Trainers for Visually Impaired Students Introduce 3D Printing

Tutorial Module 5
Introduction in CAD Software
Autodesk Fusion360

Tutorial for the T4VIS-In3D trainer course

Published by the
T4VIS-In3D project consortium

The project “T4VIS-In3D” was co-financed by the “ERASMUS+” Programme of the European Commission

The European Commission's support for the production of this publication does not constitute an endorsement of the contents, which reflect the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

This Tutorial is published by the T4VIS-IN3D project consortium.

Printed:
October 2022 by Berufsförderungswerk Düren gGmbH

The T4VIS-In3D Project Consortium:

Berufsförderungswerk Düren gGmbH (Project co-ordination)
Karl-Arnold-Str. 132-134, D52349 Düren, Germany, http://www.bfw-dueren.de

Fundacion Aspaym Castilla Y Leon
C/ Severo Ochoa 33, Las Piedras 000, 47130, Simancas Valladolid, Spain, https://www.aspaymcyl.org/

Hilfsgemeinschaft der Blinden und Sehschwachen Österreichs
Jägerstrasse 36, 1200 Wien, Austria, https://www.hilfsgemeinschaft.at/

Instituttet for Blinde og Svagsynede, IBOS
Rymarksvej 1, 2900 Hellerup, Denmark, https://www.ibos.dk

Istituto Regionale Rittmeyer per i ciechi di Trieste
Viale Miramare 119, 34136 Trieste, Italy, http://www.istitutorittmeyer.it/

NRCB
24 Landos Str., Plovdiv, 4006, P. Box 11, Bulgaria, http://www.rehcenter.org

Content

1 General

Various software products for “Computer-Aided Design” (abbrev. CAD) are available on the market. These are characterised by different specifications and areas of application. The best-known CAD application is probably “AutoCAD” from Autodesk.

This is a CAD application that primarily enables 2D drawing, i.e. it replaces the classic drawing sheet. 3D functions were added in “toolsets”.
Other well-known CAD programmes are[1]:

Name	Manufacturer	Areas of application	Link
Catia	Dassault Systems	Aerospace technology, automotive manufacturing, mechanical engineering, large-scale enterprises	http://www.3ds.com/de/produkte-und-services/catia/
Fusion360	Autodesk	Product design, toolmaking, smaller companies	https://www.autodesk.de/products/fusion-360/overview
Inventor	Autodesk	Product design, toolmaking, architecture, medium enterprises	https://www.autodesk.de/products/inventor/overview?us_oa=dotcom-us&us_si=3b99b45b-f3a0-4ec7-9bb8-e2370ab4276d&us_st=inventor&us_pt=INVNTOR
Rhino(ceros 3D)	Robert McNeel & Associates	Product design, architecture, small and medium enterprises	https://www.rhino3d.com/
Solid Edge	Siemens	Mechanical engineering, plant engineering, toolmaking, large-scale enterprises	http://www.plm.automation.siemens.com/de_de/products/velocity/solidedge/
Solid Woks	Dassault Systems	Mechanical engineering, plant engineering, toolmaking, large-scale enterprises	http://www.solidworks.de/

The decision to use Autodesk Fusion360 for tactile teaching media was based on the following reasons:

1. Good price/performance ratio, available school licences
2. Comprehensive support of 3D constructions
3. Easy to learn, available online tutorials

[1] List in alphabetical order without reference to market shares

The software Autodesk Fusion360 is a very comprehensive CAD application that allows the creation of very complex technical constructions. In fact, only a fragment of Fusion360's functionality is required for the production of tactile teaching materials. These basic functions will be covered here and be quite sufficient to create tactile location maps and simple tactile models for technology or science lessons.

Since updates to the software are released several times a year, it is possible that the graphical user interface or the menus may change and no longer be identical to the illustrations listed in this manual.

For readers who want to explore Fusion360 in more detail, the video tutorials from Autodesk are recommended. These are available at:
https://help.autodesk.com/view/fusion360/ENU/courses/AP-GET-STARTED-OVERVIEW
Alternatively, you can scan the following QR Code with your smartphone.

For most trainers working in educational institutions for people with visual impairments, the handling of a CAD programme is rather unknown. The large range of functions as well as the technical application background may even seem daunting for newcomers. However, the model shown below proves that Fusion360 is easy to use. After only 30 minutes of instruction in Fusion 360, a 16-year-old student constructed this model on his own in 18 hours. The model was based exclusively on photos and drawings from the internet. The procedures described in this tutorial were used for realisation of this model.

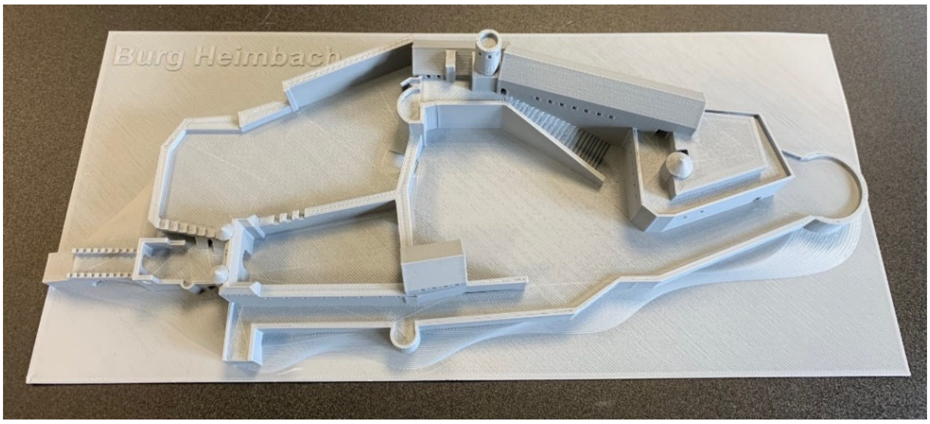


Figure 1 Building designed with Fusion360

2 Autodesk Fusion360: Download and Installation

Autodesk Fusion360 is available free of charge for private users and non-commercial use. However, this version is severely limited in functionality. For educational organisations, an educational version is available that offers the full range of functions. All that is necessary for this is to register. At this point, it is recommended to use this full version. All descriptions and illustrations are based on the full version.

Currently (in August 2021), Fusion360 can be obtained as a single-user or campus licence via download. A stable internet connection is required for installation. The licence version is checked online both during installation and each time the programme is started. Without internet connection during the installation process, Fusion360 cannot be installed. In case there is no internet connection after the installation, you can work offline for a limited period of time.

Fusion360 is cloud-based. Although one can save all constructions locally, the intended storage method is the cloud. Each administrator can create a folder with subfolders for their courses, which means that editing is thus also possible from other computers. There are even apps for smartphones and tablets.

Autodesk prescribes a minimum technical configuration for the operation of Fusion360. This can be found at: https://knowledge.autodesk.com/support/fusion-360/troubleshooting/caas/sfdcarticles/sfdcarticles/System-requirements-for-Autodesk-Fusion-360.html

However, to ensure that Fusion360 works without problems, the computer used should have a 3D-capable graphics card with at least 4 GByte VRAM and a working memory of 12 GByte. This allows changes of perspective without any delay.

3 First steps in Fusion360

After installation, Fusion can be opened either via the icon on the desktop or the start menu. The first time Fusion is started after installation, the user must enter the Autodesk ID and password. Fusion360 cannot be started without this identification. It is therefore necessary that this data is available to the users or that the login is carried out by the administrator. After the successful start, the working window opens. The largest area of the window is the Working Area. This area represents the virtual drawing sheet that can be moved, rotated and enlarged as desired.

3.1 Graphical User Interface and Essential Menus

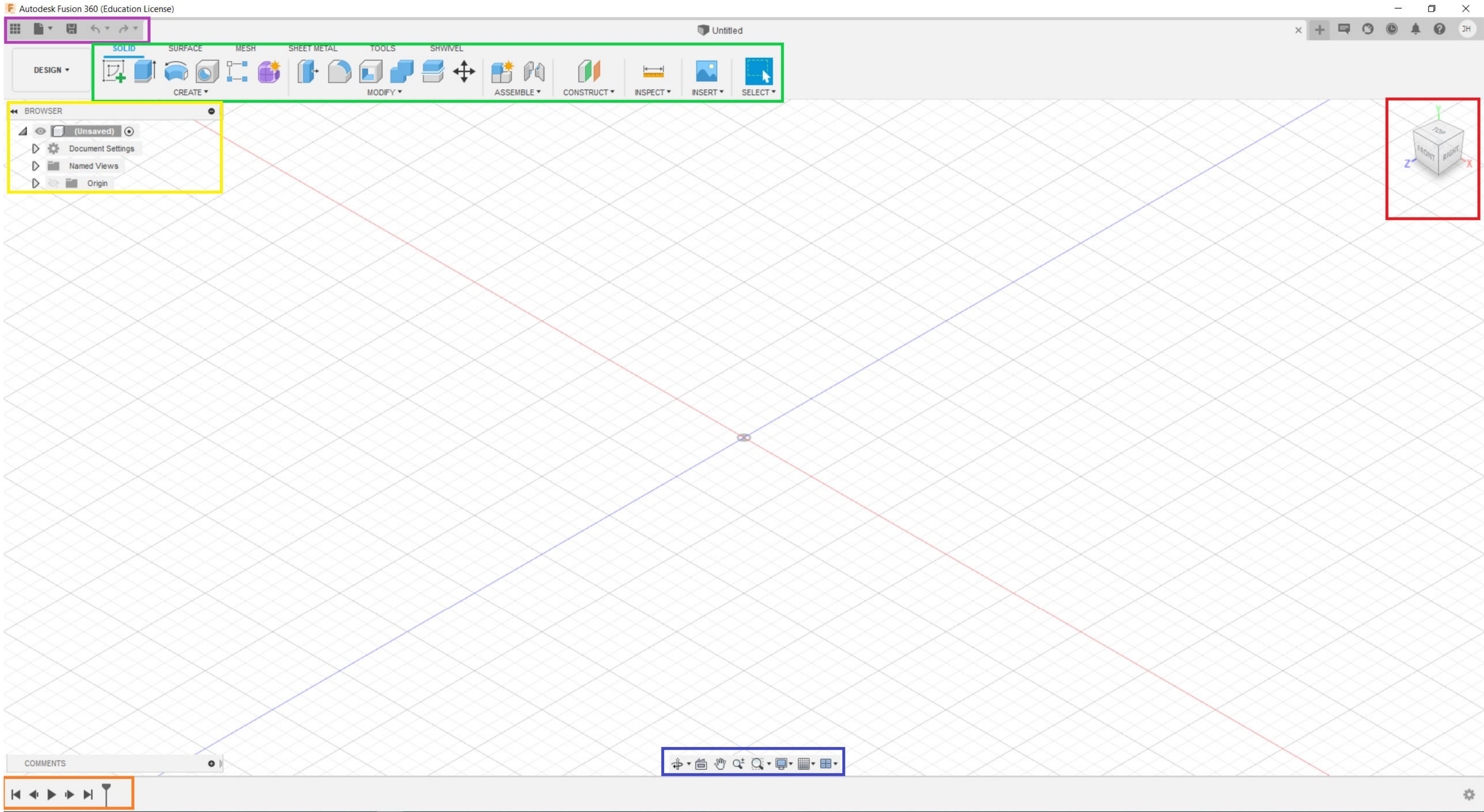


Figure 2 Graphical User Interface of Fusion360

The different controls have been highlighted in different colours for the purposes of explanation. These are outlined in the following chapters as far as they are relevant for the construction of tactile materials.

- Quick Access Toolbar (purple)
- Ribbon (green)
- Browser bar (yellow)
- View cube (red)
- Navigation bar and view settings (blue)
- Timeline (orange)

3.1.1 The Quick Access Toolbar

The quick access toolbar contains all the functions for saving, exporting and loading files.

Figure 3 Quick Access Toolbar

The quick access toolbar contains the following functions from left to right:

- Group data
- File functions
- Save (the actual construction)
- Undo last command
- Repeat last command

The last three of the above-mentioned functions are self-explanatory.

3.1.1.1 Group data

If you save your own designs in the cloud, this function offers the quickest and easiest access to the designs saved there.

By activating the button, the last used project folder appears. The subfolders or constructions created are displayed here. You can open them by double-clicking the left mouse key. You can then close the view by deactivating the button or by clicking on the X symbol in the group data window.

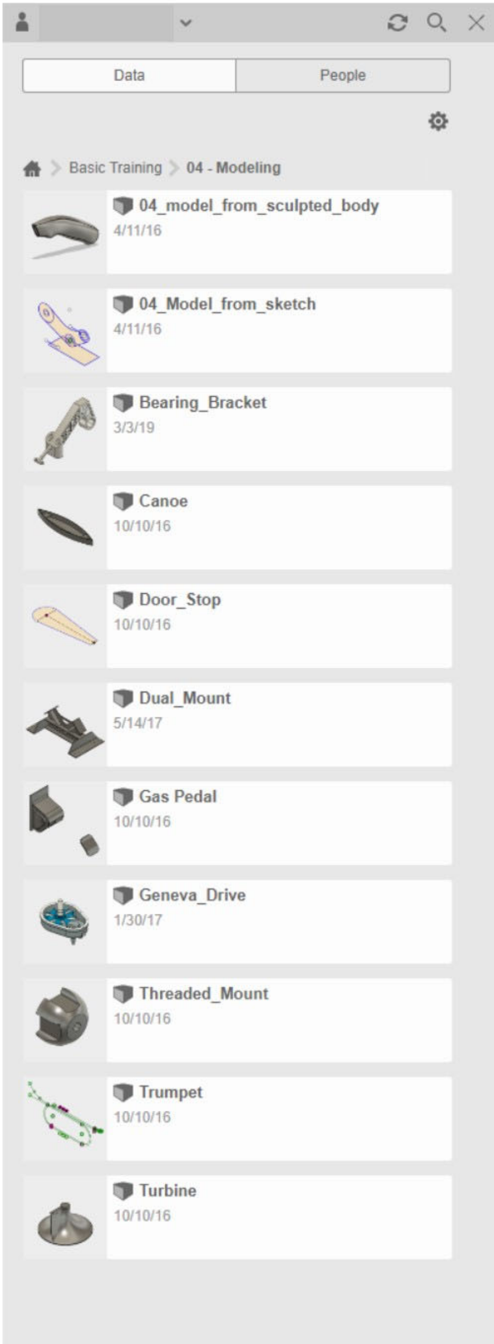


Figure 4 Open group data window with available constructions

3.1.1.2 File function

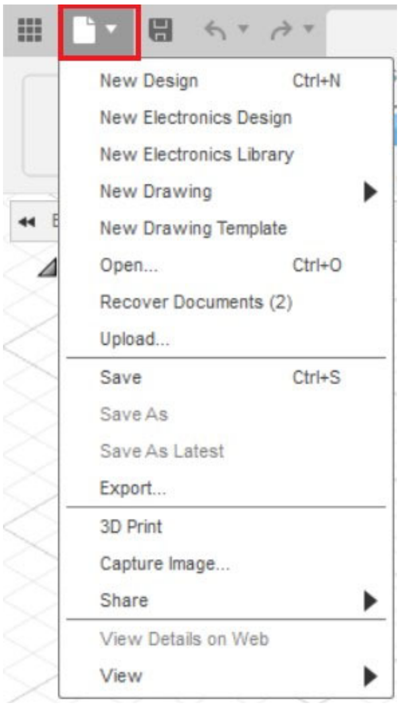


Figure 5 Menu of the “File” function

This menu is divided into four blocks arranged one below the other.
New constructions can be started in the first block. For our purposes, “New Design” is appropriate.

In the second block, the functions “Save (as)” and “Export” are relevant.
The “Save” function allows the file to be saved in the cloud. If no file name has been assigned yet, a dialogue window opens that prompts you to enter the file name and the folder in the cloud. To see the entire dialogue window, the button ▾ in the right-hand area should be activated.
The folders created under the username in the Fusion360 Cloud are displayed here. These folders are called “project” and can in turn be subdivided with several folders.

Corresponding buttons can be used to create new projects at the top level and also to create new folders.

An option to save to a local data carrier is not offered in this function.

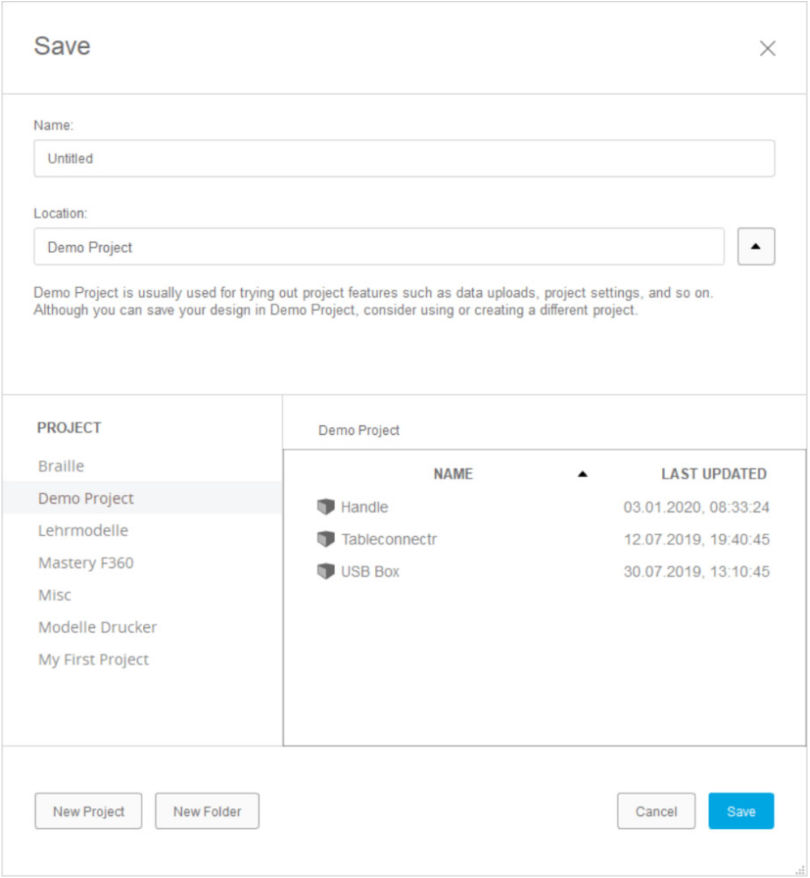


Figure 6 “Save” window

The function “Save” (the actual construction) in the quick access bar is identical to the function described above.

The “Export” function enables a file to be saved locally on a data carrier. The window of this function is similar to the window shown in 6Figure 6 “Save” window. In this case, a reference is made to a local data carrier in the destination folder.

The other functions in the remaining two blocks are less relevant for our purposes.

3.1.2 Ribbon

The Ribbon is the most powerful tool because it contains all the elements required for construction. When starting Fusion360, make sure that the menu item "Solid" is activated. If this is the case, the toolbox corresponds to the following illustration.

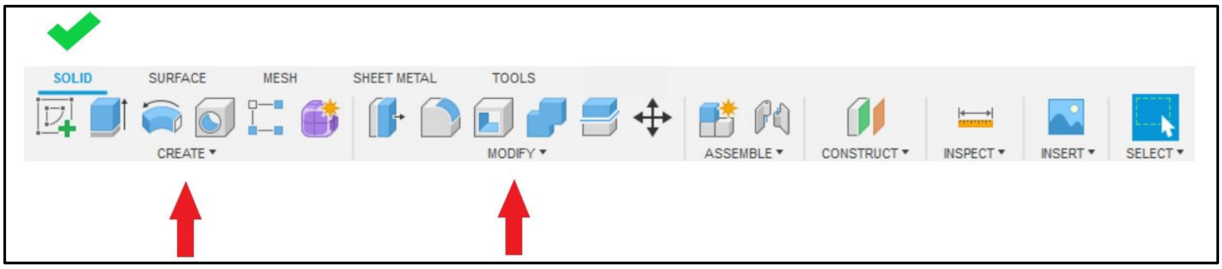


Figure 7 Representation of the ribbon in the "Solid" setting

The Ribbon is divided into several sections, which are subdivided by a grey dividing line. The most important areas are "Create" and "Modify". Above these names, you can find icons with frequently used functions. The menus can be opened by clicking on the small drop-down triangle to the right of the section name.

The functions are described in more detail in the following chapter.

3.1.3 Browser Bar

The Browser bar provides an overview of the created elements within the design. Thus, this browser palette fulfils a similar function to the Explorer in Windows. Especially for complex constructions with many different elements, this function offers a quick overview. In addition, by marking an element, it can be edited directly.

