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DESIGNING 3D MODELS IN FUSION 360

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Project "MAKER SCHOOLS: Enhancing Student Creativity and STEM Engagement by Integrating 3D Design and Programming into Secondary School Learning" (Agreement no. 2020-1-BG01-KA201-079274)

MAKER SCHOOLS
3D Design for Education

Contents

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Introduction

Modern society is based on innovations that cover all areas of human activity. Thanks to the ever more powerful personal computers and the ever more sophisticated design software products, the process of creating new products and technologies is accelerating, and in the future artificial intelligence will play an increasing role in it. The advent of mass 3D printers makes it possible for everyone to create and produce unique details provided they have the necessary modelling and design software. This guide is intended to assist students and teachers in mastering the use of the Fusion 360 3D modelling software. It provides basic knowledge on how to work with it and its basic features in the field of 3D object design. Without claiming to be exhaustive in presenting the full capabilities of Fusion 360, the learning resource includes a number of examples that demonstrate the principle of operation of the main commands of the environment and their application in the process of creating objects. The guide is intended for secondary school students, notably students studying technical sciences. However, it can be useful for everyone else who wants to get acquainted with the opportunities that Fusion 360 provides for the design and modelling of new products.

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Key features of Fusion 360

Fusion 360 is an advanced CAD (Computer-Aided Design), CAM (Computer-Aided Manufacturing) and CAE (Computer-Aided Engineering) program. It is designed to work in the cloud, providing extensive capabilities for its users in the areas of design, simulation, and manufacturing of new components and products.

Fusion 360 was created by Autodesk, a world leader in the development of engineering and 3D design products. Its widespread use is due to the great functional capabilities it provides, as well as the extremely attractive price at which it is offered. Non-commercial users and educational institutions can also use Fusion 360 free of charge.

A notable advantage of Fusion 360 is that it is constantly being improved. New features are added or existing ones are changed in a way that makes them more user-friendly.

Installing Fusion 360 and working in the cloud

To install the environment, the user needs to register on the Autodesk website dedicated to the promotion and support of Fusion 360 (Figure 1.1).

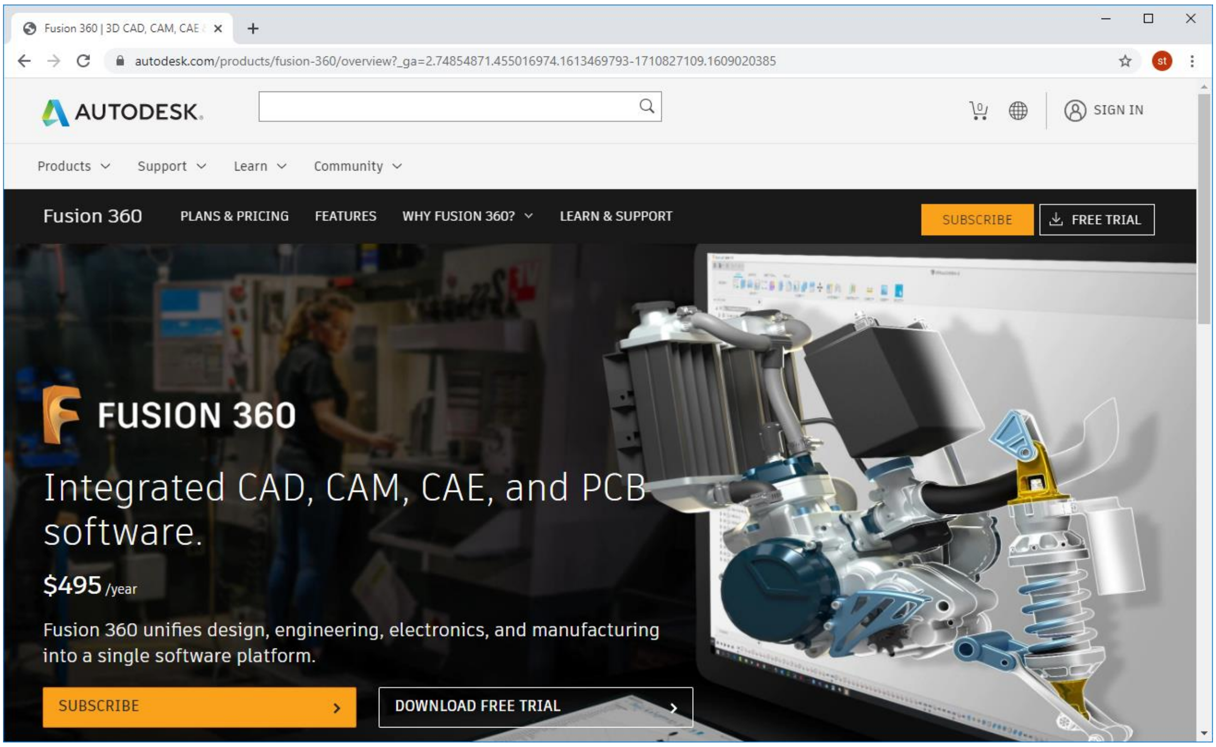

Figure 1.1

After registration, the user is given access to the resources offered by Autodesk and can download Fusion 360. If the user is registered as a student or faculty member at an educational institution (which requires the attachment of additional supporting documents), the user can also receive free access to other Autodesk software products (Figure 1.2).

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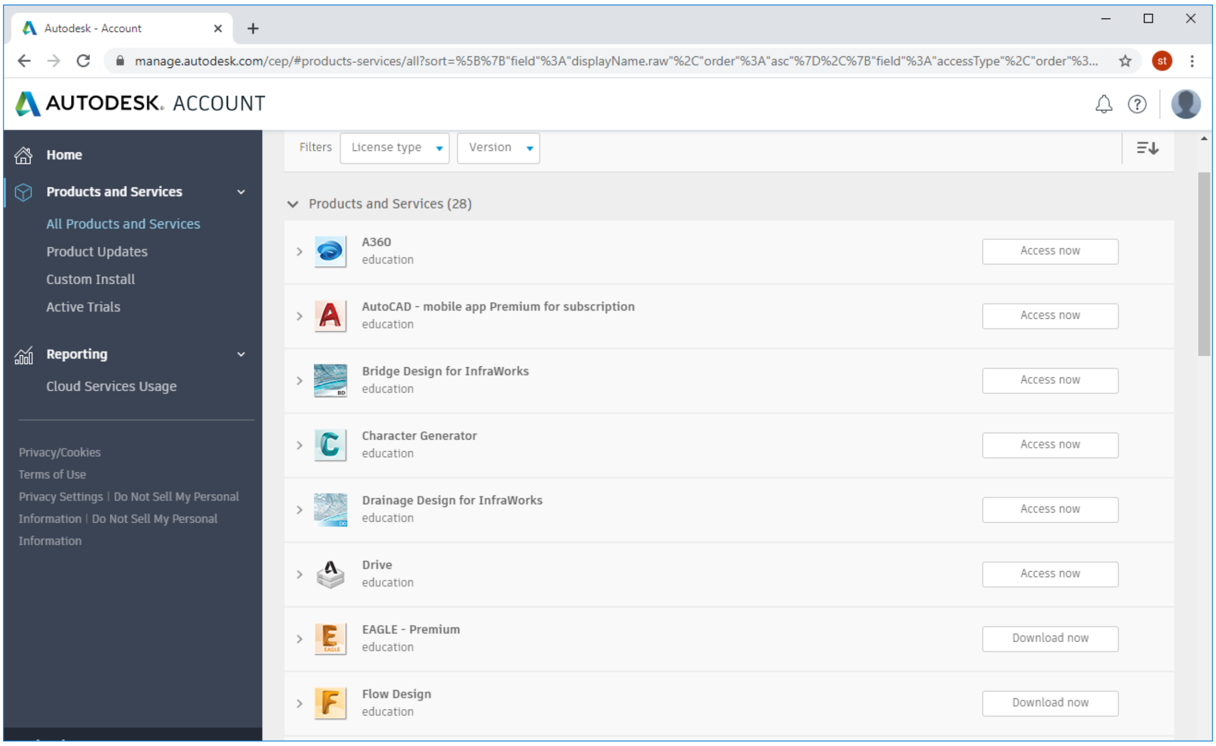

Figure 1.2

User interface of the environment

After launching Fusion 360, it is necessary to register in the environment with your account. Once this is done, the user interface of the program will be launched (Figure 1.3).

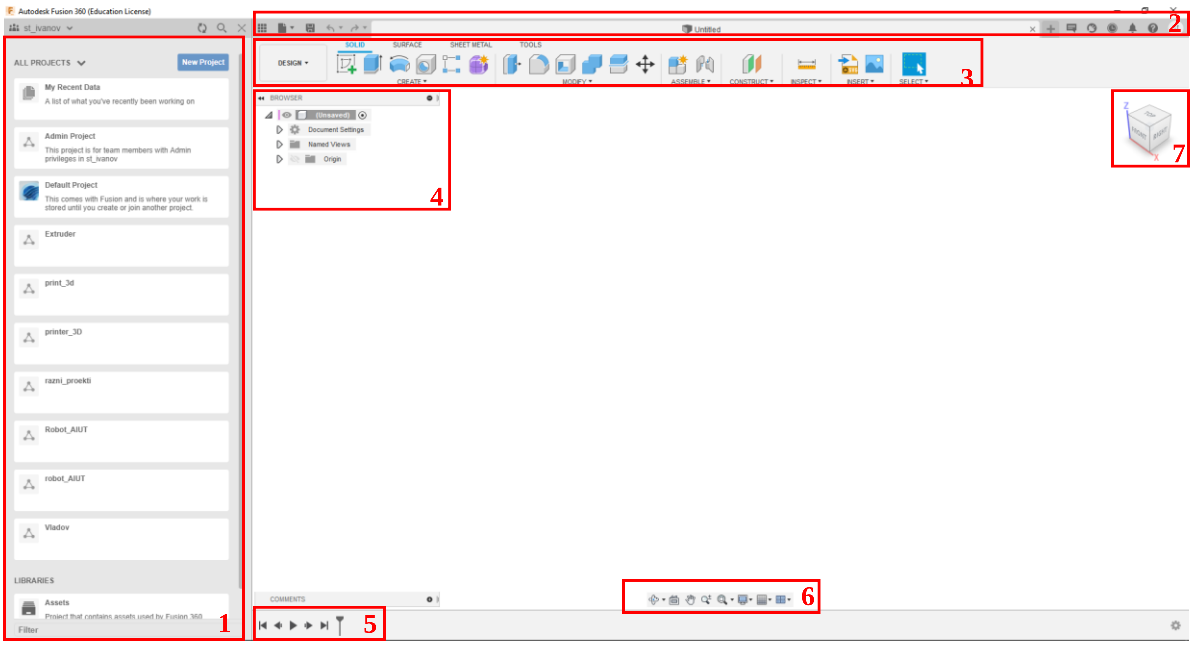

Figure 1.3

The main parts of the user interface are as follows:

1. *Data Panel* – allows easy access to different projects and files, as well as collaboration with other users in the cloud.

2. *Application Bar* – used for creating, opening, importing and exporting files, as well as for setting environment settings, accessing help information and training resources
3. *Toolbar* – contains the commands and menus used when working with the program. The toolbar changes depending on the Workspace selected and the actions currently being performed
4. *Browser* – a tree-like structure that includes all objects (components, solids, sketches, planes, etc.) contained in the current design
5. *Timeline* – a chronological presentation of the sequence of actions performed in the corresponding design. The information that is displayed depends on the active component in the browser
6. *Navigation Bar* –used to zoom in and out of the objects in the work area, and to change their visualization and the display settings
7. *View Cube* –allows for rotating the objects in the design and changing their view.

Fusion 360 supports different Workspaces, depending on the actions being performed. The choice of which workspace to activate is located in the toolbar (Figure 1.4).

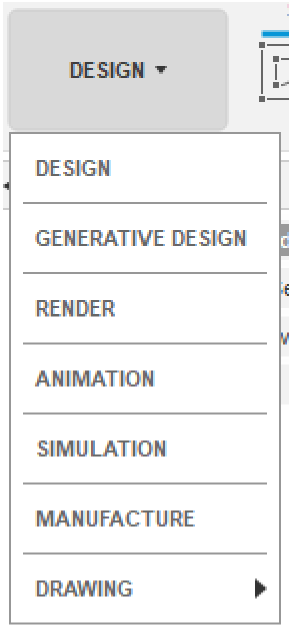

Figure 1.4

The workspaces included in the environment are:

- *Design* – the main workspace used for 3D design
- *Generative Design* – enables the use of Artificial Intelligence to automatically create optimized 3D models based on user-defined constraints
- *Render* – allows for generating realistic images of the created 3D models based on their defined physical properties
- *Animation* – allows for creating animations of the assembly sequence of models consisting of multiple components
- *Simulation* – allows for performing different types of analyses to determine the behaviour of parts and assembled units under different loads

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- *Manufacture* – allows the developed 3D models to be produced by machines with digital control or using modern additive technologies; generates a program for their production
- *Drawing* – allows for creating drawings of parts or assembly joints using the 3D models from the Design workspace or the views from the Animation workspace. Drawings represent the final stage of designing a part or a product.

Since this learning resource is intended as an introduction to the design of 3D models in Fusion 360, we will mostly present the Design workspace menus. When we start the program, the Design workspace is the first to be visualized. It provides a choice of four menus that activate different commands on the toolbar.

The SOLID menu contains commands used to create and edit 3D models (Figure 1.5).

Figure 1.5

The SURFACE menu is used to work with surfaces (Figure 1.6). Typically, it is ancillary to the main SOLID menu and its use requires some user experience in 3D modelling.

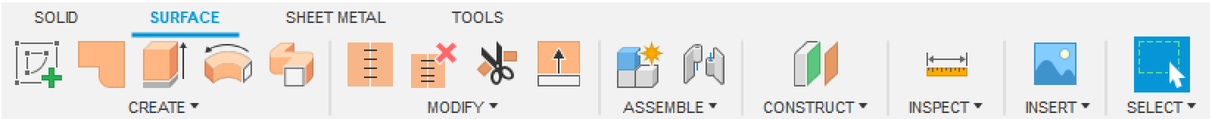

Figure 1.6

The SHEET METAL menu allows models to be made from sheet material (Fig.1.7). Compared to other 3D modelling environments, Fusion 360 has made working with sheet material relatively easy and intuitive for the user.

Figure 1.7

The TOOLS menu (Figure 1.8) contains commands that can be used to perform additional checks on parts. Here it is also possible to use add-ons that are installed in Fusion 360 usually with the task to automate the creation of machine components.

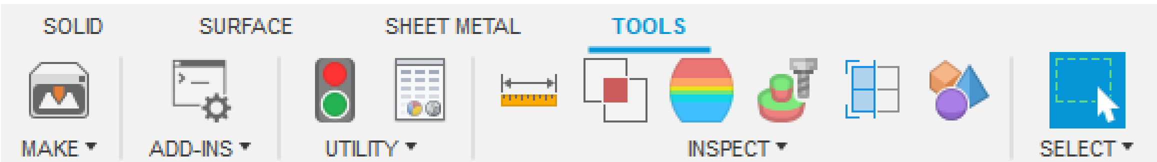

Figure 1.8

When creating a 3D model, we initially draw 2D sketches of the object's underlying geometry. When we create a new sketch, the Toolbar changes and appears as shown in Figure 1.9.

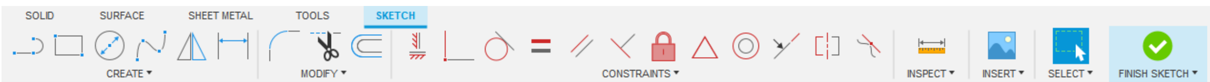

Figure 1.9

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Fusion 360 has the built-in ability to create objects using the so-called Virtual Sculpting. The FORM menu is used for this purpose (Figure 1.10). In this approach, the so-called T-Spline primitives are used and they can be manipulated similarly to virtual clay.

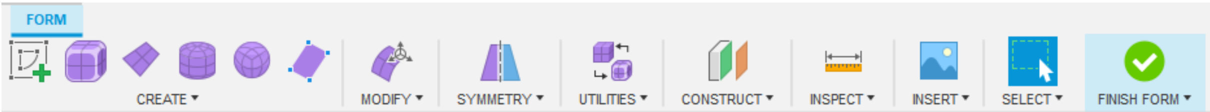

Figure 1.10

The Navigation Bar (Fig.1.11) includes tools for controlling the work area view.

Figure 1.11

Fusion 360 makes it possible to select either the standard view for working with 3D objects or multiple simultaneous views of a single object in the work area – Figure 1.12.

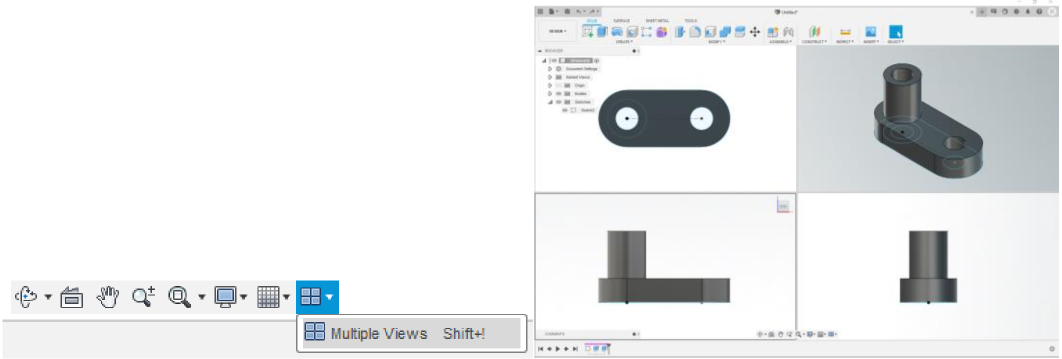

Figure 1.12

The user can turn a layout grid on or off, and to tether the cursor movement to it (Figure 1.13).

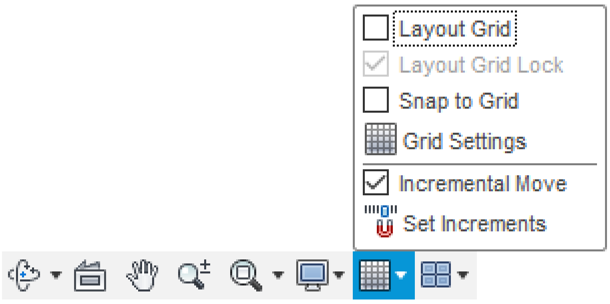

Figure 1.13

Perhaps the most widely used tools on the Navigation Bar are those used to set the style of 3D objects displayed in the workspace (Figure 1.15).

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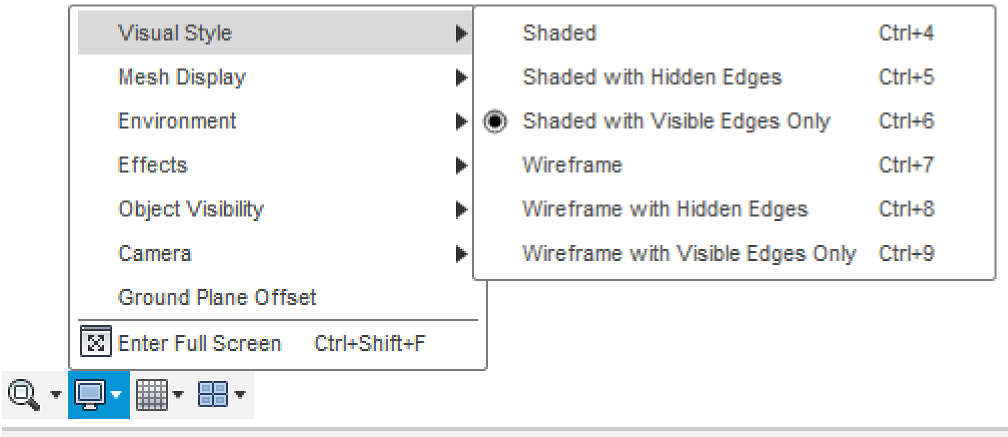
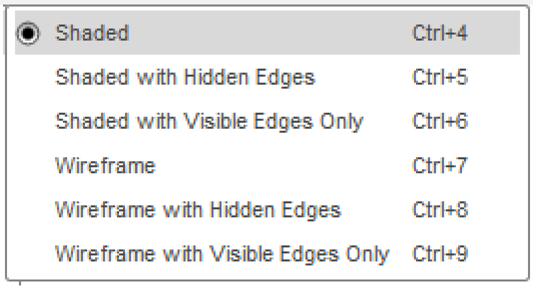
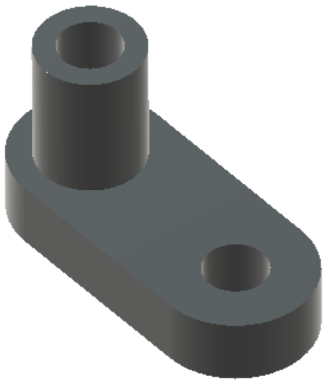
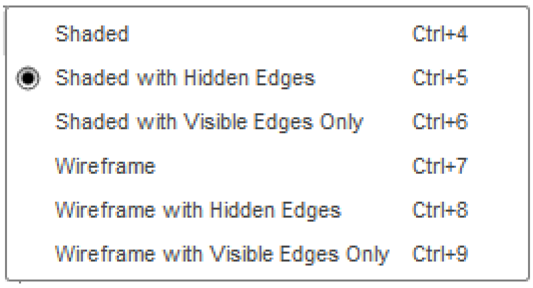
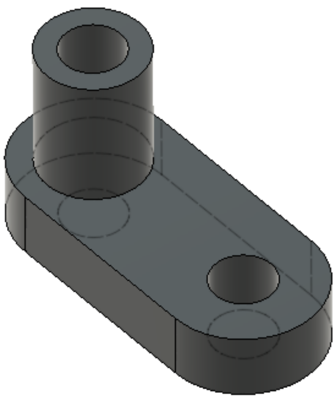
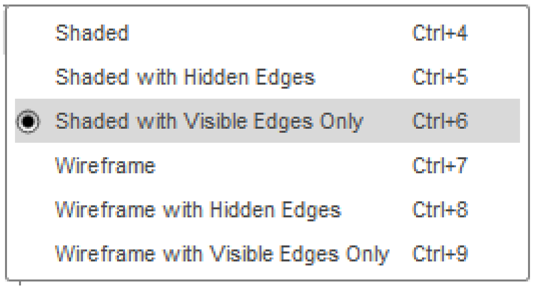


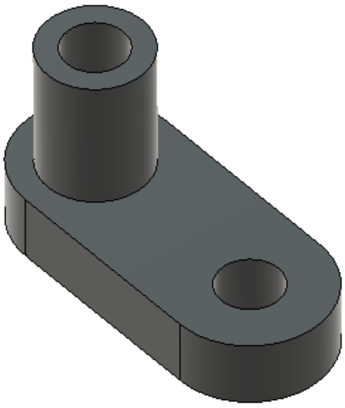
Figure 1.15

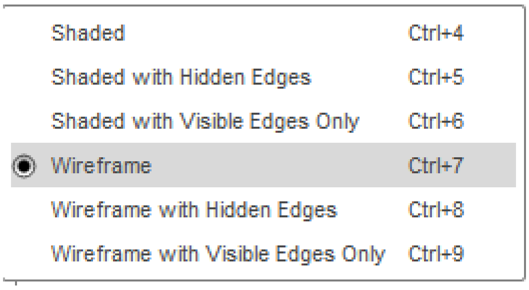
Depending on what we select in the Visual Style menu, we obtain the following basic types of rendering of 3D details – Table 1.1.

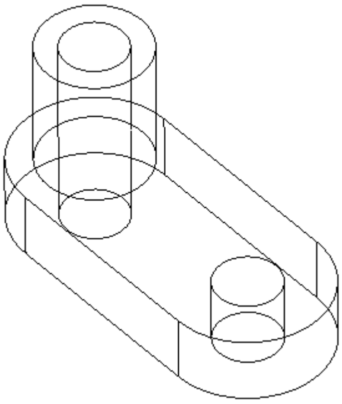
Table 1.1

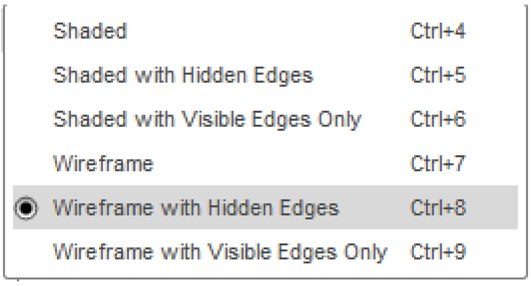
Rendering style	Visualisation of the detail
	
	

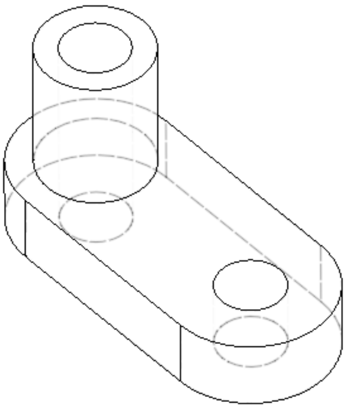


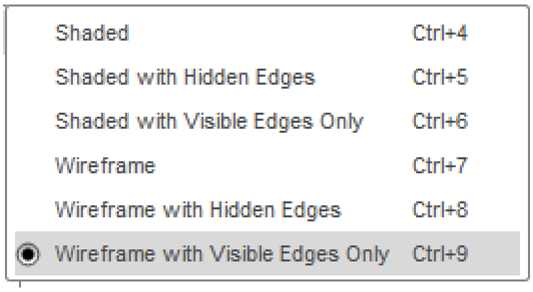


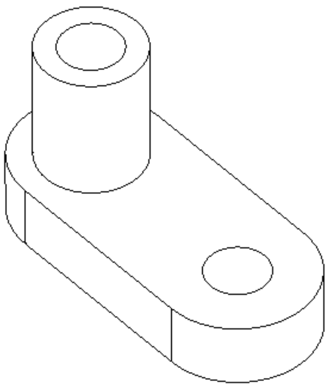












The other tools in the Navigation Bar are related to zooming the view of 3D objects, moving and rotating them.

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The Application Bar (Figure 1.16) contains menus used to set up the environment, receive notifications, and access tutorials and a gallery of 3D models published by other Fusion 360 users.

Figure 1.16

Fusion 360 is designed to be used primarily in online mode. However, the user can choose to work in offline mode when network access is unavailable for a period of time (Figure 1.17). After connecting to the network and returning to the standard online mode of operation, data synchronization is automatically performed.

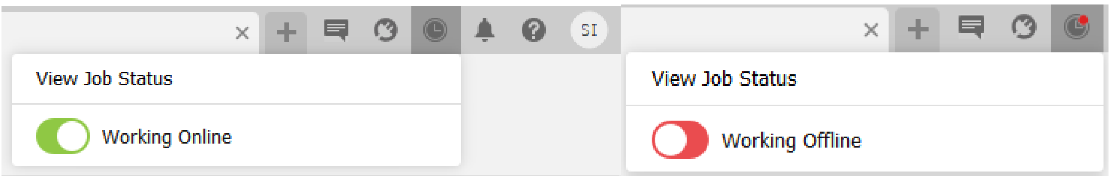

Figure 1.17

The Help menu (Fig.1.18) provides quick access to training resources, documentation for the environment, a link to a user forum and a gallery of finished projects.

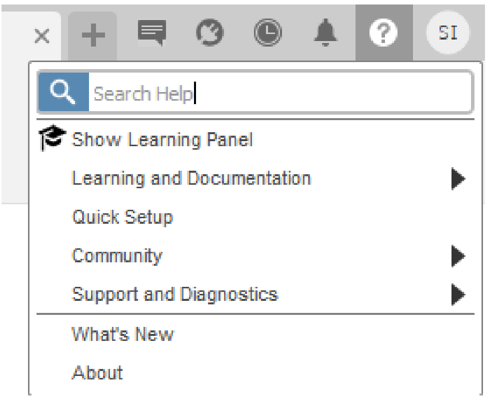

Figure 1.18

The Quick Setup command (Figure 1.19) is used to quickly set up the mouse operation and keyboard shortcuts that control the behaviour of 3D objects. The users can configure these settings to resemble those in other 3D modelling environments that they are familiar with, such as SolidWorks, Inventor, and others.

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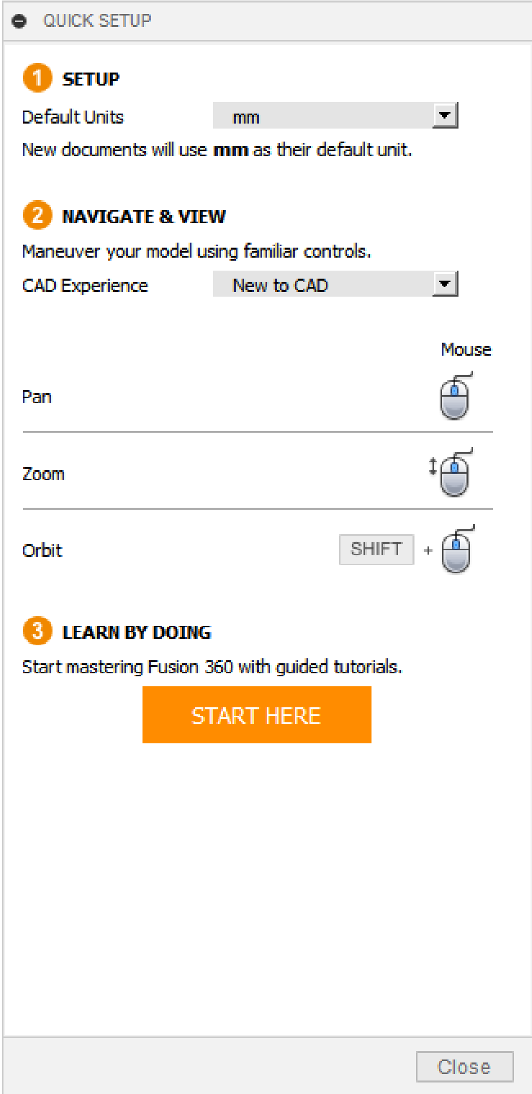

Figure 1.19

The Help menu also includes the Graphic Diagnostic command (Figure 1.20), which can diagnose the computer system and make optimizations to speed up and improve the performance of Fusion 360. Since the program can be installed on different platforms with different graphic card capabilities, it is advisable to run the Graphic Diagnostic after installing Fusion 360 and, if necessary, to perform the necessary optimizations.

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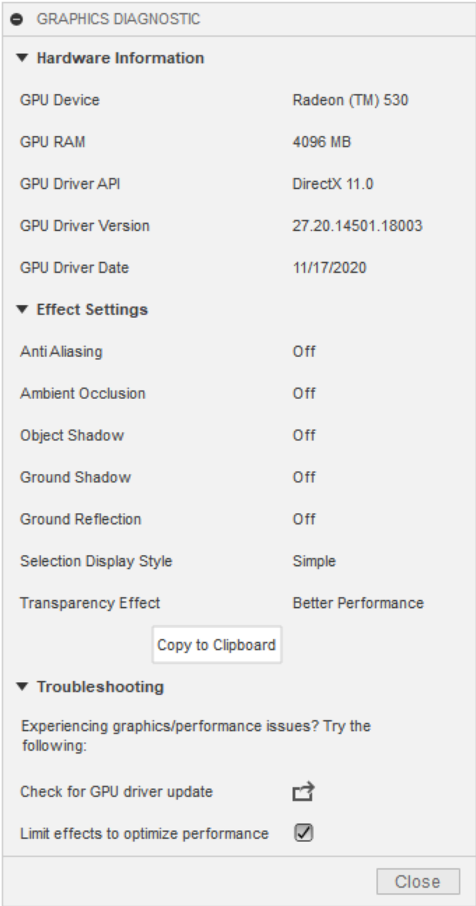

Figure 1.20

Creating and working with projects and files

When working with Fusion 360 the user can create projects, folders in projects and files. Projects are created by clicking on the New Project button (Figure 1.21) in the Data Panel.

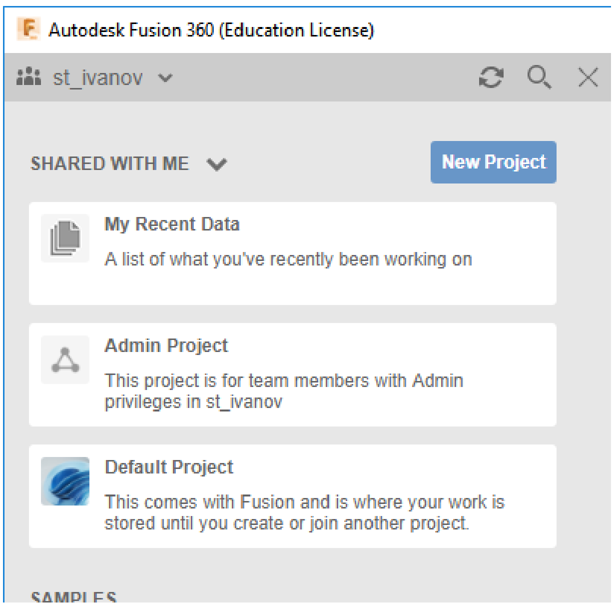

Fig .1.21

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Folders can be added to each project in order to better organize the files. This is done via the New Folder button (Figure 1.22), after entering the respective project.

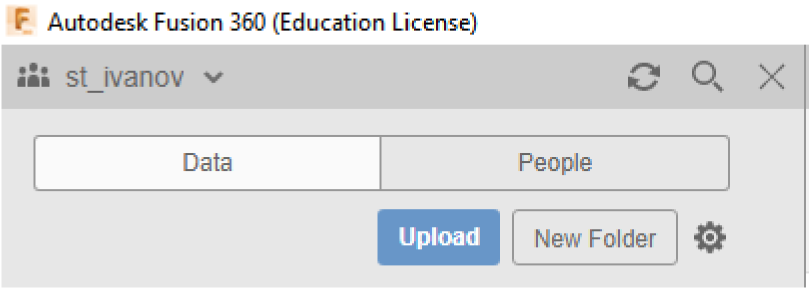

Figure 1.22

Next, the user chooses the name of the folder that will be added to the project (Figure 1.23).

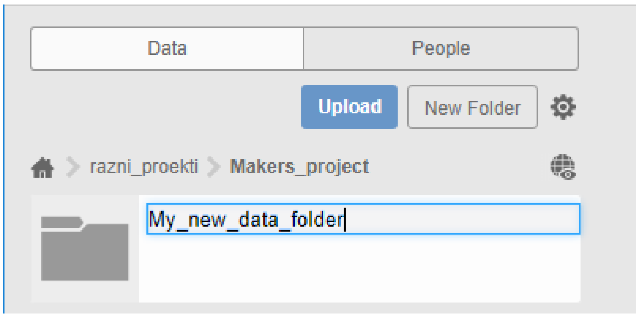

Figure 1.23

The newly created folder is empty (Figure 1.24), but it can be populated with files that are on the personal computer or are newly created.

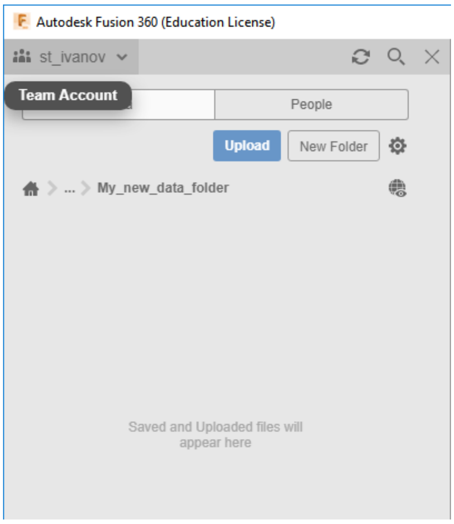

Figure 1.24

The Upload button enables us to save into the folder (and respectively in the cloud) files that are located on the personal computer (Figure 1.25).

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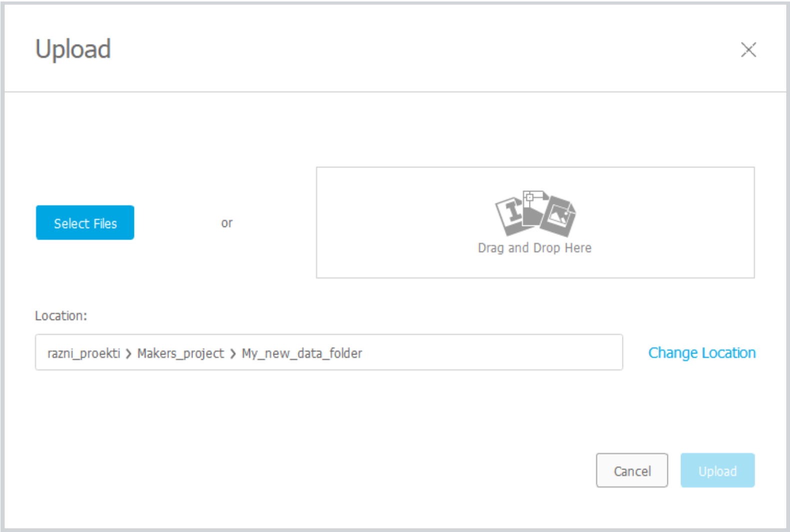

Figure 1.25

By using the commands from the Application Bar, we can perform operations to open, save, and export files, to create new designs, etc. (Figure 1.26).

Figure 1.26

When Fusion 360 is started, a default file named Untitled is always opened. The corresponding 3D design is created in the workspace and the file can be saved using the Save command (Figure 1.27) or the keyboard shortcut Ctrl+S.

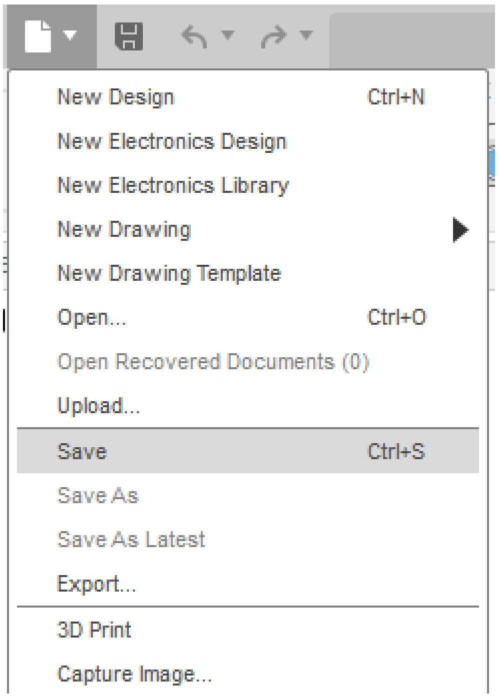

Figure 1.27

When saving the file, we must specify its name, as well as the project and folder in which the file will be stored (Fig.1.28).

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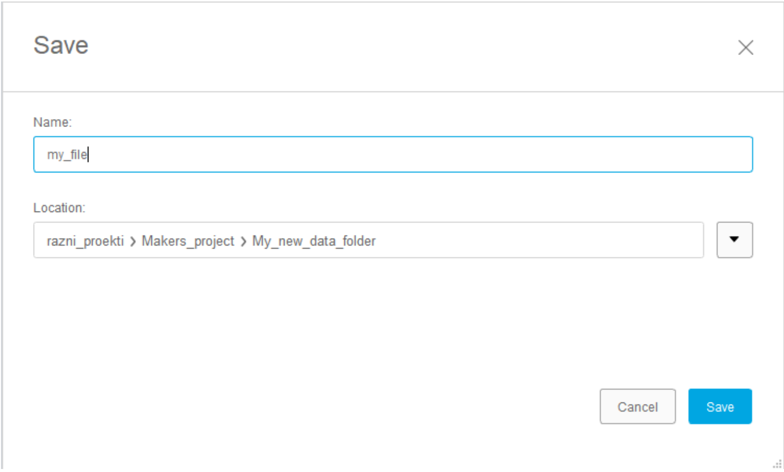

Figure 1.28

After saving, the file appears in the data panel of the corresponding folder (Figure 1.29).

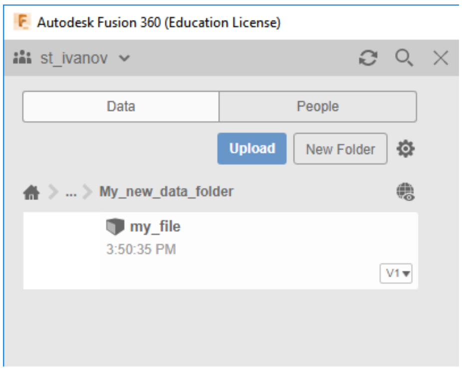

Figure 1.29

The properties of each file, the versions saved on the cloud server, and other specific features can be viewed by clicking on the drop-down menu to the side of each file in the data panel (Figure 1.30).

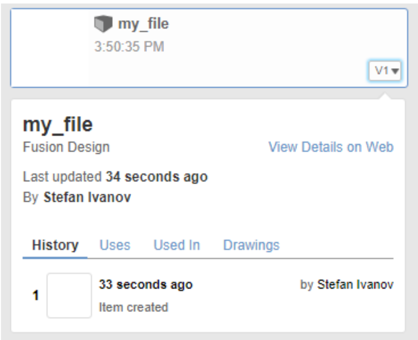

Figure 1.30

The Open command is used to open a file. The dialog window that appears gives us the option to open files that are in the cloud or hosted on our personal computer (Figure 1.31).

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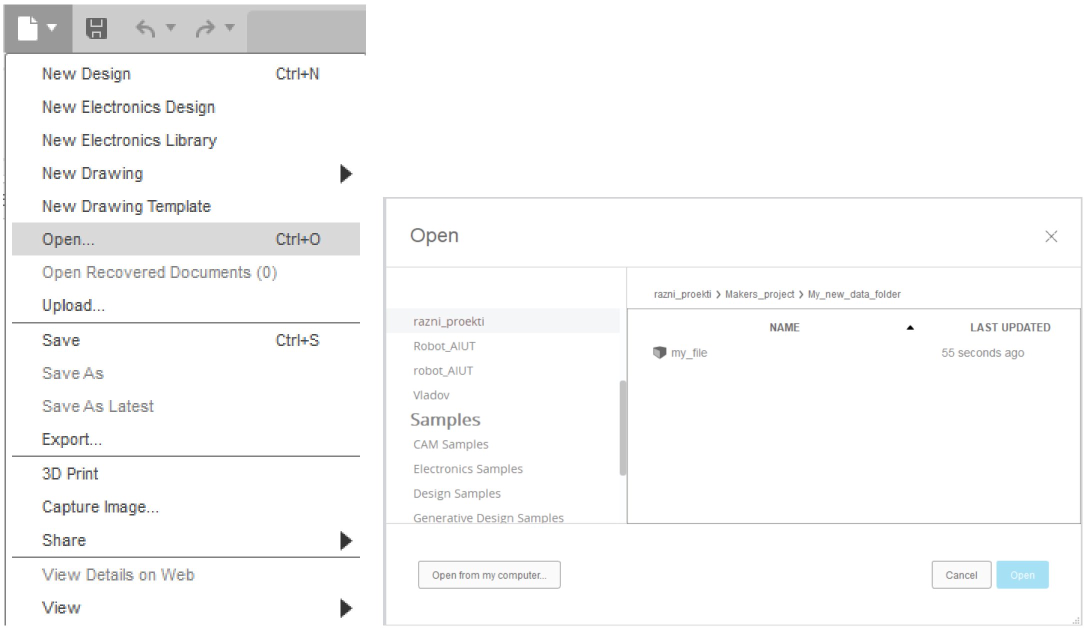

Figure 1.31

When files located on the computer's hard disk are opened, the file is simultaneously uploaded to the Fusion 360 environment. The environment allows different file types to be uploaded (Figure 1.32)

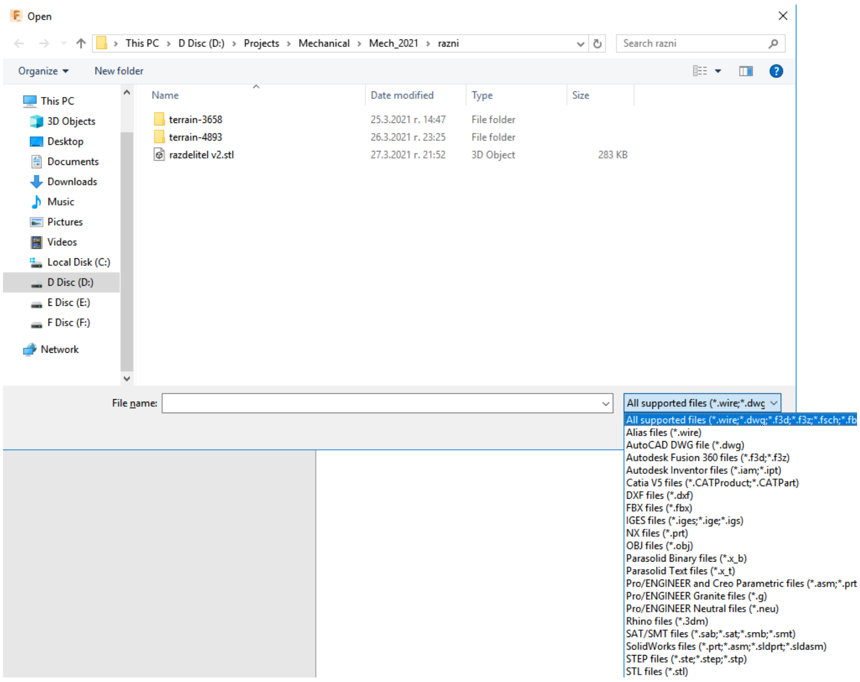

Figure 1.32

In addition to files from our personal computer, we can also open files from projects created in Fusion 360 (Figure 1.33).

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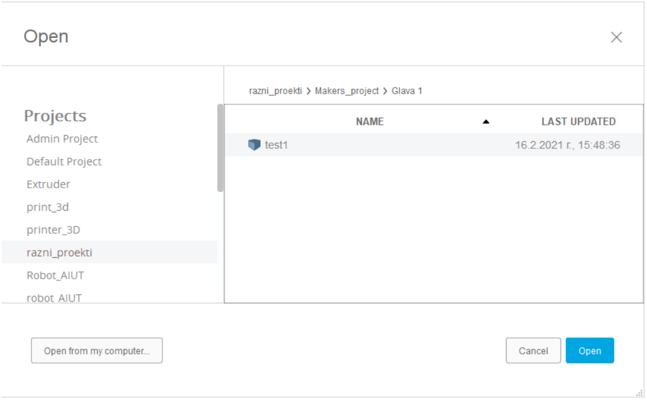

Figure 1.33

After opening the selected file, the 3D model saved in it is previewed in the workspace and can then be edited (Figure 1.34).

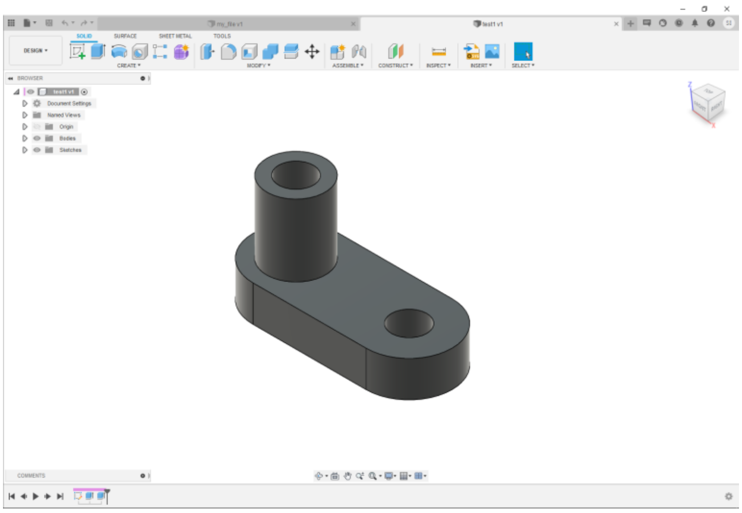

Figure 1.34

Exporting files; saving .STL files for 3D printing

The 3D model files created in the Fusion 360 environment, as already mentioned, are stored in the cloud. However, we can export designs and save them locally in our computer. This can be done with the Export menu in the Application Bar (Figure 1.35).

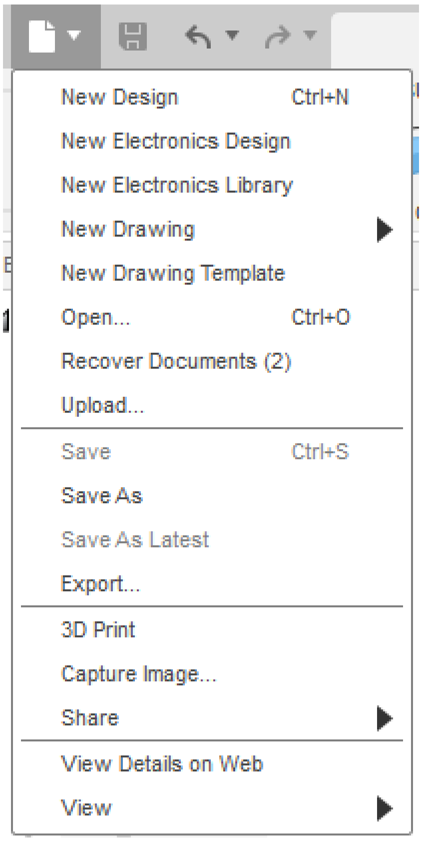

Figure 1.35

The Export menu allows the selected file to be saved in various formats (Figure 1.36). The basic format for saving Fusion 360 files has the .f3d extension. However, 3D models can also be exported to file formats used by other types of 3D design programs, such as the .stp (or .step) file format - Standard for the Exchange of Product Data - designed for data exchange between different CAD design programs.

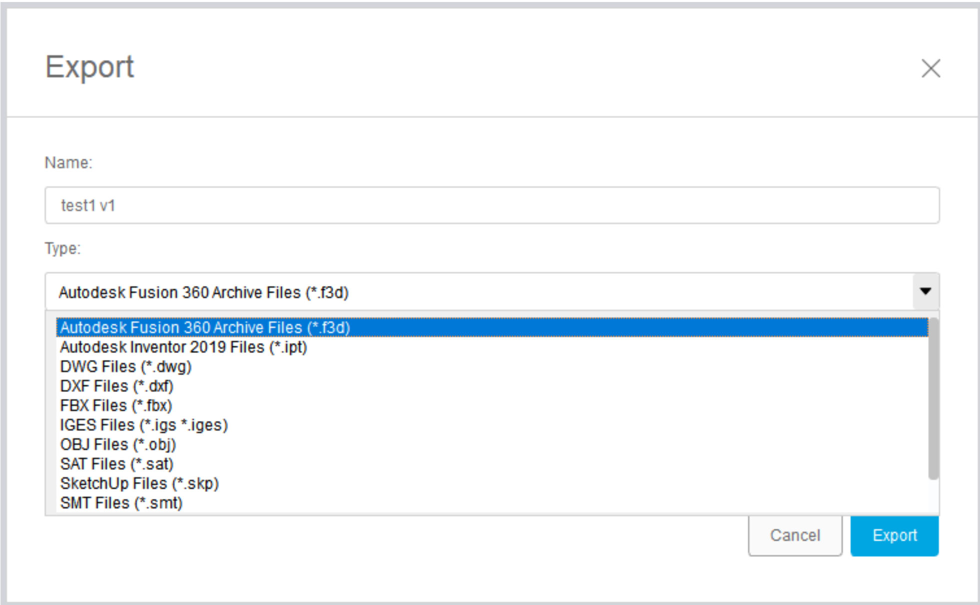

Figure 1.36

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From the Export menu it is possible to save files in the .stl format. This file format is used to save data for 3D models and is the most common file format used in 3D printing.

Since this learning resource is intended mainly to present the Fusion 360 capabilities for creating 3D models for printing, it is necessary to mention that a .stl file can be generated from a component or from a body by right-clicking on it and selecting the Save As STL command (Figure 1.37).

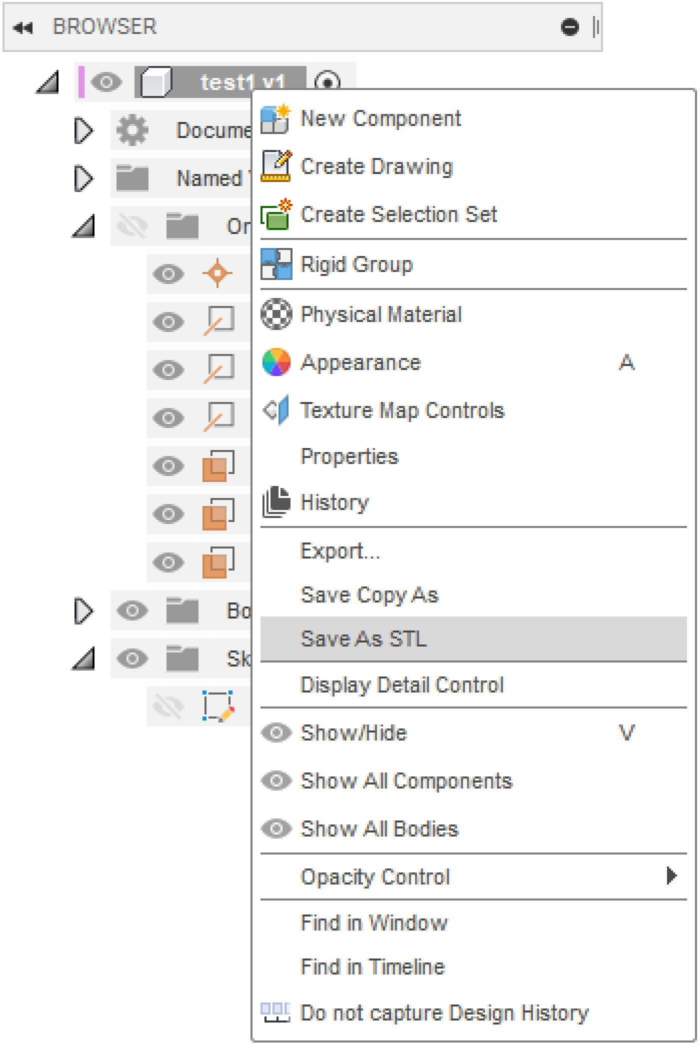

Figure 1.37

When saving the .stl file, the environment allows us to select the quality of the file and to preview the file (Fig.1.38).

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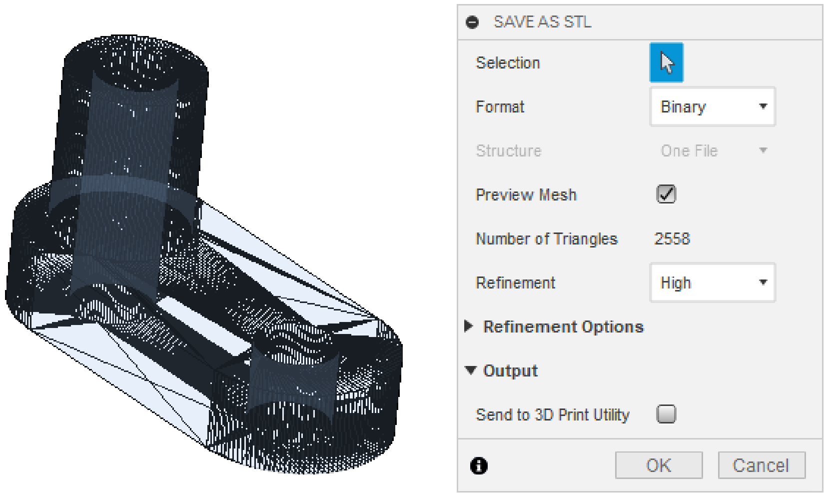

Figure 1.38

This chapter presented the main features of the Fusion 360 environment interface. The program is very intuitive. The menus are easy to work with and the number of commands used has been optimized so that a novice user can quickly become comfortable with the environment.

Additional self-study video materials on how to work with the environment interface:

https://www.youtube.com/watch?v=uQVyKsmlWTo

https://www.youtube.com/watch?v=uZIirtstwRk

https://www.youtube.com/watch?v=w1lzoXYcSQY

https://www.youtube.com/watch?v=wAb6KI5G1V8

https://www.youtube.com/watch?v=F2ybXYNUigw

https://www.youtube.com/watch?v=rJqToV33EQI

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Sketches as a basis in 3D modelling

2D sketches form the basis of 3D objects created in Fusion 360. Sketches are created using the Create Sketch command from the Toolbars (Figure 2.1).

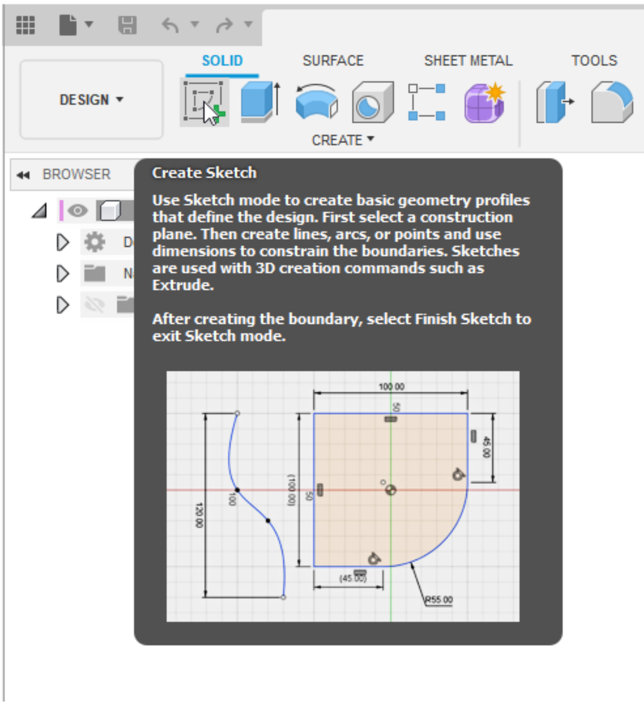

Figure 2.1

Sketches are created on planes or planar surfaces. When creating a new 3D object, the initial sketch is usually placed on one of the planes of the coordinate system (Figure 2.2).

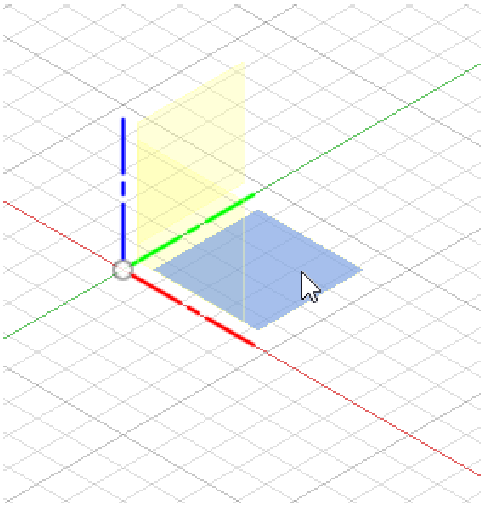

Figure 2.2

Once the plane on which the sketch will be created is selected, the toolbar changes and takes on the form shown in Figure 2.3 below.

Figure 2.3

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The commands that appear in the toolbar are used to create sketches, edit sketches and define the so-called sketch constraints. When creating sketches, commands are used to draw lines, rectangles, circles, arcs, polygons, ellipses, slots, and cubic splines and ellipses (Figure 2.4).

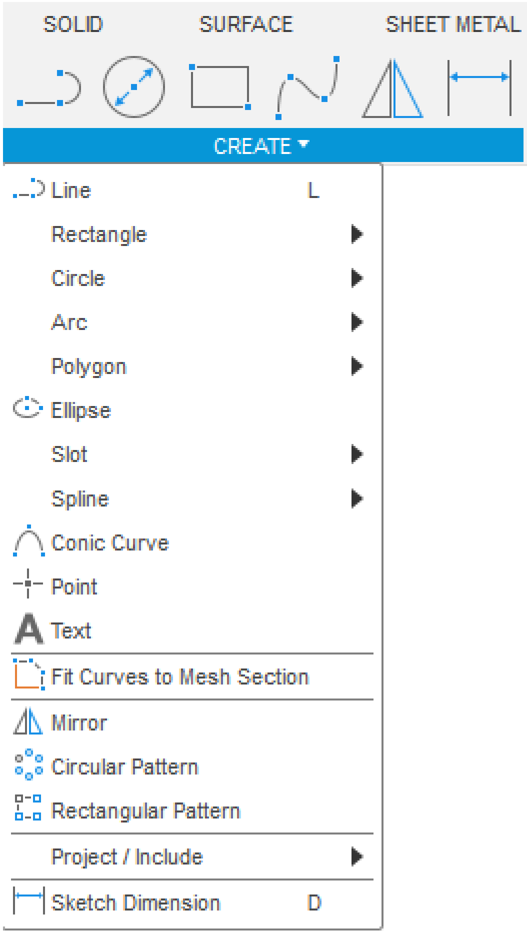

Figure 2.4

The sketch editing commands make it possible to modify already created sketches or the two-dimensional objects in them. The main editing commands allow for: creating a curvature between two intersecting lines, cutting lines, extending lines to the next object, breaking lines, scaling objects, and creating an offset image of selected objects (Figure 2.5).

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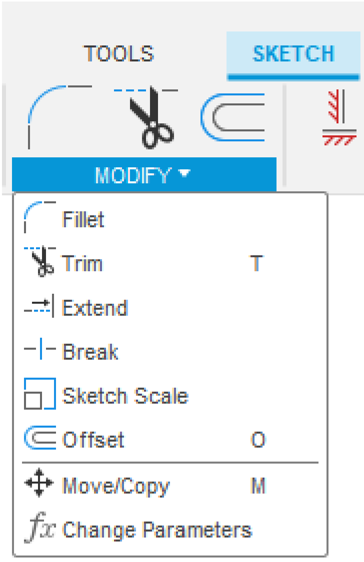

Figure 2.5

Constraint definition commands (Figure 2.6) are used to define the relationships between objects in the sketch or the position of objects in the coordinate system. The commands can define the following characteristics: horizontality/verticality of objects with respect to the coordinate system, overlapping of a point of one object with the coordinates of another object, tangent contact of a line with a circle, identity of two objects, parallelism, perpendicularity and collinearity of two lines, concentricity of two circles, etc.

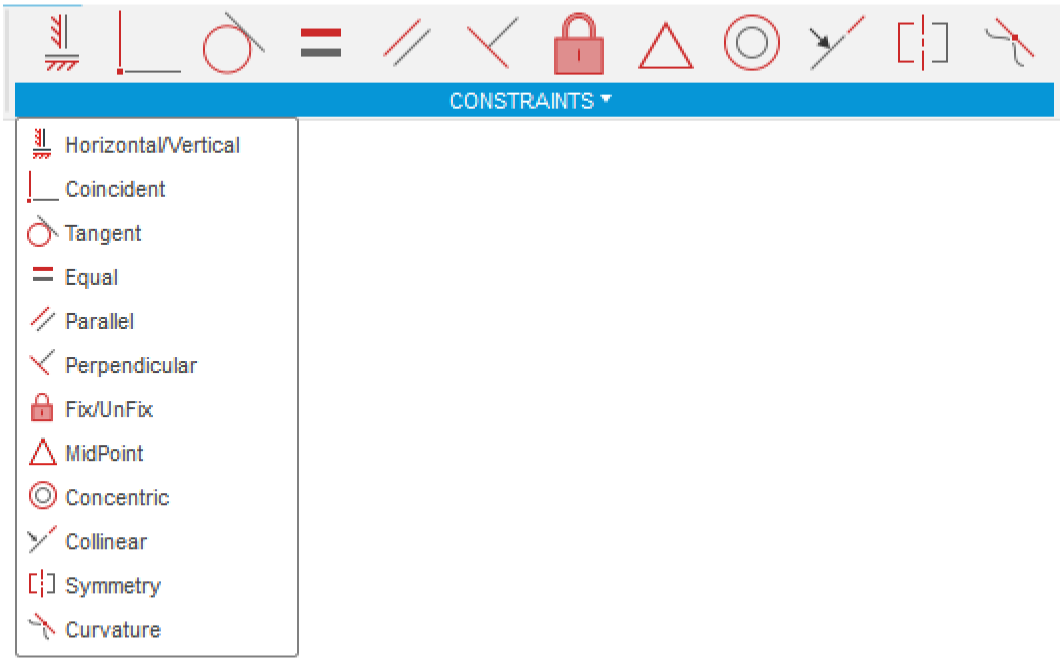

Figure 2.6

The sketch workspace also contains the so-called Sketch Palette that allows us to define additional features in the selected 2D objects or drawing commands. These features can include the way the objects are drawn, the type of lines, the binding to the drawing grid, the possibility to draw sketches in three-dimensional space, etc. (Figure 2.7).

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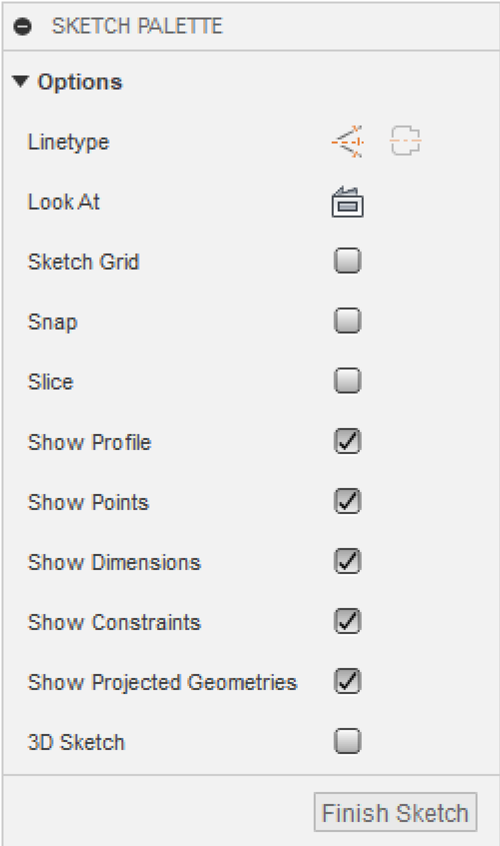

Figure 2.7

After we finish the sketch, it is necessary to exit the sketching mode. This is done by clicking on the Finish Sketch button from the Sketch Palette or on the special Finish Sketch command (Figure 2.8).

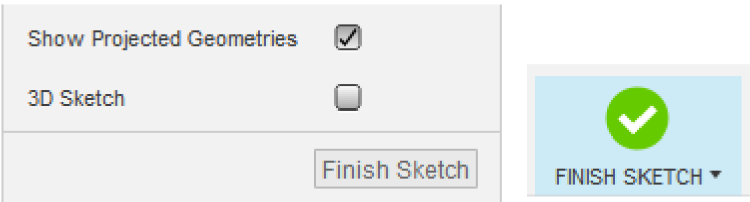

Figure 2.8

Each created sketch is displayed in the tree-like structure of the browser in the Sketches section of the main component or in the local Sketches sections of the components included in the file (if the file contains any components) (Figure 2.9). Each sketch is automatically assigned a name by Fusion 360 but the name can subsequently be changed.

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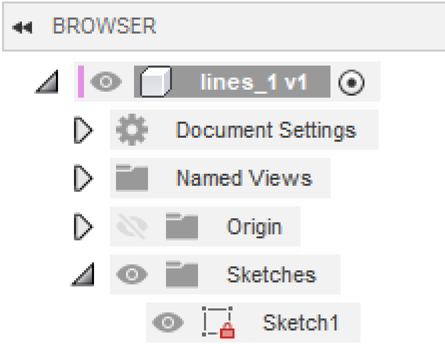

Figure 2.9

Line Sketching

Line sketching is the most basic approach to creating 2D objects in sketches. The Create Lines command also enables the creation of tangent curves to existing lines or other curves.

Before starting to draw lines, it is necessary to select the plane on which to draw. For this sketch, let us select the XY base plane (Figure 2.10).

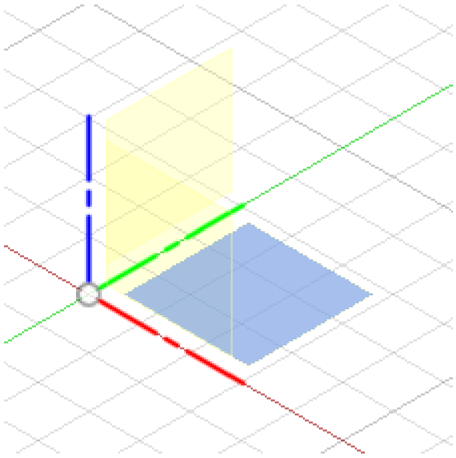

Figure 2.10

From the sketching toolbar, let us select the Create Lines and Arcs command (Figure 2.11).

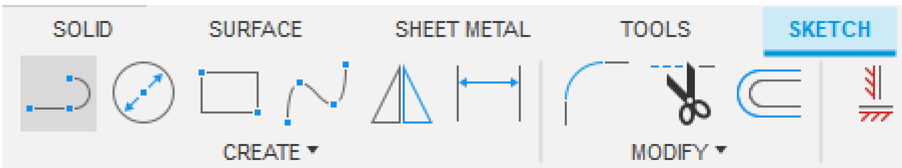

Figure 2.11

The same command can also be selected from the Create drop-down menu, which includes all the commands used for sketching (Figure 2.12).

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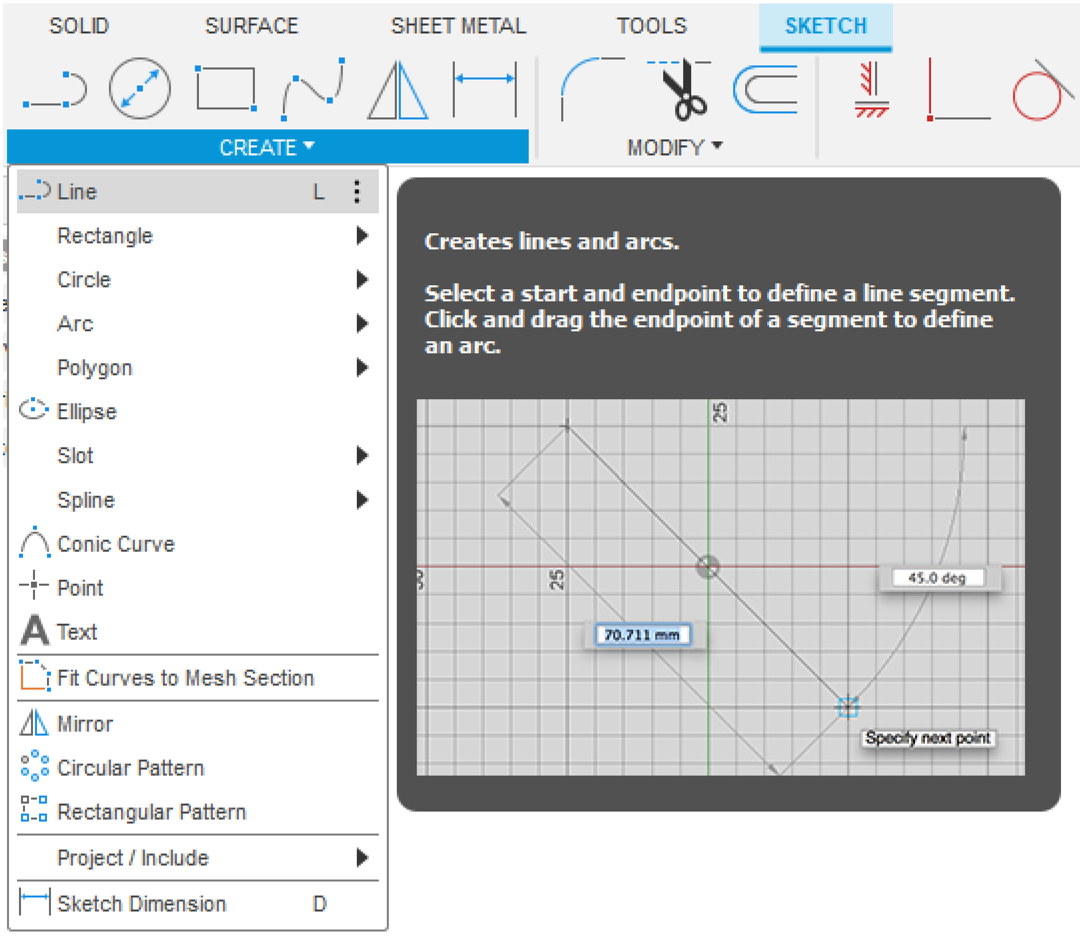

Figure 2.12

We start drawing the line by clicking in the work area where the starting point will be and dragging the mouse to draw the corresponding line. A new click sets the end of the line and starts drawing the next line, and, in addition to its size, it displays the angle with respect to the previous one (Figure 2.13).

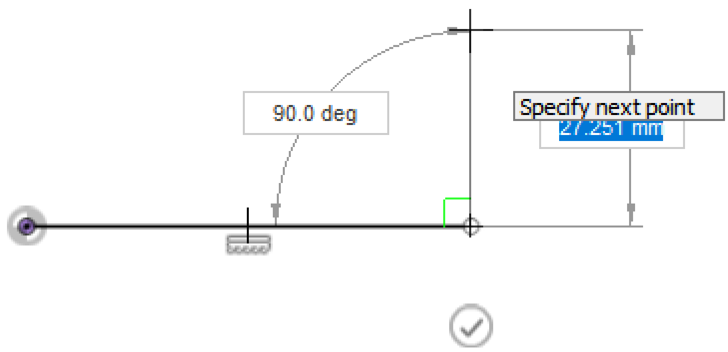

Figure 2.13

If we continue drawing lines we can construct a closed contour figure that can subsequently be used as the basis for creating a 3D solid. When we create a closed contour figure, the figure itself is recognized as a 2D surface enclosing the corresponding area (Figure 2.14).

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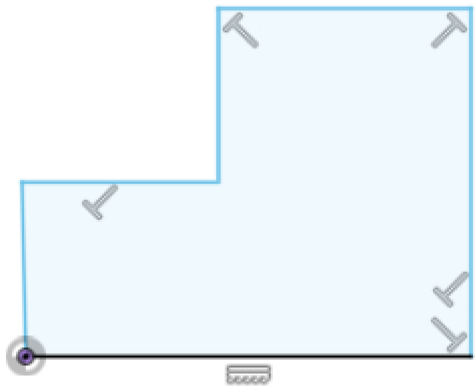

Figure 2.14

When drawing a 2D object, Fusion 360 displays the size of the object in a help window. It is necessary to use the Sketch Dimension command on the toolbar (Figure 2.15) or the Ctrl+D key combination to accurately set the dimensions of the drawn object.

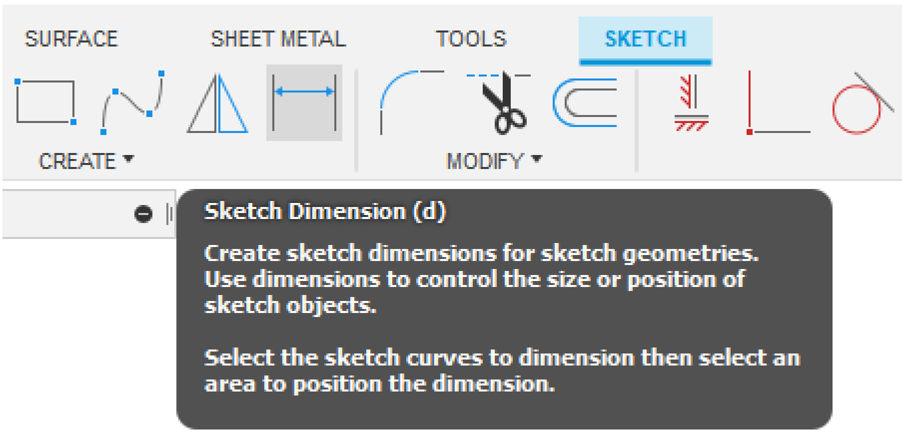

Figure 2.15

When the size of a line or another 2D object is not specified or referenced to the coordinate system, the line or 2D object are displayed in blue.

In sketch dimensioning, objects with defined parameters are displayed in black. The sizing of a sketch should continue until all objects in the sketch are rendered in black. If more dimensions than necessary are placed during the dimensioning, some of them are implemented as so-called *driven dimensions* and are displayed in brackets (Figure 2.16).

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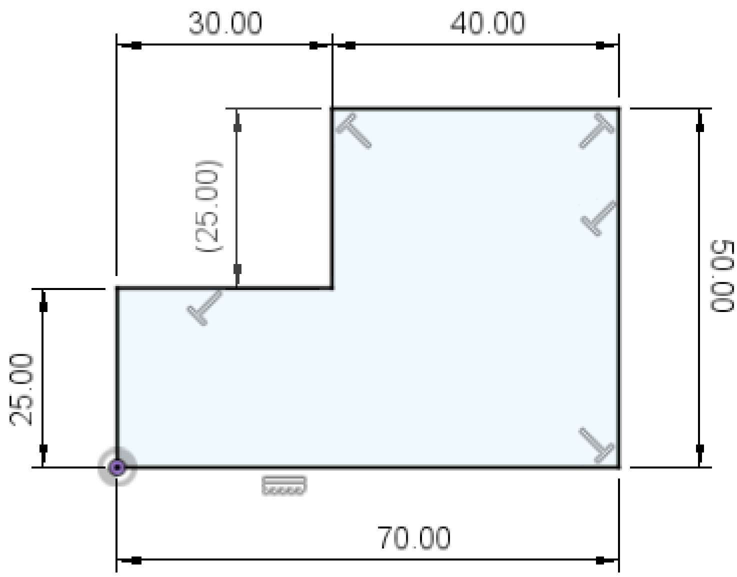

Figure 2.16

The line drawing commands allow lines to be drawn sequentially, each new one starting from the end of the previous one. If it is necessary to terminate the drawing process, this can be done in the following ways: by double-clicking with the mouse; by pressing the Esc key; or by selecting the end-draw icon, which appears immediately after clicking to end the current line (Figure 2.17).

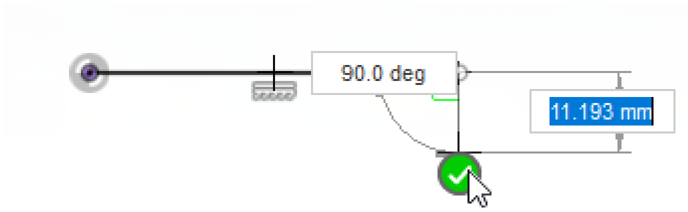

Figure 2.17

In addition to drawing straight lines, we can also use this command to draw curves, tangent to the last line or curve. This is done by holding down the mouse button after clicking, and dragging to display the corresponding curve (Figure 2.18).

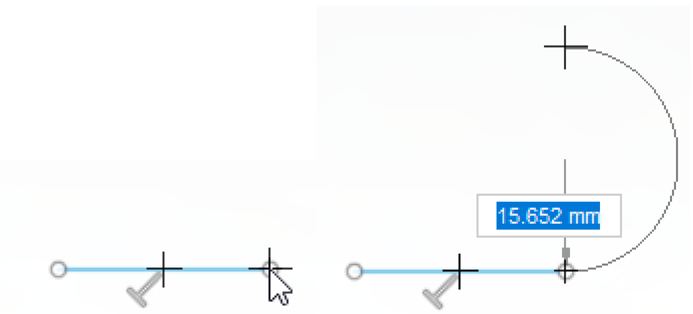

Figure 2.18

The line drawing command can also be used to close the outline of 2D objects. If the objects in the sketch do not have a closed contour, then they do not define a surface that can be used when creating the 3D models. If we have such objects in the sketch, contour closure can be performed either by dragging one contour vertex over the other or by drawing a connecting line (or arc) between them (Figure 2.19).

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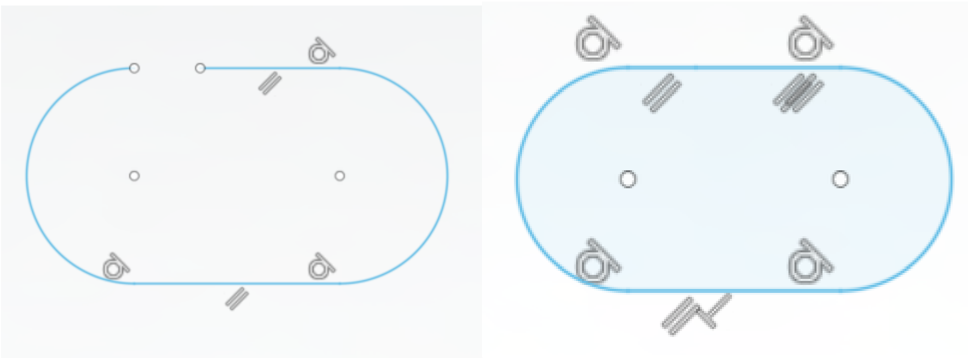

Figure 2.19

The lines that are drawn can be defined as auxiliary (Construction) or axial (Centerline) – Figure 2.20. Such lines will not be involved in defining the surface enclosed by the sketch, but they can be used as auxiliary lines when mirroring objects in the sketch or as centerlines when creating 3D parts by extrusion by rotation.

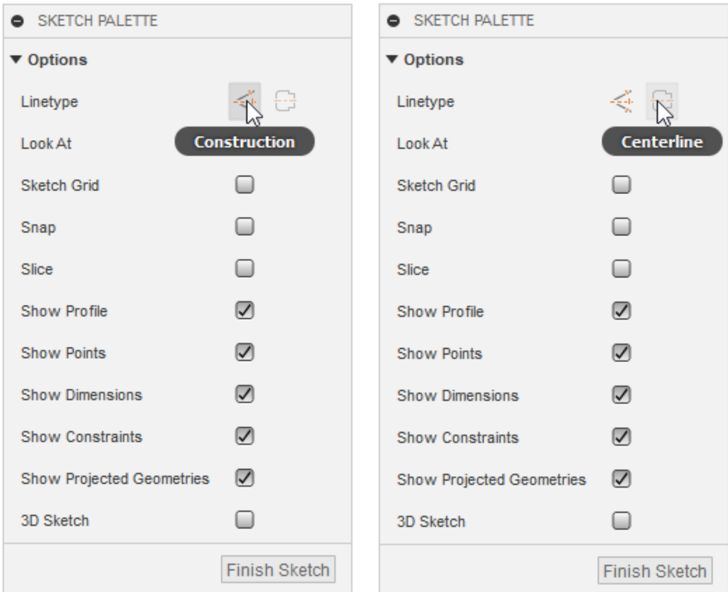

Figure 2.20

Construction lines and Centerlines lines differ from continuous standard drawing lines (Figure 2.21).

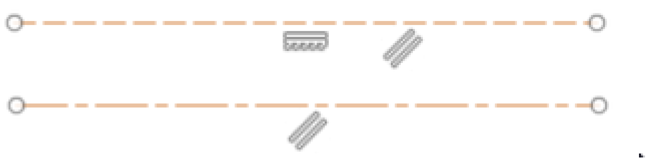

Figure 2.21

An example of the use of a Construction line is presented in Figure 2.22. Figure 2.22(a) shows a contour that is mirrored to the Construction line to produce the closed contour in Figure 2.22(b).

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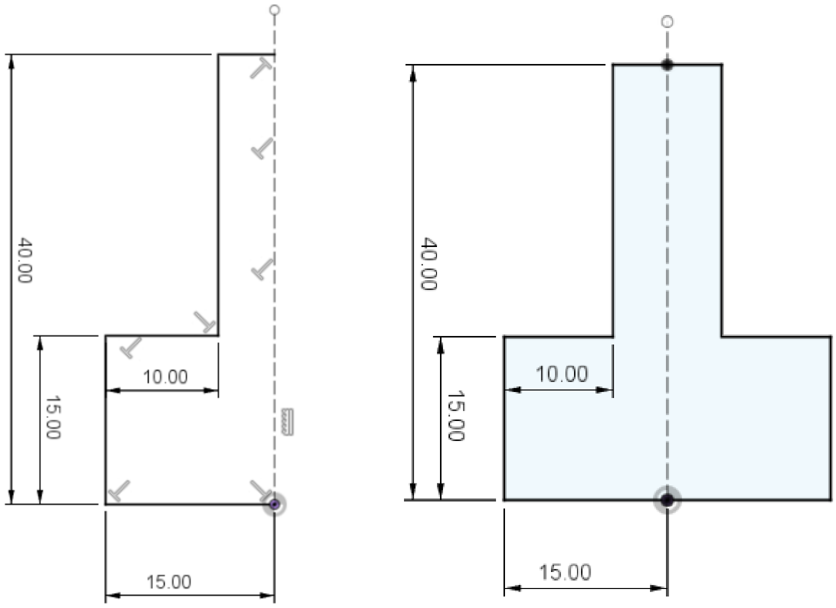

a) b)

Figure 2.22

Sketching different geometric objects

Fusion 360 allows us to draw in the sketches different geometric objects such as circles, rectangles, polygons, ellipses, etc. The drawing of geometric objects can be performed in different ways, which brings additional convenience and flexibility when creating sketches.

Drawing a circle

Drawing a circle is done by selecting the appropriate command from the toolbar or from the drop-down menu (Figure 2.23). The environment allows us to choose to define the circle by:

- a centre and a diameter
- two points defining the diameter
- three points through which the circle passes
- selecting two lines touching the circle and its diameter
- three lines touching the circle.

Regardless of which circle creation command we have selected, it can be changed from the Sketch Palette while we work.

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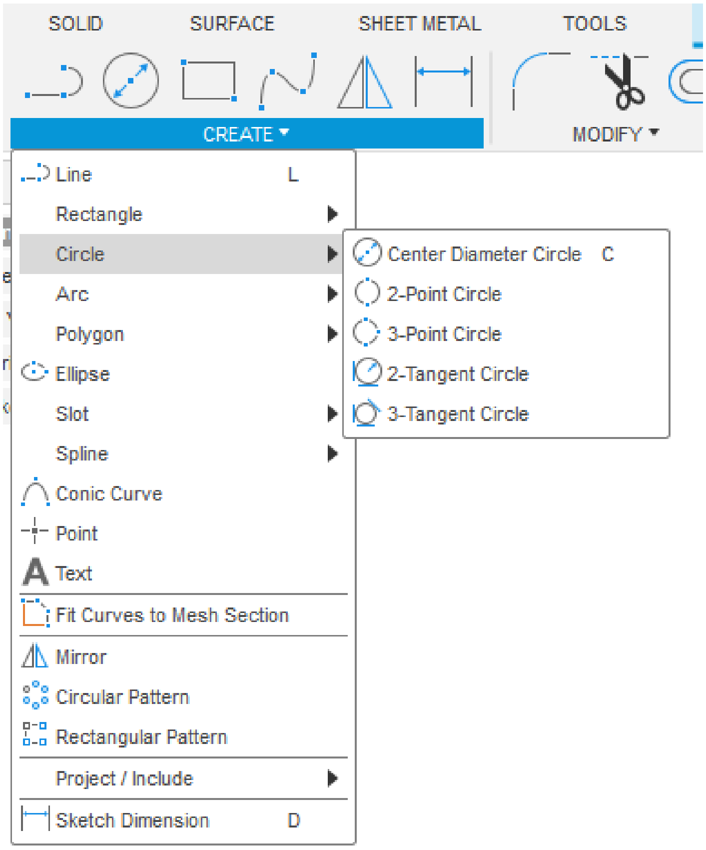

Figure 2.23

When drawing a circle with a centre and diameter, we need to set the centre of the circle anywhere in the work area, and define its diameter by dragging the mouse (Figure 2.24).

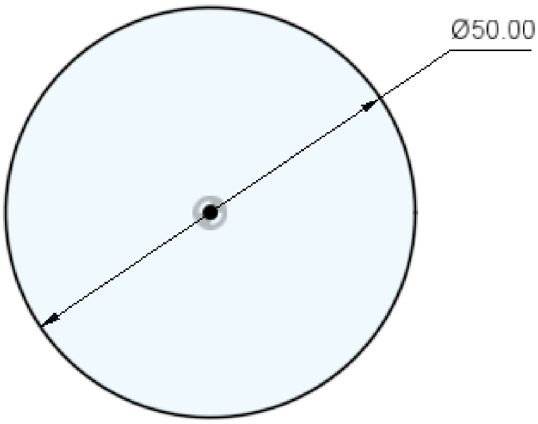

Figure 2.24

When drawing a circle using two points, the diameter of the circle is set to be equal to the segment defined by the two points, and the centre of the circle lies in the middle of this segment (Figure 2.25).

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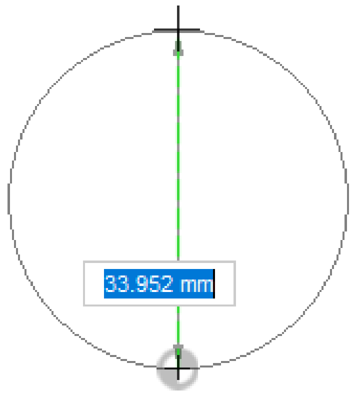

Figure 2.25

A circle can also be defined by using the mouse to click on three points through which the circle will be drawn (Figure 2.26).

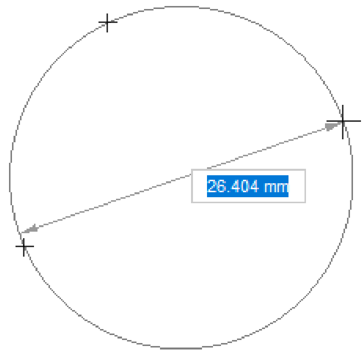

Figure 2.26

A circle can be drawn as a tangent to two lines. In order to use this command, we must have two lines already drawn in the sketch. We mark those with the mouse, and then we can define the diameter of the circle by dragging with the mouse. In this approach, the circle is tangent to the lines or their imaginary continuation (Figure 2.27).

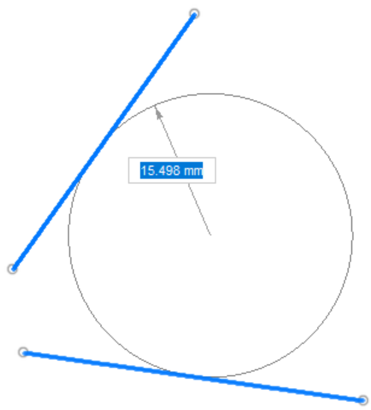

Figure 2.27

If we have three intersecting lines drawn on the sketch, we can use them to create a circle that touches all three of them (Figure 2.28).

