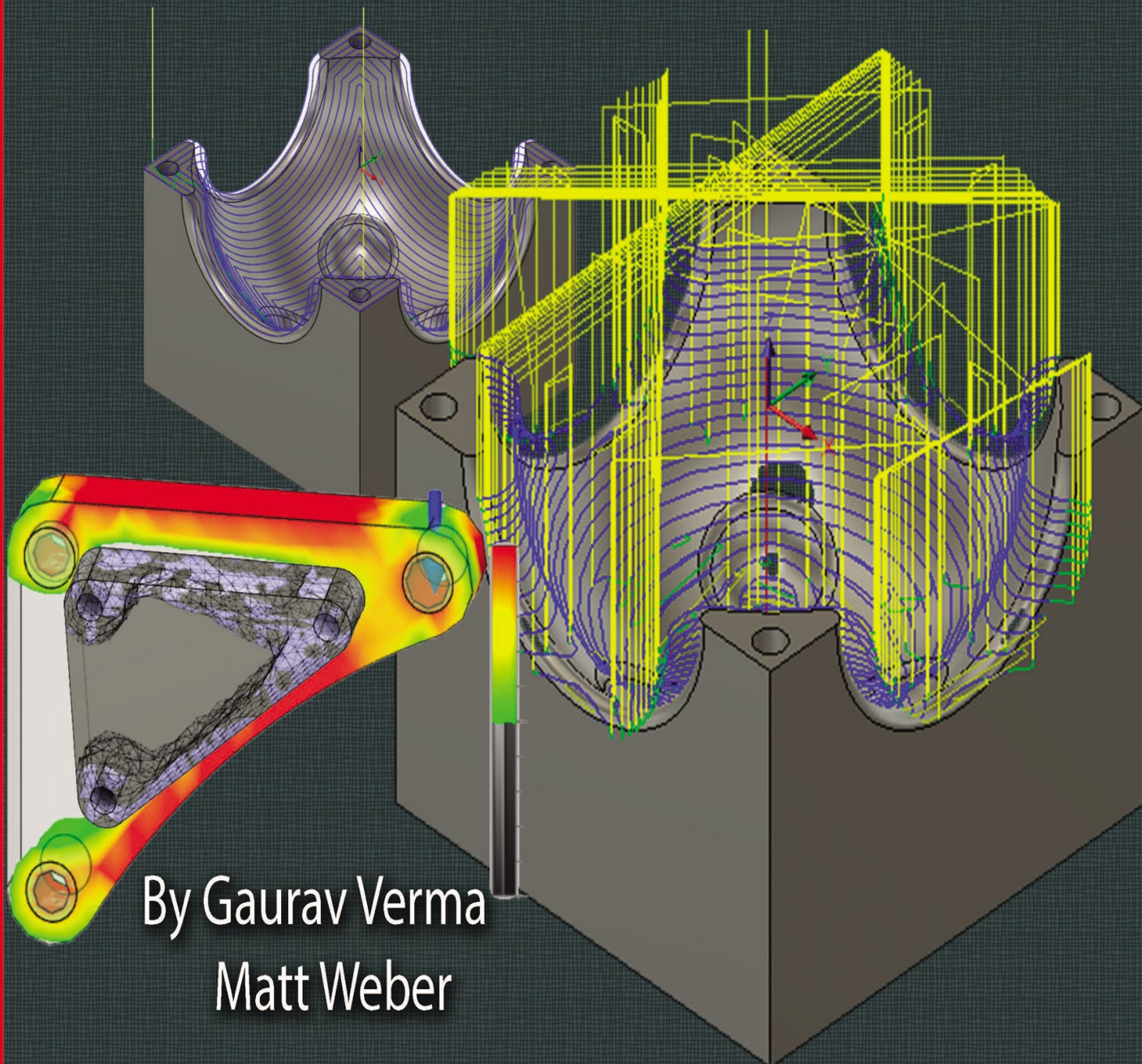


Autodesk Fusion 360 Black Book (V 2.0.15293) - Part 1

Book Covers:

Sketching, Solid Modeling, Direct Editing, Fusion Team, Basics of Engineering Drawing, Assembly Design, Surface Design, Sheetmetal, Rendering, Animation, Drafting, 3D PDF, 3D Printing, and Drawing with details of Geometric Tolerance



By Gaurav Verma
Matt Weber

Autodesk Fusion 360 Black Book
(V 2.0.15293) - Part 1

**Autodesk Fusion 360 Black Book (V 2.0.15293) - Part 1
(Colored)**

By

Gaurav Verma

Matt Weber

(CADCAMCAE Works)



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DEDICATION

To teachers, who make it possible to disseminate knowledge
to enlighten the young and curious minds
of our future generations
To students, who are the future of the world

THANKS

To my friends and colleagues
To my family for their love and support

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Preface

Autodesk Fusion 360 is a product of Autodesk Inc. Fusion

360 is the first of its kind software which combine 3D CAD, CAM, and CAE tool in single package. It connects your entire product development process in a single cloud-based platform that works on both Mac and PC. In CAD environment, you can create the model with parametric designing and dimensioning. The CAD environment is equally applicable for assembly design. The CAE environment facilitates to analysis the model under real-world load conditions. Once the model is as per your requirement then generate the NC program using the CAM environment.

The **Autodesk Fusion 360 Black Book** (V 2.0.15293) is 6th edition of our series on Autodesk Fusion 360. The book is updated on Autodesk Fusion 360 Ultimate, Student V 2.0.15293. With lots of features and thorough review, we present a book to help professionals as well as beginners in creating some of the most complex solid models. The book follows a step by step methodology. In this book, we have tried to give real-world examples with real challenges in designing. We have tried to reduce the gap between educational use of Autodesk Fusion 360 and industrial use of Autodesk Fusion 360. This edition of book, includes latest topics on Sketching, 3D Part Designing, Assembly Design, Sculpting, Mesh Design, CAM, Simulation, Sheetmetal, 3D printing, Manufacturing, and many other topics. The book covers almost all the information required by a learner to master the Autodesk Fusion 360. The book starts with

sketching and ends at advanced topics like Manufacturing, Simulation, and Generative Design. Some of the salient features of this book are :

In-Depth explanation of concepts

Every new topic of this book starts with the explanation of the basic concepts. In this way, the user becomes capable of relating the things with real world.

Topics Covered

Every chapter starts with a list of topics being covered in that chapter. In this way, the user can easily find the topic of his/her interest easily.

Instruction through illustration

The instructions to perform any action are provided by maximum number of illustrations so that the user can perform the actions discussed in the book easily and effectively. There are about **2400** small and large illustrations that make the learning process effective.

Tutorial point of view

At the end of concept's explanation, the tutorial makes the understanding of users firm and long lasting. Almost each chapter of the book has tutorials that are real world projects. Moreover most of the tools in this book are discussed in the form of tutorials.

Project

Projects and exercises are provided to students for

practicing.

For Faculty

If you are a faculty member, then you can ask for video tutorials on any of the topic, exercise, tutorial, or concept. As faculty, you can register on our website to get electronic desk copies of our latest books, self-assessment, and solution of practical. Faculty resources are available in the **Faculty Member** page of our website (www.cadcamcaeworks.com) once you login. Note that faculty registration approval is manual and it may take two days for approval before you can access the faculty website.

New

If anything is added or enhanced in this edition which was not available in the previous editions, then it is displayed with symbol **New** in table of content.

Formatting Conventions Used in the Text

All the key terms like name of button, tool, drop-down etc. are kept bold.

Free Resources

Link to the resources used in this book are provided to the users via email. To get the resources, mail us at [**cadcamcaeworks@gmail.com**](mailto:cadcamcaeworks@gmail.com) with your contact information. With your contact record with us, you will be provided latest updates and informations regarding various technologies. The format to write us mail for resources is as

follows:

Subject of E-mail as ***Application for resources of _____book.***

Also, given your information like

Name:

Course pursuing/Profession:

E-mail ID:

Note: We respect your privacy and value it. If you do not want to give your personal informations then you can ask for resources without giving your information.

About Authors

The author of this book, Gaurav Verma, has authored and assisted in more than 16 titles in CAD/CAM/CAE which are already available in market. He has authored **AutoCAD Electrical Black Books** which are available in both **English** and **Russian** language. He has also authored books on various modules of Creo Parametric and SolidWorks. He has provided consultant services to many industries in US, Greece, Canada, and UK. He has assisted in preparing many Government aided skill development programs. He has been speaker for Autodesk University, Russia 2014. He has assisted in preparing AutoCAD Electrical course for Autodesk Design Academy. He has worked on Sheetmetal, Forging, Machining, and Casting designs in Design and Development departments of various manufacturing firms.

If you have any query/doubt in any CAD/CAM/CAE package, then you can contact the authors by writing at cadcamcaeworks@gmail.com

For Any query or suggestion

If you have any query or suggestion, please let us know by mailing us on ***cadcamcaeworks@gmail.com***. Your valuable constructive suggestions will be incorporated in our books.

Chapter 1

Starting with Autodesk Fusion 360

Topics Covered

The major topics covered in this chapter are:

- *Overview of Autodesk Fusion 360*
- *Installing Autodesk Fusion 360 (Educational)*
- *Starting Autodesk Fusion 360*
- *Starting a New Document*
- *File Menu*
- *Preform*
- *Undo and Redo button*
- *User Account drop-down*
- *Help drop-down*
- *Data Panel*
- *Simulation Browser*
- *Navigation Bar*
- *Display Bar*
- *Customizing Toolbar and Marking Menu*

OVERVIEW OF AUTODESK FUSION 360

Autodesk Fusion 360 is an Autodesk product designed to be a powerful 3D Modeling software package with an integrated, parametric, feature based CAM module built into the software. Autodesk Fusion 360 is the first of its kind 3D CAD, CAM, and CAE tool. It connects your entire product development process in a single cloud-based platform that works on both Mac and Microsoft Windows; refer to [Figure-1](#).

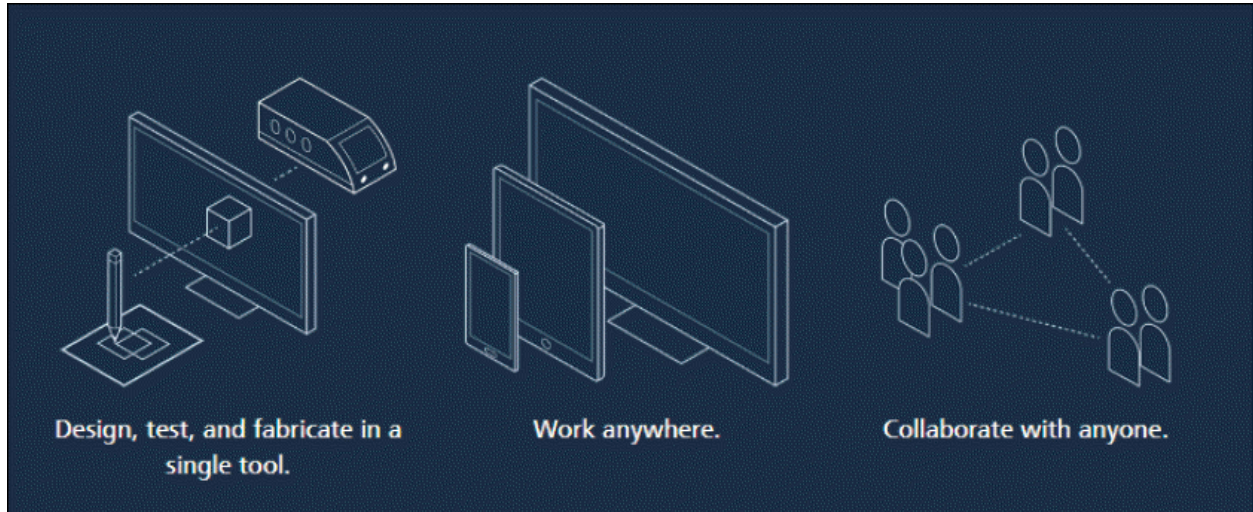


Figure-1. Overview

It combines mechanical design, collaboration, simulation, and machining in a single software. The tools in Fusion 360 enable rapid and easy exploration of design ideas with an integrated concept to production toolset. This software needs a good network connection to work in collaboration with other team members.

This software is much affordable than any other software offered by Autodesk. To use this software, one need to pay monthly subscription of Fusion 360. You can work offline in this software and later save the file on Autodesk Server. User can access this software from anywhere with an internet connection. The user is able to open the saved file and also able to share files with anyone from anywhere as long as he/she has the software and good internet connection. Also, the pricing of this software is so effective that anyone can use it for manufacturing of tools and parts. Autodesk Fusion 360 is build to work in multi body manner:

both parts and assemblies build in a single file. The procedure to install the software is given next.

INSTALLING AUTODESK FUSION 360 (STUDENT)

- Connect your PC with the internet connection and then log on to <https://www.autodesk.com/education/edu-software/overview> as shown in [Figure-2](#).

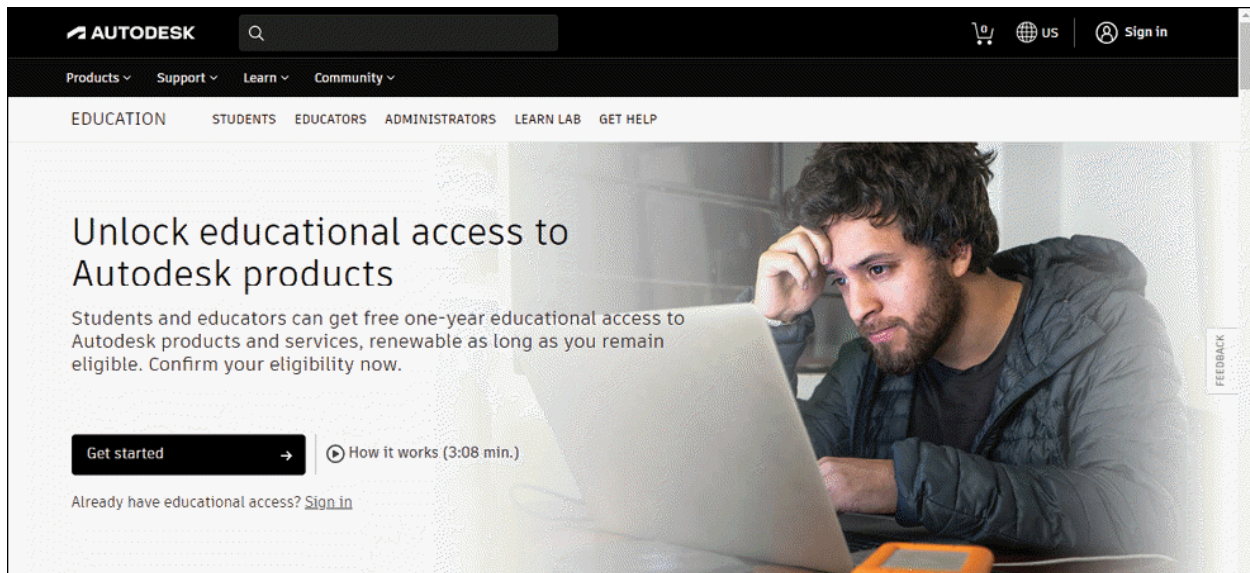


Figure-2. Autodesk Website

- Click on the **GET STARTED** button and on next pages, in **Education role** drop-down, you need to select **Student** option and provide your date of birth. (There is free subscription for students with license term of 1 year which can be renewed).
- After filling the details, click on **NEXT** button. In this page, you need to fill your personal details and then click on **Create Account** as shown in [Figure-3](#). A message will be displayed confirming the account creation; refer to [Figure-4](#).

Create account

First name

Last name

Email

Confirm email

Password

I agree to the [Autodesk Terms of Use](#) and to the use of my personal information in accordance with the [Privacy Statement](#) (including cross-border transfers as described in the statement).

CREATE ACCOUNT

ALREADY HAVE AN ACCOUNT? [SIGN IN](#)

Figure-3. Creating account

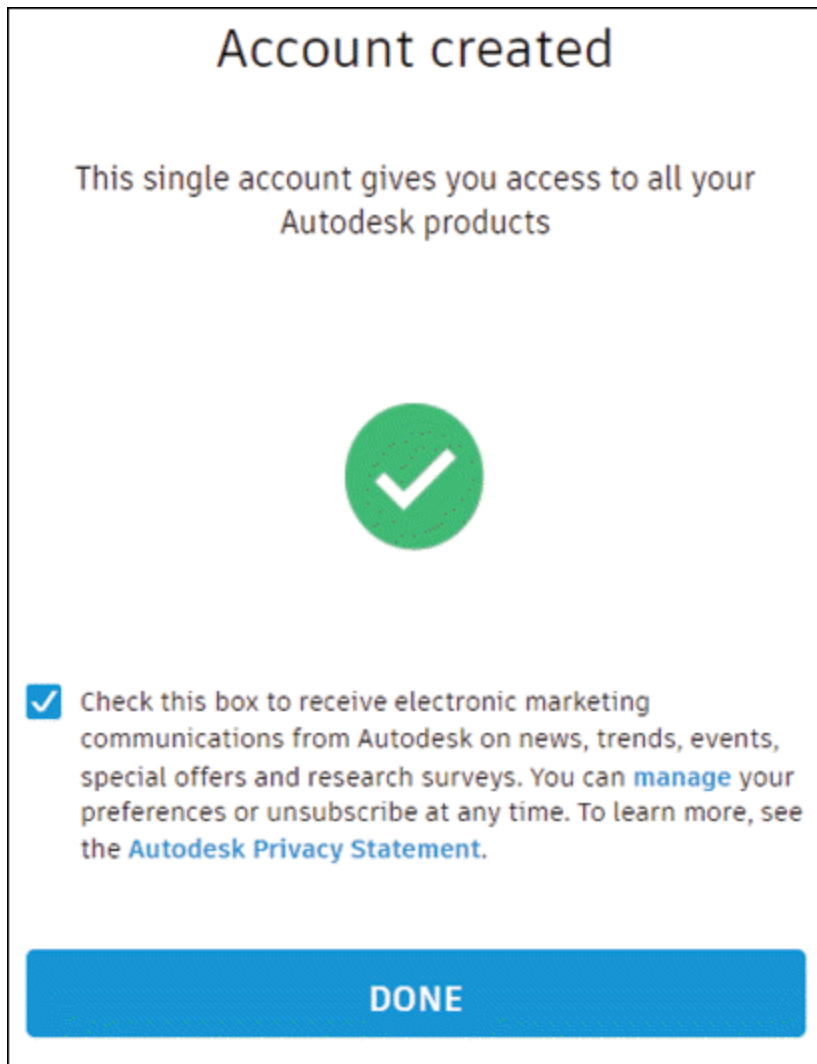


Figure-4. Account created message

- Click on the **DONE** button. You will be sent back to education page of Autodesk.
- Click on the **Get Educational Access** button from the page; refer to [Figure-5](#). The web page to confirm your educational status will be displayed.
- Provide the details in fields of this page and then click on the **CONFIRM** button. After completion of these processes, you may be asked to upload your documentation

like ID card scan or receipt to confirm your student status. Once your account is approved which can take upto 2 days, you need to sign-in to your Autodesk Account and search for the **Autodesk Fusion 360 software**.

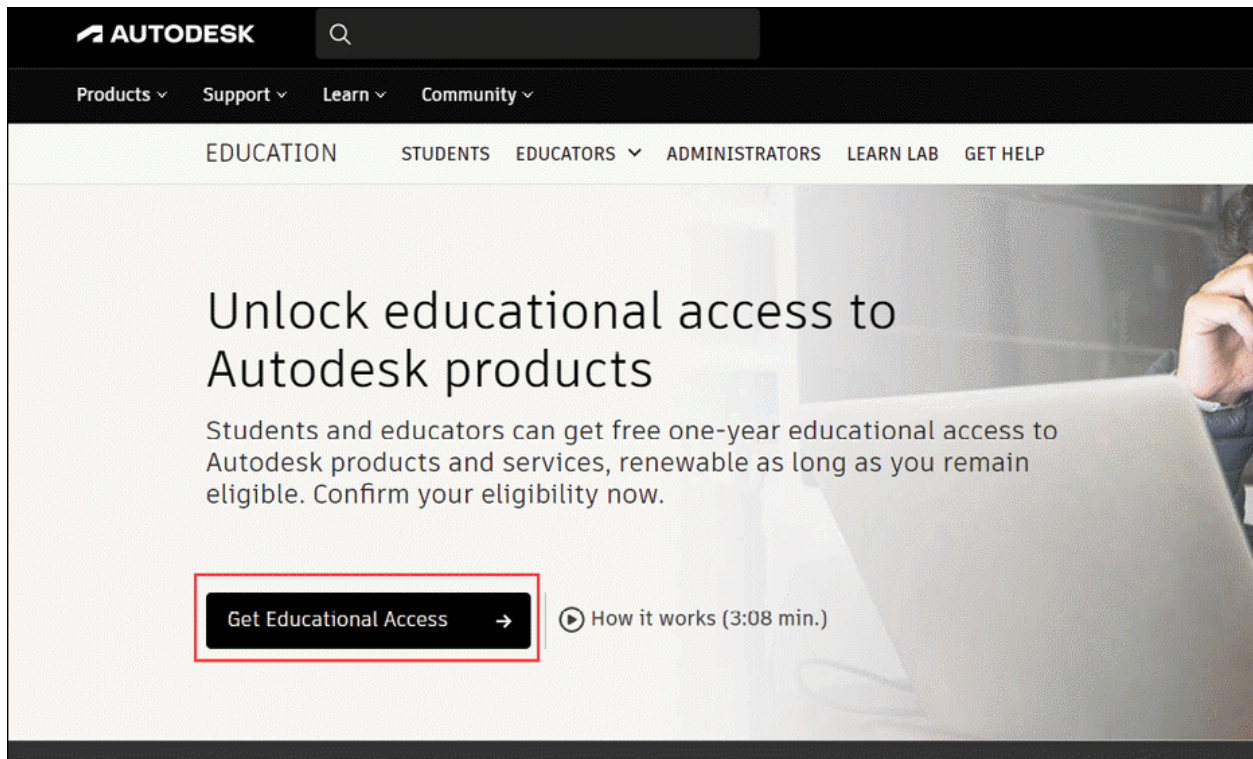


Figure-5. Get Educational Access button

- Click on the **Get product** button below the software name. The **Access** button will become active; refer to [Figure-6](#). Click on this button to download the **Autodesk Fusion 360 software**.

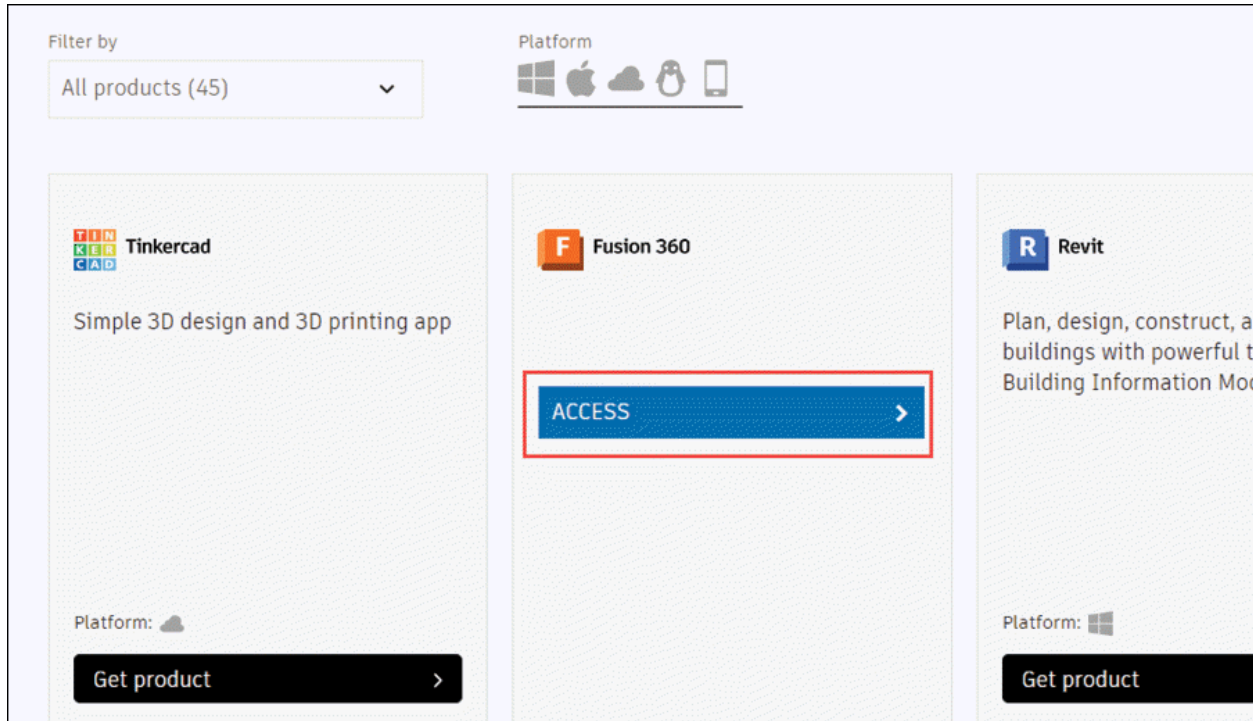


Figure-6. Access button

- Open the download setup file and follow the instructions as per the setup instruction.
- The software will be installed in a couple of minutes.

STARTING AUTODESK FUSION 360

- To start **Autodesk Fusion 360** from **Start** menu, click on the **Start** button in the **Taskbar** at the bottom left corner and then select **Autodesk Fusion 360** option from the **Autodesk** folder. Select the **Autodesk Fusion 360** icon; refer to [Figure-7](#).

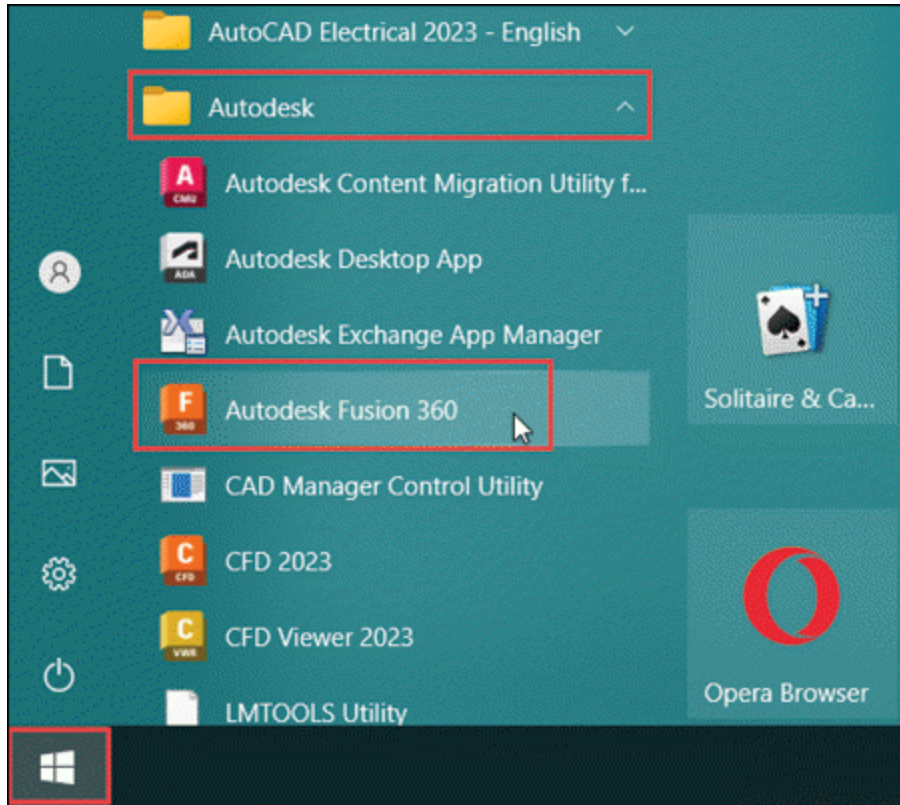


Figure-7. Start menu

- While installing the software, if you have selected the check box to create a desktop icon then you can double-click on that icon to run the software.
- If you have not selected the check box to create the desktop icon and want to create the icon on desktop now in Windows 10 or later, then drag the **Autodesk Fusion 360** icon from **Start** menu to Desktop.

After clicking on the icon, the Autodesk Fusion 360 software window will be displayed; refer to [Figure-8](#).

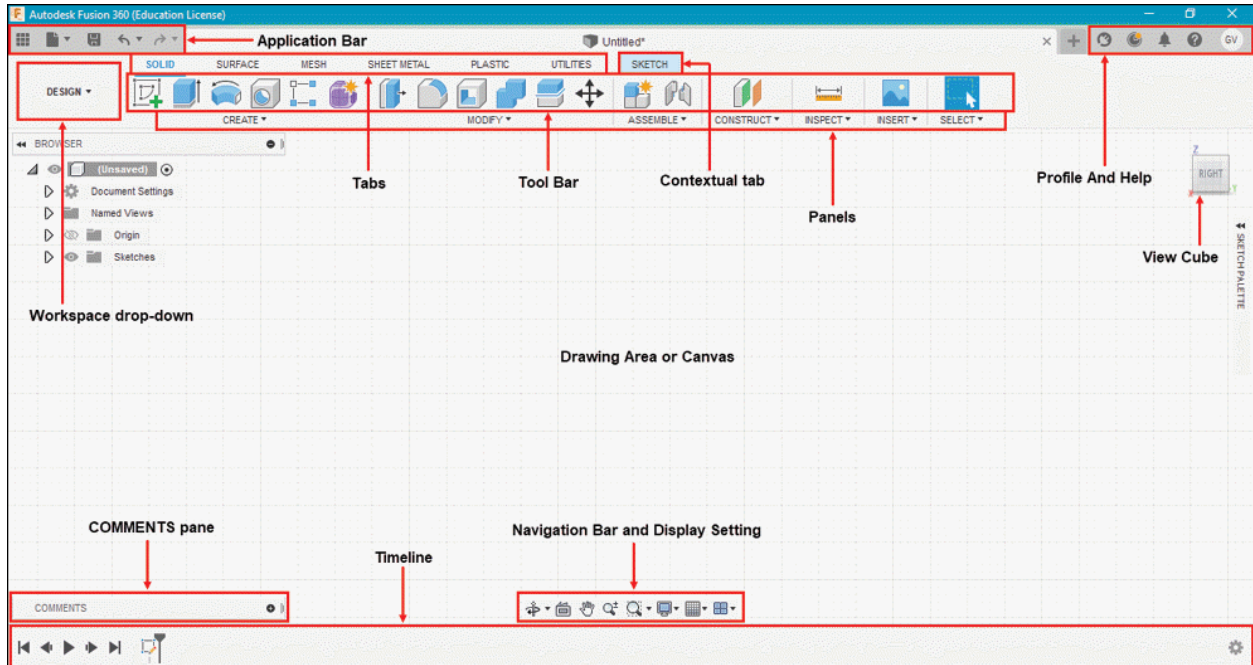


Figure-8. Autodesk Fusion 360 Application Window

STARTING A NEW DOCUMENT

- Click on the **File** drop-down and select the **New Design** tool as shown in [Figure-9](#). A new document will open.

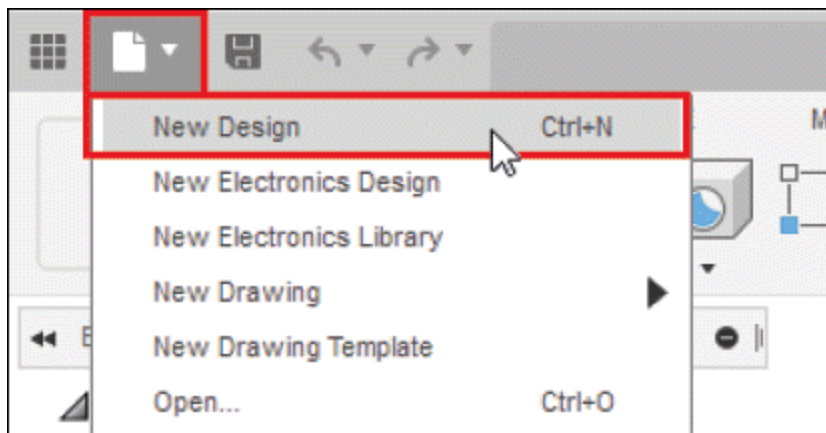


Figure-9. File menu

- Select desired workspace from the **Workspace** drop-down; refer to [Figure-10](#).

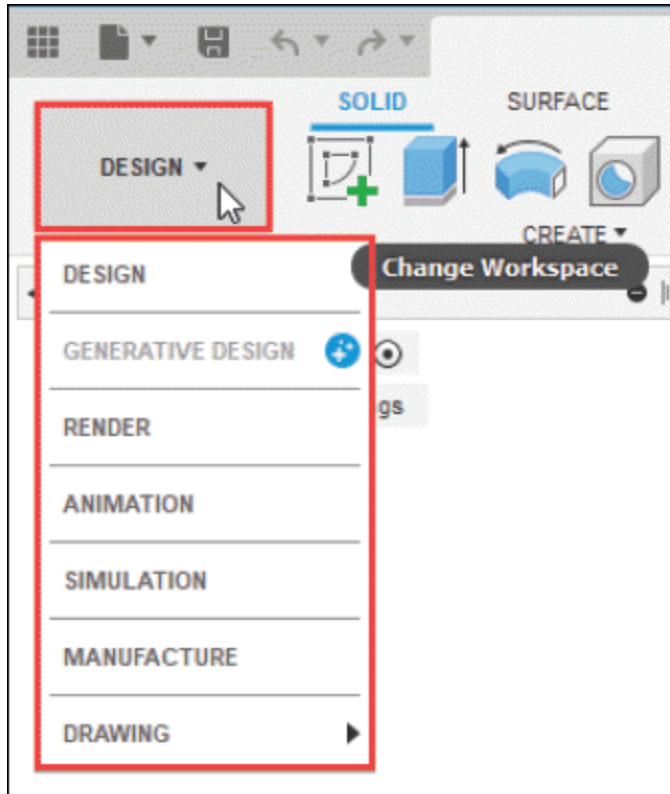


Figure-10. Workspace drop down

- There are six buttons available in **Workspace** drop-down in Educational version; **DESIGN**, **RENDER**, **ANIMATION**, **SIMULATION**, **MANUFACTURE**, and **DRAWING**.
- The **DESIGN** button is used to create solid, surface, and sheet metal designs.
- The **RENDER** button is used to activate workspace for rendering realistic model for presentation.
- The **ANIMATION** button is used to activate workspace for creating automatic or manual exploded views as well as direct control over unique animation of parts and assemblies.

- The **SIMULATION** button is used to perform Engineering Analyses.
- The **MANUFACTURE** button is used to generate G-codes for manufacturing processes like turning, milling, drilling, cutting, probing, and so on.
- The **DRAWING** button is used for generating drawings from model and animation.

If you are working on Professional version of software then another workspace **GENERATIVE DESIGN** is also available using which you can generate different iterations of designs based on specified limits and goals.

You will learn more about these workspaces later in this book.

FILE MENU

The options in the **File** menu are used to manage files and related parameters. Various tools of **File** menu are discussed next.

Creating New Drawing

The tools in **New Drawing** cascading menu are used to initialize a new drawing from animation or design; refer to [Figure-11](#). The methods to use these tools will be discussed later in the book.

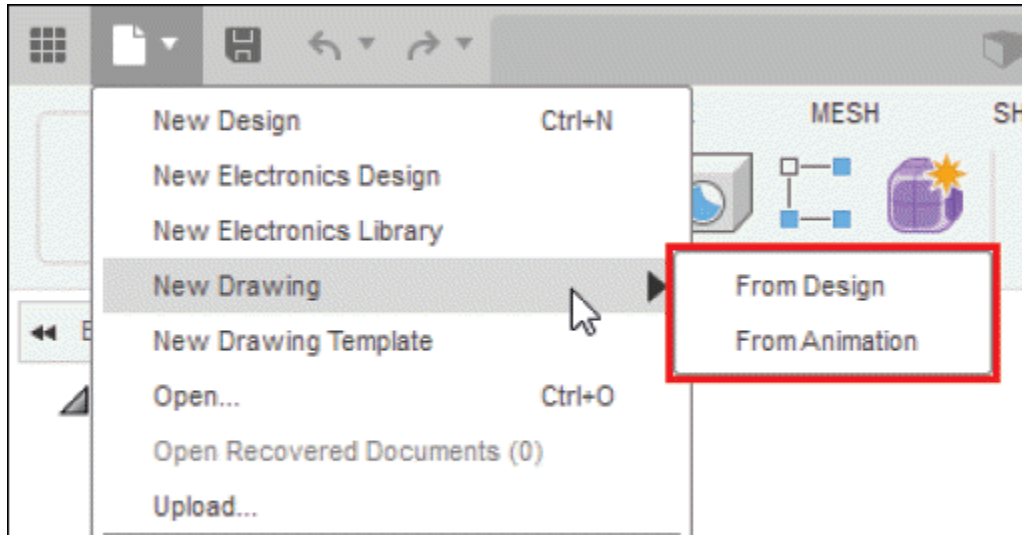


Figure-11. New Drawing cascading menu

Creating New Drawing Template

The **New Drawing Template** tool is used to create a template file for drafting. This template can either be created from scratch or you can use other drawing file as reference for template. This template can later be used for creating engineering drawings of the model. You will learn more about this tool later in the book.

Opening File

The **Open** tool is used to open files earlier saved in cloud or local drive. You can also use this tool to import supported files of other software. The procedure to use this tool is given next.

- Click on the **Open** tool from the **File** menu or press **CTRL+O** from keyboard. The **Open** dialog box will be displayed; refer to [Figure-12](#). Note that we are working in online mode of Autodesk Fusion.

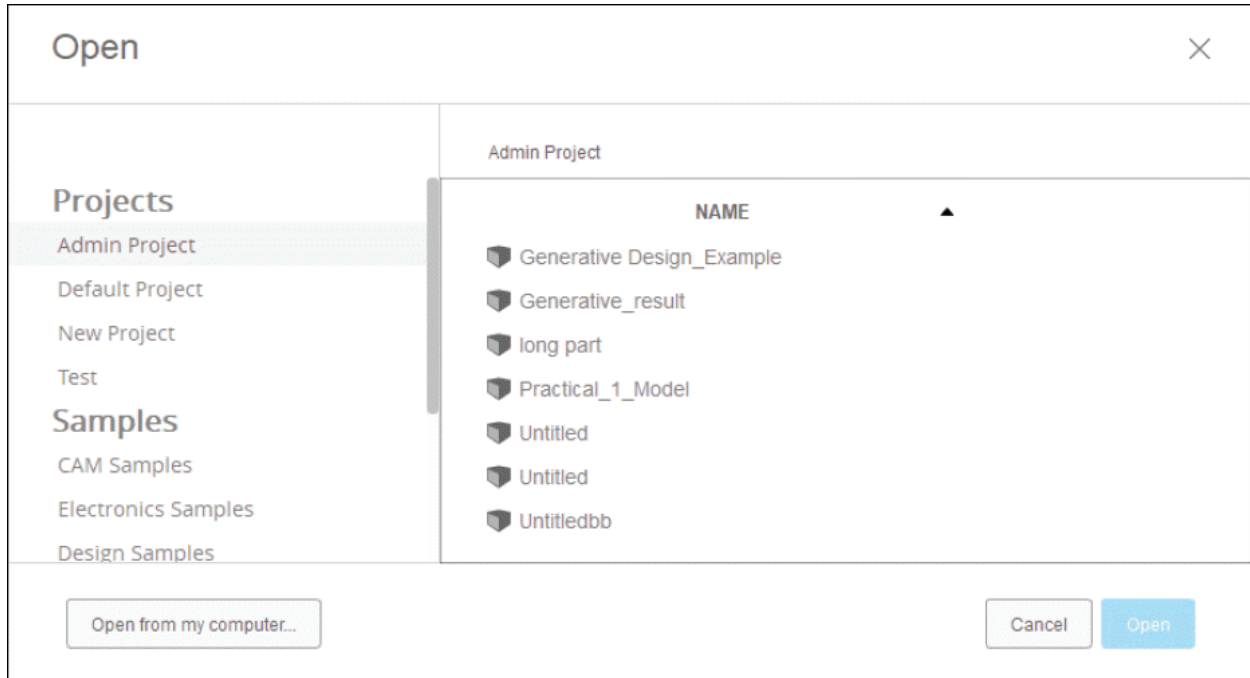


Figure-12. Open dialog box

- By default, files saved on cloud are displayed in the dialog box. Select desired file and click on the **Open** button. The file will open in Autodesk Fusion.
- If you want to open a file stored in local drive then click on the **Open from my computer** button. The **Open** dialog box will be displayed; refer to [Figure-13](#).

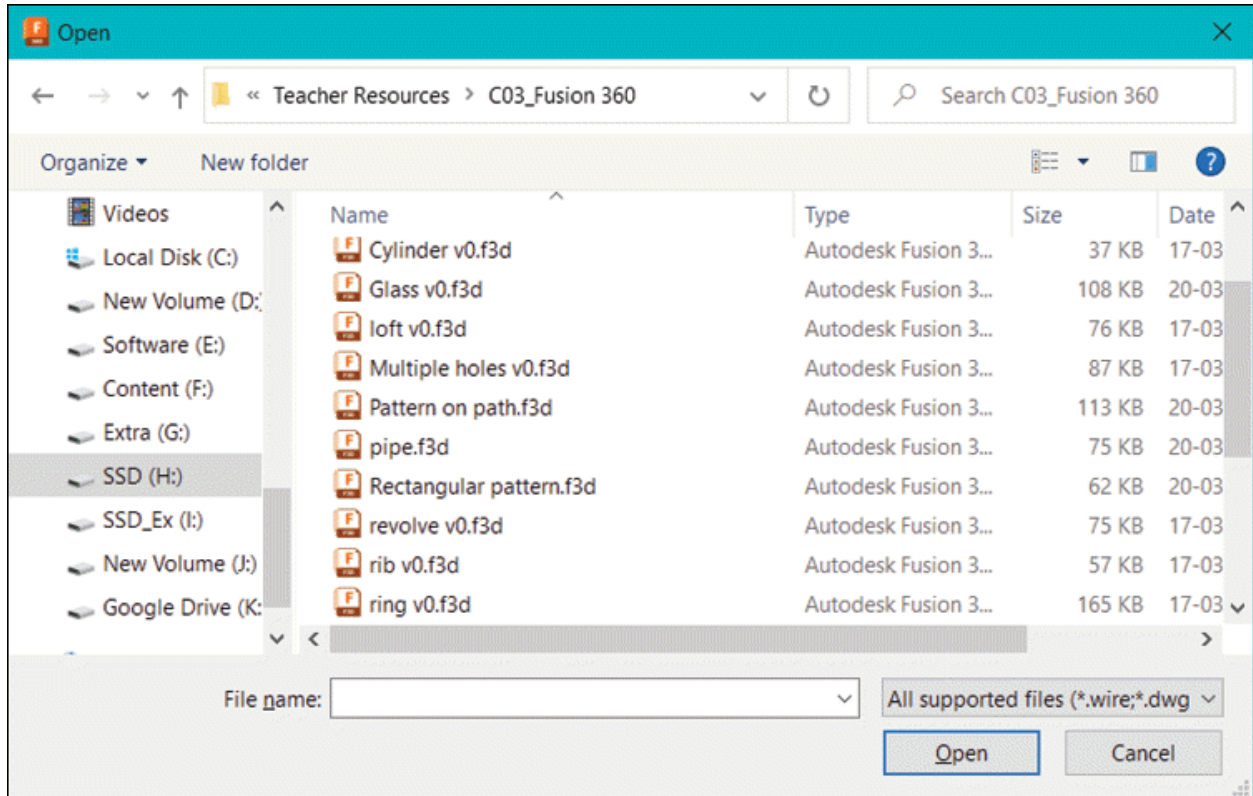


Figure-13. Open dialog box

- Select desired file and click on the **Open** button. The file will be displayed in the application window. If you have selected the file of different format from native format then the **Job Status** dialog box will be displayed notifying you the status of file import; refer to [Figure-14](#).

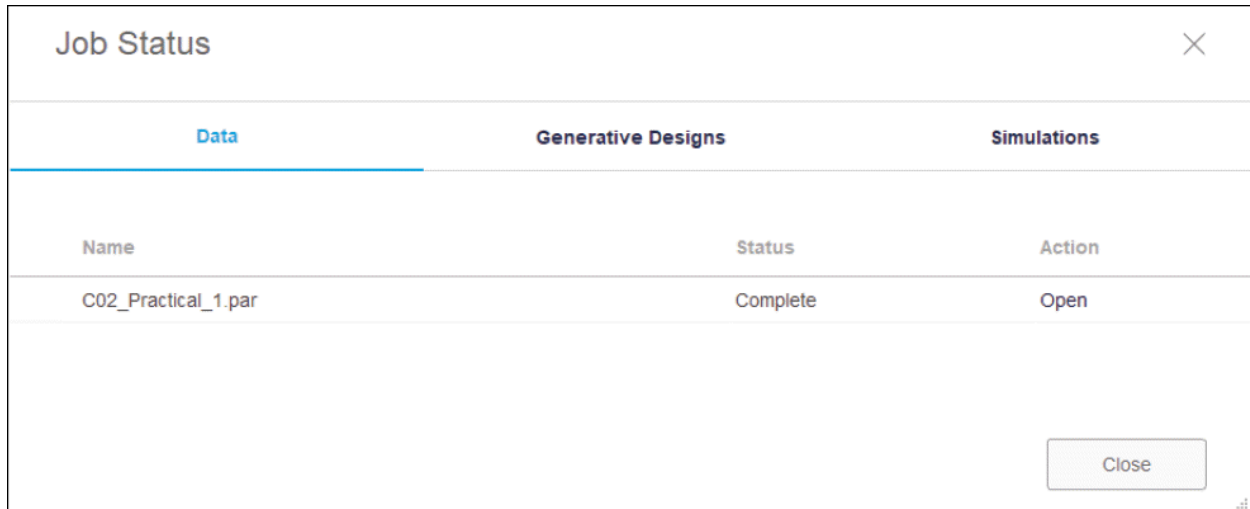


Figure-14. Job Status dialog box

- Once the status of import is complete then click on the **Open** button from the **Action** column in the dialog box. The model will be displayed; refer to [Figure-15](#) (Creo Parametric file imported in Fusion 360).

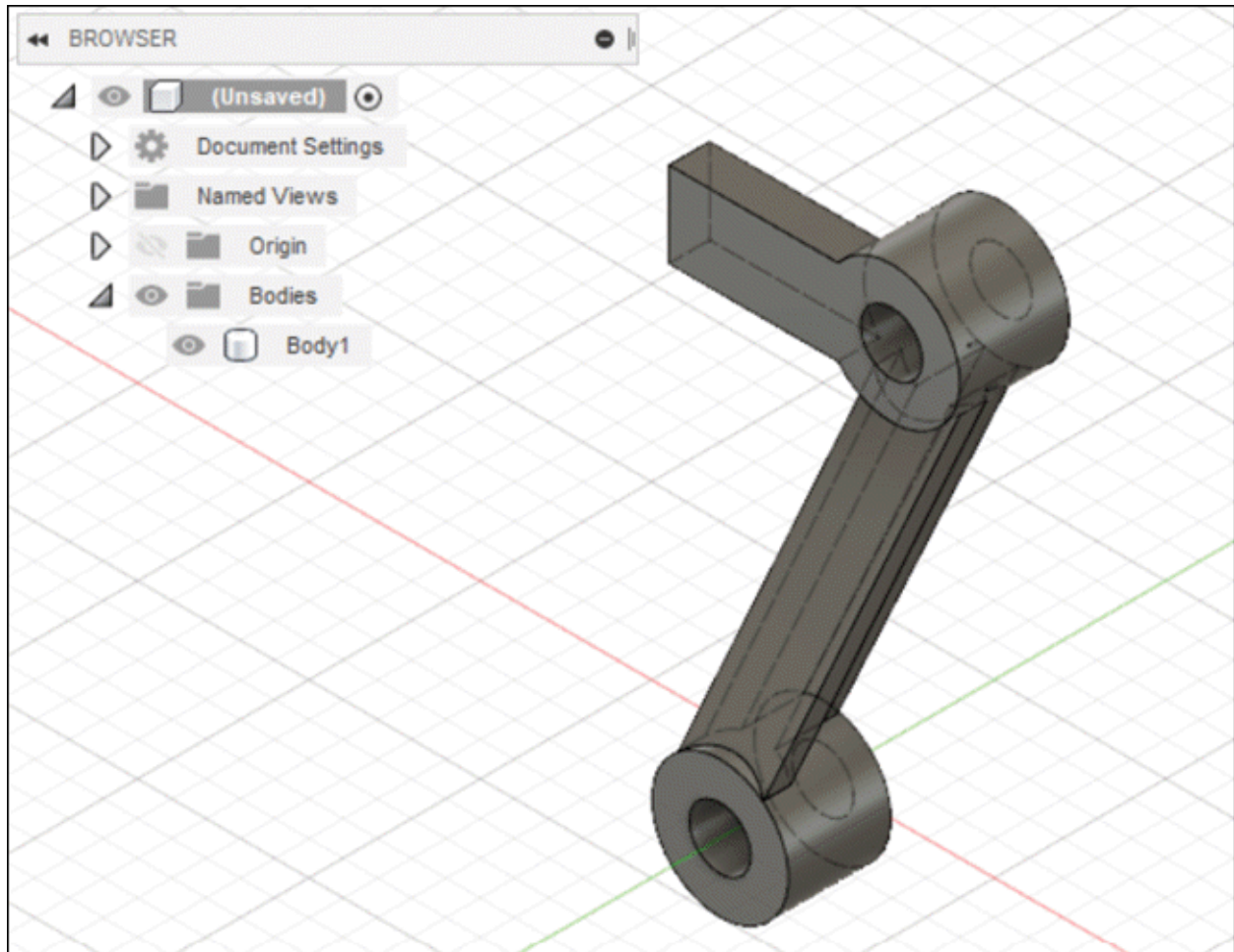


Figure-15. Model imported from Creo Parametric file

Recovering Documents

The **Recover Documents** tool is used to recover unsaved file versions which are created when software closes unexpectedly; refer to [Figure-16](#). Note that there is an auto save feature in Autodesk Fusion which saves versions of file at a specific time intervals. If software has created the auto save version of your file then only you will be able to recover the file. By default, this time interval is 5 minutes and it can be changed in **Preferences** dialog box which will be

discussed later. The procedure to use this tool is given next.

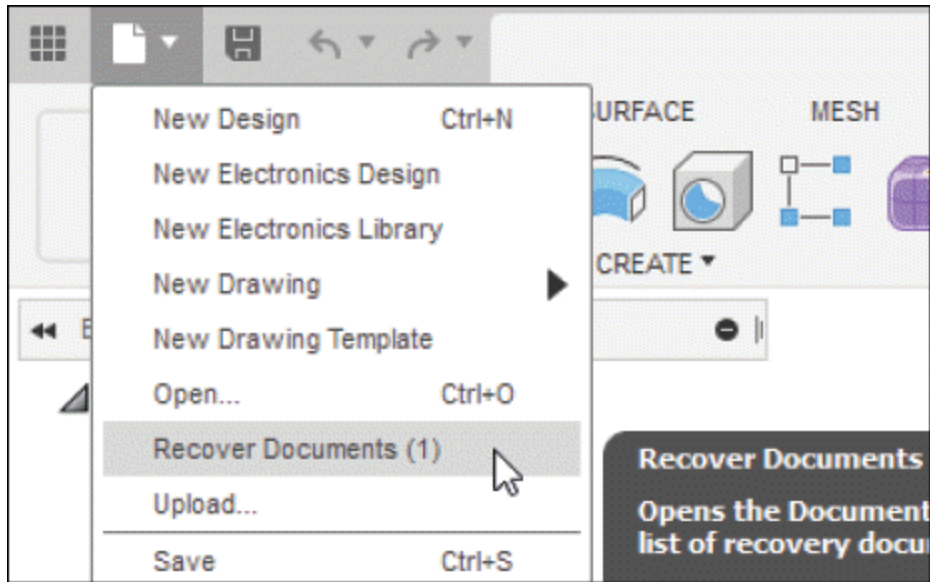


Figure-16. Recover Documents tool

- Click on the **Recover Documents** tool from the **File** menu. The **File Recovery** dialog box will be displayed where you need to select auto save version of your file; refer to [Figure-17](#).

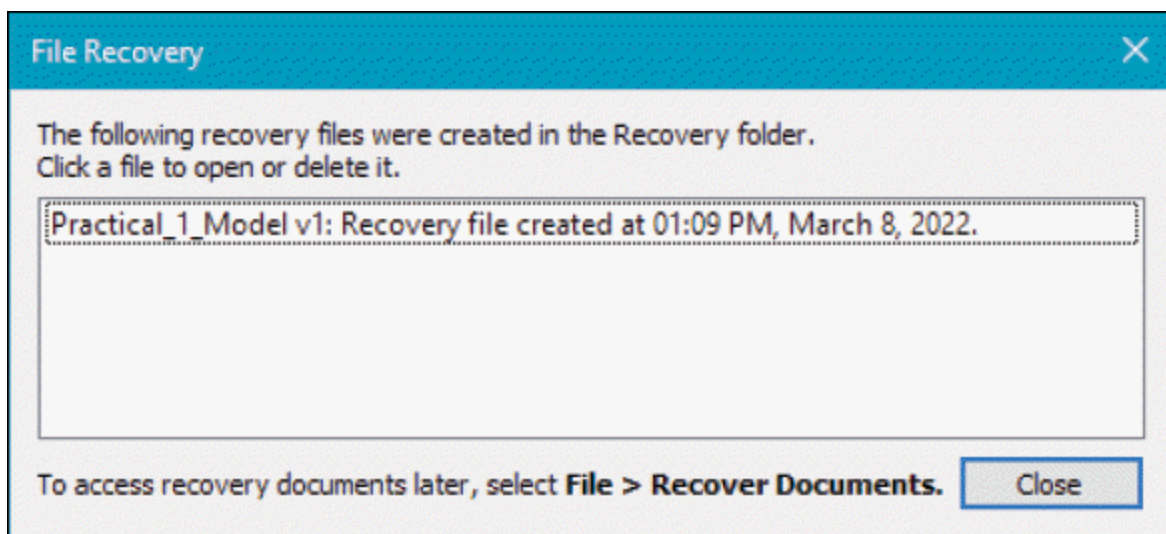


Figure-17. File Recovery dialog box

- Click on the file version that you want to be recovered and click on the **Open** button from the displayed menu. The file version will open in **Autodesk Fusion 360**. If the opened version is as desired then save it otherwise close it and open the other version from the **File Recovery** dialog box.

Upload

The **Upload** tool is used to upload files on cloud. The procedure to use this tool is discussed next.

- Click on **Upload** button from the **File** menu. The **Upload** dialog box will be displayed; refer to [Figure-18](#). Click on the **Select Files** button or Drag & Drop the files to be uploaded in the **Drag and Drop Here** area of the dialog box.
- If you want to change the location where files will be uploaded then click on the **Change Location** button and select desired location from the **Change Location** dialog box; refer to [Figure-19](#).

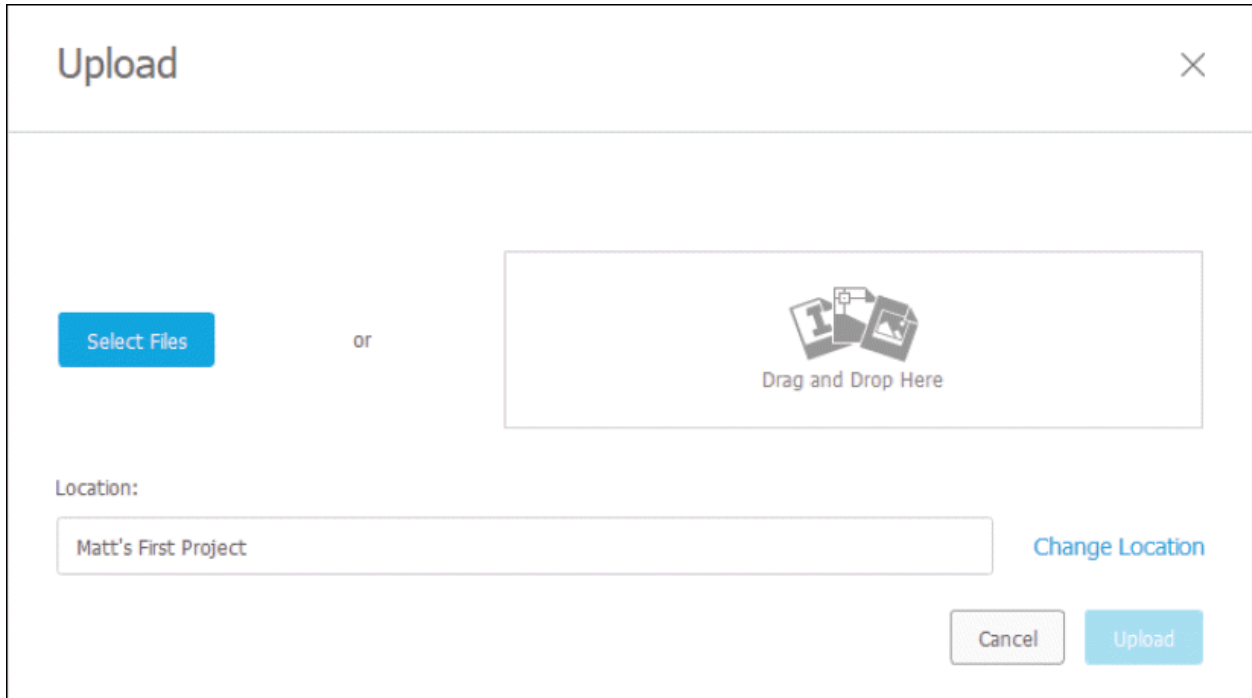


Figure-18. Upload dialog box

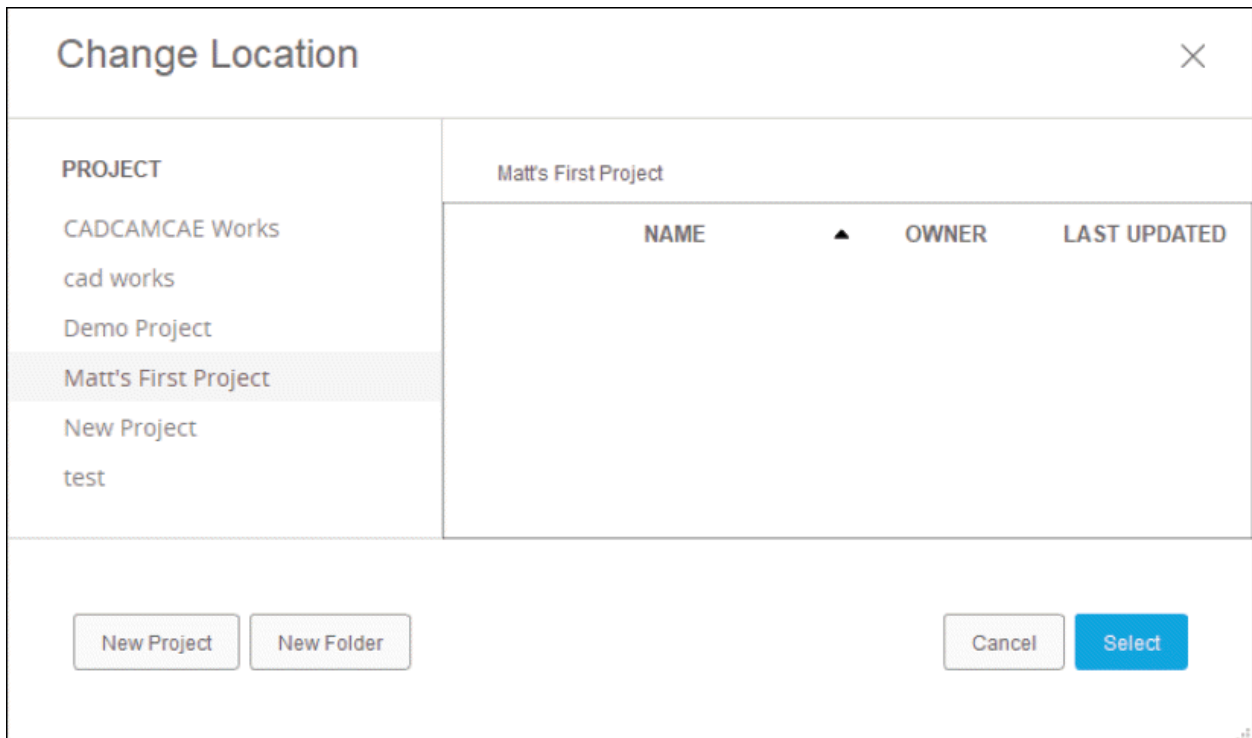


Figure-19. Change Location dialog box

- After selecting desired location, click on the **Select** button.

You will return to **Upload** dialog box with updated location. Note that you can also create new folders and new locations by using the options in **Change Location** dialog box which will be discussed later in this book.

- Click on the **Select Files** button to upload files and select multiple files while holding the **CTRL** key from the **Open** dialog box displayed; refer to [Figure-20](#).

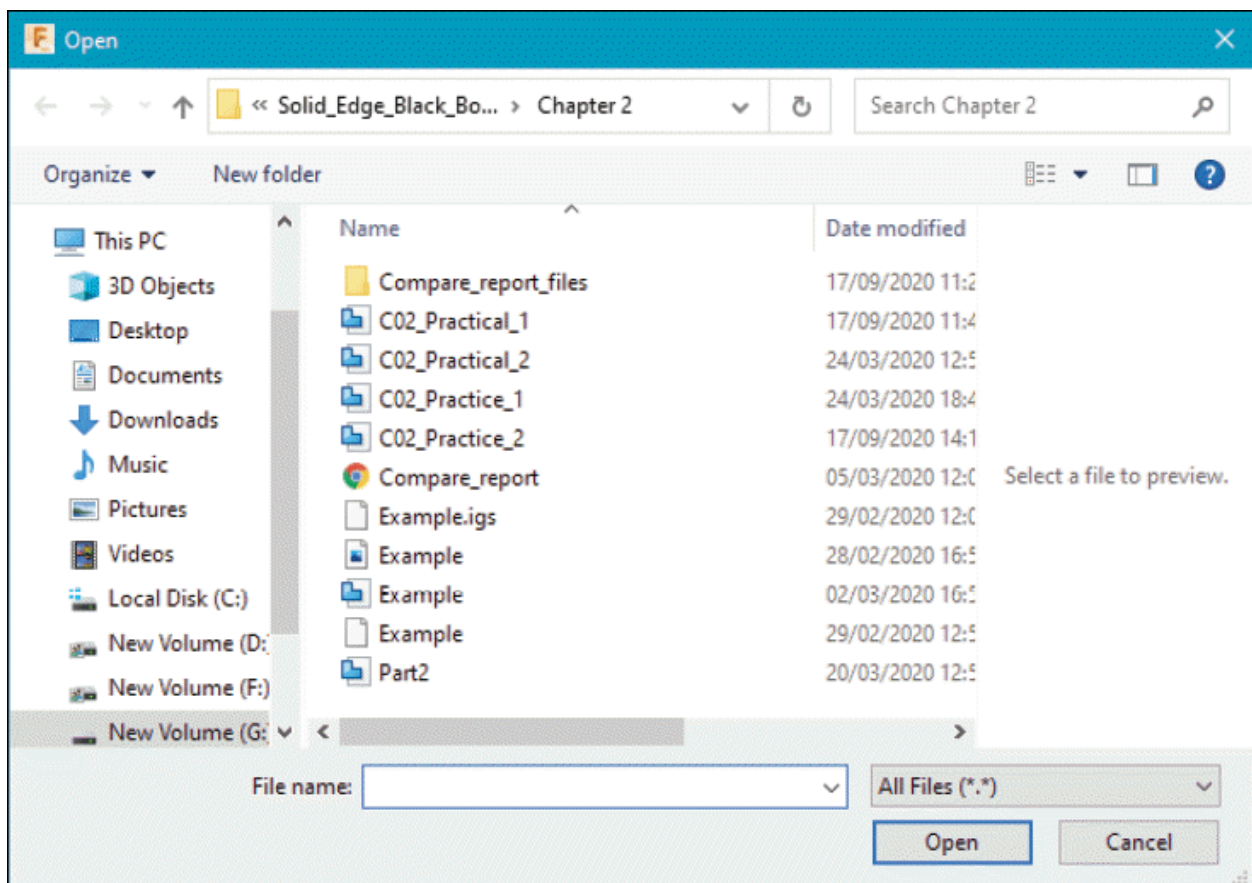


Figure-20. Open dialog box for file upload

- Click on the **Open** button from the dialog box to upload files. Or, drag the files in the dialog box and drop them. The options in the **Upload** dialog box will be updated; refer to [Figure-21](#).

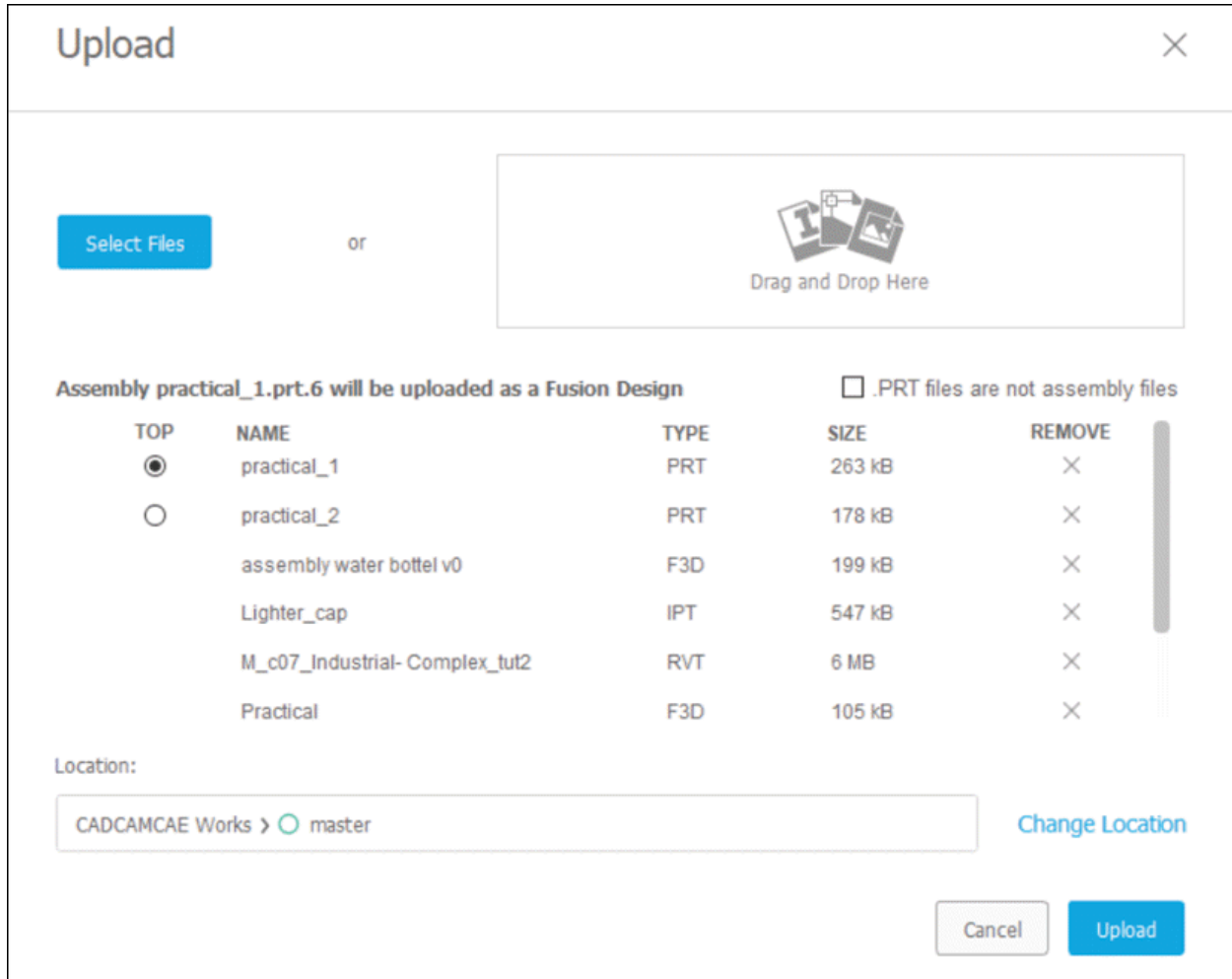


Figure-21. Updated Upload dialog box

- If you have selected a **.PRT** file of NX or Creo then select the **.PRT files are not assembly files** check box to explicitly tell software that these PRT files are not connected to any assembly.
- After specifying desired parameters, click on **Upload** button from **Upload** dialog box. Files will be uploaded and status will be displayed in **Job Status** dialog box.

Save

The **Save** tool is used to save the current file on cloud. You

can press **CTRL+S** key from keyboard to save file. The procedure to save file is given next.

- Click on the **Save** tool from **File** menu or select the **Save** button from **Application bar** as shown in [Figure-22](#). The **Save** dialog box will be displayed asking you to specify the **Name** and **Location** of your file; refer to [Figure-23](#).

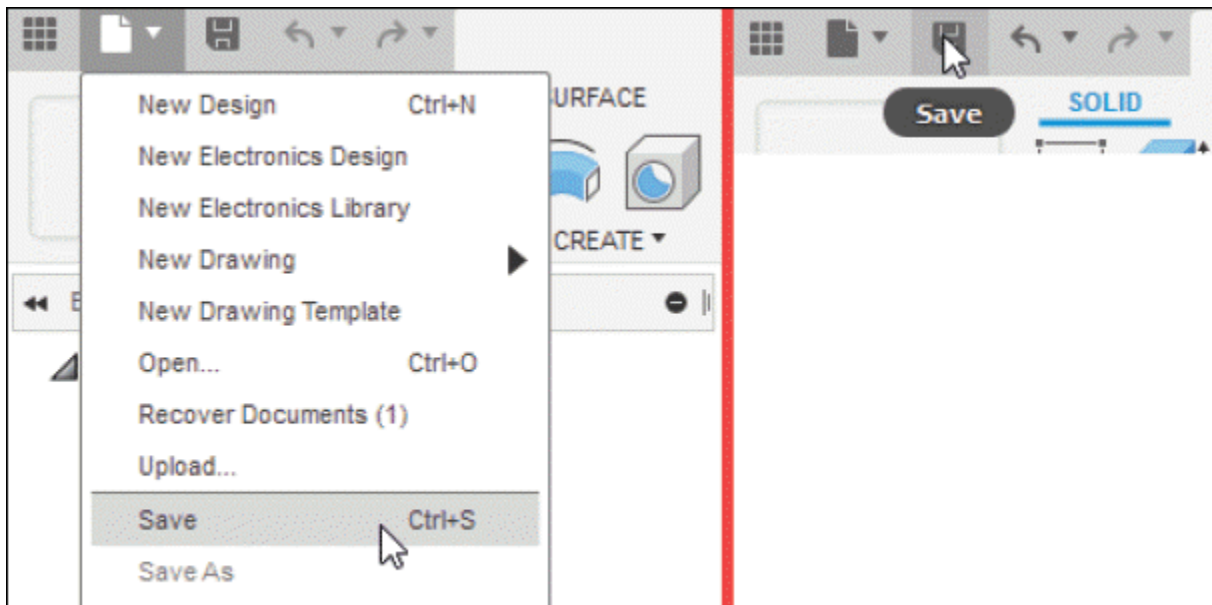


Figure-22. Save tool

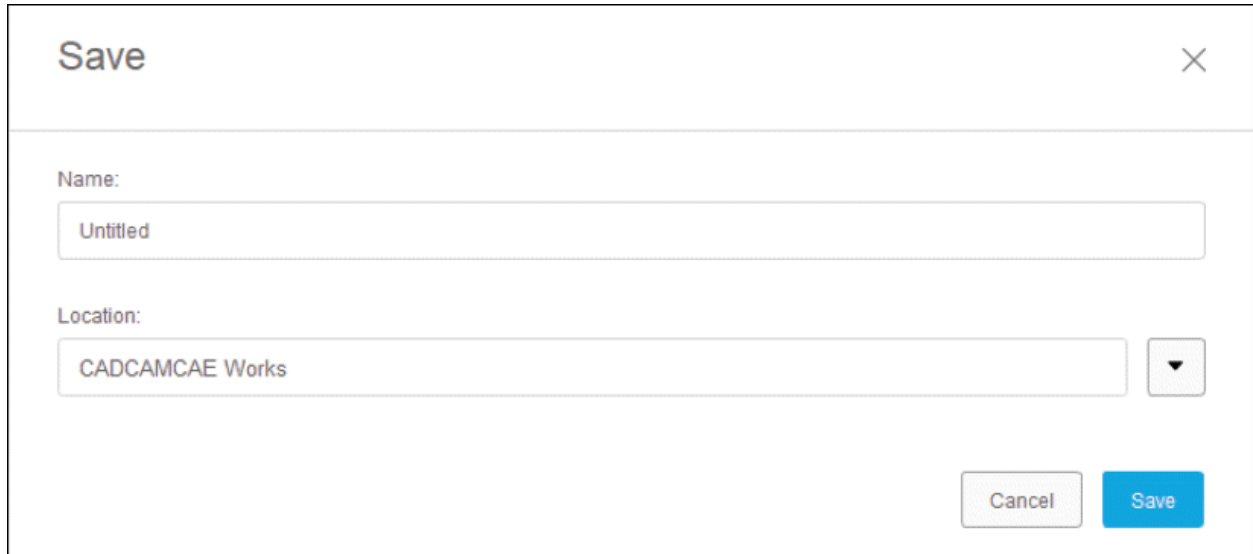


Figure-23. Save dialog box

- Specify desired name in the **Name** edit box.
- Click on the down arrow next to **Location** edit box in the dialog box if you want to select a different location on cloud. A list of locations created for/by you in Autodesk Cloud will be displayed; refer to [Figure-24](#).

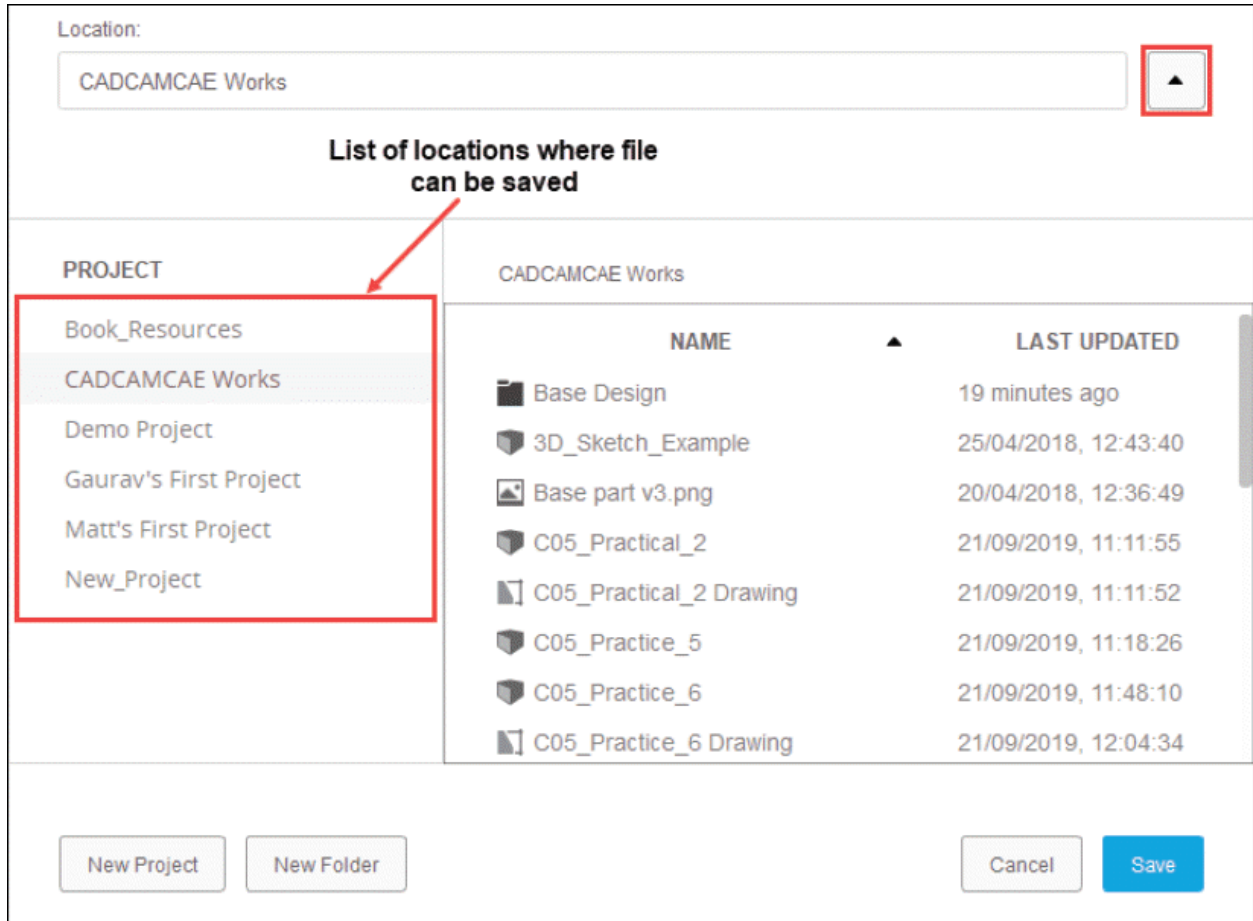


Figure-24. Locations to save files

- By default, Demo Project is added in the Project list. To add a new project, click on the **New Project** button at the bottom left corner of the dialog box. You will be asked to specify name of the new project; refer to [Figure-25](#).

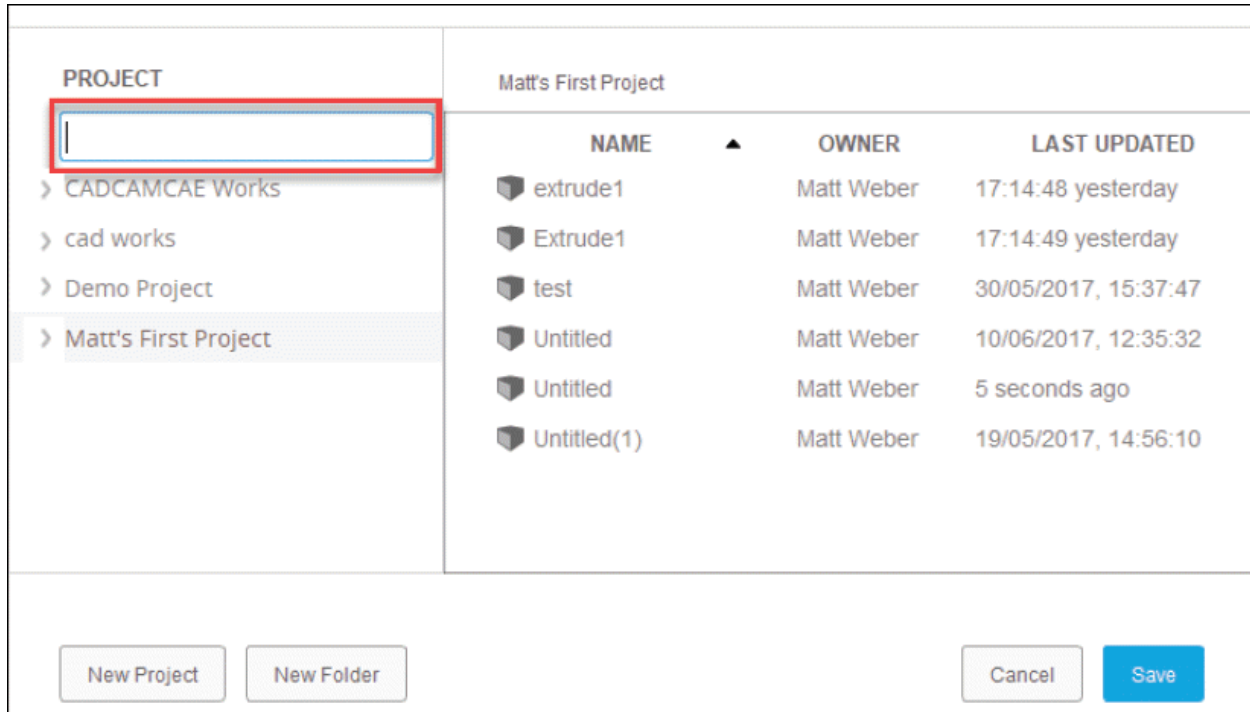


Figure-25. Specifying project name

- Specify desired name for the project in displayed edit box and click anywhere in blank area of the dialog box. The new location will be created. Select this new location to save the file; refer to [Figure-26](#).

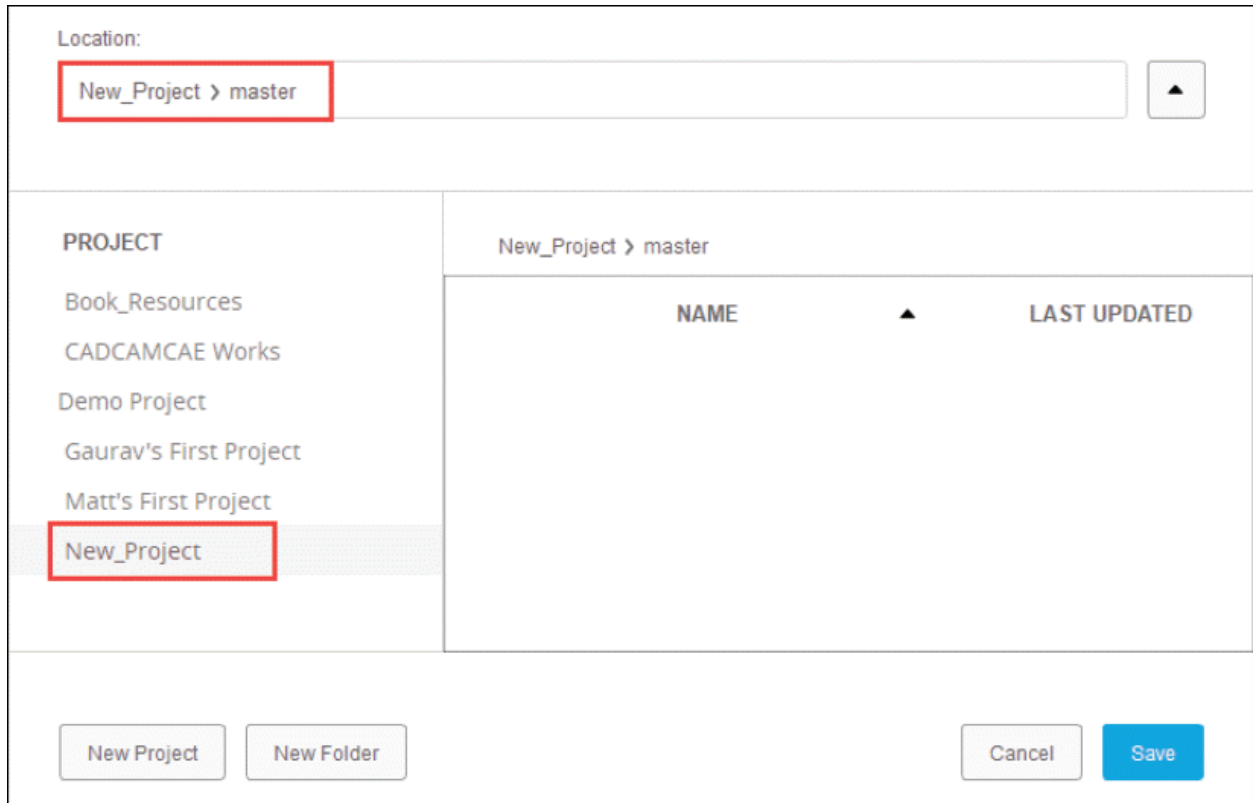


Figure-26. Location selected for saving file

- If you want to create folder in the project with desired name then click on the **New Folder** button from the dialog box. You will be asked to specify the name of the folder. Specify the name of folder and click in blank area of the dialog box.
- To save your file in the folder, double-click on folder name to enter the folder and then click on the **Save** button. The file will be saved at specified location.

Note that when next time you will use **Save** tool after making changes then the **Save** dialog box will be displayed asking you to specify description for this new version of same file; refer to [Figure-27](#). Type a short description about

what has changed in this version. Select the **Milestone** check box, type desired milestone name in the edit box, and click on the **OK** button. You will notice that file name has changed from xxxxx v0 to xxxxx v1 which describes that this is the version 1 of same file. These versions can be accessed by **Data Panel** which will be discussed later.

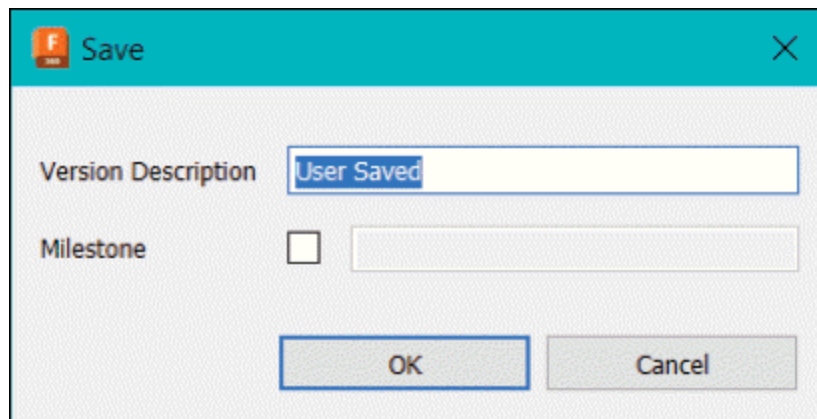


Figure-27. Save dialog box

Save As

Using the **Save As** tool, you can save the file with different name at different location. The procedure to use this tool is discussed next.

- Click on **Save As** tool from **File** menu; refer to [Figure-28](#). The **Save As** dialog box will be displayed as shown in [Figure-29](#).

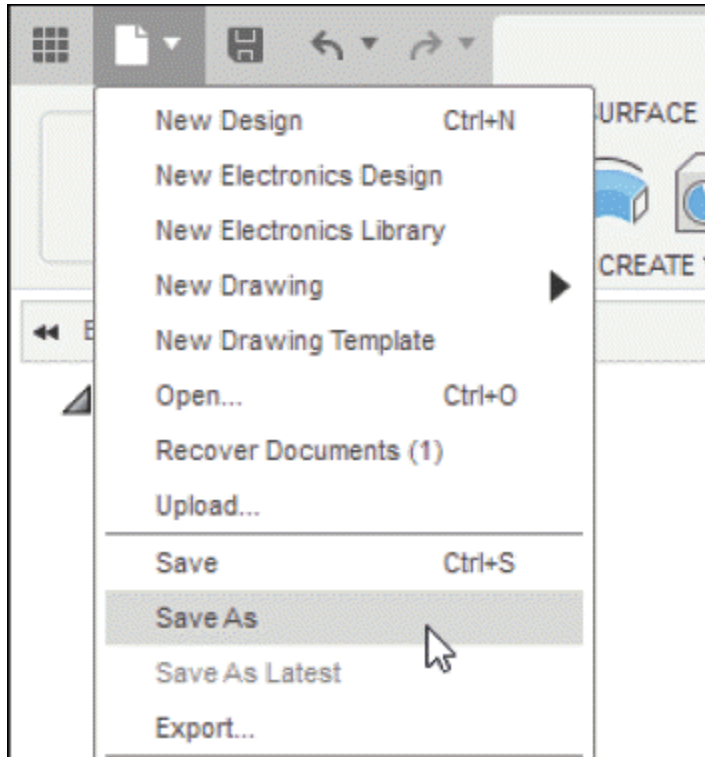


Figure-28. Save As tool

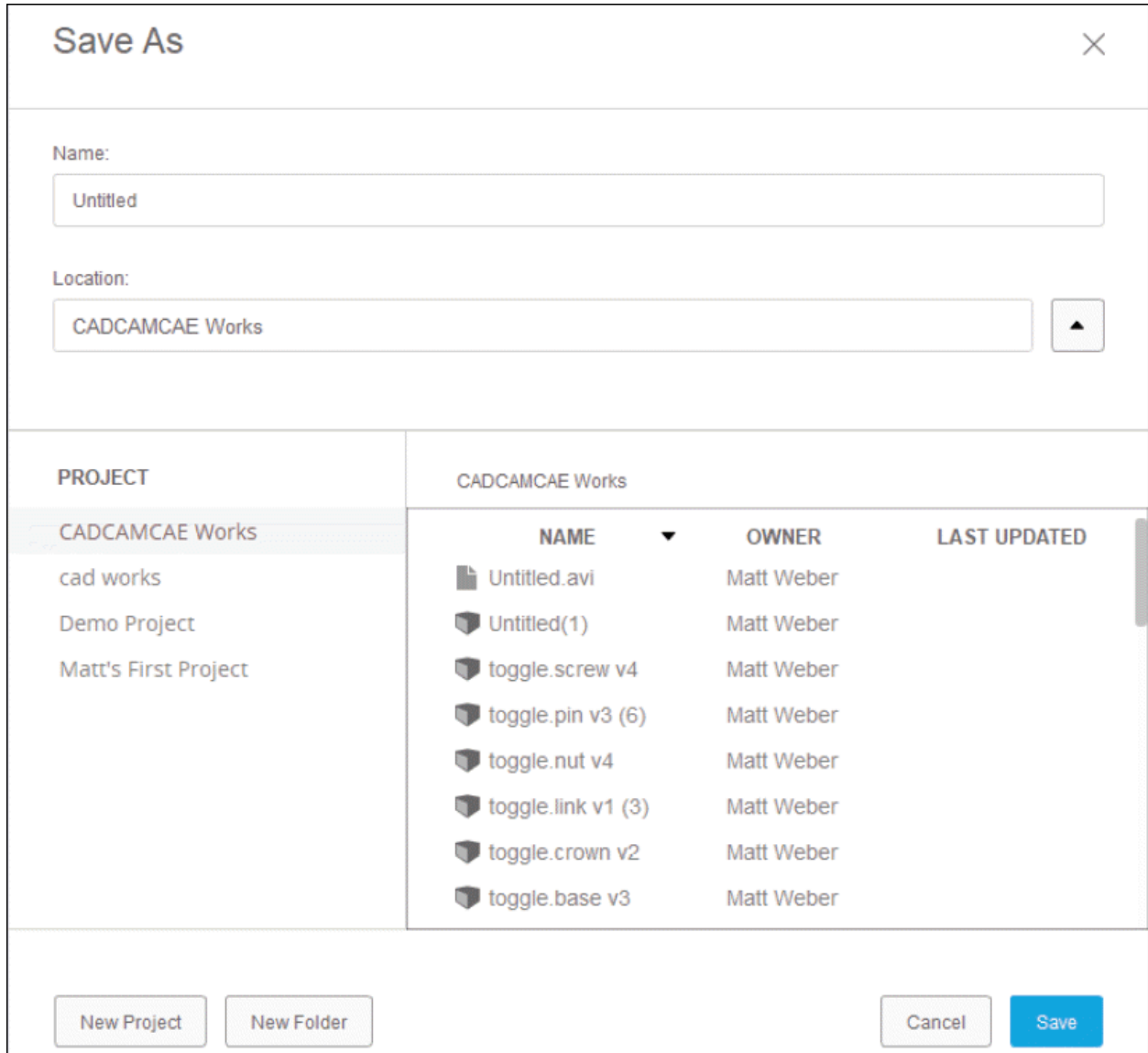


Figure-29. Save As dialog box

- Specify desired name and location for the file, and click on the **Save** button. The options in this dialog box are same as discussed in **Save** dialog box.

Save As Latest

When you are collaborating on a model with your team and after creating many versions of the model, you find that one of the previous versions is to be considered as final model

for use. In such case, open the previous version file of Autodesk Fusion using **Open** tool and the **Save As Latest** tool will be active in **File** menu to save this version as latest; refer to [Figure-30](#). On clicking the **Save As Latest** tool, an information box will be displayed showing how versions will be renumbered by making current version latest. Click on the **Continue** button from the information box. The **Save As Latest** dialog box will be displayed; refer to [Figure-31](#). Specify desired description text about this version in **Version Description** edit box. Select the **Milestone** check box if you want to mark it as achievement of one of your design goals and click on the **OK** button from dialog box. The file will be saved on cloud.