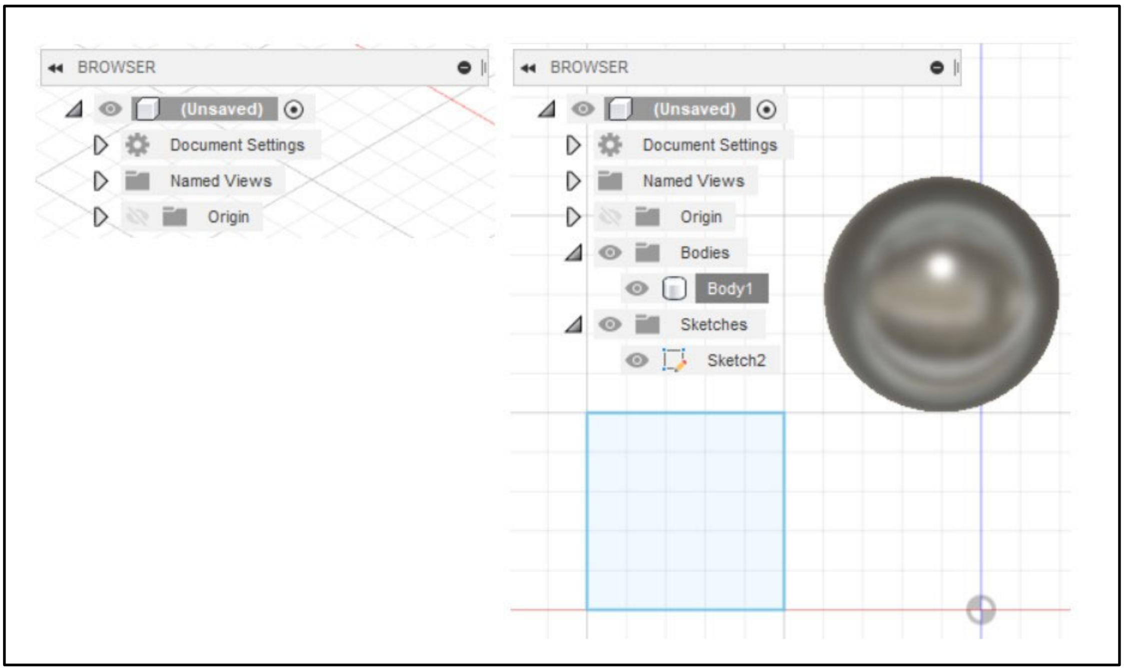


Figure 8 Browser bar. Left: New worksheet. Right: Two elements created

3.1.4 View Cube

The View Cube allows for quick changes of view on all axes. The view change allows a change in 90° and 45° steps. For the 90° steps, simply click on the full surfaces such as “Front” or “Top”. For smaller changes click on the corresponding edges or corners. As soon as the mouse pointer is on a selectable position, the view area is displayed in blue.

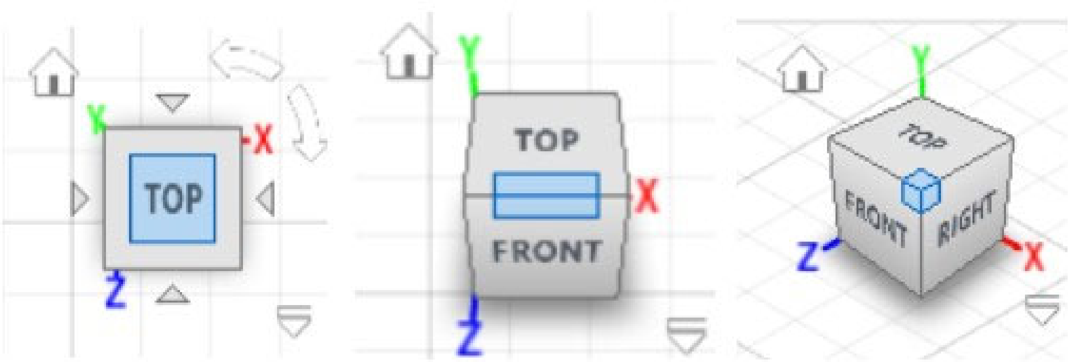


Figure 9 Changing views with the View Cube

Next to the View Cube, there are further symbols. The house symbol takes you to the “Home” view. Here the view is aligned with the centre of the construction and an diagonal view is provided. With the arrows on the upper right side, the view can be tilted by 90°.

3.1.5 Navigation Bar and View Settings

In this toolbar, alterations can be made to the views. The most important buttons are used for free rotation of the working area and for shifting.

The “Orbit” button  activates the free rotation function of the working area. By clicking and holding the left mouse button, you can rotate the working area as desired. In this mode, the mouse pointer changes and can only be used for the rotation function. To exit this mode, press the ESC key.

Moving the working area using the “Pan” button  functions in a similar way. By clicking and holding the left mouse button, the working area can be moved as desired. To exit this mode, also press the ESC key.

To return to the centre of the workspace, activate the “Home” symbol in the view cube.

3.1.6 Timeline

The Timeline is a very important function that allows changes and corrections in the design without having to delete and completely redraw. In the Timeline, each construction step is documented and displayed symbolically.

Figure 10 Timeline in Fusion360

If you want to make changes in the construction, you can jump to the corresponding construction step with the slider and change it. The constructions following this construction step will change accordingly.

In addition, the Timeline is a good methodological tool for coaching Fusion360. By moving the timeline step by step, a student can follow the steps in a construction. This facilitates the creation of Fusion360 lessons that can be followed by students through the timeline.

4 Designing with Fusion360

With CAD programmes, the user must keep in mind that they are designing in a three-dimensional space. The positioning of drawing and construction elements depends on the viewing perspective. With a classic drawing board, you look at and draw the drawing from above. With a CAD programme, however, it is possible to draw from different perspectives. For example, if the viewing angle in the View Cube is “Front”, an element is drawn from the front, i.e. as if it was standing on the drawing board. Therefore, it is recommended to choose the “Top” view.

4.1 The Beginning: Drawing of Sketches

Constructions usually begin with a two-dimensional drawing. These are created using the “Sketch” function. To do this, either select the button  in the toolbox or open the “Create” menu by clicking on the small black drop-down triangle.
Here, you select the function “Create Sketch”.

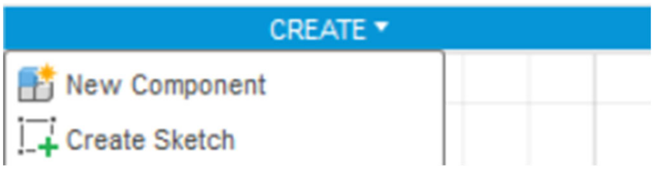


Figure 11 “Create Sketch” menu

After this selection, nothing seems to happen at first. However, if you look closely at the workspace, you will see that a square is displayed in the middle of the screen.

This is the construction plane. Click on this construction plane to create the sketch horizontally to the Z-axis.

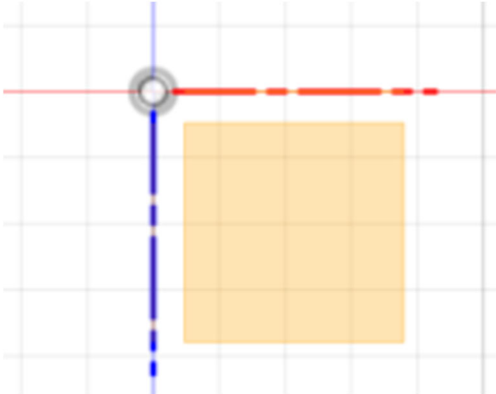


Figure 12 Construction Ipane

Subsequently, the icons in the toolbox change and the most frequently used sketching tools are listed. Again, it is also possible to open the menu by clicking on the triangle next to the “Create” selection.

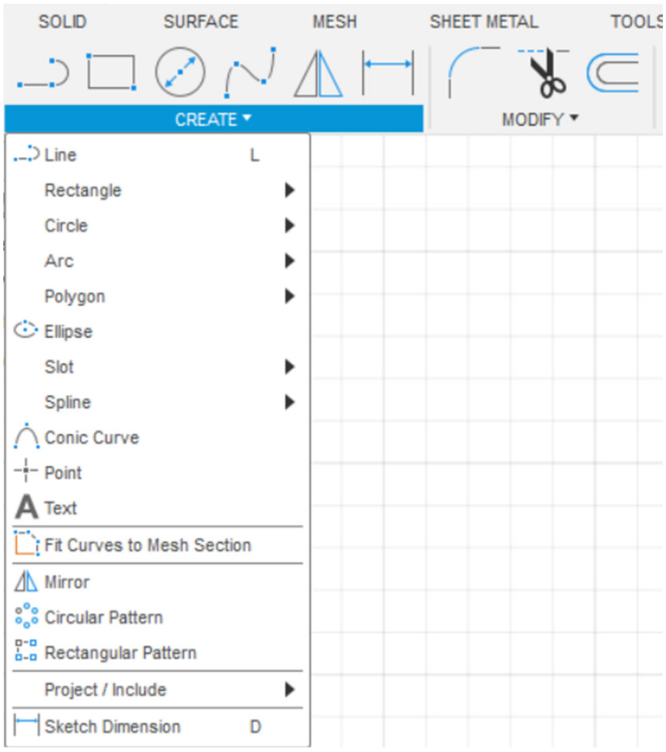


Figure 13 Toolbox and “Sketch” menu

There are several sketch elements in the sketch menu. The most popular ones are:

- Line
- Circle
- Rectangle
- Text
- Spline

A sketch can consist of only one of the upper elements, or a combination of multiple elements. As an example, let's take a building floor plan. This is where all lines, circular and rectangular elements can be created separately as several sketches or as a common sketch. In the example below, all drawing elements belong to one sketch.

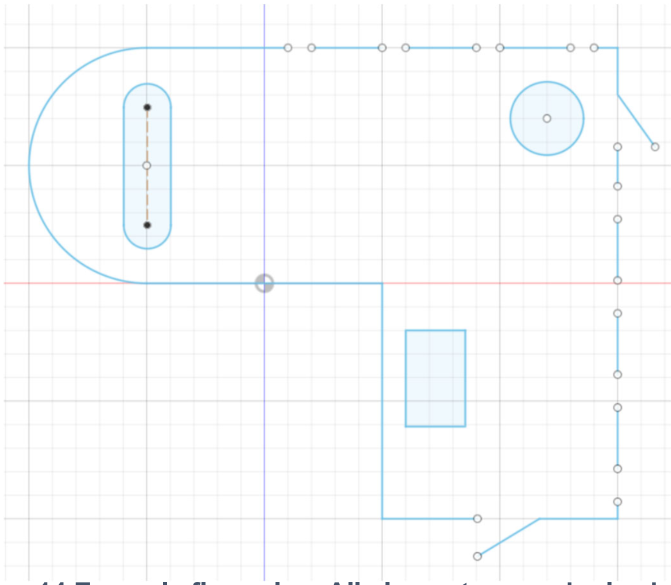


Figure 14 Example floor plan. All elements as a single sketch

The decision whether to create all elements as separate or as a unique sketch depends on the degree of the following operations. If, in the example above, the entire floor plan has to be moved, rotated or scaled, it is better that all elements belong to one sketch. If individual elements need to be changed, it is easier to create them as a separate sketch.

To create a sketch, activate the corresponding button or select the menu item. Frequently used sketching tools can also be accessed directly with a shortcut.

4.1.1 Circle, Rectangle and Line

Circle → c
Rectangle → r
Line → l

After selecting the desired sketch tool, the mouse pointer changes to a crosshair cursor. Position this crosshair cursor at the desired location and drag it while holding down the left mouse button until the desired sketch size is reached. When drawing the sketch, a text field opens. Here you can enter the desired size in millimetres. Decimal places must be entered using the decimal point.

For rectangles and squares, you can switch between the text fields using the tab key.

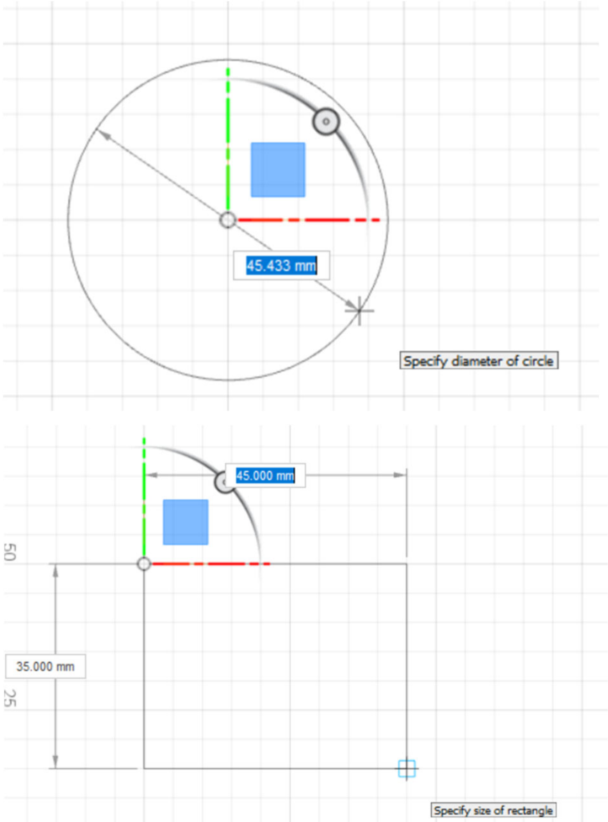


Figure 15 Sketch tool with text field for size specification

Once the sketch has been completed, it must be finalised by clicking "Finish sketch". This function is located as an icon on the far right of the toolbox or in the "Sketch Palette" dialogue window.

More complex shapes, such as floor plans, can be created using the "Line" function. This function also allows parameters to be entered via two text fields. One parameter is the length, the other is the angular dimension to the selected drawing level. By means of a **single** left mouse click, the line is drawn from a starting point to the end point. Subsequently, another line can be drawn starting from the last end point. The line function is completed by either pressing the enter key or double clicking the left mouse key.

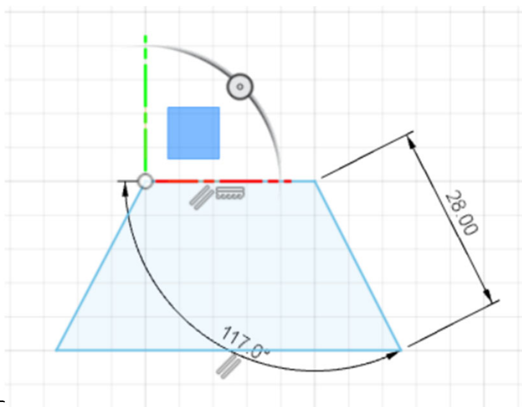


Figure 16 Trapezoid drawn using the line function

Drawing lines in particular requires some skill in using the mouse. A hasty mouse click immediately results in an unwanted line. The easiest way to remove a line is to use the "Undo command" symbol in the quick access toolbar.

4.1.2 Polygon, Spline and Text

These three tools enable the creation of more complex shapes as well as the design of text surfaces.

With the "Polygone" function, polygonal building outlines can be created very easily. The "Spline" function allows the creation of curved lines or contours. With the "Text" function, three-dimensional text elements can be created.

Polygons are created by specifying the diameter and the number of corners. In order to be able to draw a polygon, the command "Create Sketch" must first be selected again. After clicking on the construction plane, the first option "Circumscribed Polygon" is selected from the Sketch menu. Subsequently, select the centre of the polygon in the construction area and draw the polygon by holding down the left mouse button. Here, too, the diameter and the number of corners can be entered using text fields.

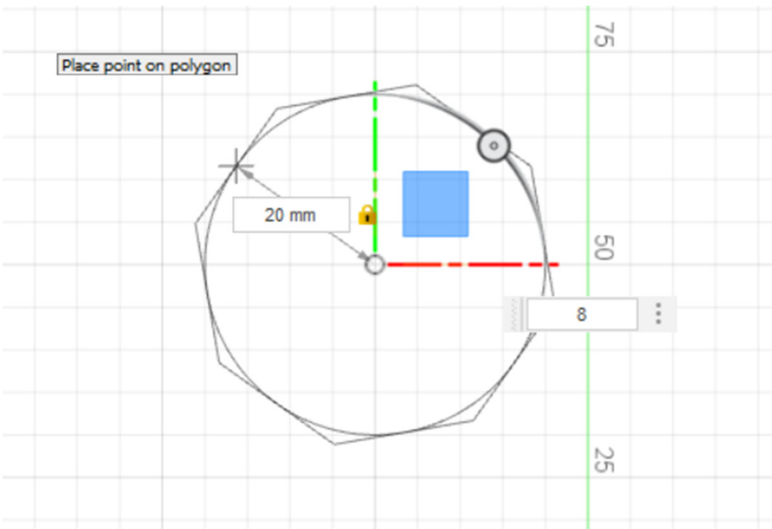


Figure 17 Circumscribed polygon

Once you have drawn the polygon, do not forget to press the "Finish Sketch" button to exit sketch mode.

The "Line" function can be used to draw a variety of floor plans. However, this function is not suitable for drawing round shapes. Instead, the "Spline" function is much better suited for this. After selecting the "Spline" function, you can use the mouse pointer to set various points of progression. Fusion360 connects these points with an arc according to the position of the previous point. The shorter the distances between the points, the smaller the radius between them.

After completing the spline, press the enter key to exit the input mode. You can now see the line together with all the black points created. On both sides of each of these black points are green points connected by a green line. This is an angular curve that allows you to change the angle of the curve defined by the black point. By keeping the left mouse button pressed, the green points can be moved. The arcs change accordingly.

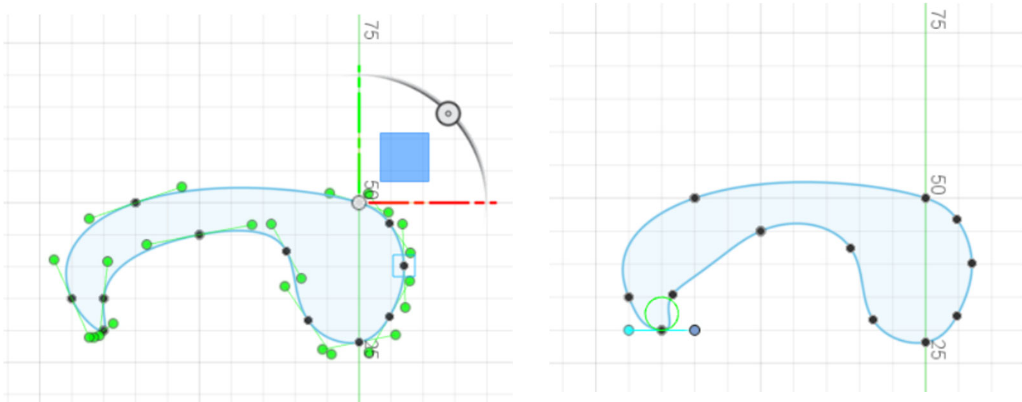


Figure 18 Spline sketch. Left: Spline sketch source. Right: Arc bottom left changed

To change arcs in the spline, the black points can be moved by holding down the left mouse button.

The sketch function “Text” allows you to insert text elements. Various fonts and text attributes are available. However, not all fonts and attributes are suitable for 3D printing as they are too fragile and do not allow for stable printing. Yet, sans-serif fonts such as Arial and Verdana are suitable, provided you do not go below a character size of 6mm. If the character size is too small, punctuation marks and umlaut dots in particular are too fine to withstand even light mechanical stress.

Inserting the text is very simple. After opening the sketch mode and selecting the text function, you can select the text orientation. Two icons are displayed for this. By default, a straight text alignment is selected with the first icon . However, it is possible to select a winding or angled text orientation with the second button . For the latter, however, a circle, spline or rectangle must be available as the required gradient path. In this description we will only deal with the straight orientation.

After selecting the orientation, draw a rectangle along the desired text position with the mouse. The left mouse button is used to mark the text position.

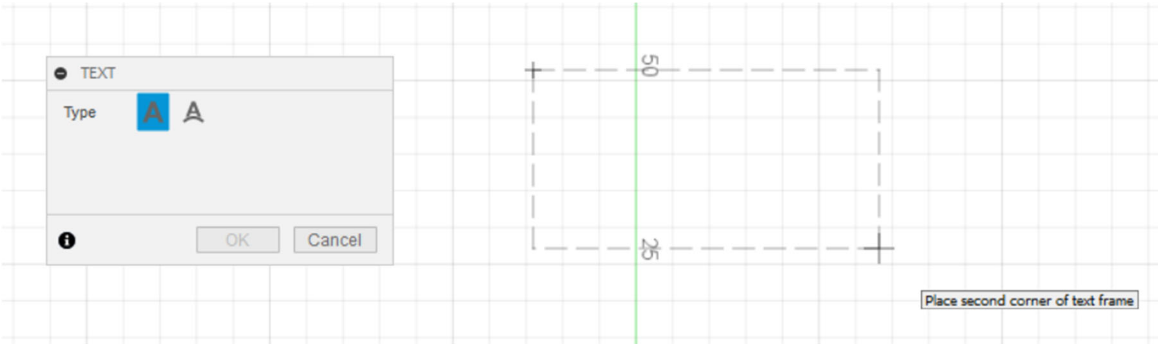
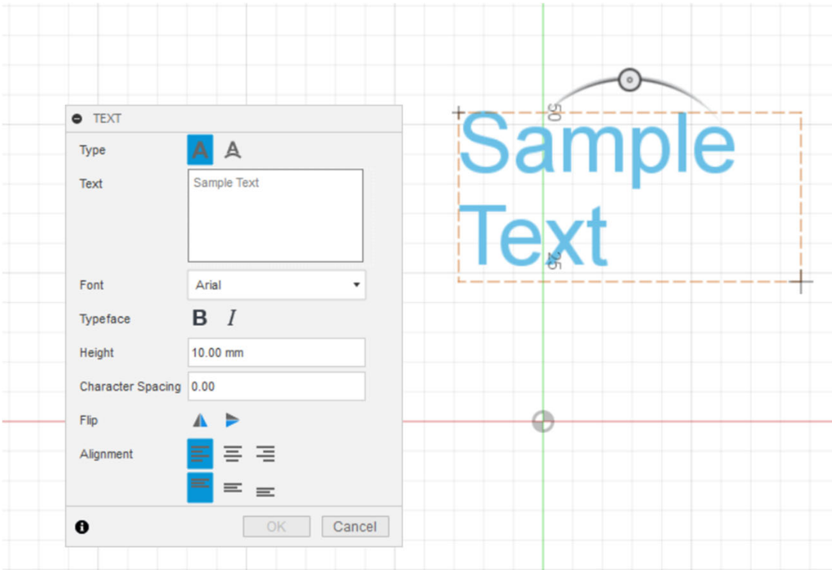


Figure 19 Text function and text positioning

When placing the position frame, adherence to the exact dimensions is not relevant at first as the text can still be repositioned precisely afterwards. As a rule, the position frame is set from top left to bottom right with the mouse.