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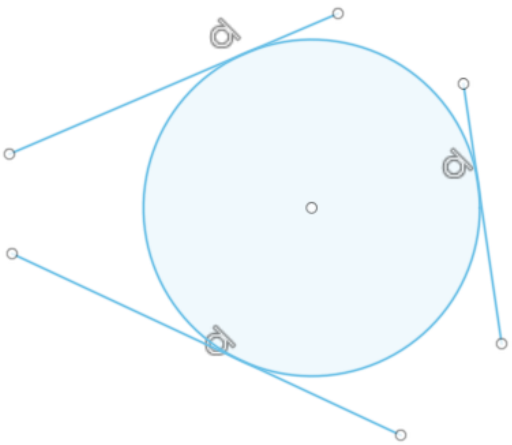

Figure 2.28

Drawing a rectangle

Three commands can be used to define and draw a rectangle (Figure 2.29):

- two points
- three points
- a rectangle centre and a diagonal.

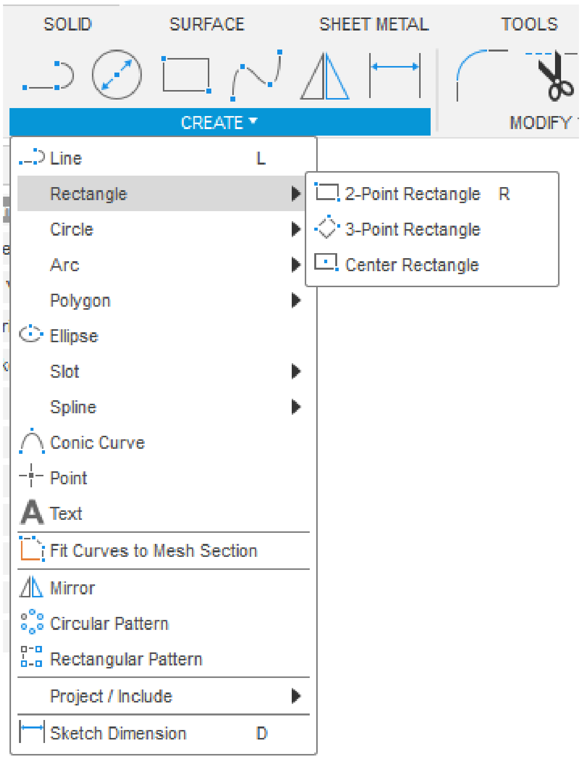

Figure 2.29

When creating a rectangle by two points, we click with the mouse to set the first point, then drag and set the dimensions of the rectangle accordingly, and with a new click we position it on the sketch (Figure 2.30). In this way, the walls of the rectangle are guaranteed to be positioned horizontally and vertically relative to the orientation of the 2D sketch plane.

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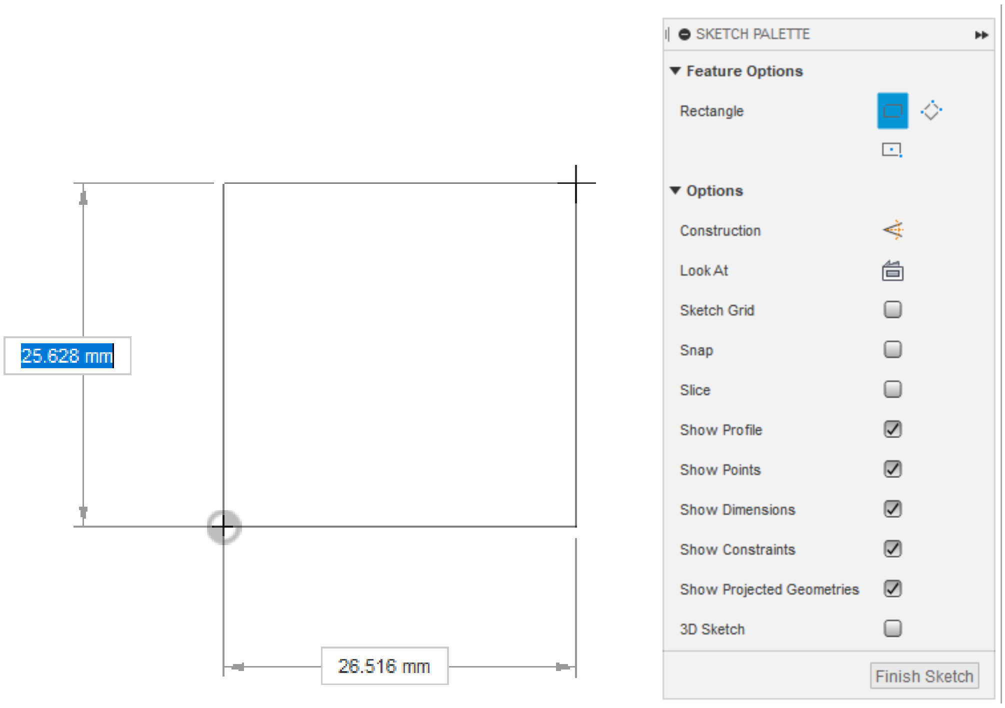

Figure 2.30

When defining a rectangle using three points, it is possible to specify its rotation angle in the sketch in addition to its dimensions (Figure 2.31).

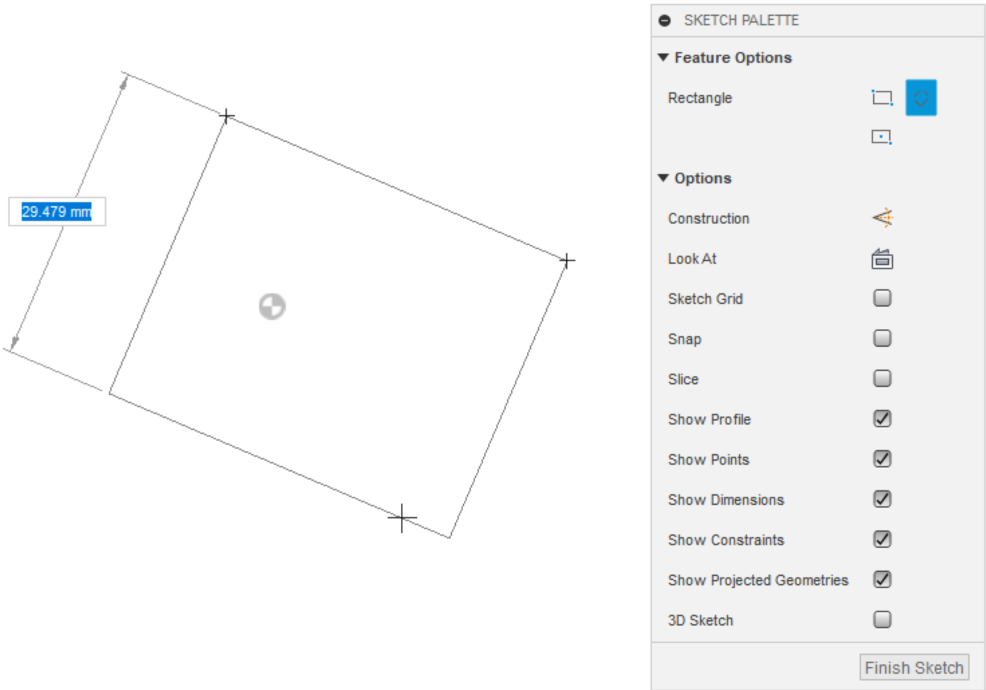

Figure 2.31

When it is necessary to position the centre of the rectangle on a precise point on the sketch, the rectangle can be drawn by setting the centre and specifying the diagonal length and angle (Figure 2.32).

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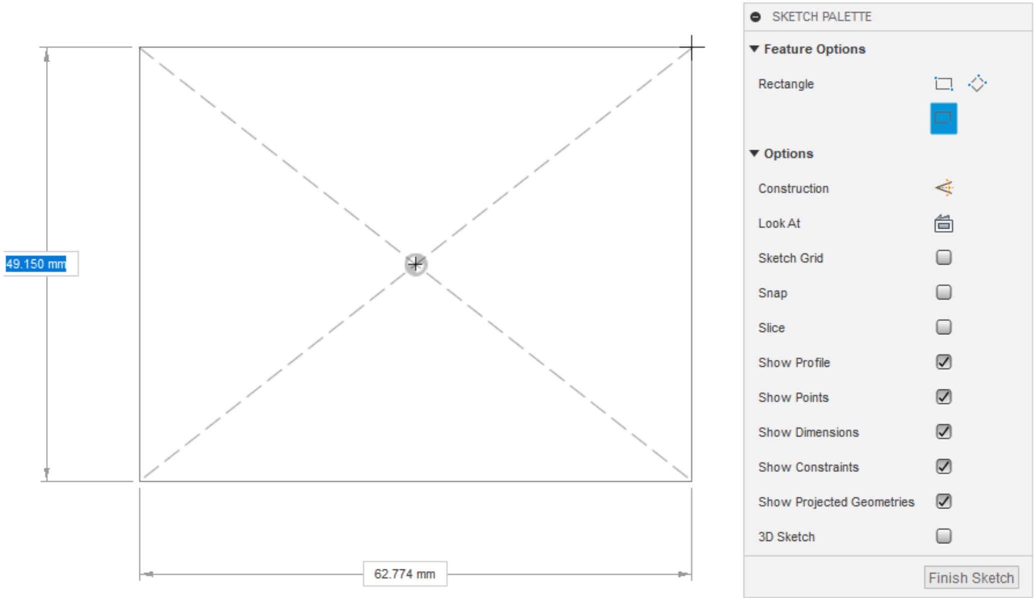

Figure 2.32

Drawing a polygon

Polygons can be drawn in the environment and defined as:

- drawn around a circle (Circumscribed Polygon)
- drawn inside a circle (Inscribed Polygon)
- defined by the length of their side.

By default, polygons are hexagonal in shape, but the number of sides can be changed when the polygon is created. To draw a polygon, we use the Polygon command and select the polygon definition type (Figure 2.33).

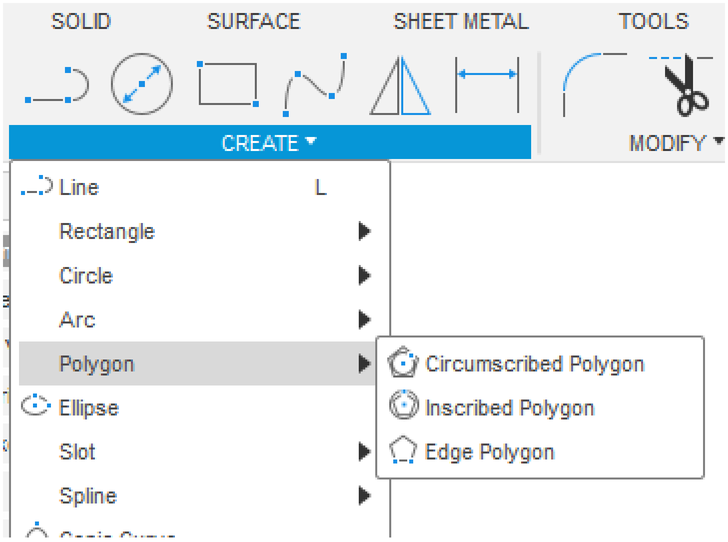

Figure 2.33

When creating polygons with sides of the same length, the radius of the circle around which the polygon is circumscribed (Figure 2.34 (a)) or in which the polygon is inscribed (Figure 2.34 (b)) can be set. Regardless of how we choose to define the polygon, it can be drawn by clicking with

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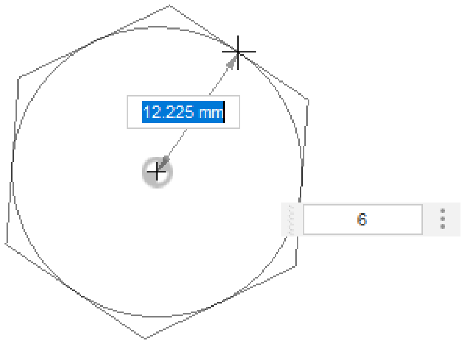
the mouse at the point that the centre of the polygon should be, and dragging the mouse to set its desired size. The number of sides can be specified in the box for setting the number of sides. By default the Polygon command draws a hexagon.



(a) (b)

Figure 2.34

Creating a polygon can also be accomplished by setting its side length by clicking and dragging with the mouse (Figure 2.35).

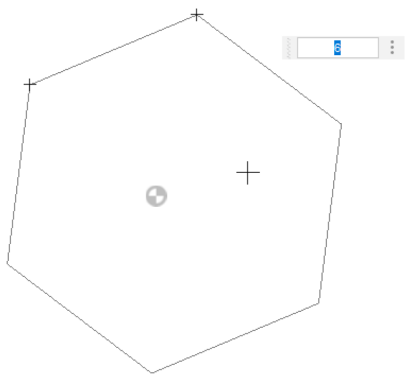


Figure 2.35

Regardless of how the polygon is created, only one size needs to be set to specify its size (Figure 36).

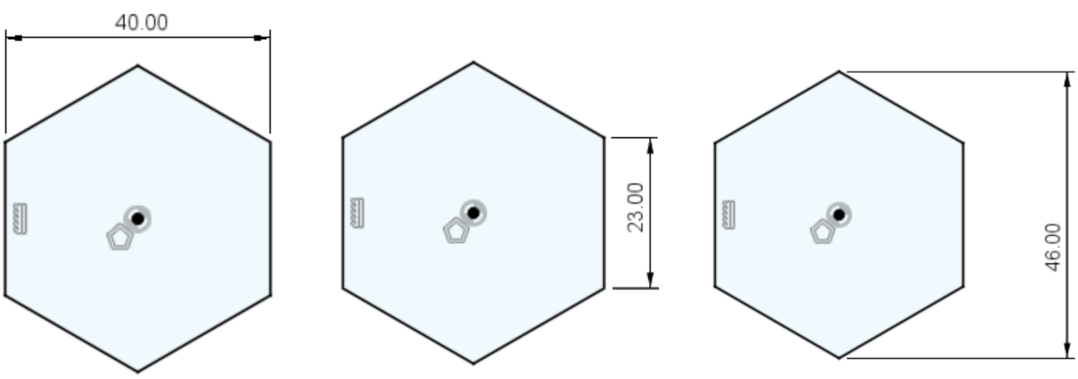


Figure 3.36

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Drawing slots

Slots are drawn with the Slot command. The command allows the slot shape to be specified by various means (Figure 2.37).

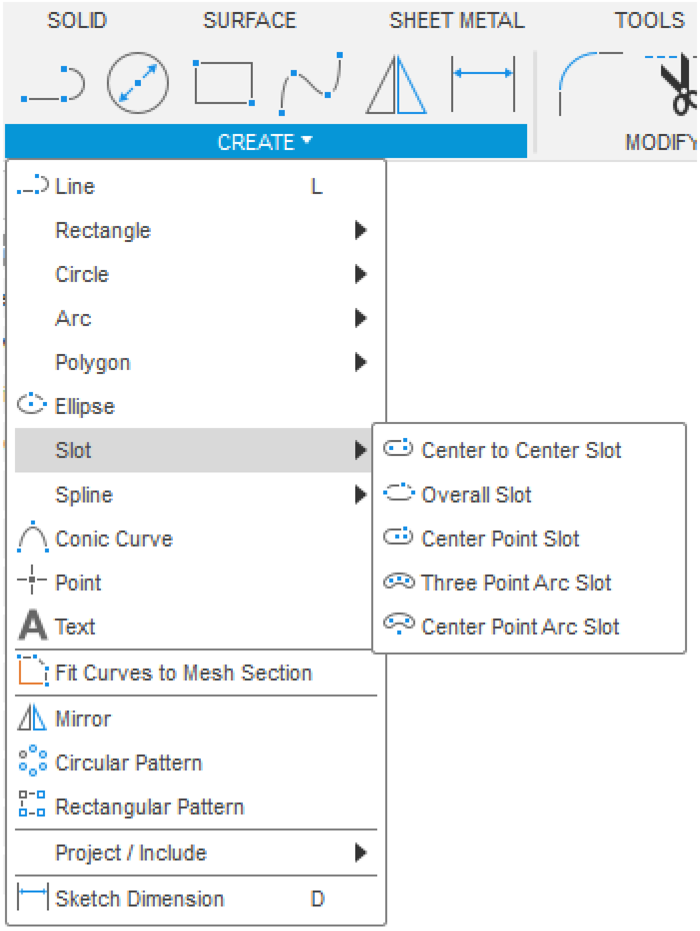

Figure 3.37

The most common way to define the slot is by the distance between the centre points of its ends (the curvatures) and its width (Figure 2.38).

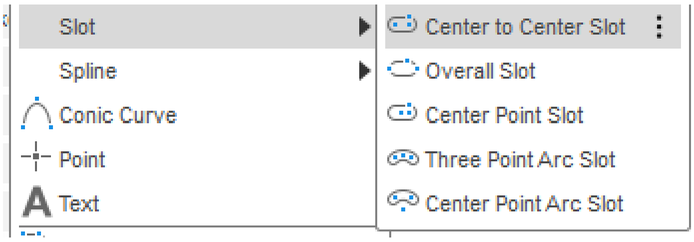

Figure 2.38

In this case, the centre points of the slot ends are indicated by clicking the mouse, and the slot width is set by dragging. The slot can be sized by defining the distance between the centre points and its width (Figure 2.39).

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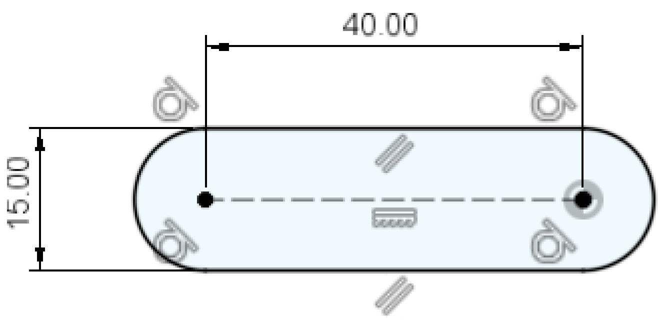

Figure 2.39

A slot can be drawn also by setting the final dimensions with mouse clicks or by setting a centre point, the distance between the slot centre point and the centre point of one of the ends, and the slot width (Figure 2.40).

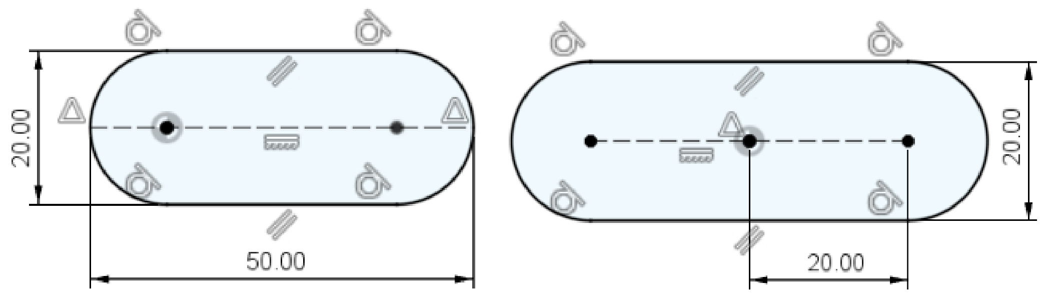

Figure 2.40

Fusion 360 allows us to draw slots that are arc-shaped. These slots are defined using the 3 points through which the central arc of the slots passes, or they can also be defined by specifying the centre of the arc radius and two points on it (Figure 2.41).

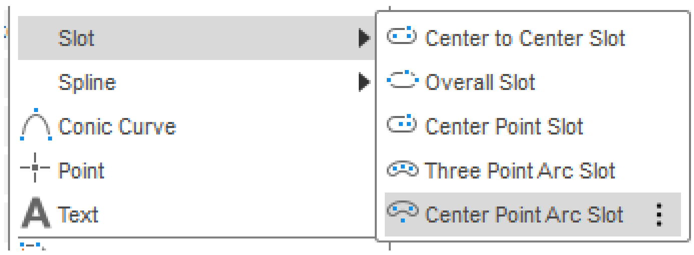

Figure 2.41

When drawing an arc-shaped slot, its dimensioning can be done in a similar way to that shown in Figure 2.42. It can be seen that, in addition to the standard linear dimensions, the environment allows radial and angular dimensioning to be used.

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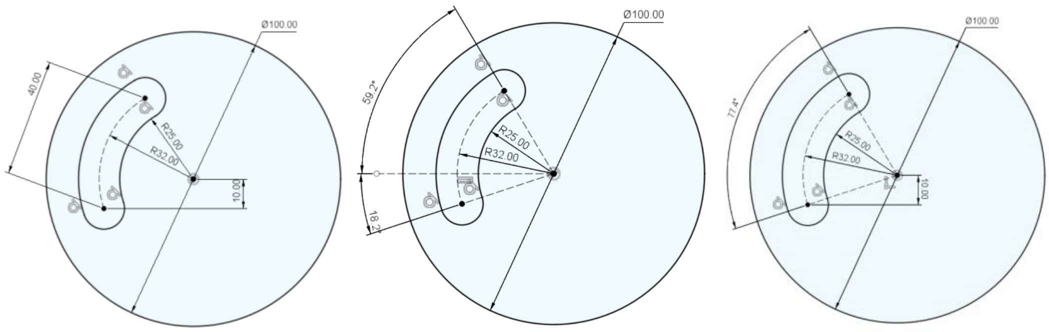

Figure 2.42

Spline sketching

In addition to straight lines, arcs and geometric objects, Fusion 360 also allows us to draw curves of arbitrary shape. They are defined using reference points. We use the Spline command to draw the curves controlled by these reference points. The Spline command allows two types of splines to be drawn (Figure 2.43) – splines passing through the reference points (Fit Point Spline) and splines not passing through the reference points but controlled by them (Control Point Spline).

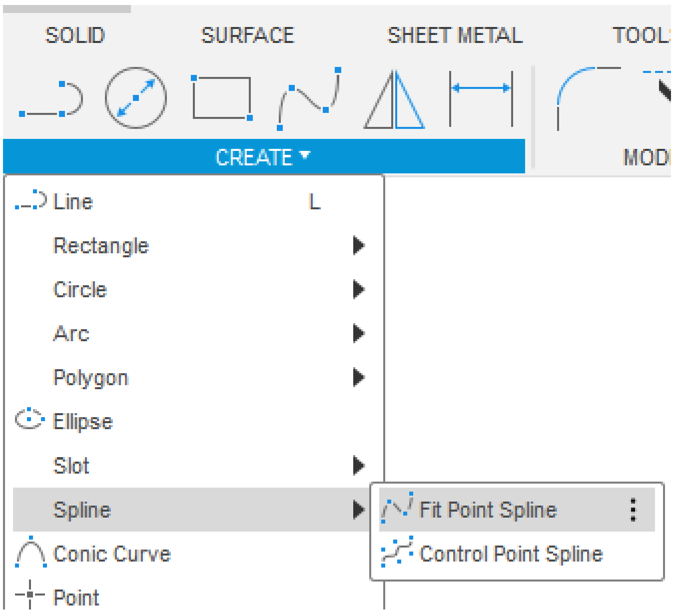

Figure 2.43

Drawing splines passing through the reference points

Drawing this type of spline is done by successively clicking the mouse to select the points through which the spline curve should pass (Figure 2.44). To terminate the drawing of the spline, we must click the ✅ icon that appears after a new point has been defined.

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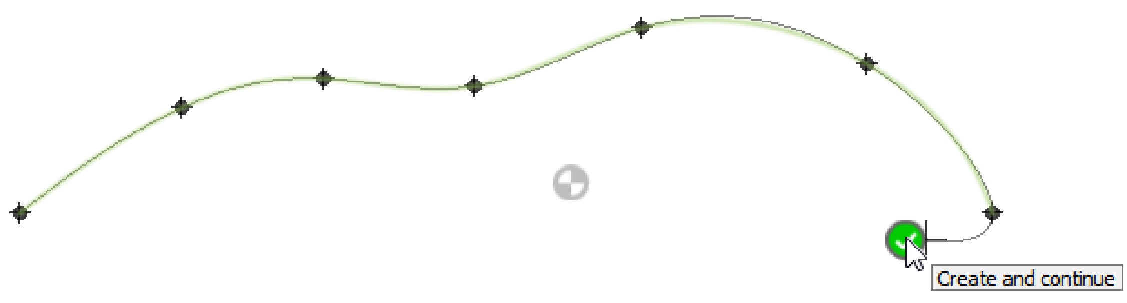

Figure 2.44

In the created spline curve it is possible to adjust the degree of the curves by means of control lines to each reference point (Figure 2.45).

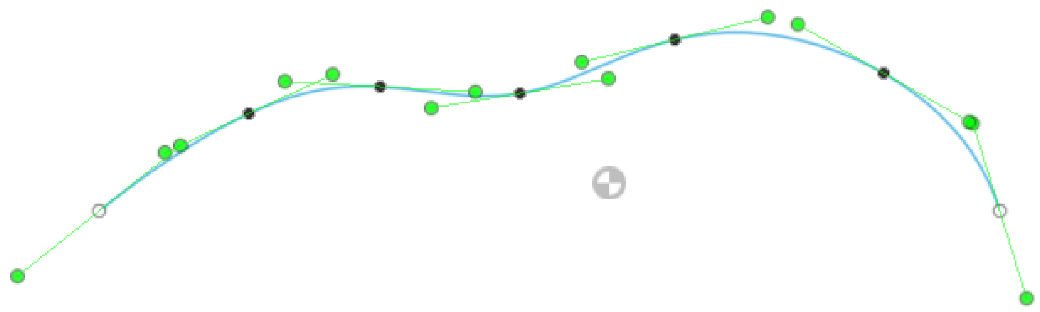

Figure 2.45

Using the control lines it is possible to edit the spline curve so that it takes the desired shape. The curve in Figure 2.46 has the same position of the reference points as Figure 2.45, but with modified connecting curves between the reference points.

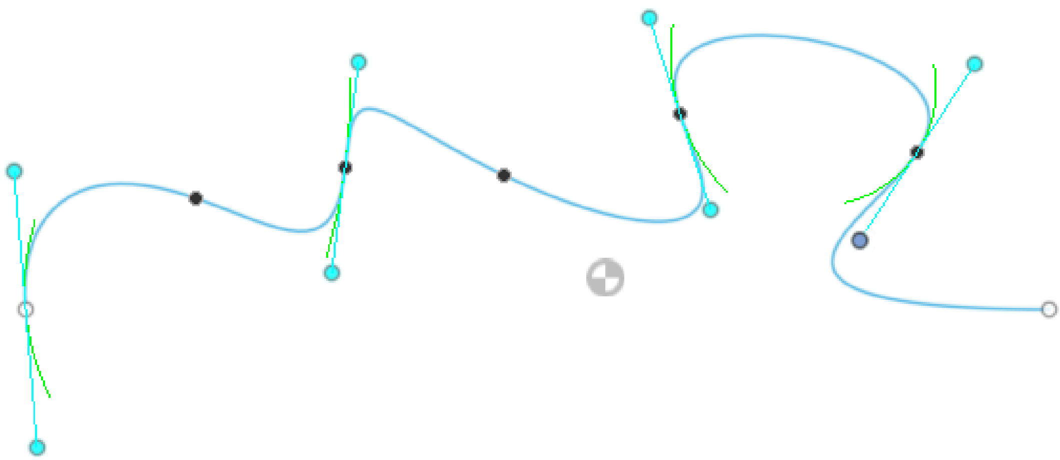

Figure 2.46

The degree of the curvature can be graphically illustrated by using the so-called Comb. The greater the curvature at a particular reference point, the greater the graphical representation of the comb (Figure 2.47).

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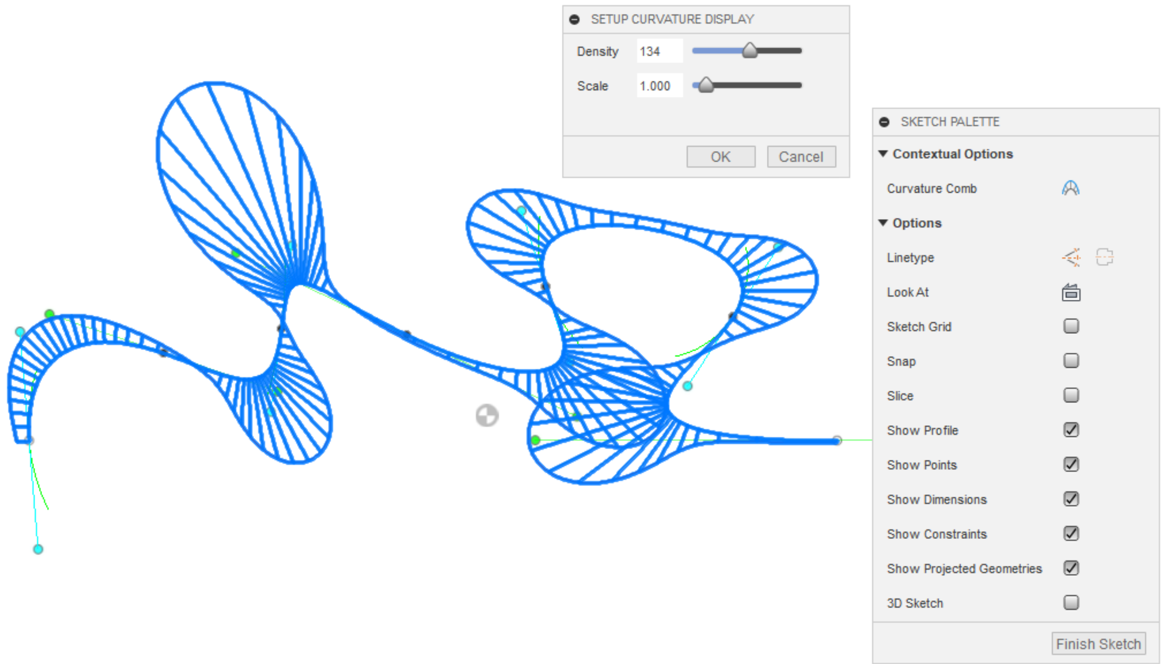

Figure 2.47

Drawing splines not passing through the reference points

When drawing this type of spline, the reference points define the shape of the curve only approximately (Figure 2.48) since the curve itself does not pass through the reference points. The shape of the curve can later be fine-tuned by changing the position of the reference points by dragging the mouse until the desired shape of the curve is obtained.

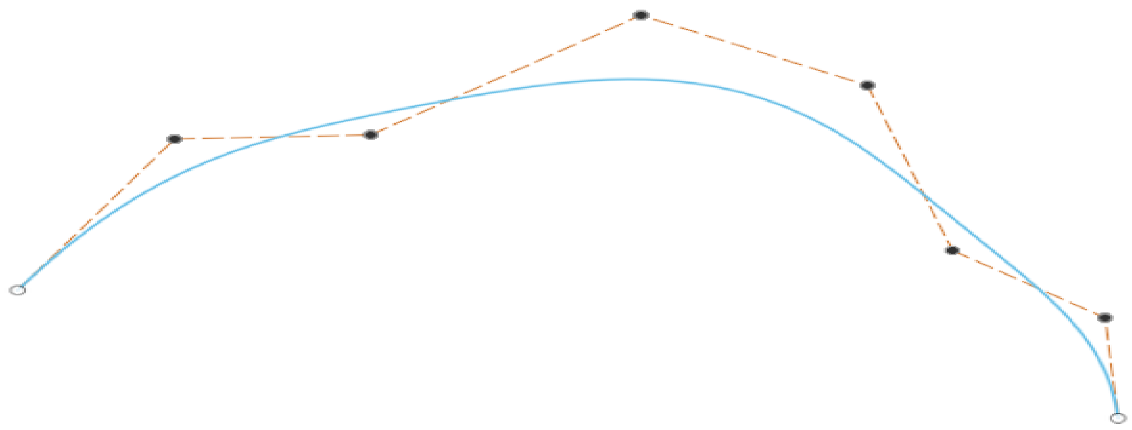

Figure 2.48

In the Sketch Palette the drawing of the curve can be specified by a third- or fifth-order polynomial (Figure 2.49). The fifth-order polynomial can be used to draw curves that are more complex in shape, while using the third-order polynomial makes curve changes smoother.

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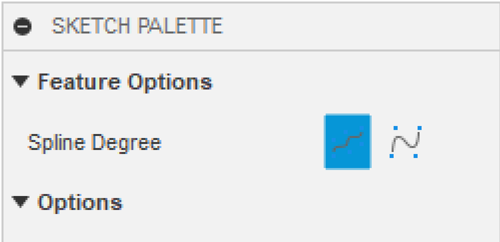

Figure 2.49

The application of splines can be demonstrated with the following example. Let us place an image of a vase in the work area using the Canvas command from the toolbar (Figure 2.50). Based on the image, a 3D model of the vase will be built based on a sketch containing splines passing through the anchor points.

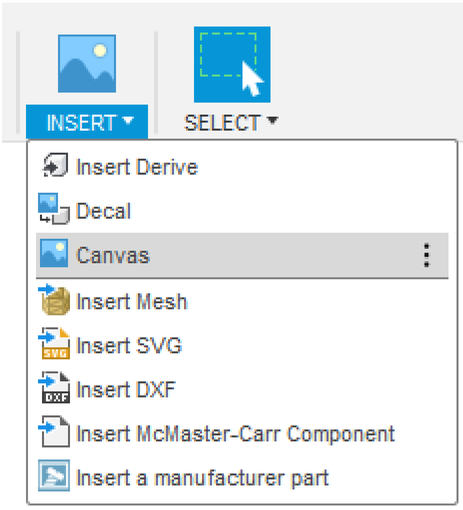

Figure 2.50

The placement of a photo or drawing with the Canvas command is done on a surface. In this case we will select one of the main construction surfaces of the 3D model. The image is selected from the computer's file system (Figure 2.51).

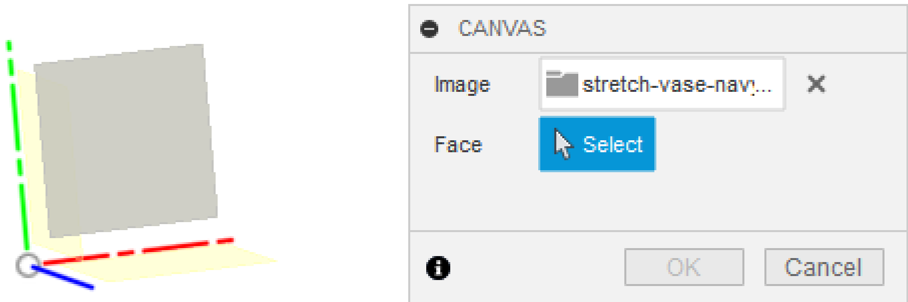

Figure 2.51

The orientation of the image should be further adjusted, and scaling should be performed if necessary (Figure 2.52).

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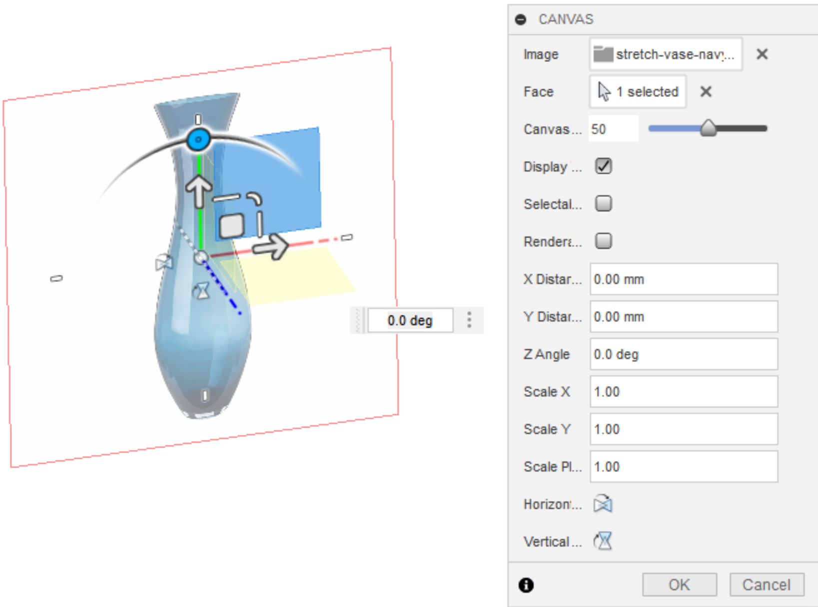
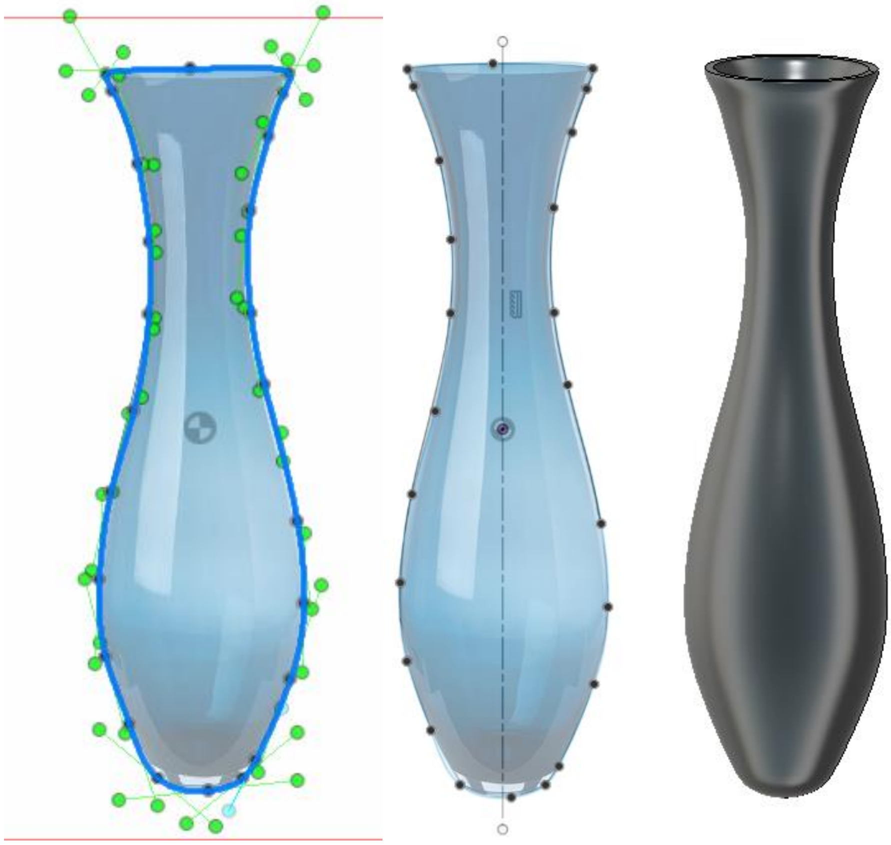


Figure 2.52

Using splines we draw the outline of the vase (Figure 2.53(a)), add a centreline (Figure 2.53(b)) and, using the 3D modelling commands (which will be introduced in the next chapter), the final model of the vase is built (Figure 2.53(c)).



(a) (b) (c)

Figure 2.53

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Defining a point in the sketch

In the sketching process, there is an option to define a precise point in the sketch (Figure2.54). Defined points can be connected to create a figure with a specific geometry or they can be used for subsequent construction of axes and planes.

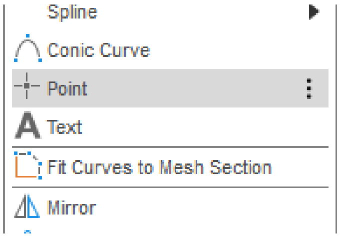

Figure 2.54

To demonstrate the use of points, let us place two points in a precise location on an existing geometric figure (Figure 2.55). The points will then be connected to one vertex of the rectangle. In this way, we obtain triangles with precisely defined dimensions.

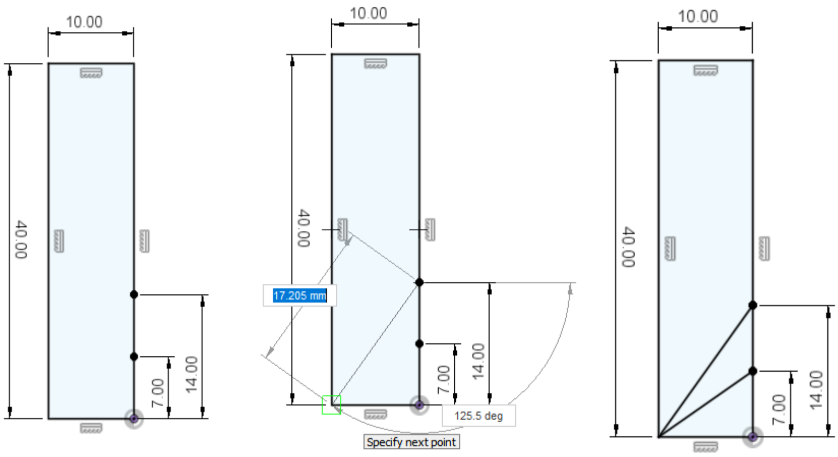

Figure 2.55

Let us create a new sketch while keeping the preview of the old one by activating the  icon next to the sketch in the browser (Figure 2.56).

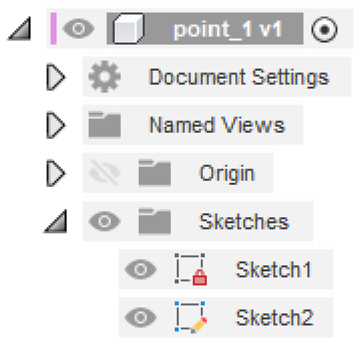

Figure 2.56

We create the new sketch in a plane perpendicular to the plane of the previous sketch. In the new sketch, we can use the set points as reference points to draw a new geometry (Figure 2.57).

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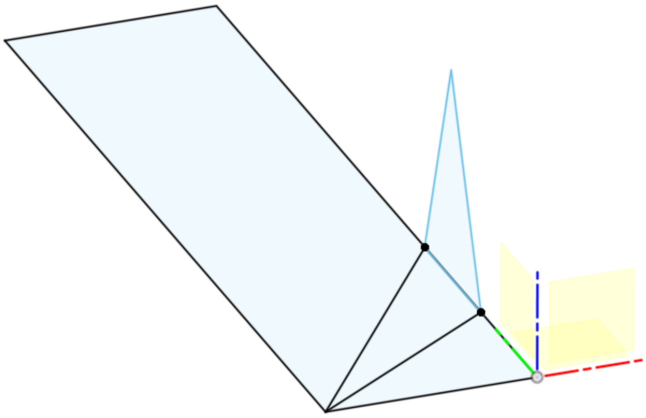

Figure 2.57

Inserting text

Fusion 360 allows text to be inserted into sketches by using the Text command (Figure 2.58). This text can subsequently be converted into 3D information containing letters and numbers that is added or inserted into the 3D objects.

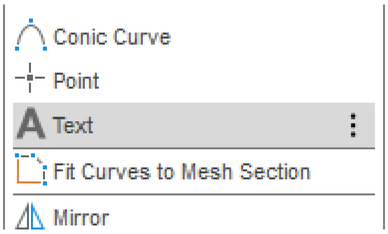

Figure 2.58

Text can be inserted into the drawing as standard text or as text following a defined line (Figure 2.59).

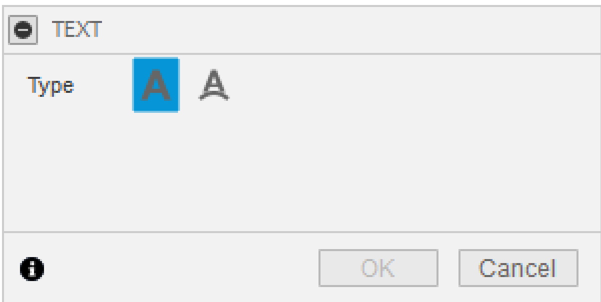

Figure 2.59

When we have chosen to insert plain text, we use the mouse to select the position of the text in the working field of the sketch, and with the additional menu we set its parameters (Figure 2.60).

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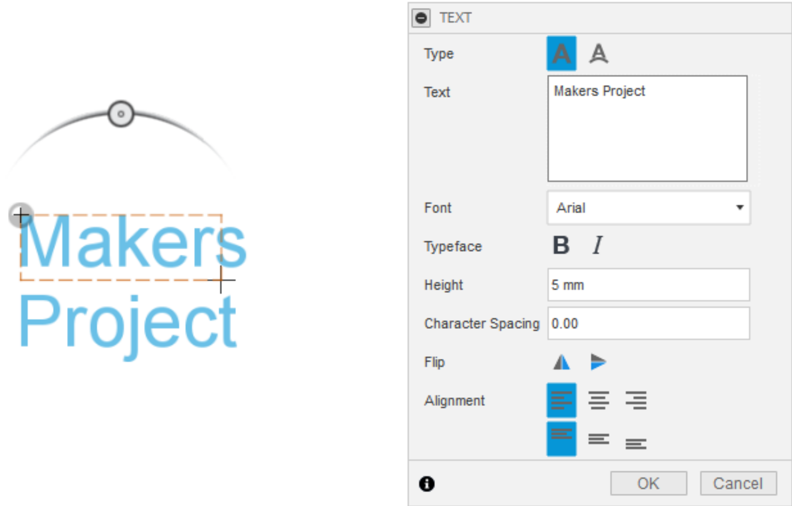

Figure 2.60

We can define text parameters such as font type and size, the spacing between letters, the alignment, and the ability to mirror the text. Additionally, the text box can be rotated to a specific angle in the work area (Figure 2.61).

Figure 2.61

When it is necessary for the text to follow a certain line, we must have the line drawn in advance in the sketch, and set it as a path for the text to follow. A straight line, a curved line or a geometrical figure can be used as a path. The line to follow can be selected using the mouse (Figure 2.62).

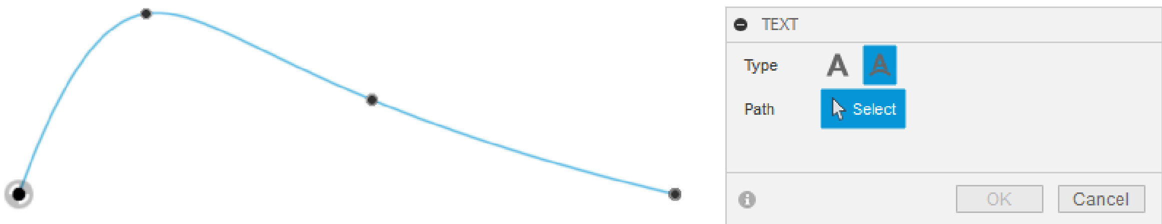

Figure 2.62

The entered text is positioned along the specified path, and we are able to set additional parameters from the auxiliary menu (Figure 2.63).

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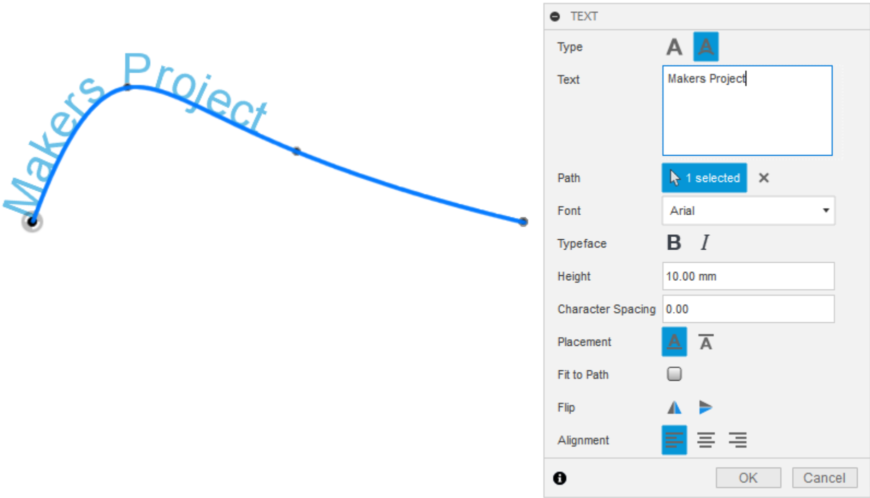

Figure 2.63

A useful option of the environment is the possibility to position the text evenly along the length of the selected path by checking the Fit to Path option (Figure 2.64).

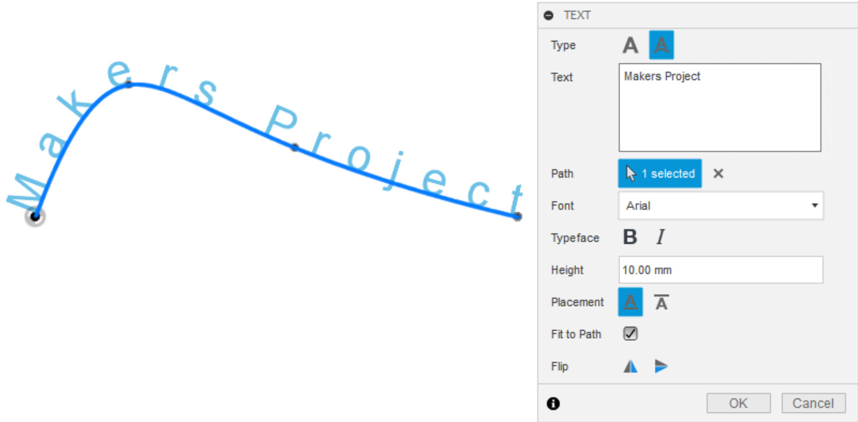

Figure 2.64

Multiplication of objects in a sketch

In sketches it is often necessary to draw objects of the same type located at a certain fixed distance from each other. To speed up the process, object multiplication commands are used. They make it possible to multiply an object a certain number of times in the sketch. Multiplication commands include Mirror, Circular Pattern, and Rectangular Pattern.

Mirroring objects

Consider the following sketch (Figure 2.65), on which we will add additional objects.

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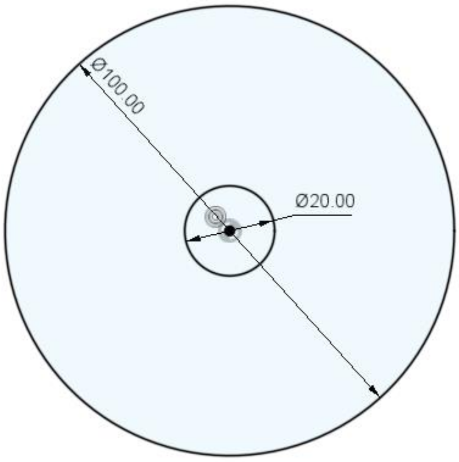

Figure 2.65

We will create a slot on the sketch and size it as shown in Figure 2.66.

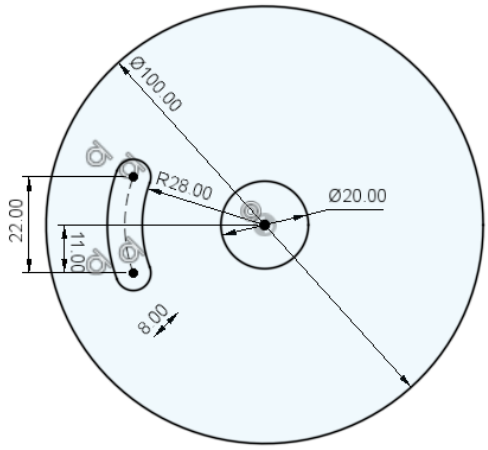

Figure 2.66

In order to mirror the created slot, it is necessary to specify a line around which the mirroring will take place. Figure 2.67 shows the creation of a vertical construction line.

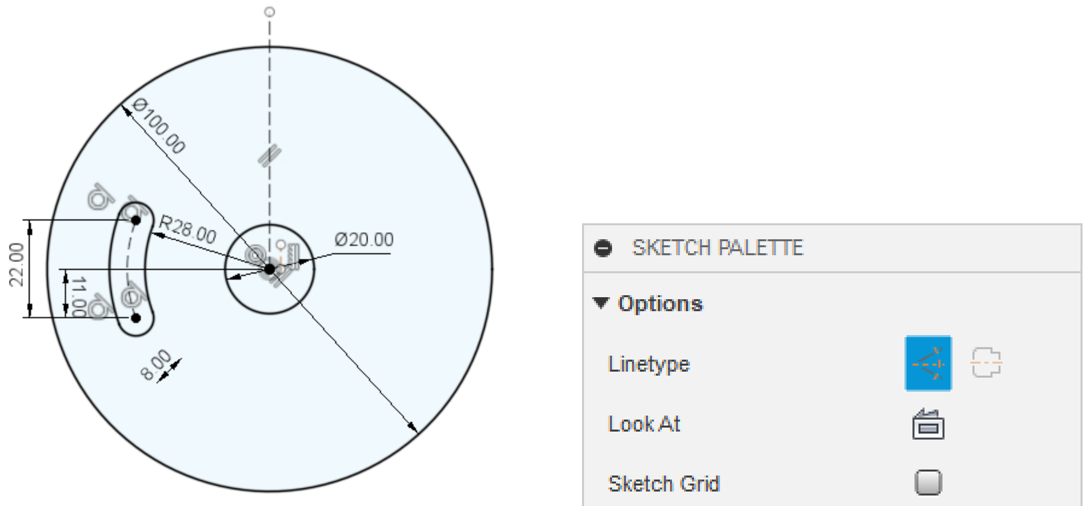

Figure 2.67

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Mirroring of the object is performed with the Mirror command from the toolbar or from the drop-down command menu (Figure 2.68).

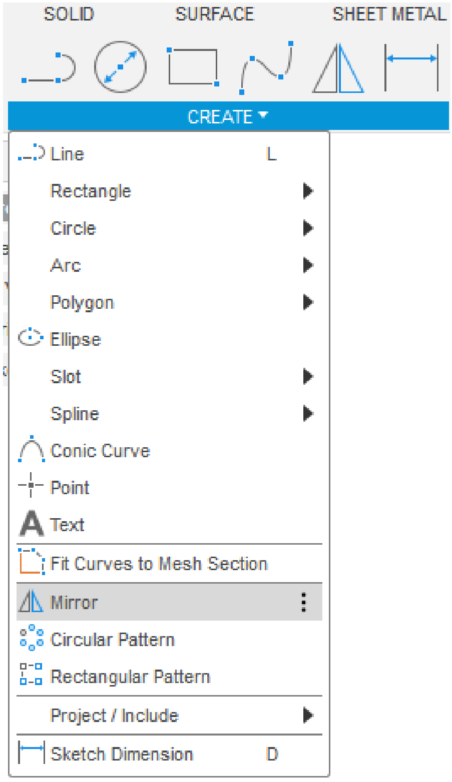

Figure 2.68

Once the command has been activated, it is necessary to use the mouse to specify the objects (or the sketch geometry elements) that will be mirrored, as well as the mirror line (Figure 2.69). The mirror line cannot overlap with any of the sketch geometry elements selected to be mirrored.

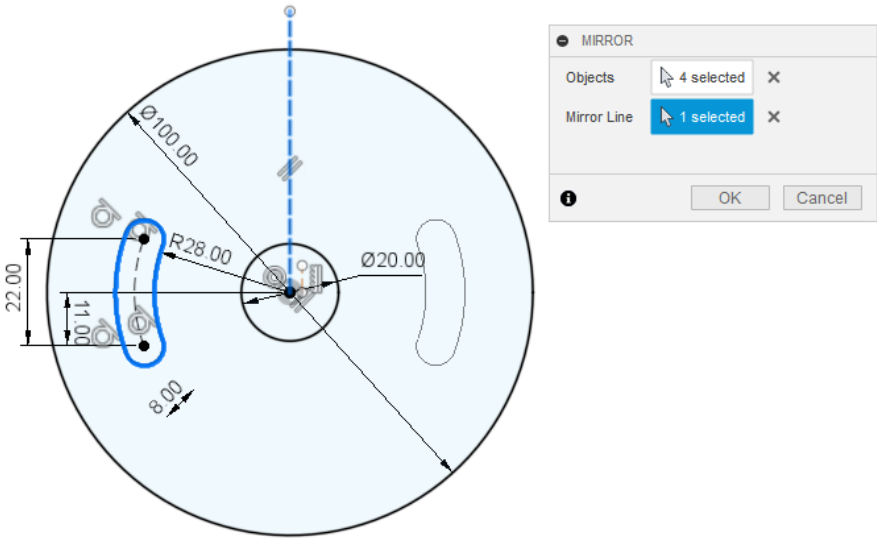

Figure 2.69

The result of the operation is presented in Figure 2.70. The new object is at the same distance from the mirror line as the mirrored object.

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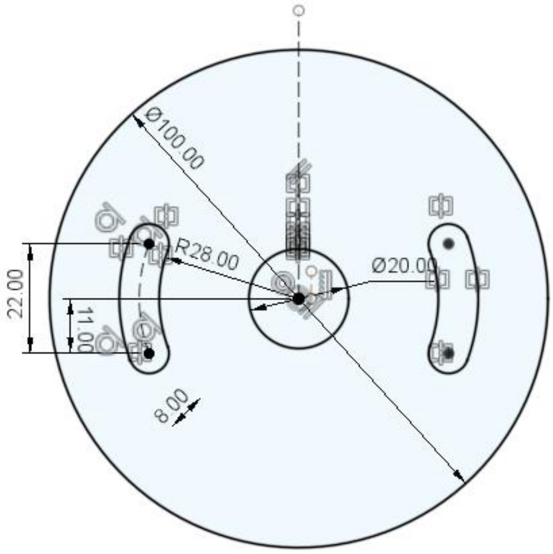

Figure 2.70

Circular multiplication of objects

Figure 2.71 shows the corresponding figure, where the small circle at its periphery will be used for mirroring.

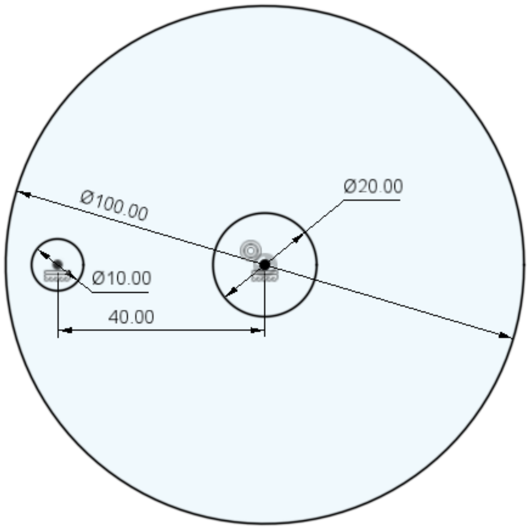

Figure 2.71

The Circular Pattern command (Figure 2.72) is used to enact circular object multiplication.

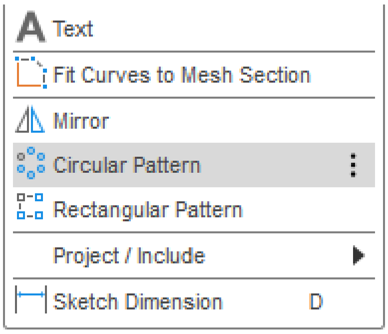

Figure 2.72

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After selecting the command, it is necessary to use the mouse to select the objects to be patterned (in this case, the circle in the periphery of the sketch) and also to specify a centre point around which the rotation will be performed (Figure 2.73).

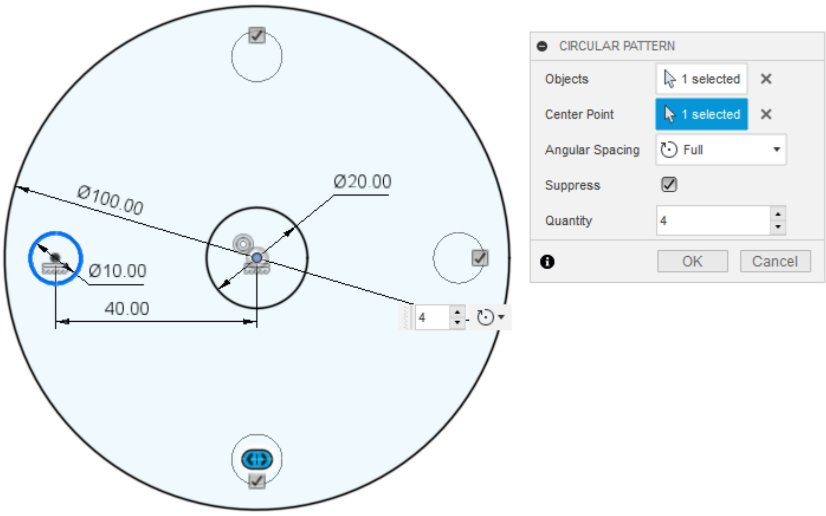

Figure 2.73

The command asks to specify the final number of objects that will appear in the sketch. If the quantity of objects is set at 4, the final result of the circular pattern will be as shown in Figure 2.74.

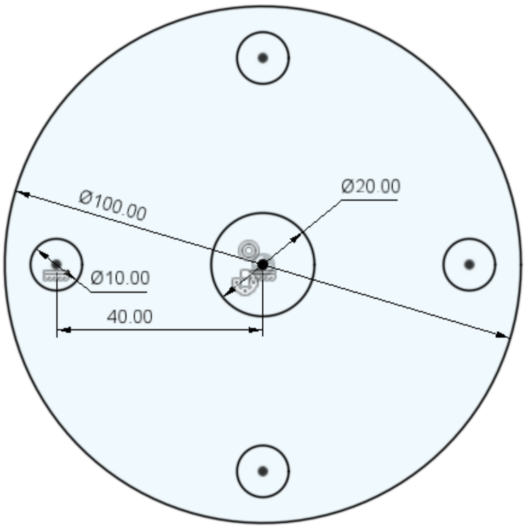

Figure 2.74

Rectangular multiplication of objects

In rectangular multiplication, there is an option to multiply a selected object vertically and horizontally (or in the direction specified by a straight line) at corresponding distance increments between the new objects. Figure 2.75 shows a figure, containing a circle at one end. The circle will be used in the pattern.

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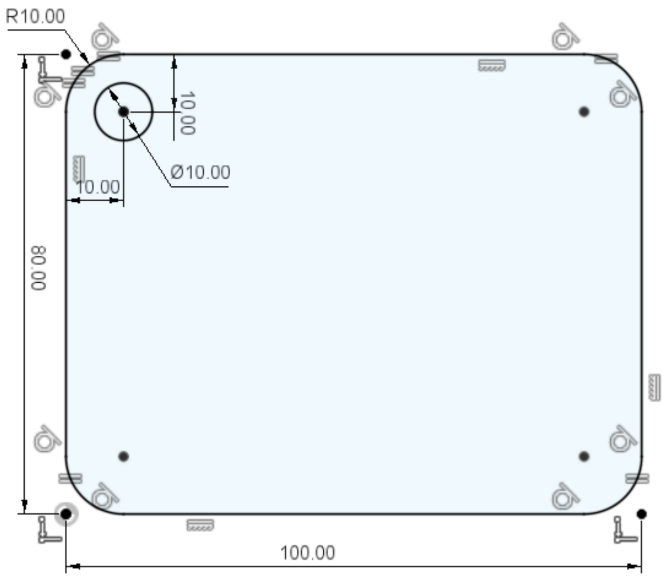

Figure 2.75

The command that should be used in this case is the Rectangular Pattern (Figure 2.76).

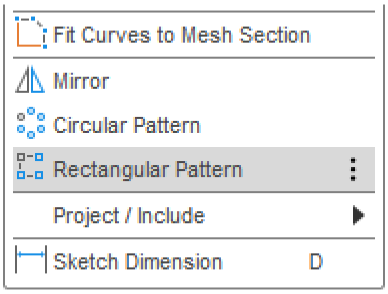

Figure 2.76

The command menu allows the multiplication to be performed for a specific distance or with a certain space between the objects (Figure 2.77(a)), and the multiplication can be one-sided or symmetrical vis-à-vis the axes (Figure 2.77(b)).

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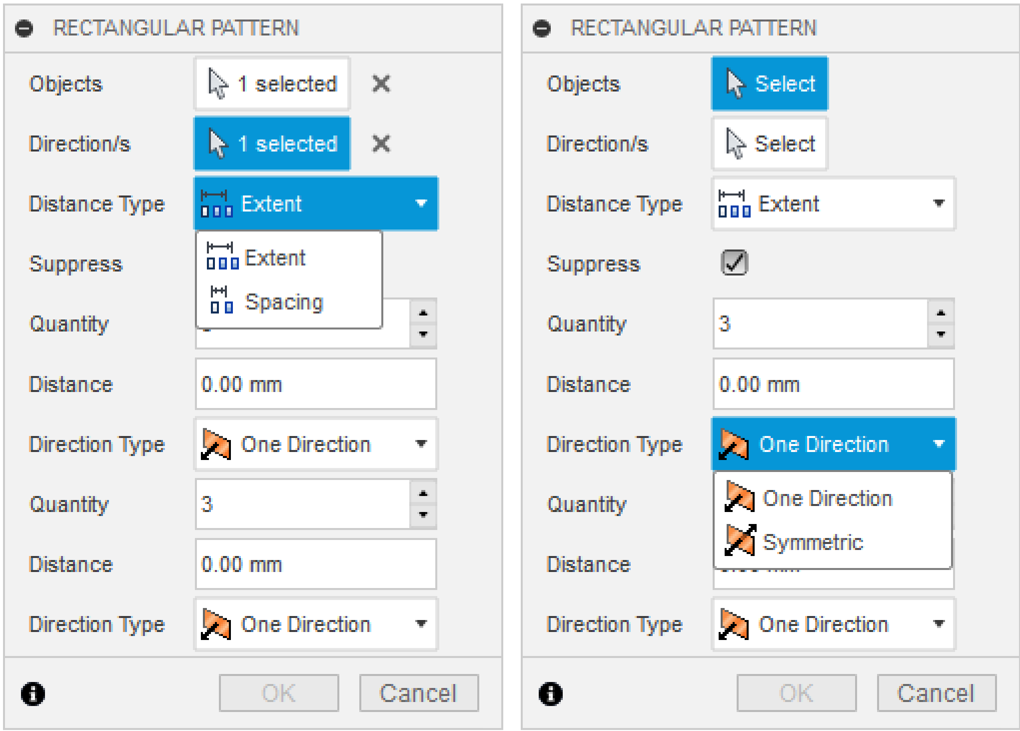

(a) (b)

Figure 2.77

For this particular case, let the multiplication take place according to the settings given in Figure 2.78. This results in 20 sketch objects, grouped into 4 rows and 5 columns.

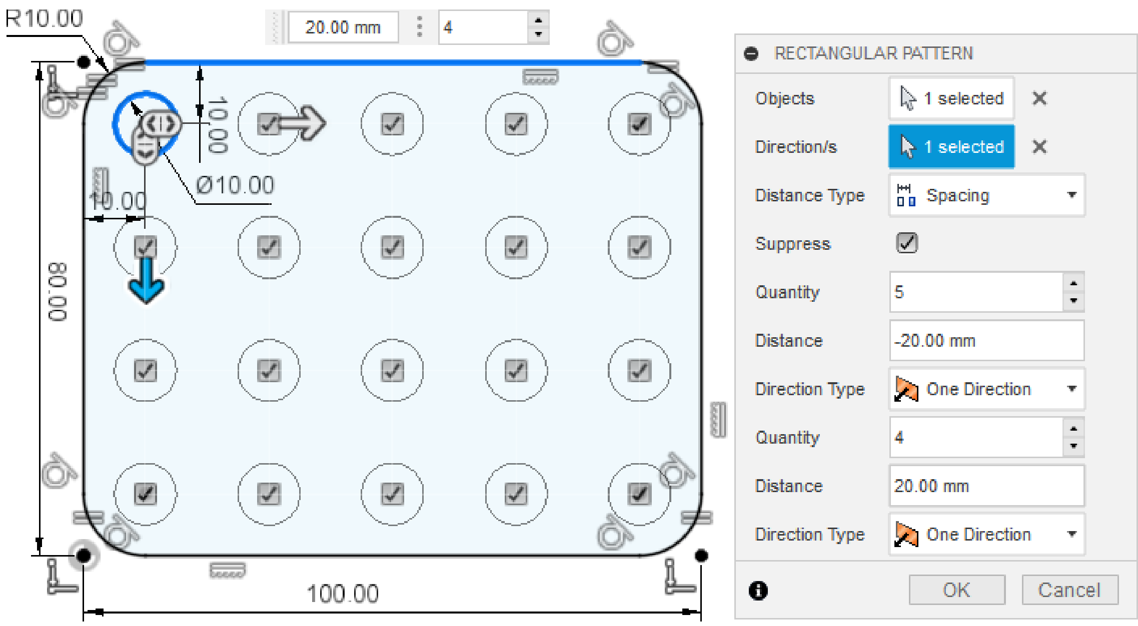

Figure 2.78

The rectangular multiplication makes it possible to create a large number of objects on the sketch very quickly and to generate complex 3D shapes (Figure 2.79). It is also possible to use checkboxes to exclude those objects that we do not want to appear on the sketch.

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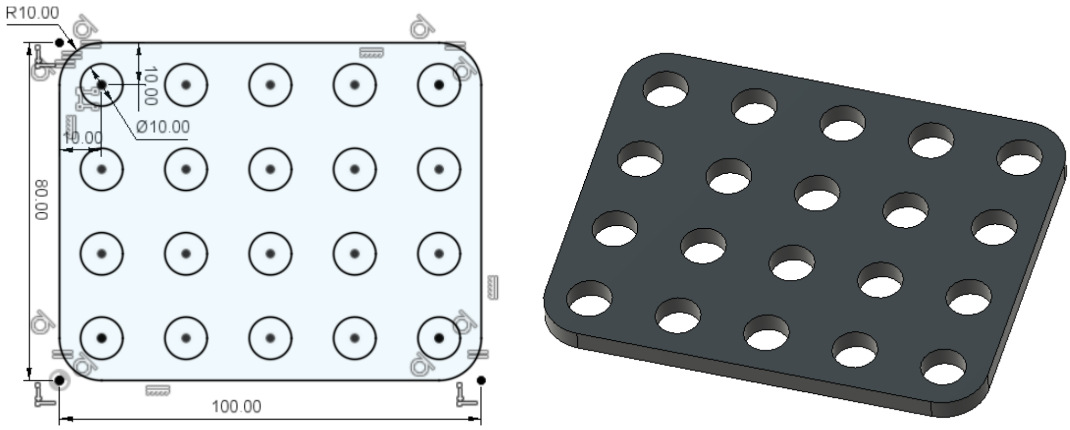

Figure 2.79

Projecting geometry from existing 3D objects onto the active sketch plane

To create sketches in Fusion 360, we can also use the geometry of existing objects, and project this geometry onto the sketch plane.

Figure 2.80 shows a 3D object and a construction plane on which a sketch will be created by projecting this 3D object.

Figure 2.80

A set of commands (Figure 2.81) is used to project the geometry of 3D objects, of which the Project command is the most widely used. Apart from it, with other commands of the same group it is possible to project the intersection of the selected object and the plane of the active sketch, to project onto a surface, etc.

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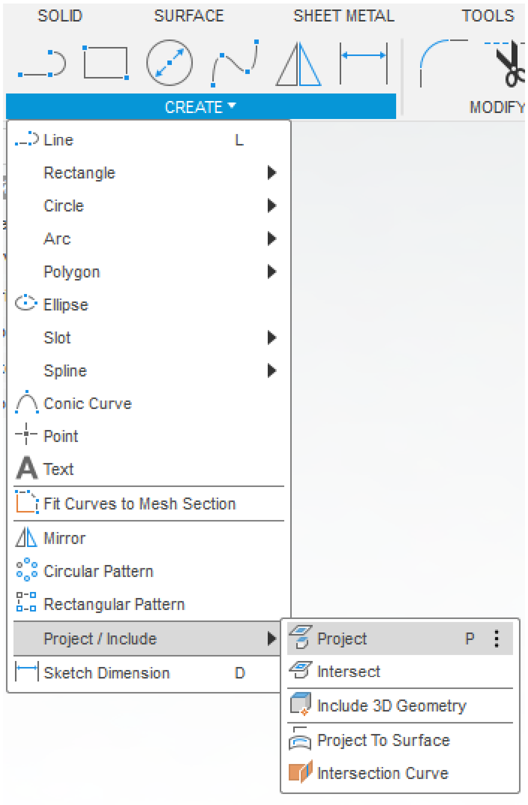

Figure 2.81

If it is necessary to select hidden edges, in the Navigation Bar we can set the object's visualization style as Wireframe (Figure 2.82). In this way we will make all edges of the object visible for projection.

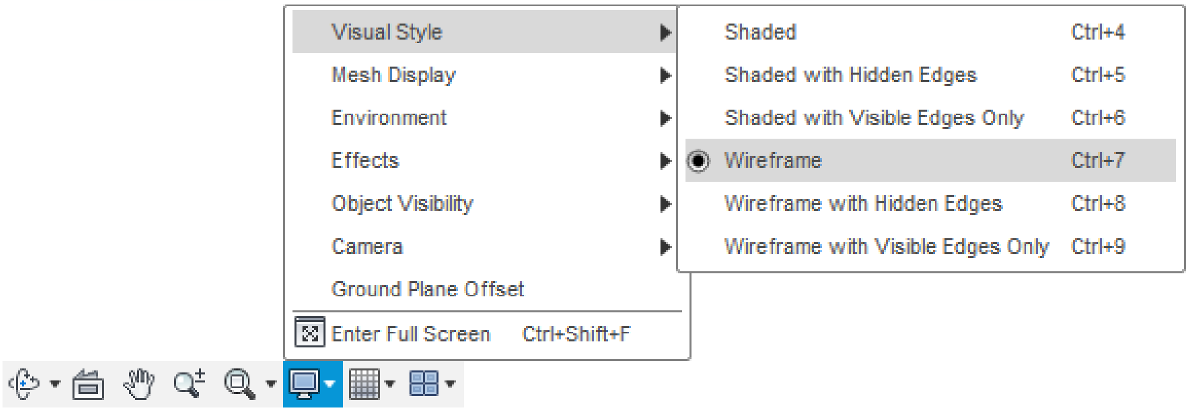

Figure 2.82

When projecting, we can choose to project only certain edges of the object or its entire body (Figure2.83).

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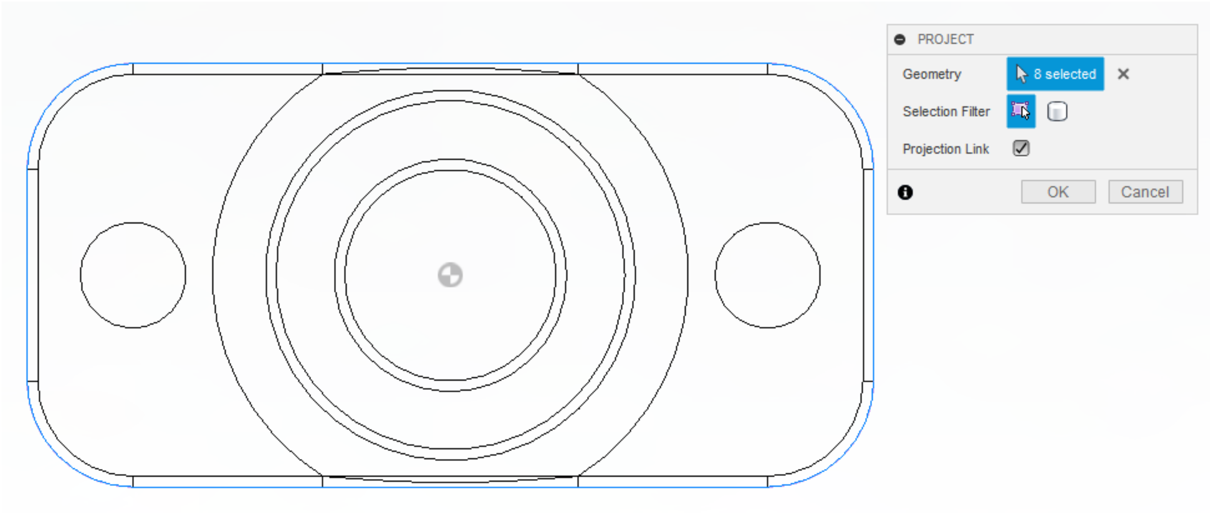

Figure 2.83

In this example, we choose to project the outermost edge of the object. The selection is made by selecting with the mouse the lines that make up the edge (Figure 2.84).

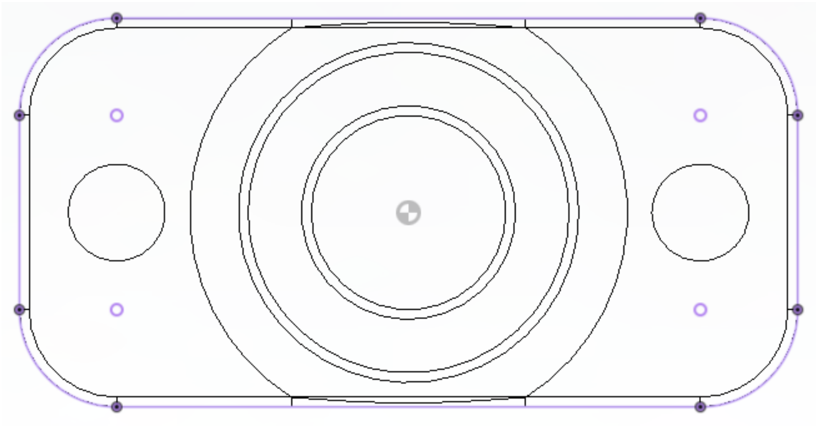

Figure 2.84

The projected lines are displayed on the sketch plane and can be used in subsequent steps in the modelling process (Figure 2.85).

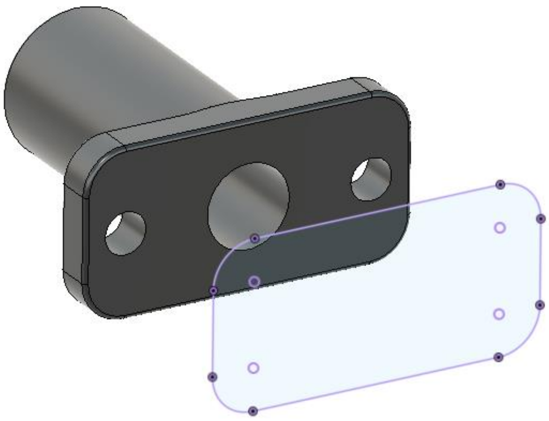

Figure 2.85

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If we want to make a projection of the object's body contours, then the body projection parameter should be ticked in the Project command menu (Figure 2.86).

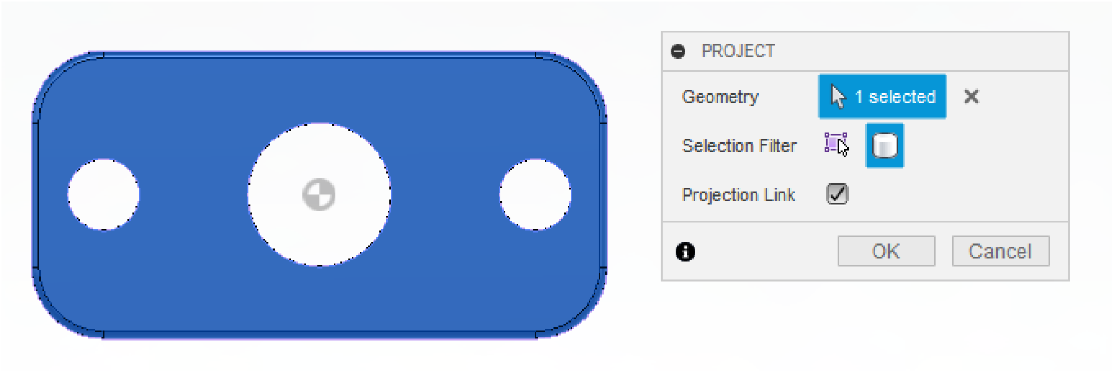

Figure 2.86

The result projecting the whole body of the object is presented in Figure 2.87.

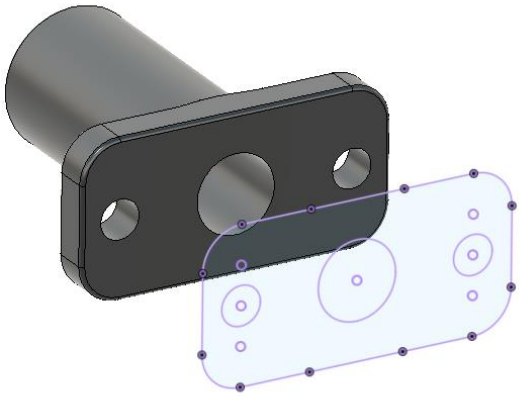

Figure 2.87

Modifying and editing the sketches

It is often necessary to modify some sketch parameters after the creation of the sketch. The commands for sketch modification are found in the Modify menu (Figure 2.88).

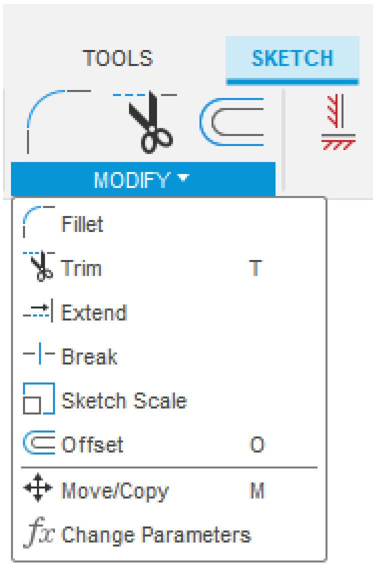

Figure 2.88

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The commands included in this menu are related to creating curvatures (rounding corners), cutting lines, extending lines, breaking lines, scaling objects, creating offset lines relative to an outline, and moving of graphic primitives.

Creating curvatures

In the drop-down menu, the Fillet command creates curvatures between two intersecting lines (Figure 2.89). The same command is also available from the toolbar under the following graphical notation .

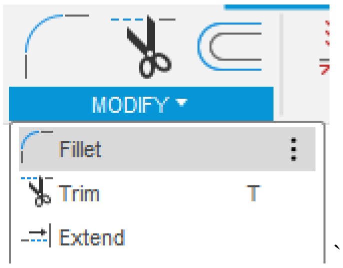

Figure 2.89

To demonstrate how the command works, let us look at the sketch in Figure 2.90. There is a rectangle drawn whose edges we will round by using the aforementioned command.

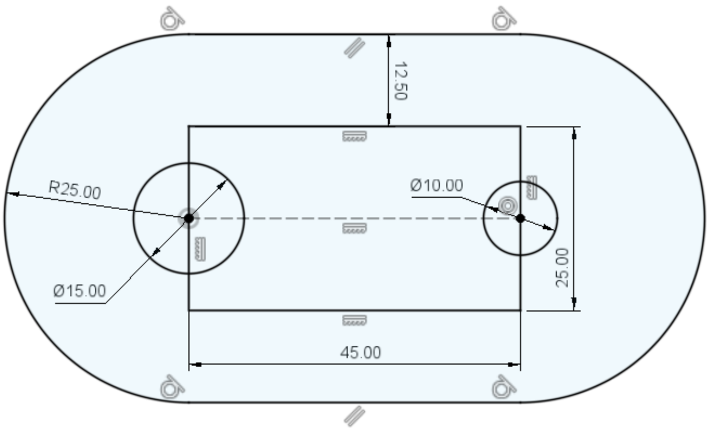

Figure 2.90

Rounding is done after activating the command, by successively selecting two intersecting lines with the mouse. In a dialog box we are invited to the set the value of the curvature. Once the command is activated, it remains active and new curvatures can be implemented between adjacent intersecting lines. In this way, the four edges of the rectangle can be rounded quickly (Figure 2.91).

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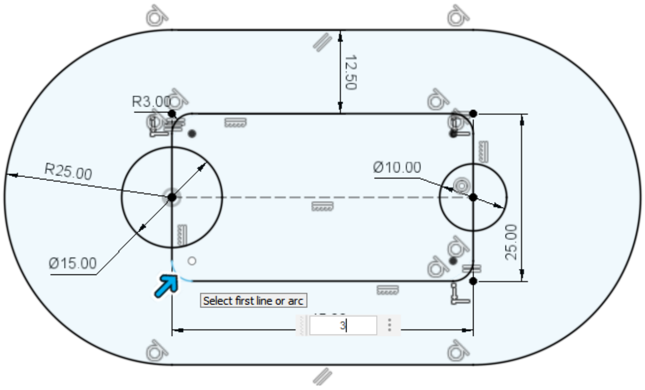

Figure 2.91

Cutting lines

By means of the Trim command  from the toolbar or from the drop-down menu (Figure 2.92) we can remove parts of the sketch geometry that are not necessary or would make it difficult to create a 3D model.

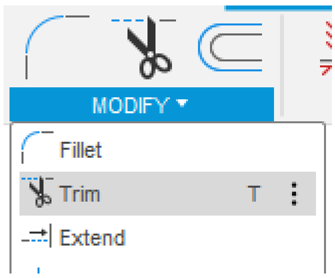

Figure 2.92

In the sketch in Figure 2.91, let us use the Trim command to remove the unnecessary parts. We need to successively select them by placing the mouse cursor over them and then clicking (Figure 2.93).

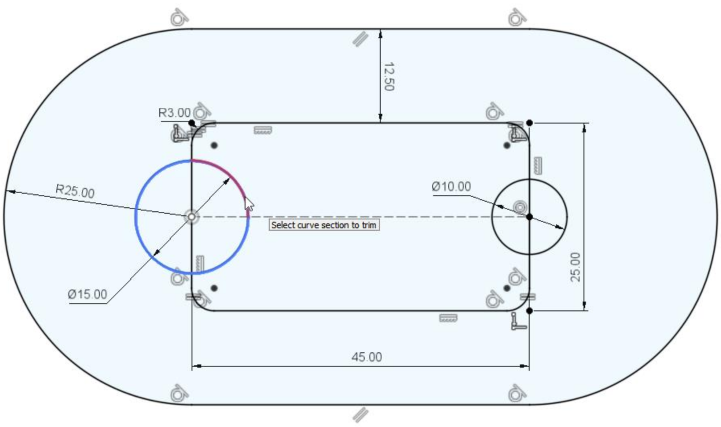

Figure 2.93

After removing the unnecessary lines, the 2D sketch object takes on the form shown in Figure 2.94. Often, when removing lines, the specified dimensions are also erased and it is therefore necessary to then resize those parts of the sketch that have lost their definition and their lines have been coloured in blue.

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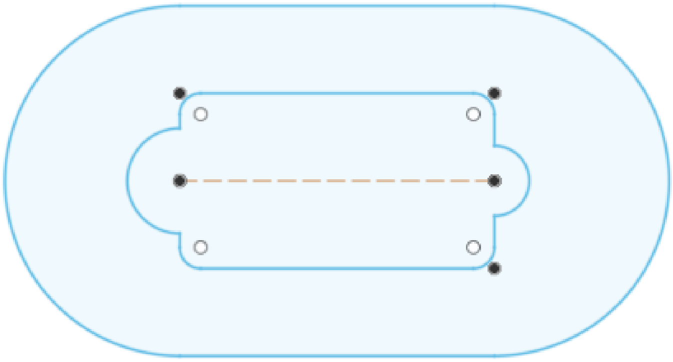

Figure 2.94

Extending lines

By using the Extend command (Figure 2.95), lines can be automatically extended to the next line that they intersect with in the two-dimensional space.

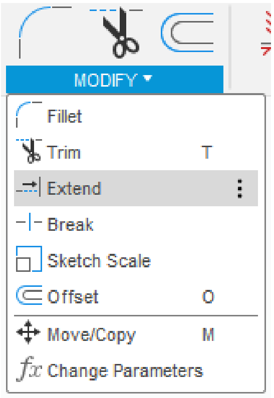

Figure 2.95

To demonstrate how the Extend command works, let us look at the sketch in Figure 2.96. It can be seen that the contour is not closed, i.e. there is no surface that can be used for extrusion in 3D modelling.

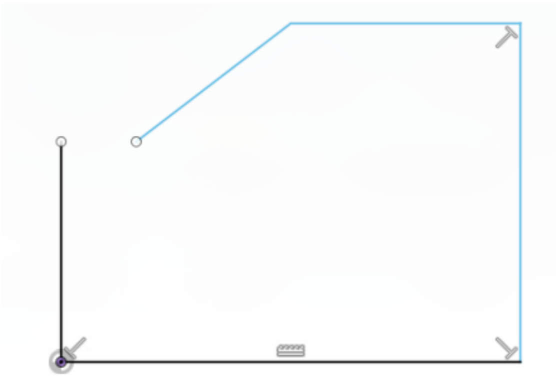

Figure 2.96

To close the outline, we activate the Extend command and point-click on one of the lines (Figure 2.97 (a)). This line will then be extended to intersect another line in the sketch. As a consequence, the contour will be closed and a surface will be obtained (Figure 2.97 (b)).

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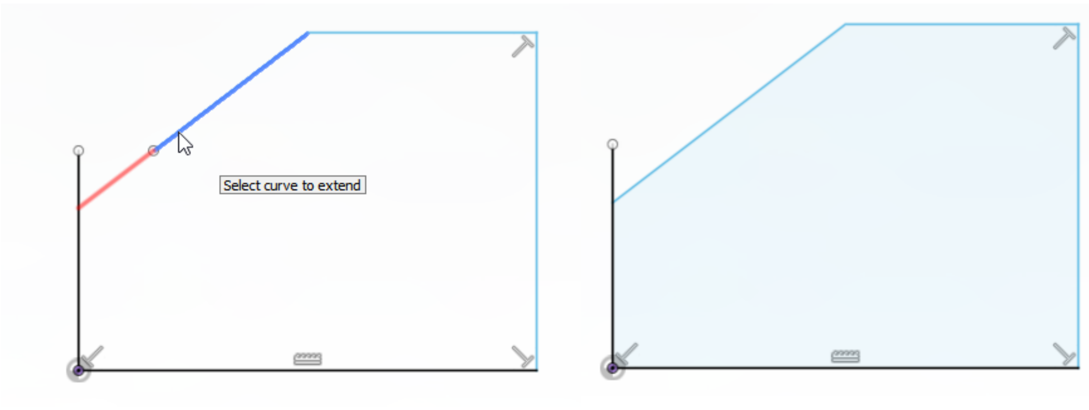

(a) (b)

Figure 2.97

Splitting a line

We can use the Break command (Figure 2.98) to split a line into two parts, producing two separate graphical objects, each of which can be manipulated and modified. Line splitting is performed relative to the point of intersection with another line.

