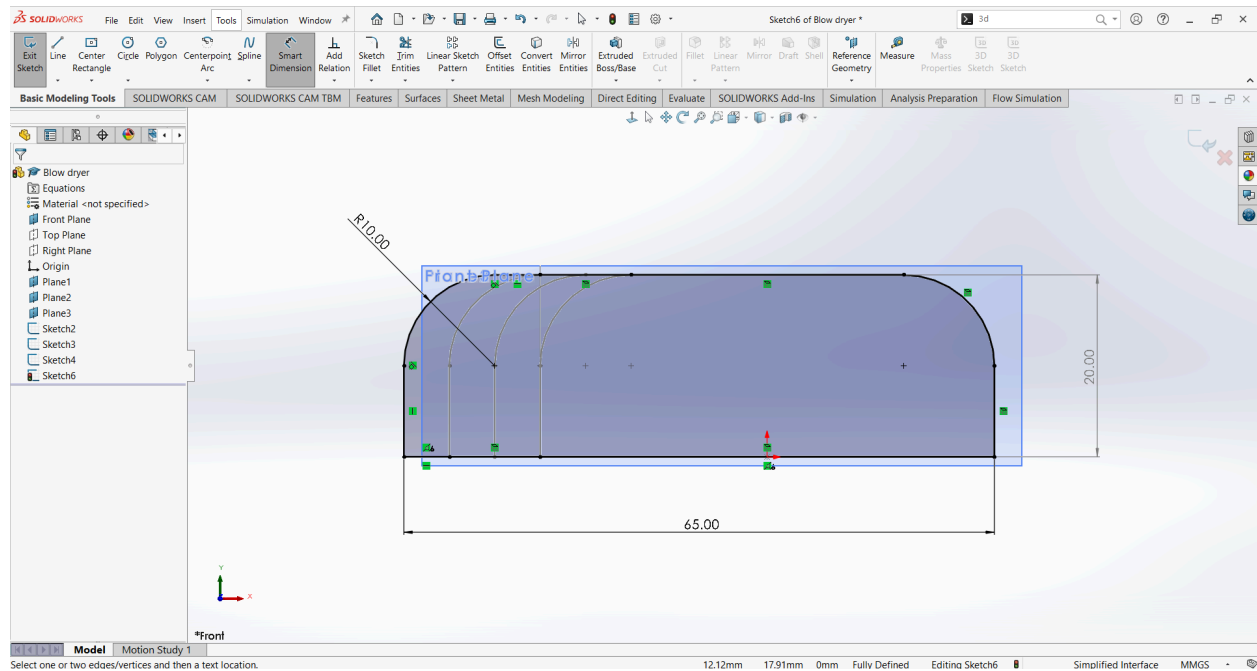
On **Plane 2**, start a new sketch. Create the sketch shown below. Use the required sketch tools, dimensions, and relations to define the profile. Make sure the lower-right point of this sketch is **Coincident** with the same lower-right point from the previous sketch on **Plane 1**. This relation keeps both sketches connected and aligned correctly. This sketch will be used as another profile for creating the main body shape of the blow dryer.



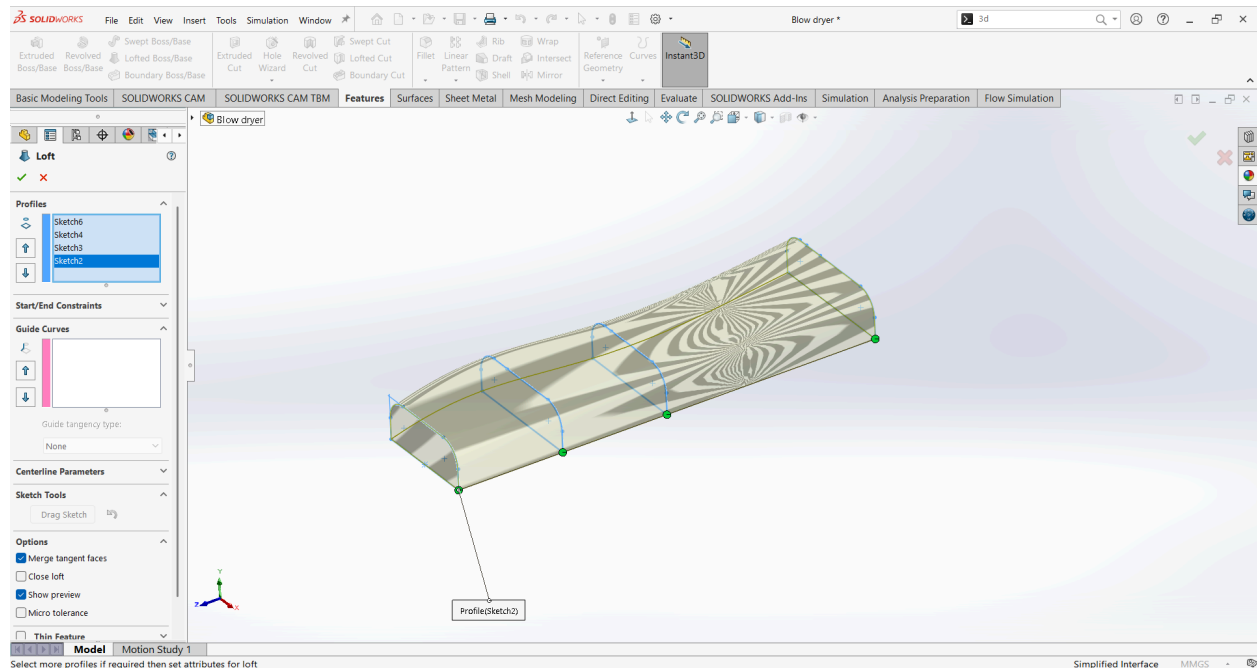
On **Plane 3**, start a new sketch. Create the sketch shown below. Use the required sketch tools, dimensions, and relations to fully define the profile. Make sure the lower-right point of this sketch is **Coincident** with the same lower-right point from the previous sketch on **Plane 2**. This relation keeps the profiles connected and aligned correctly. This sketch will be used as another profile for creating the main body shape of the blow dryer.



