Altium Designer

Essentials Training with Altium 365

Module 17: PCB Configuration

Software, documentation and related materials:

1

Table of Contents

Module 17: PCB Configuration

1 Purpose

In this exercise, you will learn how to define a board shape using several methods.

In Altium Designer, the board shape is defined by creating a board outline. The board outline is a closed polygon that represents the physical boundaries of the printed circuit board (PCB). The board outline not only defines the physical shape of the PCB but also plays a crucial role in determining the board's electrical and thermal characteristics. It influences the placement of components, routing of traces, and overall manufacturability of the PCB.

2 Shortcuts

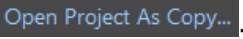Shortcuts used when working with Module 17: PCB Configuration

2	2D Mode
3	3D Mode
G	Grid
Ctrl + G	Grid Editor
Ctrl + A	Select All
D » S » D	Define Board Shape from Selected Objects (2D Mode)
K » R	Open Explorer panel

3 Preparation

1. Close all existing projects and documents.
2. Next, create a Copy / Clone of the Training Project `Module 17 PCB Configuration.`
3. Select **File » Open Project…** to open the *Open Project* dialog.
4. Enable the folder view button [icon].
5. Navigate to the predefined Training Project `Module 17 PCB Configuration (Top\Projects\Altium Designer Essentials Training Course\...).`
6. Select **Open Project as Copy…** Open Project As Copy… .
7. At the new dialog *Create Project Copy:*
   a) Add your name to the project: `Module 17 PCB Configuration – [Your Name].`
   b) Add a description: `Altium Essential Training - Module 17 – [Your Name].`
   c) Open the *Advanced* section.
   d) Select the Ellipsis Button [···] from the **Folder** configuration to open the *Choose Folder* Dialog.
      i) Select the folder with your name: `Project\For Attendees\[Your Name]`
      ii) Select **OK.**
   e) Change the Local Storage path if needed.
   f) Select **OK** to create the copy.
8. Wait until Altium Designer created the copy of the project and opened the project for you at the *Projects* panel, this may take up to 1 minute.

> Hint: For details how to Copy / Clone the predefined training project see Module 9 Making the Connection, Step 3 Preparation.

Altium TRAINING

4 Adding a New PCB

The Project template already offers a PCB. In case you start from scratch with a new PCB we explain some options how to modify the board shape.

9. Add a new PCB document to the project by opening the *Projects* panel and right-clicking on the project name. From the right-click menu, select **Add New to Project » PCB** as seen in Figure 1 below.
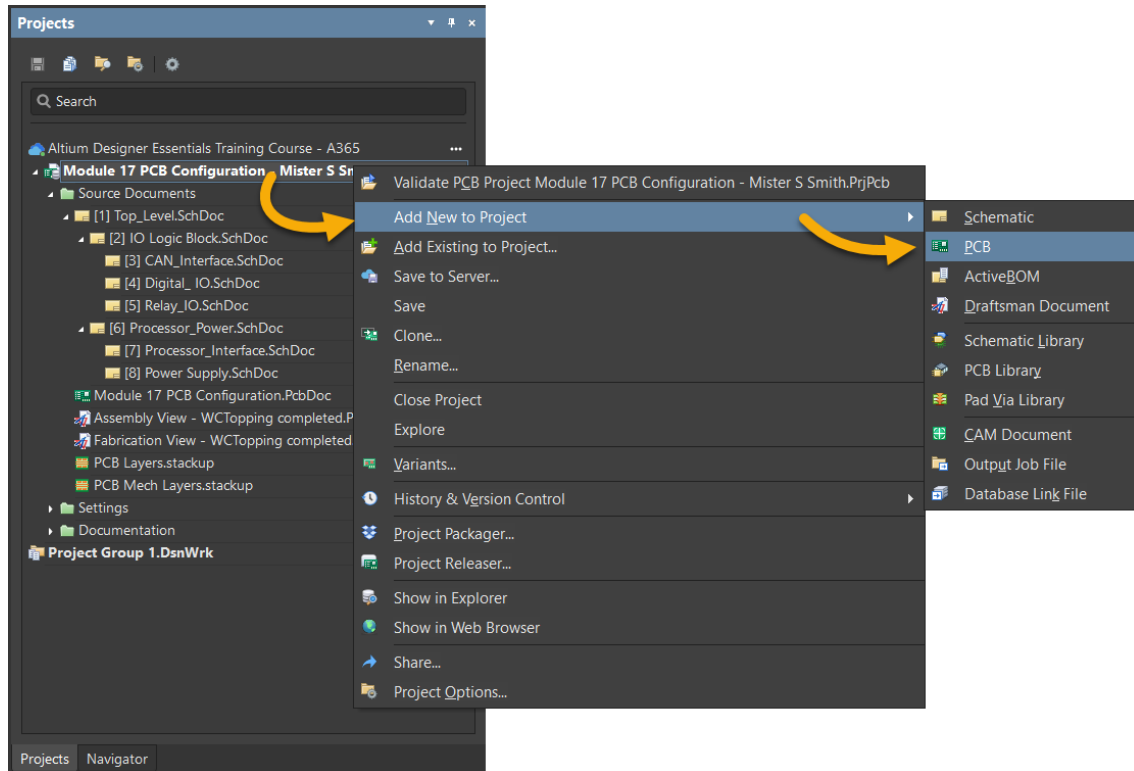

