

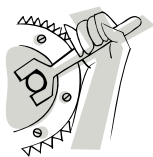
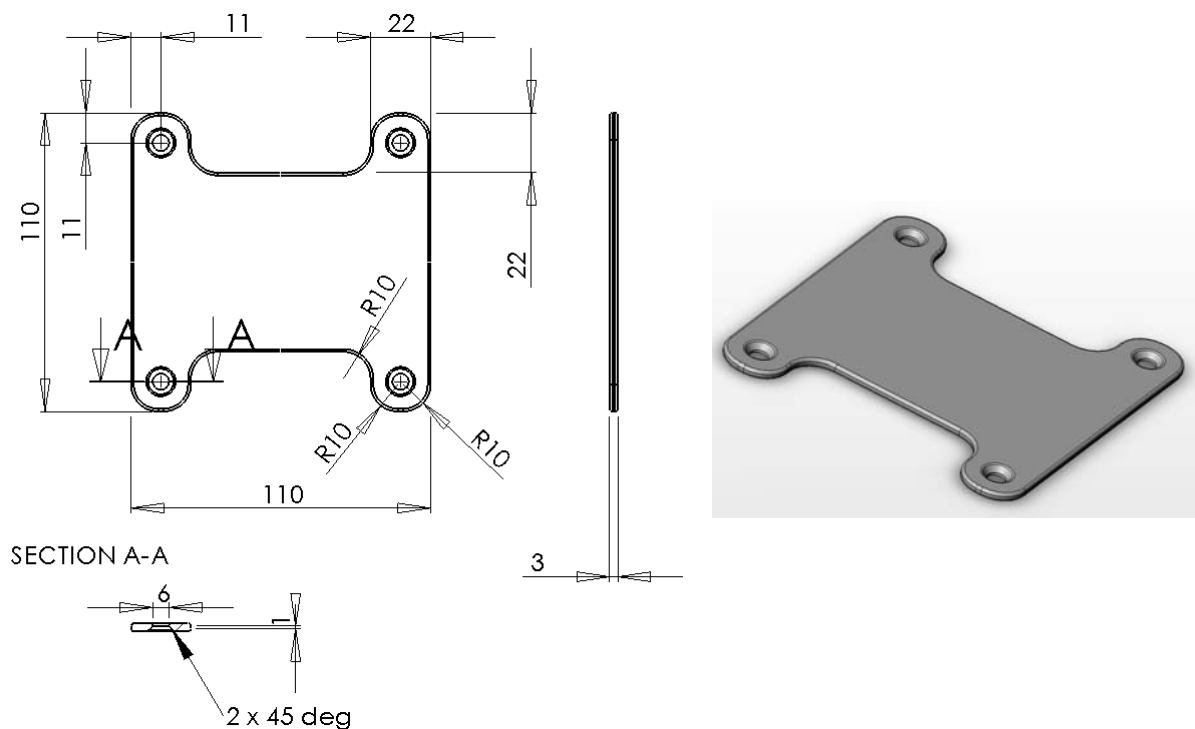
Casting Design Exercises– Basic Skills 1: Extruding and sketching

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What is new in this exercise?

- Sketch tool
- Extruded Boss/Base tool
- Extruded Cut tool
- Swept Boss/Base tool
- Chamfer tool
- Fillet tool
- Shell tool

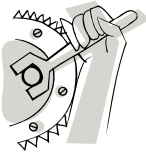
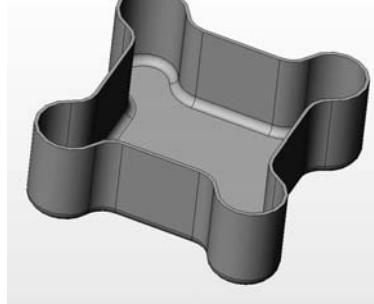
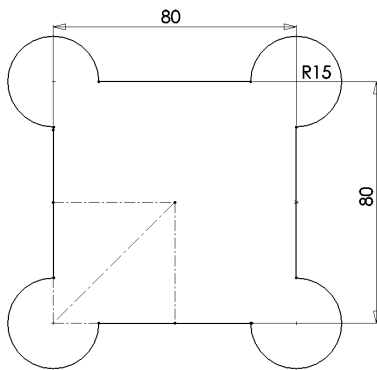
Exercise 1.1



To do:

The model is a flat, symmetric part with a chamfered hole in each corner. Produce the model by using Extruded Boss/Base tool, Extruded Cut tool and Chamfer tool. As an option make all edges round with 1 mm radius fillet. See the finished example model. Some of the edges are not filleted. Why?