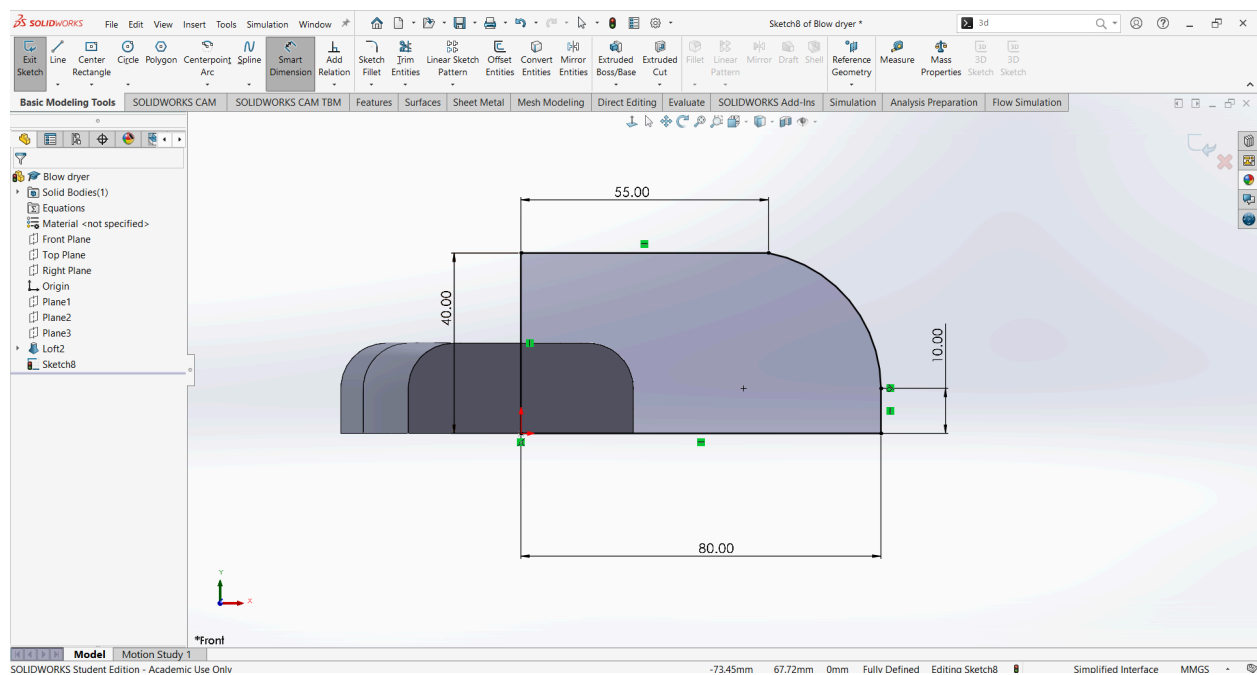
On the **Front Plane**, start a new sketch. Create the sketch shown below. Use the required sketch tools, dimensions, and relations to fully define the profile. Make sure the lower-right point of this sketch is **Coincident** with the same lower-right point from the previous sketch on **Plane 3**. This relation keeps all the profiles connected and aligned correctly. This sketch will be used as the final profile for creating the main body shape of the blow dryer.



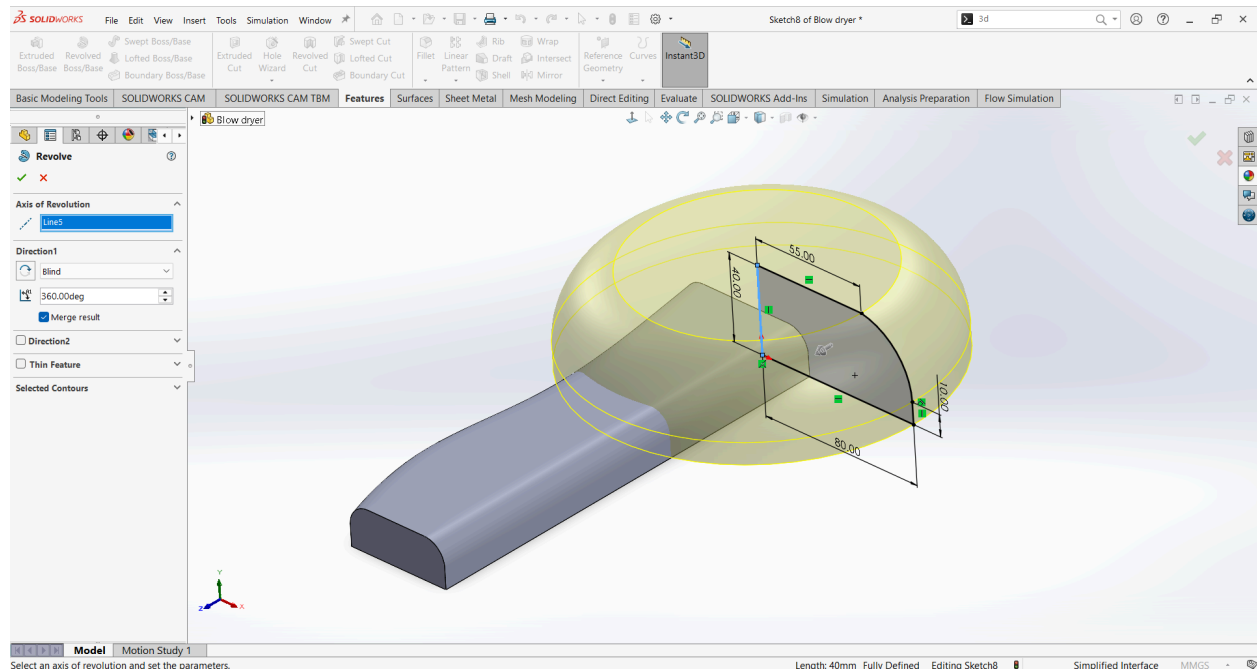
Create the next feature using **Lofted Boss/Base**. Select **Lofted Boss/Base** from the **Features** tab. In the **PropertyManager**, go to the **Profiles** selection box. Select the four previously created sketches as the loft profiles. Make sure you select the sketches in the correct order, starting from the first profile and moving through the remaining profiles. Also, make sure you select each sketch from the same side. This is important because selecting profiles from different sides can twist the loft and create an incorrect shape. After all four profiles are selected correctly, preview the loft shape. If the preview looks correct, click **OK** by selecting the green check mark.