The dialogue window for setting the text settings will open. This is based on the functions of text editors and is therefore easy to understand.

Figure 20 Setup options for text



The default “Sample Text” can be overwritten in the text field. Line breaks can be inserted by pressing the enter key. The box below allows you to select the font type. Below this is the option to select the typeface (bold or italic). For reasons of stability, italic fonts are only suitable for 3D printing non-functional models.

The font height is automatically set to 10mm. However, another value can be entered. As already mentioned above, the height value should not be less than 6mm.

After setting all options, click the “OK” button to complete the process. The editing area is now opened and the text can be repositioned with the mouse pointer. Finally, press the “Finish sketch” button.

4.1.3 Moving and copying sketches

If a sketch needs to be moved, there are two methods for achieving this:

1. Move after finishing the sketch
2. Move in edit mode of the sketch

1. Once the sketch is completed, **all** elements of the sketch to be moved have to be marked. This is done by clicking the mouse while holding down the shift key or by marking with a selection rectangle using the left mouse button. The last option is only possible, however, if two sketches do not overlap and only one of them is to be moved.

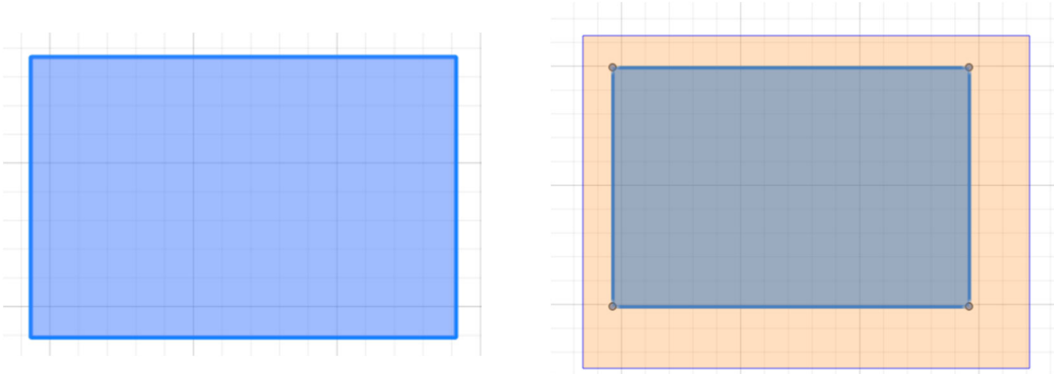


Figure 21 Marking sketch elements. Left: By mouse clicking on the relevant lines. Right: Using a selection rectangle

Once the elements have been marked, the sketch can be moved. To do this, select one of the lines by clicking on it with the left mouse button held down and move it to the desired position by moving the mouse.

2. The editing mode also offers the option for the user to copy a sketch. In addition, moving can be done very precisely by entering the parameters.

If the editing mode has been closed by clicking the “Finish sketch” button, it must be reactivated. To do this, activate the sketch in question in the browser palette by clicking on it with the mouse. Then, press the right mouse button to open the context menu as shown in Figure 22 Browser bar and context menu”. In this menu, select the option “Edit Sketch”.

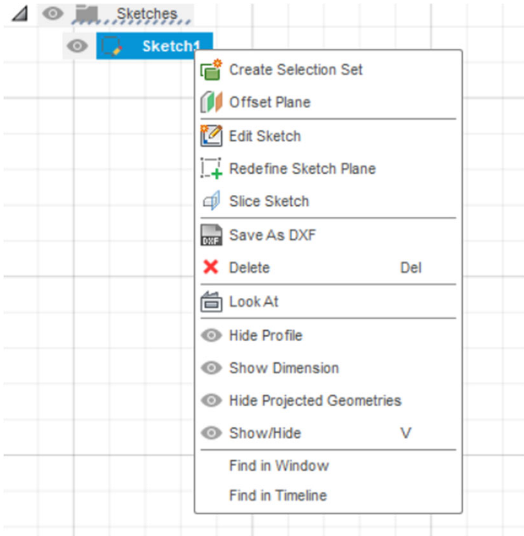


Figure 22 Browser bar and context menu

After opening the editing mode, the sketch to be moved must be marked as described in 1. However, the context menu can now be activated by hovering the mouse pointer over one of the lines of the sketch and clicking the right mouse button. This provides you with the option "Move/Copy".

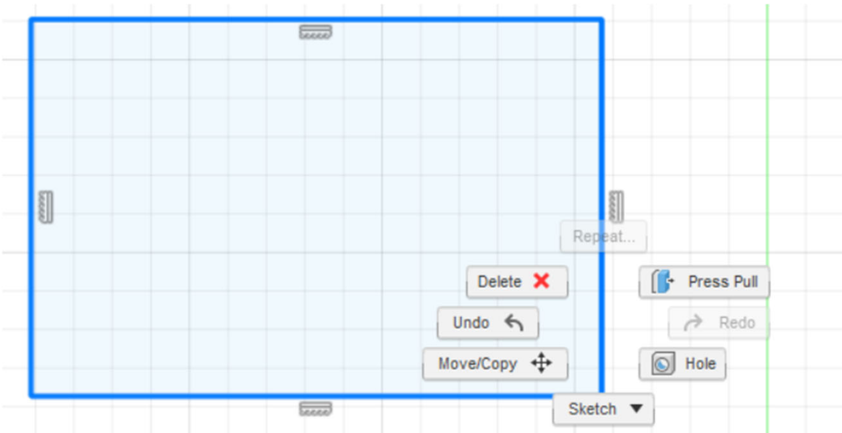


Figure 23 Context menu in sketch editing mode

This option can be selected via a mouse click and results in the "Move" widget now appearing on the sketch. Figure 24 depicts this widget containing two movement arrows and a small circle to change the angle of the sketch. By clicking on one of the arrows, the sketch can be moved by mouse along the axis. Moreover, a text field is opened. By entering a value, the sketch is moved in the direction of the arrow. If a negative value is inserted, the sketch is shifted in the opposite direction. This option is much more precise than moving the sketch with the mouse as it depends on the zoom factor. Sketches that have been zoomed out of can only be moved with little precision using the mouse.

By clicking on the small circle that is located on an arc, you can rotate the sketch. Again, a text field is opened that allows you to enter an angle measure.

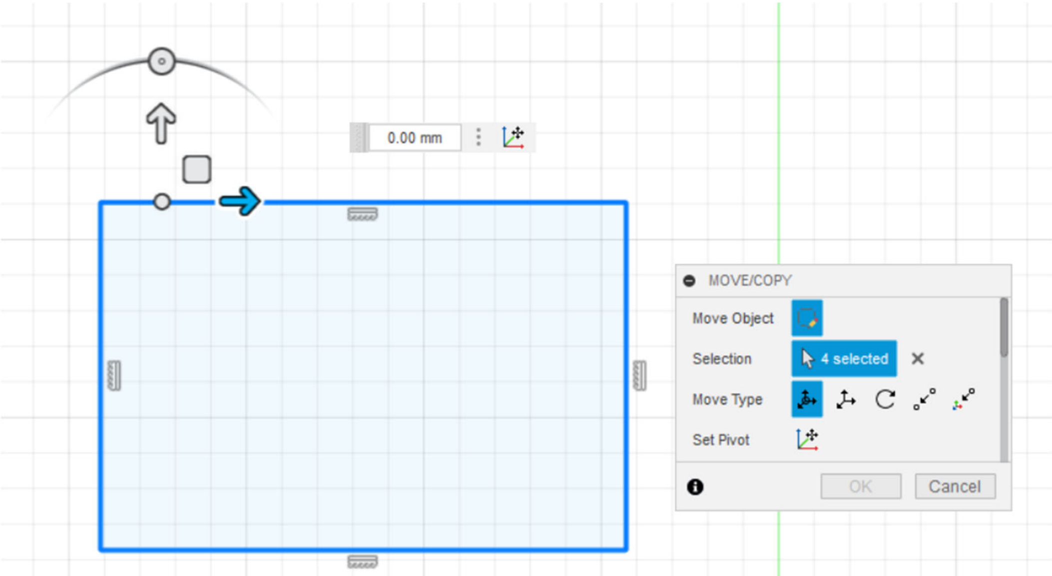


Figure 24 “Move” widget in the “Top” view

The “Move” widget depends on the view mode of the sketch. In Figure 24 “Move” widget in the “Top” view), the view “Top” has been selected. This only permits movement with the arrows in the X and Y axes. However, if you select other views, i.e. “Front”, “Back”, “Left” or “Right” in the view cube, the sketch can also be moved along the Z axis.

Copying a sketch can also be done via the context menu. Instead of selecting the “Move/Copy” function, simply choose “Copy”. Subsequently, move the mouse to the location where the copy is to be pasted. Click the right mouse button to open the context menu again and select “Paste”.

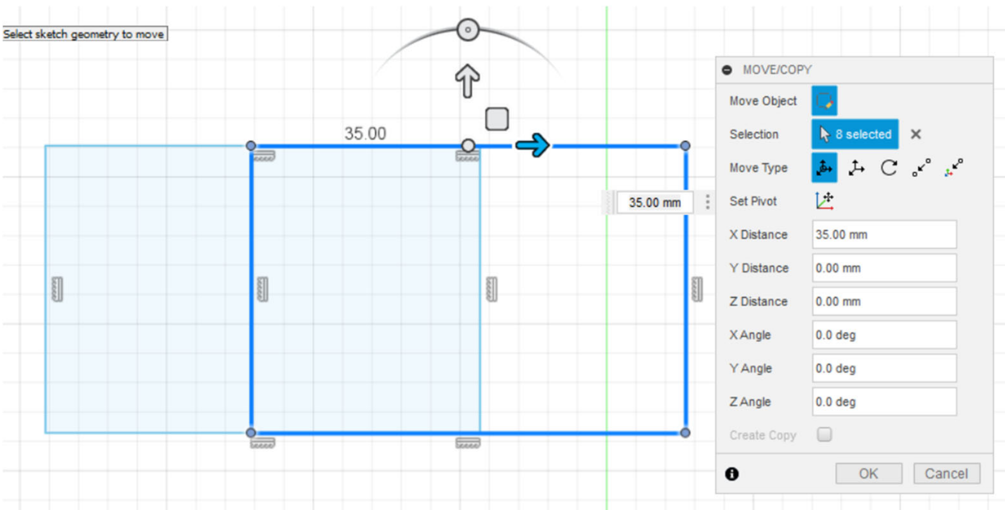


Figure 25 Paste mode after activating the “Paste” option in the context menu

As shown in Figure 25, the “Move widget” allowing for a sketch to be moved is displayed. To finish the moving/copying process, press the “Finish Sketch” button.

4.1.4 "Offset" and "Trim" function

There are two important functions in sketch mode that are particularly helpful when creating site maps and floor plans.

Imagine you are drawing a winding road for a site map. You do this, of course, using the Spline and Line function. To draw the entire width of the road, however, you would have to draw an exact parallel line. To facilitate this process, you simply need to select the "Offset" function in the "Modify" toolbox.

Once "Offset" has been activated, select the reference line using your mouse. A dialogue box with a text field and a red offset line will appear.

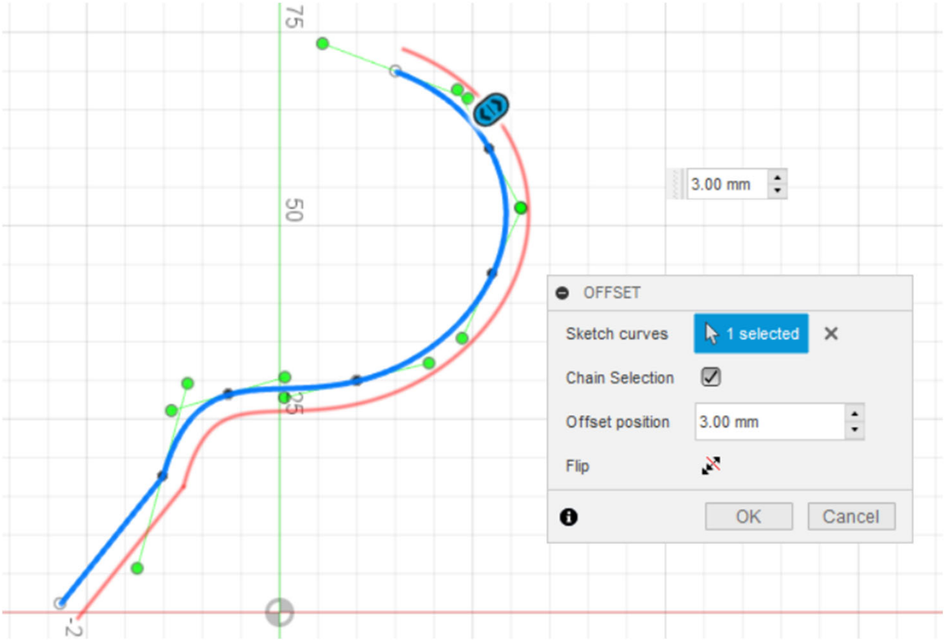


Figure 26 "Offset" function after having selected a reference-line

The distance between the two lines can be entered in the text field. A negative value relocates the red offset line to the opposite side of the drawn (blue) line. This function can also be performed with any other sketch element such as a square, circle or polygon.

The distance between the two lines is entered in the text field. With a negative value you shift the offset line to the opposite side. This function can also be performed with any other sketch element such as square, circle or polygon.

In the case of tactile site plans for buildings, it may happen that different sketch elements overlap if the floor plan is not solely in a rectangular shape. In such cases, the floor plan may consist of a combination of lines, squares, circles or polygons (see: Figure 27). Creating an offset for the entire outer wall of such a complex floor plan will not work because the offset function is only applicable to one sketch element at a time.

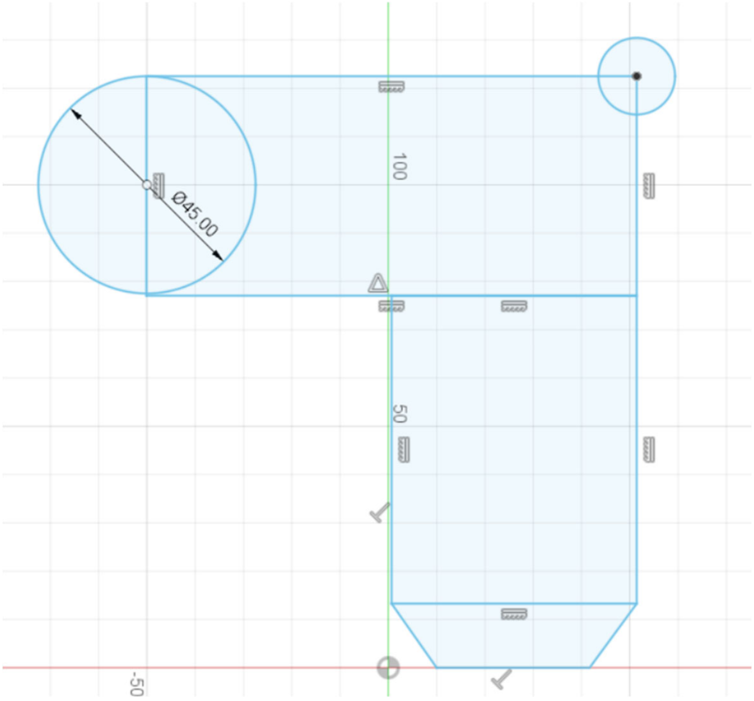


Figure 27 Example floor plan from several sketch elements

It can thus be useful to delete disturbing or superfluous lines. This can be done with the “Trim” function in the “Modify” section of the toolbox.

After activating this function, all you need to do is click on the lines to be deleted with the mouse. As soon as the mouse pointer is hovering over a deletable line, the line colour changes. To delete the selected line, click the left mouse button.

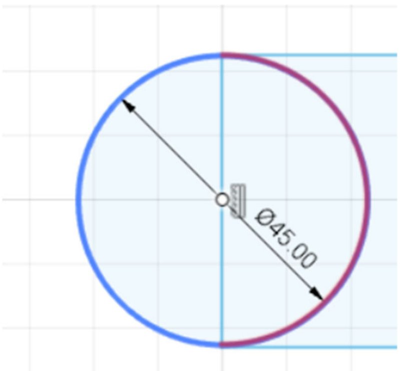


Figure 28 “Trim” function and highlighted line

This way, all superfluous lines can be deleted one after the other. To undo the deletion of a line, press the reverse button or use the key combination Ctrl + Z.

4.2 Arrangement of Sketches

Creating recurring elements on a circle while maintaining the same spacing would be very difficult to realise without the “Circular Pattern” function. In the example below, a clock face is to be created. Circles for the different hours are to be arranged on a circular line. To do this, one must first draw only one circle with the diameter of the circle line on which the clock numbers are to be arranged using the sketch function. Then, another circle the size of the clock numbers is drawn on the circle line. Other shapes could be chosen instead of a circle for the digits. Next, one selects the “Circular Pattern” function in the Sketch menu before completing the sketch.

In the “Objects” line of the dialogue window of this function, select the circle for the hour digits with the left mouse button and click on the small circle. Then, for the “Centre Point” field, select the large circle of the sketch and subsequently enter the number “12” in the “Quantity” field. Immediately, you will see a preview of the sketch. Click on the “OK” button to complete the layout.

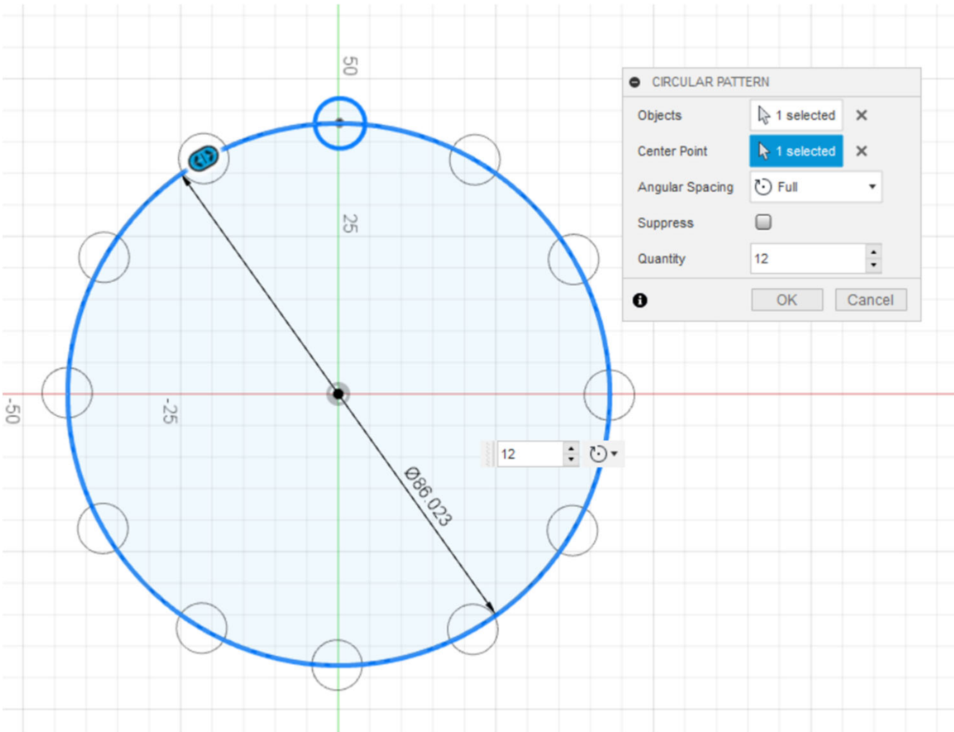


Figure 29 “Circular Pattern” function

The “Rectangular Pattern” function works in a similar way to the function described above. In this case, an arrangement can be made along the X and Y axis. The sketch to be arranged must be selected in the dialogue window. In addition, specify the direction, distance and number of duplications.