Exercise 1.2

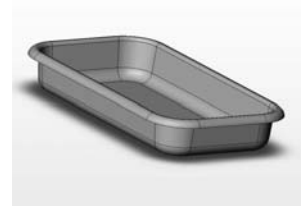
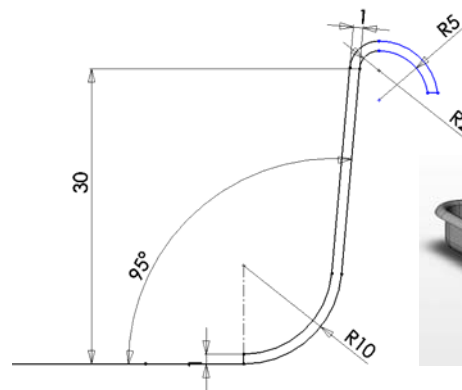
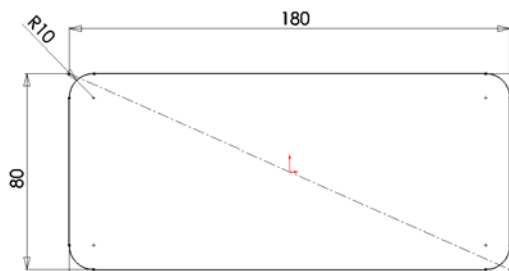


To do:

The model is a symmetric, cup-shaped part with rounded (filleted) corners in sides and in the bottom. There are different ways to prepare this part.

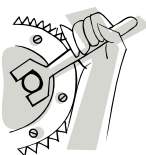
Take these steps: Sketch the shape shown in the figure above, extrude it, fillet vertical edges and bottom edges and finally make a 1,5 mm thick shell from the extruded and rounded solid. You can also produce the part by sketching the shape with already rounded corners, extruding, rounding the bottom and finally producing the shell.

Exercise 1.3



Trajectory (path of the sweep).

Profile for the cup wall, or profile to sweep along the trajectory. Sketch the profile so that it is coincident with the trajectory.

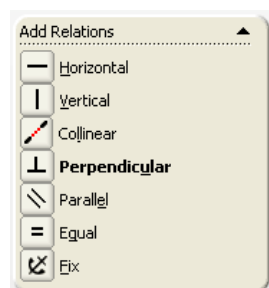
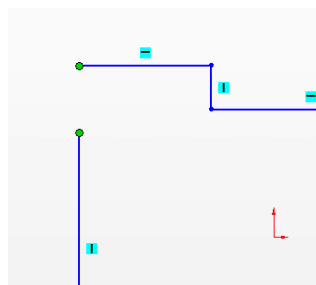
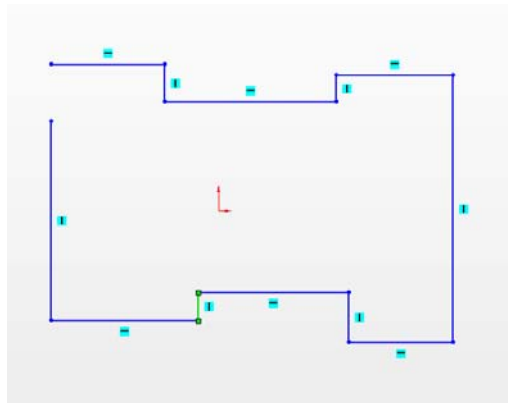
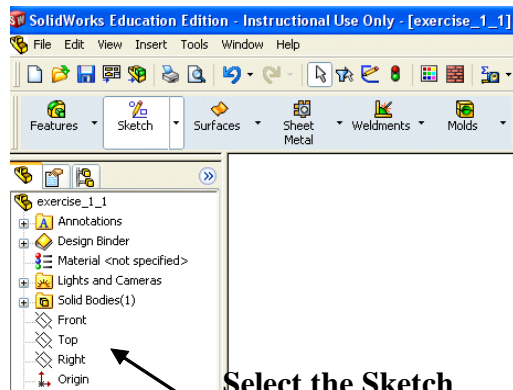


To do:

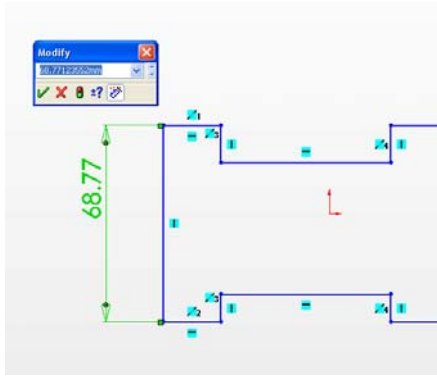
The model is a symmetric cup-shaped part with a collar around. Produce the model by using Swept Boss/Base and Extruded Boss/Base tools.