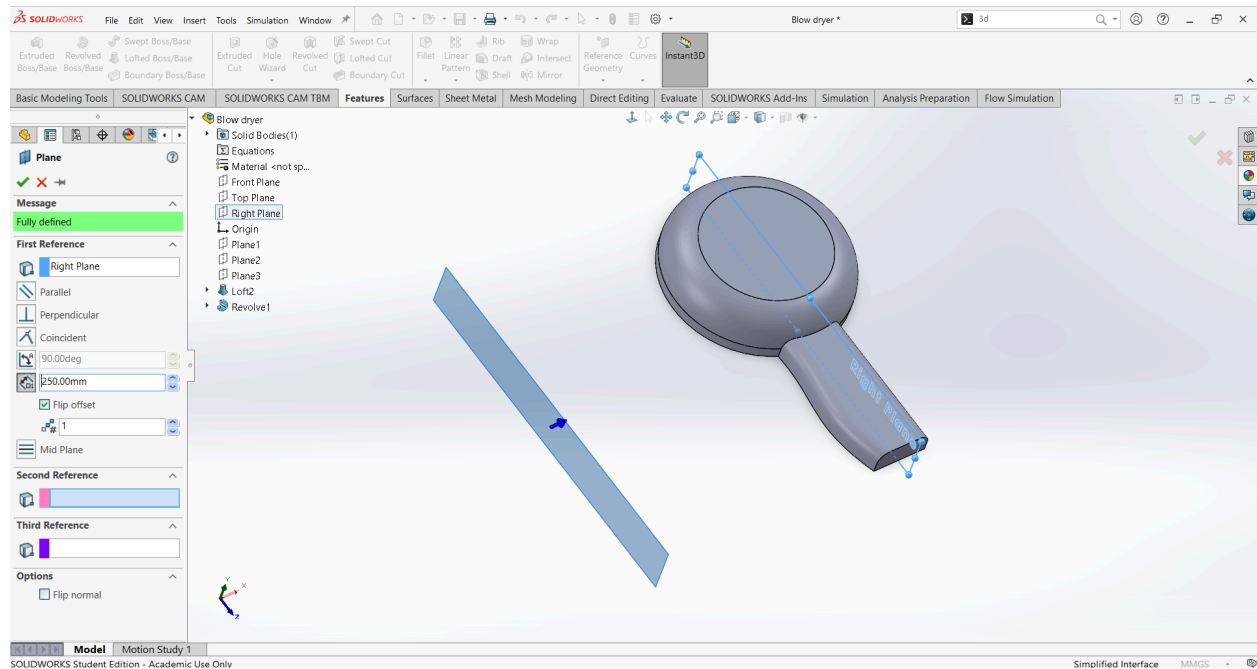
On the **Front Plane**, start a new sketch. Create the sketch shown below. Use the required sketch tools, dimensions, and relations to fully define the profile. Make sure the sketch is positioned correctly, as shown. This sketch will be used to create the next feature of the blow dryer.



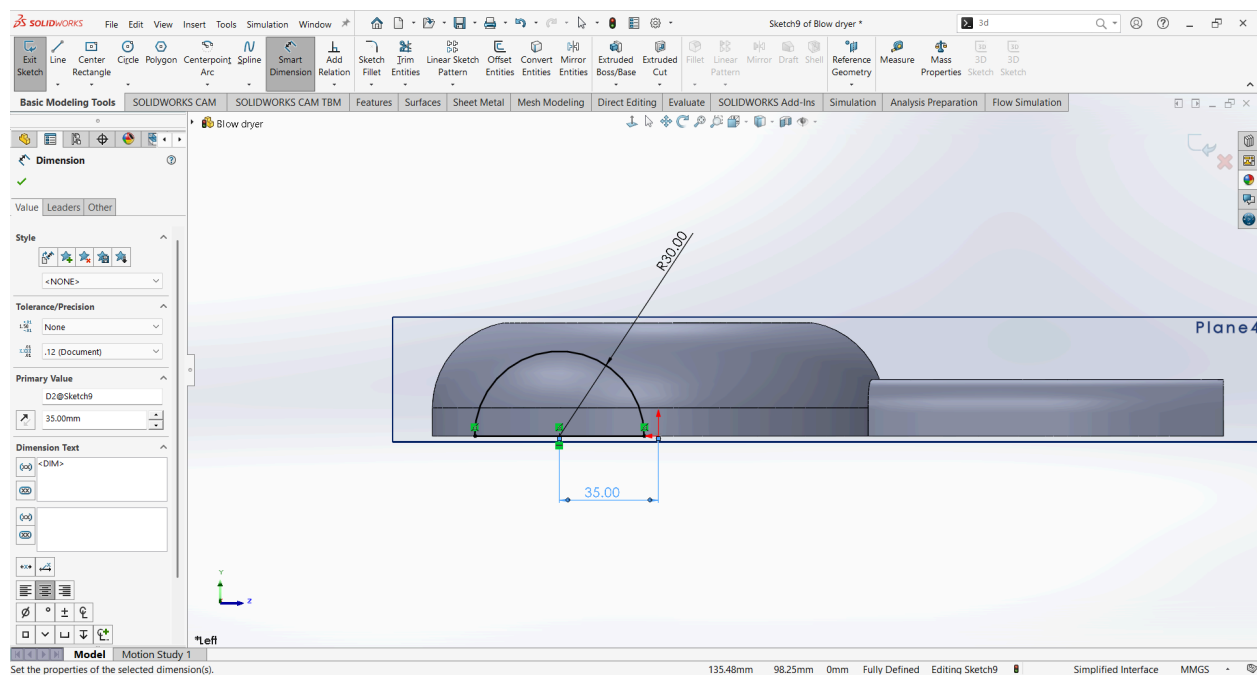
Use the previously created sketch to create a **Revolved Boss/Base**. Select **Revolved Boss/Base** from the **Features** tab. In the **PropertyManager**, select the sketch profile you want to revolve. Then select the line that passes through the origin as the axis of revolution. Set the rotation angle to **360°**. This will rotate the sketch completely around the selected line and create the revolved shape. Click **OK** by selecting the green check mark.



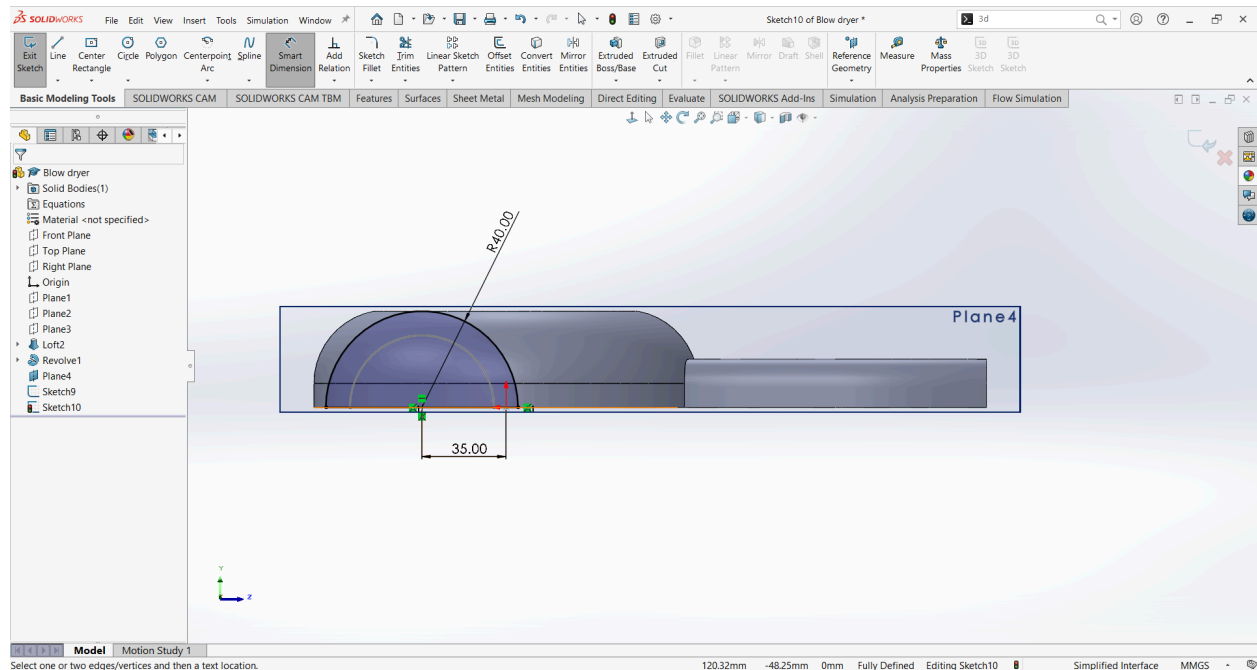
Create **Plane 4** at a distance of **250 mm** from the **Right Plane**. Select the **Right Plane** as the reference. Then set the offset distance to **250 mm**. Make sure **Plane 4** is created in the correct direction, as shown. Click **OK** by selecting the green check mark.