4.3 From a Sketch to a 3D Solid

For this purpose the “Extrusion” function in the toolbox “Solid” is used. This function can also be activated with the key command “e”. After selecting the function, mark the sketch that is to be converted with the mouse. Note that only closed sketches (these have a light blue background) can be extruded. Further details can be selected in the “Extrusion” dialogue window.

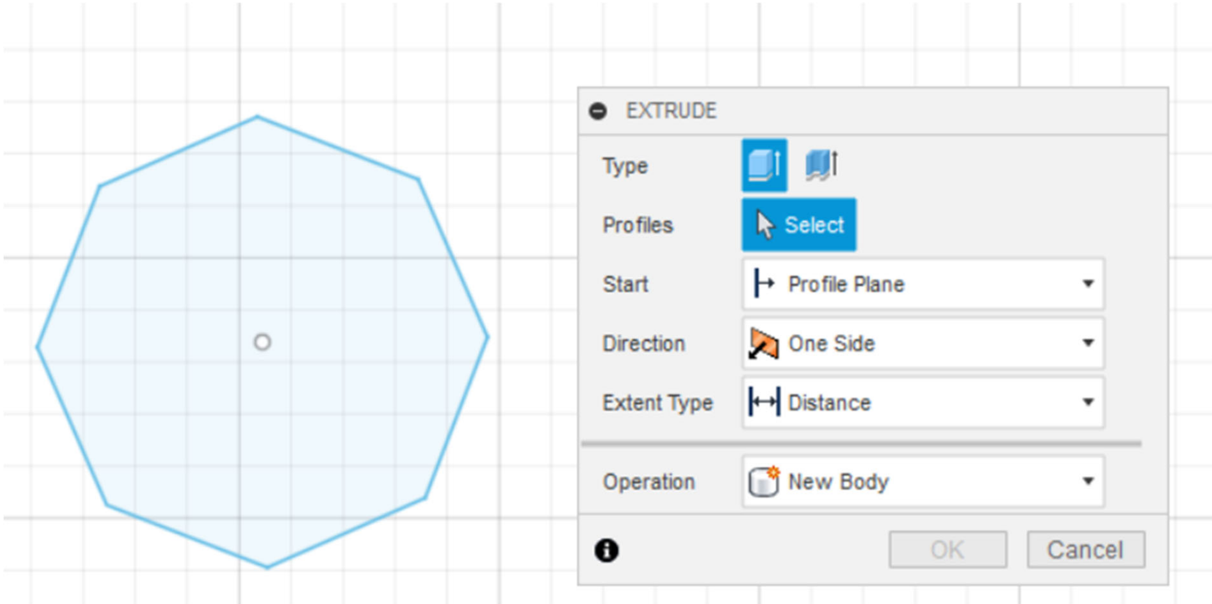


Figure 30 “Extrusion” dialogue window

For the production of tactile learning materials, the default settings of this function are completely sufficient.

After selecting the inner surface of the sketch with the mouse, the dialogue window expands to include two text fields. In the text field “Distance”, the height of the extrusion is to be indicated in millimetres. In the example shown in Figure 31, the sketch is extruded by 20mm. If a negative value is entered (e.g. “-20”), the extrusion takes place into the opposite direction.
The solid has a grey shading that distinguishes it from a sketch. The “Taper Angle” function provides the solid with a funnel shape for positive angle values and a cone shape for negative values. However, this input function is less frequently used.

The created elements are called “solids” in Fusion360.

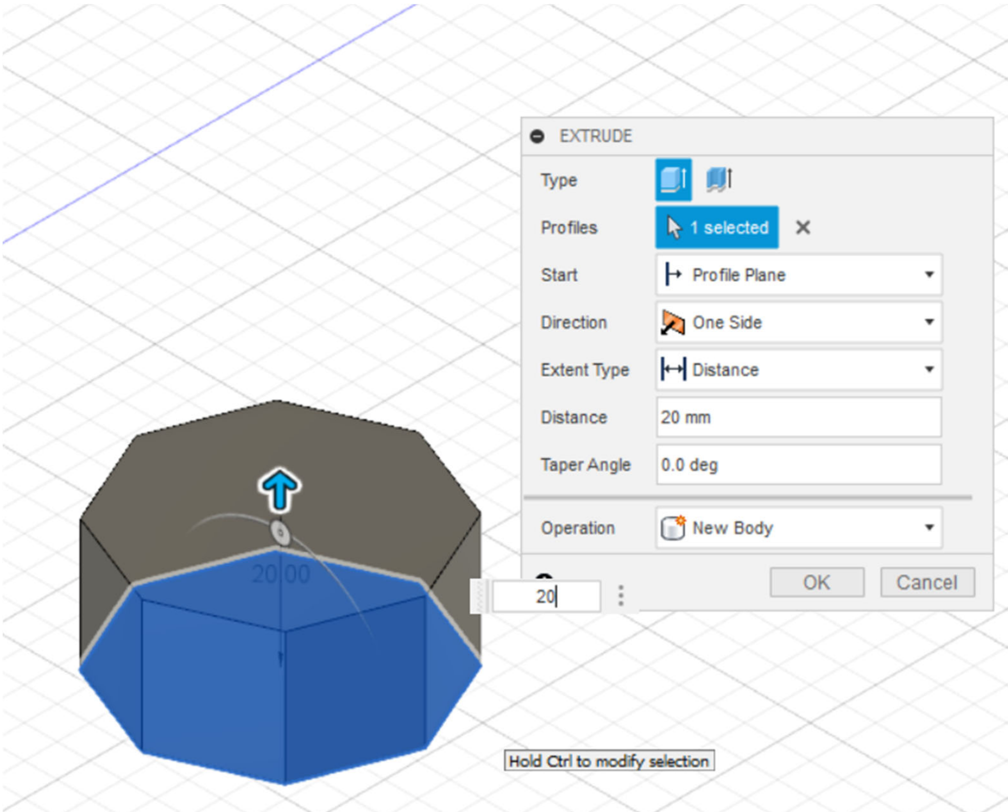


Figure 31 “Extrusion” Dialogue

As soon as a solid has been created, it is also visible in the left browser palette and consecutively numbered as solid 1, solid 2, etc.

The “Extrusion” function can be used with any closed sketch as described above. Text can be transformed to solid in the same way. For the latter, the function first needs to be activated and then the desired letter is to be selected with the mouse. Again, the desired height of the solid to be created needs to be entered into the text field “Distance”. After selecting the “OK” button, the text sketches are transformed to several solids. Each character represents a separate solid and is also listed in the browser palette.

4.4 Creating Solids Without the Sketch Function

Fusion360 further offers the possibility to directly create complex solids. This way, cuboids and cylinders can also be created without having to draw a rectangle or circle first.

The “Sphere” and “Torus” functions are particularly relevant in this regard and both very easy to use. Once the “Sphere” function has been activated, you only have to mark the position with the mouse and then enter the desired diameter.

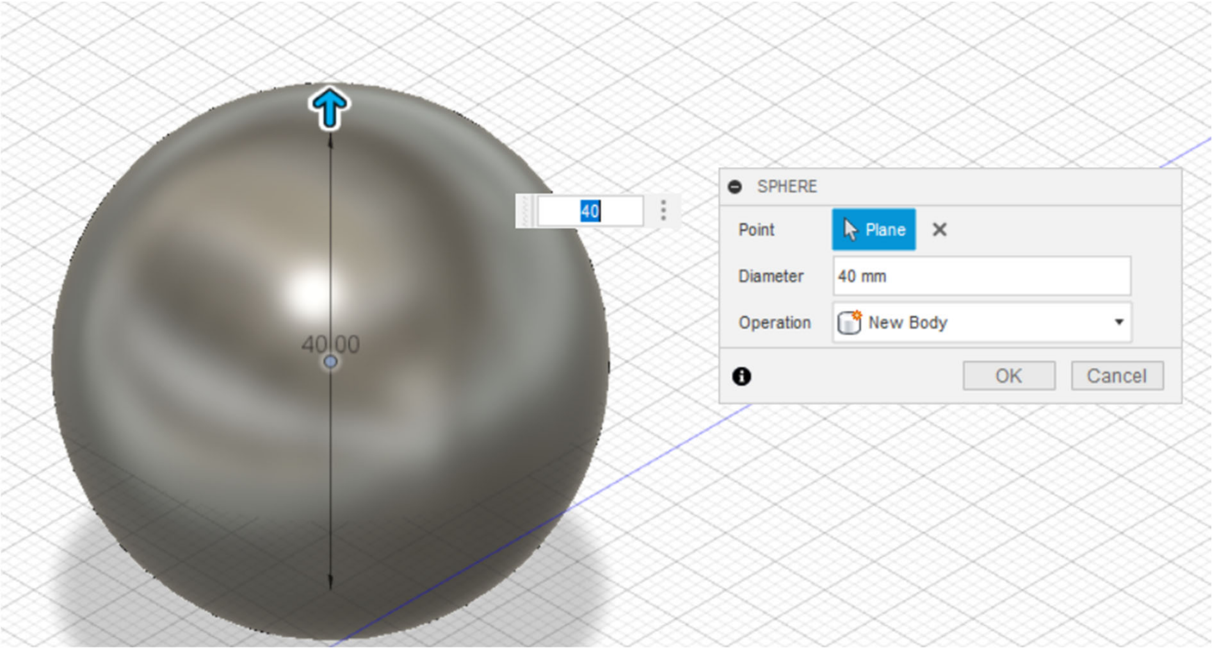


Figure 32 "Sphere" function

The "Torus" function allows for the creation of a solid that is very suitable for representing roundabouts. It would be time-consuming to construct such a solid using the sketch function.

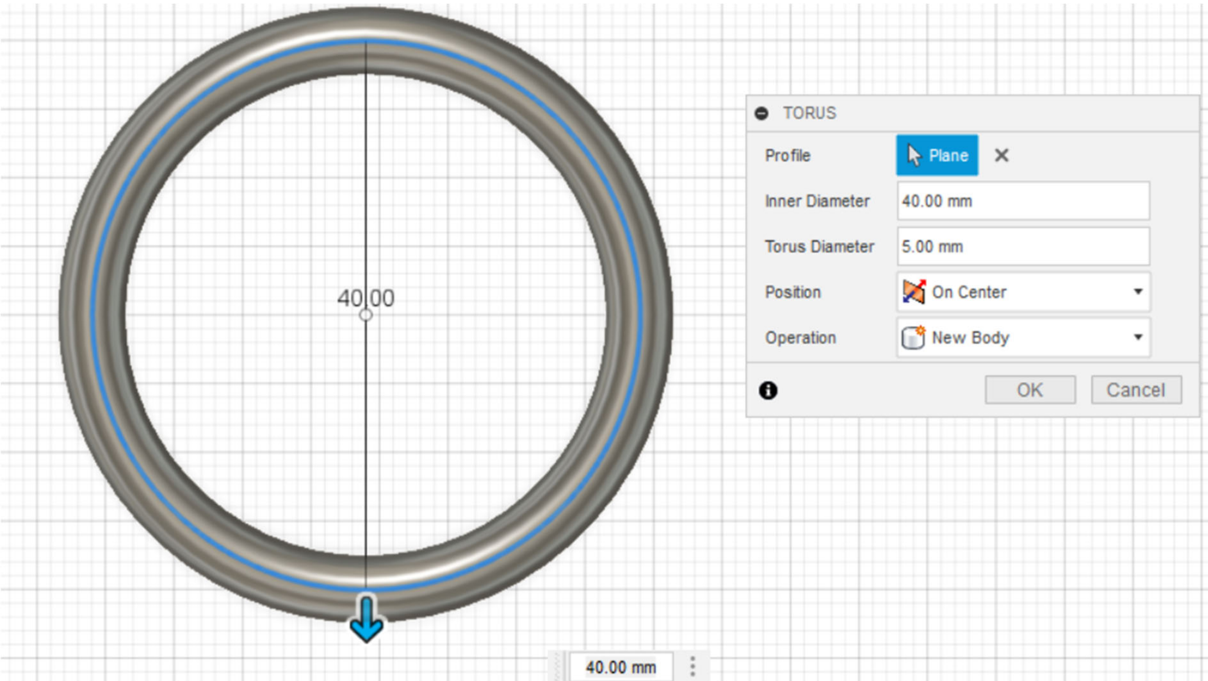


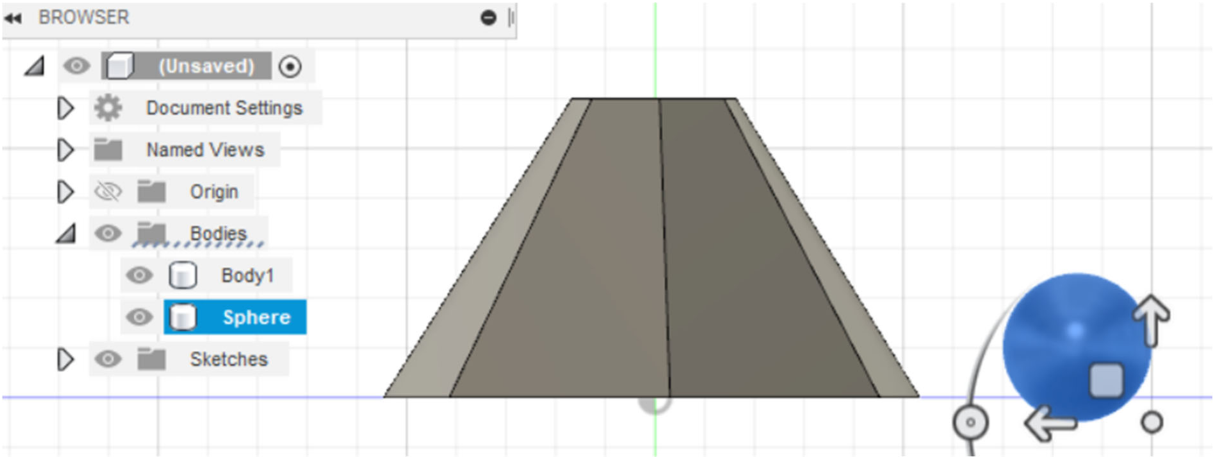
Figure 33 "Torus" function

To construct such a solid, simply enter the inner diameter in the first text box and the thickness of the torus (i.e. torus diameter) in the second text box. Note that the inner diameter in the default setting corresponds to the diameter that runs through the centre of the torus. To change the measuring point of the diameter, change the value in the “Position” field.

4.5 Moving and Copying Solids

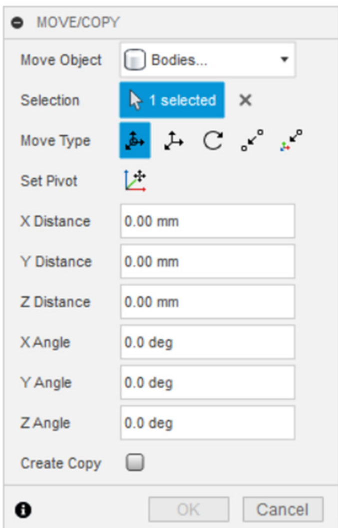
Moving and copying solids is just as easy as it is for sketches. In the example shown in Figure 34, the sphere is to be moved to the top of the polyhedron.

Figure 34 Moving solids with the motion widget



The most convenient way to achieve this is to select the solid to be moved in the browser palette by clicking on it with the mouse. Just like it is the case for sketches, the movement widget then appears. Using the direction arrows, the selected solid can be moved in the corresponding direction with the mouse. In the example provided in Figure 34, the sphere can only be moved along the Z and X axes in the selected view. In order to position the sphere exactly in the centre of the top of the polyhedron, “Top” view must be selected in the View Cube. This allows the solid to be moved along the Y-axis.

As is the case when moving sketches, a dialogue window pops up next to the movement widget, where the corresponding values can be entered manually. In order to copy the selected solid, tick the “Create Copy” check box in the lower part of this dialogue window.



Co-funded by the
Erasmus+ Programme
of the European Union

Figure 35 Dialogue window "Move/Copy"

4.6 Creating Solids with the "Revolve" and "Sweep" Functions

Fusion360 offers other helpful functions to create solids from sketches. The "Revolve" and "Sweep" functions allow for the creation of solids from sketches that originate from a revolve axis or path.

4.6.1 "Revolve" Function

This function allows the simple creation of round solids that are rotated around an axis. In the example given below, a champagne glass is to be created. To do this, one must first create only one half of the glass using the "Line" sketch tool. Subsequently, the "Revolve" function in the toolbox in the "Create" menu needs to be applied. In the first step, mark the profile to be rotated. To do this, left click on the surface area of the sketch. In the next step, select the rotation axis around which the sketch is to rotate.

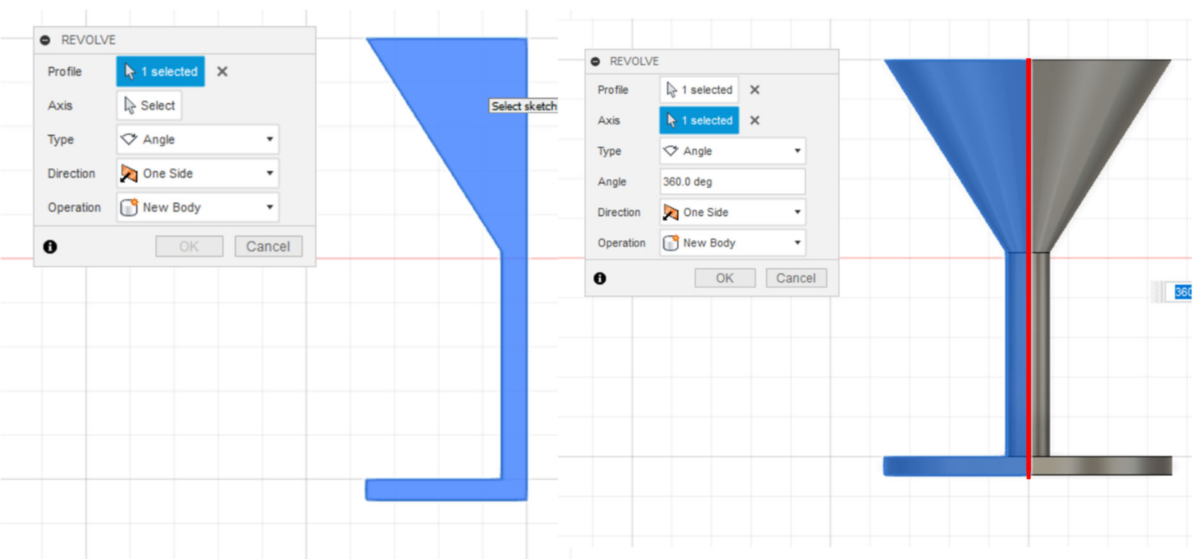


Figure 36 "Revolve" function. Left: Selection of the profile. Right: Selection of the axis (red)

Moreover, the axis also needs to be selected using the left mouse button. The angle of rotation can then be entered in the dialogue window. In the above example, the value remains 360°.

4.6.2 "Sweep" Function

The "Sweep" function allows for the creation of curved, twisted or angled solids from at least two sketch elements. The starting point for this is usually the following:

1. A line that serves as a directional path.
2. A geometric shape arranged at right angles to the path.

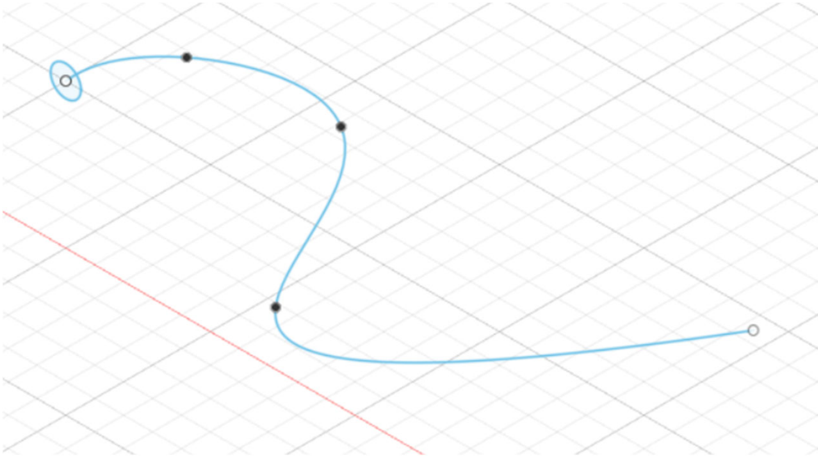


Figure 37 Elements available for the "Sweep" function

The geometric shape can be extruded using the "Sweep" function in the direction of the line, the directional path. To do this, first, select the menu item "Create" from the toolbox and then, select the function "Sweep".