Instructions

Sketching

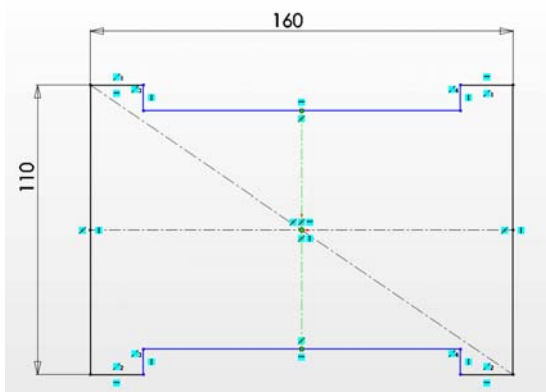
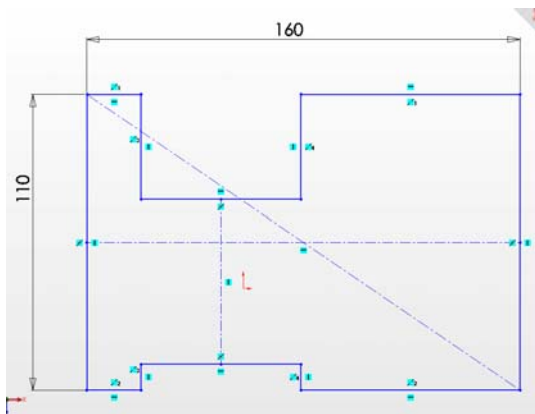


1. Start the Sketch tool by selecting the Sketch commands from the upper left corner of the SolidWorks window. A Sketch palette opens to the upper right corner of the window.
2. Select a plane for sketching. The plane can be one of the model faces, a reference plane or one of the basic planes: Front, Top or Right plane. Select the plane from the feature tree on the left side of the screen.
3. Sketch something, which looks like the desired shape. See figure on the left.
4. Notice the hole in the upper left side of the sketch. Do not put an extra line there. Use one of the sketch entity relations (Merge) to close the sketch.
5. Select the endpoints from both of the lines you wish to close. Use Control (Ctrl) key to select more than one entity. If you select one or more sketch entities a Properties window appear to the left side of the SolidWorks window into the place of the feature tree. In the Properties window you can see the relations and other properties, which are possible to change in the selected sketch entity. In this case select Merge and the two endpoints come together and close the sketch.
6. Use other sketch entity relations to make your sketch look more like the final shape. Typical sketch entity relations are: Horizontal, Vertical, Colinear, Perpendicular, Parallel, Equal, Midpoint and Coincident. Try these.



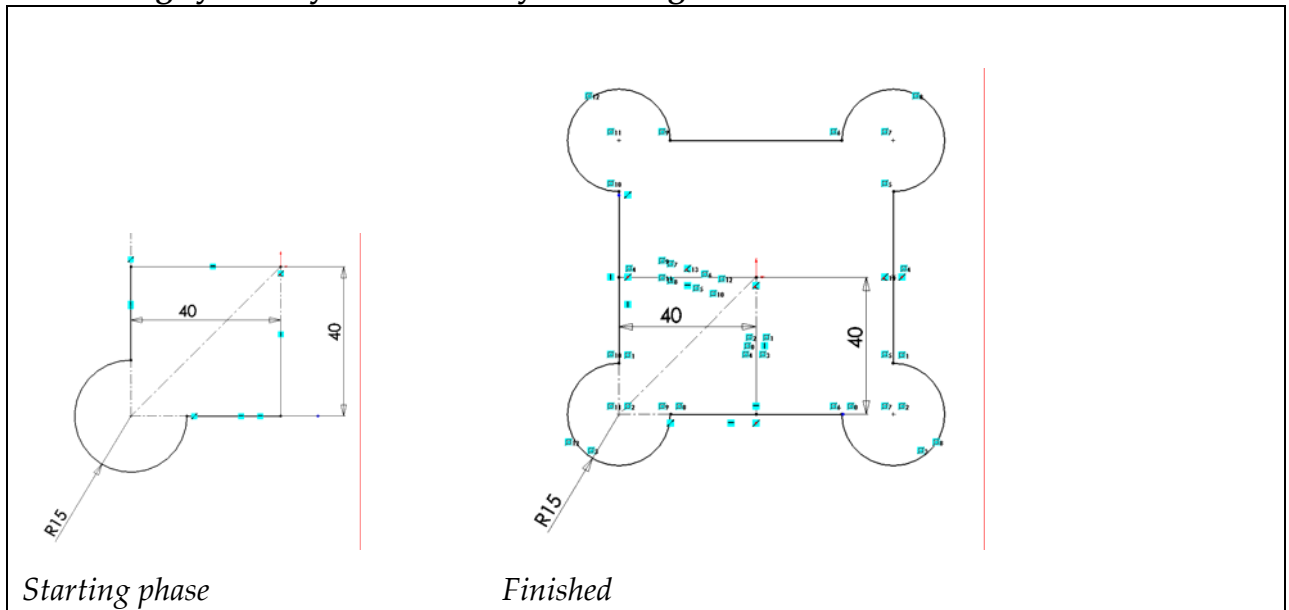
7. Finally add dimensions to the sketch. Select Smart Dimension from the Sketch palette. Select the sketch entity or the sketch entities you wish to add dimension. Once you release the mouse button a Modify window appears and you can write the dimension you wish.