Figure 1. Add new PCB to the Project

10. Go to **File » Save All** and save the PCB document as `Module 17 PCB Configuration [Your Name].PcbDoc`.

5 Pre-configuration

11. The focus should be on the new PCB document you just created. If not already opened, open it from the PCB panel.

5.1 Grid

12. With the PCB as active document we will now change the grid.

13. To simplify the drawing process, we will set a coarse grid of 500mil.

    a) Press **Ctrl + G** to open the *Grid Editor* and change the Step value to `500mil`.

    b) Select **OK** to close the *Grid Editor*.

5.2 Mechanical Layer Configuration

5.2.1   Adding a New Mechanical Layer

14. Open the *View Configurations* panel by going to **View » Panels » View Configuration** or press the **L** hotkey.

15. Right-click on layer `Mechanical 1` and select **Edit Layer** as shown in Figure 2. Change the Layer Type to `Board Shape` and the Layer Name to `Board Outline`.
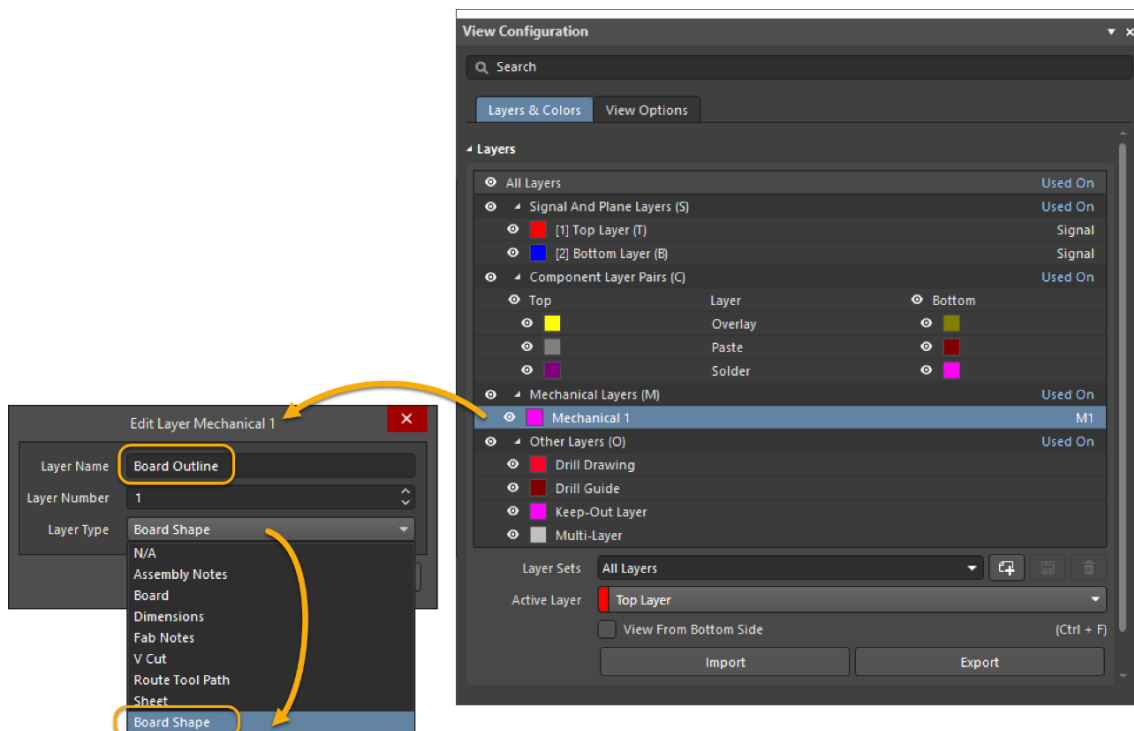
16. Select **OK** to close the dialog.