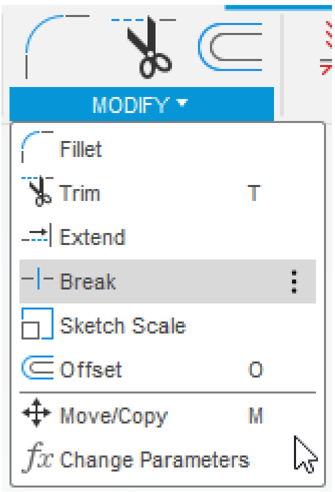

Figure 2.98

Lines that are divided into parts can be part of more complex geometric objects. Consider a sketch constructed from two geometric objects (Figure 2.99).

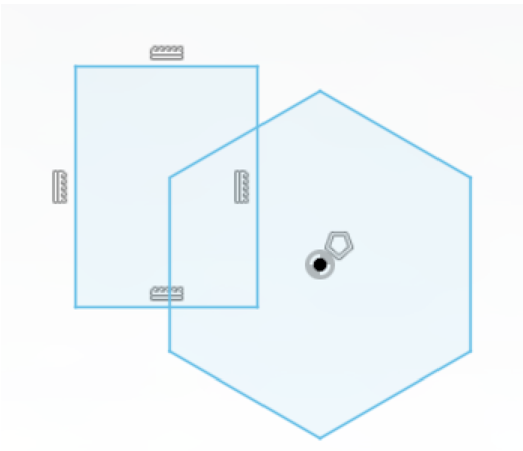

Figure 2.99

Using the Break command, the mouse cursor is used to select the line we wish to split (Figure 2.100).

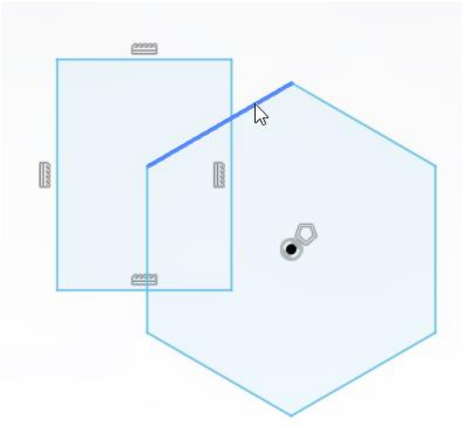

Figure 2.100

The line is split into two separate lines at the point of intersection (Figure 2.101).

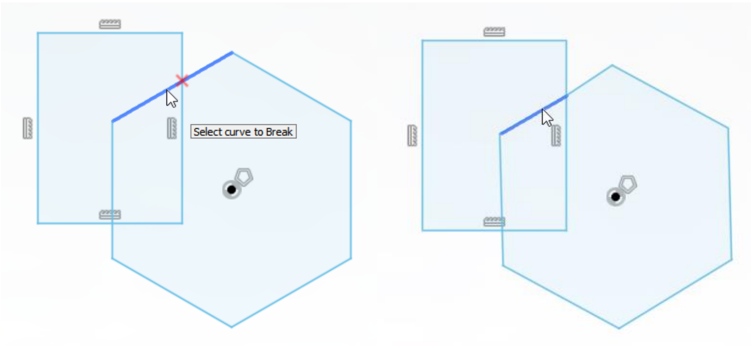

Figure 2.101

Each of the new lines can be moved or manipulated in a certain way (Figure 2.102).

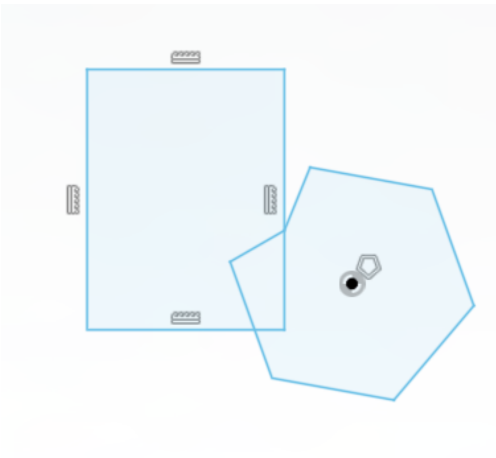

Figure 2.102

Scaling objects in the sketch

A sketch or certain objects within it can be scaled using the Sketch Scale command (Figure 2.103).

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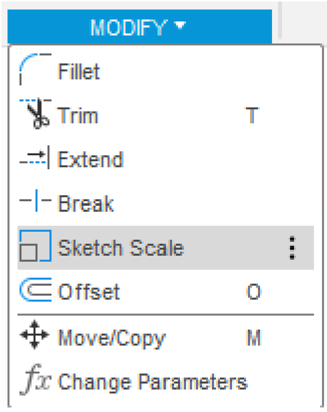

Figure 2.103

Consider the sketch shown in Figure 2.104. It consists of two objects, one of which will be scaled down using the Scale command.

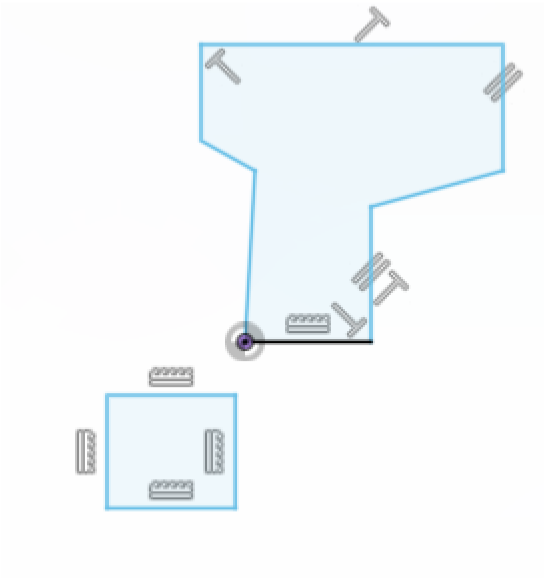

Figure 2.104

We activate the Sketch Scale command and we select with the mouse the object we wish to scale (Figure 2.105).

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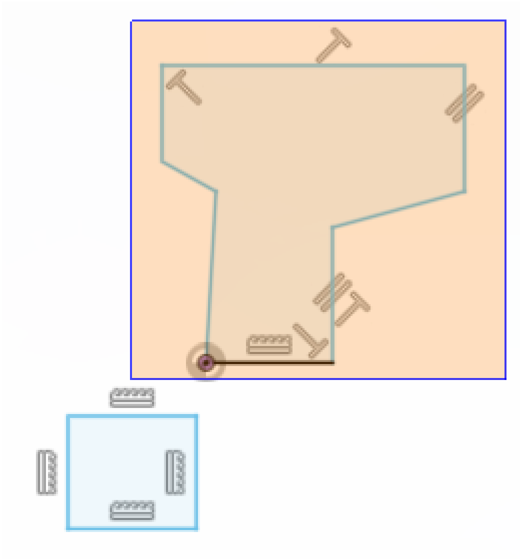

Figure 2.105

The object itself is made up of multiple lines and points. Additionally, a point must be specified against which the scaling will be performed (Figure 2.106). In the command menu, it is also necessary to specify the scaling factor (called Scale Factor). If this factor is greater than 1.00, then the object will be scaled by increasing its size, and, if it is smaller, the scaling will reduce the size of the object.

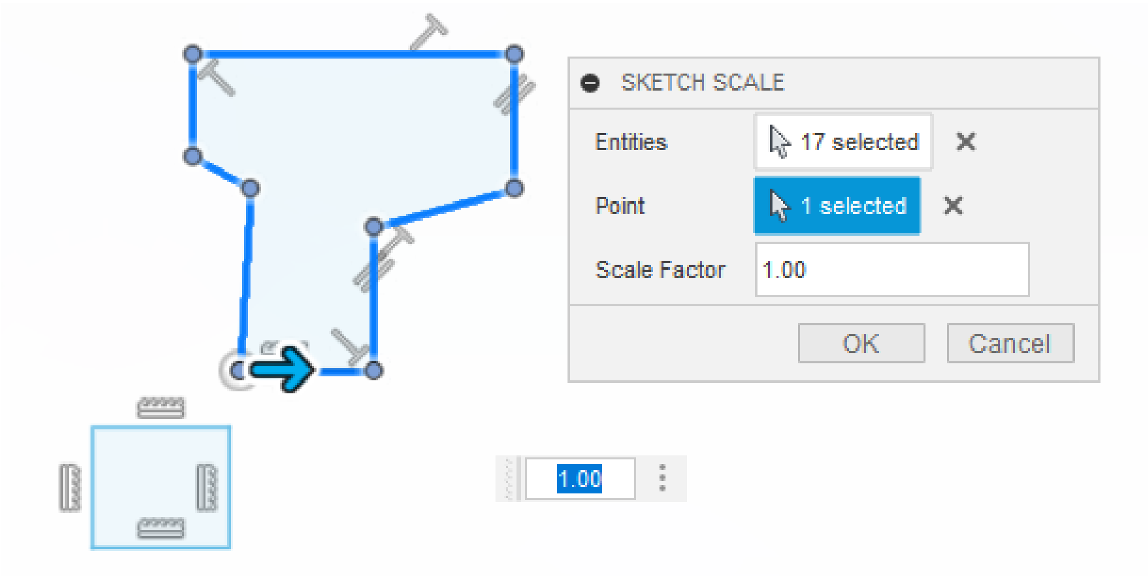

Figure 2.106

Scaling can also be done by using the arrow (starting from the scaling point). By pulling with the mouse on this arrow, the corresponding scale of the object can be set, i.e. its scaling factor can be changed (Figure 2.107).

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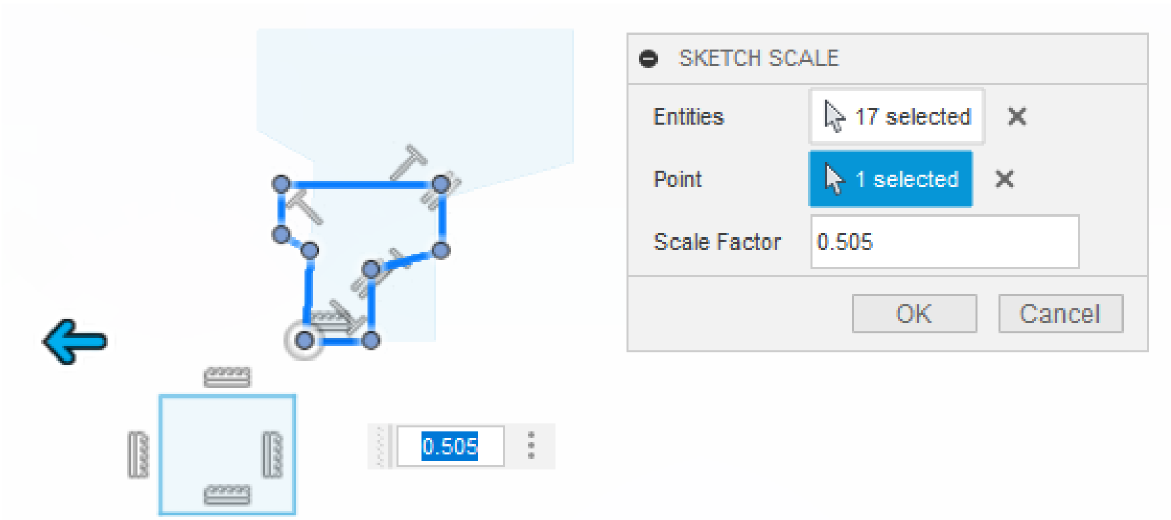

Figure 2.107

Creating an offset contour

By using the Offset command (Figure 2.108) offset contours can be created at a specified distance from the selected line or geometric shape. The offset command is very useful for easily and accurately repeating the shape of certain objects, obtaining accordingly a new location or a different scale of the new object.

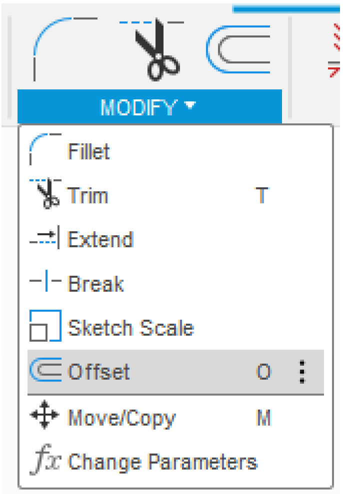

Figure 2.108

Consider the sketch presented in Figure 2.109 as a starting point.

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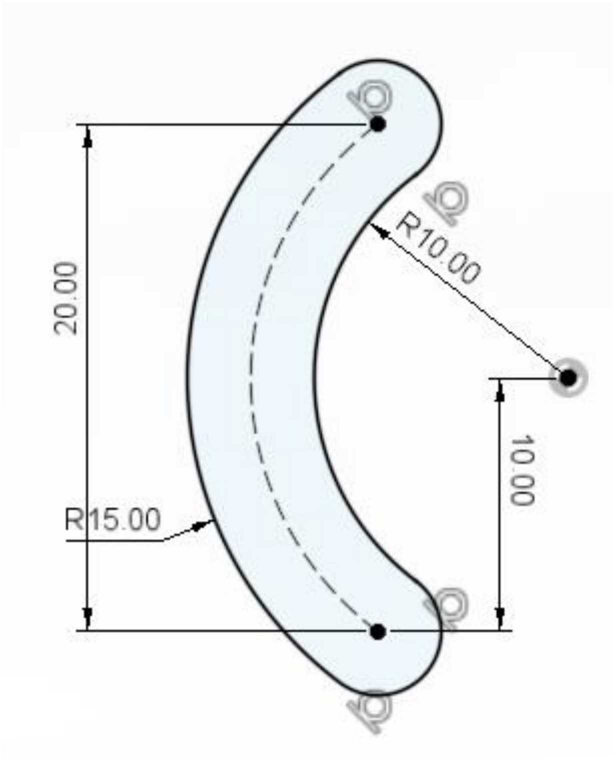

Figure 2.109

Once the Offset command is run, it is necessary to specify the contour on which the action will be applied (Figure 2.110). If necessary, the contour lines or the set of graphic primitives of which it is composed are selected in turn. In the command menu, the Chain Selection check box allows all the lines of a contour to be selected together after clicking on any of the contour lines.

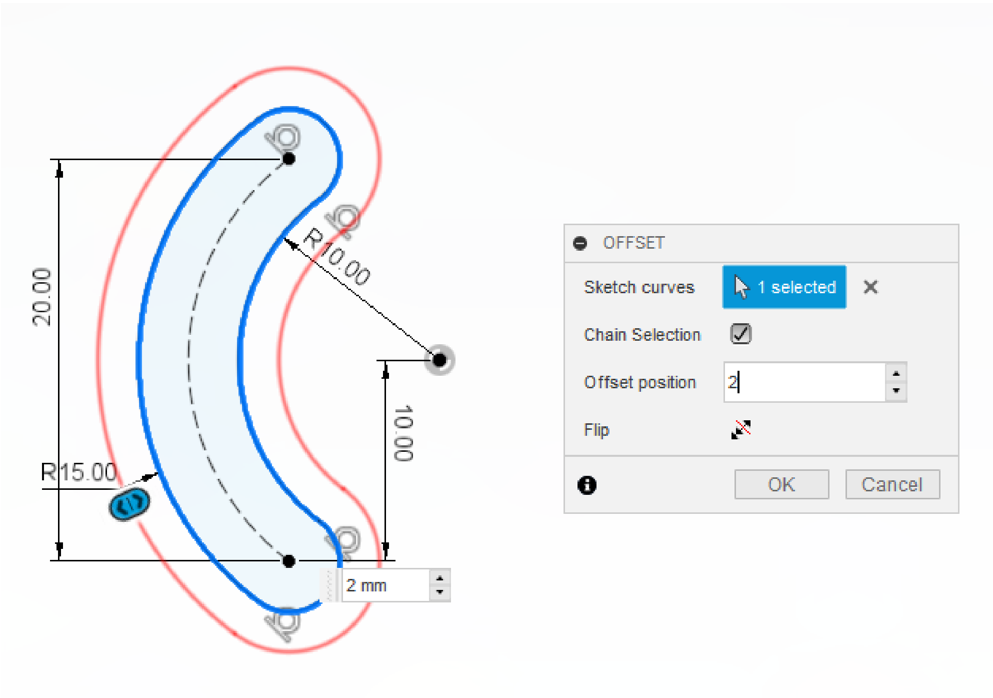

Figure 2.110

Depending on the value set in the Offset position field and its sign, the position of the offset contour changes (Figure 2.11).

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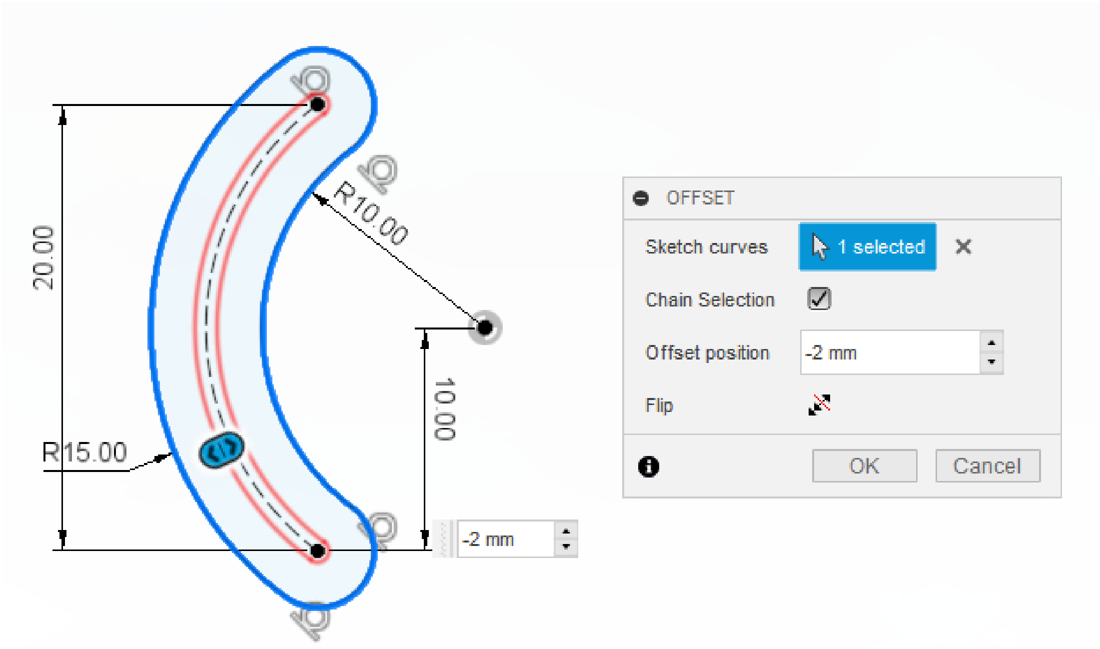

Figure 2.111

In the example under consideration, the new contour does not need to be further sized, as it appears in relation to the base contour depending on the specified offset value. In our example, the new contour can be used to define a surface as a basis of a 3D model (Figure 2.112).

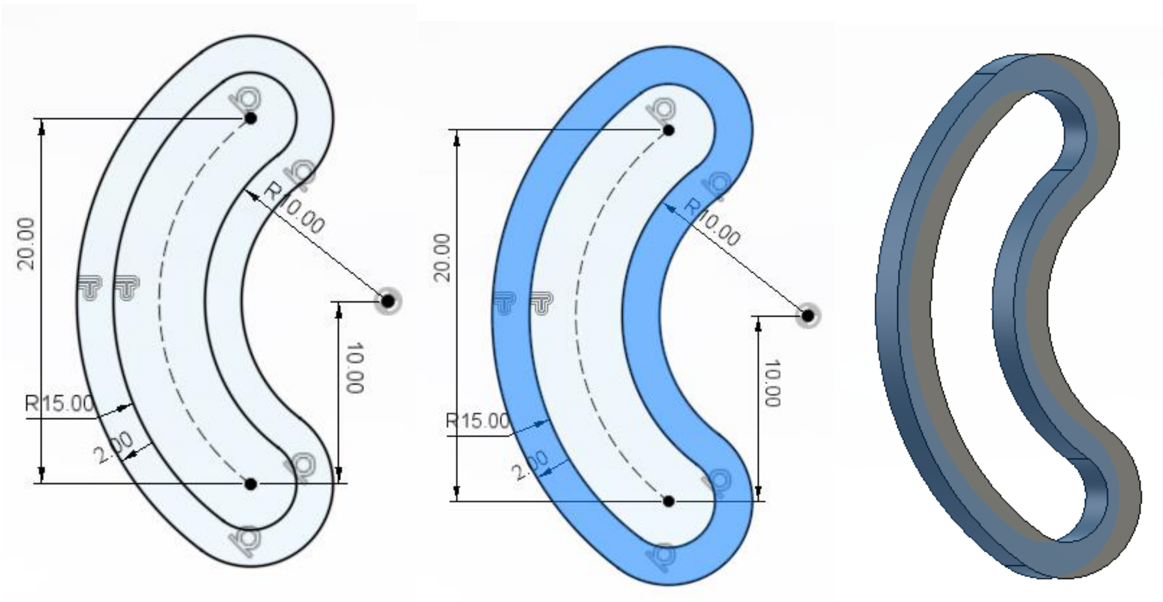

Figure 2.112

Moving and copying objects

In the process of sketching, it is often necessary to move individual objects and also to copy objects and place them in a new location in the sketch. For this purpose, we use the Move/Copy command (Figure 2.113).

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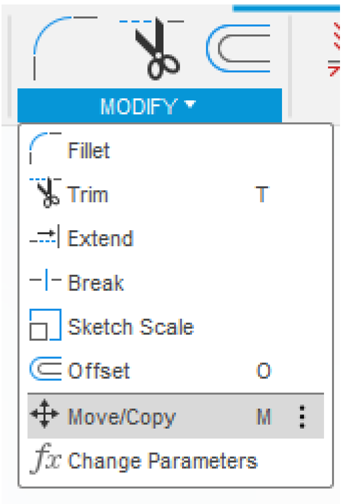

Figure 2.113

To demonstrate the operation of the command, let us consider a sketch consisting of two circles (Figure2.114). By using the Copy command, the larger circle will be moved to cover the smaller one and at the same time the two circles will touch each other.

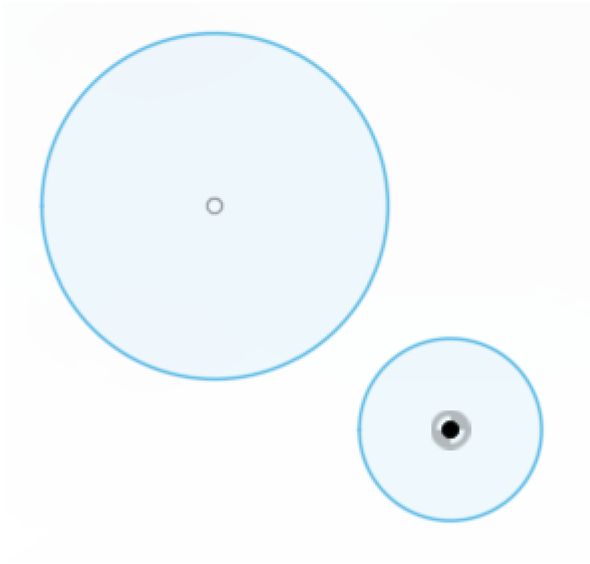

Figure 2.114

In order to move an object, it must be selected with the mouse after activating the Move/Copy command (Figure 2.115). Fusion 360 allows the move to be performed on both axes and the moved object can also be rotated at the appropriate angle.

In the command menu, there is a Create Copy checkbox. When activated, it moves a copy of the selected object, not the object itself.

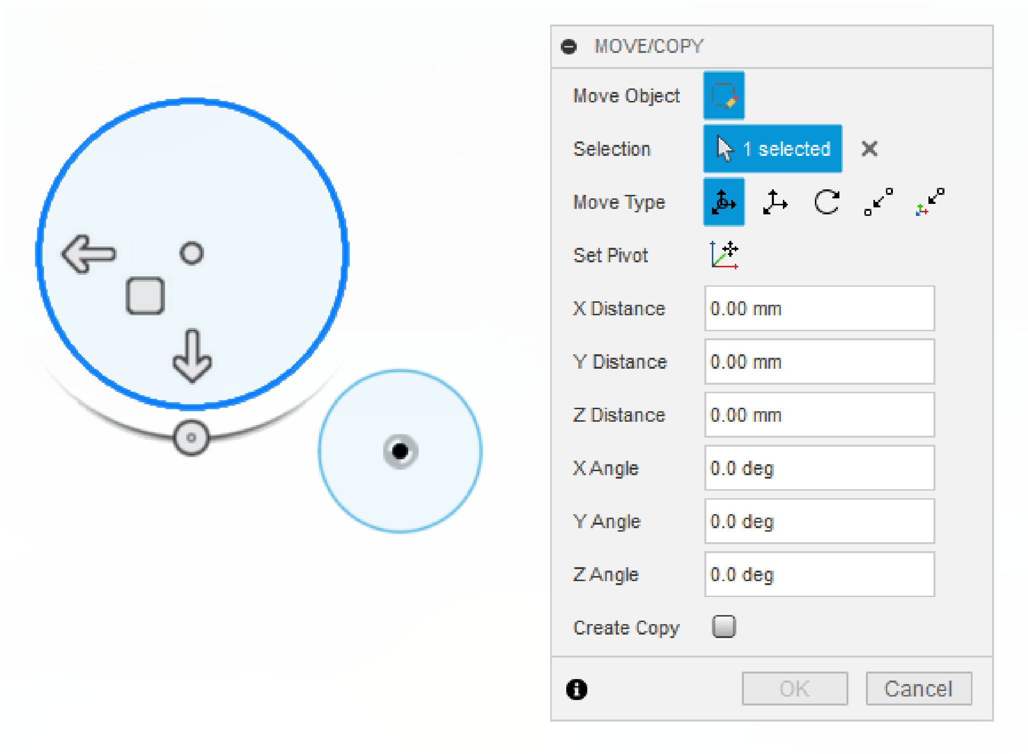

Figure 2.115

The end result of the operation is shown in Figure 2.116.

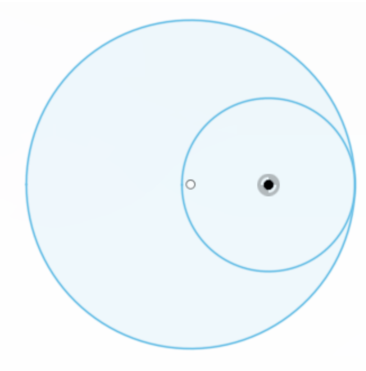

Figure 2.116

Defining constrained relationships between objects in the sketch

When creating sketches, individual lines or objects can be mutually related. These interrelationships set certain kinds of constraints that complement the sketch dimensioning. Constraints in sketches can be generated automatically by the environment during the drawing process or they can be specified afterwards with the help of the Constraints toolbar (Figure 2.117).

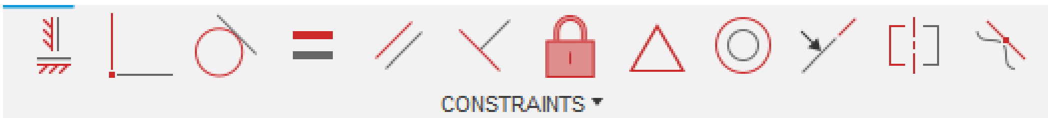

Figure 2.117

If not all Constraint commands are visible during the process, they can be seen by clicking on the drop-down menu (Figure 2.118).

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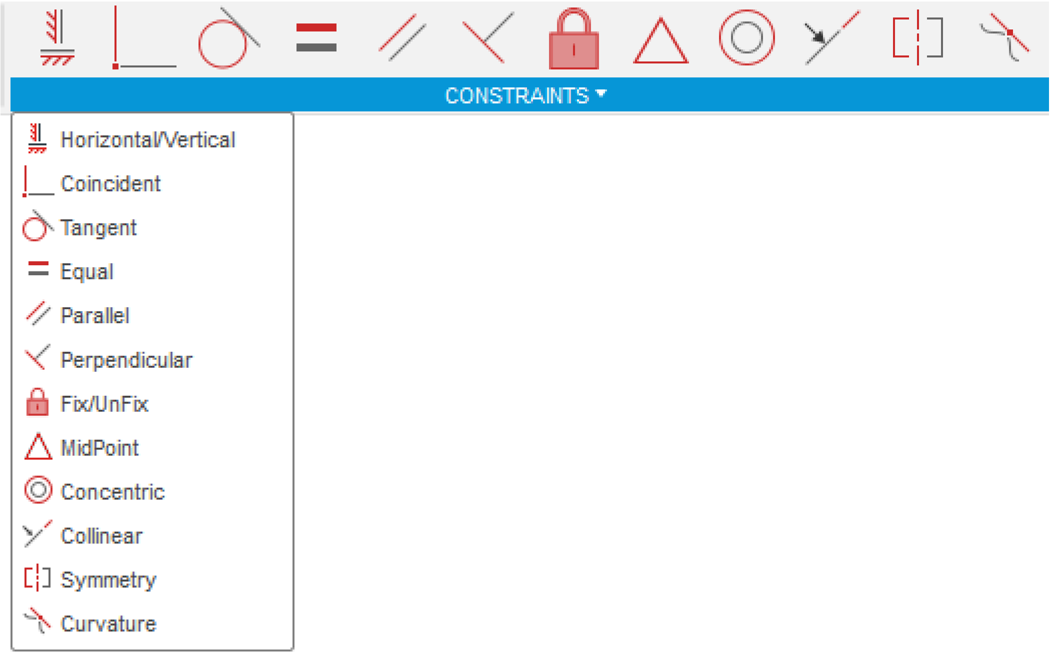
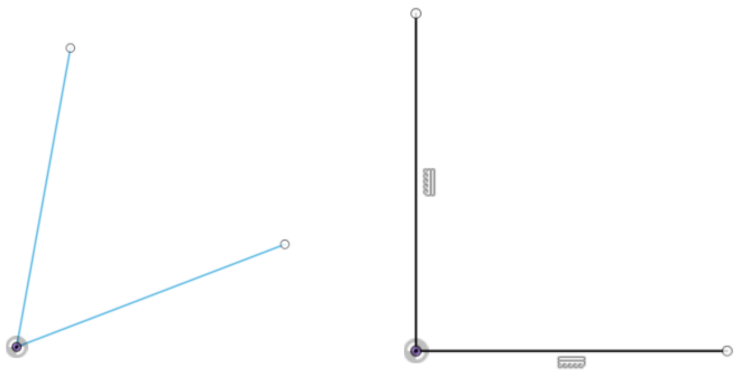


Figure 2.118

Setting line horizontality/verticality

Through the Horizontal/Vertical command  a line selected with the mouse is aligned and becomes parallel to the horizontal or vertical axis of the sketch, depending on which of the two its rotation angle is closer to.

In the sketch in Figure 2.119(a), after activating the Horizontal/Vertical command and clicking with the mouse on the two lines, they change their position in space (Figure 2.119(b)) and orient themselves horizontally and vertically.



(a) (b)

Figure 2.119

Setting a match of point and line or point and point

The Coincident command  is used to make a point lie on a line or on another point.

In the sketch in Figure 2.120 (a) there are two lines. After activating the Coincident command, we select the two endpoints of the lines (Figure 2.120 (b)) that we want to coincide.

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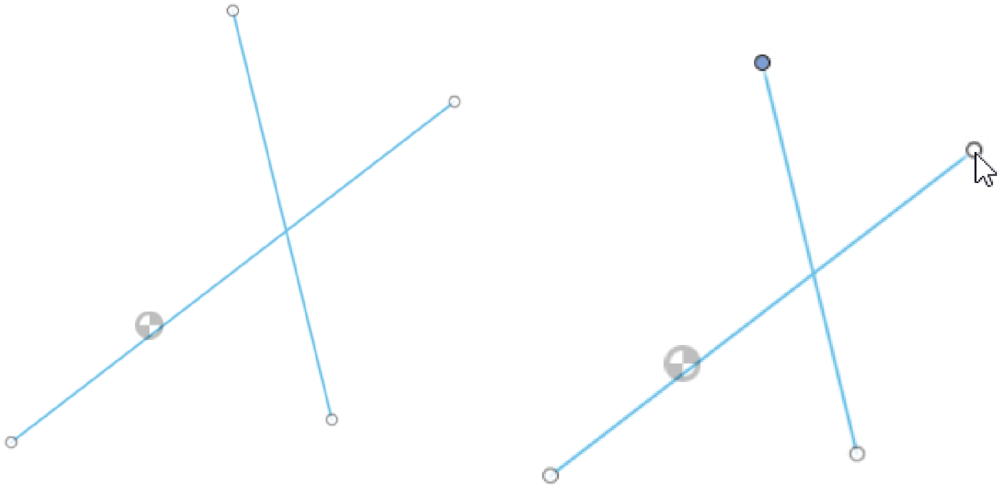


(a) (b)

Figure 2.120

After the points are specified, they are placed on top of each other (Figure 2.121)

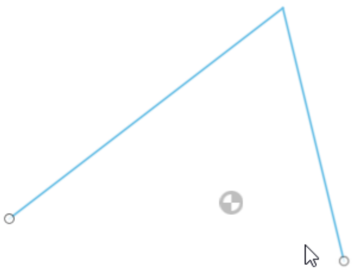
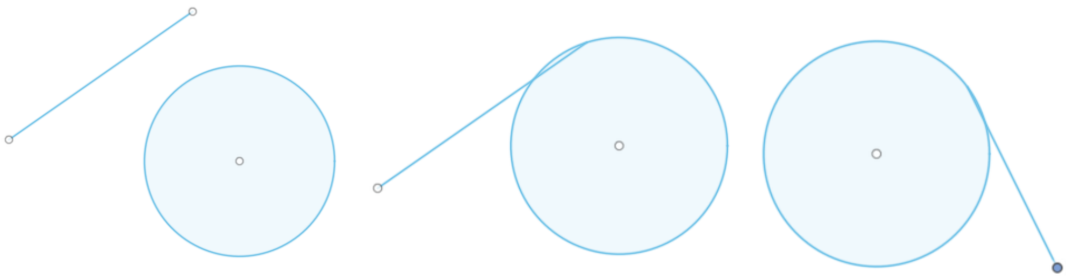


Figure 2.121

The Coincident command can also be applied to a line. Figure 2.122(a) shows a sketch containing a line and a circle. With the Coincident command, we select one endpoint of the line and specify the circle. After the selection, the point and the circle coincide (Figure 2.122(b)). It is possible to move the point, but now the new location can only be on the circle (Figure 2.122(c)).



(a) (b) (c)

Figure 2.122

Setting tangency

The Tangent  command is used to set a tangency constraint between a line and a circle or between two circles (or arcs), whereby the two objects will have a single common point.

A sketch consisting of a line and a circle is shown in Figure 2.123(a). After running the Tangent command, the straight line and then the circle should be selected with the mouse (Figure

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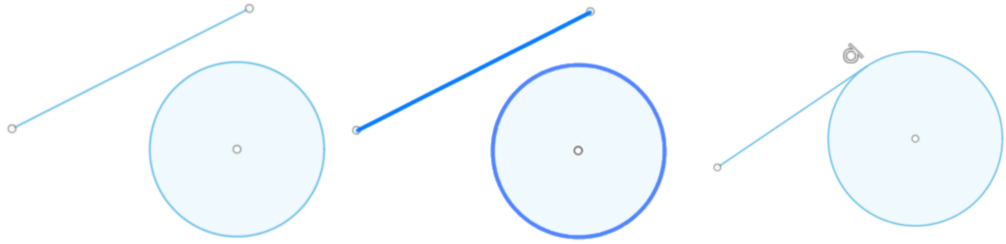
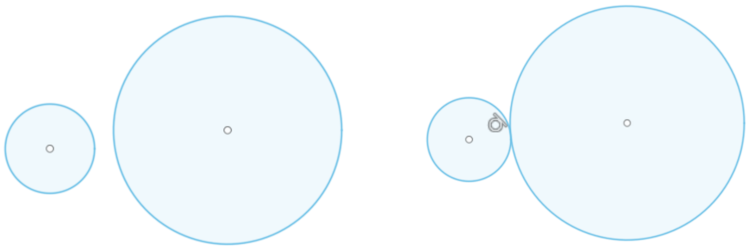
2.123(b)). The two objects are automatically made tangent to each other, and a symbol indicating the presence of a tangent relationship between them appears on the sketch (Figure 2.123(c)).



(a) (b) (c)

Figure 2.123

A tangent connection can also be specified between two circles (Figure 2.124(a)). After defining the tangency, the two circles touch each other at a single common point (Figure 2.124(b)).



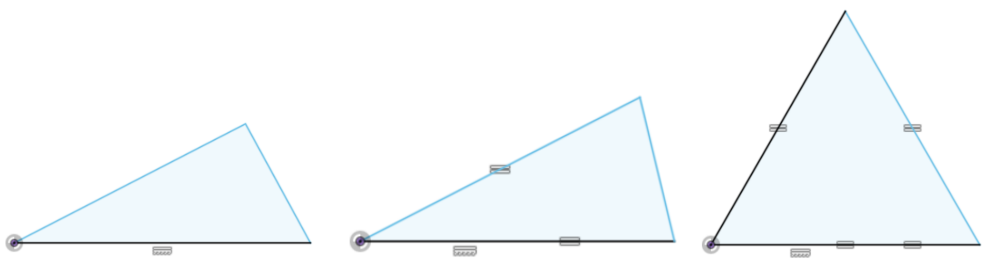
(a) (b)

Figure 2.124

Setting identical sizes

The Equal  command is used to make the sizes of two objects of the same type identical, for example two lines or two circles.

Consider a sketch of a triangle (Figure 2.125(a)). To convert this arbitrary triangle into an equilateral triangle, we need to use the Equal command to first select two of the sides (Figure 2.125(b). Then we need to click on the next two sides (one of which is the undefined side). This gives all sides the same length (Figure 2.125(c)).



(a) (b) (c)

Figure 2.125

Setting parallelism

The Parallel  command is used to make two lines parallel.

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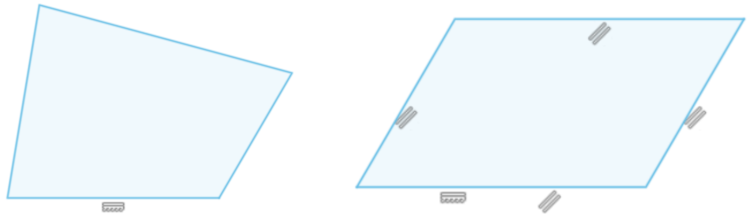
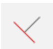
Consider a sketch of a quadrilateral figure (Figure 2.126(a)). Using the Parallel command, it is possible to convert the quadrilateral figure into a parallelogram by selecting the opposite sides of the quadrilateral (Figure 2.126(b)).

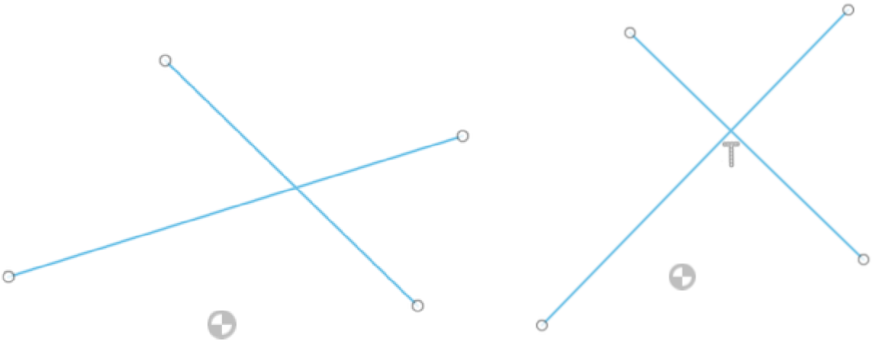


(a) (b)

Figure 2.126

Setting perpendicularity

The Perpendicular  command is used to make two lines perpendicular. The operation of the command is demonstrated in Figure 2.127. The two intersecting lines (Figure 2.127(a)) are positioned at 90° using the command (Figure 2.127(b)), and the perpendicularity symbol is displayed.



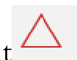
(a) (b)

Figure 1.127

Fixing geometric objects

The Fix/Unfix  command is used to fix geometric objects in the workspace. A fixed geometric object changes colour to green and its position in space relative to the coordinate system cannot be changed. To remove the fixed state of the object, the same command is used by clicking on the object. Using the Fix/Unfix command is useful for preventing certain objects in the sketch from being accidentally moved when a full dimensioning of the sketch will not be performed.

Setting a midpoint

The MidPoint  command makes two lines intersect at their exact midpoints, or a point to be located exactly in the middle of a selected line.

Consider the sketch in Figure 1.128(a). Using the MidPoint command, we select the vertical auxiliary line and the top side of the rectangle. The two lines are positioned relative to each other according to the position of their midpoints (Figure 1.128(b)).

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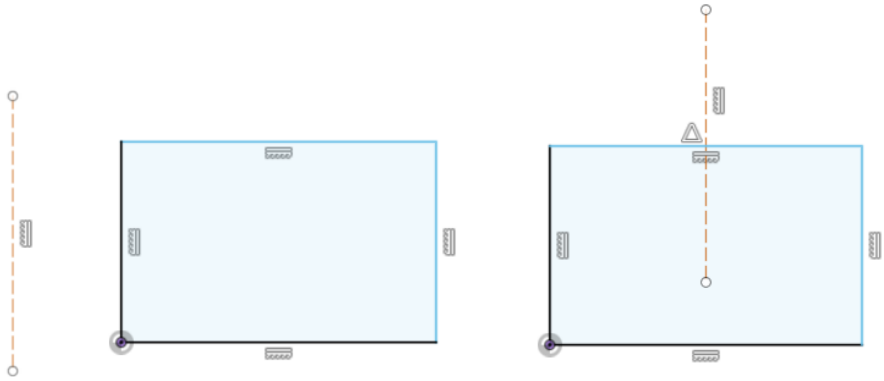
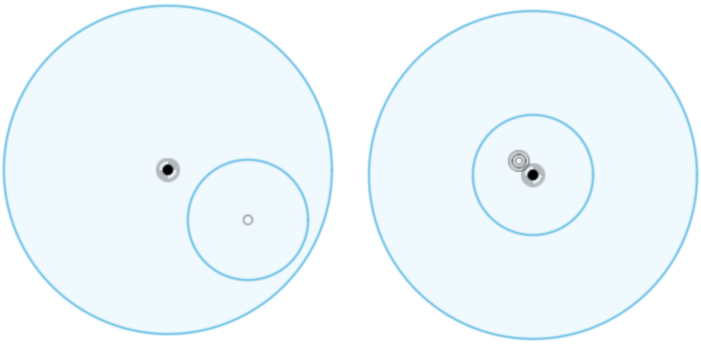


(a) (b)

Figure 2.128

Setting concentricity

The Concentric  command makes two circles or arcs concentric. A sketch containing two circles is given in Figure 2.129(a). After applying the Concentric command and pointing with the mouse at the two circles, they become concentric (Figure 1.129(b)). A concentricity symbol also appears next to the centre of the circles.

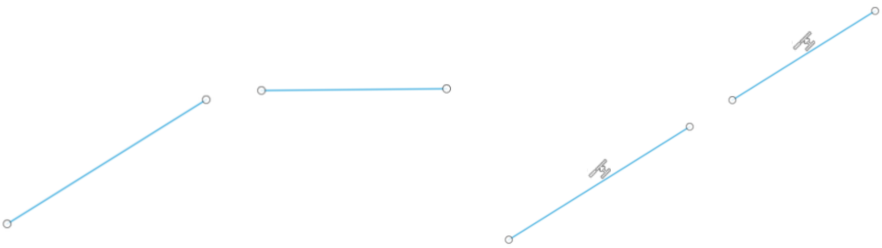


(a) (b)

Figure 1.129

Setting collinearity

The Collinear  command makes two lines coaxial (collinear). The result can be seen in Figure 2.130. Consider two lines that are at an angle to each other (Figure 2.130(a)). After activating the Collinear command, the mouse is used to select the two lines one after the other. As a result, the two lines become coaxial, with a collinearity symbol appearing next to them (Figure 2.130(b)).



(a) (b)

Figure 2.130

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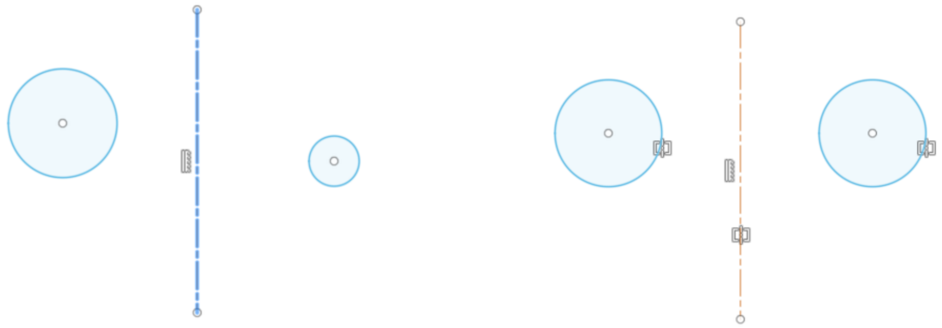
Setting symmetry

With the Symmetry  command, it is possible to position objects of the same type in the sketch symmetrically around a symmetry line and to make them identical. Figure 2.131(a) shows a sketch containing two circles and a line of symmetry between them. After activating the Symmetry command, it is necessary to point with the mouse the two circles and then the line of symmetry. As a result, the circles take the same shape and are symmetrically placed around the line of symmetry (Figure 2.131(b)).



(a) (b)

Figure 2.131

Sizing of the sketches

Object dimensioning is a key stage in the creation of sketches. If the objects' dimensions are not defined they can be easily changed with the mouse. Therefore, in order to create objects with accurate geometric dimensions, sketches need to be dimensioned.

The Sketch Dimension  command on the toolbar is used for this purpose. Each sketch has a reference point (Figure 2.132(a)) against which the position of the individual elements in the sketch is determined. This reference point is effectively a projection of the point defining the zero coordinates in three-dimensional space (Figure 2.132(b)) onto the 2D plane of the sketch.

(a) (b)

Figure 2.132

With the Sketch Dimension command, horizontal and vertical dimensions, dimensions to indicate the diameter of a circle and the radius of an arc or a curvature can be easily plotted. Angular dimensions are also automatically plotted when two lines that intersect each other are selected.

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The dimensioning process is made easier by the Constraints entered in the sketch, which define the geometry and relationship of objects to each other. In Figure 2.133 not all dimensions are plotted, but thanks to the constraints that have been introduced, the sketches are fully defined.

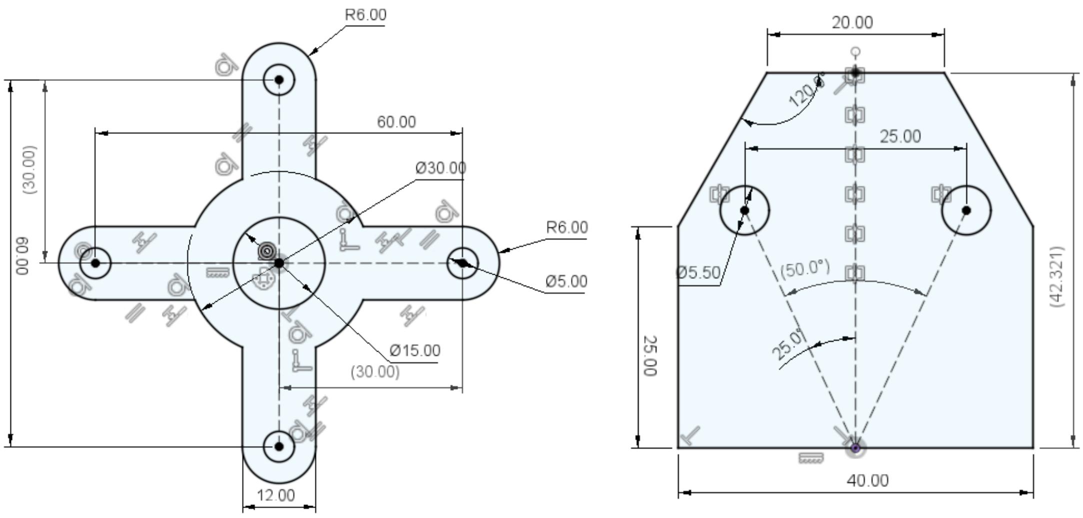

Figure 2.133

Dimension values can be defined either directly, by setting them as numbers, or by setting them as predefined parameters. The Change Parameters command from the Modify menu is used to define parameters (Figure 2.134).

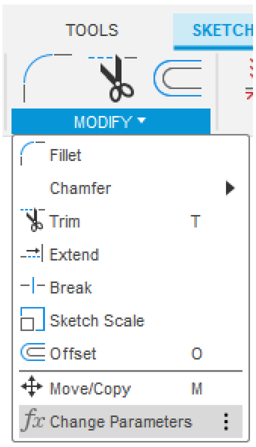

Figure 2.134

Figure 2.135 shows the parameter menu with three user-defined parameters, par1, par2 and par3, and their corresponding values defined as numbers.

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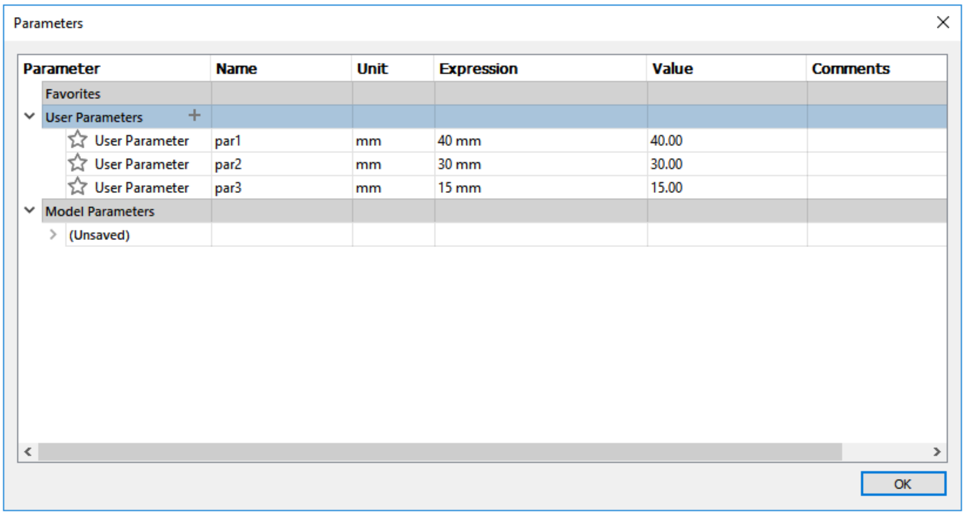

Figure 2.135

Entering new parameters is done by clicking on the "+" next to the User Parameters category. A menu opens (Figure 2.136), which needs to be filled in with the parameter name, units and value.

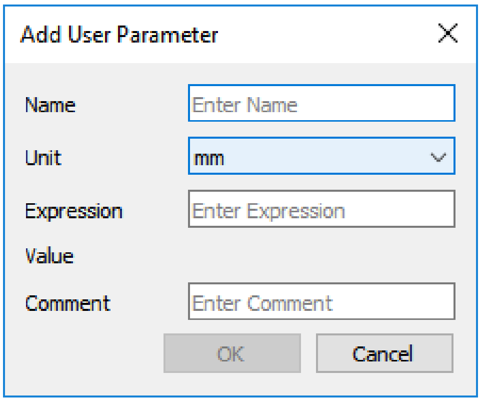

Figure 2.136

When sizing, we can use the defined parameters by writing a parameter name instead of a size value (Figure 2.137).

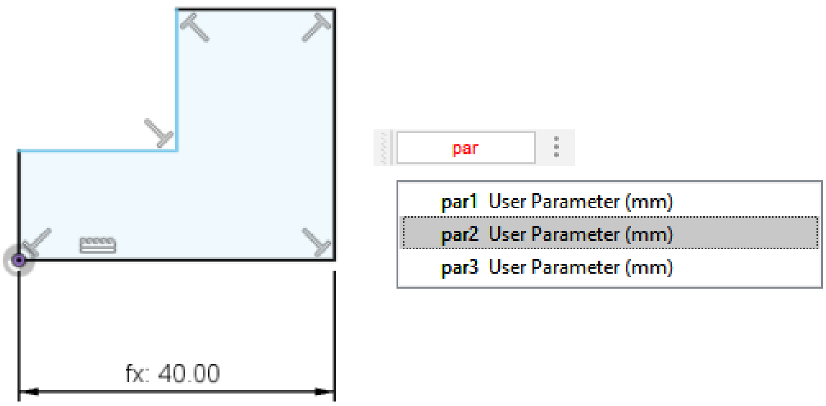

Figure 2.137

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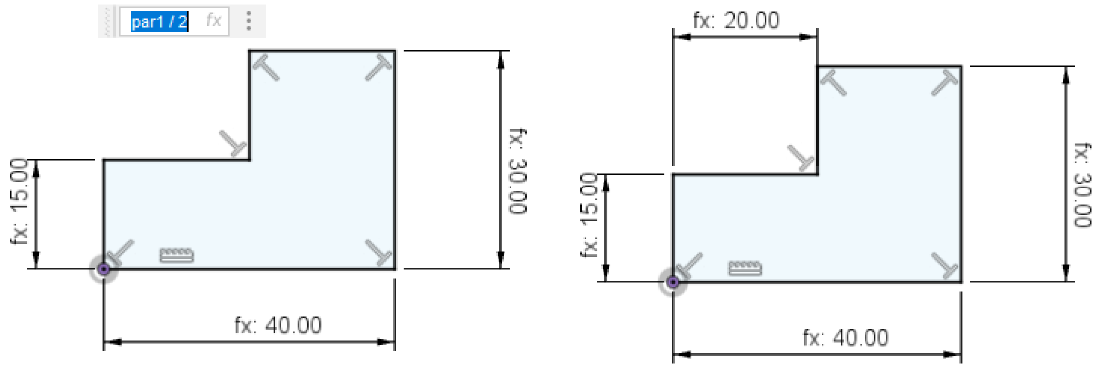
Dimension values can also be defined as mathematical expressions (Figure 2.138(a)). All dimensions set as parameters or mathematical expressions are displayed starting with *fx:* and a subsequent dimension value (Figure 2.138(b)).



(a) (b)

Figure 2.138

Three-dimensional sketching

In Fusion 360, in addition to standard two-dimensional sketching, it is also possible to use three-dimensional sketching, where the sketch contains objects located on different planes. To activate three-dimensional sketching it is necessary to select the 3D Sketch checkbox in the Sketch Palette menu (Figure 2.139).

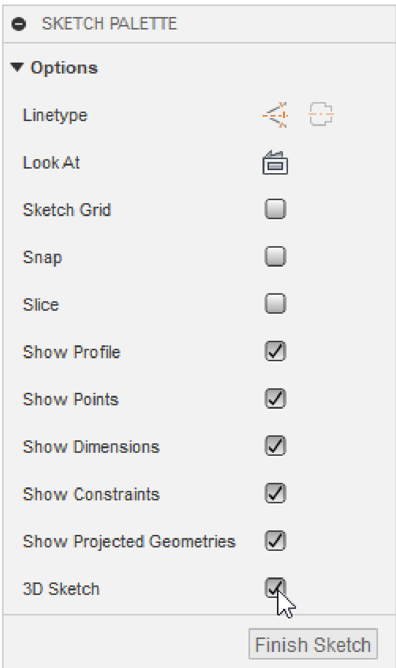


Figure 2.139

When 3D sketching is activated, the cursor changes and takes on the appearance presented in Figure 2.140.

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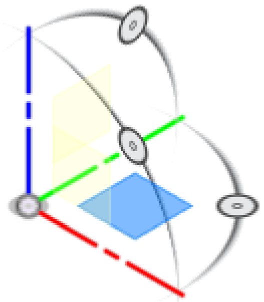
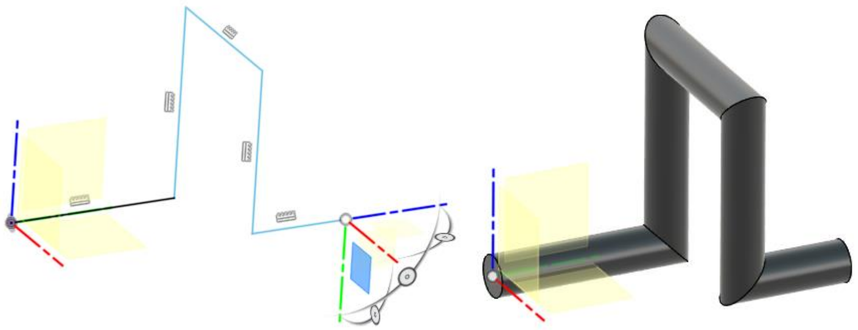


Figure 2.140

In 3D sketching, it is possible to successively switch the drawing of objects to take place in different planes, thus producing three-dimensional sketches (Figure 2.141(a)). 3D sketching is most often applied in the creation of tubular and profile constructions that follow a particular geometry in space (Figure 2.141(b)), but it can also be used for the creation of cables and connectors.



(a) (b)

Figure 2.141

Supplementary self-study video materials for creating and working with sketches:

https://www.youtube.com/watch?v=wvp5dO-2Q88&list=PLmA_xUT-8UlLx4mcCK62PNW2fz4K4_Rxy

https://www.youtube.com/watch?v=wOQydMGSq9w&list=PLmA_xUT-8UlLx4mcCK62PNW2fz4K4_Rxy&index=2

https://www.youtube.com/watch?v=J_2If5zVp84&list=PLmA_xUT-8UlLx4mcCK62PNW2fz4K4_Rxy&index=3

https://www.youtube.com/watch?v=PwVRaWxW7F0&list=PLmA_xUT-8UlLx4mcCK62PNW2fz4K4_Rxy&index=6

https://www.youtube.com/watch?v=dCEJO8j2bMM&list=PLmA_xUT-8UlLx4mcCK62PNW2fz4K4_Rxy&index=14

https://www.youtube.com/watch?v=DzcLAJ9wg1U&list=PLmA_xUT-8UlLx4mcCK62PNW2fz4K4_Rxy&index=25

https://www.youtube.com/watch?v=hh32H_qN2As&list=PL40d7srwyc_MzVZR2WdcElaX9w6BjUd8n&index=1

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Creating a 3D model

The creation of 3D models in Fusion 360, as well as in other 3D modeling software, is based on pre-drawn 2D sketches. Sketches are the basis for drawing the shape of 3D models. Fusion 360 includes various commands for creating three-dimensional models (Figure 3.1).

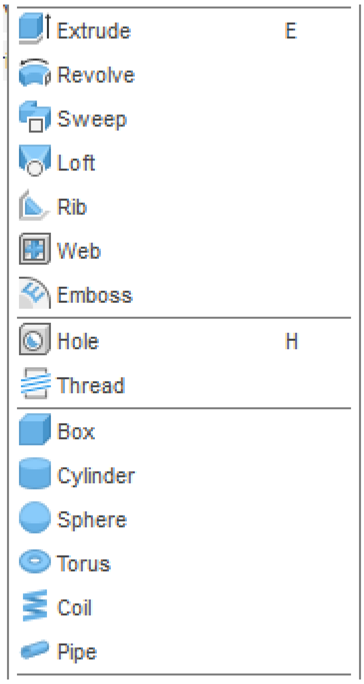

Figure 3.1

In this chapter we shall address the most important commands for creating 3D models and shall describe the most important parameters by means of which the operation of these commands is controlled.

Extrusion with Extrude command

The Extrude command is the main command used to create 3D bodies in the program. To create the three-dimensional object, a sketch is used as a base. The surface of another three-dimensional object can also be used.

In Figure 3.2 a sketch of an object is presented, with the help of which the three-dimensional body will be created.

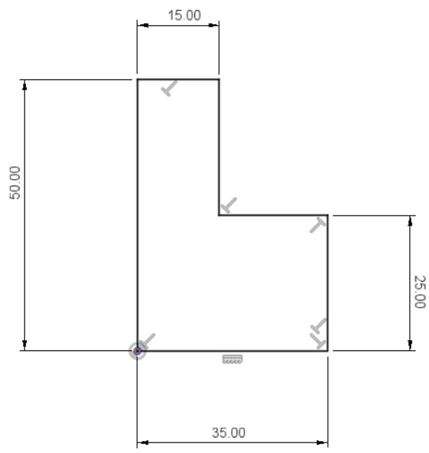

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Figure 3.2

We select the Extrude command from the toolbar (Figure 3.3).

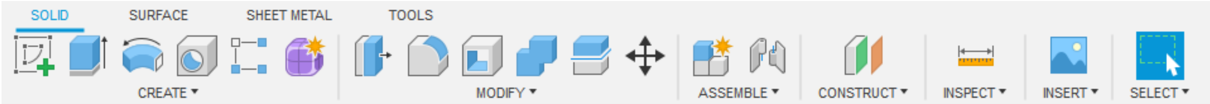

Figure 3.3

After starting the Extrude command , the sketch (or its separate parts) should be selected by clicking (Figure 3.4).

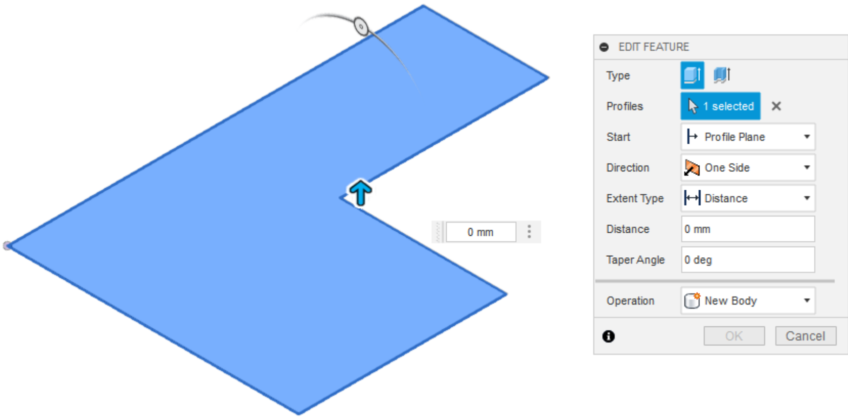

Figure 3.4

After selecting the surface, in the context menu there is a possibility to specify the extrusion distance (Figure 3.5) – in this example, we choose 10 mm.

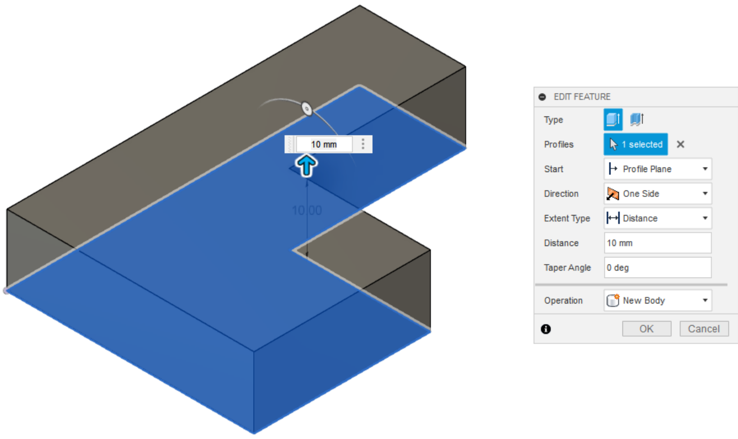

Figure 3.5

Before confirming the execution of the command by the OK button, a preview of the three-dimensional object appears in the working environment and it can be further edited with the options provided in the Extrude command menu.

The first basic option is to select and define a starting point from which the object's extrusion starts. Extrusion can start from the plane of the sketch in which the object's profile is set, from a plane parallel to the sketch's plane and shifted at a certain distance from it, or from the plane of a given object (Figure 3.6, a).

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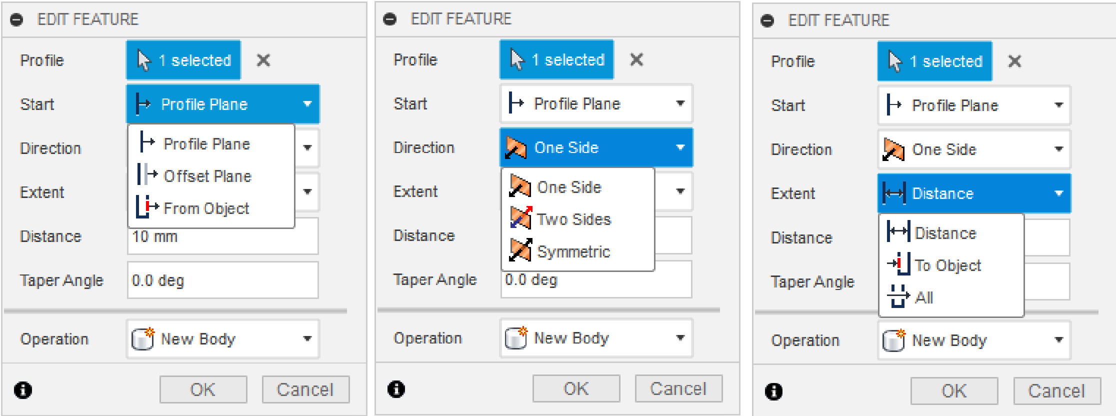
The extrusion can be performed in one direction, in two directions, with different values of the extrusion distance in each direction and symmetrically with respect to the starting point of the extrusion, or in both directions and with the same extrusion distance (Figure 3.6, b). The extrusion size can be set by specifying the distance or the object to which the extrusion is to be performed. In case we want to cut the object (remove a part of the object) a cut operation can be used (Figure 3.6, c).



(a) (b) (c)

Figure 3.6

The Extrude command also allows for the possibility of extrusion at a certain angle. We need to specify the extrusion angle in the Taper Angle field (Figure 3.7). This option is especially useful when creating objects that will be produced by casting, so that they can be then easily separated from the molds.

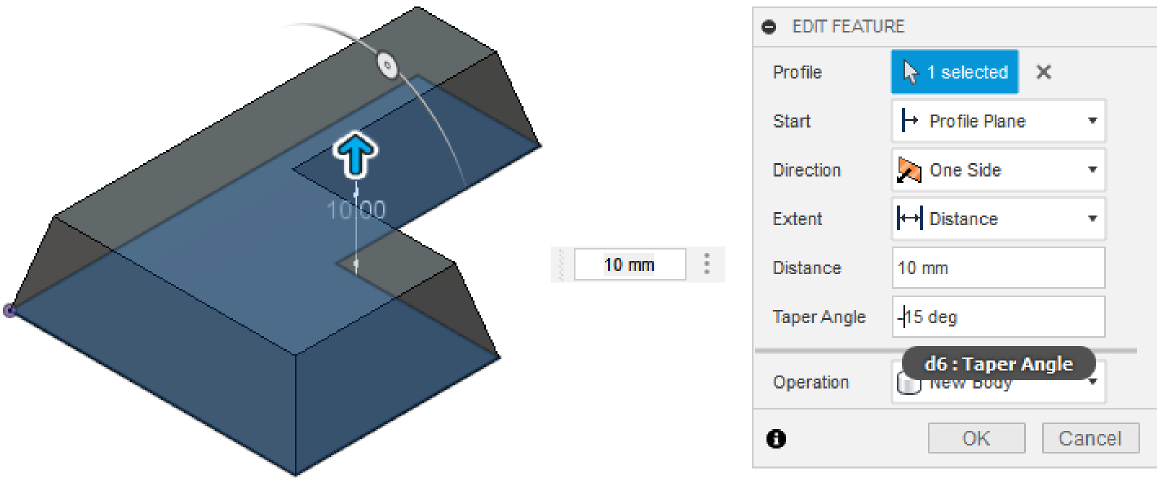


Figure 3.7

In the Operation field, we need to select the type of operation that will be performed when executing the selected command (in this case the Extrude command). There are the following options for an operation (Figure 3.8):

- New Body – creation of a new body (a separate three-dimensional object) from the other objects existing in the working field
- Join – the object created by the respective sketch or profile is added to an already existing three-dimensional object which it should touch or intersect

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- Cut – with the help of a chosen profile, an operation is performed to cut or remove part of an existing object
- Intersect – the selection of this option leads to the creation of an object representing the intersection of an existing three-dimensional body with the object to be extruded.

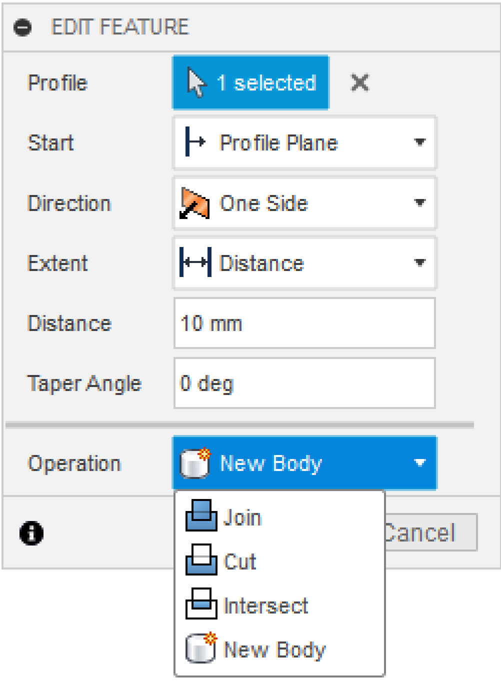

Figure 3.8

Let us create a new sketch on top of the created object, and the plane of the sketch will be one of the object's surfaces (Figure 3.9).

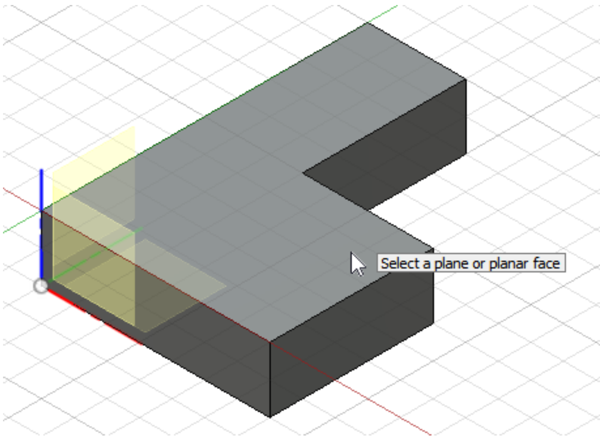

Figure 3.9

On the new sketch we will draw a circle with the dimensions presented in Figure 3.10.

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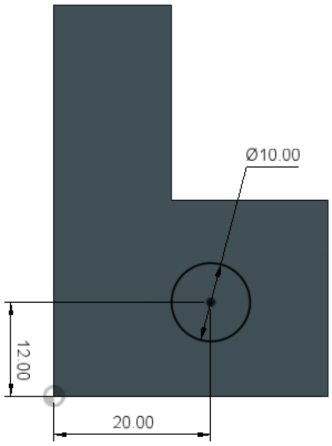

Figure 3.10

After creating the sketch, we will create a new feature of the three-dimensional object using the Extrude command. The new feature will be added to the object by using the Join operation (Figure 3.11).

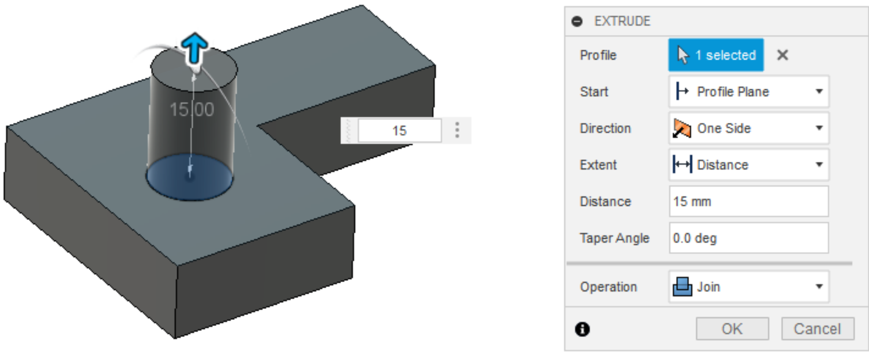

Figure 3.11

When using the Join operation, the extrusion of the profile is added to the existing object and a single body is displayed in the file browser (Figure 3.12).

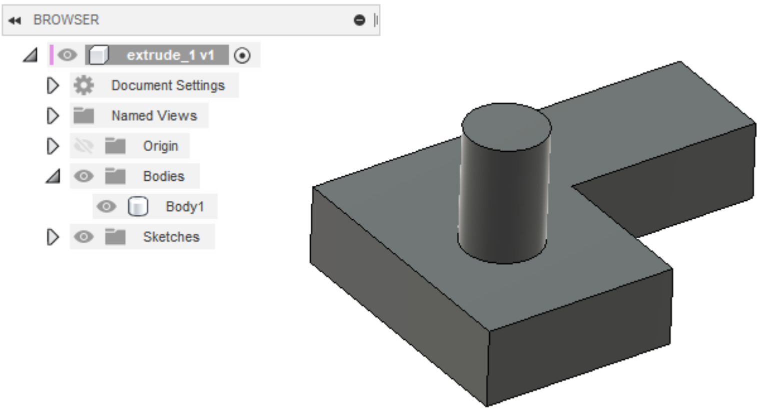

Figure 3.12

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To demonstrate the difference between the Join and New Body operations, let us edit the object created so far. Editing a certain feature of the object is done by right-clicking this feature on the Timeline of the file (Figure 3.13). By selecting the Edit Feature command we enter edit mode.

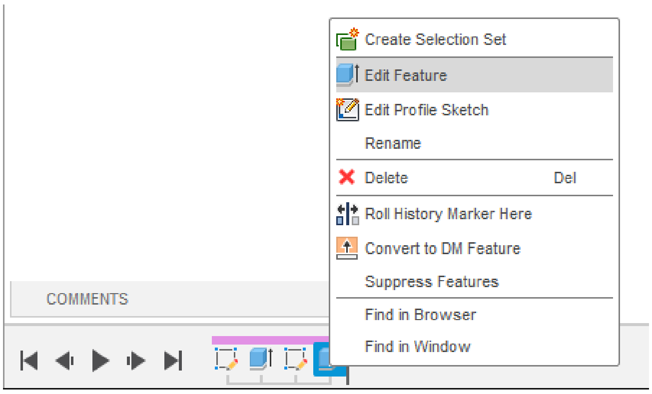

Figure 3.13

Let us change the operation from Join to New Body (Figure 3.14), without changing other parameters from the Extrude command menu.

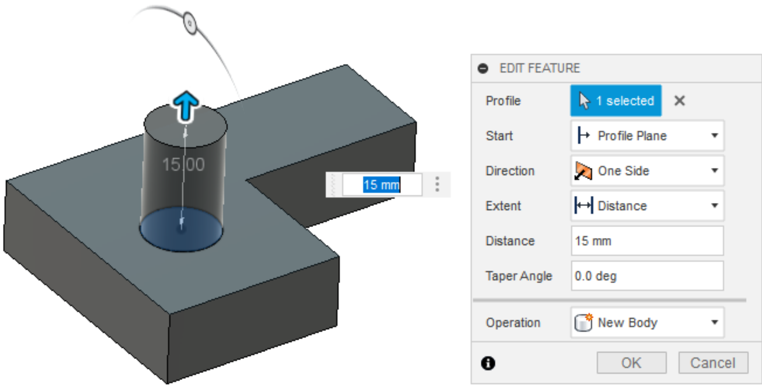

Figure 3.14

As a result of this change, a second body appears in the file browser, and it is independent from the first (Figure 3.15).

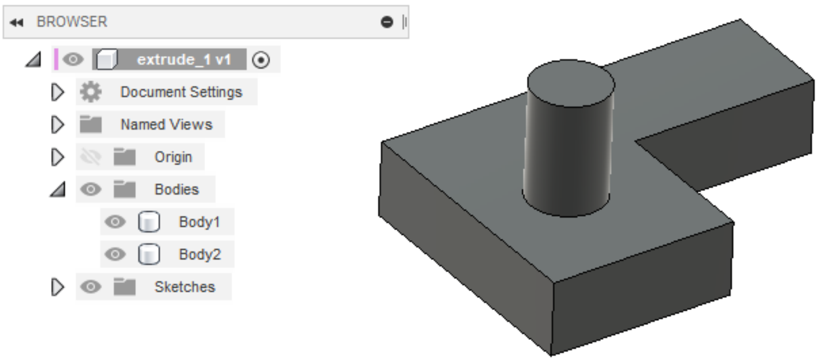

Figure 3.15

Let us again change the operation of the Extrude command and select the Cut option. The Cut operation will be selected automatically by Fusion 360 if we drag the arrow indicating the direction of extrusion through an already existing three-dimensional object. The Cut operation

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will also be enabled automatically if in the Distance field we write a value at which the direction of extrusion would pass through an existing three-dimensional object (Figure 3.16).

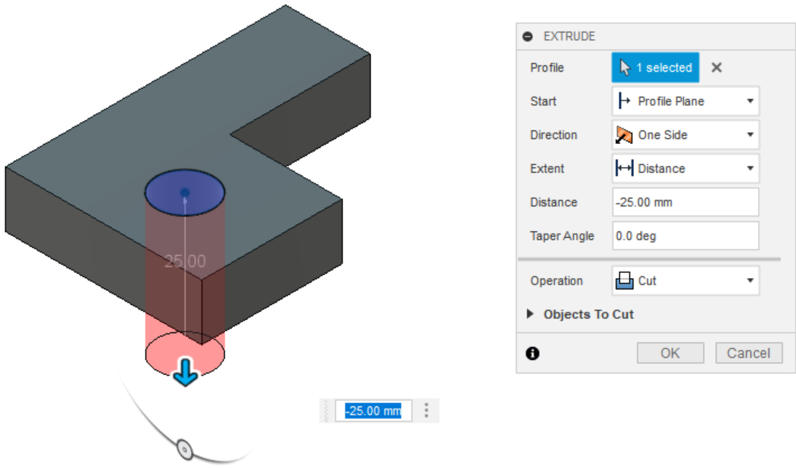

Figure 3.16

As a result of the Cut operation, part of the volume of the existing object will be removed, and the body of the object will acquire a new shape (Figure 3.17).

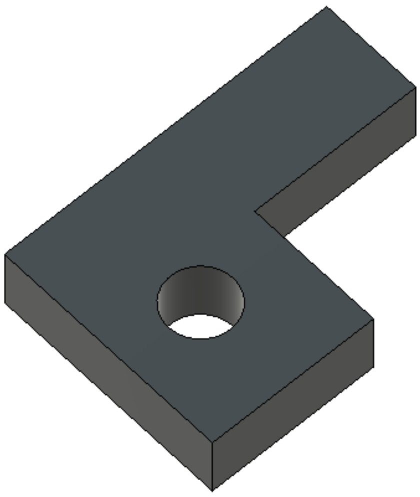

Figure 3.17

The Extrude command can also be used to create a thin-wall object from a specific 3D contour. Let us start the command and select the extrusion type to be Thin Extrude 

(Figure 3.18). Following the contour of the selected profile, a thin-walled object will be created with the parameters presented in Figure 3.19.

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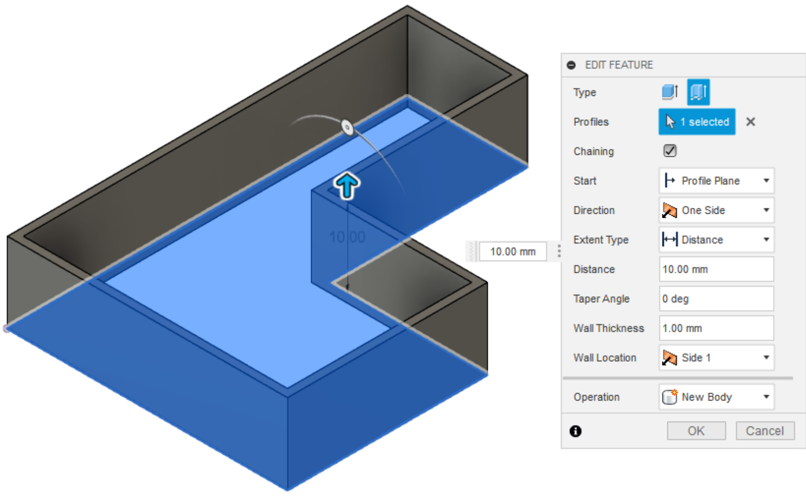

Figure 3.18

When creating the object, in addition to the other extrusion parameters, the thickness of the wall has to be selected, as well as its location (Wall Location) - outside the contour, inside it or in the center of the contour.

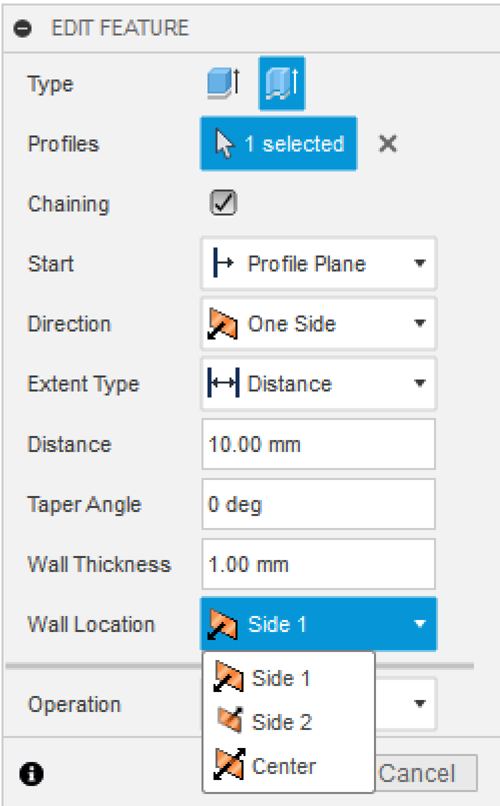

Figure 3.19

The resulting object follows the selected contour (Figure 3.20).

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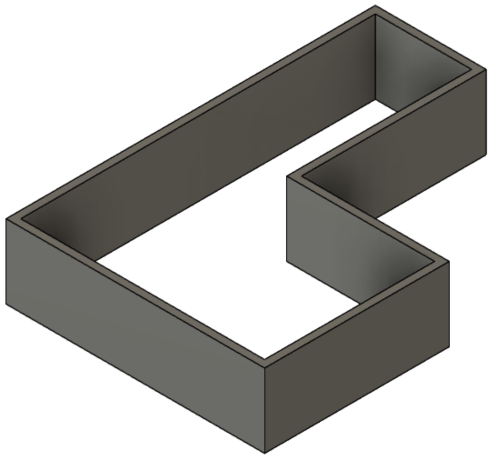

Figure 3.20

Extrusion with the Revolve command

In Fusion 360, objects can be created using extrusion through rotation (the Revolve command) . This command allows for creating three-dimensional objects with the help of a created 2D contour and axis of rotation.

The sketch presented in Figure 3.21 contains a closed contour and a construction line that will be used as the axis of rotation.

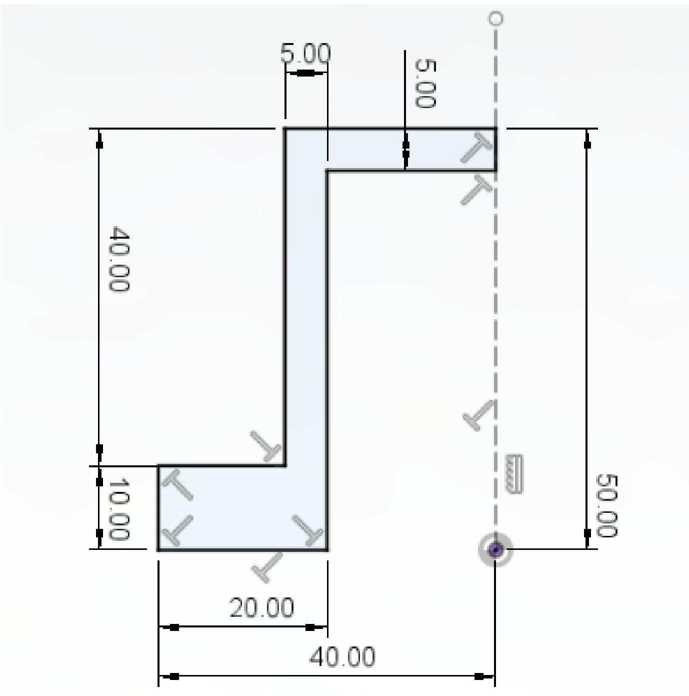

Figure 3.21

From the toolbar or from the menu the Revolve command is selected (Figure 3.22).

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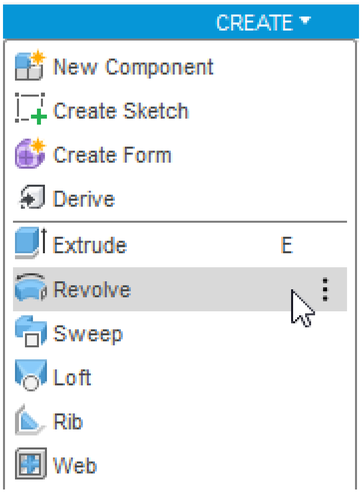

Figure 3.22

With the mouse we select the profile that will be used in the operation (Figure 3.23).

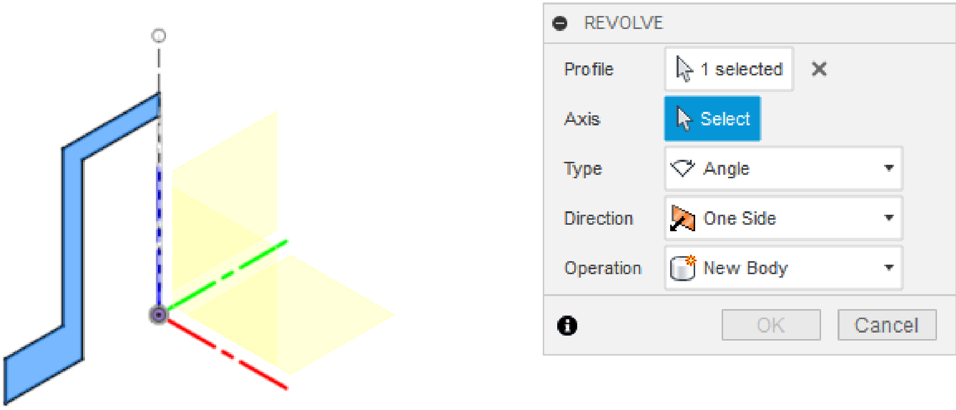

Figure 3.23

After specifying the axis of rotation, Fusion 360 will provide a preview of the created object (Figure 3.24).

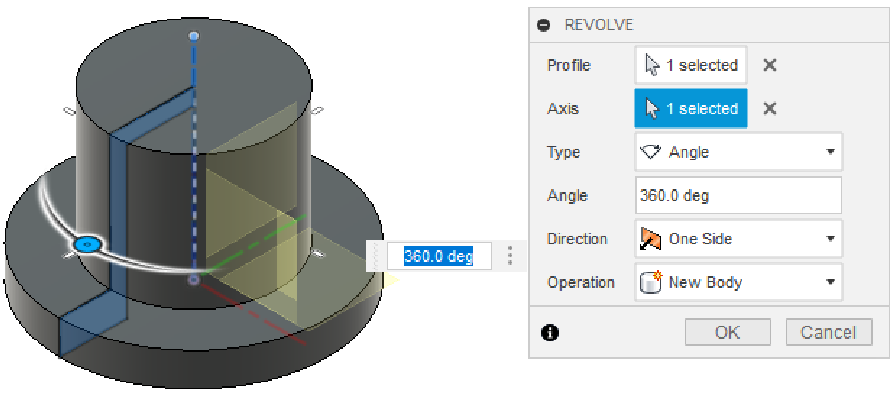

Figure 3.24

With the options in the context menu of the Revolve command (Figure 3.25) we can define the type of extrusion. The option used most often is *Angle*. It extrudes at a given angle.

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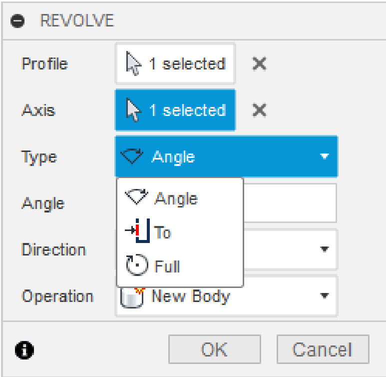


Figure 3.25

The value of the extrusion angle can also be specified using the context menu or using the mouse and the knob for angle adjustment (Figure 3.26).

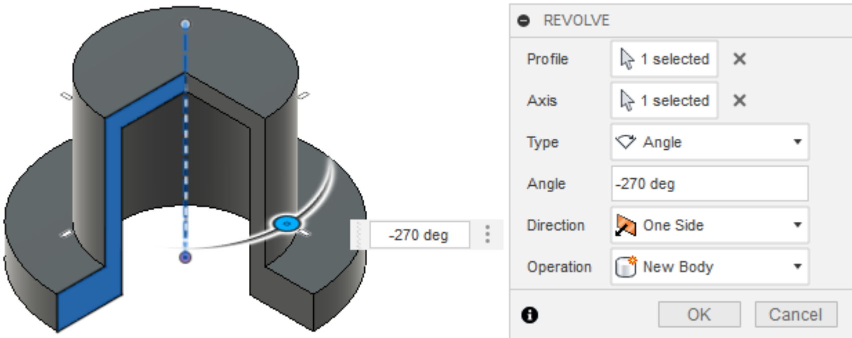
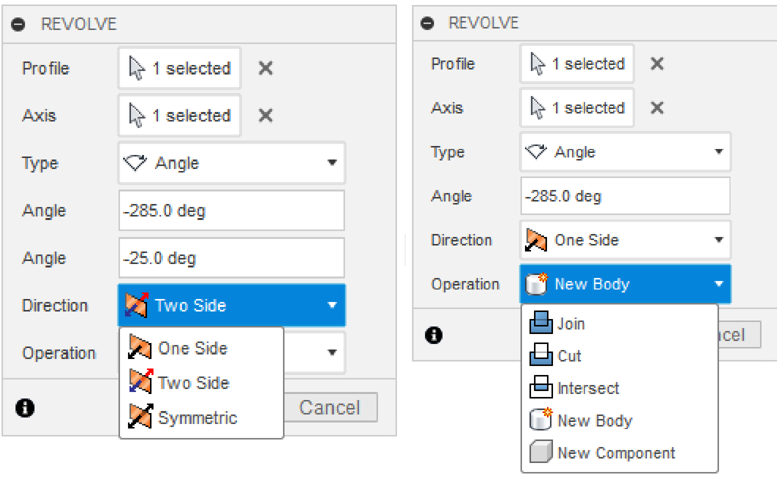


Figure 3.26

Additional features in the command context menu are the options for specifying the extrusion direction (Figure 3.27, a), as well as the operation type for creation of the three-dimensional object (Figure 3.27, b).