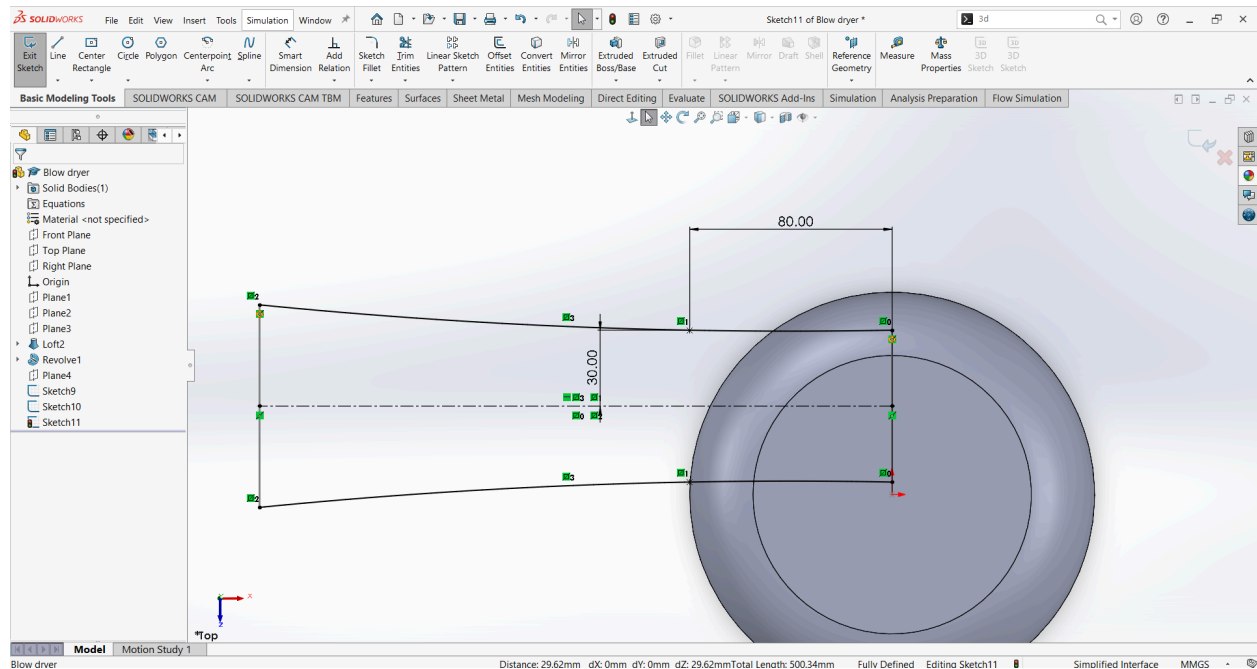
On the **Right Plane**, start a new sketch. Create the sketch shown below. Use the required sketch tools, dimensions, and relations to fully define the profile. Make sure the sketch is positioned correctly on the **Right Plane**, as shown. This sketch will be used to create the next feature of the blow dryer.



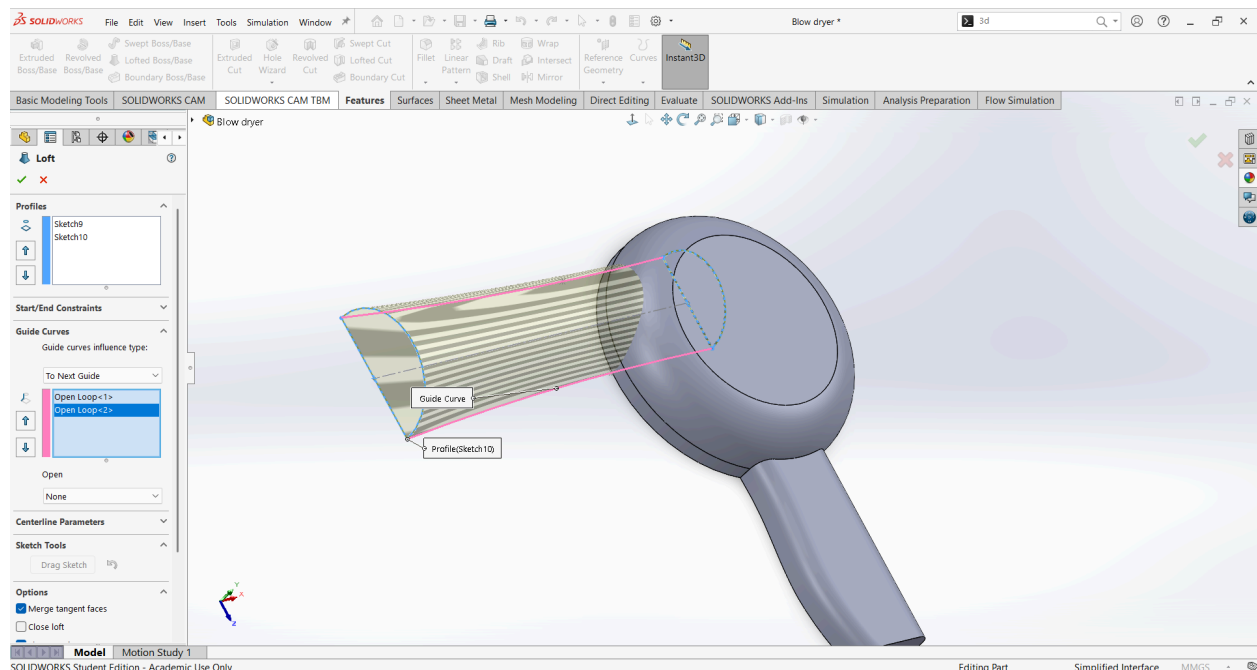
On **Plane 4**, start a new sketch. Create the sketch shown below. Use the required sketch tools, dimensions, and relations to fully define the profile. Make sure the sketch is positioned correctly on **Plane 4**, as shown. This sketch will be used together with the previous sketch to create the next feature of the blow dryer.