Figure 2. Renaming Mechanical Layer

17. Select the `Board Outline` layer to make it the active layer in the Layer Bar at the bottom of the PCB editing area.

5.2.2 Load a Layer Stack File

18. Instead of configuring the Mechanical layers from scratch it is possible to load a preconfigured template file.

19. Go to **Tools » Import Mechanical Layers... .**

20. Navigate to the folder … / `Module 17 PCB Configuration - [Your name]\Config Files.`

21. Select the File `PCB Mech Layers.stackup` and select **Open.**

22. After loading the *.stackup file, notice the new configuration of the mechanical layers, see Figure 3.

Figure 3. Mechanical Layer configuration
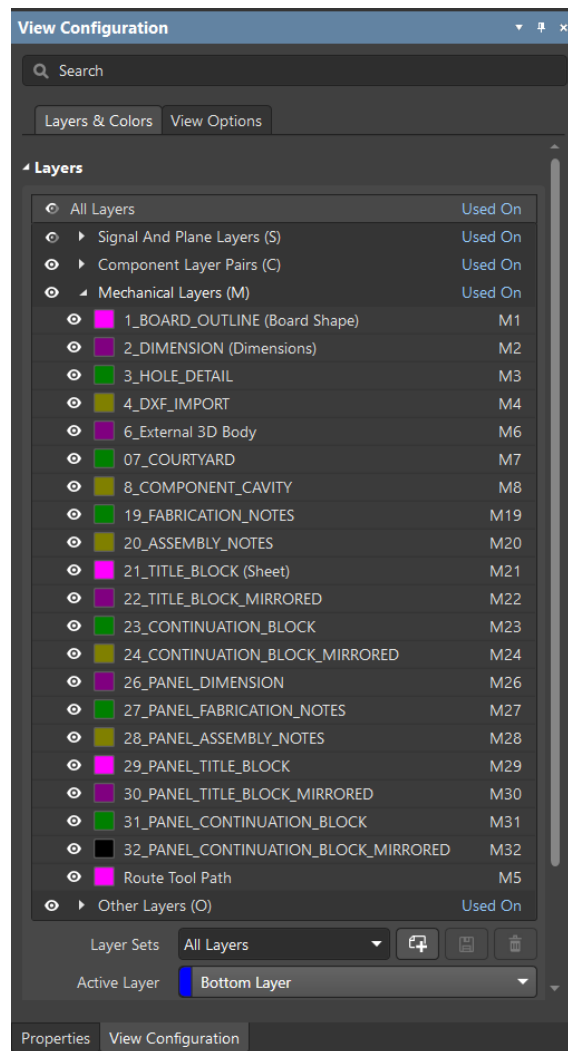
23. Set the `1_Board_Outline (Board Shape)` layer as active layer by using the drop-down list below the layer list or select the Tab `1_Board_Outline (Board Shape)` at the bottom of the PCB editing area.

> Hint: Mechanical Layers are not predefined. Any Mechanical layer can be used for any information. It is good practice to standardize the Mechanical Layers used for the PCB.

7

5.3 Layer Sets

24. Similar to the option of importing a set of mechanical layers, it is possible to load Layer Sets.

25. Still in the *View Configuration* panel, Figure 4:

    a) Select the drop-down list to see the current Layer Sets.

    b) Select **Import** and navigate to the folder …/`Module 17 PCB Configuration - [Your name]\Config Files`

    c) Select the file `PCB Layerset.layerset`

    d) Select the drop down list again to see the current Layer Sets or use the button **Layer Set (LS)** [LS] at the lower left side of the PCB Workspace.