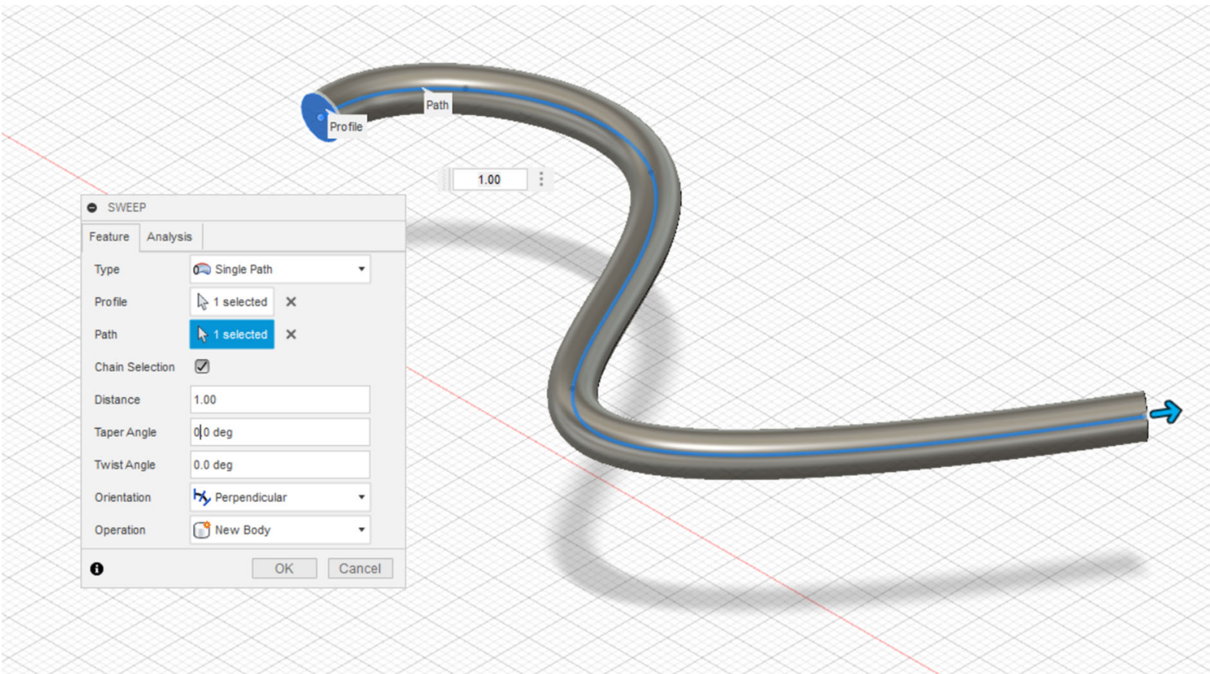


Figure 38 Window of the "Sweep" function

5 Exercise: Floor plan

The previously explained functions are completely sufficient to create a tactile room plan. The aim of this chapter is to outline how to create a tactile map of a conference room as shown in Figure 39.

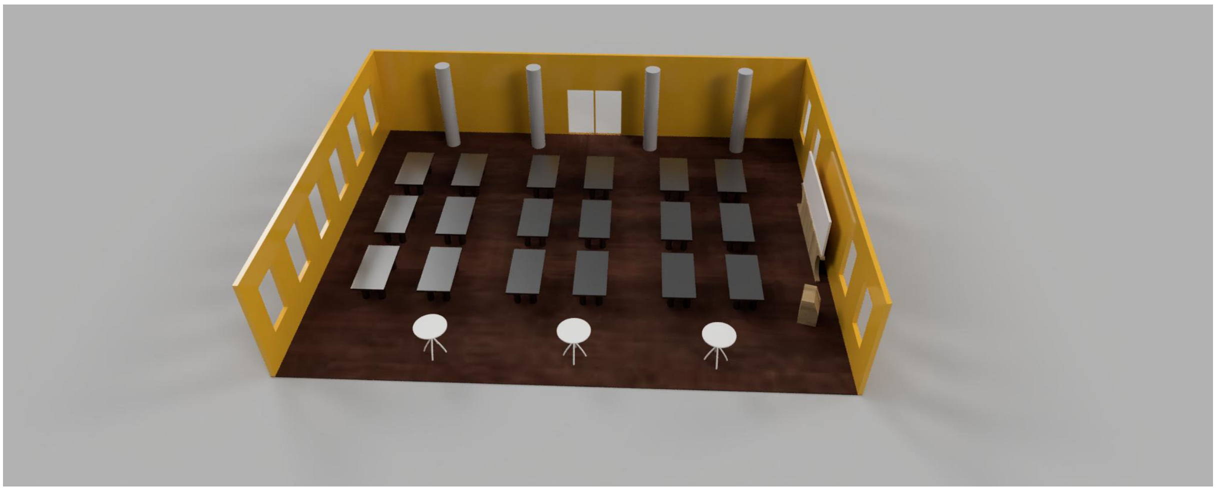


Figure 39 View of a conference room for the exercise

With regard to carrying out the exercise, it must be noted that no emphasis needs to be placed on the exact scale of the floor plan, but rather on the application of the functions of Fusion360.

The objective is to create a tactile site plan that shows blind visitors the location of the:
- entrance doors,
- tables and table arrangements,
- pillars,
- screen, and
- lectern.

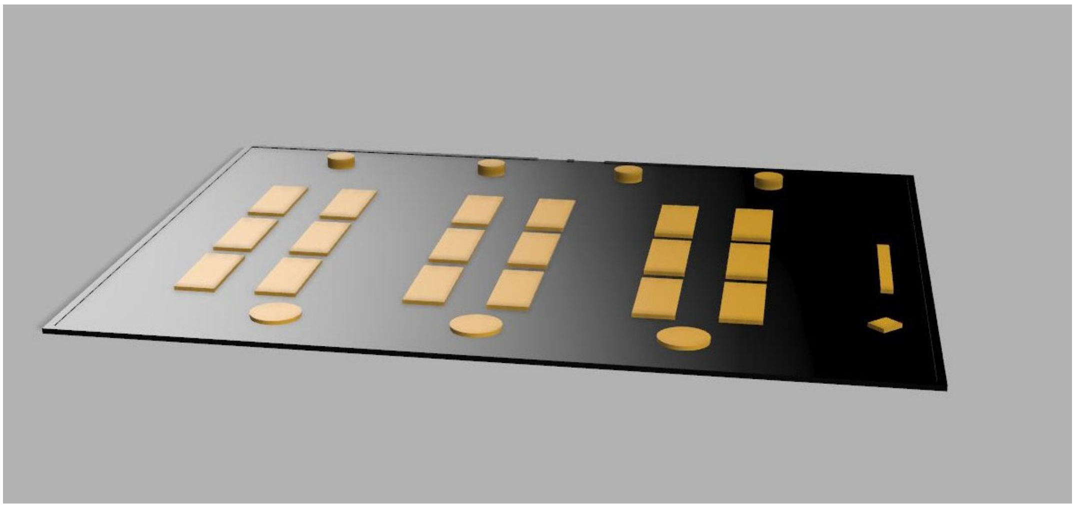


Figure 40 Aim of the exercise: tactile floor plan

As a reference for the dimensions that are used, refer to the drawing in the appendix (chapter 12).

5.1 Floor Plan and Base Plate

The base plate on which the displayable elements are located is the basis for tactile site plans. In practice, the size of the site plan is often determined by the maximum size of the 3D printer. The larger the size of the 3D printer, the more details can be displayed on the tactile site plan. As a rule, tactile site plans produced in 3D printing also follow the principles of representation known from plans on swell paper or the thermoforming process. This is sufficient in most cases. However, the 3D printing process also offers the possibility of representing special elements in full and not just in plan. In this context, mobility trainers will also learn how to use the new possibilities to their advantage.

In the present exercise, only the flat outline of the floor plan is to be created.

The minimum thickness of the base plate should not be less than 2 mm to ensure sufficient stability.

For this exercise, it is assumed that the 3D printer has a maximum size of 200x200 mm. Therefore, a base area of 200x158mm was used for the base plate. To create it, select the “Sketch and Rectangle” function in Fusion360 and draw a rectangle with the dimensions 200x158 mm.

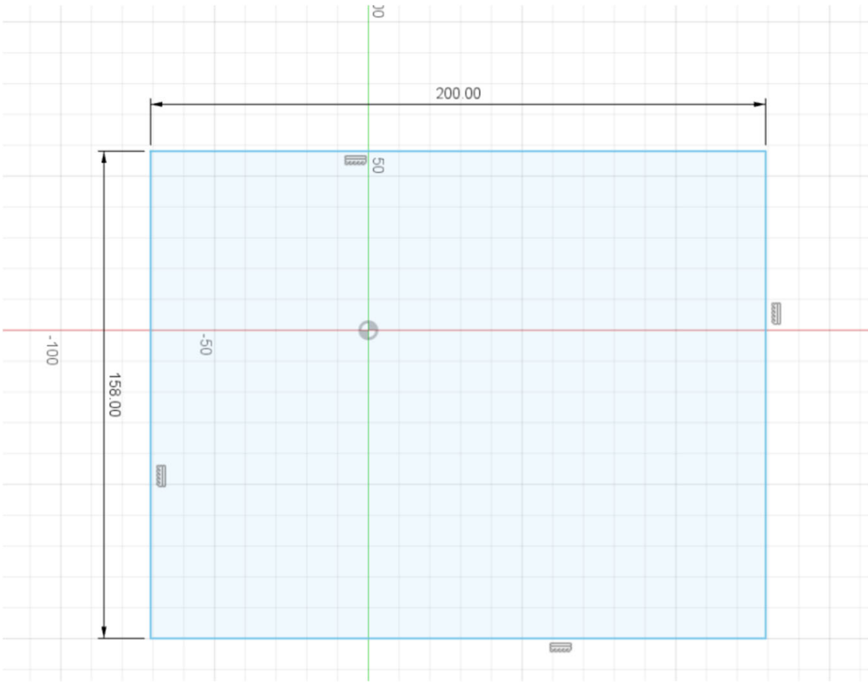


Figure 41 Rectangle for the base plate

Before finishing the sketch, select the “Offset” function in the toolbox under “Modify”. Here, choose an offset of 2mm, which is used to represent the walls.

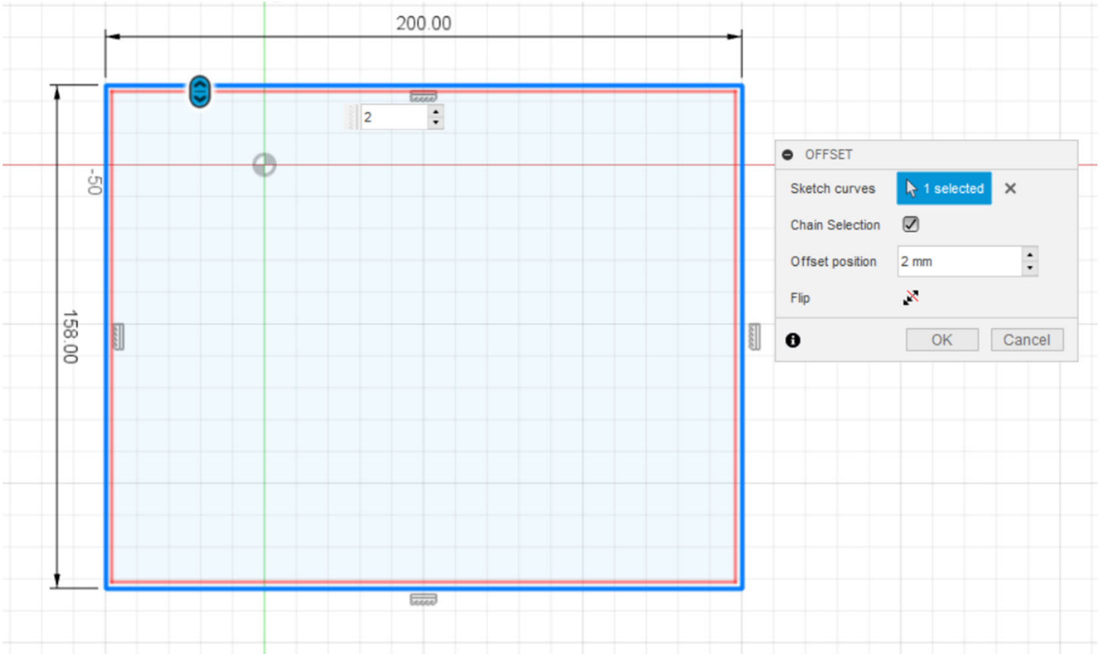


Figure 42 Offset representing the walls

The next step is to extrude the base plate and the offset of the walls. To be able to extrude the base plate by 2mm, select the “Extrusion” function in the toolbox in the application “Create”. Then, click on the inner square with the mouse and enter “2mm” for the extrusion height in the respective text field.

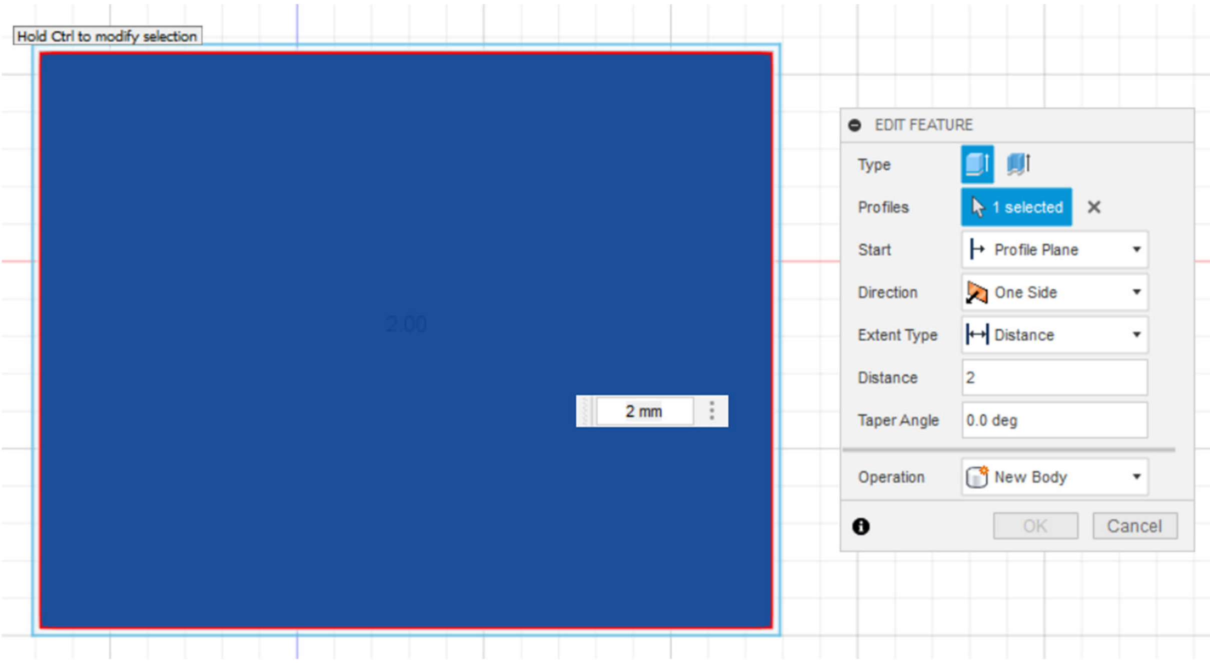


Figure 43 2mm extrusion of the base plate

Proceed in the same way with the offset for the walls. After selecting the “Extrusion” function, left click on the frame around the base plate and enter “3mm” as the offset size. The wall now overlaps the base plate by 1mm.
The next step is to create the cut-outs in the front wall for the two doors. As can be seen in Figure 39, there is a double door with a centre door jamb. To realise this in construction, first draw a sketch with two rectangles for the two door leaves. Enter “9mm” for the width of each door and “1mm” for the width of the jamb.

To draw the cut-outs, select “Create sketch” from the toolbox. Then, click on the surface of the wall body at the edge of the base plate.

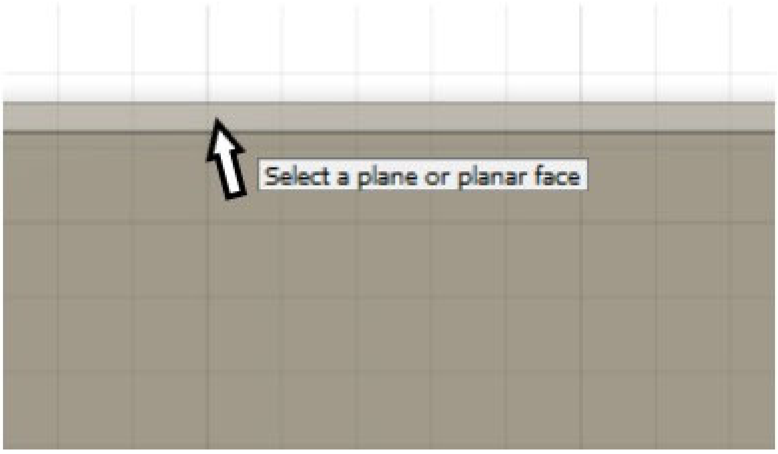


Figure 44 Selecting the sketch level for doors

This has the effect that the sketch is created at the height of the wall surface. To determine the centre of the base plate as well as the position of the door leaves, one can use the background grid. Next, the rectangles for the door leaves need to be drawn accordingly on the wall solid. The process is completed by activating the “Finish sketch” button.

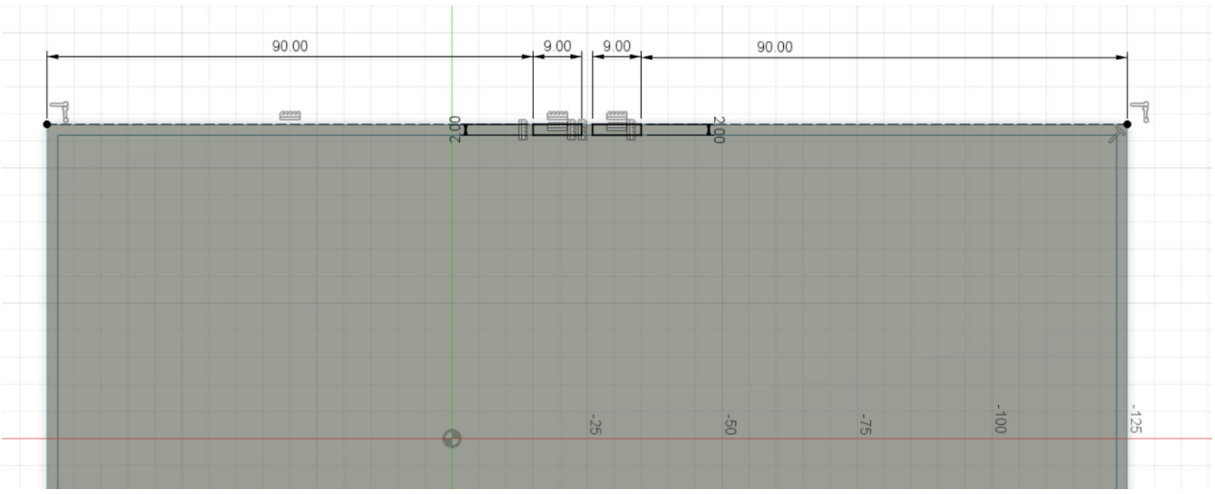


Figure 45 Grid for the positioning of the doors

Once the sketch has been completed, select the “Extrusion” function again and, while holding down the CTRL key, select both newly added rectangles by pressing the left mouse button.

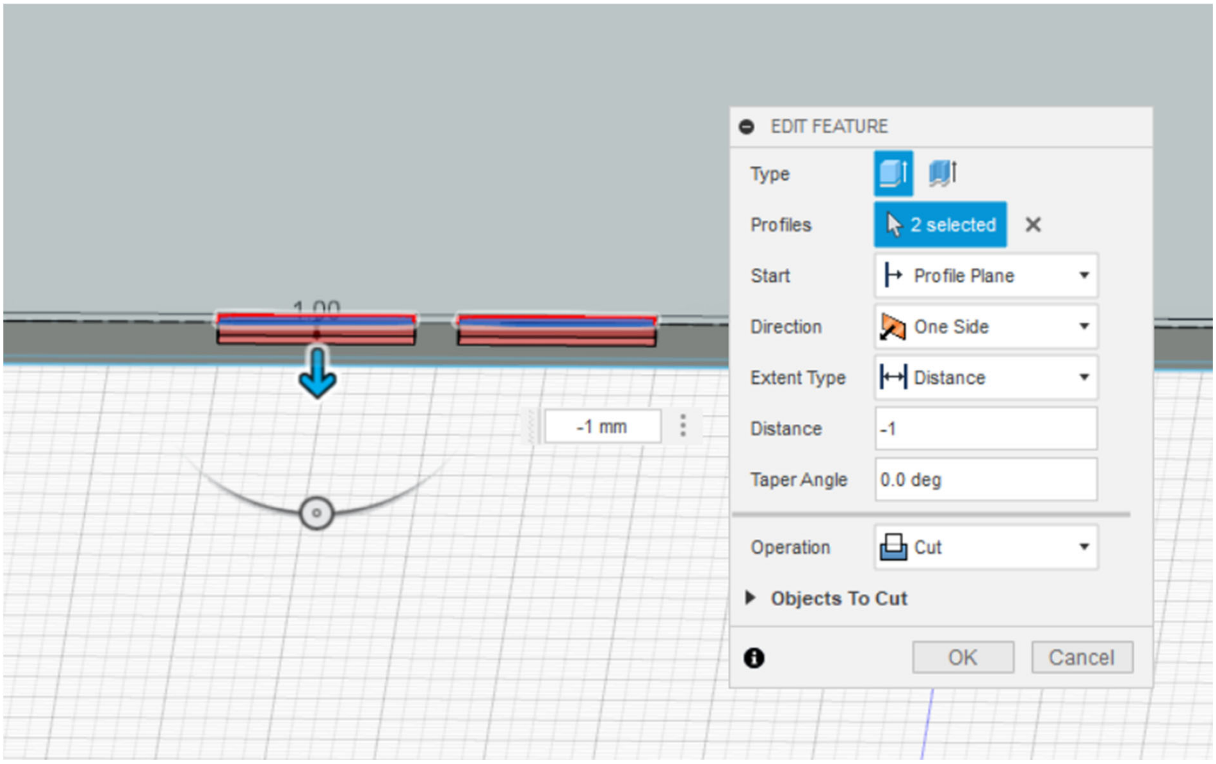


Figure 46 Extrusion with negative values for cutting solids

In the text field ("Distance"), enter the negative extrusion value "-1mm". In the selected view, you can now see a red area below the rectangles. Instead of saying "Join" or "New Solid", the "Operation" field now contains "Cut". This operation mode is always preset in Fusion360 whenever an extrusion extends into a solid. This means that the "Extrusion" function is also suitable for creating cut-outs.