(a) (b)

Figure 3.27

Let the object be created with a 360º angle of rotation (Figure 3.28).

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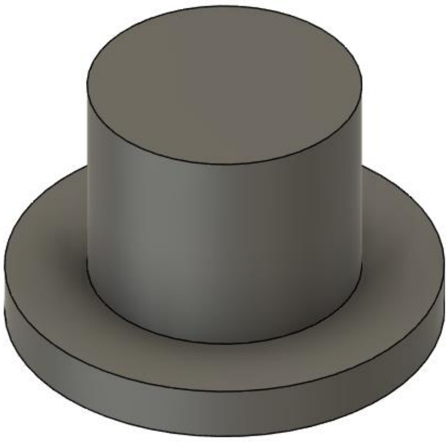

Figure 3.28

A new sketch should be created on the top plane of the object (Figure 3.29). The path determined by this sketch will be used again with the Revolve command, but this time to perform the cutting operation (Cut).

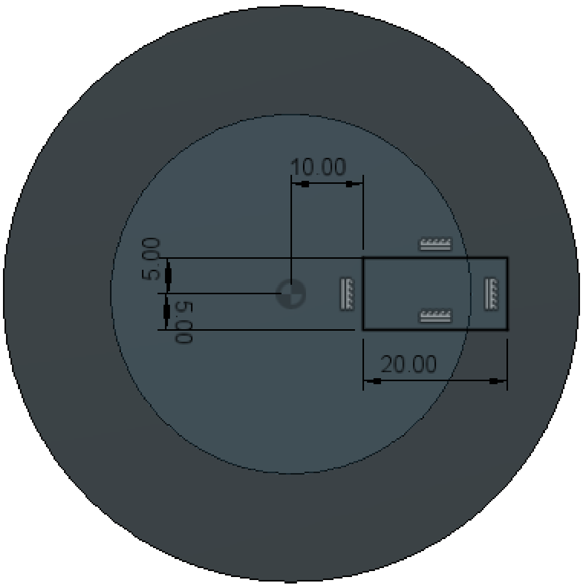

Figure 3.29

The entire contour created on the sketch is selected as a profile (Figure 3.30, a). If only part of the profile is selected (Figure 3.30, b) the cutting operation will achieve completely different results.

In Fusion 360, when the two-dimensional objects of a sketch are intersected by edges lying on the same plane as the sketch, the intersection causes the environment to interpret the two-dimensional objects as composed of multiple surfaces. Each of these surfaces can be independently selected and can participate in the operation of a command for three-dimensional modeling.

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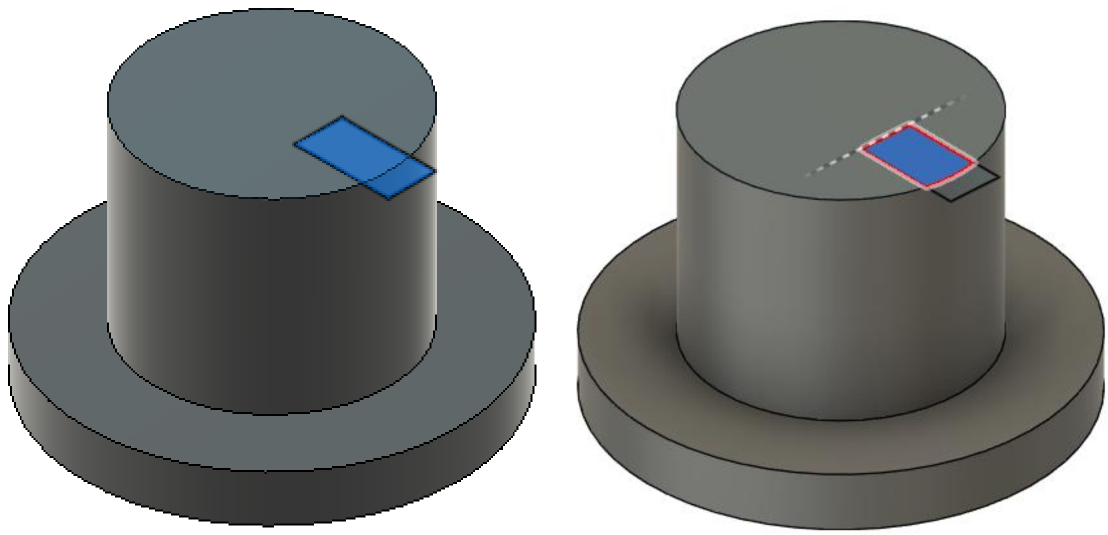

(a) (b)

Figure 3.30

We select one of the rectangle's sides as axis of rotation (Figure 3.31).

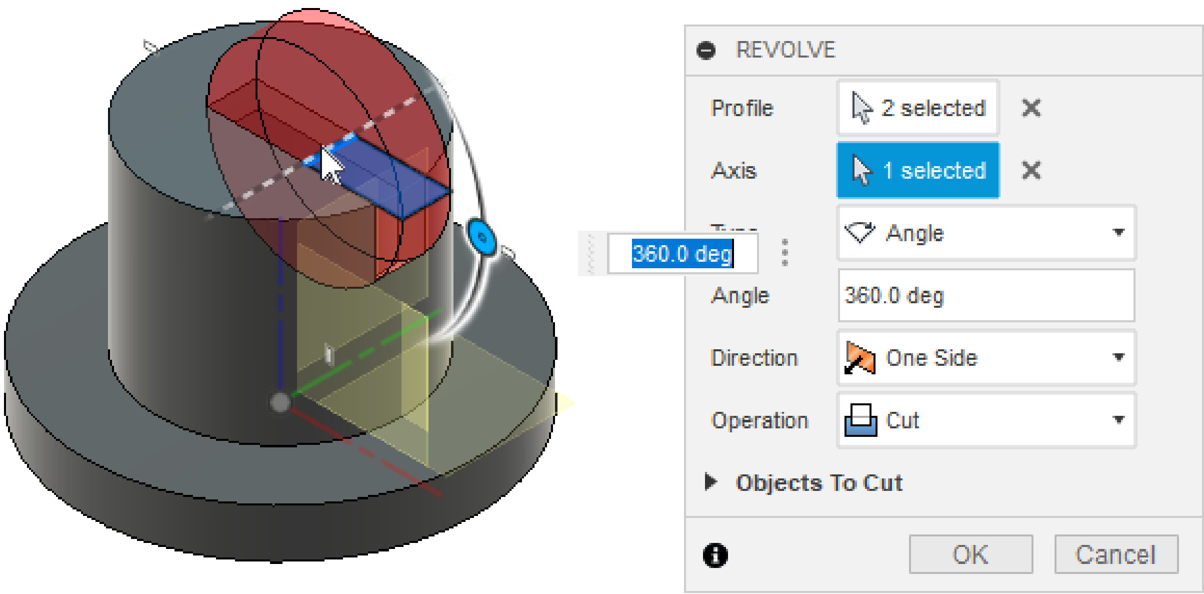

Figure 3.31

In the Revolve command context menu, we select Cut as operation type and remove part of the existing three-dimensional object. The angle of the rotation is set, and the result is a slot in the object (Figure 3.32).

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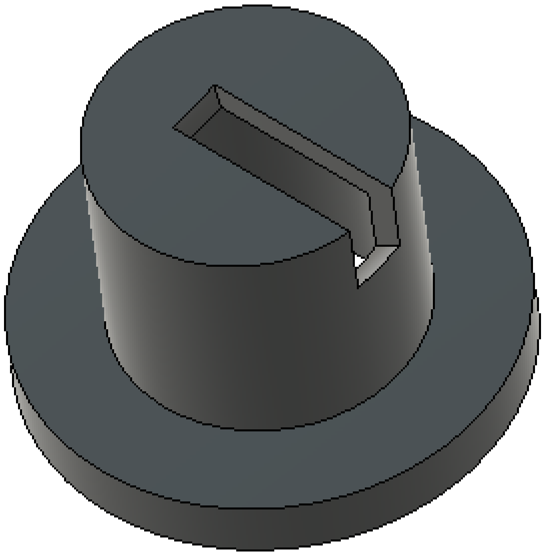

Figure 3.32

Using the Sweep command

With the Sweep command it is possible to create three-dimensional objects, with the sketch of the object's profile following a predefined path in the extruding process.

Let the path that will define the shape of the object be given as a sketch (Figure 3.33).

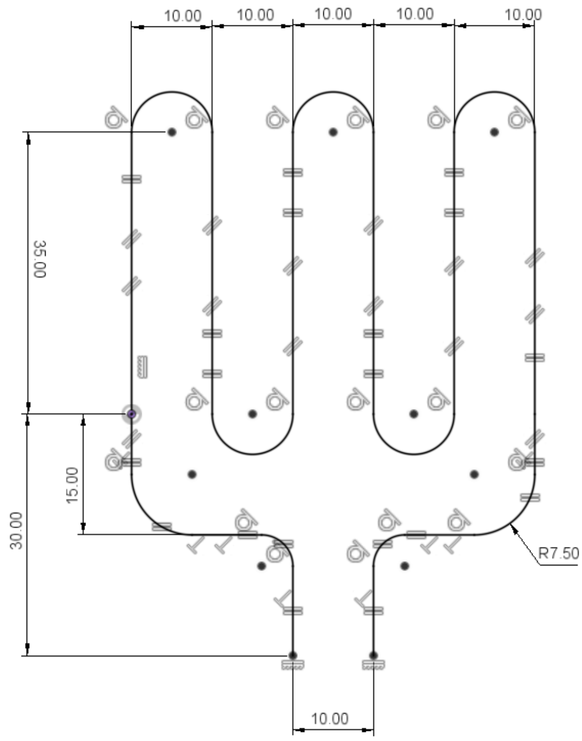

Figure 3.33

In the perpendicular plane of the displayed sketch we will create a new sketch in which the profile of the object will be created. In this case, a circle shall be used as a profile, and its center lies on the path shown above (Figure 3.34).

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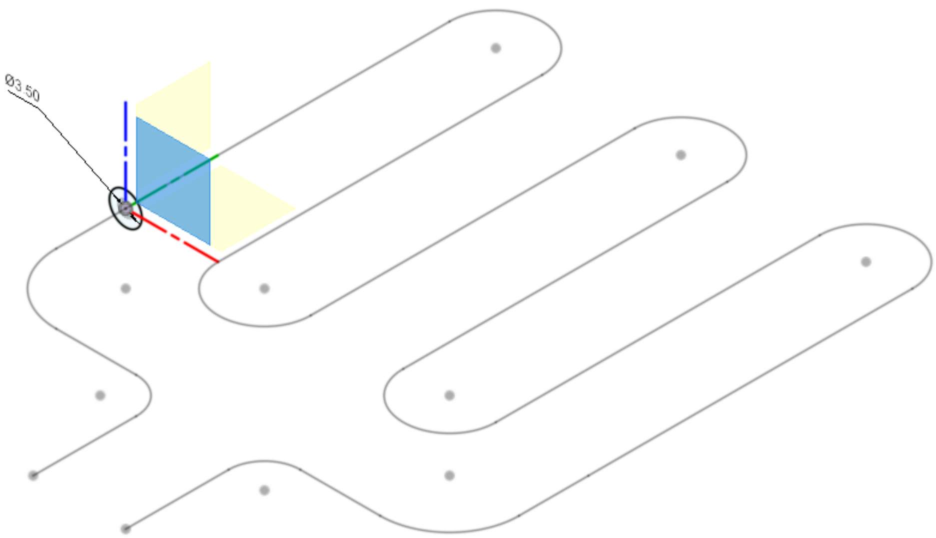

Figure 3.34

The Sweep command is selected from the drop-down menu containing commands for creating new objects (Figure 3.35).

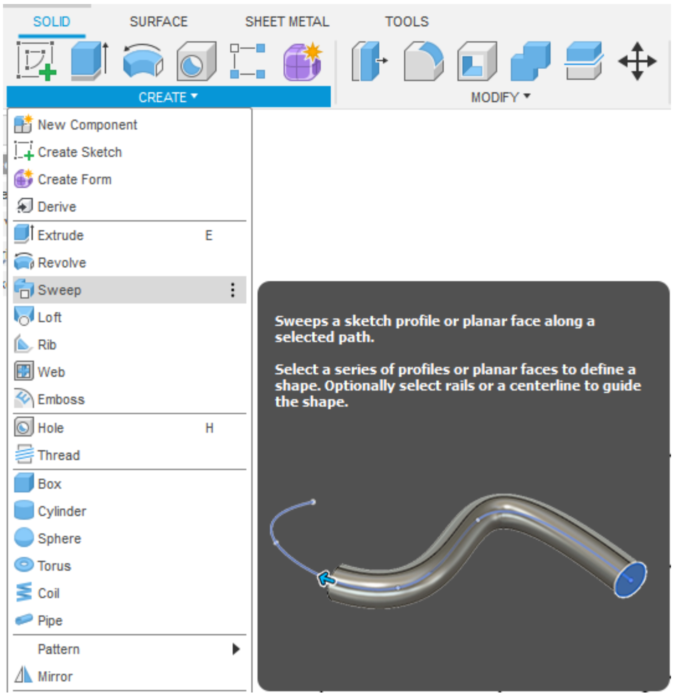

Figure 3.35

After the command is activated, a menu appears in which it is necessary to specify the previously drawn circle as a profile, and the contour created at the beginning is selected as a path that will be used to create the object (Figure 3.36).

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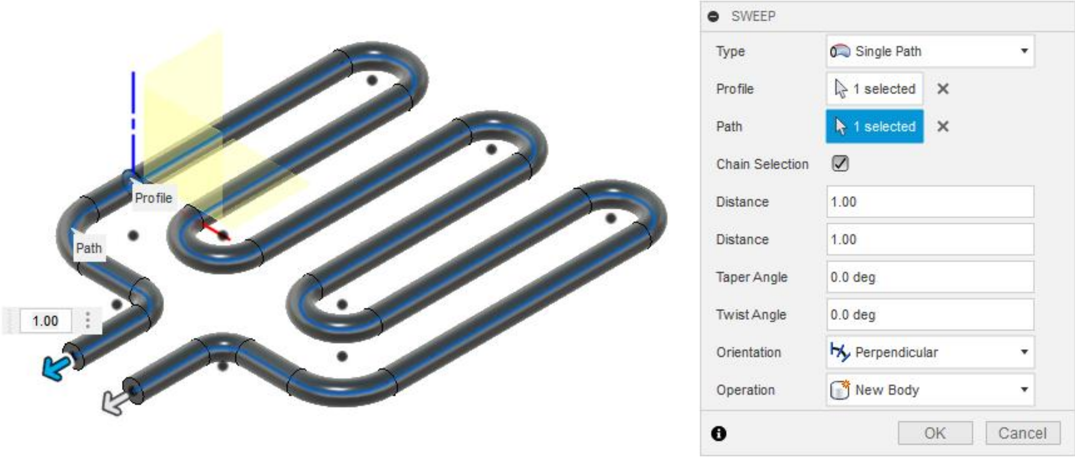

Figure 3.36

In the menu of the Sweep command, the length of extrusion of the profile, the angle of twisting, its orientation to the contour, as well as other parameters can be set.

The object obtained as a result of applying the Sweep command with the selected profile and path is presented in Figure 3.37.

Figure 3.37

The contour for the extrusion path can also be a 3D sketch. Firstly, the drawing of 3D sketches should be allowed in the work environment. One way of creating a 3D sketch is by selecting the 3D Sketch option in the SKETCH PALETTE menu (Figure 3.38, a). As a result of the selection, in the process of drawing objects the mouse cursor changes and allows the drawing to be done in different planes. There is also the possibility to change the angle of the drawn planes in the process of work (Figure 3.38, b). Sketching in 3D space provides flexibility in operation, especially when using the Sweep command. However, it requires some experience that can be gained over time.

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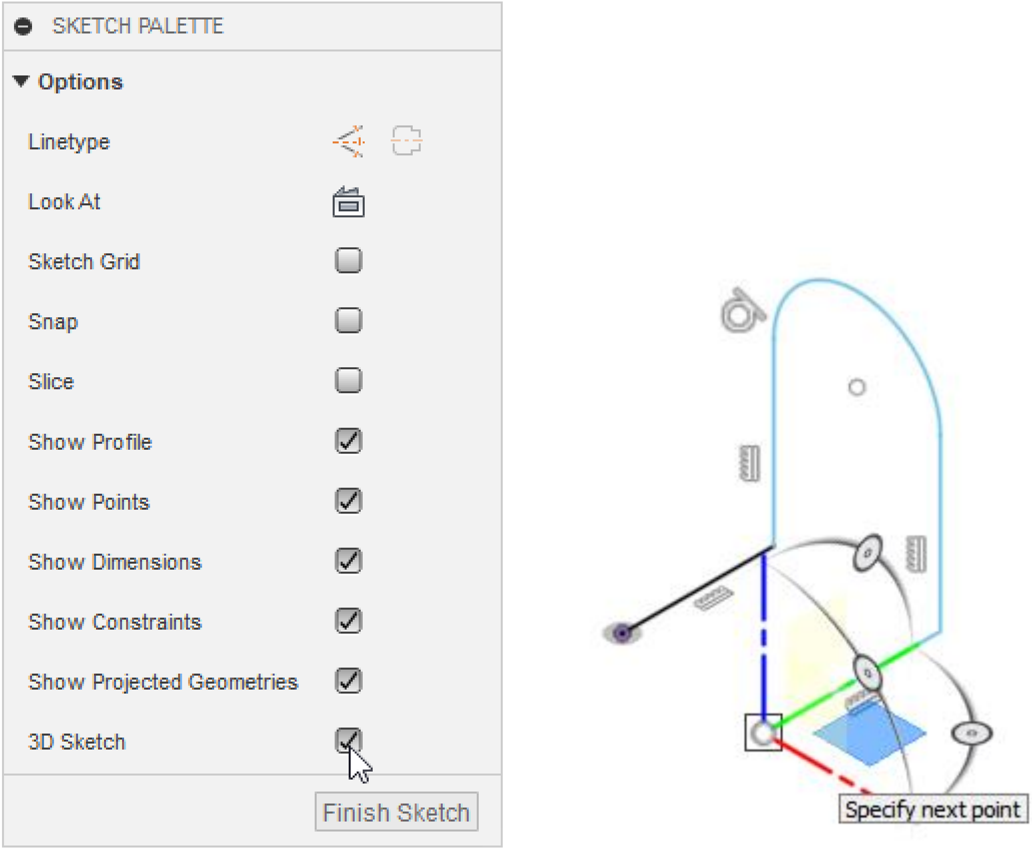

(a) (b)

Figure 3.38

Let us have a contour with the shape shown in Figure 3.39.

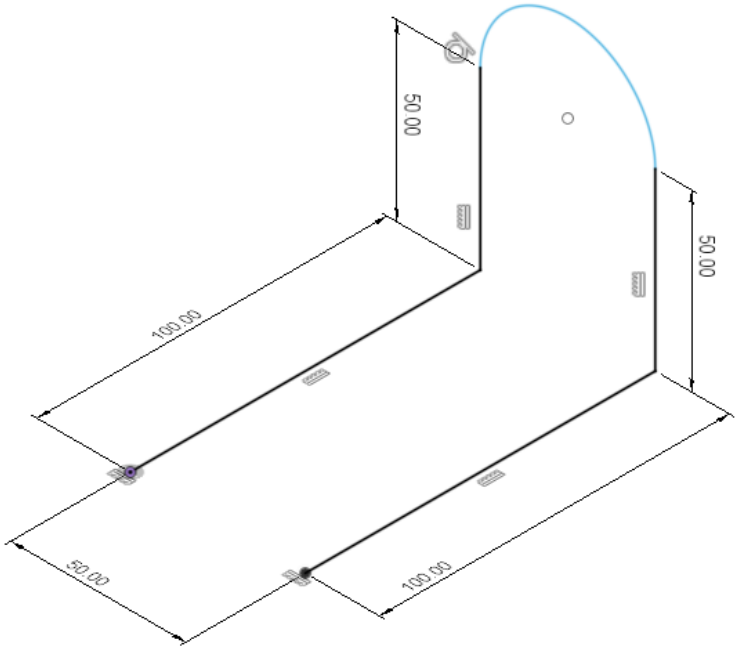

Figure 3.39

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Using the Plane Along Path command (Figure 3.40), which is a supporting command for creating construction geometry, we create a plane perpendicular to the contour shown above.

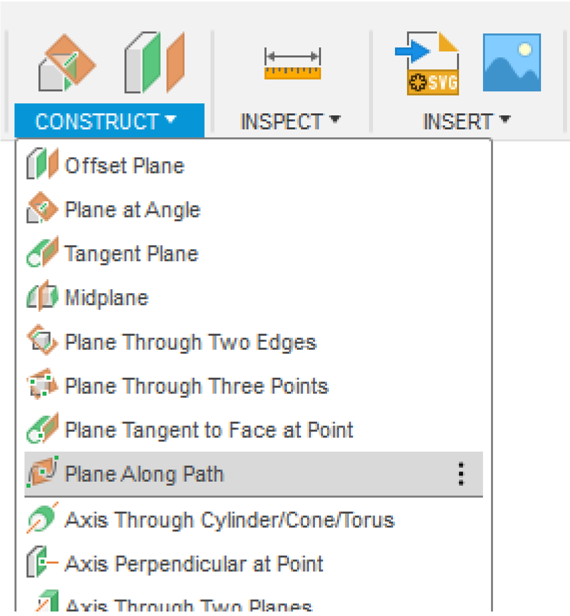

Figure 3.40

The created plane can be positioned in space in such a way as to follow the path and be perpendicular to it at its point of intersection with the contour (Figure 3.41).

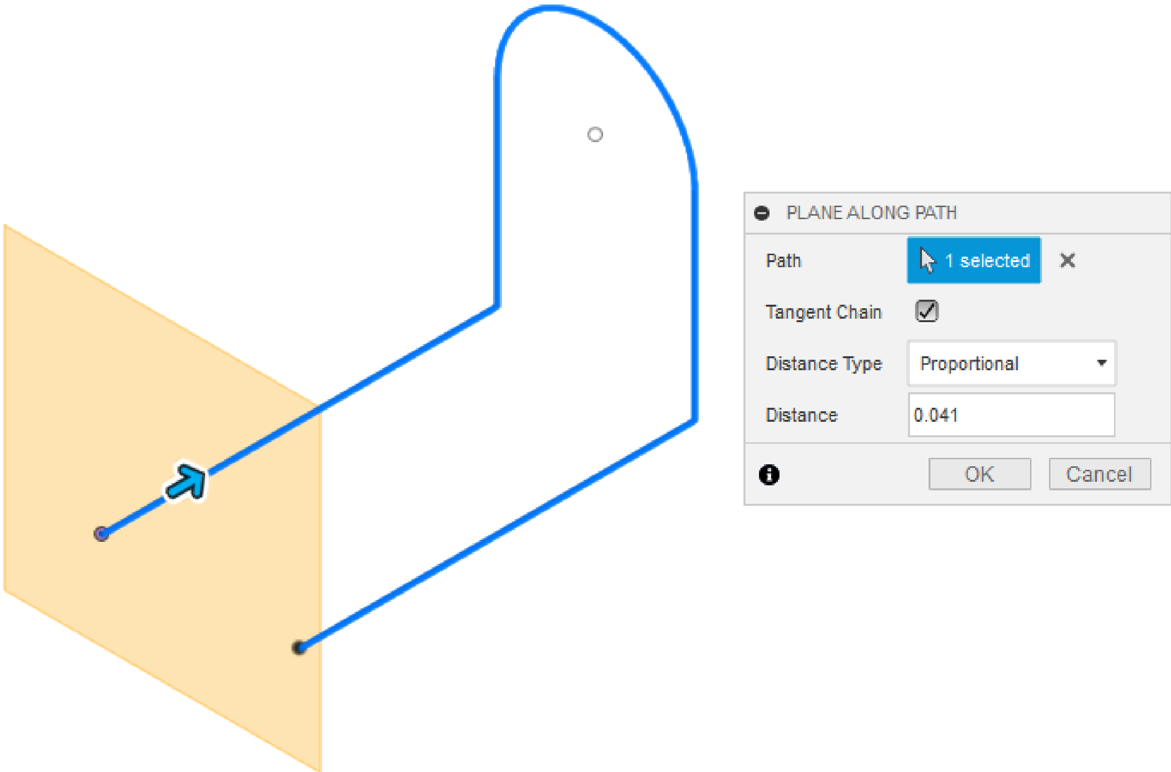

Figure 3.41

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We create a new sketch on the plane. We will draw the profile of the object in this sketch (Figure 3.42).

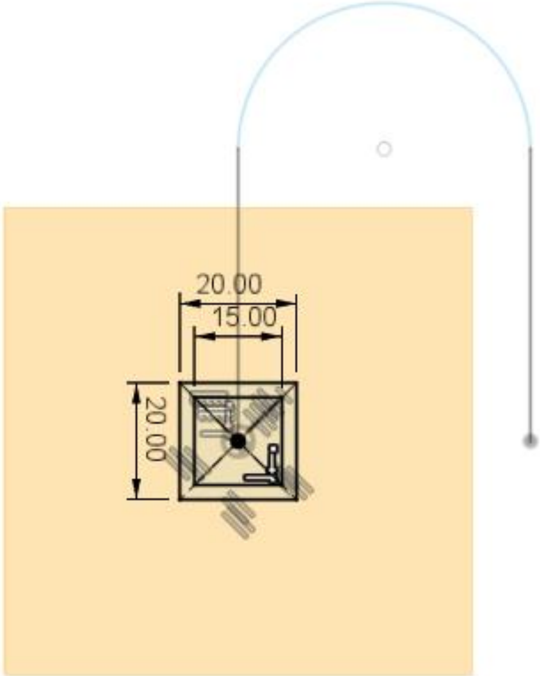

Figure 3.42

Using the sketches of the contour and the object's profile, the Sweep command is used to extrude an object in the three-dimensional space. The extrusion follows the created 3D contour (Figure 3.43).

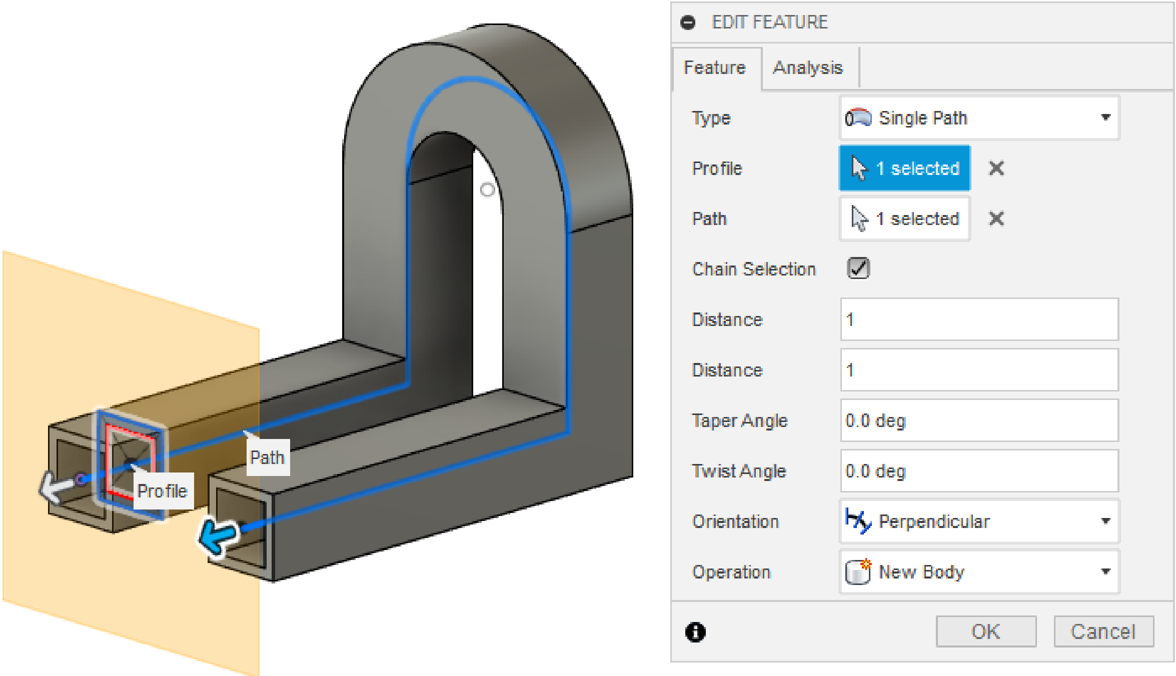

Figure 3.43

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The Sweep command allows us to select the way the command is executed (Figure 3.44, a), as well as the orientation of the selected profile vis-à-vis the path (i.e. contour) along which the extrusion is performed (Figure 3.44, b) - perpendicular to the path or parallel to the profile's initial position.

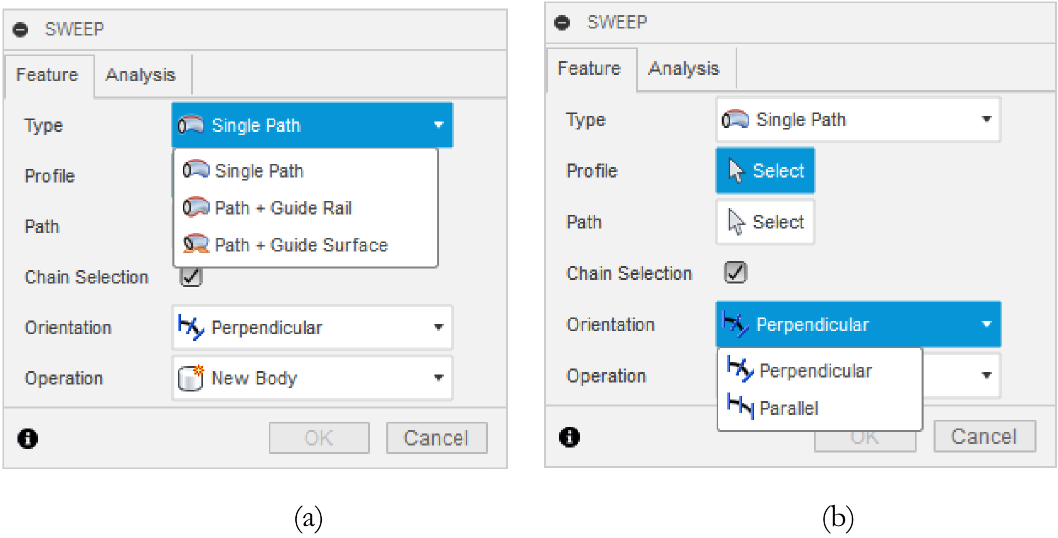

Figure 3.44

Using the Loft command

The Loft command (Figure 3.45) allows for the creation of a three-dimensional object using two or more sketches, in which the object's profiles are defined as it intersects with the planes of the sketches.

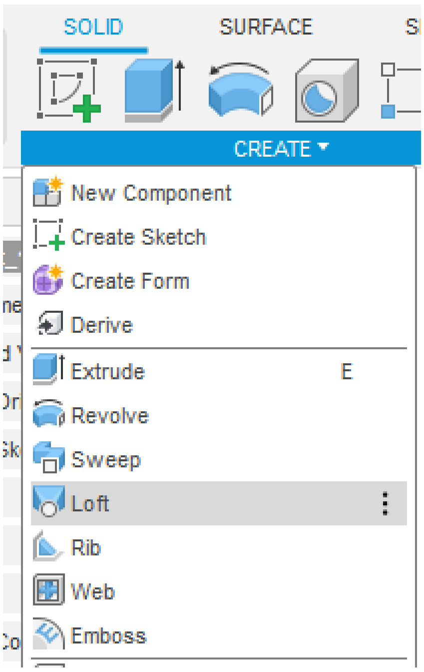

Figure 3.45

Before using the Loft command, it is necessary to create two or more sketches that will be used by the command in the extrusion process.

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For this example, let us create three sketches on three parallel planes. The Offset Plane command (listed in the set of commands for geometry construction) is used for the creation of the parallel planes (Figure 3.46).

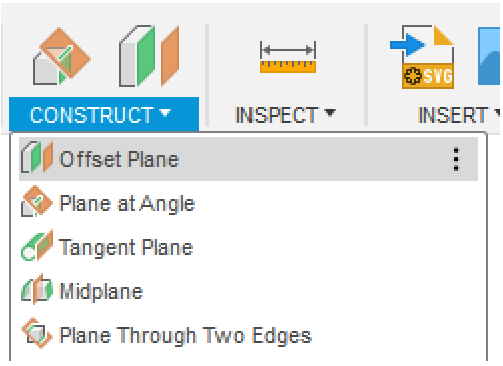

Figure 3.46

When selecting the Offset Plane command, it is necessary to specify a base plane, parallel to which we will create another plane, and then to set the distance between the base plane and the new plane (Figure 3.47).

Figure 3.47

In the same way we create the third parallel plane necessary for the creation of the sketches (Figure 3.48).

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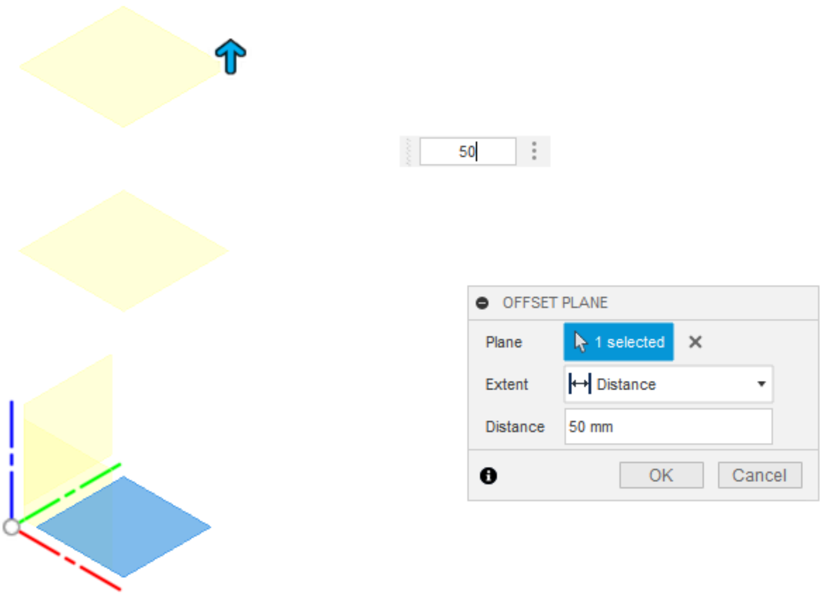

Figure 3.48

Three separate sketches, representing simple geometric shapes, will be created on the prepared parallel planes (Figure 3.49).

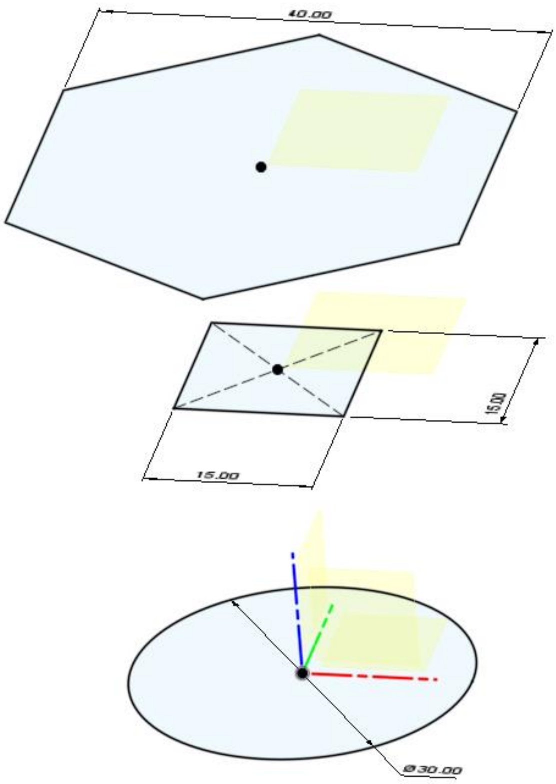

Figure 3.49

The new sketches can be seen in the browser of the current design (Figure 3.50). If any of the sketches is not visible (marked with the sign), then its visibility should be enabled in the browser (it should be designated with the sign).

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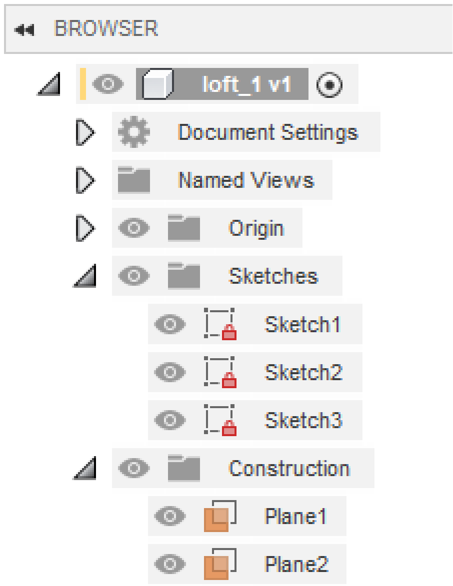

Figure 3.50

Once all the sketches that will be used in the operation are visible, the Loft command can be launched. The profiles of the geometric objects will be selected from the sketches. The command connects the profiles and creates a three-dimensional body (Figure 3.51).

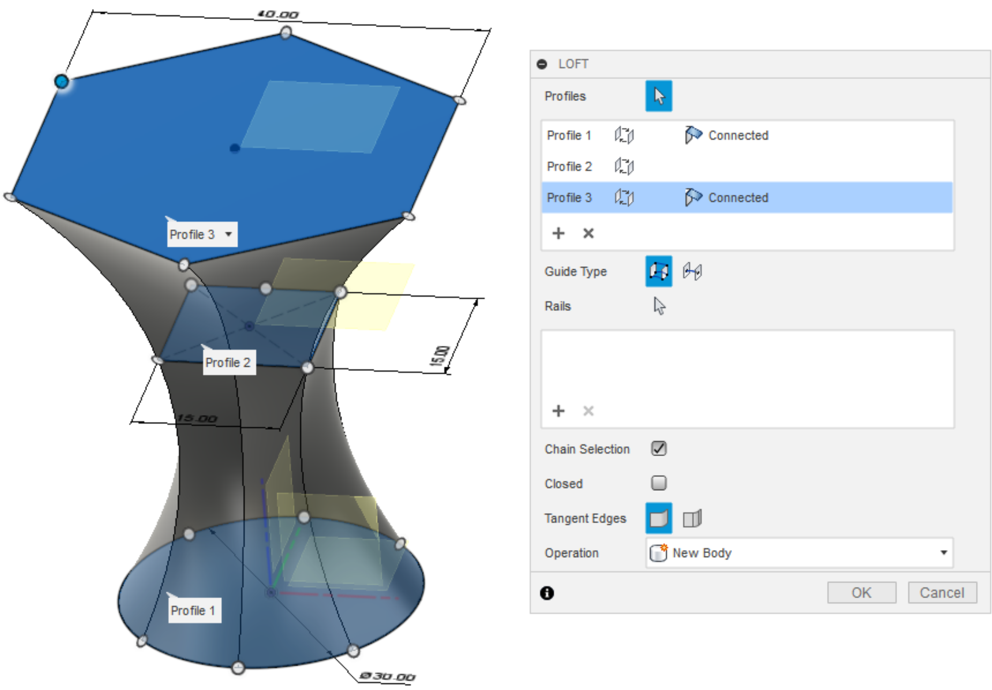

Figure 3.51

The resulting three-dimensional object has a shape that is controlled by the profiles of the geometric objects from the sketches. Fusion 360 chooses automatically how to perform the extrusion so as to ensure smooth connection of the individual profiles (Figure 3.52).

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Figure 3.52

The Loft command also allows us to control the extrusion form by defining a guiding profile. Let us look at this functionality in the Loft command. We will initially create two parallel planes (Figure 3.53).

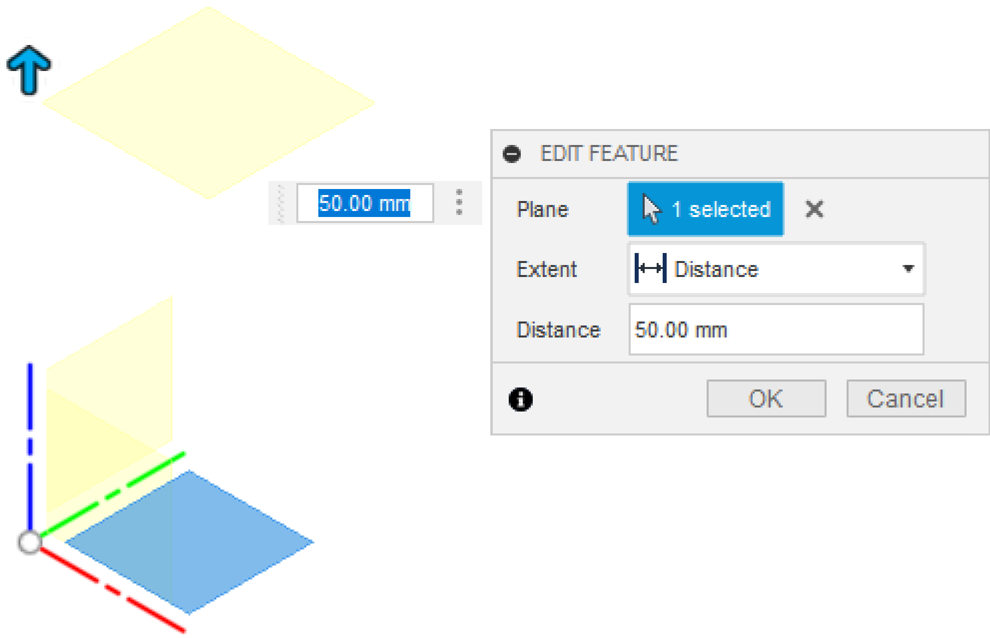

Figure 3.53

Let us draw two circles on these planes (Figure 3.54).

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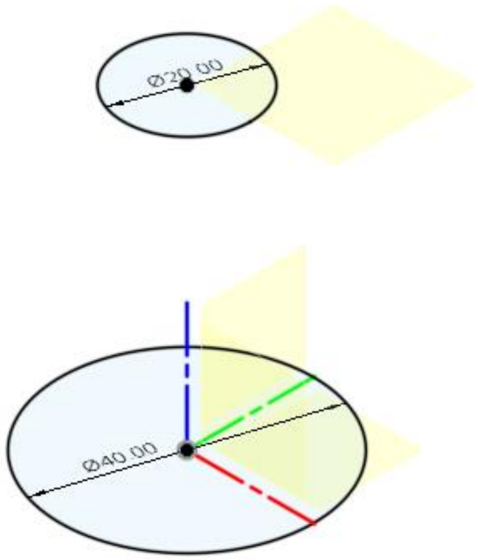

Figure 3.54

When the Loft command is used and both circles have been selected, a three-dimensional body with the shape shown in Figure 3.55 will be obtained. Fusion 360 automatically creates an object with a conical surface defined by the dimensions of the two circles.

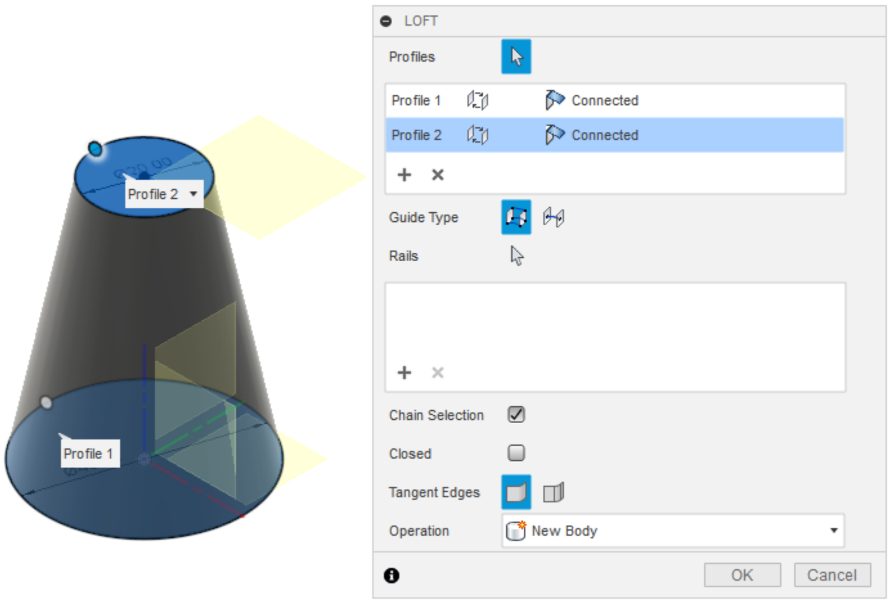

Figure 3.55

It is possible, however, for the generated 3D object to follow a shape chosen by the user. For this purpose, additional sketches need to be provided to define the contour that should be followed by the Loft command. The additional sketches are created in planes perpendicular to the planes defining the object's profile. In Figure 3.56 an additional contour is presented. It is added as a sketch to the existing sketches of the two circles.

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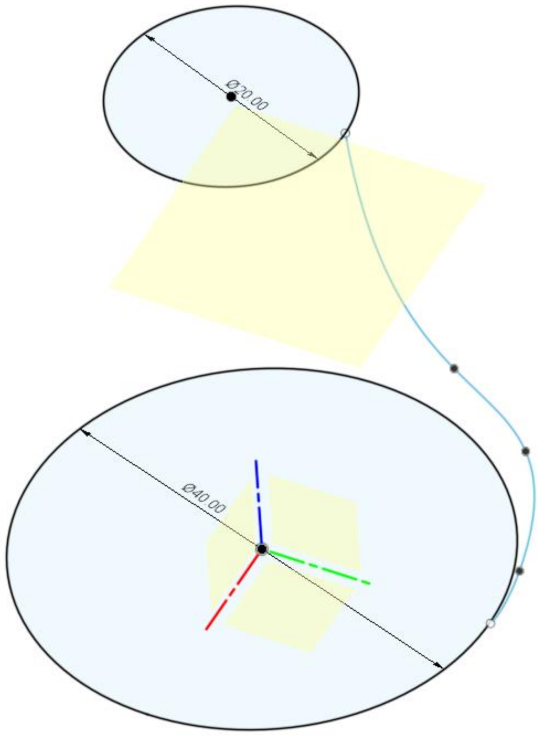

Figure 3.56

When executing the Loft command, it is necessary to select this contour in the Rails field in the command menu. The contour specified in this way will then be followed in the creation of the three-dimensional object and will define the object's resulting shape (Figure 3.57).

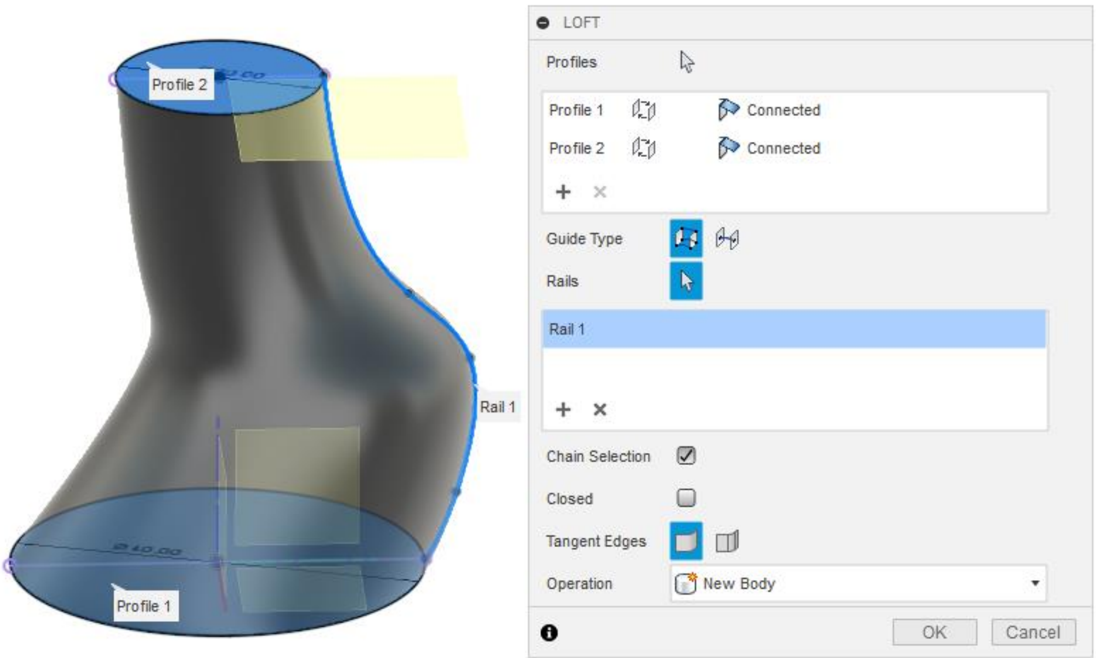

Figure 3.57

Let us edit the sketch of this contour. This is done in the Timeline by right-clicking on the guide rail and selecting the Edit Sketch command from the menu (Figure 3.58).

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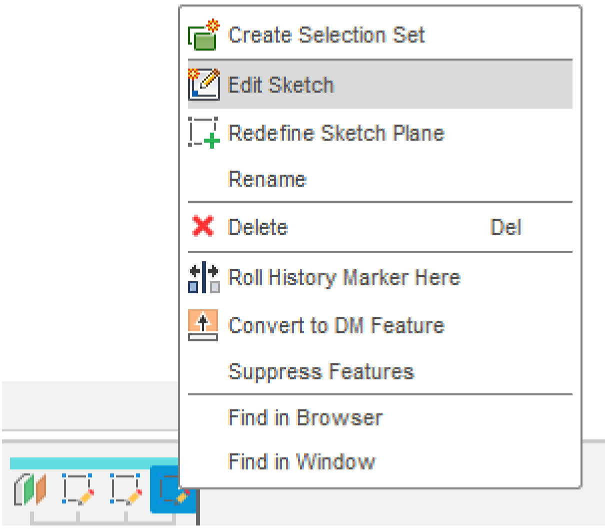

Figure 3.58

A mirror image of the existing guide rail will be added to the sketch to obtain the shape shown in Figure 3.59.

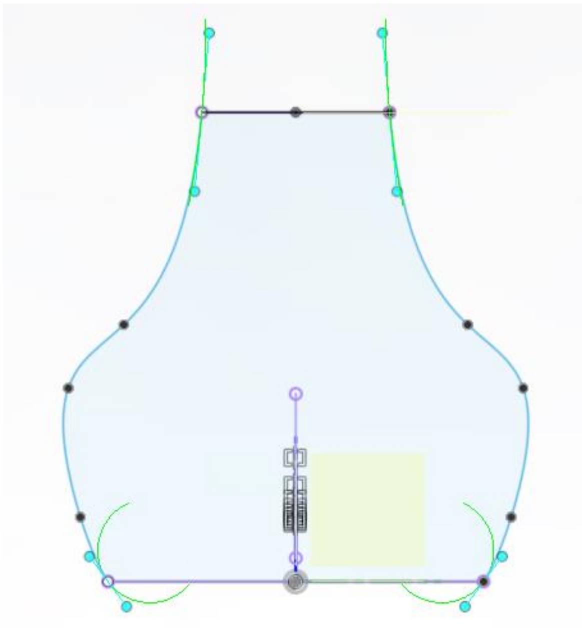

Figure 3.59

In the Loft dialog, in the Rails field, we will add the new contour to achieve symmetry of the three-dimensional object (Figure 3.60).

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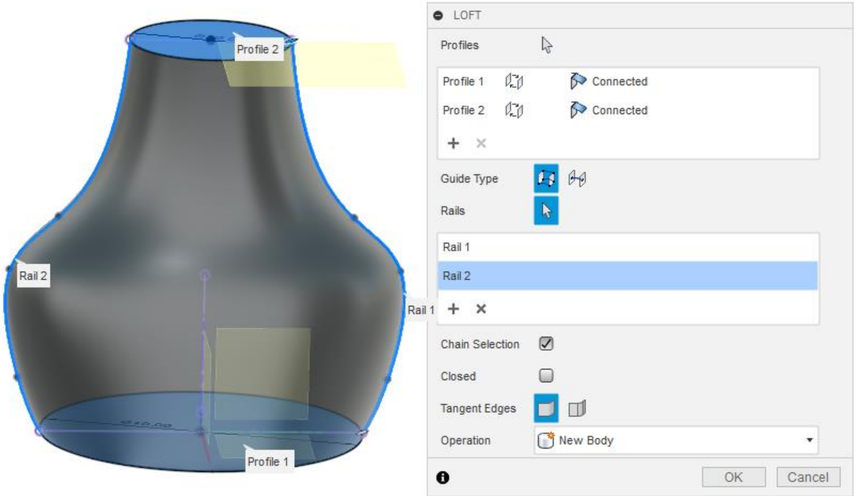

Figure 3.60

The shape of the object is controlled by the selected guide rails. Following the defined constraints of the contours, Fusion 360 seeks to create an object with a smoothly changing shape (Figure 3.61). If additional changes need to be made in the shape, they can easily be obtained by adding new profile sketches of intersections or guiding rails.

Figure 3.61

Using the Rib and Web commands

The Rib and Web commands are used to create construction features that increase the strength of volumetric or hollow bodies.

- The Rib command creates construction reinforcing ribs. This is done by using a sketch drawn on a plane. We create the rib in a direction parallel to the sketch plane. The dimensions of the rib are limited by the surfaces intersecting the sketch, as well as by the settings of the command itself.

Below is a three-dimensional object (Figure 3.62, a) on which a reinforcing rib will be built using the Rib command (Figure 3.62, b).

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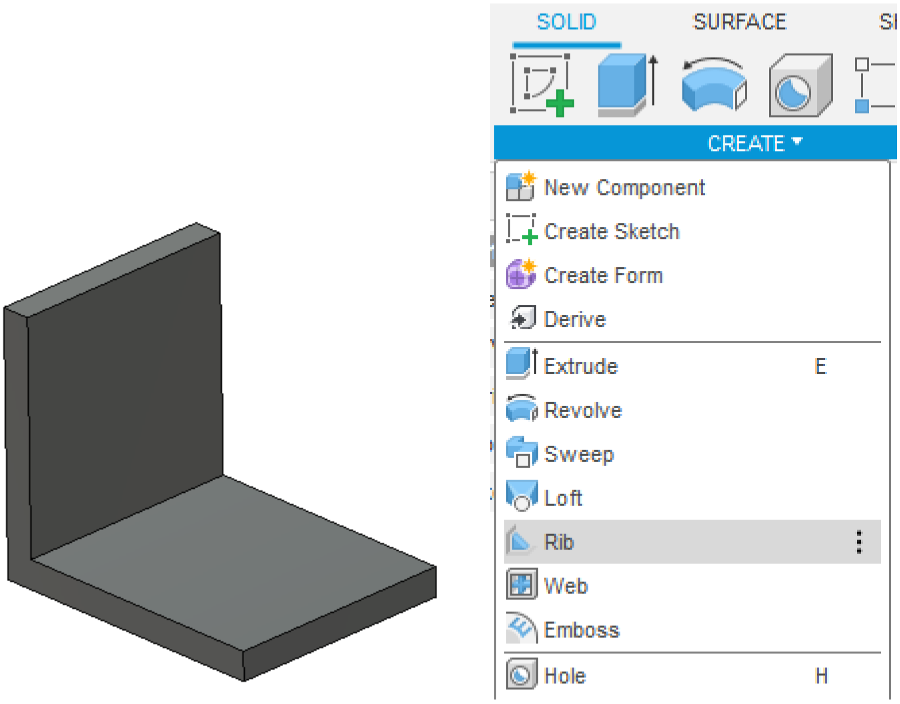

(a) (b)

Figure 3.62

Before using the Rib command it is necessary to draw a sketch defining the general rib location. To create the sketch, an auxiliary plane is drawn, which passes through the middle of the three-dimensional object. To create this plane, the Midplane command is used (Figure 3.63).

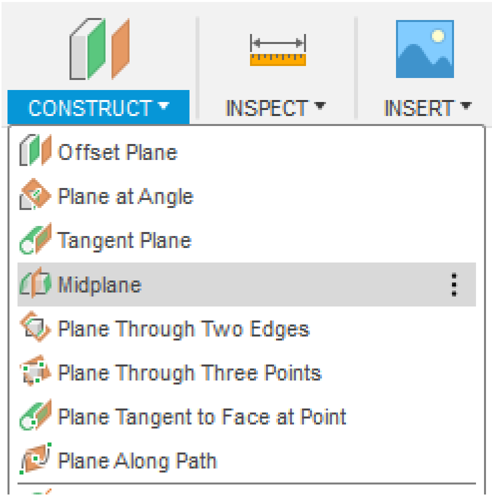

Figure 3.63

After clicking on the Midplane command, the two surfaces between which the construction plane should be created need to be specified (Figure 3.64).

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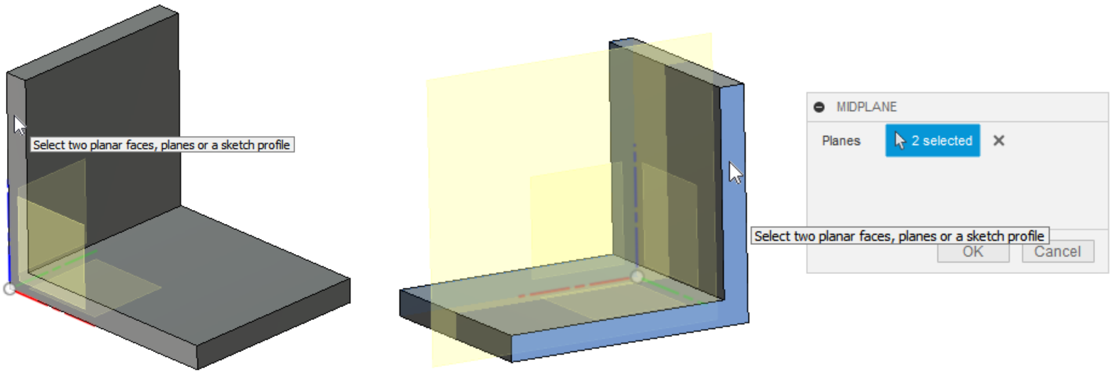

Figure 3.64

We draw a sketch on the created plane, and we create the profile of the extruded rib in this sketch. In Figure 3.65 the profile is set with a straight line, but it can be curved or take any arbitrary shape.

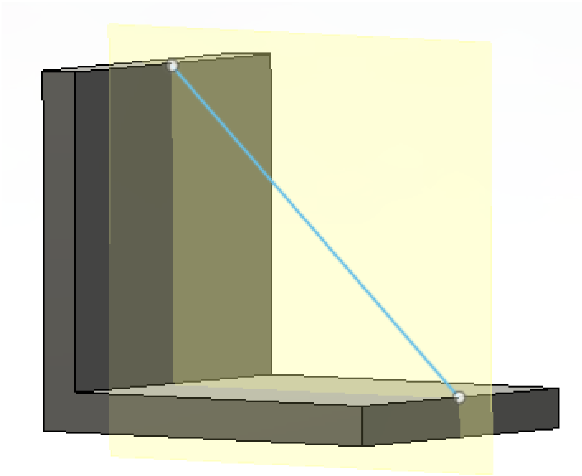

Figure 3.65

After launching the Rib command, we select the line and specify the thickness of the rib, as well as the way of its creation - one-directional or symmetrical to the sketch plane (Figure 3.66).

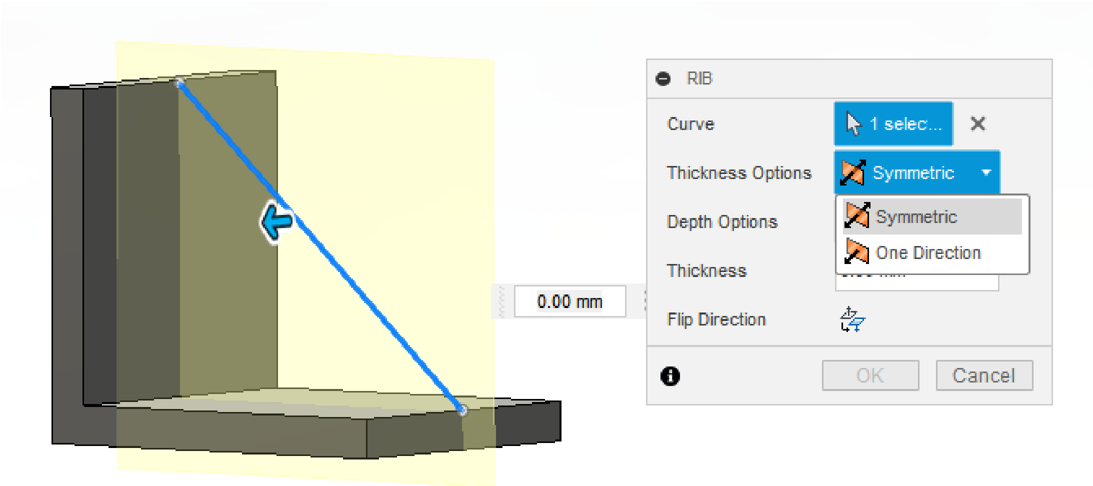

Figure 3.66

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The resulting reinforcing rib is located along the sketch's plane and reaches the faces of the 3D object (Figure 3.67).

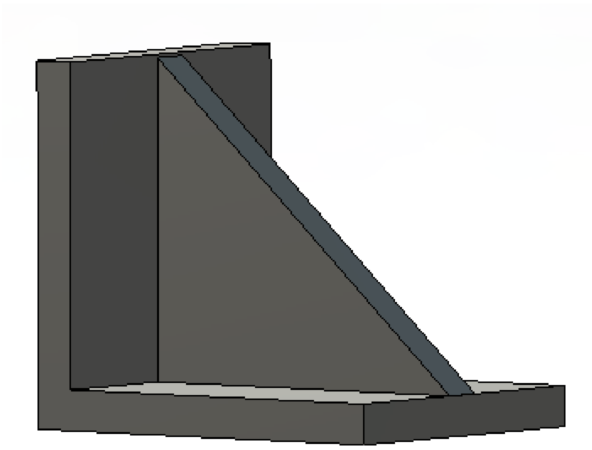

Figure 3.67

Fusion 360 allows for adjusting the rib's geometric dimensions. The depth of the rib can be specified in Depth Options (Depth is selected from the drop-down menu) (Figure 3.68).

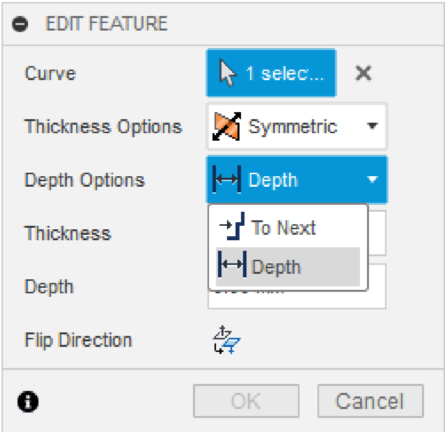

Figure 3.68

When this option is selected, the dialog menu changes, and a field appears in which the depth of the rib can be entered (Figure 3.69).

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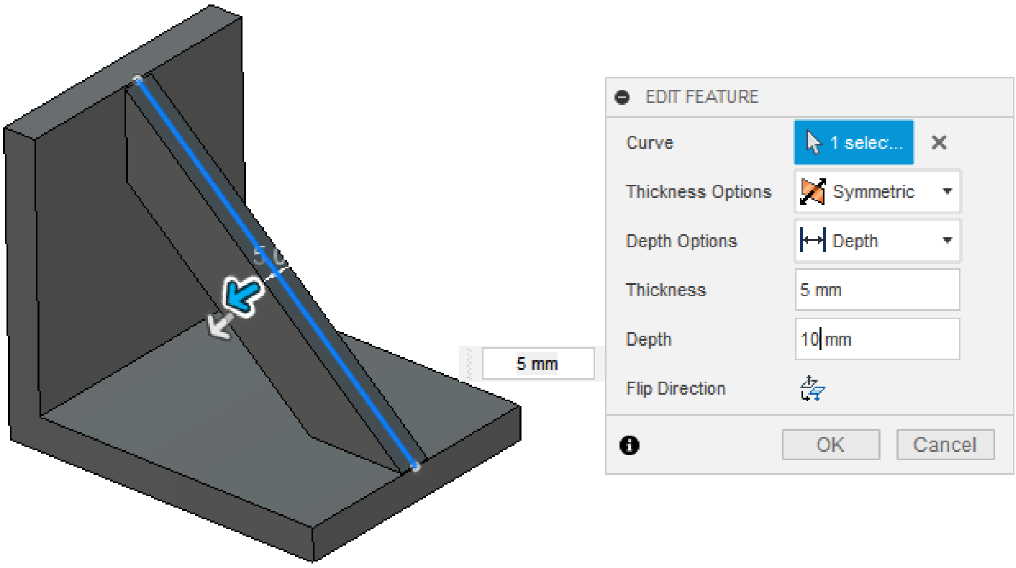

Figure 3.69

As a result, the rib acquires the shape presented in Figure 3.70.

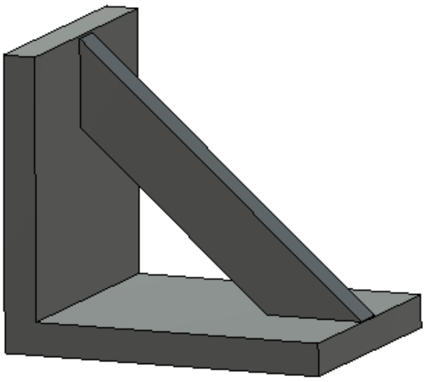

Figure 3.70

- The Web command allows for creating a grid of elements that enhance the construction of the object. We create the grid with the help of a sketch and we extrude it in a direction perpendicular to the sketch plane. The Web command is located in the Create panel in the toolbar (Figure 3.71).

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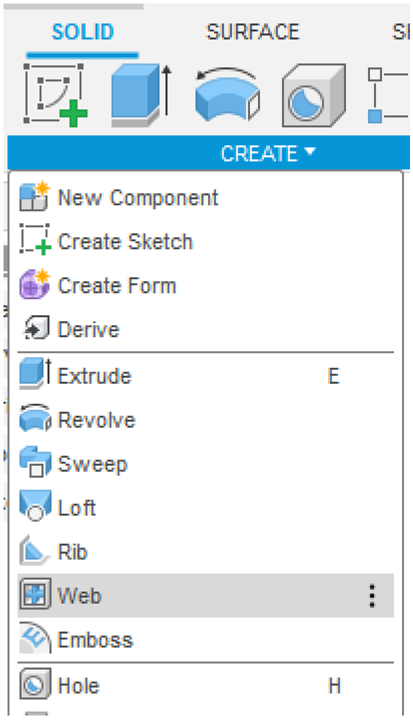

Figure 3.71

The 3D object in Figure 3.72 demonstrates the functioning of the Web command.

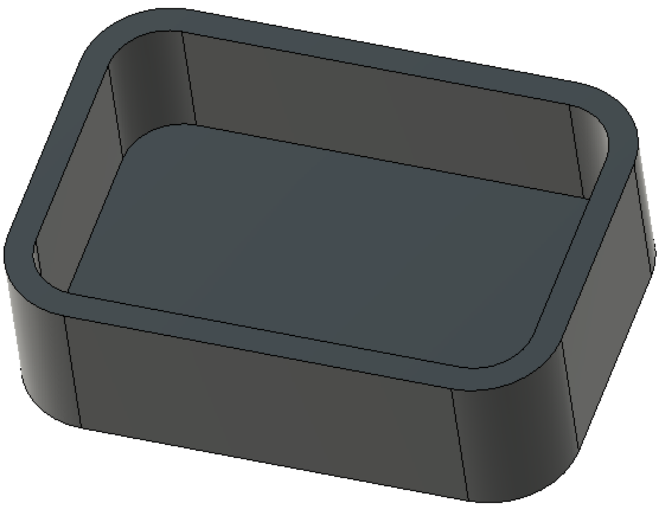

Figure 3.72

The plane on which the sketch controlling the grid shape will be created is chosen to be the surface of the given object (Figure 3.73, a). The sketch itself represents several lines perpendicular to one another (Figure 3.73, b). In this case, the lines of the sketch are drawn in such a way that they do not touch the three-dimensional object itself.

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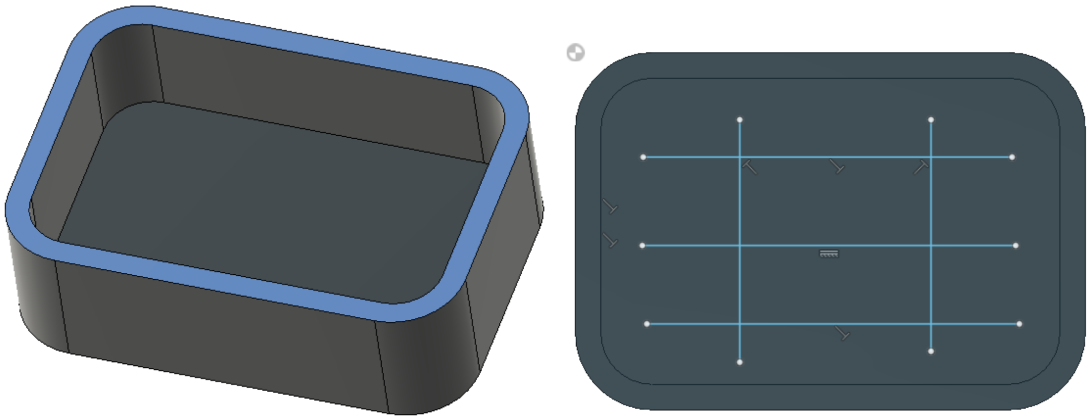


(a) (b)

Figure 3.73

After launching the Web command, the lines from the sketch are selected sequentially, and the grid's thickness and its location vis-à-vis the selected lines are specified (Figure 3.74).

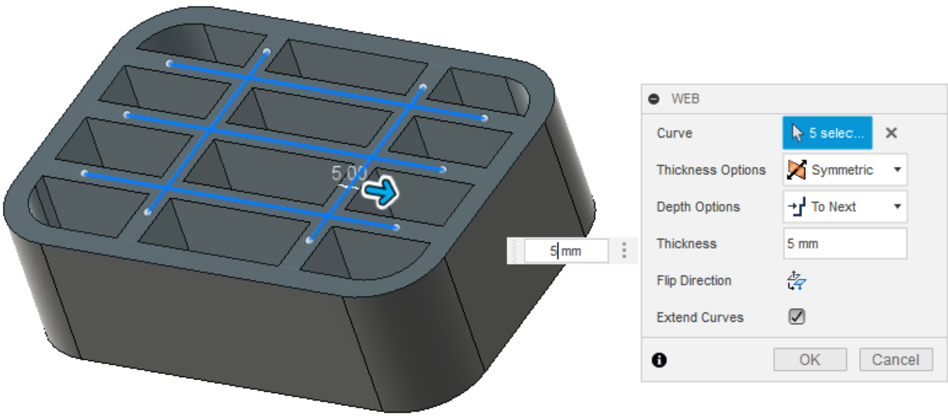
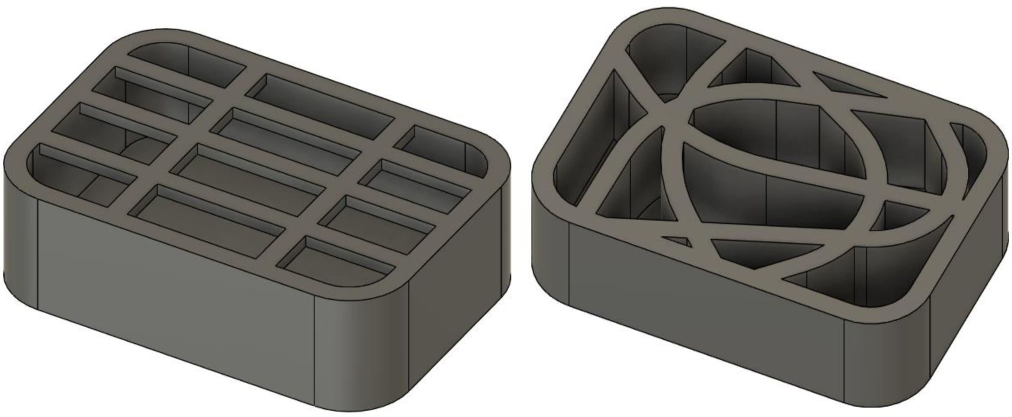


Figure 3.74

With the Web command it is also possible to specify the depth of the grid (Figure 3.75, a). The generated web can have any shape (Figure 3.75, b).



(a) (b)

Figure 3.75

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Using the Emboss command

The Emboss command (Figure 3.76) is used to place concave or convex objects on a surface. Most often, it is applied to wrap text around a 3D object.

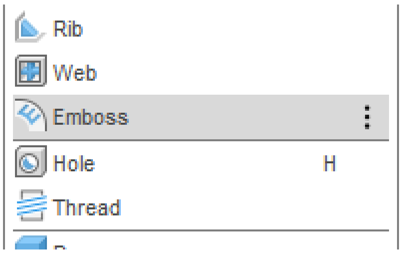

Figure 3.76

Let us assume that a sketch has been created on one of the faces of a three-dimensional object (Figure 3.77). The created sketch contains the text, presented in the figure below.

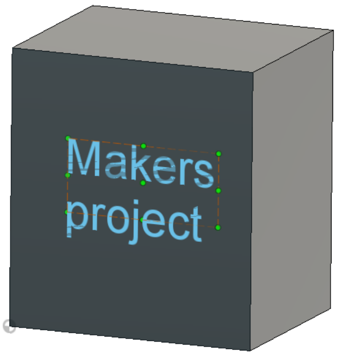

Figure 3.77

By using the Emboss command, we select the created text, and we specify the surface on which the command will be applied and the depth of the convex or concave inscription. In the example below, we have chosen a convex inscription (Figure 3.78).

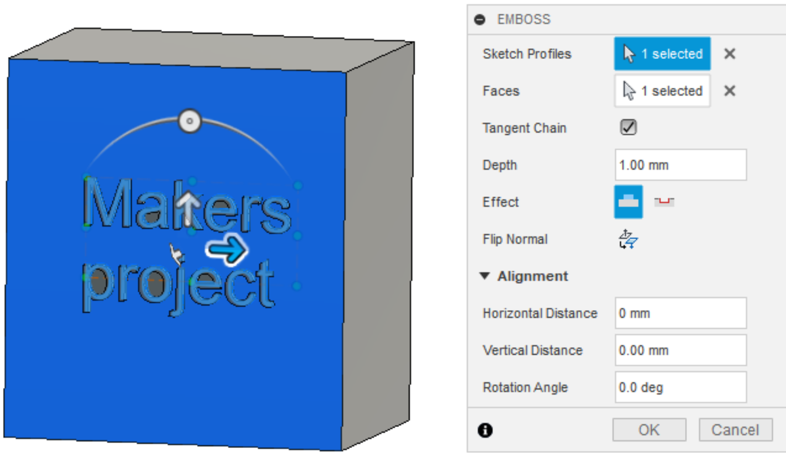

Figure 3.78

After executing the command, the desired convex inscription appears on the object (Figure 3.79)

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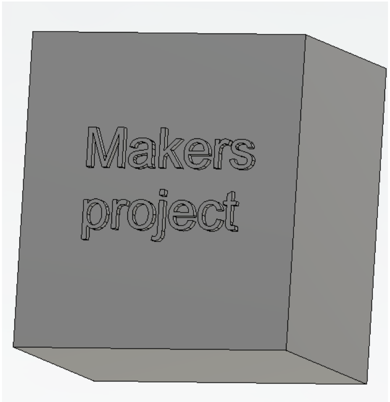

Figure 3.79

Text can be placed on any surface, not just flat surfaces. Let us use a cylindrical object instead (Figure 3.80).

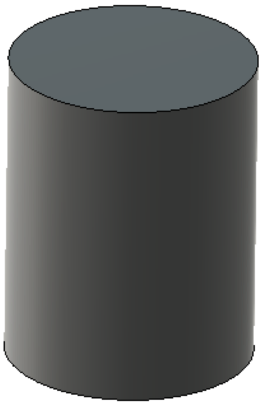

Figure 3.80

First, we use the Offset Plane command  from the set of Construction commands. A plane will be created in a direction parallel to one of the model's main planes, and located outside the body (Figure 3.81).

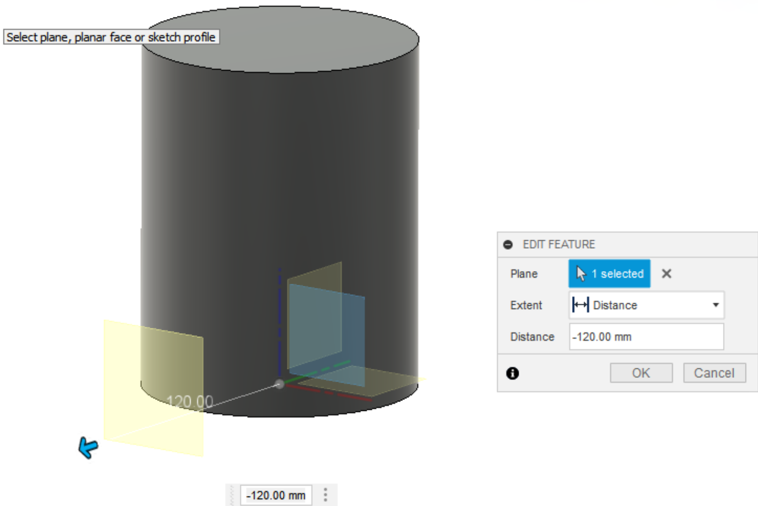

Figure 3.81

A sketch will be drawn on the created plane, containing the inscription presented in Figure 3.82.

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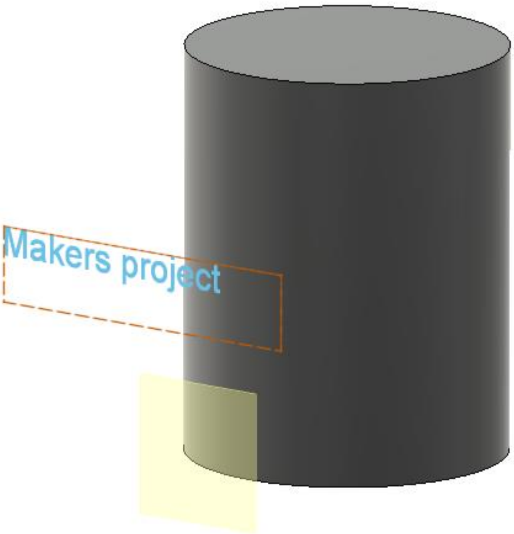

Figure 3.82

Then, by using the Emboss command, the inscription will be wrapped on the side face of the cylindrical object (Figure 3.83).

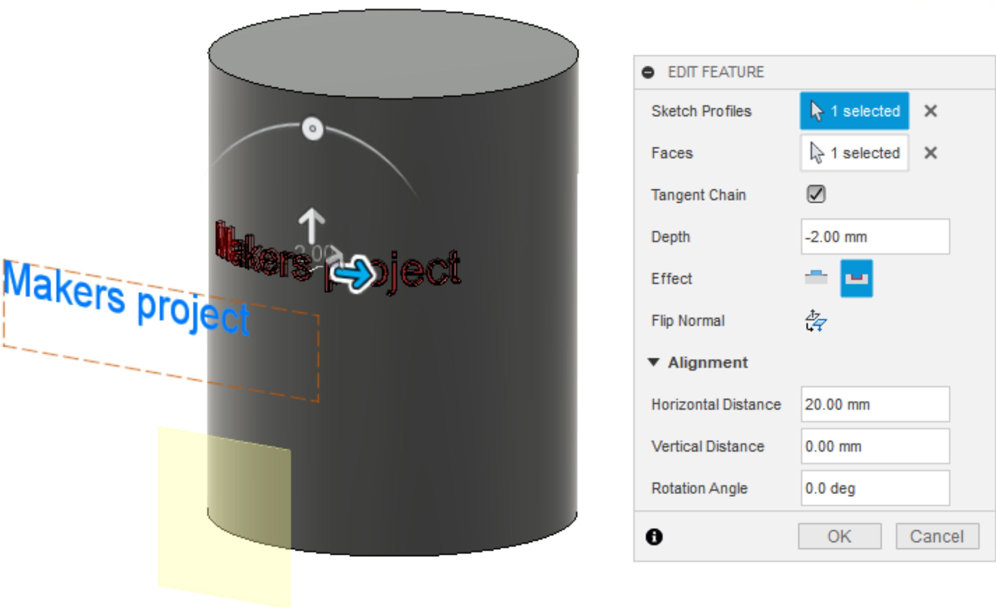

Figure 3.83

The Emboss command allows for controlling the position of the caption on the target surface and the angle of rotation, without the need to change the 2D text on the sketch. After applying all the parameters of the command, the concave inscription shown in Figure 3.84 is obtained.

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Figure 3.84

Using the Hole command

The Hole command (Figure 3.85) is a specialized command for creating different types of holes.

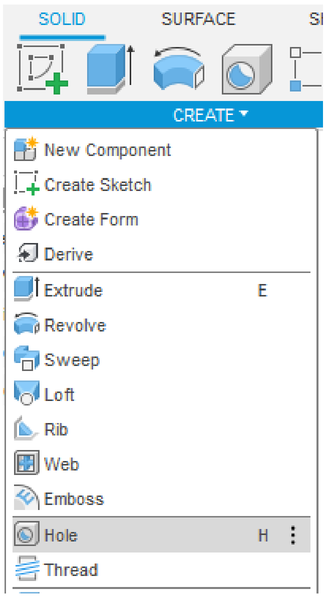

Figure 3.85

The command allows for configuring the type of hole - open-end hole, blind hole, a hole with or without thread. It also specifies the way in which the hole is expected to be drilled. The Hole command does not need a 2D sketch for its implementation. When using it, it is necessary to specify the surface of the three-dimensional body on which the hole will be placed. In Figure 3.86, after starting the Hole command, the upper body surface has been selected as the base on which the hole will be implemented.

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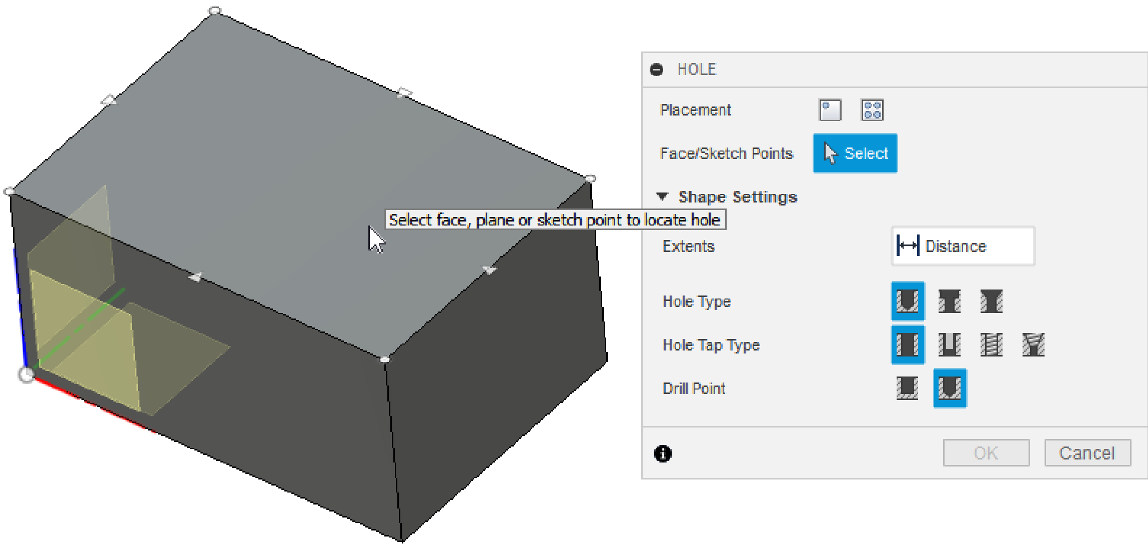

Figure 3.86

From the context menu we select the diameter, the depth, and the type of the hole. We can also set the angle at the drill tip with which the hole is expected to be drilled. If it is a blind hole, this angle appears in the shape of the hole (Figure 3.87).

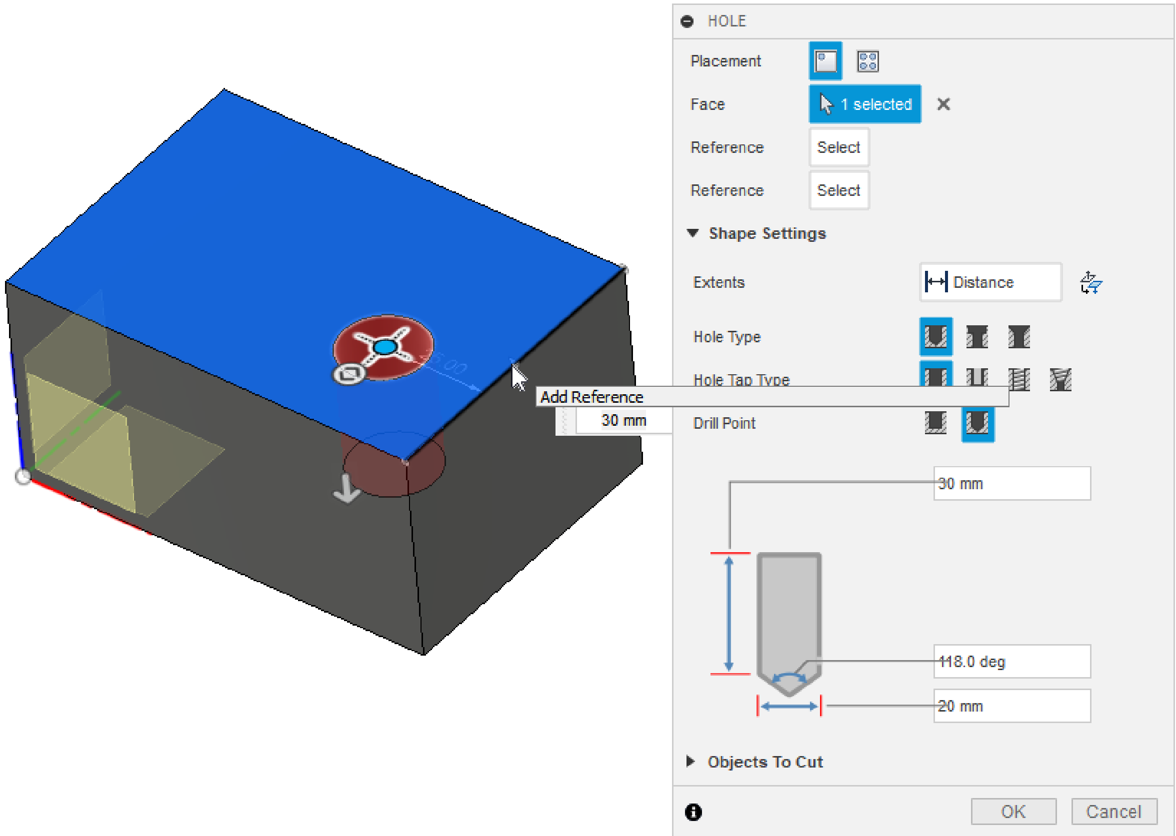

Figure 3.87

The position of the hole on the surface can be specified using the Reference feature in the dialog menu. In this case, it is necessary to indicate the edge of the three-dimensional body lying on the

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face selected for placing the hole. The Hole command allows for selecting two support edges in order to define the exact hole position (Figure 3.88).

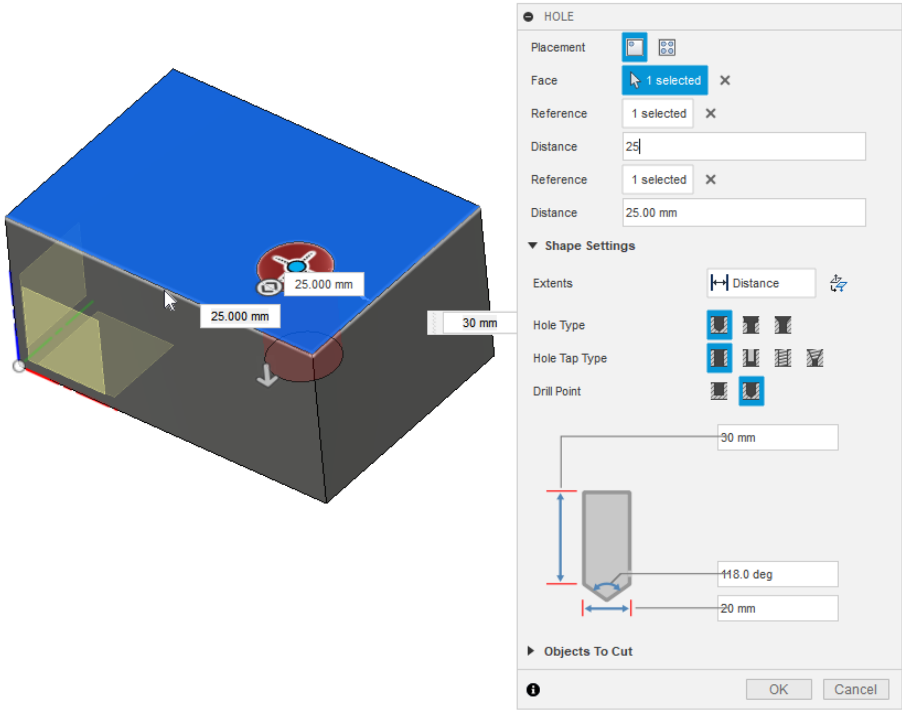

Figure 3.88

To demonstrate the Hole command, we can create a section view of the 3D object. To do this, we will select the Section Analysis command from the INSPECT menu (Figure 3.89).