Figure 4. Load predefined Layer Sets

Altium
TRAINING

6 Define Board Shape from Selected Objects

6.1 Create Circular Board Shape

26. With the Layer `1_Board_Outline (Board Shape)` as active layer.
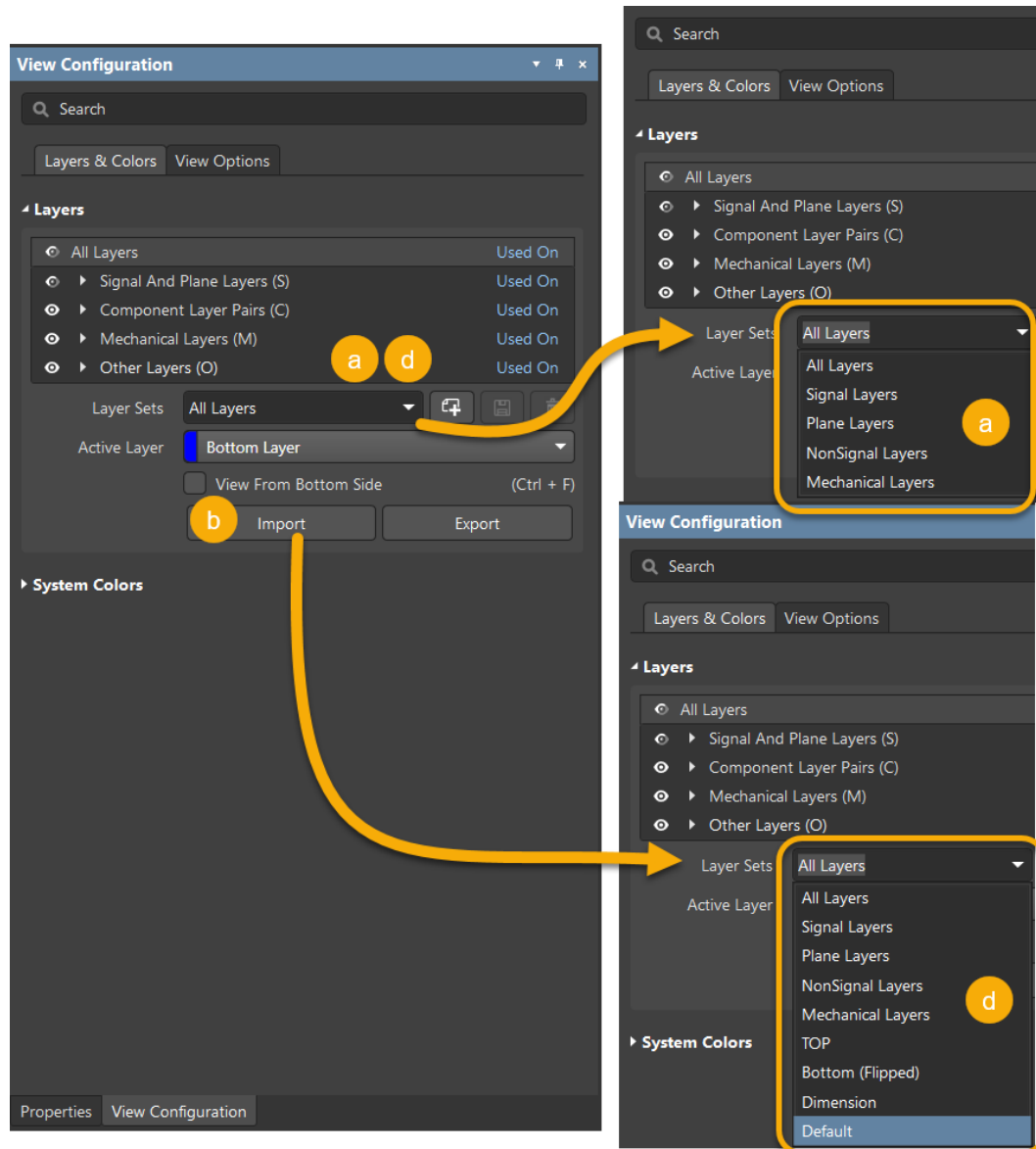
    a) Select **Place » Arc » Full Circle**.

    b) Left click anywhere near the middle of the existing PCB shape.

    c) Move the mouse either towards you or away from you to enlarge the circle. The Head Up Display shows you the radius of your circle.

    `No net assigned,    R: 1000mil    A1:0.00    A2:360.00`

    d) Left clicking again will set the size of the circle.
    For further changes, you can select the circle and change the *Radius* value of the circle at the *Properties* panel.

    e) Click the right-mouse button to terminate the command. Your PCB should now be similar to Figure 5. Your existing Board Shape may be different to the one you see at the image.
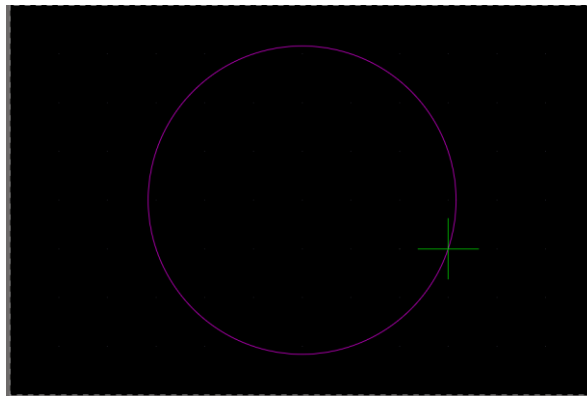
Figure 5. Placing a circle in the PCB workspace