Producing symmetry to a sketch with centerlines

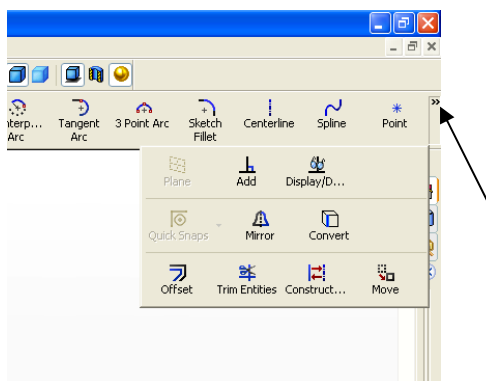


1. SolidWorks has a very handy symmetry tool in the sketch entities: the Centerline. In the exercise 1.1 you need to produce a symmetric, extruded part. Use the Centerline entity.
2. Draw centerlines like in the figure on the left. The horizontal and vertical centerlines are drawn by using the line midpoints. If you move the mouse cursor near the line midpoint, SolidWorks shows you this entity and you can select it.
3. Use midpoint relation with the sketch origin with all centerlines. The sketch becomes symmetrical and there is less to dimension. Try this by changing dimensions.

Producing symmetry to a sketch by mirroring

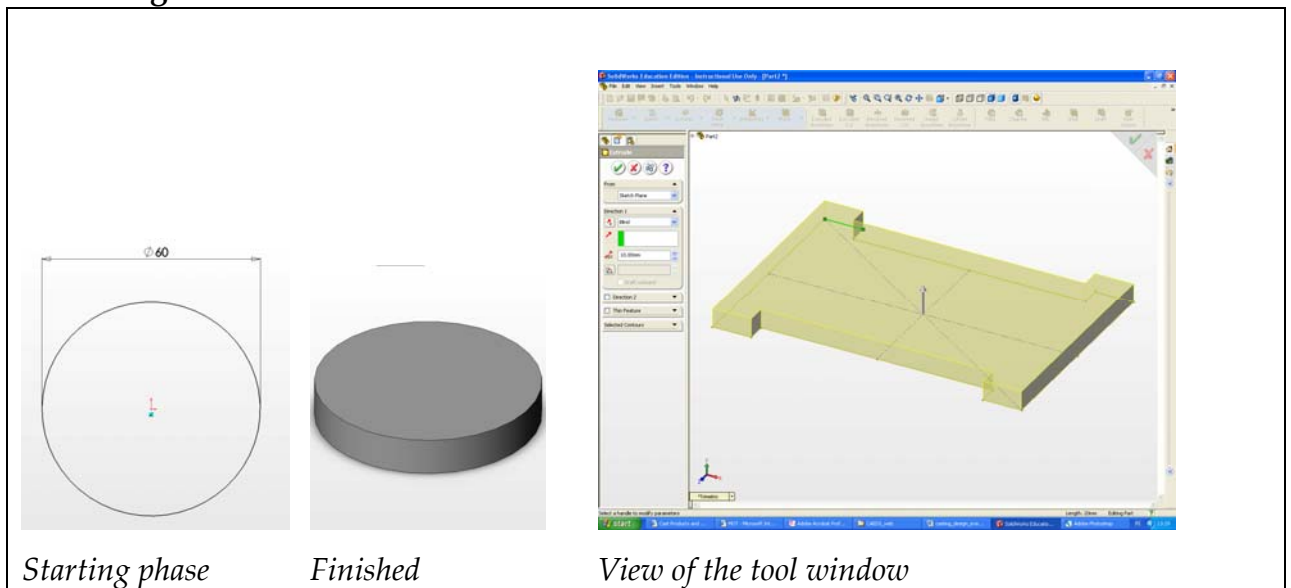


1. Sketch the entities you wish to mirror. In this example the finished part is symmetrical in two directions. There is a need to produce only a quarter of the final product.