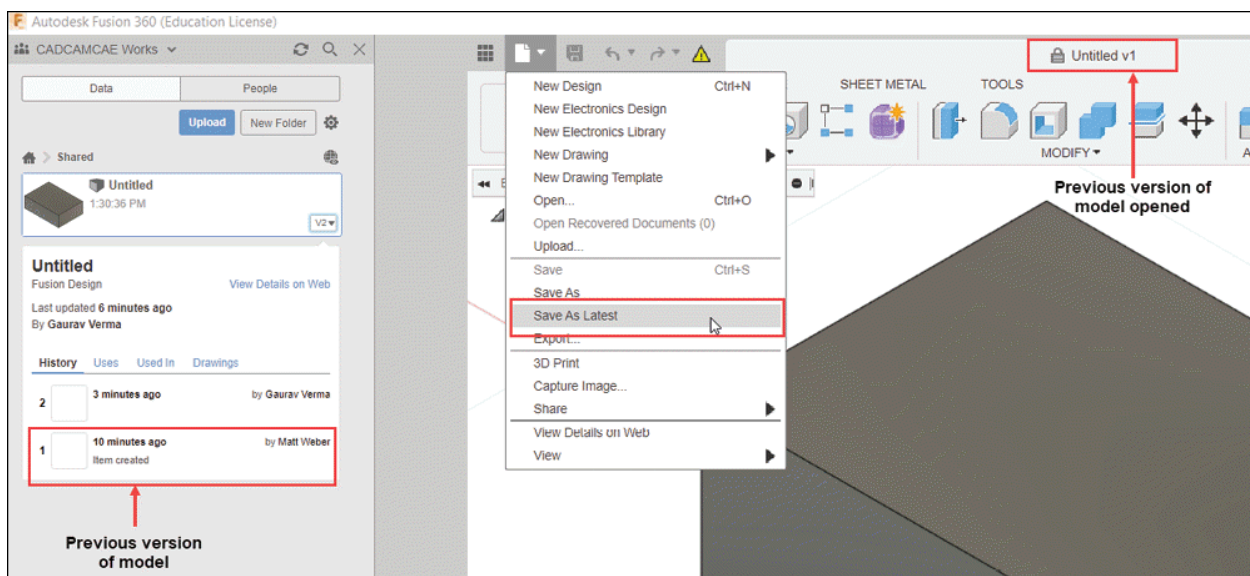


Figure-30. Save As Latest tool

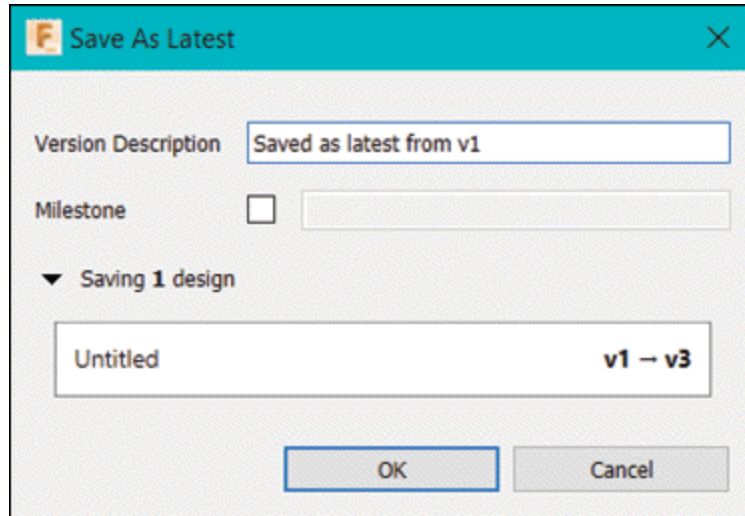


Figure-31. Save As Latest dialog box

Export

The **Export** tool is used to export the file in different formats. The procedure to use this tool is given next.

- Click on the **Export** tool from the **File** menu; refer to [Figure-32](#). The **Export** dialog box will be displayed; refer to [Figure-33](#).

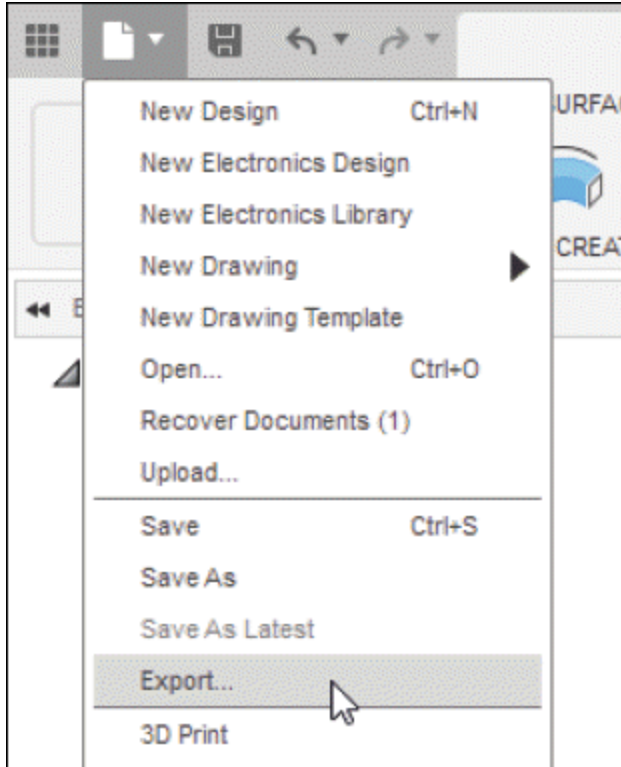


Figure-32. Export tool



Figure-33. Export dialog box

- In the **Type** drop-down, there are various options to specify

export format; **F3D, IGES, SAT, SMT, STEP, IPT, DWG, DXF**, and so on. Select desired format from the drop-down. Note that for some file formats, a cloud conversion will be required and take some time. You will be able to see progress of conversion in **Job Status** dialog box.

- Specify desired name and location for the file. Note that you can save the files only in the local drive by using this dialog box.
- Click on the **Export** button from the dialog box to save the file.

3D Print

The **3D Print** tool is used to prepare and send current model for 3D printing. This tool convert the selected body to mesh body and sends the output to 3D-Print Utility or converts it to **STL** format. The procedure to use this tool is given next.

- Click on the **3D Print** tool from the **File** menu or click on the **3D Print** tool from the **Make** panel in the **UTILITIES** tab of the **Toolbar**; refer to [Figure-34](#). The **3D PRINT** dialog box will be displayed; refer to [Figure-35](#).

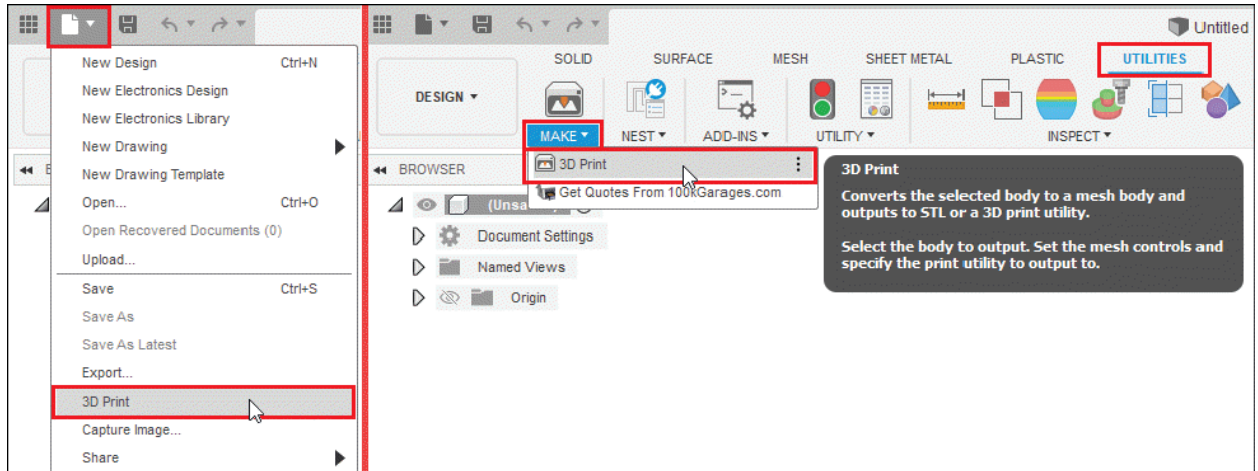


Figure-34. 3D Print tool

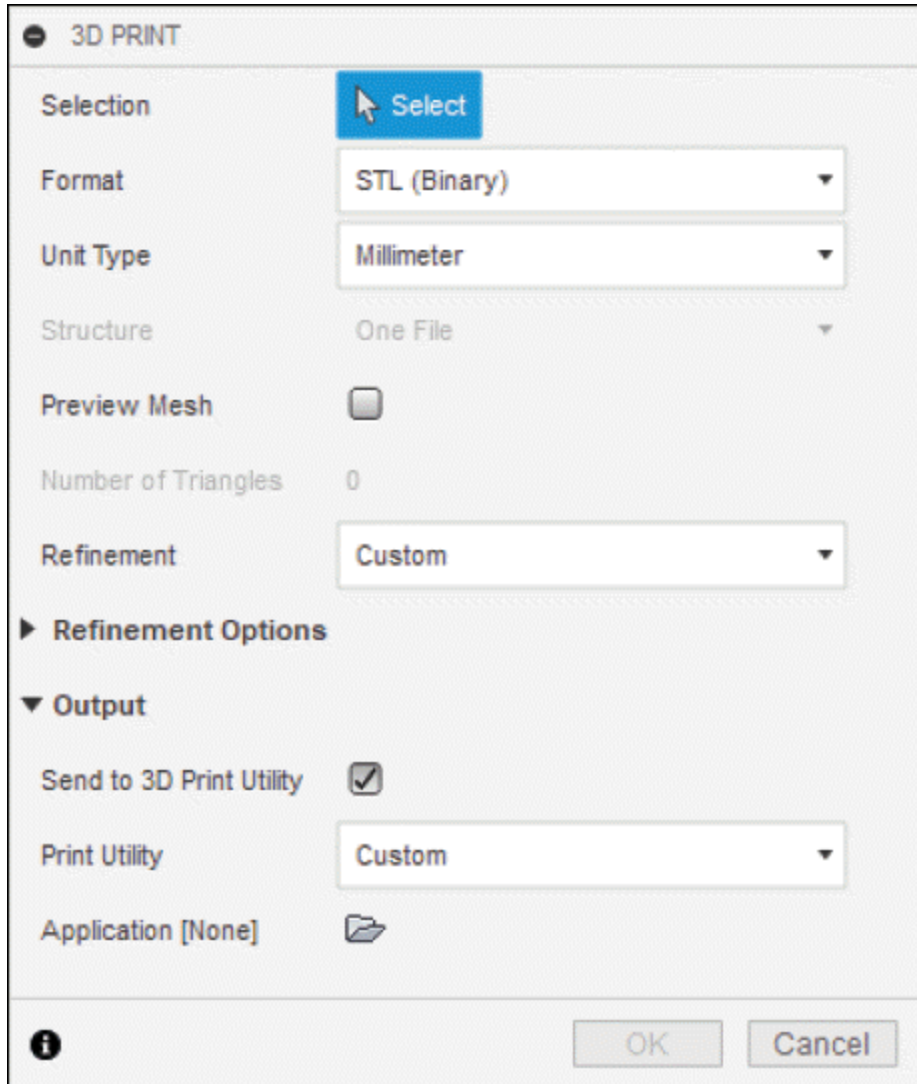


Figure-35. 3D Print dialog box

- By default, the **Selection** button is active in this dialog box and you are prompted to select model to be 3D printed. Click on the model to be 3D printed.
- Select desired format of file in which you want to output the mesh model from **Format** drop-down in the dialog box.
- Select desired unit to be used in 3D printing from **Unit Type** drop-down.

- Now, select the **Preview Mesh** check box to see the number of triangles forming in the selected body when it converts to mesh.

If you don't know why it is previewed in mesh then you need to know full name of standard file format used for 3D printing. STL format is used as standard for 3D printing which abbreviates **Standard Triangle Language**. So, your model will be broken into very small triangles and then these triangles will be arranged in the shape of your model in layers.

- Select desired level of resolution for printing in **Refinement** drop-down menu. There are four resolution in this drop-down. If you select the **High**, **Low**, and **Medium** command then the **Surface deviation**, **Normal deviation**, **Maximum Edge Length**, and **Aspect Ratio** parameters in the **Refinement Options** section will be adjusted automatically. (Click on the arrow before **Refinement Options** node to expand the section.) If you want to adjust these parameters manually then you need to select the **Custom** option in **Refinement** drop-down; refer to [Figure-36](#). Note that your 3D printing machine should be able to print the model at specified resolution. A higher resolution will take more time and power in 3D Printer. Unnecessarily high refinement should be avoided in 3D printing.

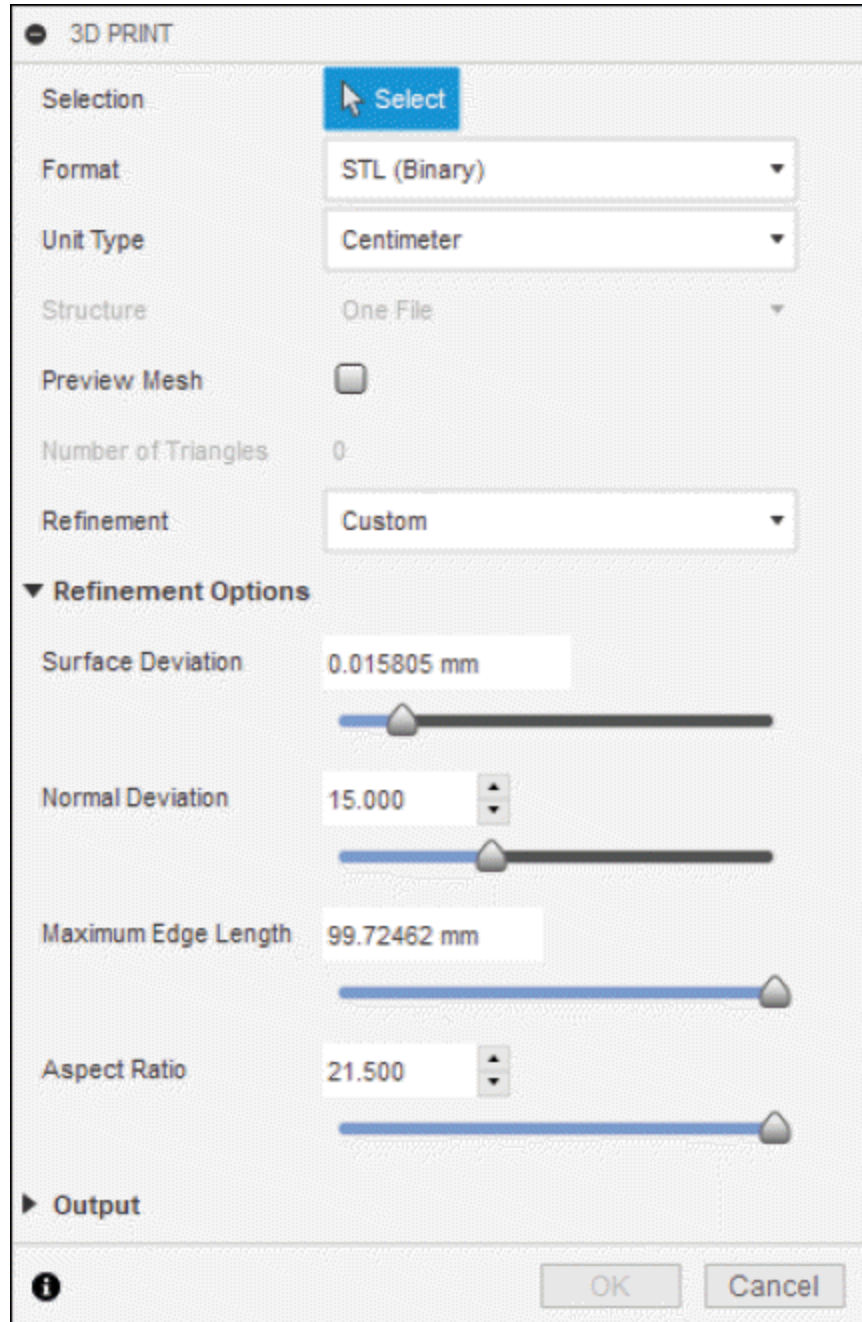


Figure-36. Refinement Options

Now, for the technical stuff :

The **Surface Deviation** is difference between real surface of model and triangulated tessellation of model. In simple language, it drives the number of triangles to be generated

along the surfaces/faces in mesh; refer to [Figure-37](#). The value specified in **Surface Deviation** edit box will be taken as maximum deviation allowed, the software will process the deviation lower than specified value but it will not go higher.

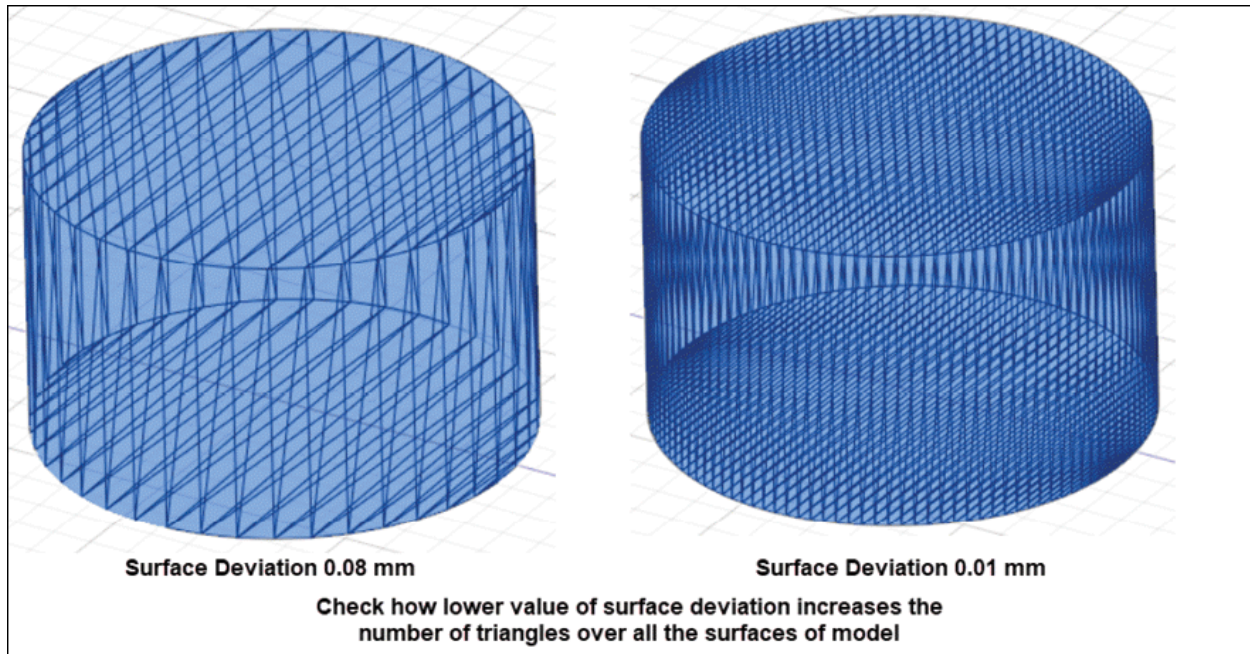


Figure-37. Surface Deviation in 3D Printing

The **Normal Deviation** is gap between two normals of consecutive triangles in tessellation. In simple words, a lower value of normal deviation will increase the number of triangles perpendicular to the edges of model in a given region; refer to [Figure-38](#).

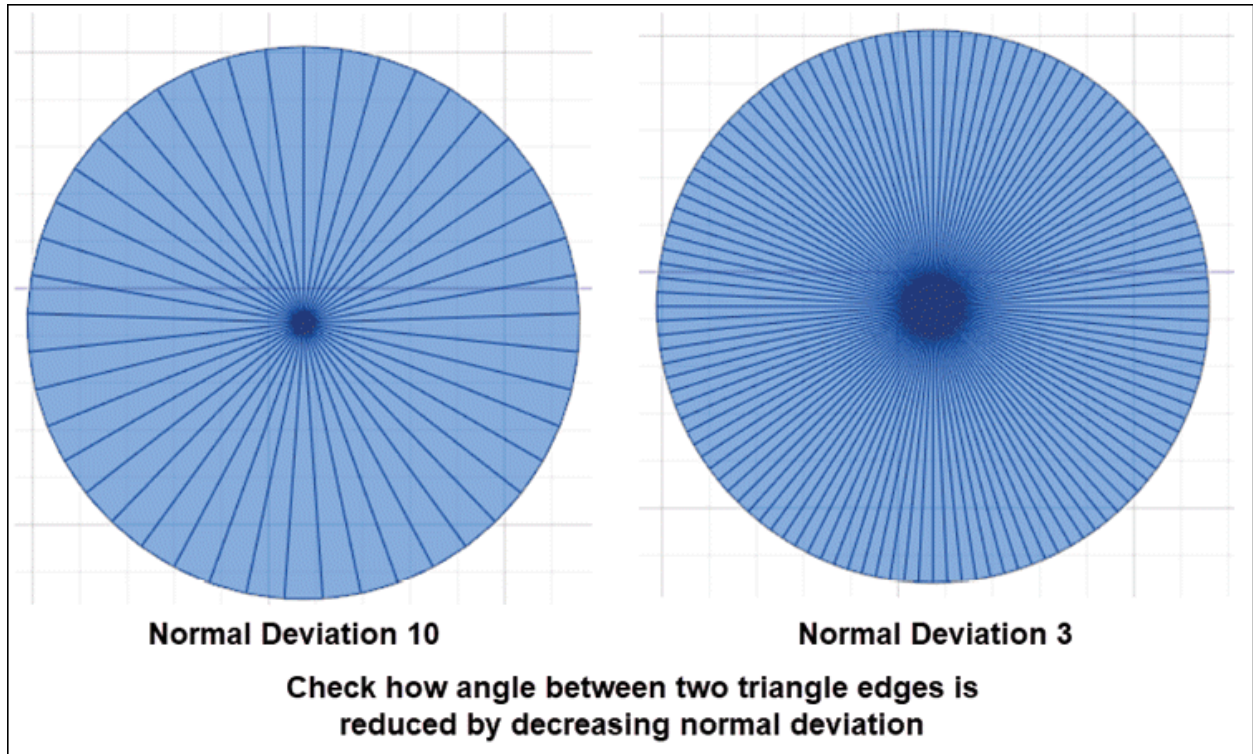


Figure-38. Normal Deviation in 3D printing

The **Maximum Edge Length** is used to define maximum length of triangle edges. If you decrease maximum edge length then more triangles of smaller edge length will be created in best possible orientations; refer to [Figure-39](#).

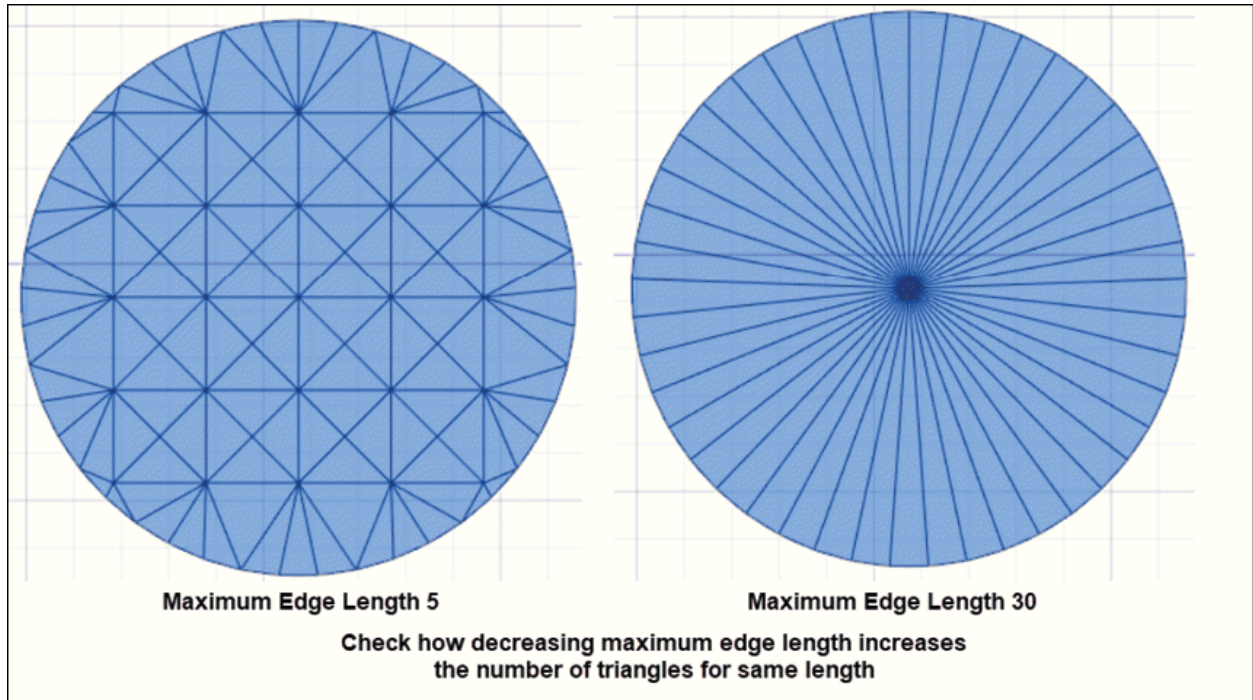


Figure-39. Maximum Edge Length in 3D Printing

The **Aspect Ratio** defines the ratio of number of triangles along width and height of the model; refer to [Figure-40](#). (If you have irregular shape model then don't bother finding aspect ratio, your software knows better!!)

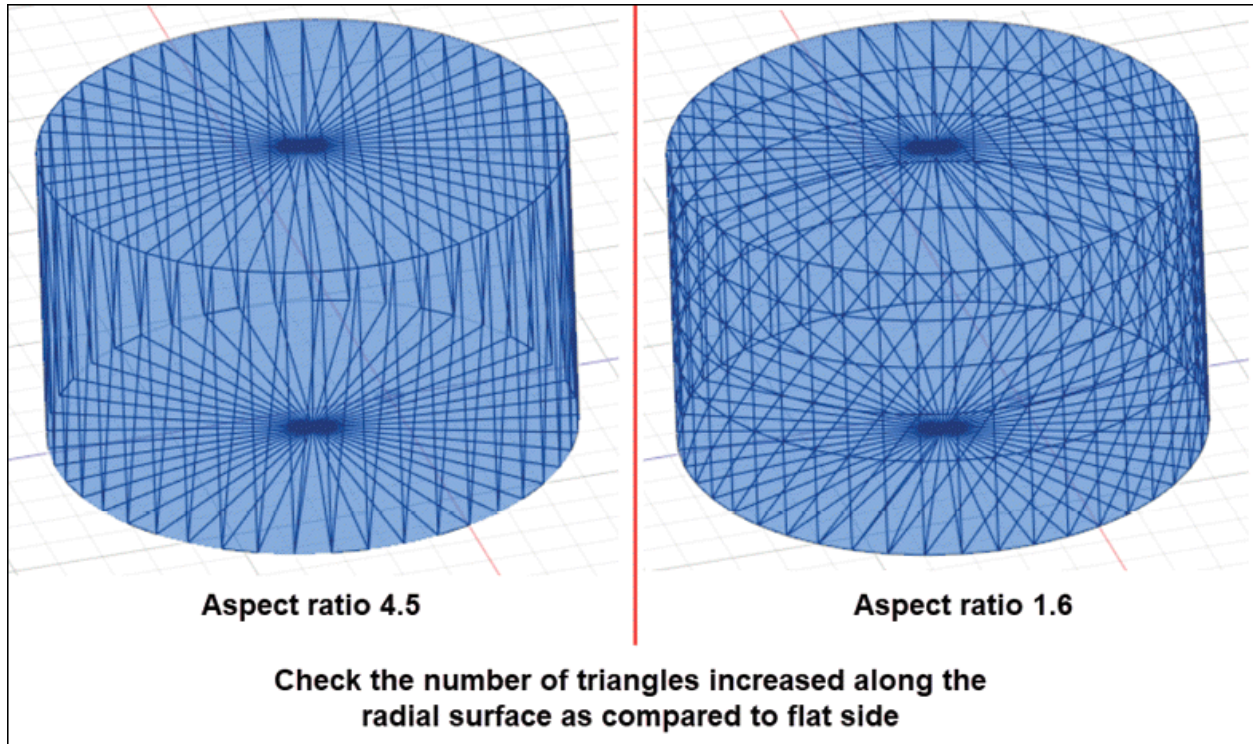


Figure-40. Aspect ratio in 3D printing

- Select the **Send to 3D Print Utility** check box in **Output** section to further edit the model in 3D printing utility. Select desired 3D Printing utility from **Print Utility** drop-down; refer to [Figure-41](#). Note that 3D printing utilities are not installed automatically during Fusion 360 installation. On selecting the utility from **Print Utility** drop-down, an option to download and install the utility will be displayed in the dialog box. Download and install desired utility.

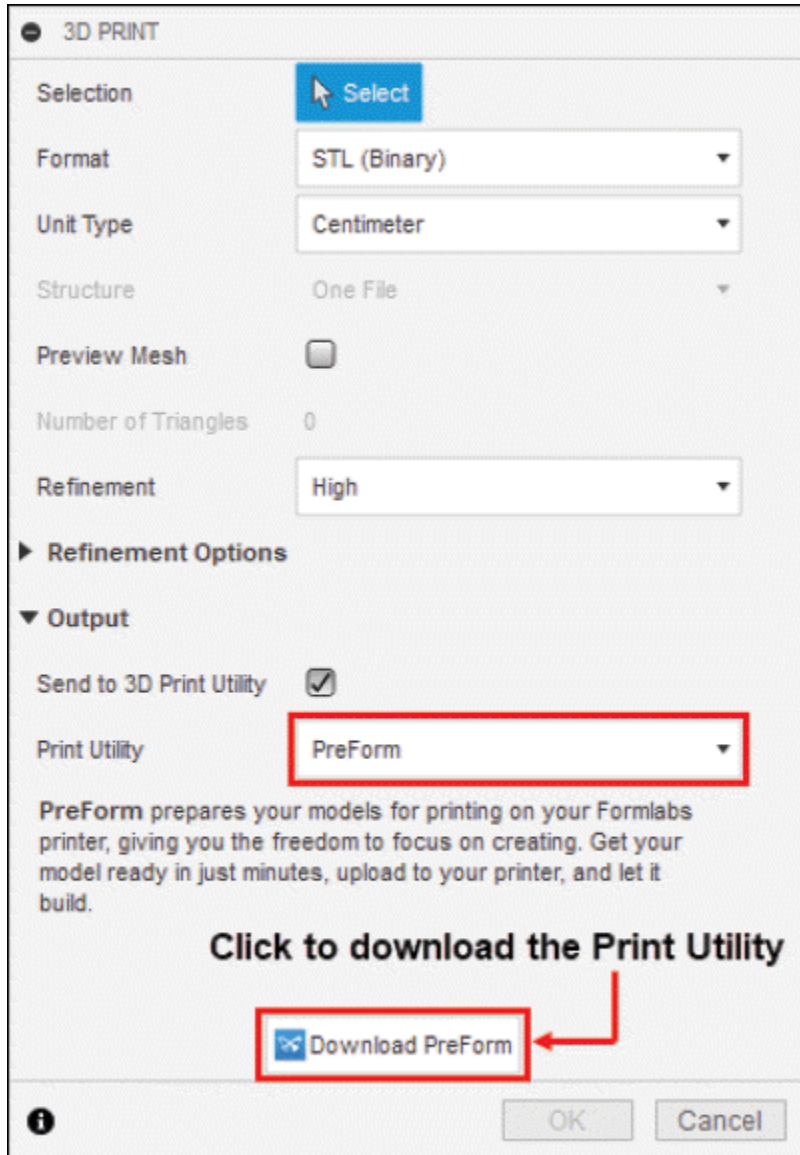


Figure-41. Option To download Print Utility

- Clear the **Send To 3D Print Utility** check box if you want to save the project file in **STL** or other 3D printing file format so that you can open this file later in any 3D Printing software.
- In our case, we have selected **PreForm** option from the **Print Utility** drop-down (Needless to say, we have

installed it already). Click on the **OK** button from the dialog box. The **PreForm** application window will be displayed; refer to [Figure-42](#).

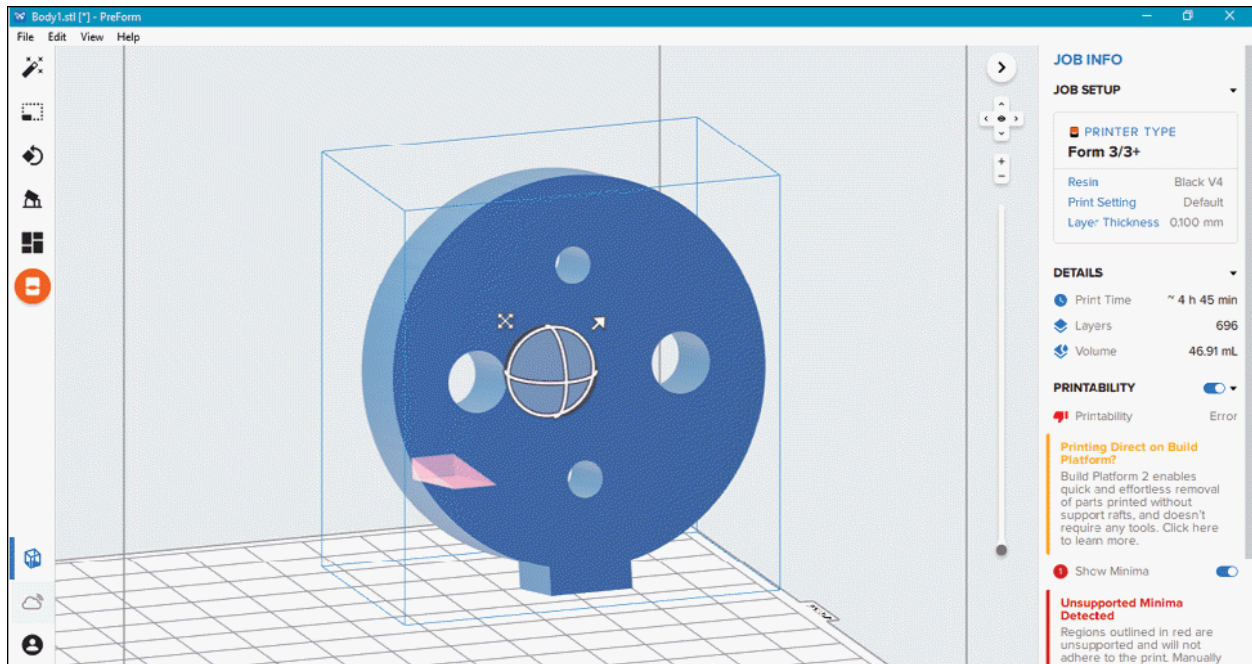


Figure-42. PreForm window

The tools of this application are discussed next.

PREFORM

Click on the **Apply** button from the **JOB SETUP** dialog box to enter the print utility, we will come back to these options later. On clicking **Apply** button, the model will be displayed in PreForm application window; refer to [Figure-43](#).

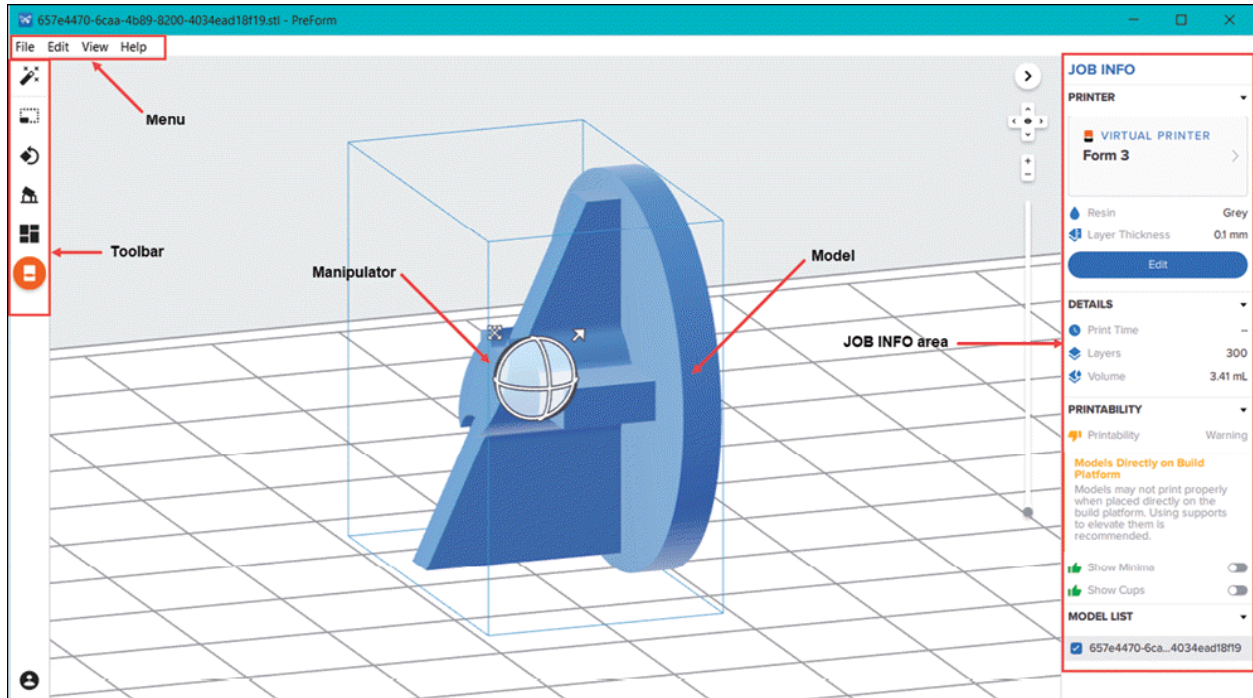


Figure-43. Interface of PreForm

Orienting the Model

There are two ways to orient a model in Preform, using **Orientation** tool from **Toolbar** or by using **Manipulator** displayed on the model. We will discuss both the ways here.

Orienting Model Using Manipulator

Click on the **Circle** handle in **Manipulators** for direction in which you want to rotate the model for orientation and drag the handle; refer to [Figure-44](#). Note that software will automatically place the flat face of model on the bed of virtual machine.

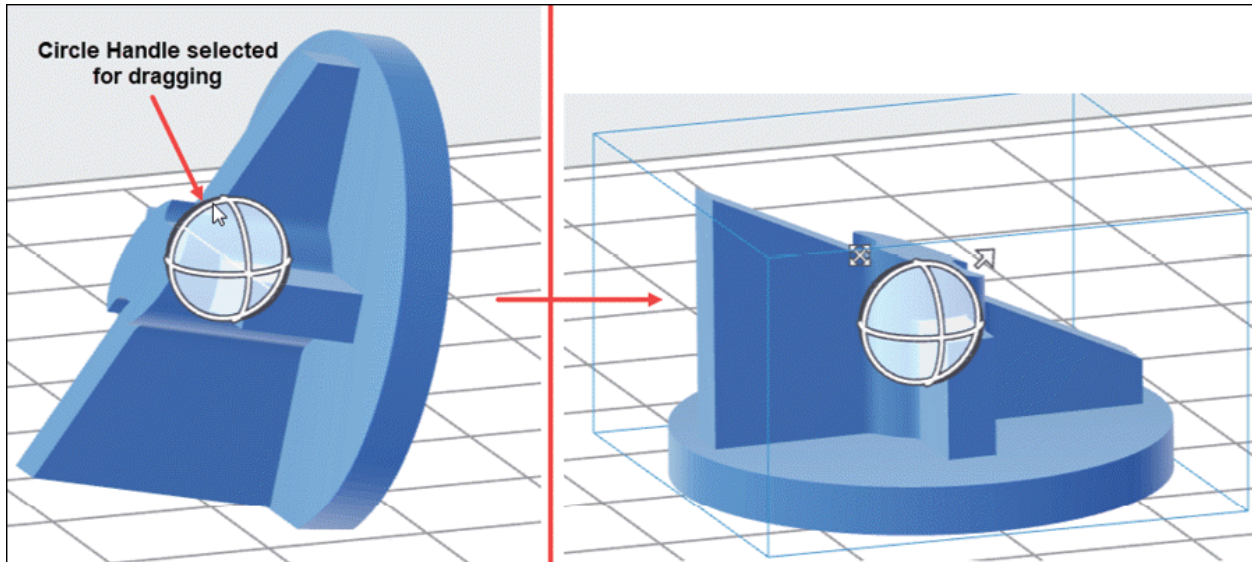


Figure-44. Orienting model using Manipulator

If you want to move the model at some other location then click on the **Move** handle in **Manipulators** and drag it to desired location; refer to [Figure-45](#).

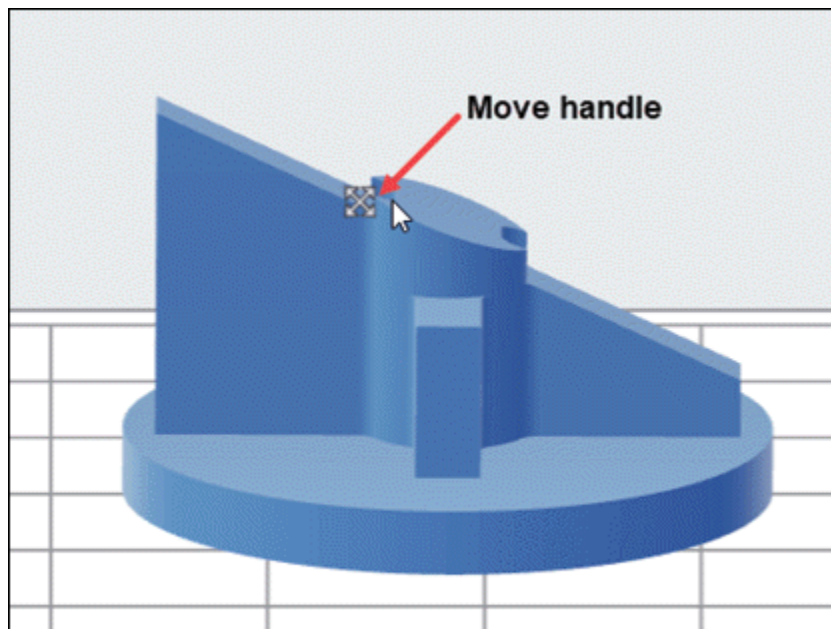


Figure-45. Move handle in Manipulator

Note that the **Manipulators** are displayed on model only

when the model is selected. If the **Manipulators** are not displayed by default in your case or it is hidden then select the **Show Manipulators** option from the **View** menu; refer to [Figure-46](#).

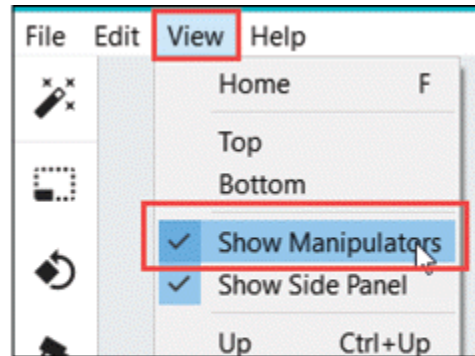



Figure-46. Show Manipulators option

Orienting Using Orientation Tool

- Click on the **Orientation** tool  from the **Toolbar** at the left in application window. The **ORIENTATION** toolbox will be displayed; refer to [Figure-47](#).

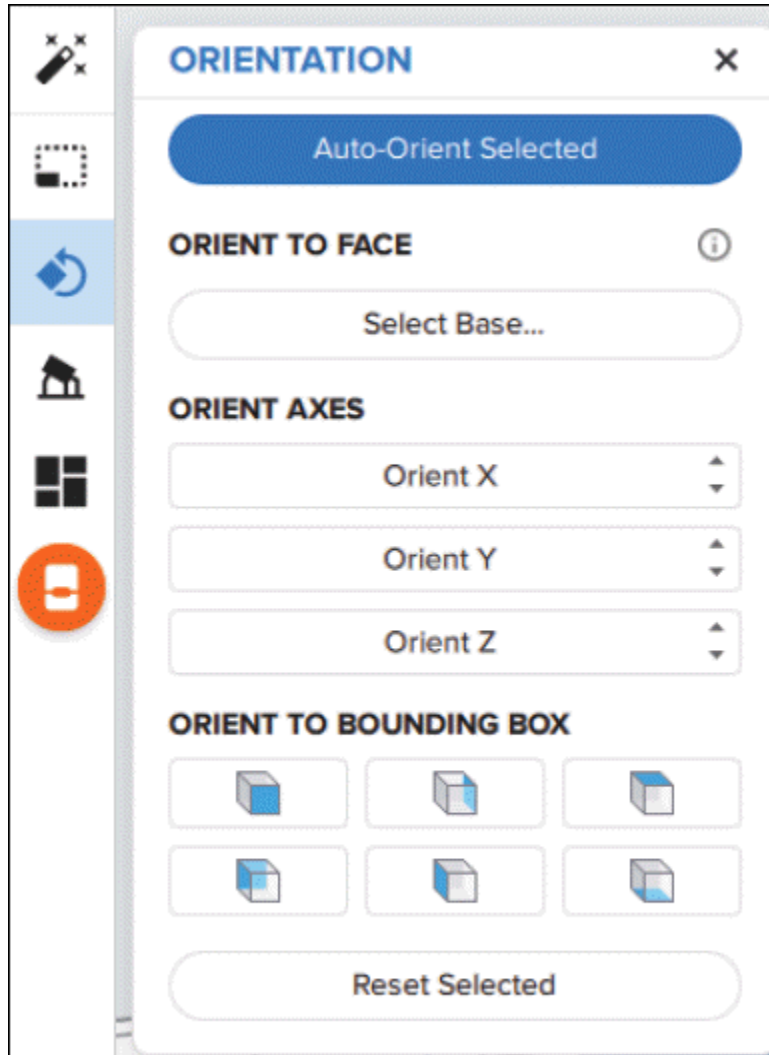


Figure-47. *ORIENTATION* toolbox

- Click on the **Auto-Orient Selected** button to automatically orient the selected model on bed. (Note: this option was clumsy while writing this book and could not generate a proper orientation so we do not recommend this tool.)
- Click on the **Select Base** button to define the face which will be placed flat on the bed. After selecting this button, click on the face to be used as base of model; refer to

[Figure-48](#). The model will be automatically oriented to bed. Click on the **Done** button from the information box displayed in top-left corner. (This is the most used and recommended method to orient the part.)

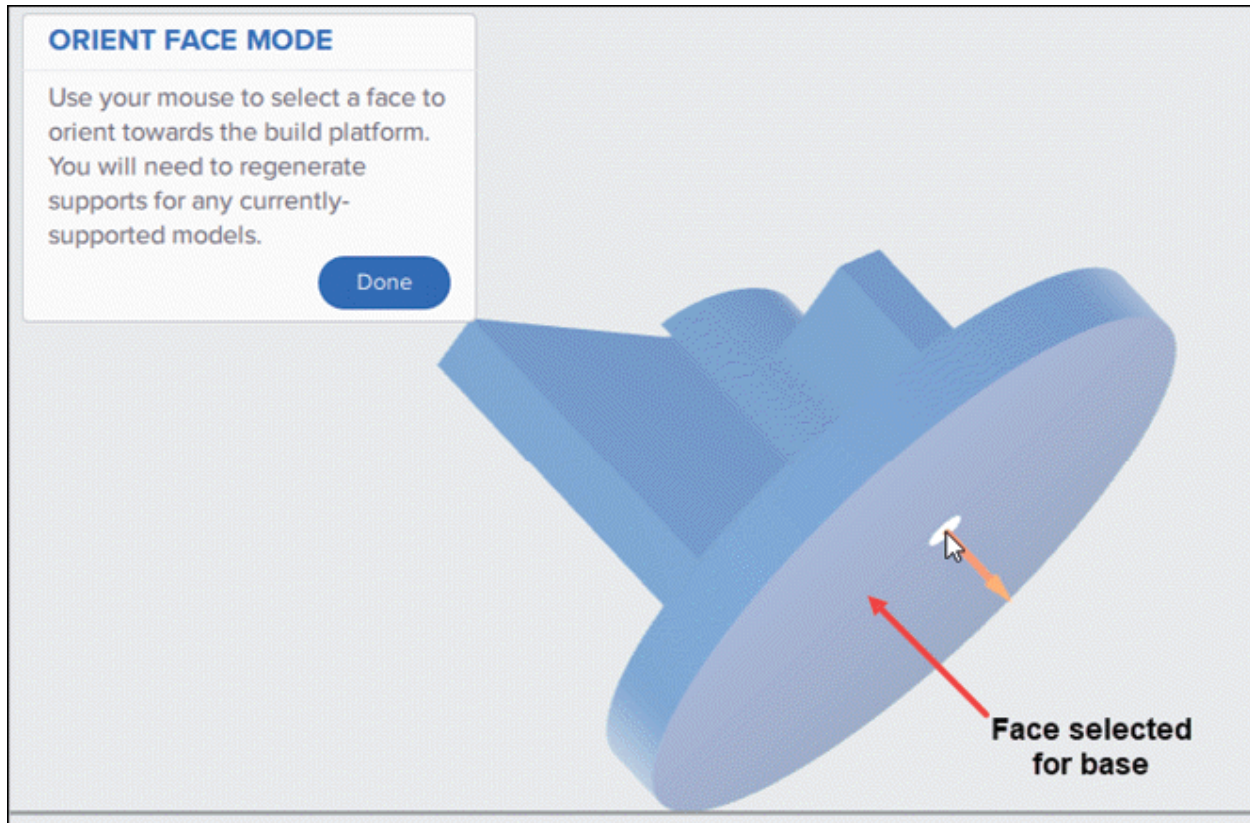


Figure-48. Selecting face for base

- You can use the spinners in **ORIENT AXES** area of the dialog box to define orientation angles for model. (These options are generally used to tweak the orientation; refer to [Figure-49](#). You can also use handles in Manipulators to tweak the orientation.)

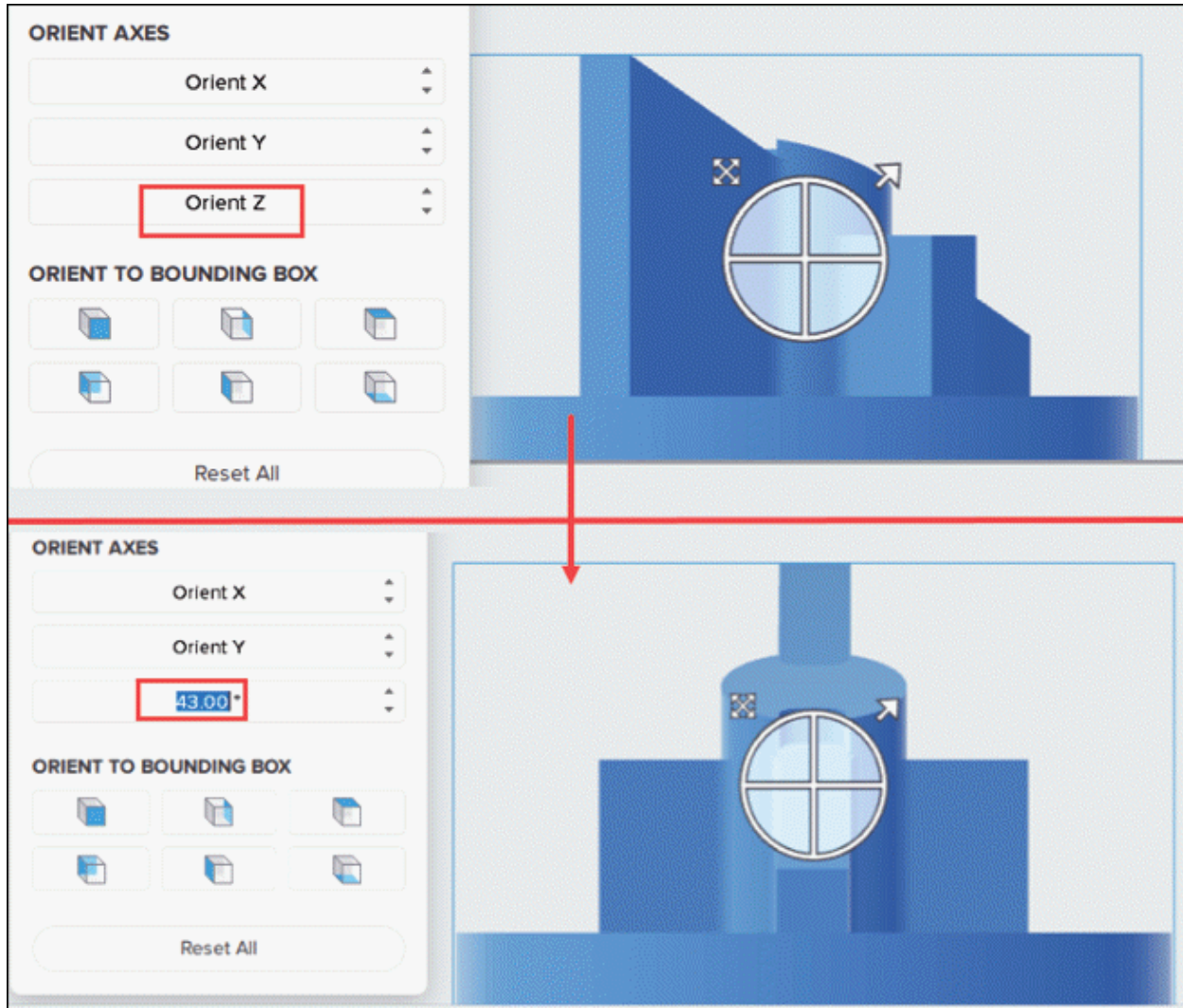


Figure-49. Tweaking the orientation of model

- The buttons in **ORIENT TO BOUNDING BOX** area of toolbox are used to orient the model along respective faces of the bounding box. (Bounding box is an imaginary cuboid which defines maximum length, width, and height occupied by the model.)
- At any time, you find that you have messed up the orientation then you can move back to original orientation by clicking on the **Reset Selected** button at the bottom in

the toolbox.

Note that there are two sides of bed for 3D printing machine called FRONT and MIXER SIDE; refer to [Figure-50](#). The front is the side from where you will be extracting the 3D printing model so this is the side of machine where you will be standing while controlling the machine. The Mixer side is the one where material cartridge of 3D printer will be located.

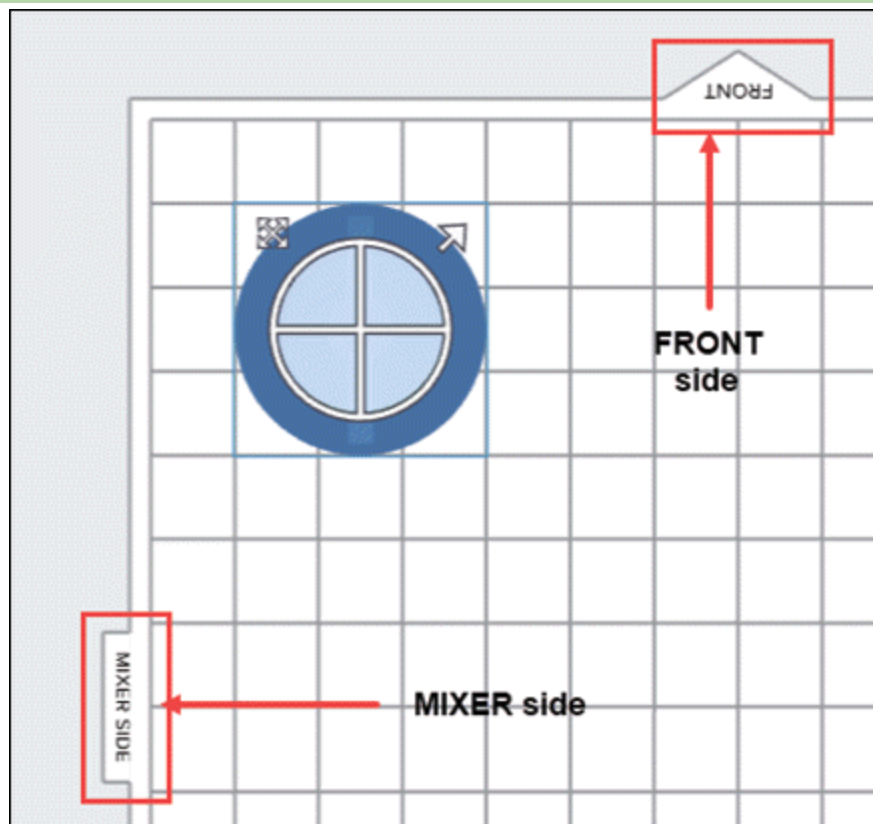


Figure-50. Sides of machine bed

Changing View

You can use the **Up**, **Down**, **Left**, **Right**, and **Home** buttons at the top-right corner of application window to switch

between different views; refer to [Figure-51](#). To dynamically change the view, right-click in modeling area, hold the button, and drag the cursor.

To zoom in and zoom out, click on the + and - buttons respectively which are available in the **View Switch** buttons or you can scroll up and scroll down with mouse.

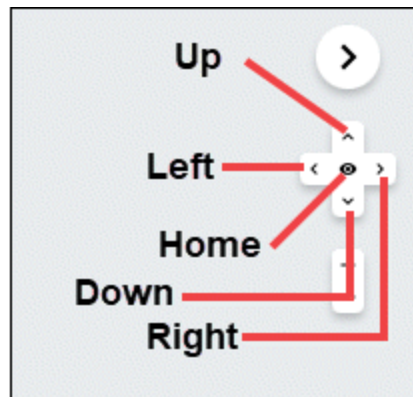


Figure-51. View Switch buttons

The shortcut keys for changing views are given next.

Up View	CTRL + Up key
Down View	CTRL + Down key
Left View	CTRL + Left key
Right View	CTRL + Right key
Home View	F
Zoom in	+ or Scroll up
Zoom out	- or Scroll down

Pan	SHIFT + Right-click drag
Dynamic View change	Right-click drag

Scaling Model

To scale up or scale down the model, click on the **Size** handle in **Manipulators**, hold the button and then drag in desired direction to scale up or scale down; refer to [Figure-52](#).

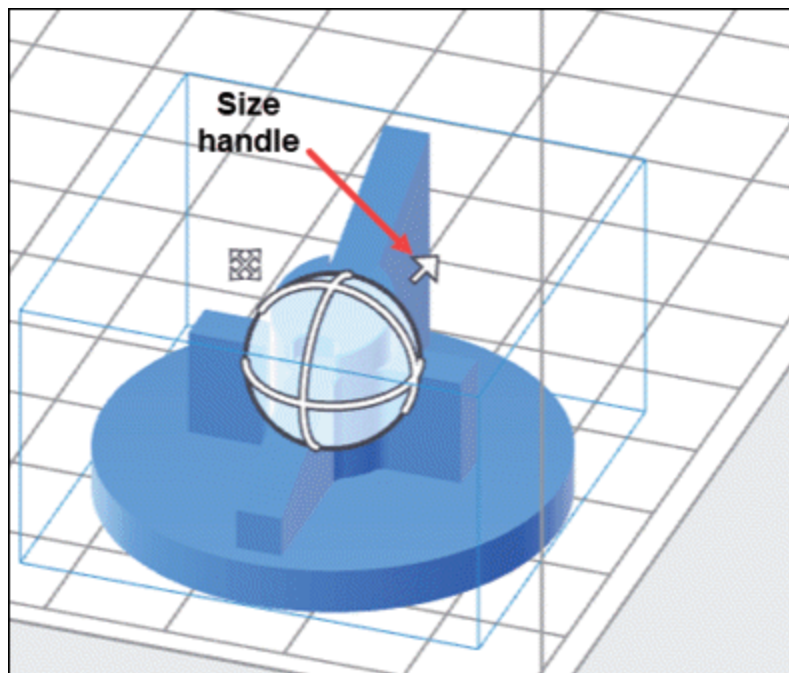


Figure-52. Size handle

OR

- Click on the **Size** button from the **Toolbar** at the left in application window. The **SIZE** toolbox will be displayed; refer to [Figure-53](#).

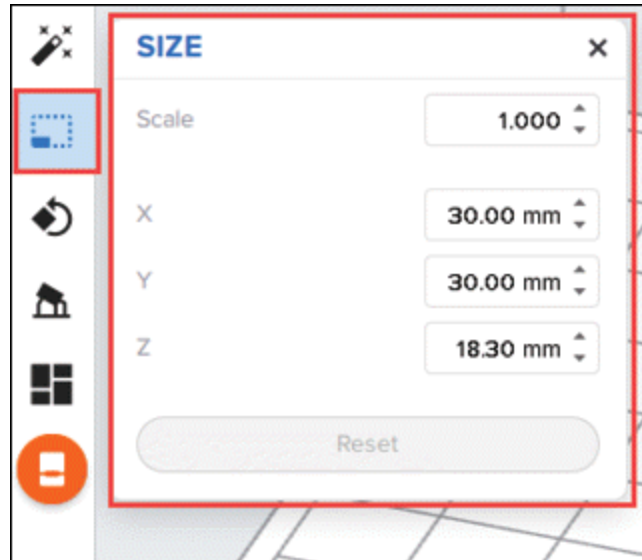


Figure-53. SIZE toolbox

- Specify desired scale value in the **Scale** edit box of toolbox or you can use the spinner to change its value.
- You can also use **X**, **Y**, or **Z** spinners to change the scale of model. Note that size of model will increase or decrease uniformly in all direction so it does not matter whether you use **Scale**, **X**, **Y**, or **Z** spinner.
- Click on the **Reset** button if you want to revert back to original scale. Close the toolbox after making desired changes.

JOB INFO Settings

The options in the **JOB INFO** area are used to set parameters related to printer. You can also check the warnings and details related to 3D printing process. Various tools and options are discussed next.

Job Settings

- Click on the **Edit Job Setup** button (**Printer Type** selection box) from the **JOB INFO** area at the right in the dialog box. The **Job Setup** window will be displayed; refer to [Figure-54](#).

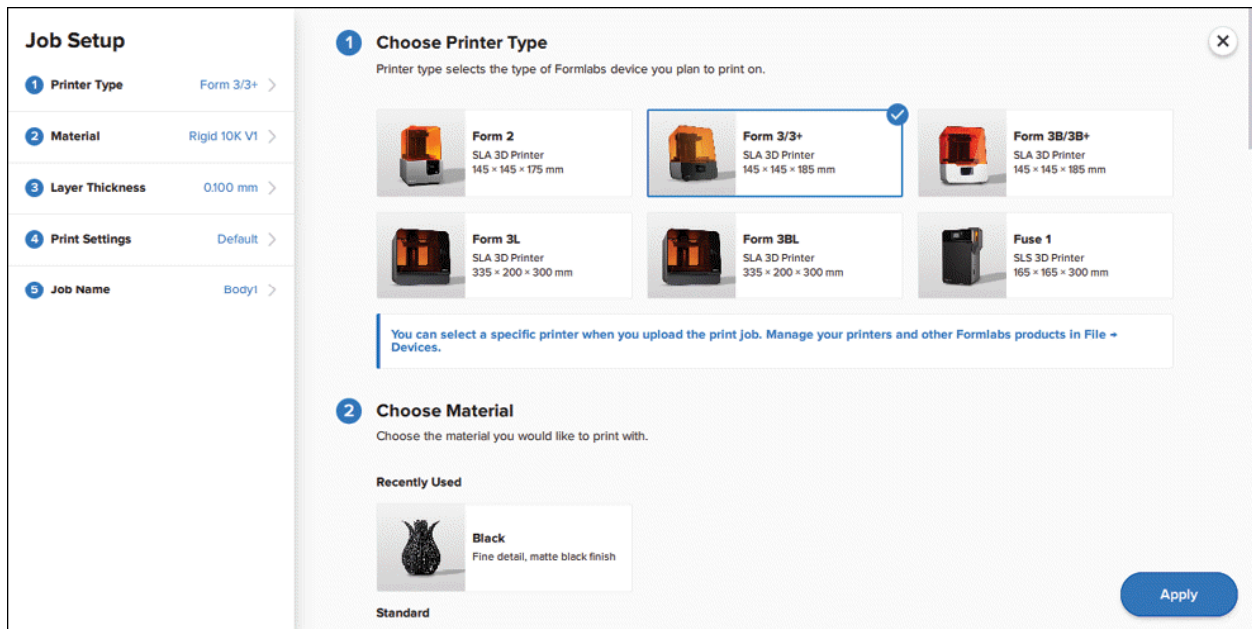


Figure-54. Job Setup window

- Click on the **Printer Type** button from left area of the window and select desired type of printer you want to use for printing.
- If you have selected a 3D printer which has options to select different types of materials then select desired type of materials and versions of resins you want to print with.
- Click on the **Layer Thickness** button and select desired thickness of layer to increase print speed or resolution. Thicker layers print faster and thinner layers print slower.
- Click on the **Print Settings** button and select desired print settings to control print speed, support print shape, and

other aspects of print performance.

- Click on the **Job Name** button and type desired name in the **Job Name** edit box to help differentiate your print on the printer and dashboard.
- After specifying desired parameters in the **Job Setup** window, click on the **Apply** button to make the changes.

Job Details

Various details of the printing job will be displayed in the **DETAILS** rollout of the **JOB INFO** area like estimated printing time, number of layers in 3D printed model, and total volume of material used in 3D printing.

PRINTABILITY Options

There are three options in the **PRINTABILITY** rollout of **JOB INFO** area; **Printability**, **Show Minima**, and **Show Cups**. The **Printability** icon defines whether the model is good for printing, there are warnings for 3D printing, or there are errors stopping from 3D printing like in [Figure-55](#), a warning is displayed telling you that you need to elevate the model and add some supports for proper extraction.

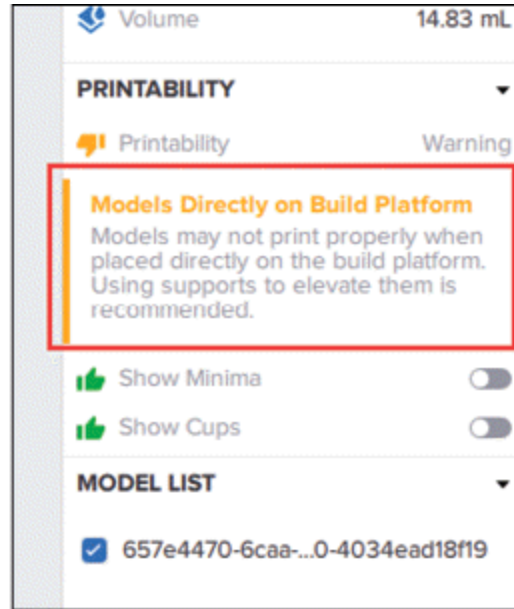


Figure-55. Warning for 3D printing

- Select the **Show Minima** radio button to display the unsupported areas of model if the model is not oriented correctly; refer to [Figure-56](#).

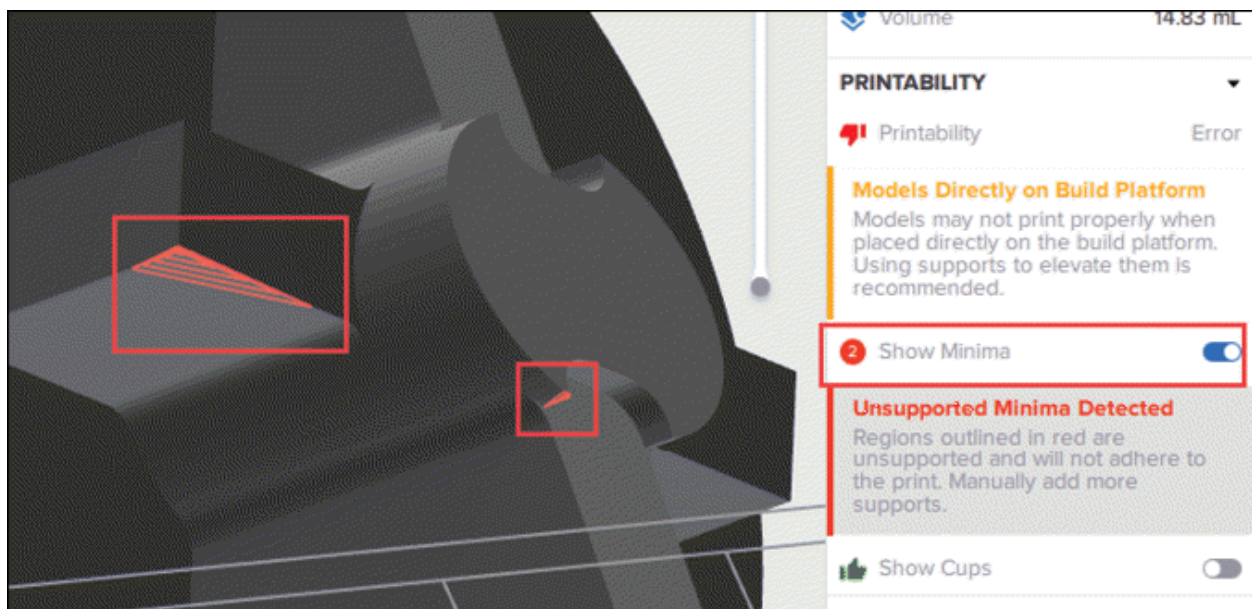


Figure-56. Unsupported minima areas of model

- Select the **Show Cups** radio button from the

PRINTABILITY rollout to display areas on model that can trap air while 3D printing. Most of the time cups are generated when there are holes/cuts perpendicular to base plane.

Adding Supports

The **Supports** tool in the **Toolbar** is used to add supports to the 3d printing object. Supports are used when the model is unstable or extraction of 3d printed model from base plate can cause damage to the model. The procedure to add supports is given next.

- Click on the **Supports** tool from the **Toolbar** or press **C** from keyboard. The **SUPPORTS** toolbox will be displayed; refer to [Figure-57](#).

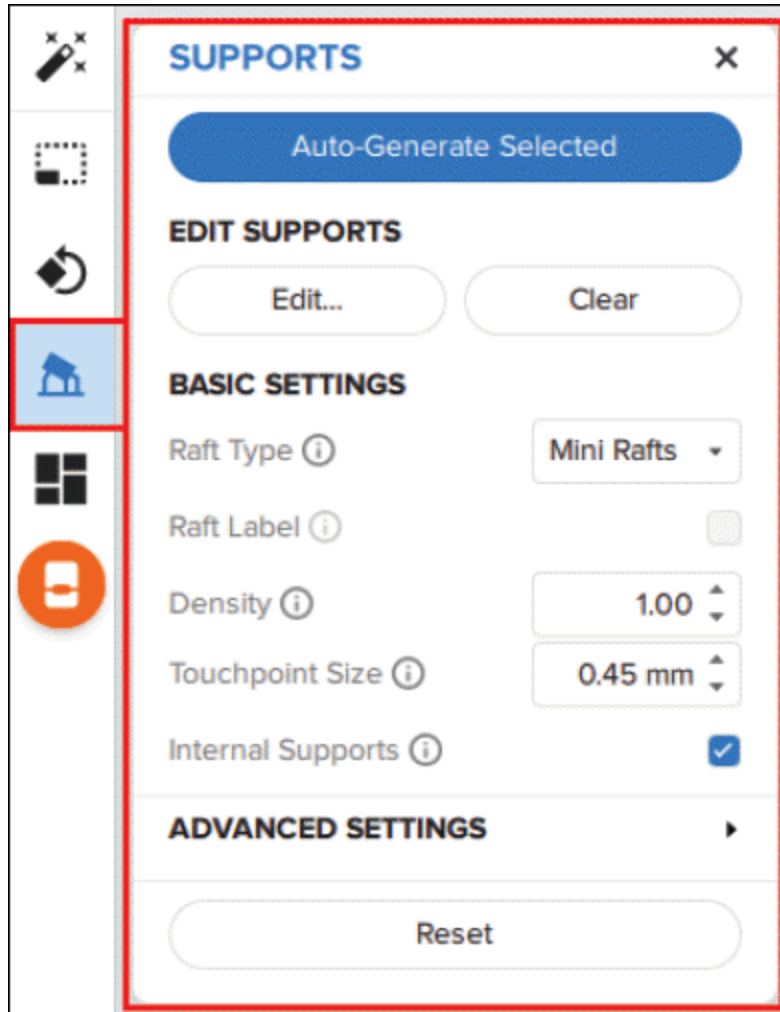


Figure-57. SUPPORTS toolbox

- Click on the **Auto-Generate Selected** button to automatically generate support features for the selected model; refer to [Figure-58](#). Note that using this button, only positions of support elements are decided. It does not automatically decides the shape and size of support elements. So, you should click on this button after you have specified the parameters for supports in the toolbox.

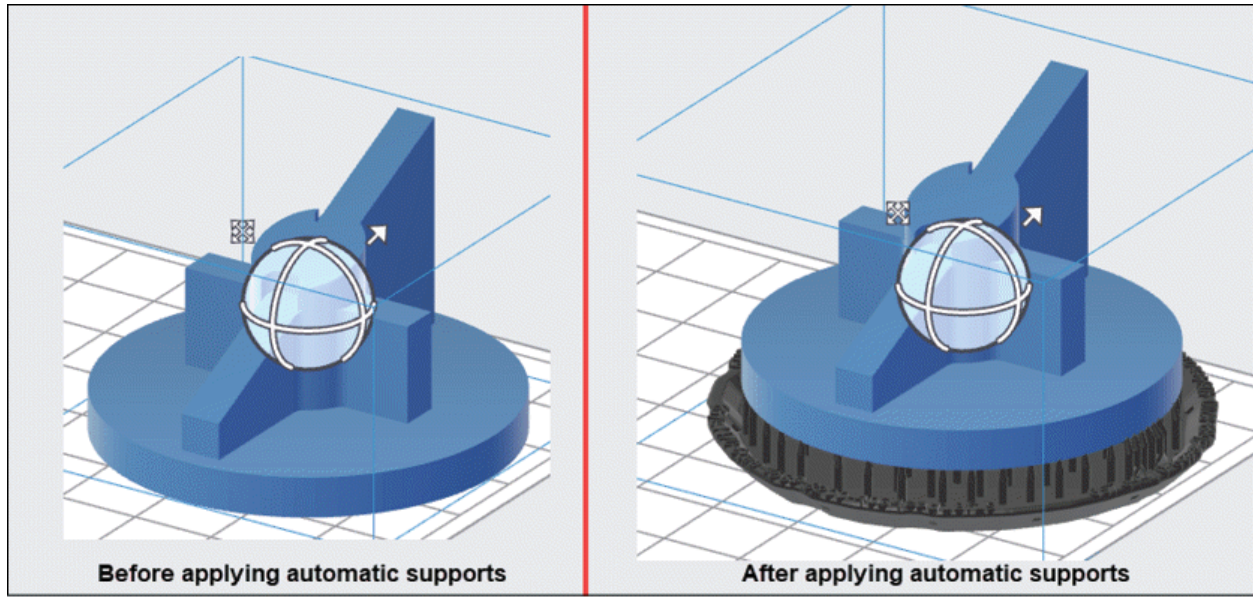


Figure-58. Applying supports

- Select the **Mini-Rafts** option from the **Raft Type** drop-down if you want to mini-rafts for supports; refer to [Figure-59](#).

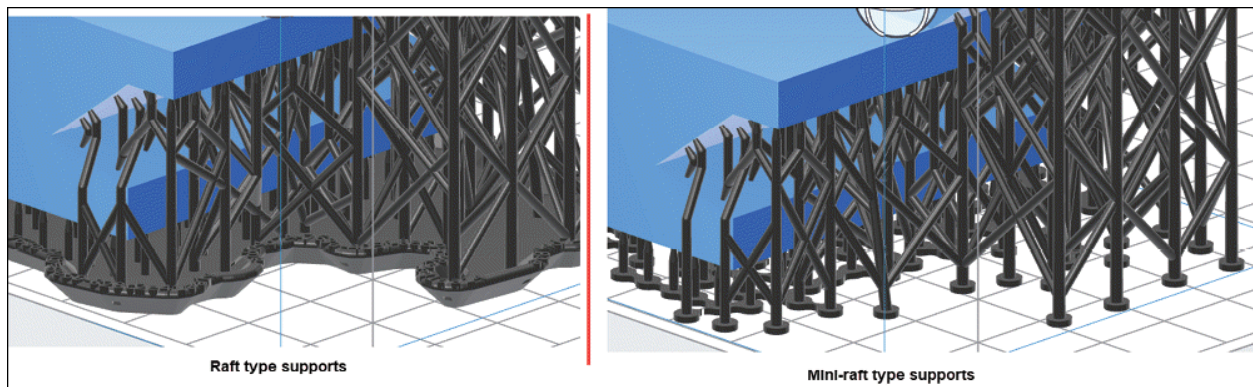


Figure-59. Raft types for supports

- Select the **Raft Label** check box if you want to 3D printing the name of file on the support raft.
- Set desired density and touch point size in respective edit boxes in the toolbox. Density is the thickness of support base and touch point size is the thickness of upper section

of support where it touches the model.

- Select the **Internal Supports** check box if you want to create internal supports whose start point and end point are on the model; refer to [Figure-60](#).

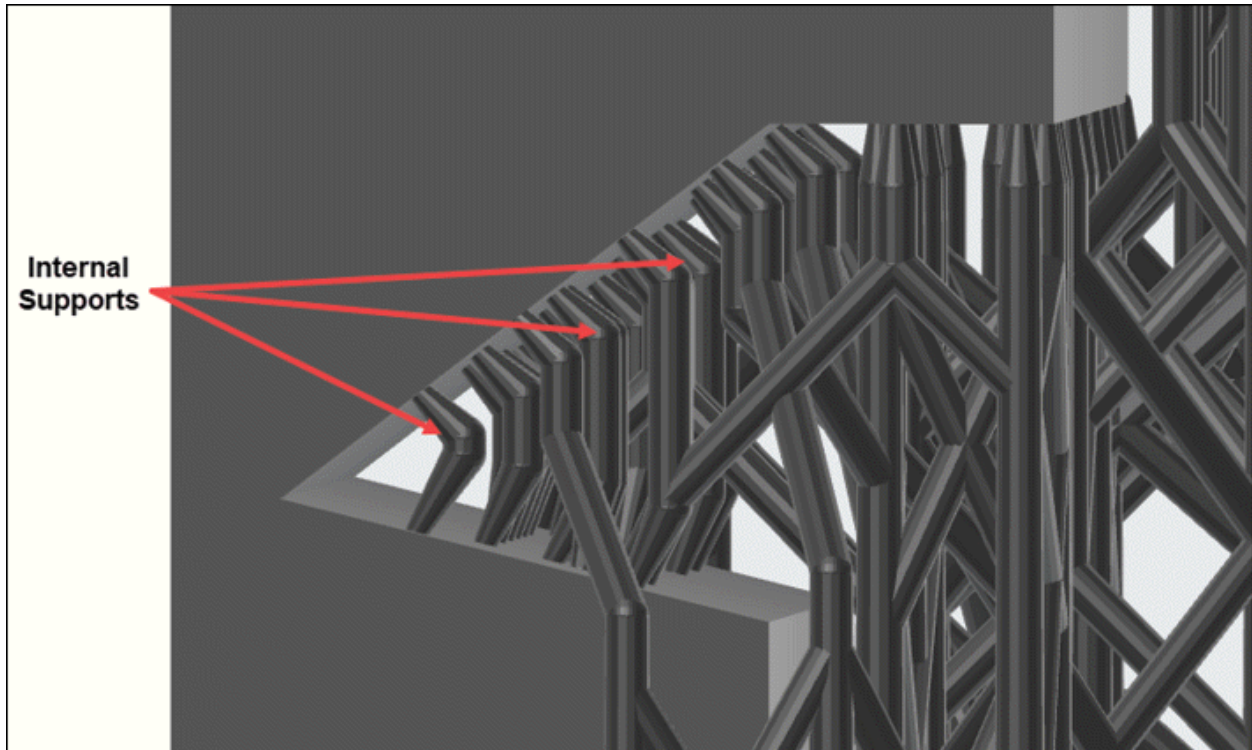


Figure-60. Internal supports created

- Specify desired parameters in the **ADVANCED SETTINGS** rollout of the toolbox if needed.

Creating Layout for Multiple Part Printing

- Click on the **Layout** tool from the **Toolbar**. The **LAYOUT** toolbox will be displayed; refer to [Figure-61](#).

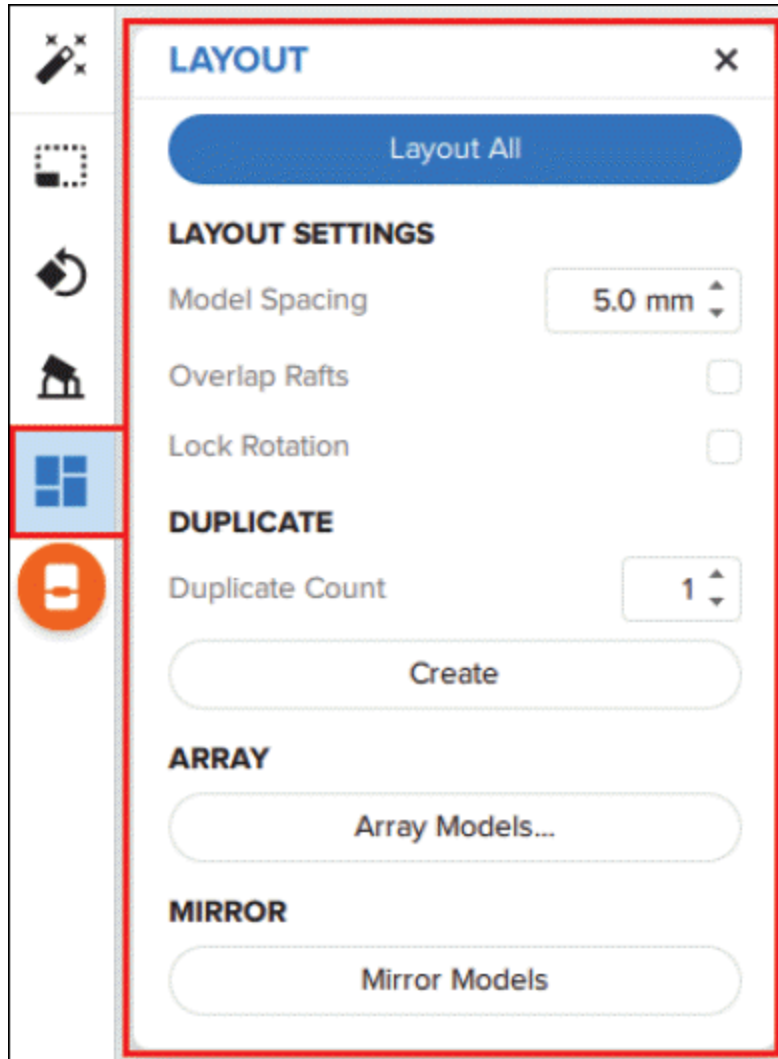


Figure-61. LAYOUT toolbox

- Set desired parameters like space between two instances of the model, raft overlap, rotation lock, and number of duplicate copies of model to be created on same bed.
- After setting desired parameters, click on the **Create** button. The copies of model will be created on the bed. Note that every time you click on the **Create** button, new copies of the model are created. So, it is better to set the duplicate count value carefully and then click on the

Create button; refer to [Figure-62](#).

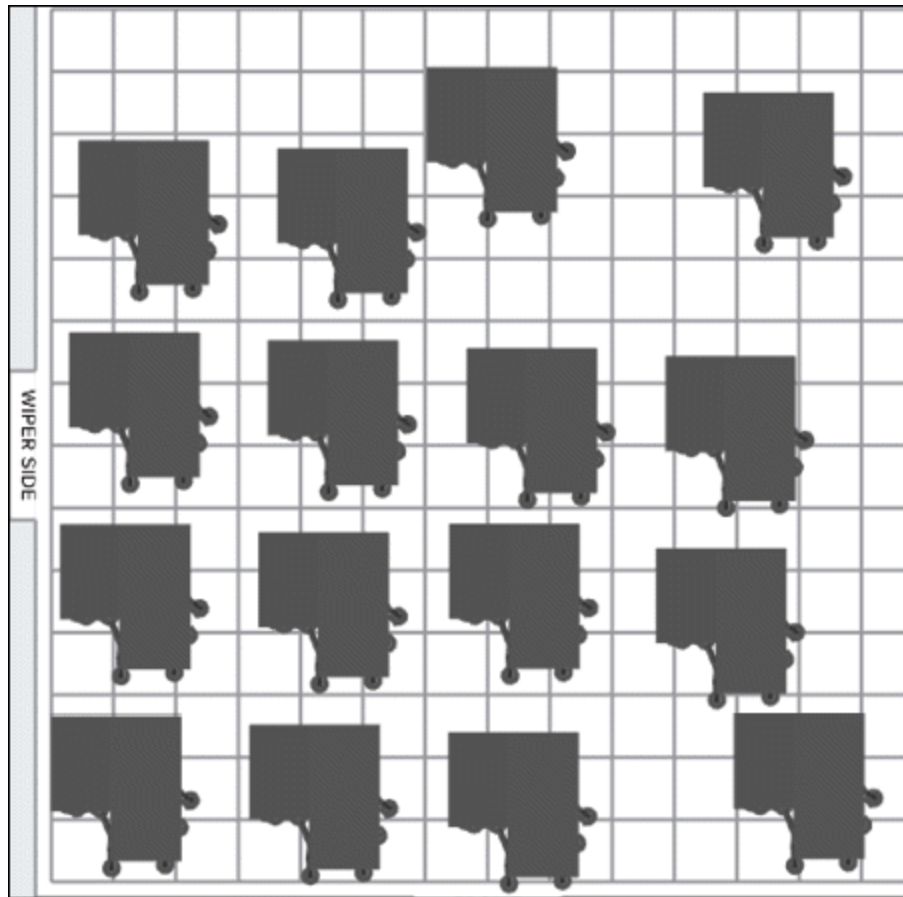


Figure-62. Duplicate copies of model

- Click on the **Layout All** button to arrange the copies of model in such a way that maximum number of copies can be accommodated on the bed; refer to [Figure-63](#). Note that after using this tool, you might be able to add more copies of the model on bed.

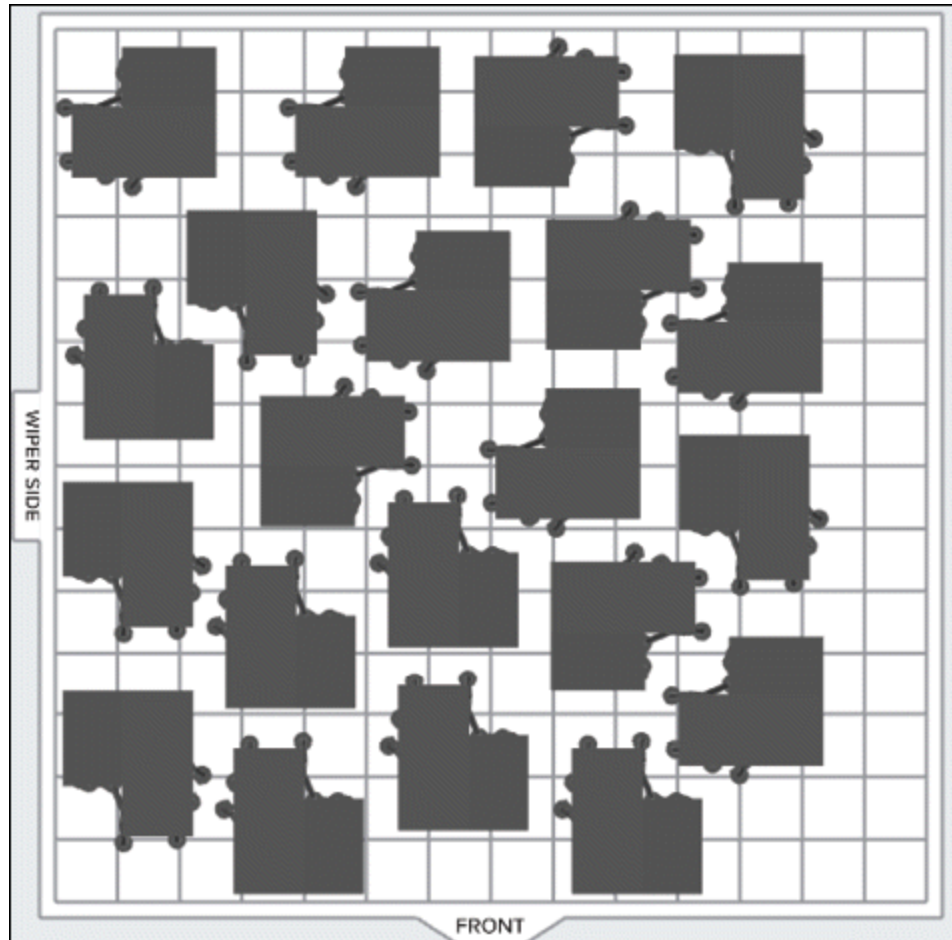



Figure-63. After using Layout All tool

- Click on the **X** button at top-right corner to exit the toolbox.

Printing

Now, we are ready to start 3D printing of model layout. If you have printer connected with your system then click on the **Start a print**  button from the **Toolbar**. The **PRINT** dialog box will be displayed; refer to [Figure-64](#). Click in the **Job Name** edit box and specify desired name for print job. If you have 3D Printer connected then it will be displayed in

the Printer list. You need to select the printer from the list. If your printer is connected to a network on FORMLABS then you need to specify account information or log in to your FORMLABS account. After setting desired parameters, click on the **Upload Job** button. The file will be uploaded on FORMLABS account for printing.

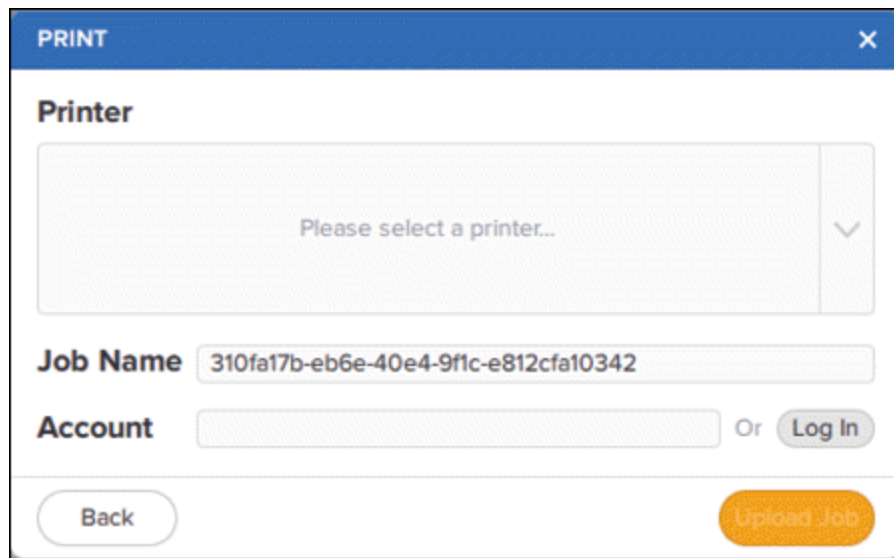


Figure-64. PRINT dialog box

- Click in the **Select Printer** box of **Printer** area in the dialog box to select desired printer. The **DEVICE LIST** dialog box will be displayed; refer to [Figure-65](#).

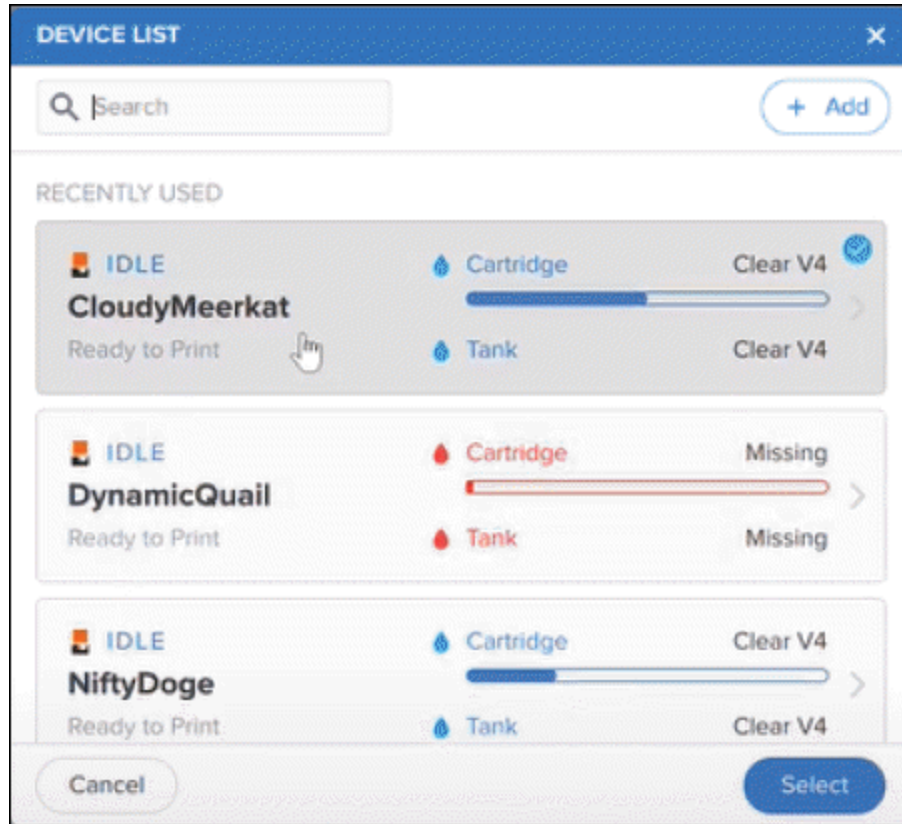


Figure-65. DEVICE LIST dialog box

- Select desired printer from the list and click on the **Select** button from the **PRINTER DETAILS** dialog box displayed; refer to [Figure-66](#).