27. The circle should be highlighted indicating that it's selected. If it is not, press **Ctrl + A** to select it.

28. To create a circular board shape, go to the menu **Design » Board Shape » Define from Selected Objects**. The black area, which is the board outline, should now be limited to the circle as shown in Figure 6.
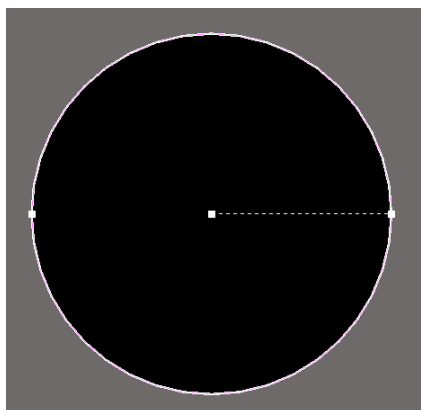

Figure 6. Defining the Board Shape from Selected Primitives

6.2 Create Rectangle Board Shape

29. Go to **Place » Rectangle** for drawing a rectangle of any shape inside of the existing circular PCB. Press G to change the Grid if needed.

> Hint: Try changing the corner style by pressing **[Shift]+[Spacebar]** while drawing the rectangle. Examine the *Status Bar* in the bottom center of the window to see the current mode as you toggle through the three corner style choices.

> Hint: Instead of using the rectangle command you could also use the Line command to create the rectangle, or any other shape you need.

> Caution: If you're having issues placing the lines for the rectangle, this may be because you're in **Look Ahead Mode**. Hit the **1** key to disable this mode, which may help with line placement.

Ensure that the Shape you create with the Line command is closed by looking for a white circle at the cursor when you are completing the polygon as shown in Figure 7.
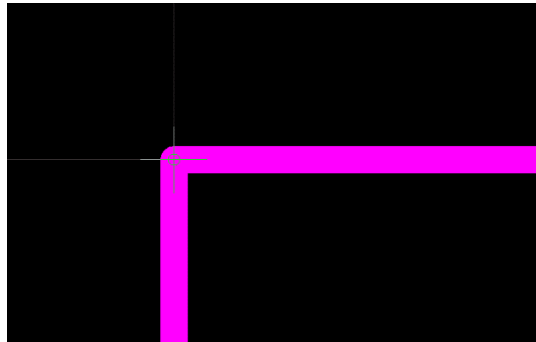

Figure 7. Drawing a rectangle with lines to define our Board Shape

30. After you complete the rectangle, right-click to terminate the command.
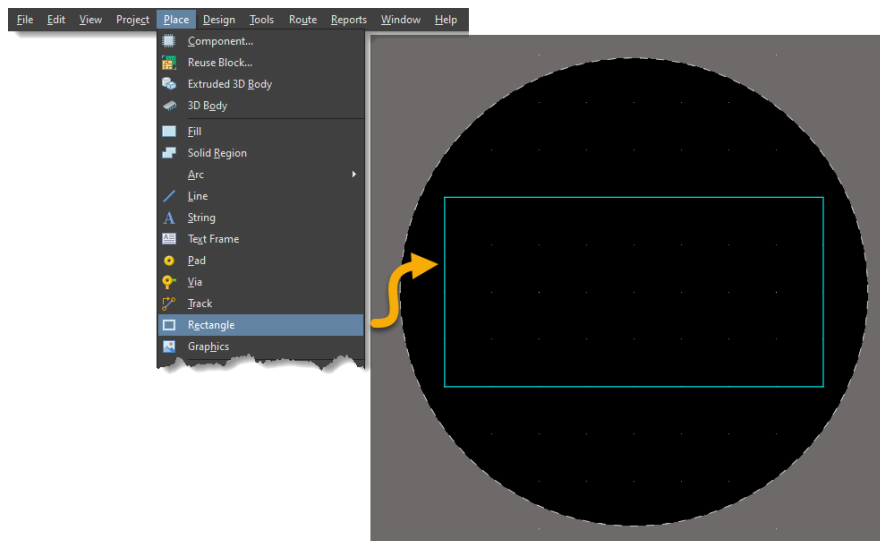

Figure 8. Rectangle for Board Shape

31. Select the lines of the newly defined rectangle by using a selection rectangle.

32. Now we'll create a board shape to match your rectangle by going to **Design » Board Shape » Define from selected objects**.

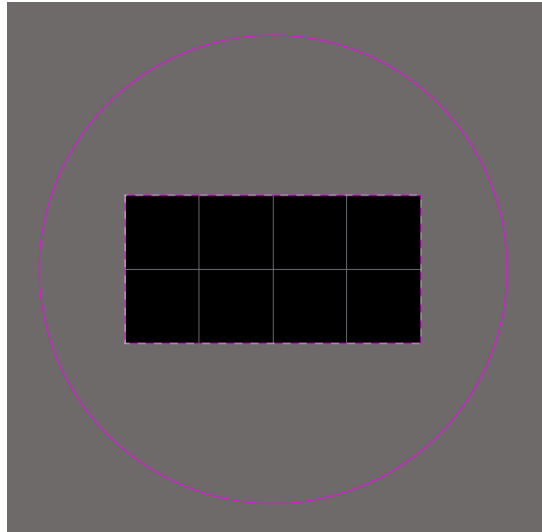33. Your PCB should now be rectangular and similar to Figure 9.