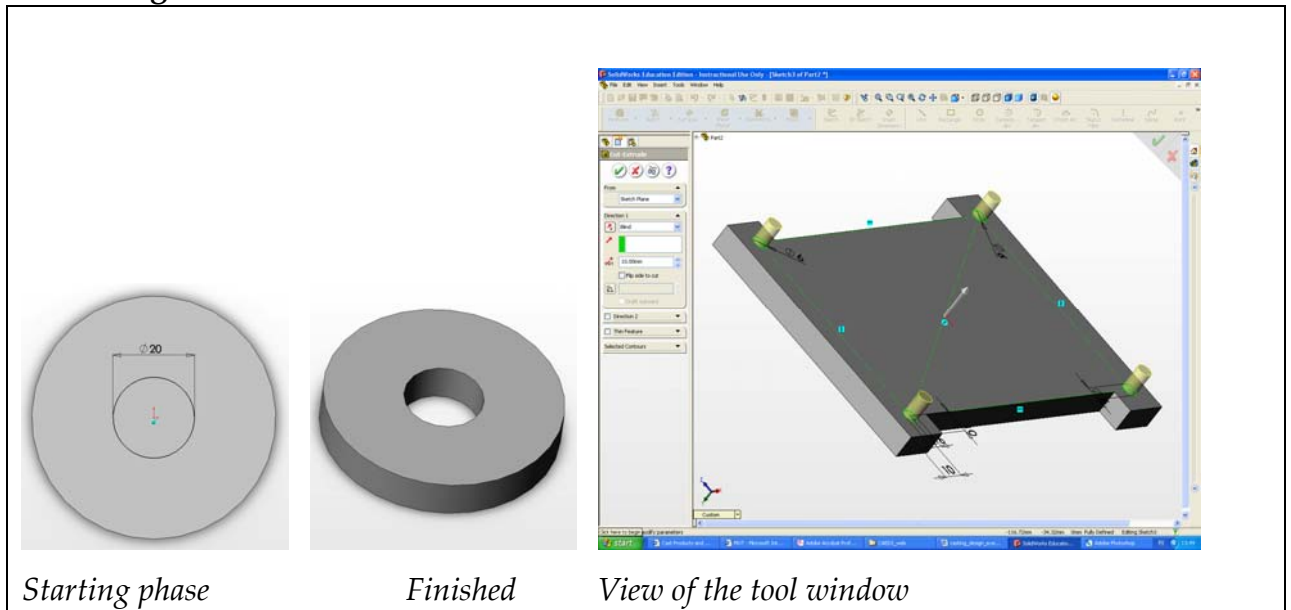
2. Open the Mirror tool. You can find it under the Tools menu. Tools->Sketch Tools->Mirror. Or from the Sketch palette. SolidWorks hides the tools that do not fit into the window. You can find the hidden tools from the place shown in the figure on the left.
3. With the mirror tool active, select the entities you wish to mirror and the axis to mirror about. You can mirror only to a one direction at a time.

Extruding a solid

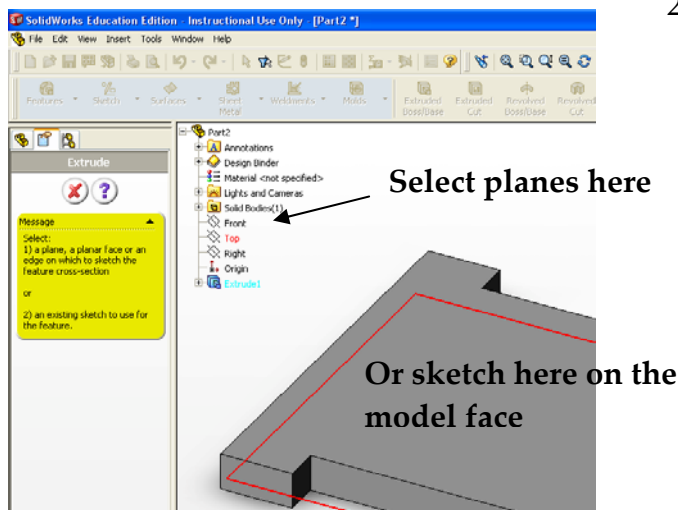


1. Select Features from the upper left corner of the SolidWorks window. A Features palette appears to the upper right part of the window. Select Extruded Boss/Base from the palette.
2. This tool needs you to prepare a sketch, which is then extruded. SolidWorks gives you two options: To select an existing sketch or to select a plane to start sketching. If you already have a sketch, select it and you will enter the state shown in the figure on the left. If not, start sketching. When you have finished the sketch, exit the sketching state and you will be able to start giving the extruded boss properties as when started with the already existing sketch.
3. Give a dimension to the extrude. There are different options to constraining and dimensioning the extrude. See SolidWorks help file to see the options and try them. When there is only one feature and one sketch, the number of possible options is limited.