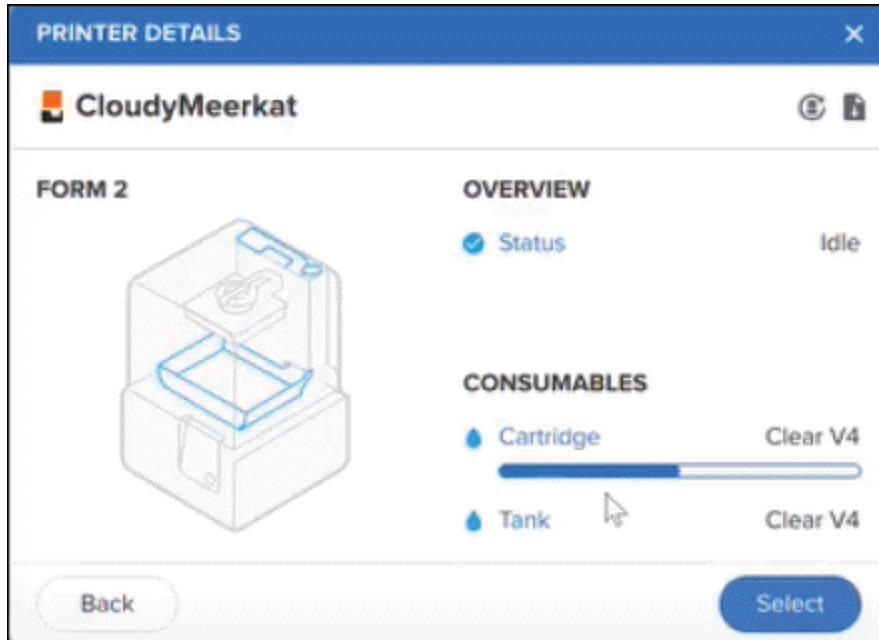


Figure-66. PRINTER DETAILS dialog box

- If you want to add a new printer in the list of **DEVICE LIST** dialog box, click on the **+Add** button from the dialog box. The **ADD DEVICE** dialog box will be displayed; refer to [Figure-67](#). Specify the IP address of network printer in the **IP address** input boxes and click on the **Connect** button. The printer will be added in the list.

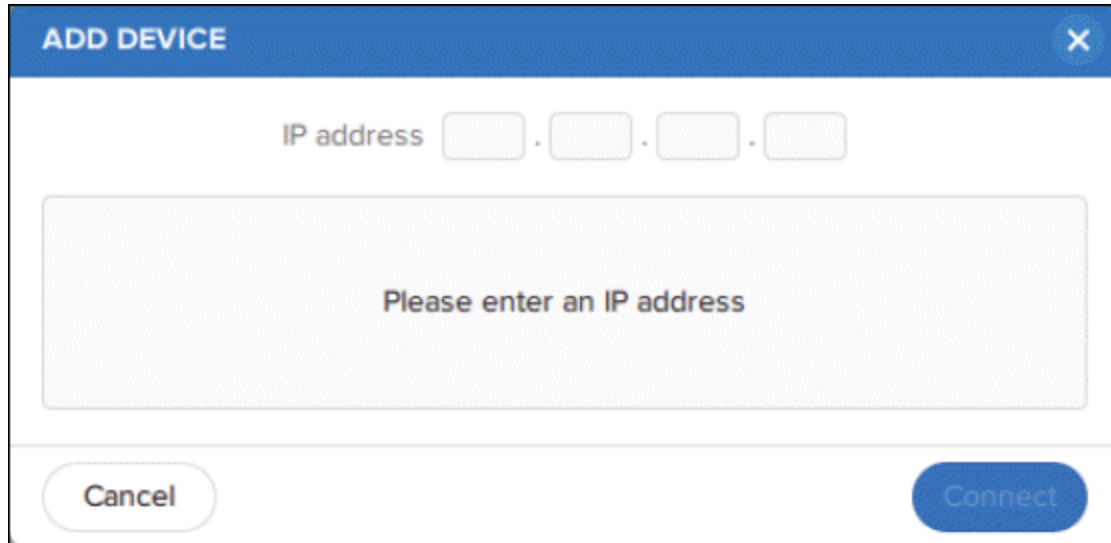


Figure-67. ADD DEVICE dialog box

After performing desired operations, click on the **Quit PreForm** tool from the **File** menu in the application window to exit the PreForm application.

Capture Image

The **Capture Image** tool is used to capture the image of model in current state. The procedure is discussed next.

- Click on the **Capture Image** tool from **File** menu; refer to [Figure-68](#). The **Image Options** dialog box will be displayed; refer to [Figure-69](#).

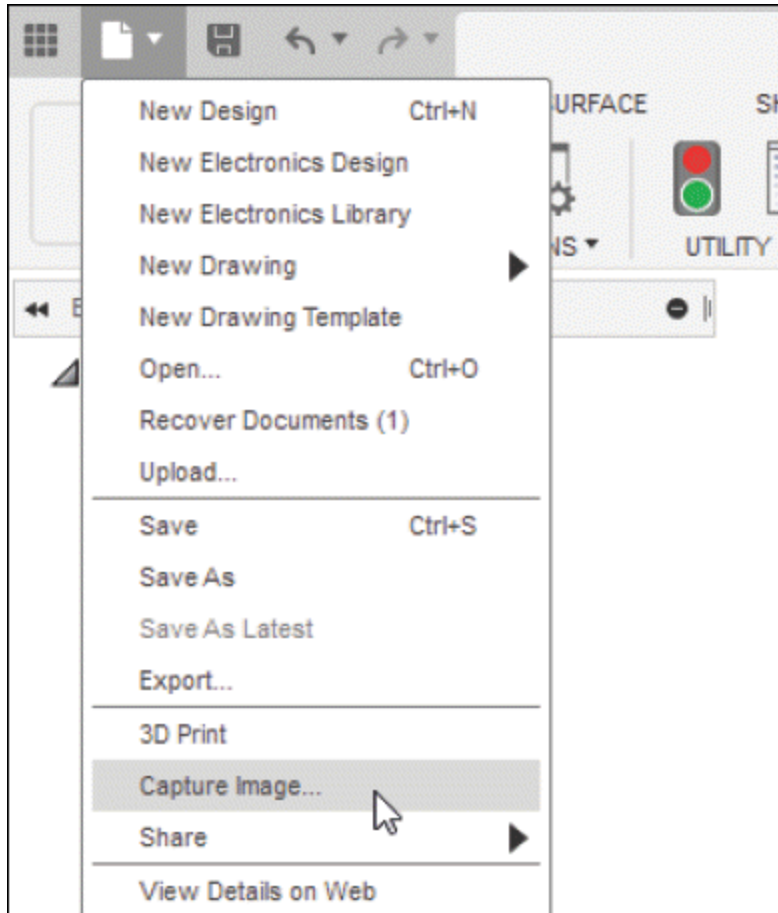


Figure-68. Capture Image tool

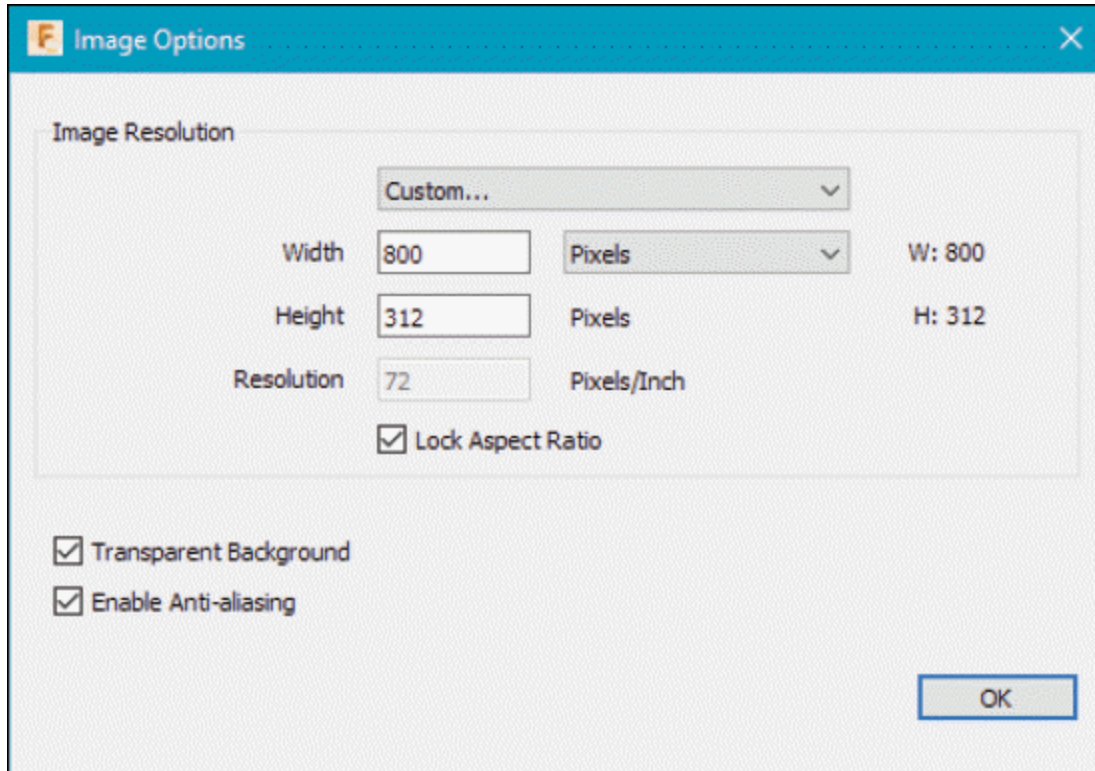


Figure-69. Image Options dialog box

- Select desired option from the drop-down in **Image Resolution** area to define size of image. All the other parameters of **Image Resolution** area will be defined automatically.
- If you want to set image resolution other than the standard one then select the **Custom** option from the drop-down and define the parameters. By default, you are asked to specify size in pixels but you can also define the image size in inches by selecting the **Inches** option from the drop-down next to **Width** edit box. Now, you will be able to define pixel density in the **Resolution** edit box manually. Clear the **Lock Aspect Ratio** check box to specify values

of height and width individually.

- Select **Transparent Background** check box for setting the background of the image to be transparent.
- Select the **Enable Anti-aliasing** check box to smooth jagged lines or textures by blending the color of an edge with the color of pixels around it.
- After setting desired parameters, click on the **OK** button from **Image Options** dialog box. The **Save As** dialog box will be displayed; refer to [Figure-70](#).

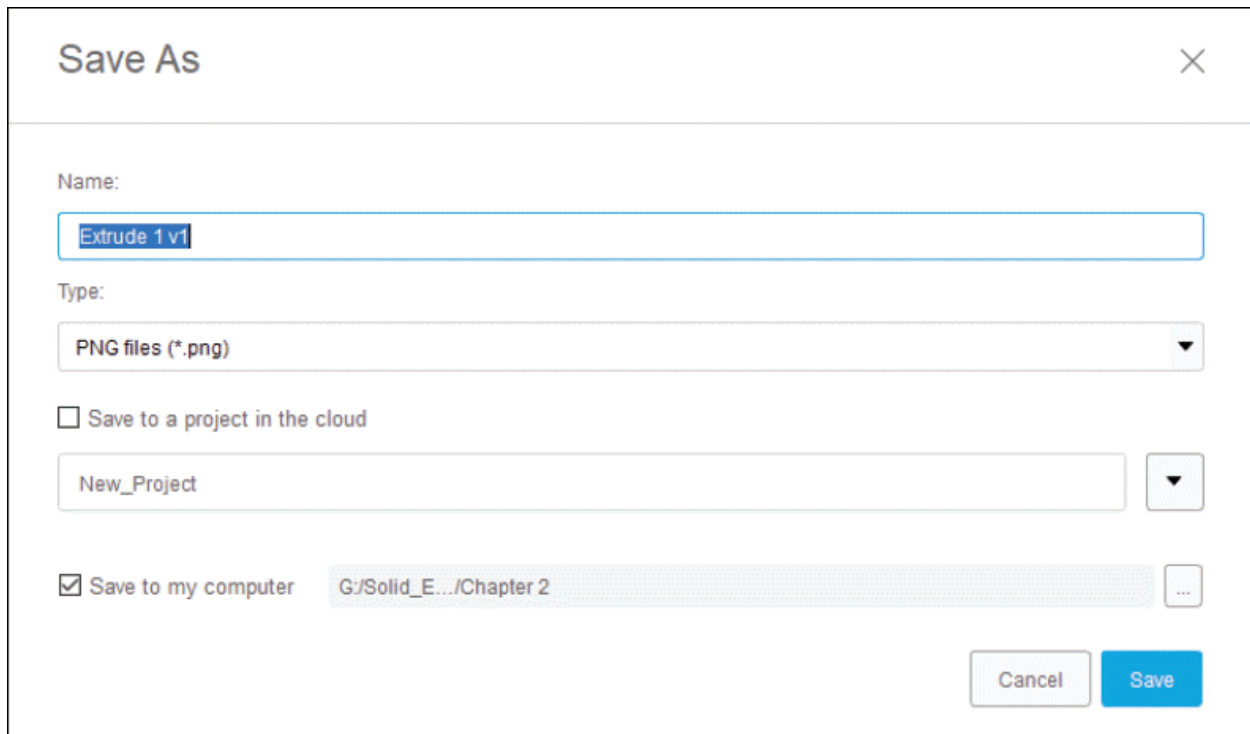


Figure-70. Save As dialog box for image capturing

- Select desired format from the **Type** drop-down for image. There are three formats available: PNG, JPG, and TIFF.
- Set the other parameters as discussed earlier and click on the **Save** button to save the image.

Share

The **Share** tools are used to share the file or project on various portals. Click on the **Share** tool from **File** menu. Four options will be displayed in **Share** cascading menu; refer to [Figure-71](#).

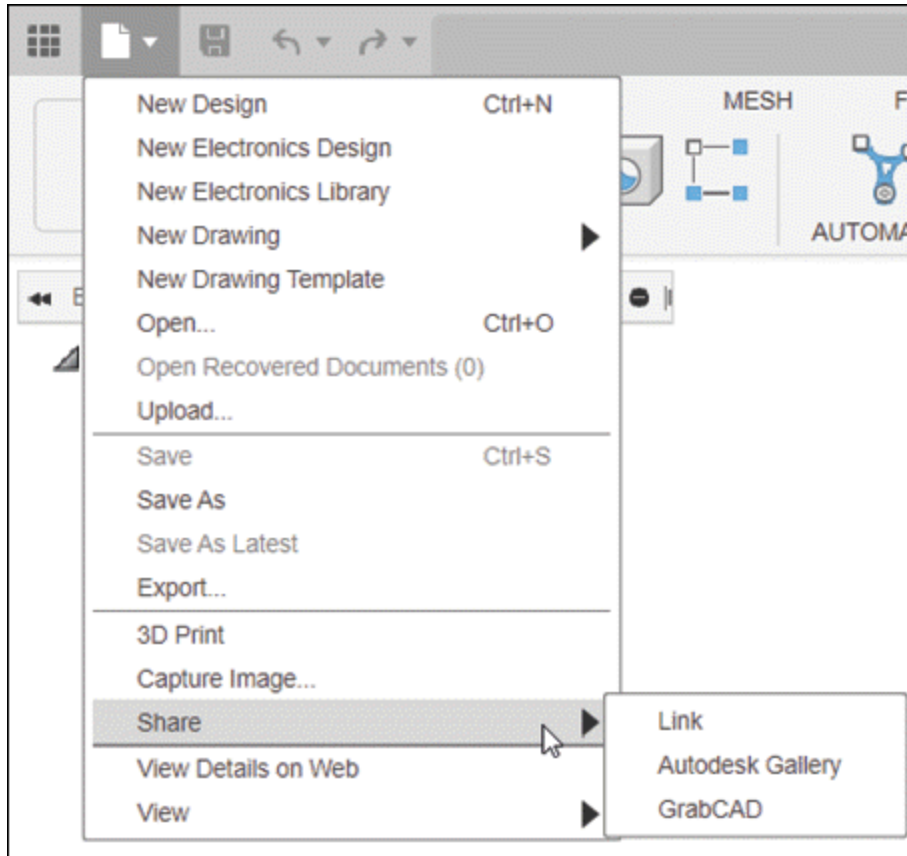


Figure-71. Share cascading menu

Share Public Link

The **Link** tool is used to share the project link with anyone. The procedure to use this tool is discussed next.

- Click on **Link** tool of **Share** cascading menu from **File Menu**.

- On selecting this tool, the **Share Public Link** dialog box will be displayed, refer to [Figure-72](#).



Figure-72. Share Public Link dialog box

- Click on the toggle buttons to activate or de-activate respective functions.
- After activating the sharing function, copy the link from **Copy** box and share the link via E-mail or other method with anyone. Set the **Allow item to be downloaded** toggle button to allow downloading of file.
- Activate the **Require a password to access the public link** option and specify desired password in the edit box displayed below it to secure your link with a password.

Share To Autodesk Gallery

The **Autodesk Gallery** tool is used to share the file online to the gallery of Autodesk where the file is accessible to everyone registered with the gallery. The procedure to use this tool is given next.

- Click on **Autodesk Gallery** tool from **Share** cascading menu. The **SHARE TO AUTODESK GALLERY** dialog box will be displayed; refer to [Figure-73](#).

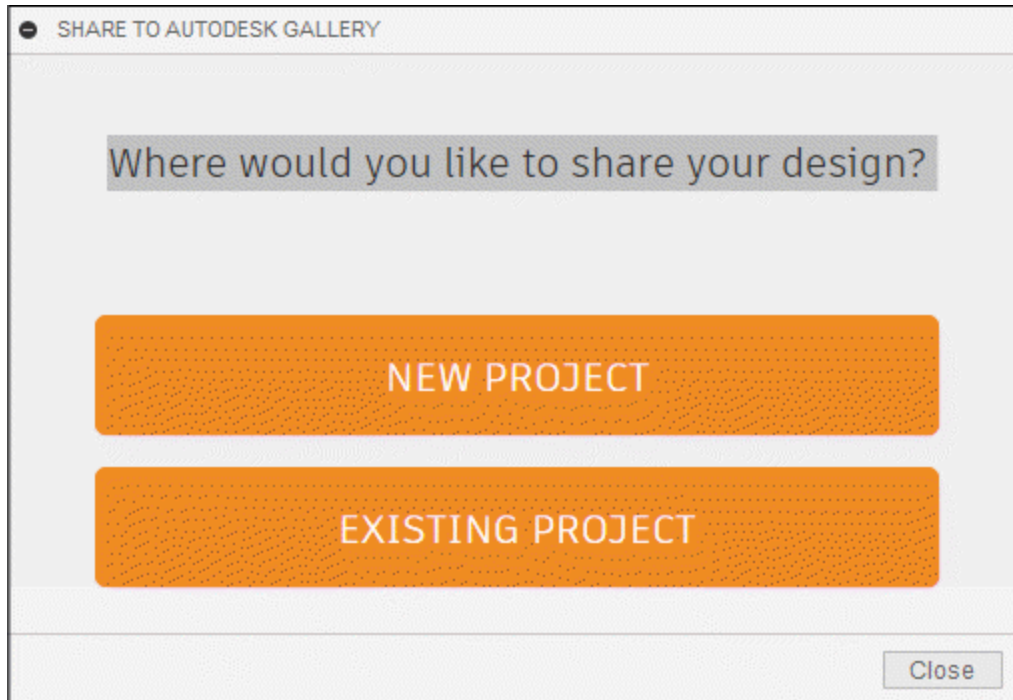


Figure-73. SHARE TO AUTODESK GALLERY dialog box

- Click on the **NEW PROJECT** button from the dialog box. The options will be displayed as shown in [Figure-74](#).

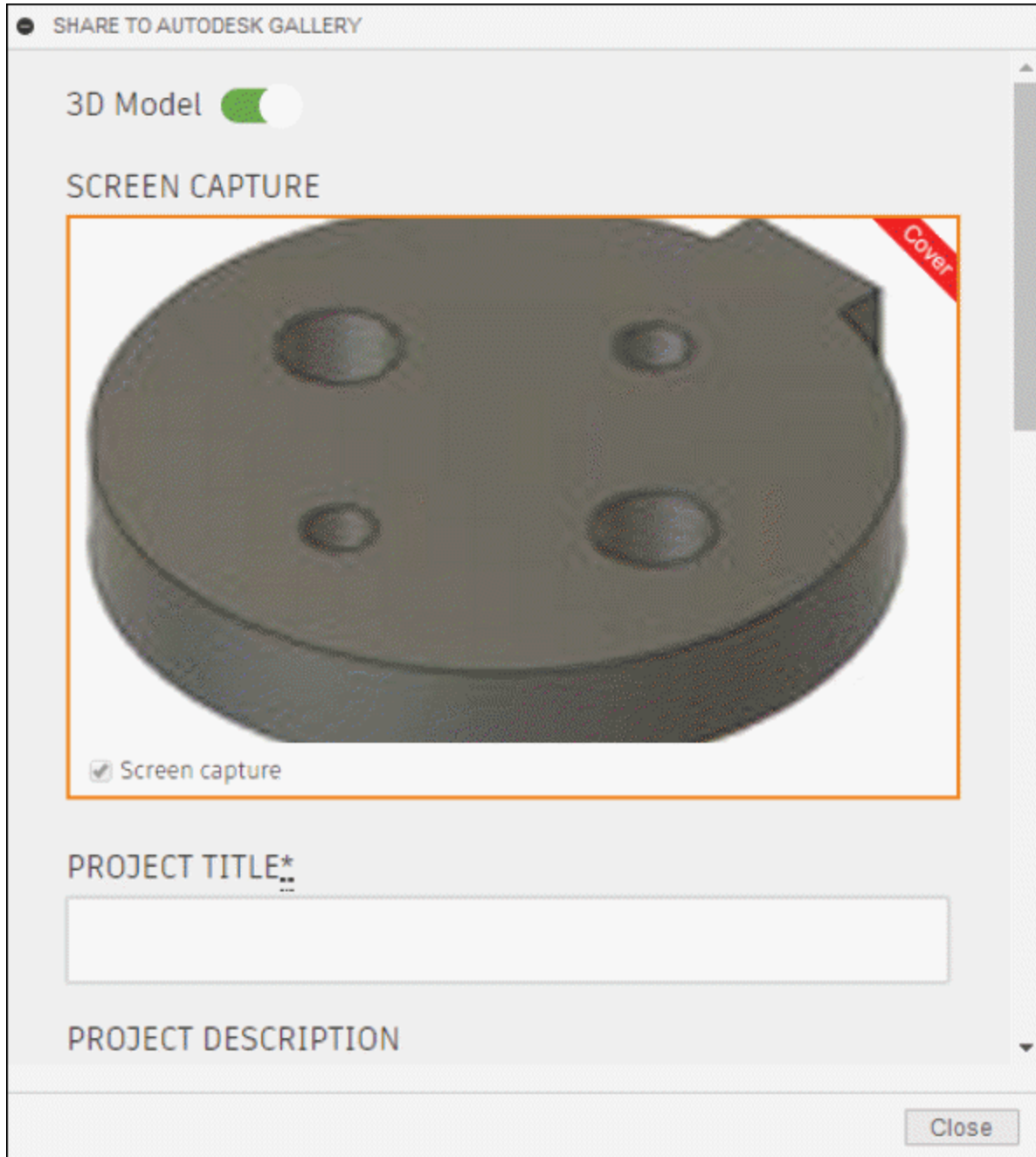


Figure-74. Screen Capture Of 3D Model

- Scroll down in the dialog box and specify the information as required.
- Once all desired information is specified, click on the **Publish** button. **Your project has been successfully**

shared message will be displayed.

- Click on the **Go and take a look!** button to check your model in gallery.

Publish to GrabCad

Autodesk Fusion 360 allows you to publish your model directly to **GrabCad** portal. The procedure to use this tool is given next.

- Click on **GrabCad** tool from **Share** cascading menu. The **PUBLISH TO GRABCAD** dialog box will be displayed; refer to [Figure-75](#).
- Login to GrabCAD platform with your GrabCAD account details to upload or share the project. If you do not have an account on GrabCAD then you can create the account by clicking on **Sign Up** button at the bottom in dialog box and following the instructions.
- Once you have login to your account, the dialog box will be displayed as shown in [Figure-76](#).

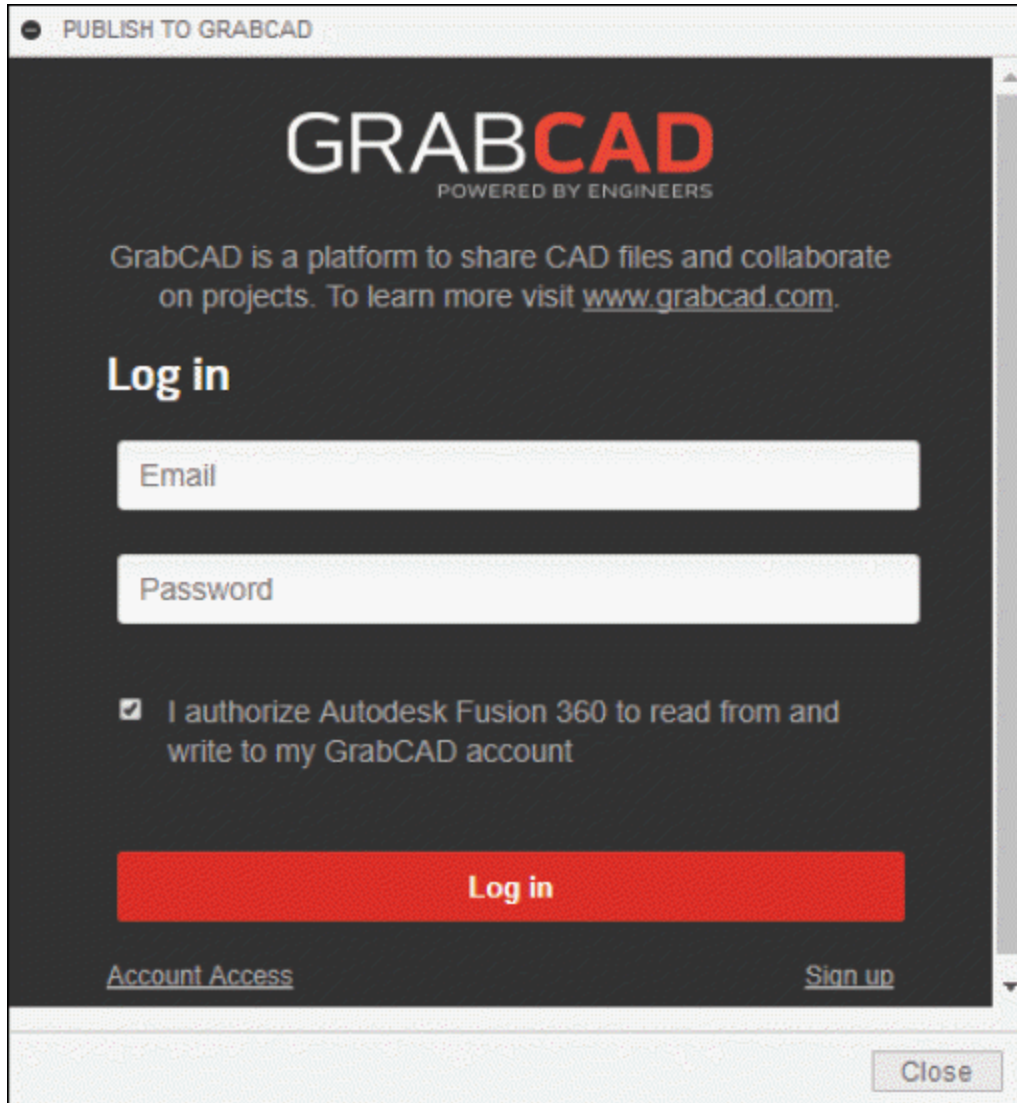


Figure-75. PUBLISH TO GRABCAD dialog box

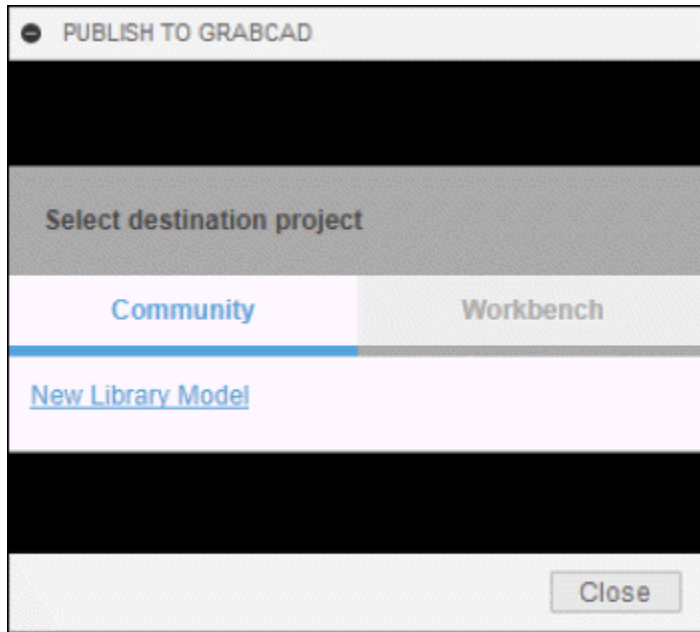


Figure-76. Select Destination Project

- Click on the **New Library Model** link button if you want to save model in community. If you want to share model in a workbench project then click on the **Workbench** tab in the dialog box and then click on the **New Workbench Project** link button. Based on your selection, the respective information will be displayed in the dialog box; refer to [Figure-77](#).

PUBLISH TO GRABCAD

New Library Model

Model name *

Description *

Create model Cancel

Close

Figure-77. New Library Model Details

- Set desired parameters and click on the **Create model** button. In the next page, click on the **Publish Here** button. Your model will be uploaded to GrabCAD.

View details On Web

The **View Details On Web** tool is used to display the details of current file on Autodesk cloud in web browser; refer to [Figure-78](#). You can use this option only after you have saved your file on Autodesk cloud.

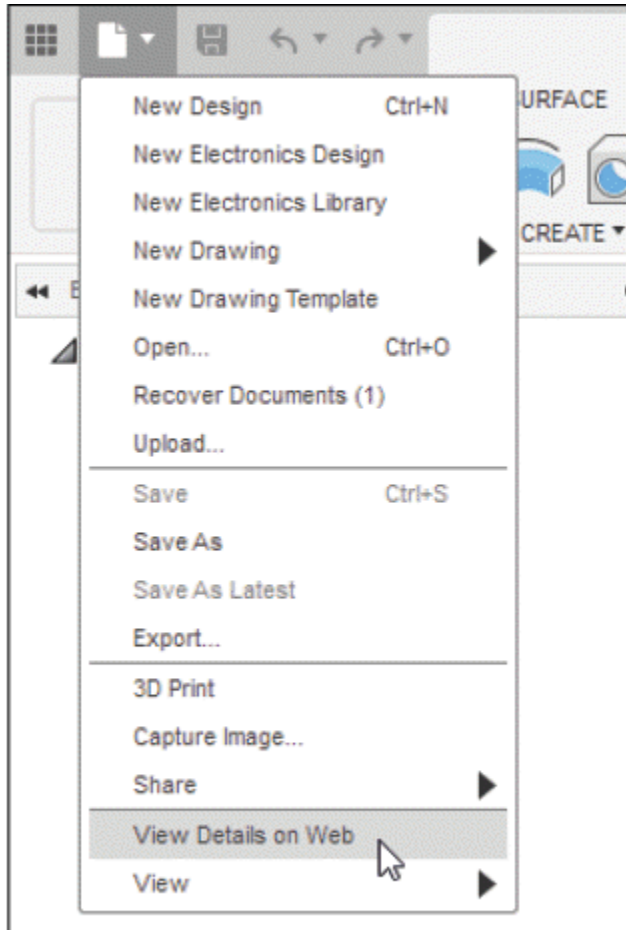


Figure-78. View Details on Web tool

View

The **View** tool is used to show and hide various elements of Fusion 360 application window.

- Click on the **View** tool from **File** menu. The **View** cascading menu will be displayed; refer to [Figure-79](#).

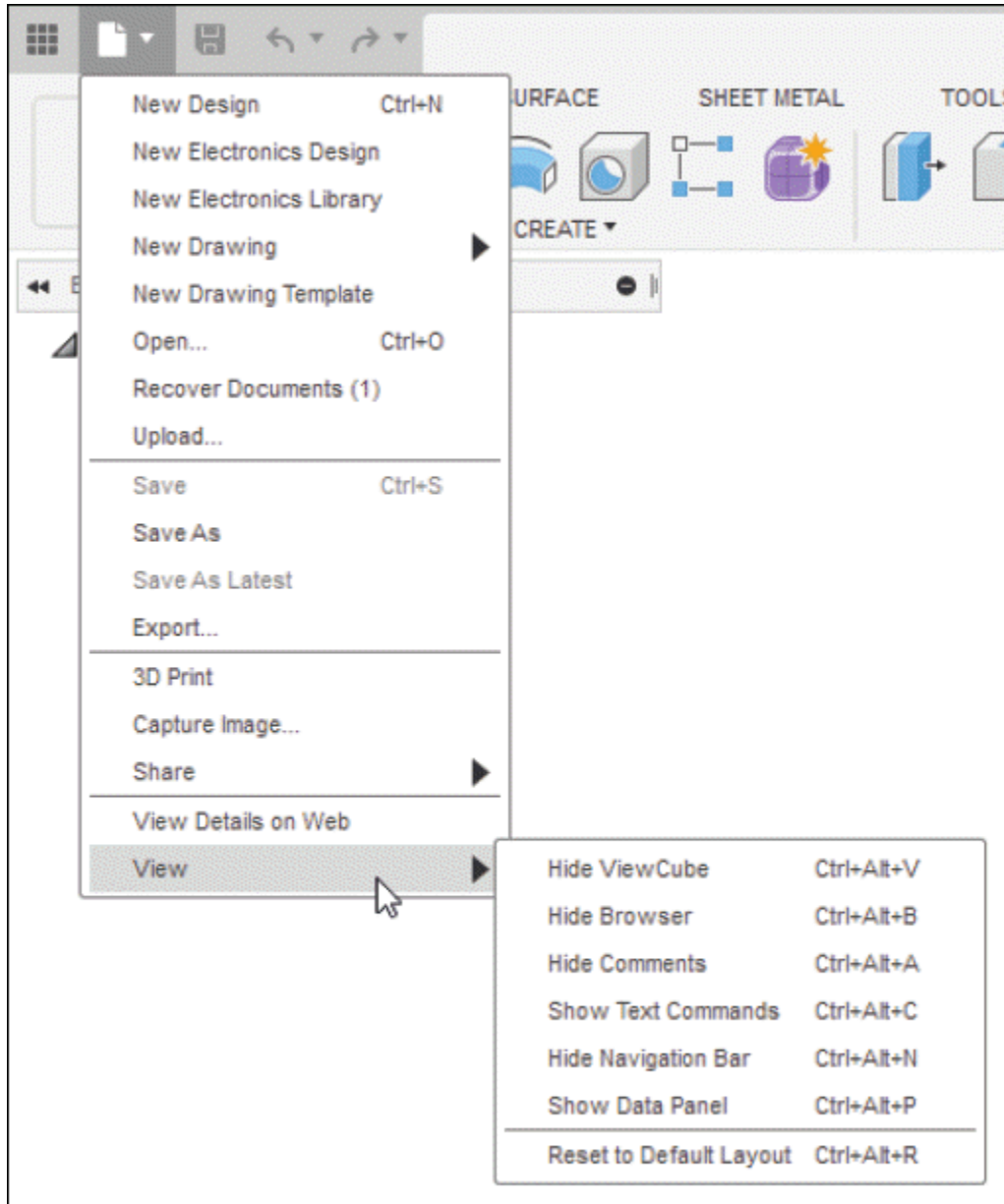


Figure-79. View cascading menu

Various tools in this cascading menu are given next.

Hide ViewCube

The **Hide ViewCube** tool is used to hide the **ViewCube**. If the **ViewCube** is already hidden then **Show ViewCube** tool will be displayed in place of it. On again selecting this tool,

View Cube will be displayed. You can also show/hide **ViewCube** by pressing the **CTRL+ALT+V** keys.

Hide Browser

The **Hide Browser** tool is used to hide the **Browser** from the current screen. If the **Browser** is already hidden then **Show Browser** tool will be displayed in place of it. This tool can also be activated by pressing **CTRL+ALT+B** together.

Hide Comments

The **Hide Comments** tool is used to show and hide the **Comments** from Fusion 360 software. If the **Comment** is already hidden then **Show Comment** tool will be displayed in place of it. This tool can also be used by pressing **CTRL+ALT+A** together.

Show Text Commands

The **Show Text Commands** tool is used to show **Text Commands Bar**. If the **Text Commands Bar** is already displayed then **Hide Text Commands** tool will be displayed in place of it. This tool can also be activated by pressing **CTRL+ALT+C** together.

Hide Navigation Bar

The **Hide Navigation Bar** tool is used to hide the **Navigation bar**. If the **Navigation Bar** is already hidden then **Show Navigation Bar** tool will be displayed in place of it. This tool can also be activated by pressing **CTRL+ALT+N** together.

Show Data Panel

The **Show Data Panel** tool is used to show the **Data Panel**. If the **Data Panel** is already showing then **Hide Data Panel** tool will be displayed in place of it. This tool can also be activated by pressing **CTRL+ALT+P** together.

Reset To Default Layout

The **Reset To Default Layout** tool is used to reset user interface to default settings. This tool can also be used by pressing **CTRL+ALT+R** together.

Scripts and Add-Ins

The **Scripts and Add-Ins** tool is used to run and manage **Scripts** and **Add-Ins** in Autodesk Fusion 360. The procedure to use this tool is discussed next.

- Click on **Scripts and Add-Ins** tool from **ADD-INS** drop-down in **UTILITIES** tab of **Ribbon**. The **Scripts and Add-Ins** dialog box will displayed; refer to [Figure-80](#).

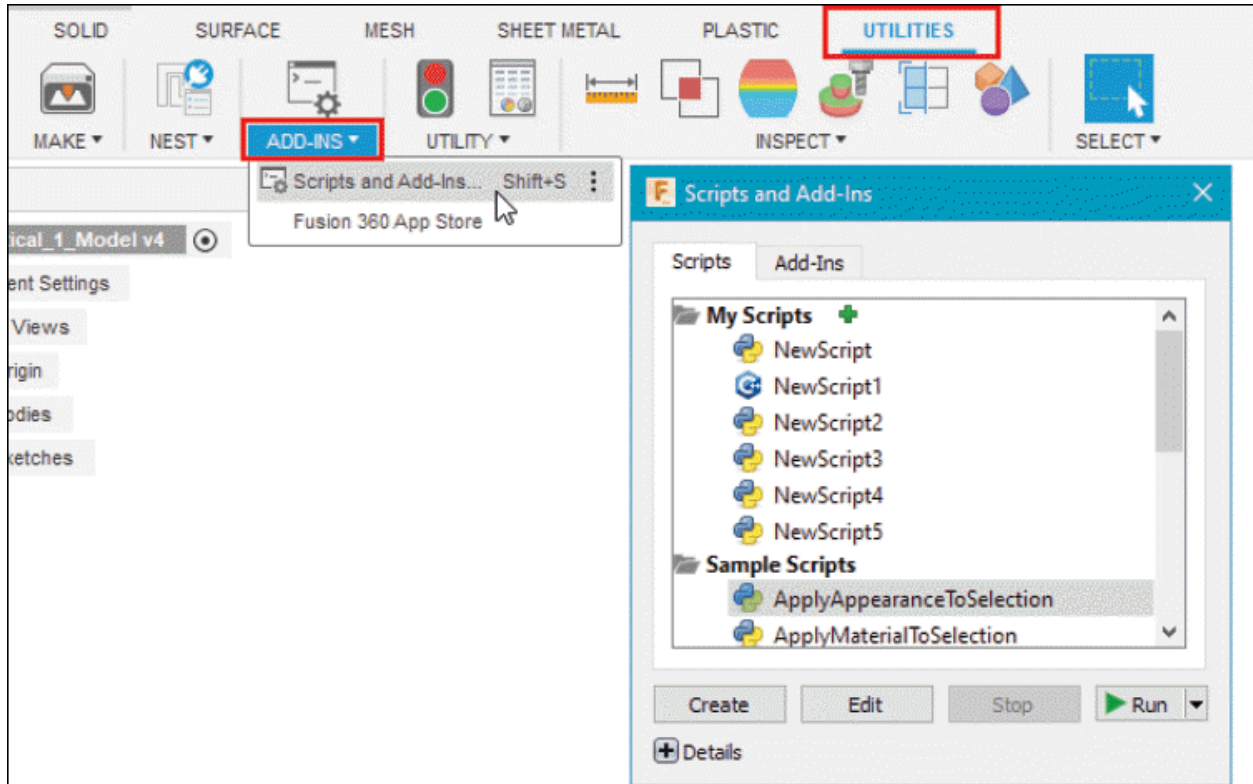


Figure-80. Scripts And Add-Ins

There are two tabs in this dialog box; **Scripts** and **Add-ins**. Select the **Script** tab to run Python scripts in Autodesk Fusion. Select the **Add-Ins** tab to run the third party applications inside Autodesk Fusion 360.

Creating Script and Add-Ins

- To create a script or Add-Ins, click on the **Create** button from **Scripts and Add-Ins** dialog box. The **Create New Script or Add-In** dialog box will be displayed; refer to [Figure-81](#).

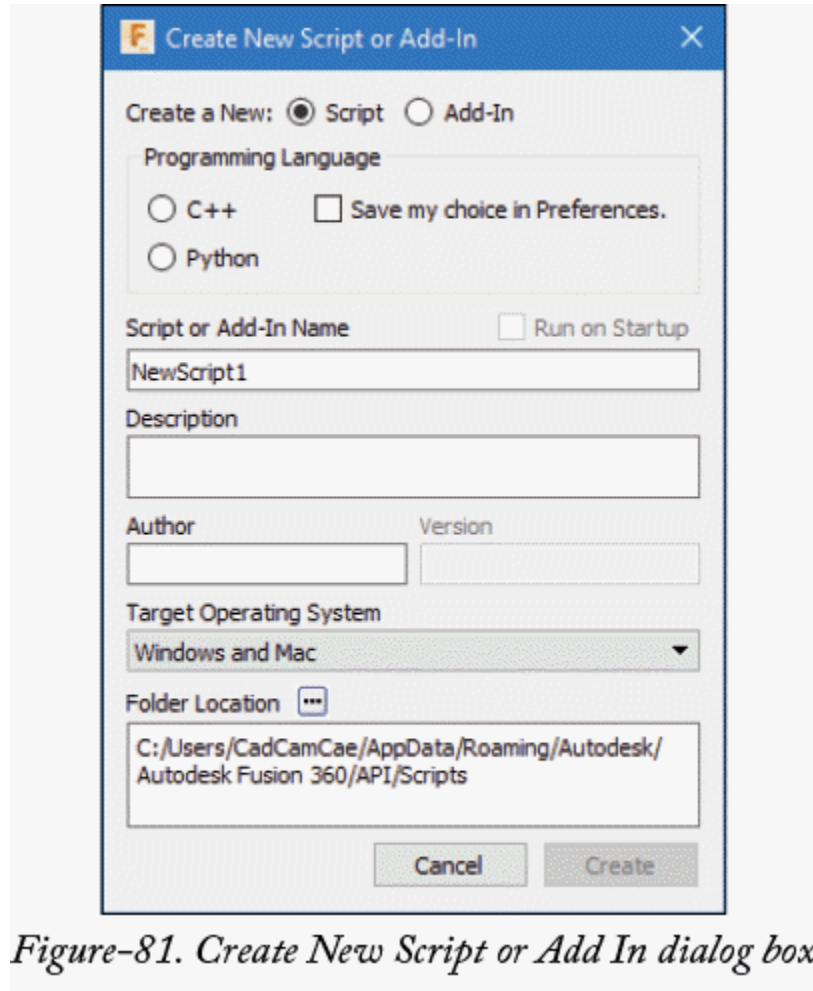


Figure-81. Create New Script or Add In dialog box

- Select **Script** radio button to create a script or select the **Add-In** radio button to create an Add-In.
- Select desired radio button from the **Programming Language** area to define the language in which you will be writing script/Add-in codes.
- Set the other parameters as required and click on the **Create** button. A new script/Add-in will be added in the dialog box.
- Select the newly created script/Add-in and click on the **Edit** button. If the script/Add-in is coded in **Python** then

an information box will be displayed asking you to download and install VS Code. Download & install the application and once installation is complete, codes will be displayed in application where you can edit it; refer to [Figure-82](#).

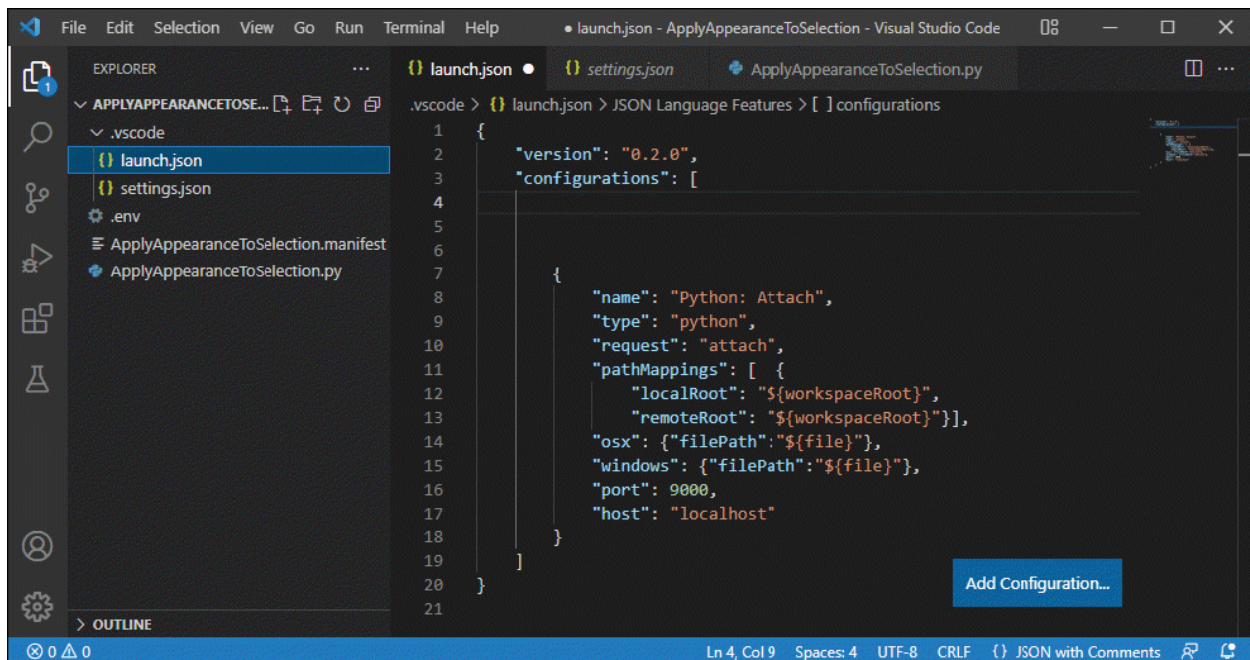


Figure-82. Visual Studio Code application window

- If you have opted for any other language then system will ask you for respective Programming software to display codes. If you like coding then you can check **Learning Python, 5th Edition by Mark Lutz** as reference book for Python programming.
- Click on the **Run** button from the **Scripts and Add-Ins** dialog box to run selected script.
- If you want to debug a script/Add-in then select it from the **Scripts and Add-Ins** dialog box and click on the **Debug**

button in **Run** drop-down; refer to [Figure-83](#). Programming interface will be displayed based on language of selected script/Add-In.

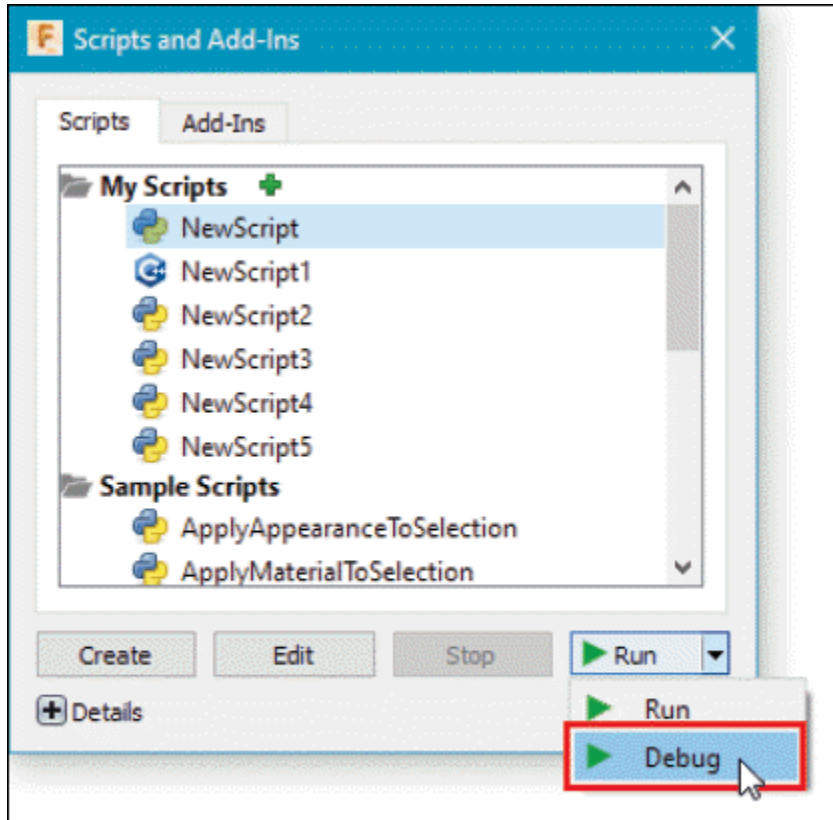


Figure-83. Debug button

Fusion 360 App Store

The **Fusion 360 App Store** tool in **ADD-INS** drop-down of **UTILITIES** tab in **Ribbon** is used to download and install third party Apps in Autodesk Fusion. On clicking this tool, a web page is displayed in your default web browser; refer to [Figure-84](#).

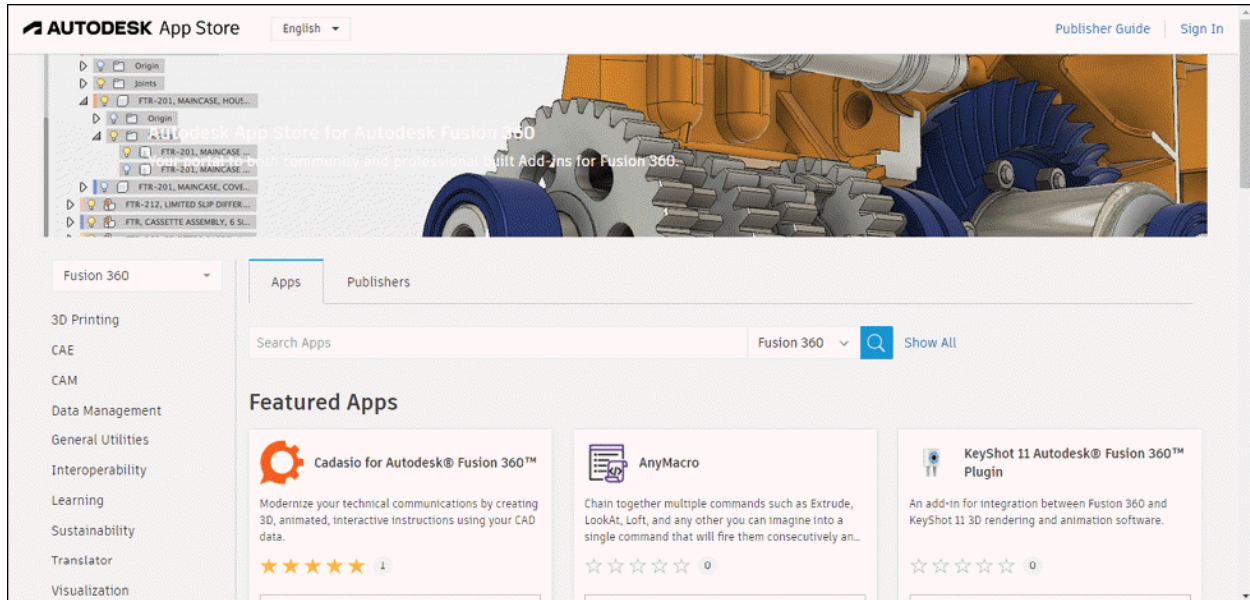


Figure-84. Fusion 360 Store Webpage

- Login to your Autodesk account after clicking **Sign In** button at the top right in the web page.
- Select desired app, set the operating system option, and click on the **Download** button from web page; refer to [Figure-85](#). The software will start downloading.

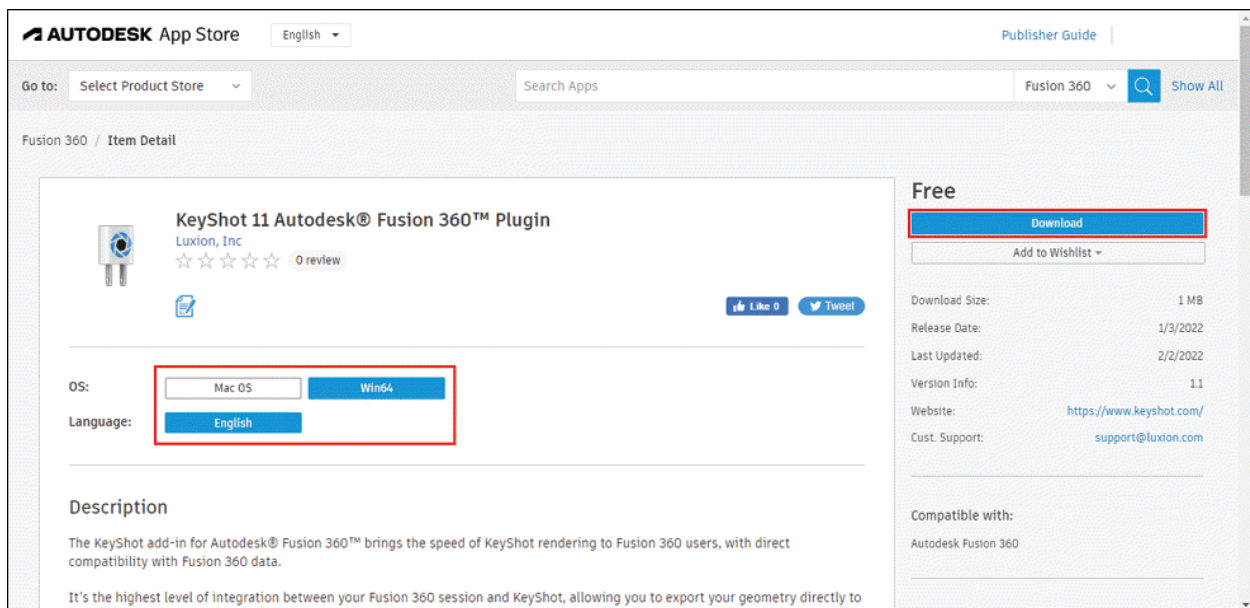


Figure-85. Downloading App

- Once downloaded, double-click on the setup file to install it and follow the instructions as displayed during installation.
- Once the app is installed, you can access it from **Add-Ins** tab of **Scripts and Add-Ins** dialog box; refer to [Figure-86](#).

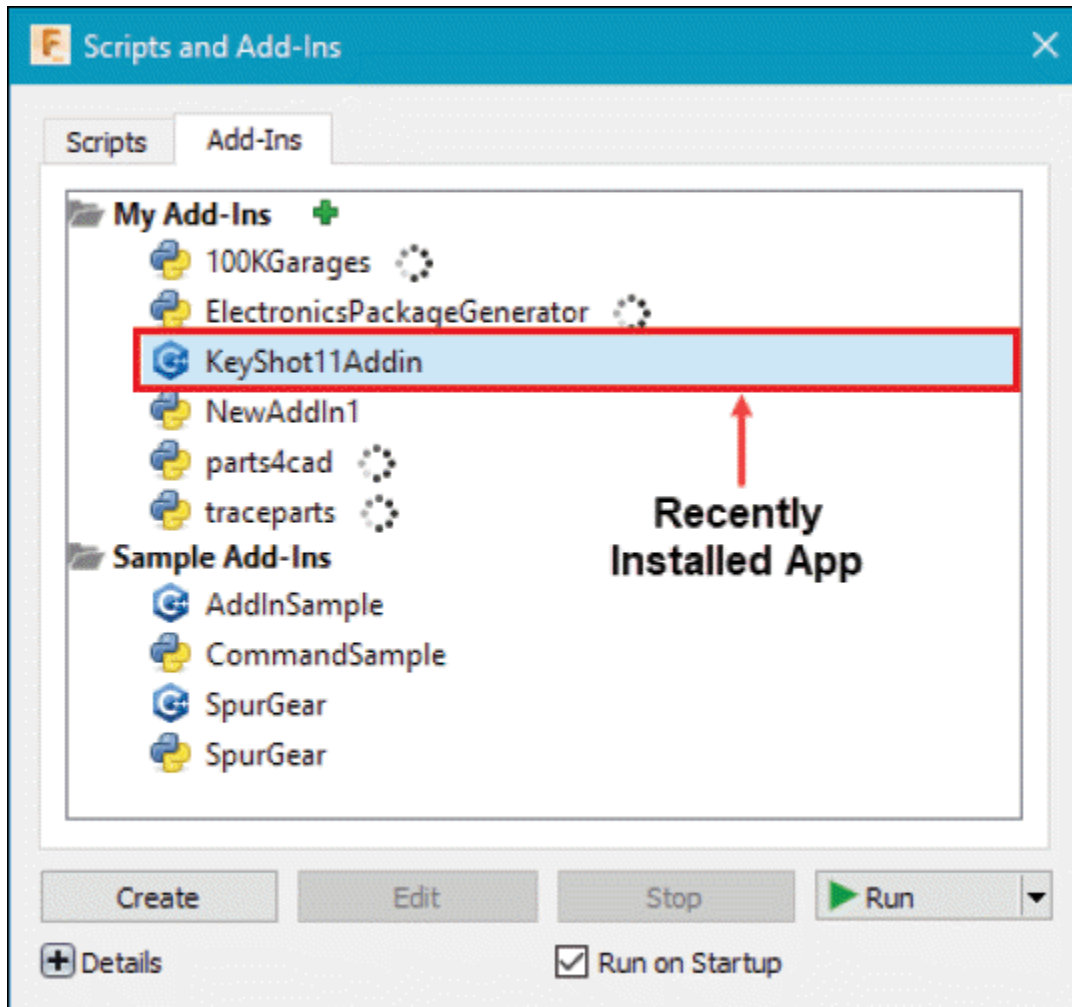


Figure-86. Installed App

UNDO AND REDO BUTTON

The **Undo** tool is used to revert back to condition before performing most recent action. The procedure to use this tool is discussed next.

- Click on **Undo** tool in **Application bar** or press **CTRL+Z** keys; refer to [Figure-87](#).
- You can't undo some actions, like clicking commands on the **File** tab or saving a file.

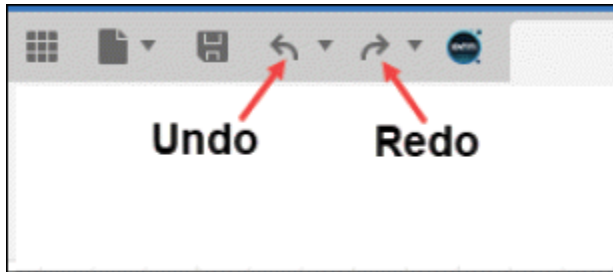


Figure-87. Undo And Redo Button

- The **Redo** tool is used to redo an action reverted by **Undo** tool. To use this tool, select the **Redo** button in **Application bar** or press **CTRL+Y** keys.
- The **Redo** button activates only after you have undone an action.

USER ACCOUNT DROP-DOWN

The tool in **User Account** drop-down are used to manage user account details and preferences for Autodesk Fusion 360; refer to [Figure-88](#). The tools in this drop-down are discussed next.

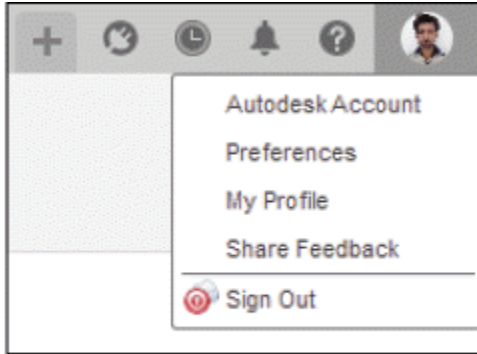


Figure-88. User Account drop down

Preferences

The **Preferences** tool is used to set the preferences for various functions of the software like you can set units, material libraries, display options, and so on. The procedure to use this tool is given next.

- Click on **Preferences** tool from **User Account** drop-down; refer to [Figure-88](#). The **Preferences** dialog box will be displayed where you can specify various parameters for application; refer to [Figure-89](#).

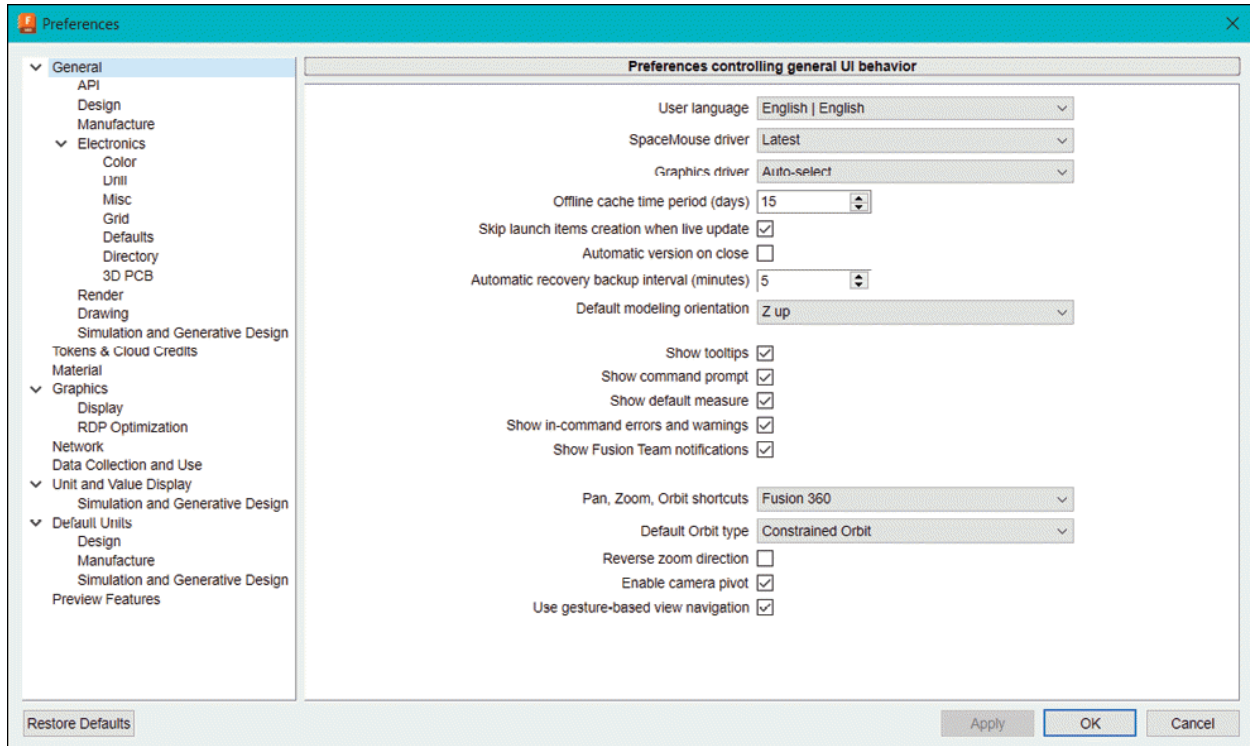


Figure-89. Preferences dialog box

General

In **General** node, you can specify the preferences for language, graphics, mouse functioning, default workspace, and so on. Click on the **General** node at the left in the dialog box to display general options. Various important options are discussed next.

- Click on the **User language** drop-down and select desired language for Autodesk Fusion interface.
- Select desired version of the SpaceMouse SDK used by the Fusion 360 from **SpaceMouse driver** drop-down.
- Click on the **Graphics driver** drop-down to select the graphics driver to be used by Autodesk Fusion.

- **Offline cache time period (days)** edit box/spinner is used to specify the number of days up to which your documents can be in the cache memory of Autodesk Fusion before you go back to online mode of Autodesk Fusion. If you specify high value then more local memory will be used by Autodesk Fusion to save temporary copy of your documents. If you specify a low value here then system will prompt you soon to go back online. The maximum number of days specified here can be 360 and minimum number of days can be 7.
- Whenever Autodesk Fusion is updated, launch icons are created automatically in various locations. Select the **Skip launch items creation when live update** check box to skip creating new launch icons.
- Select the **Automatic version on close** check box to automatically save the newer version of file when you close Autodesk Fusion.
- Specify desired time in minute after which a recovery copy of your model will be created automatically in the **Recovery time interval (min)** edit box.
- Click on the **Default modeling orientation** drop-down and select desired direction option to define default orientation of model. In most of the CAD software, Z axis upward is the default orientation of model. To set the common orientation, select the **Z up** option from the drop-down.

- Select the **Show tooltips, Show command prompt, Show default measure, Show in-command errors and warnings,** and **Show Fusion Team notifications** check boxes to display respective interface elements.
- Select desired software style from the **Pan, Zoom, Orbit shortcuts** drop-down to define which software style for shortcuts of pan, zoom, and orbit should be used. If you are switching from Alias, Inventor, Solidworks, or Tinkercad to Autodesk Fusion then you can select respective option from the drop-down to use familiar shortcuts.
- Select desired option from the **Default Orbit type** drop-down to define the default orbit type for rotating model view. There are two options available for orbit; **Constrained Orbit** and **Free Orbit**.
- Select the **Reverse zoom direction** check box to reverse the zoom direction of mouse scroll and other zoom methods.
- Select the **Enable camera pivot** check box to display camera pivot point used as center for orbiting.

Note that by default the pivot point is placed on center of mass point of model but if you want to change the position of pivot point then right-click in blank area of canvas and select the **Set Orbit Center** option. The green point will get attached to cursor and you will be asked to define the location of orbit point. Click at desired location to place the orbit center.

API Options

- Click on the **API** option under **General** node in the left of the dialog box and set desired parameters to define default location and language of scripts/Add-Ins.

Design Options

- Click on the **Design** option under **General** node in the left of the dialog box to set the parameters related to model creation. The options in the dialog box will be displayed as shown in [Figure-90](#).
- Select the **Active Component Visibility** check box to display only currently active component and hide rest of the components.
- Select the **Capture Design History** option from **Design History** drop-down if you want to keep detail of every operation you perform on the model in model history. If you want to save space or want to hide operation details while sharing file then create the model after selecting **Do not capture Design History** option from the **Design History** drop-down. Note that changes made in most of the options will be applied only after you have restarted Autodesk Fusion.

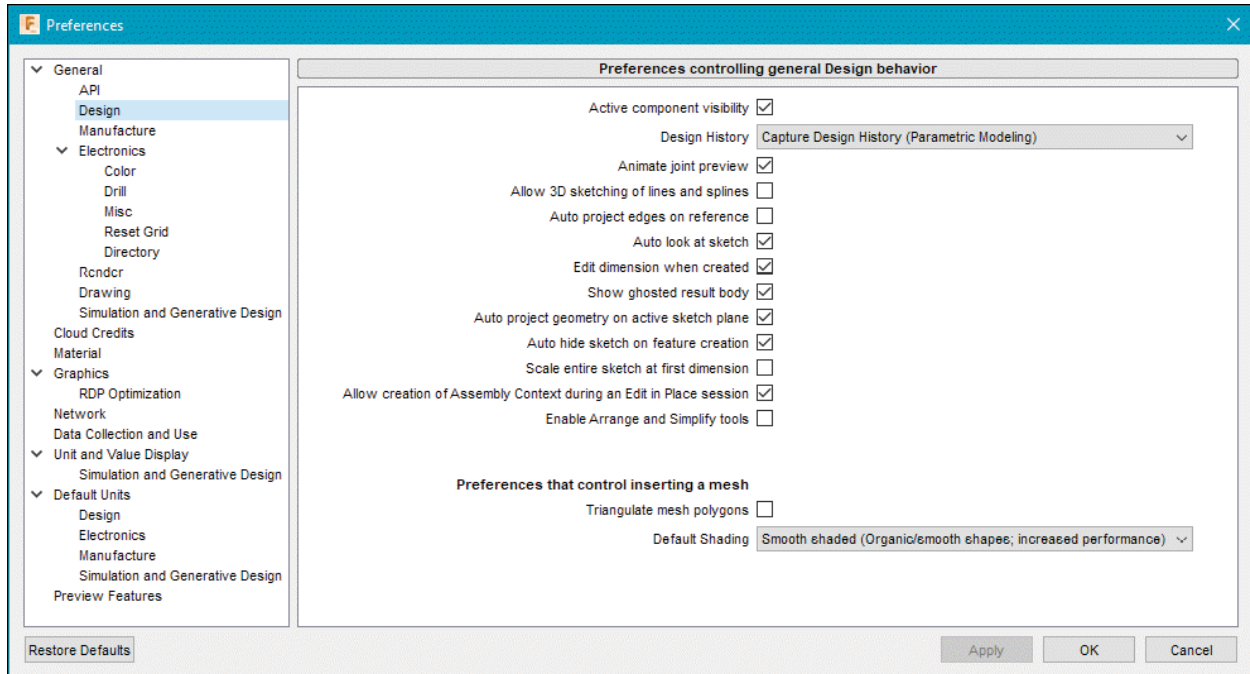


Figure-90. Design Page

- Select the **Animate joint preview** check box if you want to check small animation of how joint will work. The animation can give you clues about degrees of freedom which are constrained by selected joint.
- Select the **Allow 3D sketching on lines and splines** check box if you want to create lines and splines which are not confined to single plane.
- Select the **Auto project edges on reference** check box if you want to automatically project edges of model when they are used as reference for dimensioning or constraining of other sketch elements. Refer to [Figure-91](#).

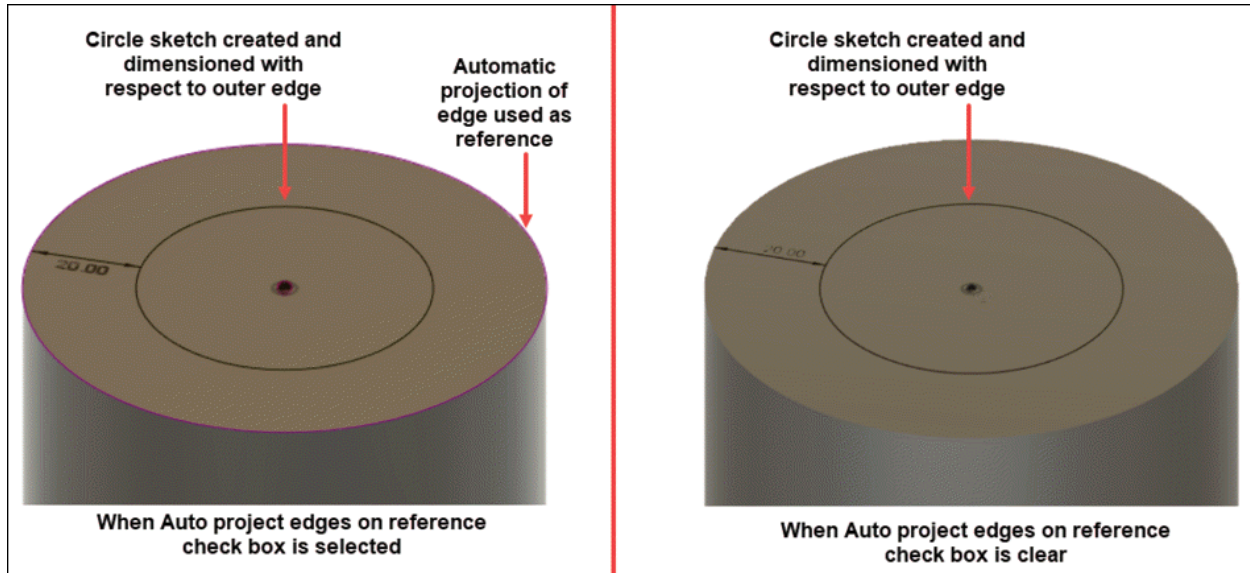


Figure-91. Use of Auto project edges on reference check box

- Select the **Auto look at sketch** check box to automatically make sketching plane parallel to screen.
- Select the **Edit dimension when created** check box if you want to edit the values of dimensions when you are applying dimensions to objects.
- Selecting the **Show ghosted result body** check box will create a ghost image of model before a sculpting operation is performed on it. Note that the ghost image will be displayed until you are in sculpt mode; refer to [Figure-92](#). When you exit the model after making changes then the ghost image will not be displayed.

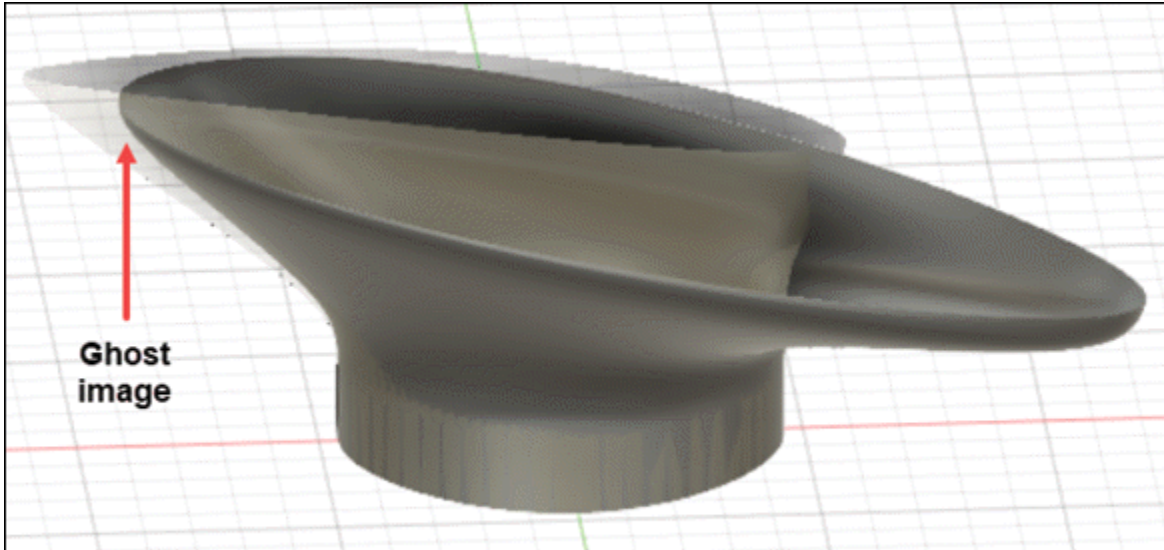


Figure-92. On selecting Show ghosted result body check box

- Select the **Auto project geometry on active sketch plane** check box if you want to project geometry on the sketching plane by default.
- Select the **Auto hide sketch on feature creation** check box to automatically hide the sketch after it has been used to create a feature.
- Select the **Scale entire sketch at first dimension** check box to automatically fit the sketch objects in view area with reduced dimensions after you specify first dimension of the sketch; refer to [Figure-93](#). Note that all the other objects reduce in same ratio as the first dimension reduces its respective entity.

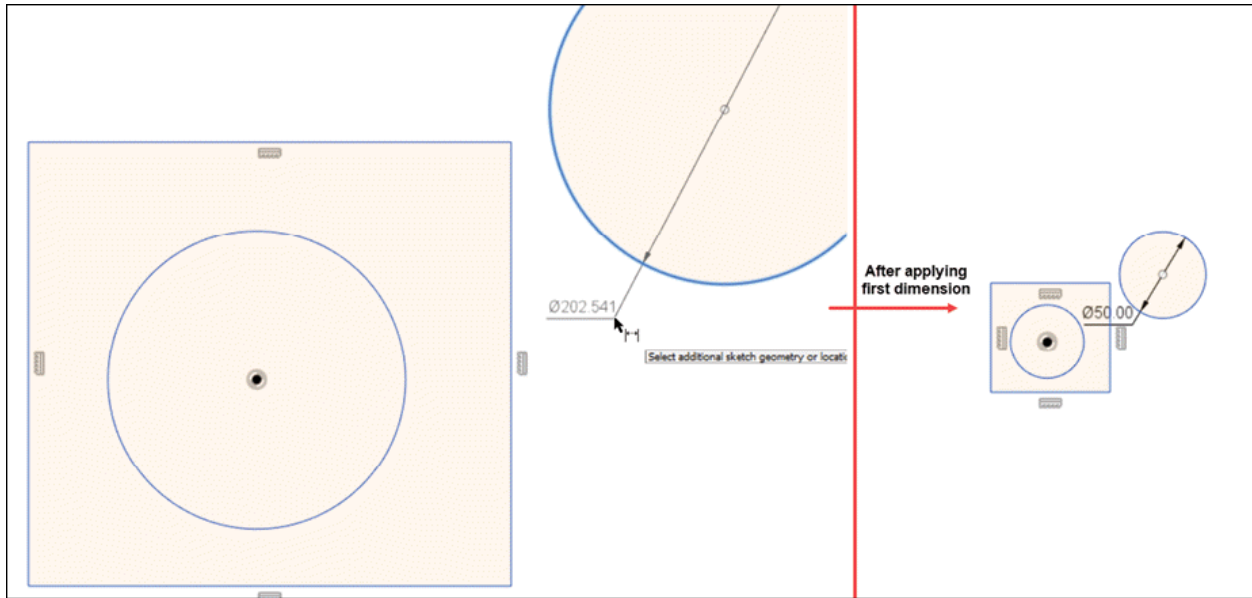


Figure-93. When scale entire sketch at first dimension check box is selected

- Similarly, set the other options as required on different nodes of the dialog box. We will work with options in this dialog box during rest of the book whenever a change in application settings is required.
- After setting the parameters, click on the **OK** button from the dialog box and restart Autodesk Fusion if needed.

Autodesk Account

The **Autodesk Account** tool in **User Account** drop-down is used to manage your Autodesk account profile in default web browser.

My Profile

The **My Profile** tool in **User Account** drop-down is used to manage your project data stored on cloud via web browser.

Work Offline/Online

The **Work Offline/Online** toggle button is displayed on

clicking **Job Status** button at the left of **User Account** drop-down; refer to [Figure-94](#). Click on this button to toggle between offline and online mode of Autodesk Fusion.

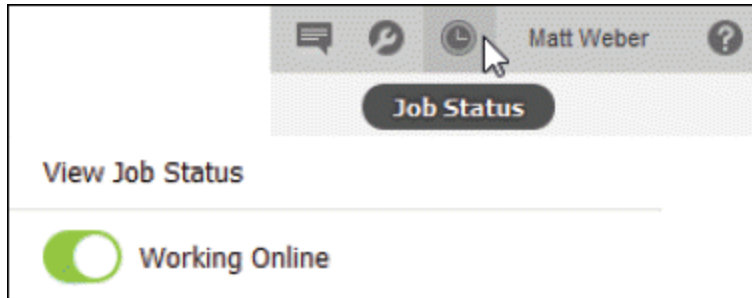


Figure-94. Work Offline Online

Extensions

Click on the **Extensions** button to display the list of current available extensions in the **Extension Manager** dialog box for adding more functionality to Autodesk Fusion; refer to [Figure-95](#). If you want to use these extensions then select the extension to be used and click on **Purchase** button or activate button if available.

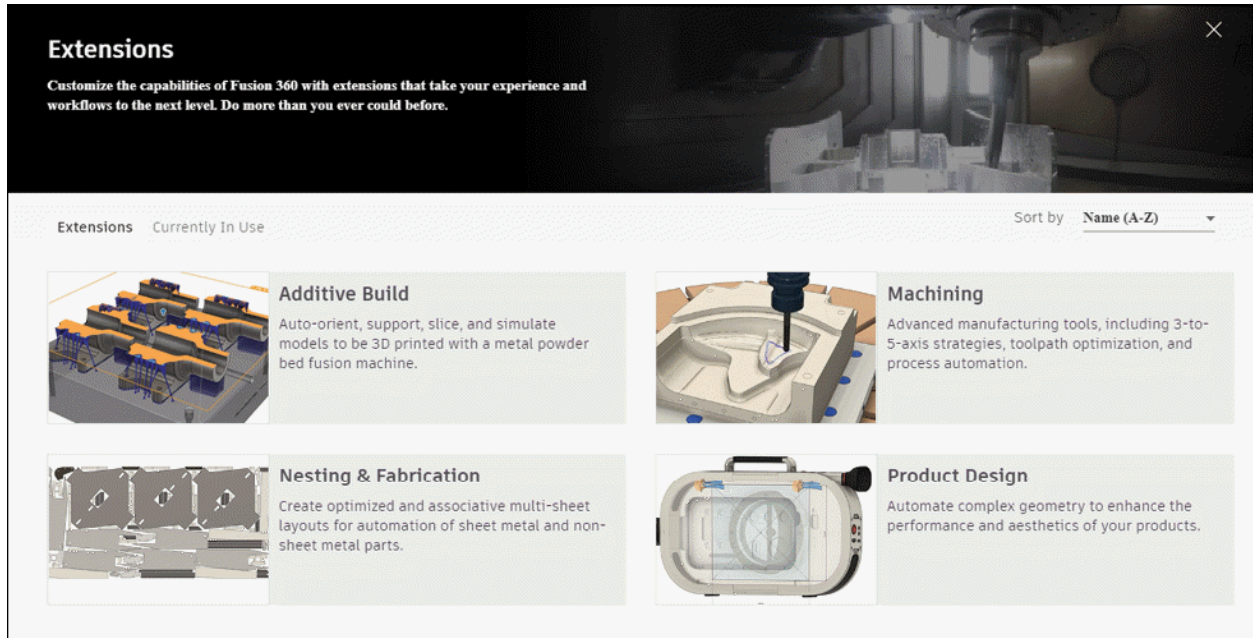


Figure-95. Extension Manager dialog box

HELP DROP-DOWN

When you face any problem working with Autodesk Fusion 360, the tools in **Help** drop-down can be useful; refer to [Figure-96](#). Tools in the drop-down are given next.

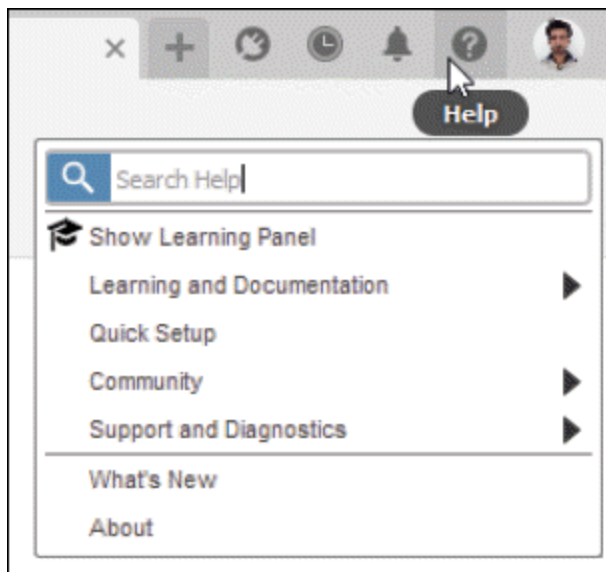


Figure-96. Help drop down

Search Help Box

When you are facing any problem regarding tools and terms in the software, then you need to type keywords for your problem in **Search Box** and press **ENTER** to find a suitable solution to your problems; refer to [Figure-97](#).

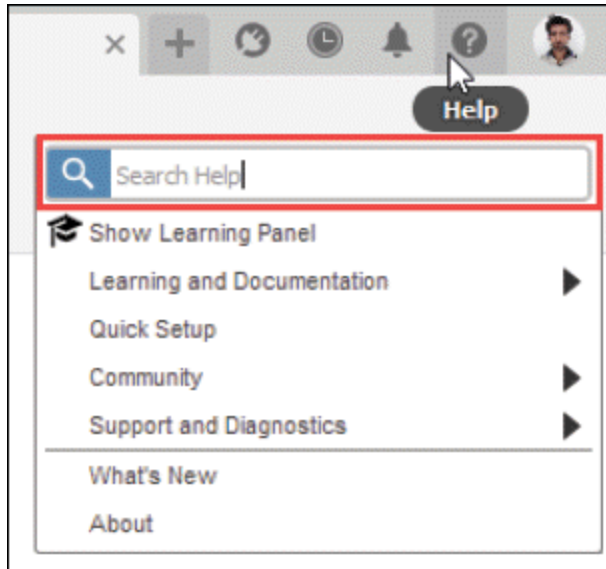


Figure-97. Search Box

Learning and Documentation

The tools in the **Learning and Documentation** cascading menu are used to display video tutorials, help files, and fusion 360 api related content in web browser; refer to [Figure-98](#). On clicking the **Self-Paced Learning** button in the **Learning and Documentation** cascading menu of **Help** menu, the website of Autodesk Fusion 360 will be displayed; refer to [Figure-99](#).

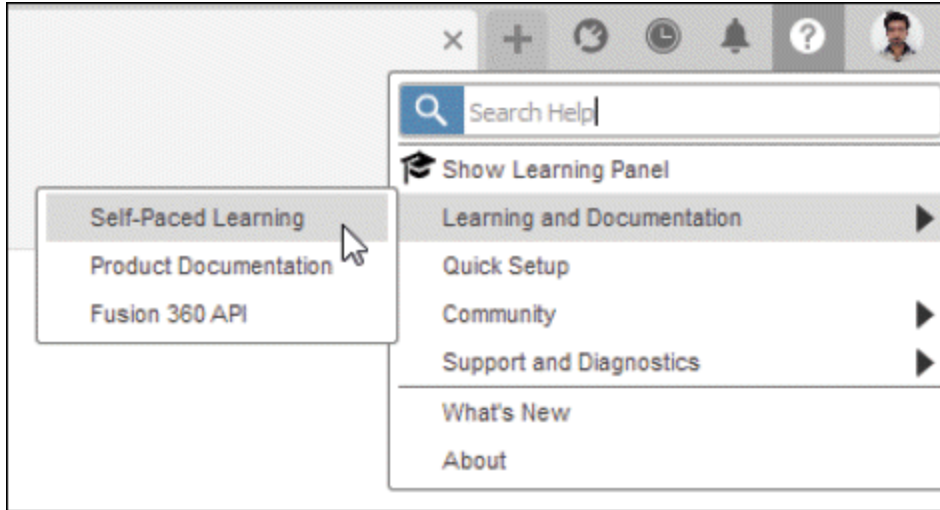


Figure-98. Learning and Documentation

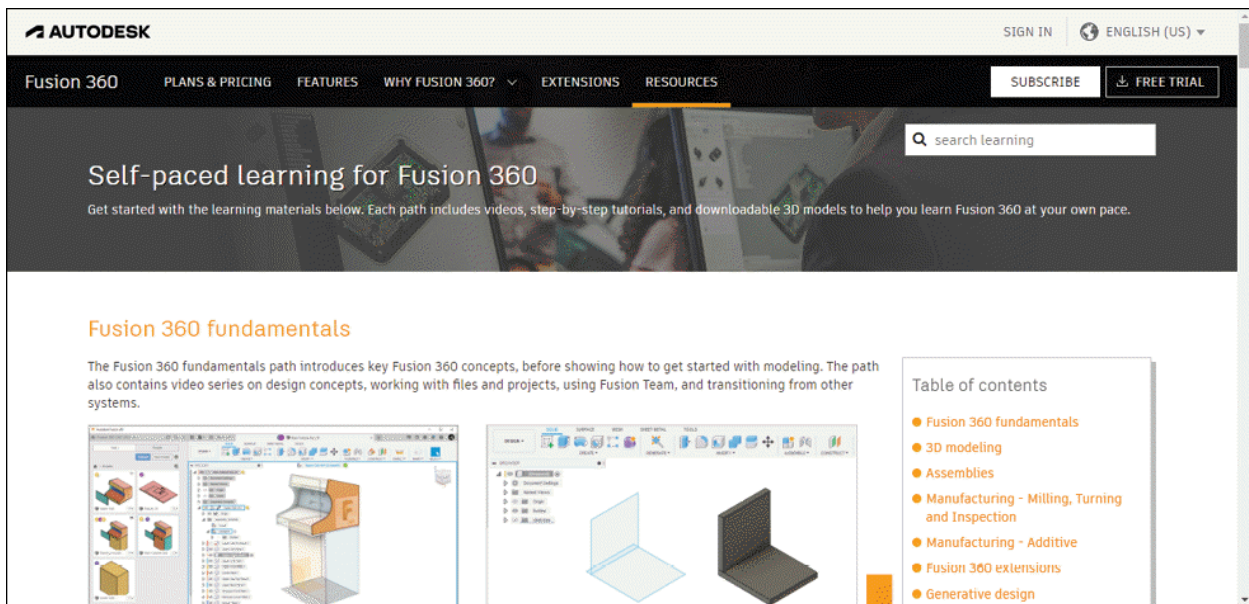


Figure-99. Fusion 360 Learning Website

Similarly, you can use **Product Documentation** and **Fusion 360 API** tools to learn about Fusion 360 interface, design methodology, and APIs.

Quick Setup

When you are new to Autodesk Fusion 360 then this tool will help you to understand this software terminology and basic

settings.

- Click on **Quick Setup** tool from **Help** menu. The **QUICK SETUP** dialog box will be displayed; refer to [Figure-100](#).

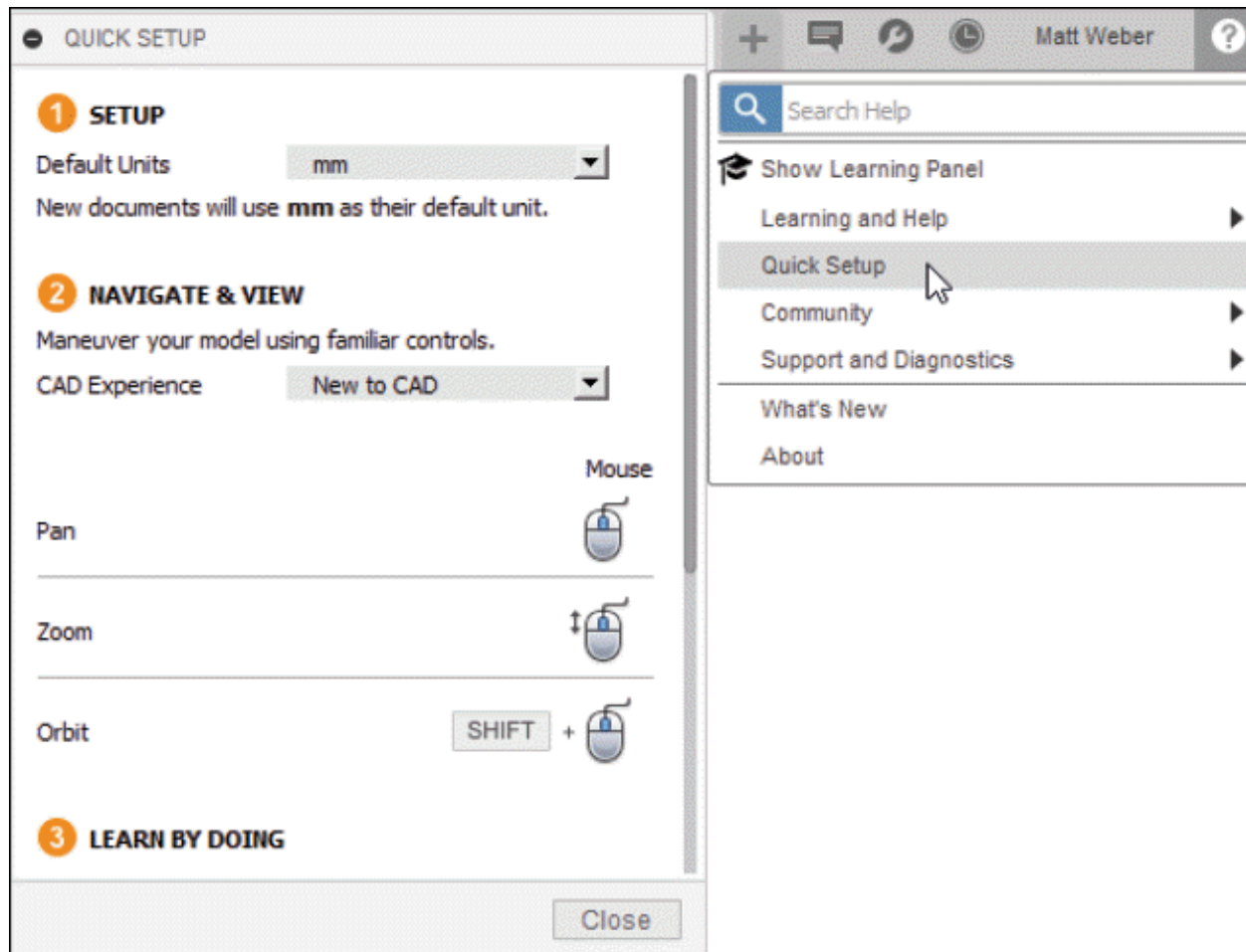


Figure-100. Quick Setup

- Set desired unit in the **Default Units** drop-down.
- The navigation functions of mouse can be set as per Autodesk Inventor, SolidWorks or Alias software by selecting respective option from the **CAD Experience** drop-down.
- If you are new to CAD then you can select the **New to CAD**

option in **Cad Experience** drop-down. The mouse will function as per Autodesk Fusion 360.

Similarly, you can use **Community Forum, Feedback Hub, Gallery, Roadmap, Blog** tool to get help from Autodesk Fusion 360 users or provide feedback.