Figure 9. Example Board Shape defined from the rectangle

34. Since we changed our board shape, we can delete the outer circle by selecting it, and hit **Delete**.

6.3 Position the Origin Marker

Next, we will position the Origin Marker (0/0 coordinate).

35. Execute **Edit » Origin » Set**

   a) With a Crosshair attached to your cursor, move your cursor to lower left corner of the PCB.

   b) Left click at the lower left corner to place the Origin marker.

> Hint: It is easier to place the Origin marker if the snap to the Board Shape is active `Board Shape`.
> Press **CTRL+E** to open the Snap Options and activate `Board Shape`.
> Close the options with **ESC.**

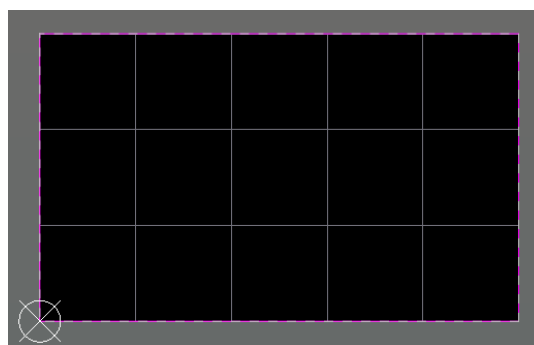36. Your PCB should now loke similar to what you see at Figure 10.

Figure 10. Rectangular board with Origin marker

37. Remove the rectangle for the board by pressing **Ctrl + A** to select the outline, then press the **Delete** key to remove the outline.

11

7 Defining Board Shape from DXF/DWG

38. Open the *Explorer* panel **K » R** and browse to the training folder `Mechatronik 3D Models – DXF`

39. Select the DXF and download the file, for example, to your Windows Desktop.

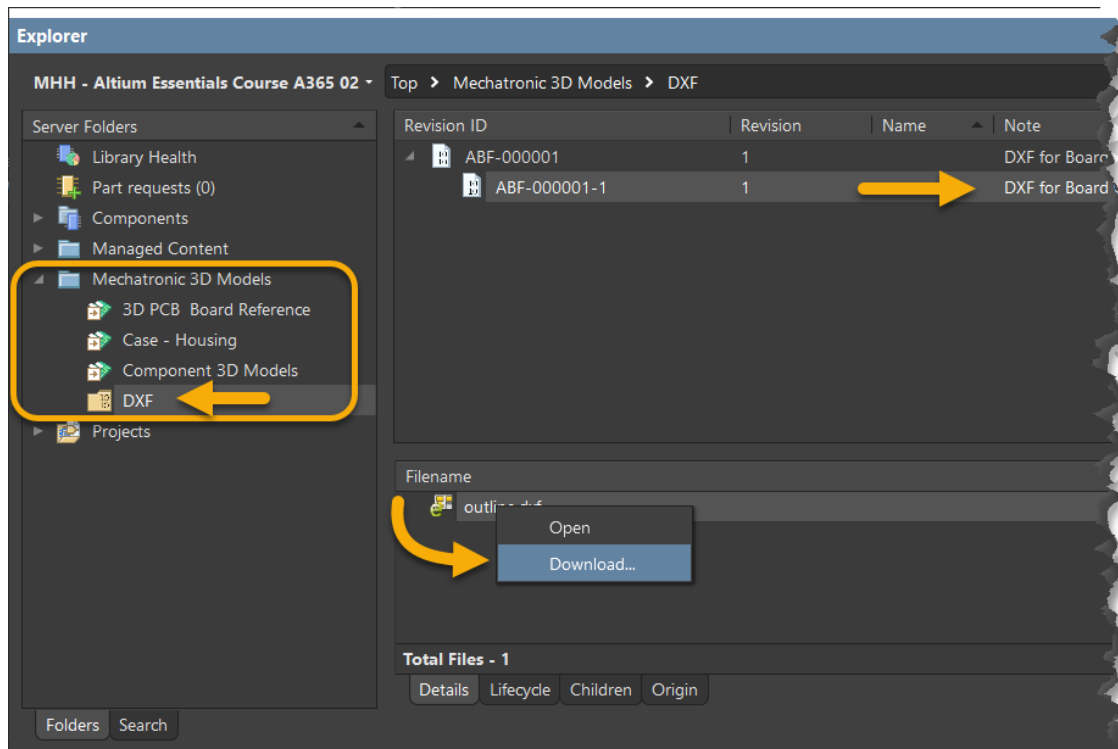40. Select **OK** *at the* Dialog *Download from server.*

Figure 11. Download of Outline DXF from A365 workspace

41. Go to **File » Import » DXF/DWG** and navigate to the Windows Desktop, subfolder `Desktop\A3D-000002-1\Released` and open the `outline.dxf`. (the folder structure / name may be different for your PC).

