SolidWorks Tutorial 5

Tic-Tac-Toe

Preparatory Vocational Training
and Advanced Vocational Training
In this tutorial we will create a Tic-Tac-Toe game. The game consists of two plates that are on top of each other. In the top plate, there are holes for inserting small cylinders marked ‘X’ or ‘O’. In this exercise we repeat a lot of tools we already know and add a few others: working with configurations and the use of standard parts. Some new features in this tutorial include working with tolerances and fittings and working with patterns.

Top Plate

Work plan

First, we will create the top plate. We will do this according to the drawing below.

We will execute the following steps:

1. First, we will create the top plate with dimensions 60 x 60 x 10.
2. Then, we will make four counter bore holes.
Finally, we will create a pattern of 9 holes.

1. Start SolidWorks and open a new part.
2. Set the units for the part as MMGS at the bottom right of the SolidWorks screen.

3. Select the **Top Plane**.
   - Click on the **Sketch** tab in the **CommandManager**.
   - Click on **Rectangle**.

4. Draw a rectangle:
   1. Click on **Center Rectangle** in the **PropertyManager**.
   2. Click on the **origin**.
   3. Click at a random point to get the second corner.
5 Add a horizontal dimension to the sketch, as in the illustration on the right.

Change the dimension to 60 mm.
Push the <Esc> key on the keyboard to end the command.

6 Set the length of the horizontal and vertical lines to the same length:

1 Select a vertical line.
2 Push the <Ctrl> button and click on a horizontal line.
3 Click on Equal in the PropertyManager.

Tip: Remember that a blue field in the PropertyManager is a selection field. You can add elements by clicking on them in your model and you can also delete elements from it (e.g., when you have selected a wrong element).

To remove an element from the list, click on the element in the field and push the <Del> key on your keyboard. SolidWorks often asks you if you really want to remove the element from the selection field to prevent inadvertent deletions.

Tip: The sketch is not fully defined. You can determine this from the color of the lines in the sketch:

- Blue means: the sketch is not fully defined.
- Black means: the sketch is fully defined.

You can check if a sketch is fully defined in the status bar at the bottom of the screen. In SolidWorks it is not mandatory to make a fully defined sketch, but it is a
good practice to do this because it can help you to avoid a lot of problems when creating a model later.

In addition to the colors blue and black, a line in a sketch can turn red or yellow.

- **Red** or **Yellow** means: the sketch is *over-defined*.

Try the following: set the dimension of the height of the square. The **Make Dimension Driven?** message appears:

![Make Dimension Driven?](image)

You have entered too much information because:

- The dimension you added says the height is 60 mm.
- The relation between the two lines you have created before says the height is equal to the width, which is also 60.

The height is defined twice now, and this creates a conflict in SolidWorks. You must resolve this inconsistency. In the menu that is shown above, the best thing to do is choose **Cancel**. The dimension will not be set.

Did you make an *over-defined* sketch anyway? Then, throw away (delete) dimensions and/or relations, so that the sketch is no longer *over-defined*. 
7 Click on the **Features** tab in the **CommandManager**, and then on ** Extruded Boss/Base**.

1. Set the thickness of the plate to 10 mm.
2. Click on **OK**.

8 Next, we will make a sketch in which we will determine the exact position of the holes:

1. Select the **top plane** of the plate.
2. Click on the **View Orientation** icon.
3. Click on **Normal To**.

9 Draw another rectangle with a dimension of 46 mm. Follow the steps 4 to 6 again if you need help.

10 Click on **Exit Sketch** in the **CommandManager**.

We will not use this sketch to make a feature.
11 Start up a new sketch:

1. Select the top plane of the plate again.

2. Click on Circle in the CommandManager.

3,4 Draw a circle like the one in the illustration.

12 Set the dimension between the circle and one of the diagonal lines that you drew previously:

1. Click on Smart Dimension in the CommandManager.

2. Click on the center of the circle.

3. Click on the diagonal line.

4. Set the dimension.

5. Change it to 15 mm.

6. Click on OK.

13 Next, set the dimension to the other diagonal line (15 mm) and the diameter of the circle (Ø8 mm).

Push the <Esc> key to close the Smart Dimension command.
14 To set an exact fitting to the hole (Ø8), execute the following steps:

1. Select a dimension (it turns blue).
2. Be sure that Tolerance/Precision is visible in the PropertyManager. Click on the double arrows to reveal it.
3. Set Tolerance type to Fit.
4. Select a fitting of D10 in the Hole fit field.
5. Click on OK.

Tip: In this and the following tutorials, we will be using the commands from the CommandManager more often.

At this point, you should be getting used to working with SolidWorks and might find it more convenient to use the quick menu. This quick menu can be activated by pushing the S on the keyboard. The most important and most frequently used commands will appear. You will see the commands and functions that are associated with the part of the menu in which you are working, so you will see different commands/functions when you are in sketch mode than when you are in feature mode.

15 Make a hole in this sketch click on the Features tab in the CommandManager and then on Extruded Cut.

Set the depth of the hole in the PropertyManager to Through all and click on OK.
16 We will complete the hole pattern now.
1 Select the hole you just created.
2 Click on the **Linear Pattern** icon in the **CommandManager**.

17 Next, set the following features:
1 Select ONE of the diagonal lines.
2 Check to make sure that the line appears in the selection field.
3 Set the distance between the copies to 15 mm.
4 Set the number of copies to 3.
5 Whenever the copies are placed on the wrong side, click on **Reverse Direction**.

18 Repeat these steps in the area named **Direction 2**. For this purpose, select the other diagonal line. If the preview looks good to you, click on **OK**.

19 We will now create the mounting holes for the bolts. Click on **Hole Wizard** in the **CommandManager**.