Support and Diagnostics

When you are having a problem in Fusion 360 and want **Technical support** team to look at software problem data, then you can create the log files and send it to the support team of Autodesk.

- To create log files, select **Diagnostic Log Files** tool under the **Support and Diagnostics** cascading menu in **Help** drop-down. The **Diagnostic Log Files** dialog box will be displayed; refer to [Figure-101](#).

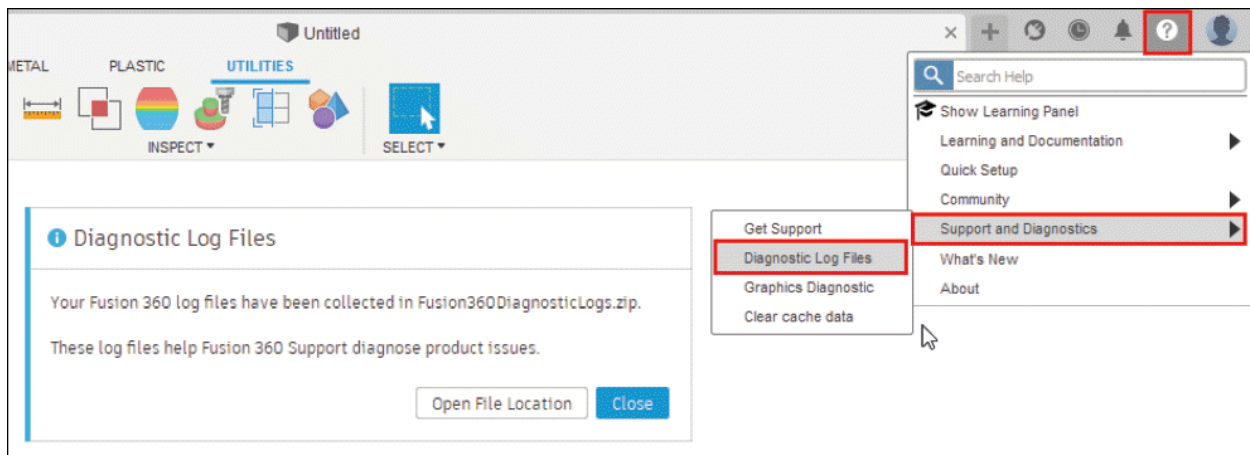



Figure-101. Diagnostic Log Files dialog box

- Click on **Open File Location** button and select the log file which you want to send to technical support team for the solution of problem.

Similarly, you can use **Graphics Diagnostic**, **Clear user cache data**, and other tools from **Help** drop-down in Autodesk Fusion 360.

DATA PANEL

The **Data** panel is used to manage Autodesk Fusion 360 projects. The projects you save in cloud are shown in the **Data** panel. You can easily access the saved project files from anywhere and anytime with the help of internet access.

To show or hide the **Data** panel, click on the **Show Data Panel** or **Hide Data Panel** button  at upper left corner of the Fusion 360 window; refer to [Figure-102](#). A box will be displayed with files of current project. In this box, there are two sections; **Data** and **People**.

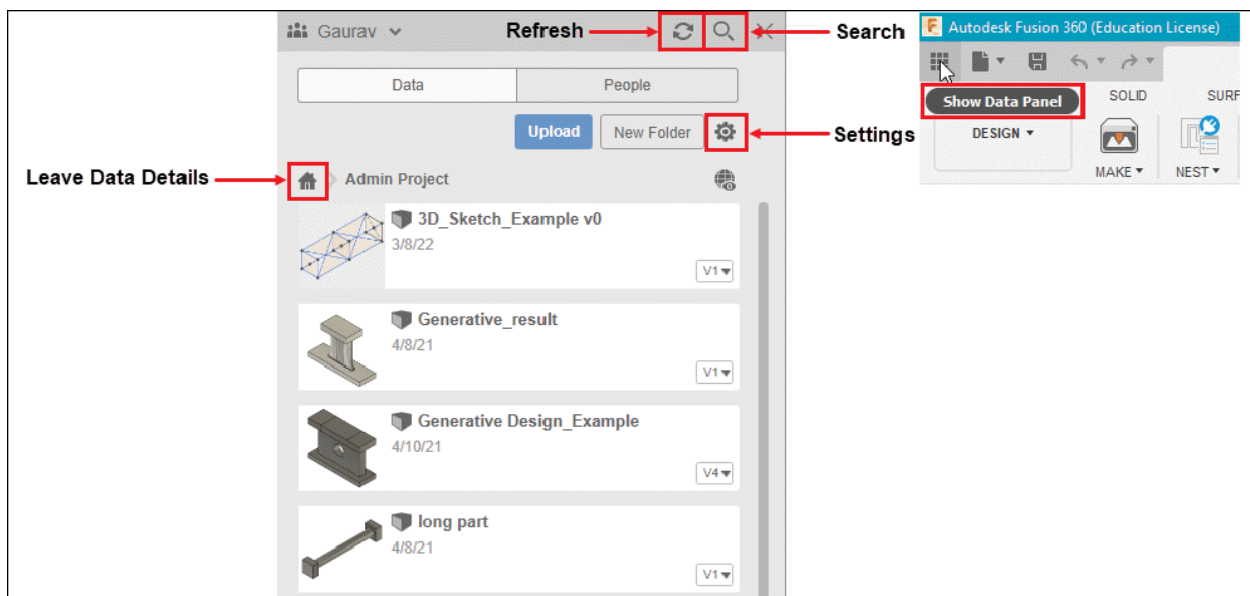


Figure-102. Data Panel

In **Data** section, the files which you have saved earlier in current project will be displayed. You can open any file by

double-clicking on it.

In **People** section, the information of users will be displayed who have access the files of current project; refer to [Figure-103](#).

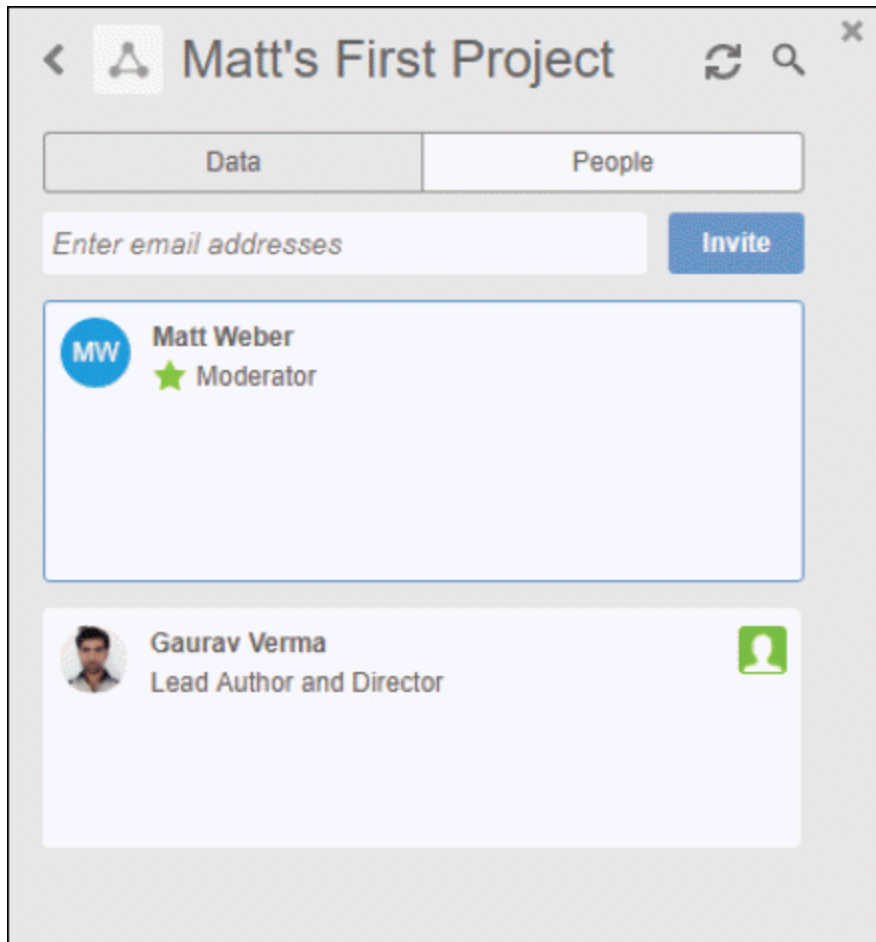


Figure-103. People Section In Data Panel

Working in a Team

Using the tools in **User** drop-down of **Data Panel**, you can create a new team or work in other's team to create designs in collaboration with others; refer to [Figure-104](#). Various tools of this drop-down are discussed next.

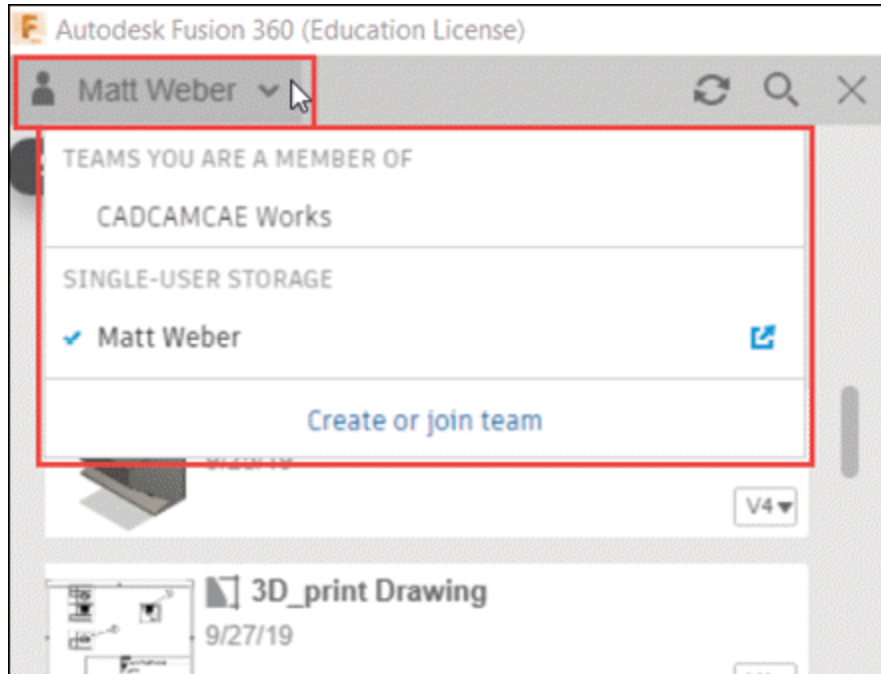


Figure-104. User drop-down

Creating a Team

- To create your own team for a given project, click on the **Create or join team** tool from the drop-down. An information box will be displayed about Fusion Team; refer to [Figure-105](#). Note that this information box will be displayed for only one time when you are creating team using educational version of software as Educational version can create only 1 team for students & educators.

Create or Join Team

In Fusion 360, everything happens inside a team. A team is a collaborative environment where you can store design data and work on your own or with collaborators.

Whether you are the only member of your team or collaborating, you are always in control of your data and can define who can access it.



Close

Next

Figure-105. Create or join team information box

- Click on the **Next** button from the information box. The options to create or join teams will be displayed in a dialog box; refer to [Figure-106](#).

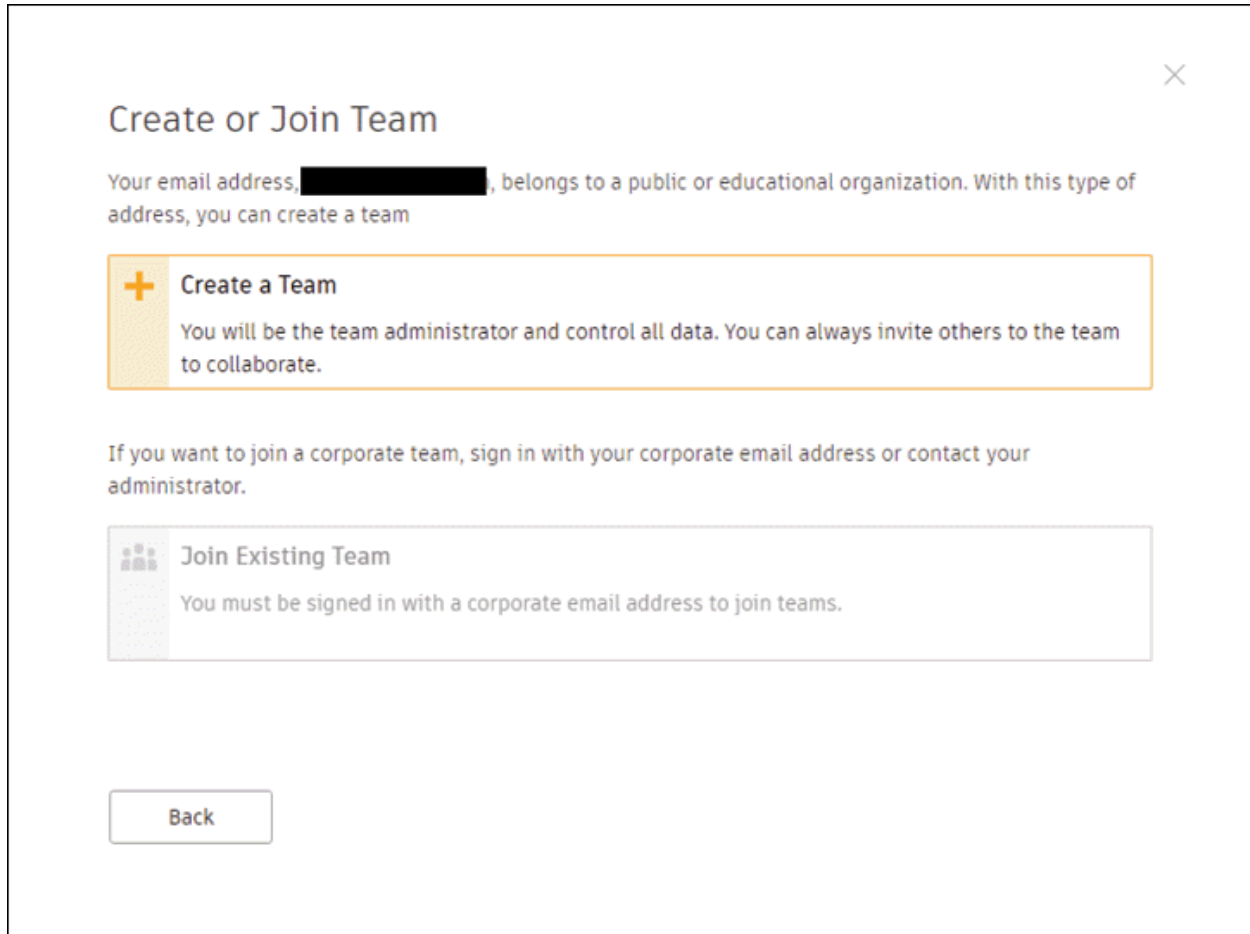


Figure-106. Create a team or join existing team options

- Click on the **Create a Team** option from the dialog box displayed. You will be asked to specify name of the team in next page of dialog box; refer to [Figure-107](#).

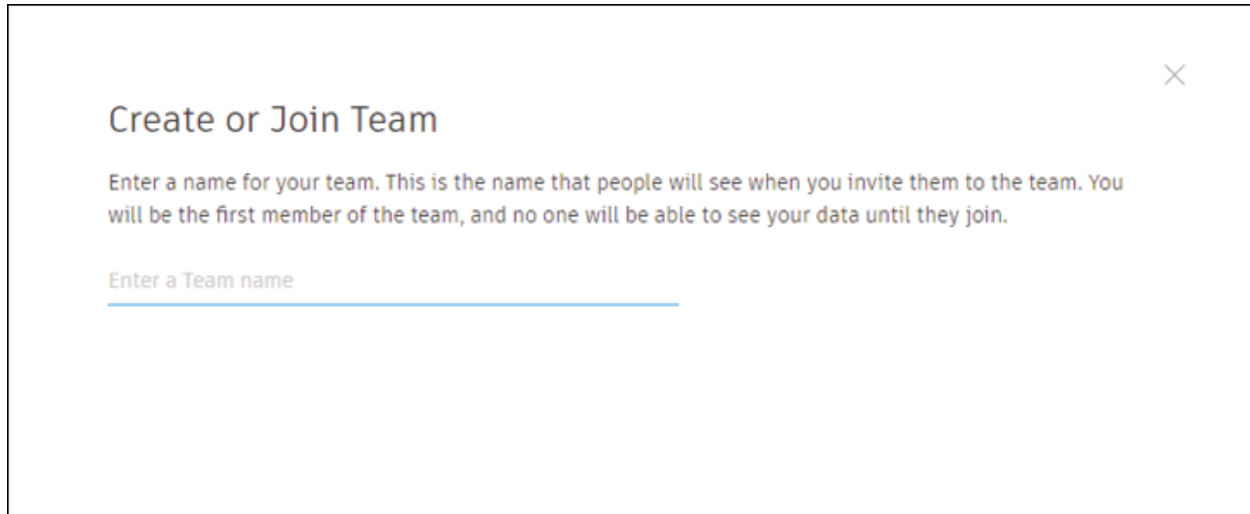


Figure-107. Specifying name of team

- Specify desired name for the team to be created and click on the **Next** button. The next page in dialog box will ask you whether to make the team discoverable on cloud server or make it private.
- Select desired radio button from the dialog box and click on the **Create** button. A message box will be displayed tell you that a team with your specified name is created. Click on the **Go to team** button from the information box. Name of the team will be added in the **Data Panel**; refer to [Figure-108](#).

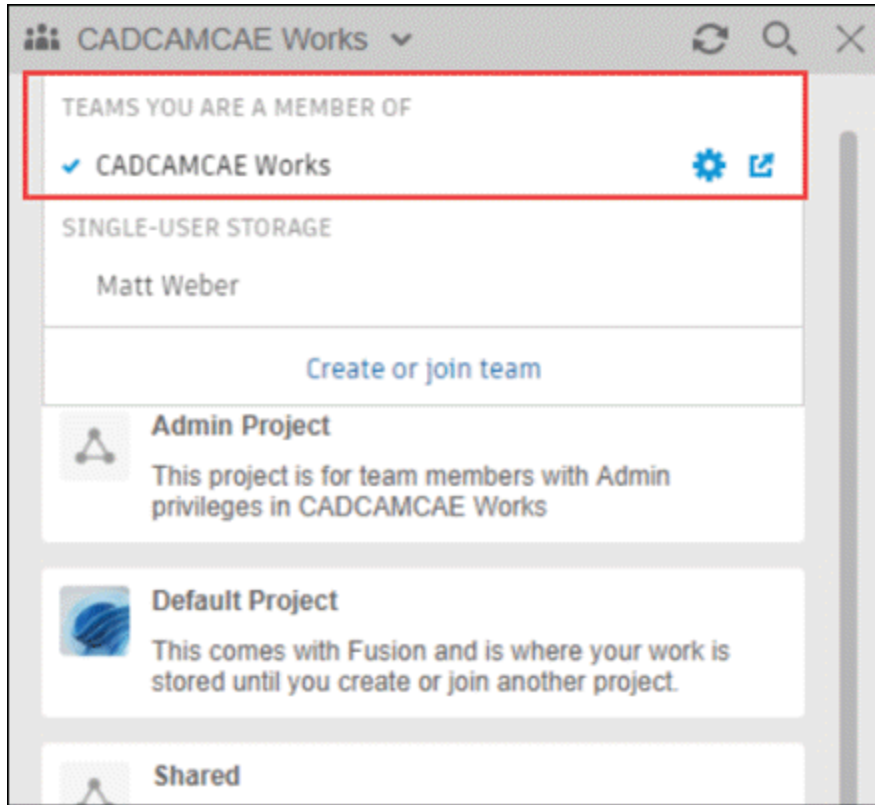


Figure-108. Team added in Data Panel

- Click on the **Open Administrator Console on the Web** button (similar to common settings button icon) displayed next to name of the team. The administration page for team will be displayed in the default web browser; refer to [Figure-109](#).

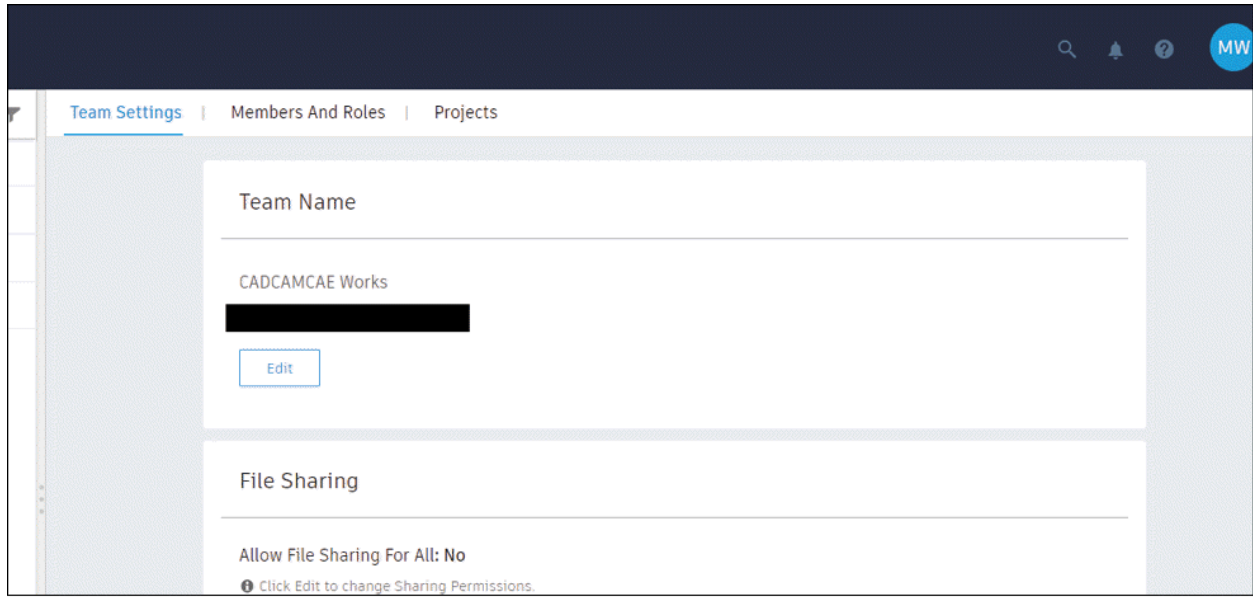


Figure-109. Administration page for team

- Specify desired parameters in this page to define settings related to file sharing, access, and collaboration. Note that you can set auto-approval settings for specified domains by using the **Add domain** button on this page. You will be asked to specify the text in email ID displayed after @ like for `matt@cadcamcaework.com` the `@cadcamcaework.com` is domain.
- To add members in your team, click on the **Members and Roles** tab in the web page. The options will be displayed as shown in [Figure-110](#).

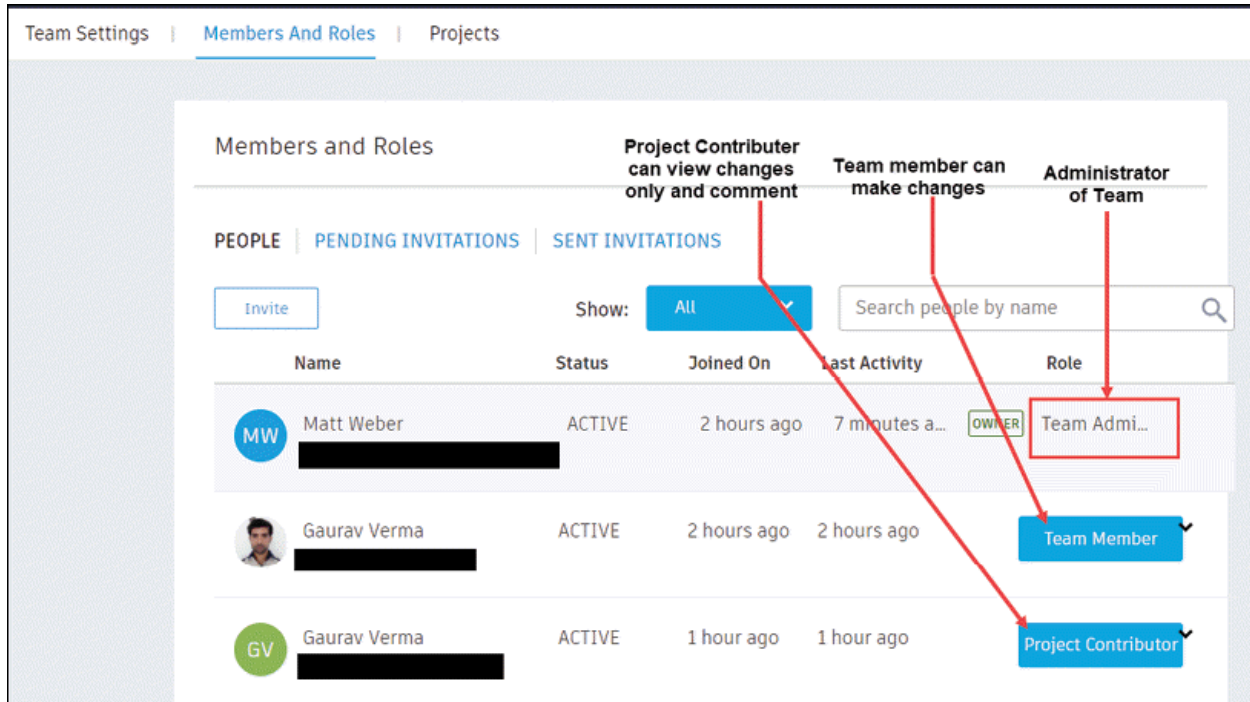


Figure-110. Members and roles page

- To invite a new member, click on the **Invite** button on this page. The **Invite Team Members** dialog box will be displayed; refer to [Figure-111](#).

Invite Team Members

Invited Team Members can create Projects and will have access to all Projects in the Team subject to Project Admin's approval.

Enter email addresses

Enter a comma-separated list of email addresses.

Cancel Invite

Figure-111. Invite Team Members dialog box

- Specify the email id's of team members separated by comma (,) in the **Enter email addresses** edit box and click on the **Invite** button. Once they accept invitation on their email ids, they will become members of the team. You can later assign the roles to members as desired using the Administrator page. You can now logout and close the browser to exit the admin panel of Fusion team.

After creating a team, make sure you are log-in by team name in the **Data Panel** and create a new project using **New Project** button in the **Data Panel**. This new project will be shared with all the team members for modifications based on roles assigned to them; refer to [Figure-112](#).

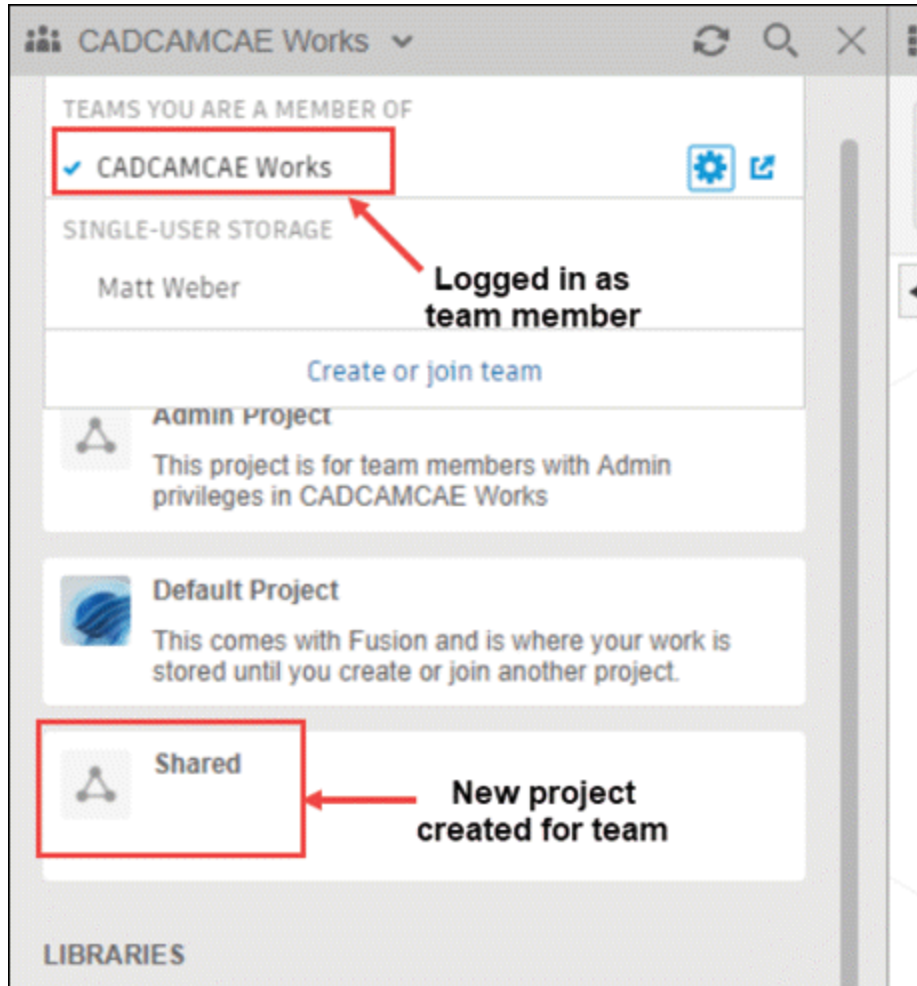


Figure-112. Sharing project with team

Cloud Account of User

To open the cloud account of user, you need to click on the **Open on the Web** button next to the name of user in **Data Panel**; refer to [Figure-113](#). On selecting this button, details of user's cloud storage will be displayed in web browser. You can check and modify the files as desired. Note that you can delete any file available in Cloud account using the tools in web browser which is not possible in **Data Panel**; refer to [Figure-114](#).

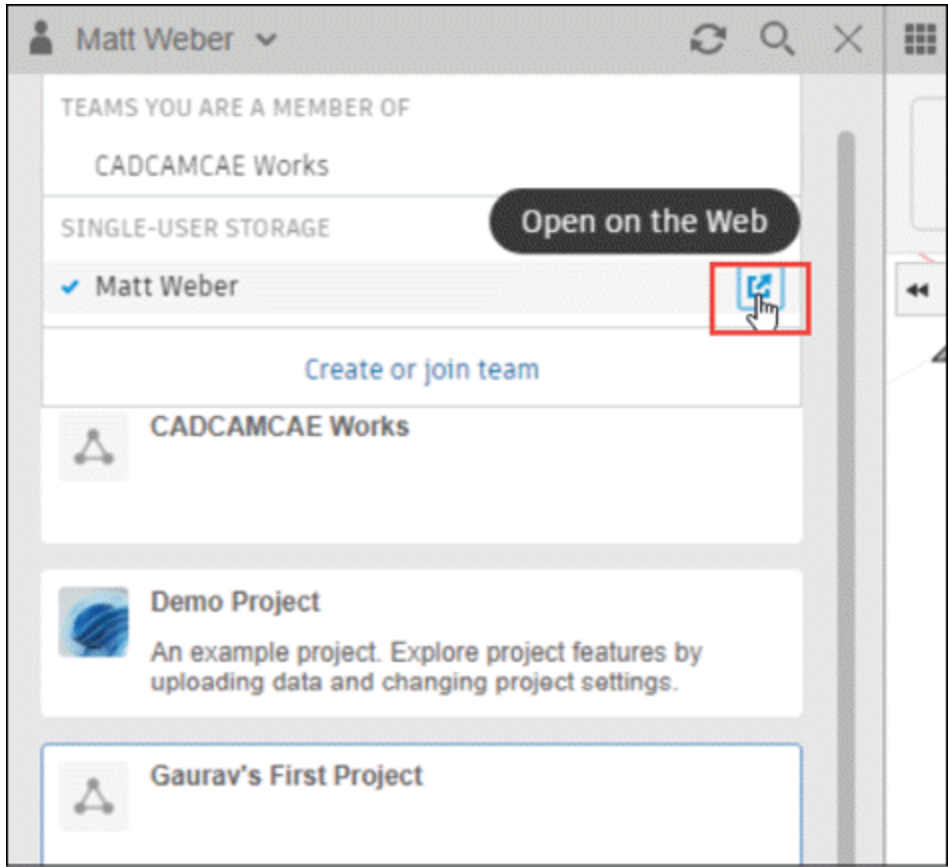


Figure-113. Open on the web

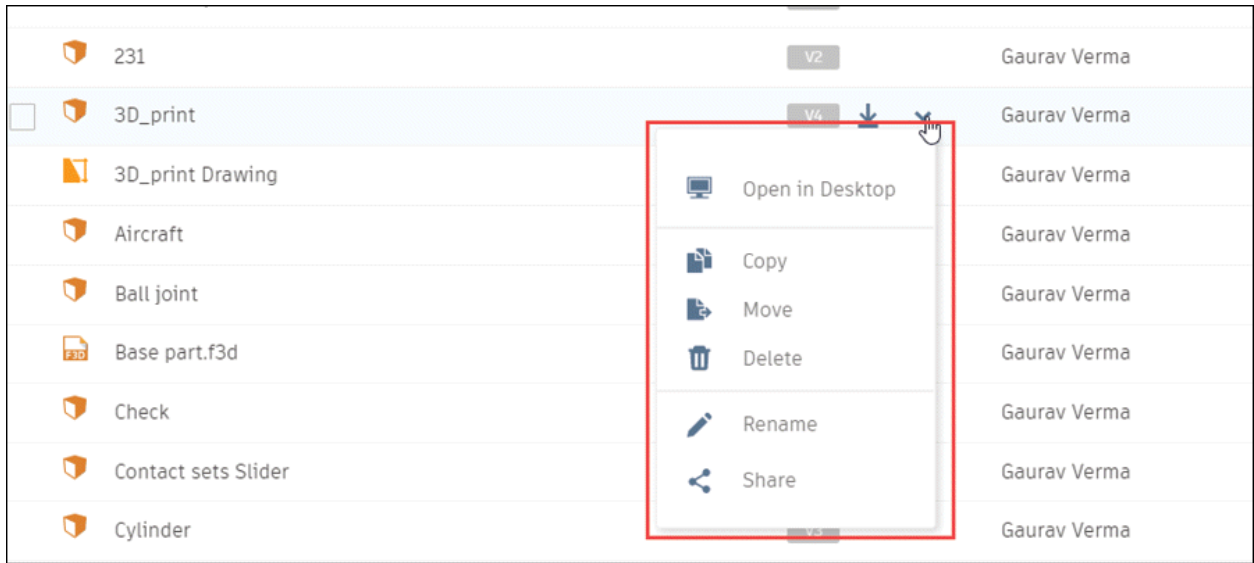


Figure-114. Shortcut menu for modifying files in cloud account using web browser

BROWSER

The **BROWSER** presents an organized view of your design steps in a tree like structure on the left of the Fusion 360 screen; refer to [Figure-115](#). When you select a feature or component in the **BROWSER**, it is also highlighted in the graphics window.

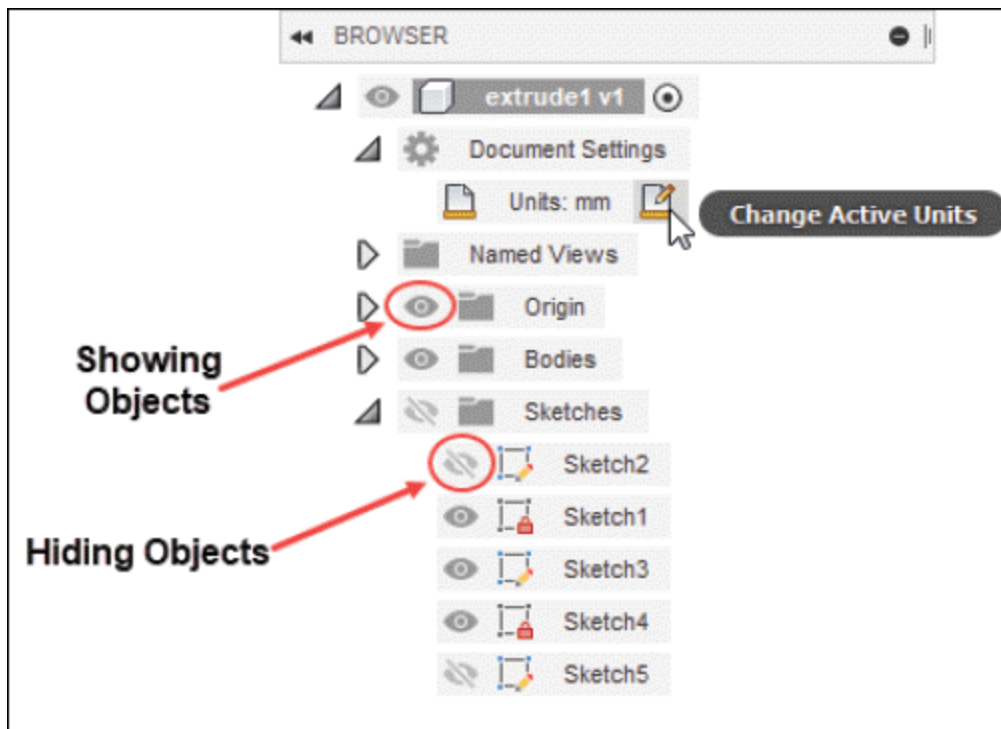


Figure-115. Browser

- Click on the **Eye** button to toggle the visibility of respective object.
- If you select any object in **BROWSER** then it is highlighted in blue color which indicates that the respective object is active for various operations.
- To change the units of the design or sketch, select the **Change Active Units** button as highlighted in the above figure. The **CHANGE ACTIVE UNITS** dialog box will be

displayed. Select desired unit and click on the **OK** button.

NAVIGATION BAR

The **Navigation Bar** is available at the bottom of graphics window of Fusion 360. It provides access of navigation commands for design. To start a navigation command, click on any tool from the **Navigation Bar**; refer to [Figure-116](#). There are various tool in **Navigation Bar** to navigate the model which are discussed next.

- **Orbit**- The **Orbit** tools are used for rotating the current model. There are two types of orbit tools i.e. **Free Orbit** and **Constrained Orbit**. **Free Orbit** is used to rotate the design view freely and **Constrained Orbit** is used to rotate the view in constrained motion relative to current origin point. The **Constrained Orbit** tool is displayed on clicking down arrow next to **Free Orbit** tool. After selecting this tool, click at desired location on model and drag the cursor to rotate model view. You can do the same function by using **SHIFT + Middle Mouse Button** dragging.

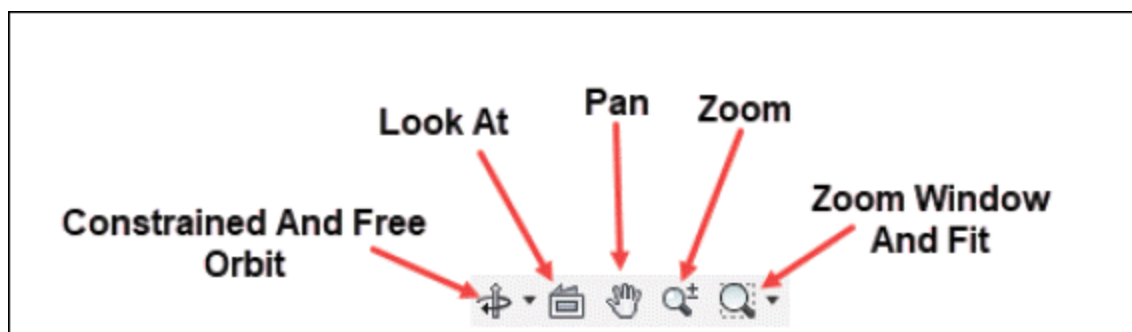


Figure-116. Navigation Bar

- **Look At-** This tool is used to view selected face parallel to screen. To use this tool, select desired face of model, and then click on the **Look At** tool from **Navigation Bar**.
- **Pan-** It is used to move the design parallel to the screen. You can do the same function by middle mouse button drag.
- **Zoom-** It is used to increase or decrease the magnification of the current view. After selecting this tool, click on the model, hold, and drag mouse upward/downward to zoom in/out. You can do the same function by rolling the mouse wheel up/down.
- **Zoom Window** and **Fit - Zoom Window** tools are used for magnification of selected area. **Fit** tool is used to position the entire design on the screen. You can do the same function as **Fit** tool does by pressing **F6** from keyboard.

DISPLAY BAR

The tools of **Display Bar** are used to visualize the design in different viewports; refer to [Figure-117](#).

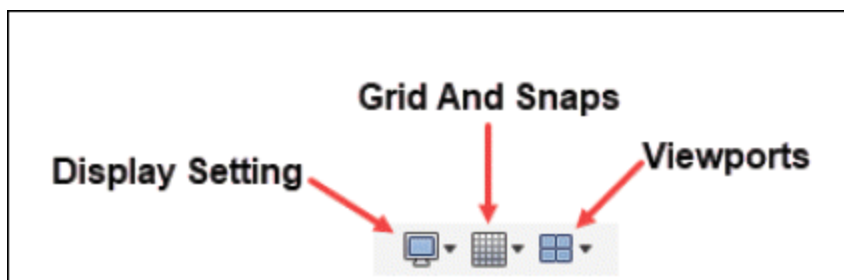


Figure-117. Display Bar

Display Settings

The options in the **Display Setting** flyout are used to enable or disable various commands related to **Visual Style**, **Visibility of objects**, **Camera settings**, and so on; refer to [Figure-118](#).

- Select desired visual style of 3D model from the **Visual Style** cascading menu. Various styles available are Shaded, Shaded with Hidden Edges, Shaded with Visible Edges only, Wireframe, Wireframe with Hidden Edges, Wireframe with Visible Edges only. (You can open a sample file of 3D model and check the effects of different visual styles.)
- The **Face Groups** check box available in the **Mesh Display** cascading menu is used to display or hide mesh group faces. You will learn more about this option while working on mesh.

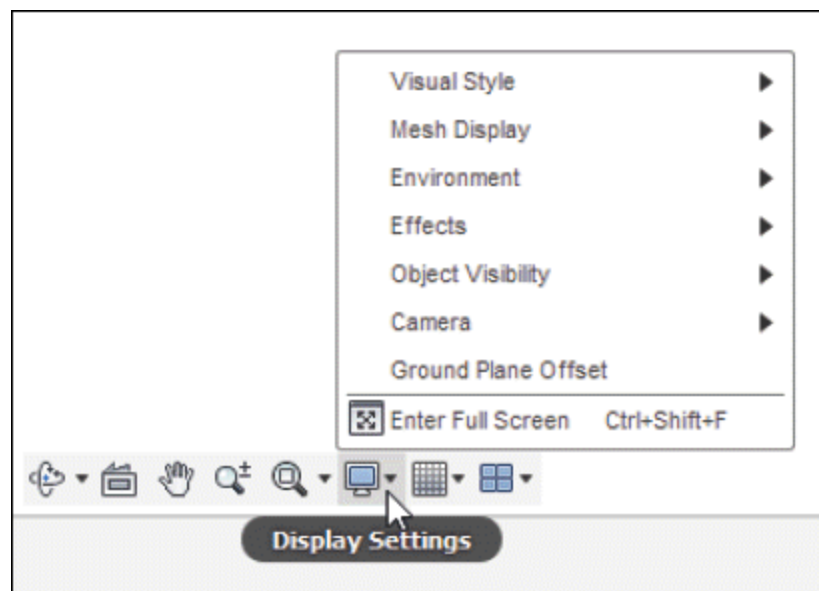


Figure-118. Display Settings flyout

- Select desired option from the **Environment** cascading menu to define color scheme of the canvas.
- Select desired check boxes for effects that you want to add in display style from the **Effects** cascading menu. Similarly, you can display or hide objects from the **Object Visibility** cascading menu.
- The options in **Camera** cascading menu are used to change the display to Orthographic, Perspective, or Perspective with Ortho Faces.
- Click on the **Ground Plane Offset** tool from the **Display Settings** flyout. The **GROUND PLANE OFFSET** dialog box will be displayed; refer to [Figure-119](#). Clear the **Adaptive** check box and set desired offset value if you want to change the location of ground plane. Note that position of ground plane is generally changed for rendering which will be discussed later.

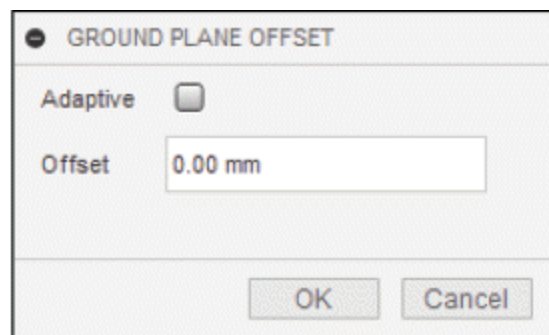


Figure-119. GROUND PLANE OFFSET dialog box

- If you want to run Autodesk Fusion 360 in full screen then press **CTRL+SHIFT+F** from keyboard.

Grids and Snaps

-
- The **Grid and Snaps** flyout is used to activate or deactivate various options like **Layout Grid**, **Layout Grid Lock**, **Snap to Grid**, and so on. You can also define the size of grid and snapping increments by using **Grid Settings** and **Set Increments** tools in this flyout.

Viewports

- The options in **Viewports** flyout are used to switch between four viewports and single viewport in the Fusion 360 screen. Click on the **Multiple Views** option from the **Viewports** flyout to display the current model in different views in four viewports. You can click in each of the viewport and reorient the model. If you want to set these views to synchronous again then select the **Synchronize Views** check box from **Viewports** flyout. If you want to move back to single viewport then select the **Single View** tool in **Viewports** flyout. By pressing **SHIFT+!** keys, you can toggle between single viewport and multiple viewports modes.

ViewCube

The **ViewCube** tool is used to rotate the view of model to access different faces of model. Click at desired location on model and drag the **ViewCube** to perform a free orbit. There are six faces in **ViewCube**; **Top**, **Bottom**, **Front**, **Back**, **Right**, and **Left** to access different orientations of model. To access the standard orthographic and isometric views, click on the faces and corners of the cube; refer to [Figure-120](#).

Right-click on the **ViewCube** to access options related to perspective mode and other view modes.

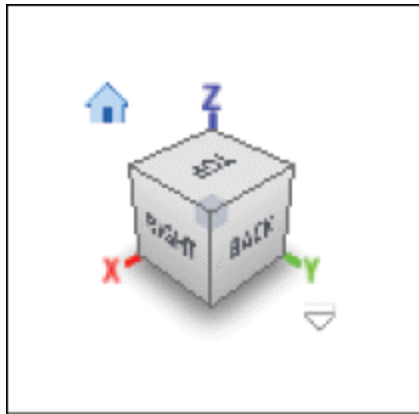


Figure-120. ViewCube

Mouse Functions

You can use the mouse shortcuts to zoom in/out, pan the view, and orbit the view; refer to [Figure-121](#).

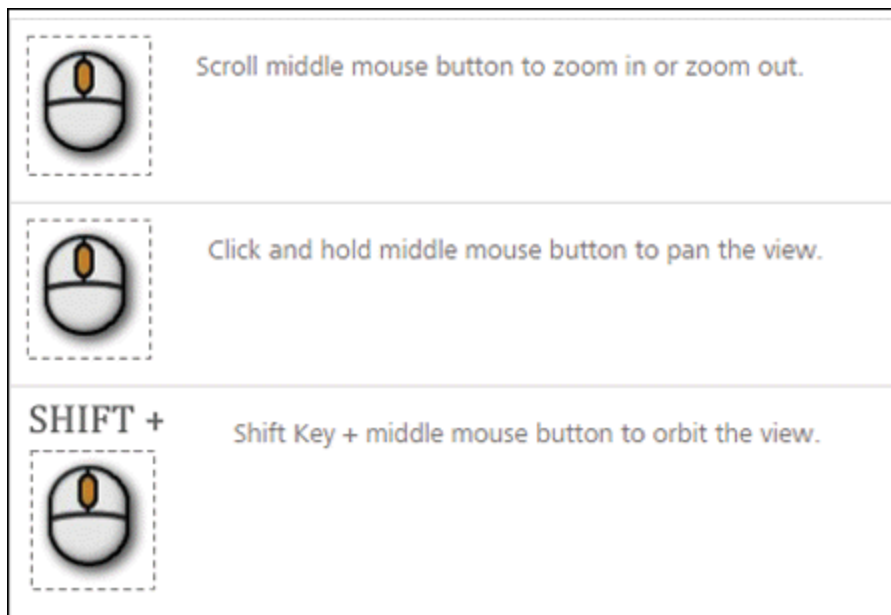


Figure-121. Mouse shortcut keys

- Scroll the **Middle Mouse Button** downward/upward to zoom in or zoom out, respectively. Note that this function

can be reversed by using options in **Preferences** dialog box.

- Click and hold the **Middle Mouse Button** to pan the view.
- Use the **Shift + Middle Mouse Button** to orbit the view.
- Press the left mouse button to select any object or tool.
- Press right mouse button (right-click) to access shortcut menus in the software.

Timeline

The **Timeline** bar is very useful for saving time while editing. It records the design feature in chronological order; refer to [Figure-122](#). Note that the **Capture Design History (Parametric Modeling)** option must be active in the **Design** node of **Preferences** dialog box to access **Timeline** bar.



Figure-122. Timeline

If you want to edit any feature of model then double-click on the respective feature from **Timeline** bar. The respective dialog box/interface element will be displayed. Make desired changes and apply the parameters. The final design will be generated automatically with the applied changes; refer to [Figure-123](#).

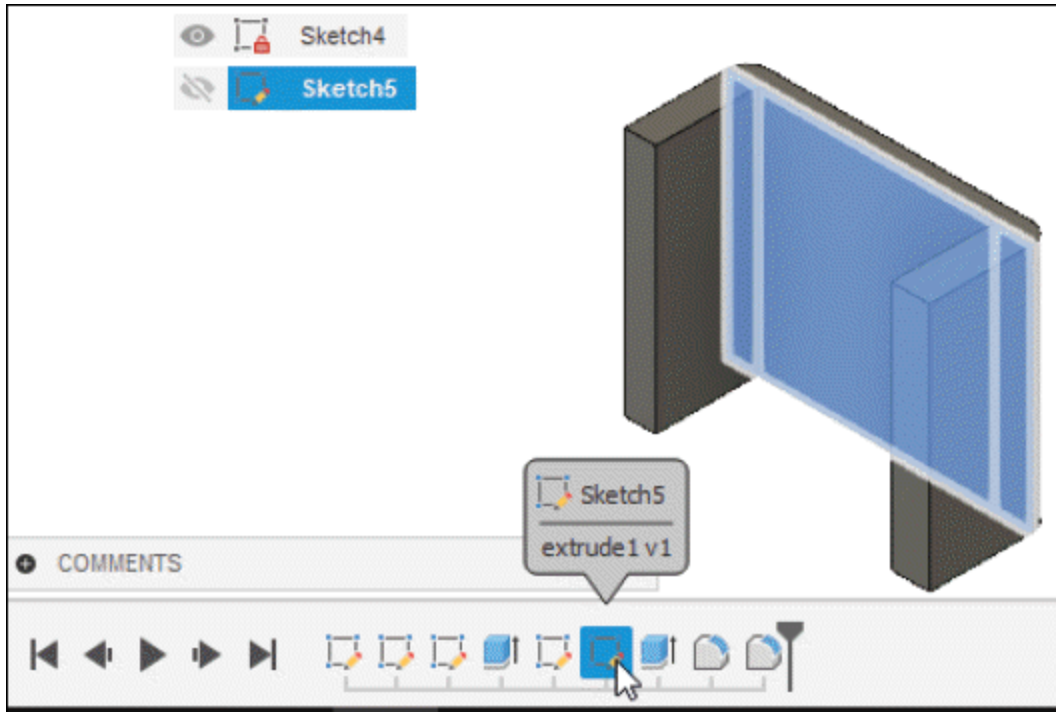


Figure-123. Editing Model From Timeline

CUSTOMIZING TOOLBAR AND MARKING MENU

In Autodesk Fusion 360, you can add or remove any tool in the displaying panels of **Toolbar**. You can also add or remove tool from right-click marking menu. These operations are discussed next.

Adding Tools to Panel

- Click on the **Three dot** button next to tool to be added in the panel from the drop-down of **Toolbar**. The shortcut menu to customize toolbar will be displayed; refer to [Figure-124](#).

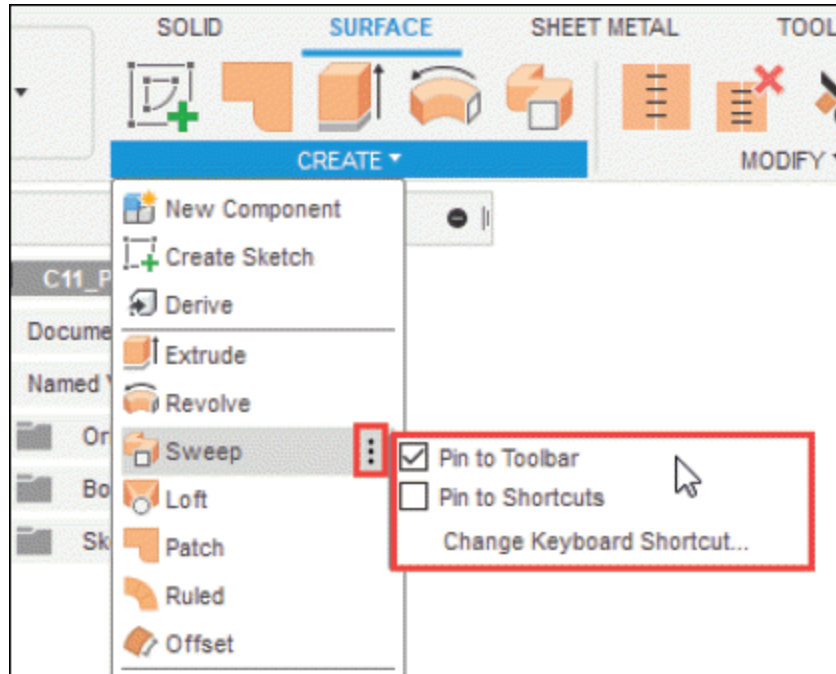


Figure-124. Shortcut menu for tool customization

- Select the **Pin to Toolbar** check box from the shortcut menu. The tool will be added in the panel.

Assigning Shortcut Key

- Click on the **Change Keyboard Shortcut** option from the shortcut menu as shown in [Figure-124](#). The **Change Keyboard Shortcut** dialog box will be displayed; refer to [Figure-125](#).

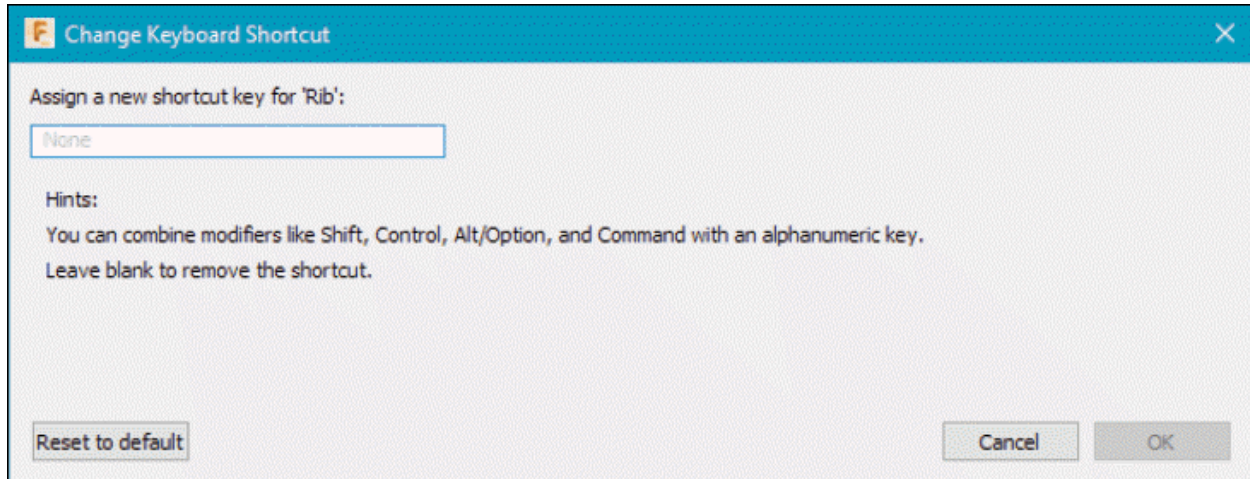


Figure-125. Change Keyboard Shortcut dialog box

- Use desired combine modifier like **SHIFT**, **CTRL**, or **ALT** and then press desired alphanumeric key to create a shortcut button.
- If you want to remove shortcut key then use **Backspace** to delete shortcut key from edit box. Click on the **OK** button from the dialog box to apply the settings.

Adding or Removing tool from Shortcut menu/Marking menu

- Select the **Pin to Shortcuts** check box from the shortcut menu for tool that you want to add in right-click shortcut menu/marking menu; refer to [Figure-126](#).

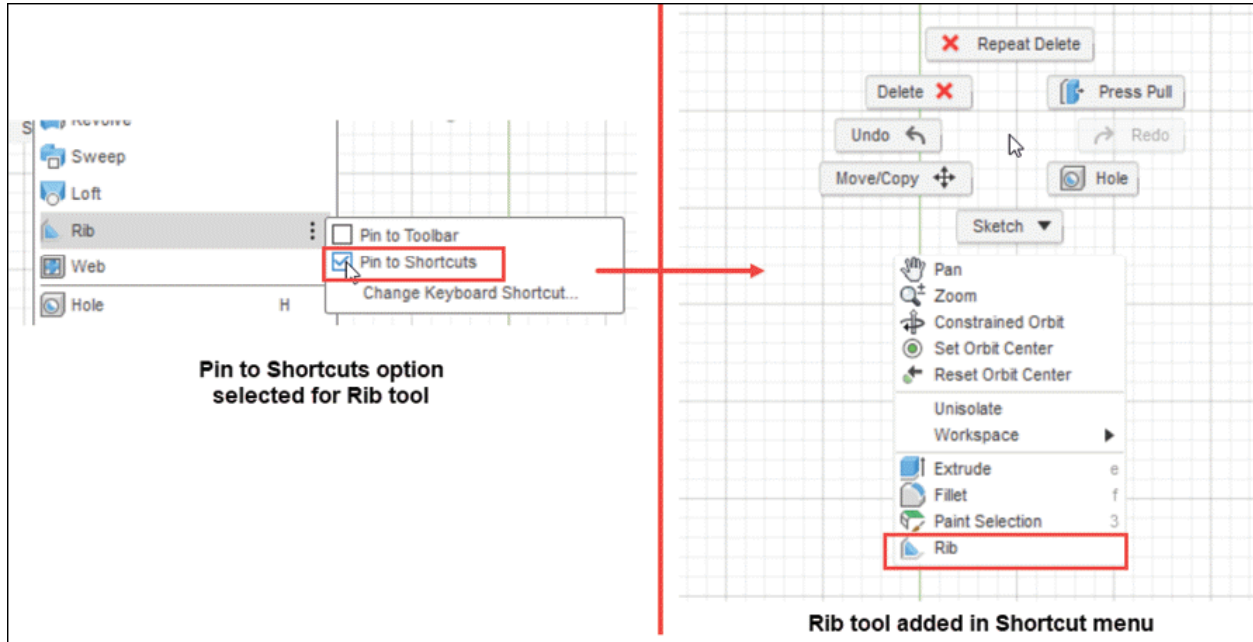


Figure-126. Adding tool to shortcut menu

- Clear the **Pin to Shortcuts** check box from the shortcut menu for tool that you want to remove from right-click shortcut menu/marking menu.

Note that the shortcut menu is workspace specific so what you find in shortcut menu for **DESIGN** workspace will be different from **MANUFACTURE** workspace, and so on.

Removing Tool from Panel in Toolbar

To remove a tool from panel in **Toolbar**, select the tool and drag it to canvas; refer to [Figure-127](#).

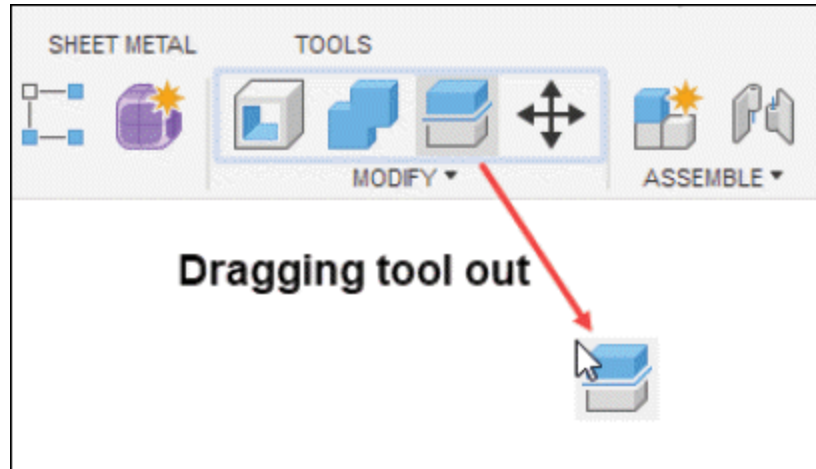


Figure-127. Removing tool from panel

Similarly, you can change position of tool by dragging it within the panel.

Resetting Panel and Toolbar Customization

Right-click on any of the tool in **Toolbar**. A shortcut menu will be displayed as shown in [Figure-128](#).

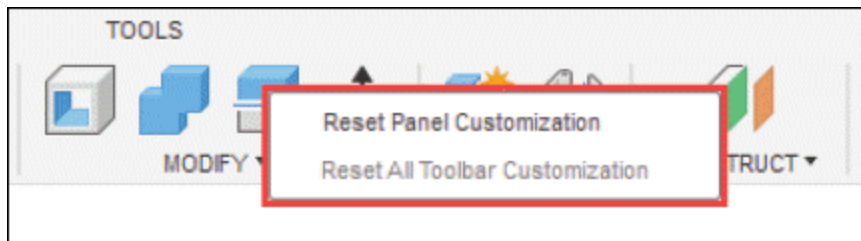


Figure-128. Shortcut menu for Resetting Toolbar

Click on the **Reset Panel Customization** option to reset current panel to original state. Note that this option will be available only when you have added or removed a tool from the panel.

Click on the **Reset All Toolbar Customization** option from the shortcut menu if you want to reset the complete toolbar

including all the panels to its original state.

SELF ASSESSMENT

Q1. is used to manage the fusion 360 project.

Q2. Which tool is used to share the files to GrabCAD?

Q3. The.....button is used for producing CNC codes.

Q4. Which of the following workspace is not available in Autodesk Fusion 360 application?

- a. Patch Workspace
- b. Model Workspace
- c. Assemble Workspace
- d. Drawing Workspace

Q5. Discuss the use of **Save** tool with example.

Q6. Discuss the process of exporting a project in computer with example.

Q7. Discuss the use of **3D Print** tool with example.

Q8. Which tool is used to generate **Toolpaths** for model?

Q9. How to reset the default layout?

Q10. Explain the process of creating **Script** with example.

Q11. Mark the following statements as True or False:

- a. Autodesk Fusion 360 software can be used to perform CAD operations.
- b. Autodesk Fusion 360 software can be used to perform CAM operations.

- c. Autodesk Fusion 360 software can be used to perform CAE operations.
- d. Autodesk Fusion 360 software can be used to perform PLM operations.
- e. Autodesk Fusion 360 software can be used to perform direct 3D printing on 3D Printer.

Q12. Which of the following tool is used to create a new part model in Autodesk Fusion?

- a. New Design
- b. New Drawing->From Animation
- c. New Drawing->From Design
- d. New Drawing->New Drawing Template

Q13. Which of the following workspace is not available in educational version of Autodesk Fusion 360?

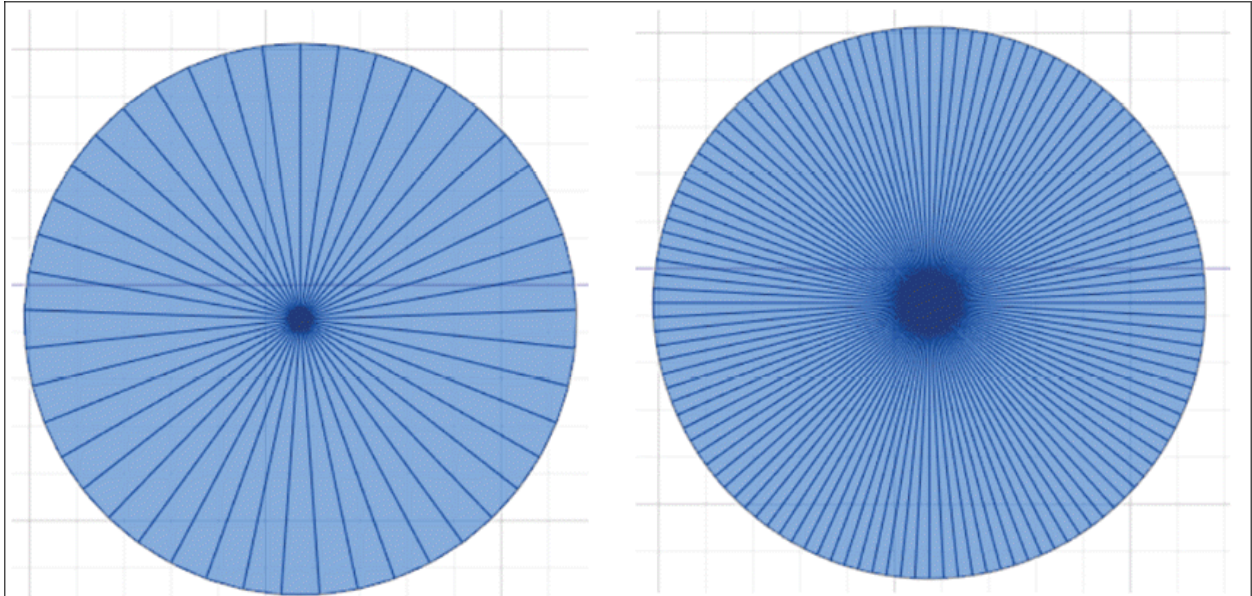
- a. Animation
- b. Manufacturing
- c. Generative Design
- d. Simulation

Q14. Which of the following tool is used to save the file in local drive in Autodesk Fusion 360?

- a. Save
- b. Save As
- c. Export
- d. Upload

Q15. STL format is used as standard for 3D printing which abbreviates Standard Triangle Language. (T/F)

Q16. Which parameter of an STL file can decrease the angular distance between two edges of triangle as shown in next figure (from left to right)?



- a. Surface Deviation
- b. Normal Deviation
- c. Aspect Ratio
- d. Maximum Edge Length

Chapter 2

Sketching

Topics Covered

The major topics covered in this chapter are:

- ***Introduction***
- ***Starting Sketch***
- ***Sketch Creation Tools***
- ***Sketch Editing Tools***
- ***Sketch Palette***
- ***Constraints***
- ***Sketching Practical and Practice***

INTRODUCTION

In Engineering, sketches are based on dimensions of real world objects. These sketches work as building blocks for various 3D operations. In this chapter, we will be working with sketch entities like; line, circle, arc, polygon, ellipse, and so on to create base feature for various 3D operations. Note that the sketching environment is the base of 3D Models so you should be proficient in sketching.

There is no separate workspace for sketching and tools to

create sketches are available in **Design** workspace. So, we will be working in **Design** workspace in this chapter. We will learn about various tools of toolbar used in sketching and 3D Sketching.

STARTING SKETCH

- To start a new sketch, click on the **Create Sketch** tool from **CREATE** drop-down in **SOLID** tab of **Toolbar**; refer to [Figure-1](#). The three primary planes will be displayed on canvas screen; refer to [Figure-2](#).

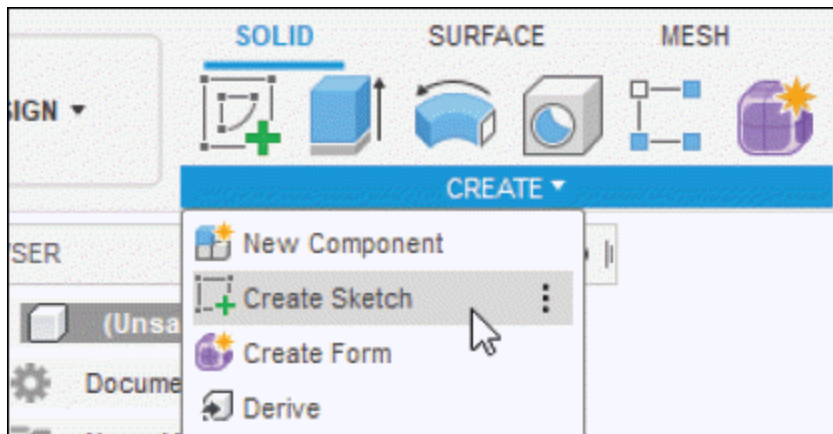


Figure-1. Create Sketch tool

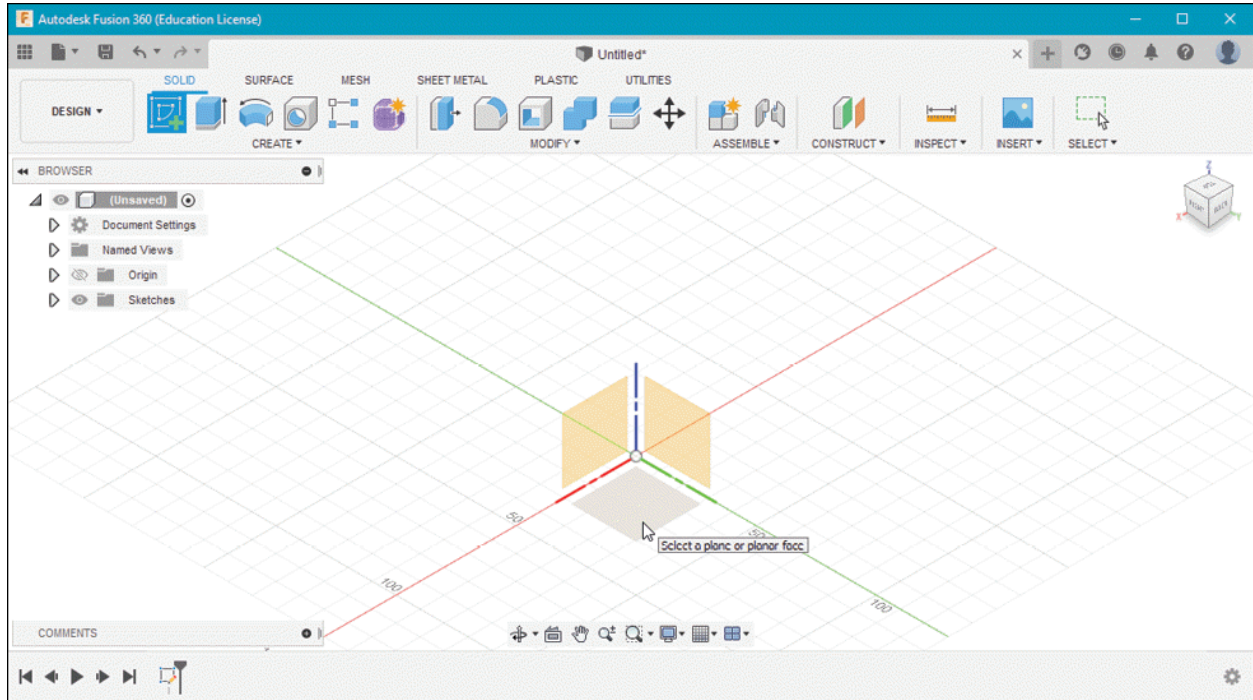


Figure-2. Select sketch plane

- Click on desired plane from the canvas screen. The selected plane will become parallel to the screen and act as current sketching plane. Also, the tools to create sketch will be displayed in **SKETCH** contextual tab in the **Toolbar**; refer to [Figure-3](#). Now, we are ready to draw sketch on the selected plane.

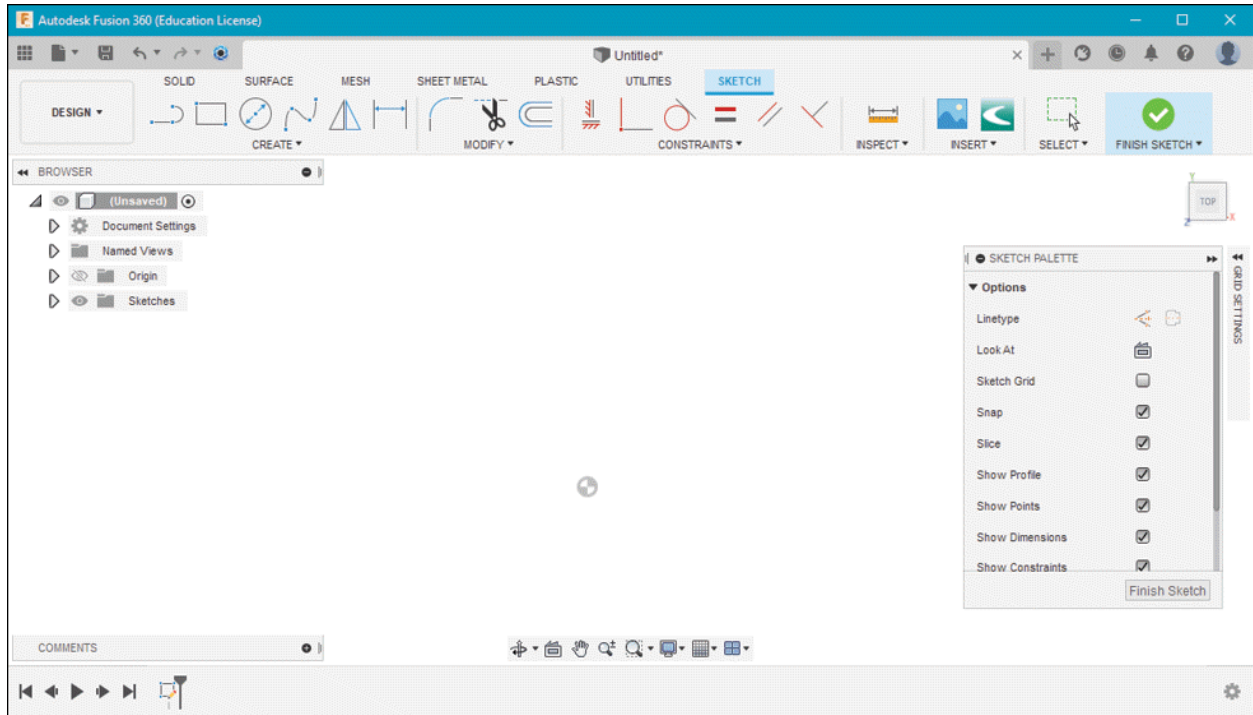


Figure-3. Sketching mode

First, we will start with the sketch creation tools and later, we will discuss the other tools.

SKETCH CREATION TOOLS

There are various tools in **CREATE** drop-down for creating sketch entities. These tools are discussed next.

Line

In Autodesk Fusion 360, the **Line** tool is used to create a line or arc. The procedure to use this tool is discussed next.

- Click on **Line** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-4](#). You can also press **L** key to select the **Line** tool. You will be asked to specify start point of the line.

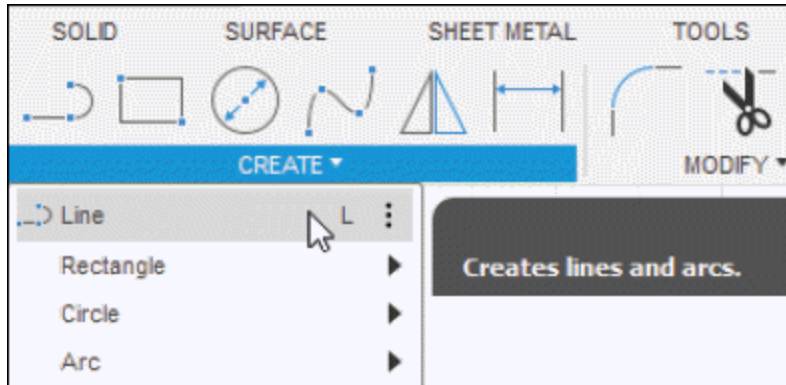


Figure-4. Line tool

- Click at desired location to specify start point and move the cursor away from selected point in desired direction in which you want to create the line. **Angle** and **Length** input boxes will be displayed with the preview of line.
- Specify desired value of length in the **Length** input box. Press **TAB** key to toggle to **Angle** input box; refer to [Figure-5](#).

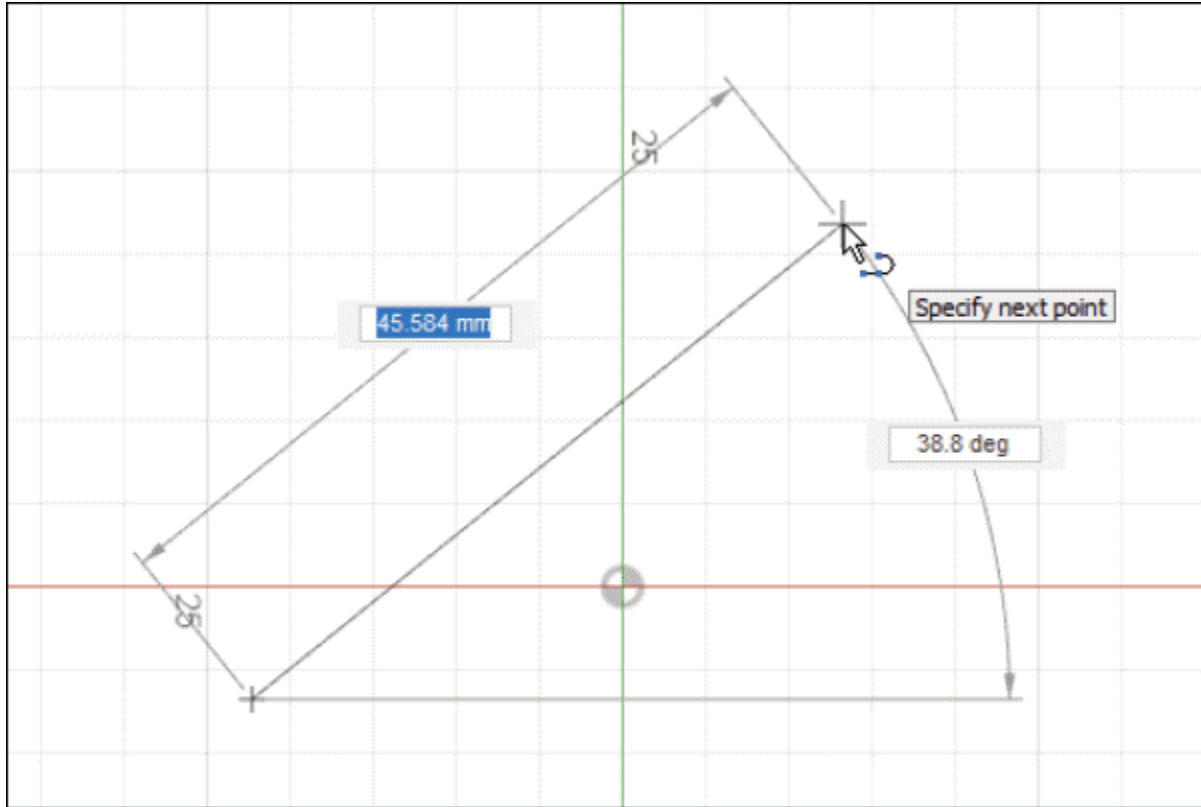


Figure-5. Creating the line

- If you are creating a freestyle line then click on the screen at desired location without specifying any value in **Length** and **Angle** edit boxes. Press **ESC** after specifying end points to exit the tool.
- To create a line for construction purpose, click on the **Construction** button from **SKETCH PALETTE** displayed at the right in the application window after activating **Line** tool; refer to [Figure-6](#). (You will learn more about **SKETCH PALETTE** later in this chapter.) You can also right-click on the line after creating it and then click on **Normal/Construction** button from **Marking menu** to do the same; refer to [Figure-7](#).

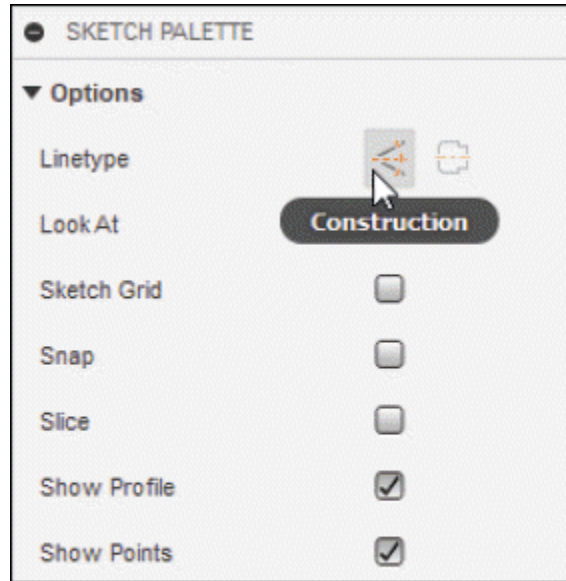


Figure-6. Construction button in SKETCH PALETTE

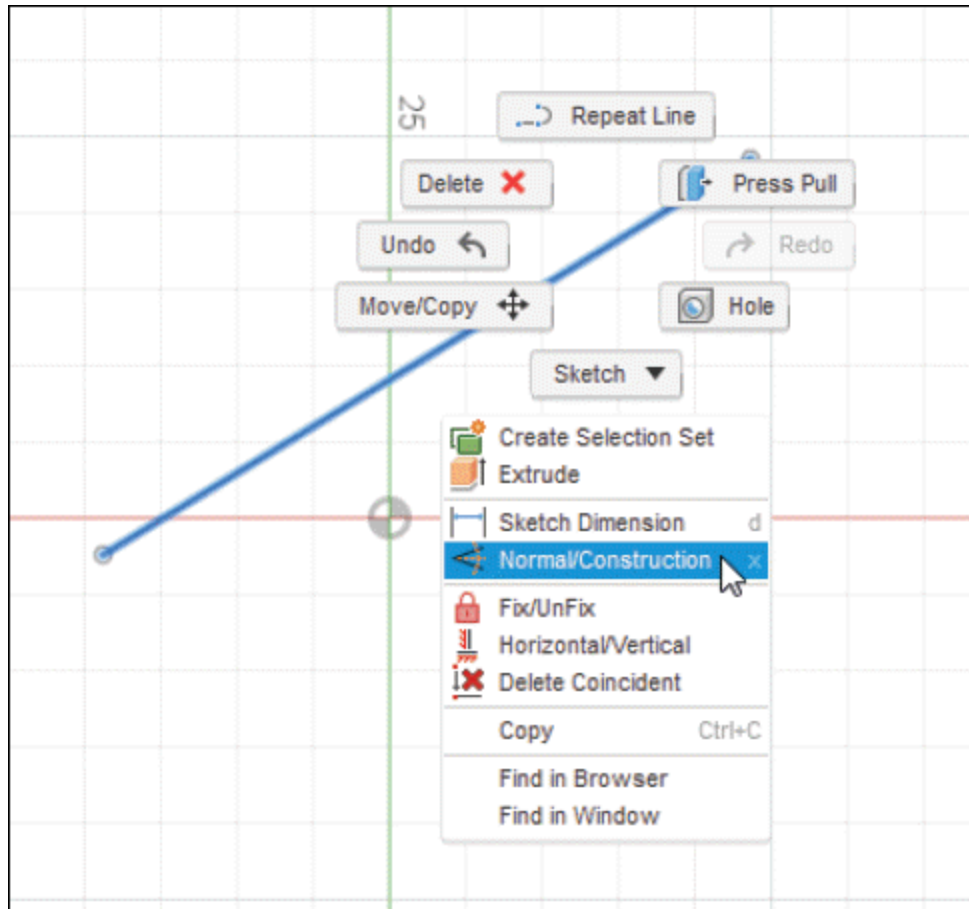


Figure-7. Marking menu

Marking menu

It is the radical display of most frequently used tool and tools related to selected entities. This menu also provides a quick access to the tools of the toolbar. This menu is the fastest way to activate the tool. To access this menu, right-click anywhere on the screen within the graphics window. To select any tool of this menu, move the cursor towards the tool and the tool will be highlighted. Click on it to activate the tool.

Creating Arc using Line Tool

You can create an arc using the **Line** tool which has one of its end point shared to another entity. In simple words, the starting point of arc created by **Line** tool will lie on end point of another line or arc. The procedure to create arc is given next.

- Click on the end point of a line or curve to define start point of arc, hold the **LMB** (Left - Mouse Button), and drag to the point where you want to specify the end point of arc; refer to [Figure-8](#).

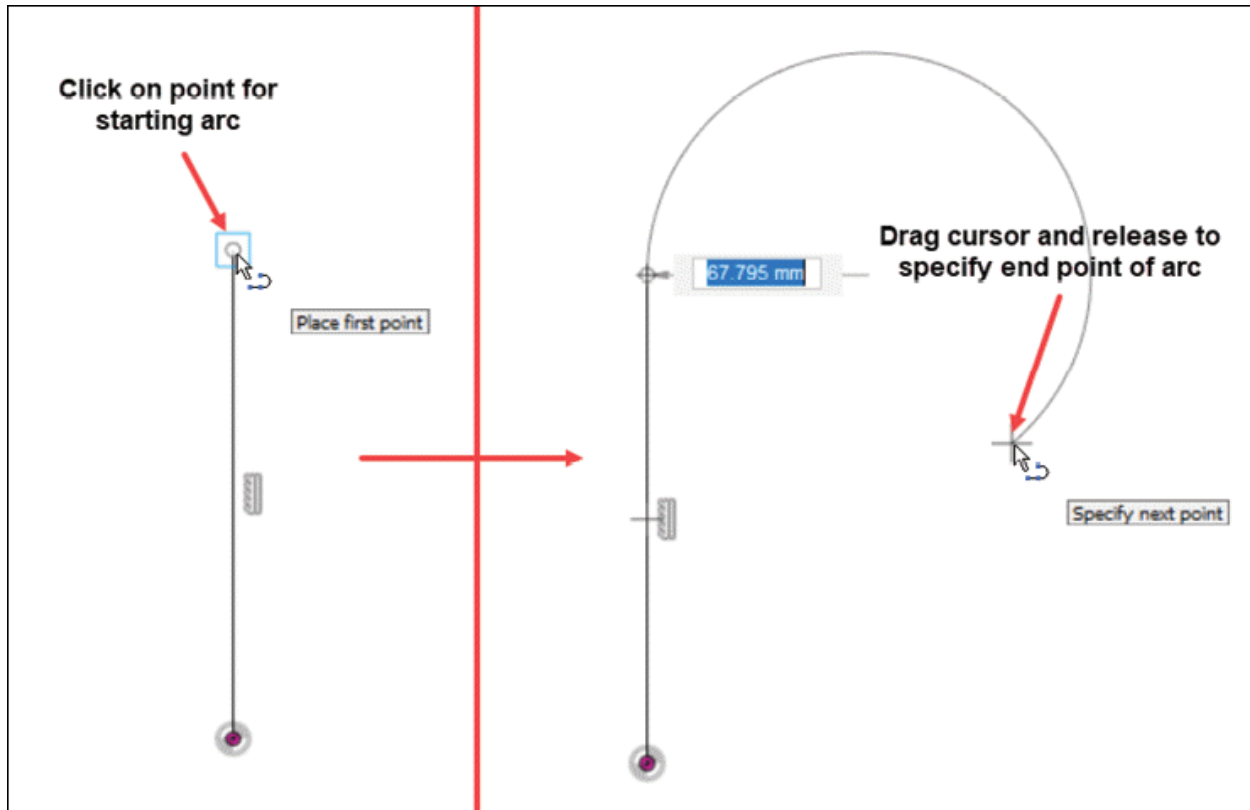


Figure-8. Creating line arc

Rectangle

The tools in **Rectangle** cascading menu are used to create rectangles. There are three tools in **Rectangle** cascading

menu; **2-Point Rectangle**, **3-Point Rectangle**, and **Center Rectangle**; refer to [Figure-9](#).

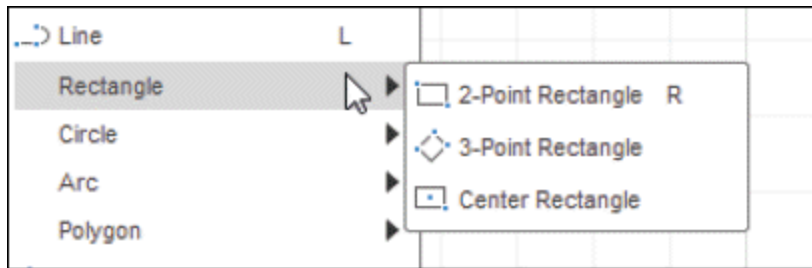


Figure-9. Rectangle cascading menu

You can also activate the **Rectangle** tool by pressing **R** key while in **Sketch** mode. The procedures to use these tools are discussed next.

2-Point Rectangle

- Click on the **2-Point Rectangle** tool of the **Rectangle** cascading menu from **CREATE** drop-down in **SKETCH** contextual tab. You will be asked to specify the first point.
- Click on the sketch canvas to specify first point of rectangle. You will be asked to specify the other corner point; refer to [Figure-10](#).

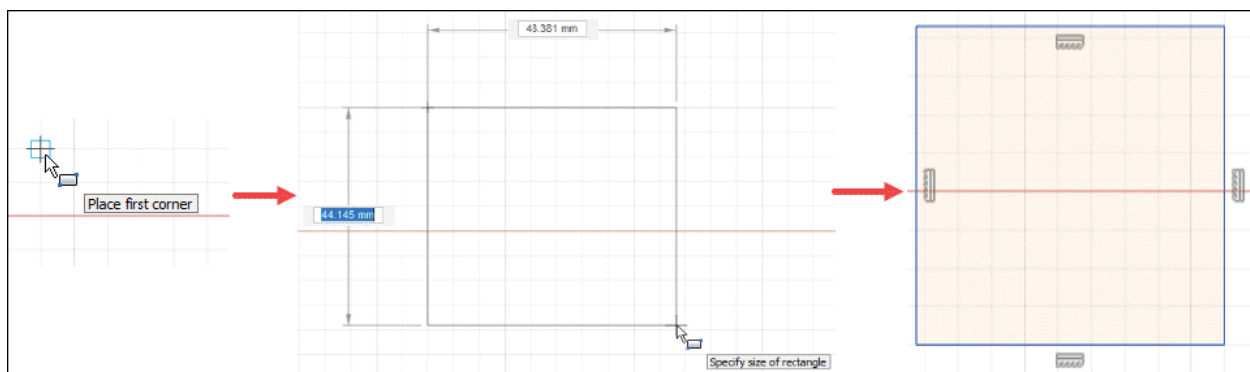


Figure-10. Creation of 2-point rectangle

- Specify the length and width of rectangle in the dynamic

input boxes. To switch between the length dynamic input box and width dynamic input box, press **TAB** key.

- After specifying the parameters, press **ENTER** to create the rectangle. To exit the tool, press **ESC** key.
- If you want to create a freestyle rectangle then click on the screen to select the first corner point and other diagonal corner point of the rectangle on the sketch canvas.
- To add dimensions, right-click on the line of rectangle and click on **Sketch Dimension** tool in **Sketch** drop-down from the **Marking menu**. You can also activate the **Dimension** tool by pressing **d** from keyboard after selecting the line. You will learn more about dimensioning later in this chapter.

3-Point Rectangle

- Click on the **3-Point Rectangle** tool of **Rectangle** cascading menu from **CREATE** drop-down. You will be asked to specify location of first corner point of rectangle.
- Click at desired location to specify first corner point of the rectangle on the sketch canvas; refer to [Figure-11](#).

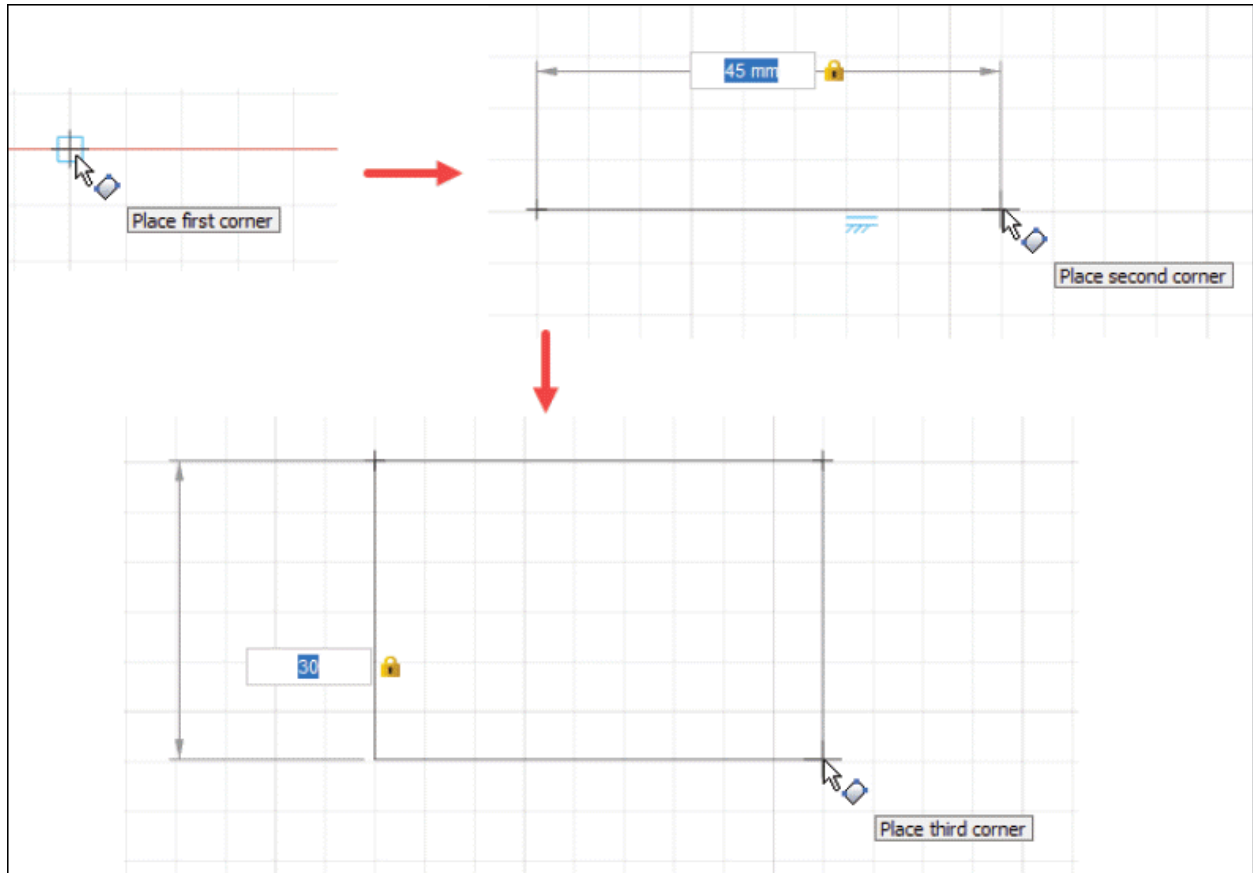


Figure-11. Creation Of 3-Point Rectangle

- Next, click to specify the second corner point (end point of base line of rectangle) on the canvas or specify the dimension in dimension box.
- Next, click to specify the third point of the rectangle to define the length of rectangle and complete the rectangle creation.

