Activity 1 – Modeling a Cast Part

In this activity, you will construct the model of a casting. When completed, your model should look like the following illustration. While building this model, take time to note the techniques used for obtaining the features.

1. Create a metric part file.

2. Create a global parallel reference plane 120.4 mm parallel to the reference plane shown.

3. Create a global reference plane at a 10-degree angle to the plane created in the previous step.
4. Create a Sketch on the first global plane you created. This sketch will represent the center of the plane. Include the edge of the reference plane shown in the profile environment for the sketch profile.

Note:
This is done because you only know the angle of the plane and the distance between the center of the part and the center of the mounting boss. When you get into the profile environment on the angled plane, you have no way of determining exactly where the center of the plane is located. This sketch locates the exact center of the plane.

5. Create a Protrusion for the angled mounting boss on the angled global reference plane. Make the extent symmetric at 35 mm. Position the circle at the midpoint of the sketch draw on the first parallel global plane.
Tip:
Sometimes it is convenient to model the critical design areas of the part first. In this case, the critical areas are the mounting holes. You will start by modeling the material in which the mounting holes will be placed.

6. Mirror the protrusion as shown in the illustration.

7. Using the Revolved Protrusion command, create the final mounting feature as shown.
8. Next, create the 25-degree symmetric revolved connecting body of the model. The profile and resulting model are shown in the illustration.
Note the progress of the model in the following illustration.

This is the model at this point in the construction.
9. To construct the support arm portion of this model, you need a path for a swept feature. Construction surfaces can help create the path. However, before you construct these surfaces, you need some sketches for exact location of the construction surfaces. The following illustrations provide details for the two sketches.

![Sketch and corresponding plane](image)

![Sketch and corresponding plane](image)

10. On the Constructions toolbar, use the Extruded Surfaces command to project construction surfaces from these sketches. Use symmetric extents long enough to insure their intersection.
11. On the Constructions toolbar, use the Intersection Curve command to create an intersection curve from these two surfaces. This will be the path for a swept protrusion that will connect the material.

12. Using the intersection curve as the sweep path and a simple circle (26 mm diameter) at one end point of the sweep path, create the swept protrusion as shown.

Note the progress of the model shown in the following illustration.
13. Create the revolved cutout as shown. The angle of the sweep must go completely through the part.

14. Construct an offset surface from the top surface of the part as shown in the following illustration. The offset goes into the part 8 mm. This construction surface will be used for a cutout extent that will be constructed in the following steps.
15. Construct the cutout shown in the following illustration. Use the construction surface created as the ‘To’ extent.

16. Add the three mounting holes.
17. Add the large rounds (15 mm) at the ends of the swept feature as shown (both sides of the model).

18. Apply 2 mm rounds to the edges as shown in the following illustration.
19. Apply 2 mm chamfers to the edges as shown.
The following illustration shows the completed model.
20. Save and close the file. This completes the activity.
Activity 2 – Modeling a Plastic Part

In this activity, you will model a plastic part. When completed, your plastic part should look like the following two illustrations. While building this model, take time to note the techniques used for obtaining the features.
1. Create a new metric part file.

2. Create a 125 mm wide protrusion that extends symmetrically from the reference plane. The profile and resulting feature are shown in the following illustrations.

![Diagram showing the creation of a 125 mm wide protrusion.]

3. Create a sketch that consists of two straight lines. These lines represent the start point and end point of the dome-shaped swept protrusion you will construct later in the activity.
4. Model two construction surfaces from the previous sketch. Be sure the construction surfaces completely intersect the solid model.

5. Turn off the sketches and click the Intersection Curve command to find the intersection of the construction surfaces with the solid body. In the model shown in the illustration, construction surfaces are turned off – only the solid and the intersection curves are shown.
6. Construct a sketch on a parallel plane at the endpoint of the intersection curve you just created. Include the intersection curve (not the reference plane) as the base of the profile. You may have to zoom-in to make sure you select the intersection curve and not the reference plane. They do not occupy the same location.

Tip:
On the ribbon bar, use the Keypoints option to place the parallel plane exactly at the endpoint of the intersection curve.

7. Construct a sketch on another parallel plane at the other intersection curve endpoint. The profile must be connected to the included intersection line and not the model edge.
8. Add a sketch on a plane that is parallel to the reference plane shown and intersects the right corner of one of the sketches. The result is shown in the following illustration.
9. Add a swept protrusion to the model as shown.

10. Create the sketch as shown. This will be used as cross-section1 in a swept cutout.
11. Create the sketch as shown. This will be used as cross-section 2 in a swept cutout.

12. Construct a Swept Cutout as shown.
13. On a reference plane parallel to and 15 mm above reference plane A, add the sketch shown in the following illustration.
14. Offset a construction surface as shown.
15. Construct a cutout as shown. Use the reference plane on which the sketches were drawn, and use the offset surface as the From/To extent for the cutout.

16. Add the three counterbore holes as shown. The hole diameter is 3 mm, counterbore diameter is 5 mm, and counterbore depth is 1.5 mm.
17. Construct a circular Pattern of the holes you placed in the previous step. Pattern the two holes located on the outside corners, not the middle hole. The following illustration provides the remaining details.
18. Add 8 mm rounds to the four edges circled in the illustration.
19. Add a 1.5 mm thin wall to the model, leaving the bottom face open as shown.

![THINWALL diagram](image)

(1.5mm - bottom face open)

20. Add a reference plane 15 mm below and parallel to reference plane A.

![Reference plane A and parallel reference plane](image)

Parallel reference plane - 15 mm below plane A

Reference Plane A

21. On the plane you created in the previous step, construct the sketch shown in the illustration.
22. Offset a construction surface as shown.

23. Construct a cutout as shown in the following illustration.
24. Apply the 2 mm round as shown in the following illustration.
25. The following images are of the completed model.
26. Save and close the file. This completes the activity.