Extruding a cut



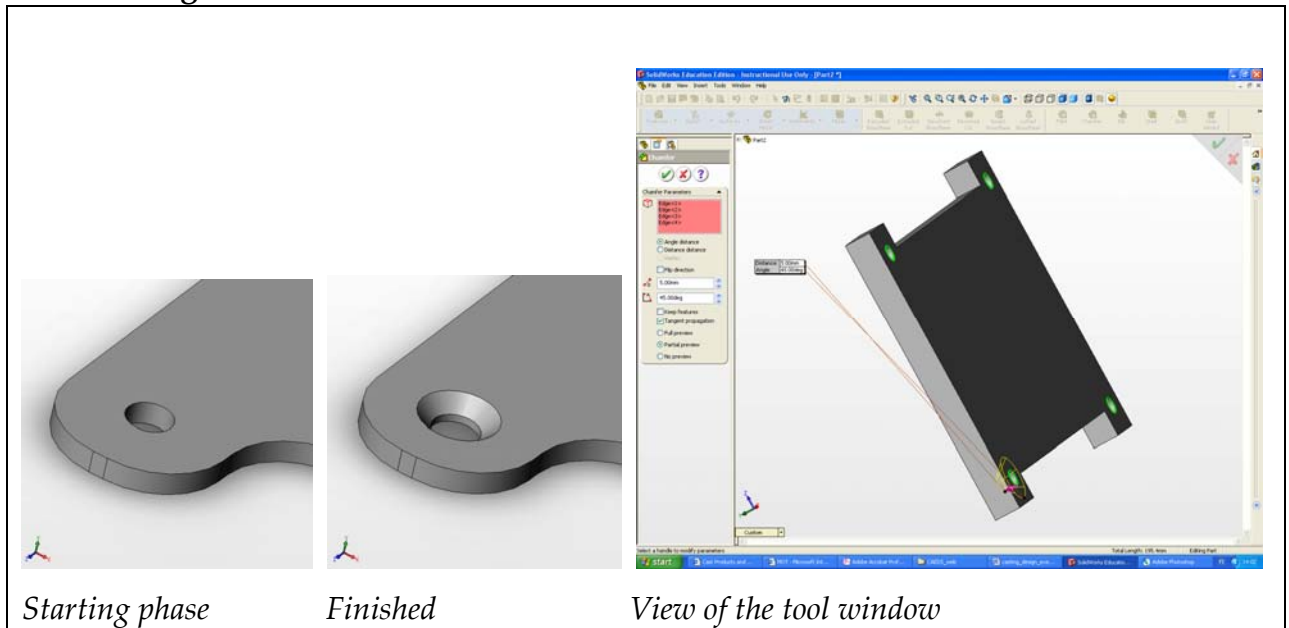
1. Select Features from the upper left corner of the SolidWorks window. A Features palette appears to the upper right part of the window. Select Extruded Cut from the palette.



2. It is obvious that you need something to cut before you can add a Extruded Cut feature. This tool needs you to prepare a sketch to give dimensions to an extruded cut. You can use one of the solid model faces as a sketch plane or use one of the model basic planes: Front, Top or Right plane. Solid Works asks you to select an existing sketch or to sketch one. You can select basic planes from the feature tree that has appeared to the SolidWorks model window.

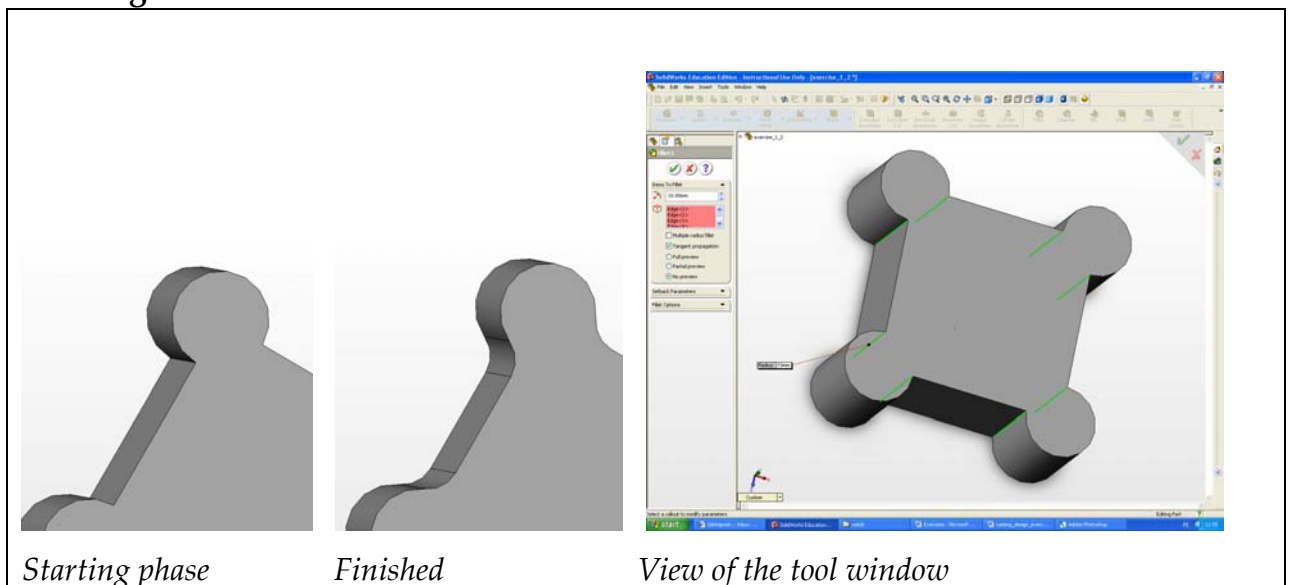
3. After you have selected the sketch or prepared the sketch and exit the sketching state you will enter the Extruded Cut feature state. Enter the dimensions and other properties. You will see the result in the model window. Notice that you can change the extruded cut direction. The direction shown in the figure is wrong and will cause an error.
4. If you are preparing a hole that should extend trough a feature, use some other than Blind direction. For example use the 'Up to surface' option. Try this.