Center rectangle

- Click on the **Center Rectangle** tool of the **Rectangle** cascading menu from **CREATE** drop-down. You will be asked to specify center point of the rectangle.
- Click on the canvas to specify the center point of the

rectangle; refer to [Figure-12](#). You will be asked to specify corner point of rectangle.

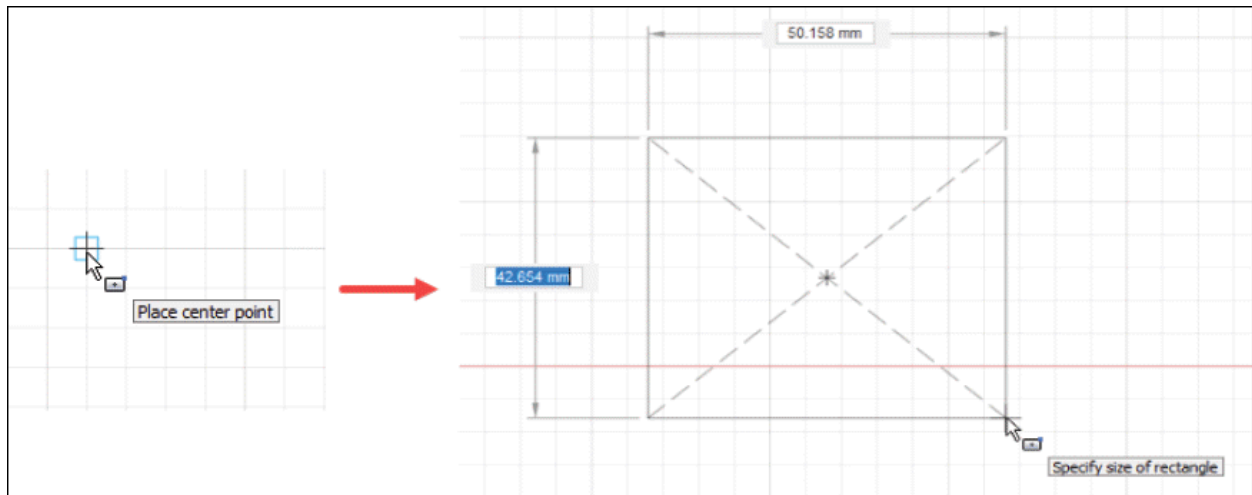


Figure-12. Creation of center rectangle

- Type the dimension for rectangle in the edit boxes and press **ENTER** to create the rectangle.
- If you want to create freestyle rectangle then click on the screen to select the center point and corner point of rectangle.

Note that you can switch between different types of rectangles from the **Rectangle** section of **SKETCH PALETTE** after activating the **Rectangle** tool.

Circle

There are five tools in **Circle** cascading menu to create circles; **Center Diameter Circle**, **2-Point Circle**, **3-Point Circle**, **2-Tangent Circle**, and **3-Tangent Circle**; refer to [Figure-13](#).

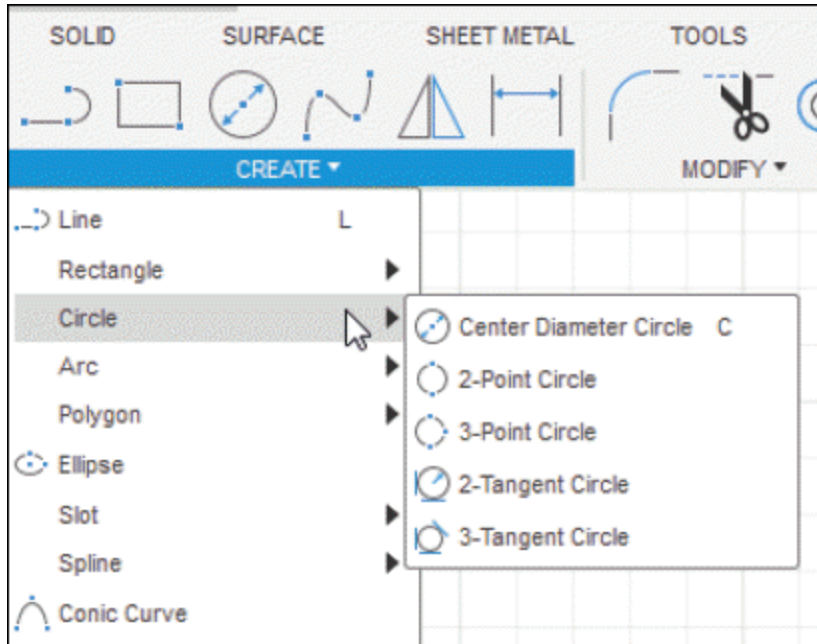


Figure-13. Circle cascading menu

The shortcut key to use the **Center Diameter Circle** tool is by pressing **C** key from keyboard. The tools to create circle are discussed next.

Center Diameter Circle

The **Center Diameter Circle** tool is used to create circle specifying center location and diameter value. The procedure to use this tool is given next.

- Click on the **Center Diameter Circle** tool from **Circle** cascading menu in the **CREATE** drop-down of **SKETCH** contextual tab. You will be asked to specify center point of circle.
- Click to specify the center point for the circle; refer to [Figure-14](#). You will be asked to specify circumferential point of circle to define diameter.

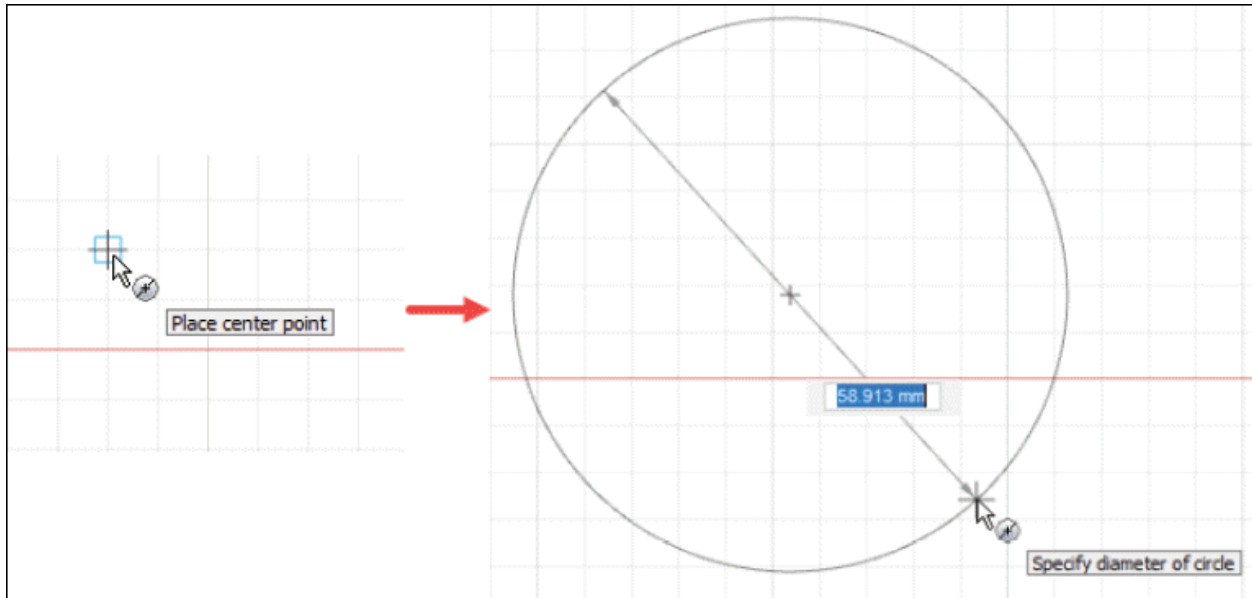


Figure-14. Creation of center diameter circle

- Click to specify the circumferential point or enter desired value of diameter in the dynamic input box.

2-Point Circle

The **2-Point Circle** tool is used to create circle by specifying two circumferential points. The procedure to use this tool is discussed next.

- Click on the **2-Point Circle** tool of **Circle** cascading menu from **CREATE** drop-down and specify the first point for circle; refer to [Figure-15](#).

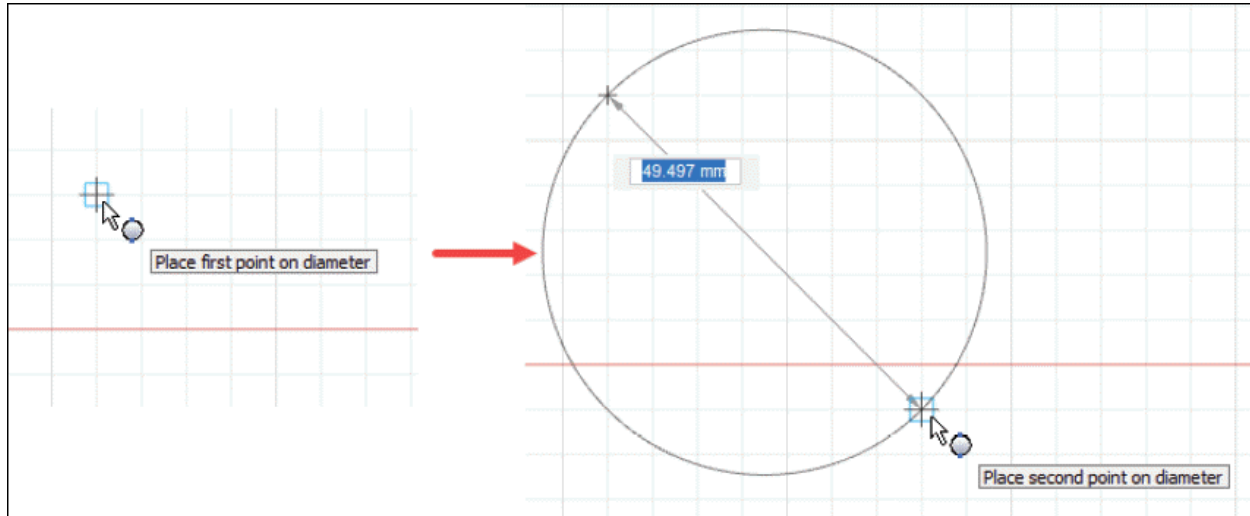


Figure-15. Creation of 2-point circle

- Specify the second point on the canvas screen or type the diameter dimension in the dynamic input box and press **ENTER** key.

3-Point Circle

The **3-Point Circle** tool is used to create circles using 3 circumferential points as references. The procedure to use this tool is discussed next.

- Click on **3-Point Circle** tool of **Circle** cascading menu from **CREATE** drop-down.
- Click on the screen to specify the first point, second point, and third point of circle; refer to [Figure-16](#). Note that you can also select on other entities to define circumferential points for circle.

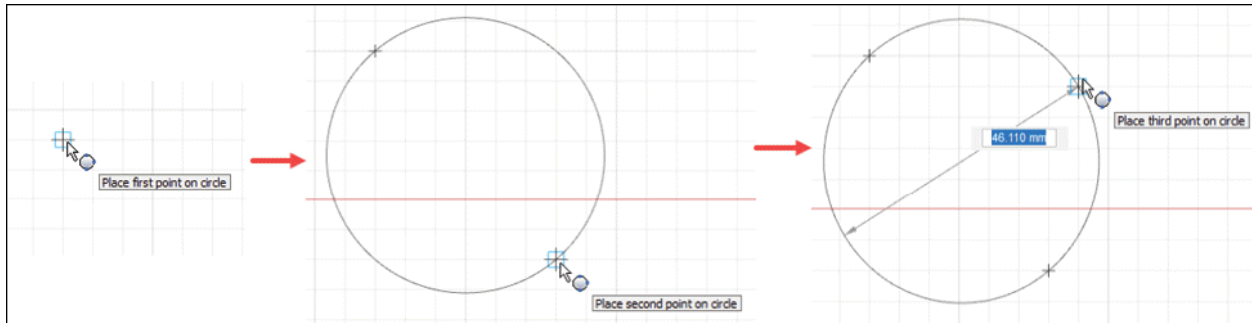


Figure-16. Creation of 3-point circle

- After specifying points, right-click on the screen and click **OK** button from **Marking menu** to complete the process. You can also press **ESC** key to exit the circle tool.
- If you want to add dimensions then right-click on the selected circle and click on the **Sketch Dimension** tool from **Marking menu**. You can also select **Sketch Dimension** tool from **SKETCH** drop-down.

2-Tangent Circle

The **2-Tangent Circle** tool is used to create a circle which is tangent to two selected lines. The procedure to use this tool is discussed next.

- Select **2-Tangent Circle** tool of **Circle** cascading menu from **CREATE** drop-down.
- Click at desired locations on lines that should be tangent to the circle; refer to [Figure-17](#). Preview of circle will be displayed.

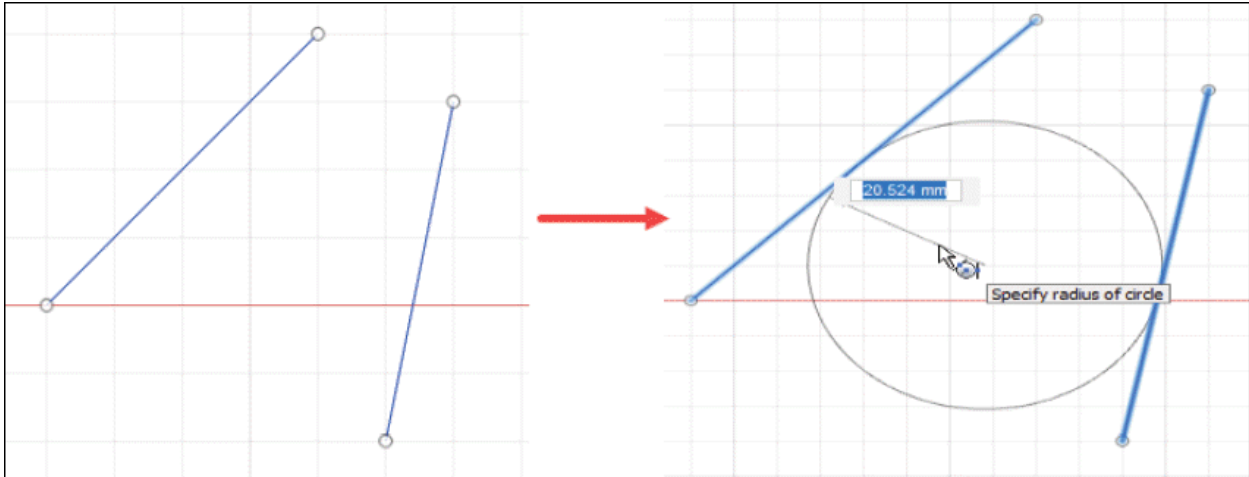


Figure-17. Creation of 2-tangent circle

- Type desired radius value of the circle in the dynamic input box and press **ENTER** to create a circle or click at desired location to define diameter of the circle.
- Press **ESC** to exit the tool.

3-Tangent Circle

The **3-Tangent Circle** tool is used to create a circle using three tangent lines. The procedure to use this tool is discussed next.

- Click on **3-Tangent Circle** tool of **Circle** cascading menu from **CREATE** drop-down.
- Select three lines to be tangent to the circle. You can also use window selection to select the lines; refer to [Figure-18](#). The preview of circle will be displayed.
- Right-click on the screen and click **OK** button from **Marking menu** to create the circle. You can also press **ESC** key to exit the tool.

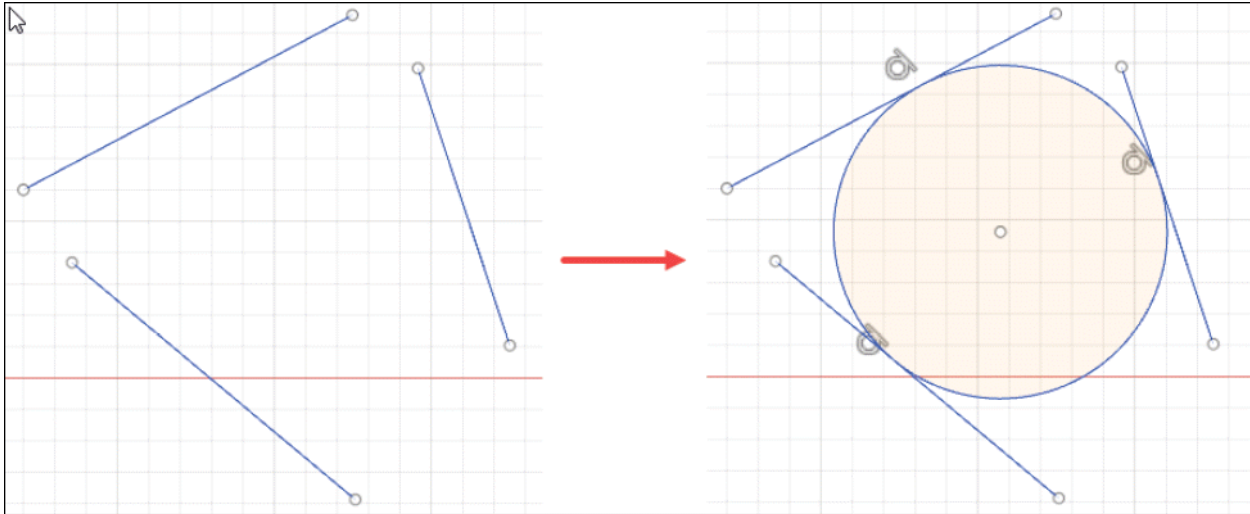


Figure-18. Creation of 3-tangent circle

Arc

The **Arc** tool is used to create arcs. There are three tools in the **Arc** cascading menu; **3-Point Arc**, **Center Point Arc**, and **Tangent Arc**; refer to [Figure-19](#).

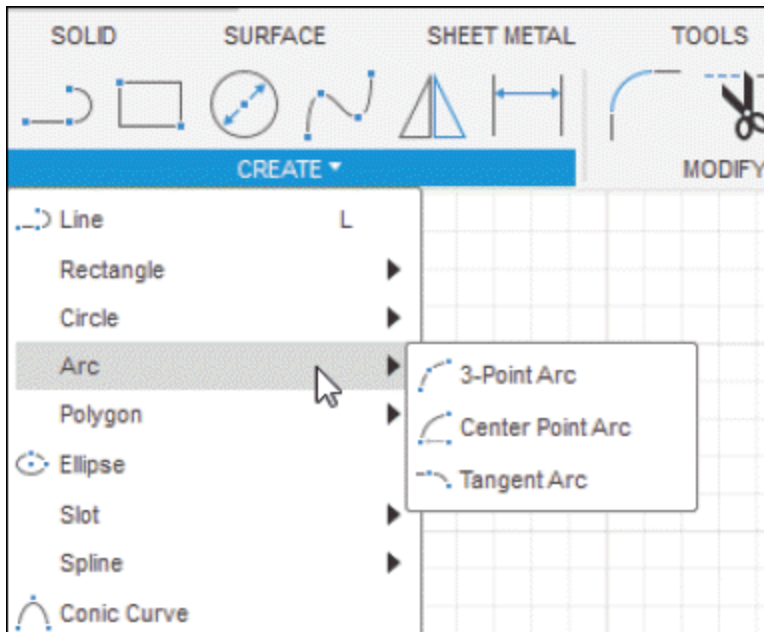


Figure-19. Arc cascading menu

The tools of **Arc** cascading menu are discussed next.

3-Point Arc

The **3-Point Arc** tool is used to create an arc using three points (start point, end point, and circumferential point). To use this tool, you need to specify two end points and one circumferential point of the arc. The procedure to use this tool is discussed next.

- Click on **3-Point Arc** tool of **Arc** cascading menu from **CREATE** drop-down. You will be asked to specify points for arc.
- Click on desired location to specify the start point of the arc; refer to [Figure-20](#).

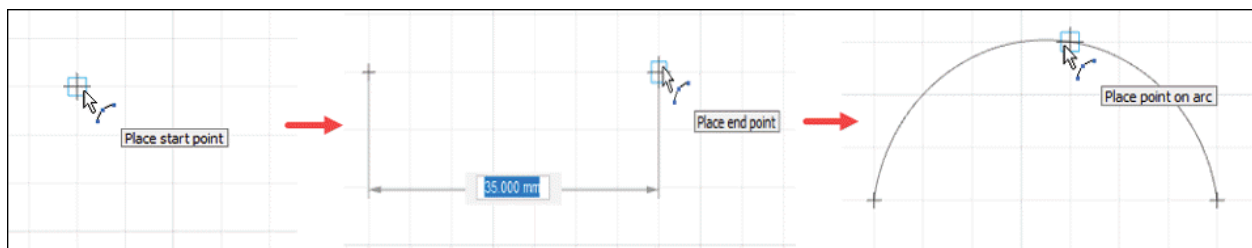


Figure-20. Creation of 3-point arc

- Click on the screen to specify the end point of the arc.
- Click at desired location to define the radius of arc. The arc will be created.
- Press **ESC** to exit the tool.

Center Point Arc

The **Center Point Arc** tool is used to create arc by defining center point, start point, and end point of the arc. The procedure is given next.

- Click **Center Point Arc** tool of **Arc** cascading menu from

CREATE drop-down. You will be asked to specify center point of the arc.

- Click on the screen canvas to specify the center point of arc.
- Click to specify the start point of the arc or type desired value of radius in dynamic input box.
- Enter desired angle value in dynamic input box to define length of arc or click on the screen to specify the end point of the arc; refer to [Figure-21](#).

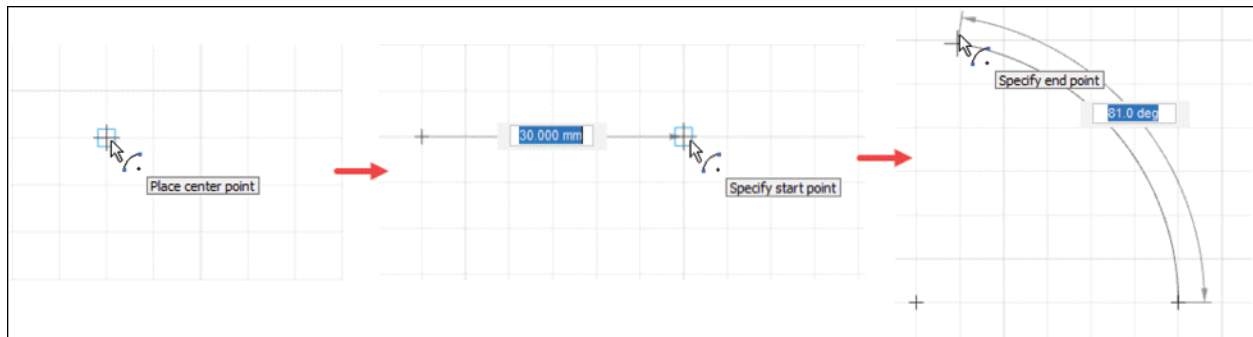


Figure-21. Creation of center point arc

- The arc will be displayed. Right-click on the screen and click **OK** button from **Marking menu** to exit the tool. You can also press **ESC** key to exit the tool.

Tangent Arc

The **Tangent Arc** tool is used to create arc tangent to selected entity.

- Click on the **Tangent Arc** tool of **Arc** cascading menu from **CREATE** drop-down.
- Click near the end point of an entity (line, arc, or curve) from screen to specify the start point of the arc; refer to

[Figure-22](#). A rubber-band arc will be attached to cursor starting from selected point.

- Click at desired location to specify the end point of the arc.

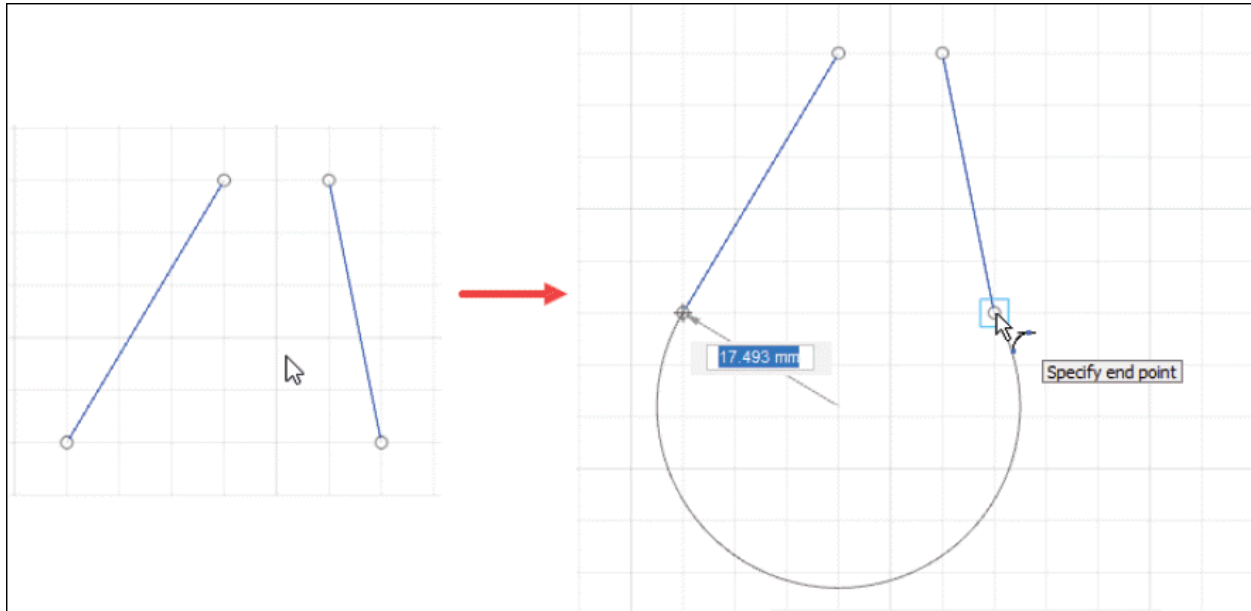


Figure-22. Creation of tangent arc

- Right-click on the screen and click **OK** button from **Marking menu** to complete the process. You can also press **ESC** key to exit the arc tool.

What happens when you try to create an arc at corner location where two entities meet and selected point is common to both the entities; refer to [Figure-23](#). Will the arc be created tangent to line 1 or line 2? What if the arc is being created tangent to line 1 but we want to create it tangent to line 2. In such conditions, after selecting end point, move the cursor in line with entity to be tangent for few millimeters and then specify the end point at desired location; refer to [Figure-24](#).

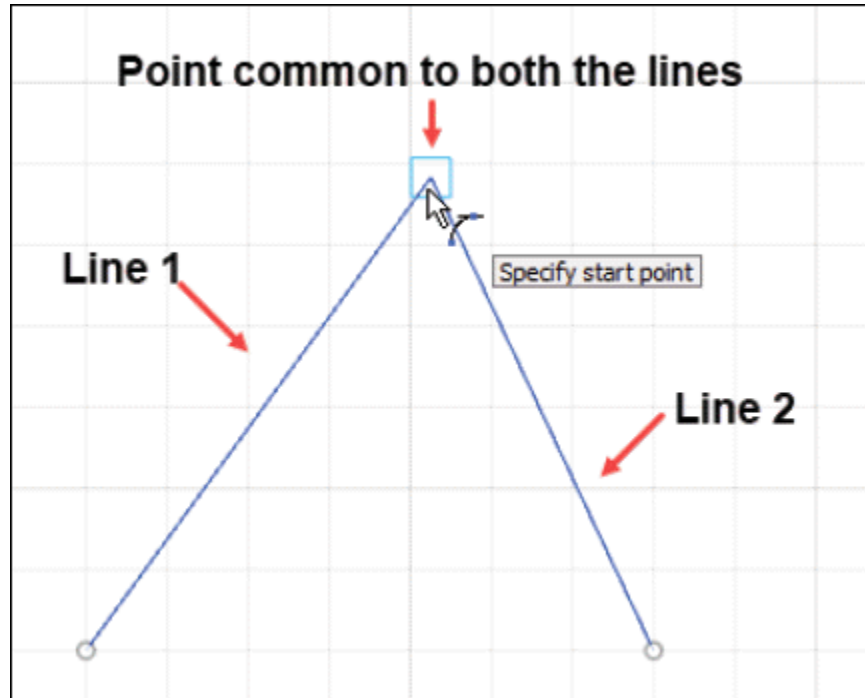


Figure-23. Tangent Arc Problem At Corner Point

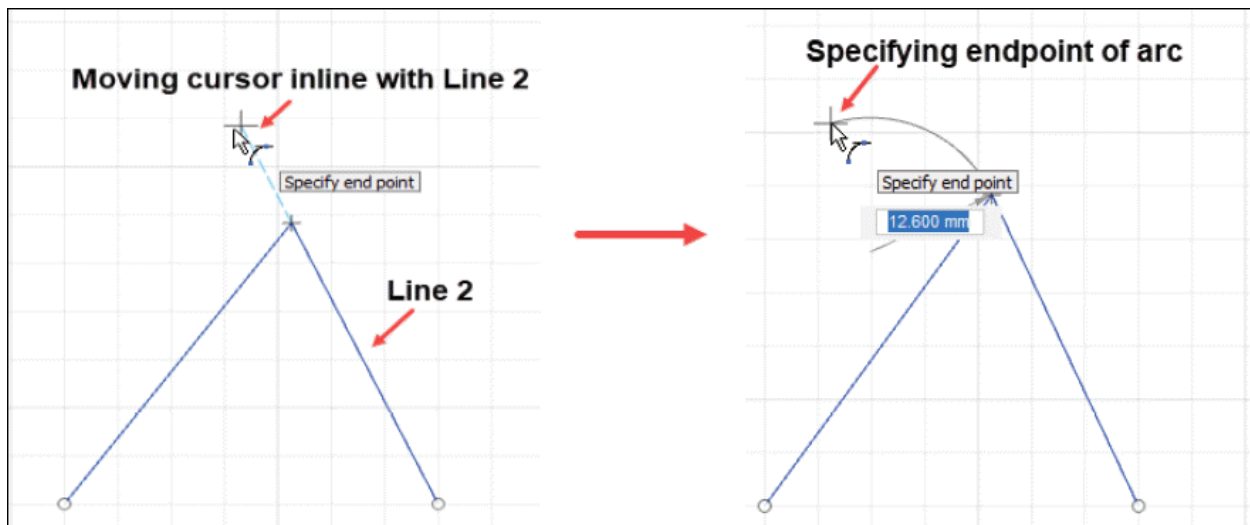


Figure-24. Creating Arc Tangent To Line

Polygon

The **Polygon** tool is used to create polygon of desired number of sides. There are three tools to create polygon in **Polygon** cascading menu; **Circumscribed Polygon**,

Inscribed Polygon, and **Edge Polygon**; refer to [Figure-25](#). The tools to create the polygons are discussed next.

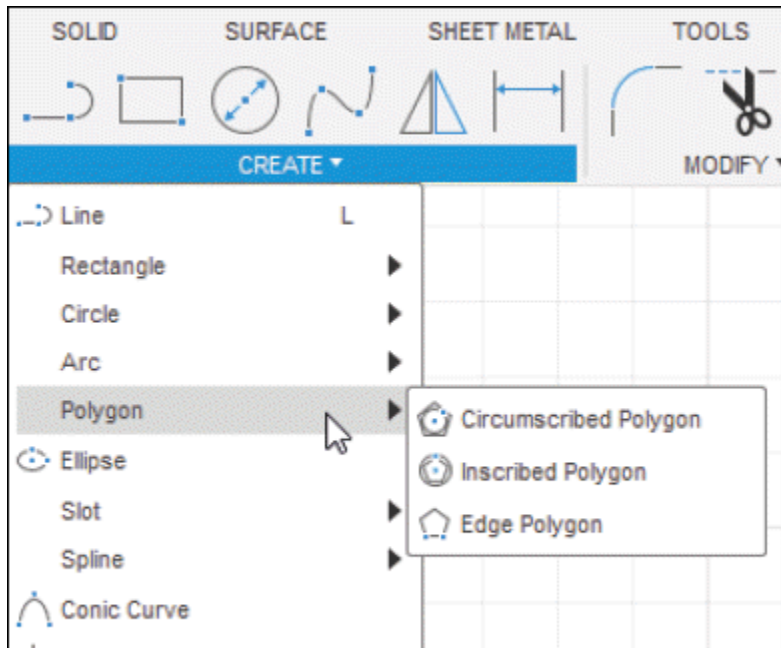


Figure-25. Polygon cascading menu

Circumscribed Polygon

Use the **Circumscribed Polygon** tool to create a polygon formed outside the reference circle. Note that midpoints of polygon lines will lie on the reference circle. The procedure to use this tool is discussed next.

- Click on the **Circumscribed Polygon** tool of **Polygon** cascading menu from **CREATE** drop-down; refer to [Figure-25](#). You will be asked to specify center point of the reference circle.
- Click on the screen to specify the center point of the polygon; refer to [Figure-26](#). You will be asked to specify radius of reference circle.

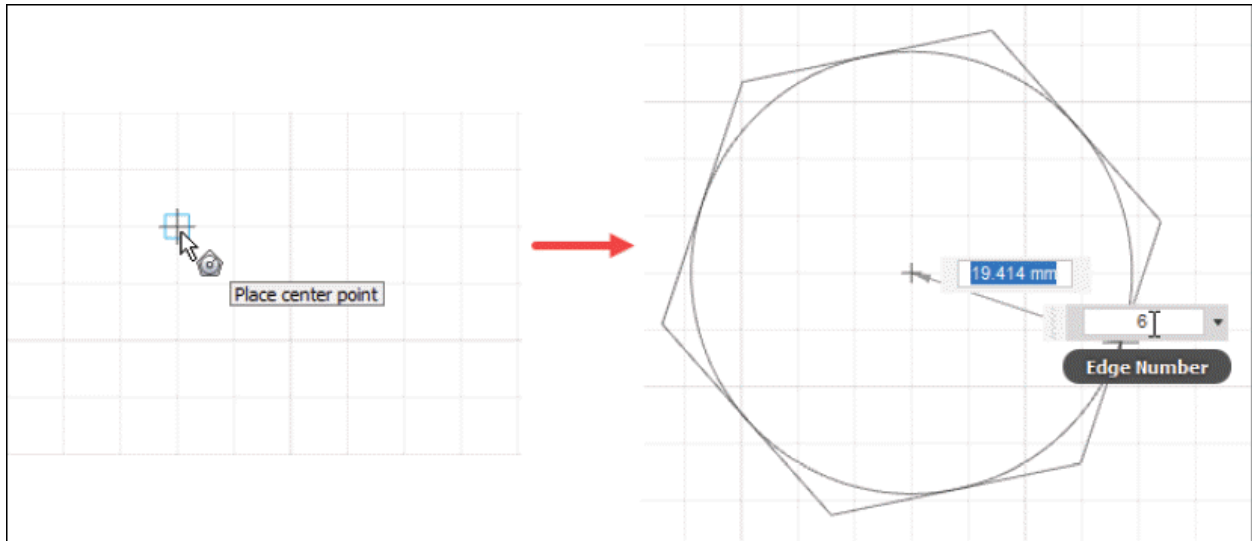


Figure-26. Creation of circumscribed polygon

- Type the value of radius in the edit box and press **TAB** to activate **Edge Number** edit box.
- Specify desired number of edges of the polygon in **Edge Number** edit box.
- After specifying the parameters, press **ENTER** to create polygon.

Inscribed Polygon

The **Inscribed Polygon** tool is used to create a polygon formed inside the reference circle. The procedure to use this tool is discussed next.

- Click on **Inscribed Polygon** of **Polygon** cascading menu from **CREATE** drop-down. You will be asked to specify center point of reference circle.
- Click on the screen to specify the center point of the circle; refer to [Figure-27](#).

- Specify the distance between center point of polygon and vertex of polygon edge in dynamic input box.
- Press **TAB** and specify the value of number of edges in **Edge Number** dynamic input box.
- Press **ENTER** to create polygon.

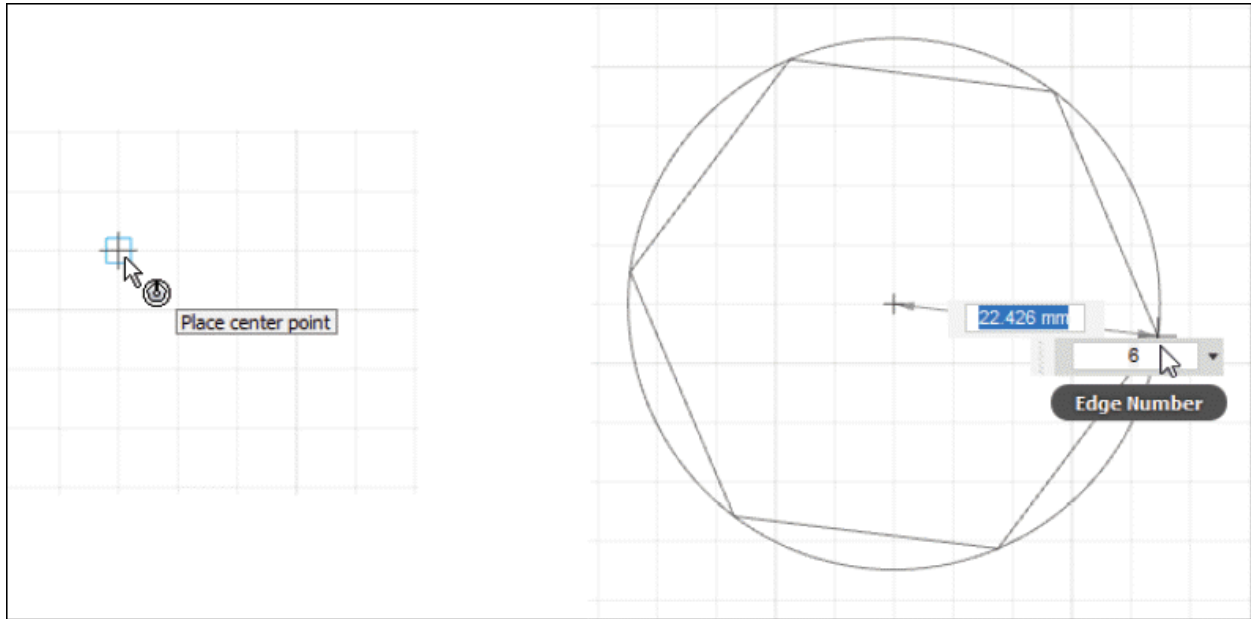


Figure-27. Creation of inscribed polygon

Edge Polygon

The **Edge Polygon** tool is used to create a polygon by defining single edge and number of edges of the polygon. The procedure to use this tool is discussed next.

- Click on the **Edge Polygon** tool of **Polygon** cascading menu from **CREATE** drop-down. You will be asked to specify start and end points of the edge line.
- Click on the screen to select the start point and end point of edge. You can also specify the value of angle and length

of the edge in dynamic input boxes; refer to [Figure-28](#). After specifying the value in dynamic input boxes, move the cursor to desired side of edge where you want to create the polygon.

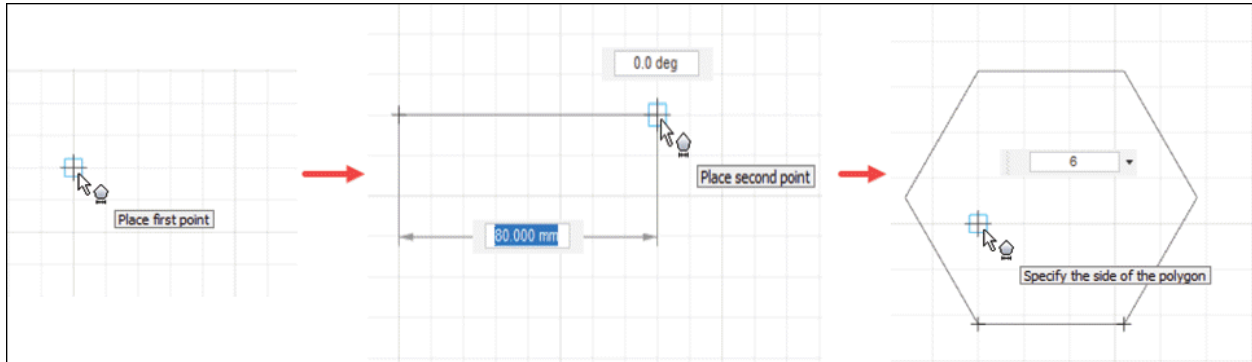


Figure-28. Creation of edge polygon

- Specify the value of number of edges of polygon in **Edge Number** input box.
- Press **ENTER** or click in desired side to create the polygon.

Ellipse

The **Ellipse** tool is used to create ellipses in the sketch. The procedure to create an ellipse is discussed next.

- Click on **Ellipse** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-29](#). You will be asked to specify center point for ellipse.

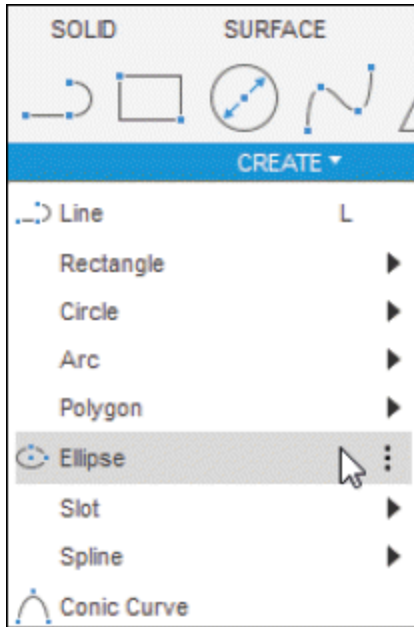


Figure-29. Ellipse tool

- Click on the screen to specify the center point for ellipse.
- Move the cursor in desired direction and specify the value of major diameter in dynamic input box. You can also specify the value of angle in **Angle** dynamic input box if you want major axis to be inclined.
- Click on the screen and move the cursor upward/downward. You will be asked to specify length of minor axis. Enter desired value of minor diameter in the input box; refer to [Figure-30](#). The ellipse will be created.

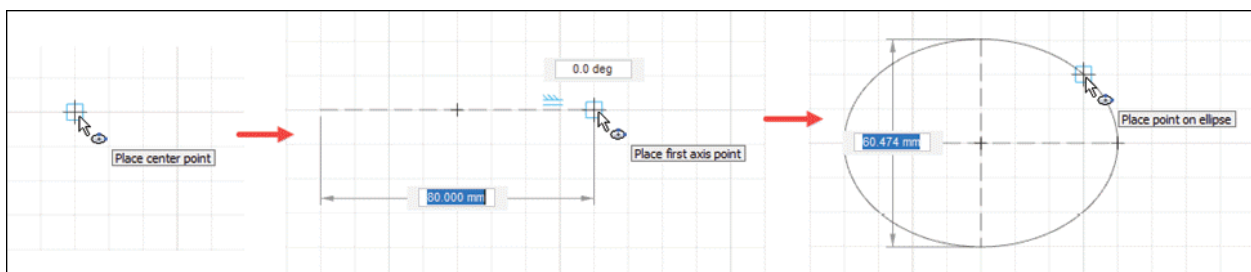


Figure-30. Creation of ellipse

- Right-click on the screen and click **OK** button to exit this tool.

Slot

The **Slot** tool is used to create slots. There are five tools in **Slot** cascading menu; **Center to Center Slot**, **Overall Slot**, **Center Point Slot**, **Three Point Arc Slot**, and **Center Point Arc Slot**; refer to [Figure-31](#).

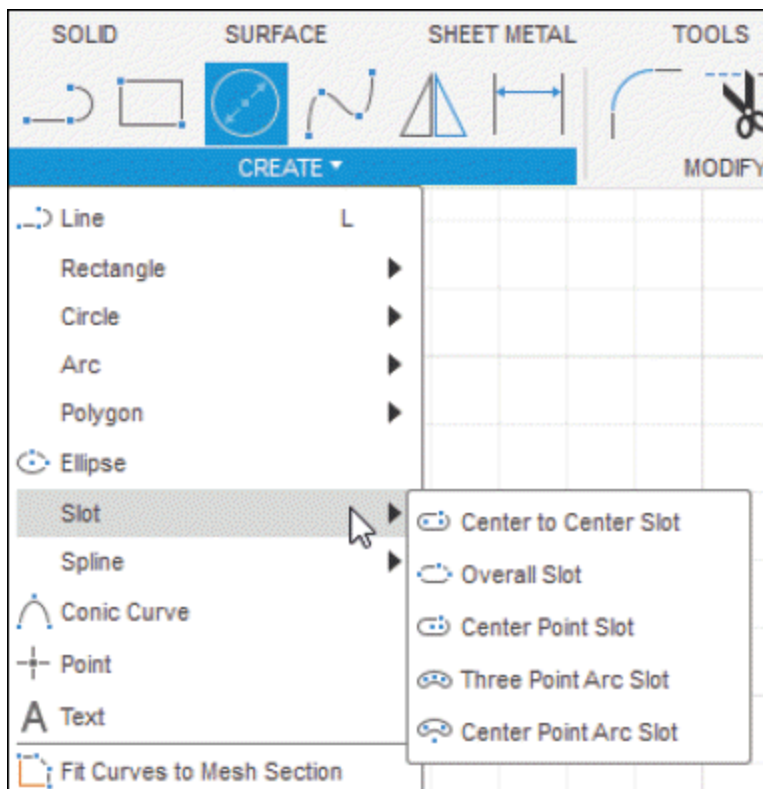


Figure-31. Slot cascading menu

The tools of **Slot** cascading menu are discussed next.

Creating Center To Center Slot

The **Center To Center Slot** tool is used to create the linear slot defined by the distance of slot arc centers and width of

slot. The procedure to use this tool is discussed next.

- Click on **Center To Center Slot** tool of **Slot** cascading menu from **CREATE** drop-down.
- Click on the screen to specify the center point of first arc of slot. You will be asked to specify center point of other arc of slot.
- Click to specify the point or specify desired parameters in the input boxes.
- Move the cursor upward/downward. You will be asked to specify width of slot/diameter of slot arc.
- Enter the value in dynamic input box or click on the screen; refer to [Figure-32](#). The slot will be created.

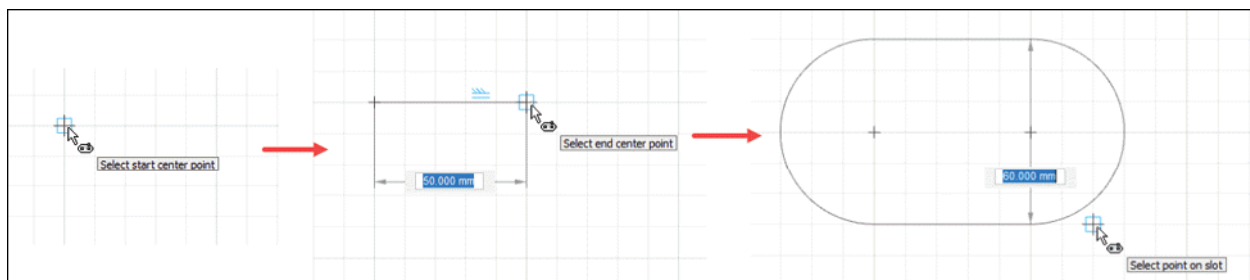


Figure-32. Creation of center to center slot

Creating Overall Slot

The **Overall Slot** tool creates a linear slot defined by total length and total width. The procedure to use this tool is discussed next.

- Click on **Overall Slot** tool of **Slot** cascading menu from **CREATE** drop-down.
- Click on the screen to specify the start point of the slot;

refer to [Figure-33](#). You will be asked to specify end point of slot.

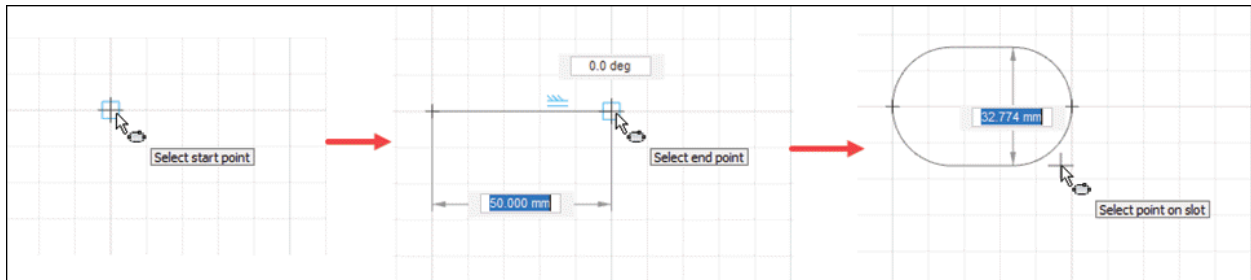


Figure-33. Creation of overall slot

- Click on the screen to specify the end point of the slot or specify the distance between start point and end point in the dynamic input box.
- Move the cursor upward/downward and specify the width of slot in dynamic input box or click at desired location.
- Press **ENTER** to create the slot and exit the tool.

Creating Center Point Slot

The **Center Point Slot** tool is used to create a linear slot which is defined by center point of slot, location of arc center, and width of slot. The procedure to use this tool is discussed next.

- Click on **Center Point Slot** tool of **Slot** cascading menu from **CREATE** drop-down.
- Click on the screen to specify the center of slot; refer to [Figure-34](#).

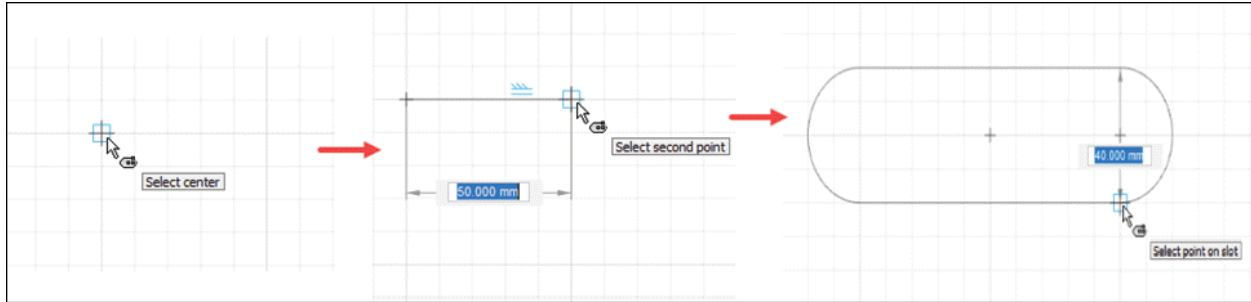


Figure-34. Creation of center point slot

- Click to specify the half length of the center to center distance of the slot.
- Specify the value of width for slot in the dynamic input box and press **ENTER** key. The slot will be created.

Creating Three Point Arc Slot

The **Three Point Arc Slot** tool is used to create a slot along specified arc with defined width. The procedure to use this tool is given next.

- Click on the **Three Point Arc Slot** tool of **Slot** cascading menu from **CREATE** drop-down. You will be asked to specify the start point of center arc.
- Click at desired location. You will be asked to specify end point of center arc.
- Click to specify the end point of arc. You will be asked to specify a circumferential point of the arc.
- Click at desired location and move the cursor upward/downward. Preview of arc will be displayed with dynamic width value.
- Type desired value of slot width in dynamic input box and

the press **ENTER** to create the slot; refer to [Figure-35](#).

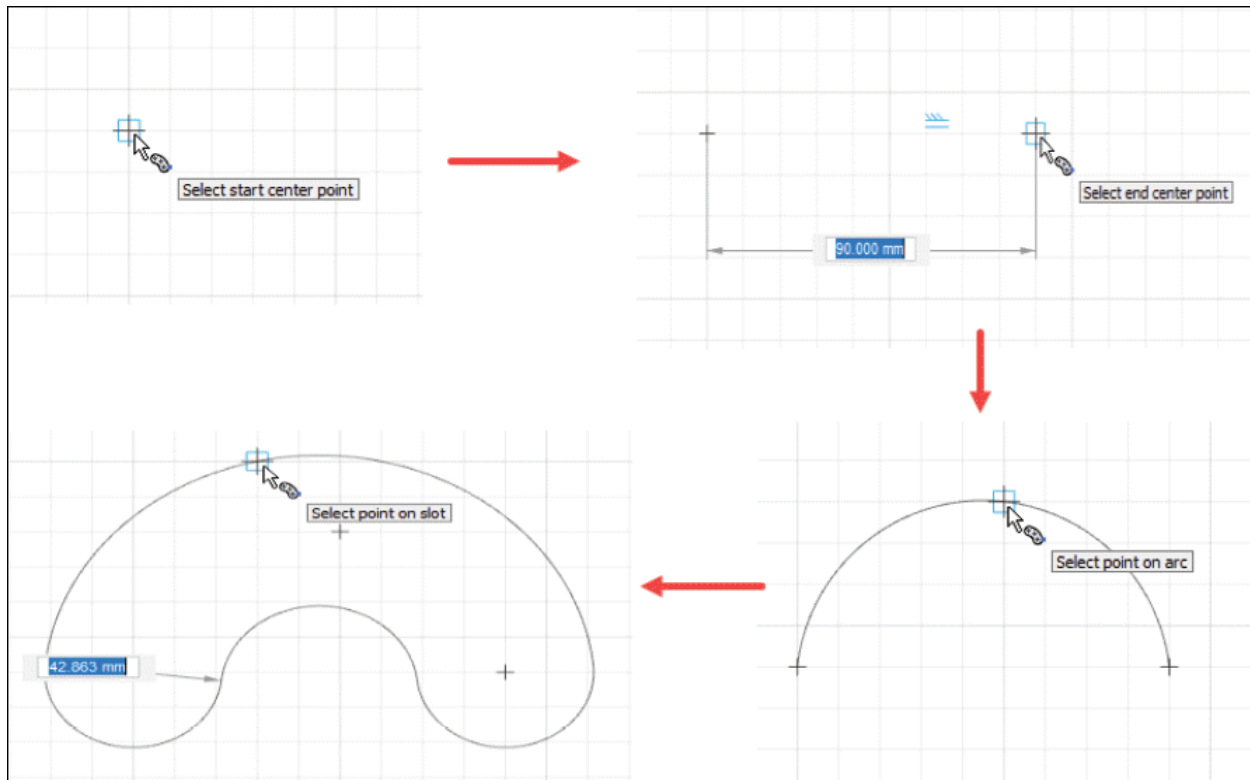


Figure-35. Creation of 3-point arc slot

Creating Center Point Arc Slot

The **Center Point Arc Slot** tool is used to create an arc slot by specifying center point of arc, start & end point of arc, and width of slot. The procedure to use this tool is given next.

- Click on the **Center Point Arc Slot** tool of **Slot** cascading menu from **CREATE** drop-down in the **Toolbar**. You will be asked to specify the center of arc.
- Click at desired location. You will be asked to specify the start point of the arc.
- Click at desired location or type desired value of arc radius

in the dynamic input box. You will be asked to specify the end point of arc.

- Click at desired location to specify end point or type desired value of angular span of arc in the dynamic input box. You will be asked to specify the width of slot.
- Click to specify the slot width or enter desired value in dynamic input box to create the slot; refer to [Figure-36](#).

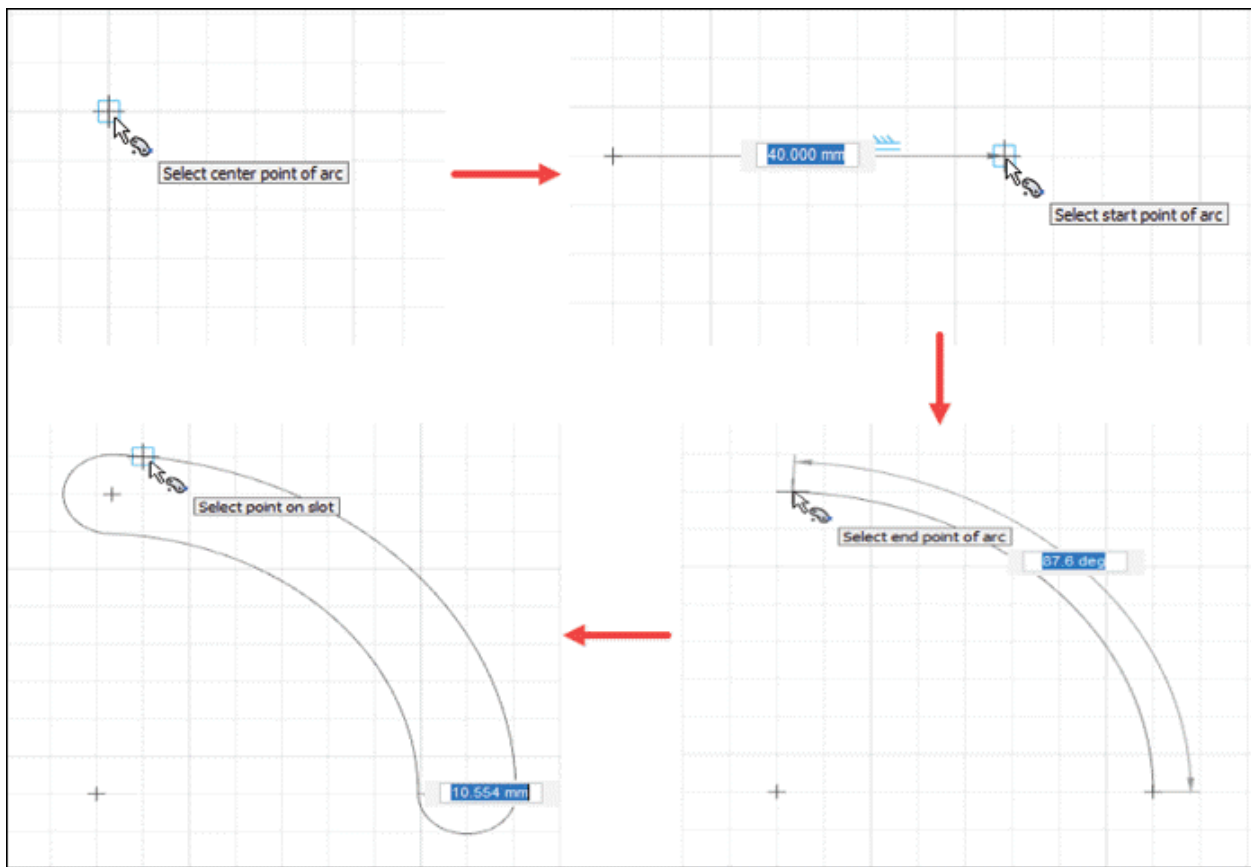


Figure-36. Creation of center point arc slot

Spline

The **Spline** tools are used to create spline curves. There are two tools in **Spline** cascading menu; **Fit Point Spline** and **Control Point Spline**; refer to [Figure-37](#).

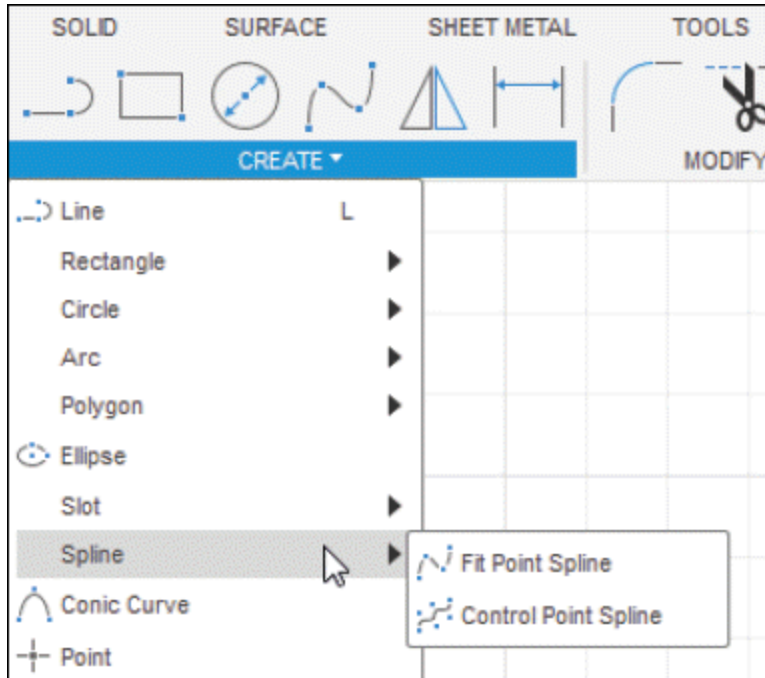


Figure-37. Spline cascading menu

The tools of **Spline** cascading menu are discussed next.

Fit Point Spline

The **Fit Point Spline** tool is used to create spline curve through the selected points. It is a freeform curve creation tool. The procedure to use this tool is discussed next.

- Click **Fit Point Spline** tool of **Spline** cascading menu from **CREATE** drop-down in the **Toolbar**; refer to [Figure-37](#).
- Click on the screen to specify the start point for the spline. You will be asked to specify next point of spline.
- Specify other points at desired locations to form spline of desired shape; refer to [Figure-38](#).

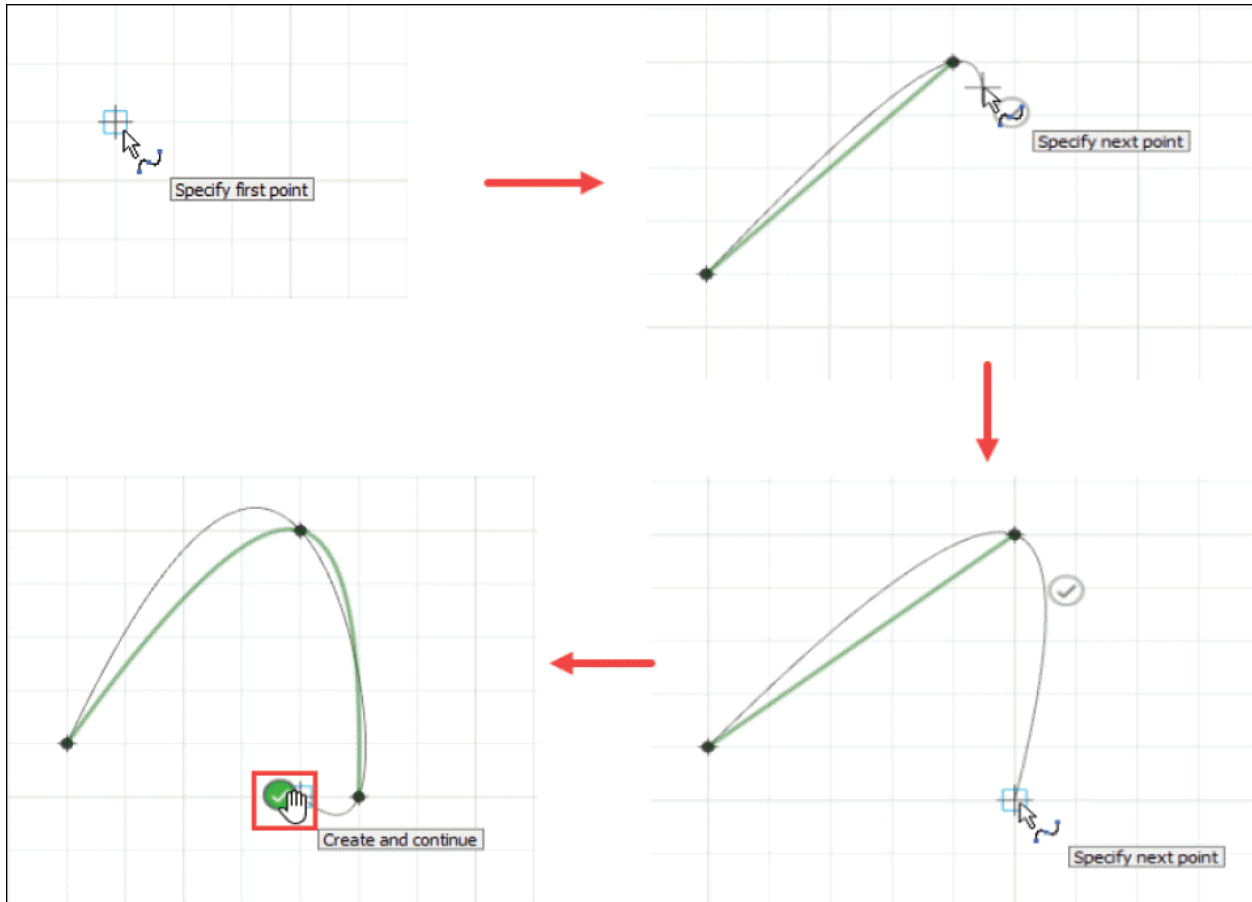


Figure-38. Creation of fit point spline

Control Point Spline

The **Control Point Spline** tool is used to create the spline curve driven by the selected control points. The procedure to use this tool is discussed next.

- Click on **Control Point Spline** tool of **Spline** cascading menu from **CREATE** drop-down in the **Toolbar**; refer to [Figure-37](#).
- Click on the screen to specify the start point for the spline. You will be asked to specify next point of spline.
- Specify the other points at desired locations to form spline

of desired shape; refer to [Figure-39](#).

- After specifying desired number of points, click on the **Tick (Create and Continue)** button to end the spline path.
- Press **ENTER** or **ESC** to exit the tool.

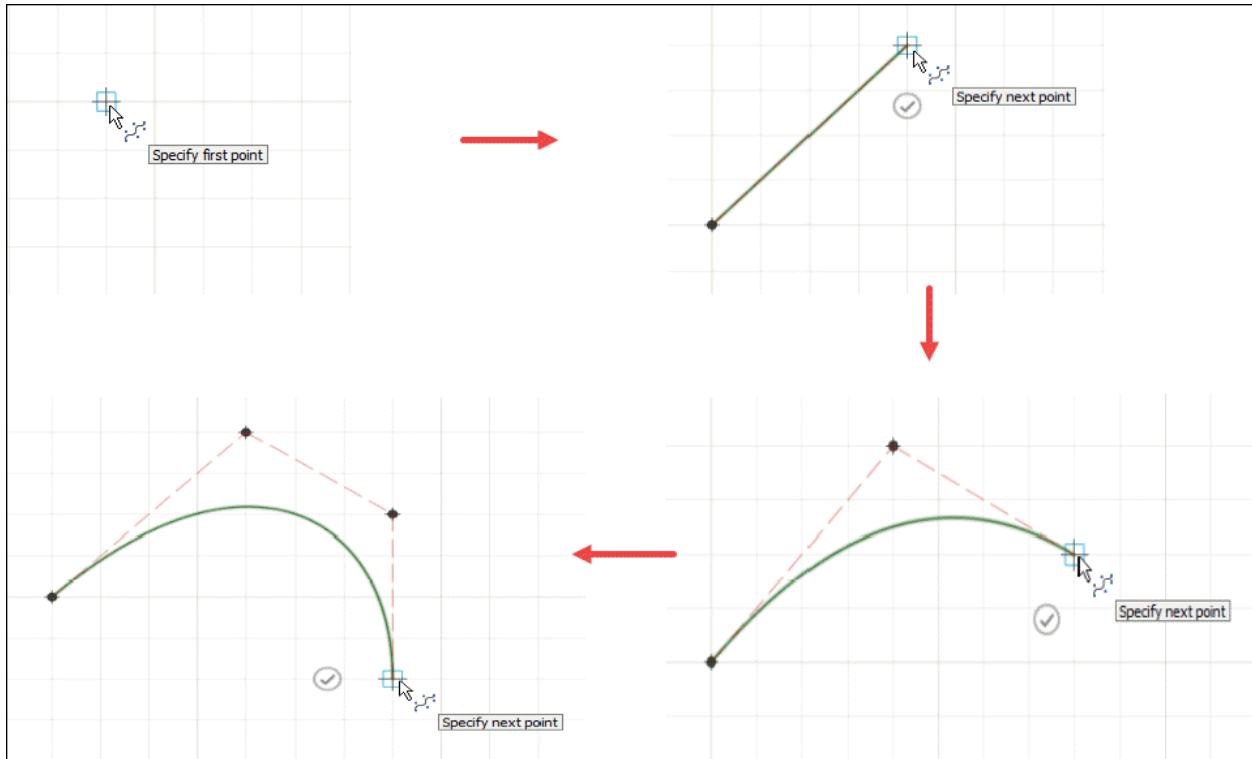


Figure-39. Creation of control point spline

Conic Curve

The **Conic Curve** tool creates a curve driven by points and rho value. The shape of the curve can be elliptical, parabolic, or hyperbolic depending on the value of Rho. The procedure to use this tool is discussed next.

- Click on **Conic Curve** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-40](#).

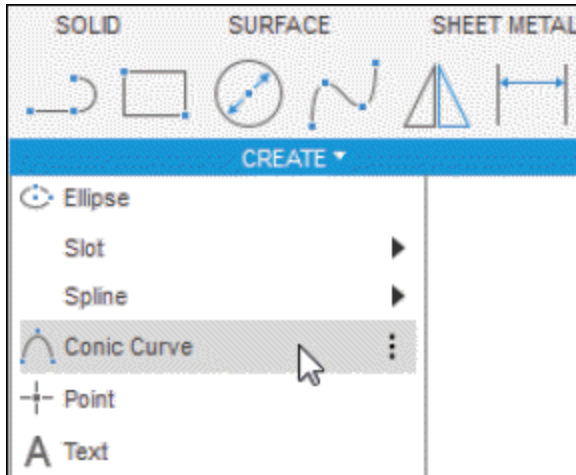


Figure-40. Conic Curve tool

- Click on the screen to specify the start point of the conic curve.
- Click on the screen to specify the end point of the conic curve.
- Click on the screen to specify control point.
- Enter the **Rho** value in **Rho** dynamic input box for the conic curve; refer to [Figure-41](#). The curve will be created. The value of rho lies between **0** to **0.99**.

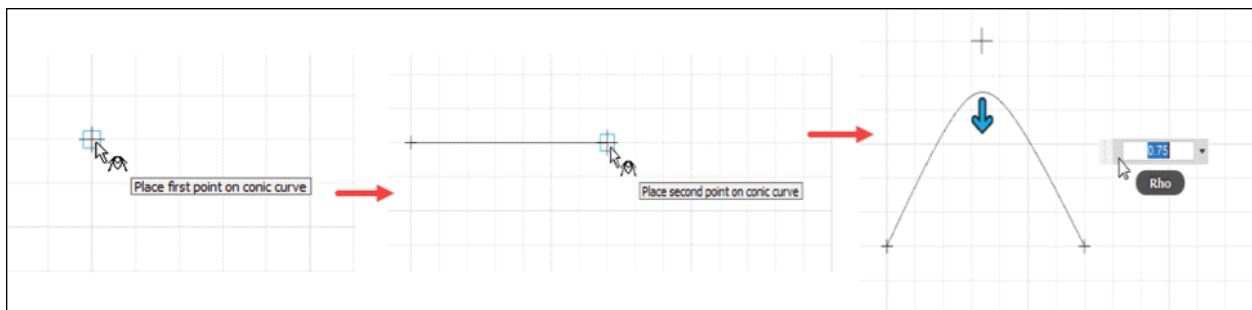


Figure-41. Creation of conic curve

Point

The **Point** tool is used to create sketch points. Sketch points

find their major usage when creating surfaces. The point gives the flexibility to parametrically change the surface design.

- Click on **Point** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-42](#).

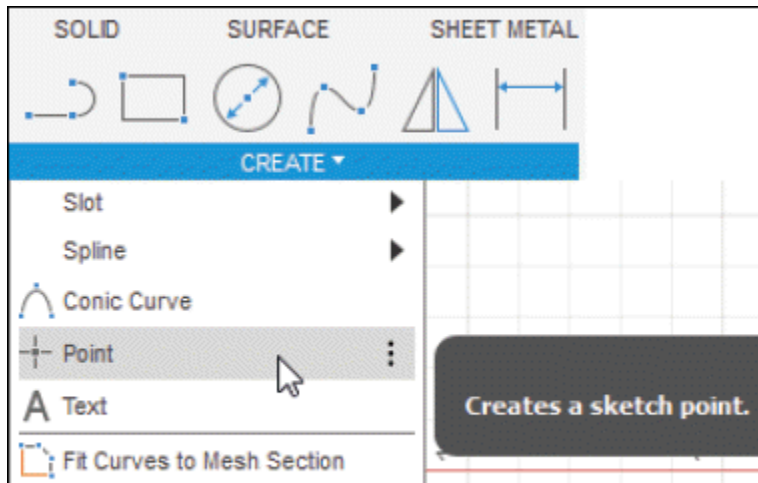


Figure-42. Point tool

- Click at desired locations in the sketch canvas to create points; refer to [Figure-43](#).

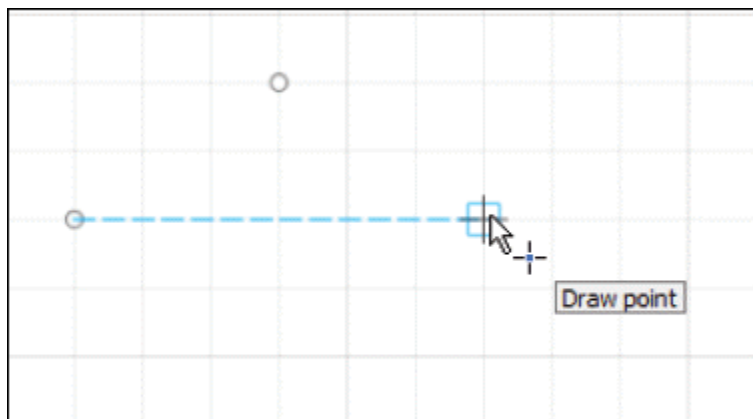


Figure-43. Creation of point

- Press **ESC** key to exit this tool or click **OK** button from

Marking menu.

Text

The **Text** tool is used to create text which can be used for embossing/engraving on a solid model. The **Text** tool is also used to create notes and other information for the model.

- Click on **Text** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-44](#). The **TEXT** dialog box will be displayed along with the symbol of text attached to the cursor asking you to specify first corner of the text frame; refer to [Figure-45](#).

Note that **Text** option is selected by default in the **Type** section of the dialog box. Select **Text** option to create the text inside a rectangular frame; refer to [Figure-45](#) and select **Text On Path** option to create the text along selected path; refer to [Figure-46](#).

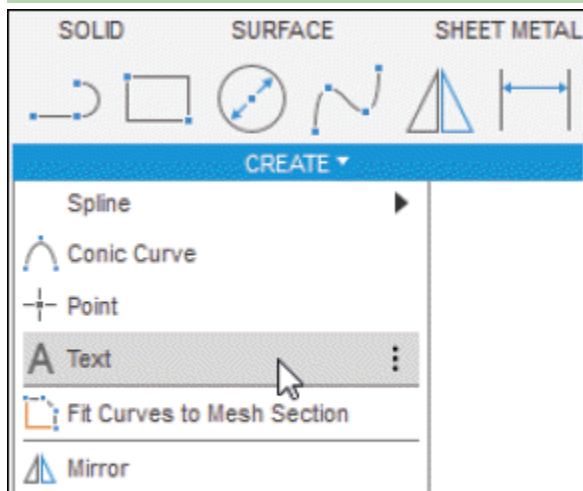


Figure-44. Text tool

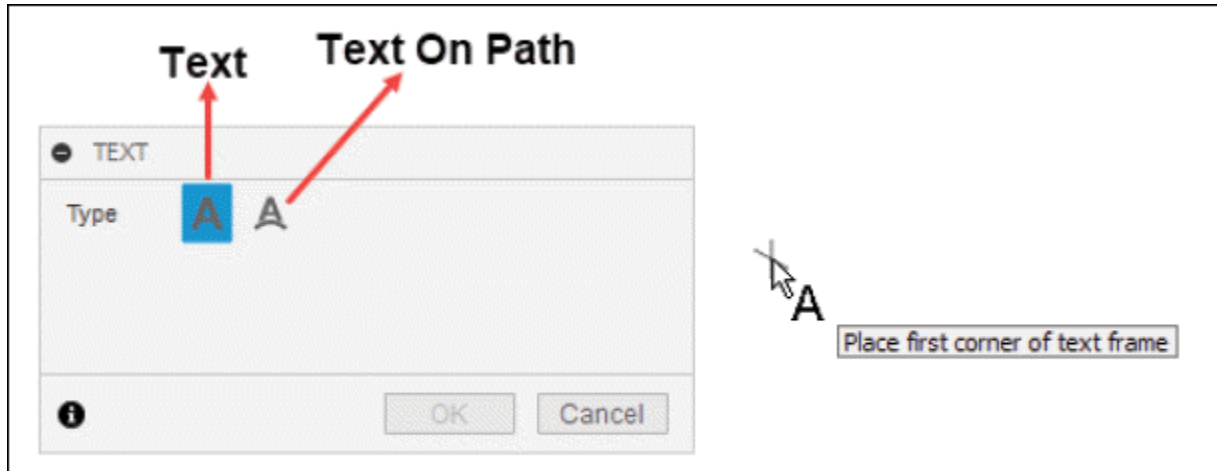


Figure-45. TEXT dialog box with text symbol attached to cursor

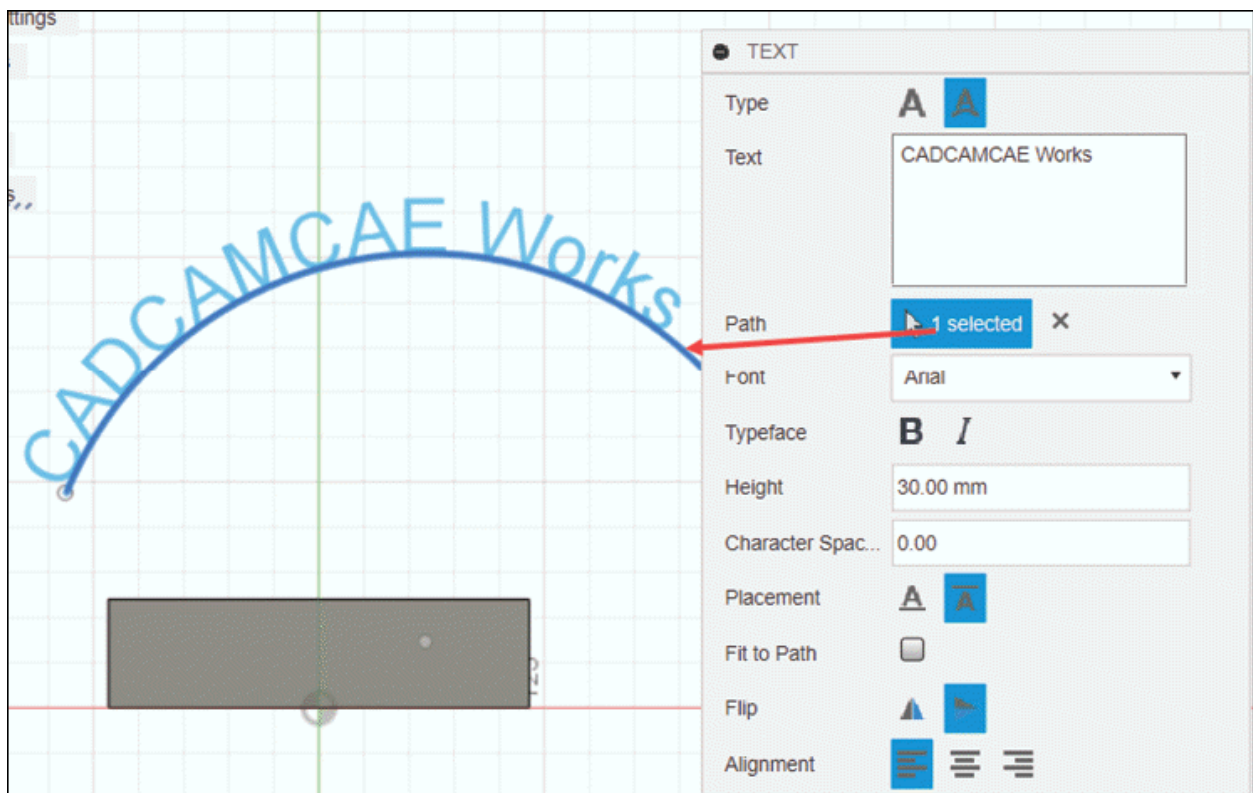


Figure-46. Creating text on path

- Click at desired location to place first corner of text frame. You will be asked to specify second corner of text frame.
- Move the cursor away and click at desired location to place

second corner of text frame. The text frame will be created and the **TEXT** dialog box will be updated; refer to [Figure-47](#).

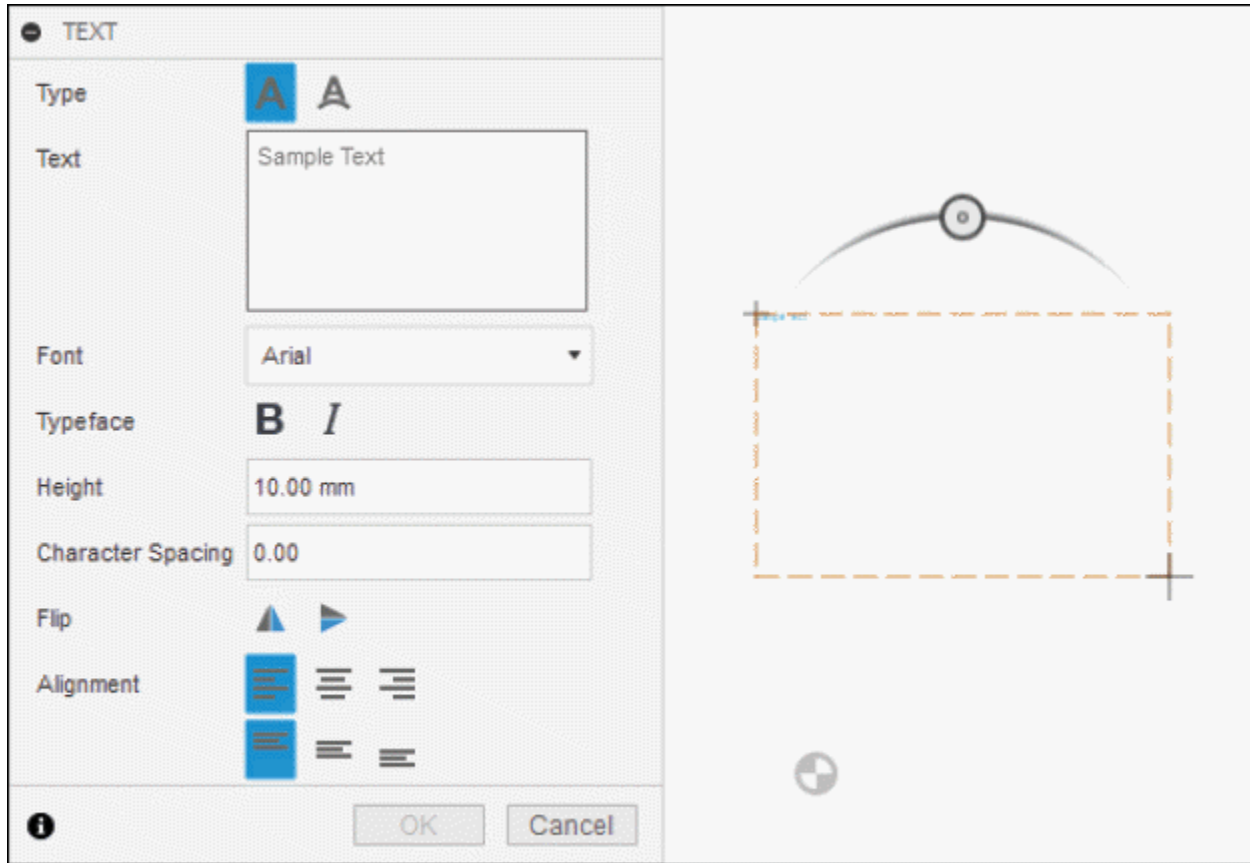


Figure-47. Updated TEXT dialog box with frame created

- Click in the **Text** edit box and type desired text. Preview of text will be displayed in real-time; refer to [Figure-48](#).

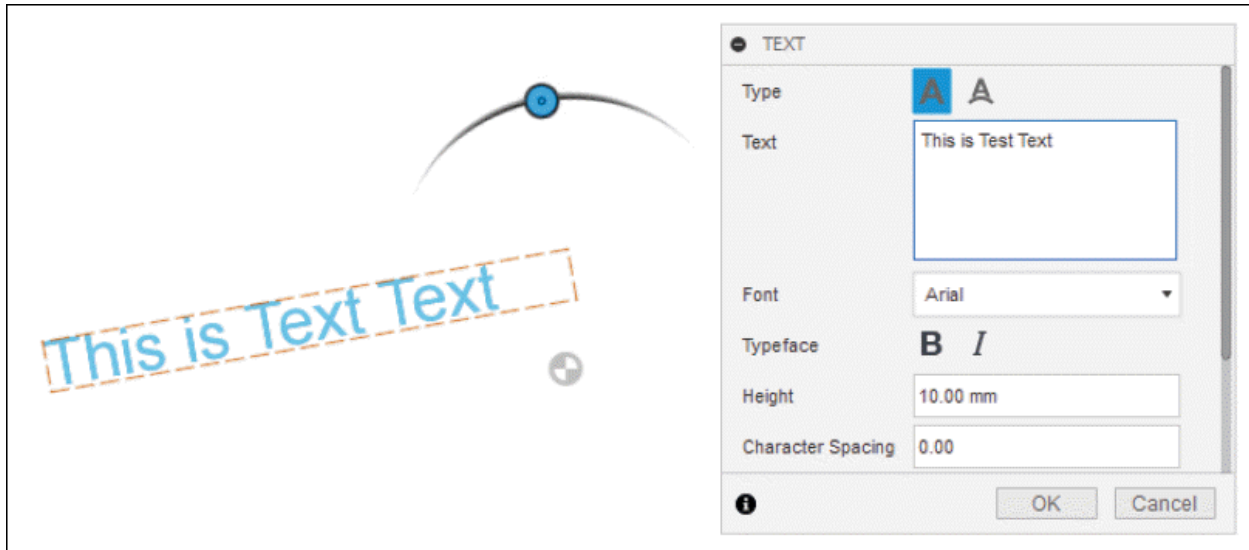


Figure-48. Preview of text created in a rectangular frame

- Click on the **Font** drop-down and select desired font for text. There are various single line text fonts available in this drop-down which are used for CAM to name or mark the part. These fonts are of **.shx** format like **cdm.shx**.
- Select desired style for text in **Typeface** section. You can set the text as bold and italic by selecting respective buttons.
- Click in **Height** edit box and type desired value of height for text.
- Click in the **Character Spacing** edit box and type desired value to specify change in default spacing between characters.
- If you want to flip the text then select desired button from the **Flip** section.
- Select **Align Left**, **Align Center**, or **Align Right** option

from Alignment section to align the text horizontally and select **Align Top**, **Align Middle**, or **Align Bottom** option from **Alignment** section to align the text vertically.

- Click **OK** button of the **TEXT** dialog box to complete the process.
- If you want to create the text on path then create the sketch for path and select **Text On Path** option from **Type** section of the dialog box; refer to [Figure-45](#). The **TEXT** dialog box will be updated; refer to [Figure-49](#).

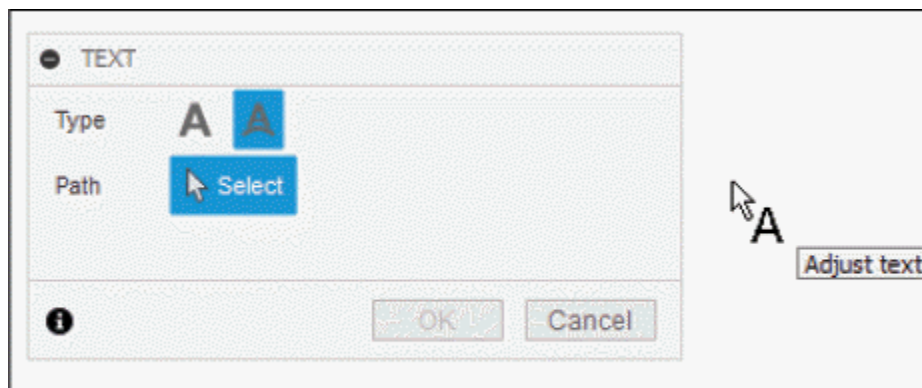


Figure-49. TEXT dialog box with Text On Path options

- The **Path** selection button is active by default and you are asked to select the path.
- Select the path created for the text to follow. The **TEXT** dialog box will be updated and the selected path will be displayed in the **Path** section of the dialog box; refer to [Figure-50](#).

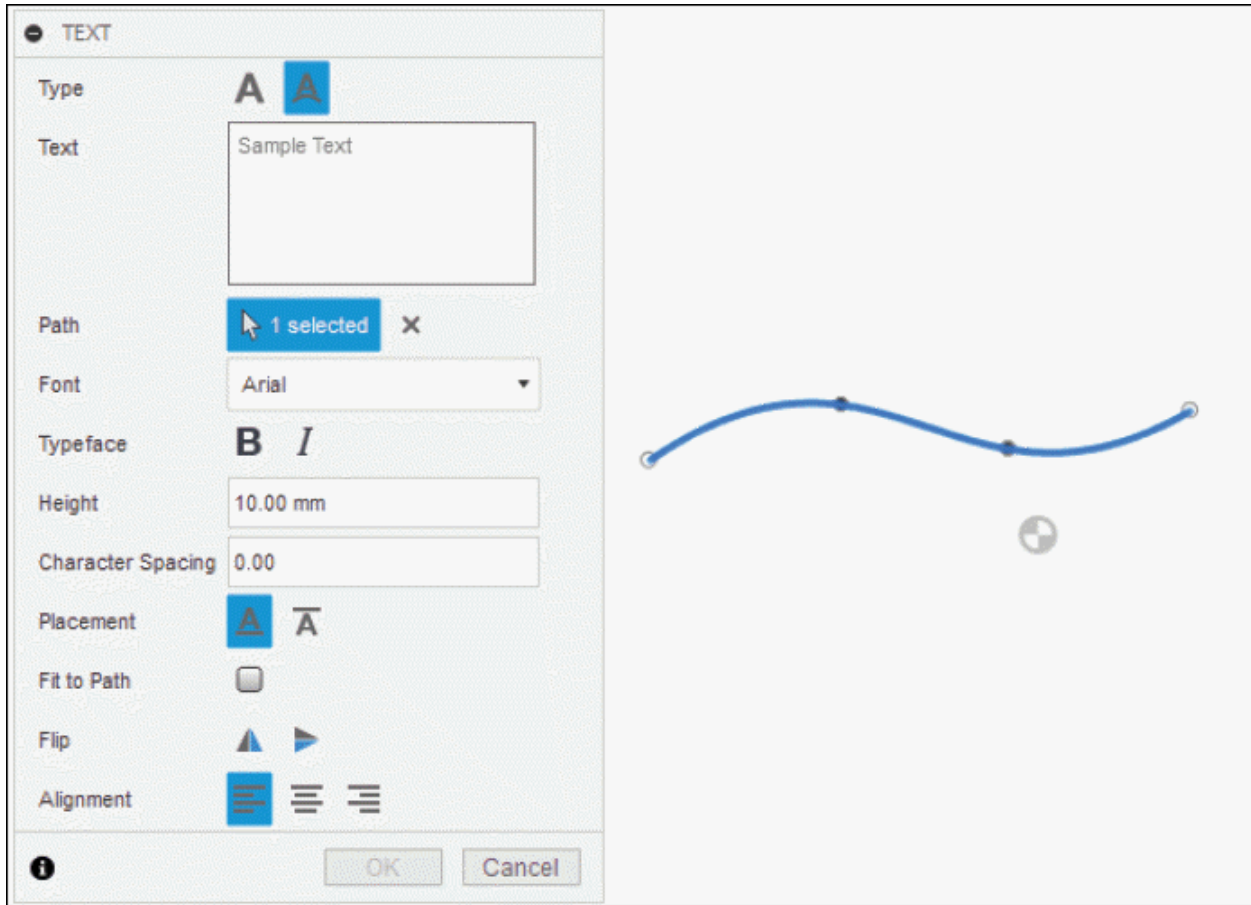


Figure-50. Updated TEXT dialog box with path selected

- Click in the **Text** edit box and type desired text. Preview of text will be displayed in real-time; refer to [Figure-51](#).