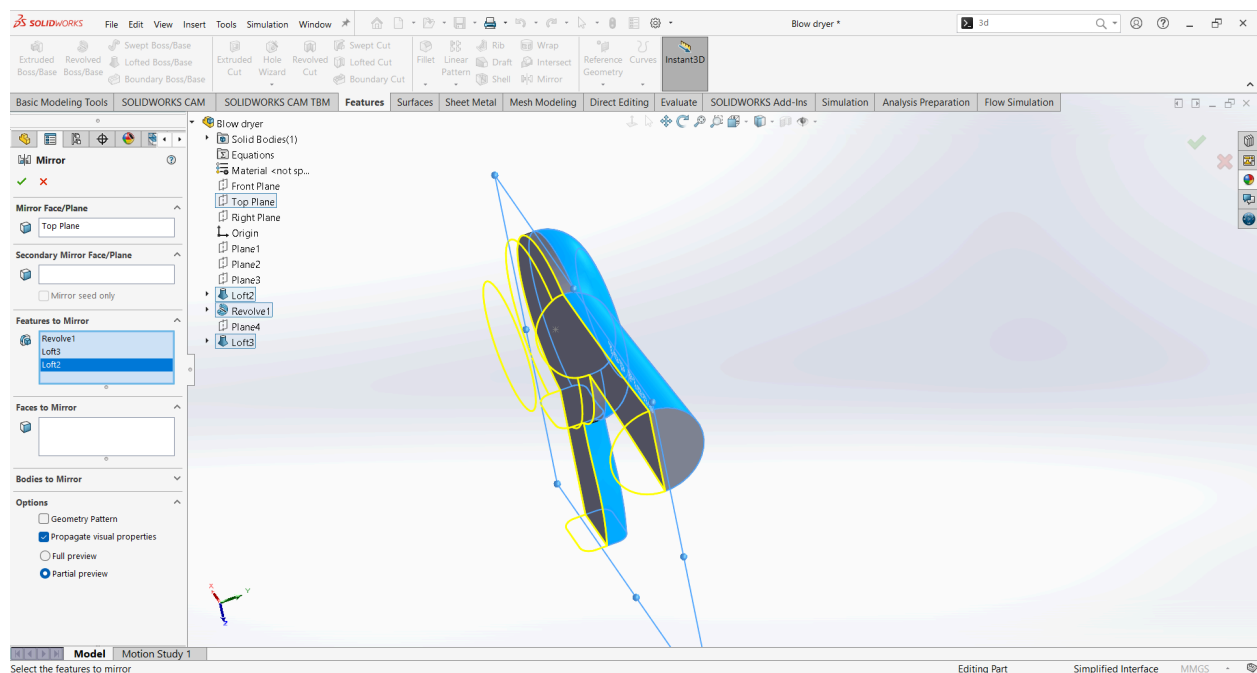
On the **Top Plane**, start a new sketch. Create the two **Spline** curves shown below. You can create the first spline manually, and then use **Mirror Entities** to create the second spline on the opposite side. Make sure both splines are positioned correctly and are symmetrical. Then define a **Pierce** relation between the start and end points of the splines and the previously created **Sketch6** and **Sketch7**. This relation connects the splines to the profile sketches and ensures that the guide curves follow the correct path. These splines will be used as guide curves for the next loft feature.



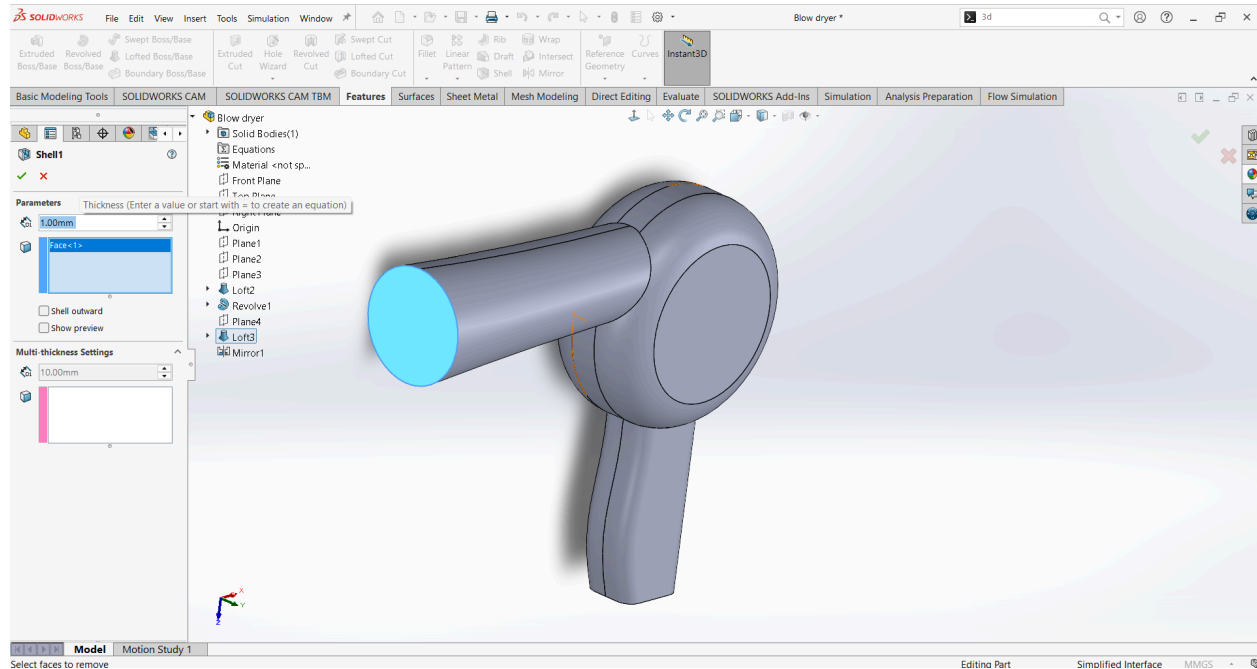
Use **Lofted Boss/Base** to create the next feature. Select **Lofted Boss/Base** from the **Features** tab. In the **PropertyManager**, go to the **Profiles** selection box. Select **Sketch6** and **Sketch7** as the loft profiles. Make sure the profiles are selected in the correct order and from the same side to avoid twisting the loft. Then go to the **Guide Curves** selection box. Select the two previously created **Spline** curves as the guide curves. The guide curves will control the shape of the loft between **Sketch6** and **Sketch7**. Check the preview to make sure the shape looks correct. Then click **OK** by selecting the green check mark.