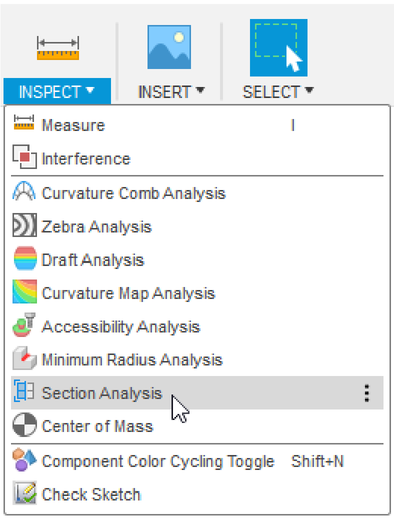

Figure 3.89

We create the section view by selecting a parallel plane. By dragging with the mouse or setting an offset we can cut the body at the desired place (Figure 3.90).

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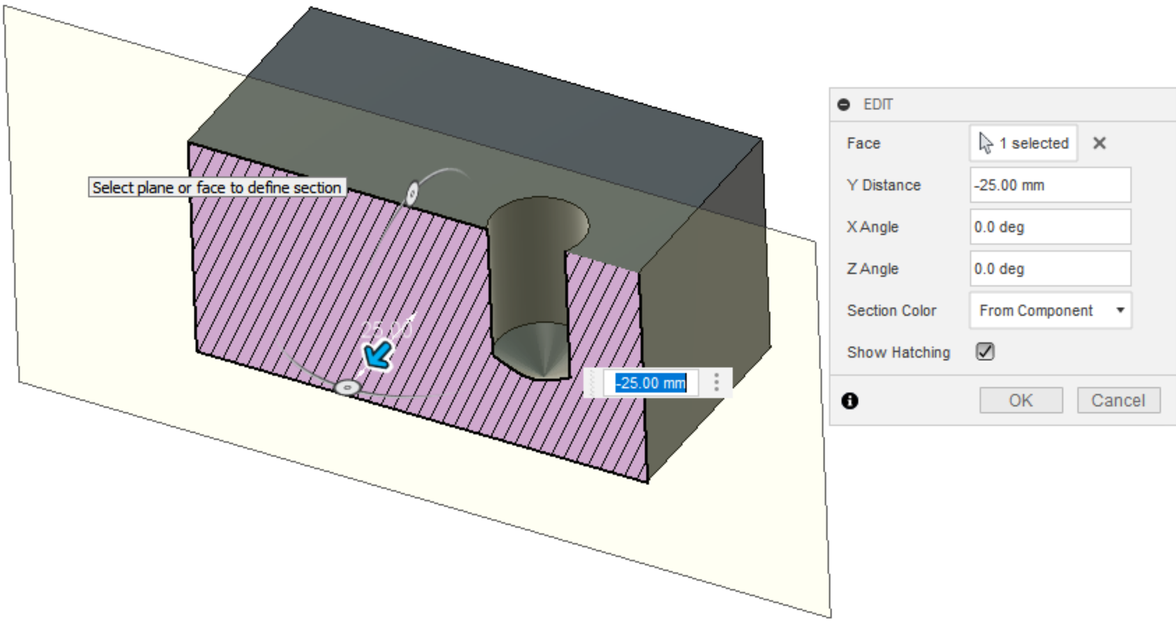

Figure 3.90

The hole shape and its depth can be seen in the created section. If the created hole does not meet our expectations, it can be easily edited again by running the Hole command from the Timeline. The section view appears in the browser menu under Analysis (Figure 3.91).

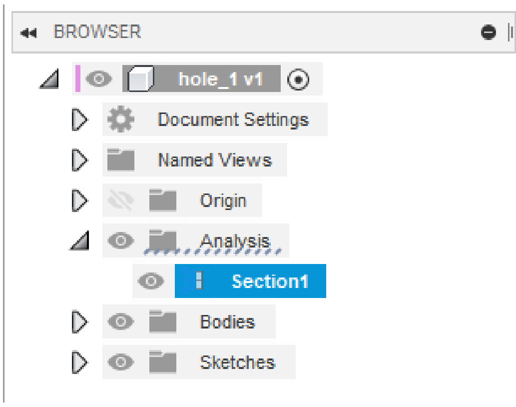

Figure 3.91

The section view can be hidden by deactivating the visualization badge  with the mouse. When the visualization badge takes the following form: , the section view will be hidden from the work area.

With the Hole command, it is also possible to create threaded holes in objects. To do this, we can select an option for creating a threaded hole from the command context menu and configure its properties (Figure 3.92).

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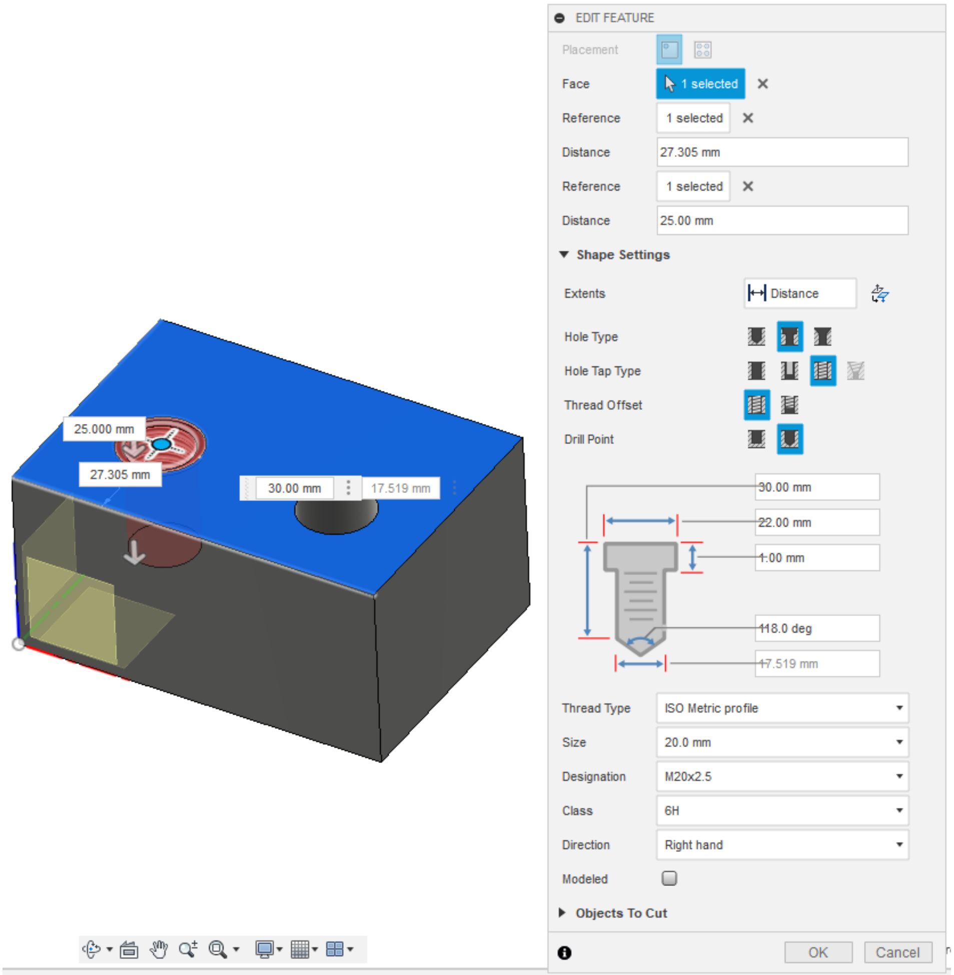
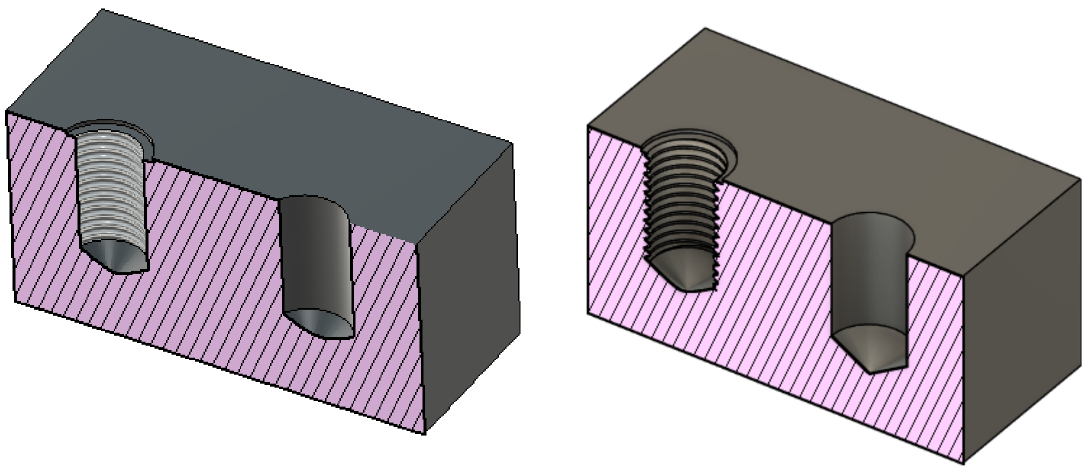


Figure 3.92

The thread can be realized as a cosmetic one (Figure 3.93, a) or as a modelled one (3-dimestional) (Figure 3.93, b).



(a) (b)

Figure 3.93

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Using the Thread command

The Thread command is used in Fusion 360 to create different types of threads on cylindrical surfaces (Figure 3.94).

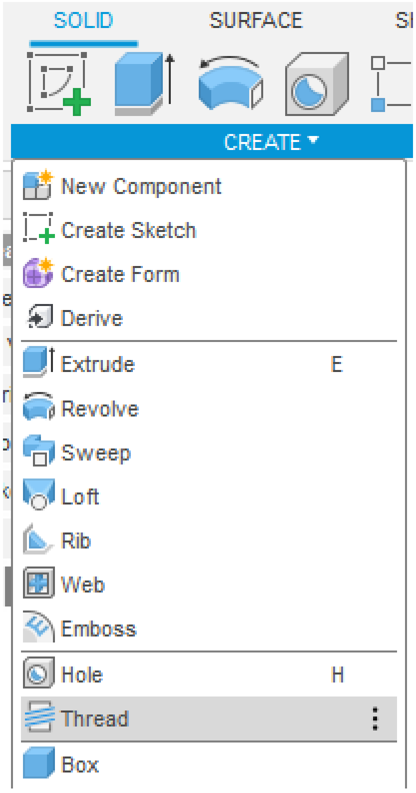

Figure 3.94

To demonstrate the creation of an external thread, let us use the three-dimensional object with the shape shown in Figure 3.95.

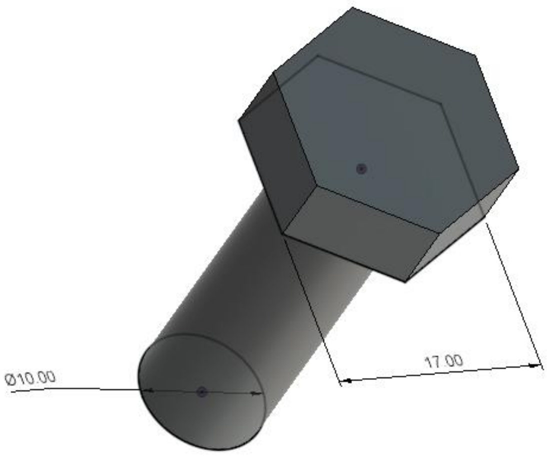

Figure 3.95

The Thread command is selected. From the context menu we designate the object's cylindrical surface as the face on which the thread will be placed (Figure 3.96).

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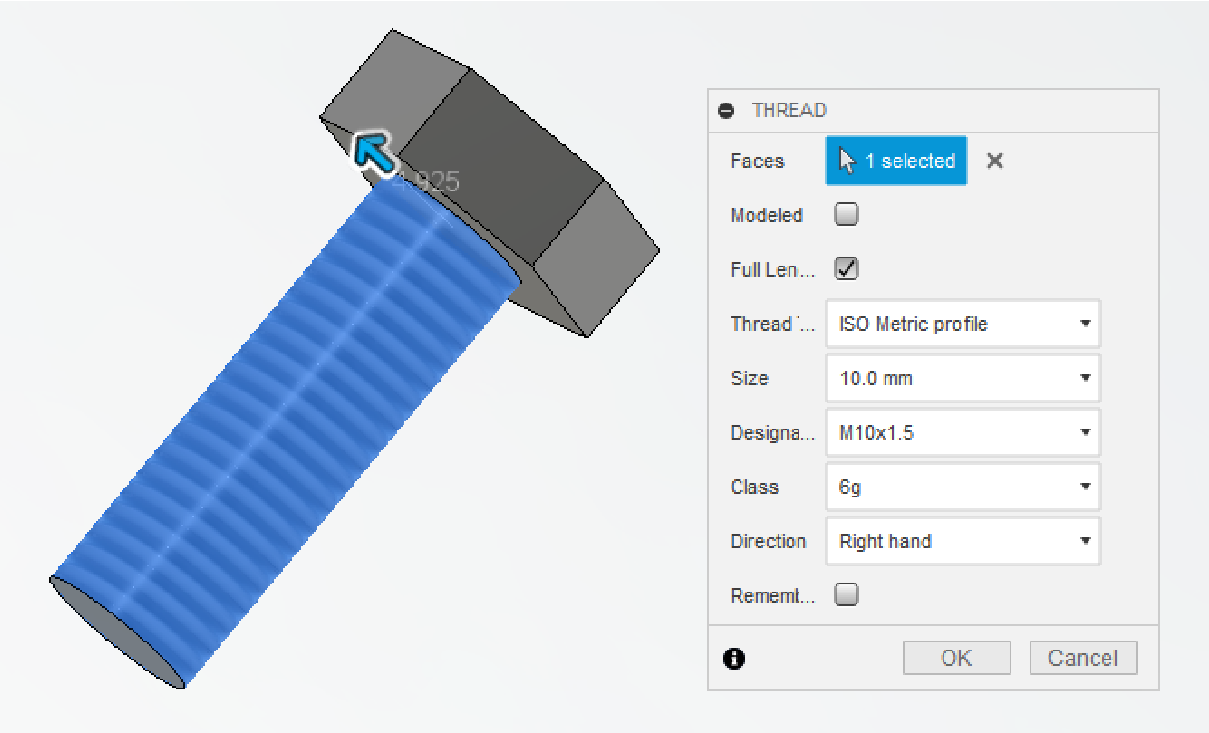
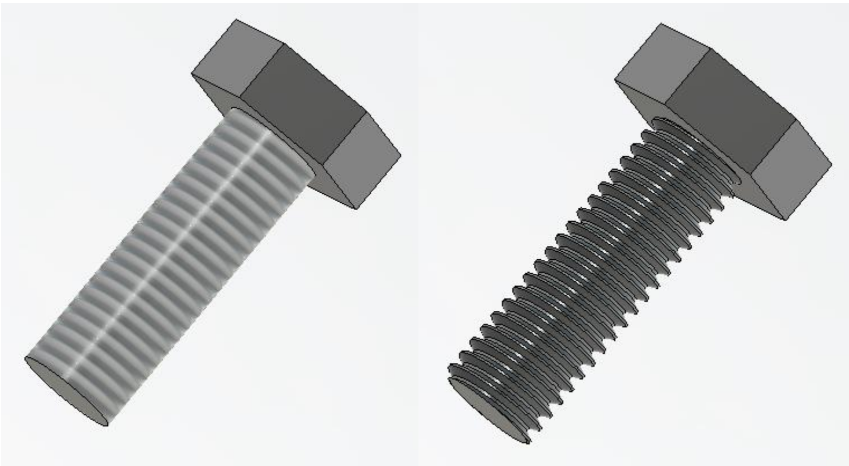


Figure 3.96

The Fusion 360 environment is intelligent enough to offer the appropriate thread based on the diameter of the cylindrical surface. We can change the type of thread, the size, the pitch, the accuracy class, as well as the direction of rotation - right or left thread. The thread, it can be visualized as cosmetic (Figure 3.97, a) or as three-dimensionally modelled (Figure 3.97, b).



(a) (b)

Figure 3.97

By deactivating the Full Length parameter (Figure 3.98), the thread length can be controlled. This is useful when it is not necessary to place a thread on the entire surface.

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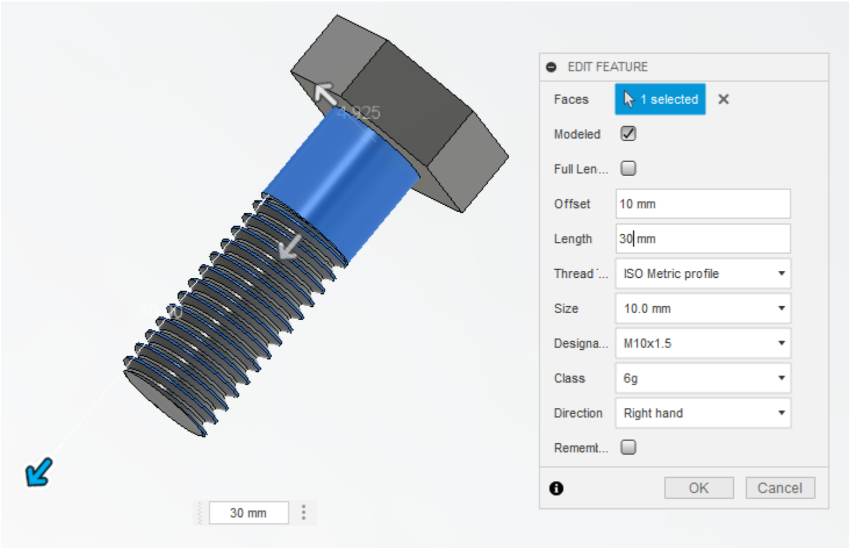

Figure 3.98

In Figure 3.99 we see the final model.

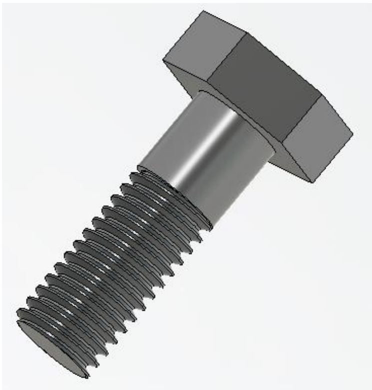

Figure 3.99

The Thread command can also be used to create internal threads. The process is similar to the creation of the external thread discussed above.

In Figure 3.100 bellow we show a three-dimensional body.

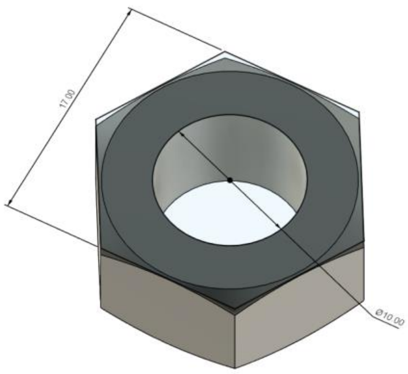

Figure 3.100

The thread is applied on the inner cylindrical surface. Based on the diameter the Thread command will suggest the type of thread that can be created (Figure 3.101).

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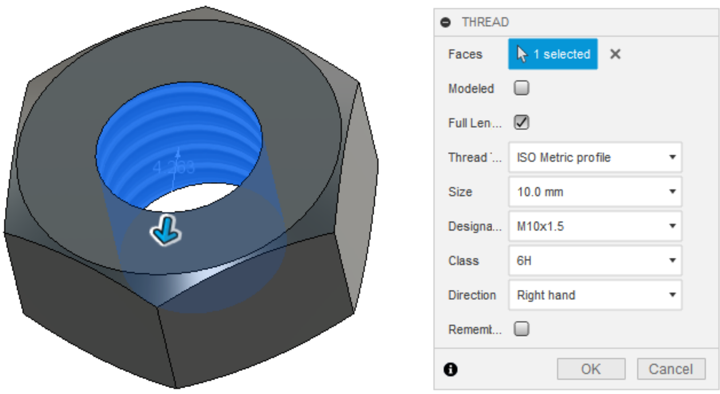

Figure 3.101

The automatically generated thread can be changed and its parameters can be controlled via the context menu. Internal threads can also be defined as cosmetic or fully modelled (Figure 3.102).

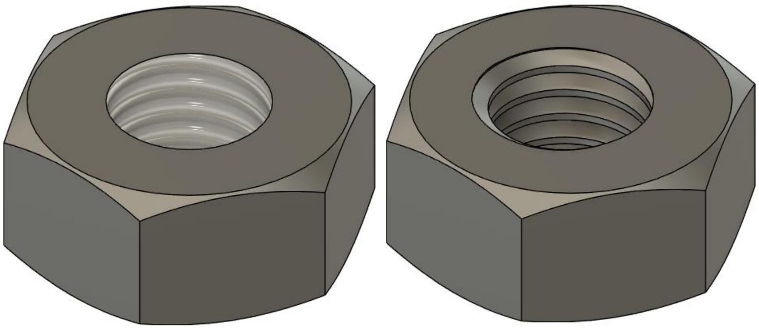

Figure 3.102

Using the Box, Cylinder, Sphere, Torus, Coil and Pipe commands

Fusion 360 can use specialized commands to create primitive three-dimensional geometric objects, such as boxes, cylinders, spheres, etc. These objects can be created directly without the need to draw 2D sketches beforehand, which speeds up the design process, eliminates the possibility of errors and ensures that the objects will have a perfect geometric shape.

The set of commands for creating geometric objects can be found in the Create menu (Figure 3.103).

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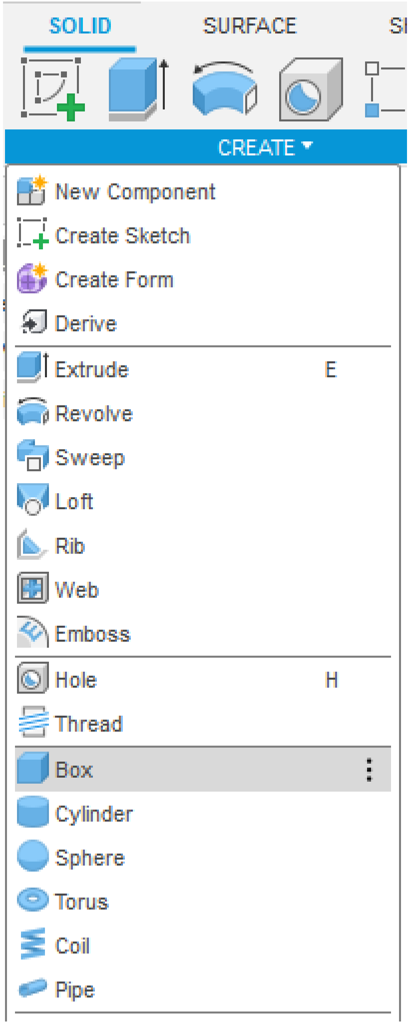

Figure 3.103

Creating a primitive Box

A parallelepiped can be drawn by selecting the Box command from the menu Create - Box . After starting the command, it is first necessary to specify the plane that will be the base of the parallelepiped (Figure 3.104).

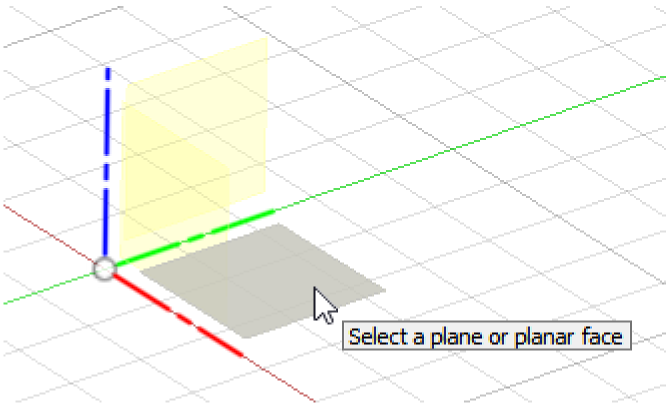

Figure 3.104

On the selected plane, we use the mouse to draw the shape's rectangular base, and we set its dimensions (Figure 3.105).

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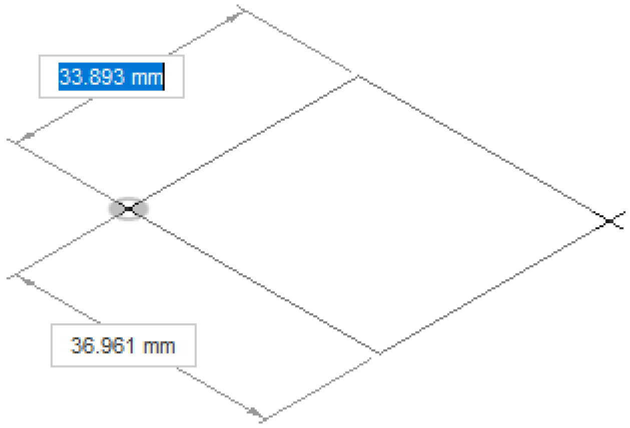

Figure 3.105

After setting the rectangular base shape, the command's dialogue menu opens, where new values for the length, width and height of the parallelepiped can be specified (Figure 3.106).

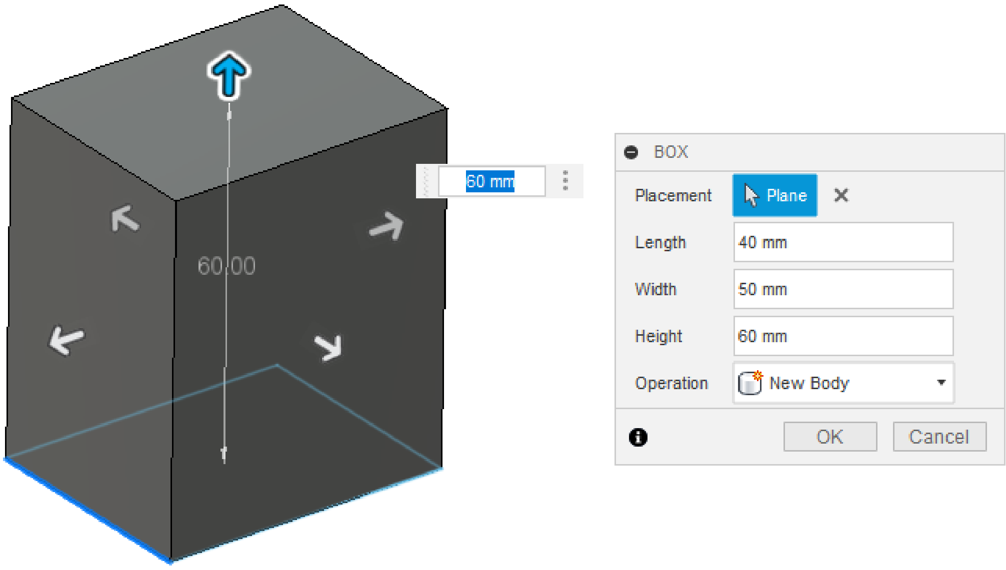

Figure 3.106

Creating a Cylinder

A cylinder can be created with the Cylinder command (Figure 3.107).

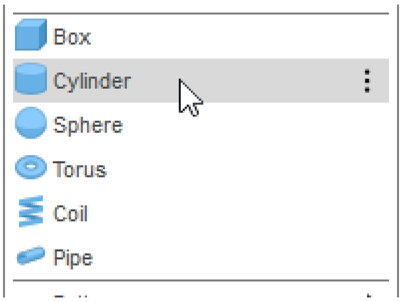

Figure 3.107

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Similarly to the creation of a parallelepiped, here it is necessary to specify the plane in which the cylinder base will be created. On this plane, we set the diameter of the cylinder's base by dragging the mouse (Figure 3.108).

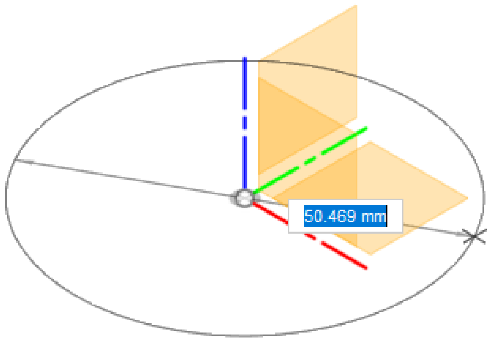

Figure 3.108

The cylinder's diameter and height can be specified in the command menu (Figure 3.109).

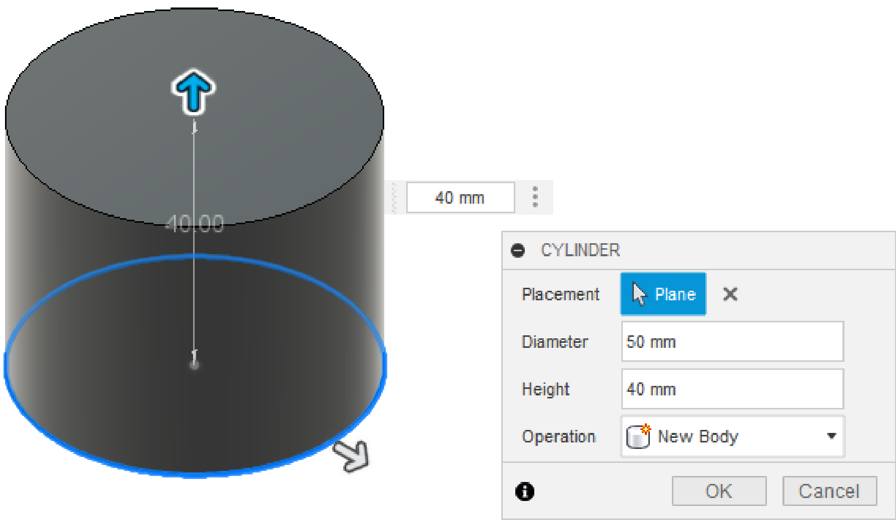

Figure 3.109

Creating a sphere

The specialized command Sphere 

is used to create a sphere.

The position of the center of the sphere in space, usually on a plane, is selected with the mouse. We create the spherical object and we can specify its diameter (Figure 3.110).

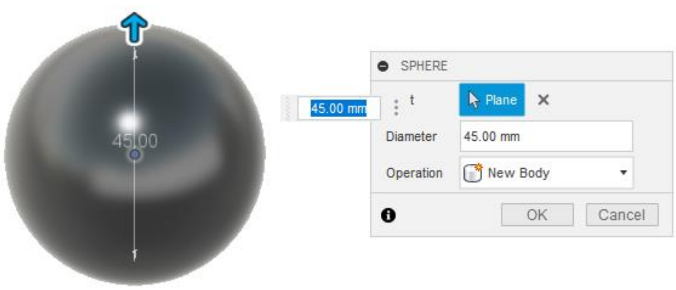

Figure 3.110

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Creating a primitive torus

The torus is a stereometric object with a rotating surface and a hole in the middle. The surface is defined by the rotation of a geometric shape around an axis passing through the middle of the hole and not intersecting the surface of the torus.

We can create a toroid with the Torus command Torus . After we activate the command, we use the mouse to select the plane on which the circle defining the torus should lie. The shape of the torus appears in the preview, and we can specify its diameter and thickness in the command menu (Figure 3.111).

Figure 3.111

The command allows you to set the location of the torus relative to its inner diameter (Figure 3.112).

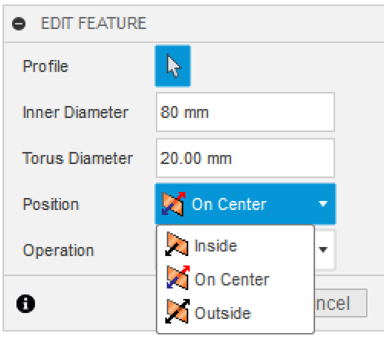
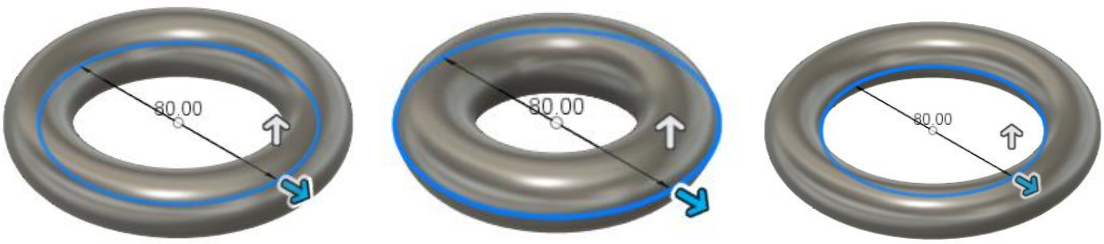


Figure 3.112

The position of the torus can be centered relative to its inner circumference (Figure 3.113, a), inside it (Figure 3.113, b) or outside it (Figure 3.113, c).



(a) (b) (c)

Figure 3.113

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Creating a coil or a spiral

We use the Coil command Coil to create a 3D object in the shape of a coil or a spiral. Springs with different shapes and principles of operation can be easily designed using this command.

The Coil command requires the selection of a plane on which the diameter of the created object can be defined using the mouse (Figure 3.114).

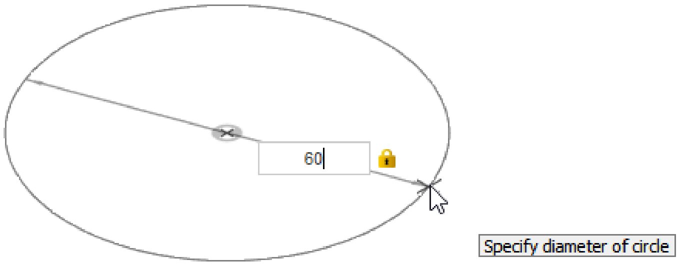

Figure 3.114

Once we have defined the diameter, a preview of the object and a command menu dialogue appear. The dialogue box allows us to adjust the parameters (Figure 3.115).

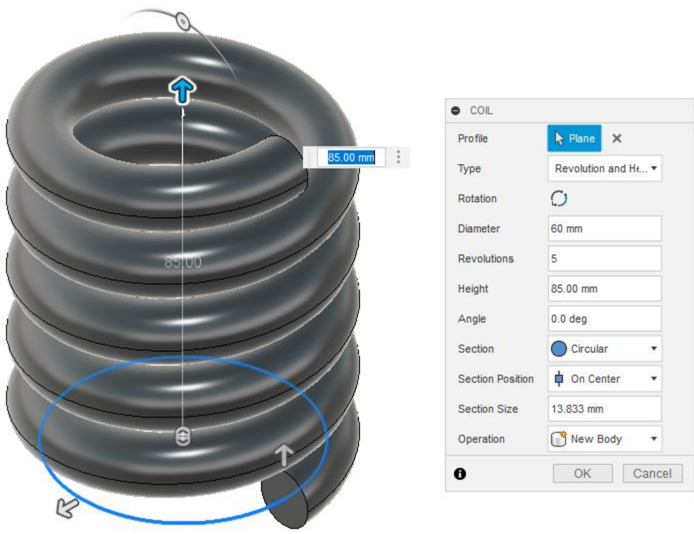

Figure 3.115

With the Coil command (Figure 3.116) the object can be created by selecting the following options:

- *Revolution and Height* – the number of revolutions and the height of the object are set. The pitch between two adjacent revolutions is obtained automatically from the input parameters
- *Revolution and Pitch* – the number of revolutions and the pitch size between them are specified, and the height of the object is obtained automatically based on them
- *Height and Pitch* – the height of the object and the pitch size between two adjacent revolutions are set. Based on this, the number of revolutions is automatically determined
- *Spiral* – this option is used to create a spiral object, the revolutions of which are in one plane.

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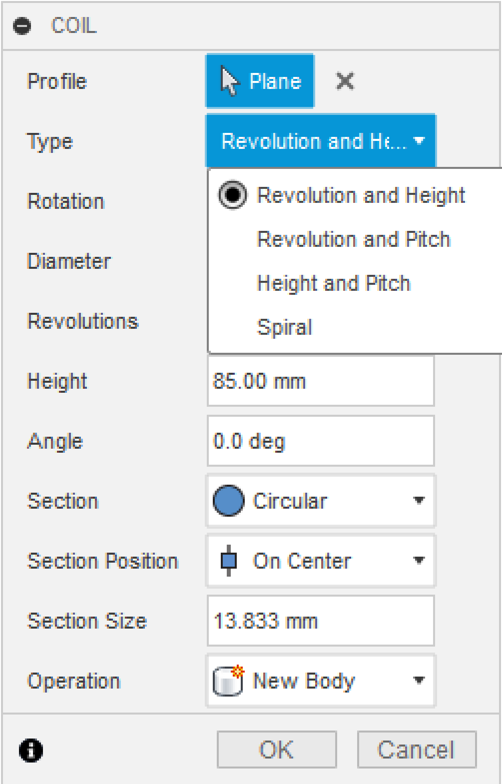
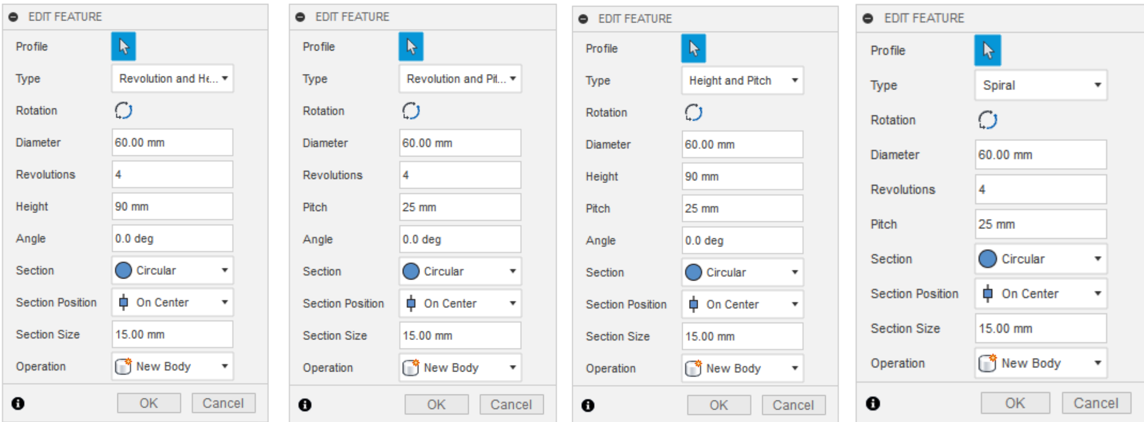


Figure 3.116

Depending on the chosen option, the menu structure of the Coil command also changes. In Figure 3.117 we show the different command menus that can appear.



(a) (b) (c) (d)

Figure 3.117

When Spiral is selected as the object type, we can perform the modelling of coil springs with the appropriate shape and number of revolutions (Figure 3.118).

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Figure 3.118

Regardless of the type of the desired object, the Section option allows for creating it with different cross sections (Figure 3.119). The available shapes are circular, square and triangular.

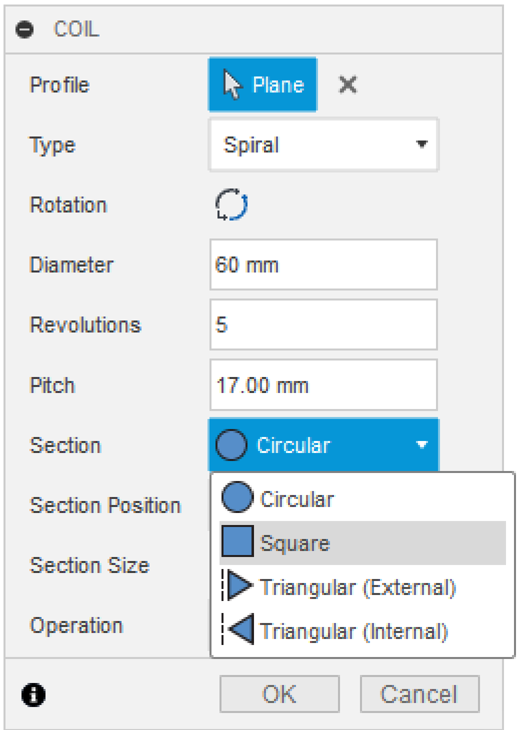

Figure 3.119

Depending on the cross section, the shape of the three-dimensional object also changes (Figure 3.120).

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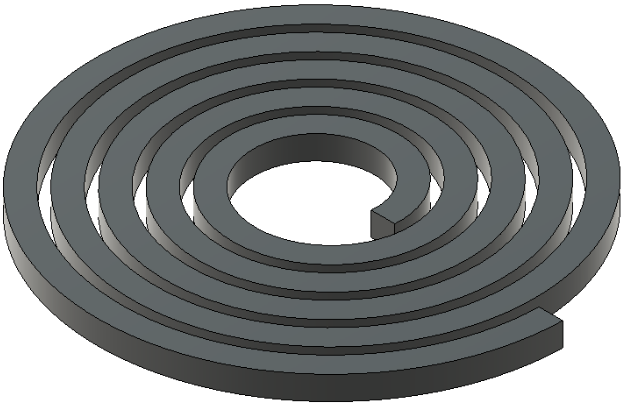

Figure 3.120

When creating a coil, the cross section of its revolutions can be positioned on the inner side, on the outer side or centered vis-à-vis the circle that determines the shape (Figure 3.121).

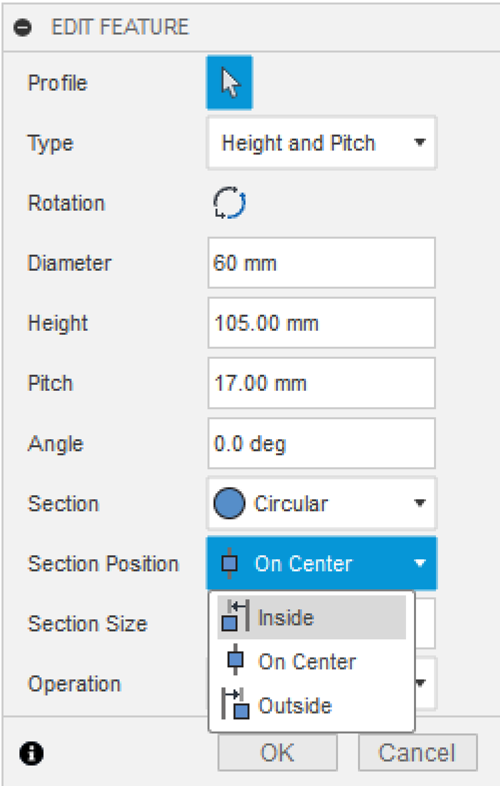

Figure 3.121

By choosing the location of the section, the dimensions of the three-dimensional object itself change. In Figure 3.122 we show the generated objects with the cross sections positioned on the inner side of the circle (a), on the outer side (b) or centered (c).

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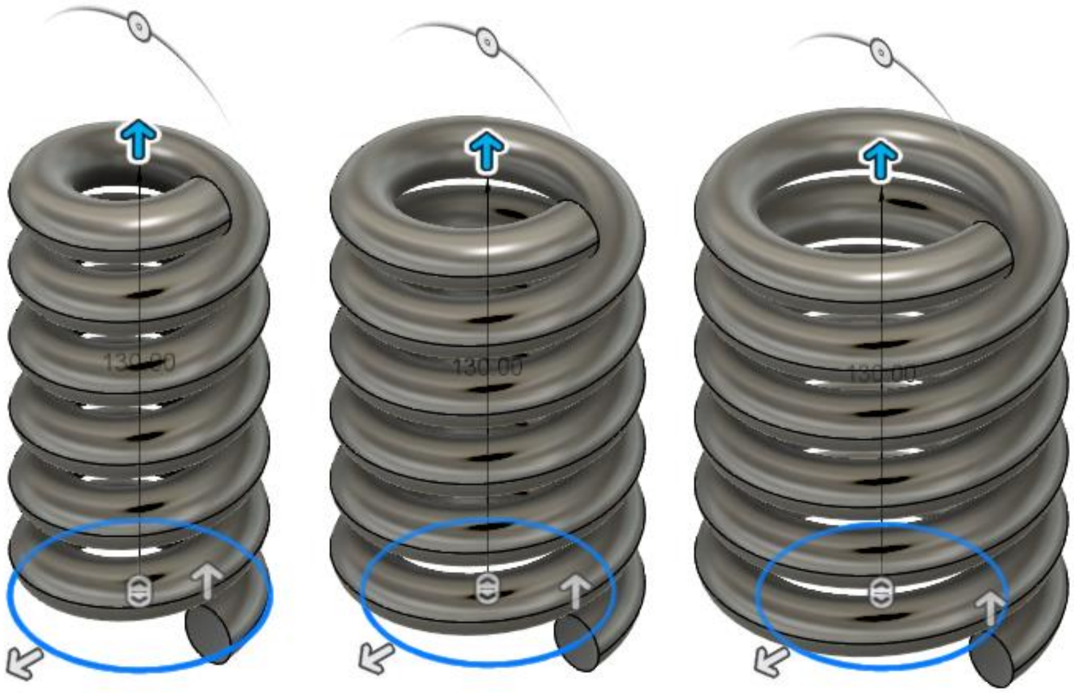

(a) (b) (c)

Figure 3.122

Creating objects with a tubular shape

The creation of tubular objects is performed with the Pipe command (Figure 3.123).

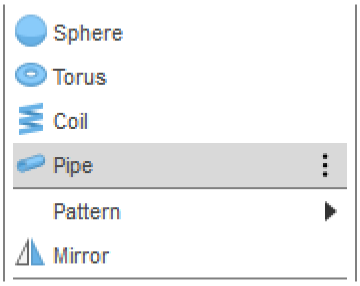

Figure 3.123

Before creating the three-dimensional object, it is necessary to draw a contour that will be followed by the tubular object. To produce a contour, in this example we create a new sketch and select the Spline command from the Create menu (Figure 3.124).

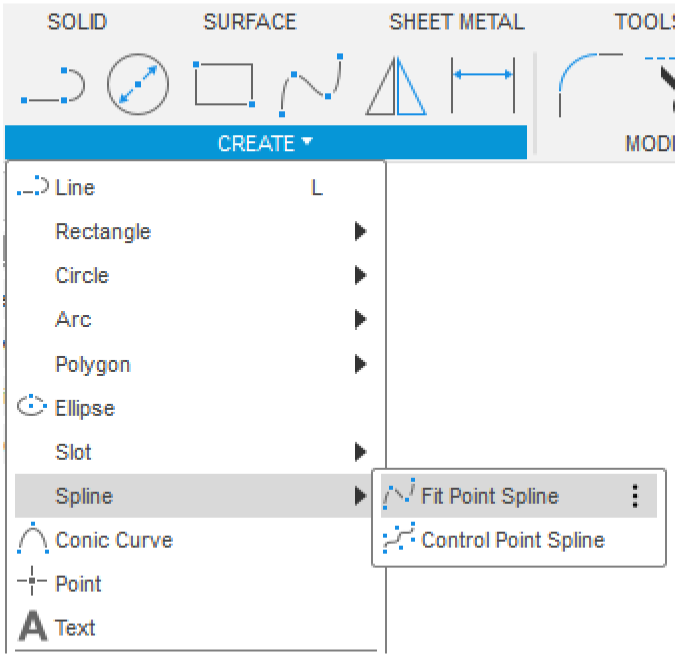

Figure 3.124

By using the Spline command, we draw a contour (Figure 3.125) that will set the shape of the tubular object.

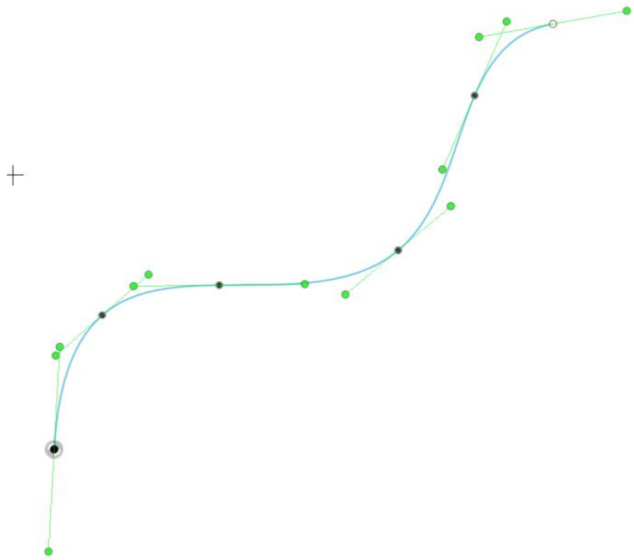

Figure 3.125

The Pipe command is started. In the menu, the previously created contour is selected as a path (Figure 3.126). Under the Distance option we specify whether the entire path will be used to define the body (entering 1 in the field means 100% following the path). In the Section Size field the diameter of the generated pipe is set.

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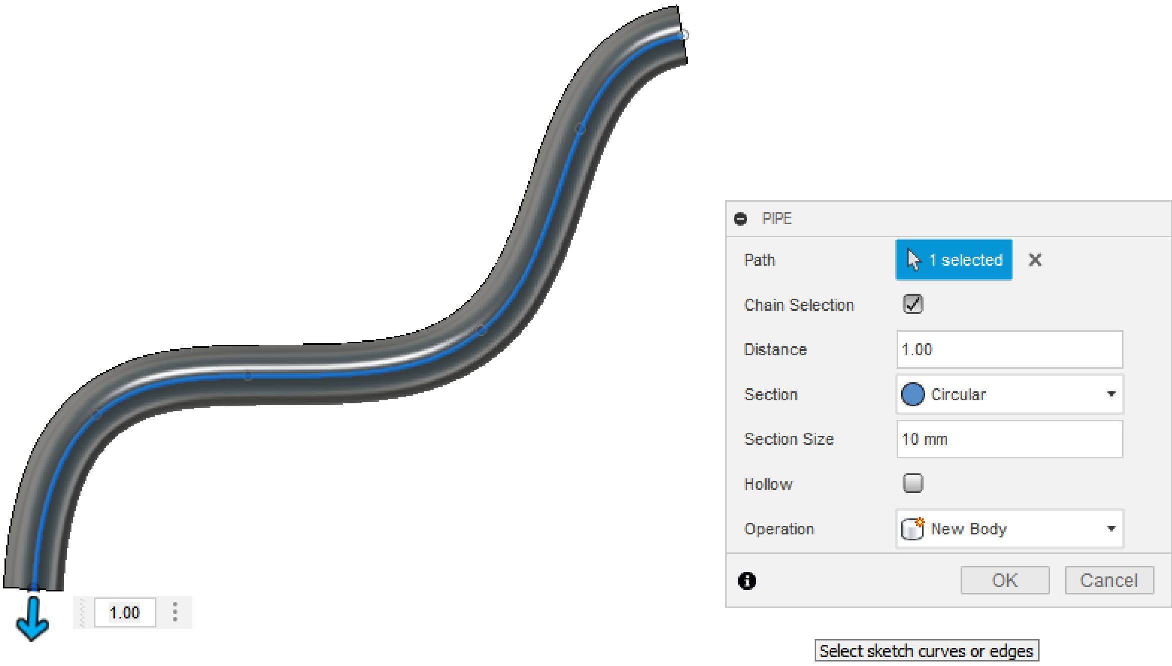

Figure 3.126

When the Hollow option is highlighted in the command menu, the generated object acquires an aperture. The wall thickness of the tubular body can be set in the field Section Thickness (Figure 3.127).

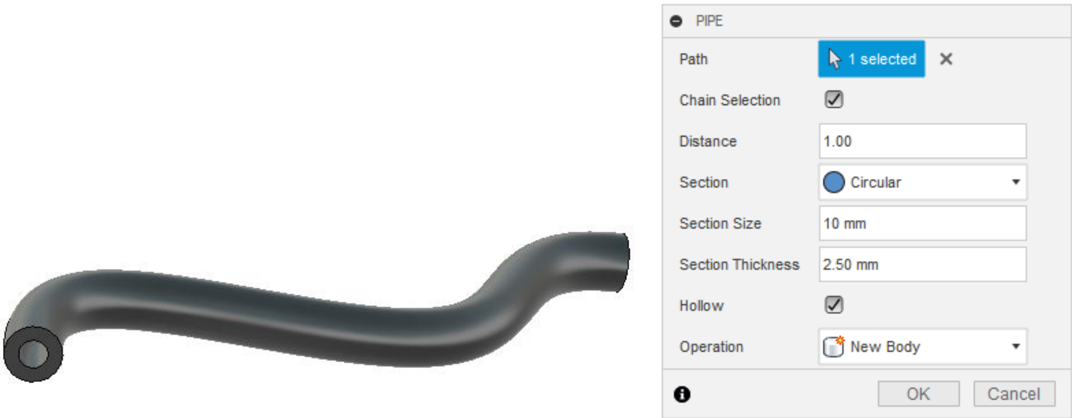

Figure 3.127

The shape of the pipe's cross-section can also be changed. It can be made circular, square or triangular (Figure 3.128).

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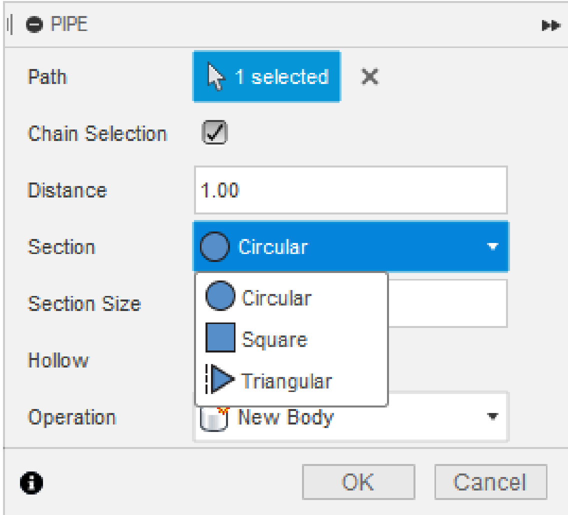

Figure 3.128

Replicating 3D objects using the Pattern and Mirror commands

Fusion 360 includes commands that can be used to replicate already created components, bodies and features, thus accelerating the work of creating multiple objects of the same type and ensuring the uniformity of their features and parameters. Replication of objects can be done using the Pattern and Mirror commands (Figure 3.129).

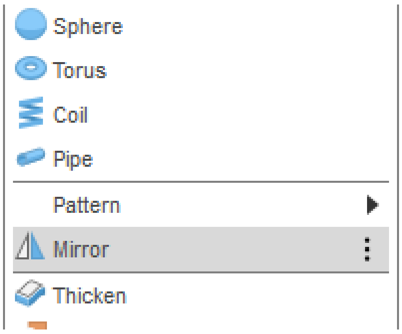

Figure 3.129

Replication of objects using the Mirror command

The replication of objects using the Mirror command () makes it possible to mirror an existing object or its characteristic in relation to a plane or planar surface.

The Mirror command allows for mirroring (Figure 3.130):

- faces of three-dimensional shapes
- three-dimensional bodies
- features of the three-dimensional objects
- components.

The number of newly obtained objects or features is equal to the number of objects or features selected to be mirrored.

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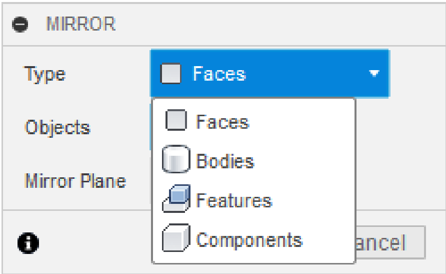

Figure 3.130

To demonstrate the operation of the Mirror command, we present a three-dimensional object in Figure 3.131 below.

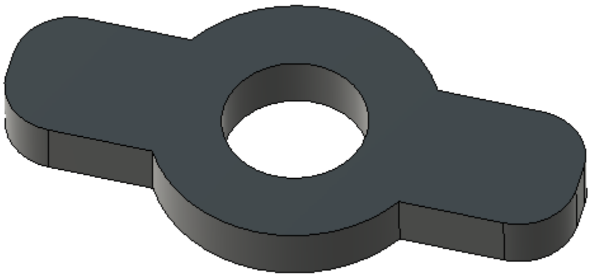

Figure 3.131

We create a new sketch on the object's upper surface. The sketch contains a circle placed as in Figure 3.132.

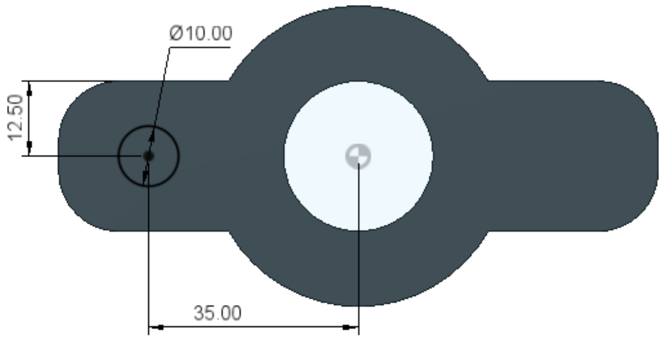

Figure 3.132

By using the Extrude command, we create a hole in the 3D object based on the circle in the sketch (Figure 3.133).

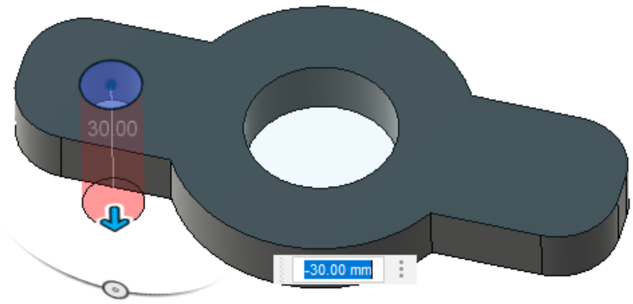

Figure 3.133

The creation of the sketch and the hole appear as successive steps in the Timeline (Figure 3.134).

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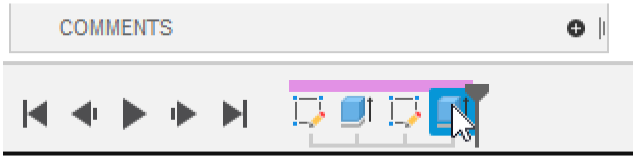

Figure 3.134

These individual steps represent the characteristics of the three-dimensional object. The feature indicating the hole's extrusion will be used in the Mirror command. Before the mirroring can be performed, it is necessary to have a visible plane or planar surface in the working field, against which the mirror reflection will be implemented. If the main planes are not visible, their visibility should be activated by the browser (Figure 3.135).

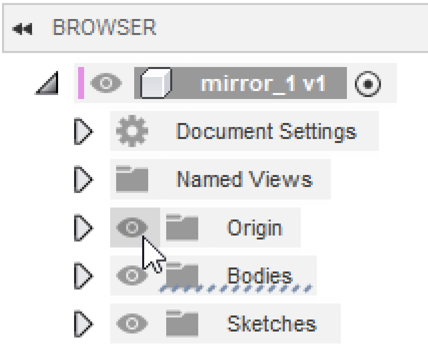

Figure 3.135

Once the main planes are visible, the Mirror command is launched. In the command menu (Figure 3.136) the object's features are selected as the type of objects to mirror. Using the mouse, the surface of the created hole is selected, and one of the main construction planes is selected as the Mirror Plane.

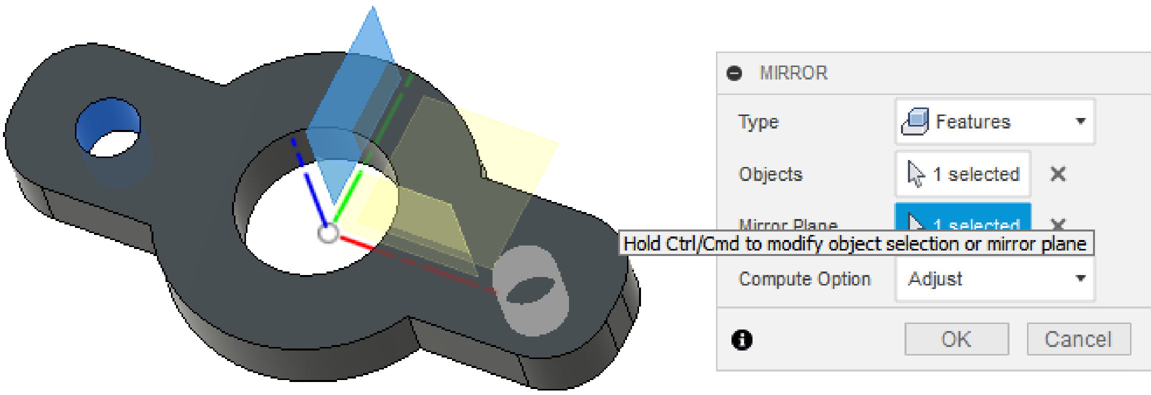

Figure 3.136

After the execution of the command, we create a new hole in the three-dimensional object, identical to the original one and symmetrically located vis-à-vis the mirror plane (Figure 3.137).

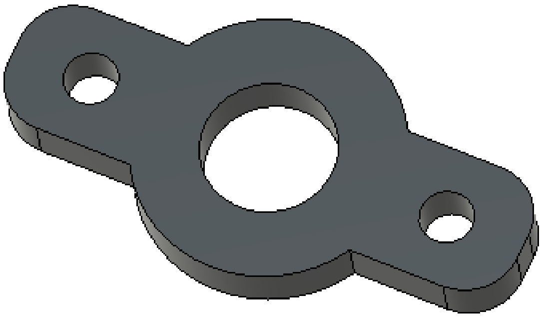

Figure 3.137

Replication of objects using the Pattern commands

The Pattern submenu is located on the Toolbar and in the Create menu. Using the commands included in this submenu, it is possible to mirror selected objects or features in a linear sequence, in a circle or following a pre-selected profile (Figure 3.138).

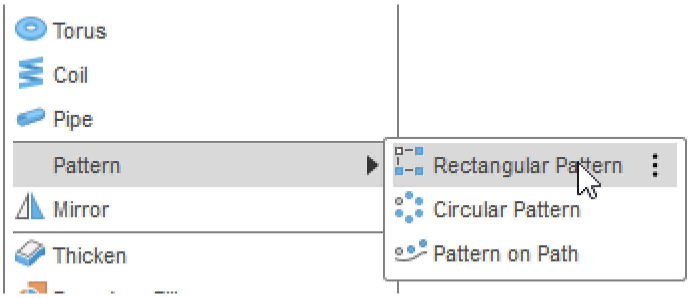

Figure 3.138

- Using the Rectangular Pattern command

This command allows the selected object (or objects) to be mirrored following a linear arrangement in two perpendicular directions.

To demonstrate the command, we present a three-dimensional object on whose upper surface we have created a sketch of a circle (Figure 3.139).

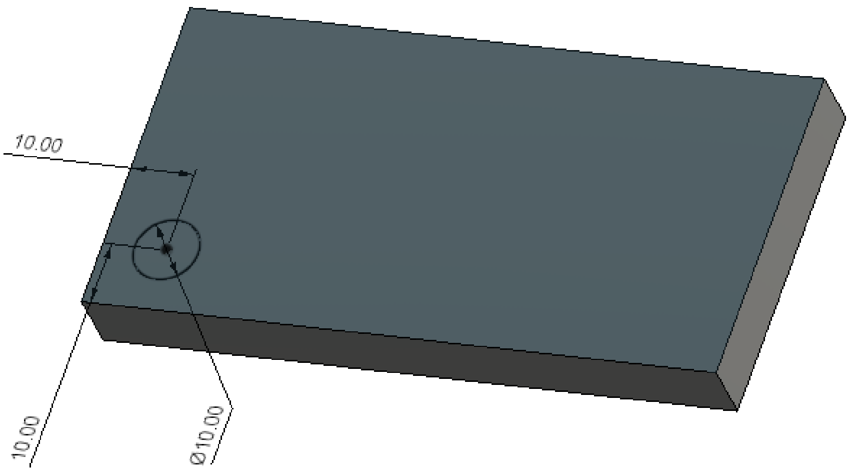

Figure 3.139

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Using the Extrude command and the circle in the sketch, we create a hole in the 3D object (Figure 3.140).

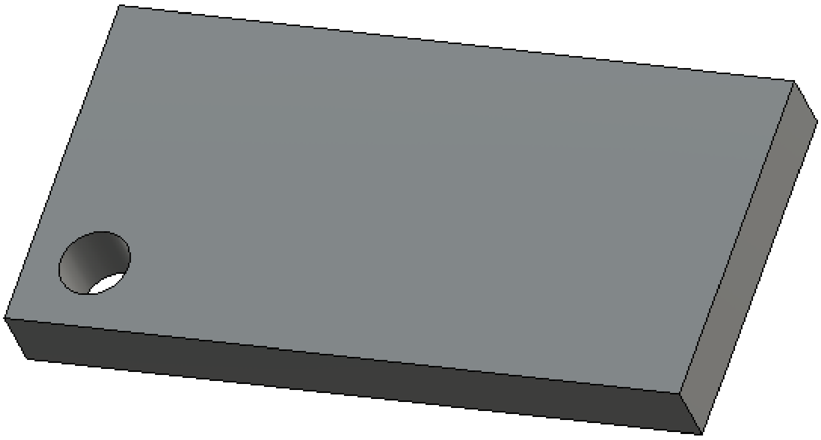
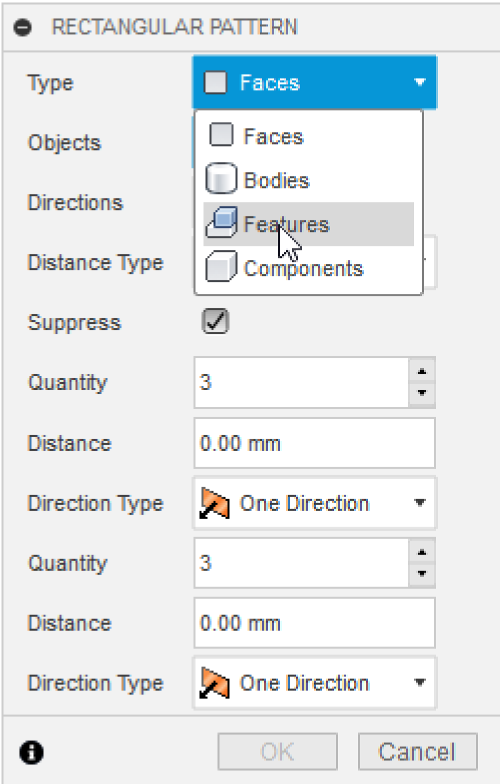


Figure 3.140

After starting the Rectangular Pattern command, a menu opens in which we can select the type of object to be mirrored (Figure 3.141). In the example, the Features option is selected as Type.



Fi gure 3.141

When the Features option is selected as Type, the particular feature of the object that we need to replicate can be selected from the Timeline by clicking on the step where this feature was created (Figure 3.142).

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Figure 3.142

When the corresponding feature (in this case the hole) is selected, it is marked in the body view (Figure 3.143).

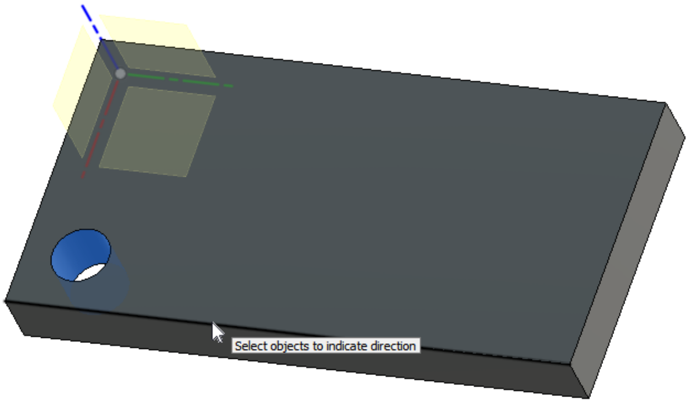

Figure 3.143

In order to indicate the direction in which the selected object will be mirrored, it is necessary to select an edge of the three-dimensional body or the construction axis. This is done in the Directions field of the command menu (Figure 3.144). The objects that are obtained in the mirroring process can be positioned relative to each other depending on the specified total length of the pattern or by selecting the spacing distance between them.

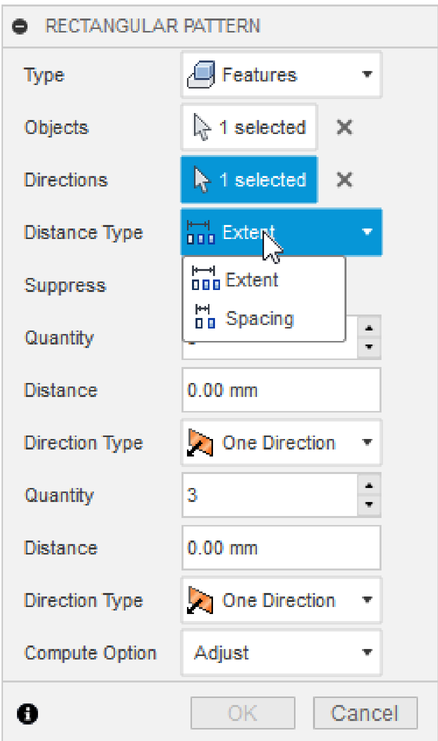

Figure 3.144

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The location of new objects in the pattern can be done in two directions - one that is selected by us and one perpendicular to the selected direction. For both directions, we can specify how many copies of the object are to be placed (Figure 3.145).

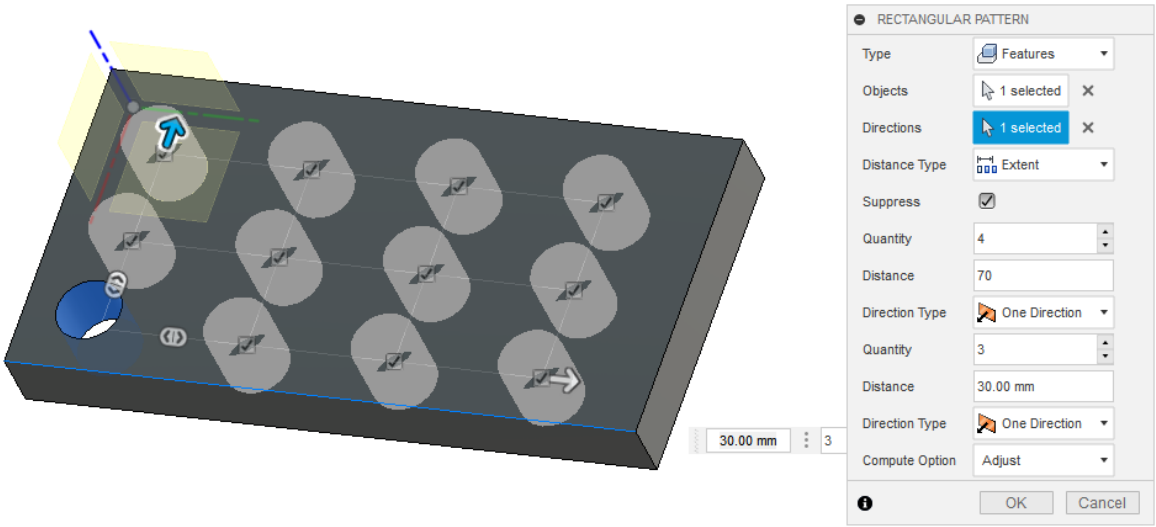

Figure 3.145

After all settings are completed, multiple copies of the selected object (or feature) are generated. The environment automatically calculates their locations based on the options we selected in the command menu (Figure 3.146).

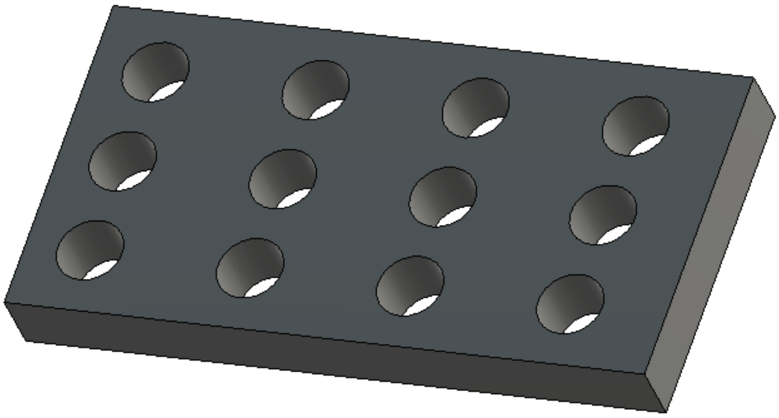

Figure 3.146

In addition to mirroring features of the object's structure, the Rectangular Pattern command can also be used to mirror entire three-dimensional bodies. In this case, new copies of the mirrored bodies appear.

A three-dimensional object appearing as a three-dimensional body in the design browser is shown in Figure 3.147.

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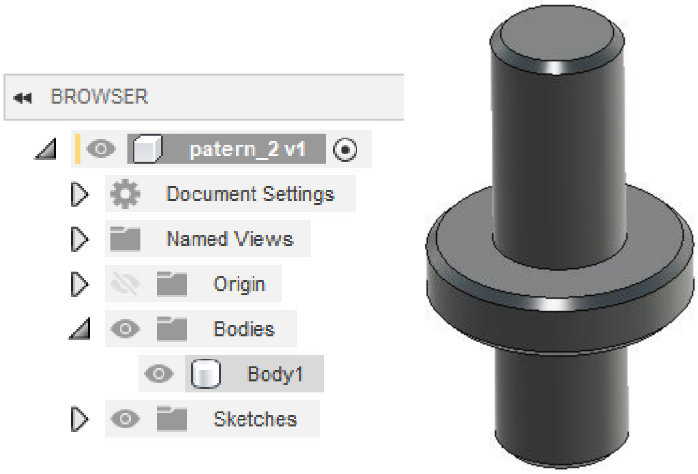

Figure 3.147

After launching the Rectangular Pattern command, in the menu that appears we need to choose Bodies in the Type field (Figure 3.148). We then need to select the presented three-dimensional body as an object to be mirrored.

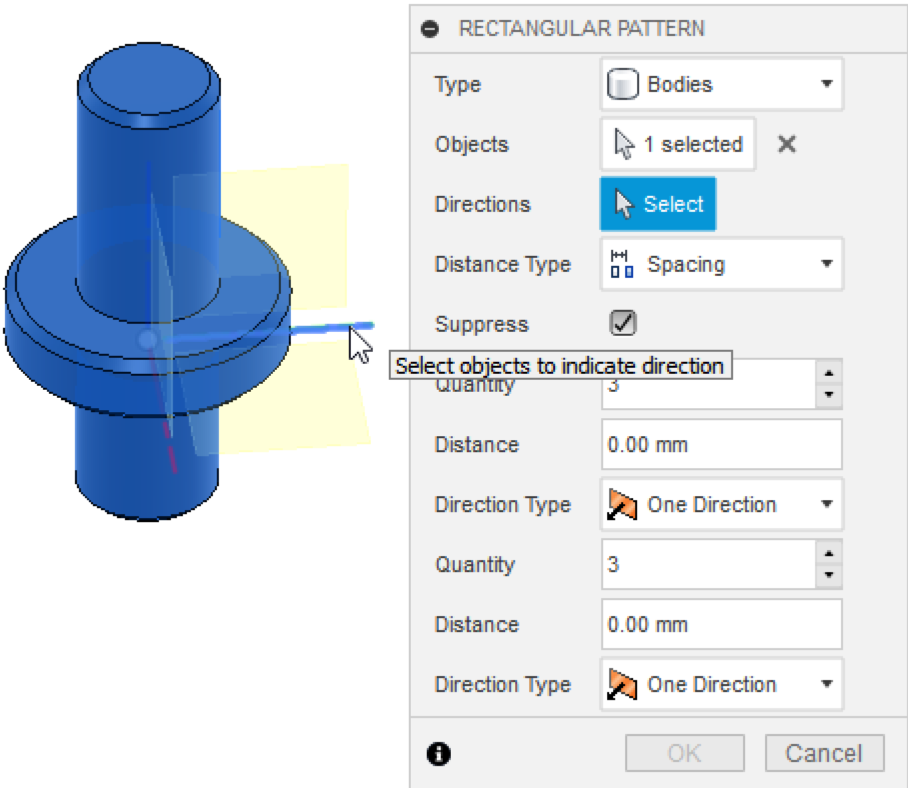

Figure 3.148

We also need to choose the direction in which we want to mirror the selected object, the distance between the individual objects in the direction of mirroring and in a direction perpendicular to it, as well as the number of objects in both directions (Figure 3.149).

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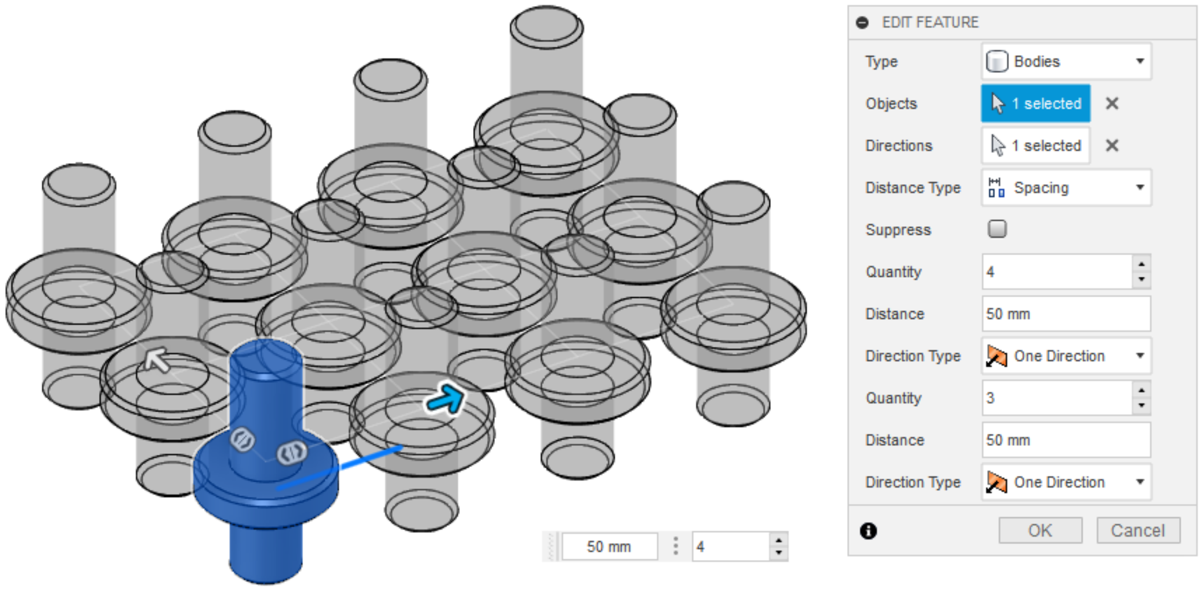

Figure 3.149

If we check the Suppress field, it is possible to exclude some of the objects from the final result (Figure 3.150).

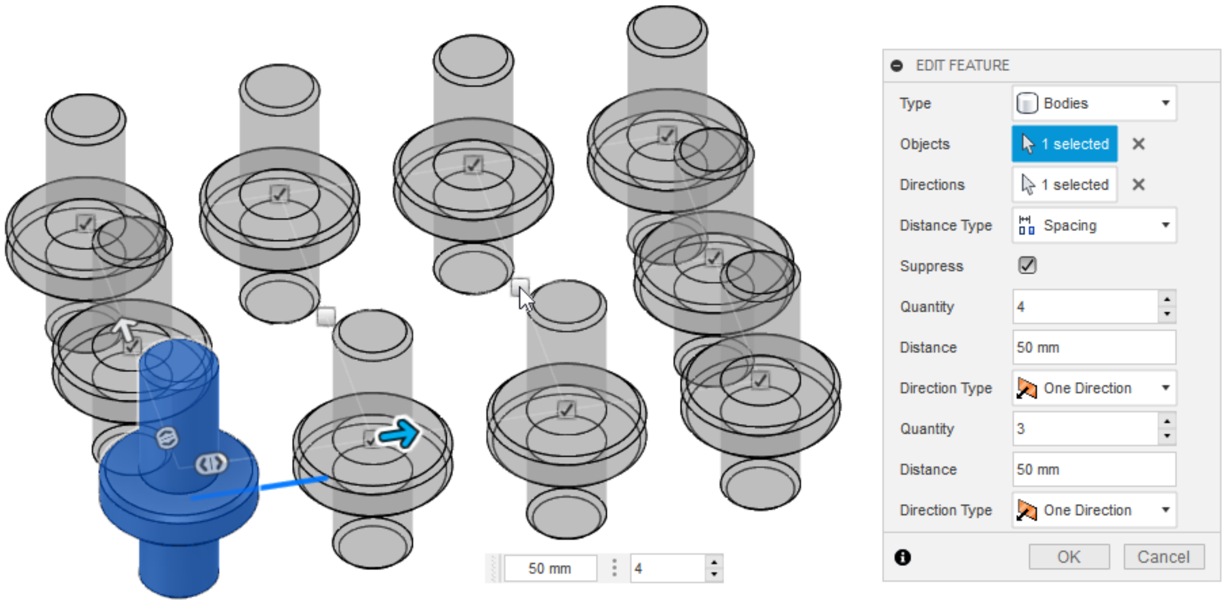

Figure 3.150

The generated objects appear in the current design browser as separate three-dimensional bodies (Figure 3.151, a), each of which is completely identical to what is being replicated (Figure 3.151, b).

In addition to mirroring bodies, the Rectangular Pattern command can be used to mirror components consisting of multiple bodies or to include other components hierarchically.

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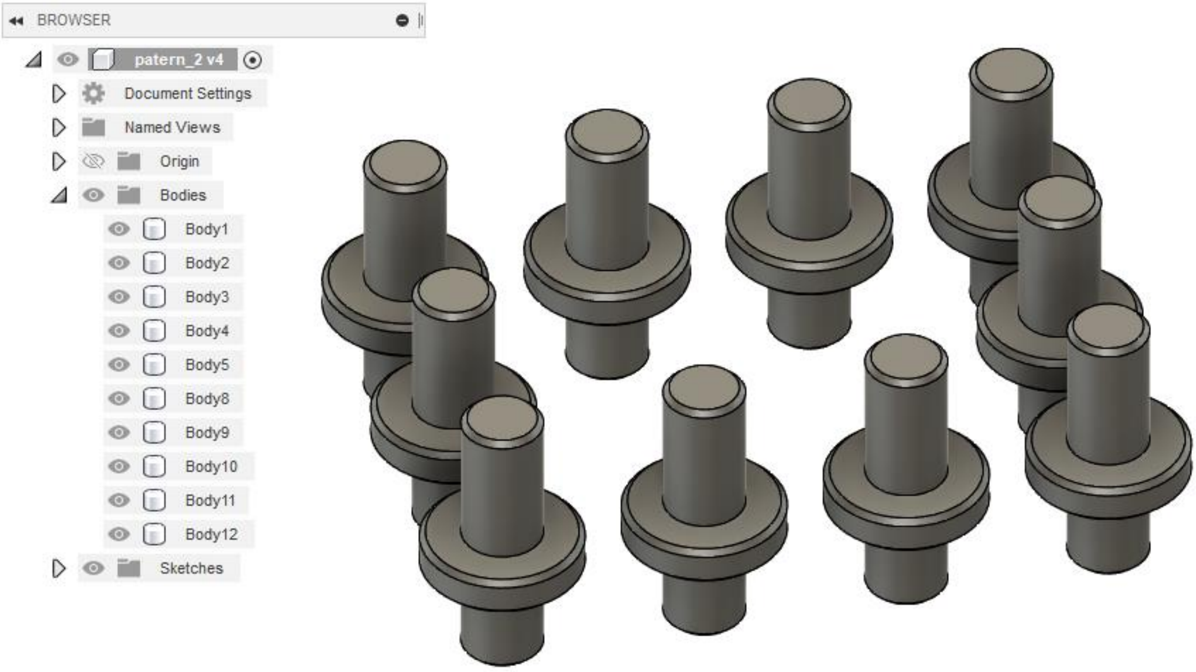

(a) (b)

Figure 3.151

Using the Circular Pattern command

In Fusion 360, the use of the Circular Pattern command allows us to replicate objects along a circle. The object shown in Figure 3.152 demonstrates the effect of the command.

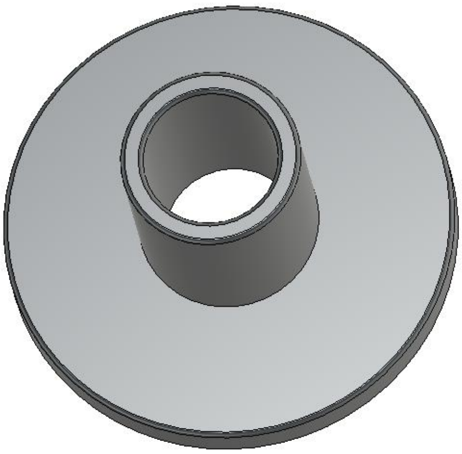

Figure 3.152

We add to the object a sketch of a circle with dimensions and location (Figure 3.153). We use the circle to make a hole in the 3D object. The hole is now the object that will be replicated with the Circular Pattern command.

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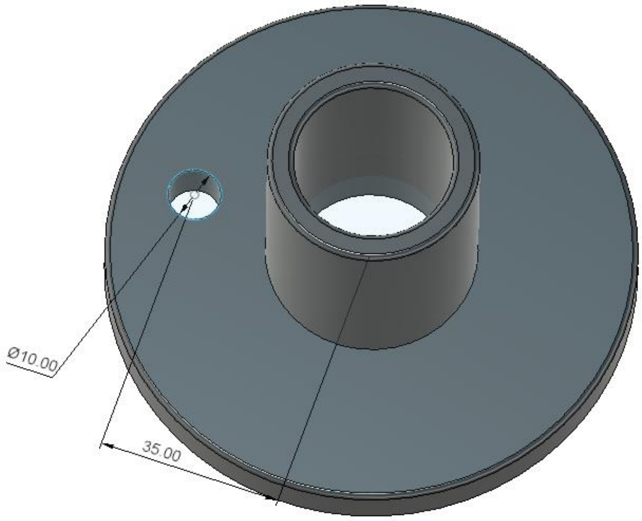

Figure 3.153

Next, we activate the Circular Pattern command (Figure 3.154).

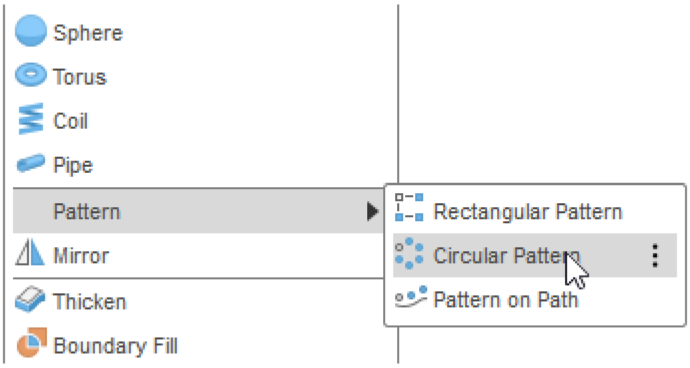

Figure 3.154

In the command menu there are several fields that specify how objects are mirrored in a circular pattern:

- *Type* – the field indicates the type of the object that will be replicated
- *Objects* – this field is used to indicate with the mouse the object (or objects) that are selected for mirroring
- *Axis* – the field sets the axis around which the circular pattern will be executed. The distance between the reproduced object and this axis is assumed to be the radius of the circle along which the new copies of the object will be placed. The center of the circle lies on the selected axis
- *Angular Spacing* – in this field we can indicate whether the reproduction will be performed along the whole circumference of the circle or only on a certain segment of it, specified by the angle of the segment;
- *Suppress* – this field allows us to remove some of the generated copies of the reproduced object.

After selecting the Type of objects (in this case it is *Features*), we specify that the hole that we have created in the three-dimensional body will be the object to be mirrored. We select the construction axis of the body to be the axis around which the circular pattern should be

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implemented. It is perpendicular to the plane on which the circle of the hole is drawn (Figure 3.155).

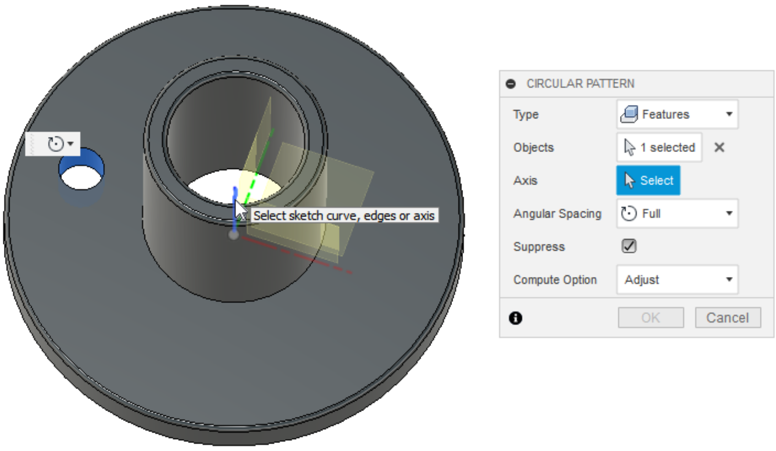

Figure 3.155

We also need to indicate the final number of objects (Figure 3.156), as well as their location on the circle (in this case we have indicated the Full option in the Angular Spacing field).

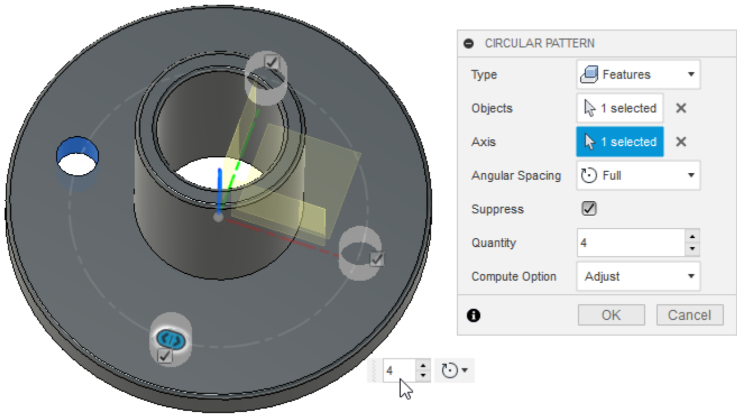

Figure 3.156

After executing the command, we obtain a body with holes evenly spaced around the selected construction axis (Figure 3.157).

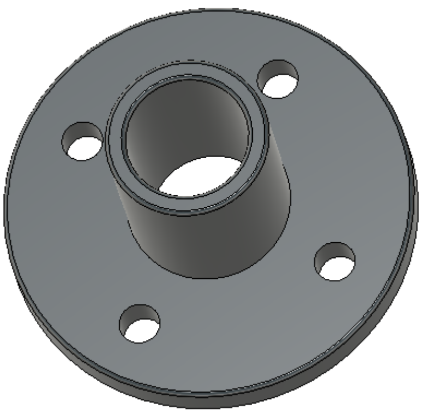

Figure 3.157

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The Circular Pattern command also allows mirroring to be performed only in a partial circle defined by a certain angle. For this purpose, in the Angular Spacing field we select the Angle option (Figure 3.158).