In the present example, lower the wall to the level of the floor plate. This creates the cut-out for the two door leaves.

5.2 Pillars and Furniture

Next, it is the aim to create the pillars, conference tables, screen, lectern and bistro tables. This can easily be done by means of the the solid functions "Box" and "Cylinder" in the "Create" menu.

First, create the four pillars in the front area. To do so, select the function "Cylinder" and position the mouse on the base plate. With the left mouse button pressed, draw the diameter. Enter a diameter of 8mm and a height of 3mm in the dialogue window.

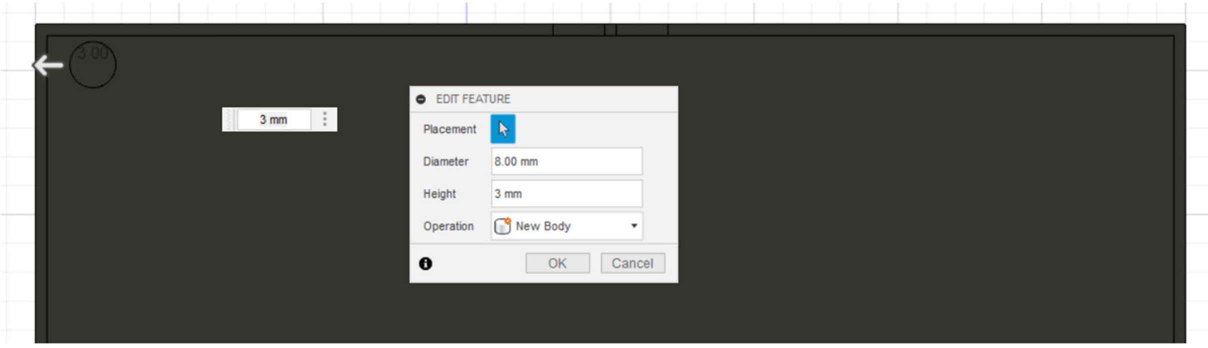


Figure 47 First pillar created using the "Cylinder" function, 8mm diameter, 3mm height

Next, move the pillar to the desired position. To do so, use the grid in the background as a guide. Select the pillar in the browser bar by left-clicking. Then, activate the context menu with the right mouse button. There, select the option "Move/Copy". By means of the appearing move widget, drag the pillar to the right by clicking on the arrow pointing to the right.

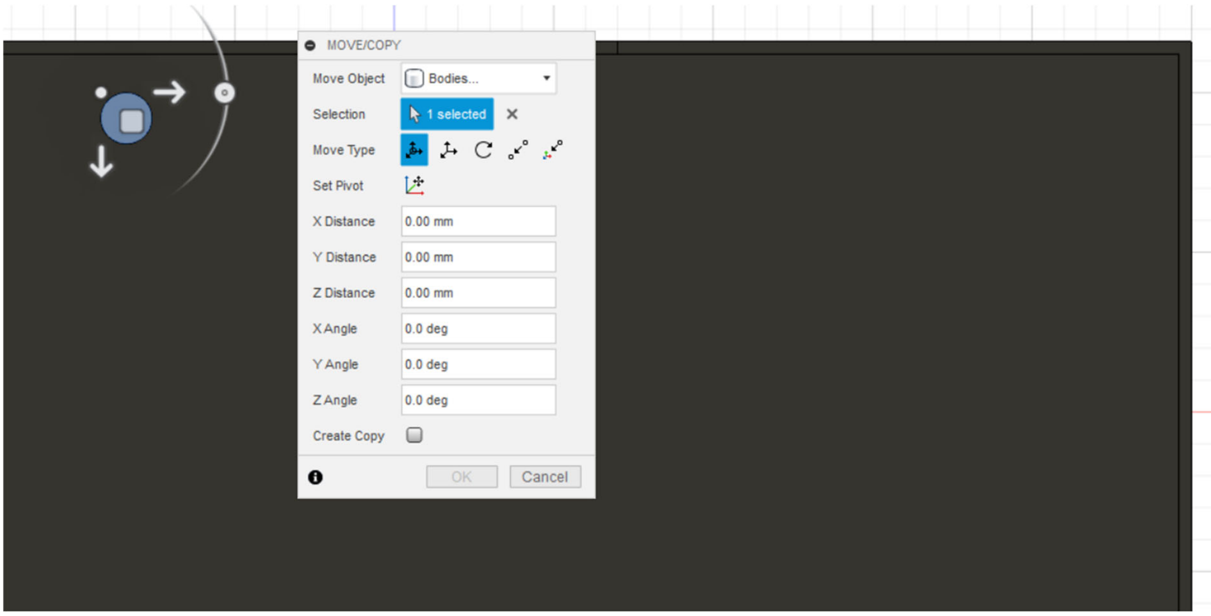


Figure 48 Moving the pillar with the "Move" widget

You can either move the object with the mouse or by entering a value for the X axis in the dialogue window. After moving the first pillar to the correct position, you can repeat the previous step three more times to create the remaining three pillars. Alternatively, use the "Copy" function. This is an easier option given that you do not have to enter the size of the column three times. To copy an element, select the body of the pillar again with the mouse via the browser palette as described above. Again, activate the context menu using the right mouse button and select "Move/Copy". Then, however, tick the check box "Create Copy" and enter "45mm" in the "X Distance" field to move the element by 45mm on the X axis.

This way, a copy of the pillar is created and positioned 45mm to the right of the original solid.

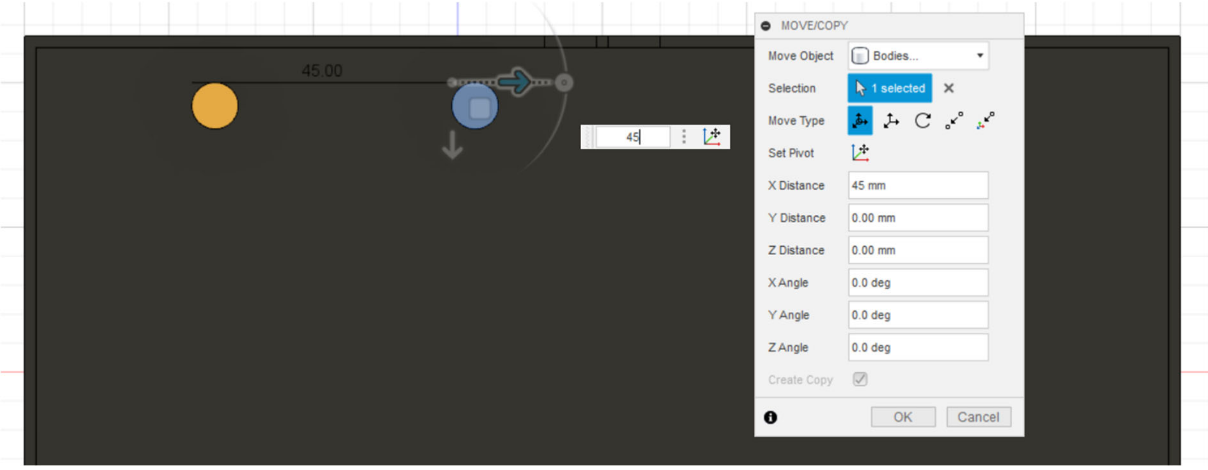


Figure 49 Copying a solid and simultaneously moving it

Repeat this process two more times for all four columns to have been created.

Next, create the 18 conference tables. For this step, proceed in the same way as described above. First, create a table with the “Box” function and then copy it.

After activating the “Box” function, position the mouse below the first column and draw a rectangle with the dimensions 10x25mm. Set the height to 1mm. Regarding the “Operation” field, make sure that the option “New Body” has been chosen. Complete the process by pressing “OK”.

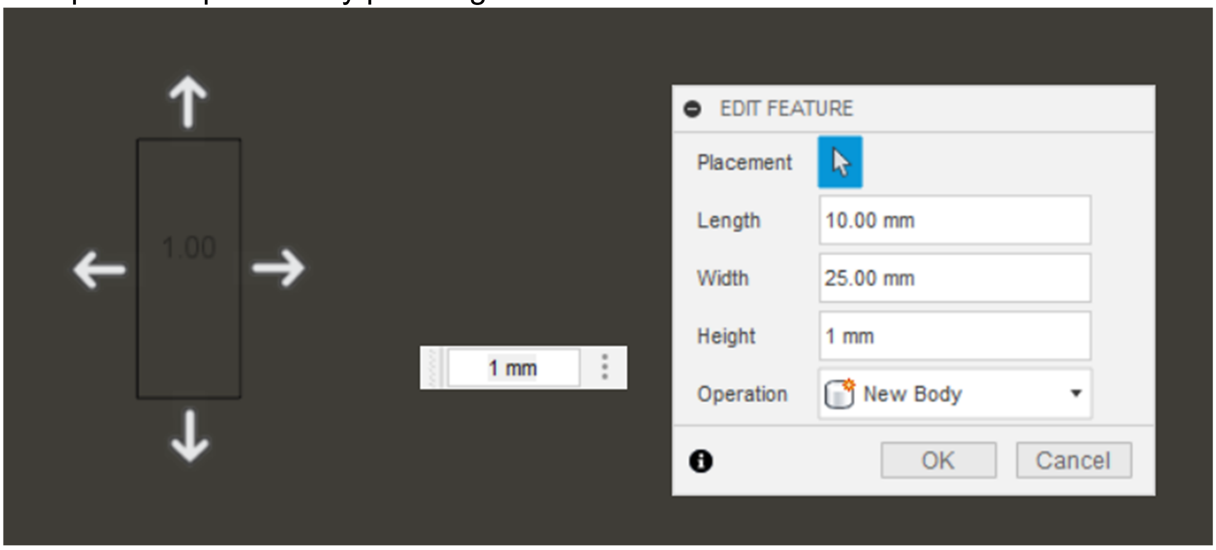


Figure 50 Table created using the “Box” function

Now, copy and move this table several times. In the above example, first copy the table once and move it downwards. Then, repeat this a second time to create the left row of 3 tables.

To do this, left click on the table in the browser palette. It should be the last solid in the list. Activate the context menu by pressing the right mouse button, and select “Move/Copy” again. Here, make sure to tick the check box “Create Copy” and enter “30mm” as value for moving the object on the Z axis.

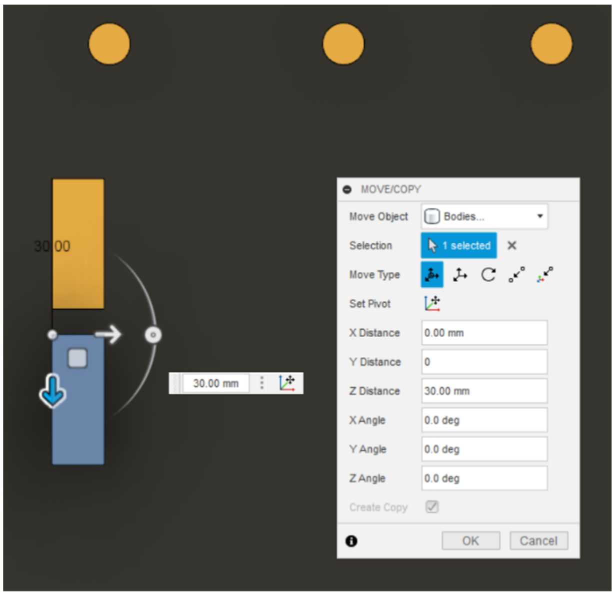


Figure 51 Copying the first table on Z axis

Repeat this process with the second table. This gives you the first row of tables. Then, copy the entire row and move it to the right to obtain the first table block. To do this, select the three tables in the browser palette at the same time by left-clicking on the last three entries “Body...” while holding down the shift key. The selected elements will be highlighted in blue. Once again, activate the context menu using the right mouse button and tick the check box “Create Copy”. Move the table group on the X axis by 20mm.

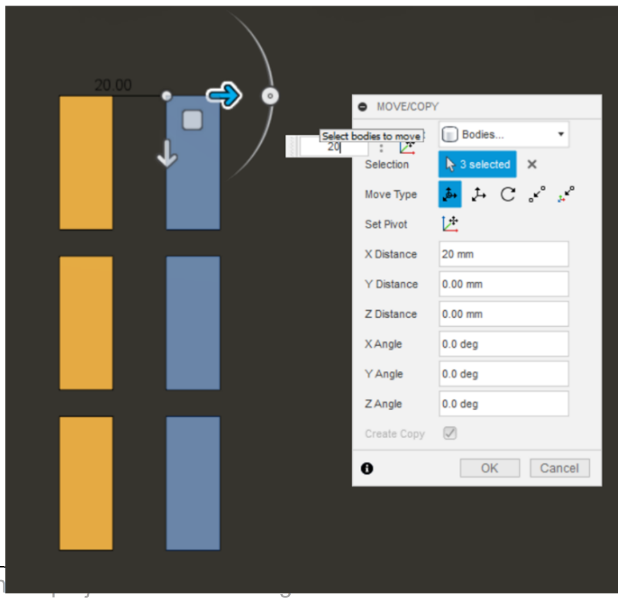


Figure 52 Copying multiple solids

Then, you can copy and move the entire block by selecting all six tables, copying them and moving them on the X axis. Enter "54mm" as the value for moving the tables. The second block should now appear. Repeat this process to create the third block as well.

Finally, the screen and the lectern need to be created. Again, select the "Box" function to create the screen as well as the tables by entering the following values:

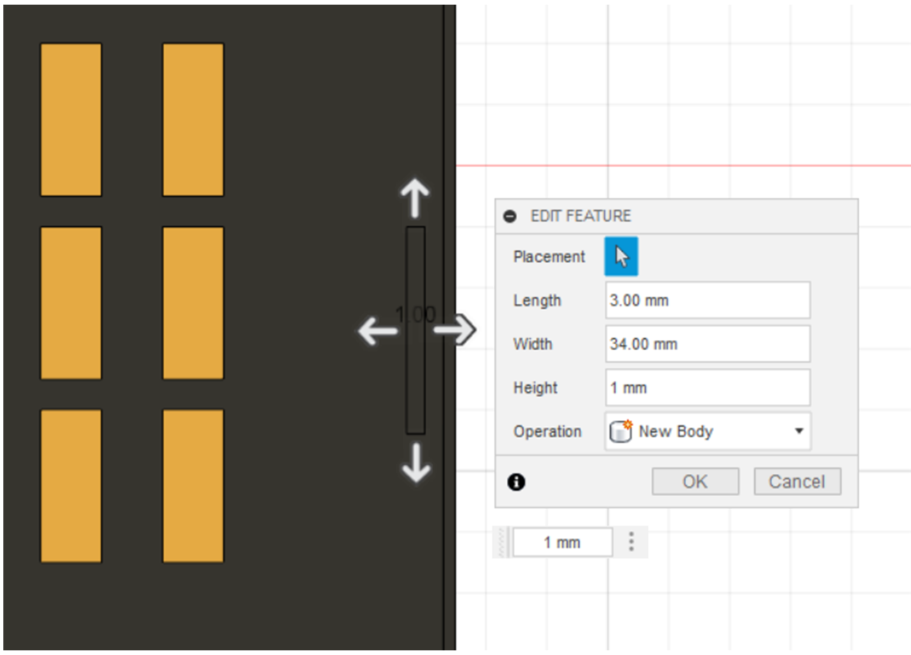


Figure 53 Designing the screen

Create the lectern in the same way using the following settings:

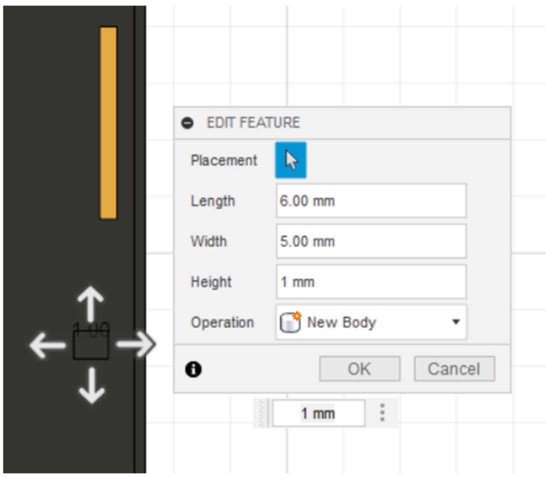


Figure 54 Designing the lectern

The bistro tables are created in the same way as the table groups. First, draw one of the tables. Then, copy it twice and move it 47mm to the right.

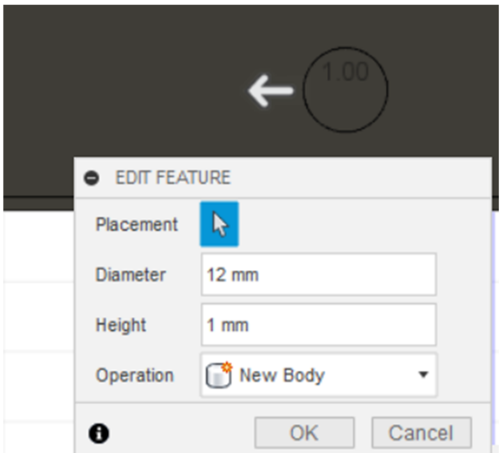


Figure 55 Data of bistro tables

5.3 Combining the Solids

The elements arranged on the base plate have each been created as a “New Body”, which allows for them to be copied. Accordingly, 28 bodies should be listed in your browser bar. In order to export the created model as an STL file, you need to combine the bodies. For this, use the “Modify” menu from the tool palette. Open the menu by clicking on the triangle and then select the “Combine” function. The dialogue window appears next to the floor plan.

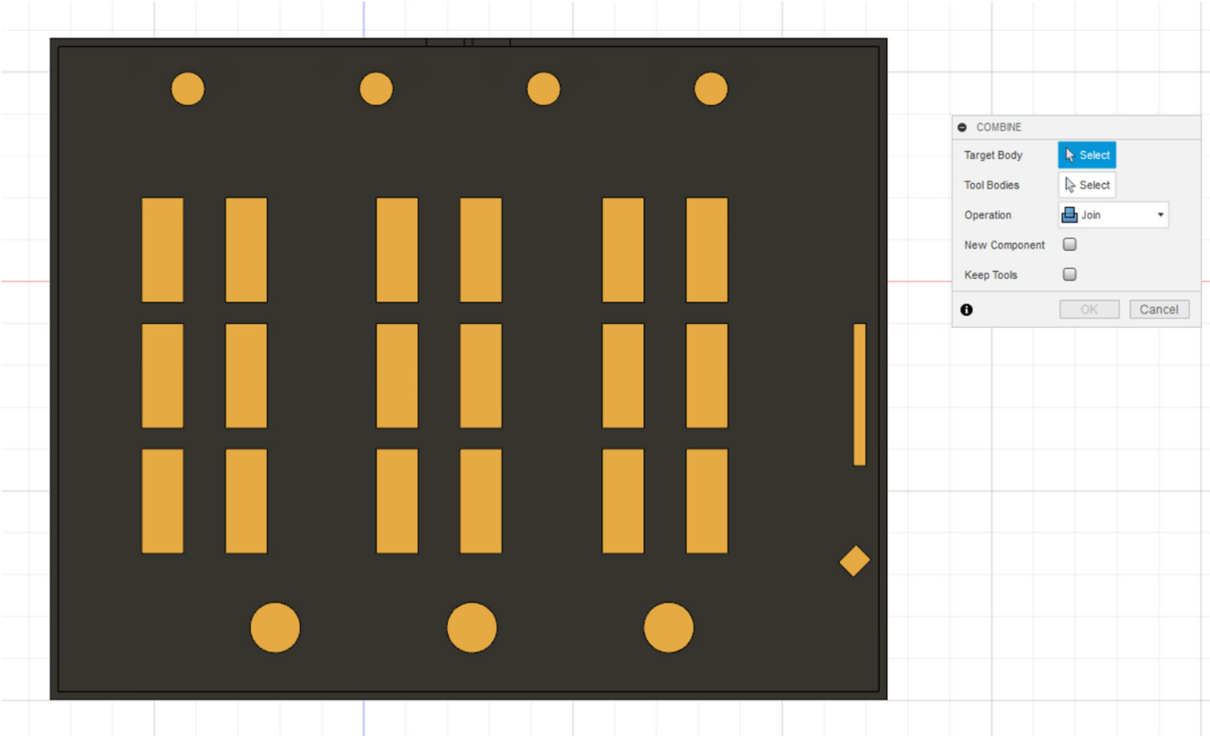


Figure 56 "Combine" function

In the "Target Body" field of the dialogue window, select the solid body into which the other bodies are to be integrated. To do this, left-click on the base plate – you will then see that phrase "1 selected" will appear in the "Target Body" line. The target bodies are selected in this line by left-clicking on all the elements on the base plate. The selected elements are displayed in blue and the number of selected bodies in the line "Tool bodies" is increased accordingly. Selecting tool bodies can also be done via the browser palette.