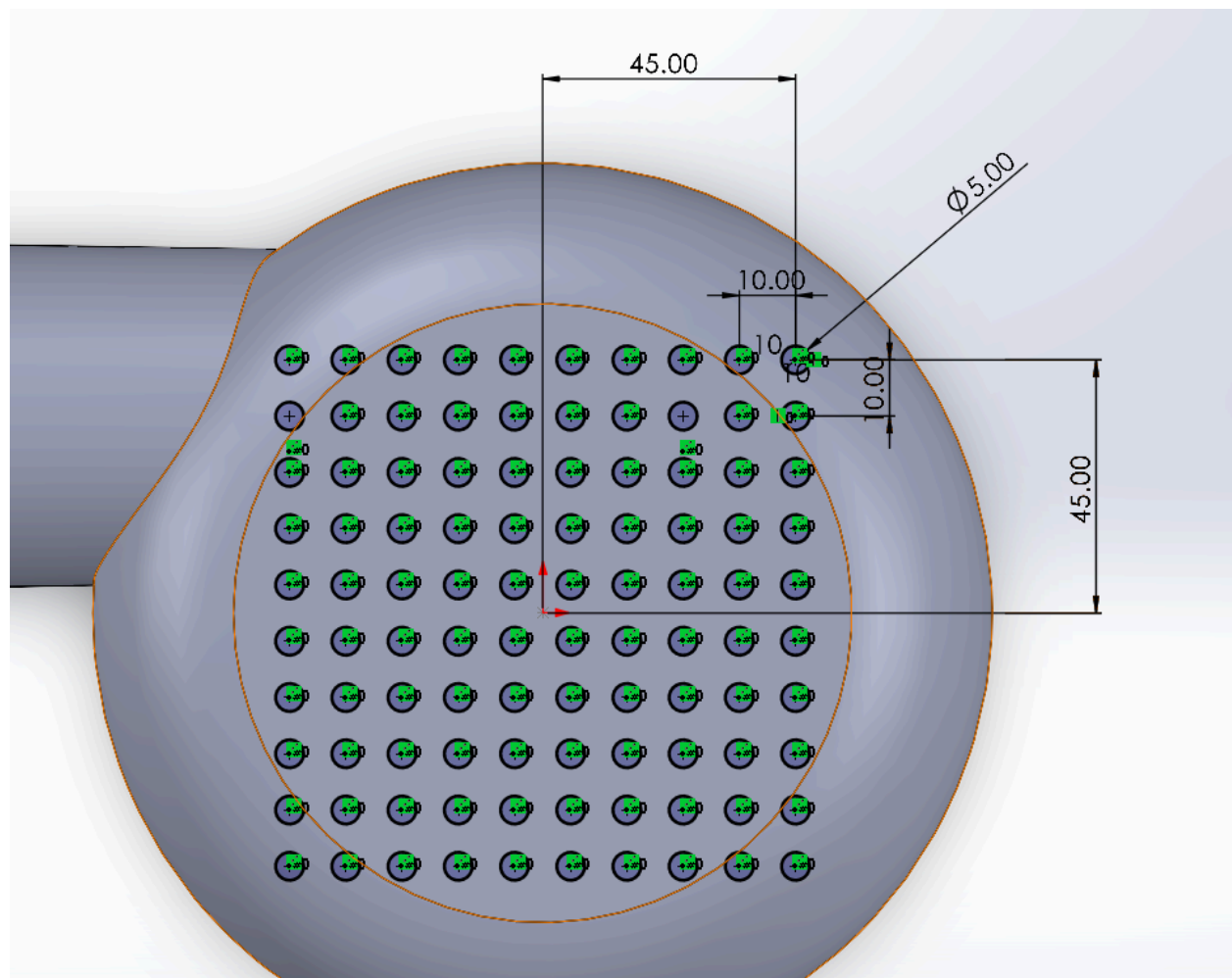
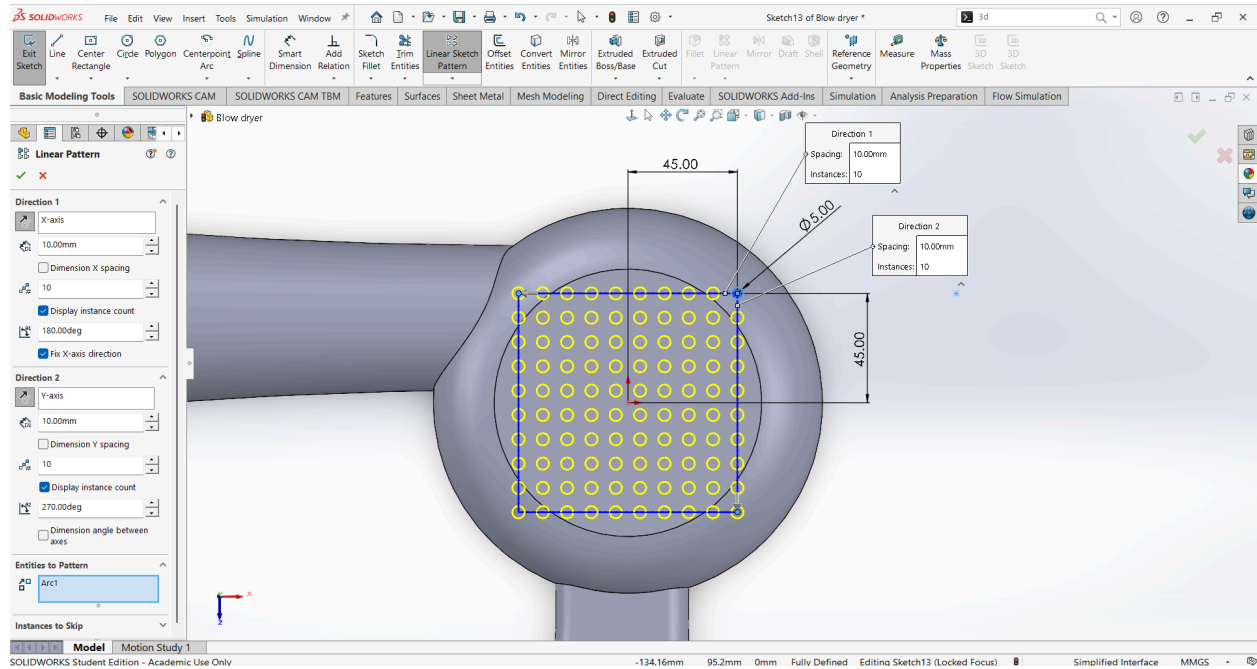
Use the **Mirror** tool to mirror all the created features about the **Top Plane**. Select **Mirror** from the **Features** tab. In the **PropertyManager**, select the **Top Plane** as the **Mirror Face/Plane**. Then select all the features you want to mirror. Make sure all required features are included in the selection. This will create a mirrored copy of the selected features on the opposite side of the **Top Plane**. Click **OK** by selecting the green check mark.