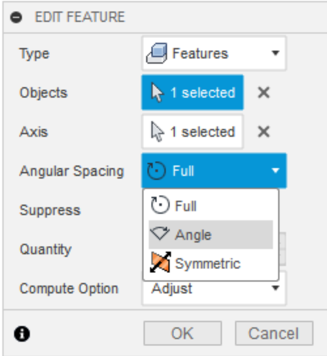

Figure 3.158

The command menu changes. A field appears in which we can enter the angle delineating the segment in which the replicated objects will be located (Figure 3.159). If it is necessary to change the position of the objects relative to the mirrored object, it is enough to change the sign of the angle.

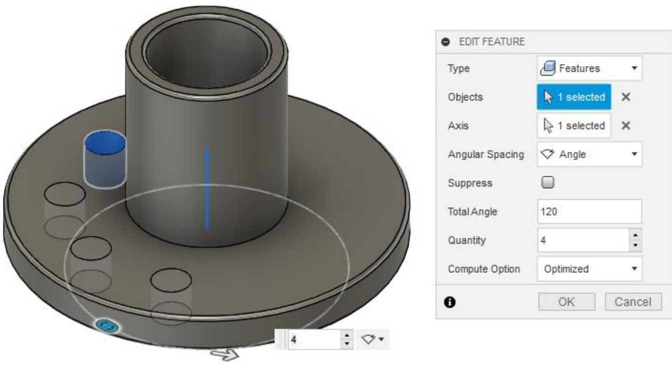

Figure 3.159

The mirrored objects, together with the object that is being replicated, appear in the specified segment (Figure 3.160).

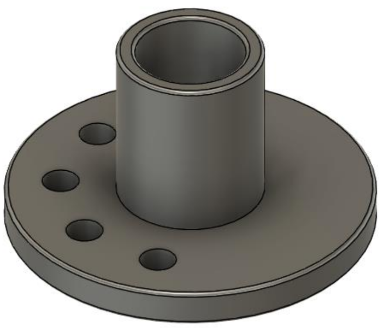

Figure 3.160

Using the Pattern on Path command

The Pattern on Path command (

) allows the multiplied objects to follow a predefined contour. To demonstrate the way the command works, let us have a three-dimensional body and a contour with a nonlinear shape defined in a sketch (Figure 3.161).

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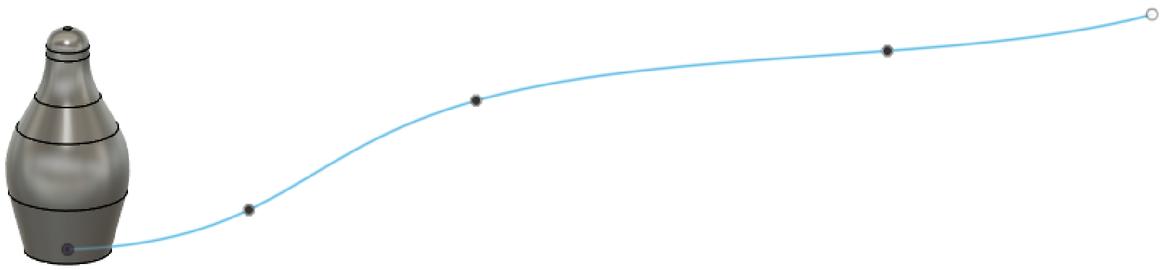

Figure 3.161

After starting the command, a menu appears, in which we can select the object to be mirrored, the path (or contour) along which the mirroring will be performed, as well as the number of objects to be obtained as a result of the command (Figure 3.162).

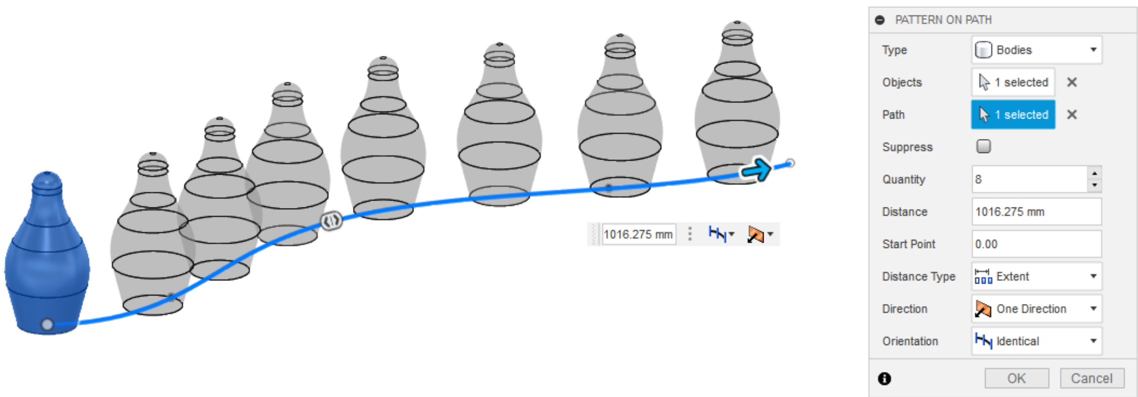

Figure 3.162

After executing the command, the selected body is multiplied. Many bodies of identical type are created the working field, arranged in a way identical to the shape of the contour (Figure 3.163).

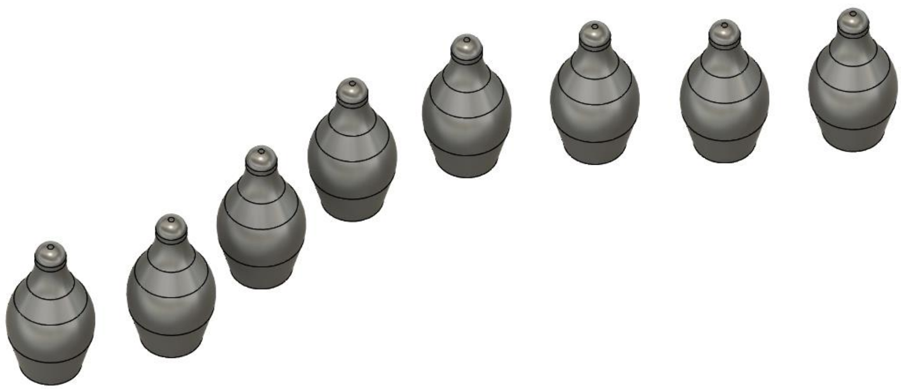

Figure 3.163

Commands for creating construction geometry

In Fusion 360 the modelling of various three-dimensional bodies often requires the use of commands that create construction geometry. This geometry is necessary in the case of creating parallel sketches, defining a plane to mirror objects, setting central axes for cylindrical objects, etc. The commands for creating construction geometry are located in the Construct menu (Figure 3.164).

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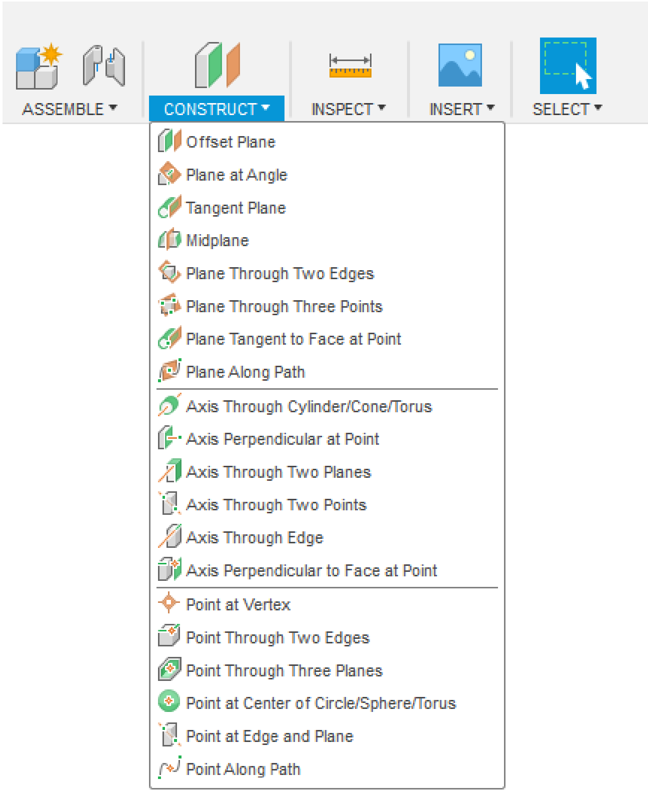

Figure 3.164

Creating an offset plane

The Offset Plane command creates a plane parallel to another plane or planar surface. We create the plane by pointing the existing plane or planar surface with the mouse and setting the distance between this plane and the new one (Figure 3.165).

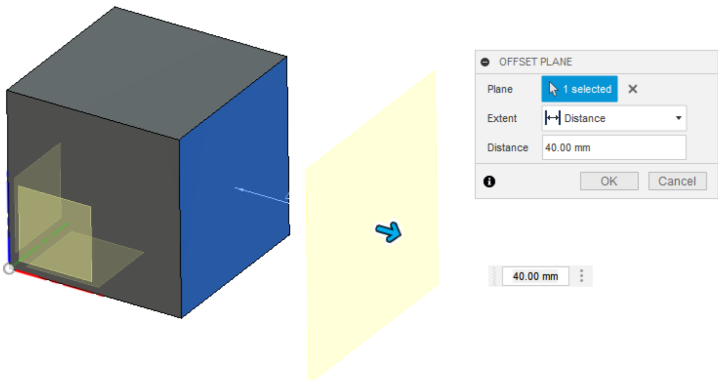

Figure 3.165

Creating a plane at an angle

The Plane at Angle command creates a new construction plane at a certain angle to an existing plane or planar surface. We create the plane by specifying a line, a body's edge or a construction axis lying on the selected plane or planar surface. For the created plane it is necessary to indicate the angle at which it is located in relation to the starting plane or planar surface (Figure 3.166).

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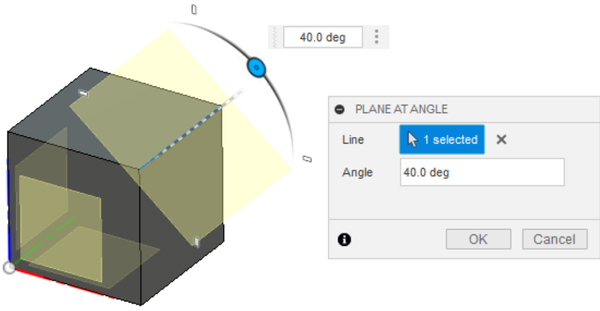


Figure 3.166

Setting a tangent plane to a cylindrical body

Using the Tangent Plane command, it is possible to create a plane that is tangent to a cylindrical or conical body. The position of the plane is controlled by the mouse, always touching the surface of the cylindrical or conical body (Figure 3.167).

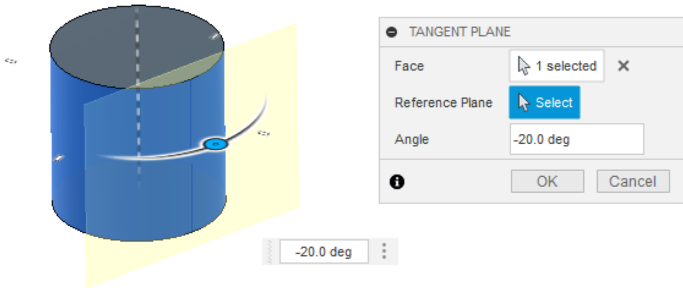
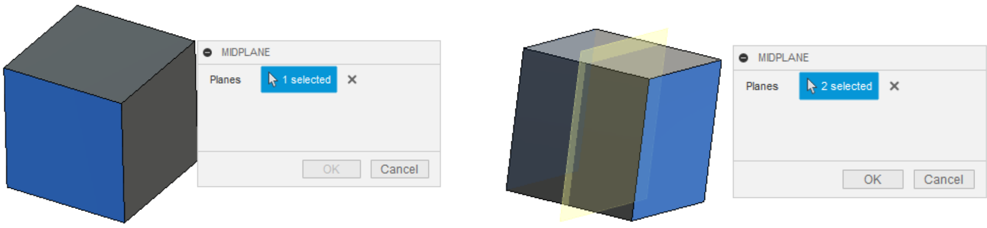


Figure 3.167

Creating a midplane lying between two other planes.

The Midplane command creates a plane that lies between two other planes or object faces. Using the mouse, we initially select the first planar surface (Figure 3.168, a). Then we specify the second surface, thus creating a construction plane lying between the two selected surfaces (Figure 3.168, b).



(a) (b)

Figure 3.168

Creating a plane through two edges

In Fusion 360, a construction plane can also be created by specifying two edges (or axes in the design). The environment allows us to use the Plane Through Two Edges command to select edges through which it is possible to create a plane (Figure 3.169). Fusion 360 does not allow us to select edges through which a construction plane cannot be created.

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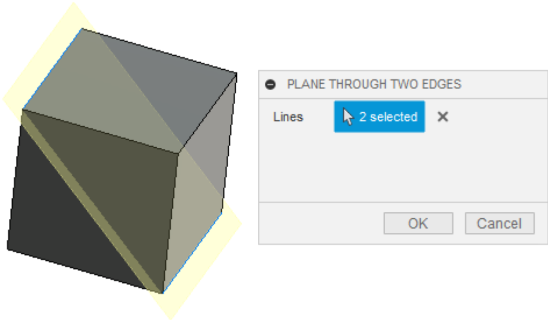

Figure 3.169

Creating a plane through three points

The Plane Through Three Points command creates a plane passing through three selected points in the workspace. Most often these points are the vertices in the geometry of a three-dimensional body (Figure 3.170).

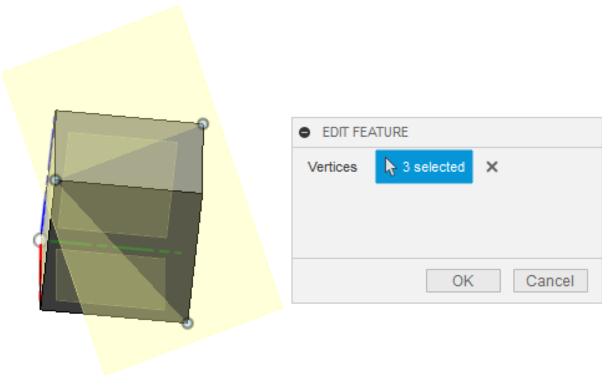

Figure 3.170

Creating a plane along a path

The Plane Along Path command creates a plane that is perpendicular to a contour at its point of intersection (Figure 3.171).

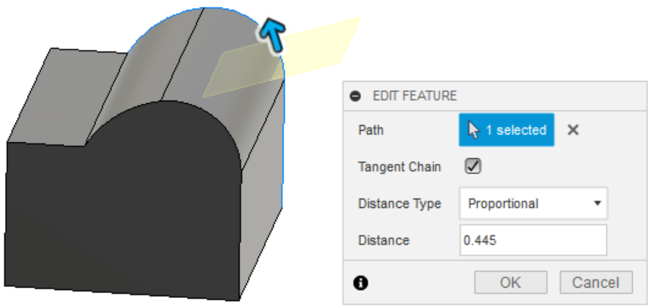

Figure 3.171

The plane can be moved with the mouse along the selected contour. When moving, it changes its orientation in space in such a way as to always be perpendicular to the contour at the point of intersection (Figure 3.172).

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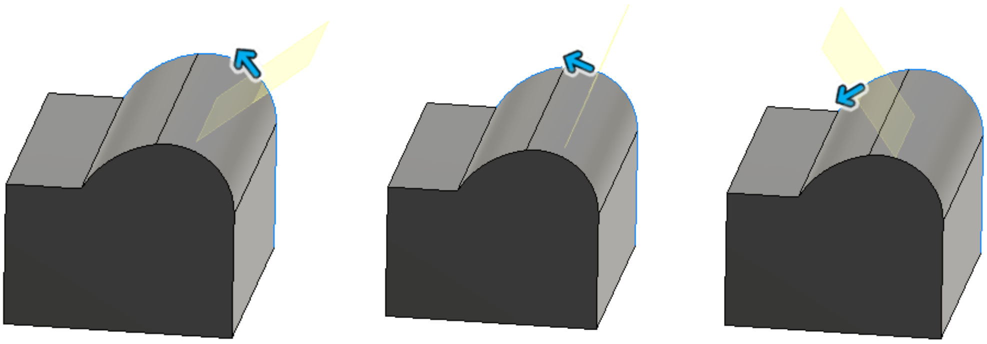

Figure 3.172

Creating a central axis through cylinder/torus

The Axis Through Cylinder/Torus command creates a central axis passing through the object's cylinder (Figure 3.173).

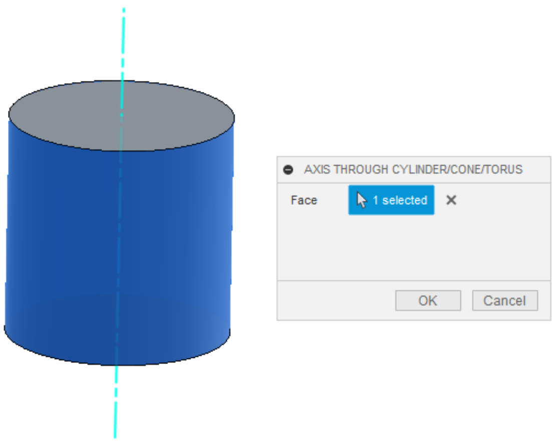

Figure 3.173

Creating an axis perpendicular at point

The Axis Perpendicular at Point command creates an axis that is perpendicular to a surface at a point indicated by the mouse (Figure 3.174).

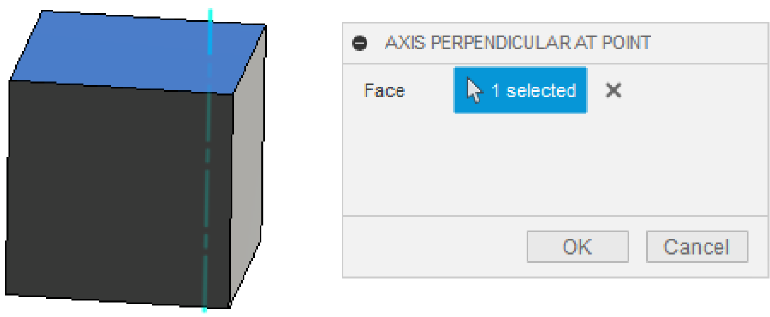

Figure 3.174

Creating an axis along two intersecting planes

With the Axis Through Two Planes command we can create an axis along the intersection of two planes. In the example below (Figure 3.175) the two planes are defined by two of the lateral surfaces of the body.

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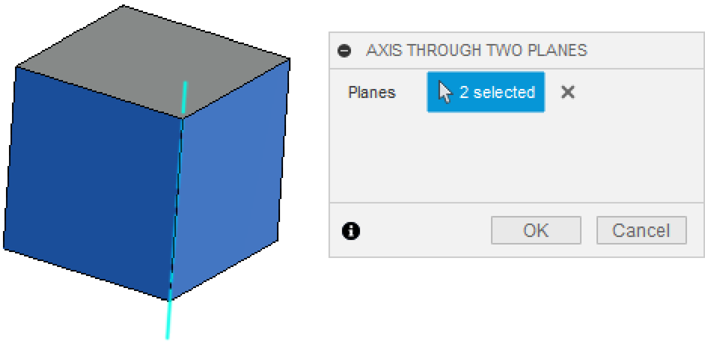

Figure 3.175

Creating an axis passing through two points

The Axis Through Two Points command is used to create an axis defined by two points in the workspace. After executing the command, we use the mouse to select two points in the workspace (or vertices in a three-dimensional body) and Fusion 360 connects them to a common axis (Figure 3.176).

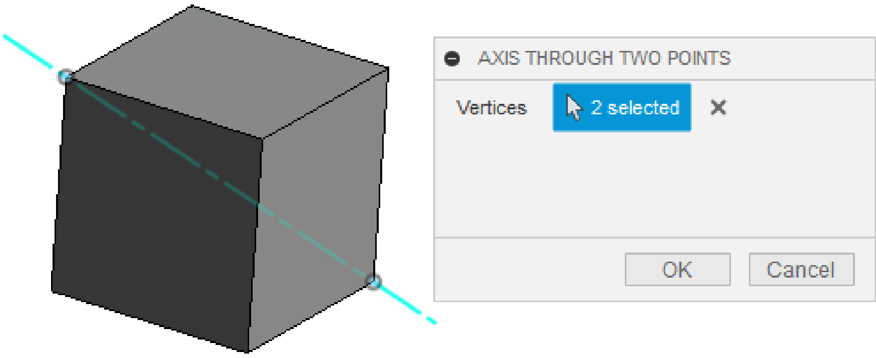

Figure 3.176

Creating an axis along an edge

The Axis Through Edge command creates an axis along the edge of a three-dimensional body (Figure 3.177).

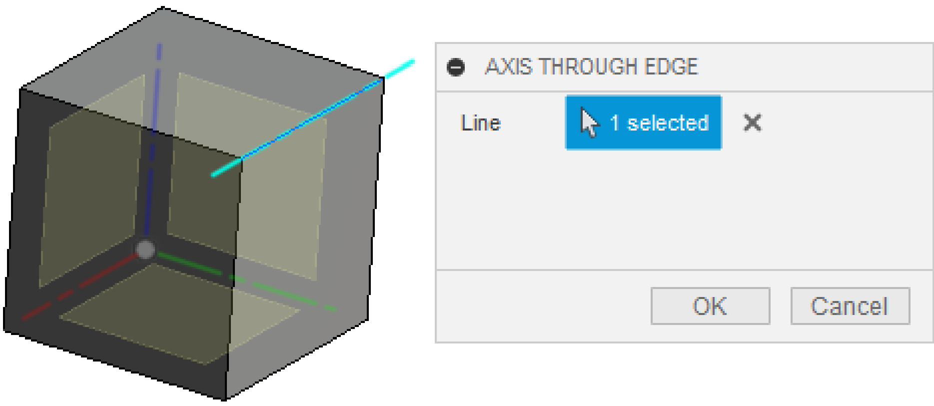

Figure 3.177

Creating an axis perpendicular to a plane at a point

The Axis Perpendicular to Face at Point command creates an axis perpendicular to a plane or surface specified by the mouse, which also passes through a point selected by the mouse (Figure 3.178).

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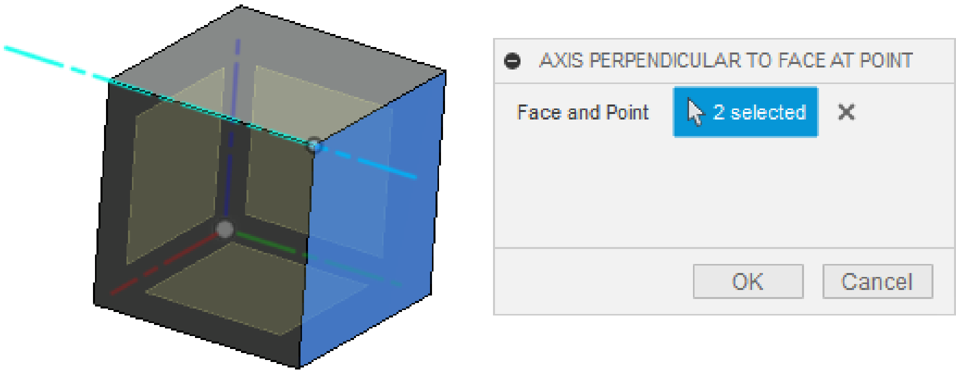

Figure 3.178

Creating a point at vertex of a body

The Point at Vertex command creates a point on a specific vertex of a three-dimensional body. Figure 3.179 shows the command's principle of operation.

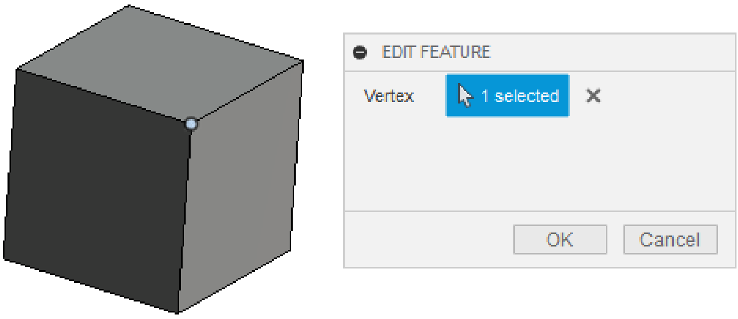

Figure 3.179

Creating a point at the intersection of two edges

The Point Through Two Edges command creates a point at the intersection of two body edges or at the intersection of an edge with a construction axis (Figure 3.180).

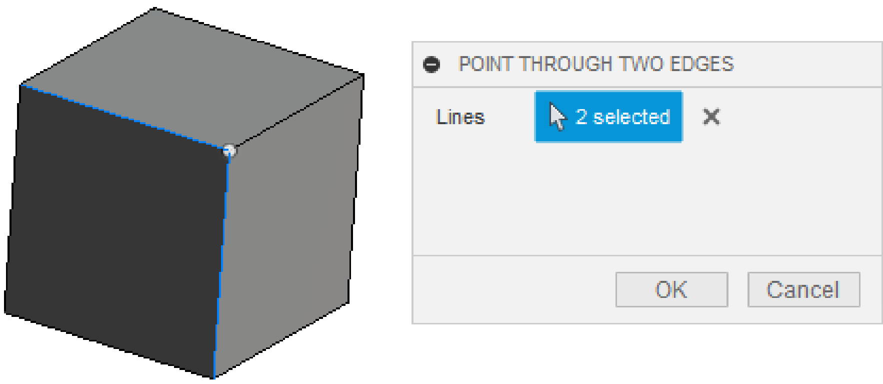

Figure 3.180

Creating a point at the intersection of three planes

The Point Through Three Planes command creates a point in space where three planes (or planar surfaces) intersect. It is mandatory to ensure that the selected planes are not parallel to each other (Figure 3.181).

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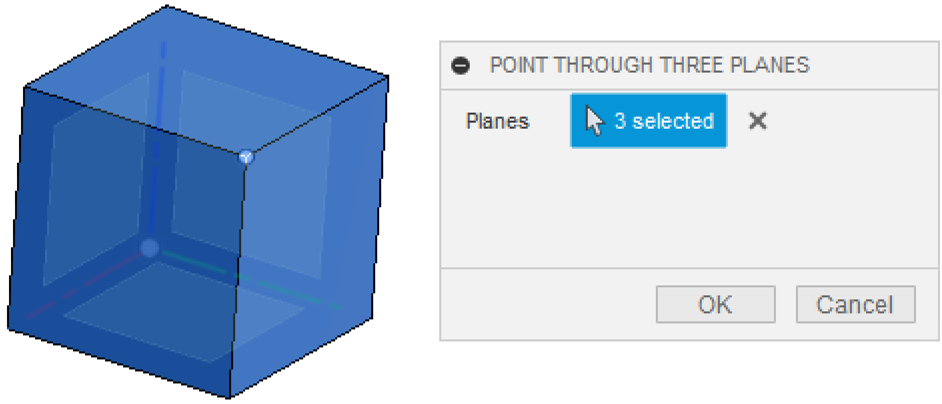

Figure 3.181

Creating a point at the center of a circle, sphere or torus

Using the Point of Circle/Sphere/Torus command, we create a point whose position in space is determined by selecting a circle from a body or by selecting a body of a sphere or a torus (Figure 3.182).

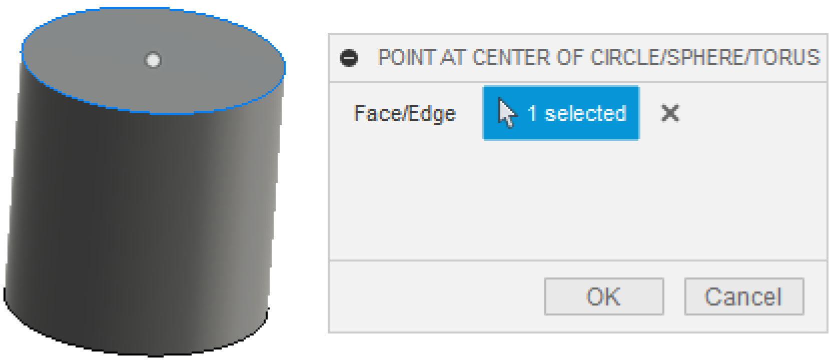

Figure 3.182

Creating a point at the intersection of an edge and a Plane

The Point at Edge and Plane command creates a construction point that lies at the intersection of a three-dimensional body's linear edge or axis and a plane or face (Figure 3.183).

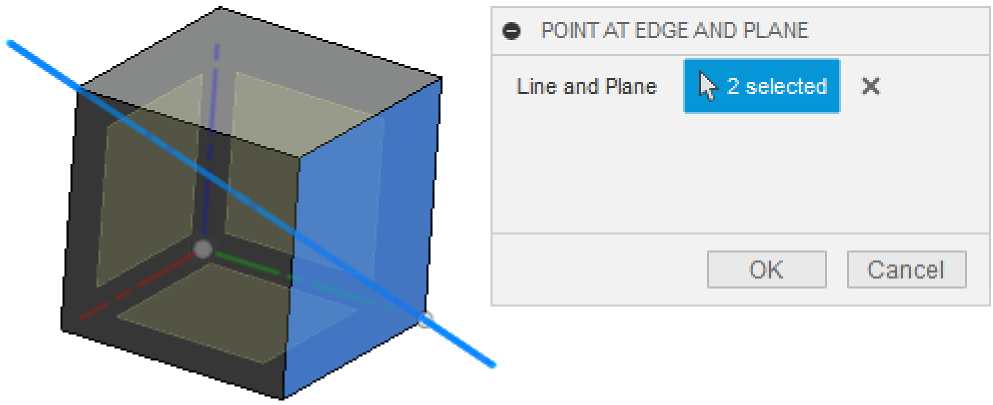

Figure 3.183

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Creating a point along a path

Using the Point Along Path command creates a construction point lying at a specified distance along a path (Figure 3.184).

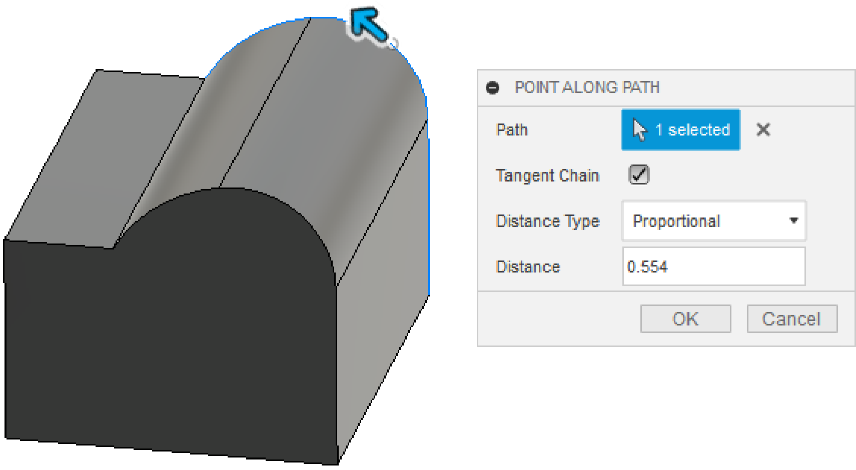

Figure 3.184

Additional video materials for self-preparation (working with the environment interface):

- https://www.youtube.com/watch?v=0ByAnvgcRV8
- https://www.youtube.com/watch?v=aXS5mgL4Yi8
- https://www.youtube.com/watch?v=4Bwm3758-qk
- https://www.youtube.com/watch?v=_F-l6E8XgBk
- https://www.youtube.com/watch?v=dgwdJsnx5mg
- https://www.youtube.com/watch?v=JmCC4TcIEmo
- https://www.youtube.com/watch?v=W0-Vk8qty2w

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Editing a 3D model

The editing of already created objects is a routine operation in CAD design. When creating three-dimensional objects, it is often necessary to edit some of their features, such as planes, edges, etc., and this is an essential part of creating the final three-dimensional object. In Fusion 360, the commands for editing objects are divided into a separate group, collectively called Modify (Figure 4.1).

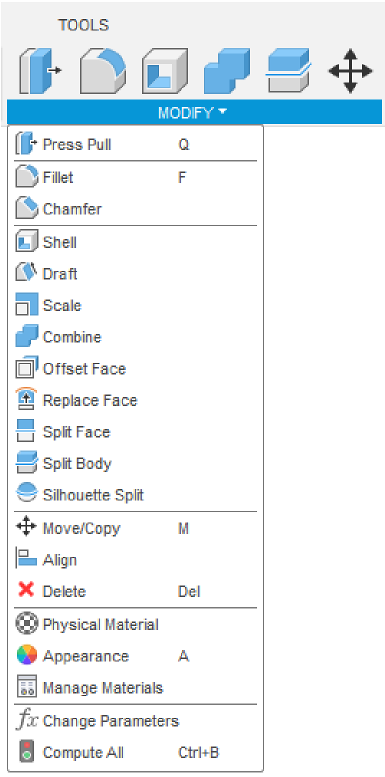
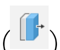


Figure 4.1

The group of commands includes both those that assist the editing of three-dimensional objects shapes' and commands that change the objects' physical features and appearance.

Editing by using the Press Pull command

The Press Pull command () is used to quickly modify individual surfaces or edges of a 3D object.

Let us take a look at a three-dimensional body with the shape presented in Figure 4.2.

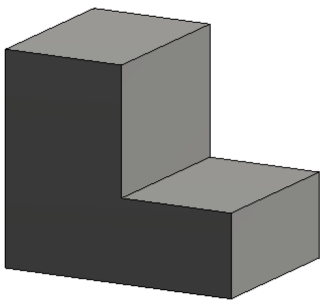


Figure 4.2

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After starting the Press Pull command, we use the mouse to select one of the body's surfaces (Figure 4.3).

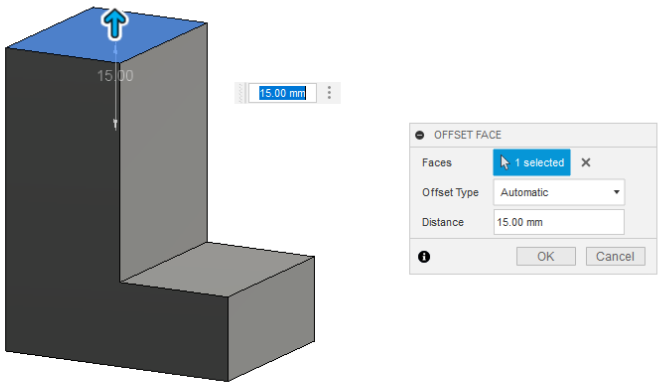
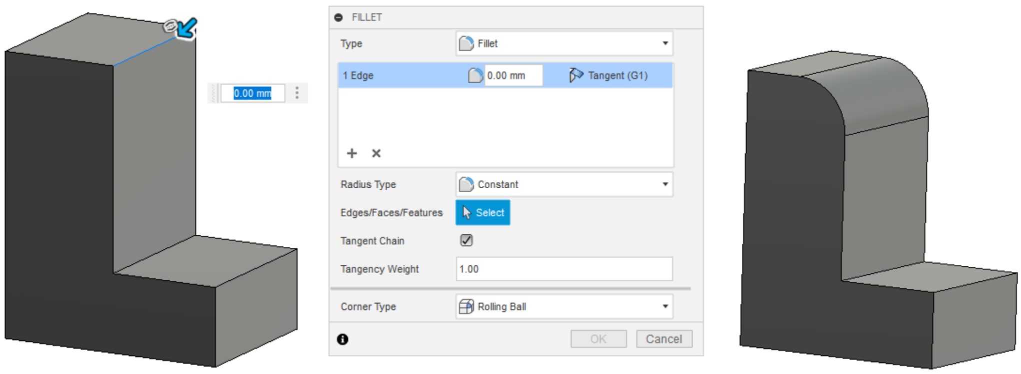


Figure 4.3

We can move the selected surface with the mouse. Alternatively, we can move it by selecting the desired distance in the command menu.

If, instead of selecting a surface, we select an edge (or several edges) (Figure 4.4, a) of the three-dimensional body, then the Press Pull command can be used to round (set a fillet) the selected edge (Figure 4.4, b).



(a) (b)

Figure 4.4

Using the Fillet and Chamfer commands to create rounding and chamfers

When creating three-dimensional models, certain edges often need to be rounded or chamfered. These operations are performed with the help of Fillet и Chamfer commands (Figure 4.5).

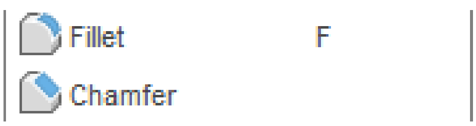


Figure 4.5

Setting of Fillets

To create a fillet, let us take the object presented in Figure 4.6. The object has a cylindrical shape and has multiple edges that can be rounded using the Fillet command.

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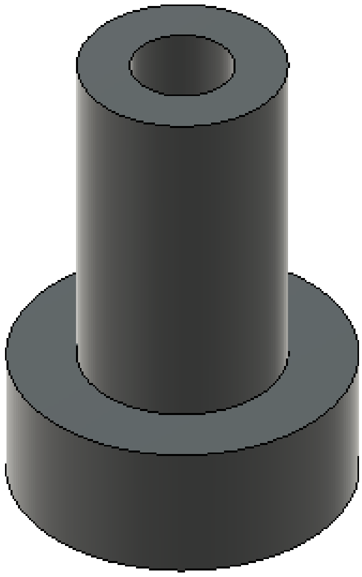

Figure 4.6

After we launch the Fillet command, we have to select with the mouse the edges of the object on which the fillet should be applied. The command allows us to define multiple fillets, and each of them can have a different radius (Figure 4.7).

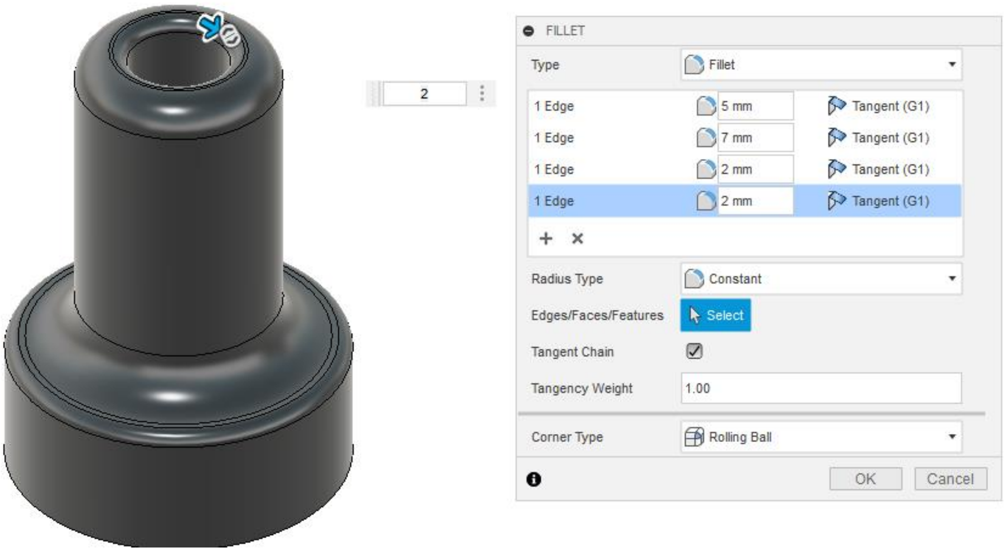

Figure 4.7

The command allows us to select the type of rounding (Figure 4.8, a) – a simple rounding controlled by defined rules in a new menu, or a full rounding defined on a selected object's surface. It also allows us to control the rounding radius (Figure 4.8, b) – the radius can be a constant (set as the length of the chord) or be variable in shape along the rounding.

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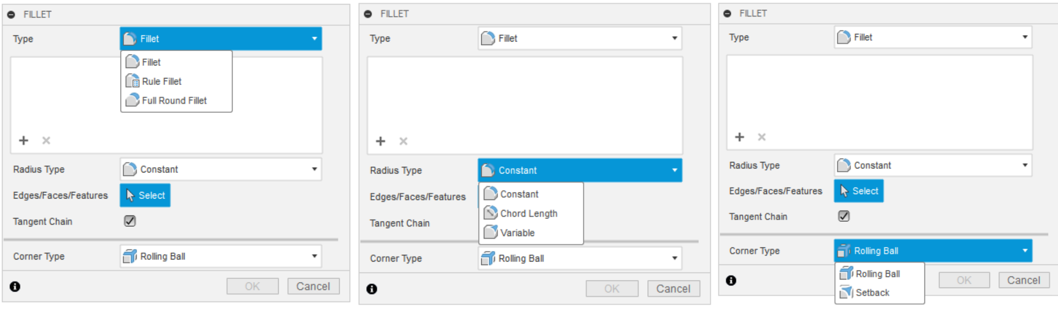

(a) (b) (c)

Figure 4.8

We can also choose the way of rounding vertices that are obtained by the rounding of three mutually intersecting edges of the object (Figure 4.8, c). The operation of the available options is presented in Figure 4.9.

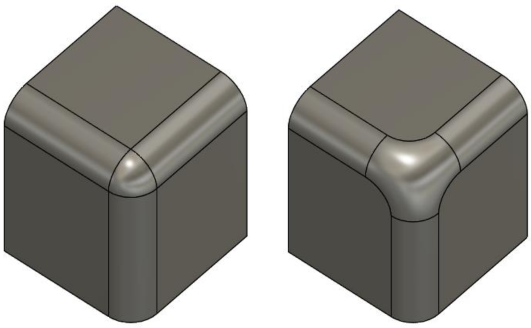

Figure 4.9

Setting of chamfers

In practice (most often when creating rotating parts) it is necessary to chamfer the edges of the parts by setting a chamfer. In Fusion 360 the Chamfer command makes it possible to create chamfers on edges, specified with the mouse. The command creates chamfers of different sizes on successively selected edges (Figure 4.10).

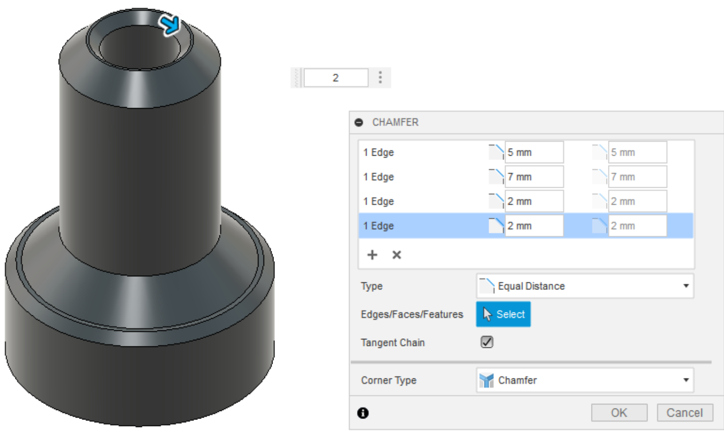

Figure 4.10

The Chamfer command allows us to select the type of chamfer (Figure 4.11, a). In the most typical case, chamfering will be done at an angle of 45°, but this angle can also be different. In the latter case, a two-dimensional setting can be used, either with a size and an adjacent angle.

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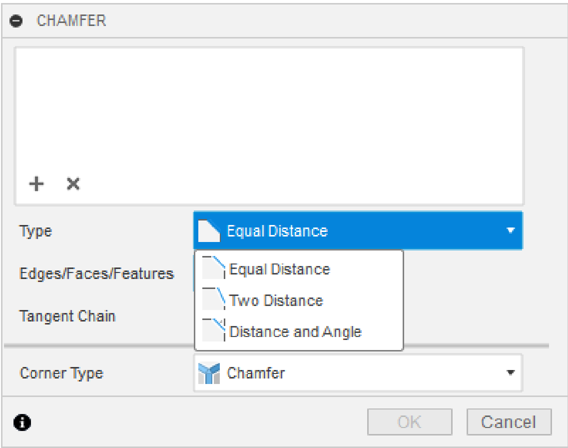
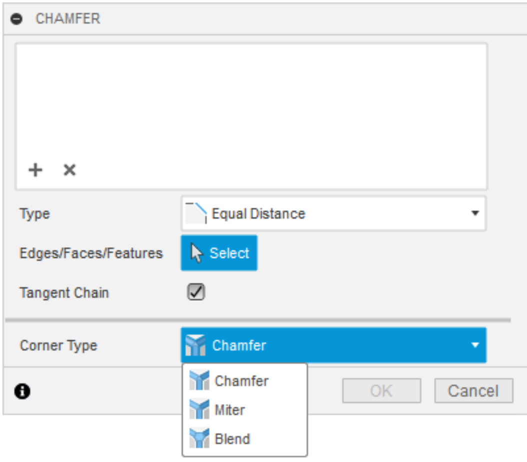
It is also possible to select the way to chamfer vertices that are defined by three edges (Figure 4.11, b).



(a)



(b)

Figure 4.11

There are three options for chamfering vertices, whose effects are presented in Figure 4.12.

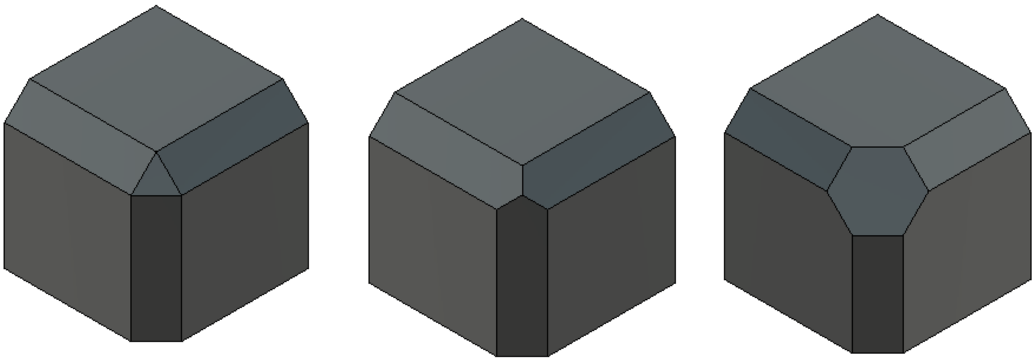


Figure 4.12

Using the Shell command

The Shell command () creates thin-walled bodies based on an existing solid three-dimensional body.

To demonstrate the operation of the command, in Figure 4.13 we present a solid volumetric body.

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Figure 4.13

After launching the Shell command, we need to specify the surface that will be removed when creating the thin-walled body. In this case, it is the uppermost surface of the body (Figure 4.14). In the command menu we need to specify the thickness of the walls.

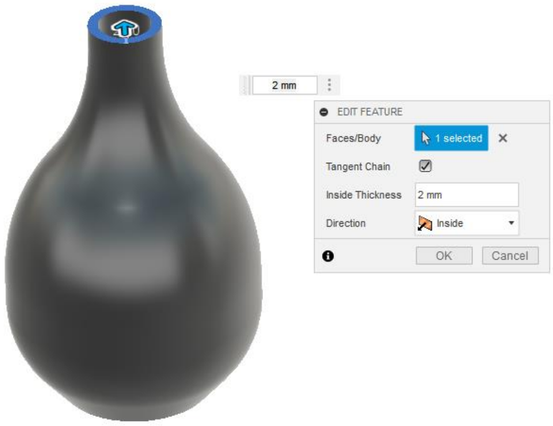

Figure 4.14

After the command is executed, the solid body is transformed into a thin walled one. Using the Section Analysis command (Figure 4.15) the object can be cut in a direction and location selected with the mouse. This is done in order to visualize the internal structure of the object.

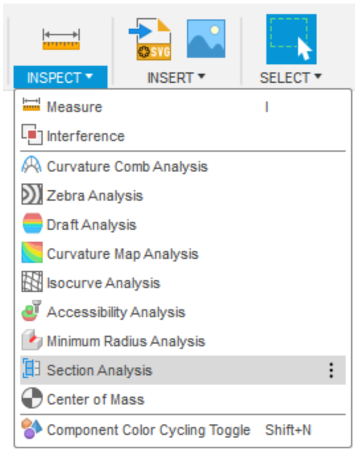

Figure 4.15

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With the help of the obtained section view, we can see that the thin-walled body has been created with a hollow interior and with the wall thickness set in the Shell command menu (Figure 4.16).

Figure 4.16

The command menu allows us to create the walls of the body internally to the contour of the original solid body, externally to it, or symmetrically on both sides of the contour (Figure 4.17). When external location of the walls is selected, the thin-walled body practically envelops the volume of the original solid body.

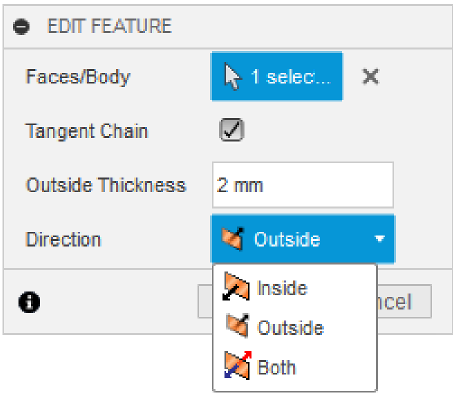
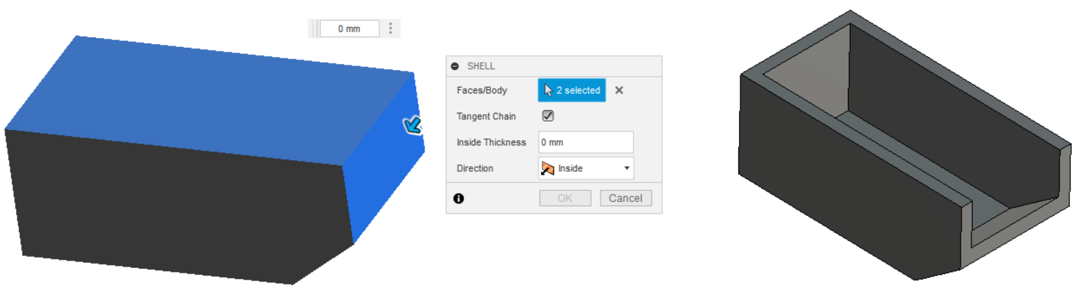


Figure 4.17

When creating a thin-walled body, the Shell command allows us to select for removal more than one surface from the original solid body. In Figure 4.18, (a) we show a three-dimensional object on which, using the Shell command, two of the surfaces have been selected for removal. In this case, the result of the command action is as presented in Figure 4.18, (b).



(a) (b)

Figure 4.18

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Using the Draft command

The Draft command (Figure 4.19) is used to set the inclination of surfaces relative to the selected direction of inclination.

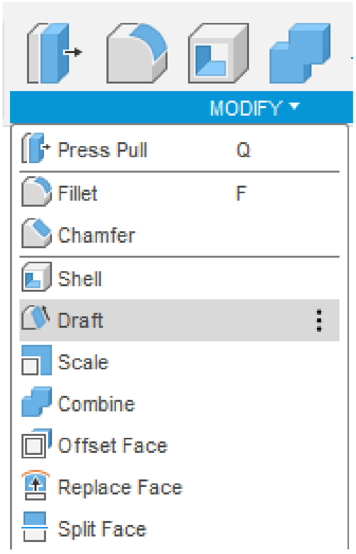

Figure 4.19

To demonstrate the principle of operation of the Draft command, Figure 4.20 presents a rotating body.

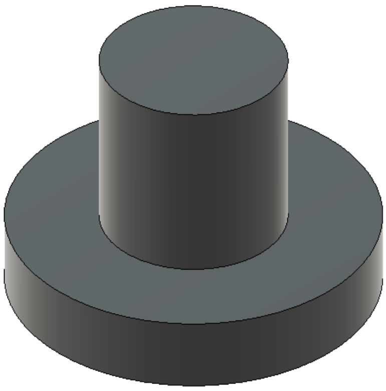

Figure 4.20

After starting the command, in the Pull Direction option in the menu we have to select the direction in which the slope will be applied. The direction is perpendicular to a plane surface indicated by the mouse (Figure 4.21).

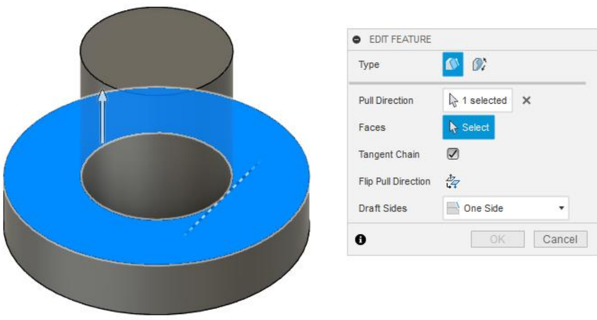

Figure 4.21

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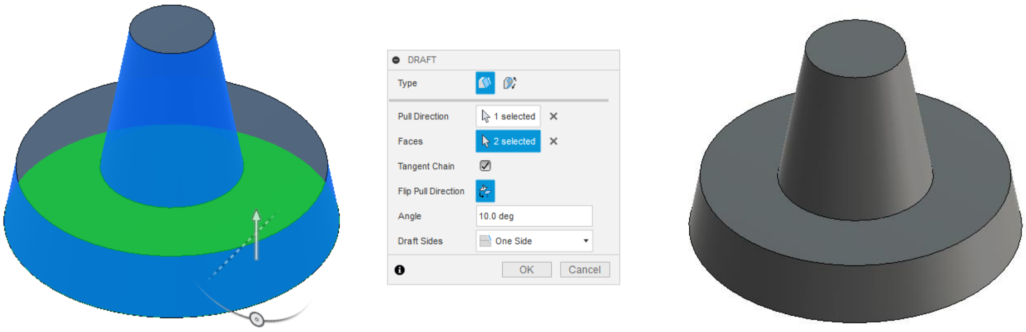
In the Faces option the surfaces on which the slope will be set are indicated – in this example the two cylindrical surfaces of the object are selected (Figure 4.22, a). The angle of inclination is also set, and the object acquires the shape presented in Figure 4.22, b.



(a) (b)

Figure 4.22

One of the main applications of the Draft command is in the production of foundry molds and setting the so-called casting inclinations (draft angles) of the mold.

Using the Scale command

We use the Scale command (Figure 4.23) to scale created objects. The command modifies the size of the object along the axes of the coordinate system with certain coefficients of modification.

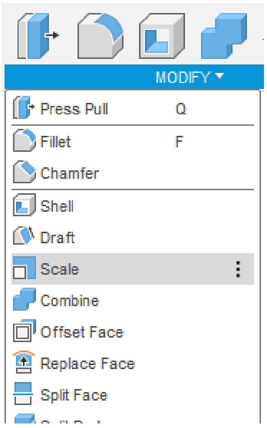


Figure 4.23

In Figure 4.24 we present two three-dimensional bodies. The bodies are of comparable size.

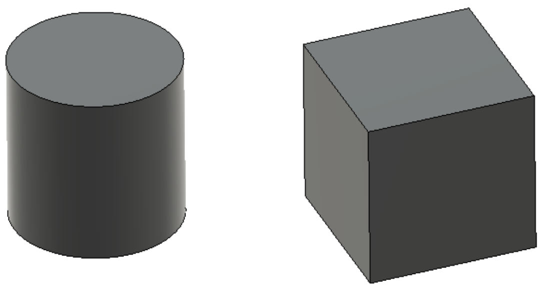


Figure 4.24

We run the Scale command in order to increase the size of one of the bodies. In the Entities field in the command menu we select the object using the mouse (Figure 4.25). The point at

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which the scaling should start is specified in the Point field. We choose the Scale Type to be the same on all axes and we set the scaling factor (Scale Factor field).

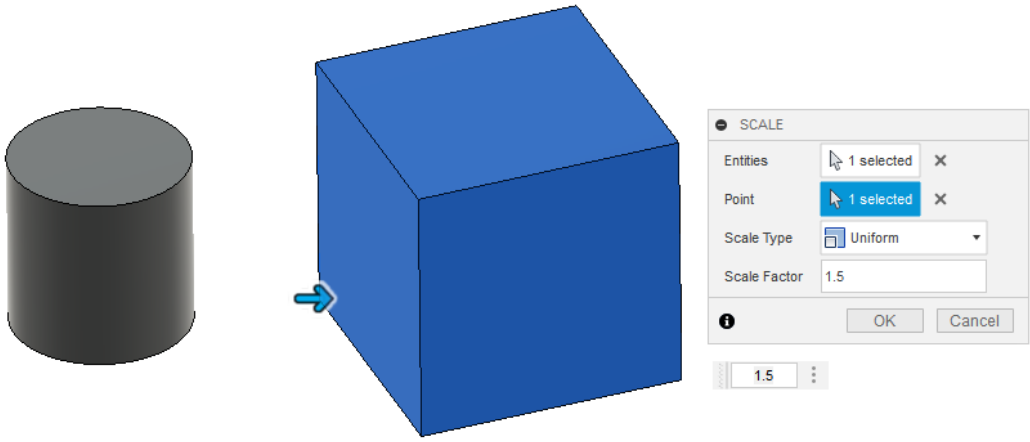

Figure 4.25

The scaling of the object can be done with a different scaling factor for each of the coordinate axes. This is possible when the Scale Type is chosen to be Non-Uniform (Figure 4.26).

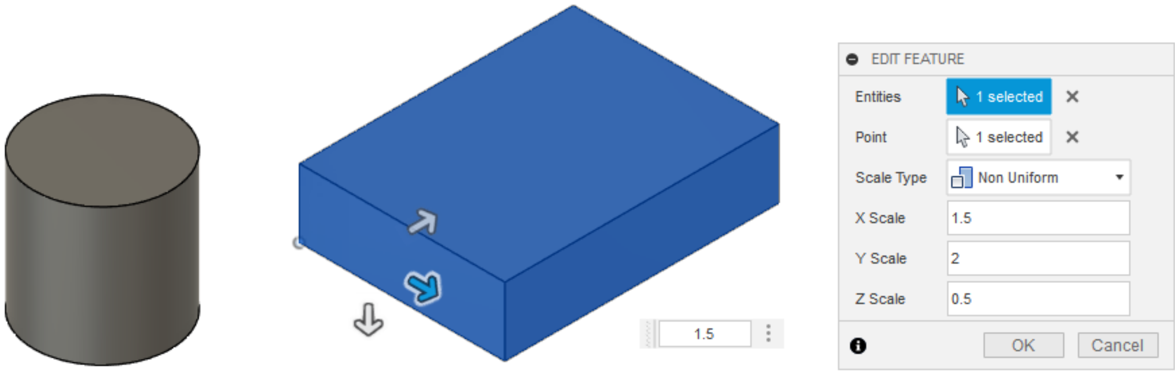

Figure 4.26

Using the Combine command

Using the Combine command () in Fusion 360, it is possible to perform different types of combinations of two or more objects. The result is a new object that includes the features of the objects involved in the operation.

Let us have two objects (Figure 4.27).

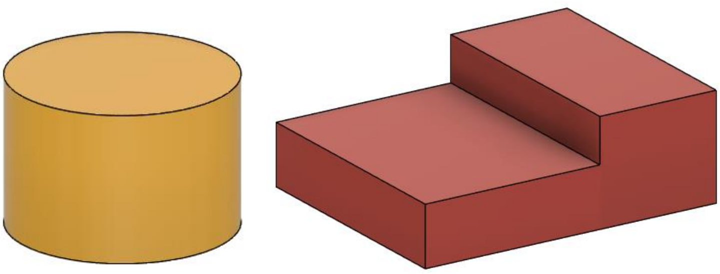

Figure 4.27

Let the objects be arranged in such a way that there is an overlap between them (Figure 4.28, a). The overlap can be seen in the section view of the objects (Figure 4.28, b).

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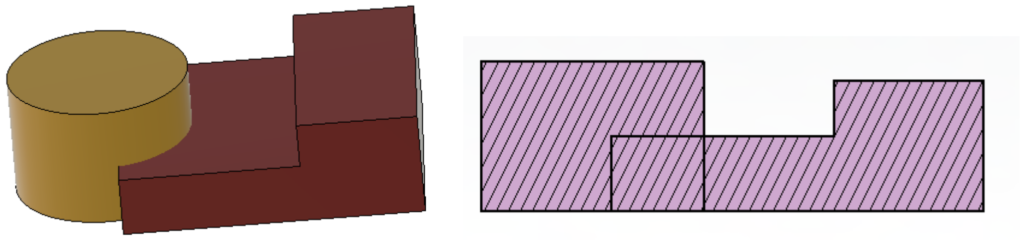


a) b)

Figure 4.28

After launching the Combine command, in the menu option we select the step-shaped body to be the Target Body, and the cylindrical body to be a Tool Body (Figure 4.29).

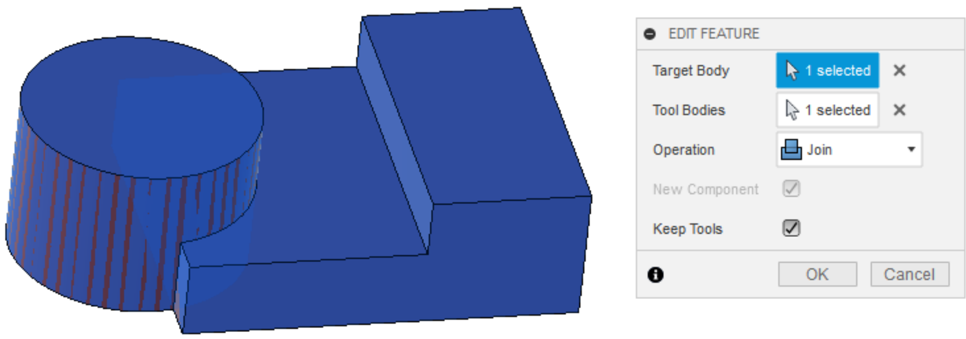
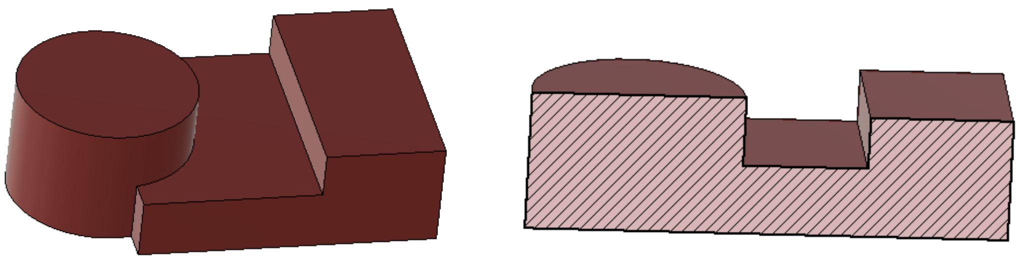


Figure 4.29

If we choose the operation to be Join, then the two objects will be combined into one (Figure 4.30, a). The section view of the new object (Figure 4.30, b) shows that a common body was obtained.



(a) (b)

Figure 4.30

If we select the Cut operation (Figure 4.31), then in this case one of the bodies is in fact a "tool" by means of which material will be removed from the other body.

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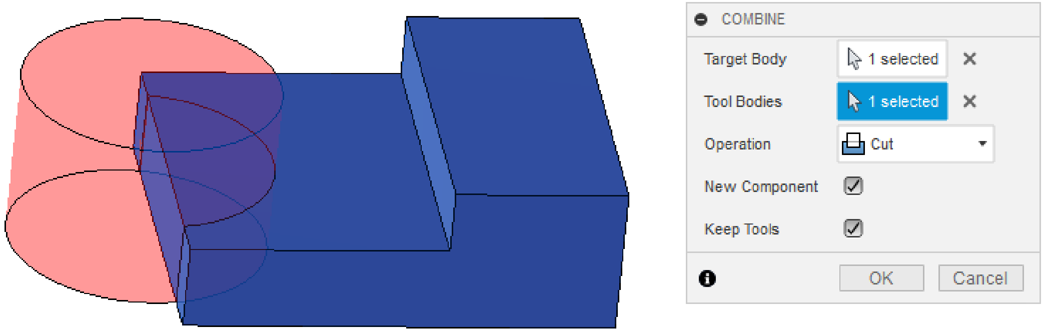

Figure 4.31

When cutting, the tool body can be preserved or removed from the project. The result of the cutting operation is presented in Figure 4.32.

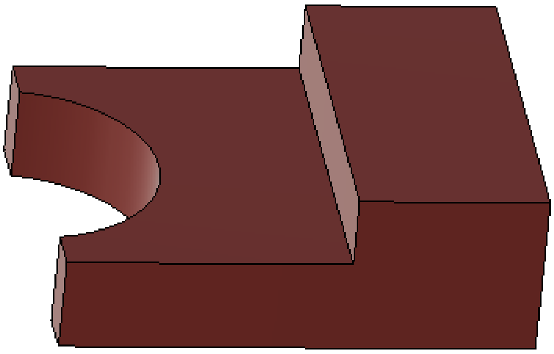

Figure 4.32

The Combine command also allows us to create a new body, which includes only the part where the two bodies overlap. To achieve this, we need to select Intersect in the Operation option (Figure 4.33).

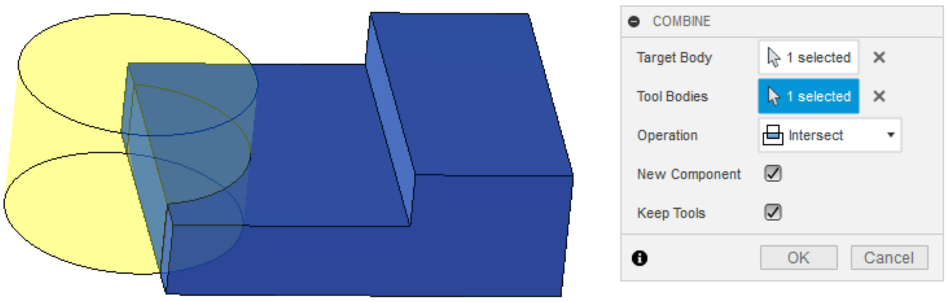

Figure 4.33

The result of the Intersect operation is presented in Figure 4.34.

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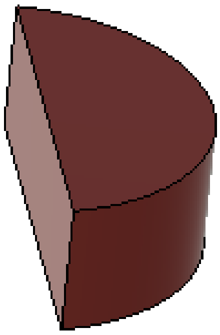

Figure 4.34

Using the Offset Face command

The Offset Face command (Figure 4.35) allows surfaces from one body to be displaced in one direction or another.

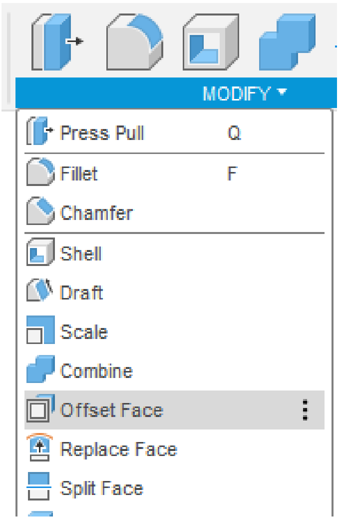

Figure 4.35

To demonstrate the action of the command, let us have a body as in Figure 4.36.

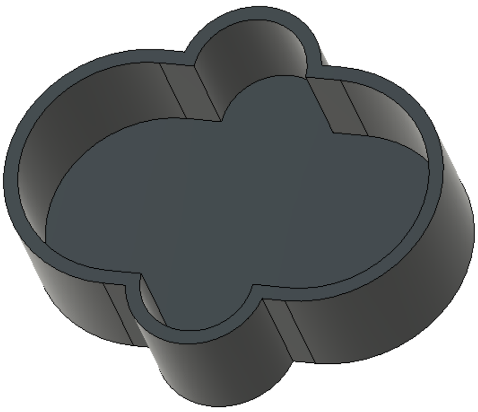

Figure 4.36

After we start the command, we need to select the surface that will be moved (Figure 4.37, a). The new shape of the object can be obtained by dragging with the mouse or by setting the offset distance in the command menu (Fig4.37, b).

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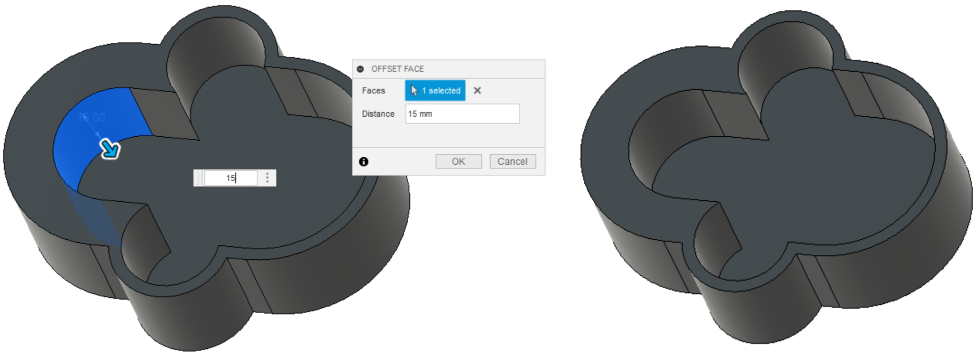

(a) (b)

Figure 4.37

Using the Replace Face command

The Replace Face command (Figure 4.38) allows the surface to be replaced or transformed.

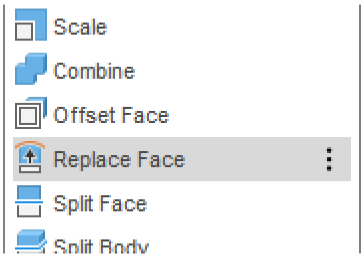

Figure 4.38

To demonstrate the operation of the command we present a three-dimensional object in Figure 4.39. We will replace the upper surface.

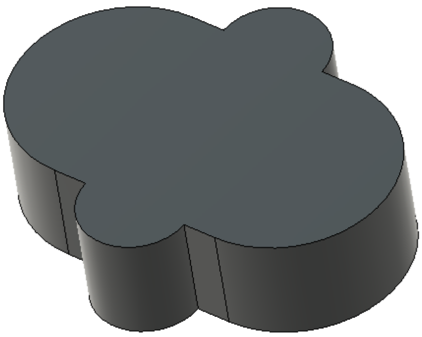

Figure 4.39

For replacement we use a surface with a complex shape, as shown in Figure 4.40. The surface itself has no volume and as an object it is in fact only a surface.

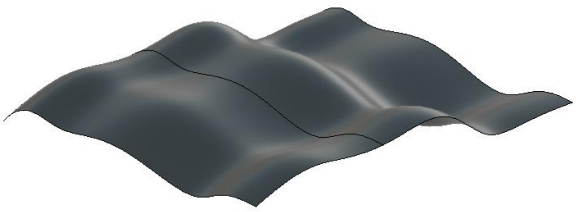

Figure 4.40

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Before replacing the surface, we position the two objects as shown in Figure 4.41

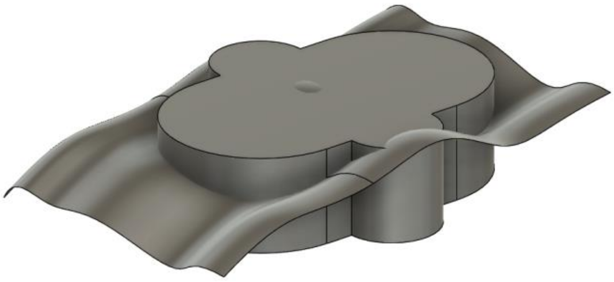


Figure 4.41

We run the Replace Face command. In the Source Faces menu option we select the top surface of the 3D body, and we specify the complex surface as Target Faces (Figure 4.42).

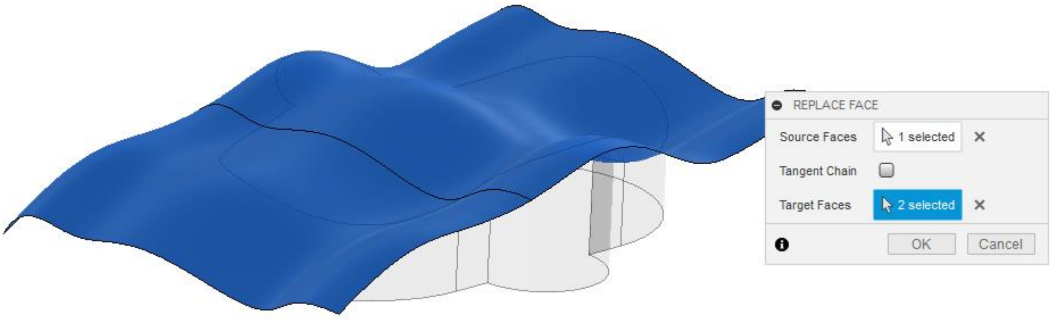


Figure 4.42

The resulting object (Figure 4.43) acquires a complex shape of its upper surface, which mimics the shape of the complex surface involved in the operation.

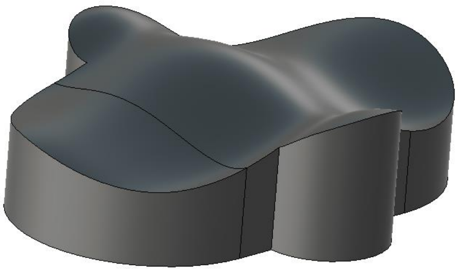


Figure 4.43

Using Split Face command

The Split Face command (Figure 4.44) is used to divide one surface into two or more parts.

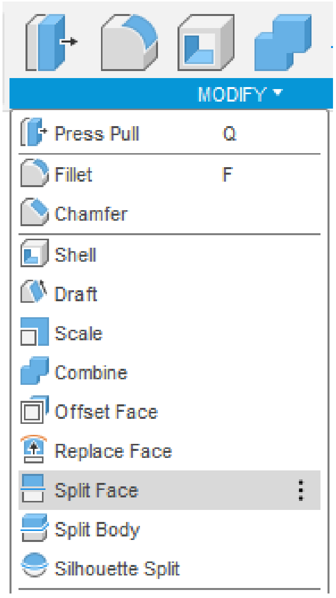
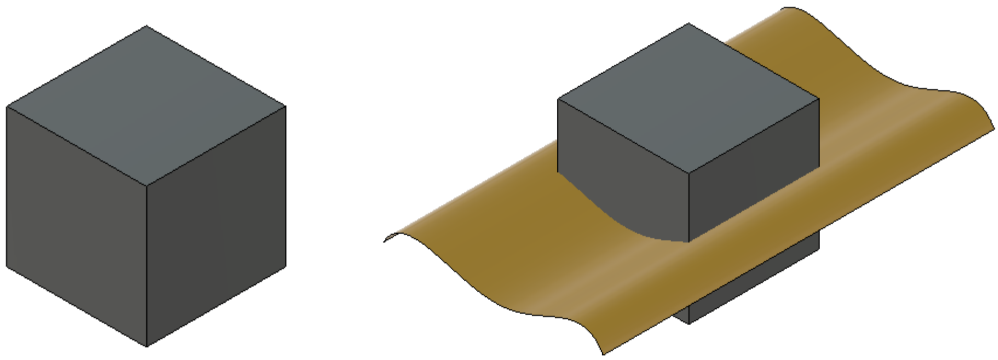


Figure 4.44

Let a three-dimensional body (Figure 4.45, a) be intersected by a surface (Figure 4.45, b), which will be used as a tool for the separation of the three-dimensional body's surface into two parts.



(a) (b)

Figure 4.45

After launching the Split Face command, we use the mouse to select the body's surface that will be split into separate parts. We select the surface passing through the body to be the Splitting Tool (Figure 4.46).

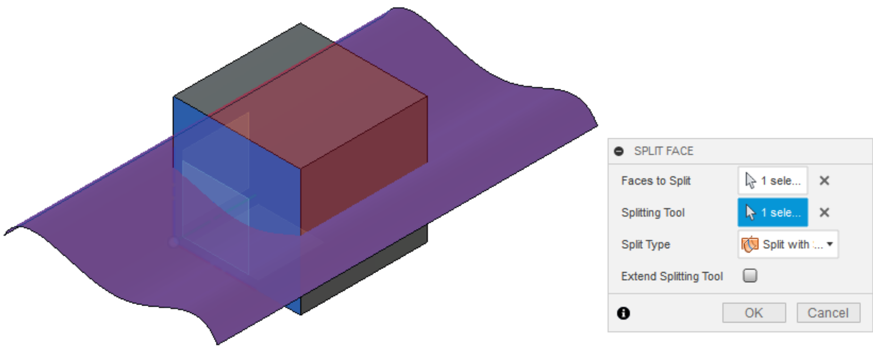


Figure 4.46

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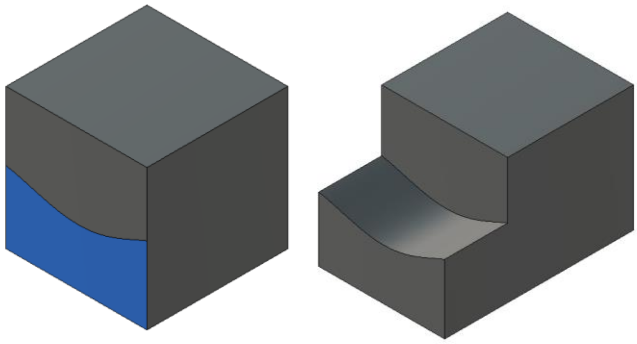
The selected surface is divided into two parts (Figure 4.47, a), after which each part can be easily moved or extruded (Figure 4.47, b).



(a) (b)

Figure 4.47

Using the Split Body command

The Split Body command (Figure 4.48) allows a three-dimensional body to be split into parts, appearing as separate bodies in the Fusion 360 file browser. In practice, this command is the opposite of the command combining two or more bodies into one. Often in the process of work the two commands are used sequentially in order to create and edit details with complex shapes.

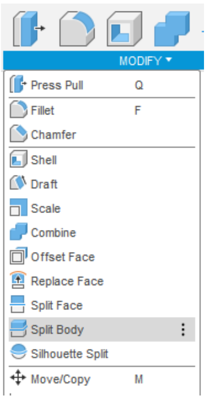


Figure 4.48

To demonstrate the operation of the command, a three-dimensional body intersected by a surface is presented below in Figure 4.49.

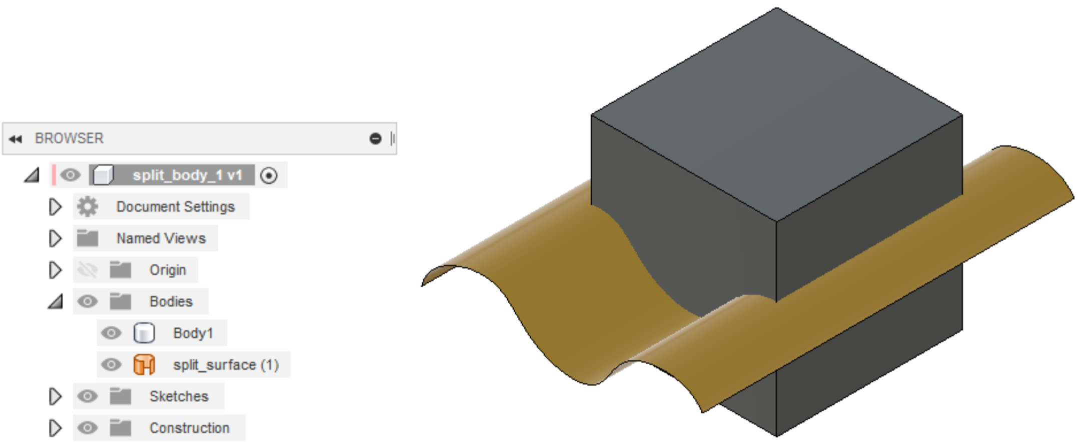


Figure 4.49

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In the dialogue box of the Split Body command, we use the mouse to successively select first the body which will be split into parts (Body to Split) and then the tool with which this action will be performed (Splitting Tools). The Splitting Tool in this case is the surface intersecting the body (Figure 4.50).

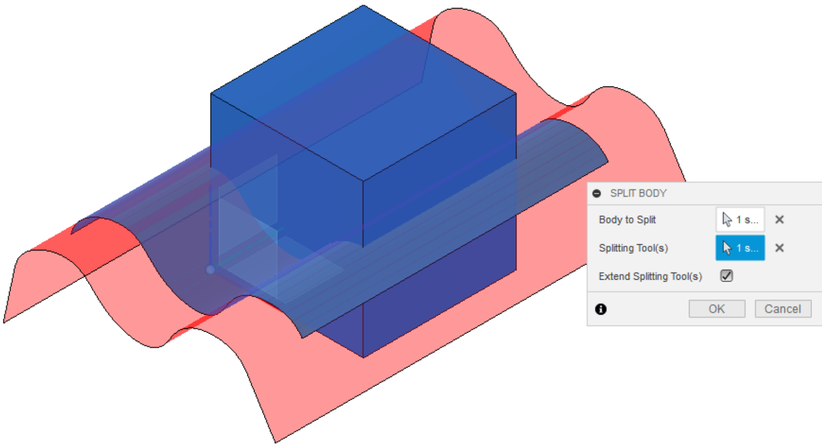

Figure 4.50

After executing the command, a new three-dimensional body appears in the browser (Figure 4.51).

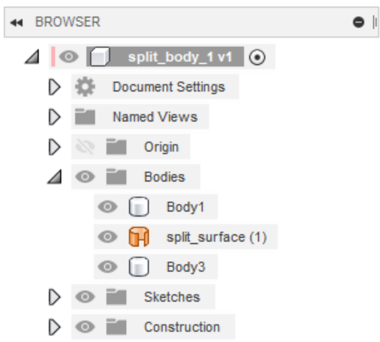

Figure 4.51

The two three-dimensional bodies are derived from the original body and together they have the same volume as the original body (Figure 4.52).

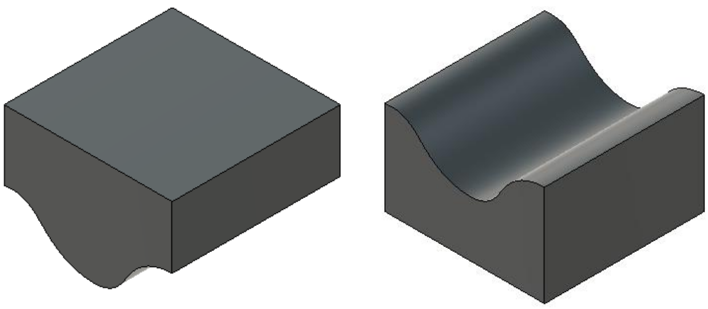

Figure 4.52

Using the Move/Copy command

The Move/Copy command ( ) moves a component, body, surface, or object from a sketch in the design workspace. The command can also be used to make copies of the selected objects.

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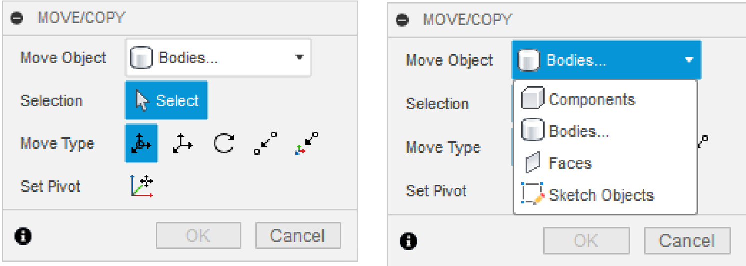
After starting the command, a menu appears (Figure 4.53, a), and we have to choose what type of object will be moved or copied (Figure 4.53, b).



(a) (b)

Figure 4.53

The following command options described in Table 4.1 can be used when moving.

Table 4.1

Type of movement	Action
- Free Move	Free movement of the object, both on the x, y, z axes in the point of rotation, and in a rotational movement around them.
- Translate	Movement along the x, y, z axes of the design or along the x, y, z axes of the respective component.
- Rotate	Rotation around a selected axis or edge of the object
- Point to Point	Relocation of the object by selecting one of its points (usually a vertex) and placing it at another point in the work area indicated by the mouse.
- Point to Position	Relocation of the object by selecting one of its points (usually a vertex) and setting the coordinates in the coordinate system of the design or component to which the point and the object should be moved.

When free movement of the selected object is performed, we can define a reference point (Set Pivot), against which the displacement or rotation are performed.

To demonstrate the operation of the Move/Copy command, let us have the three-dimensional body shown in Figure 4.54.

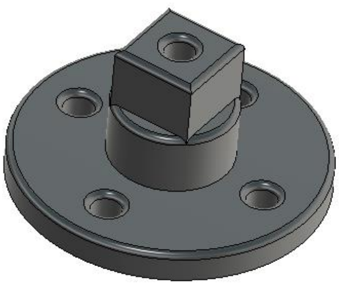


Figure 4.54

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After activating the command, in the Selection field we can select the object with the mouse. When free movement is performed, the environment allows for movement along one of the selected axes, as well as rotation around them. The displacement distances as well as the angles of rotation can be set interactively with the mouse or set manually in the corresponding fields of the command menu (Figure 4.55).

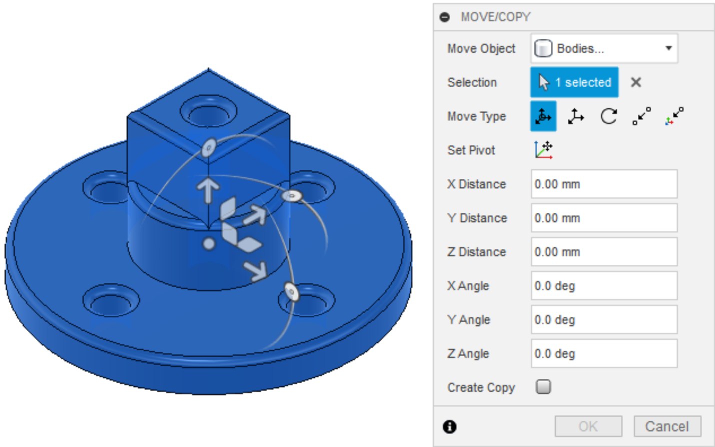

Figure 4.55

Now let the X-axis rotate at 120 degrees (Figure 4.56).

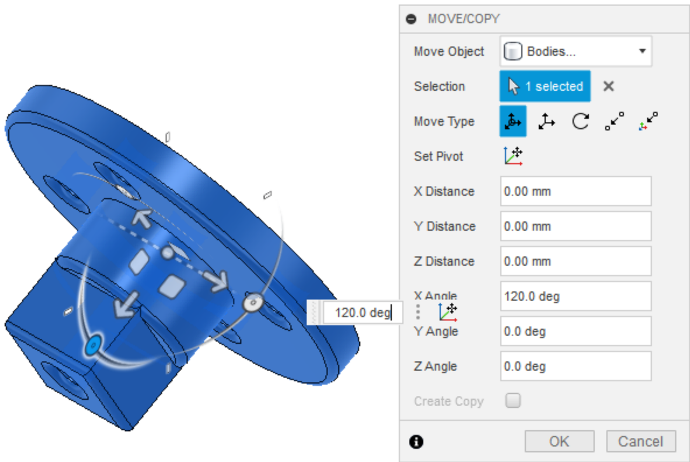

Figure 4.56

As a result, the object acquires a new position in space (Figure 4.57).

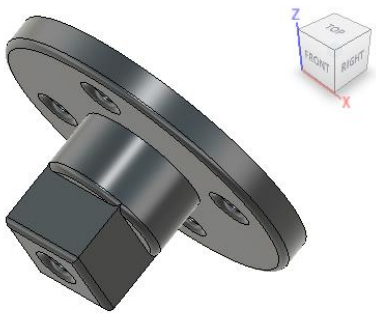

Figure 4.57

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The Move/Copy command can be used also if an object needs to be copied. In this case, after selecting the object to be copied, it is necessary to check the Create Copy menu option (Create Copy ☑). Then the new object can be placed in the workspace by dragging (Figure 4.58).

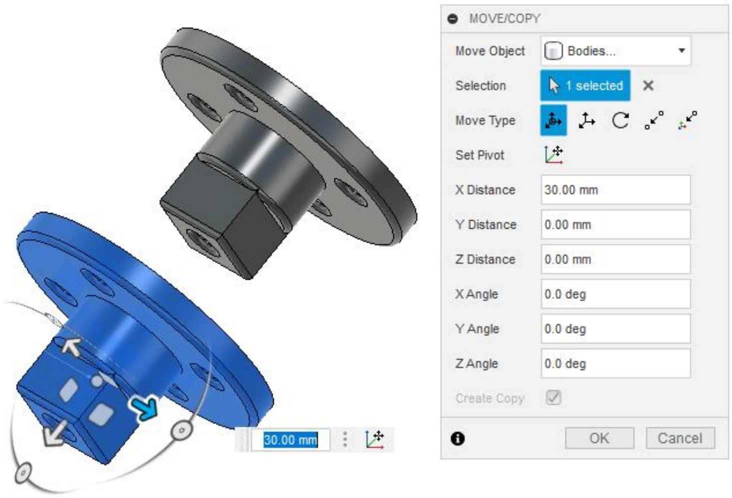

Figure 4.58

When an object has been copied, the new object appears in the browser (Figure 4.59).

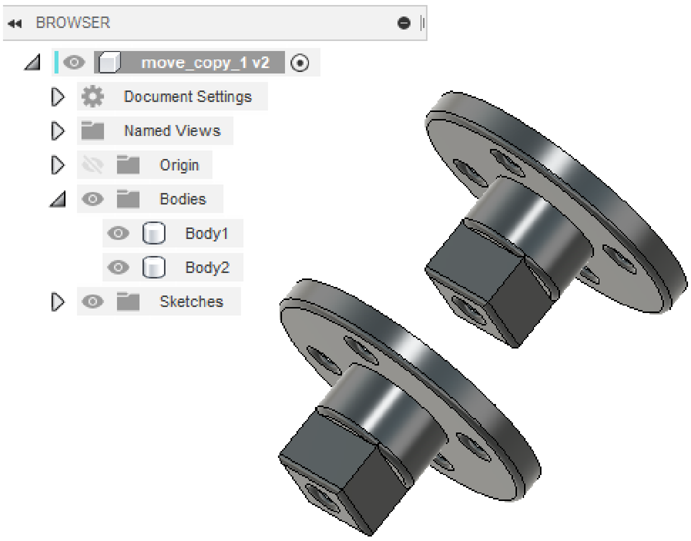

Figure 4.59

One very useful feature of the Move/Copy command is the ability to move surfaces from a three-dimensional body. We can select one of the side surfaces of the three-dimensional body shown below. In order to be able to select a surface, we need to select Faces in the Move Object field (Figure 4.60).