Chamfering features



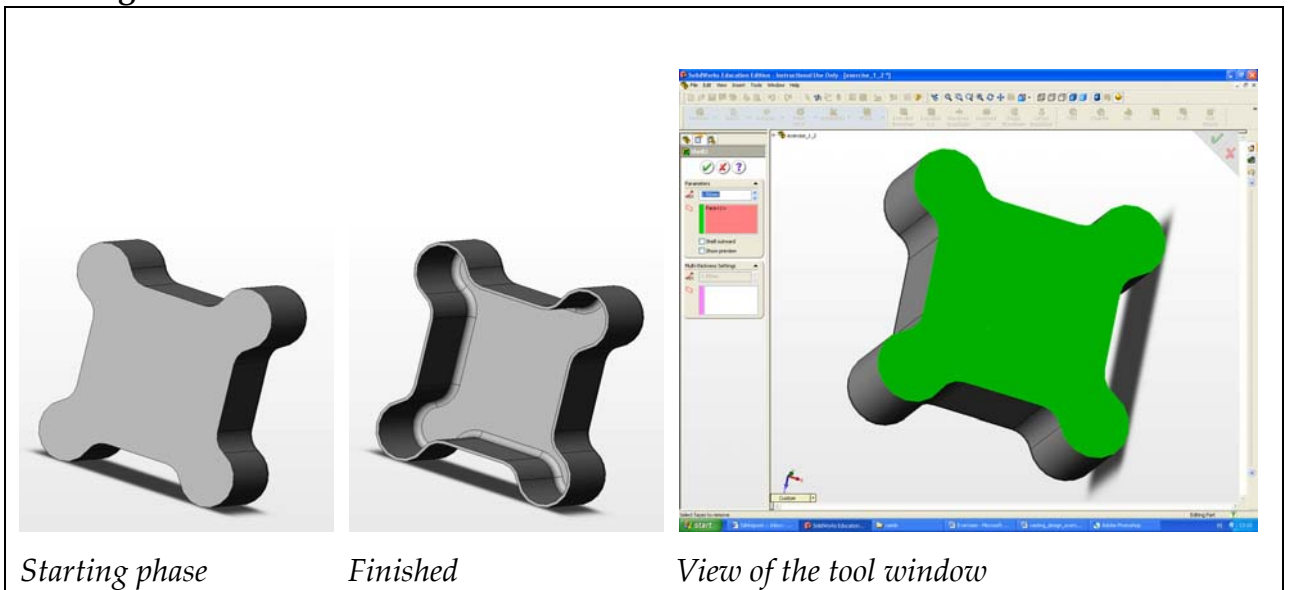
1. Select Features from the upper left corner of the SolidWorks window. A Features palette appears to the upper right part of the window. Select Chamfer from the palette. You can select Chamfer also from the Insert menu. Insert->Features->Chamfer.
2. Give the dimensions and other options as you wish. One by one select the entities you want to chamfer and press OK. Try different options. If you mistakenly select a wrong entity, selecting it one more clears the selection. You can also delete entities from the Chamfer parameters window.