42. The *Import from AutoCAD* dialog box will open. Change the settings as shown in Figure 12.

    a) Select the Scale `mm`.

    b) *Layer mapping for* **Layer 0**: `Not Imported`.

    c) *Layer mapping for Layer* **board_shape**: `1_Board_Outline (Board Shape)` layer.

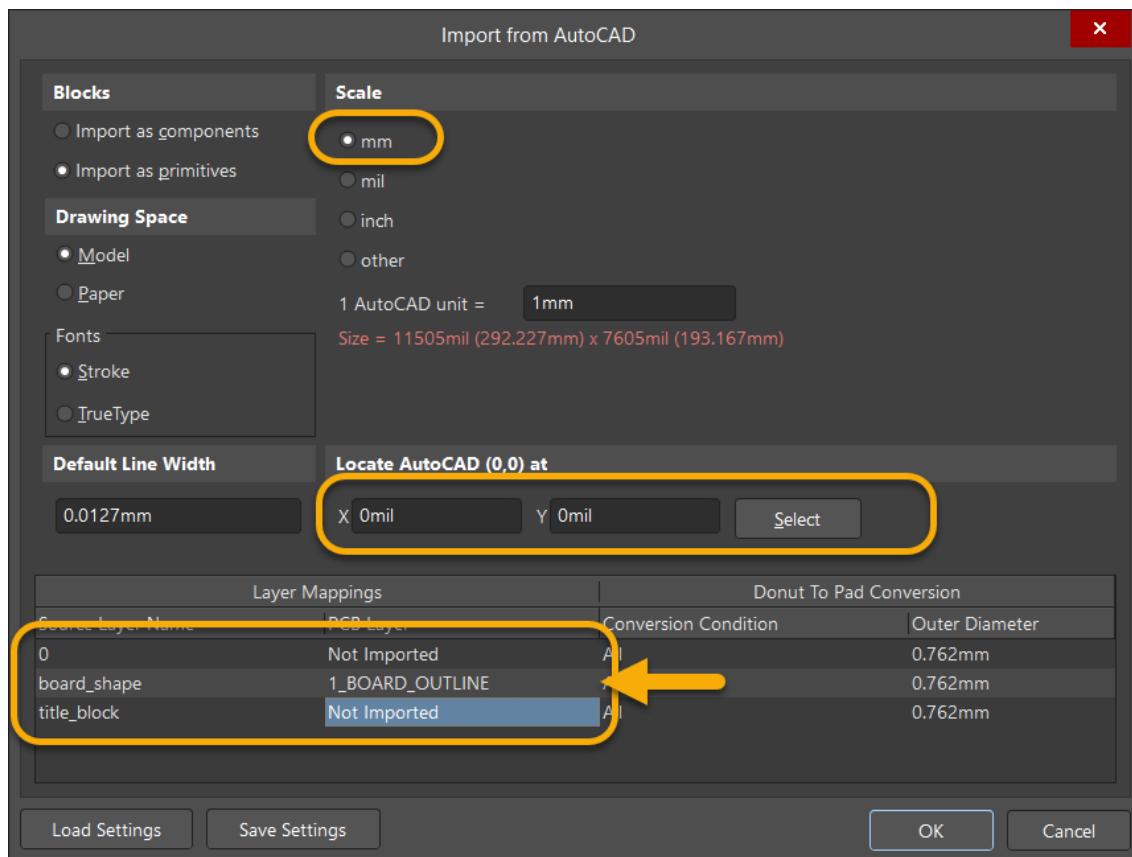   d) *Layer Mapping for Layer* **title_block**: `Not Imported`.

Figure 12. Import DXF dialog

> Hint: To ensure the proper scale selection, double-check the size. This will update based on the selected scale units and should match the dimensions you are expecting.