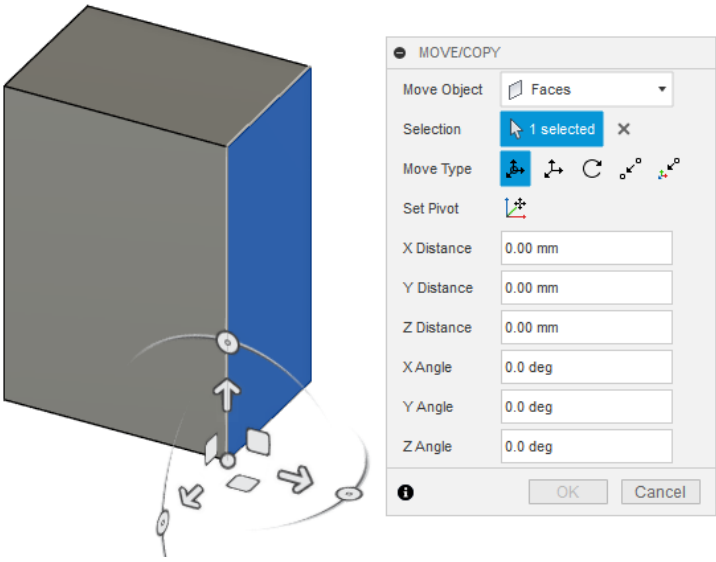

Figure 4.60

It is also necessary to set a reference point against which the displacement and rotation will be performed.

We set rotation by 20 degrees on one of the axes of the selected surface, and as a result the body changes its shape (Figure 4.61).

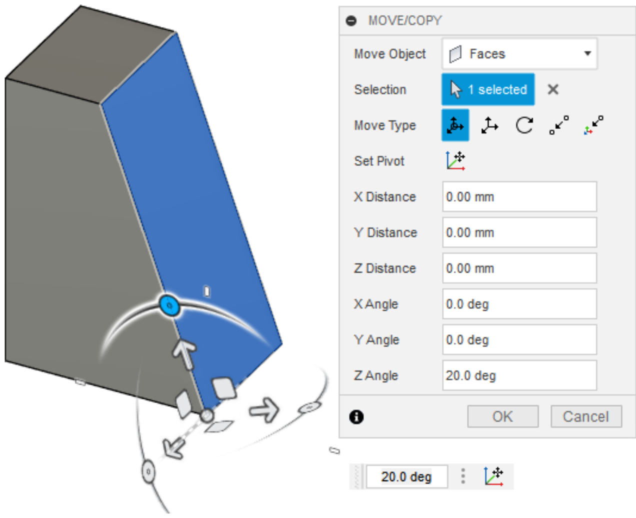

Figure 4.61

Using the Align command

The Align command (Figure 4.62) is used to align bodies or components in the design workspace.

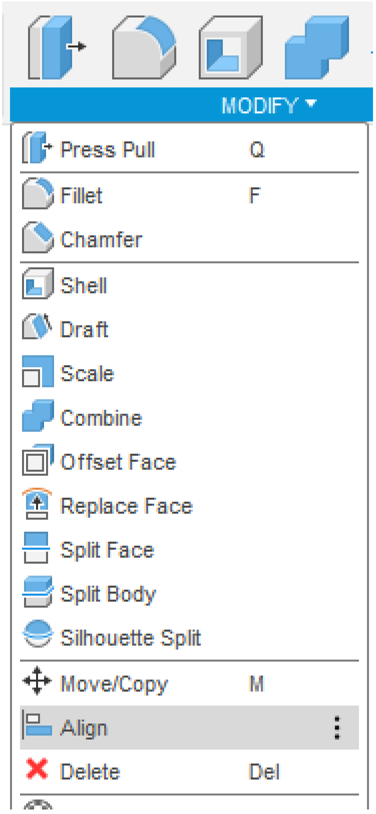

Figure 4.62

In Figure 4.63 we present two bodies on which, after starting the Align command, we select one distinctive point, towards which the alignment should be performed. In the command dialogue box, the distinctive point selected using the option *From* is aligned to the other body's distinctive point set in the *To* option of the menu.

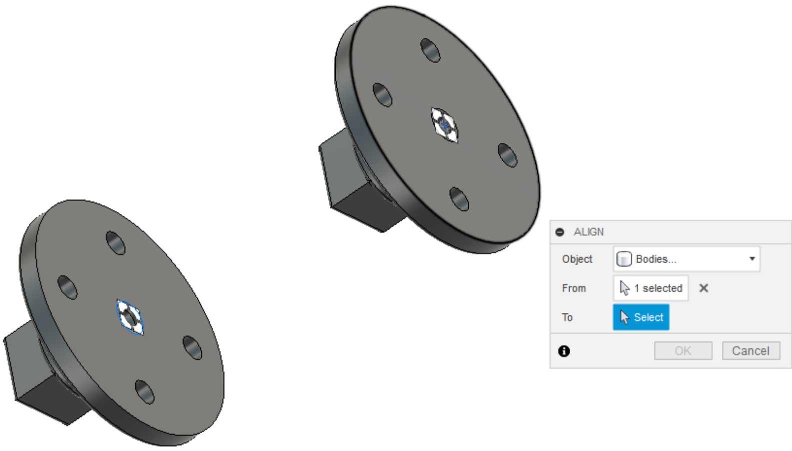

Figure 4.63

When aligning, it is possible for the environment to arrange the objects in an unexpected way or make them overlap (Figure 4.64, a). Therefore, the Flip and Angle options are provided in the command menu to reverse the position of an aligned object or to change its orientation in space. When the Flip option is activated, the desired location of the objects can be obtained (Figure 4.64, b)

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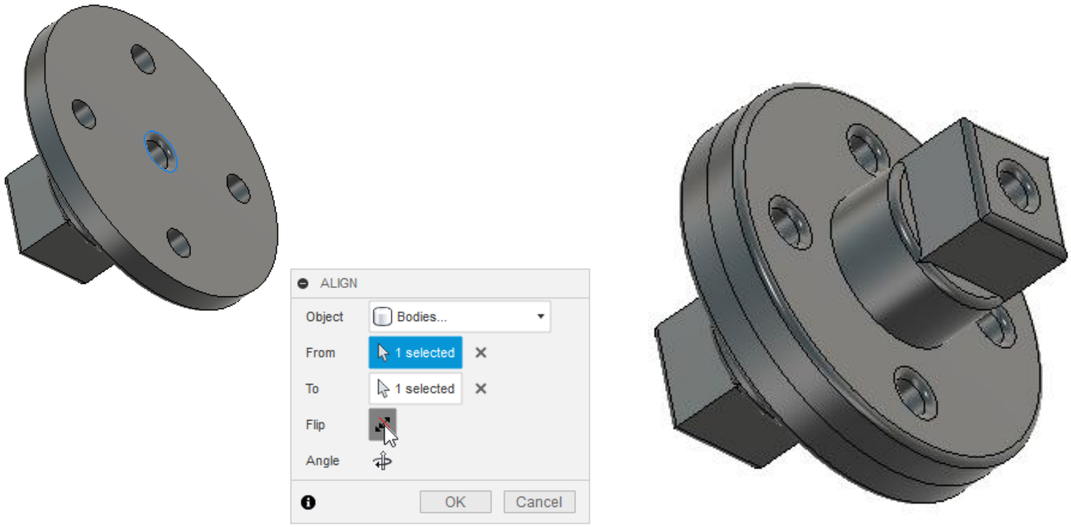


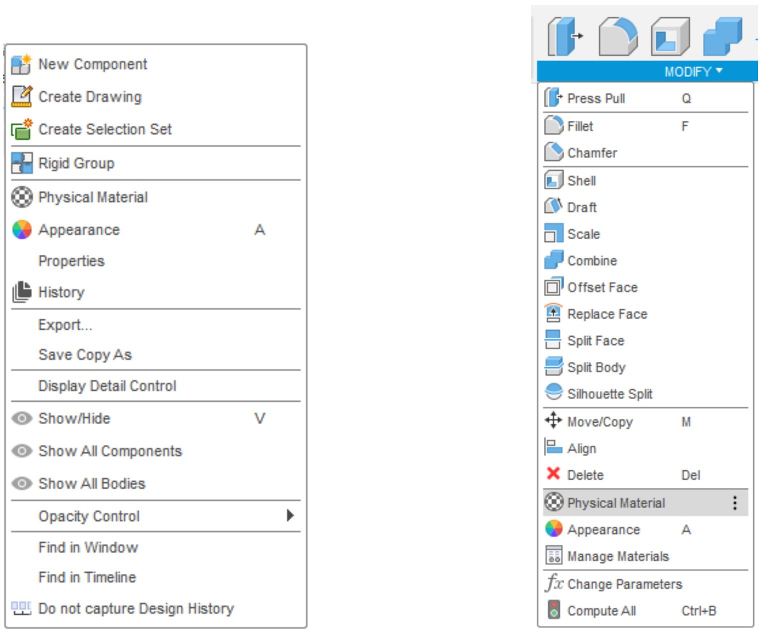
(a) (b)

Figure 4.64

Using the Physical Materials command

Using the Physical Materials command, we can define the physical properties on the objects – bodies and components. This is done by assigning a material from which they should be produced. Defining physical properties of objects is necessary for performing simulations in the working environment. Based on the objects' physical properties we can determine their reaction to the impact of different forces and loads.

The Physical Materials command can be activated either by right-clicking on the corresponding body or component (Figure 4.65, a) or via the main Modify menu (Figure 4.65, b).



(a) (b)

Figure 4.65

After running the command, a dialogue box containing a library of various materials that can be assigned to objects will open (Figure 4.66).

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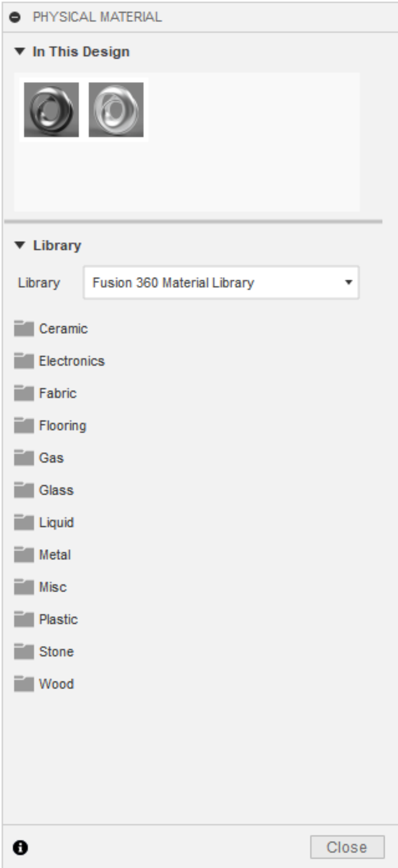

Figure 4.66

The assignment is done by selecting the respective material with the mouse and dragging it over the object (Figure 4.67). The material appears in the list of materials used in the design.

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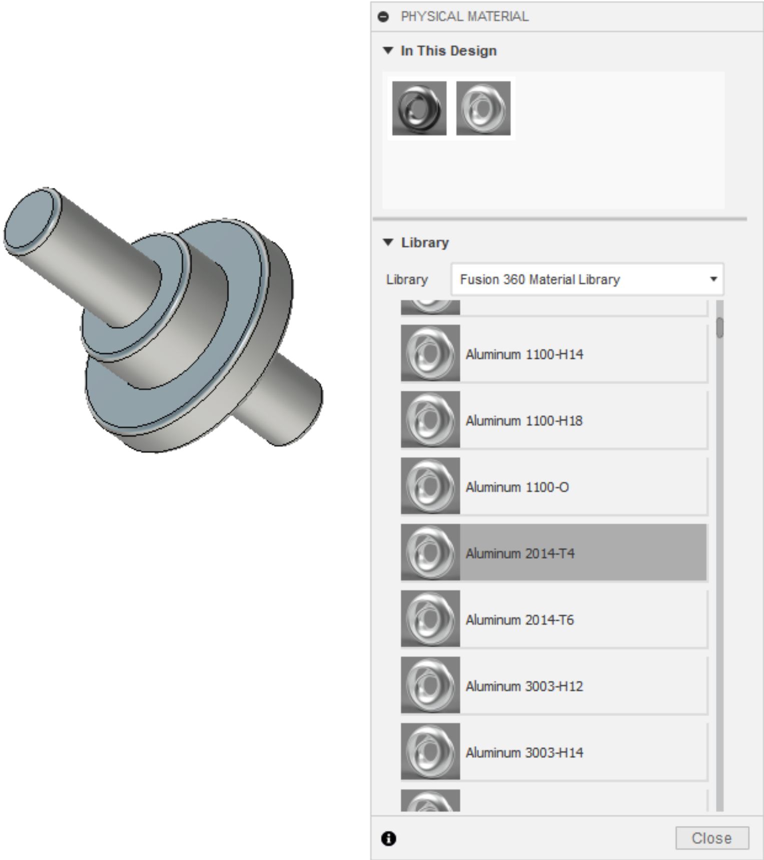

Figure 4.67

Using the Appearance command

The Appearance command is used to alter the appearance of a 3D object by changing its texture. The command menu contains a library with a large number of textures that resemble the appearance of different materials or represent possible colours that can be applied to bodies and components. Many of the textures included in the library need to be downloaded to become available.

The Appearance command can be activated by right-clicking on the corresponding body or component (Figure 4.68, a) or from the main menu Modify (Figure 4.68, b).

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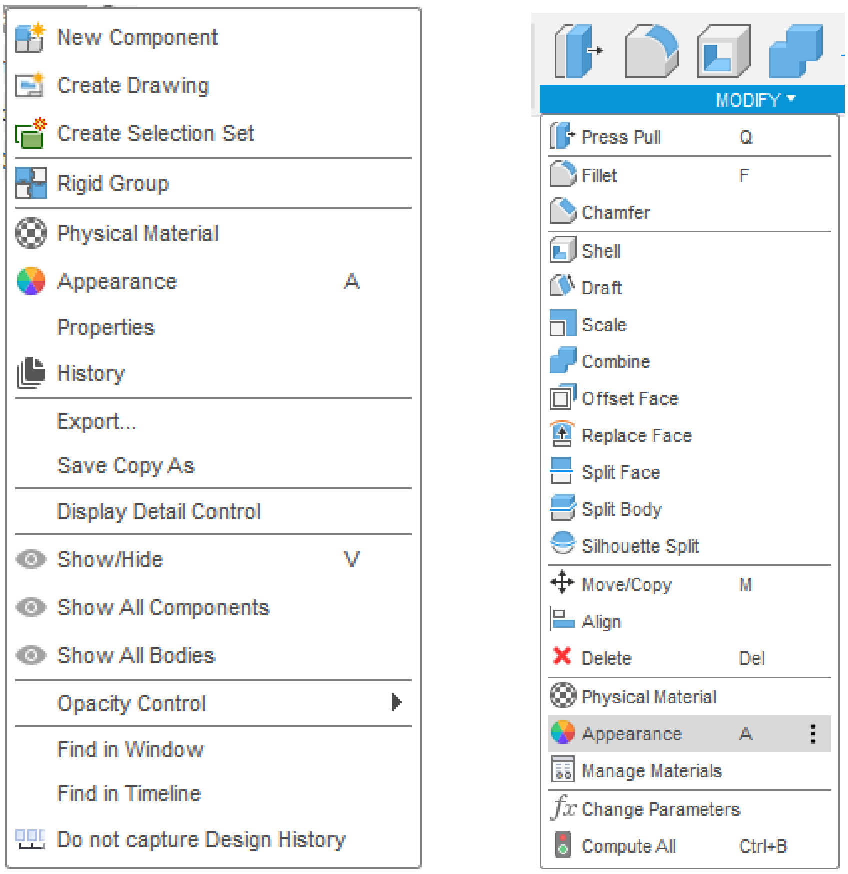

(a) (b)

Figure 4.68

When the command is activated, the selected texture is assigned to the corresponding body or component by dragging the mouse. As a result, the surface changes completely (Figure 4.69).

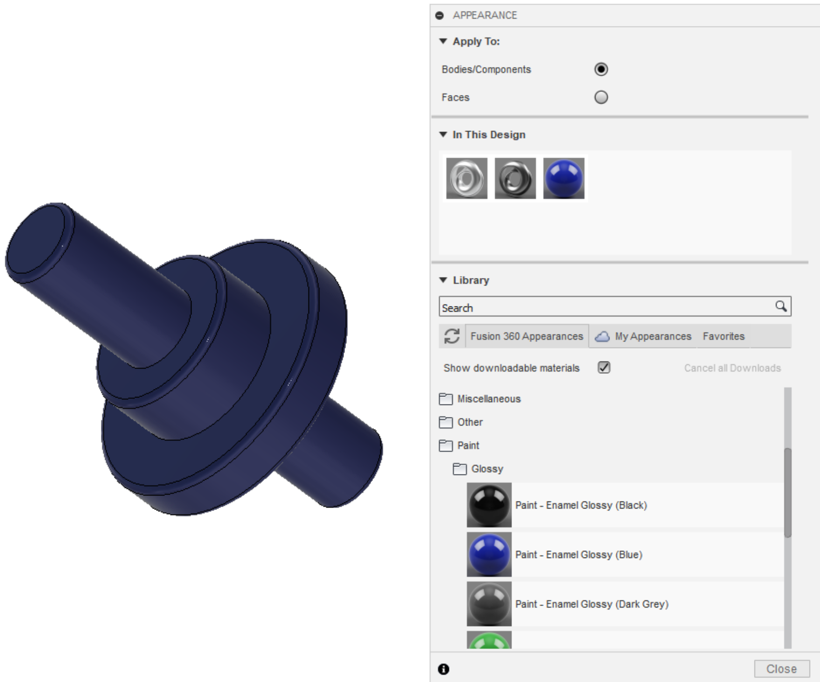

Figure 4.69

The command also allows us to selectively change the appearance of certain surfaces. To do this, we need to activate the Faces option in the command's menu (Figure 4.70).

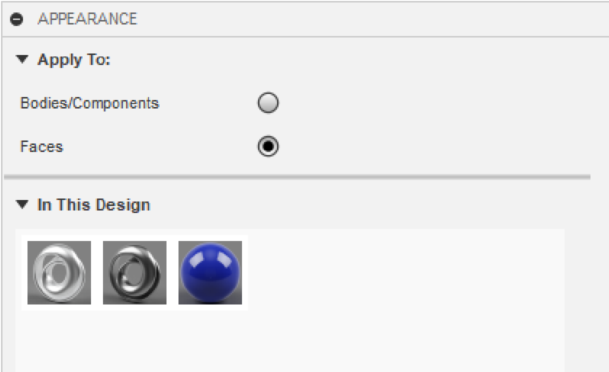

Figure 4.70

In this case, only the surfaces selected with the mouse change their colour (Figure 4.71).

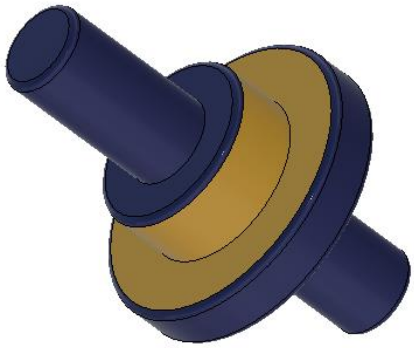

Figure 4.71

Using the Change Parameters command

The Change Parameters command is used to parametrically set object dimensions. The command can be activated from the list of commands in the Modify menu (Figure 4.72).

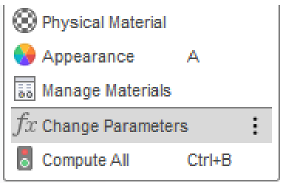

Figure 4.72

The use of parameters in the modelling process allows us to easily change the object's size and shape only by changing the value of certain parameters. After activating the command, a table-like dialogue box opens, in which the user-defined parameters can be entered (Figure 4.73).

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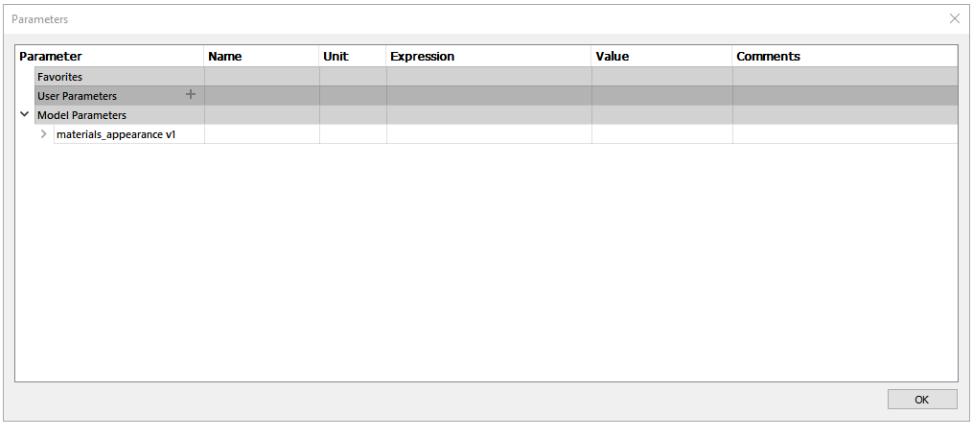
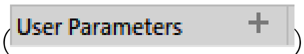


Figure 4.73

A new parameter can be added by pressing the "+" sign on the User Parameters section (
). A menu for entering a parameter opens, and it requires a name of the parameter in the Name field and a value in the Expression field. The value can be a constant (number) or a formula (Figure 4.74).

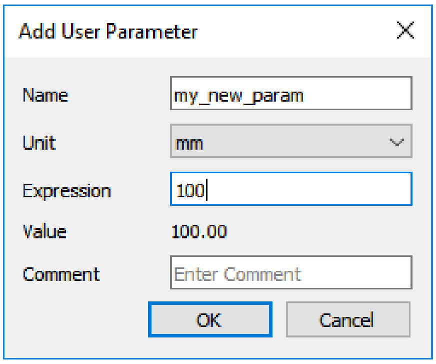


Figure 4.74

To demonstrate the effect of the parameters, in the table shown below we have entered three user parameters with their respective values (Figure 4.75).

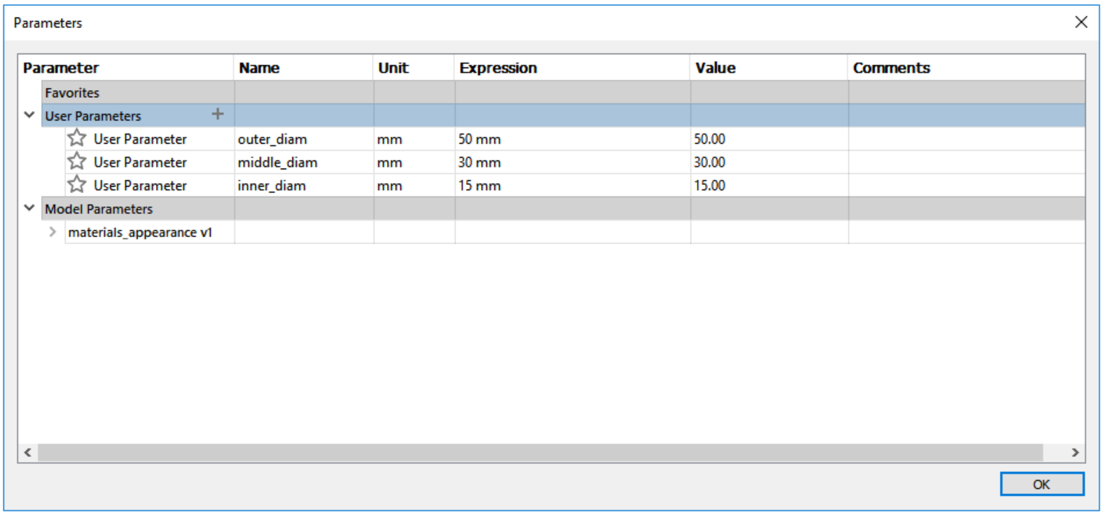


Figure 4.75

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We have created a sketch with three concentric circles. We have set the dimensions of the circles by entering a parameter name instead of a direct value in the size field (Figure 4.76).

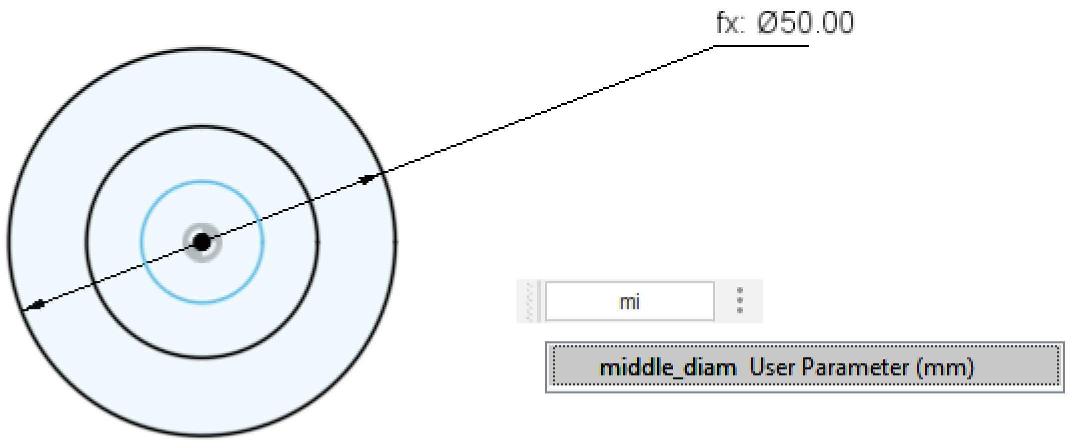

Figure 4.76

When a dimension is set as a parameter, the symbol fx appears in front of the dimension name (Figure 4.77).

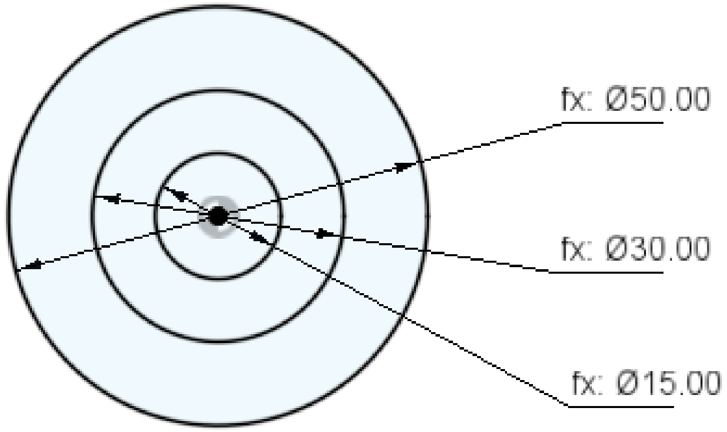

Figure 4.77

When it is necessary to change certain dimensions, this can be easily done in the table with design parameters. If we change the values of the parameters, the corresponding dimensions where these parameters are used will be updated automatically.

Figure 4.78 shows the table with parameters, where we have changed the values of two of the parameters.

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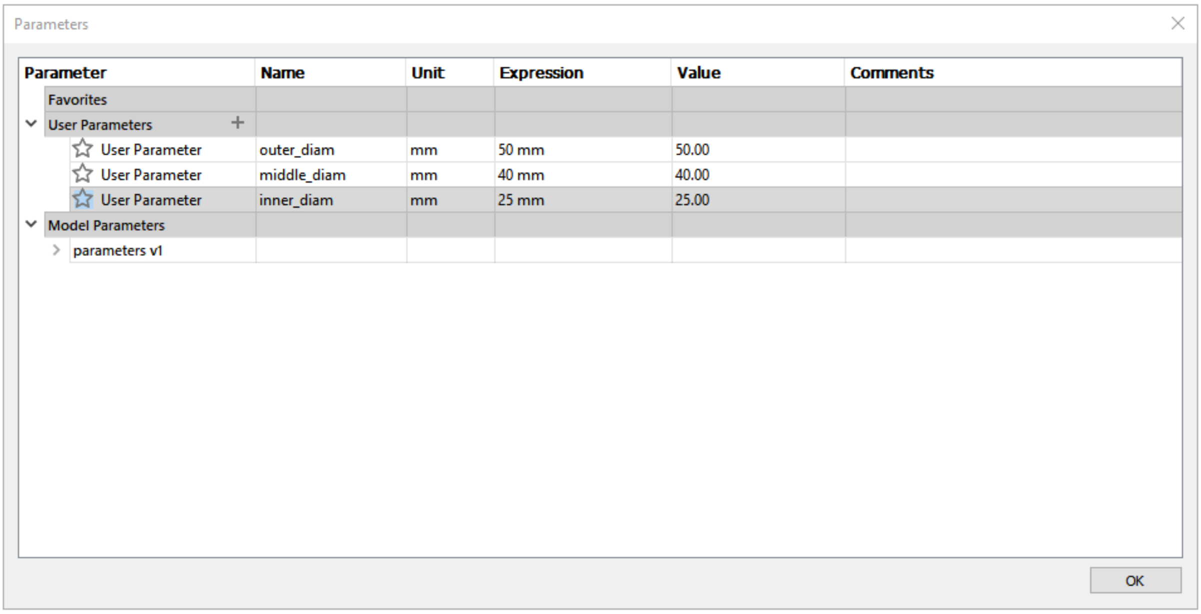


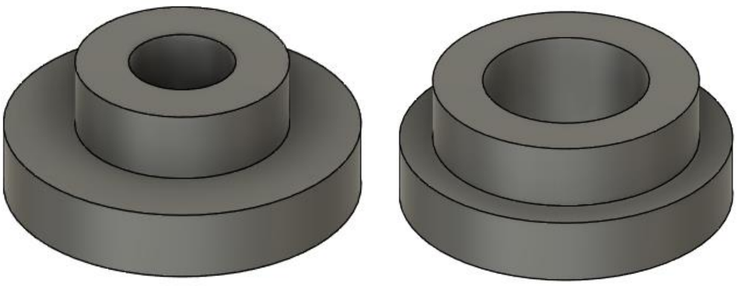
Parameters

Parameter	Name	Unit	Expression	Value	Comments
Favorites					
User Parameters					
User Parameter	outer_diam	mm	50 mm	50.00	
User Parameter	middle_diam	mm	40 mm	40.00	
User Parameter	inner_diam	mm	25 mm	25.00	
Model Parameters					
parameters v1					

OK

Figure 4.78

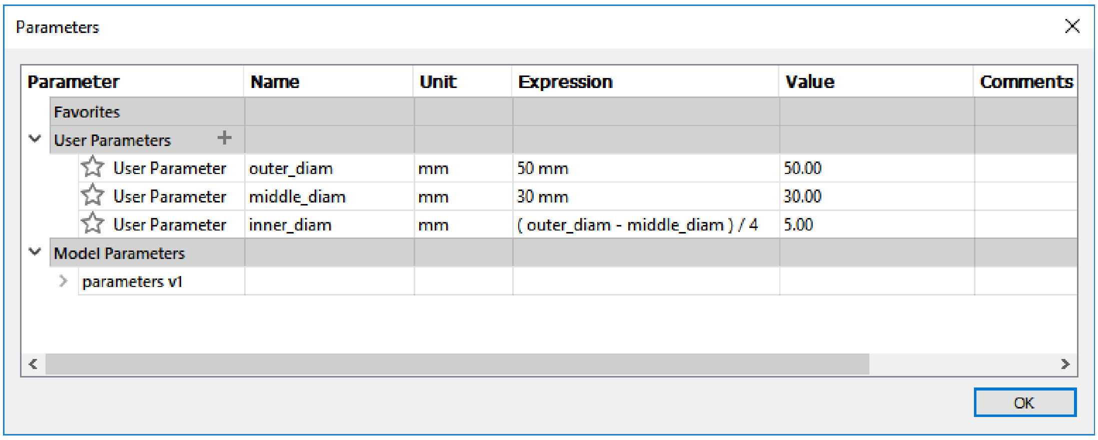
We have extruded a body from the three concentric circles. In one case (Figure 4.79, a) its shape is controlled by the values of the parameters given in Figure 4.75. In the other case (Figure 4.79, b) the shape is defined by the values of the parameters given in Figure 4.78.



(a) (b)

Figure 4.79

The parameters' values can be defined as expressions. In Figure 4.80 one of the parameters is defined as an expression in which the other two parameters are used.



Parameters

Parameter	Name	Unit	Expression	Value	Comments
Favorites					
User Parameters					
User Parameter	outer_diam	mm	50 mm	50.00	
User Parameter	middle_diam	mm	30 mm	30.00	
User Parameter	inner_diam	mm	(outer_diam - middle_diam) / 4	5.00	
Model Parameters					
parameters v1					

OK

Figure 4.80

In this case, we obtain a body shape as presented in Figure 4.81.

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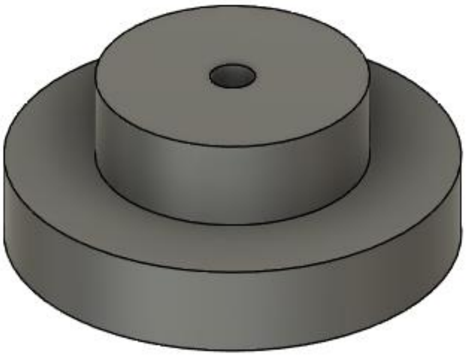

Figure 4.81

Inspecting 3D objects' features

In Fusion 360 there are many opportunities to explore and inspect various features of the created objects. To demonstrate these possibilities, let us use the three-dimensional object presented in Figure 4.82

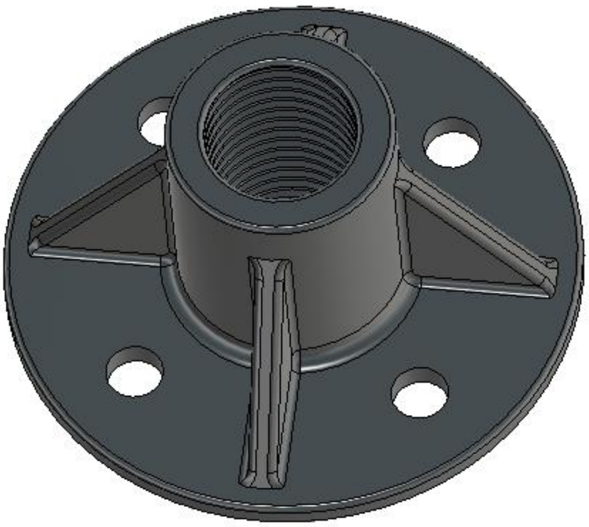

Figure 4.82

Determining the physical properties of the object

The determination of the object's physical properties is performed automatically by the environment based on the selected material from which the body is intended to be produced. To define the physical properties, the corresponding object should be right-clicked in the browser, and the Properties command should be activated (Figure 4.83).

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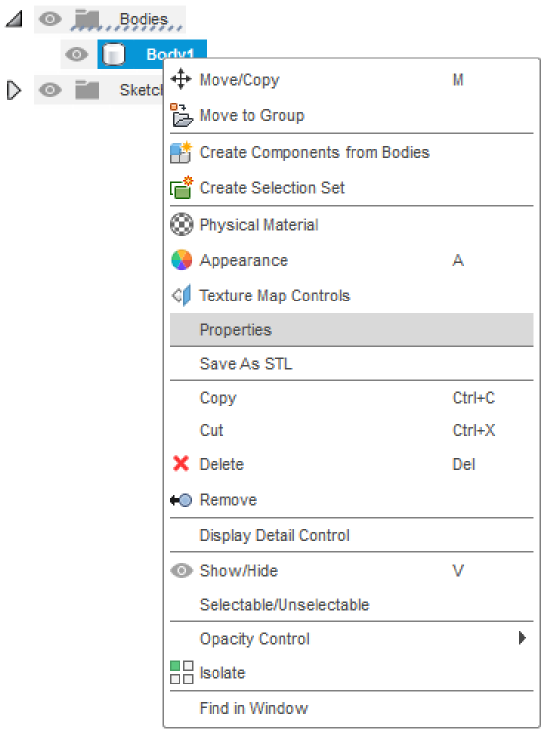
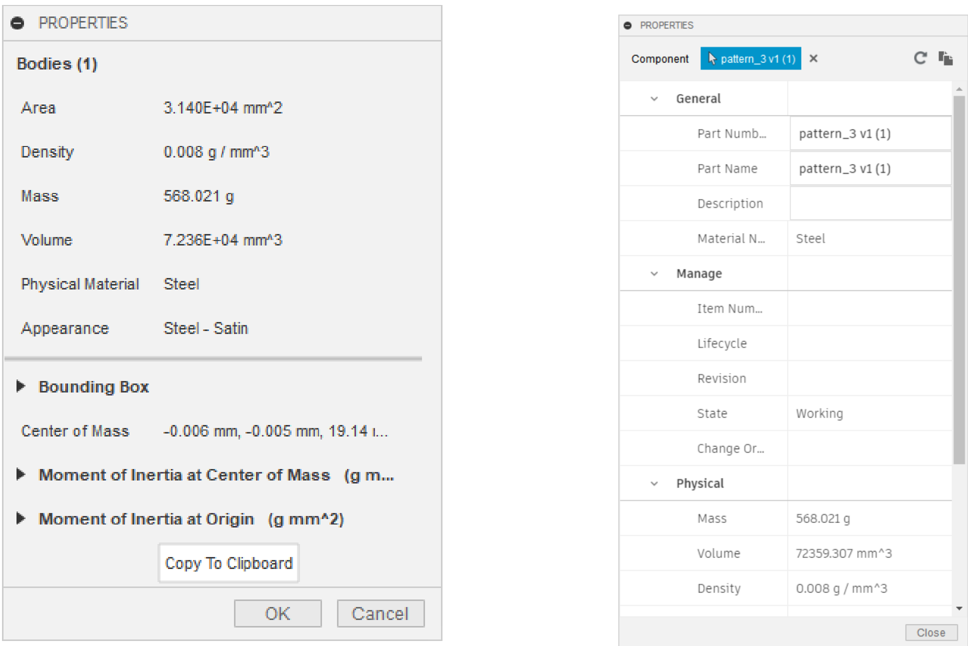


Figure 4.83

The Properties command displays information about the object's surface, volume, density and mass when it is produced from the selected material. Additionally, information about the object maximum overall dimensions along each of the axes is displayed, as well as information about the moments of inertia.

Information about the properties of objects can be displayed both for bodies (Figure 4.84, a) and for components (Figure 4.84, b).



(a) (b)

Figure 4.84

The Inspect menu and its more important commands

The Inspect menu (Figure 4.85) provides a wide range of opportunities to inspect various features related to the object's appearance and shape.

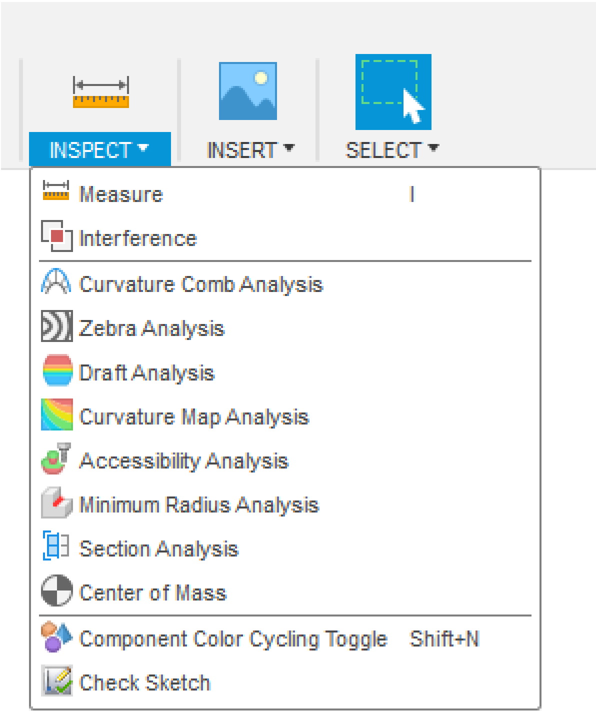


Figure 4.85

- Determining the object's dimensions

The Measure command is used to measure and determine different dimensions associated with 3D objects. It is most often used to determine the distances between vertices, points, planes and other geometric figures of which objects are composed. We obtain the necessary information by using the mouse to select the geometry we are interested in (Figure 4.86).

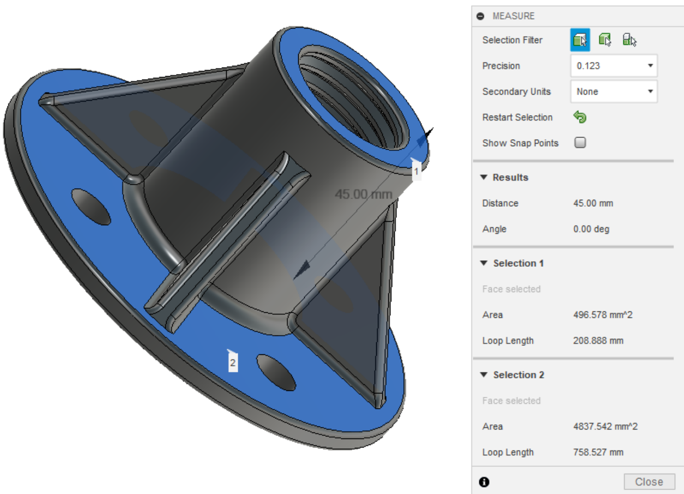


Figure 4.86

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- Draft Analysis command

With the help of this command (Figure 4.87) we analyze whether the angles of inclination of the individual parts of the object correspond to predefined limits. The command can be used to check the so-called casting inclinations of details that will be made by casting.

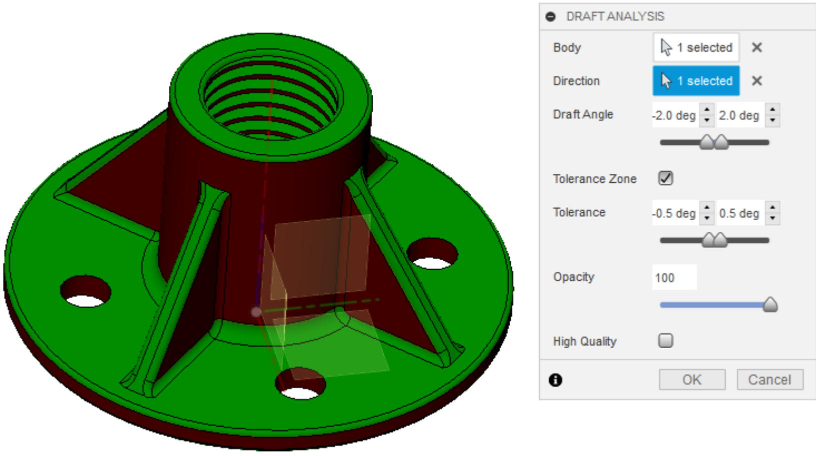

Figure 4.87

- Accessibility Analysis command

This analysis colours the object based on whether its respective parts are accessible from a certain direction (Figure 4.88).

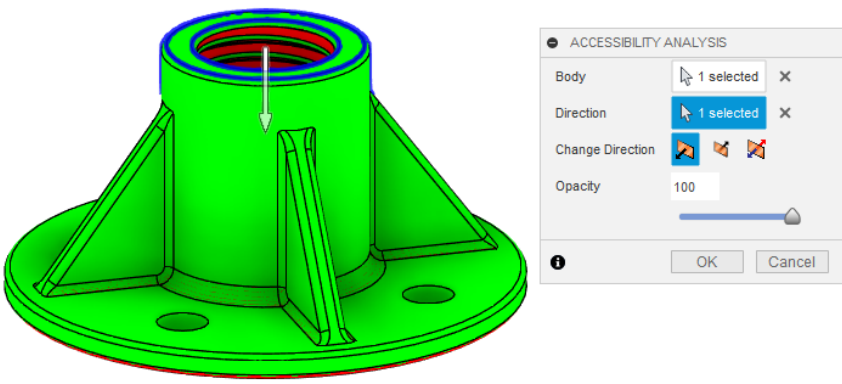

Figure 4.88

- Minimum Radius Analysis command

This analysis (Figure 4.89) allows us to determine in advance whether the part can be produced using a tool with a certain radius. It indicates the places where with the selected tool it will be impossible to achieve the required shape and accuracy in the dimensions of the part.

The analysis is useful for a preliminary assessment of a part's shape before it is sent for production and mechanical processing.

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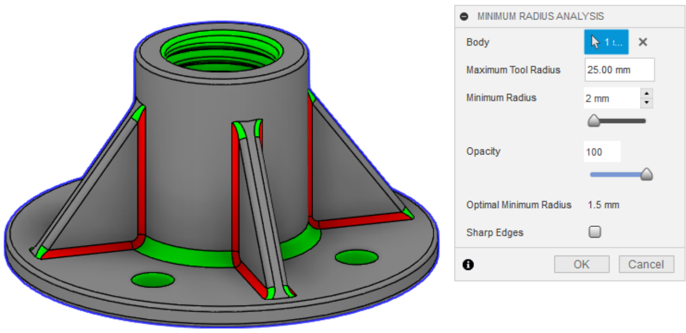

Figure 4.89

- Command for creation of section views (Section Analysis)

This command (Figure 4.90) is extremely useful when it is necessary to inspect the inside of an object. It cuts the object along an axis we have selected, and in the place we have specified. In the resulting section view, inaccuracies can be detected, both in single parts and in assembled parts.

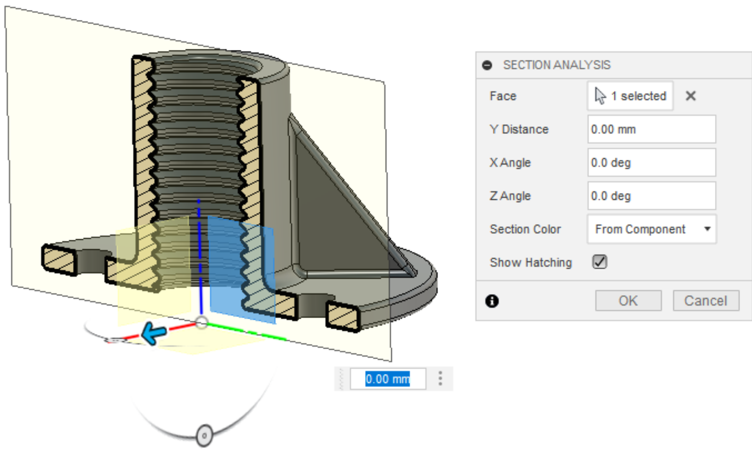

Figure 4.90

- Zebra Analysis command

With the help of this analysis (Figure 4.91) we can detect unwanted distortions in the shape of the object. In the case of smoothly changing shapes, we can also monitor for sections where unwanted sharp changes occur in the shape.

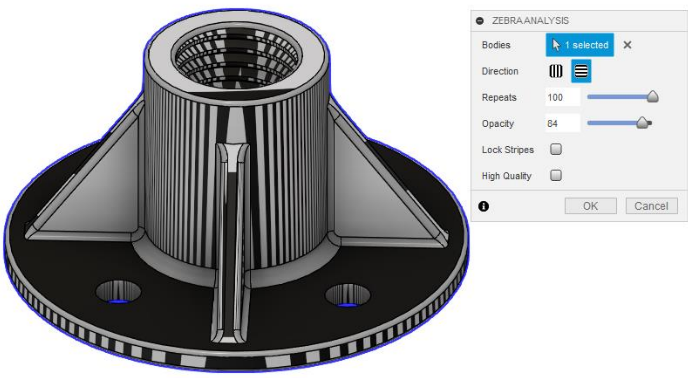

Figure 4.91

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- Determining the center of gravity of the object

The Center of Mass command determines where the center of gravity of the object is located. The center of gravity is marked accordingly in the three-dimensional space (Figure 4.92).

Figure 4.92

- Component Colour Cycling Toggle Command

This command colours differently the individual components in the design. Often, many components are included in the design and uniform components cannot be easily differentiated (Figure 4.93, a). With the help of colouring, each component can acquire an individual colour and in the process of work its detection in the design will be facilitated (Figure 4.93, b).

(a) (b)

Figure 4.93

Another great advantage of component colouring is that the corresponding commands in the Timeline related to component creation also acquire the colour of the component for which the command has been used (Figure 4.94).

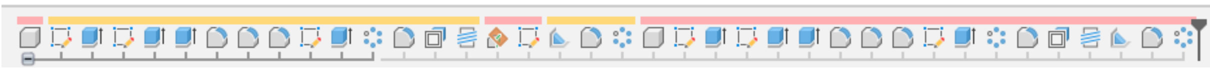

Figure 4.94

Supplementary self-study video materials for working with the environment interface:

https://www.youtube.com/watch?v=KxlDsMQG7y4

https://www.youtube.com/watch?v=s5rWJMyLpj8

https://www.youtube.com/watch?v=vRqeaqlgYko

https://www.youtube.com/watch?v=glq6t8M-uRQ

Creating assemblies

Like other 3D modelling environments, Fusion 360 provides the opportunity to create assemblies involving multiple interconnected objects. Unlike other environments, where separate file types are used to create assemblies, Fusion 360 allows assembled objects to be placed in the same file format as single bodies. For this purpose, the individual bodies are differentiated into so-called components, which in turn can be virtually assembled with each other, subject to certain constraints set at the stage of assembly.

Components in Fusion 360

In Fusion 360 a component is understood as a container composed of various 3D bodies, sketches, construction geometries, and other types of data. Each component has its own coordinate system, which is different from the basic coordinate system of the design. In practice, even the design itself is interpreted as a component in the Fusion 360 environment. A component may also contain other components. An analogy of components is the operating system's directories, which can contain different file types, as well as other directories. One of the main characteristics of components is that they can participate in assemblies, as joints can be created between the individual components and not between the three-dimensional bodies contained in the components.

A new component can be created with the New Component command (Figure 5.1). It is good practice in Fusion 360 to create each new three-dimensional object in the design as a separate component.

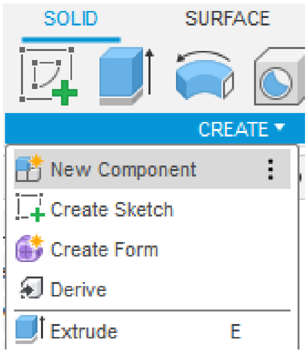

Figure 5.1

When creating a new component (Figure 5.2) it is necessary to specify its name and to indicate its parent component (in the example below the main component of the design is selected).

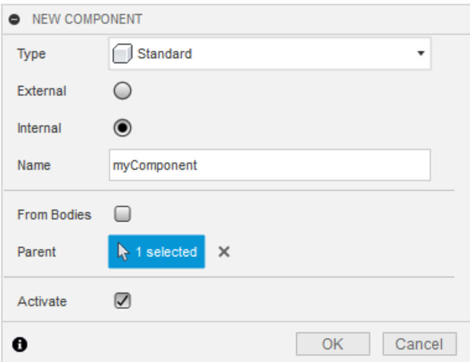

Figure 5.2

The created component appears in the browser (Figure 5.3).

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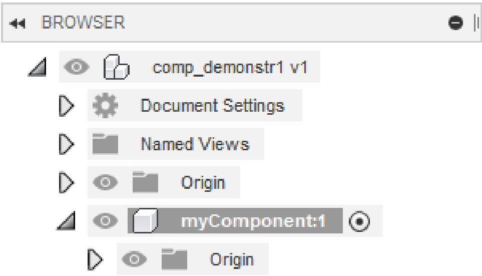

Figure 5.3

Sketches, 3D bodies, structural geometries and other types of data can be created in each component. The component's coordinate system is different from that of the main design (Figure 5.4).

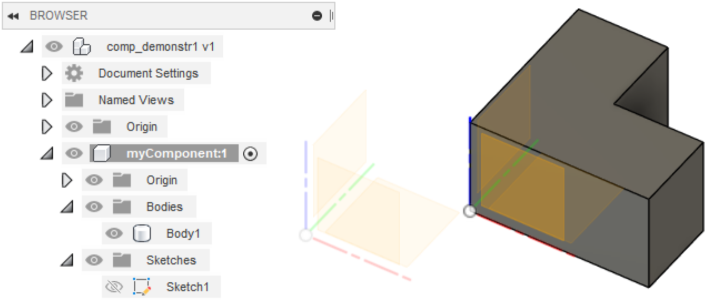

Figure 5.4

A design can have many components, each containing its own data and having its own coordinate systems (Figure 5.5).

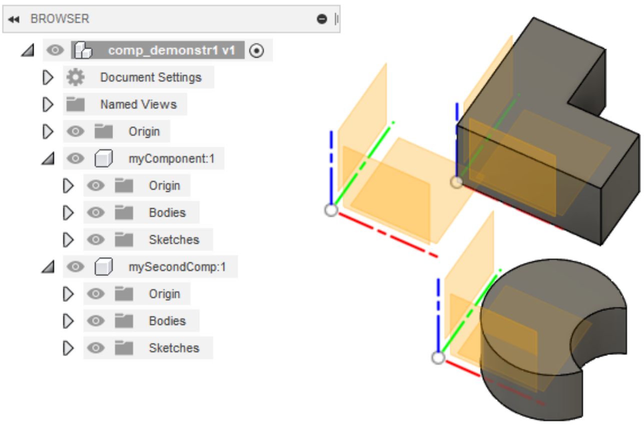

Figure 5.5

Components can be selected and at any given moment one of the components in the design can be active, while the other components go into a visualization mode showing only on their contours (Figure 5.6).

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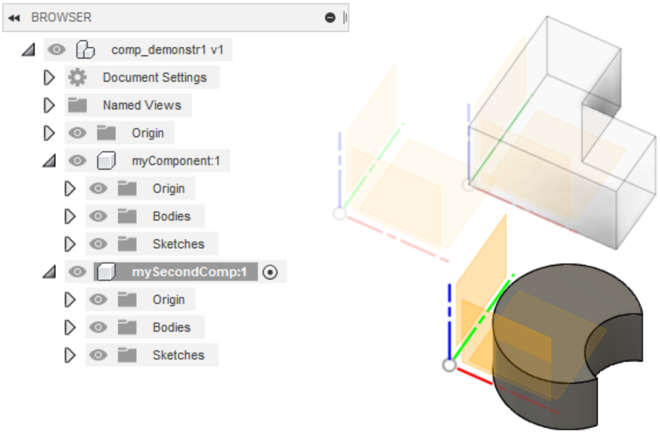

Figure 5.6

In Fusion 360, components may be:

- Single component – for example 

- Sub assembly – for example 

Sub-assemblies include other components that are typically linked with joints.

Using the Joint command

In the Assemble menu (Figure 5.7) we can activate commands to create assemblies. The most important command in this menu is the Joint command, which defines an assembly joint between two components and sets the parameters of the assembly.

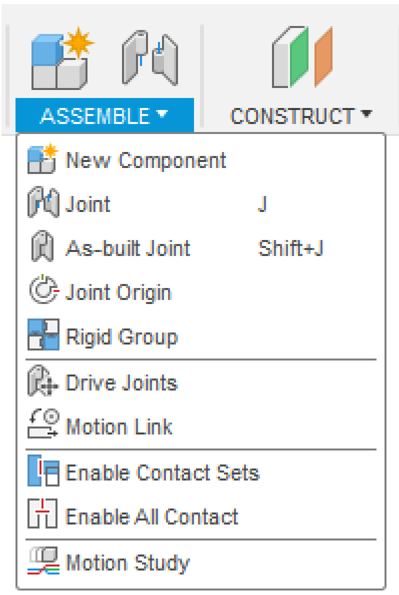

Figure 5.7

When the Joint command is run, the menu presented in Figure 5.8 opens.

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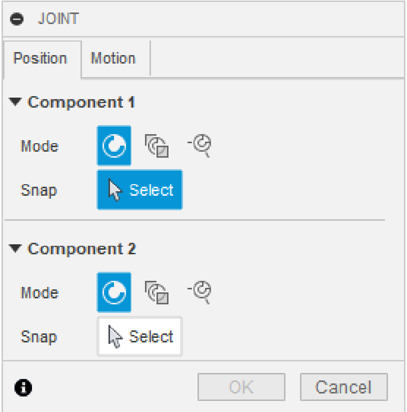

Figure 5.8

The assembly of two components can be done in three main ways, using different connection points between the two components (Table 5.1).

Table 5.1

Joint type (Mode)	Main purpose
Simple	A connection point is created on a face, edge, or point of the component.
Between two faces	A connection point is created between two faces.
Two Edge Intersection	A connection point is created at the point of intersection of the two edges of the component.

We set the joint type in the Motion field (Figure 5.9) of the Joint command, defining which translation and rotation movements around the coordinate axes are allowed.

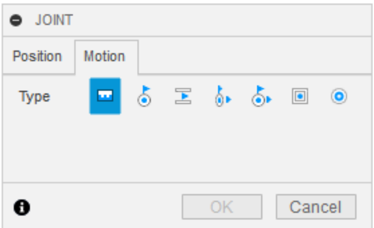

Figure 5.9

In Fusion 360 it is planned to use the following types of joints depending on the movements that are allowed between the components (Table 5.2).

Table 5.2

Type of movements	Action
Rigid	The two components are fixed to each other and no movement is allowed.
Revolute	Rotational movement along one of the axes at the point of connection is allowed.
Slider	Only movement along one of the coordinate axes is allowed.

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Cylindrical	Simultaneous rotational and translational motion along one of the coordinate axes passing through the point of connection is allowed
Pin-Slot	Rotational movement on one of the coordinate axes and translational motion along another axis are allowed.
Planar	Rotational movement along one of the coordinate axes and translational motion along the other two axes are allowed.
Ball	Rotational movements along the three axes are allowed.

- Rigid Joint

Below we present two objects, which should be assembled immovably (Figure 5.10).

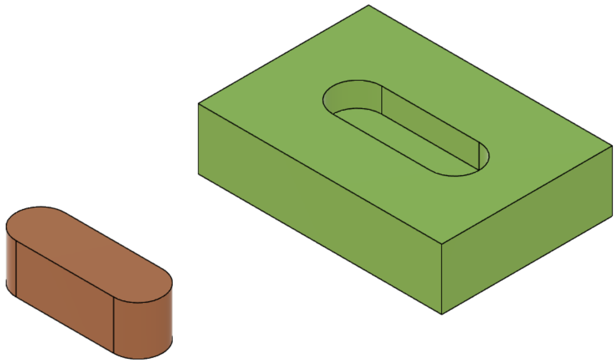

Figure 5.10

After running the Joint command, in the opened dialogue box we select the Snap option for the first component (Figure 5.11).

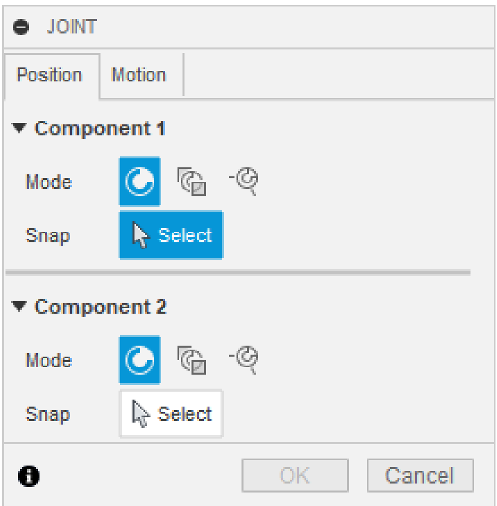

Figure 5.11

Then, for the first component we specify a specific point, which will be used for the assembly joint's creation (Figure 5.12).

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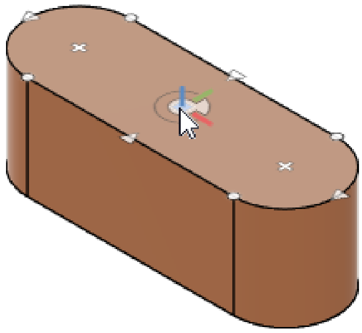

Figure 5.12

In the menu of the Joint command, we will activate the Snap option for the second component and we will indicate a specific point on the second object for the creation of the joint (Figure 5.13).

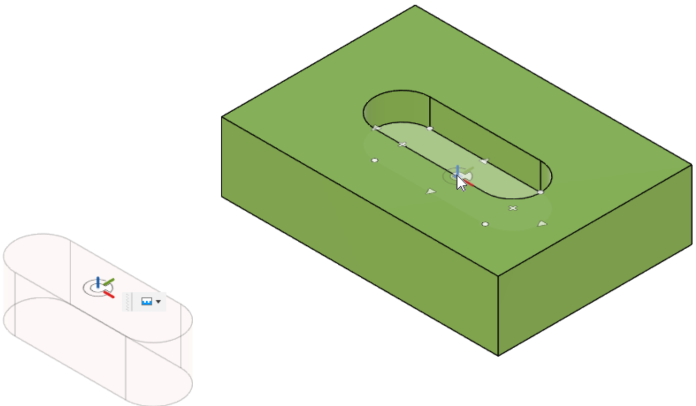

Figure 5.13

As a result, the first object will be assembled with the second at the selected joining points (Figure 5.14).

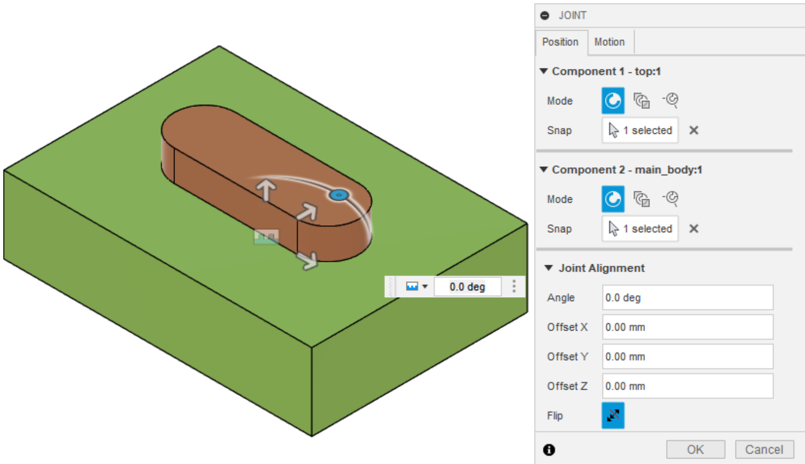

Figure 5.14

To specify that the joint is fixed, we will open the Motion option in the Joint command menu and we will specify the type as *Rigid* (Figure 5.15).

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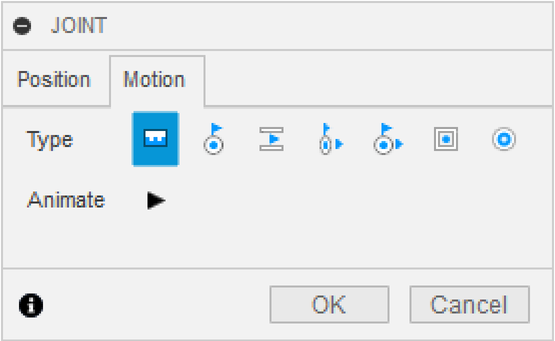

Figure 5.15

A section indicating the presence of an assembly joint in the design (and also specifying the joint type) will appear in the design browser (Figure 5.16).

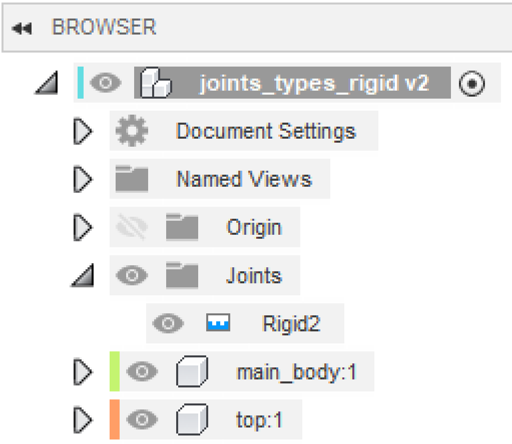

Figure 5.16

- Revolute Joint

Let us look at the objects presented in Figure 5.17.

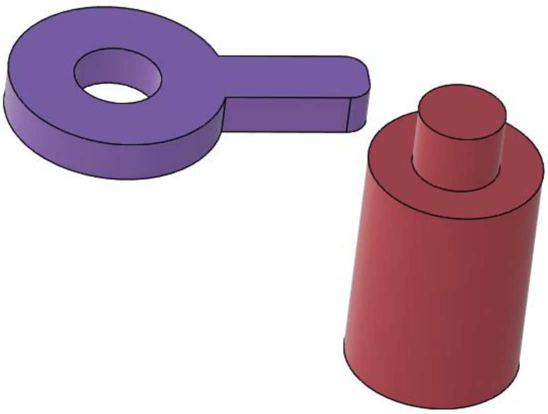

Figure 5.17

The two objects will be assembled by selecting a joining point on one object. The point lies on one of the object's sides, on the central axis of the hole (Figure 5.18).

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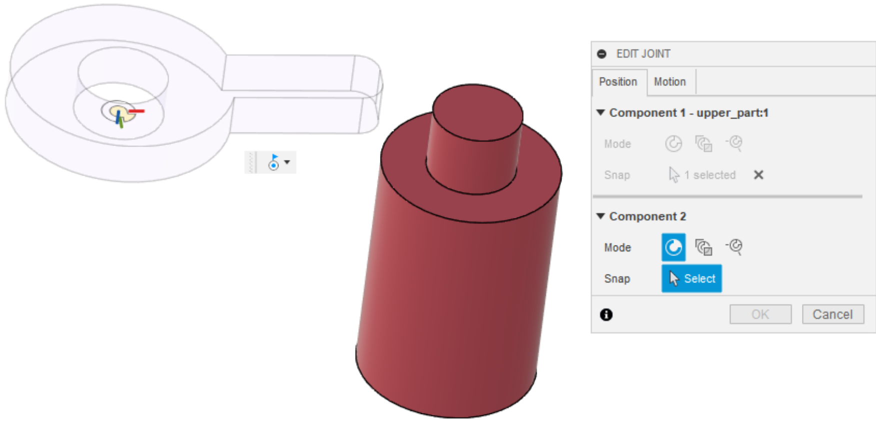

Figure 5.18

For the other object, a point lying on the object's central axis will be chosen as the assembly point, as shown in Figure 5.19. If the selection is not possible, we will press the Ctrl key on the keyboard to select the desired point.

Figure 5.19

The two components are assembled by selecting Revolute as the motion type (Figure 5.20).

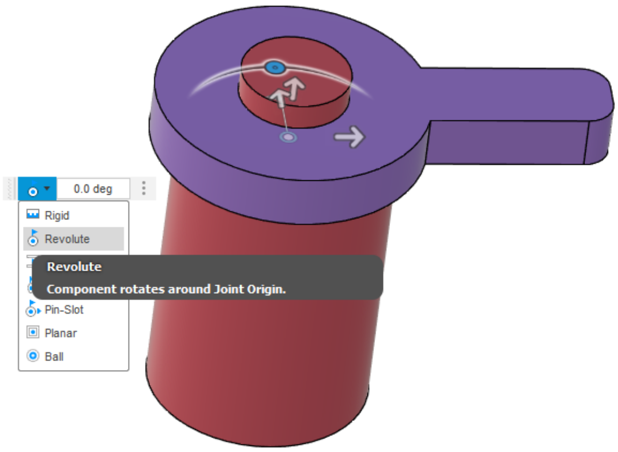

Figure 5.20

To demonstrate the rotation of the upper component around the cylindrical neck of the lower one it is necessary to lock the lower component in place. This is done by right-clicking on the lower component in the browser. In the menu that appears, the Ground command should be selected (Figure 5.21).

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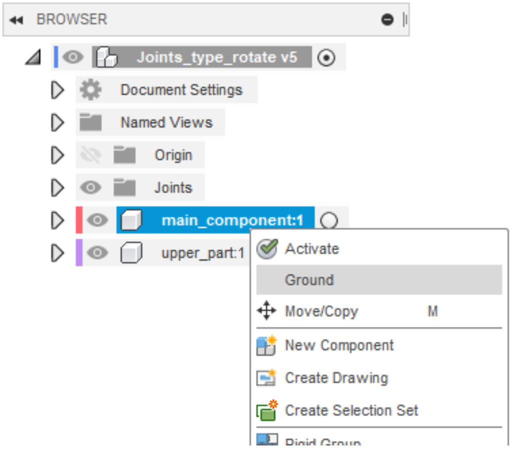

Figure 5.21

Now the upper body can rotate around the lower at the appropriate angle (Figure 5.22).

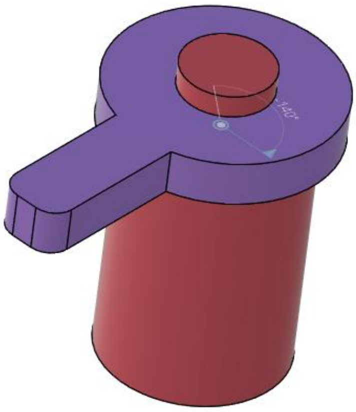

Figure 5.22

- Slider Joint

To demonstrate the slider joint, the following objects are presented in Figure 5.23 below.

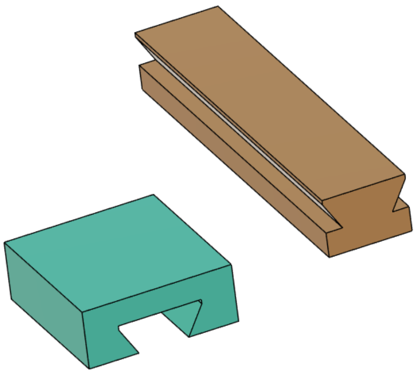

Figure 5.23

For the first component we select the joining point to lie on one side of the object, exactly in the middle of the slot created in it (Figure 5.24).

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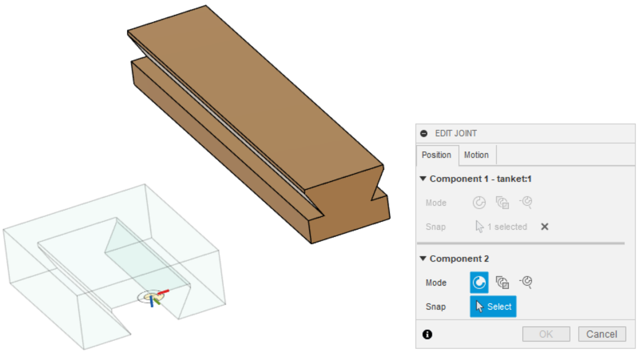

Figure 5.24

The joining point of the second object lies in the middle of the profile, which should pass through the slot in the first object (Figure 5.25).

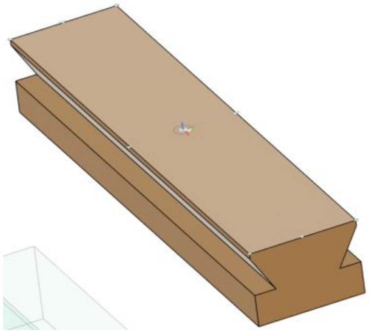

Figure 5.25

During the assembly process, the components may be arranged in an unexpected way based on Fusion 360's internal logic for positioning objects during assembly (Figure 5.26).

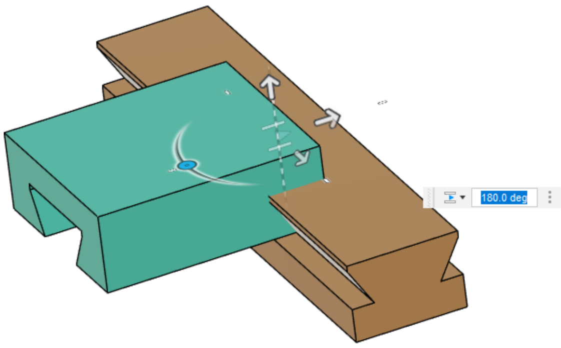

Figure 5.26

In that case, it will be necessary to manually set the angle of rotation of one object against the other or the offset along the individual axes. This is done using the Joint Alignment parameter from the command menu (Figure 5.27).

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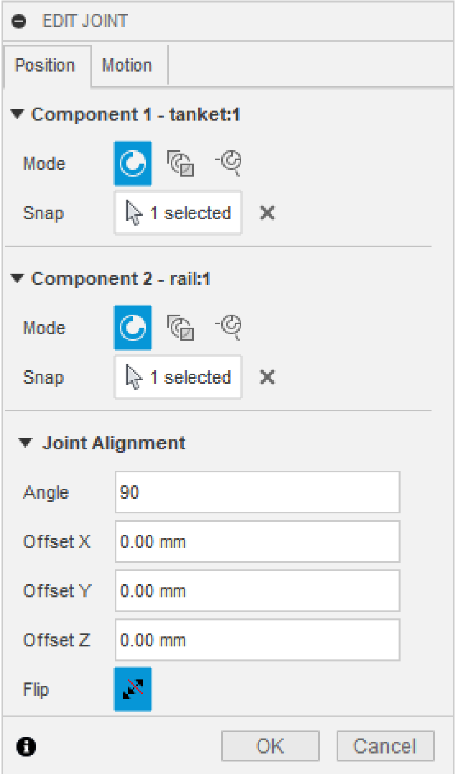

Figure 5.27

Slider (Figure 5.28) should be chosen as the type of motion in the connection of the two components.

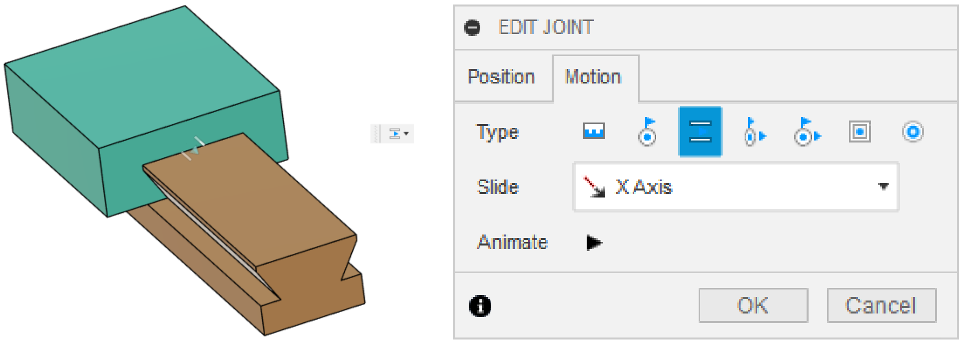

Figure 5.28

Before the implementation of the sliding movement of the one of the components vis-à-vis the other, one of the components should be locked in place using the Ground command (Figure 5.29).

The sliding movement is implemented along the selected axis, and translational motion along the other axes will not be possible. All rotational movements along the axles of this type of joint are prohibited.

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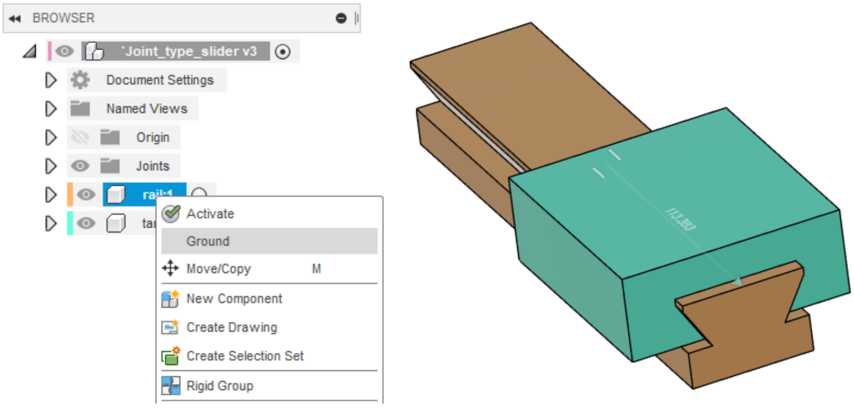

Figure 5.29

- Cylindrical Joint

Cylindrical joints are used to create bolted joints. A bolt and an object with a threaded hole in it are shown in Figure 5.30 below.

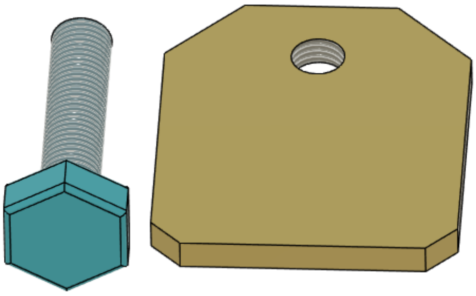

Figure 5.30

The points shown in Figure 5.31 are selected as joining points.

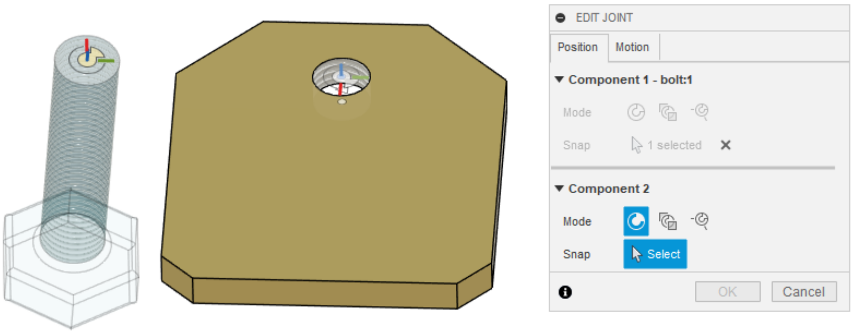

Figure 5.31

We can simulate the operation of a bolted joint when *Cylindrical* is selected as the type of motion. A translational motion and at the same time a rotation can be performed along one of the axes (Figure 5.32).

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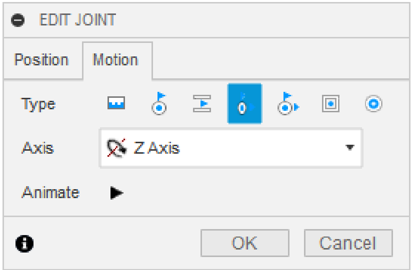

Figure 5.32

In this joint configuration, if one of the components has been locked in place (in this case the nut), then the bolt can move and occupy the desired position in the joint (Figure 5.33).

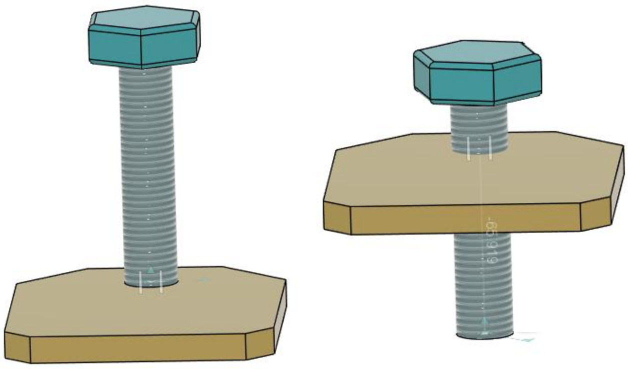

Figure 5.33

- Type Pin Slot Joint

The Pin Slot type of joint allows rotation on one axis and translational motion along another axis. To demonstrate the principle of operation of this type of joining, we present three components that are assembled as shown in Figure 5.34. The assembly was performed using two revolute joints and one Pin Slot joint. The Pin Slot joint is defined between the slot carved into the plate and the pin of the other component entering in it.

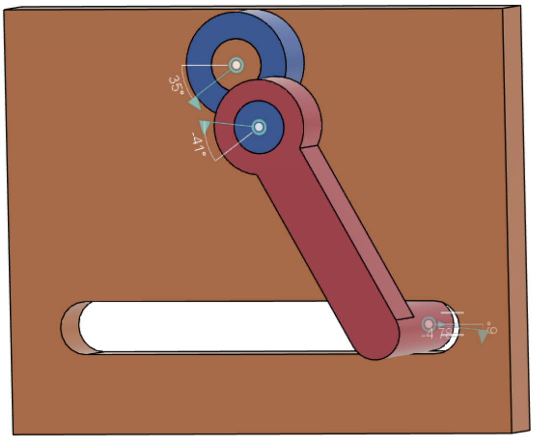

Figure 5.34

When defining the Pin Slot joint, we need to select the *Pin Slot* motion type in the dialogue box, the axis of rotation (in this case it is perpendicular to the slot in the plate), as well as the axis of movement – along the slot (Figure 5.35).

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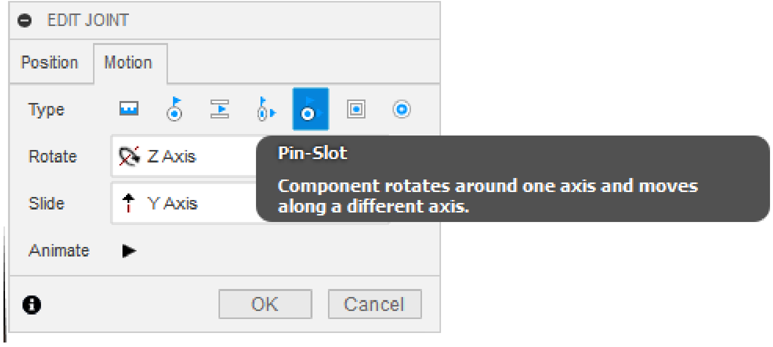

Figure 5.35

With these settings it is possible to simulate reciprocating motions of the other component's pin inside the slot of the plate (Figure 5.36).

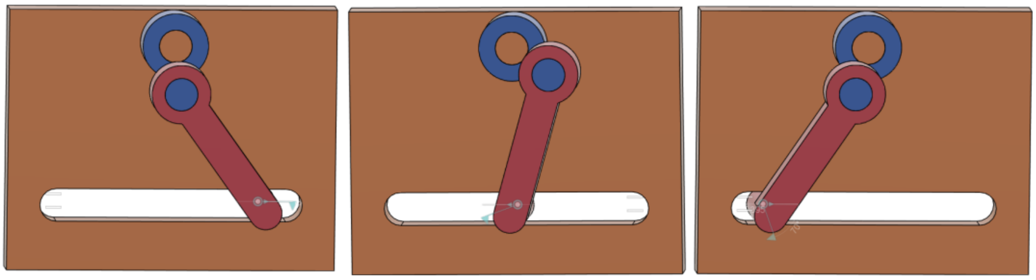

Figure 5.36

- Planar Joint

Two translational axes and one rotational axis are allowed for this type of joint. To demonstrate the connection, let us have two components (Figure 5.37), which are connected by means of the respective assembly points.

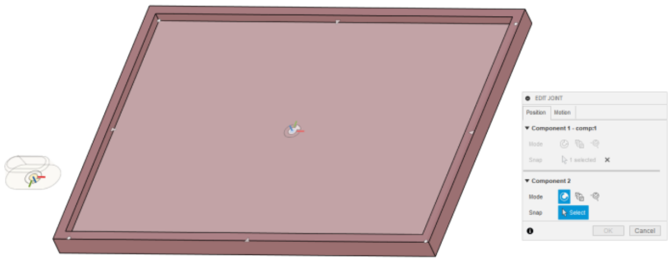

Figure 5.37

We select *Planar Joint* as type of motion (Figure 5.38).

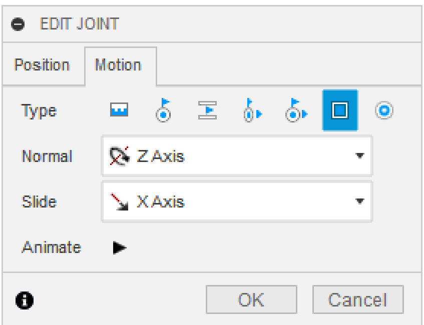

Figure 5.38

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With this type of joint, the motion of one component can be simulated on a planar surface of another component, while the component always remains in contact with the plane (Figure 5.39).

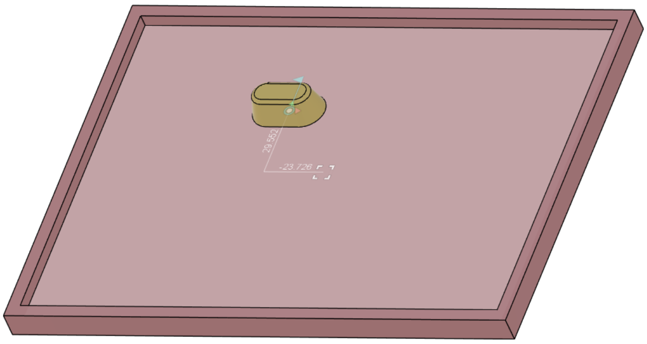

Figure 5.39

- Ball Joint

In the case of a Ball Joint, rotational movements along the three axes are allowed, while translational motions are disallowed. The components in Figure 5.40 demonstrate a ball joint.

Figure 5.40

The components are assembled by selecting the spherical surfaces with the mouse. In the centers of these surfaces we locate the joining points by means of which the two bodies will be assembled with each other (Figure 5.41).

Figure 5.41

We select *Ball* as the Motion Type (Figure 5.42).

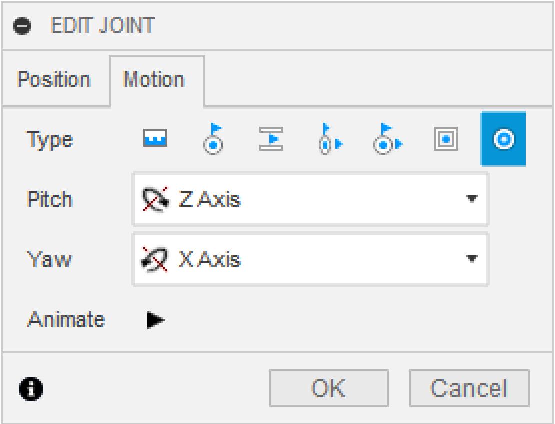

Figure 5.42

The assembled Ball Joint provides a high degree of flexibility in the location of one object relative to the other (Figure 5.43).

Figure 5.43

Using the As-built Joint command

The As-built Joint command (Figure 5.44) is used for the assembly of objects that are already positioned in the desired way against each other and it is not necessary to explicitly define assembly points.

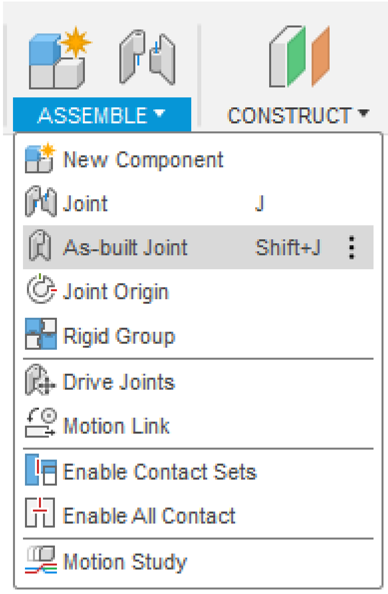

Figure 5.44

To demonstrate the work of the command, below we present a model of a kitchen table. It consists of five components that are arranged in such a way that the design has a finished look (Figure 5.45).

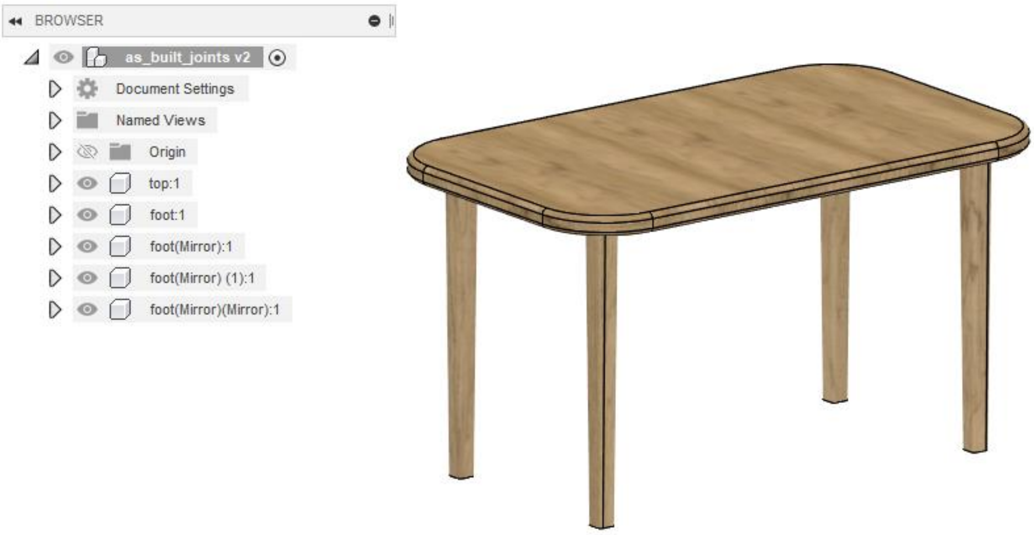

Figure 5.45

To preserve the arrangement of the individual components and to eliminate the possibility of accidental displacement of the components, it is necessary to define joints between them. The easiest way to do this is to use the As-built Joint command (Figure 5.46). With its help, we successively select two components and a connection is automatically created between them (Figure 5.46).

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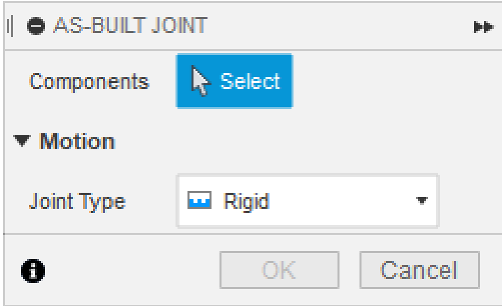

Figure 5.46

During the first assembly, we select one leg of the table and the table's upper part (Figure 5.47).