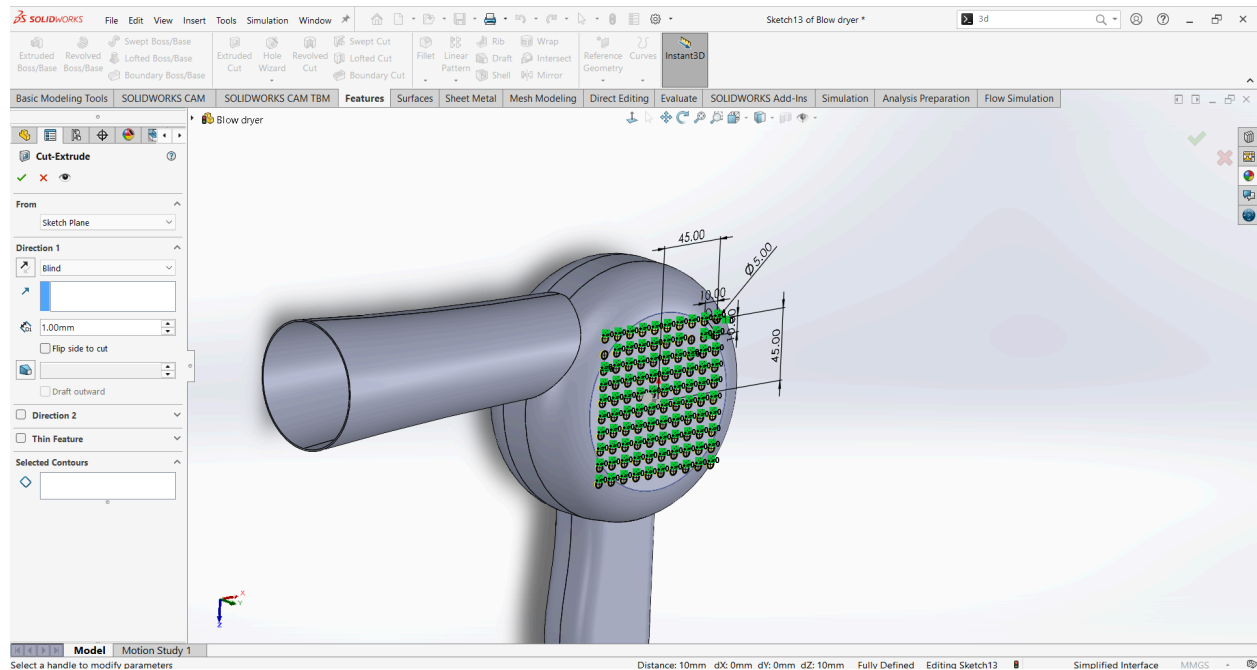
Use the **Shell** tool to hollow out the model. Select **Shell** from the **Features** tab. In the **PropertyManager**, set the shell thickness to **1 mm**. Under **Faces to Remove**, select the face shown. This will remove the selected face and create a hollow body with a wall thickness of **1 mm**. Check the preview to make sure the shell is created correctly. Click **OK** by selecting the green check mark.



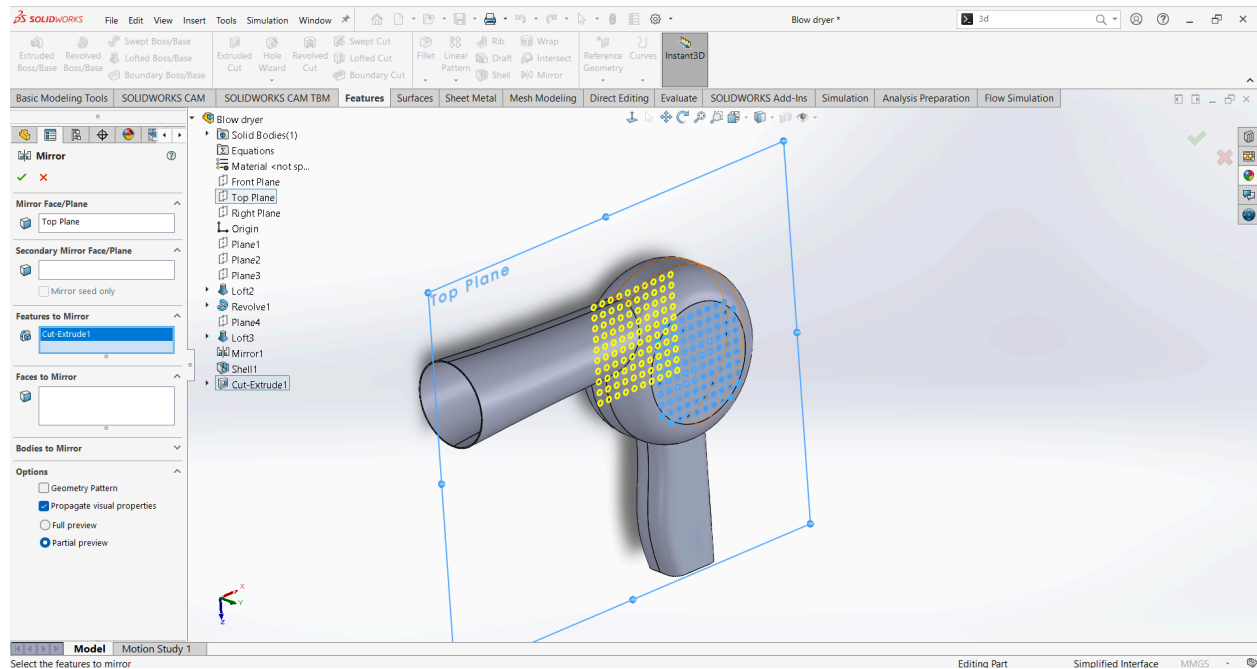
On the top face of the model, start a new sketch. Select the **Circle** tool. Create one circle in the position shown. Then use **Linear Sketch Pattern** to repeat the circle. In the **PropertyManager**, set the first direction to the **X direction**. Set the spacing distance to **10 mm**. Set the number of instances to **10**. Then activate the second direction and set it to the **Z direction**. Set the spacing distance to **10 mm**. Set the number of instances to **10**. This will create a pattern of **100 circles** in total, arranged as **10 circles in the X direction** and **10 circles in the Z direction**. Click **OK** by selecting the green check mark.