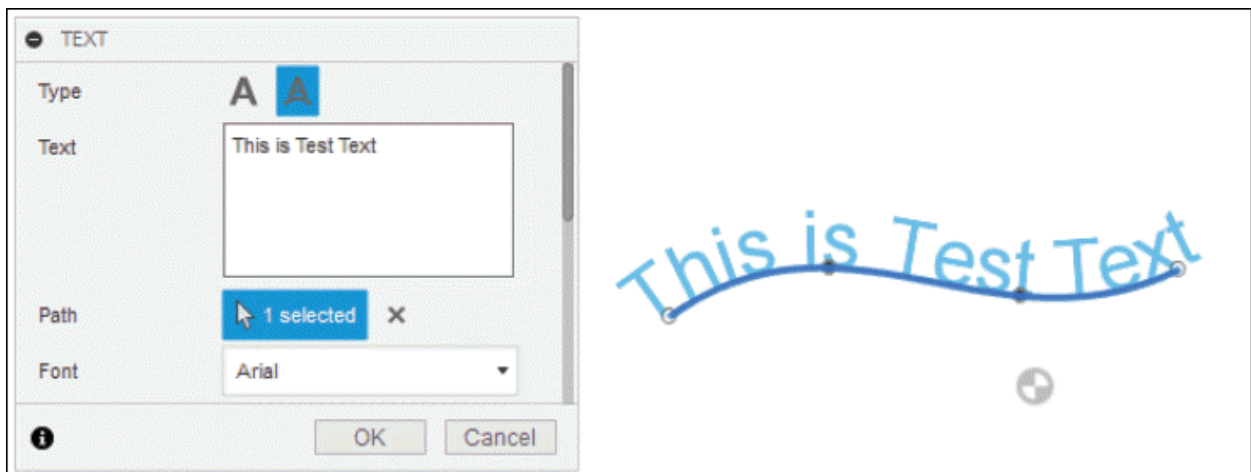


Figure-51. Preview of text created on selected path

- Select the placement of the text above or below the path by selecting respective button from **Placement** section of the dialog box.
- Select **Fit to Path** check box to distribute the text evenly along the entire path.
- Specify the other parameters in the dialog box as discussed earlier. Click the **OK** button from the dialog box to complete the process.

After creating the text, you can edit it anytime by double-clicking on it in sketch mode.

Mirror

The **Mirror** tool is used to create mirror copy of the selected entities with respect to a reference called mirror line. The procedure to create the mirror entities is given next.

- Click on **Mirror** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-52](#). The **MIRROR** dialog box will be displayed; refer to [Figure-53](#).

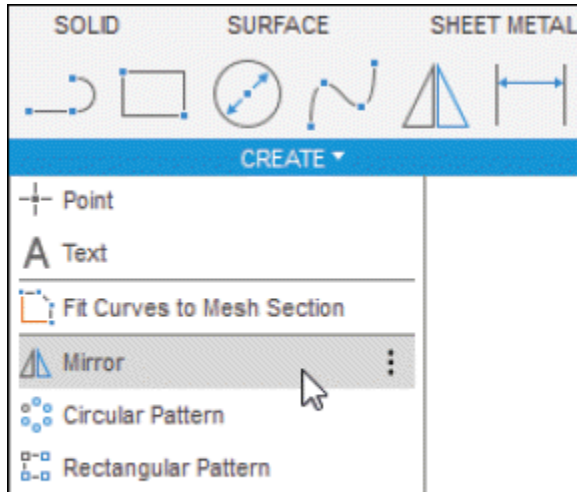


Figure-52. Mirror tool

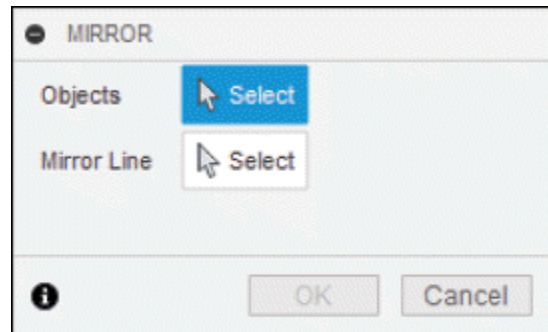


Figure-53. MIRROR dialog box

- The **Objects** selection button is active by default and you are asked to select the objects. Select the entities or sketch whose mirror copy is to be created.
- Click on **Select** button of **Mirror Line** section and select the reference line for creating mirror copy. The preview of mirror will be displayed; refer to [Figure-54](#).

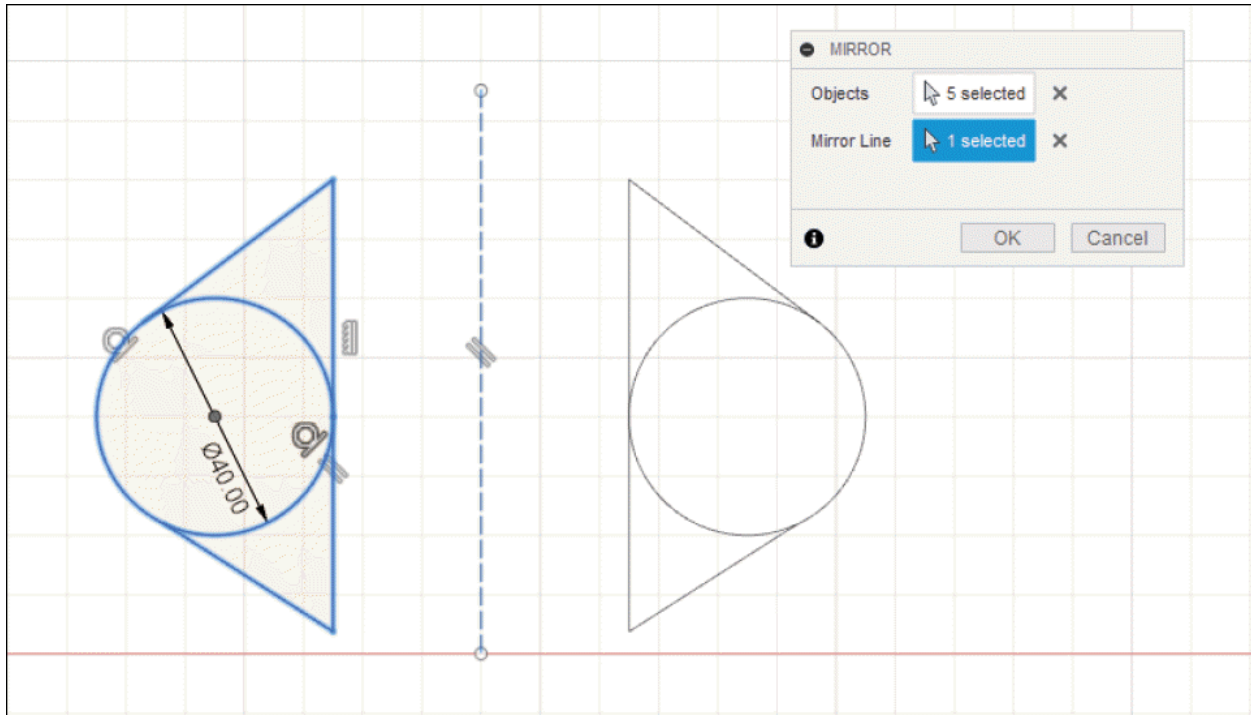


Figure-54. Procedure of mirror

- If preview of mirror is as per your requirement then click on **OK** button from **MIRROR** dialog box to complete the process.

Sometimes, we need to create multiple copies of the sketch entities like in sketch of keyboard or piano. For such cases, Autodesk Fusion 360 provides two tools, **Circular Pattern** and **Rectangular Pattern**. These tools are discussed next.

Circular Pattern

The **Circular pattern** tool is used to create multiple copies of an entity in circular fashion. The procedure to use this tool is discussed next.

- Click on **Circular Pattern** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-55](#). The **CIRCULAR**

PATTERN dialog box will be displayed; refer to [Figure-56](#).

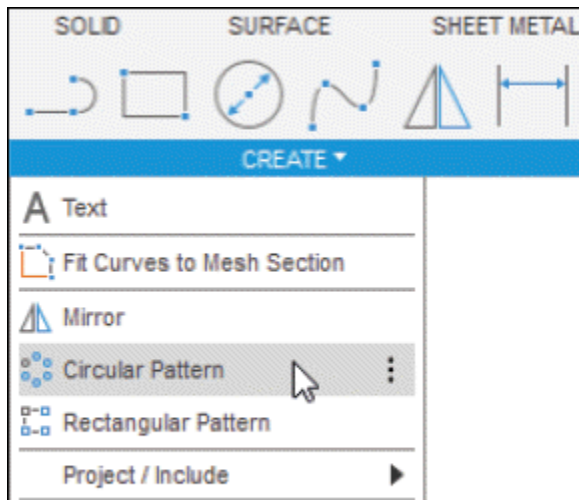


Figure-55. Circular Pattern tool

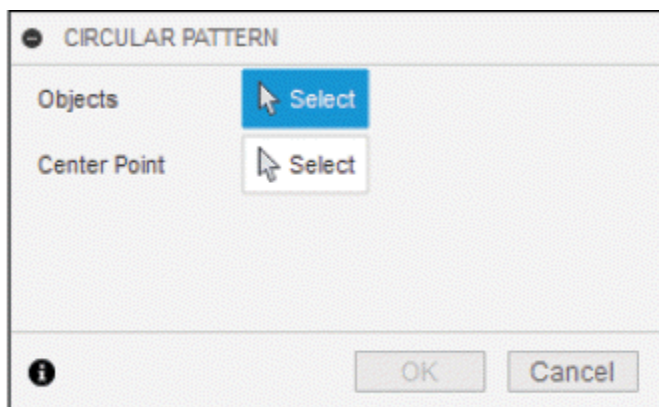


Figure-56. CIRCULAR PATTERN dialog box

- The **Objects** selection button is active by default and you are asked to select the objects. Select the entities or sketch to apply the circular pattern.
- Click on **Select** button for **Center Point** section and select the reference point to be used as center point for circular pattern. The preview of circular pattern will be displayed; refer to [Figure-57](#).

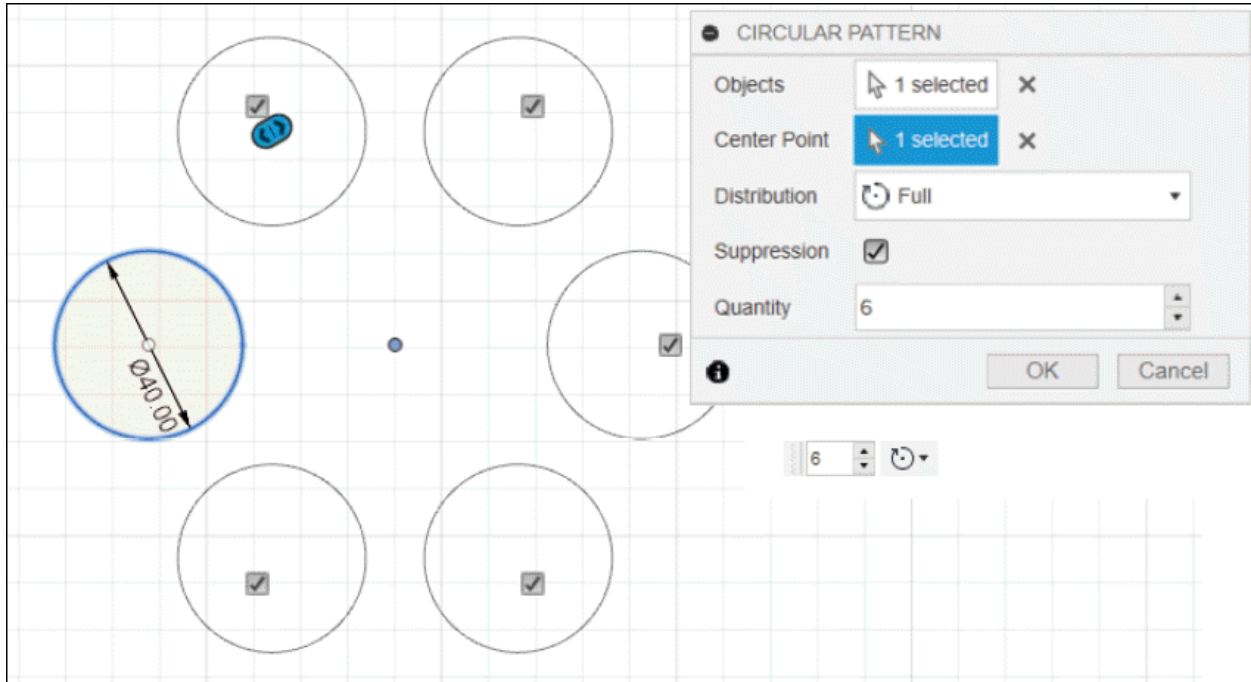


Figure-57. Creation Of circular pattern

- Select **Full** option from **Angular Spacing** drop-down if you want to create a pattern over full 360 round. Select the **Angle** option from **Angular Spacing** drop-down if you want to define the span of pattern in angle value. Select the **Symmetric** option from **Angular Spacing** drop-down if you want to create a circular pattern symmetric to the center line; refer to [Figure-58](#).

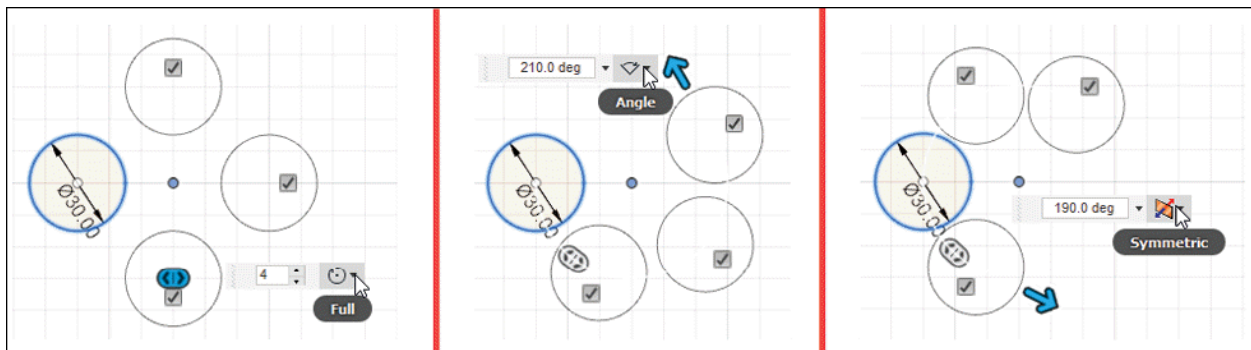


Figure-58. Tools of Type drop down

- Select the **Suppression** check box if you want to suppress any copy in pattern and then clear the check box for that entity in the preview to be suppressed. Clear the **Suppression** check box from the dialog box to generate all instances of pattern.
- Click in the **Quantity** edit box and type desired value for number of copies of object to be created in the circular pattern. You can also set the quantity by moving drag handle.
- After specifying the parameters, click on **OK** button from **CIRCULAR PATTERN** dialog box to complete the process.

Rectangular Pattern

The **Rectangular Pattern** tool is used to create multiple copies of an entity in horizontal and vertical direction. You can create pattern in two linear directions at the same time. The procedure to create rectangular pattern is given next.

- Click on the **Rectangular Pattern** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-59](#). A **RECTANGULAR PATTERN** dialog box will be displayed; refer to [Figure-60](#).

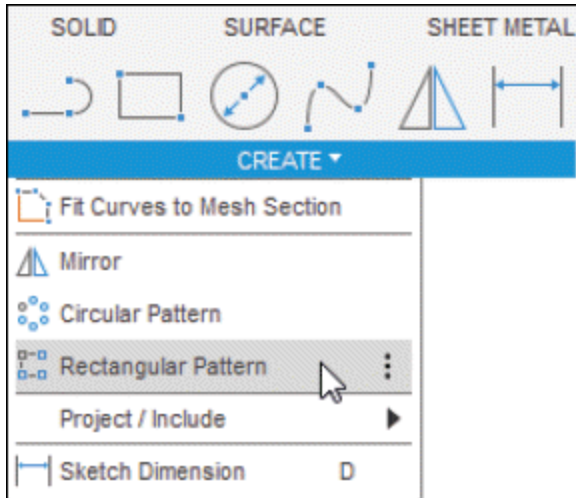


Figure-59. Rectangular Pattern tool

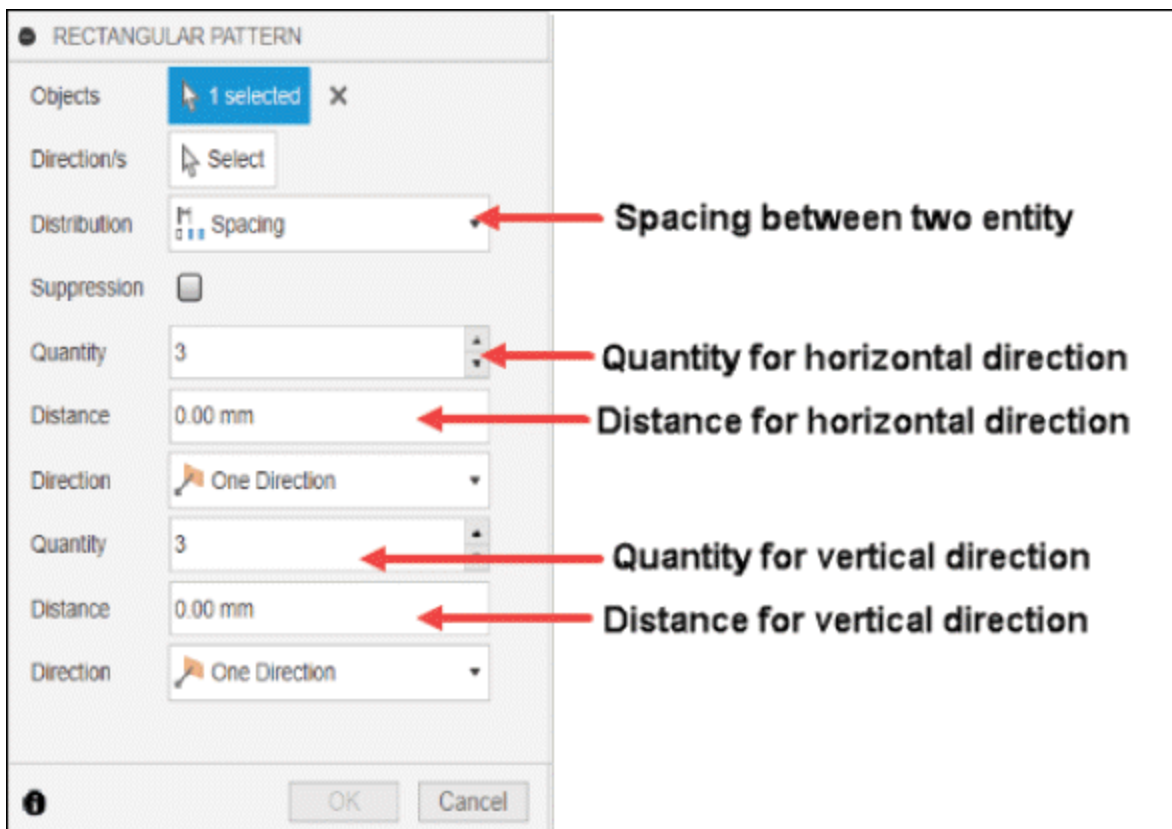


Figure-60. RECTANGULAR PATTERN dialog box

- The **Objects** section button is active by default and you are asked to select the objects. Select the sketch or entities for

rectangular pattern.

- Click on **Select** button of **Direction/s** section and select the reference lines for directions of rectangular pattern. If you do not select reference lines then patterns are automatically created along horizontal and vertical axes; refer to [Figure-61](#).

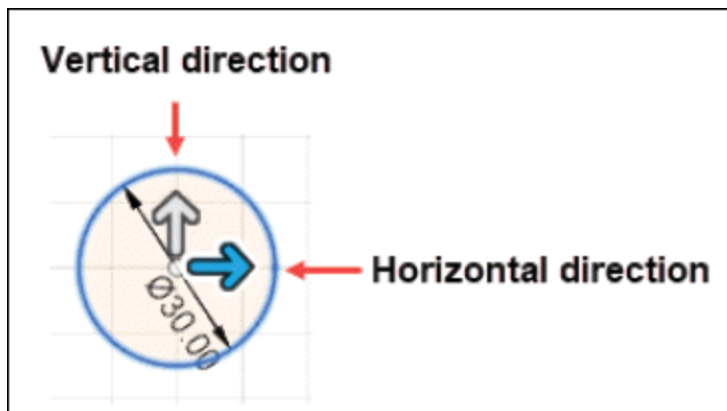


Figure-61. Rectangular pattern direction

- Select **Extent** option from **Distance Type** drop-down if you want to specify the total span in which object copies will be created. Select **Spacing** option from **Distance Type** drop-down to set distance between two consecutive copies of object. In our case, we are selecting **Extent** option.
- Select the **Suppress** check box if you want to suppress any entity in the pattern.
- Click in the **Quantity** edit box and type desired value for number of copies.
- Click in the **Distance** edit box and type desired distance

within which copies will be created; refer to [Figure-62](#).

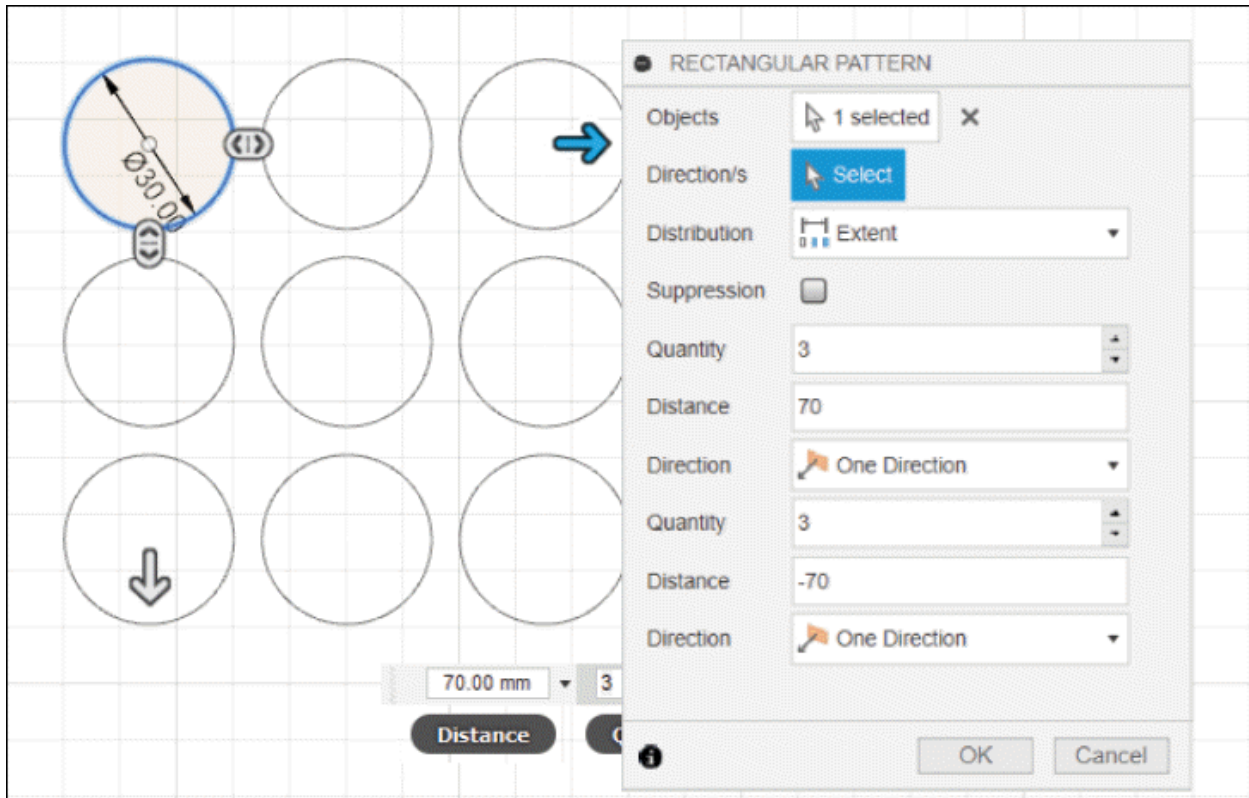


Figure-62. Creation of rectangular pattern

- Select **One Direction** option from **Direction Type** drop-down if you want to create copies in only one direction. Select **Symmetric** option from **Direction Type** drop-down if you want to create copies in both sides of selected direction.
- Similarly, specify the parameters for other direction.
- After specifying the parameters, click **OK** button from **RECTANGULAR PATTERN** dialog box to complete the process.

Project/Include

The tools in this cascading menu are used to project

edges/curves of selected solids or surfaces on the sketch. These tools will be discussed later in this book while working with 3D models. Hover on the **Project/Include** option of **CREATE** drop-down from **Toolbar** to display **Project/Include** cascading menu; refer to [Figure-63](#).

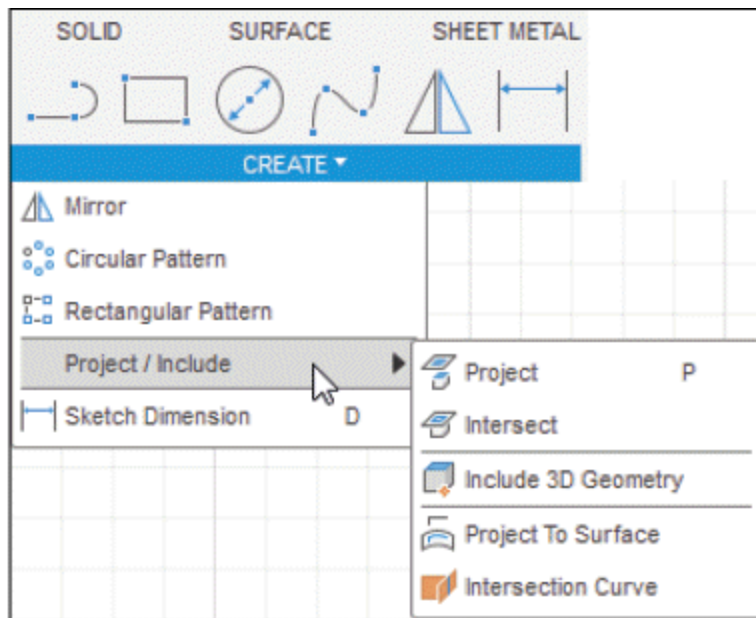


Figure-63. Project or Include cascading menu

Sketch Dimension

The **Sketch Dimension** tool is used to dimension the entities automatically based on selections. This tool can create different type of dimension like horizontal, vertical, diameter, radius, or inclined dimensions. The procedure to use this tool is discussed next.

- Click on the **Sketch Dimension** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-64](#).

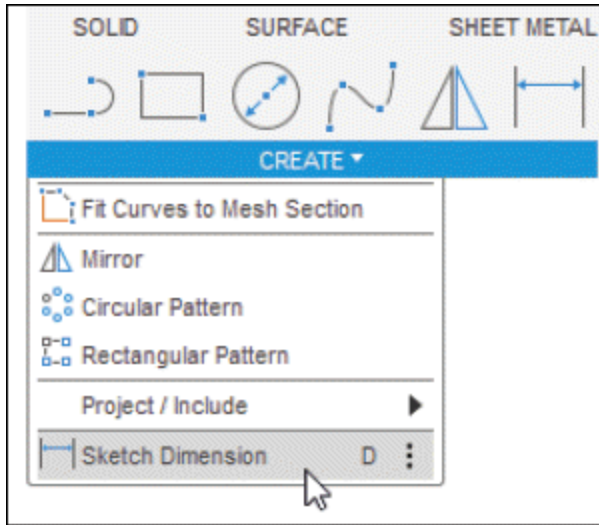


Figure-64. Sketch Dimension tool

- Select the entity you want to dimension and then click at desired distance to place dimension; refer to [Figure-65](#).

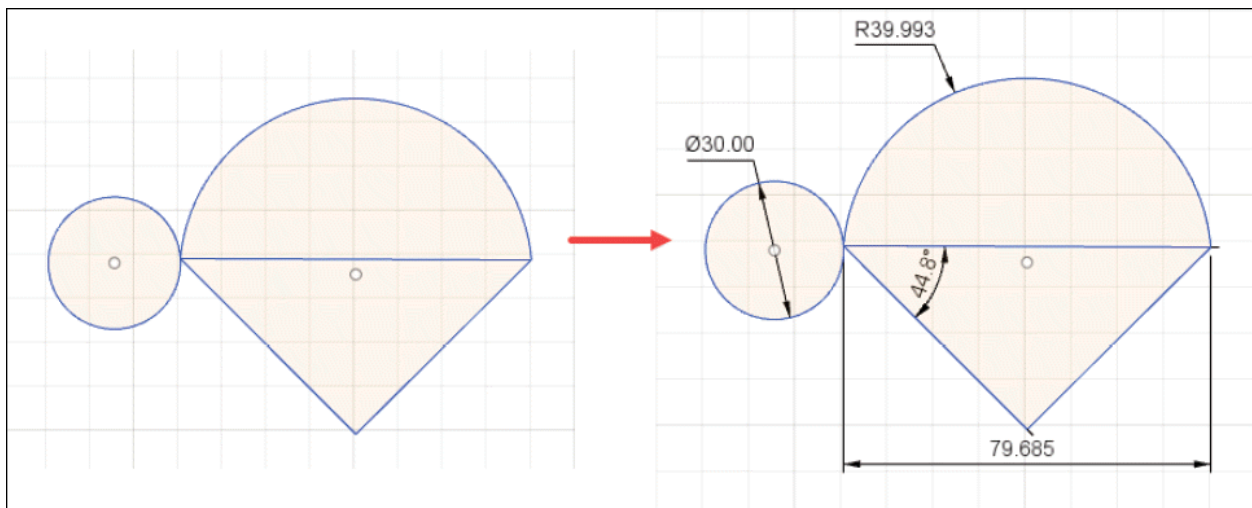


Figure-65. Applying the dimension

Based on how you move the cursor, the dimension will be horizontal, vertical, or inclined. So, after selecting an inclined line, you need to move the cursor in different ways starting from the object to get different type of dimensions. If you want to create inclined dimension for a line then move

the cursor perpendicular to the line after selecting it. If you want to create horizontal dimension of an inclined line then move the cursor straight along vertical axis. In the same way, move the cursor straight horizontal to create vertical dimension of an inclined line. This technique can also be applied while dimensioning two points.

SKETCH EDITING TOOL

The standard tools to edit sketch entities are categorized in this section. The tools in this section are discussed next.

Fillet

The **Fillet** tool is used to create fillet at the corners created by intersection of two entities. Fillet is sometimes also referred to as round. The procedure to use this tool is discussed next.

- Click on **Fillet** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-66](#). You will be asked to select the entities.

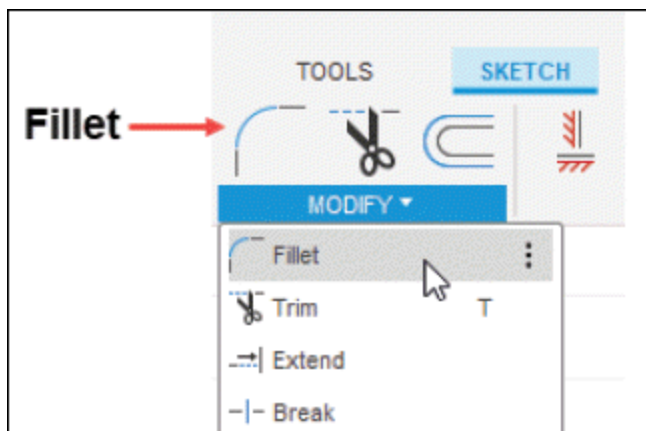


Figure-66. Fillet tool

- Select the first entity and then second entity.

- Enter the value of radius in floating window; refer to [Figure-67](#). The fillet will be created.

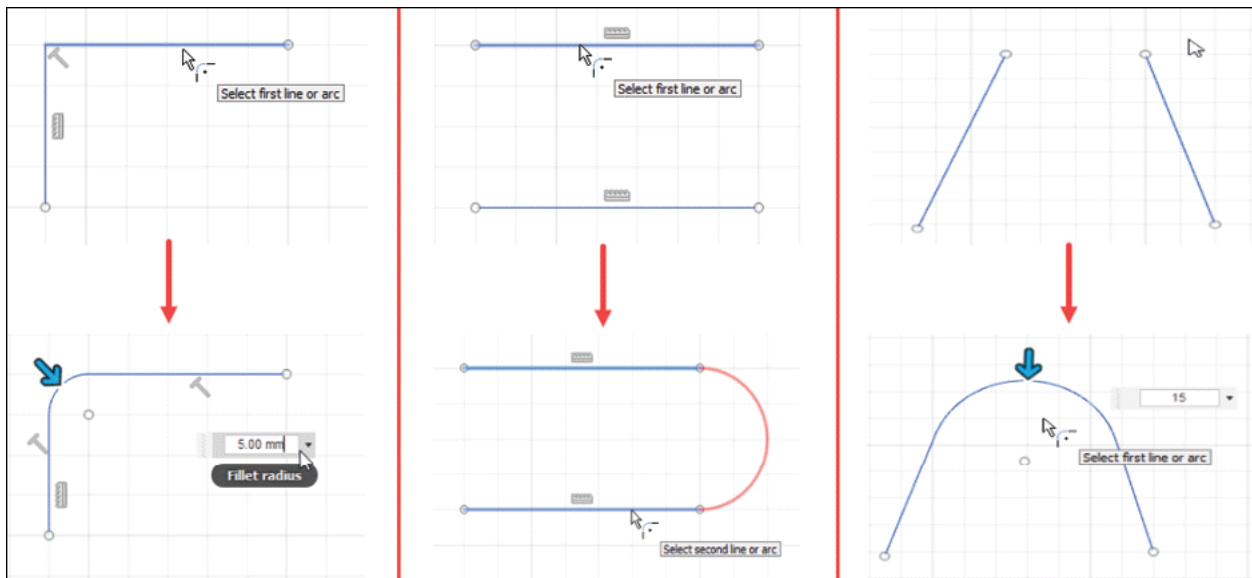


Figure-67. Creation of fillet

Chamfer

The **Chamfer** tool is used to bevel the corners of sketch geometry in an active sketch. There are three tools in **Chamfer** cascading menu of **MODIFY** drop-down in the **Toolbar**; refer to [Figure-68](#). The procedures to use these tools are discussed next.

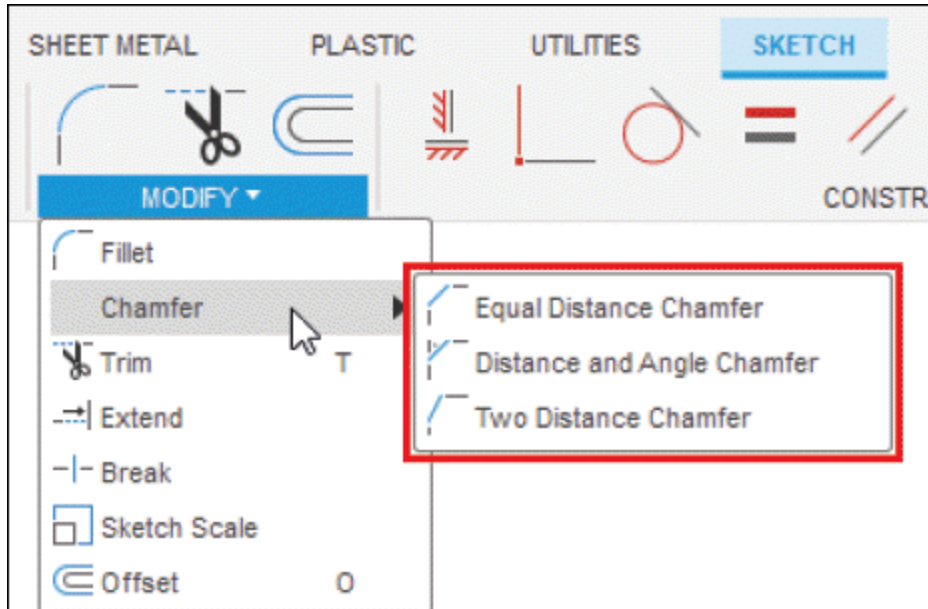


Figure-68. Tools in Chamfer cascading menu

Equal Distance Chamfer

- Click on the **Equal Distance Chamfer** tool from **Chamfer** cascading menu; refer to [Figure-68](#).
- Hover the cursor on the vertex of the sketch or select the first line and hover the cursor on the second line. The preview of chamfer will be displayed; refer to [Figure-69](#).

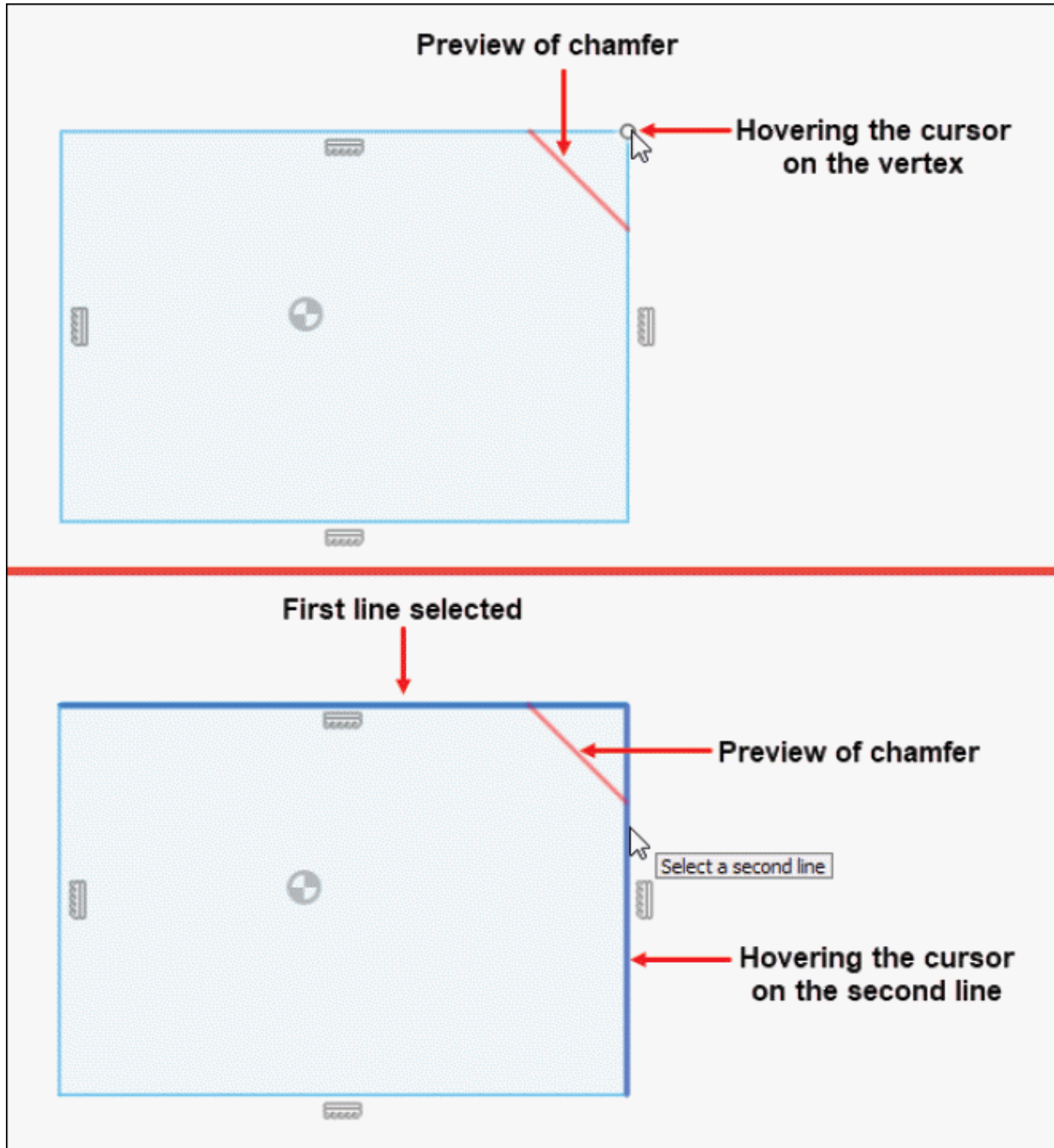


Figure-69. Preview of equal distance chamfer

- Click on the vertex or on the second line of the sketch and type desired distance value in the **Distance** edit box or drag the distance manipulator handle to specify the

distance of chamfer; refer to [Figure-70](#).

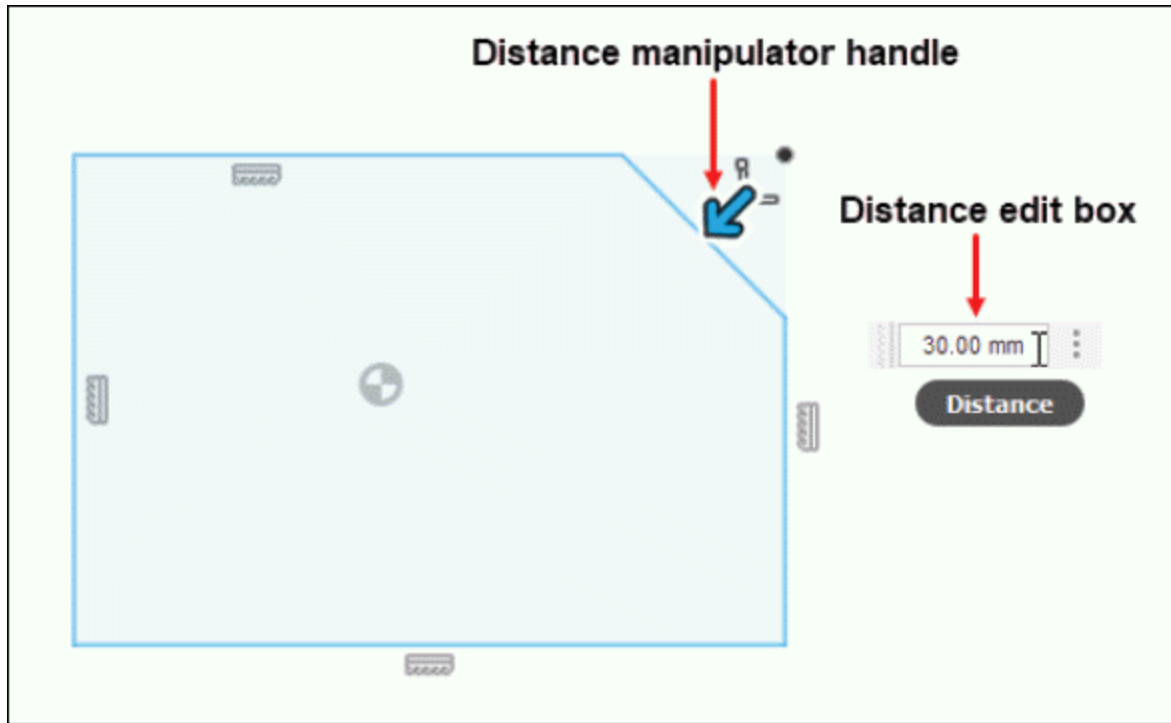


Figure-70. Specifying the distance of chamfer

- After specifying the chamfer distance, press **Enter**. The equal distance chamfer will be created; refer to [Figure-71](#).

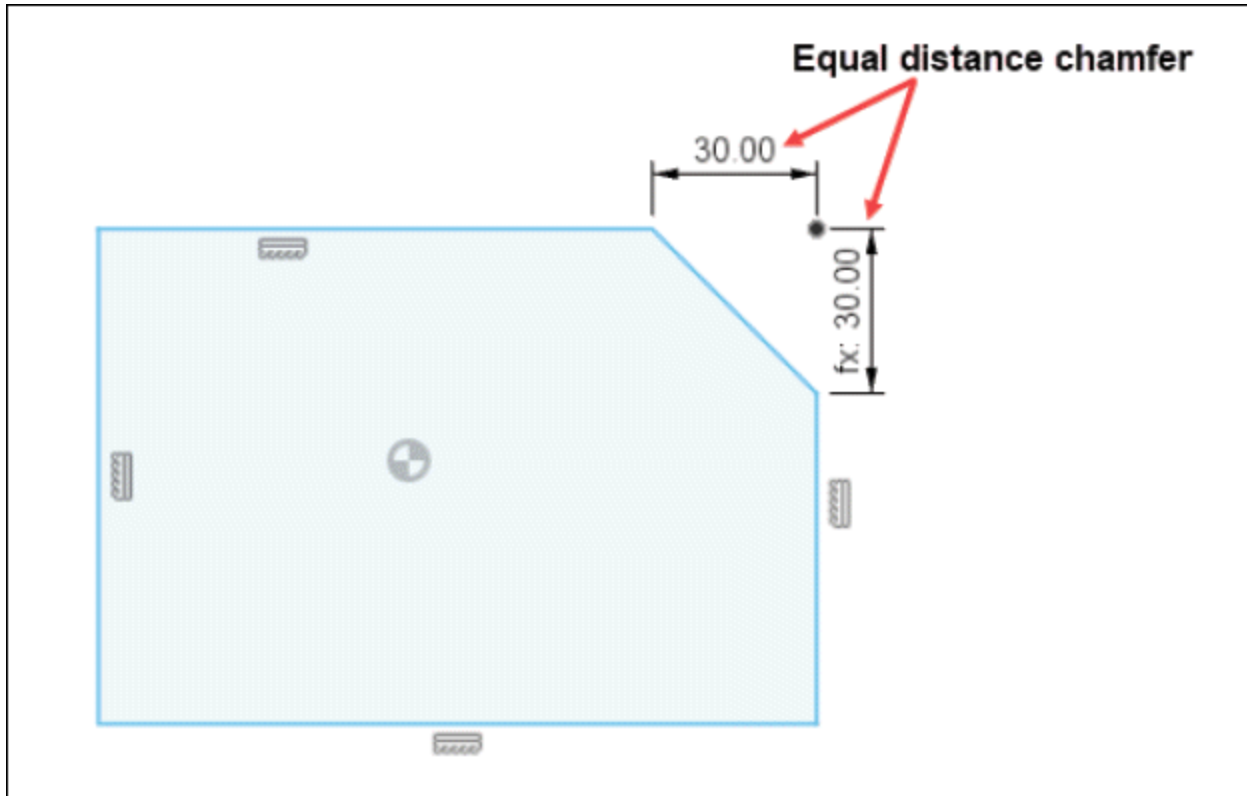


Figure-71. Equal distance chamfer created

Distance and Angle Chamfer

- Click on the **Distance and Angle Chamfer** tool from **Chamfer** cascading menu; refer to [Figure-68](#).
- Hover the cursor on the vertex of the sketch or select the first line and hover the cursor on the second line, the preview of chamfer will be displayed as shown in [Figure-69](#).
- Click on the vertex or on the second line of the sketch and type desired distance value in the **Distance** edit box or drag the distance manipulator handle to specify the distance of chamfer; refer to [Figure-72](#).

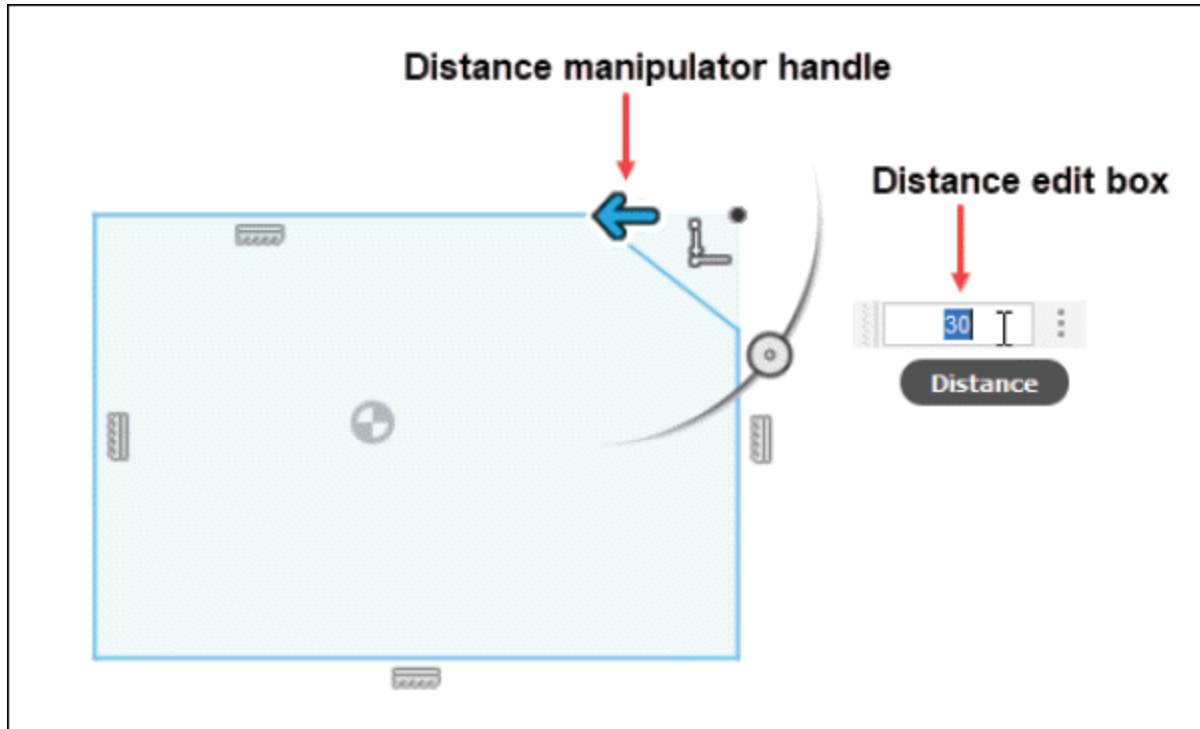


Figure-72. Specifying the distance of chamfer

- After specifying the chamfer distance, press **Tab** key from the keyboard.
- Type desired angle value in the **Angle** edit box or drag the angle manipulator handle to specify the angle of chamfer; refer to [Figure-73](#).

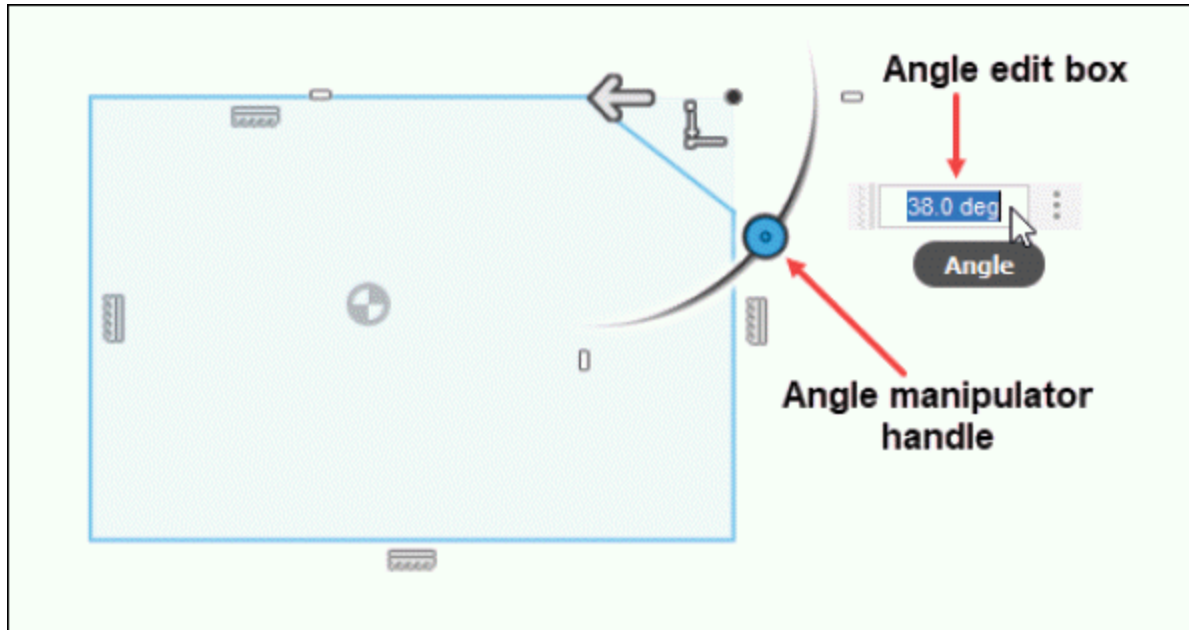


Figure-73. Specifying the angle of chamfer

- After specifying the chamfer angle, press **Enter**. The distance and angle chamfer will be created; refer to [Figure-74](#).

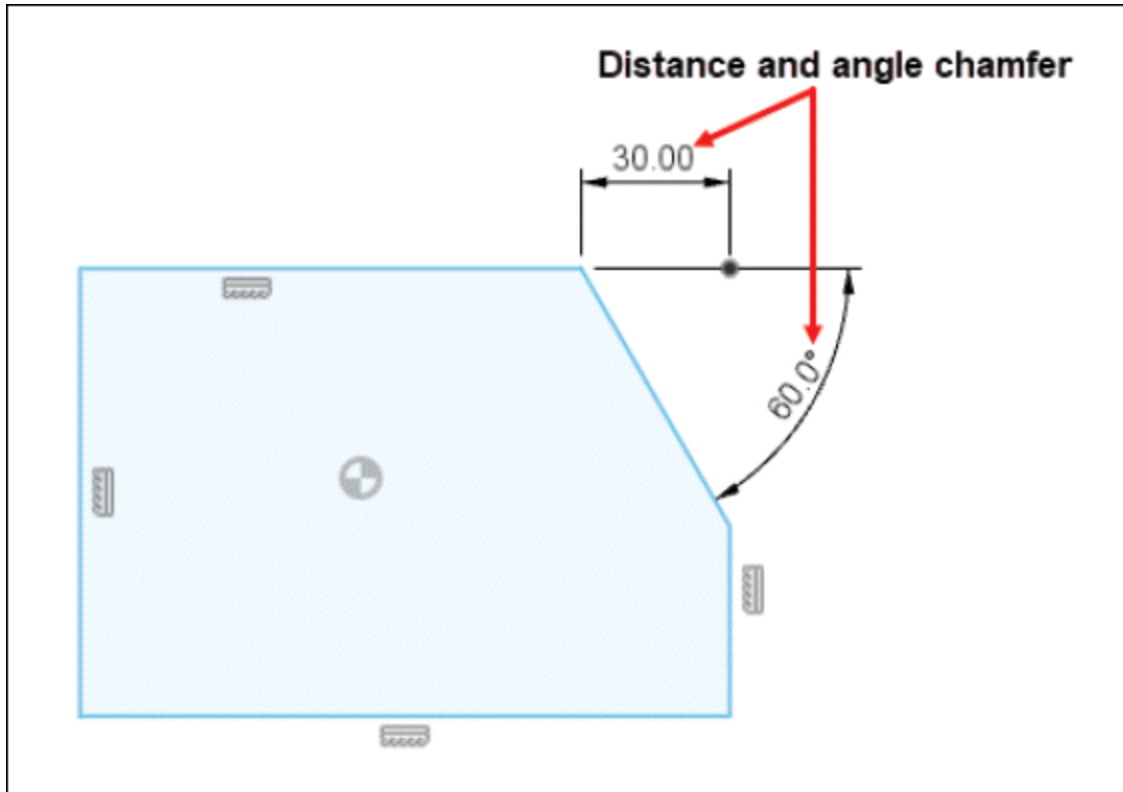


Figure-74. Distance and angle chamfer created

Two Distance Chamfer

- Click on the **Two Distance Chamfer** tool from **Chamfer** cascading menu; refer to [Figure-68](#).
- Hover the cursor on the vertex of the sketch or select the first line and hover the cursor on the second line, the preview of chamfer will be displayed as shown in [Figure-69](#).
- Click on the vertex or on the second line of the sketch and type desired value for the first distance in the **Distance 1** edit box or drag the highlighted distance manipulator handle to specify the first distance of chamfer; refer to [Figure-75](#).

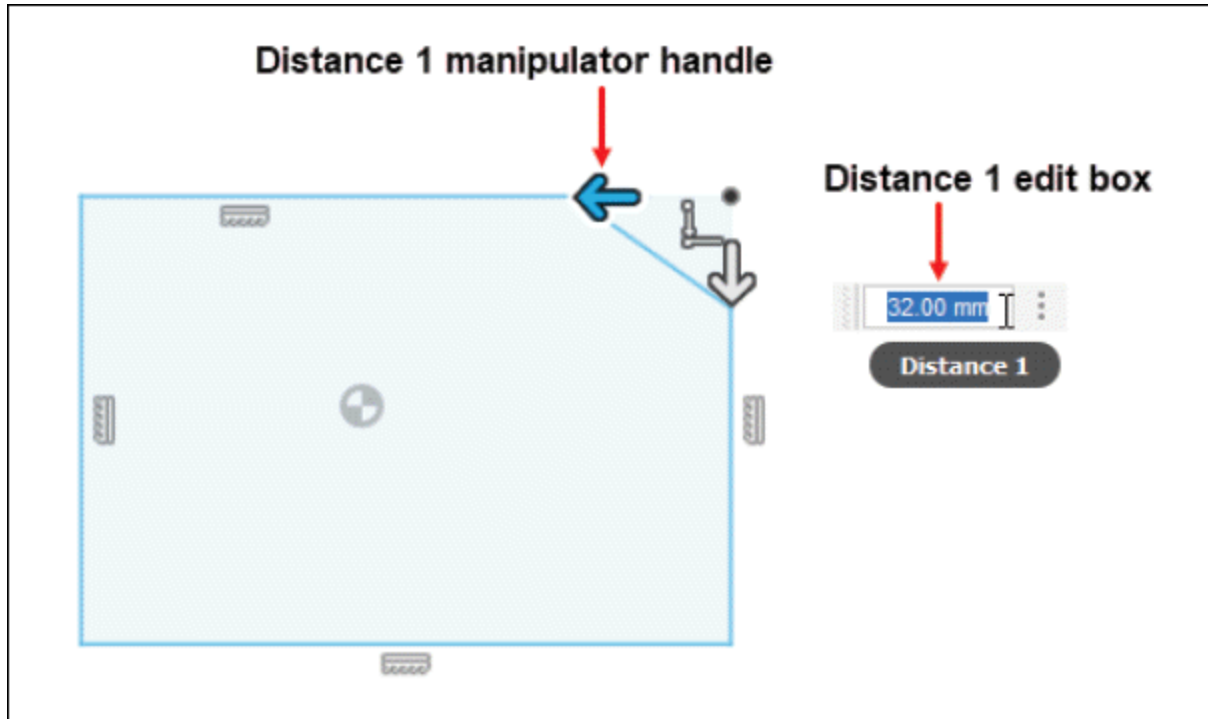


Figure-75. Specifying the first distance of chamfer

- After specifying the first distance, press **Tab** key from the keyboard.
- Type desired value for the second distance in the **Distance 2** edit box or drag the highlighted distance manipulator handle to specify the second distance of chamfer; refer to [Figure-76](#).

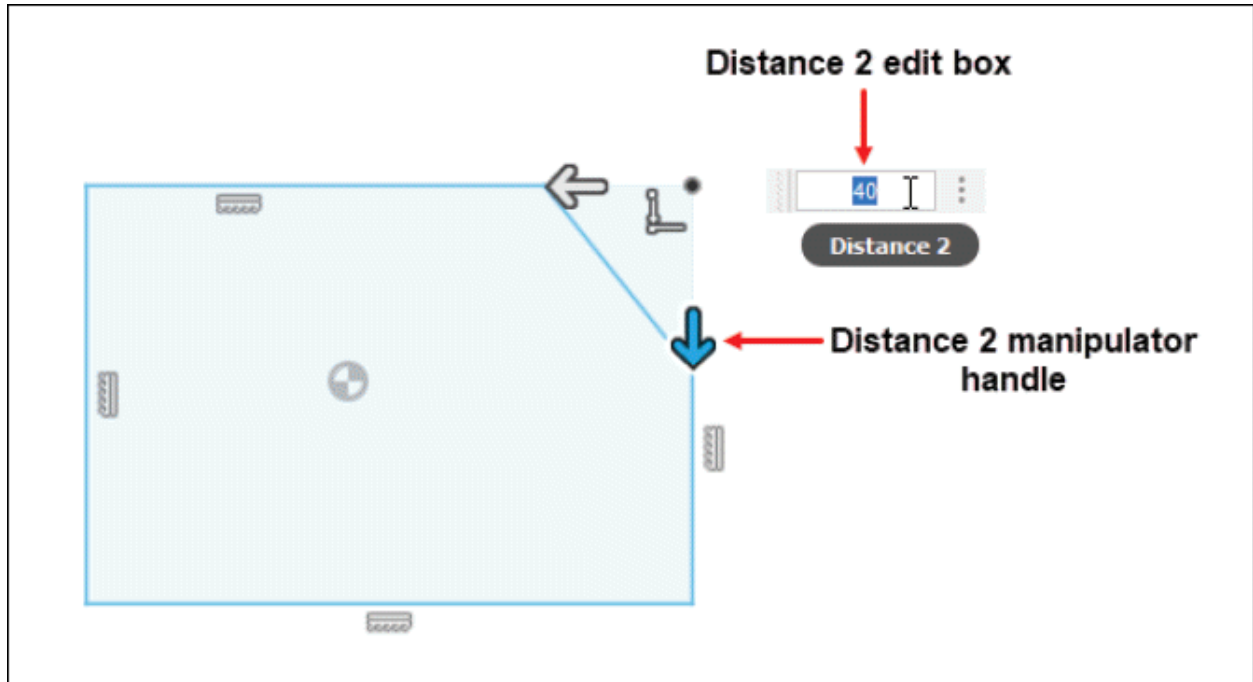


Figure-76. Specifying the second distance of chamfer

- After specifying the second distance of chamfer, press **Enter**. The two distance chamfer will be created; refer to [Figure-77](#).

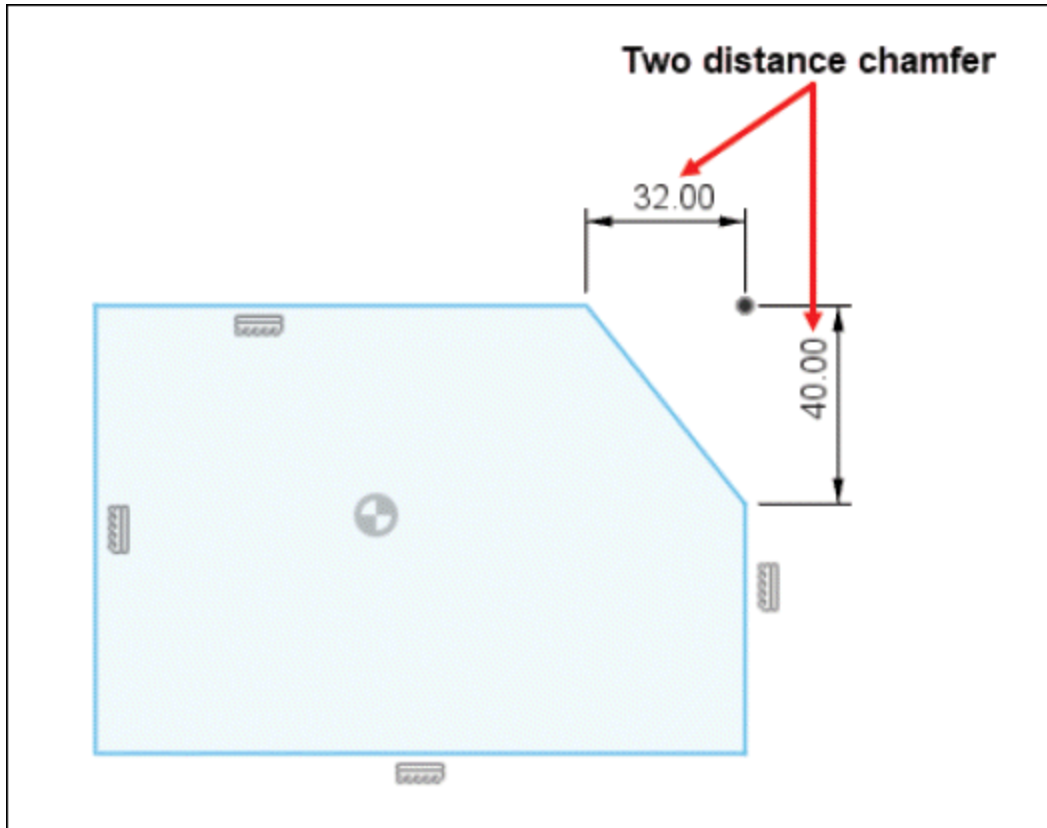


Figure-77. Two distance chamfer created

Trim

The **Trim** tool is used to remove unwanted portion of a sketch entity. While removing the segments, the tool considers intersection point as the reference for trimming. The procedure to use this tool is discussed next.

- Click on **Trim** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-78](#).

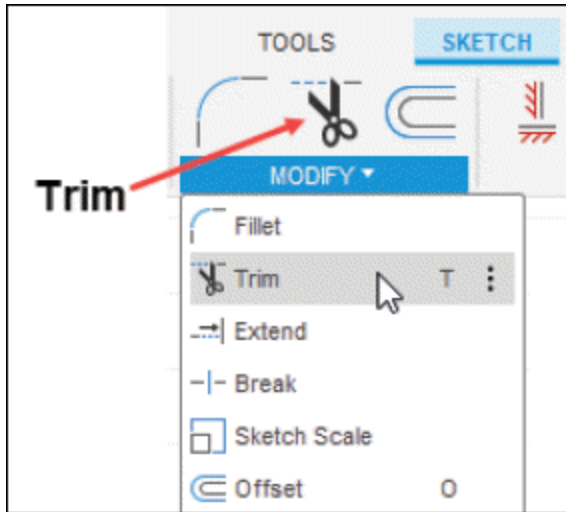


Figure-78. Trim tool

- Click on the entity to trim. You can also trim multiple entities by clicking on them. [Figure-79](#) shows the procedure of trimming.

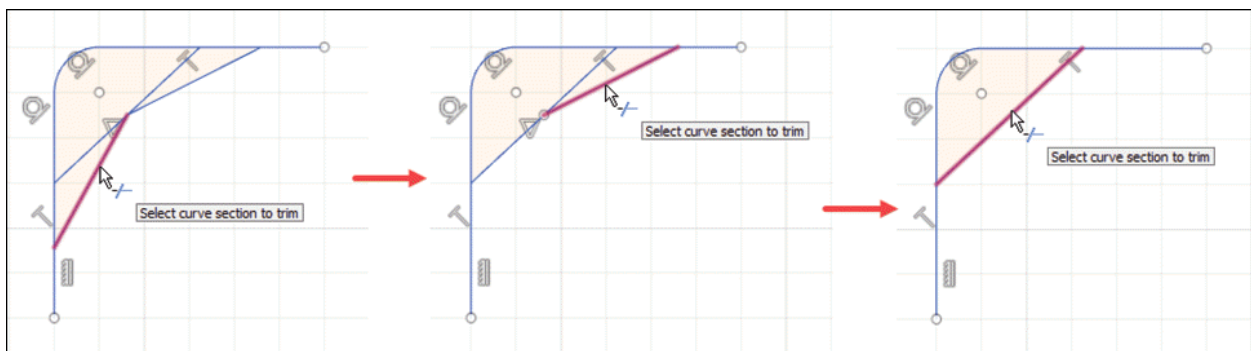


Figure-79. Procedure of trim tool

- Press **ESC** key to exit this tool.

Extend

The **Extend** tool does the reverse of **Trim** tool. This tool is available below the **Trim** tool. The **Extend** tool extends the sketch entities up to the nearest intersecting entities. The procedure to use this tool is discussed next.

- Click on the **Extend** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-80](#).

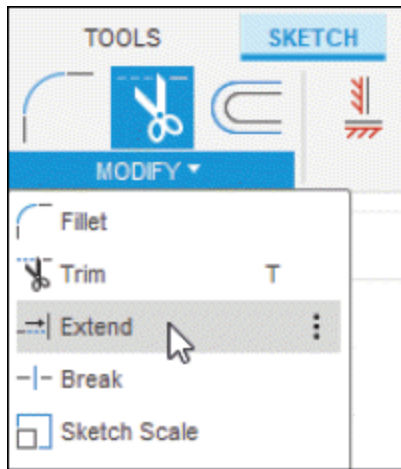


Figure-80. Extend tool

- Hover the cursor on the entity that you want to extend. Preview of the extension will display.
- Click on the entity if the preview is as per your requirement. [Figure-81](#) shows the process of extending entities.

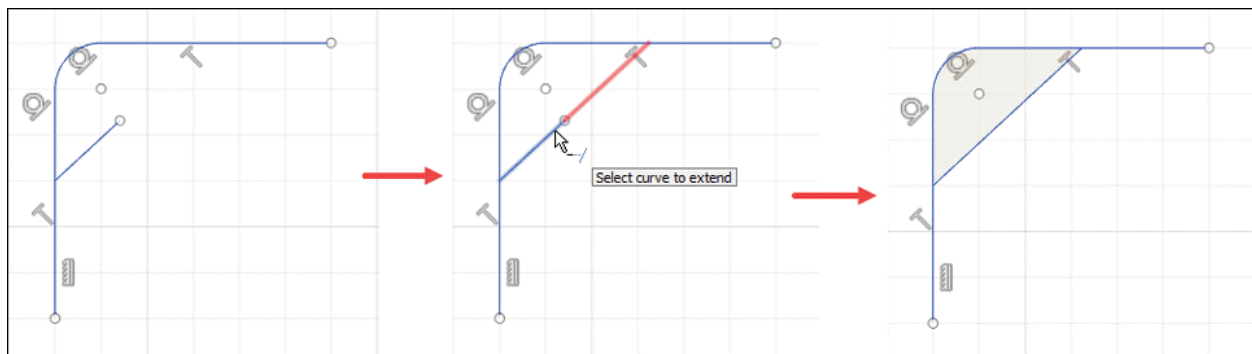


Figure-81. Procedure of extend tool

- Press **ESC** key to exit this tool.

Break

The **Break** tool is used to break the entity at selected

reference points. The procedure to use this tool is discussed next.

- Click on **Break** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-82](#).

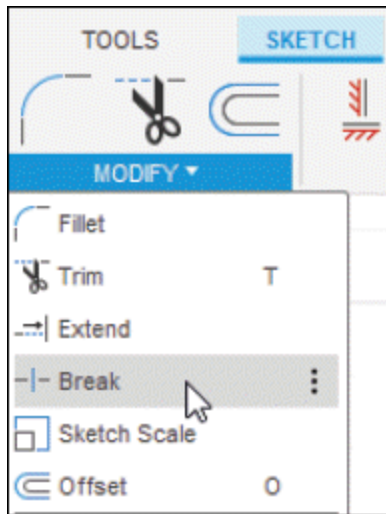


Figure-82. Break tool

- Hover the cursor on the entity. The preview of break will be displayed. Note that there must be any intersecting curve on selected entity to apply break.
- If preview is as per your requirement then click on the entity to break; refer to [Figure-83](#).

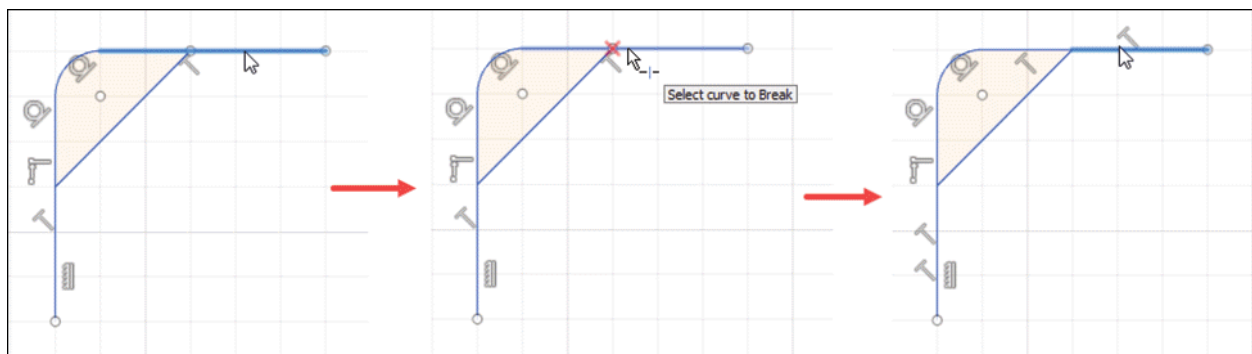


Figure-83. Creation of break

- Press **ESC** key to exit this tool.

Sketch Scale

The **Sketch Scale** tool is used to enlarge or reduce the size of selected entities with respect to the selected point based on specified scale factor. The procedure to use this tool is discussed next.

- Click on the **Sketch Scale** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-84](#). The **SKETCH SCALE** dialog box will be displayed; refer to [Figure-85](#).

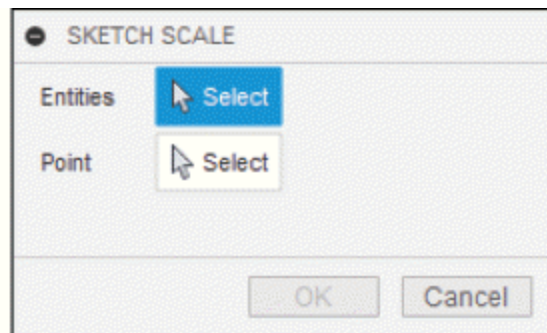
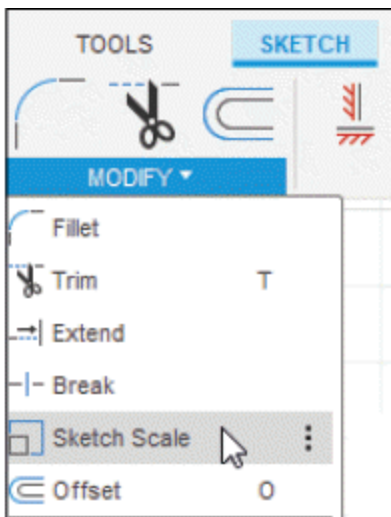


Figure-84. Sketch Scale tool Figure-85. SKETCH SCALE dialog box

- The **Entities** selection box is active by default. Click on the entities to be scaled.
- Click on **Select** button of **Point** section and select the reference point about which selected entities will be scaled up/down. The updated **SKETCH SCALE** dialog box will be displayed; refer to [Figure-86](#). Make sure the point to be selected as reference has not been selected during entities

selection.

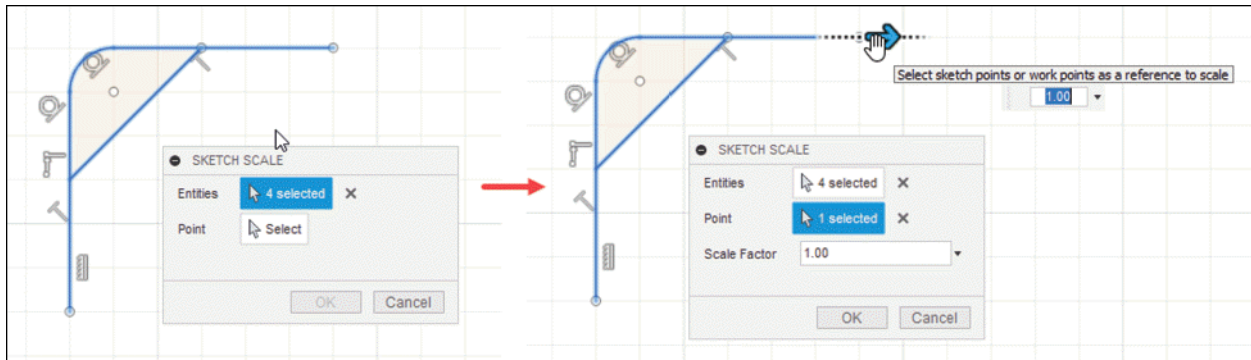


Figure-86. Procedure of sketch scale

- Click in the **Scale Factor** edit box or dynamic input box and type desired value for scale. The preview will be displayed. You can also set the scaling value by moving the drag handle from screen. A value between **0** to **1** will reduce the size and value higher than **1** will increase the size of selected entities.
- If preview is as per your requirement then click on the **OK** button of **SKETCH SCALE** dialog box to complete the process.

Offset

The **Offset** tool is used to create copies of the selected entities at a specified distance. If you are the user of **AutoCAD** then this is the most common tool used for creating layouts. The procedure to use this tool is discussed next.

- Click on **Offset** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-87](#). The **OFFSET** dialog box will be

displayed; refer to [Figure-88](#).

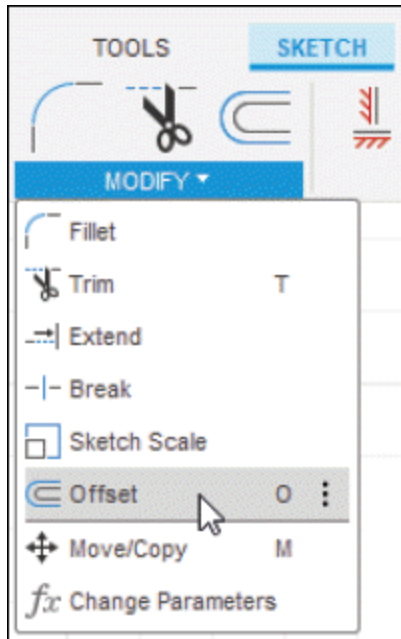


Figure-87. Offset tool

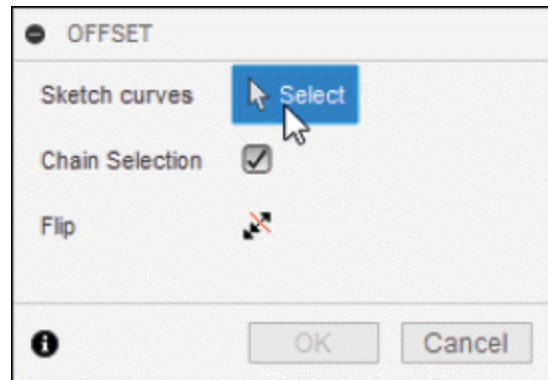


Figure-88. OFFSET dialog box

- The **Sketch curves** selection button is active by default. Select the entities for offset. The updated **OFFSET** dialog box will be displayed.
- Select the **Chain Selection** check box if you want to select all the entities in current chain. Otherwise, clear the check box if you want to select only segments of chain for offset.
- Click in the **Offset position** edit box and type desired distance for offset; refer to [Figure-89](#). You can also set the offset distance by moving the drag-handle from screen.

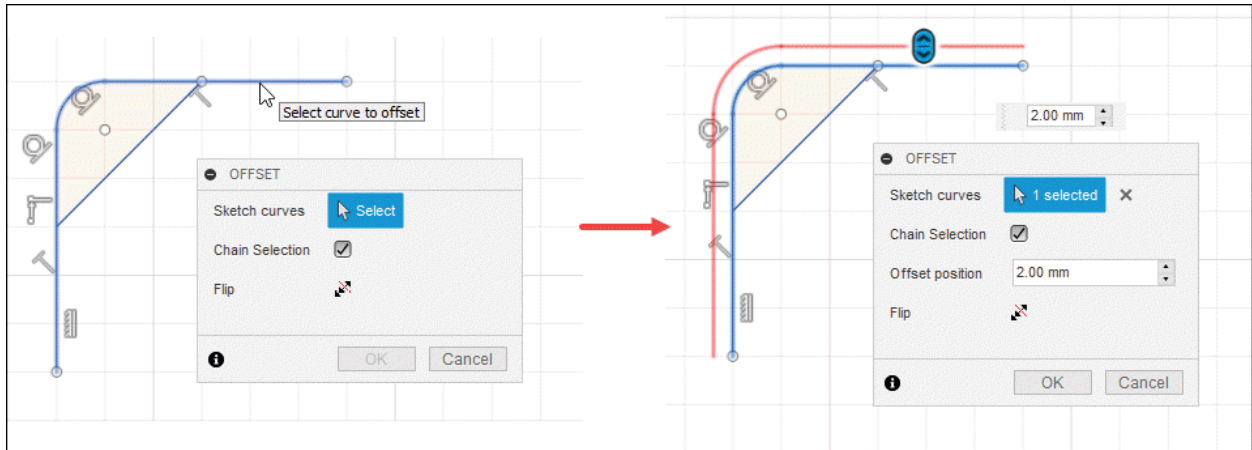


Figure-89. Procedure of offset

- Click **OK** button of **OFFSET** dialog box to finish the process.

SKETCH PALETTE

When sketch mode is active, the **SKETCH PALETTE** dialog box is displayed at bottom right corner of Fusion 360 screen. This dialog box contain commonly used sketch options and sketch constraints; refer to [Figure-90](#).

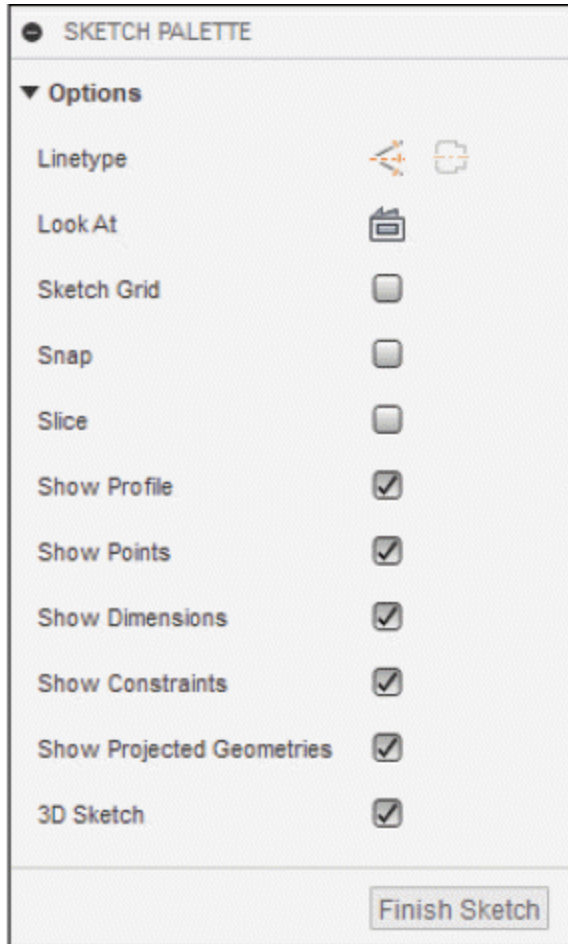


Figure-90. SKETCH PALETTE dialog box

The various tools of Sketch Palette are discussed next.

Options

In **Options** section of **SKETCH PALETTE** dialog box, there are various options to activate or deactivate sketch functions. These options are discussed next.

- **Linetype-** Select **Construction** button from **Linetype** section while creating any sketch object like line, circle, rectangle, etc. The object will be created as construction geometry and select **Centerline** button from **Linetype**

section while creating sketch of only line and rectangle object to create them as centerline. The object will be created as centerline geometry; refer to [Figure-91](#). Click again on these buttons to return to normal sketching mode.

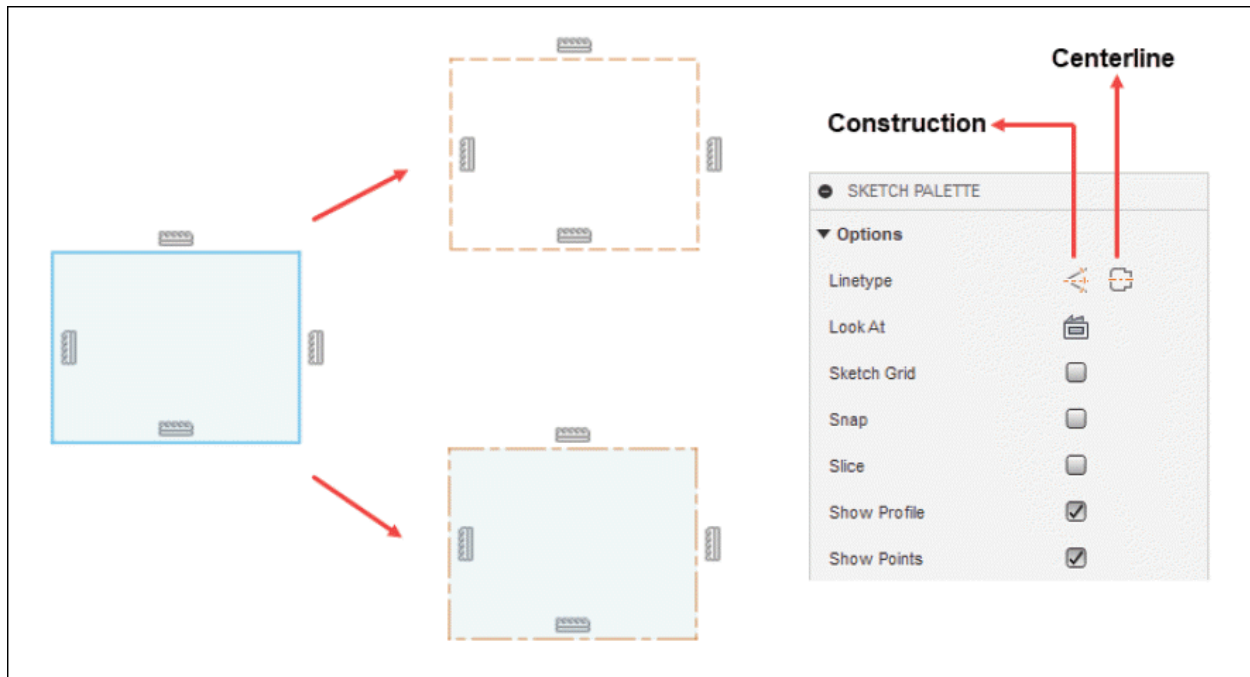


Figure-91. Creating construction and centerline geometry

- **Look At-** If due to any reason, you have rotated the model while sketching and want to make the sketching plane parallel to screen again then click on the **Look At** button once.
- **Sketch Grid-** This option controls the display of sketch grid. To view grid, select the **Grid** check box otherwise clear the check box to hide.
- **Snap-** This option controls the grid Snap. To activate grid snapping, select **Snap** check box and clear the **Snap** check

box to disable. Note that the snapping of key points like end point, intersection point, mid point, etc. will still be active even if you clear the **Snap** check box.

- **Slice-** This option cuts the solid/surface part using **Sketch Plane** if it is coming before sketching plane while creating sketch. This tool does not change the geometry because it is only a display option. To enable, select the **Slice** check box and clear the **Slice** check box to disable.
- **Show Profile-** This option is used to display closed profiles in the sketch. This tool is useful to check geometry when you are going to apply protrusion tools like **Extrude** or **Revolve** later on the sketch. If your sketch is not a closed profile then you cannot create solid protrusions using that sketch. To enable this tool, select the **Show Profile** check box and to disable it clear the check box.
- **Show Points-** The **Show Points** option is used to display points at the end of different open sketch entities. Select the check box to enable this option.
- **Show Dimensions-** This option is used to display the dimensions of entities. Select the check box to enable this option.
- **Show Constraints-** This option is used to display or hide the constraints in the sketch. Constrains are used to restrict the movement of object in defined fashion. To display constraints in the sketch, select the **Show Constraints** check box and clear the check box to hide

constraints.

- **Show Projected Geometries-** This tool is used to display the projected geometries on the sketch. Select the check box to enable and clear the check box to disable this option. Note that projected geometries are generally displayed in dark pink color.
- **3D Sketch-** This option enables to pull sketch objects out of the sketch plane and create a 3D sketch. To enable this tool, select the **3D Sketch** check box and to disable it clear the check box.

Feature Options

The **Feature Options** section of **SKETCH PALETTE** is displayed when sketch creation tools like rectangle, circle, slot, polygon, and so on are active which have variation in defining their parameters; refer to [Figure-92](#). Select desired option from the **Feature Options** section to switch to different input style for same tool.

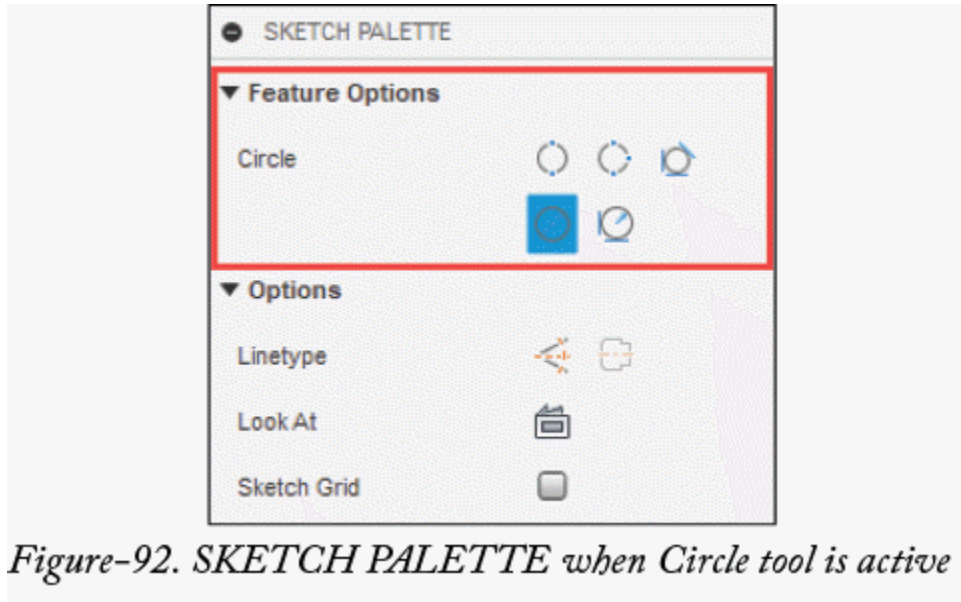


Figure-92. SKETCH PALETTE when Circle tool is active

CONSTRAINTS

The tools in **CONSTRAINTS** panel of **SKETCH** contextual tab are used to apply constraints to the sketch; refer to [Figure-93](#). Constraints are used to restrict the shapes/position of sketch entities with respect to other entities.

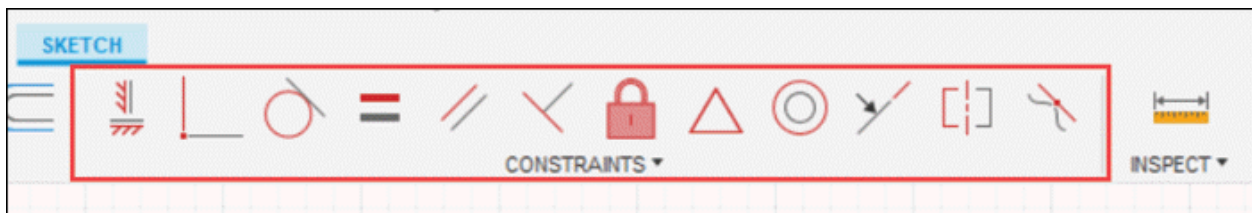


Figure-93. CONSTRAINTS panel

- Make sure the **Show Constraints** check box is selected from **SKETCH PALETTE** dialog box before working with constraints; refer to [Figure-94](#).

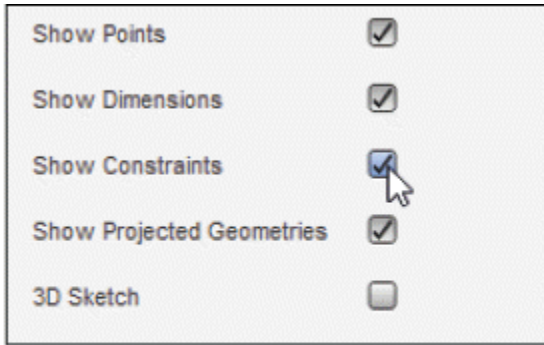




Figure-94. Show Constraints check box

Various constraints available in Autodesk Fusion are discussed next.

 The **Horizontal/Vertical Constraint** makes one or more selected lines to become horizontal/Vertical. To apply this constraint, click on the **Horizontal/Vertical** button and select the entities. Press **ESC** to exit the tool. You can also make two points horizontally or vertically in-line by using this constraint.

What if you were trying to make a line vertical constrained but it is becoming horizontal constrained. In such cases, you should drag the end point of line towards vertical reference and then apply the Horizontal/Vertical constraint.

 The **Coincident Constraint** makes selected point to be coincident with a line, arc, circle, or ellipse. To apply this constraint, select the **Coincident** button from **CONSTRAINTS** panel in **SKETCH** contextual tab and select the entities for coincident. Press **ESC** to exit the tool.

 The **Tangent Constraint** makes selected arc, circle,

spline, or ellipse to become tangent to other arc, circle, spline, ellipse, line, or edge. To apply this constraint, click on the **Tangent** button and select required entity. Press **ESC** to exit the tool; refer to [Figure-95](#).