Once all elements have been selected, the construction should look as shown below:

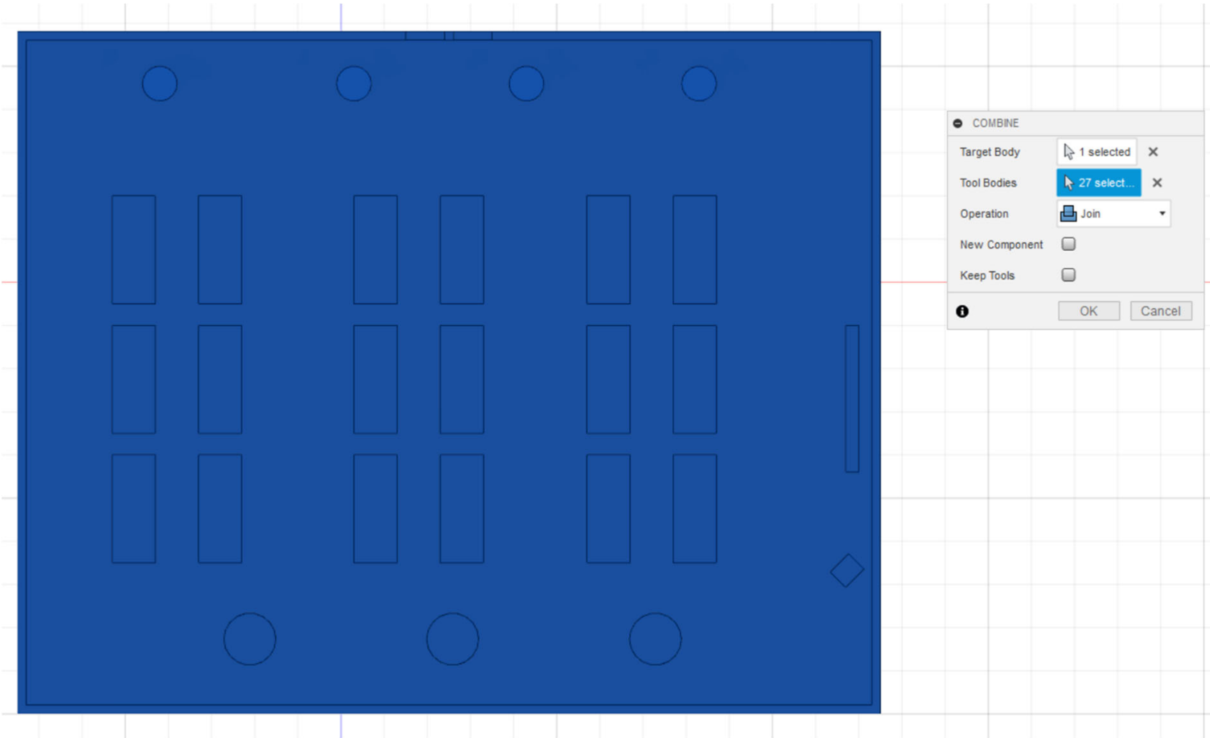


Figure 57 All target and tool solids selected

By pressing the “OK” button, the selected bodies are united. Only one body should then be visible in the browser palette. However, in case there are still several bodies listed, these bodies have either not been selected or have not been combined because they are not connected. For example, there could be a gap between two bodies. This must be checked from different perspectives and, if necessary, a body must be repositioned accordingly by using the “Move/Copy” function.

6 Creating STL files

Creating STL files from a construction in Fusion360is relatively easy. However, it is important that the solid to be converted is as combined as possible. If it consists of several, non-combined solids, errors may occur in the STL file. Moreover, it could happen that individual solids are not exported.

The model presented in chapter 5 fulfils the necessary requirements Unless it has already been done, load the file into Fusion360. Once the construction is loaded, select "Tools" from the menu in the toolbox. This will cause the display to change. Then, select the "3D Print" function from the "Make" menu.

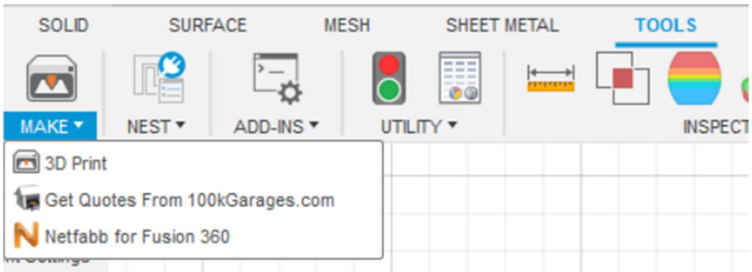


Figure 58 "3D Print" function in the "Make" menu

A new dialogue box appears. In case the check box "Send to 3D Print Utility" is activated, deactivate this function. Then, left click on the floor plan. This results in an altered display of the item and the construction is shown as a mesh.

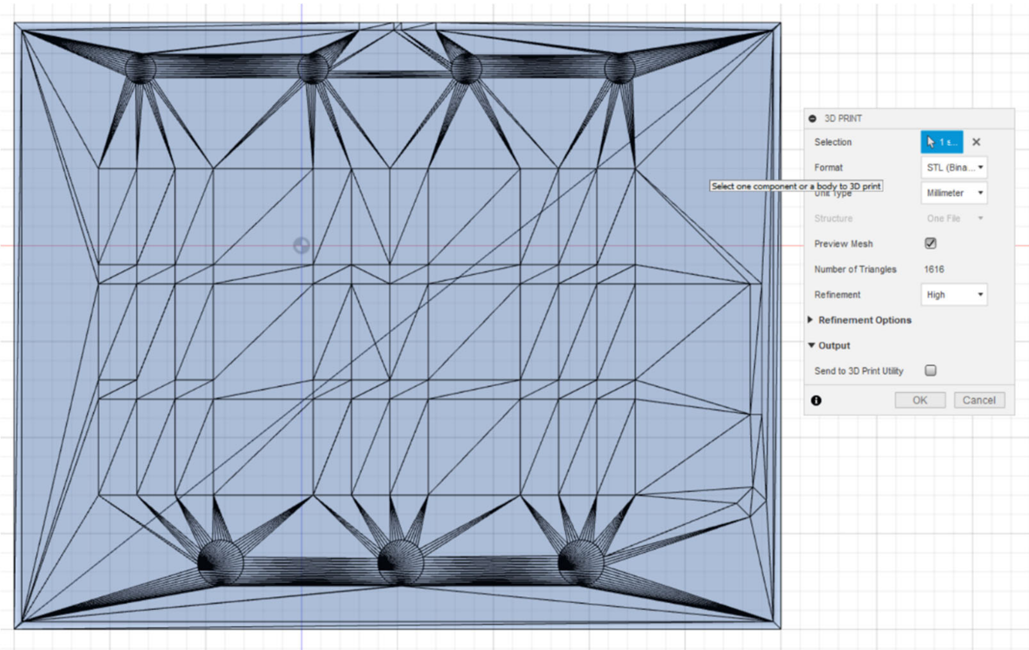


Figure 59 "3D Print" function after selecting the relevant solid

Press the "OK" button and then enter the file name and storage location for the STL file.

7 Creating Tactile Site Maps from Graphics

In Fusion360 it is further possible to include graphics, such as floor plans or photos, as a basis for the construction of site maps. In principle, a graphic is used as a template for tracing with the sketch function.

As an example, the conference room floor plan presented in chapter 5 is to be integrated into Fusion360 and traced.

First, draw a rectangle in a new project that roughly corresponds to the size of the floor plan. In the present example, the rectangle has a size of 200x158mm. After creating the rectangle, close the sketch using the “Finish sketch” button.

Next, select the menu item “Insert” from the toolbox. There, select the “Canvas” function. A data window will pop up. Select the source directory containing the graphic of the floor plan. You will then see the working area of Fusion360 appear together with a dialogue window. The first field contains the file name of the floor plan graphic you have just selected. Now, left click on the rectangle you have just drawn to let Fusion360 know that a graphic should appear at the respective location.

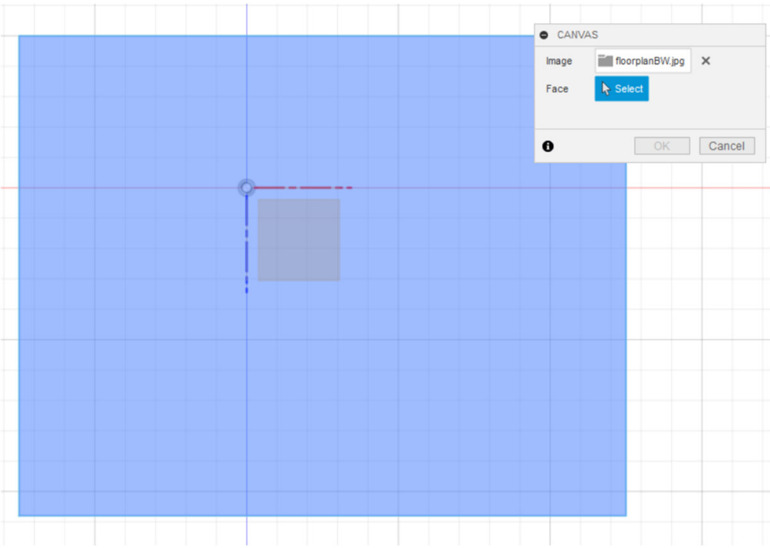


Figure 60 Selection window for the canvas area

The selected graphic is then opened in the selected canvas area. In the dialogue window that appears, detailed adjustments can be made to the positioning and transparency of the graphic. The insertion process is completed by pressing “OK”.

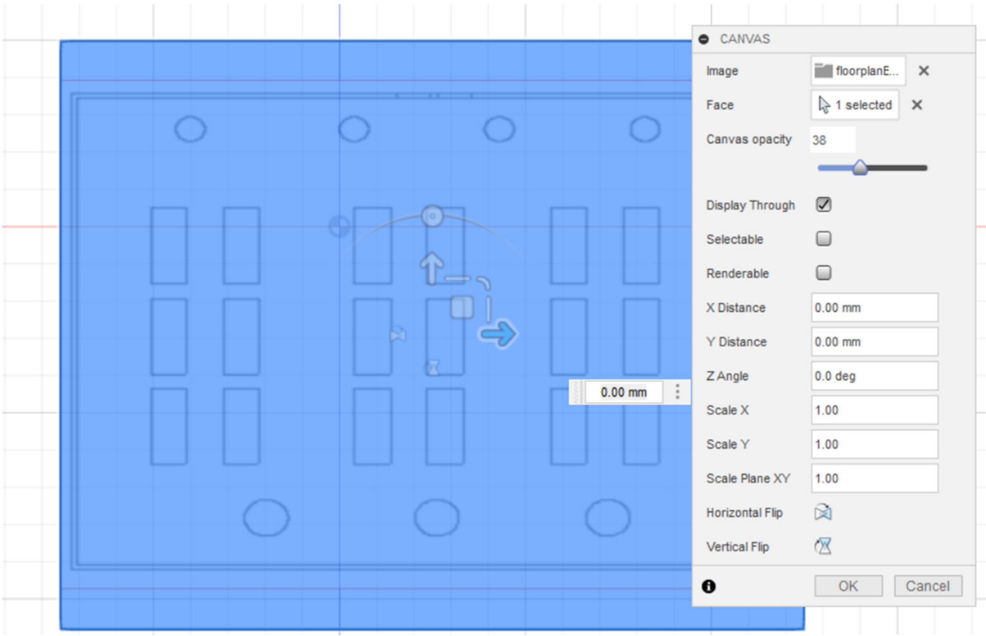


Figure 61 Adjustments for the inserted graphic

Once the graphic has been inserted, trace the floor plan with the "Sketch" function. Using the functions "Extrusion" and "Solid" described above, a tactile location map can be produced.

8 Creating Tactile Braille Models

In Fusion360, it is possible to create solids with Braille. However, this requires a chargeable add-in from the Autodesk Store. To install this add-in, select the “Tools” menu from the toolbox. In the menu bar, select the “Add-Ins” function and click on “Fusion360 App Store”.

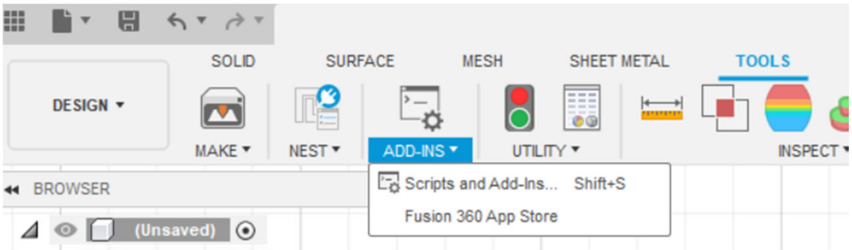


Figure 62 “Add-Ins” feature

In the Fusion360 App Store, enter the word “braille” in the search field. Thanks to autocorrection, the available add-in “Braille Creator” will be added automatically. Left-click on this mentioned add-in. This will transfer you to a page containing a description of the add-in. Currently (September 2022) this addin costs 2.99 US$ and can only be obtained for Microsoft Windows operated computers.

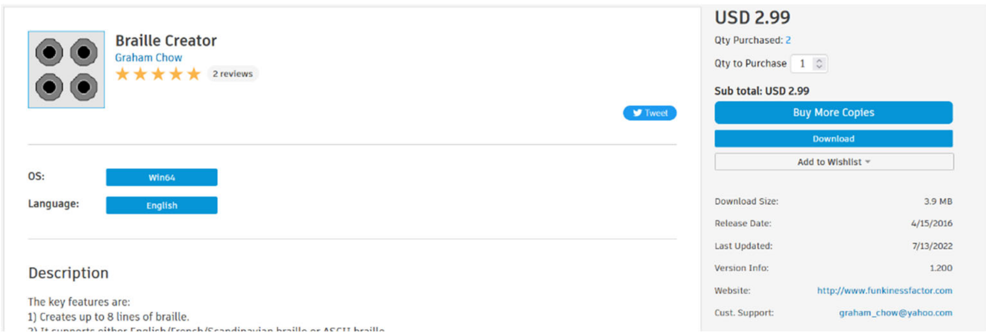


Figure 63 Add-In Store with “Braille Creator” App

After purchasing the full version or selecting the 30-day trial version, you will receive an installation file for download. Close Fusion360 before starting the installation process.

Note: This Add-in was **updated in July 2022**, after the source version no longer worked after an update of Fusion360. Hence, it might be necessary to download the new version from the Autodesk app store and repeat the installation process. To facilitate this process, save the Braille Creator installation file on a data medium. The Braille Creator supports **Unified English Braille and meanwhile French and Scandinavian letters.** Special characters, country-specific characters and umlauts from other languages are not supported yet. The generated braille characters are created in the standard dimensions.

To use the Braille Creator, simply press the “s” key to open the Model Toolbox. Enter the letters “br” and the autocorrection will immediately open the available add-ins containing this combination of letters.

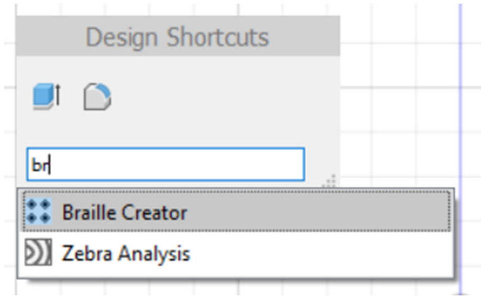


Figure 64 Model toolbox after pressing the "s" key

Activate the Braille Creator with the mouse. This will open a new dialogue window.

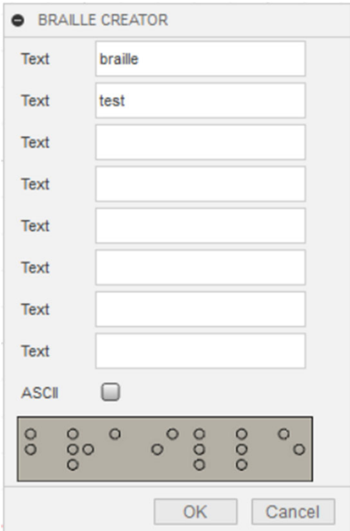


Figure 65 Braille Creator window

Enter the desired text in the respective lines. To generate the braille and insert it into Fusion360, press "OK" button. This process may take a few moments as the Add-in needs to perform complex operations to generate the braille dots. The text provided is inserted in Fusion360as a plate and forms a solid that can be copied or moved the same way other solids can be move and copied as well.

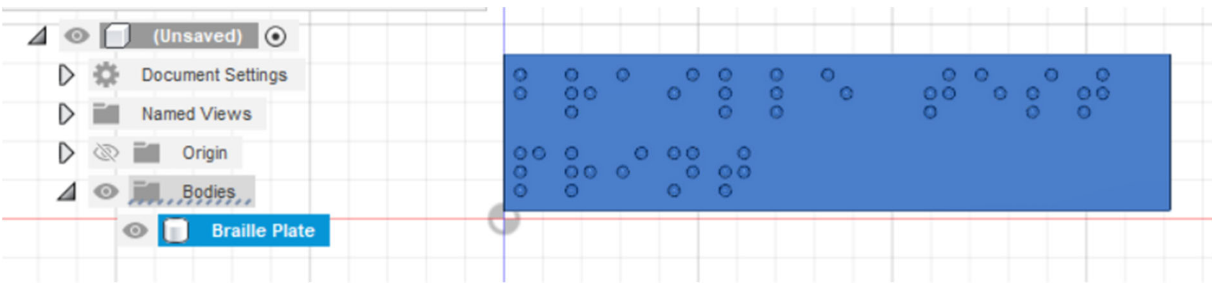


Figure 66 Generated braille and display in the browser bar

Co-funded by the
Erasmus+ Programme
of the European Union

9 Editing STL files

Solids from mesh files in STL or OBJ format can be processed with Fusion360 with certain restrictions. This can be useful if only one STL or OBJ file is available, and the contained bodies are to be changed. The problem, however, is that you are dealing with a solid consisting of a mesh of polygons. A simple cuboid in a mesh file comprises many interconnected polygons. Without conversion, these files cannot be edited.

Note: For large mesh objects with over 30,000 polygons, editing with Fusion360 can be problematic.

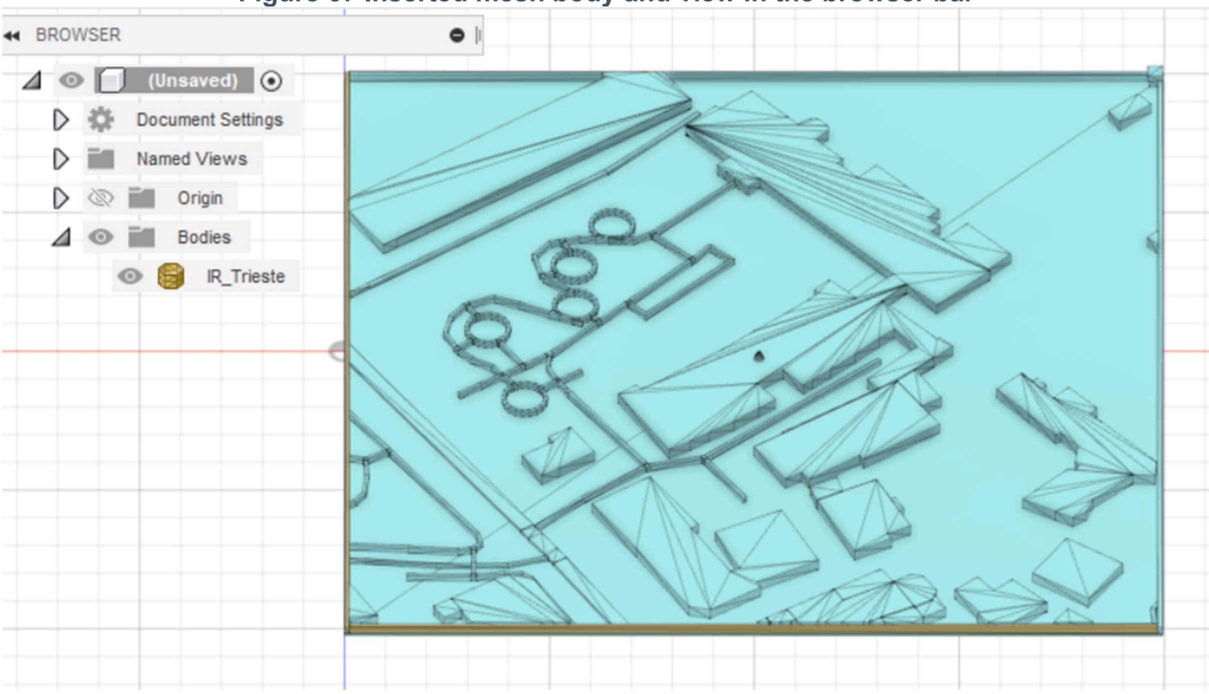
The aim of the present example is to change a site plan of the Institut Rittmeyer, which was created with Touch Mapper (https://touch-mapper.org/en/area?map=B3f149e6b9b8ac55).