Filleting features



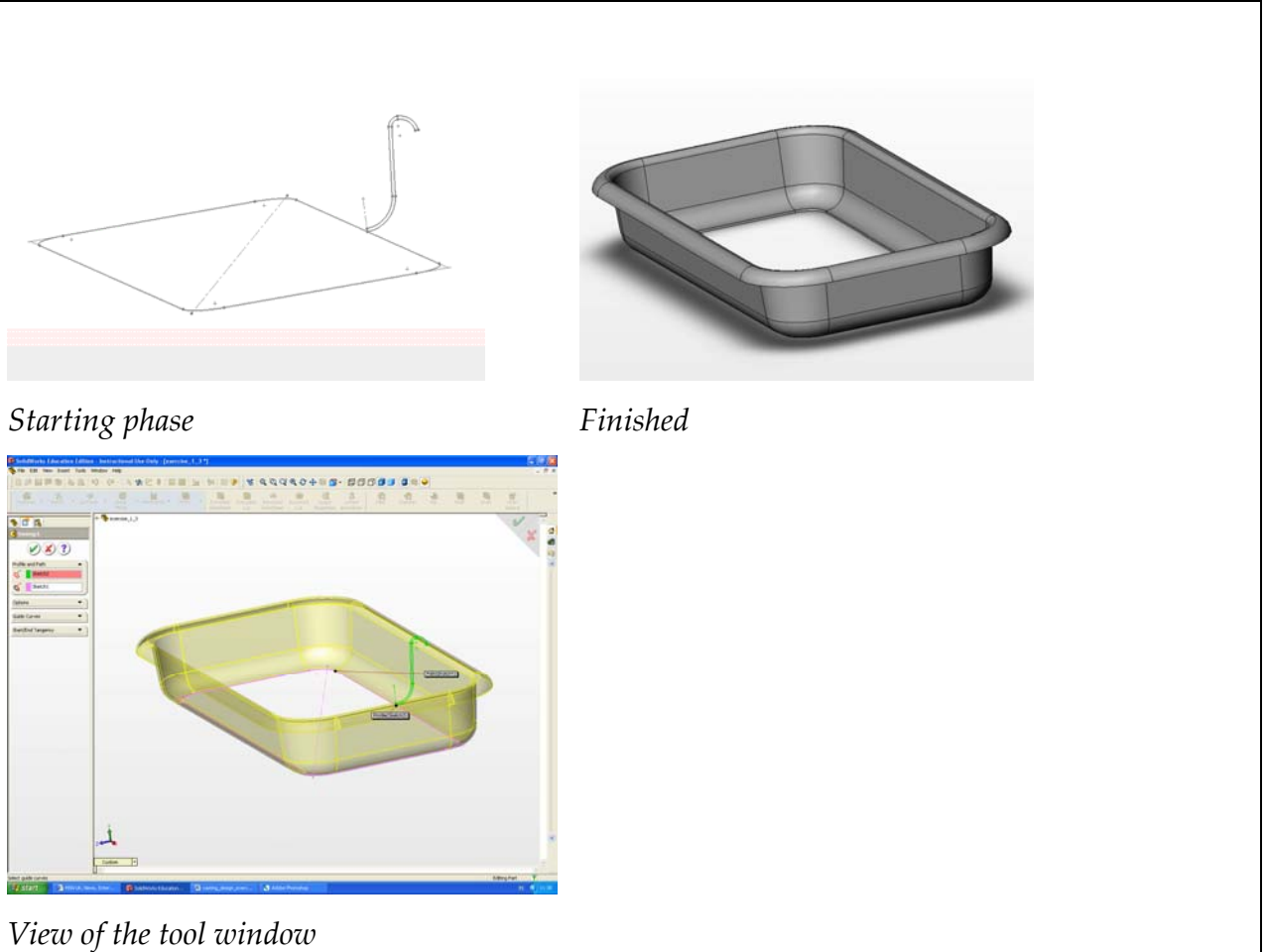
1. Select Features from the upper left corner of the SolidWorks window. A Features palette appears to the upper right part of the window. Select Fillet from the palette. You can select Fillet also from the Insert menu. Insert->Features->Fillet/Round.
2. Give a radius for the fillet. Select all the edges you wish to fillet. Be careful not to select edges that cross each other. Try to fillet edges in a different order and see what happens. In the example in the figure 'View of the tool window' it is better to choose the vertical lines first and after that fillet the vertical edges.

Making a shell from a solid



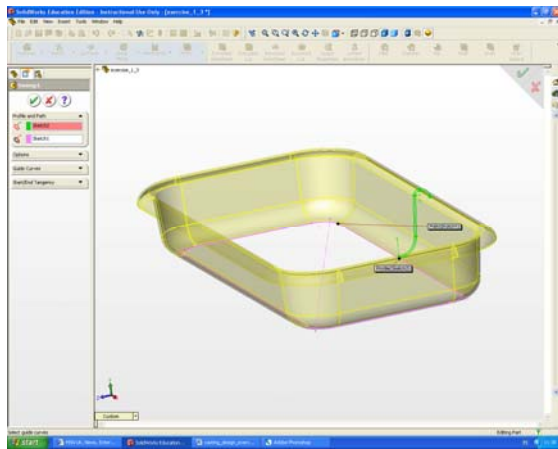
1. Select Features from the upper left corner of the SolidWorks window. A Features palette appears to the upper right part of the window. Select Shell from the palette. You can select Shell also from the Insert menu. Insert->Features->Shell.
2. Insert the wall thickness to the shell.
3. If you wish to remove one or more faces, select them. If you do not select any faces the solid will be hollow having the wall thickness you already have chosen.

Sweeping a solid



Starting phase

Finished



View of the tool window

1. Select Features from the upper left corner of the SolidWorks window. A Features palette appears to the upper right part of the window. Select Shell from the palette. You can select Shell also from the Insert menu. Insert->Features->Shell.
2. The tool needs you to prepare two sketches: One for path and one for profile. The profile should lie on the path. The profile and path are perpendicular to each other.
3. You can set different options for this tool. See SolidWorks help for more instructions. For example it is possible to use guide curves and set the end and start tangency for open profiles.