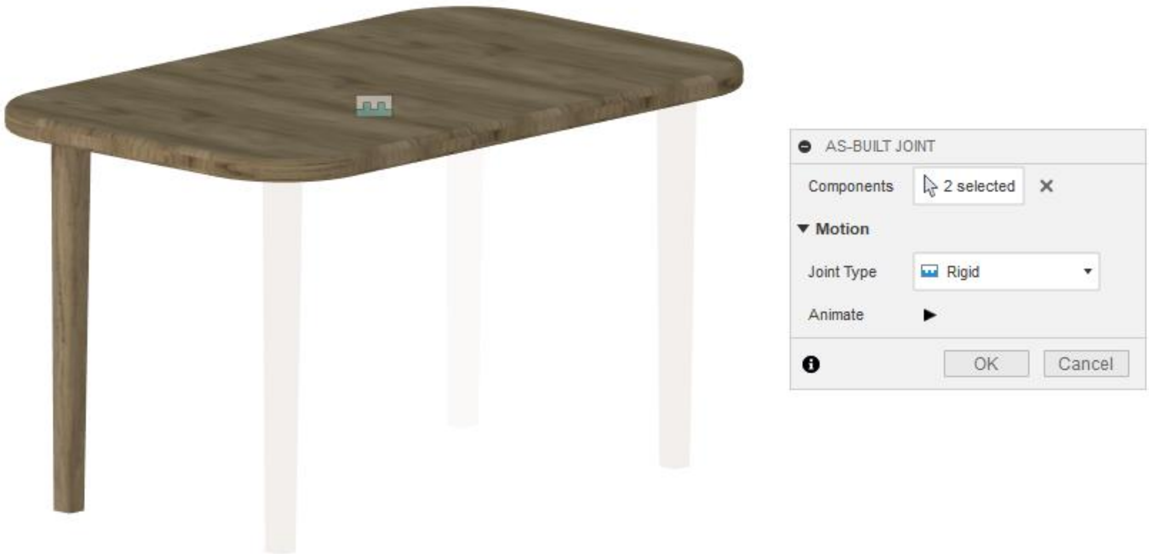

Figure 5.47

The other three legs are sequentially assembled with the upper part of the table (Figure 5.48).

Figure 5.48

The type of assembly joint can be specified during the assembly process - in this case the *Rigid* option is selected. The assemblies obtained in this way become visible in the browser (Figure 5.49).

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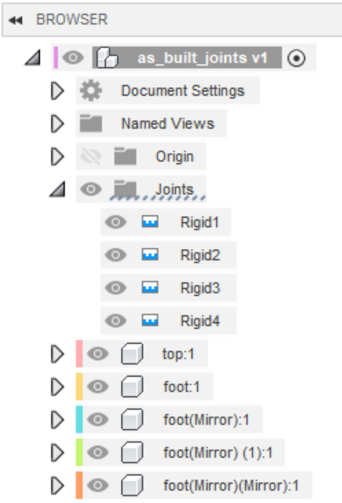

Figure 5.49

Using the Rigid Group command

With the Rigid Group command (Figure 5.50) it is possible to define a set of components as connected objects and thus to preserve their position in the group relative to each other.

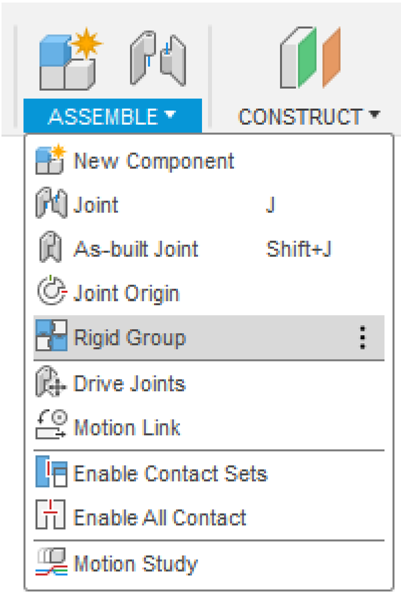

Figure 5.50

To demonstrate the operation of the command, in Figure 5.51 below we present a design containing a set of components that are arranged appropriately but are not assembled.

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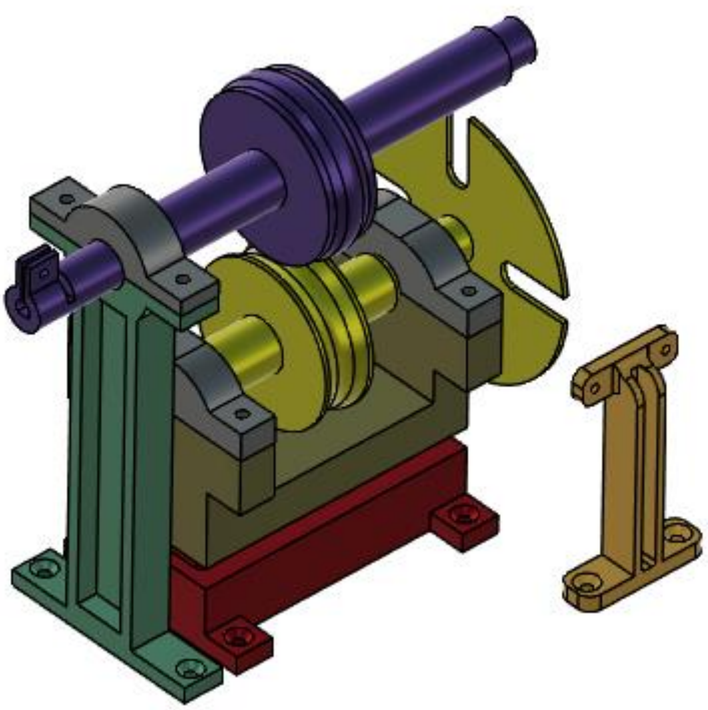


Figure 5.51

Instead of assembling the components together to maintain their position relative to each other, this can be done by joining the components together and at the same time using the Rigid Group command. For this purpose, it is necessary to select with the mouse the components that will participate in the rigid group (Figure 5.52).

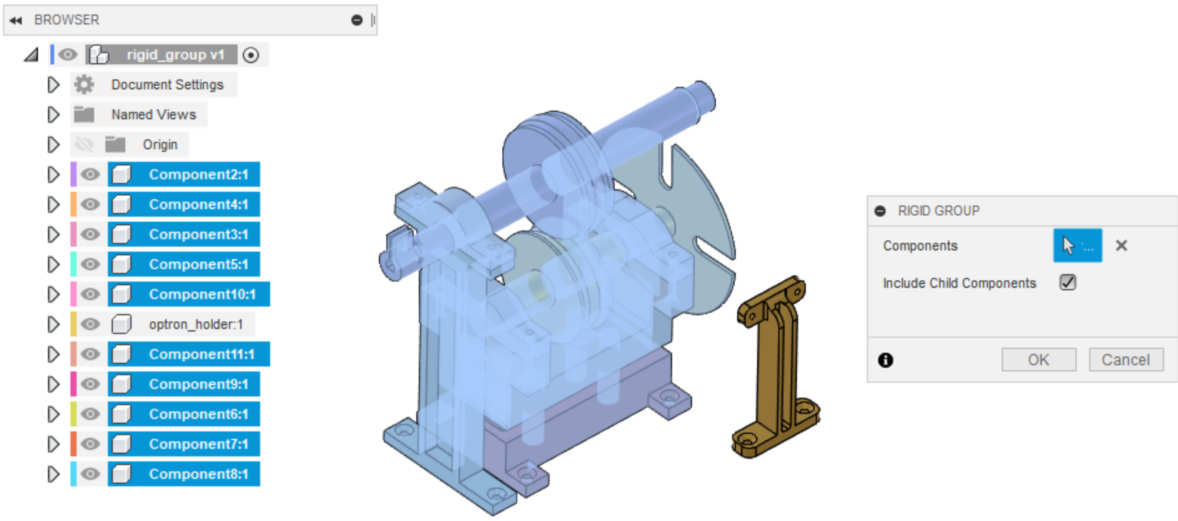
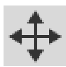


Figure 5.52

Once the command is executed, let us use the Move/Copy command  to relocate one of the objects included in the rigid group (Figure 5.53).

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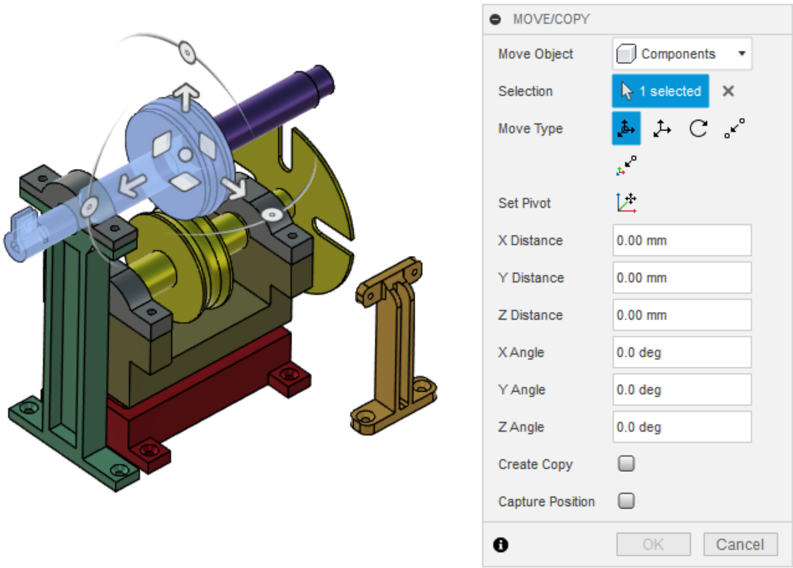

Figure 5.53

Along with the selected component, all related components in the Rigid Group will be moved (Figure 5.54).

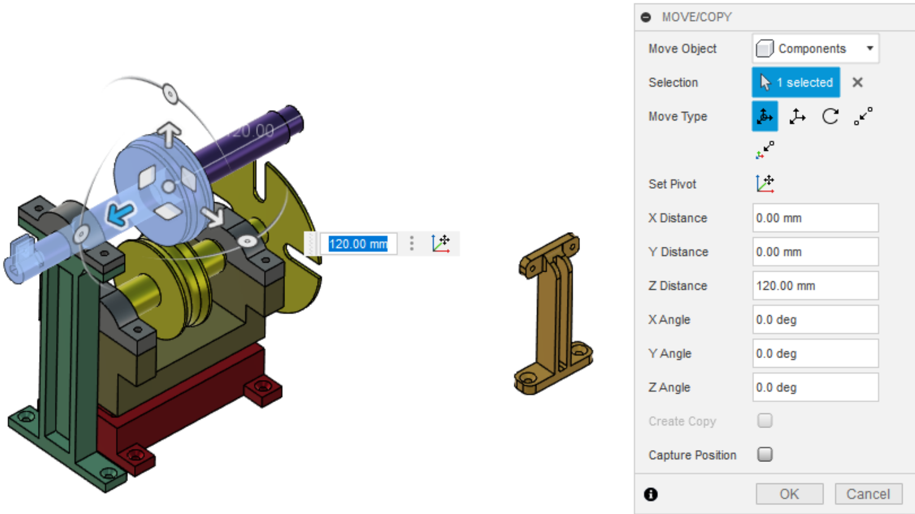

Figure 5.54

Defining connections between motions and the end positions of motions

To simulate the operation of an assembly joint whose parts can change their position vis-à-vis each other, we can define restrictions in these motions. If no movement restrictions are introduced, there is a high probability that the components will be unnaturally arranged and will occupy the same volume in the three-dimensional model, i.e. will overlap (Figure 5.55).

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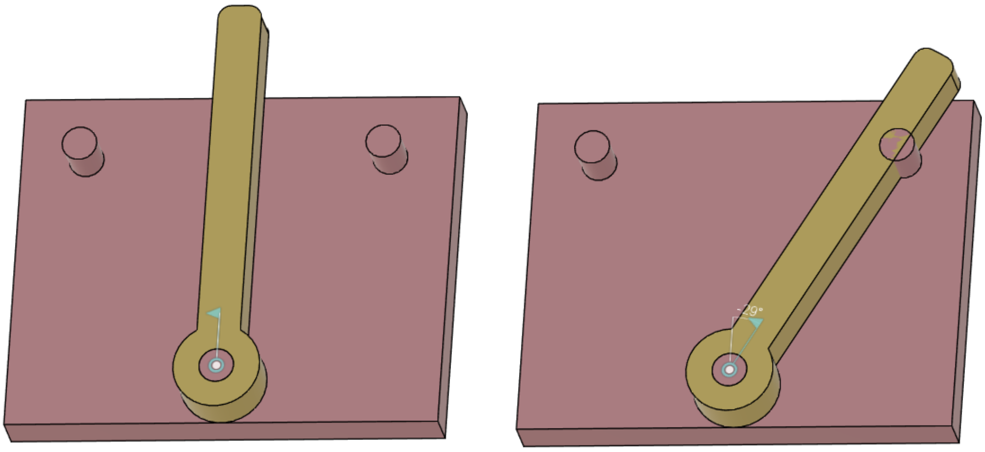

Figure 5.55

To eliminate the possibility of such an overlap, Fusion 360 has introduced the definition of contact surfaces. The Enable Contact Sets command (Figure 5.56) allows the creation of contact surfaces between two objects.

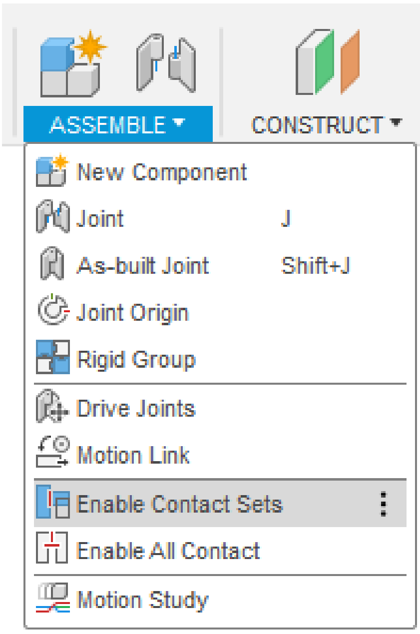

Figure 5.56

Once enabled, contact surfaces between two objects can be created with the help of the New Contact Set command (Figure 5.57).

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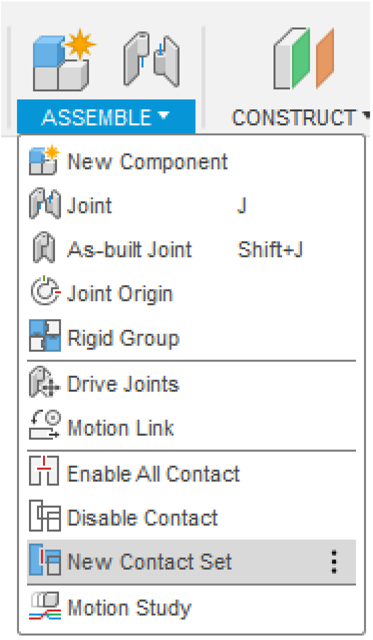

Figure 5.57

After the New Contact Set command is activated, it is necessary to select with the mouse the objects between which contact surfaces will be defined (Figure 5.58).

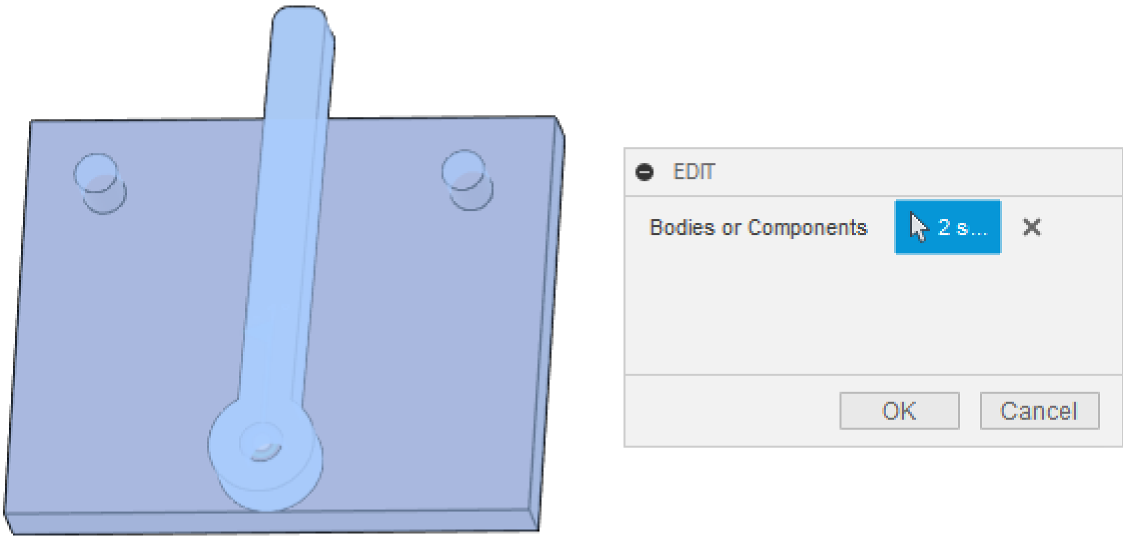

Figure 5.58

When there are defined contact surfaces between two objects, the environment will not allow any overlap. Any movement of one object relative to the other will be performed only until the two objects touch their surfaces. In this way we can achieve a realistic simulation of the movements in one assembly joint (Figure 5.59).

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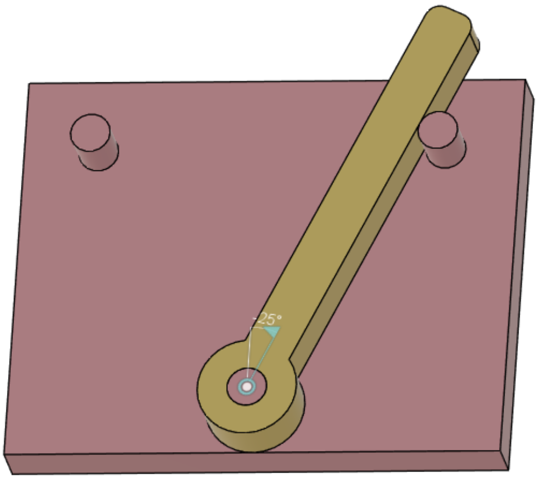

Figure 5.59

Fusion 360 allows us to create contact surfaces for all components in the design. This is done with the Enable All Contact command (Figure 5.60).

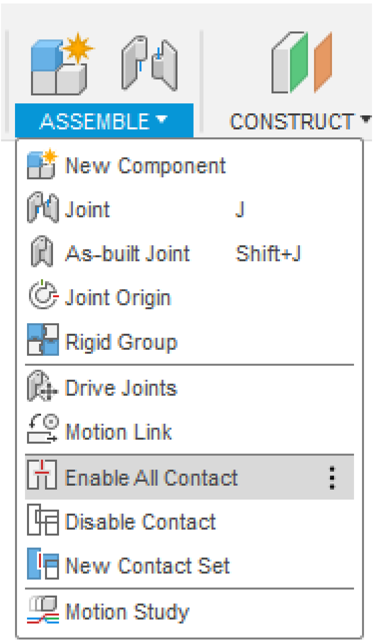

Figure 5.60

In this case, however, if the design includes multiple components, Fusion 360 will need to perform a large number of calculations and checks, which can affect the speed of the program.

Defining a Motion Link between two components

When simulating actuators, the Fusion 360 environment allows us to perform realistic simulation of the motion of interconnected parts by specifying the gear ratio between them. This is achieved by using the Motion Link command (Figure 5.61).

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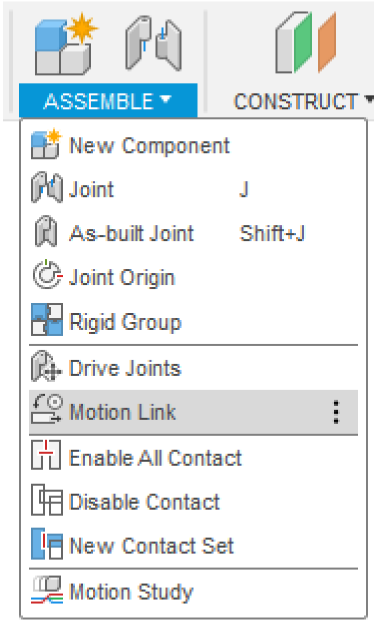

Figure 5.61

Below we present a gear realized by two gearwheels with different number of teeth (Figure 5.62).

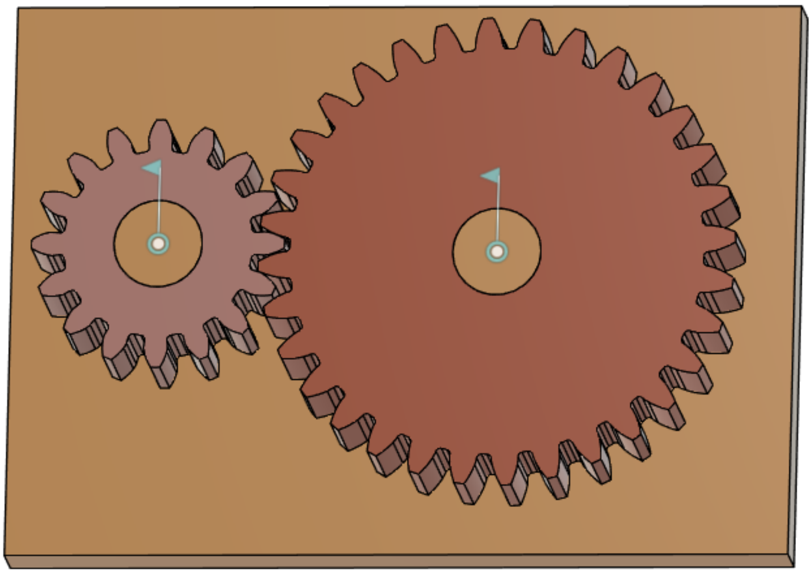

Figure 5.62

By using the Motion Link command (Figure 5.63), we can set the ratio between the angle of rotation of one gear and the angle of rotation of the other gear. In this way, the simulation of the gear operation will not let the teeth of one wheel overlap with those of the other in the process of rotation.

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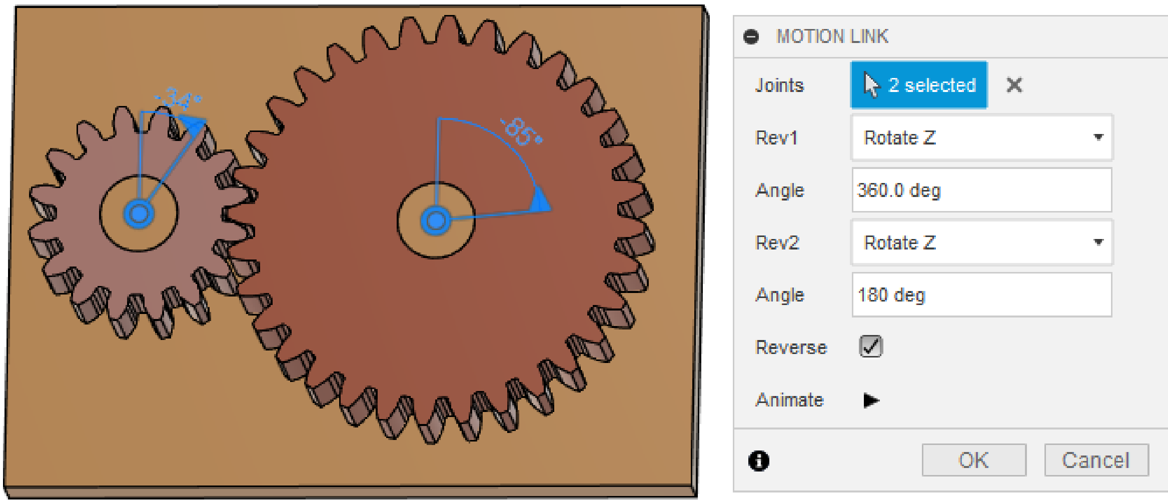

Figure 5.63

Simulating the sequence of motions

With the help of the Motion Study command (Figure 5.64) the sequence of performing complex movements can be simulated by setting the points (angles of rotation) of the individual components.

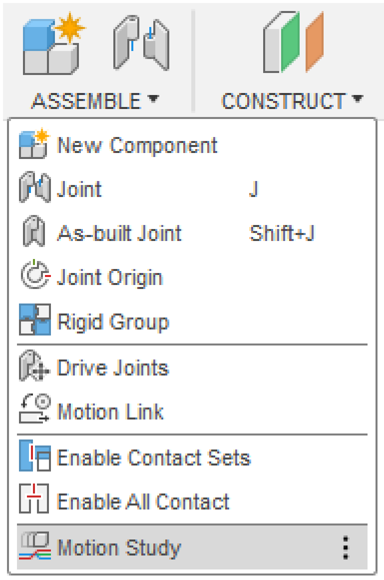

Figure 5.64

In motion simulation we can specify the direction of implementation of the defined scenario, the speed of execution of the motions and the duration of the simulated motions (Figure 5.65).

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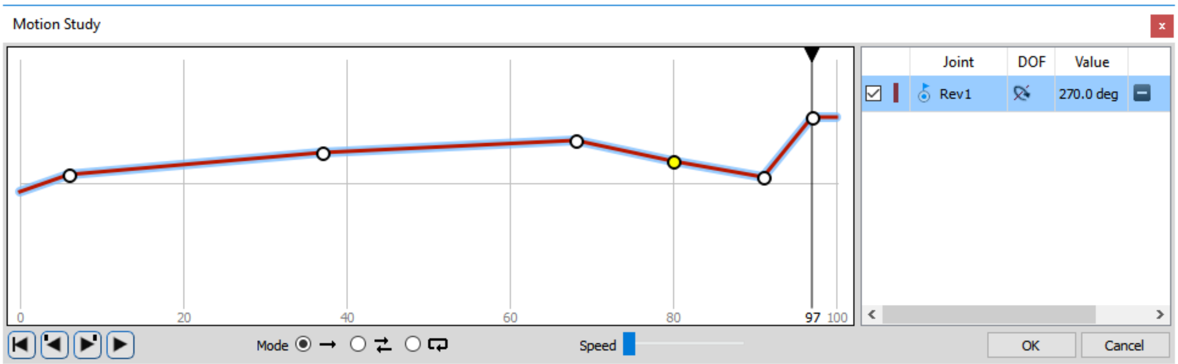


Figure 5.65

Supplementary self-study video materials for working with the environment interface:

https://www.youtube.com/watch?v=Bw08O6XsfDI&t=10s

https://www.youtube.com/watch?v=vjft_uppasc

https://www.youtube.com/watch?v=8DWIZKRLC9Y

https://www.youtube.com/watch?v=t41QmQszcbE

https://www.youtube.com/watch?v=fr-9-ux9r24

https://www.youtube.com/watch?v=jSaLy4RMnfY

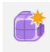
https://www.youtube.com/watch?v=1VE64VQLAgQ

https://www.youtube.com/watch?v=cfwOseuR6O4&t=1387s

Other features of Fusion 360

The Fusion 360 features discussed in previous chapters are just some of the possibilities that the environment provides. This chapter briefly addresses some of the most interesting additional features of the environment that contribute to its popularity and widespread.

Creating 3D objects in the Form mode

In Fusion 360 it is possible to create three-dimensional shapes using the so-called digital sculpture approach. The command used for this purpose is Create Form . It activates the Form menu (Figure 6.1) whose commands can be used to create artistic forms.

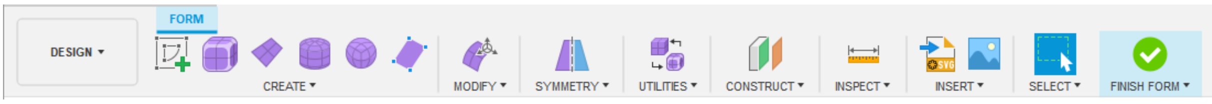


Figure 6.1

The Form menu includes many commands that are used to create various objects made of surfaces (Figure 6.2). The surfaces are the so-called T-splines. A T-spline is a mathematical model for defining free-form surfaces in computer graphics. They allow us to create forms for conceptual projects, which are the basis of the so-called "Digital sculpture".

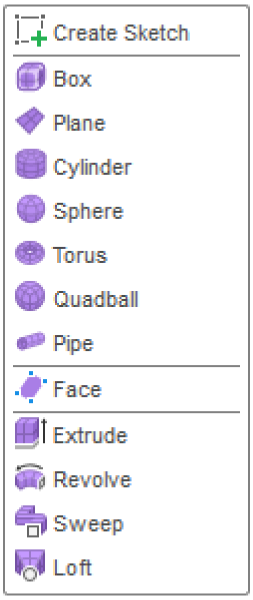


Figure 6.2

The Modify menu includes commands for editing T-spline forms. We can perform actions such as shape editing, adding edges and vertices, splitting one surface or gluing it to another, creating links between edges and vertices, rounding and aligning shapes, and many others basic actions for editing the created forms (Figure 6.3).

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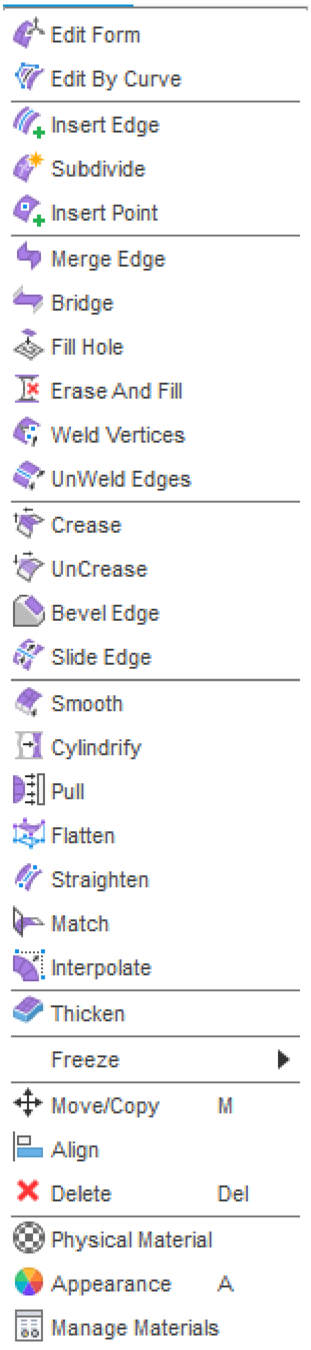

Figure 6.3

After activating the Form command, we create a cylindrical object using the Cylinder command (Figure 6.4). In the menu of the command, it is necessary to indicate the initial diameter of the object and the number of surfaces from which it is built.

The created object is the basis of a digital sculpture. Through modifications of this object we will acquire the final shape.

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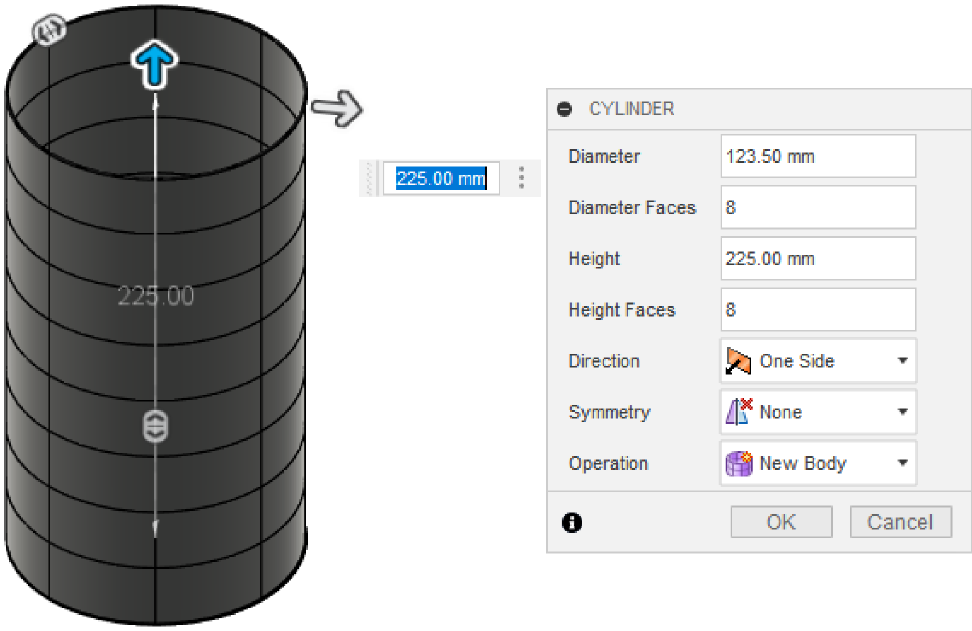

Figure 6.4

After the necessary modifications have been made, the object acquires the shape of Figure 6.5, a. It is still a shape composed of T-splines. To obtain the standard three-dimensional body shape, the T-spline shape should have a completely closed structure or be assigned a thickness (via the Thicken command) (Figure 6.5, b). With the help of T-spline shapes, we can create artistic three-dimensional objects that would be difficult to model in any other way (Figure 6.5, c).

(a) (b) (c)

Figure 6.5

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Working with surfaces

A separate menu in Fusion 360 includes commands for working with two-dimensional surfaces (Figure 6.6). When we are working with surfaces, the idea is to initially create the shape of an object as a set of surfaces, and then to combine or transform the surfaces into a three-dimensional object. Surface commands can also be used to create nonlinear surfaces when cutting or modifying a three-dimensional object.

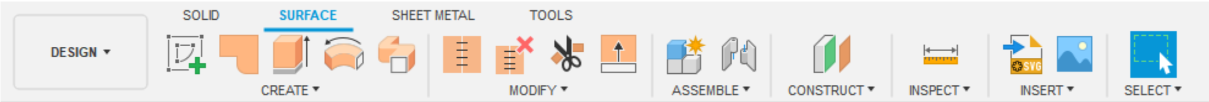

Figure 6.6

Similarly to creating three-dimensional objects and working with surfaces, commands can be used for extrusion, revolved extrusion or path following (Figure 6.7).

Figure 6.7

The editing commands (Figure 6.8) make it possible to stitch and split surfaces, change surface dimensions, etc.

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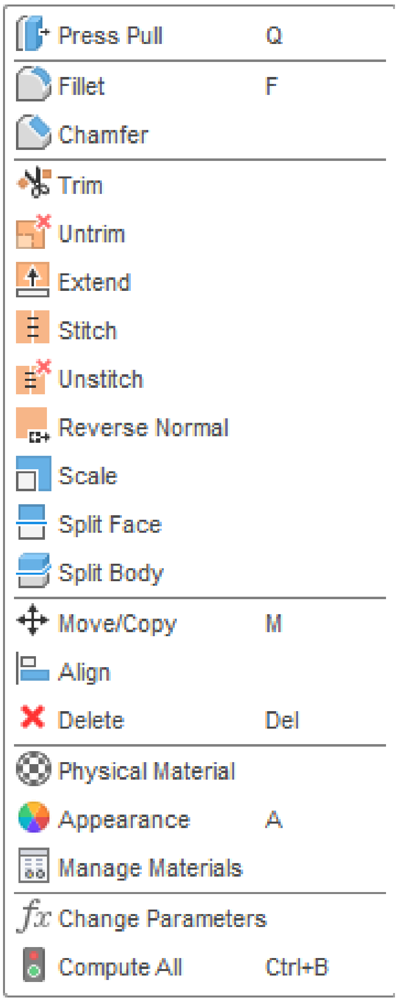
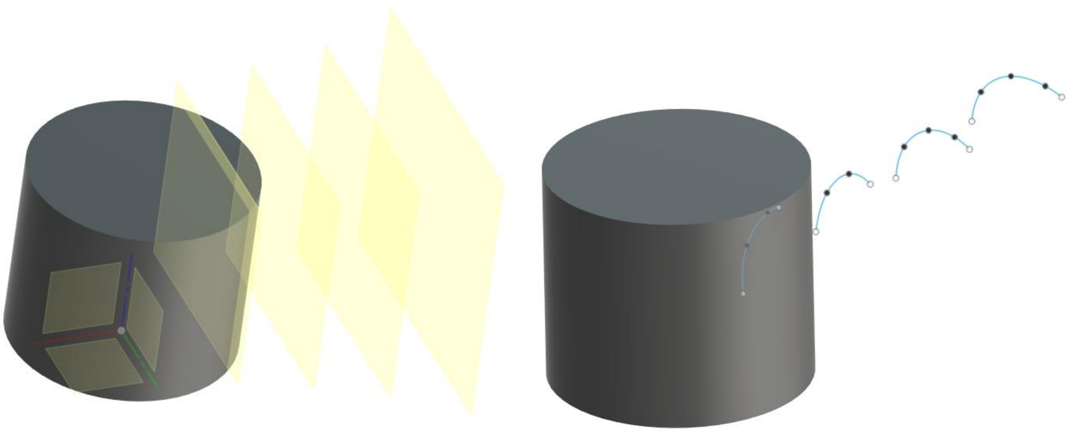


Figure 6.8

To demonstrate the work with surfaces, below we present a cylindrical object. In three construction planes (Figure 6.9, a) we have created sketches and we have drawn curves in them (Figure 6.9, b). The curves will be used to create a fan blade.



(a) (b)

Figure 6.9

To create the fan blade we will use the Loft command (Figure 6.10).

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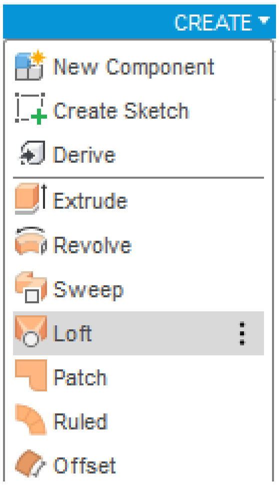

Figure 6.10

We run the Loft command and select the profiles (in this case, the created curves) through which the generated surface should pass (Figure 6.11).

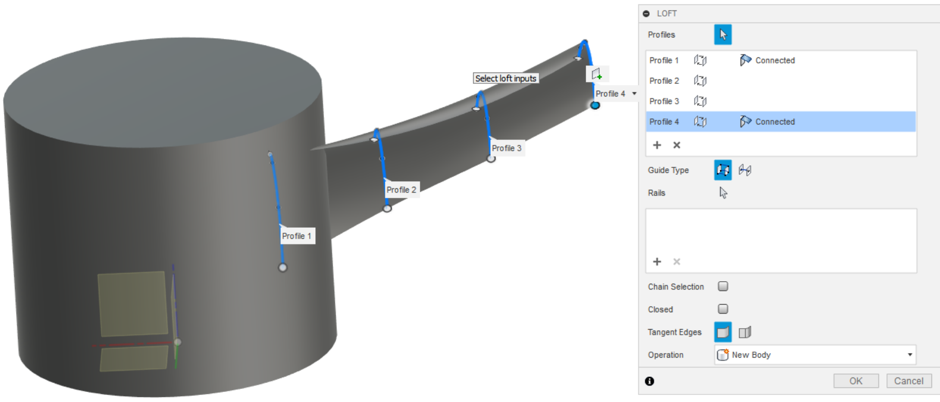

Figure 6.11

Once the surface is created, it appears in the design browser as a surface body (Figure 6.12). It has a shape but no thickness – in this case Body2 (1).

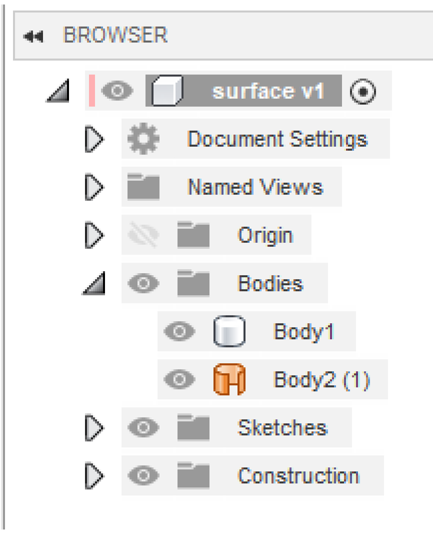

Figure 6.12

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In order to convert the created surface body into a standard three-dimensional body, it is necessary to define its thickness. This can be done with the Thicken command (Figure 6.13).

Figure 6.13

In the menu of the Thicken command we will specify the desired thickness (Figure 6.14).

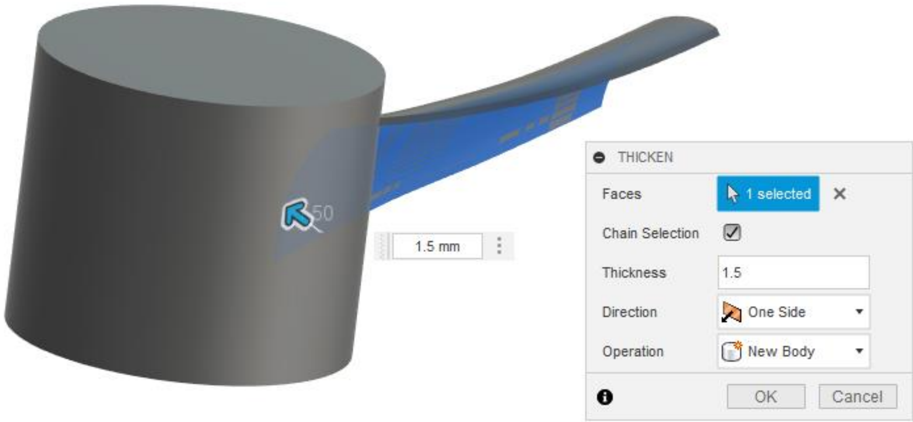

Figure 6.14

As a result, a new body will be created in the browser (Figure 6.15). It is already a standard three-dimensional body – in this case Body3.

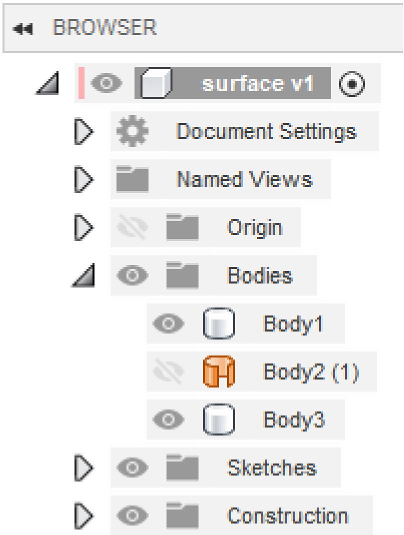

Figure 6.15

The created body can be replicated and modified to obtain the final version of the designed fan blade (Figure 6.17).

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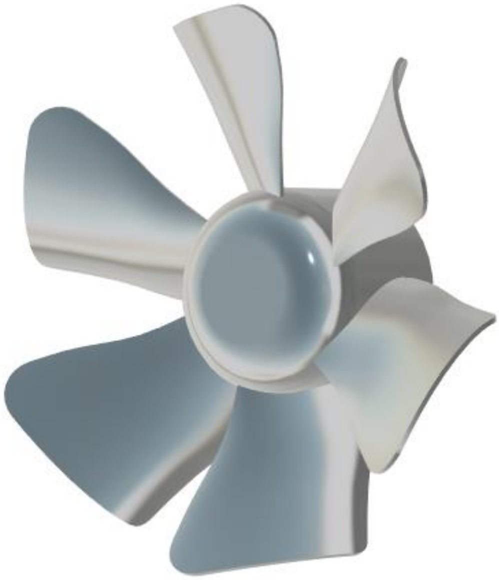

Figure 6.17

Creating models from sheet material

A special menu in Fusion 360 is dedicated to the work and creation of objects from sheet material. With the help of the commands in this menu (Figure 6.18) we can create and modify bodies built by virtually bending sheet material with certain parameters.

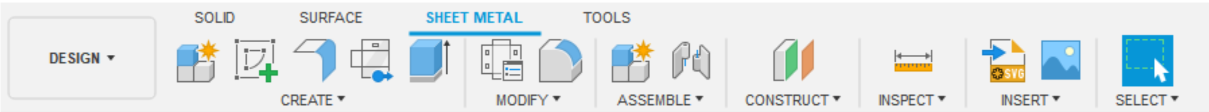

Figure 6.18

The work with sheet material is extremely easy. The Flange command is used to create the geometric shape of the objects (Figure 6.19), and specifically to “bend” the sheet material following a certain angle or contour.

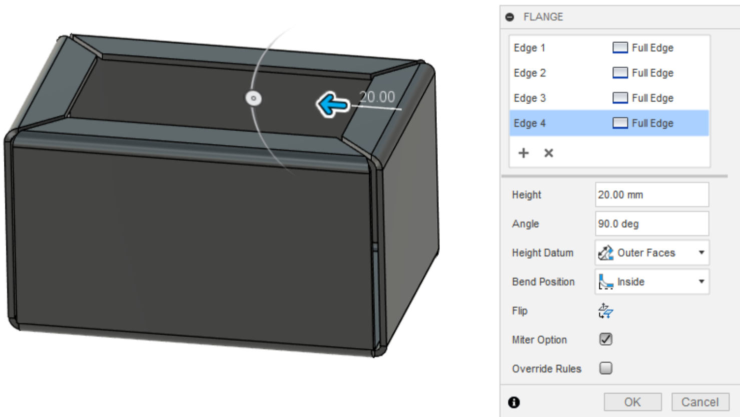

Figure 6.19

Bodies that are created with sheet material can be further modified, e.g. holes can be made in them (Figure 6.20).

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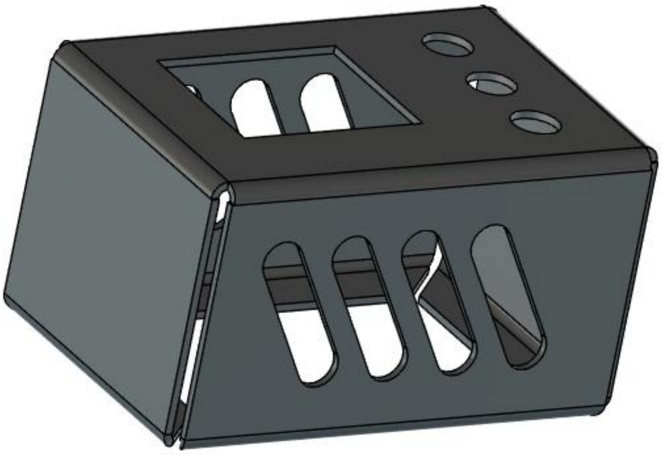

Figure 6.20

Bodies made of sheet material can be “unfolded” to generate a path along which the sheet material should be cut to produce the designed body (Figure 6.21).

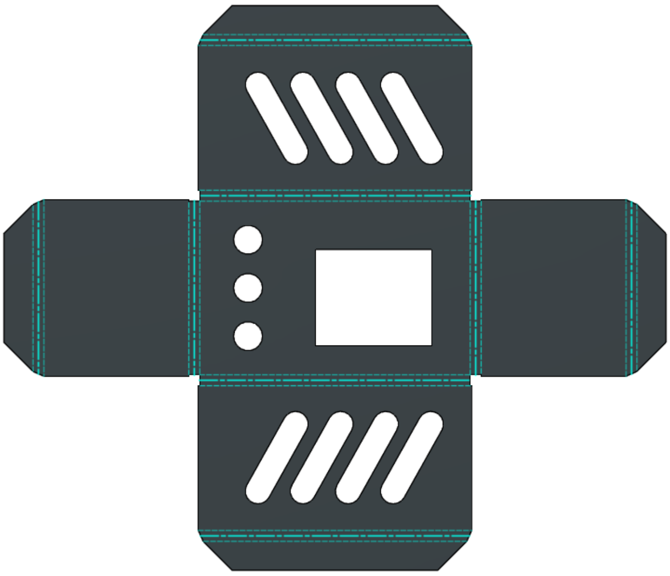

Figure 6.21

Creating realistic images of models

A useful feature of the environment is the ability to create realistic views of the designed components, considering the materials from which they are created and the texture that these materials have. The Render workspace menu (Figure 6.22) makes it possible to create these realistic images

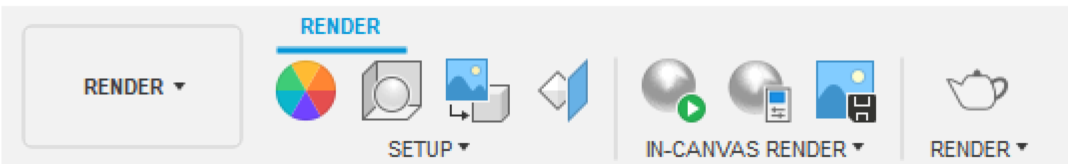

Figure 6.22

Image rendering can be done locally on a personal computer or in the cloud (for the cloud it is necessary to pay the relevant credits). Different formats and dimensions of the final images are supported (Figure 6.23).

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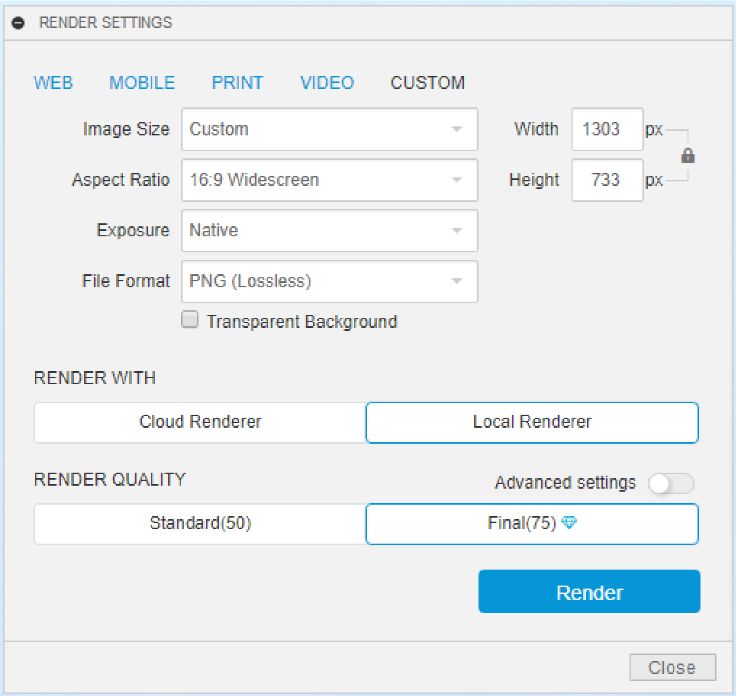

Figure 6.23

The appearance of an image obtained in the rendering process is very close to a photo of a real detail (Figure 6.24).

Figure 6.24

Simulating the behaviour of models under structural loads

Fusion 360 provides capabilities for performing various types of simulations to check and test how a three-dimensional object would respond to different structural loads. This makes it possible to ensure that the part we produce on the basis of the design will have the required strength. The simulations are performed in a separate workspace – Simulation (Figure 6.25).

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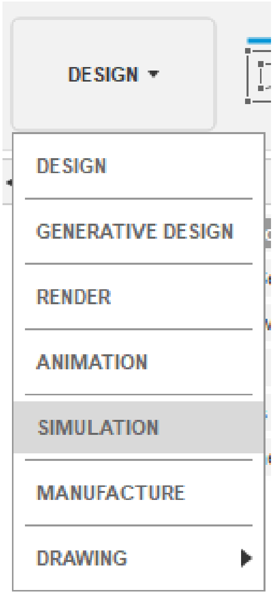

Figure 6.25

The simulations that can be performed are as follows (Figure 6.26):

- Static Stress
- Modal Frequencies
- Electronics Cooling
- Thermal, Thermal Stress
- Structural Buckling
- Nonlinear Static Stress
- Event Simulation
- Shape Optimization
- Injection Molding Simulation.

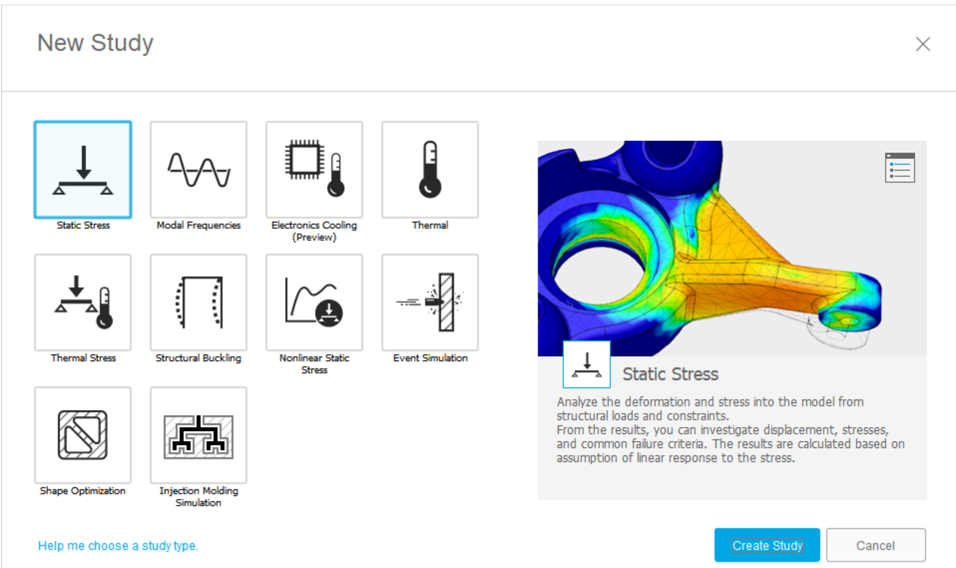

Figure 6.26

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When simulations are performed in the work environment, the main command bar changes (Figure 6.27) to correspond to the sequence of necessary actions that are performed during the simulations.

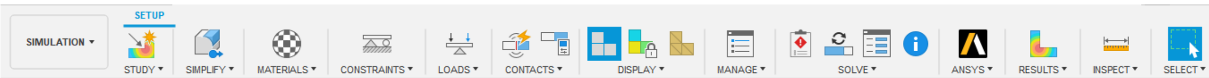

Figure 6.27

When simulating the impact of a static structural load on a specific object, it is necessary to select the plane that will be under stress. The fixed part of the object and the magnitude of the load forces should then be specified (Figure 6.28).

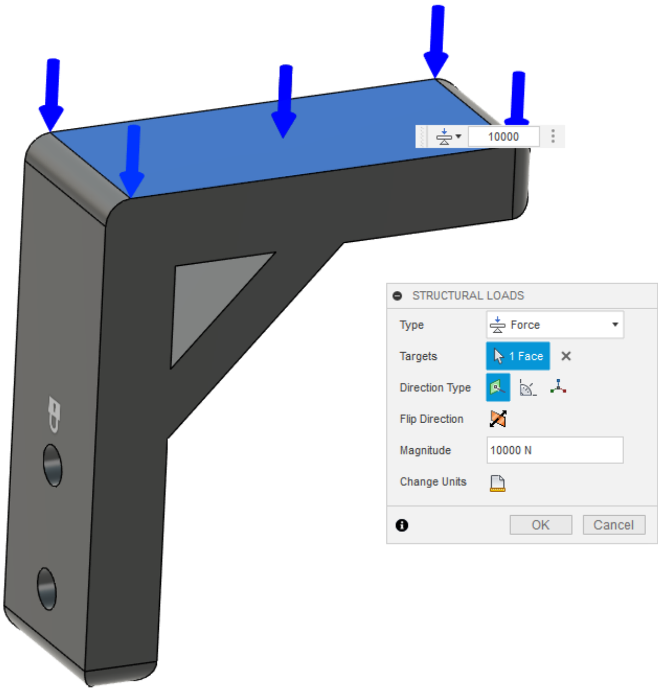

Figure 6.28

During the simulation, the environment returns to us information about the changes in the object's structure resulting from the structural load (Figure 6.29). The information contains an analysis of the object's strength and visually demonstrates the places where problems may occur due to the load.

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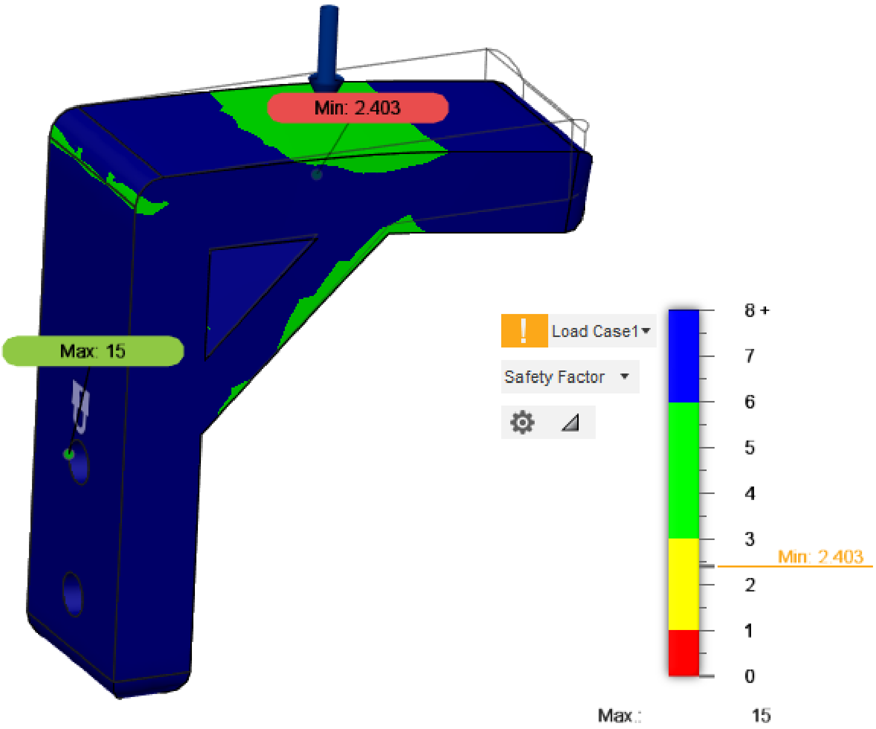

Figure 6.29

When working with simulations, the appearance of the browser changes so as to fully reflect the applied loads and constraints on the model. It also allows for quick access to change individual settings (Figure 6.30).

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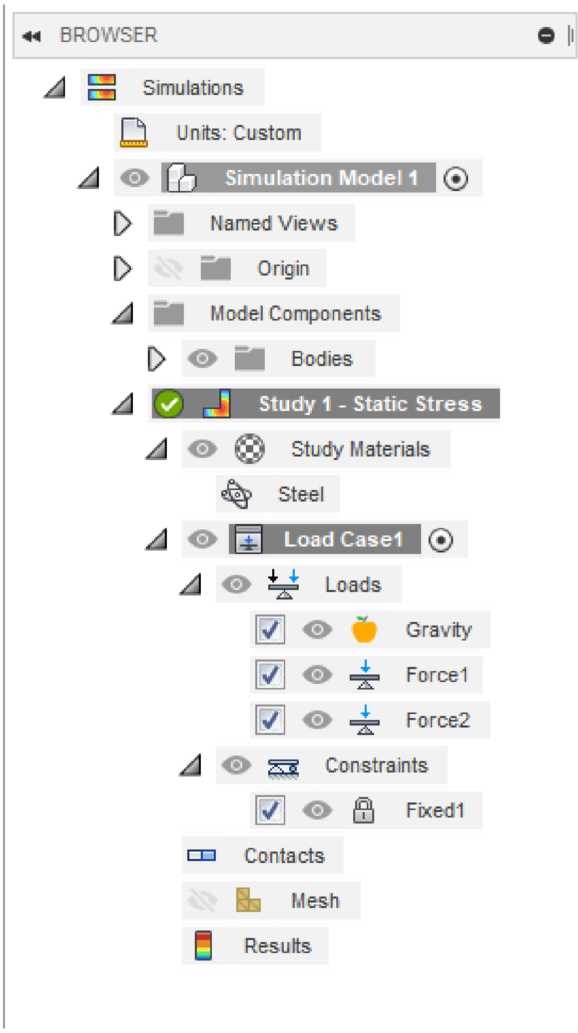

Figure 6.30

Creating animations of the models' operation and their assembly

The creation of animations helps to visualize the way in which the assembly of a mechanical system containing many components is achieved. Animations also help create presentation videos of the mechanical system to fully demonstrate its characteristics. Animations are separated in an individual workspace in Fusion 360. The menu for working and creating animations is made as convenient and user-friendly as possible (Figure 6.31).

Figure 6.31

On the figure below we present an assembled system of multiple components (Figure 6.32).

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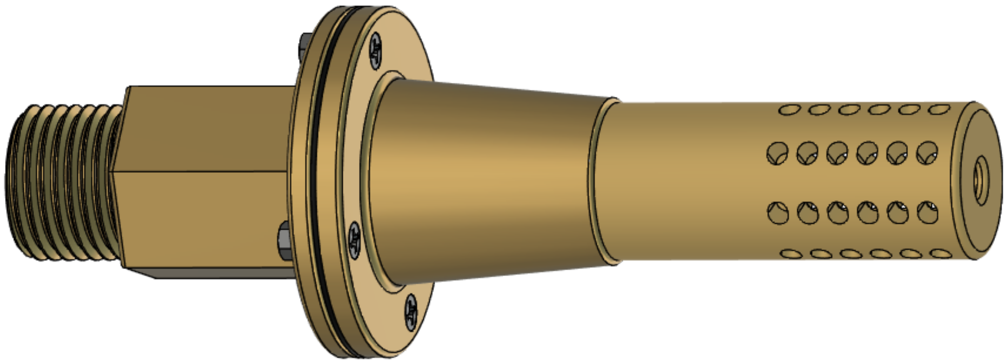

Figure 6.32

By using the commands for creating animations we can disassemble the components of the system and specify the way in which the assembly was performed (Figure 6.33).

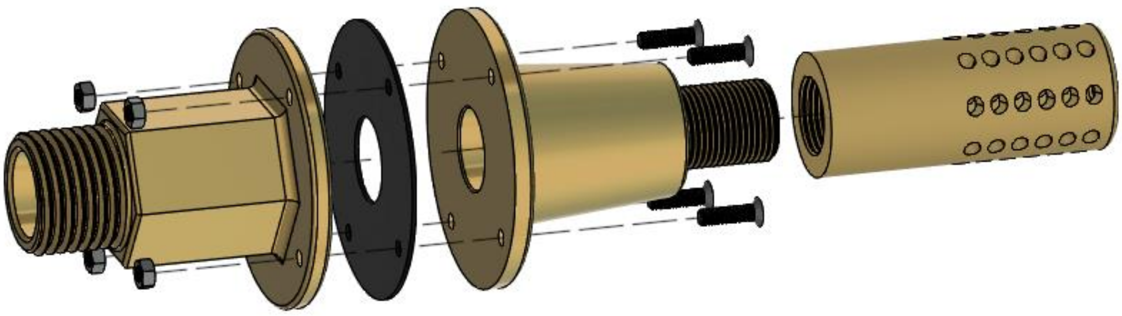

Figure 6.33

Generating technical documentation of the models

The process of designing the model of a part (object) ends with the technical documentation to support its production – its drawing, with marked dimensions and other specifications necessary for the production. With Fusion 360, it is possible to create a drawing from a 3D design or from an animation of an assembly joint (Figure 6.34).

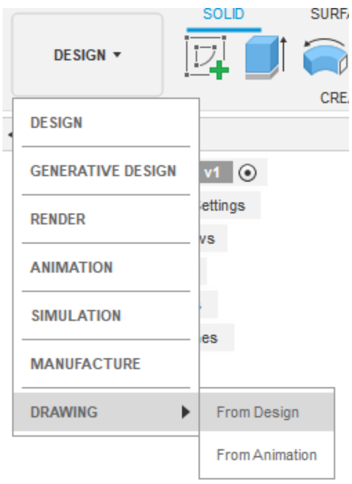

Figure 6.34

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The workspace for creating drawings contains commands that help arrange in the drawing the different views of the part or of the assembly, size them, add sections, and provide additional information necessary for production (Figure 6.35).

Figure 6.35

In Figure 6.36 we presented a three-dimensional object.

Figure 6.36

The object can be easily dimensioned in a drawing that can guide its production (Figure 6.37).

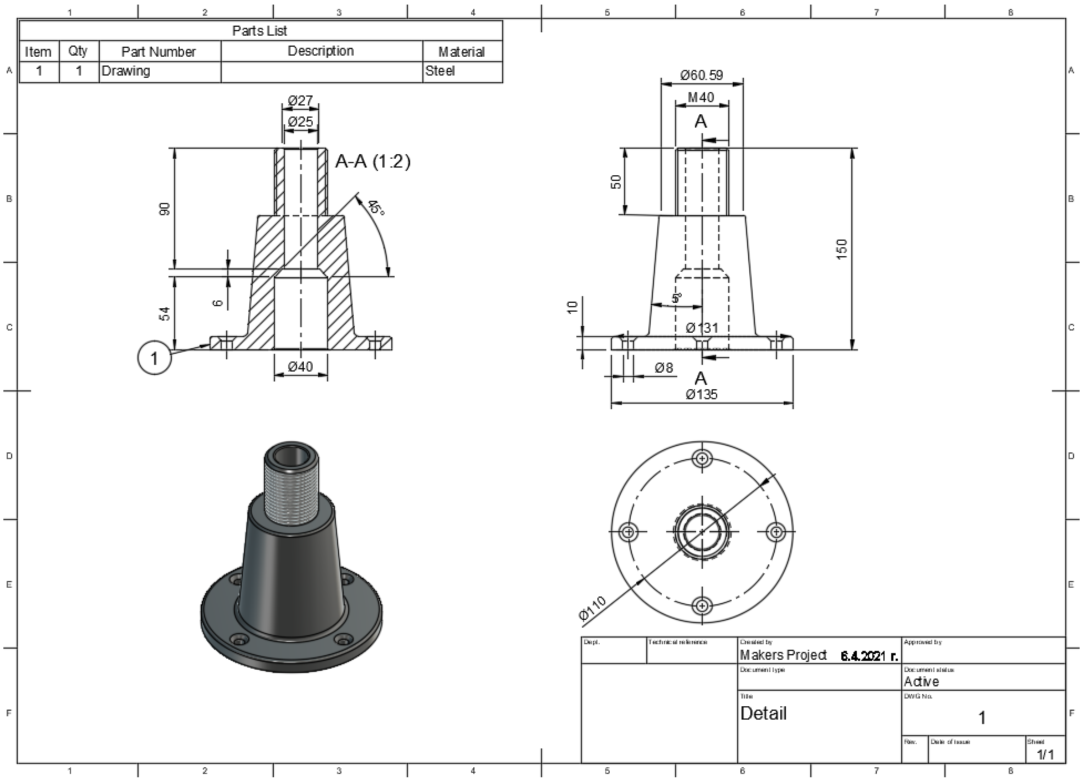

Figure 6.37

Generating NC programs for the models' manufacturing

In Fusion 360 there is a built-in CAM (Computer-Aided Manufacturing) part, with the help of which it is possible to create drivers for various machines with digital software control. The menu of the Manufacture workspace (Figure 6.38) includes commands that define basic operations in the parts' manufacturing, simulate the tool's operation and generate a G code to control the selected CNC (Computer Numerical Control) machine.

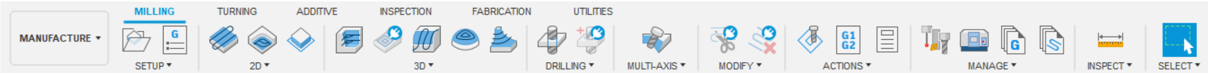

Figure 6.38

When defining the settings, we select tools with the help of which the part will be processed and we specify their dimensions (Figure 6.39).

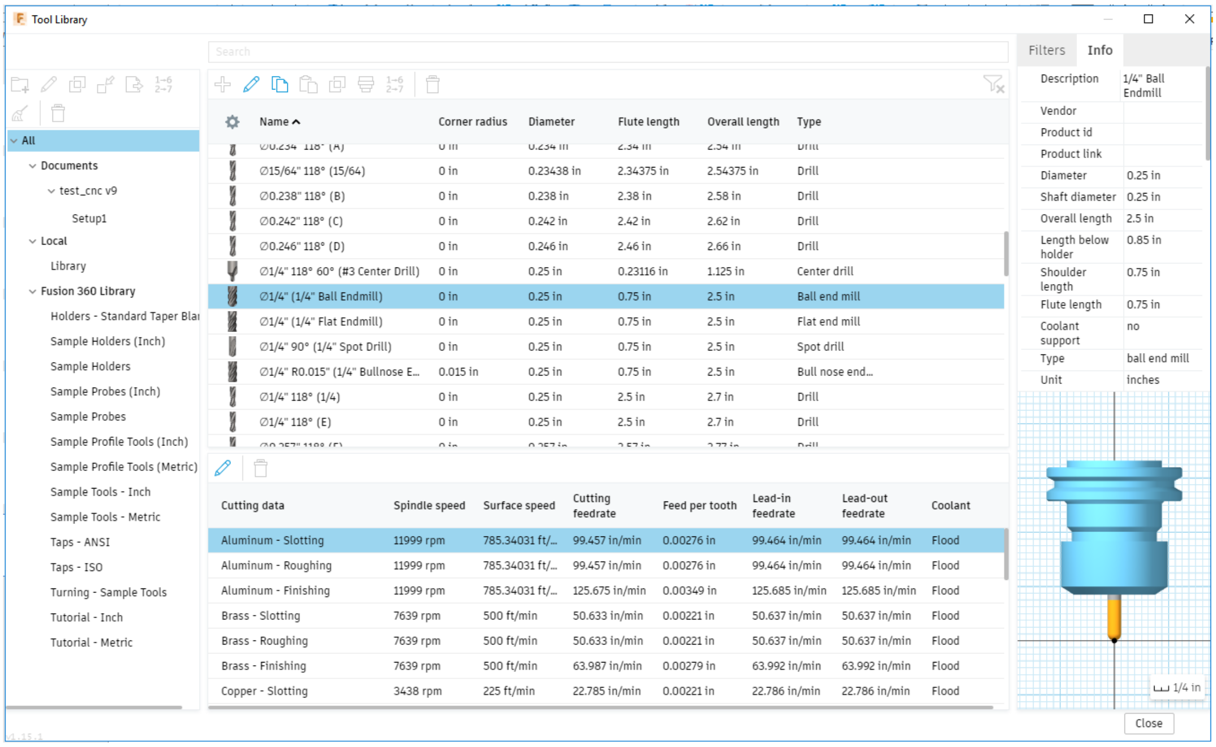

Figure 6.39

The processing of details is divided into a sequence of actions. Firstly, we specify the size of the initial workpiece from which the selected part will be produced. Then, for each operation, the corresponding tool and operating modes of the machine are selected (Figure 6.40).

After specifying all work and production processes, it becomes possible to simulate the production process. The simulation process can detect probable issues that may occur during processing and production. The production machine itself can be simulated, too, in addition to the tool path. This allows us to get a realistic idea of the upcoming production process.

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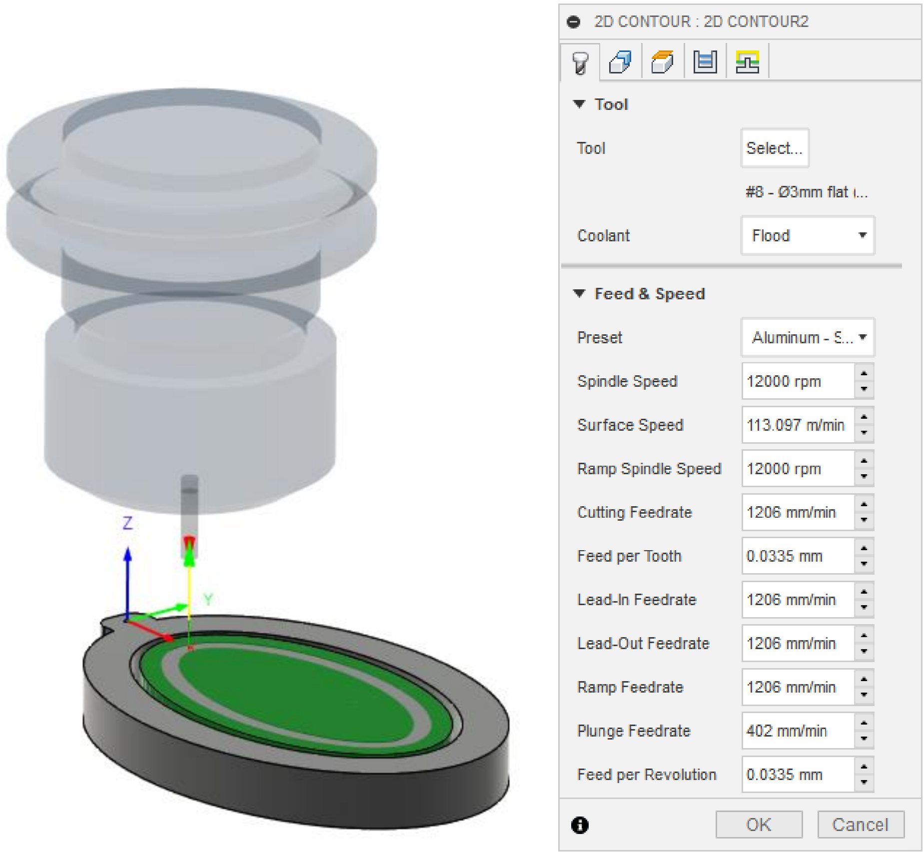

Figure 6.40

The environment supports different types of machines such as: lathes, milling machines, laser guillotines, etc. A large set of post-processors available for download from the Autodesk site can be used to generate the control G code.

Creating optimized 3D models using Generative Design

Working with the so-called Generative Design is an innovative approach to the design of three-dimensional models. It uses artificial intelligence to optimize the models' shape. In Fusion 360 the work with "generative design" is performed as a cloud service, for which the corresponding number of credits is paid from the ones available in the user profile of the designer. The Generative Design workspace has a set of commands (Figure 6.41) that prepare a pre-designed 3D model to be optimized through the Autodesk cloud service.

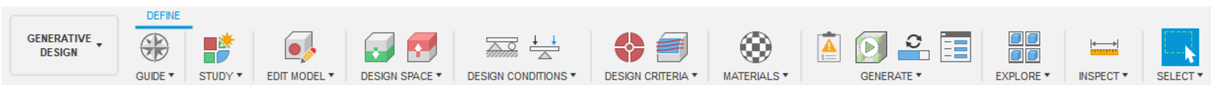

Figure 6.41

The design browser also changes its structure, storing the individual settings used to optimize the model (Figure 6.42).

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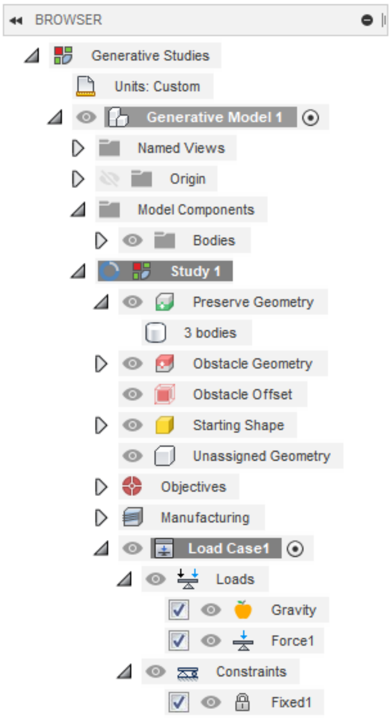

Figure 6.42

In Figure 6.43 we show a model of a three-dimensional body selected for optimization.

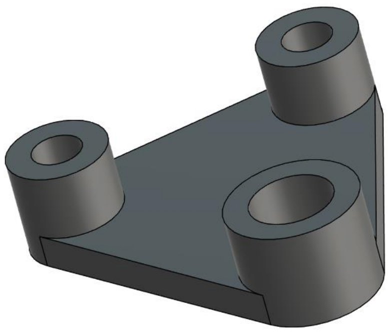

Figure 6.43

Initially, it is necessary to define the areas that will keep their geometry unchanged, as well as the areas where no material will be deployed in the process of generating the object shape (Figure 6.44).

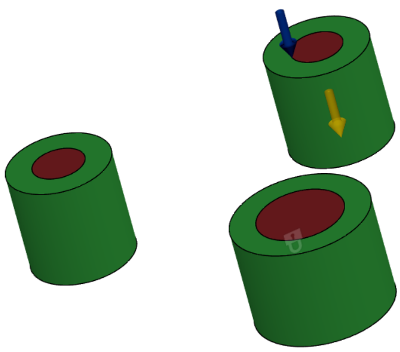

Figure 6.44

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We also need to define constraints related to the nature and the force of the structural loads that will be applied on the object. Based on this, the optimizer determines the required amount of material and its location in the final object. The cloud service generates many variants of the optimized form that the user can choose from (Figure 6.45)

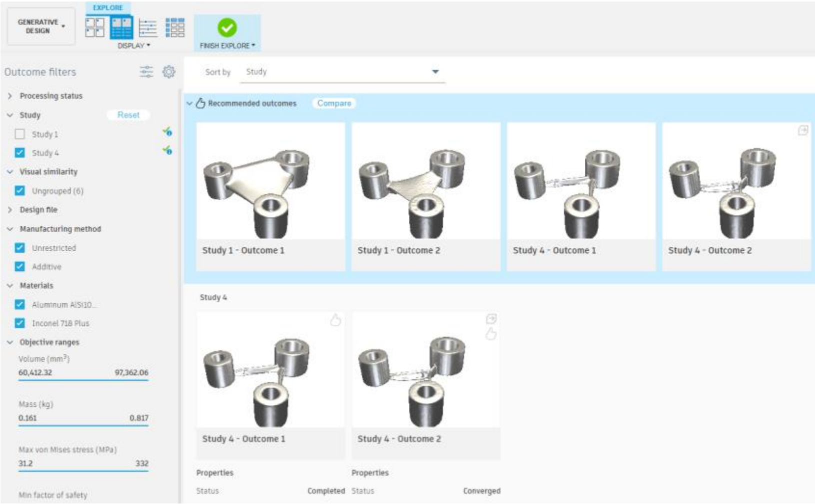

Figure 6.45

In Figure 6.46 we present two of the generated optimized shapes of the workpiece from Figure 6.43.

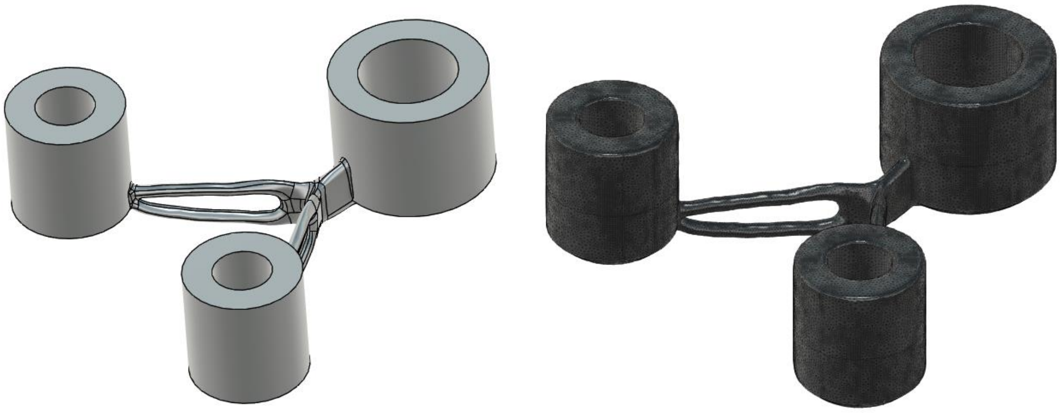

Figure 6.46

Using generative design, it is possible to design objects in a way ensures optimal strength of the part while using a reduced amount of material.

Developing electronic circuit boards with the help of Fusion 360

Fusion 360 also provides the ability to develop and design printed circuit boards for electronic devices. When creating a product we can handle both its mechanical development and the development of its control electronics. Like in other Printed Circuit Board design environments, the development of an electronic device begins with the creation of its circuit (Figure 6.47).

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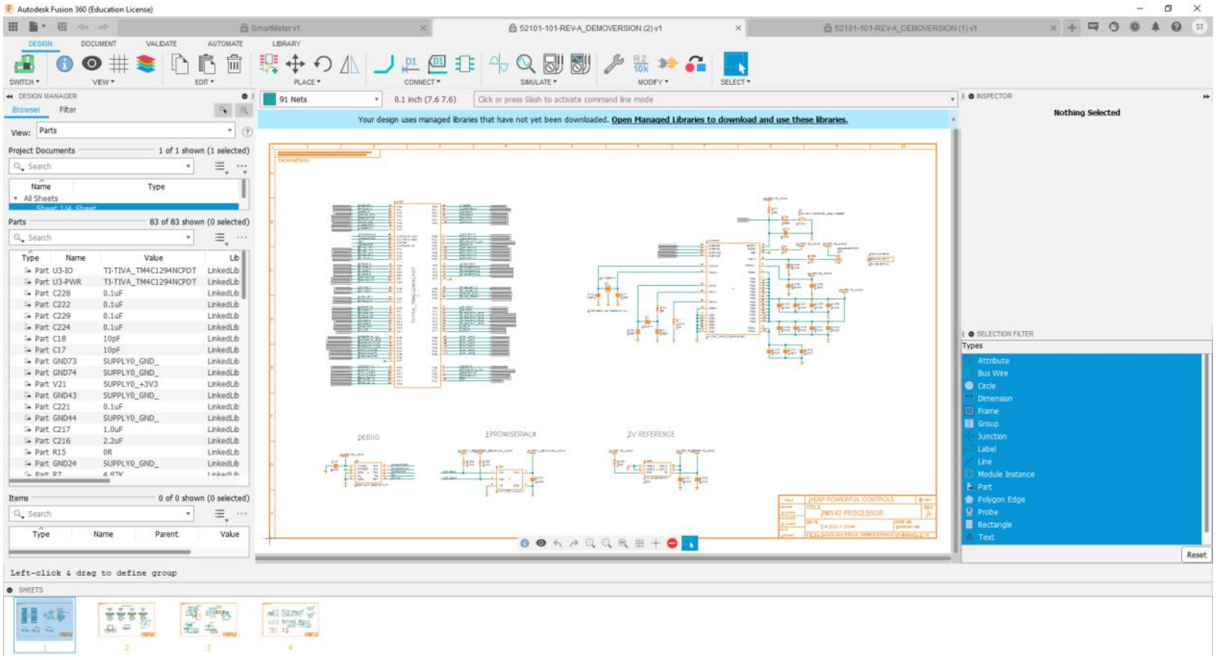

Figure 6.47

After the scheme has been created and the housings of the used components have been specified, the creation of the printed circuit board can start (Figure 6.48). The working environment includes a large number of commands that facilitate the process of wiring the tracks on the board and setting the necessary restrictions.

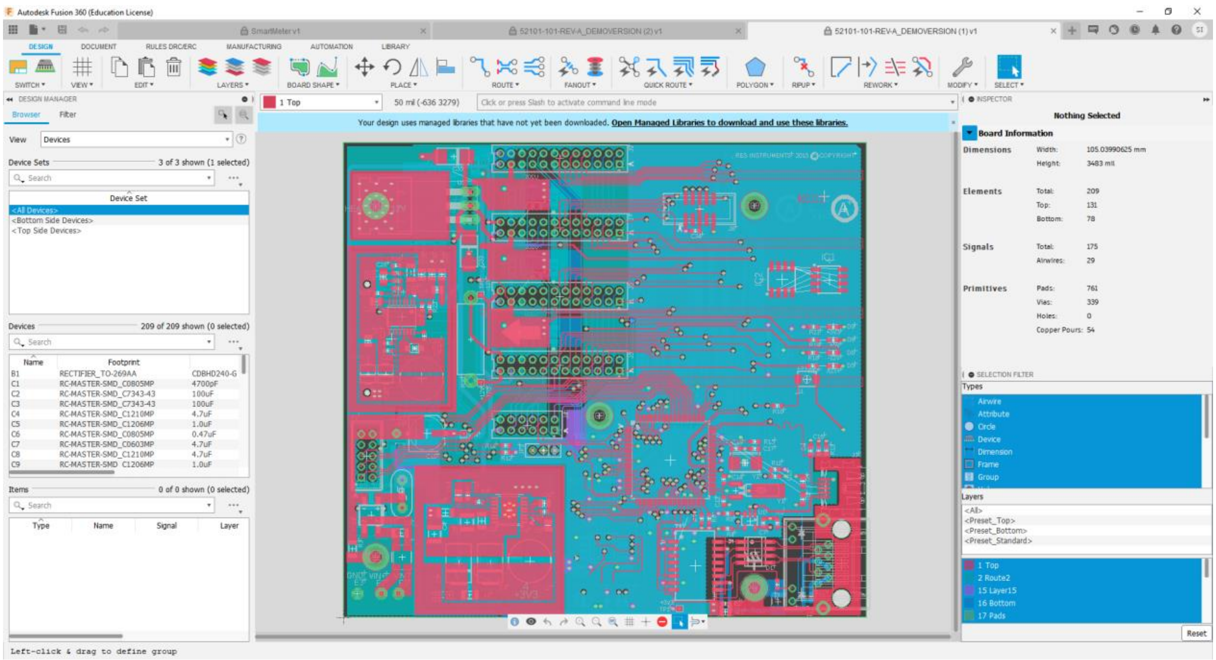

Figure 6.48

The developed Printed Circuit Boards can be integrated into the mechanical design of the created product. Fusion 360 generates a realistic three-dimensional model of the board and the electronic components on it (Figure 6.49).

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Figure 6.49

Using additional modules added to Fusion 360

Fusion 360 allows the user to install add-ins created by both Autodesk and other companies or users. Expansion modules are added with the Add-Ins command from the Utilities menu (Figure 6.50).

Figure 6.50

The add-ins can be downloaded from the Autodesk website, from the corresponding add-ins page. Both paid and free add-in modules are available. They offer features that are not supported in the basic version of Fusion 360. Such functions include creation of gears and gear wheels, creation of chain wheels, design of shafts, creation of tubular and profile constructions, etc.

One of the useful extension modules allows the creation of gears. During its installation the module creates an additional command in the work environment. When activated, the command opens a menu that sets all the necessary parameters for creating a gear (Figure 6.51). All settings are defined by mouse clicks.

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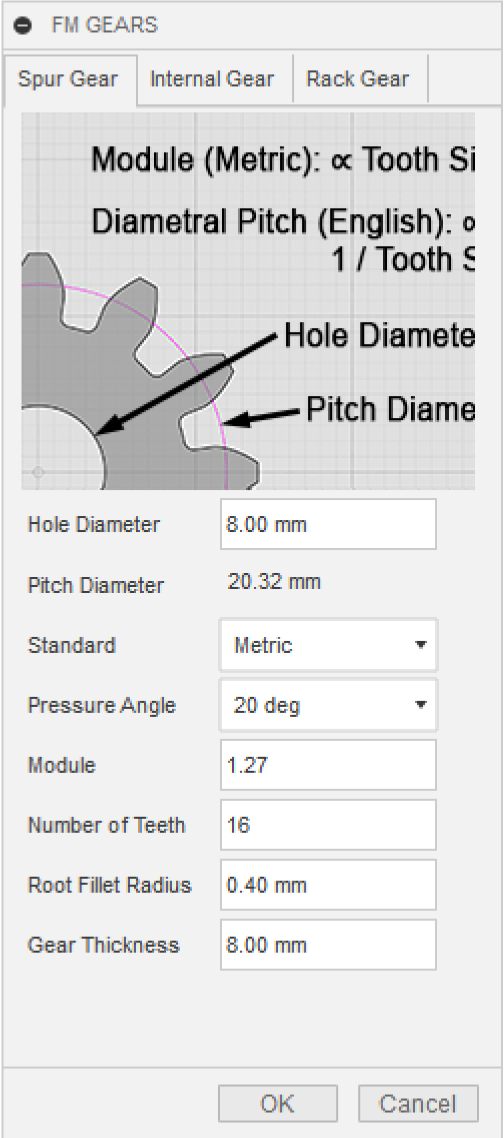

Figure 6.51

The generated gear (Figure 6.52) is a standard three-dimensional object that can be easily modified using Fusion 360 commands.

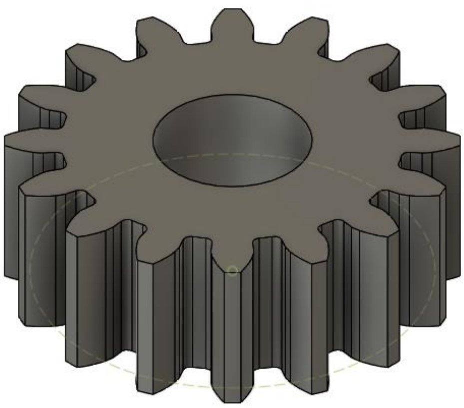

Figure 6.52

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Additional self-study video materials:

https://www.youtube.com/watch?v=EO_2QsekC8g&t=2s

https://www.youtube.com/watch?v=KThyzlM_UFw

https://www.youtube.com/watch?v=4a9YCrnypNA

https://www.youtube.com/watch?v=8HwM07hmXhM

https://www.youtube.com/watch?v=HmgwKewMcw8

https://www.youtube.com/watch?v=hIIT2WPEnuk

https://www.youtube.com/watch?v=Z5qqw_PmAAc

https://www.youtube.com/watch?v=406aQFy6IbM

https://www.youtube.com/watch?v=cgKe3JGNIEg

https://www.youtube.com/watch?v=NXu8vVYvjrg

https://www.youtube.com/watch?v=_Hl-WY_1UZE

https://www.youtube.com/watch?v=FRp4fdEEUTM

https://www.youtube.com/watch?v=2LzWy4DYfx8

https://www.youtube.com/watch?v=dwCrOwEw7u8

https://www.youtube.com/watch?v=jcUF92XDenQ

https://www.youtube.com/watch?v=IE2aQiEbwjQ

https://www.youtube.com/watch?v=bZnHQTPP-Ps

https://www.youtube.com/watch?v=Mr178Y0Mqgc

https://www.youtube.com/watch?v=xhhVjKQOECY

https://www.youtube.com/watch?v=OHUvn5Dwu2Q

https://www.youtube.com/watch?v=wC_lI0wxxS4

https://www.youtube.com/watch?v=GUDhet2TKHQ

https://www.youtube.com/watch?v=whGKwsEY4Vo

https://www.youtube.com/watch?v=Do_C_NLH5sw

https://www.youtube.com/watch?v=HRPP1yvDP7Q

https://www.youtube.com/watch?v=a6bDLMWlS98

https://www.youtube.com/watch?v=PSSt8wswNJQ

https://www.youtube.com/watch?v=zh03X5Wh5MM

https://www.youtube.com/watch?v=DqlSDB3x10w&t=77s

https://www.youtube.com/watch?v=5k82bYJ58wA

https://www.youtube.com/watch?v=hpAGHGVRgnw

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Conclusion

The examples presented in this learning resource demonstrate the basic knowledge needed to work with Fusion 360. This knowledge should be sufficient for users to be able to create three-dimensional objects and even complex mechanical systems. The consolidation of this knowledge is a matter of practice and experimentation with the programming environment. The purpose of this learning resource was to present the basics of three-dimensional modelling. Therefore, the creation of more complex geometric shapes requires users to search for additional information on the Internet, where there are a large number of video tutorials covering all aspects of working with Fusion 360. The knowledge gained from this learning resource can be successfully applied when working with 3D printers and to design parts for 3D printing. For students, teachers, and people for whom 3D printing is a hobby, this resource had the task of three-dimensional modelling enjoyable and easy, demonstrating many basic examples that can be easily reproduced.

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