43. Click **OK**.

44. A dialog box will appear stating that the import is done. Click **OK** on this dialog box.

45. After clicking on **OK**, the board shape will be imported similar to Figure 13.
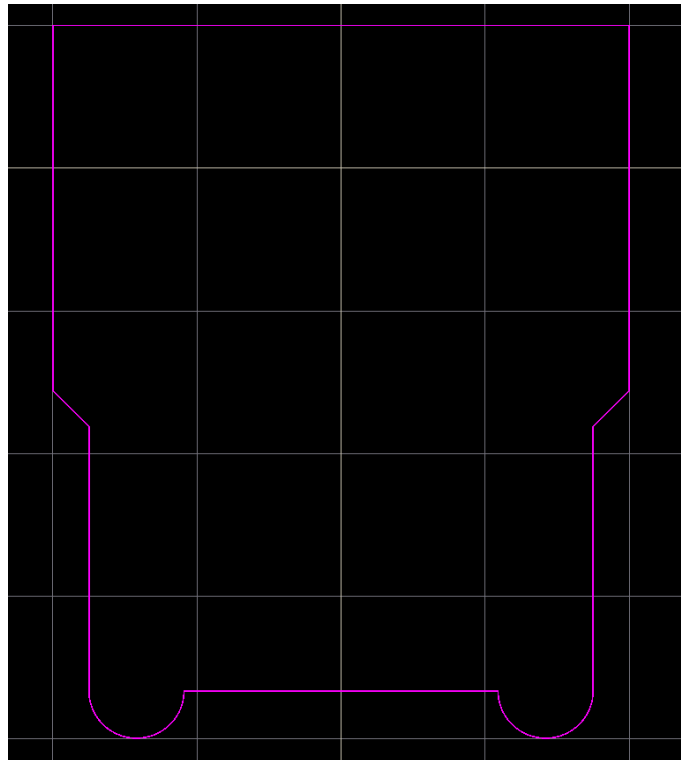

Figure 13. Imported DXF outline

46. Select all the line segments in the outline by pressing **Ctrl + A**.

47. Select **Design » Board Shape » Define from Selected Objects** to create the new board shape as shown in Figure 14.
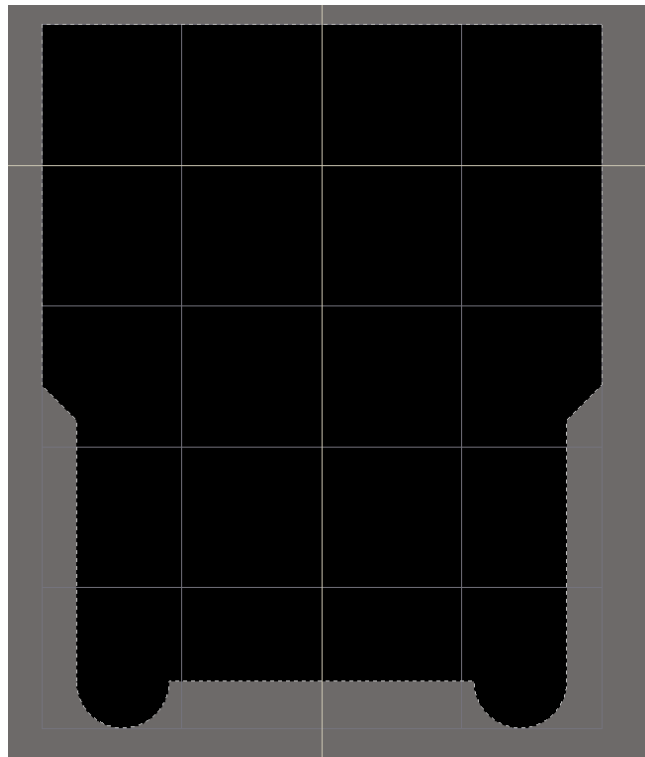

Figure 14. Final Board Shape created from a DXF outline

14

7.1 Optional Actions

48. Feel free to reposition the origin. This can be done with **Edit » Origin » Set.**

49. Feel free to Include a Board Cutout to the PCB.
Select **Design » Board Shape » Define Board Cutout** and draw the cutout you need.

50. Feel free to view your PCB in 3D, by hitting the **3** key as shown in Figure 15.

Figure 15. Example of a rectangular board shape in 3D

51. Return to 2D using the **2** key.

52. Save all documents using **File » Save All.**

53. Save the modifications to the server:
    a) At the *Project* panel, next to the Project name you find the command
    Save to Server Save to Server ○ .
    b) Select **Save to Server.**
    c) At the dialog *Save [Project Name]:*
       i) Add the comment `Module 17: PCB Configuration - [Add Your Name] -`
          `Finished.`
       ii) Click on **OK.**

54. When ready, close the project and any open documents.

Altıum.
TRAINING

Congratulations on completing the Module!

Module 17: PCB Configuration

from

**Altium Designer Essentials Training
with Altium 365**

Thank you for choosing **Altium Designer**

Altium
TRAINING