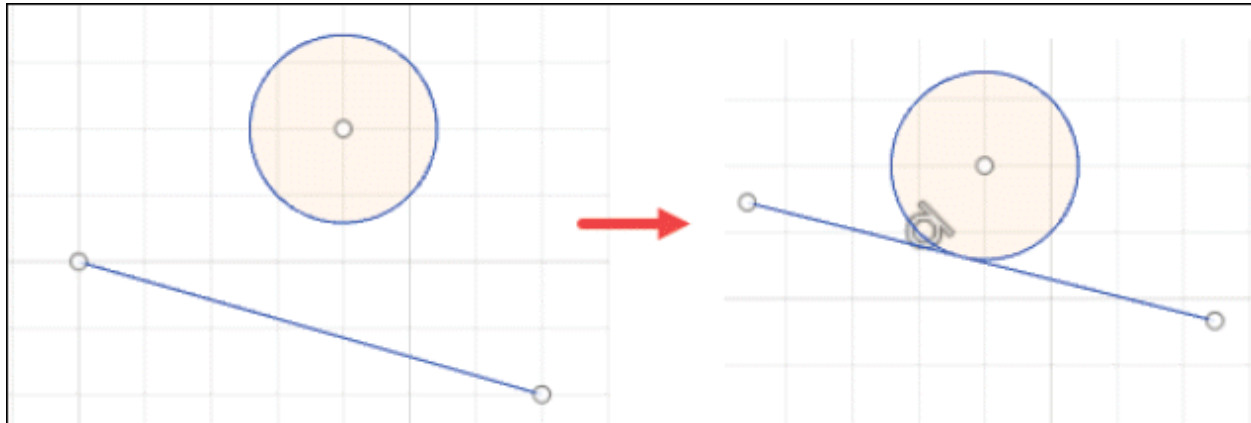



Figure-95. Applying tangent constraint

 The **Equal Constraint** makes the selected lines to have equal length and the selected arcs or circles to have equal radii/diameter. To apply this constraint, click on the **Equal** button and select required entity to apply the equal constraint; refer to [Figure-96](#). Press **ESC** to exit the tool.

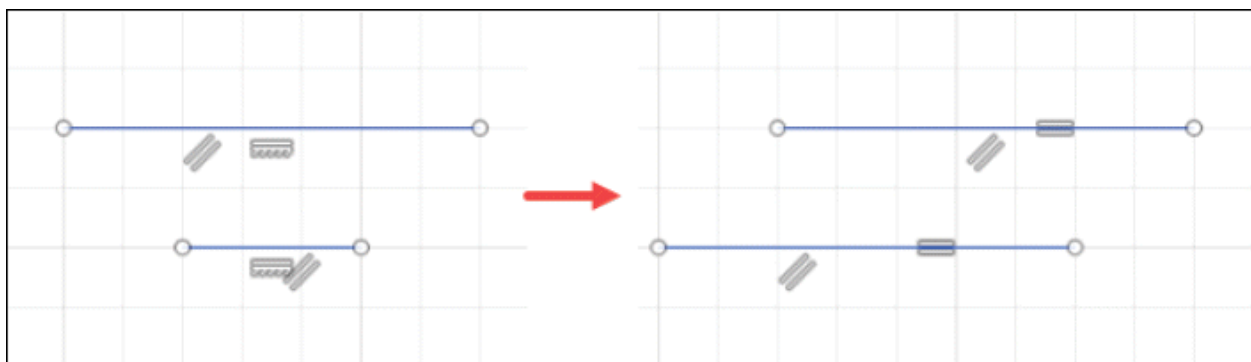


Figure-96. Applying equal constraint

 The **Parallel Constraint** makes the selected lines

parallel to each other. To apply this constraint, click on the **Parallel** button and then select desired entities; refer to [Figure-97](#). Press **ESC** to exit the tool.

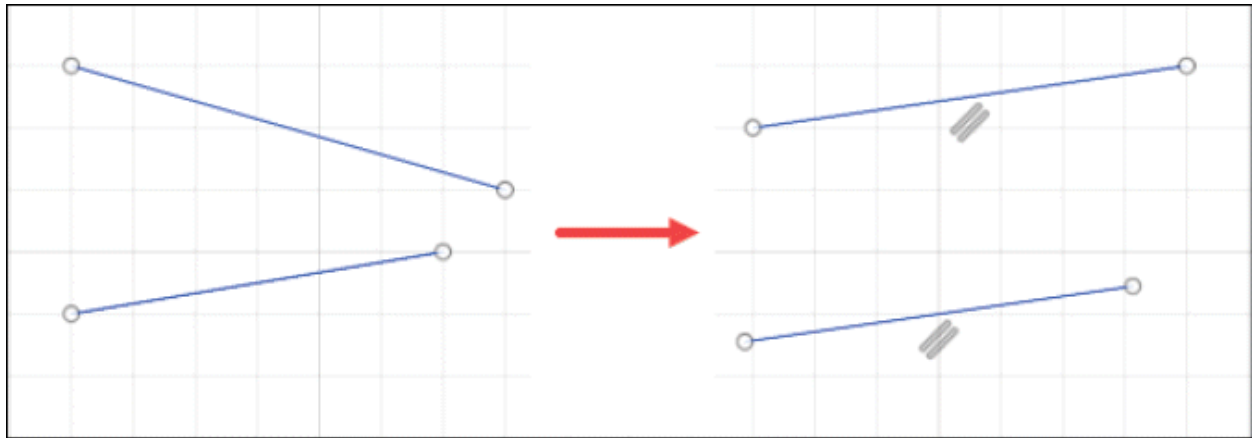


Figure-97. Applying parallel constraint

The **Perpendicular Constraint** makes the selected lines perpendicular to each other. To apply this constraint, click on the **Perpendicular** button and select the entities; refer to [Figure-98](#). Press **ESC** to exit the tool.

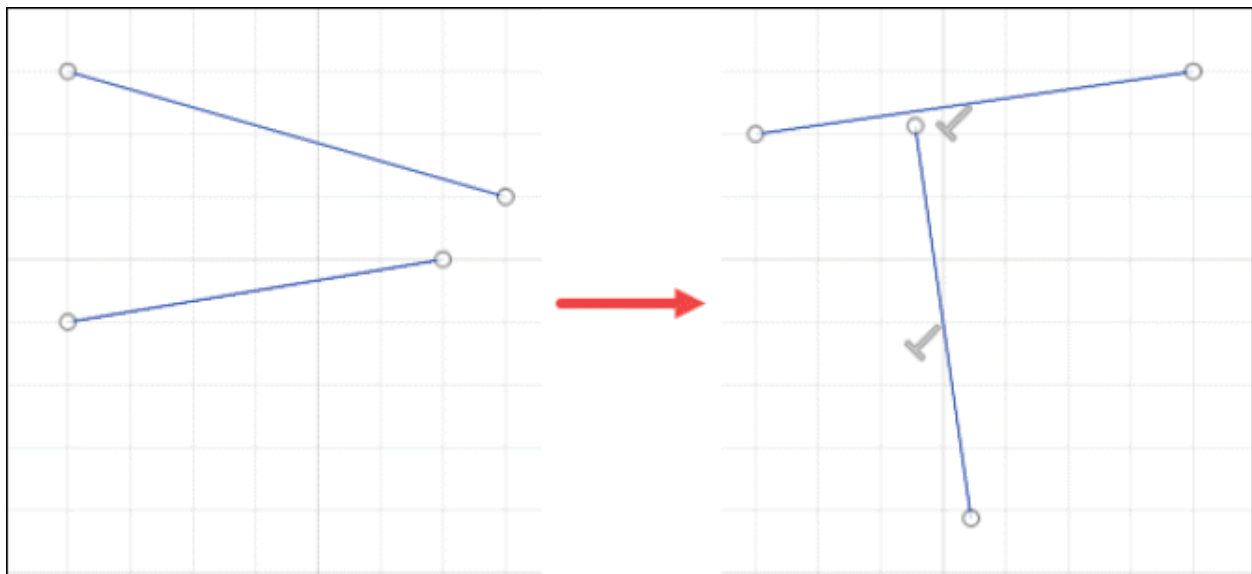




Figure-98. Applying perpendicular constraint

 The **Fix/Unfix Constraint** makes the selected entity to be fixed at current position. If you apply this constraint to a line or an arc, its location will be fixed but you can change its size by dragging the endpoints or specifying value. To apply this constraint, click on the **Fix/Unfix** button and select the entities. Press **ESC** to exit the tool.

Fix option is useful when you want to restrict the movement of an entity while applying other constraints. Like in [Figure-99](#), if you want the arc to move in place of line then you can fix the line by applying **Fix** constraint before applying **MidPoint** constraint.

 The **MidPoint Constraint** makes selected point of an entity to be placed on the midpoint of another sketch entity. To apply this constraint, click on the **MidPoint** button. Select the point and then select the entity whose midpoint is to be used for constraining; refer to [Figure-99](#). Press **ESC** to exit the tool.

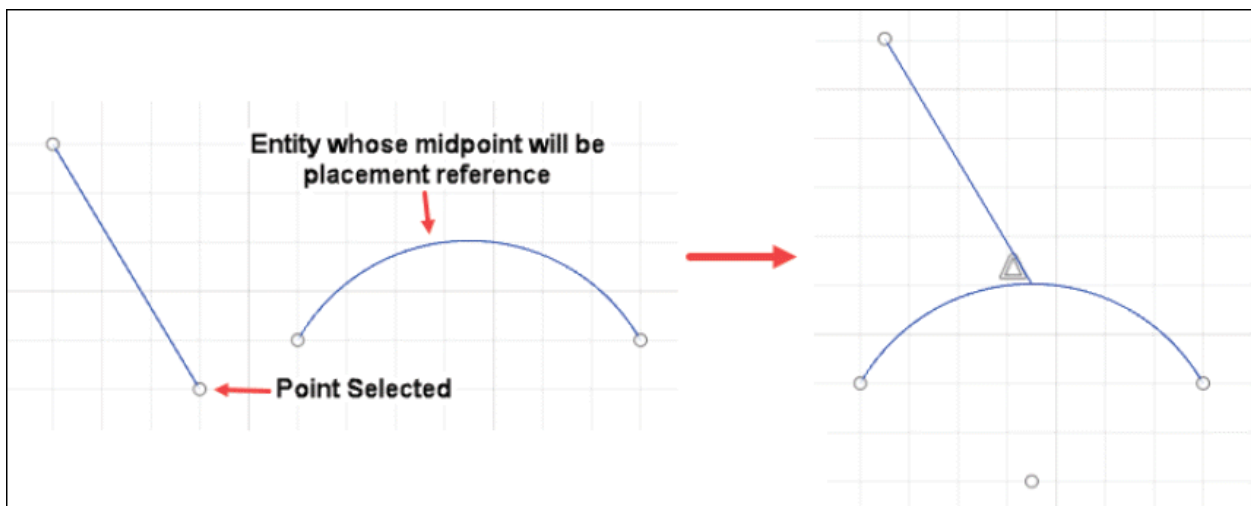



Figure-99. Applying midpoint constraint

 The **Concentric Constraint** makes selected arc or circle to share the same center point with other arc, circle, point, vertex, or circular edge. To apply this constraint, click the **Concentric** button and then select the two entities to apply the concentric constraint; refer to [Figure-100](#). Press **ESC** to exit the tool.

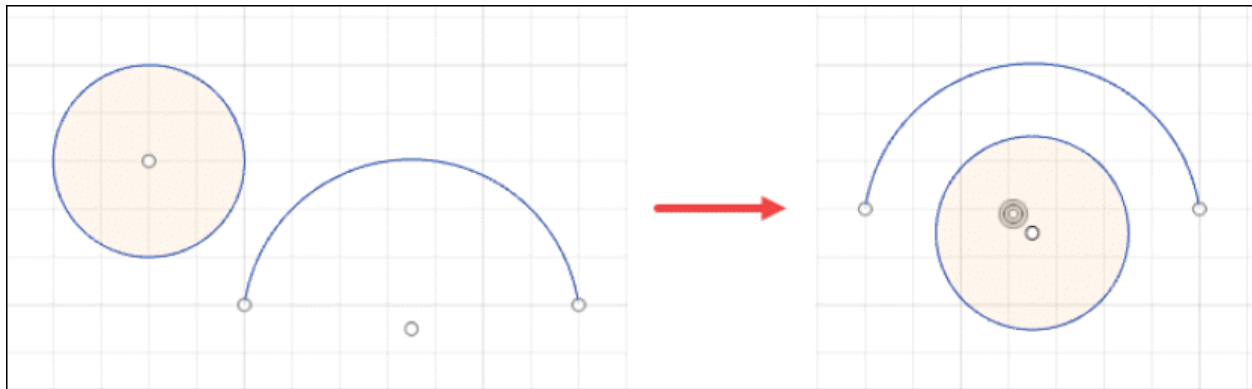



Figure-100. Applying concentric constraint

 The **Collinear Constraint** makes the selected lines to lie on the same infinite line; refer to [Figure-101](#). To use this constraint, select the **Collinear constraint** button and select the entities. Press **ESC** to exit the tool.

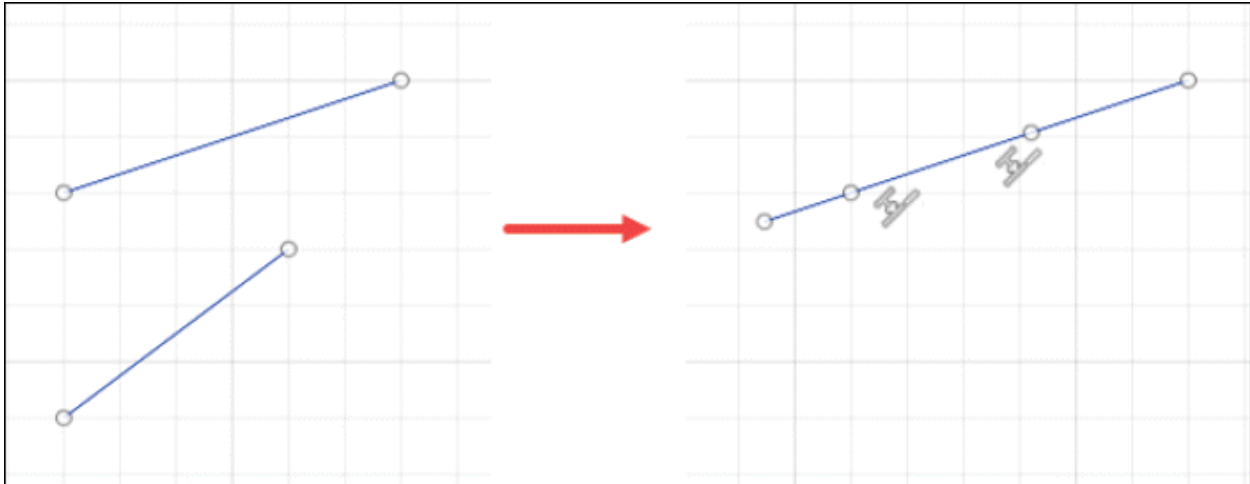



Figure-101. Applying collinear constraint

 The **Symmetry Constraint** makes two selected lines, arcs, points, and ellipses to remain symmetric about a reference line. To apply this constraint, click on the **Symmetric** button and select the two objects to be symmetric about the reference line. You will be asked to select reference object. Select the line to be used as symmetry reference. The constraint will be applied; refer to [Figure-102](#). Press **ESC** to exit the tool.

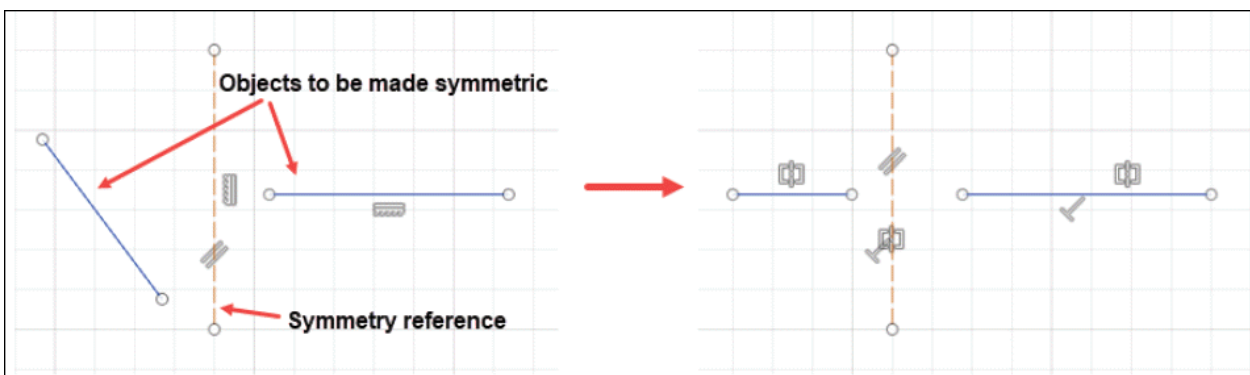



Figure-102. Applying symmetry constraint

 The **Curvature Constraint** is used to create a curvature continuous (G2) condition between selected sketch entities

which meet at sharp vertex. To use this tool, click on **Smooth** button and select desired entities near sharp vertex; refer to [Figure-103](#). Press **ESC** to exit the tool.

While applying curvature constraint to splines, make sure you select the splines not the control handles.

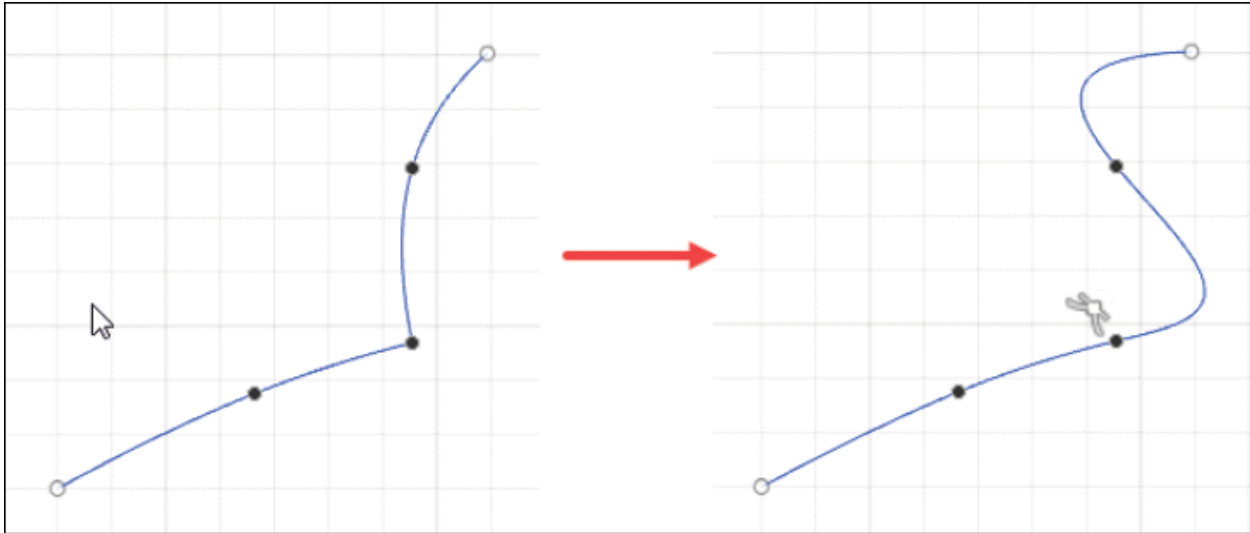


Figure-103. Applying curvature constraint

Except for Horizontal/Vertical and Fix constraint, you can also select an external entity such as an edge, plane, axis, or sketch curve of an external sketch as reference to apply the constraint.

PRACTICAL 1

Create the sketch as shown in [Figure-104](#). Also, dimension the sketch as per the figure.

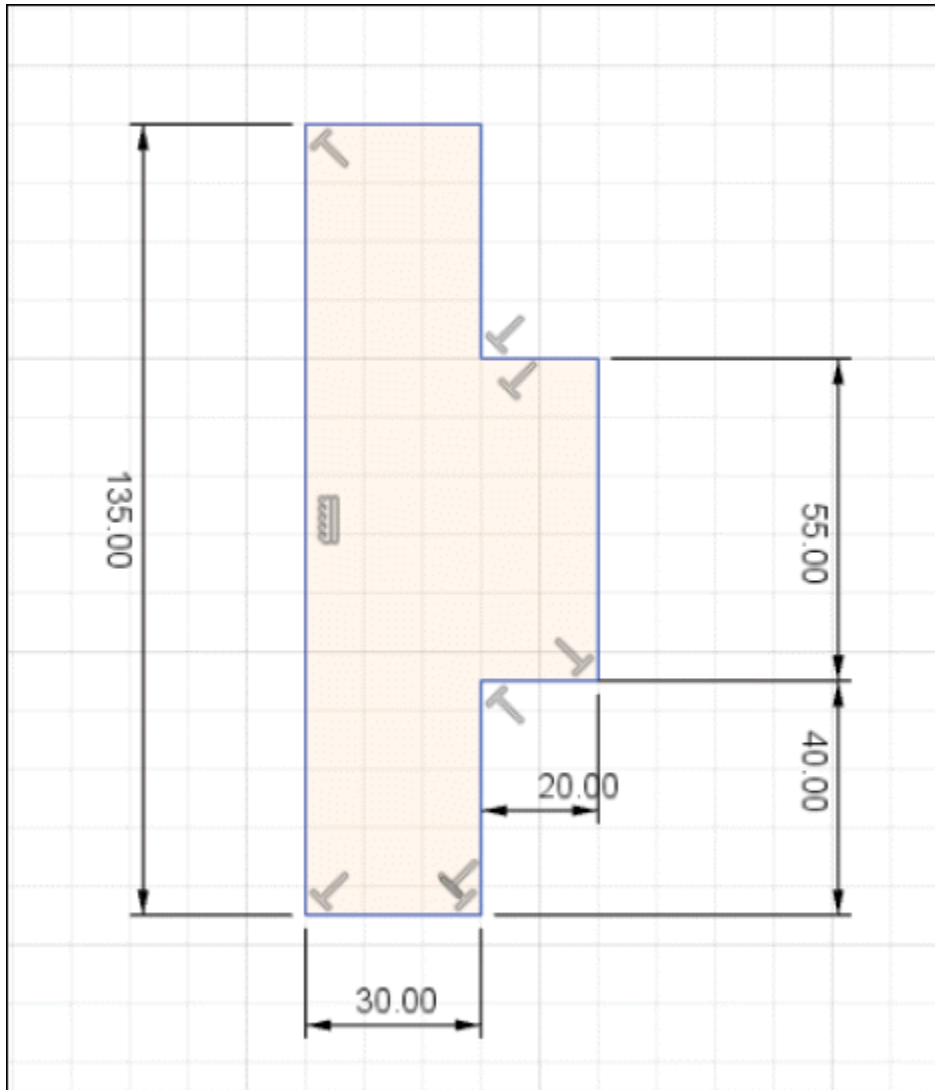


Figure-104. Practical 1

Steps to be Performed:

Below is the step by step procedure of creating the sketch shown in the [Figure-104](#).

Starting Sketching Environment

- Start **Autodesk Fusion 360** if not started already.
- Select the **New Design** tool from **File** menu to start a new design; refer to [Figure-105](#).

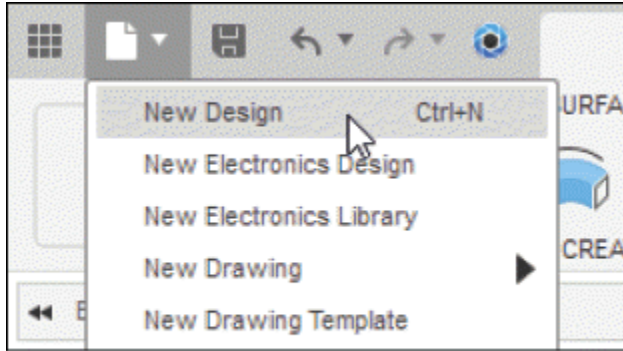


Figure-105. New Design tool

- Select the **Create Sketch** tool from **CREATE** drop-down; refer to [Figure-106](#). You will be asked to select a sketching plane; refer to [Figure-107](#).

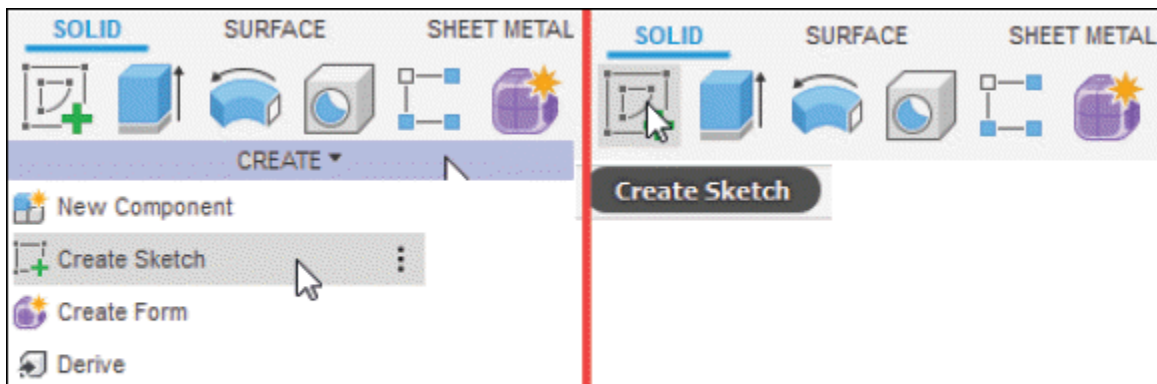


Figure-106. Create Sketch tool

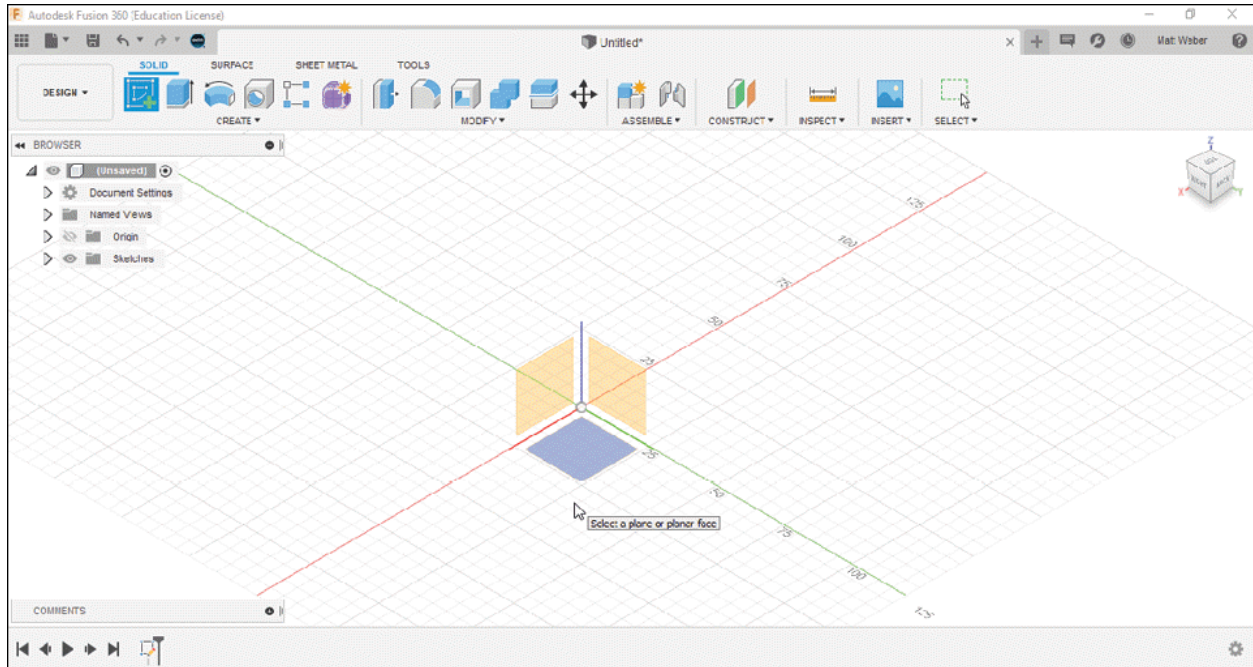


Figure-107. Select plane for sketch

- Select the “**Top**” (XY) plane for sketching.

Creating Lines

- Click on the **Line** button from **CREATE** drop-down or press **L** key from keyboard to select line. The **Line** tool will become active and you will be asked to select the start point.
- Click on the origin and move the cursor towards right; refer to [Figure-108](#).

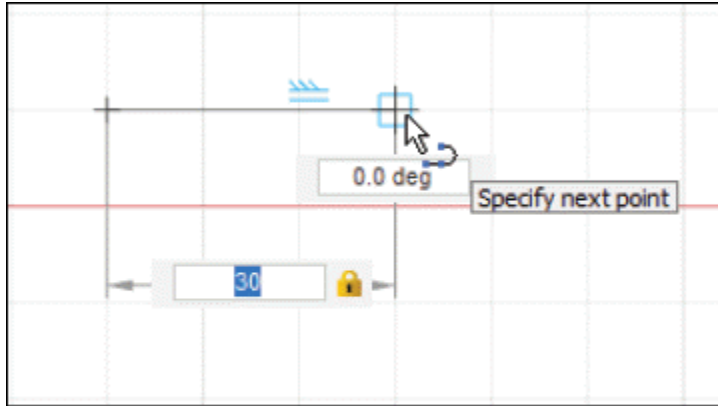


Figure-108. Starting creation of line

- Type the value **30** in the dimension box displayed near line and press **TAB** key to lock the dimension. Click on the screen to create a line.
- Move the cursor vertically upwards and type the value **40** in the dimension box and press **TAB** key to lock the dimension and then click on the screen to complete the line.
- Move the cursor towards right and type the value **20** in the dimension box. Lock the dimension as discussed earlier and click on screen; refer to [Figure-109](#).

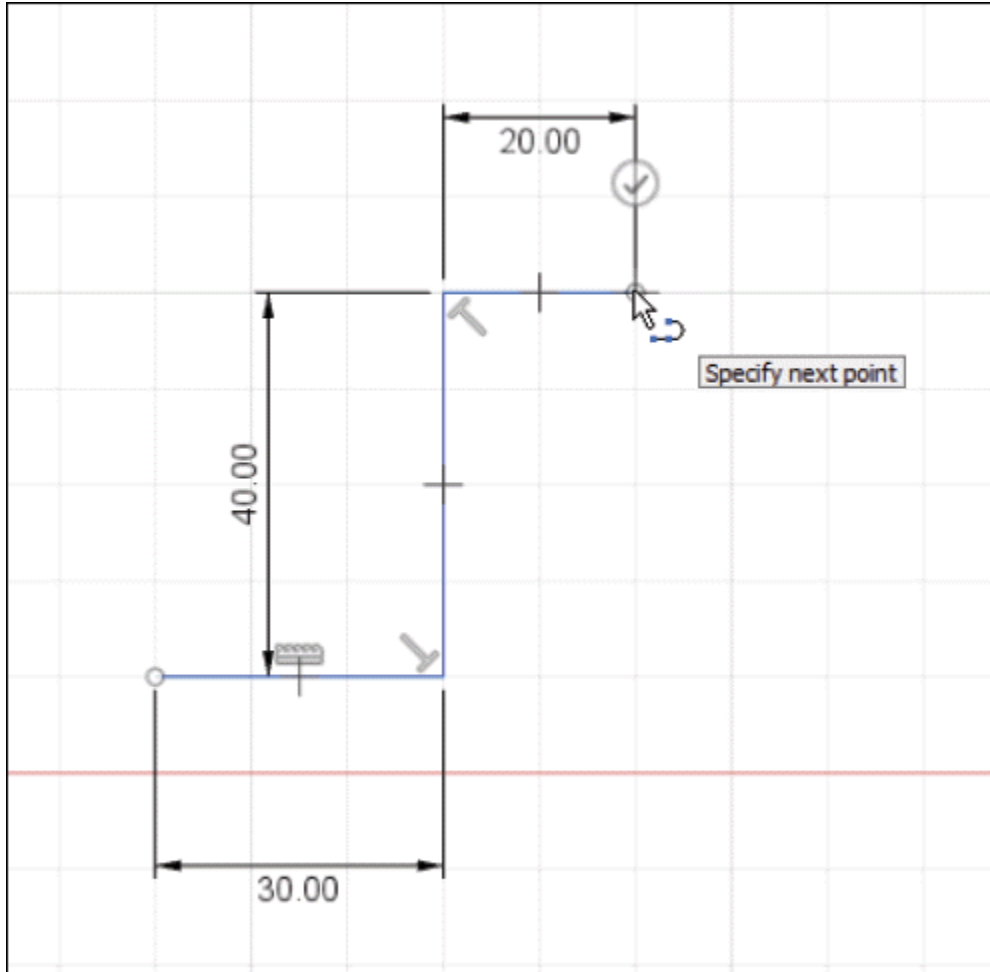


Figure-109. Sketch after specifying 20 value

- Move the cursor upward and type the value **55** in the dimension box. Press **TAB** key to lock the dimension and then click on the screen to complete the line.
- Move the cursor left and type the value **20** in the dimension box. Press **TAB** key to lock the dimension and then click on the screen to complete the line.
- Move the cursor upwards and type the value **40** in the dimension box. Press **TAB** key to lock the dimension and then click on the screen to complete the line.

- Move the cursor towards left and type the value **30** in the dimension box. Press **TAB** key to lock the dimension and then click on the screen to complete the line.
- Move the cursor downward and click on the origin to close the sketch. Press **ESC** to exit the line tool.

The sketch after performing the above steps is displayed as shown in [Figure-110](#).

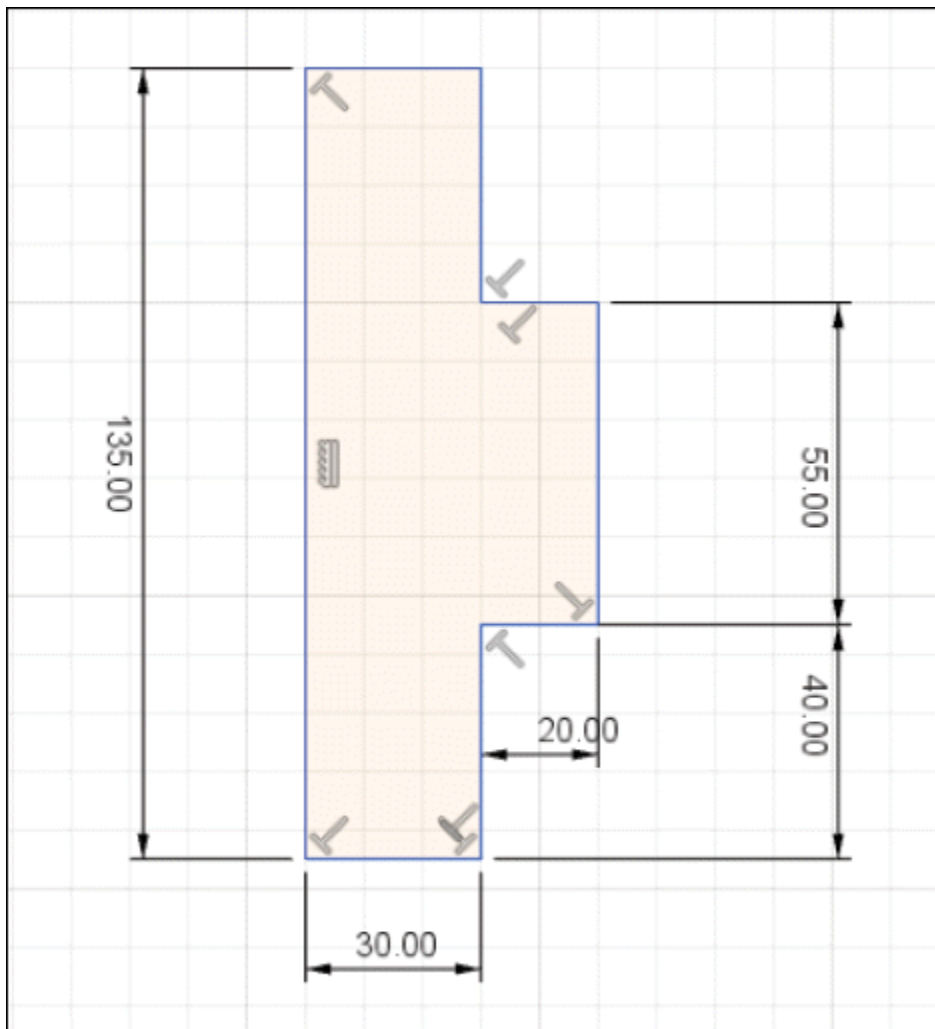


Figure-110. Final Practical 1

In this practical, you will create the sketch as shown in [Figure-111](#).

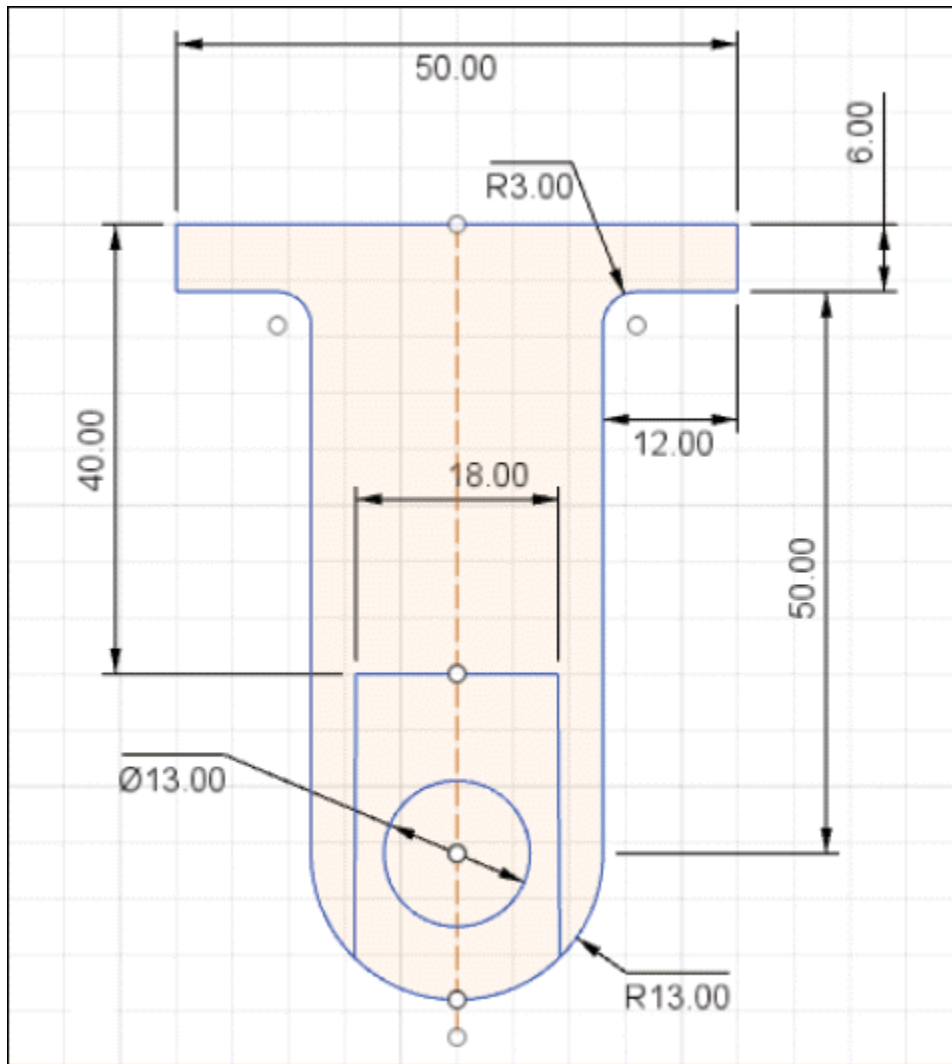


Figure-111. Practical 2

Steps to be performed:

Below is the step by step procedure of creating the sketch shown in the [Figure-111](#).

Starting Sketching Environment

- Start **Autodesk Fusion 360** if not started already.

- Select the **New Design** tool from **File** menu to start a new design.
- Select the **Create Sketch** tool from **CREATE** drop-down.
- After selecting this tool, you need to select the **plane** for creating sketch.
- Select the **Top Plane** to create the sketch.

Creating Lines

- Click on the **Line** button from **CREATE** drop-down or press **L** key. You are asked to specify the start point of the line.
- Click on the origin and move the cursor towards right. Type the value **25** in the dimension box and press **TAB** key to lock the dimension. Click on the screen to complete the line.
- Move the cursor downward and type the value **6** in the dimension box. Lock the dimension as discussed earlier and click on screen.
- Move the cursor to the left and type the value **12** in the dimension box. Lock the dimension as discussed earlier and click on screen.
- Move the cursor downwards and type the value **50** in the dimension box. Till this point, our sketch should display like [Figure-112](#).



Figure-112. Sketch after creating lines

Creating Arcs

- Select the **Tangent Arc** tool from **Arc** cascading menu in **CREATE** drop-down.
- Click on the end point of the vertical line recently created and move the cursor downwards in such a way that the end point of arc is in-line with the origin.
- Specify the dimension value **13** in the dimension box and press **ENTER** to complete the arc; refer to [Figure-113](#).

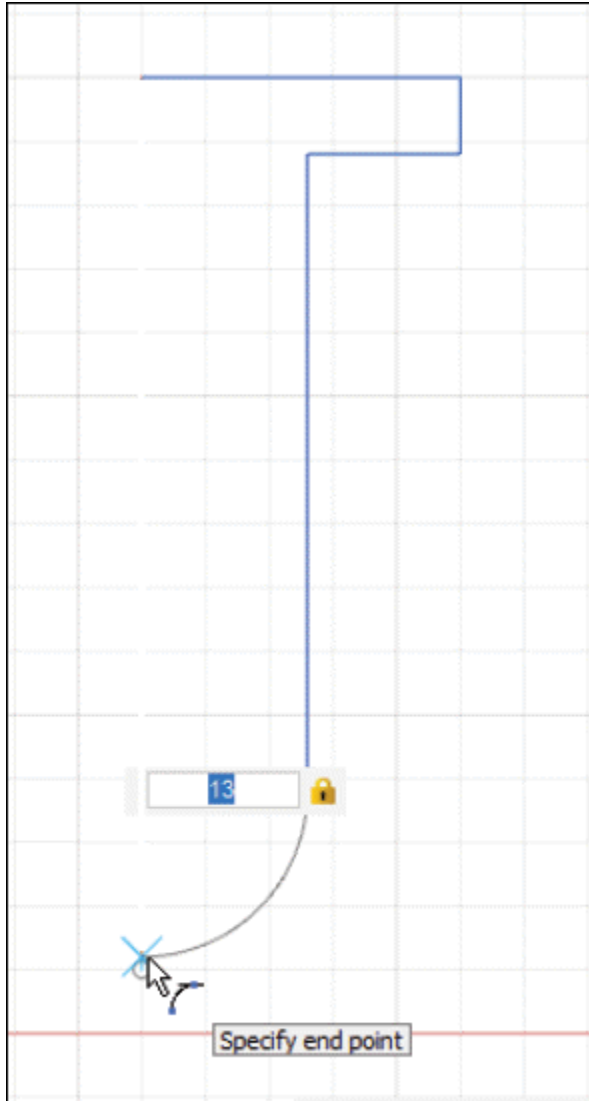


Figure-113. Arc creation

Note that you can also create tangent arc by dragging the cursor while creating line using **Line** tool. To specify dimension of tangent arc, you need to drag the cursor while creating line, hold the left click, specify the radius value for tangent arc, and then release the left click.

Creating Fillet

- Click on the **Fillet** tool from **MODIFY** drop-down/panel in

the **SKETCH** contextual tab in the **Toolbar**.

- Select the lines as shown in [Figure-114](#).
- Enter the value as **3** in the radius box.

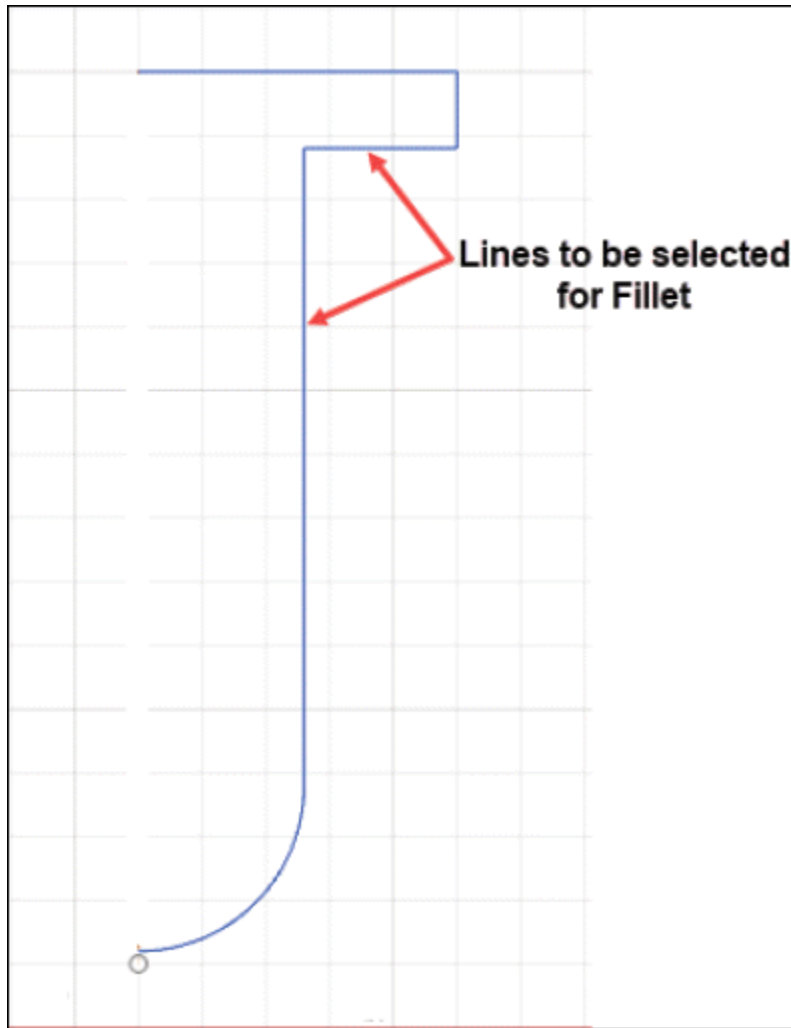


Figure-114. Line selection for fillet

Creating Mirror Copy

- Press **L** key and create a line from origin to end point of the Arc.
- Press **ESC** key from keyboard to exit the tool.
- Now, select the newly created line, right-click on it, and

select **Normal Construction** from **Marking menu** or select the newly created line and press **x** from keyboard to convert line into construction geometry. Press **ESC** to exit selection.

- Select the **Mirror** tool from **CREATE** drop-down and select all the entities we have sketched except construction line; refer to [Figure-115](#).

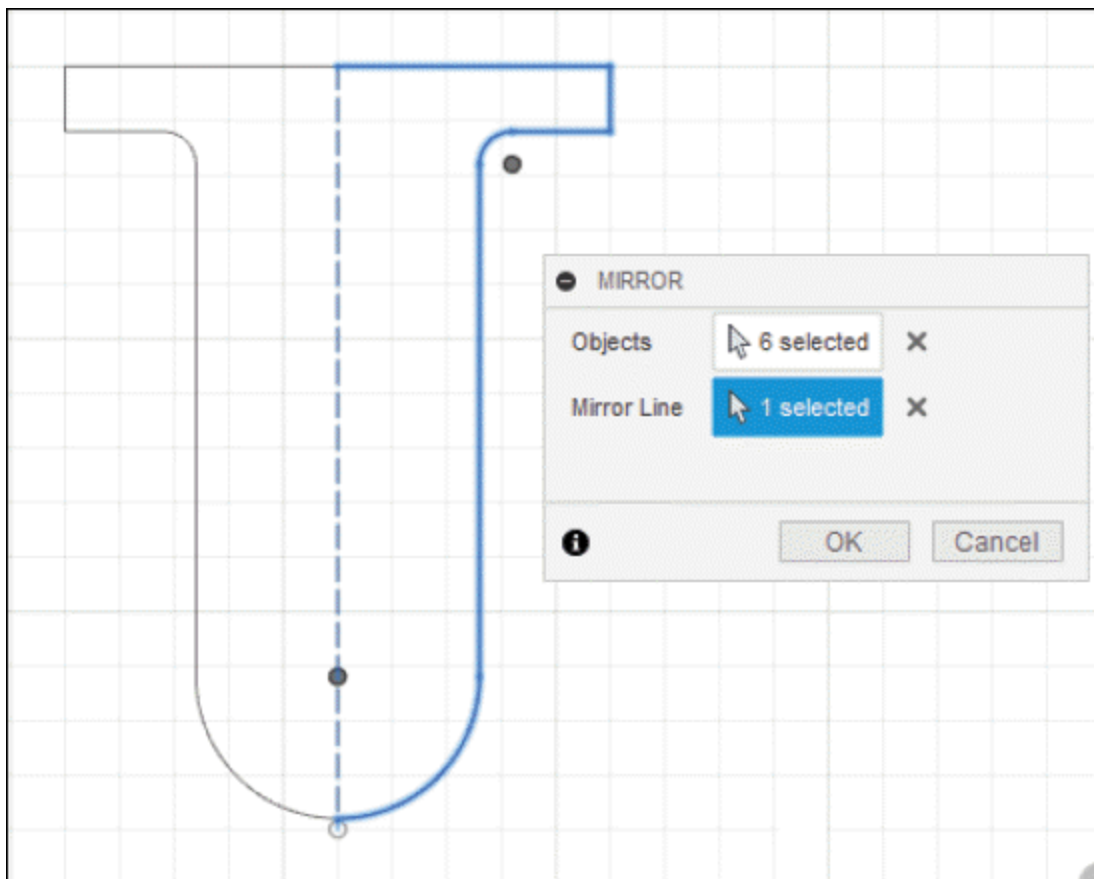


Figure-115. Entities selected for creating mirror

- Click on the **Mirror Line** selection button in the dialog box and select the construction line from sketch. Preview of mirror feature will be displayed.

- Click **OK** button from the **MIRROR** dialog box.

Creating Circle and Lines to complete.

- Click on the **Center Diameter Circle** tool from **Circle** cascading menu.
- Click at the center point of the bottom arc and move the cursor; refer to [Figure-116](#).

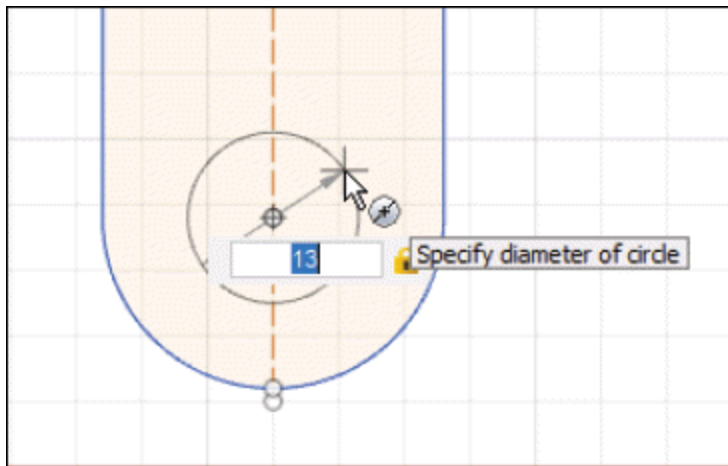


Figure-116. Circle creation

- Enter the dimension as **13** in the input box.
- Click on the **Line** tool from **CREATE** drop-down and click anywhere on the center line (except mid point and end points) to specify the start point of the line; refer to [Figure-117](#).

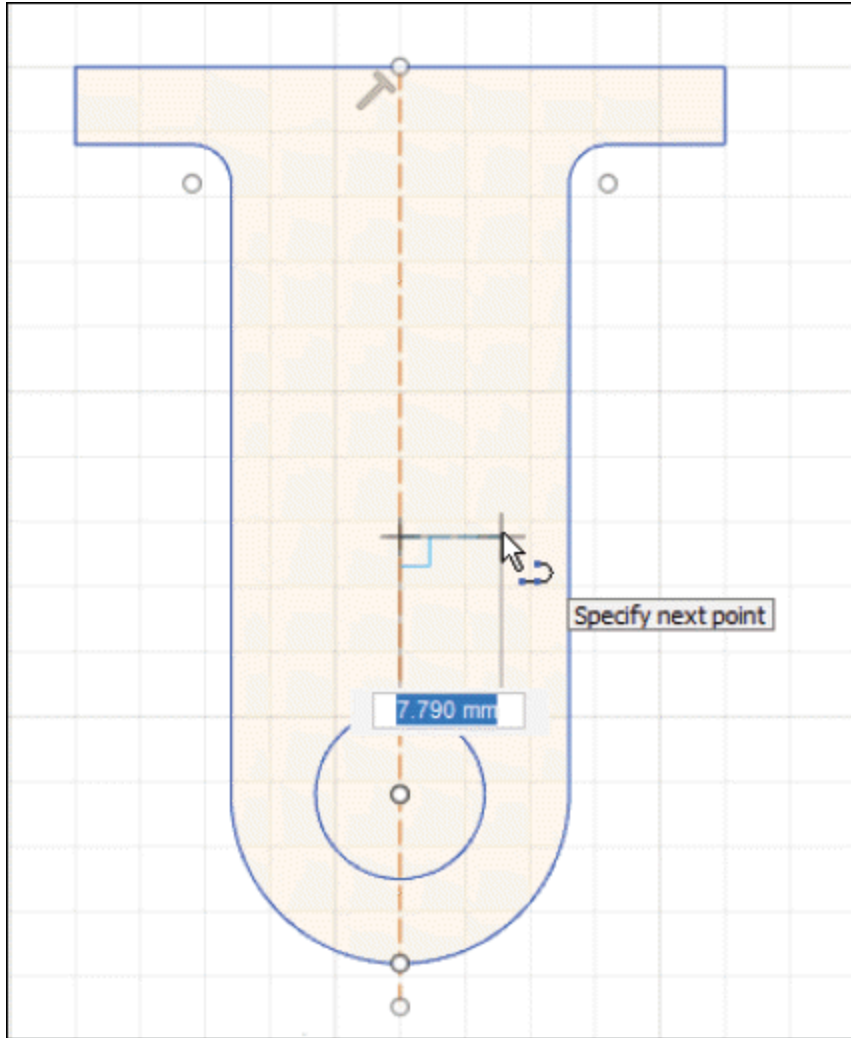


Figure-117. Point selected on centerline

- Move the cursor horizontally towards right and specify the value as **9**. Press **TAB** and click on the screen.
- Move the cursor vertically downwards and click when line passes the arc; refer to [Figure-118](#).

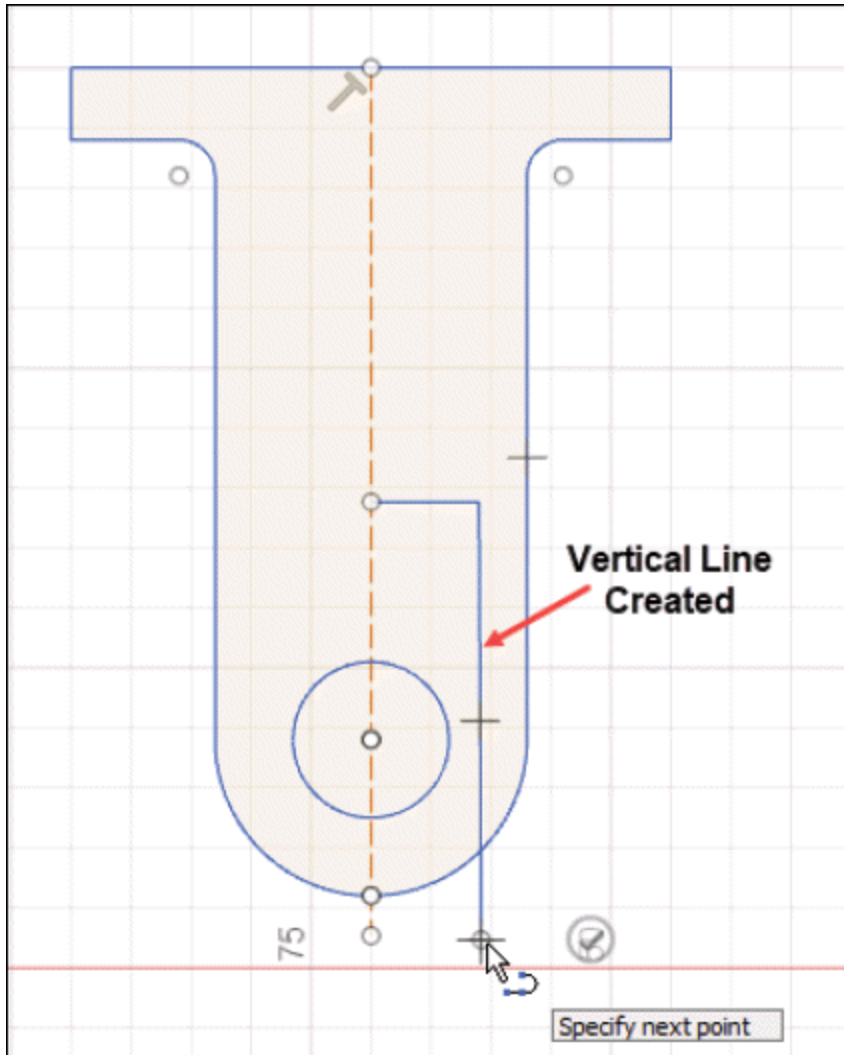


Figure-118. Vertical line created

- Press **ESC** to exit the tool. Click on the **Trim** tool from **MODIFY** drop-down and remove the extra portion outside the arc. Press **ESC** to exit the **Trim** tool.
- Mirror both the lines as we did earlier. The sketch should display as shown in [Figure-119](#).

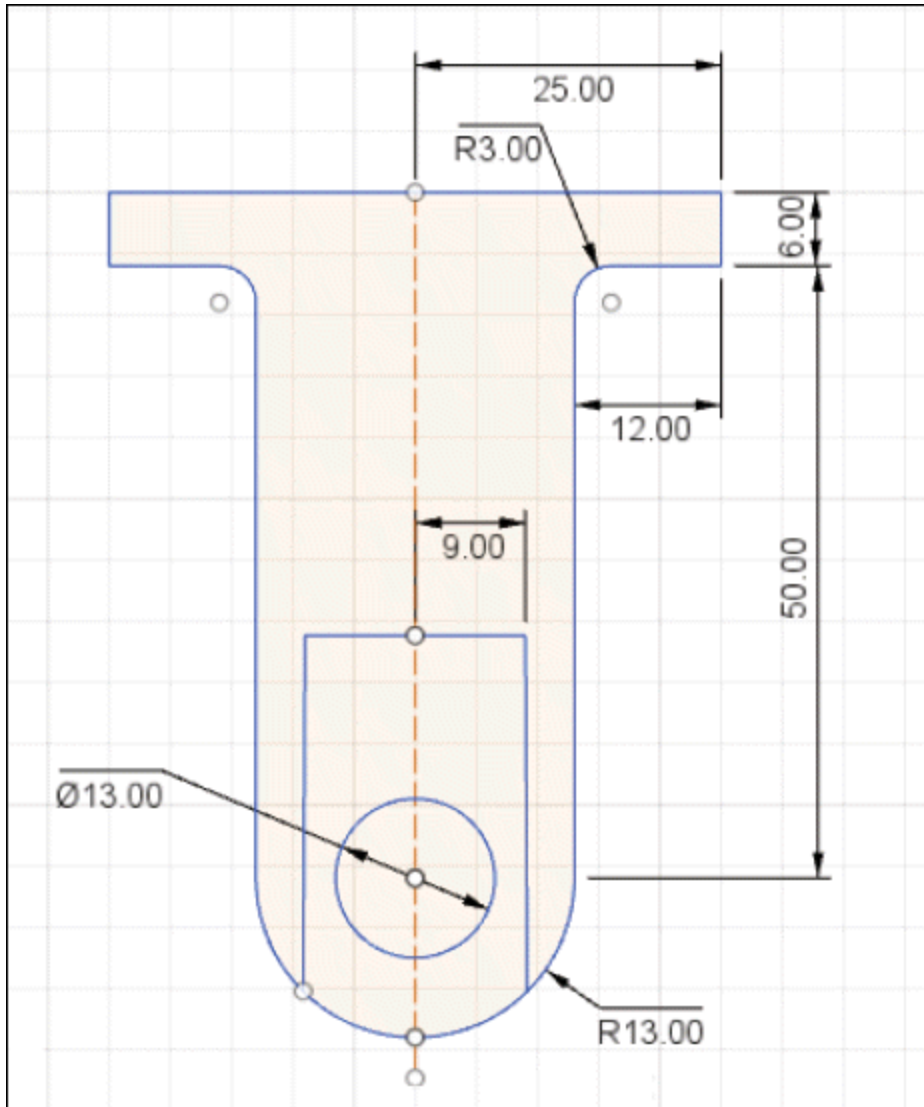


Figure-119. Sketch after all sketching operation

Dimensioning Sketch

Some of the dimensions are applied automatically while creating sketch. To create rest of the dimensions, click on the **Sketch Dimension** tool from **CREATE** panel/drop-down. Dimension symbol will get attached to cursor.

- Click on the arc and place the dimension at proper spacing if not created already. Type the radius in dynamic input

box as **13** and press **ENTER**.

- Click on the two lines as shown in [Figure-120](#).

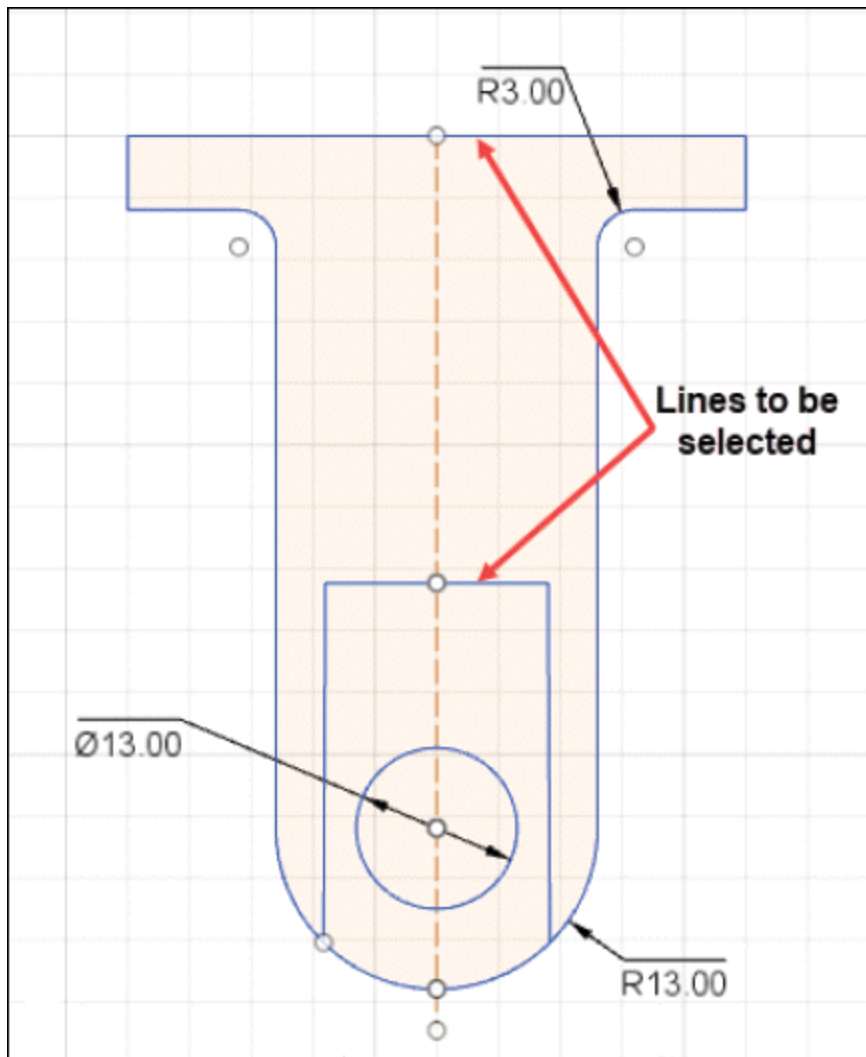


Figure-120. Lines to be selected for dimensioning

- Click to place the dimension at its proper place. In the dynamic input box, enter the value as **40**.

Similarly, dimension all the entity in the sketch until the sketch is fully defined. The final sketch after dimensioning will be displayed as shown in [Figure-121](#). You can delete extra dimensions before applying real dimensions.

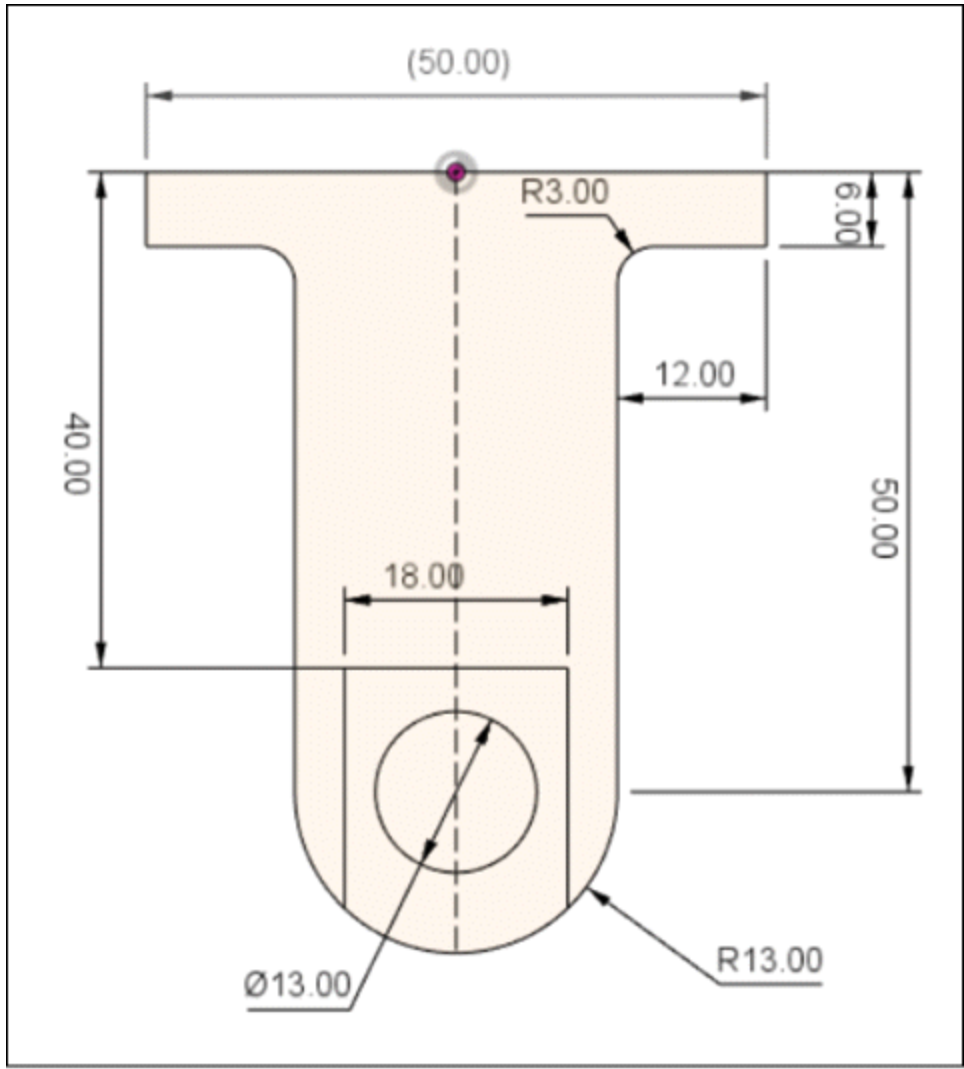


Figure-121. Final sketch

FULLY DEFINED, UNDER-DEFINED, AND OVER-DEFINED SKETCHES

Till this point, you have worked on various sketching tools and you have created sketch for two practical sessions. You might have noticed that some entities are displayed in blue color while some are displayed in bold black color. Also, there are dimensions which are enclosed in bracket. Here, we will discuss why these colors and different dimensions

appear.

Fully Defined Sketches

A fully defined sketch is the one whose specifications can not be changed unintentionally. In some complex sketches, when you change the dimension of one entity, the dimension of other entity gets changed automatically. If you have the sketch fully defined then the dimension of the entities will not change unintentionally. In technical terms, a fully defined sketch is the one in which entities have zero degree of freedom. In Autodesk Fusion, the sketch that is fully defined will be displayed in bold black color; refer to [Figure-122](#). In a fully defined sketch, if you try to drag any entity of the sketch then it will not move because its degree of freedom is zero. A fully defined sketch is always recommended for creating features in CAD.

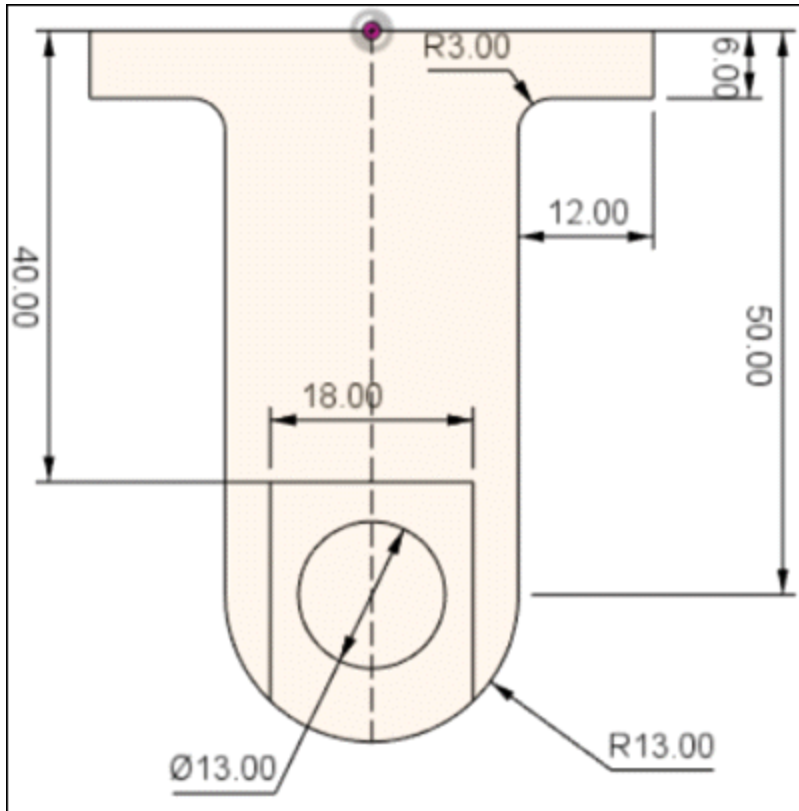


Figure-122. Fully defined sketch

Under Defined Sketches

An under-defined sketch is the one in which all the required dimensions are not applied. An under-defined sketch is displayed in blue color in Autodesk Fusion 360. When you change the value of a dimension or drag an entity in the under-defined sketch then other entities may change shape/size accordingly. An under-defined sketch is not recommended for CAD modeling.

Over Defined Sketches

An over defined sketch is the one where more dimensions are applied than those required to fully define the sketch. In over-defined sketches, extra dimensions are bound by

parenthesis like the **50** dimension as shown in [Figure-121](#).

PRACTICAL 3

Create the sketch as shown in [Figure-123](#). Also, dimension the sketch as per the figure.

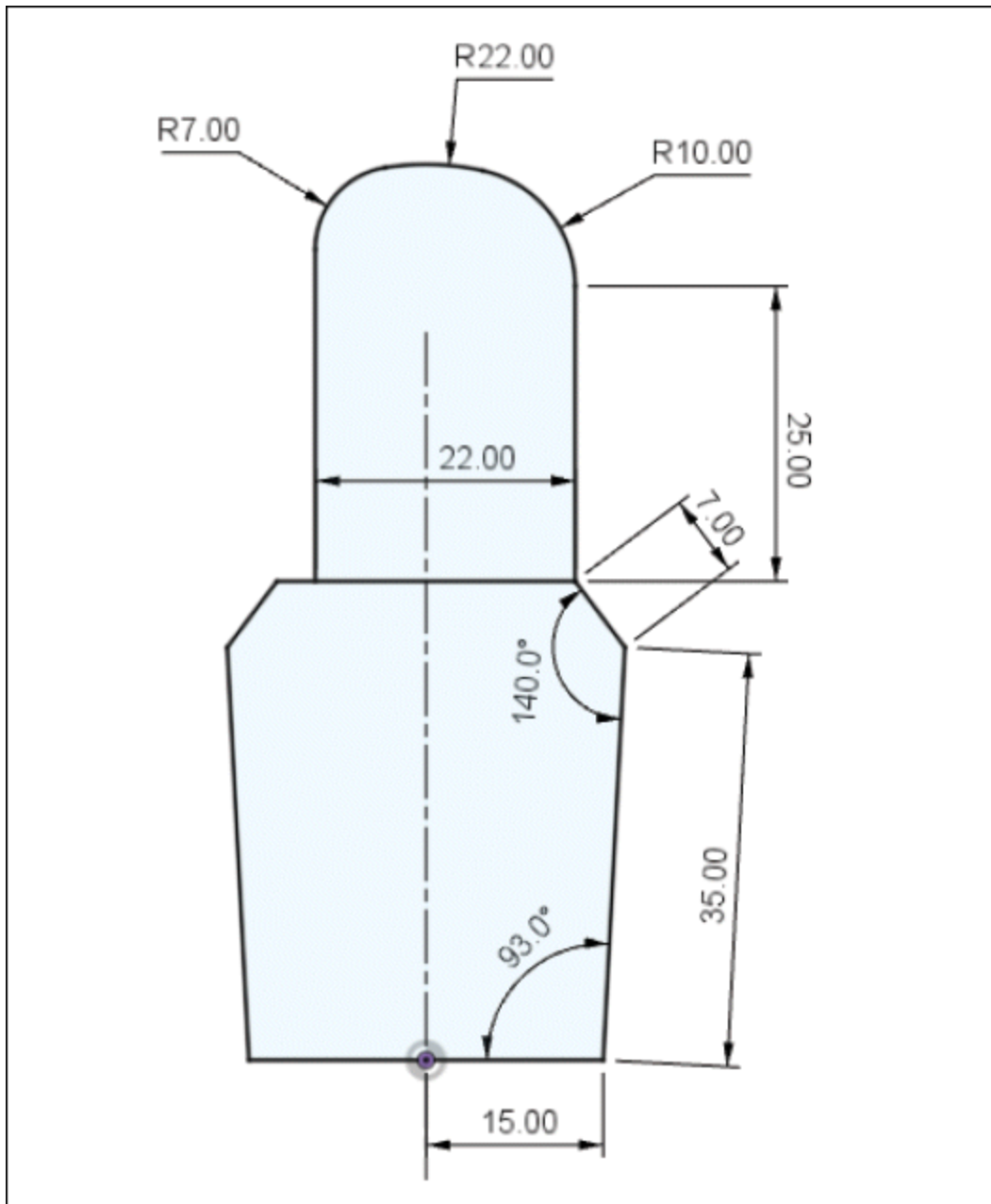


Figure-123. Practical 3

Steps to be performed:

Below is the step by step procedure of creating the sketch shown in [Figure-123](#).

Starting Sketching Environment

- Start **Autodesk Fusion 360** if not started already.
- Select the **New Design** tool from **File** menu to start a new design.
- Select the **Create Sketch** tool from **CREATE** drop-down.
- On selecting this tool, you will be asked to select a plane for creating sketch.
- Select the **Top Plane** to create the sketch.

Creating Lines

- Select **Line** tool from **CREATE** drop-down and create a vertical line starting from origin and moving upward. (No need of dimensions, just make it a long one as it is going to be centerline.)
- Select the line and right-click on this line. Marking menu will be displayed. Select **Normal/Construction** tool to make this line as a center line; refer to [Figure-124](#). Press **ESC** to exit the tool.

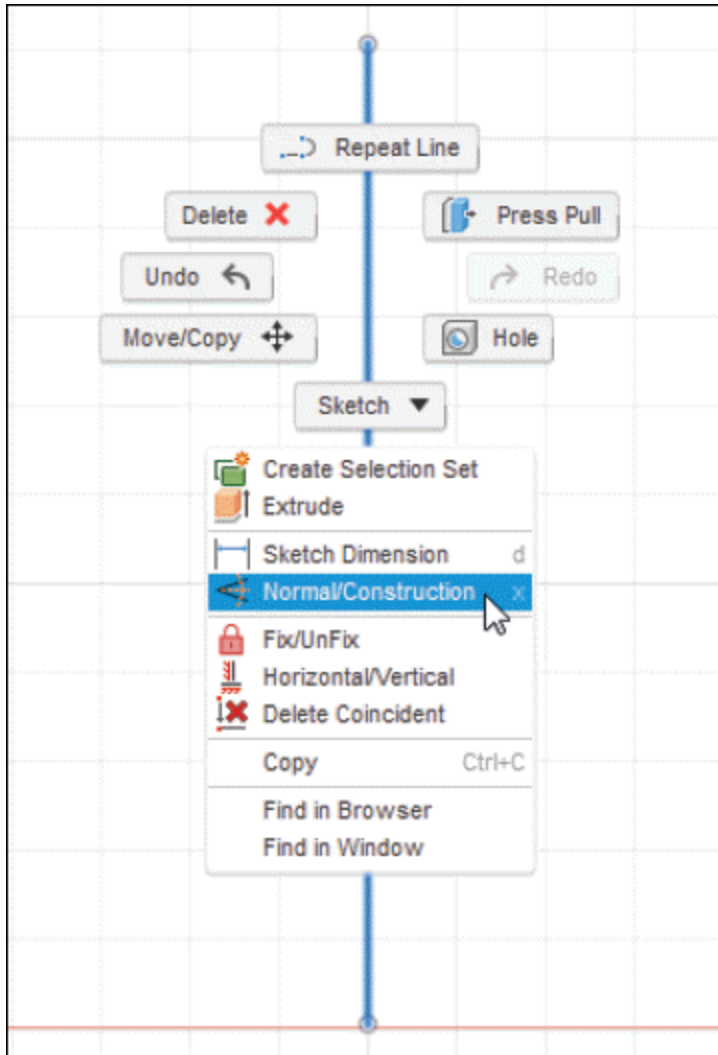


Figure-124. Creating centerline

- Press **L** key to start **Line** tool again and click at the end point of center line to create a line.
- Move the cursor towards right and specify the value as **15** in the dimension box. Press **TAB** key to lock the dimension and click on the screen.
- Move the cursor upward with some outward angle. Specify length as **35** in dimension box. Press **TAB** key to lock the dimension and switch to angle dimension. Specify the

angle value as **93** degree and click on the screen.

- Move the cursor to the left, specify the value as **7** in the length input box and **140** degree in angle input box; refer to [Figure-125](#).

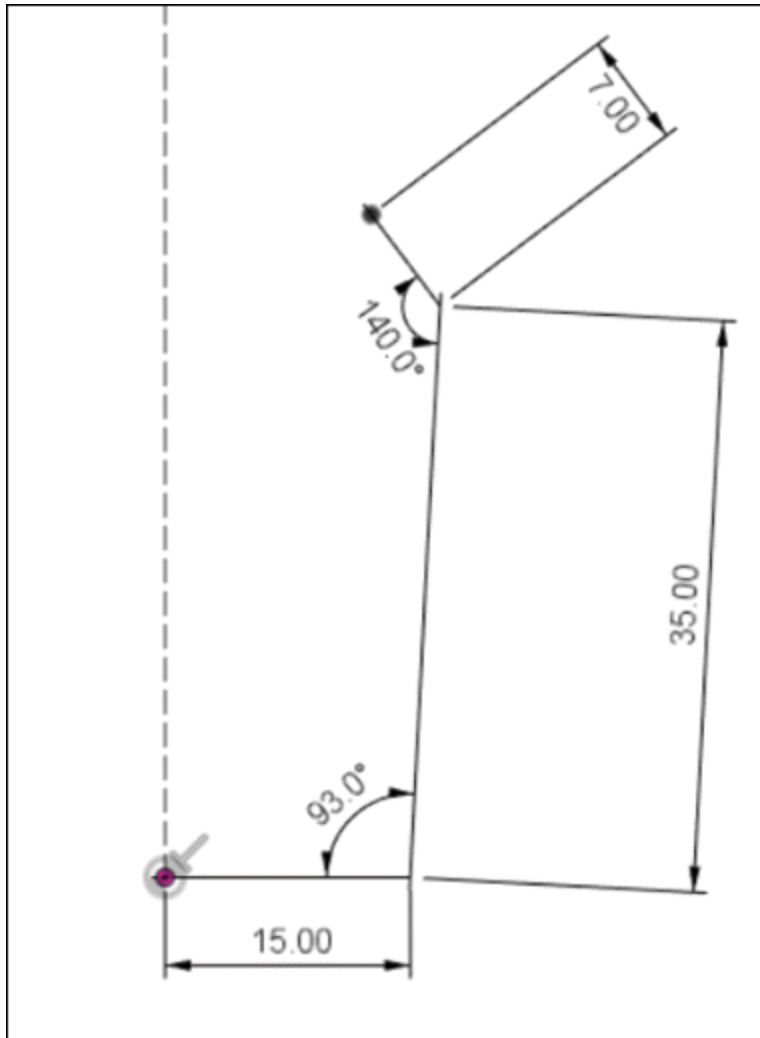


Figure-125. Creating lines with dimensions

Creating Mirror copy

- Select the **Mirror** tool from **CREATE** drop-down and select the entities we have sketched except center line for mirror.
- Click on the **Mirror Line** selection button and select the

center line of sketch as mirror line. Preview of mirror will be displayed; [Figure-126](#).

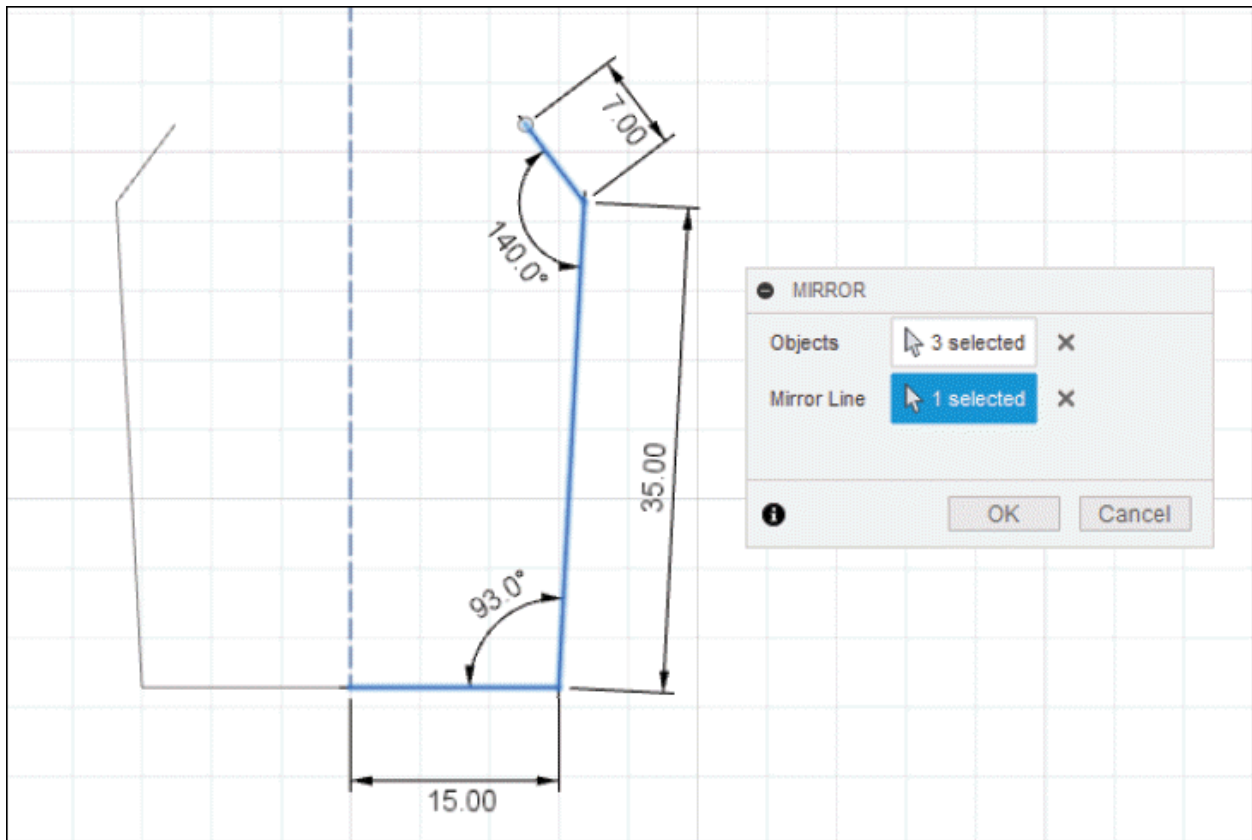


Figure-126. Preview of mirror

- Click **OK** button from **MIRROR** dialog box to complete the mirror copy.
- Now, select **Line** tool and create a line joining the open ends of sketch; refer to [Figure-127](#).

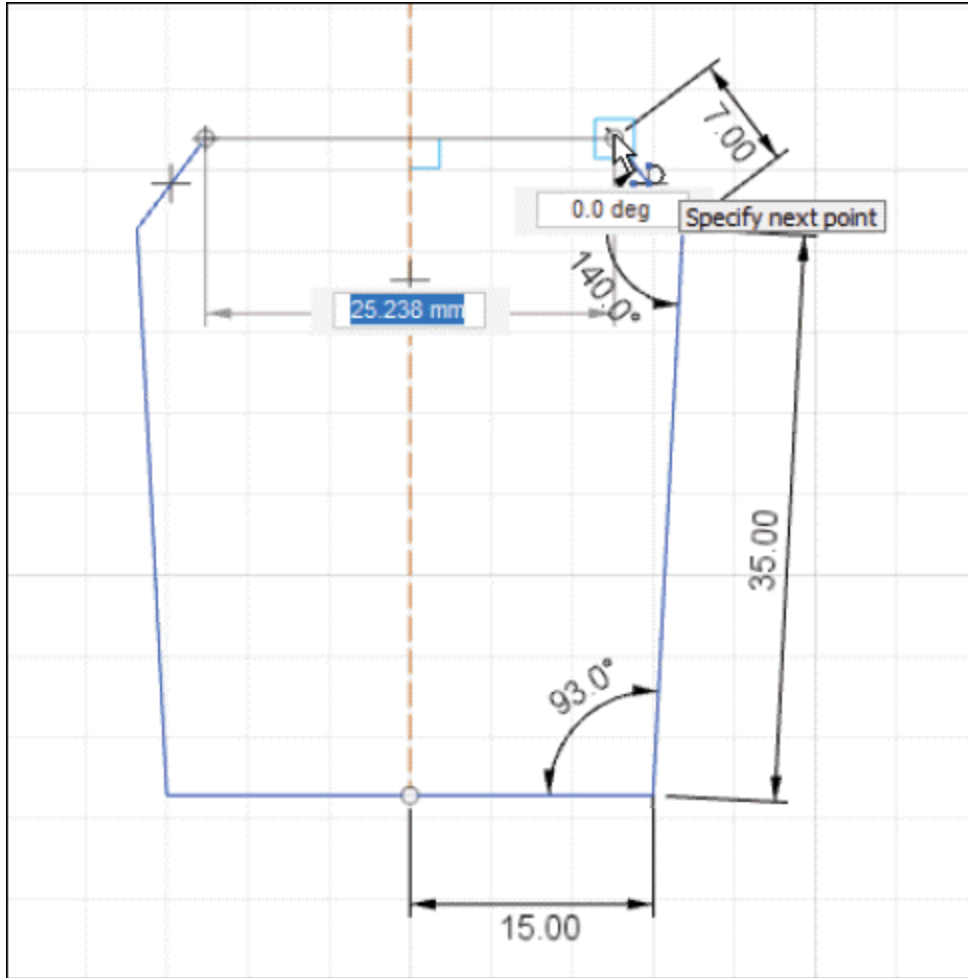


Figure-127. Joining the open ends of sketch

- Create a vertical line from one end of the sketch of length **25 mm** as shown in [Figure-128](#).

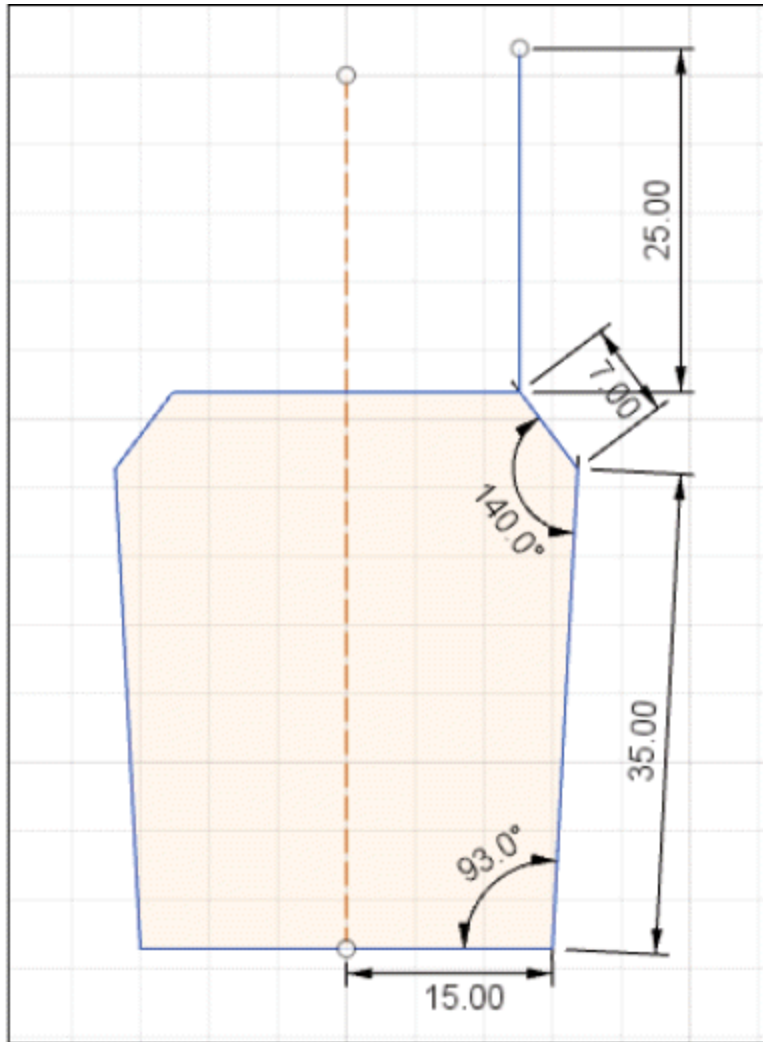


Figure-128. Creating line from right side of sketch

Creating Arcs

- Click **Tangent Arc** tool from **Arc** cascading menu in **CREATE** drop-down.
- Click on the end point of vertical line recently created and move the cursor upwards. Create three consecutive arc joining each other as shown in [Figure-129](#). Press **ESC** to exit the tool.

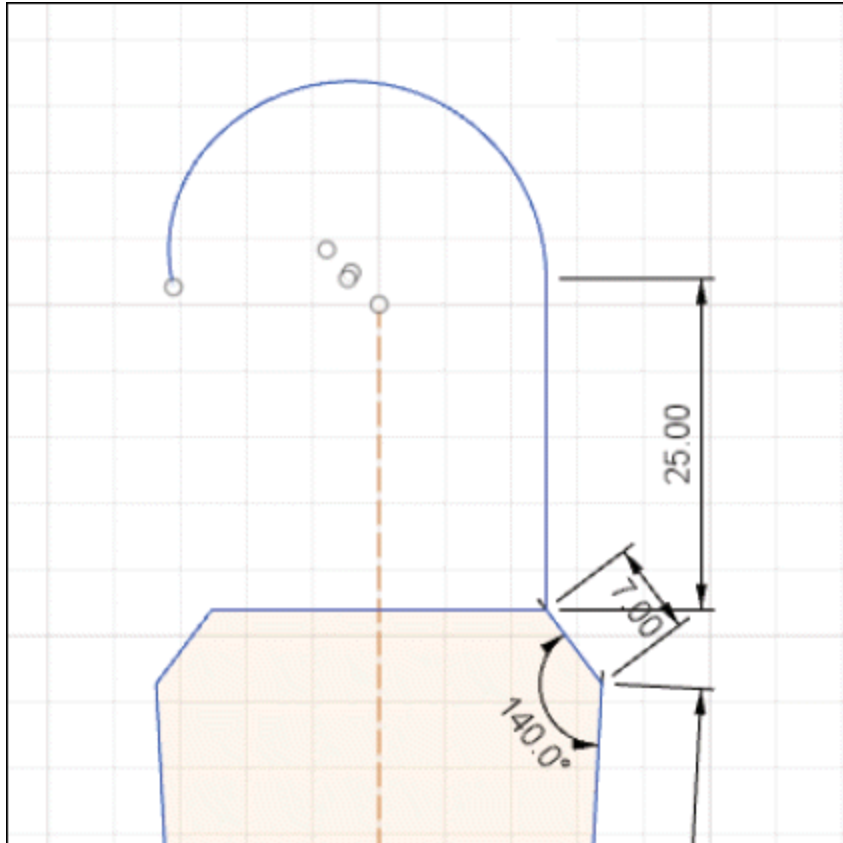


Figure-129. Creating three consecutive arcs

- Select **Sketch Dimension** tool from **CREATE** drop-down or press **D** key to select this tool.
- Apply radius dimensions to three arcs as **10**, **22**, and **7**, respectively; refer to [Figure-130](#). Make sure horizontal and vertical constraints are applied to the straight lines in the sketch.