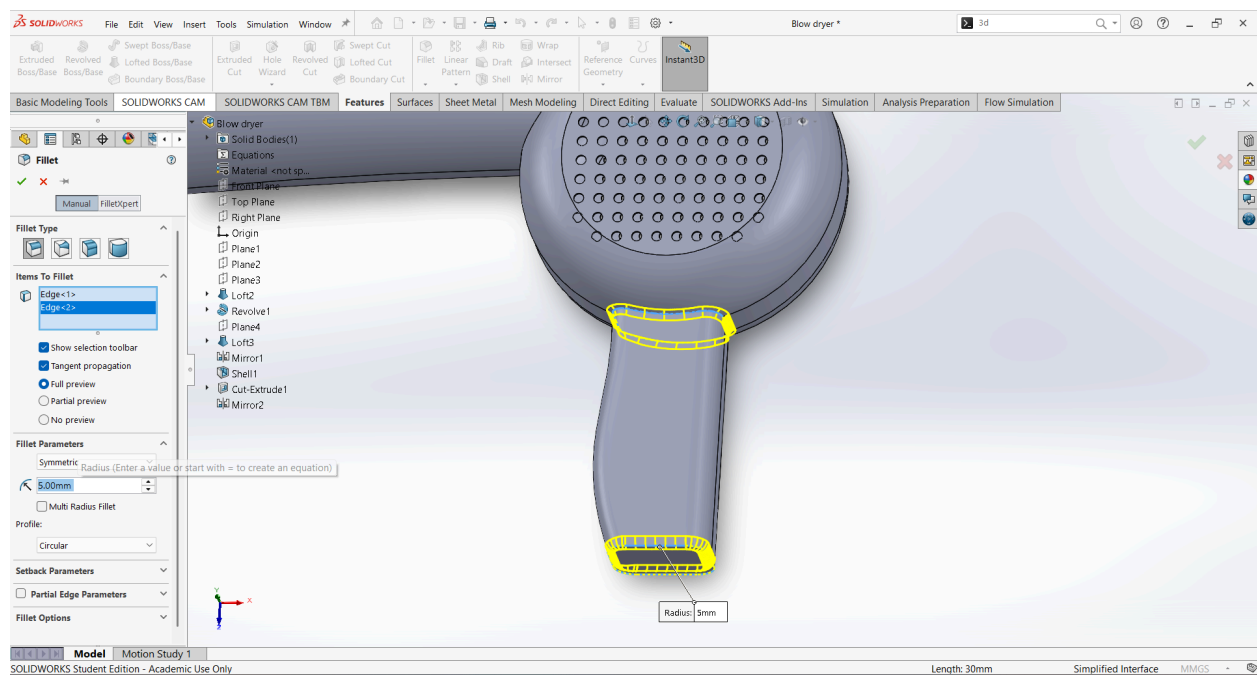
Use the previously created sketch to create an **Extruded Cut**. Select **Extruded Cut** from the **Features** tab. In the **PropertyManager**, set the cut depth to **1 mm**. Make sure the cut direction is correct, as shown. This will cut the circular pattern **1 mm** into the top face of the model. Click **OK** by selecting the green check mark.



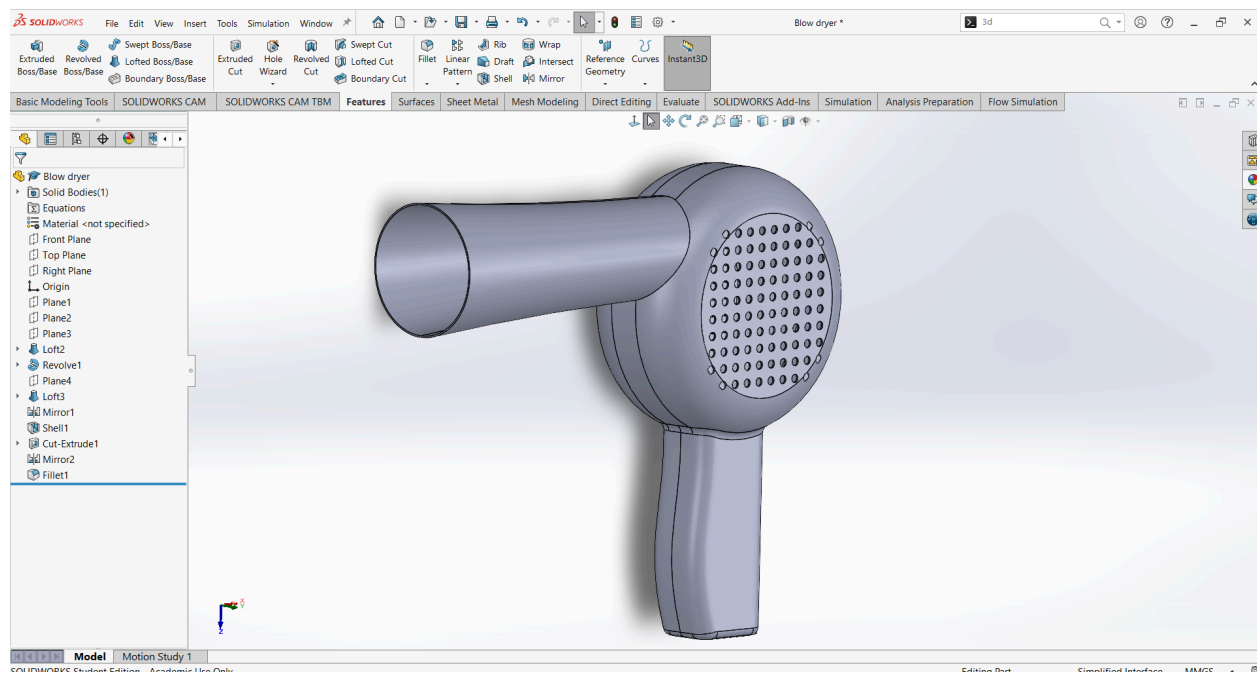
Use the **Mirror** tool to mirror the previously created **Extruded Cut** about the **Top Plane**. Select **Mirror** from the **Features** tab. In the **PropertyManager**, select the **Top Plane** as the **Mirror Face/Plane**. Under **Features to Mirror**, select the previously created **Extruded Cut** feature. This will create the same cut pattern on the opposite side of the model. Check the preview to make sure the mirrored cut is positioned correctly. Click **OK** by selecting the green check mark.



Create a **Fillet** on the selected edges. Select the **Fillet** tool from the **Features** tab. In the **PropertyManager**, set the fillet radius to **5 mm**. Then select the edges shown. This will round the selected sharp edges and make the model smoother and more realistic. Check the preview to make sure the fillet is applied correctly. Click **OK** by selecting the green check mark.



This is the final result.



If you want to dive deeper into SolidWorks, you can check out my full courses on the discount: cadandengineering.com/solidworks