The site plan is already available as an STL file on a data medium.

To import the file, select the “Insert” menu from the toolbox. Then, select the submenu “Insert Mesh”.

This forwards you to a Windows window for opening files. There, select the relevant file. The mesh body now appears in Fusion360 and is displayed in the browser palette under the heading “Body” as a mesh body.

Figure 67 Inserted mesh body and view in the browser bar



As depicted in Figure 67, the paths are shown as narrow cuboids. Delete these cuboids between the main buildings as well as the cone on the roof of the central building.

To change network files in Fusion360, the corresponding functions must be selected.

If only individual elements of the mesh file are to be deleted, separate them. However, if bodies need to be added to the mesh body or further changes are required, the mesh body must be converted.

To do so, use the “Mesh” menu from the toolbox.

9.1 Removing Mesh Elements

To delete the paths, first select the “Mesh” menu. In the modified toolbox, select the “Modify” menu and subsequently select the “Separate” function.

Figure 68 “Modify” menu in the “Mesh” main menu

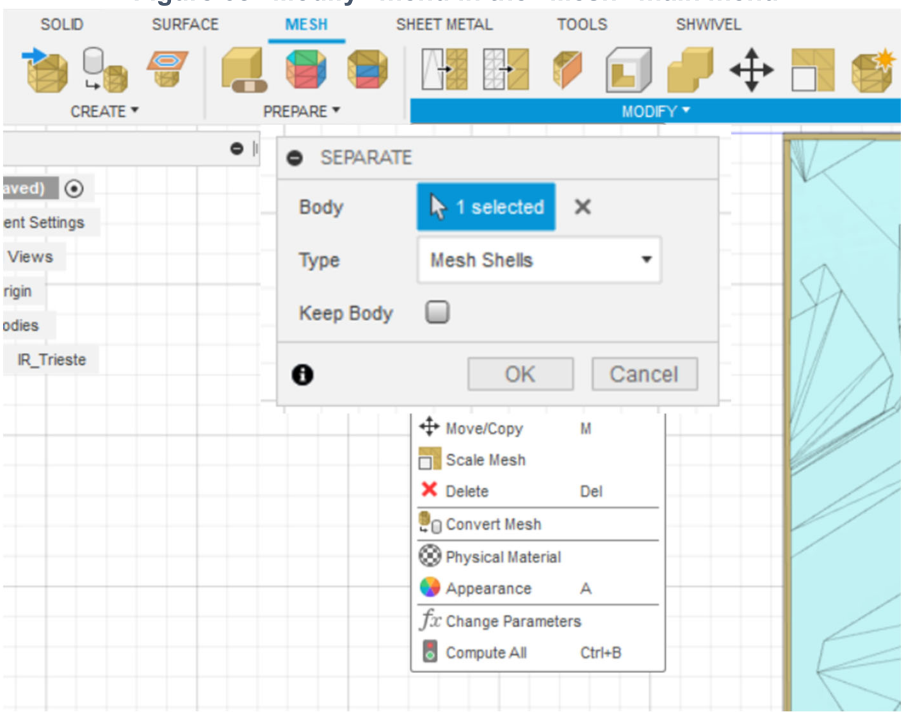


Figure 69 “Separate” window

Left-click on a surface of the base plate of the mesh body to select it as the body to be separated.
After pressing “OK”, the net body is displayed in different colours.

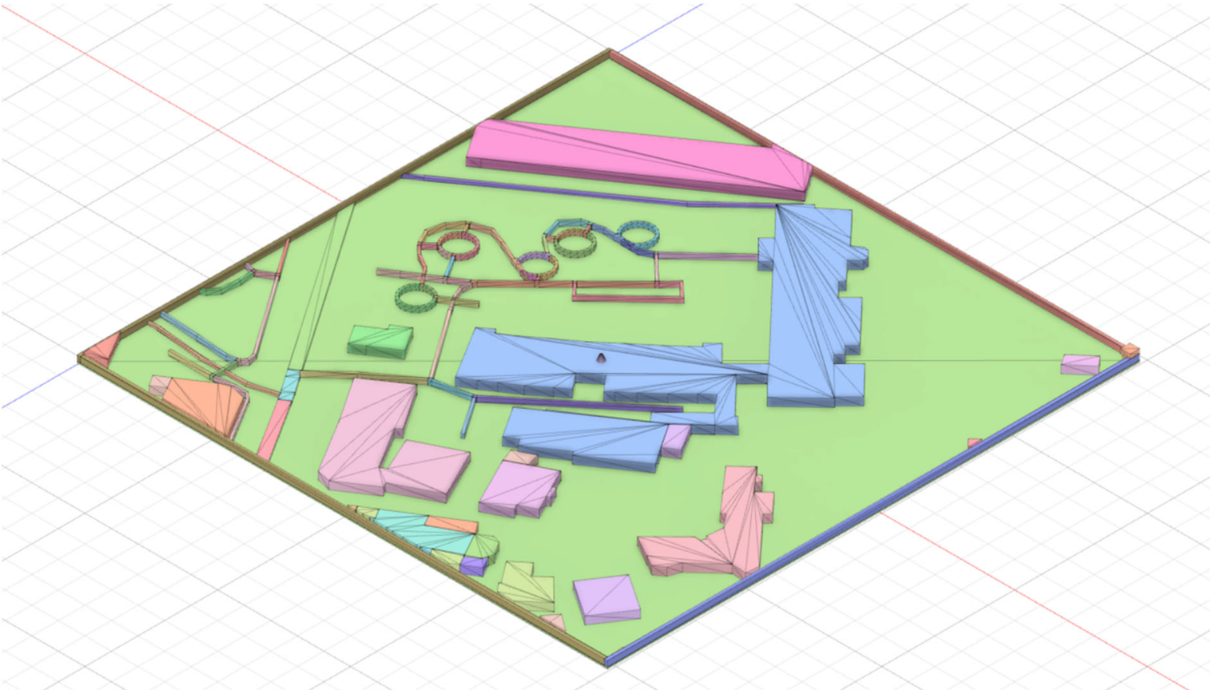


Figure 70 Display of the mesh body after separation

Now it is possible to select the individual mesh bodies by left-clicking. Once this has been done, open the context menu using the right mouse button. By selecting the function “Remove”, the selected element is deleted.

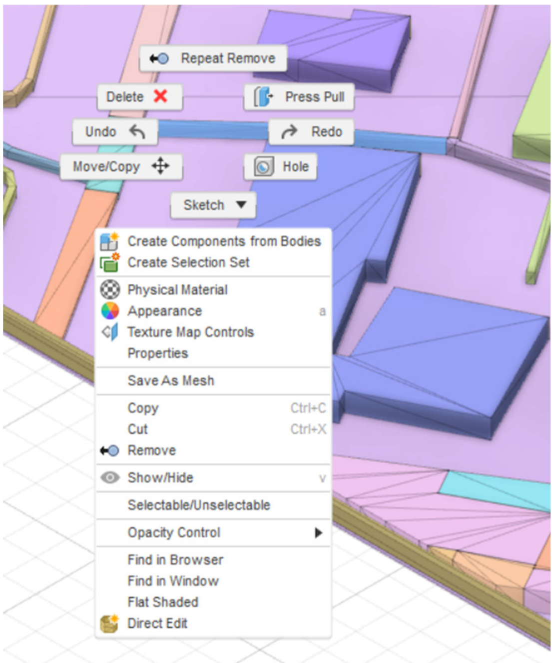


Figure 71 “Remove” function in the context menu after selecting an element

In the example shown in Figure 71, the street element located between the menu items “Undo” and “Redo” was selected. Proceed in the same way with the remaining street elements. Then, select the cone on the roof and remove it using the “Remove” function from the context menu. Finally, you have a mesh body without street elements in the centre.

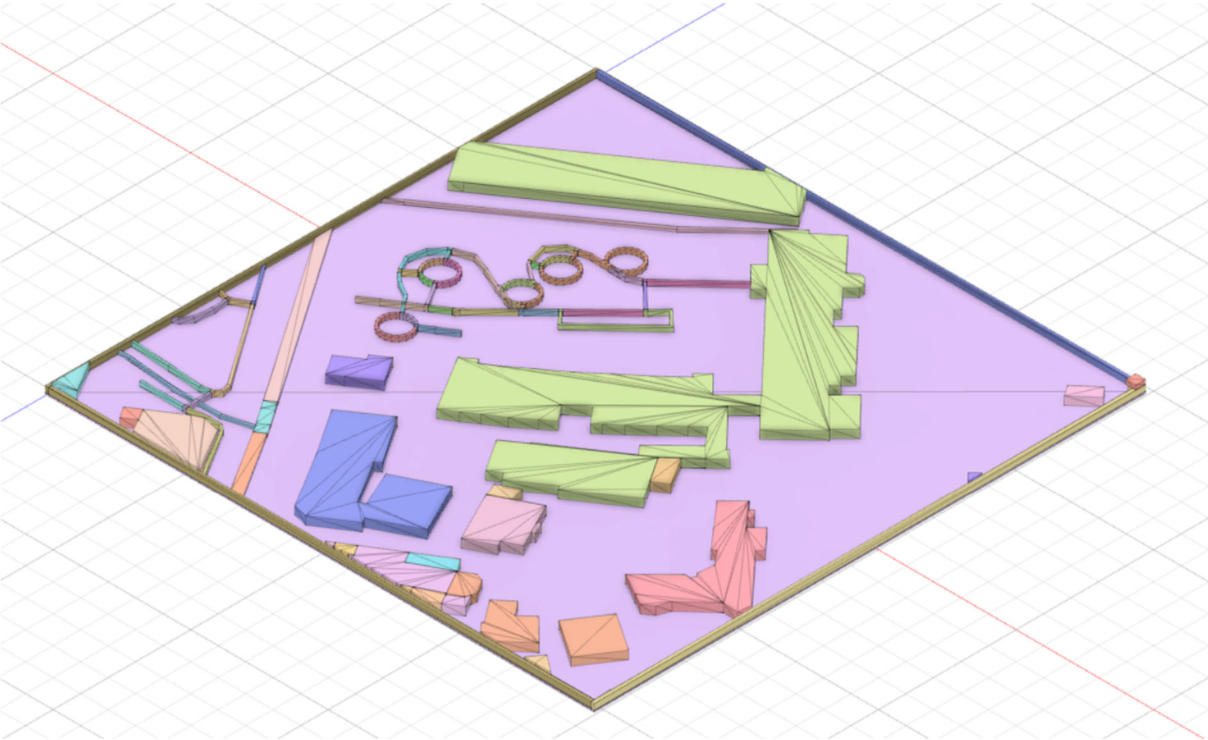


Figure 72 Mesh body with deleted elements

In order to print the altered mesh body, you first have to combine all the mesh elements again. To do so, select the “Combine” function form the “Modify” menu and select “Combine”. A dialogue window similar to the one described in Chapter 5.3. will appear. There, select the target body using the mouse: Left-click on the base plate and select all the other elements – one after the other – as “Tool bodies”. To combine the mesh body, press the “OK” button.

Using the “Export” menu in the toolbox, you initiate the same process for producing an STL file as shown in Figure 58 “3D Print” function in the “Make” menu. This will create a new STL file that can be prepared for printing with the slicer.

9.2 Editing Mesh Elements

In order to integrate a mesh element into one’s own construction or to connect additional solids to it, the mesh needs to be converted first. Once again, select the submenu “Modify” from the main menu “Mesh”. Then, select the “Convert Mesh” function.

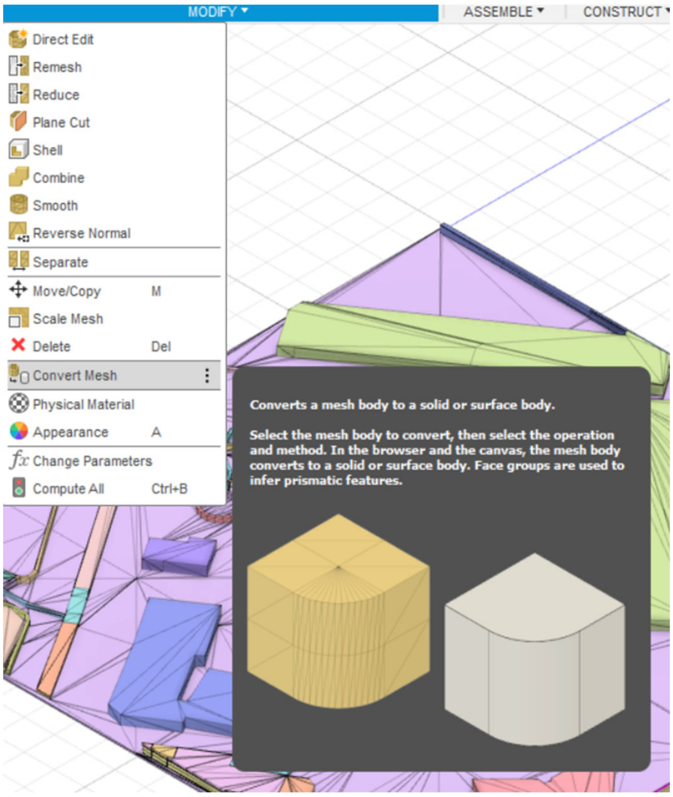
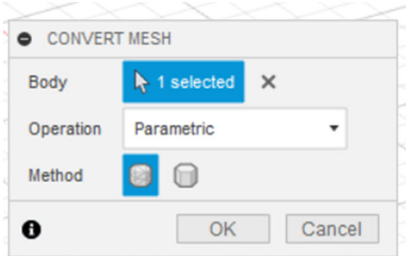


Figure 73 "Convert Mesh" function

After activation, a dialogue window pops up. Select an area of the network body by left-clicking on it. Press "OK" to start the conversion.

Figure 74 "Convert Mesh" window



Once the conversion is complete, the solids are no longer displayed as individual solids but in the structure of the polygons. This can be very confusing for further editing.

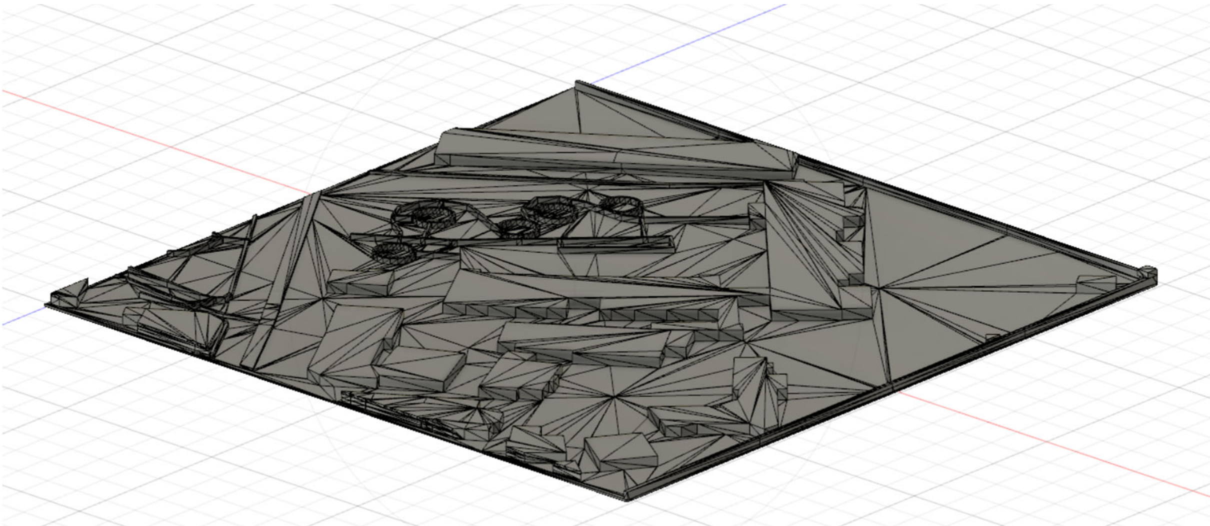


Figure 75 Converted mesh

Nonetheless, further processing is carried out as described in chapter 4.3.

10 List of Figures

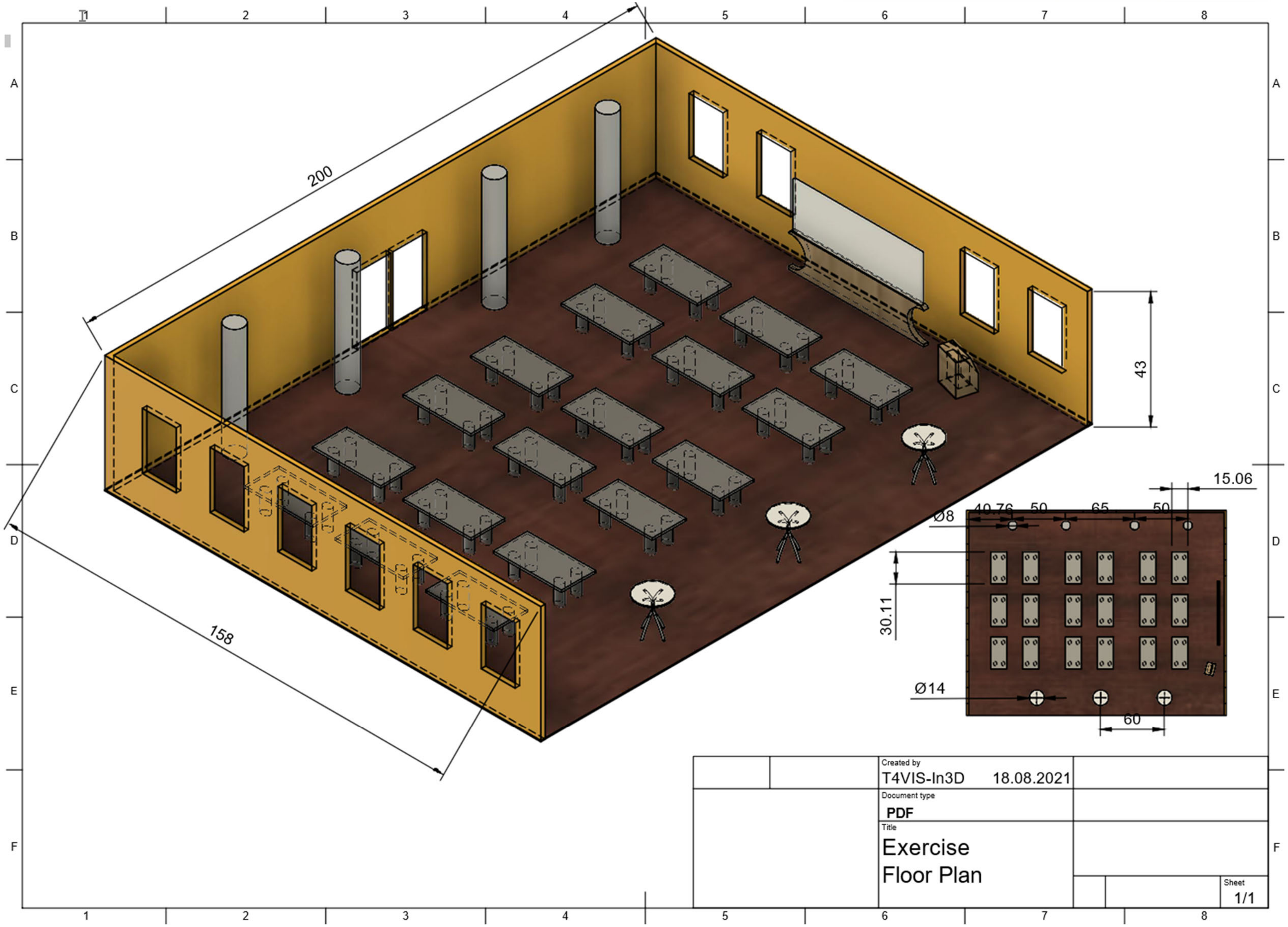
11 References

Cline L. S. (2019): *Fusion 360 für Maker*, Heidelberg, dpunkt.verlag, 1. Edition

Ridder D. (2019): *Autodesk Fusion 360*, Frechen, mitp Verlags GmbH, 1. Edition

Tickoo S. (2018): *Autodesk Fusion360 A Tutorial Approach*, Schererville, CADCIM Technologies

12 Appendix


200
158
43
15.06
Ø8
40.76
50
65
50
30.11
Ø14
60
Created by
T4VIS-In3D
18.08.2021
Document type
PDF
Title
Exercise
Floor Plan
Sheet
1/1