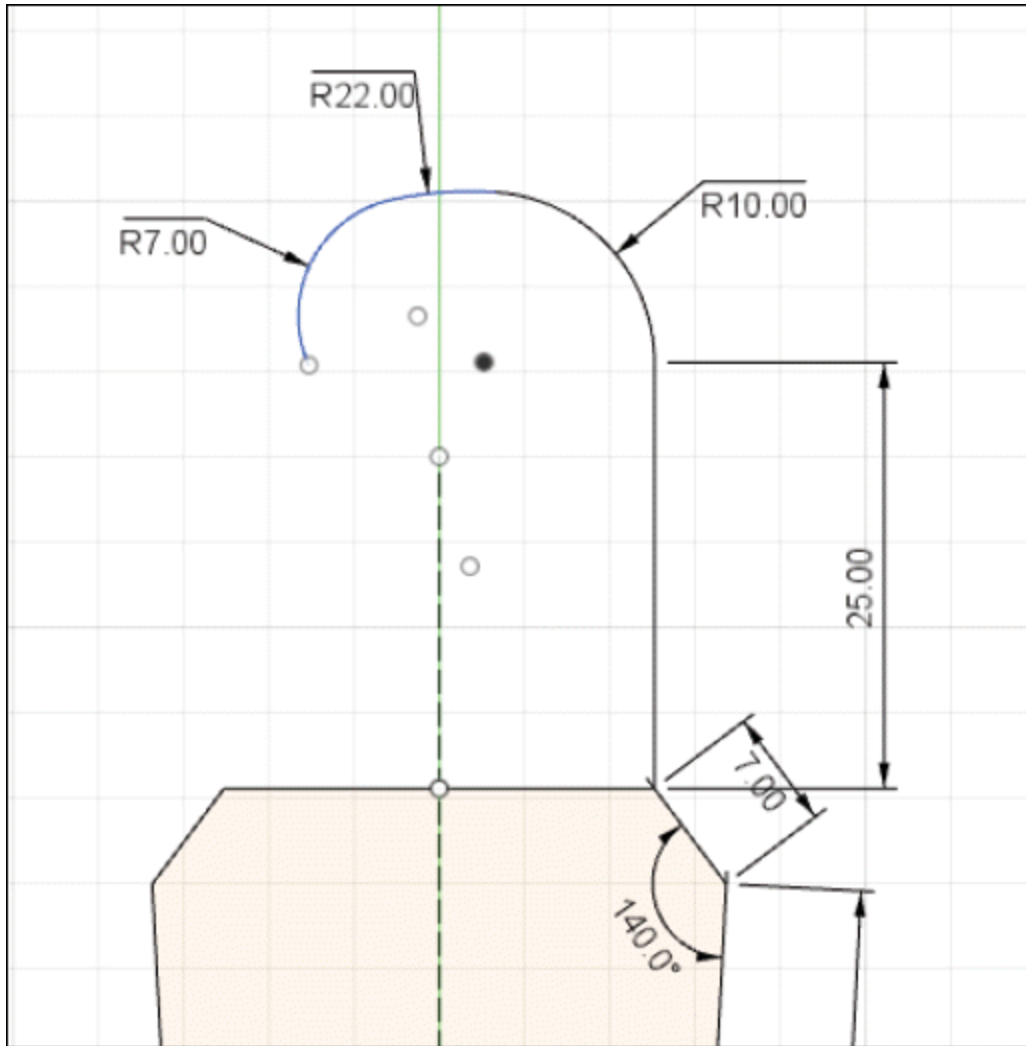


Figure-130. After sketch dimension on arc

- Select the **Line** tool and create a line joining end of recently created arc to horizontal line in the sketch; refer to [Figure-131](#). Press **ESC** tool to exit the command.

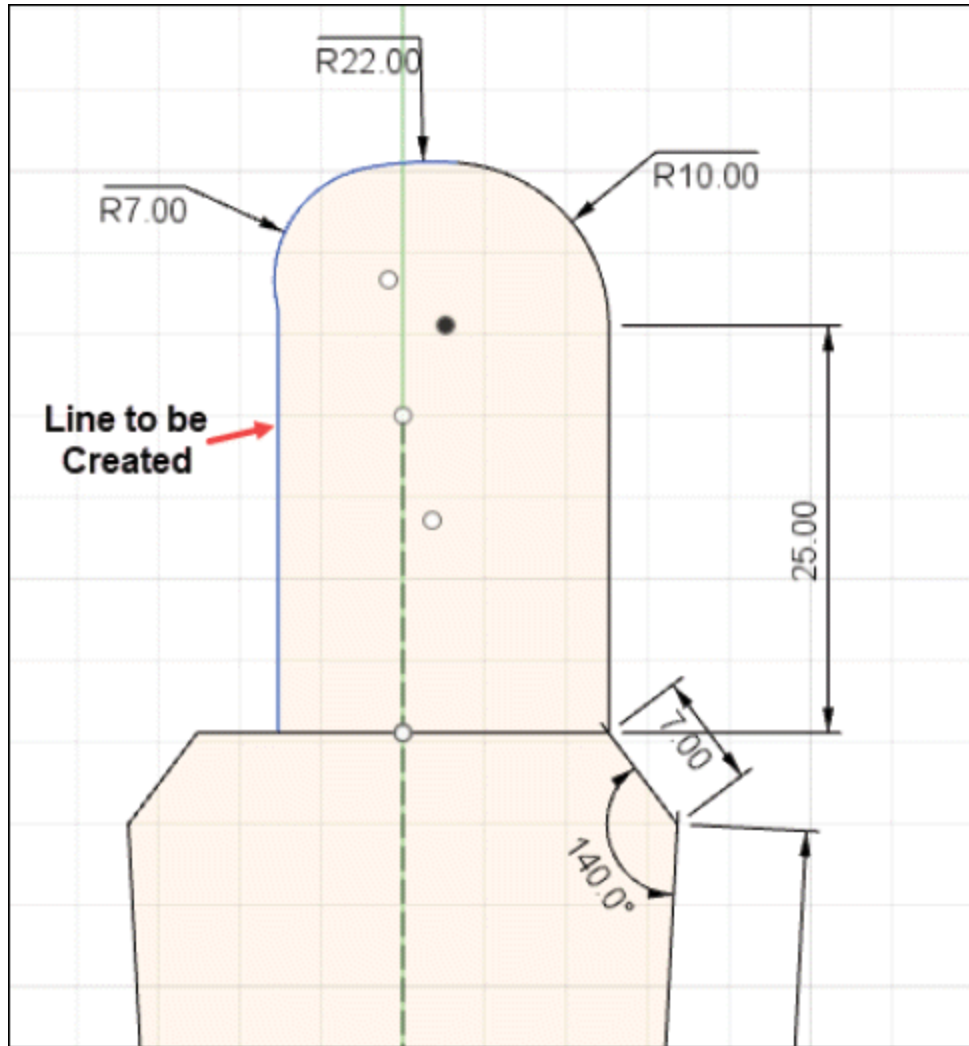


Figure-131. Line to be created

You may find that there is not a smooth transition between newly created line and arc as in our case. In such cases, apply the tangent constraint between line and arc; refer to [Figure-132](#).

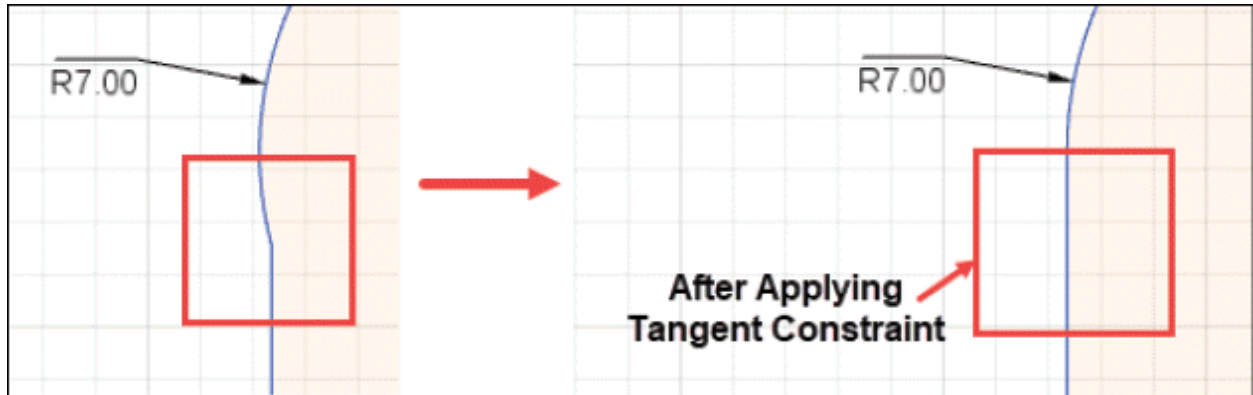


Figure-132. Applying tangent constraint for smooth transition

Dimensioning Sketch

- Select the unwanted dimensions and delete them by pressing **DELETE** key.
- Click on the **Sketch Dimension** tool or press **D** key to select the tool.
- Select the entity and apply the dimension; refer to [Figure-133](#).

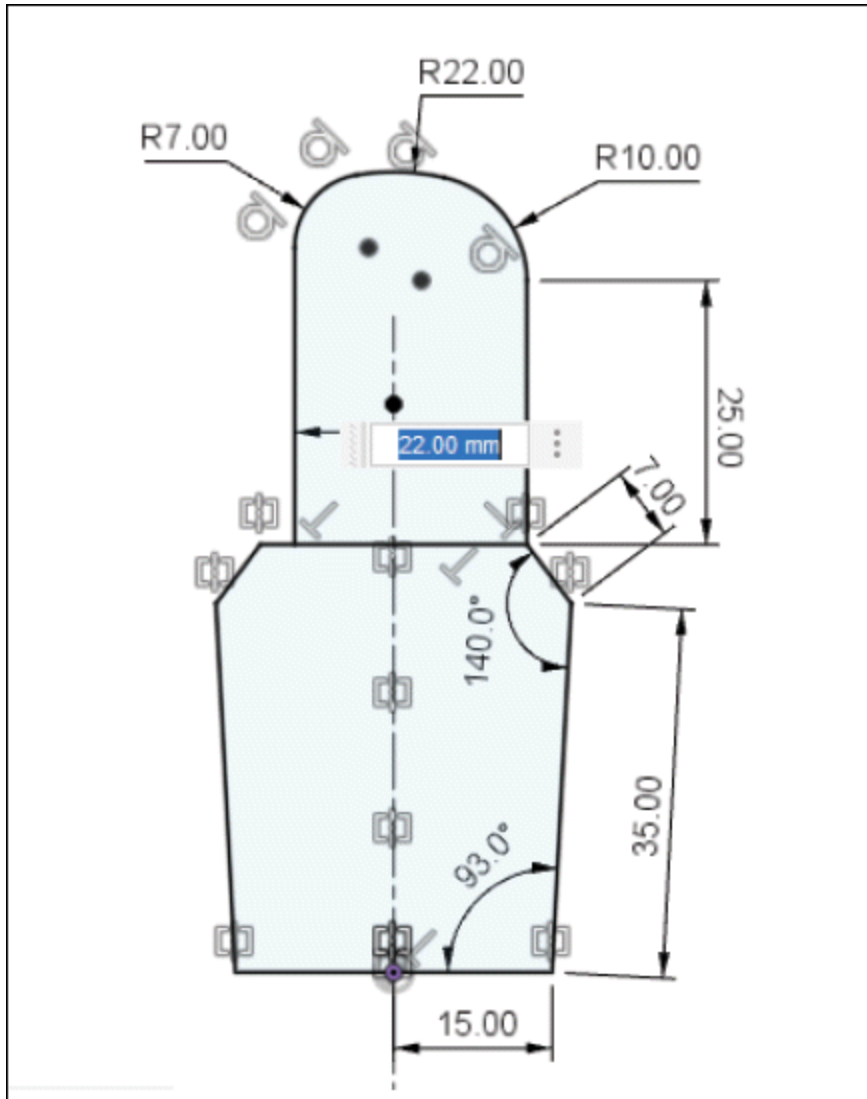


Figure-133. Select entities for dimension

- Drag the dimension to proper place.

Similarly, dimension all the other entities in the sketch until it is fully defined. The final sketch after dimensioning will be displayed as shown in [Figure-134](#).

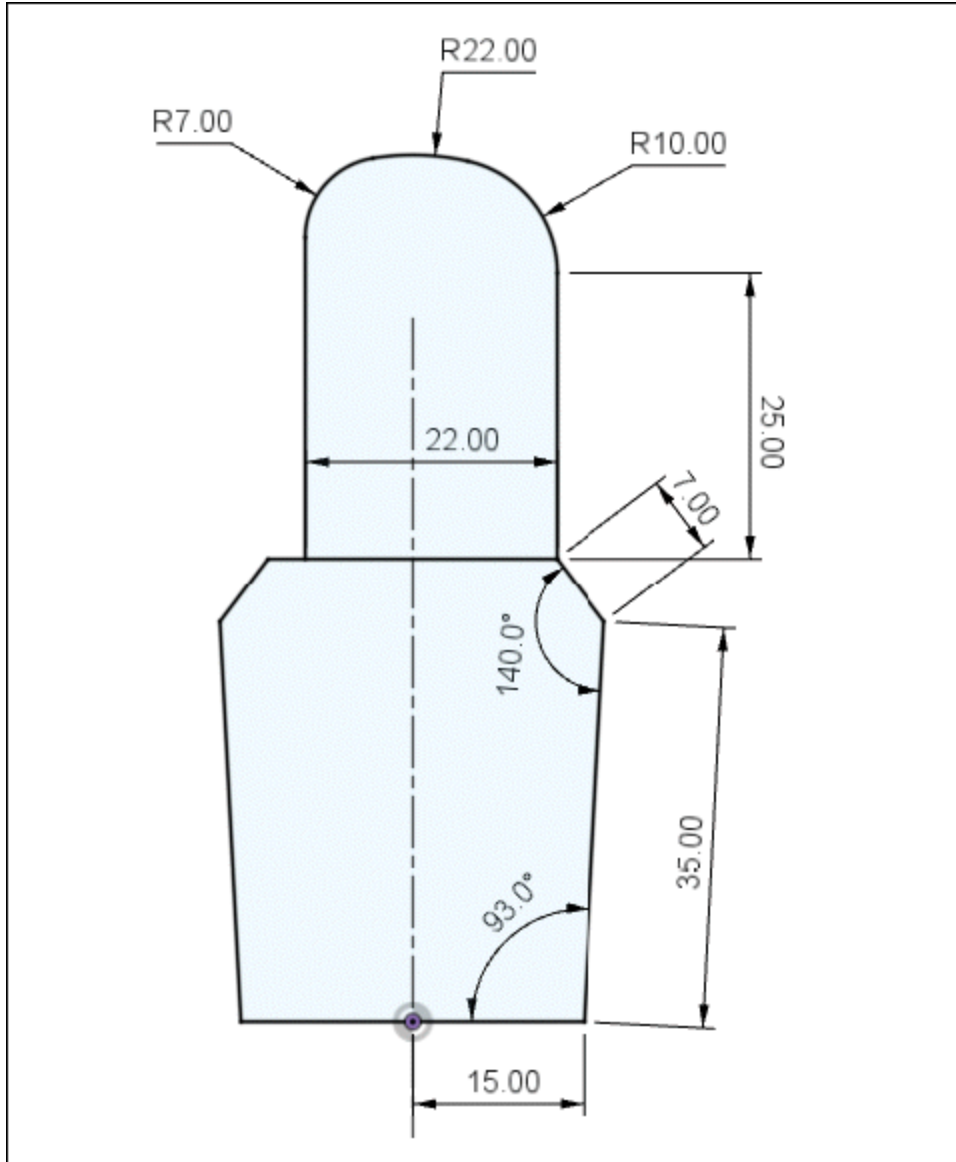


Figure-134. Final sketch of practical 3

Following are some sketches for practice.

PRACTICE 1

Create the sketch as shown in [Figure-135](#).

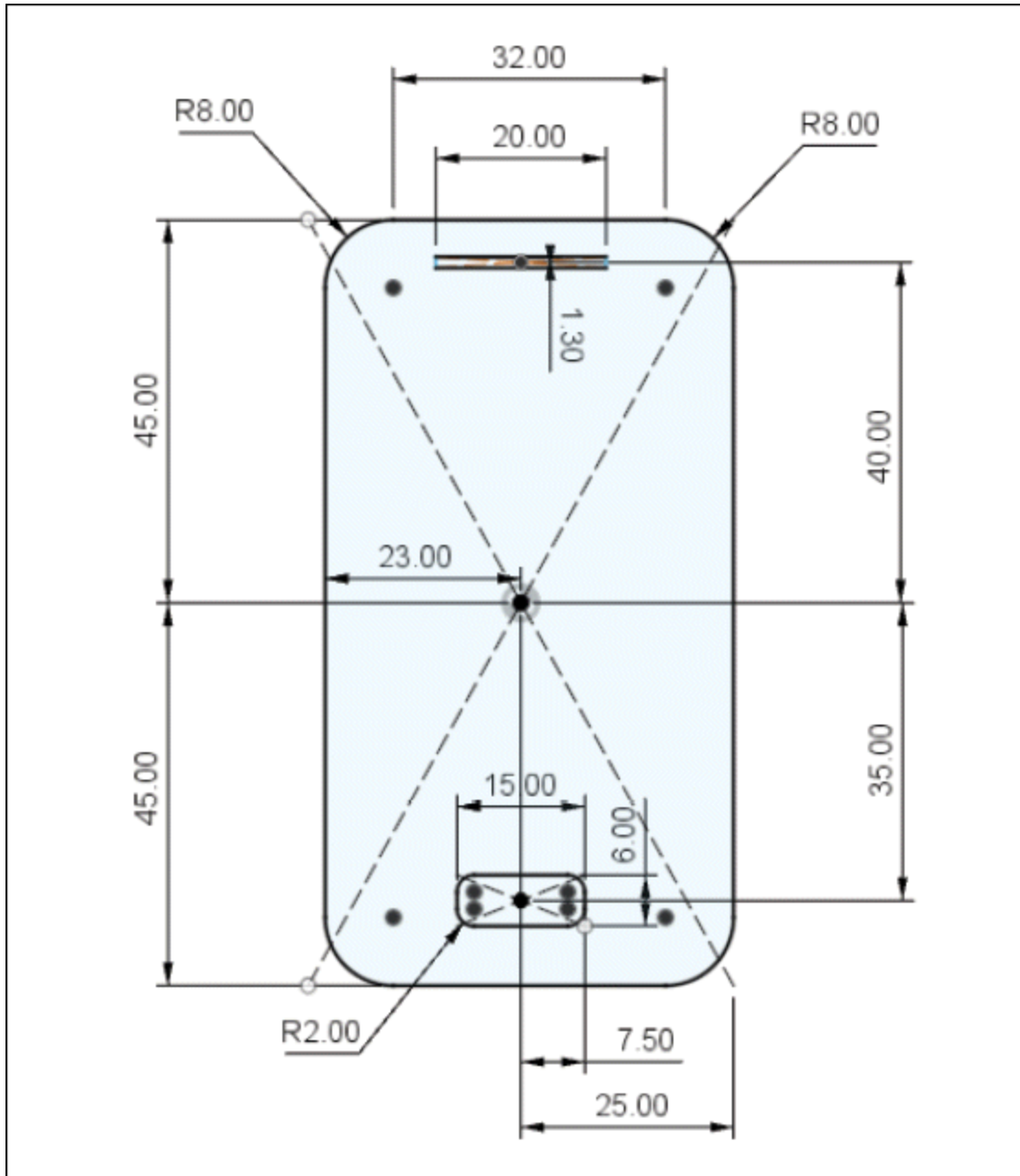


Figure-135. Practice 1

PRACTICE 2

Create the sketch as shown in [Figure-136](#).

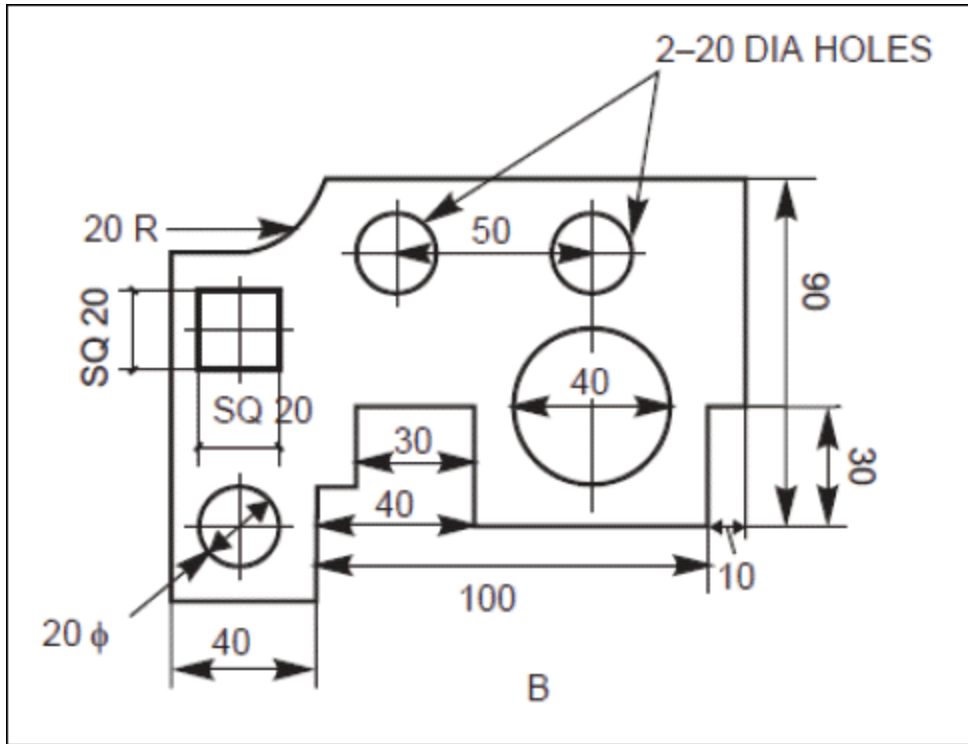


Figure-136. Practice 2

PRACTICE 3

Create the sketch as shown in [Figure-137](#).

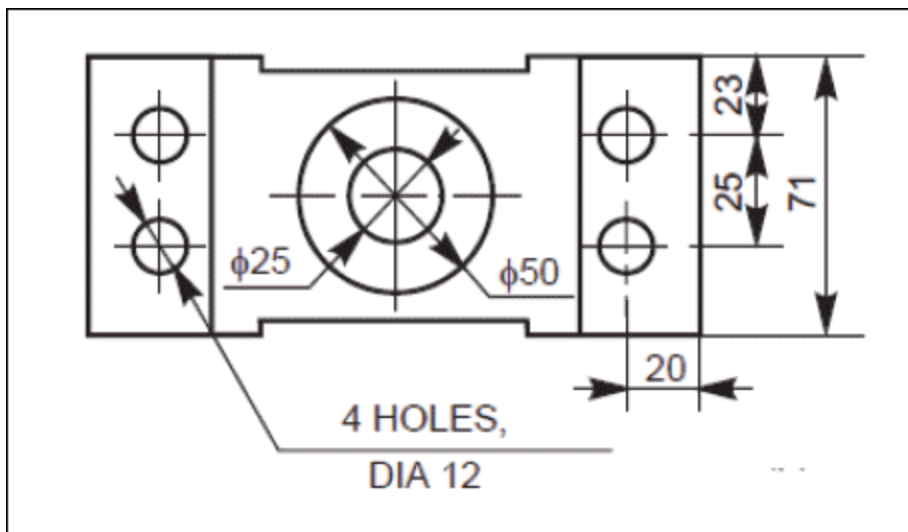


Figure-137. Practice 3

PRACTICE 4

Create the sketch as shown in [Figure-138](#).

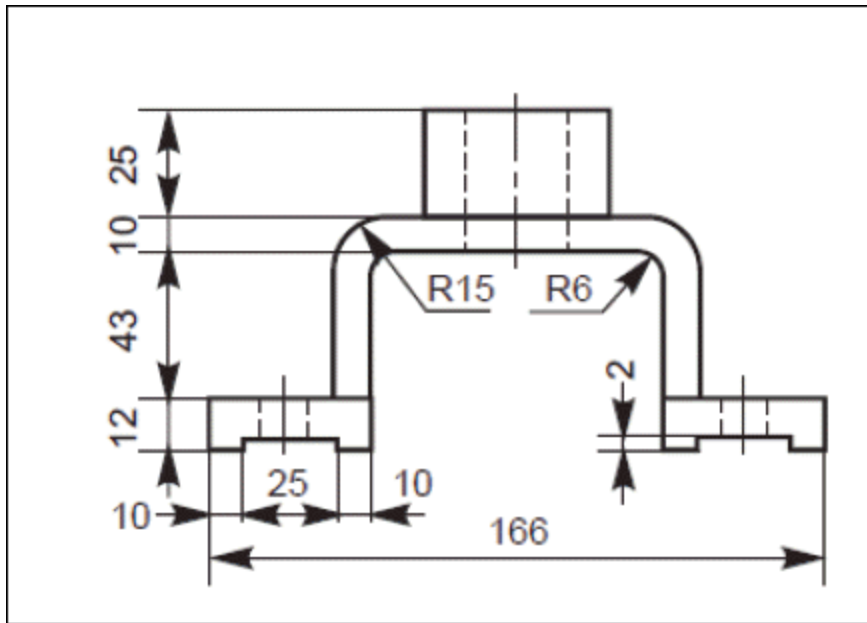


Figure-138. Practice 4

To get more exercise, mail us at cadcamcaeworks@gmail.com

SELF ASSESSMENT

Q1. In Autodesk Fusion 360, which of the following workspace has tool to start a new sketch?

- a. RENDER
- b. DESIGN
- c. DRAWING
- d. SIMULATION

Q2. Which of the following shortcut keys can be used to convert selected line into a construction line?

- a. L

b. c

c. y

d. x

Q3. While the **Line** tool is active in sketch, if you drag the cursor then a tangent arc is created. (T/F)

Q4. Which of the following tool is not available in Autodesk Fusion Sketch mode to create an arc?

a. Center Point Arc

b. 3-Point Arc

c. Tangent Arc

d. Elliptical Arc

Q5. Which of the following **Polygon** tool should be used to create polygon if distance between two parallel edges of polygon and number of edges are given?

a. Circumscribed Polygon

b. Inscribed Polygon

c. Edge Polygon

d. All of the above.

Chapter 3

3D Sketch and Solid Modeling

Topics Covered

The major topics covered in this chapter are:

- ***3D Sketches***
- ***Selection Tools***
- ***Solid Modeling***
- ***Create drop-down***
- ***Construction Geometry***
- ***Project/Include Tools***

3D SKETCHES

3D Sketches are those sketches which are not confined to single plane. Assume a line connecting end points of two lines in YZ and ZX planes; refer to [Figure-1](#). Such a sketch can be called 3D Sketch; refer to [Figure-2](#).

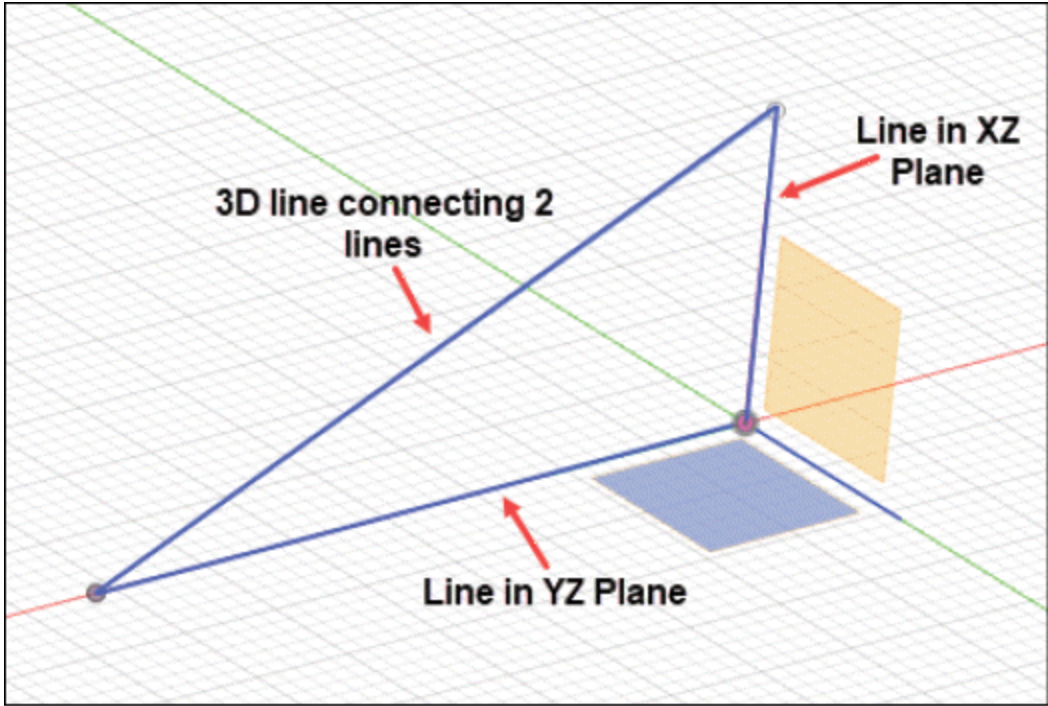


Figure-1. 3D line

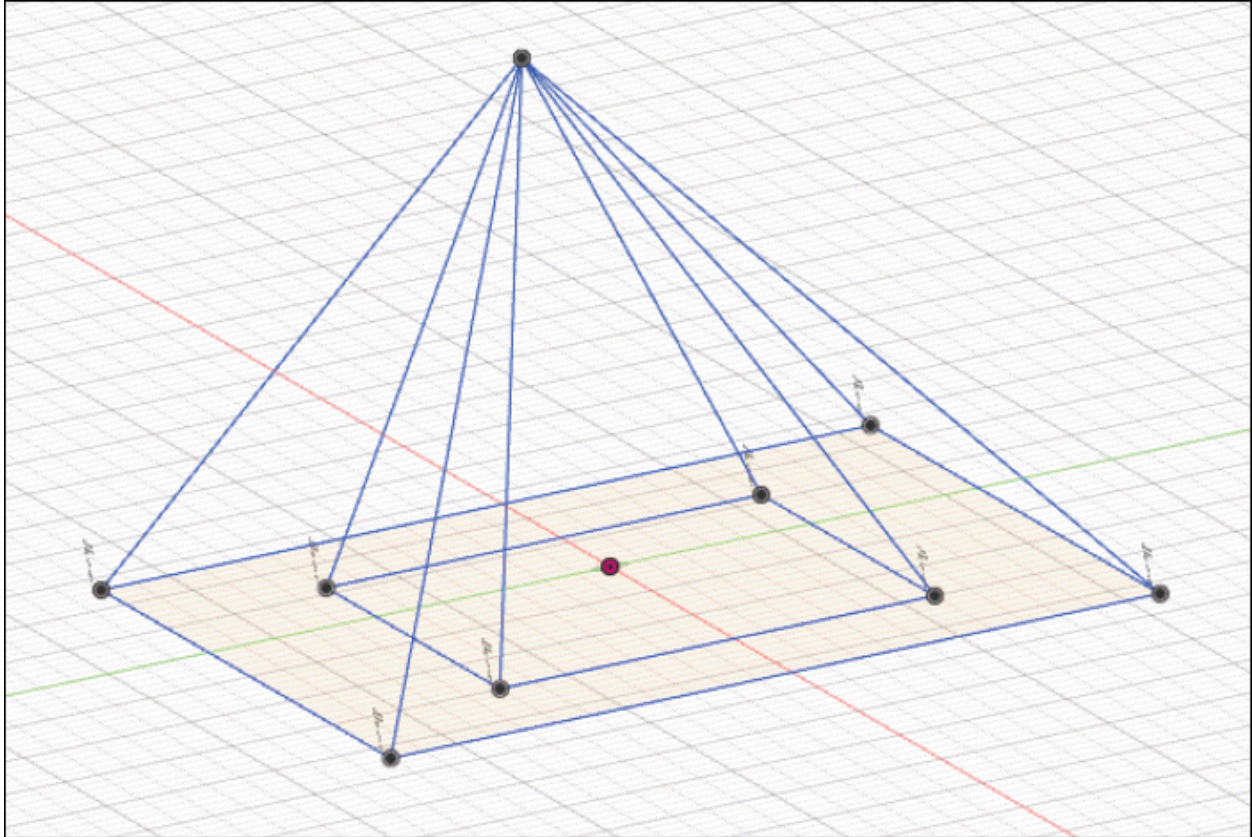


Figure-2. Example of 3D sketch

In Autodesk Fusion 360, there are no specific tools for creating 3D sketches except a check box to enable or disable 3D sketching in **SKETCH PALETTE**. So, we are back to basics of plane orientations for creating 3D sketches. The process of creating a 3D sketch is explained next with the help of an example.

Creating 3D Sketch

Create the sketch of frame as shown in [Figure-3](#).

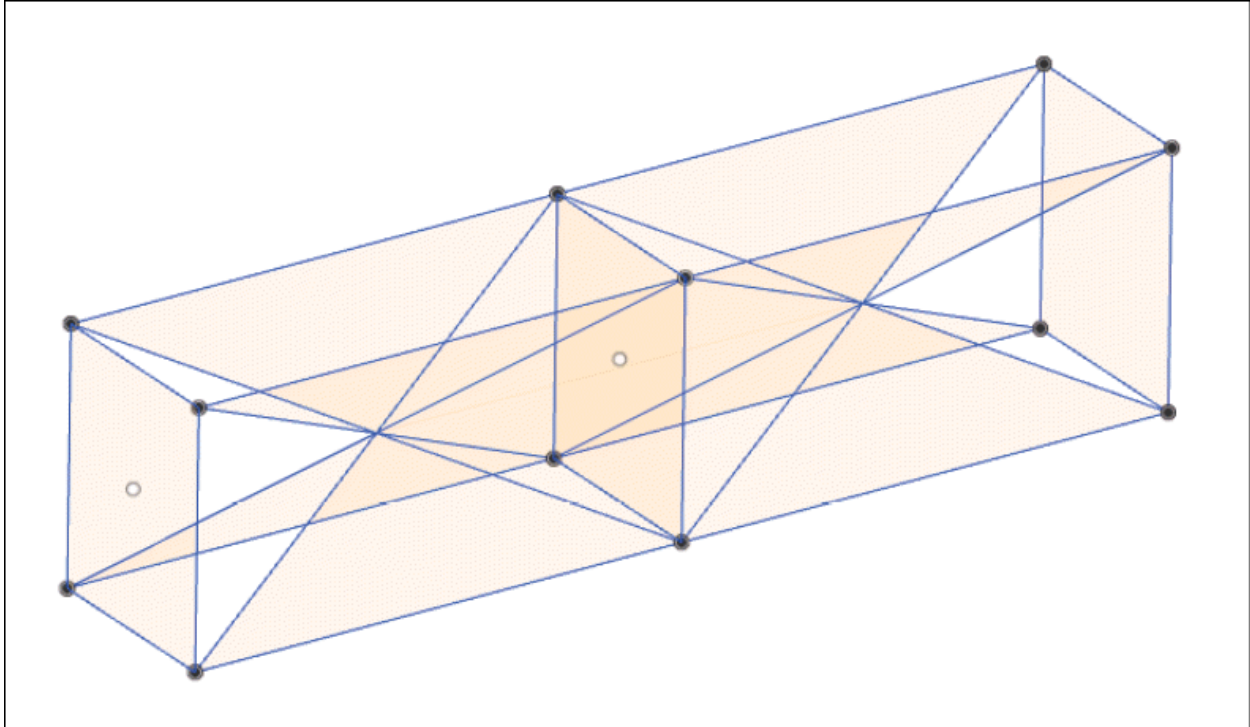


Figure-3. Load bearing frame sketch

Creating Base Sketches and Planes

- After starting a new design file in Autodesk Fusion, click on the **Create Sketch** button from the **CREATE** drop-down. You will be asked to select a sketching plane.
- Select the **XY** plane as sketching plane and create a center point rectangle as shown in [Figure-4](#).

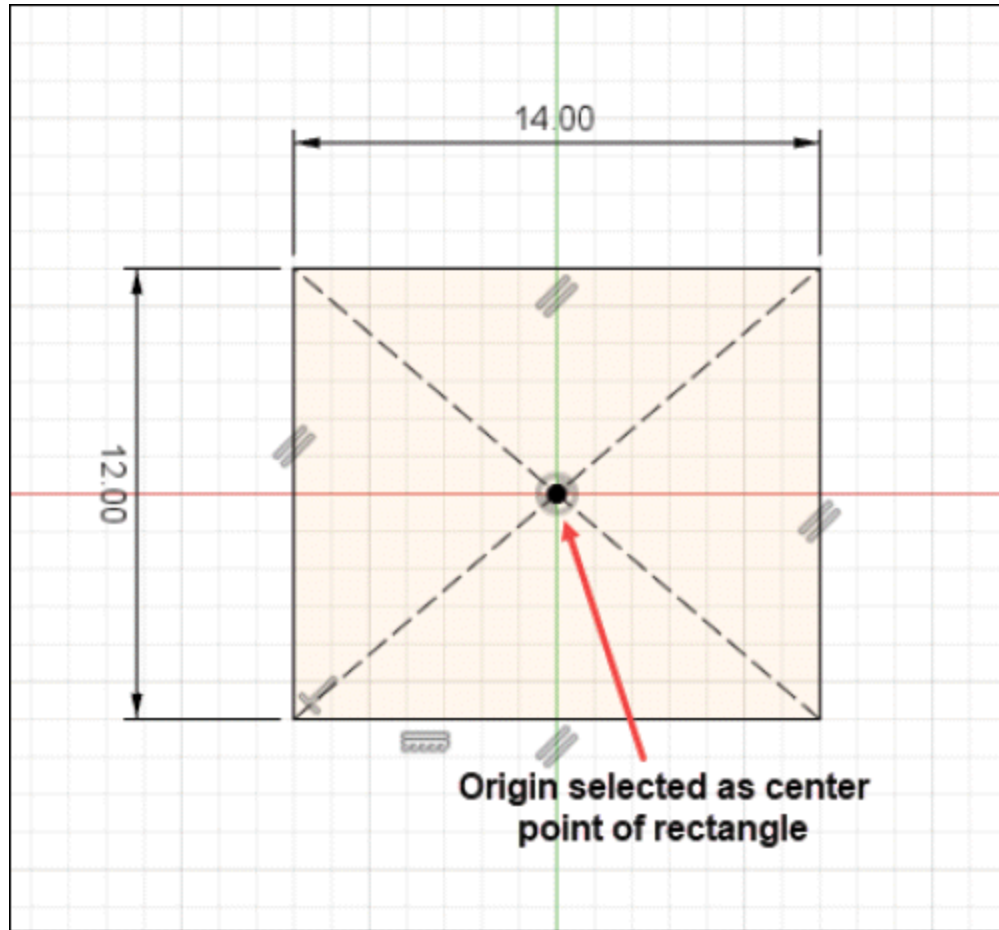


Figure-4. Rectangle created for 3D sketch

- Nope! Don't exit the sketching environment yet!! Select the whole sketch by window selection; refer to [Figure-5](#) and press **CTRL+C** from keyboard. This will keep a copy of current sketch in clipboard memory of software.
- Click on the **Finish Sketch** button from **FINISH SKETCH** panel of the **Toolbar** to exit sketching environment.

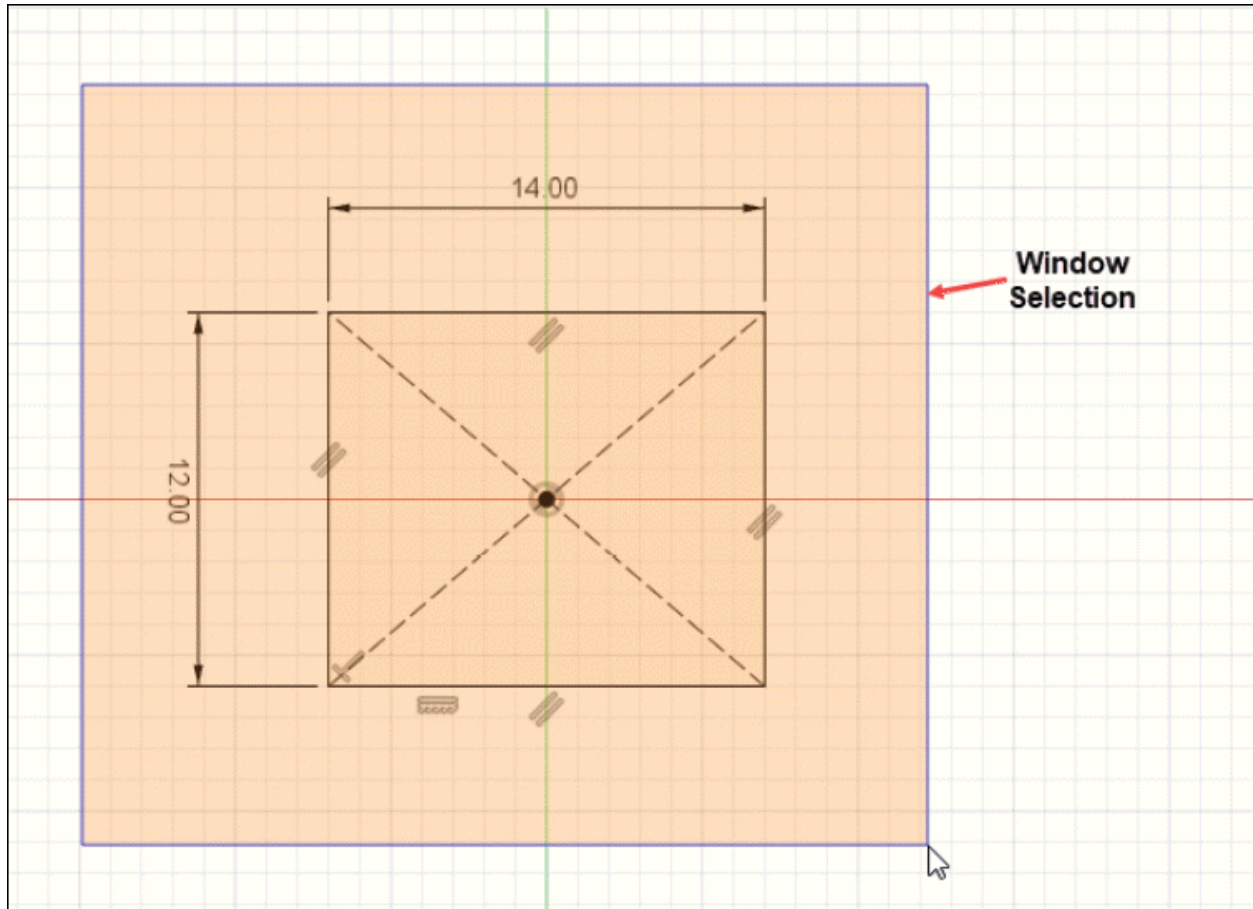


Figure-5. Window selection

- Click on the **Offset Plane** tool from the **CONSTRUCT** drop-down of the **SOLID** tab in the **Toolbar** and select **XY Plane** from **Origin** node of **BROWSER**; refer to [Figure-6](#). (You will learn more about the plane and construction geometries later in this chapter.)

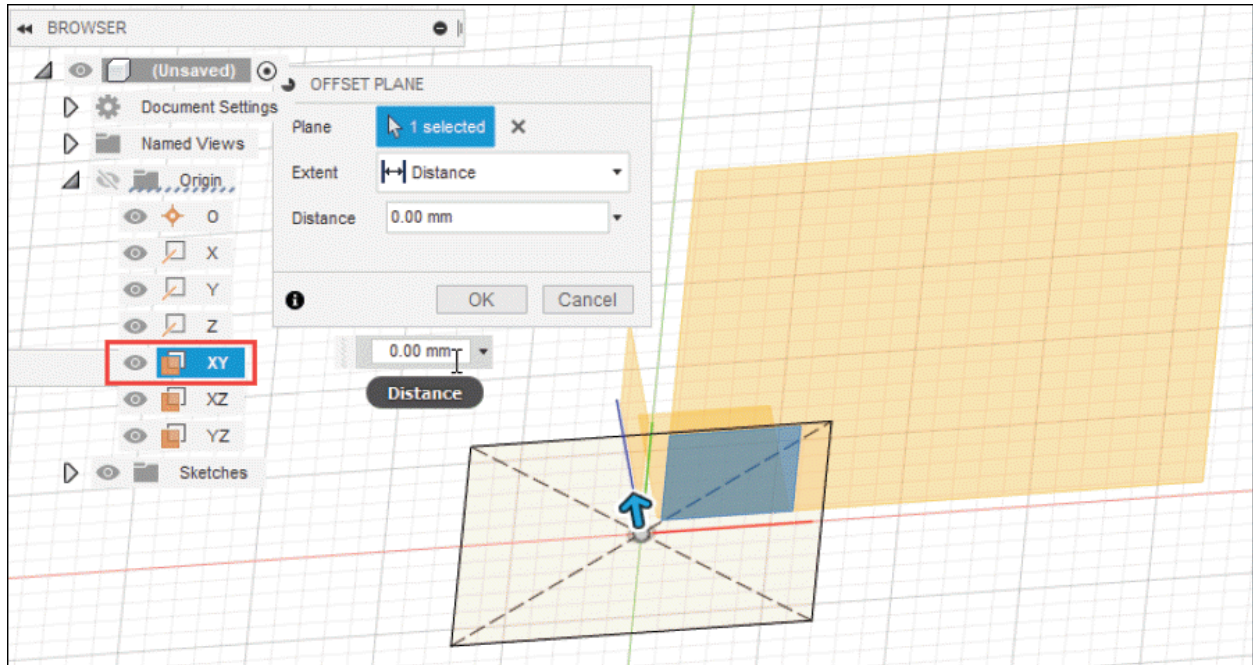


Figure-6. Creating offset plane

- Specify the distance value as **300** in **OFFSET PLANE** dialog box and click on the **OK** button. A new plane will be created at a distance of **300** from XY plane.
- Click again on the **Offset Plane** tool from the **CONSTRUCT** drop-down in the **Toolbar** and create another plane at an offset distance of **300** from earlier created plane; refer to [Figure-7](#).

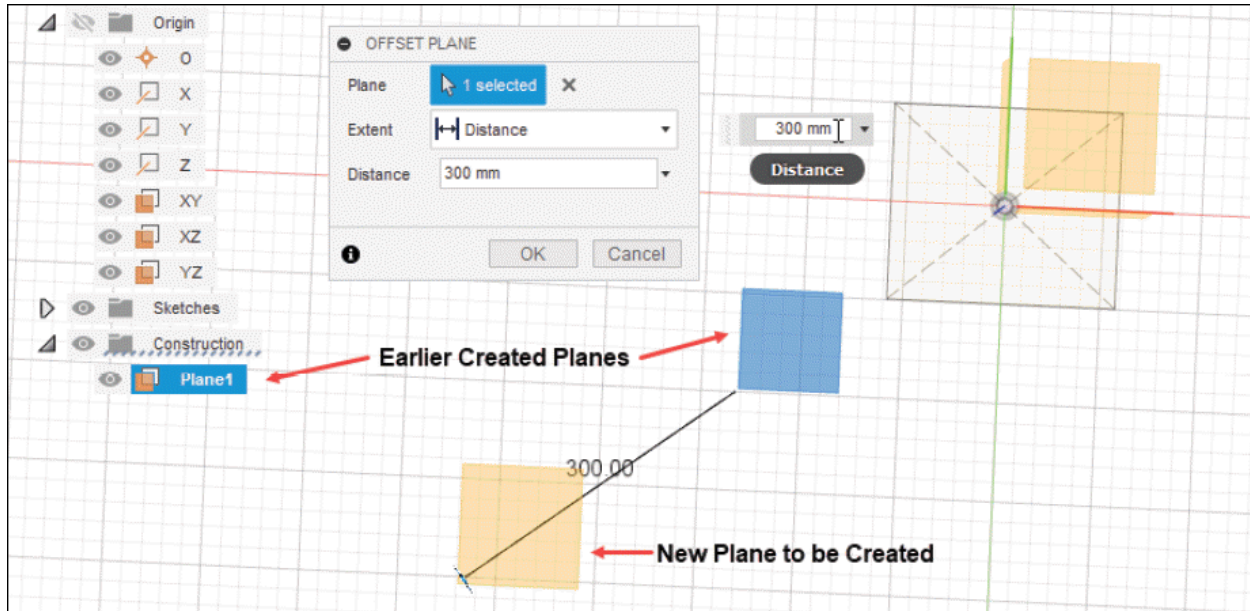


Figure-7. New plane to be created

- Select **Plane1** from the **Construction** node in **BROWSER** and click on the **Create Sketch** button from **CREATE** drop-down in the **Toolbar**. The sketching environment will be activated.
- Press **CTRL+V** from keyboard to paste earlier copied sketch. The **MOVE/COPY** dialog box will be displayed with sketch pasted; refer to [Figure-8](#).

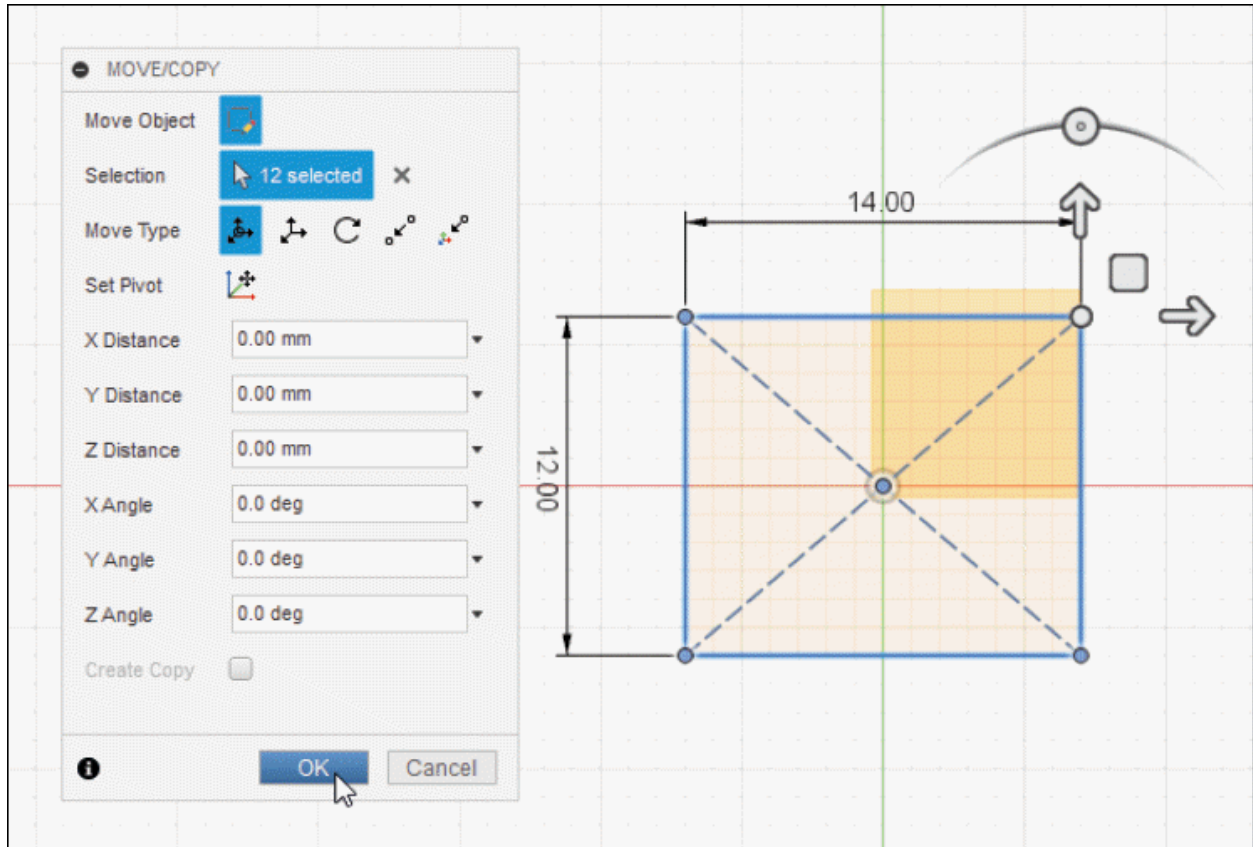


Figure-8. Sketch pasted with MOVE COPY dialog box

- There is no need to change any parameter because our sketch is well constrained and same as we wanted. Click on the **OK** button from the dialog box and exit the sketching environment.
- Create the same sketch on other offset plane. The sketches will be displayed as shown in [Figure-9](#).

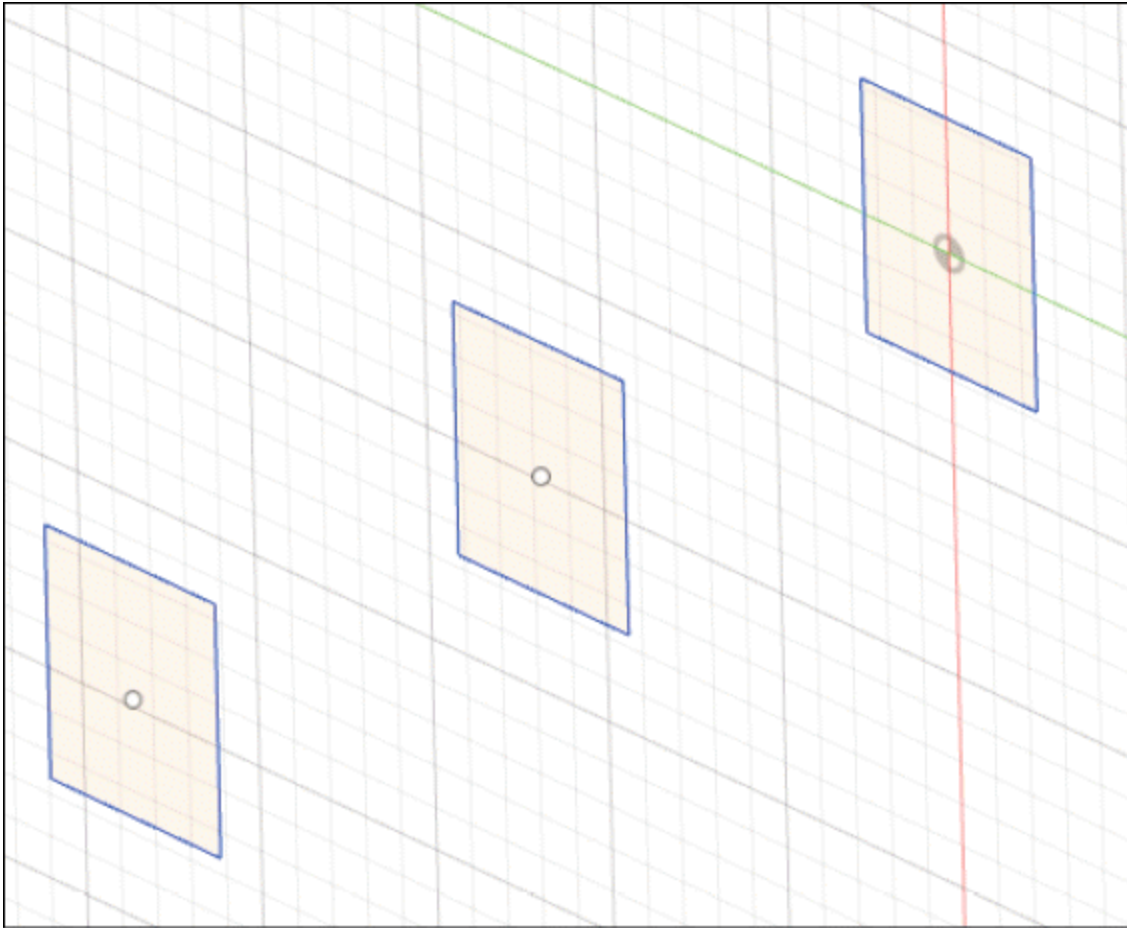


Figure-9. Sketches after copy paste

Creating 3D Lines

(Till the time, this book is authored, we are able to use only lines and splines for creating 3D sketches.)

- Click on the **Preferences** option from the **User Account** drop-down at the top-right corner of the application window. The **Preferences** dialog box will be displayed. (This dialog box has been discussed earlier in Chapter 1.)
- Select the **Design** option from **General** node at the left in the dialog box. The options will be displayed as shown in

Figure-10.

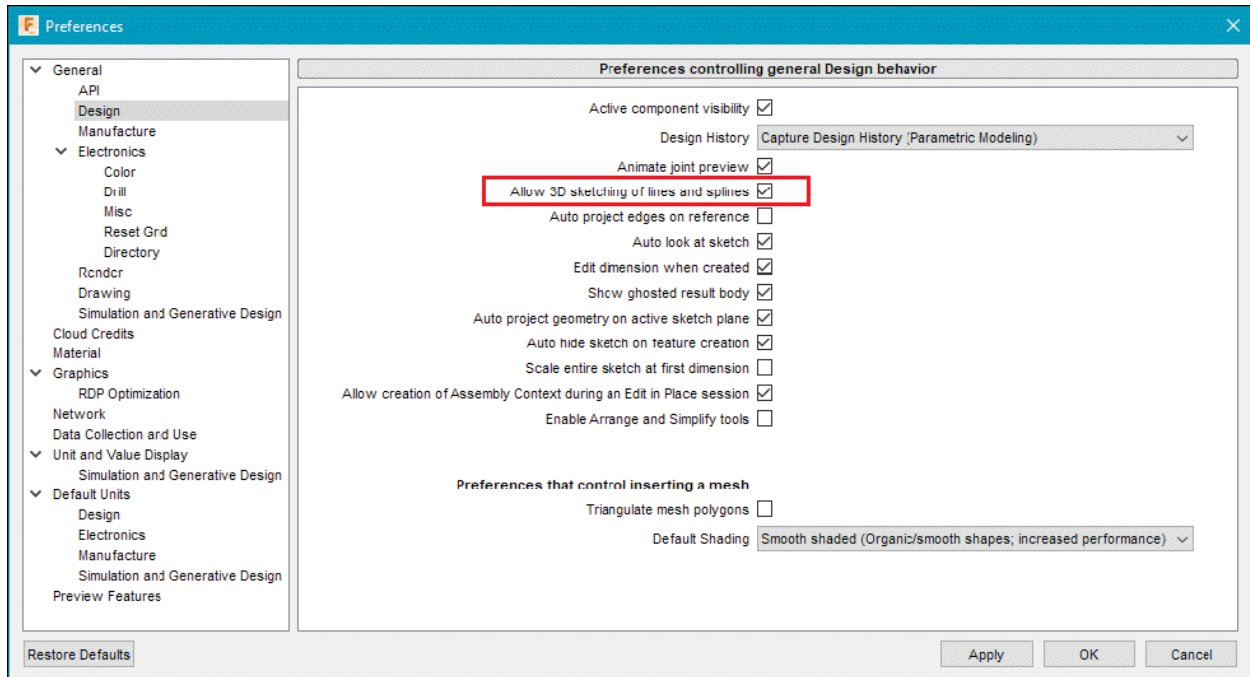


Figure-10. Preferences dialog box for 3D sketching

- Select the **Allow 3D sketching of lines and splines** check box as highlighted in above figure and click on the **OK** button from the dialog box.

Now, we are ready to create 3-dimensional lines.

- Click on the **Create Sketch** tool from the **CREATE** drop-down in the **Toolbar** and select any of the sketching plane. The sketching plane will become parallel to screen.
- Rotate the model by using **SHIFT+Middle Mouse Button (MMB)**.
- Click on the **Line** tool from **CREATE** drop-down and hover the cursor on earlier created sketches. You will find that the snap points of all the sketches get highlighted when you hover the cursor over them.

- Now what left is just clicks and clicks. Join various points of sketches to form frame sketch as shown in [Figure-11](#).

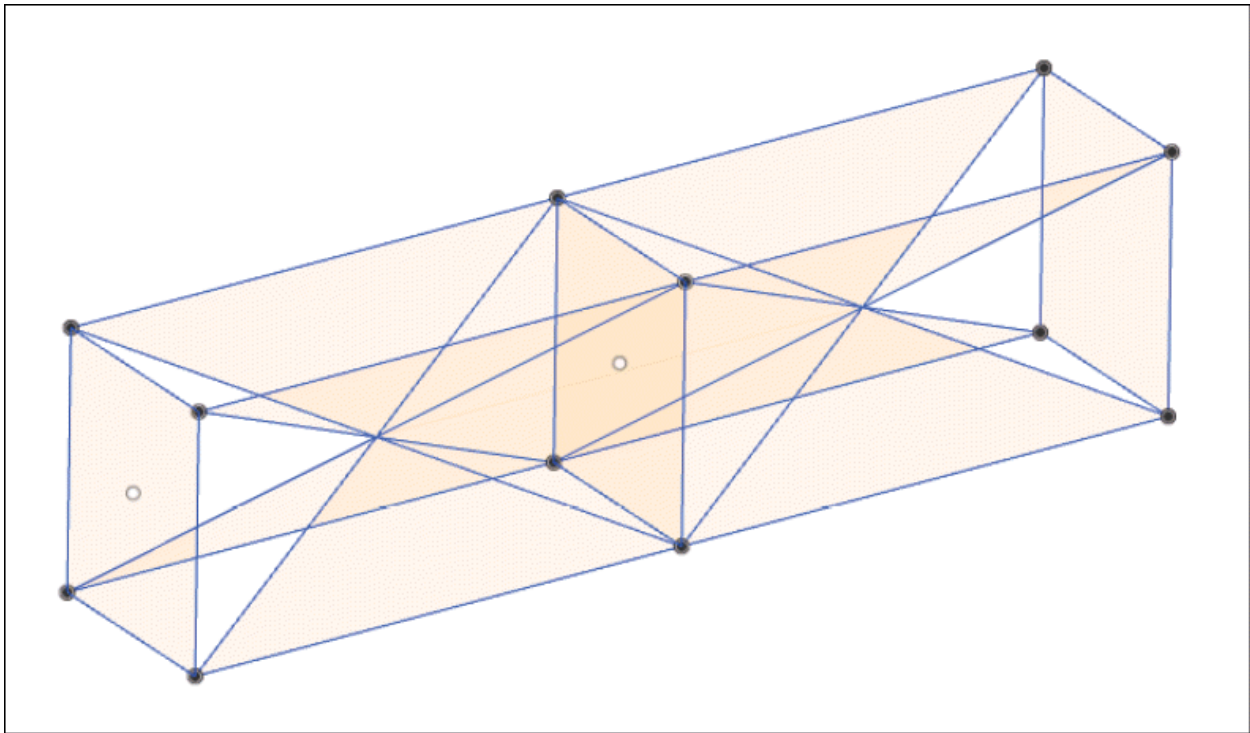


Figure-11. Sketch created for frame

- Click on the **Finish Sketch** tool from the **FINISH SKETCH** panel of the **Toolbar** to exit the sketching environment. So, it is the property of snap points for various elements that allow creating 3D sketch entities.

SELECTION TOOLS

You have earlier selected the objects by clicking on them or by making a window to select them. In Autodesk Fusion, sometimes you will reach in situations where only features are to be selected or only faces are to be selected. In such cases, we apply selection filters. You can also define the pattern in which you want to select objects using the options

in **SELECT** drop-down of the **Toolbar**; refer to [Figure-12](#). These tools are discussed next.

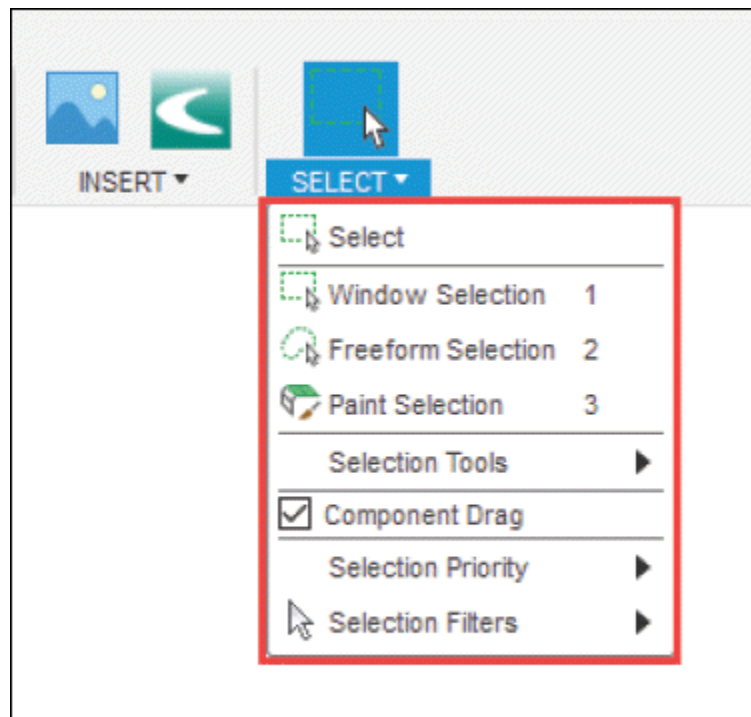


Figure-12. SELECT drop down

Select Tool

This tool is always active while working in Autodesk Fusion. Because of this tool, you are able to select any tool from toolbar or object from canvas.

Window Selection

This tool is active by default when you start working on Autodesk Fusion. This tool is used to activate window selection mode. You can activate this tool by selecting it from **SELECT** drop-down or by pressing **1** key from keyboard (Note that 1 is not to be pressed from **NUMPAD** on your keyboard). After activating window selection mode,

there are two ways in which you can create window to select objects; **Window selection** and **Cross Window selection** (sometimes called Cross selection). Click in the canvas and drag the cursor towards right to perform window selection. Release the button when you have formed window of desired size. Click in the canvas and drag the cursor towards left to perform cross window selection. Both the selection methods and their uses are given in [Figure-13](#) and [Figure-14](#).

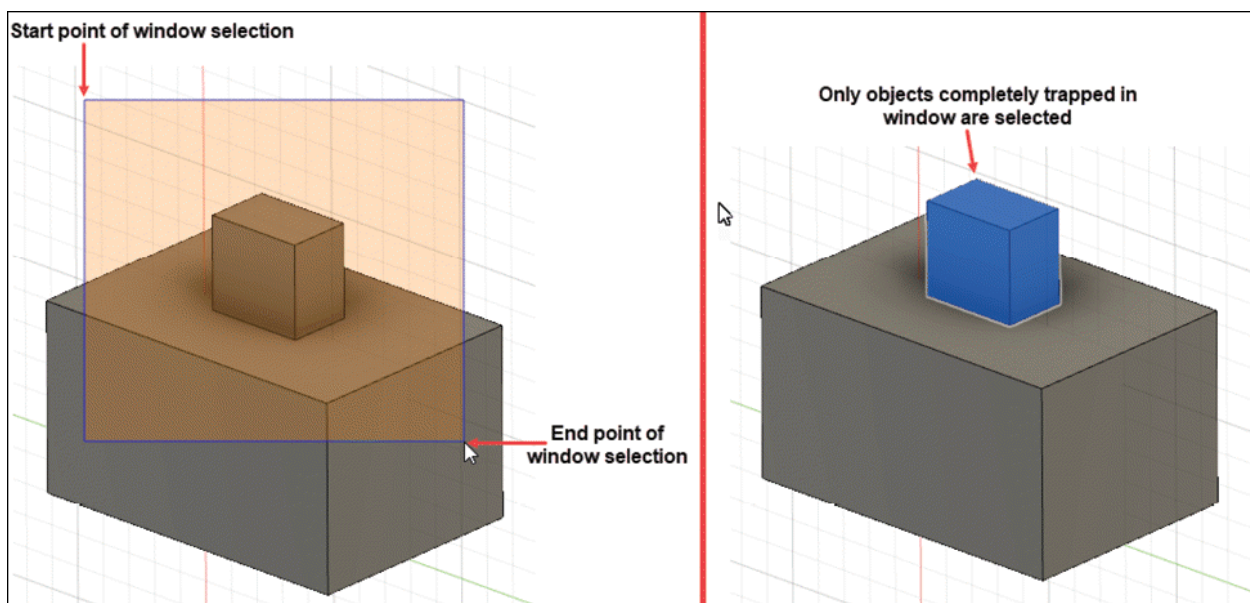


Figure-13. Window selection mode

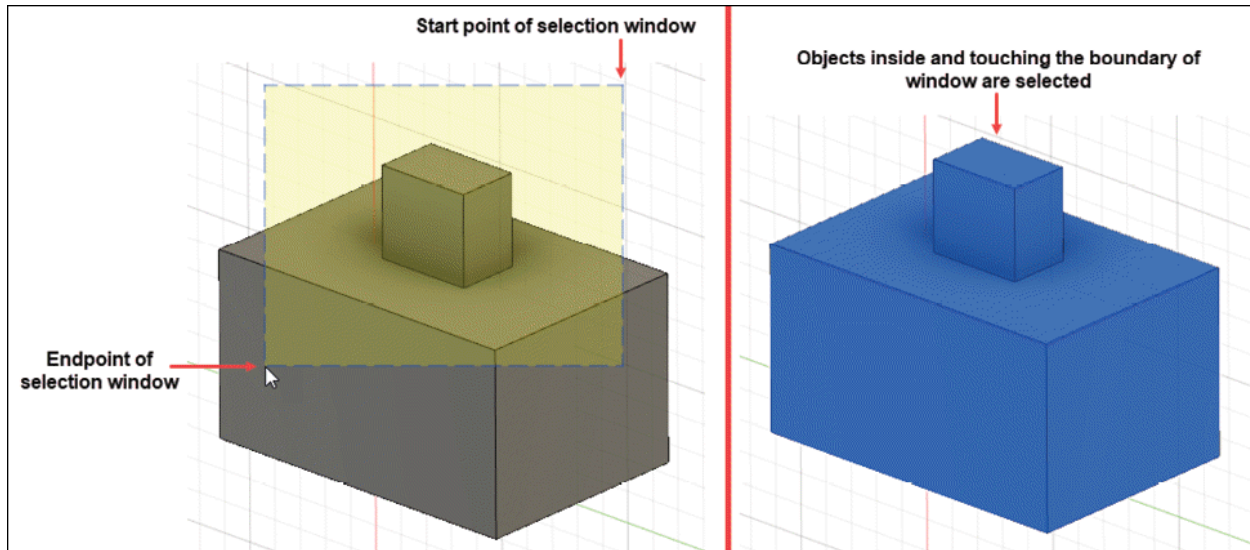


Figure-14. Cross window selection mode

Freeform Selection

Click on the **Freeform Selection** tool from **SELECT** drop-down or press **2** from keyboard to activate this selection mode. Click and drag the cursor to create a freeform boundary for selection. Dragging cursor towards right will make objects completely inside the boundaries selected and dragging cursor towards left will make objects inside or touching the boundaries selected.

Paint Selection

Click on the **Paint Selection** tool from **SELECT** drop-down or press **3** from keyboard to activate this selection mode. In this selection mode, all the objects over which cursor passes while dragging will be selected.

Selection Tools

The selection tools discussed earlier are conventional selection tools. You can also select the objects by their

names, types, and sizes. These tools are available in the **Selection Tools** cascading menu of **SELECT** drop-down; refer to [Figure-15](#).

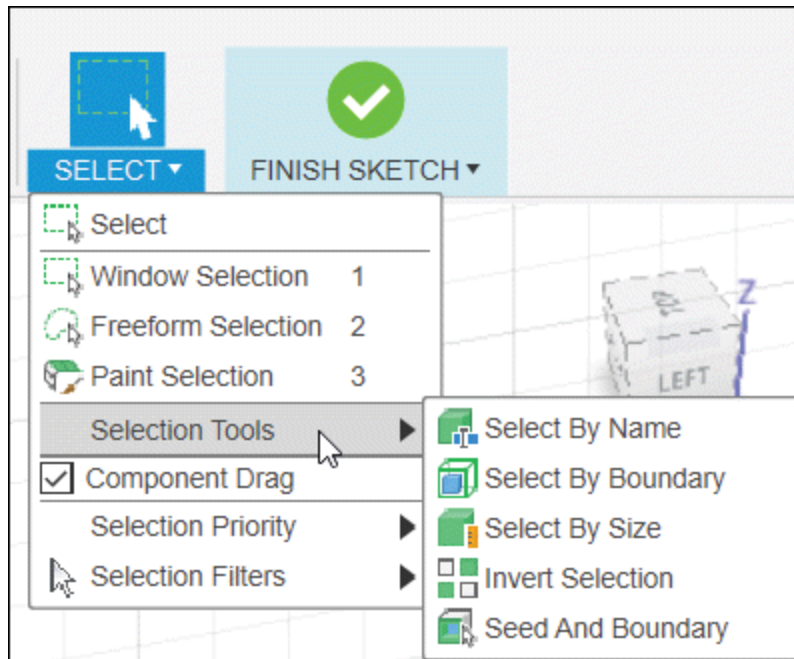


Figure-15. Selection Tools cascading menu

Select By Name Method

- Click on the **Select By Name** tool from the **Selection Tools** cascading menu of **SELECT** drop-down in the **Toolbar**. The **SELECT BY NAME** dialog box will be displayed; refer to [Figure-16](#).

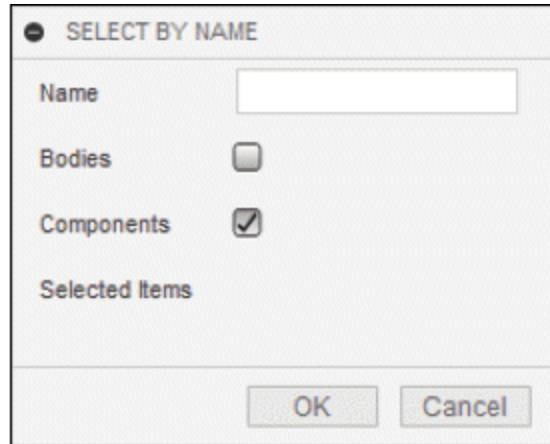


Figure-16. SELECT BY NAME dialog Box

- Type the name of object to be selected in the **Name** edit box of the dialog box.
- Select the **Bodies** check box to allow selection of bodies and select the **Components** check box to allow selection of components.
- Click on the **Find** button in the dialog box. All the components and bodies matching the specified name will be selected; refer to [Figure-17](#).

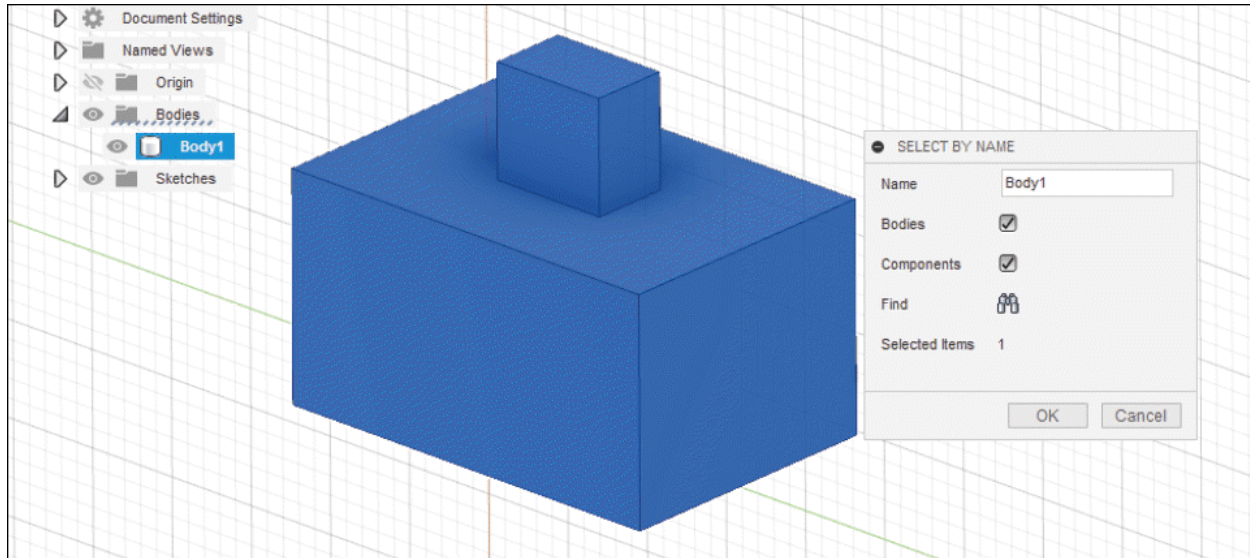


Figure-17. Bodies selected by name

- Click on the **OK** button from the dialog box to complete the process.

Select By Boundary Method

The **Select By Boundary** method is used to select objects which fall under defined shapes of cylinder, box, or sphere. The procedure to use this method is given next.

- Click on the **Select By Boundary** tool from the **Selection Tools** cascading menu of **SELECT** drop-down. The **SELECT BY BOUNDARY** dialog box will be displayed; refer to [Figure-18](#).

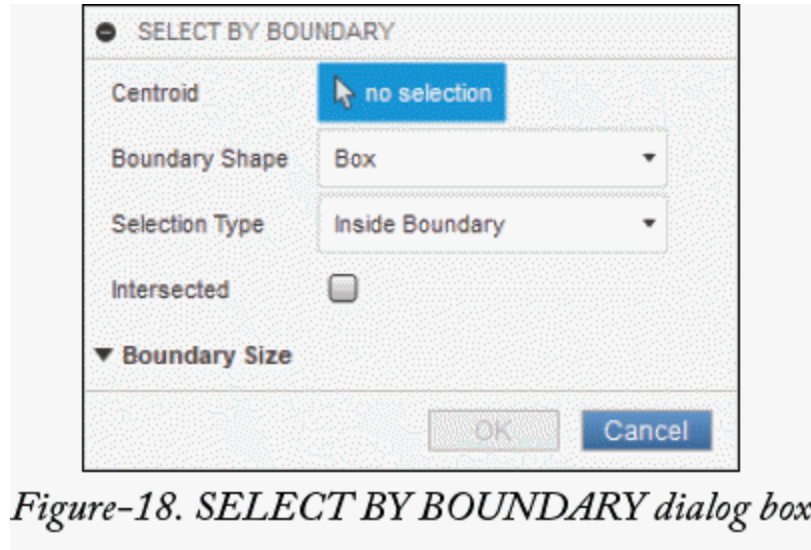


Figure-18. SELECT BY BOUNDARY dialog box

- Select desired option from the **Boundary Shape** drop-down to define the shape of selection boundary.
- Click on desired point in the model to place centroid of boundary shape. Preview of boundary will be displayed; refer to [Figure-19](#).

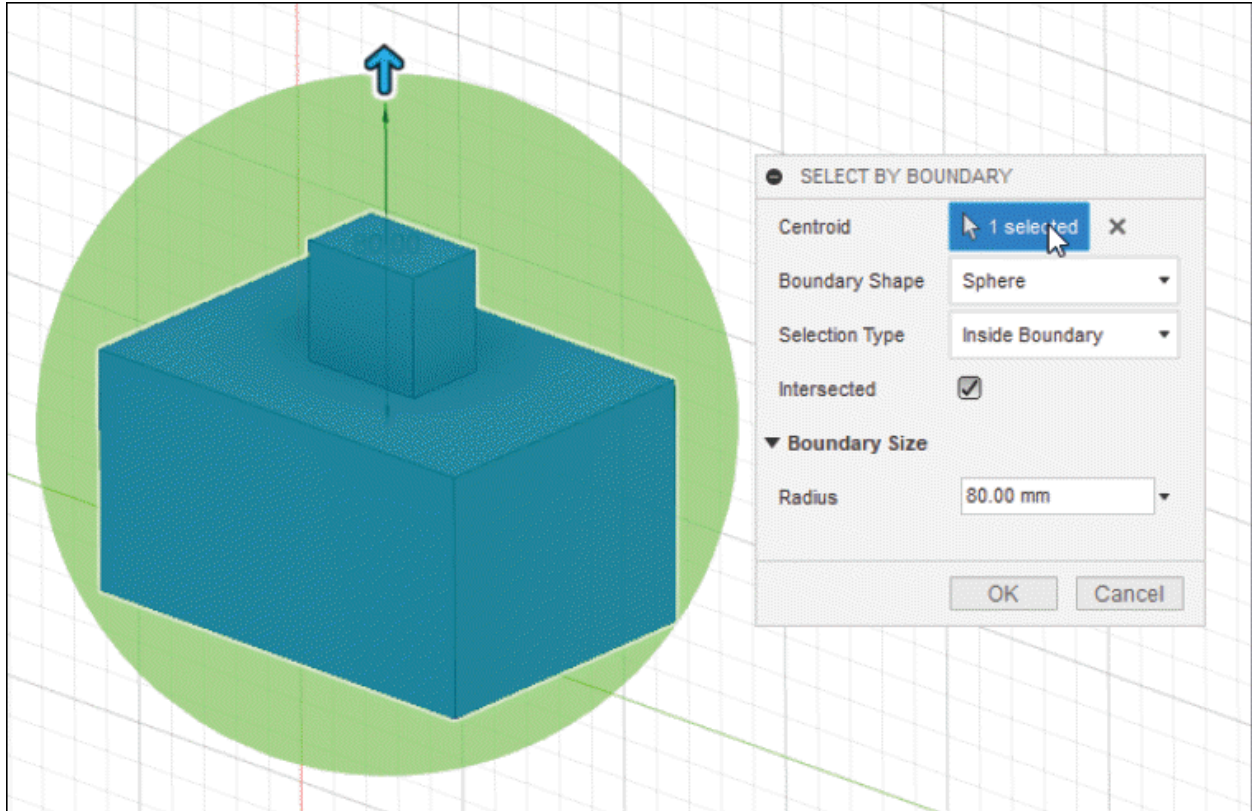


Figure-19. Sphere boundary preview

- Specify desired parameters in the **Boundary Size** section to define size of boundary shape.
- Select the **Inside Boundary** option from the **Selection Type** drop-down if you want to select the objects inside the boundary. Select the **Outside Boundary** option from the **Selection Type** drop-down if you want to select the objects outside the boundary shape.
- Select the **Intersected** check box if you want to select all the objects that are intersecting with created boundary shape.
- Click on the **OK** button from the dialog box to make selection.

Select By Size Method

The **Select By Size** method is used when you want to select the entities based on their sizes. The procedure for using this method is given next.

- Click on the **Select By Size** tool from the **Selection Tools** cascading menu of the **SELECT** drop-down in **Toolbar**. The **SELECT BY SIZE** dialog box will be displayed; refer to [Figure-20](#).

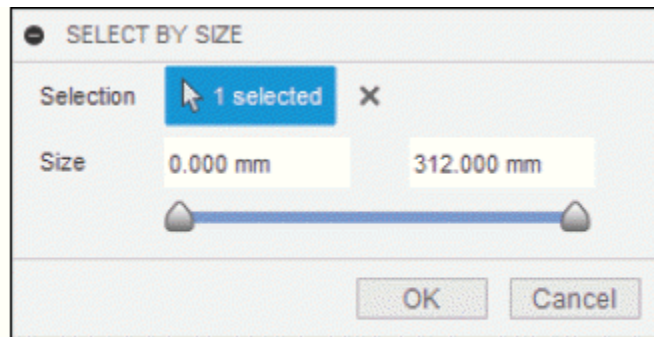


Figure-20. SELECT BY SIZE dialog box

- Specify the range of size within which entities will be selected and then click on the **OK** button.

Selection by Seed and Boundary

The **Seed and Boundary** tool is used to select faces on a solid or surface by using a seed surface/face as template for selection and outlines of a face/surface as boundary. The procedure to use this tool is given next.

- Click on the **Seed and Boundary** tool from the **Selection Tools** cascading menu of the **SELECT** drop-down in the **Toolbar**. The **SEED AND BOUNDARY** dialog box will be

displayed; refer to [Figure-21](#).

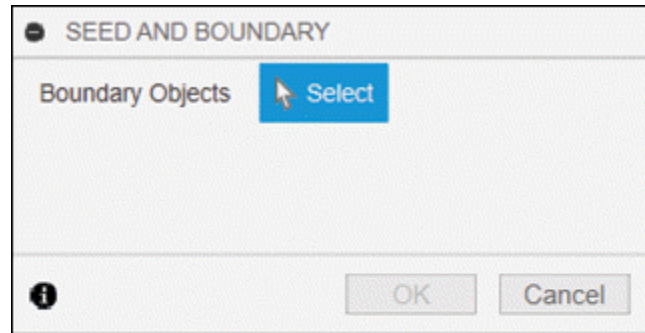


Figure-21. SEED AND BOUNDARY dialog box

- Select the face of model to be used as boundary for selection. (No objects outside this boundary will be selected by the tool). On selecting face for boundary, the **SEED AND BOUNDARY** dialog box will be updated and selection button for seed object will be displayed; refer to [Figure-22](#).

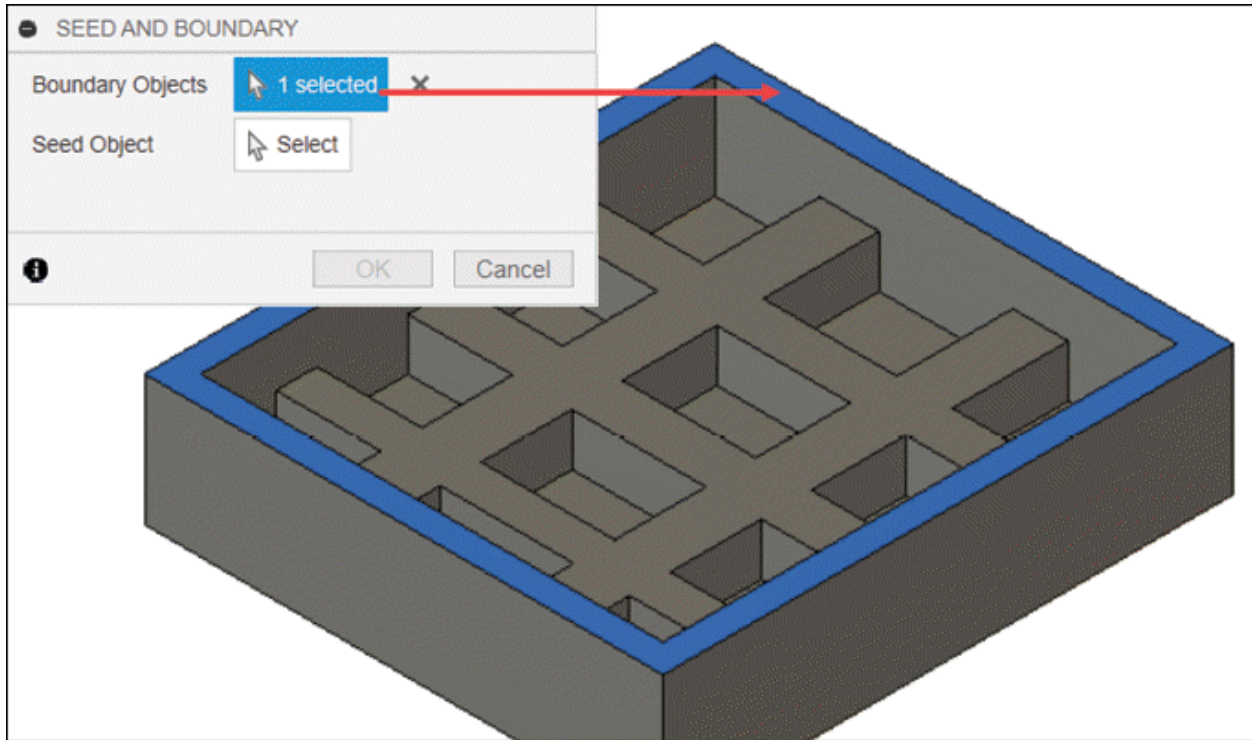


Figure-22. Boundary face selected

- Click on the **Select** button for **Seed Object** in dialog box and then select the face from model within the boundary to be used as reference for selecting nearby faces. The **Update** button will be displayed in the dialog box; refer to [Figure-23](#).

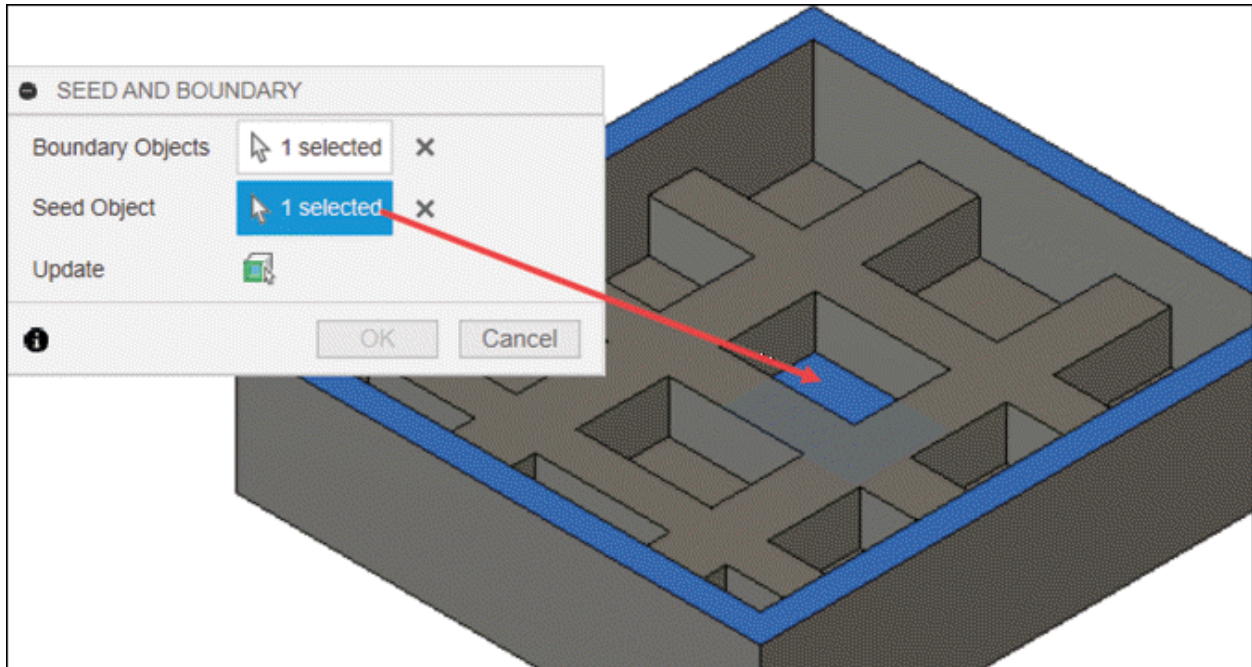


Figure-23. Face selected for seed object

- Click on the **Update** button to update the selection based on seed object. All the nearby faces within boundary will get selected; refer to [Figure-24](#).

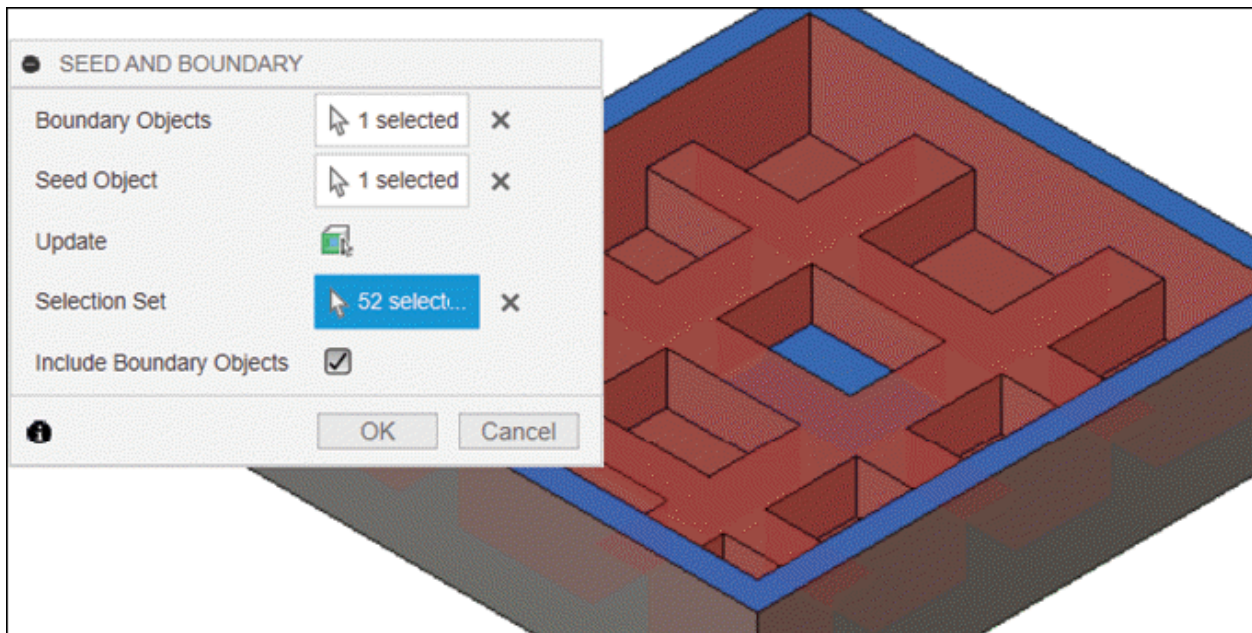


Figure-24. After updating selection

- Clear the **Include Boundary Objects** check box if you do not want to boundary faces in selection.
- Click on the **OK** button to complete selection.

Component Drag

Select the **Component Drag** check box to enable the movement of components by dragging in the canvas. Hold the **Alt** key to move the components by dragging in the canvas.

Selection Priority

The options in **Selection Priority** cascading menu of **SELECT** drop-down in **Toolbar** are used to define which entities will be given preference while selecting objects; refer to [Figure-25](#). Select the **Select Body Priority** option if you want to select bodies in place of faces/edges. Similarly, select other options in the cascading menu to select respective entities.

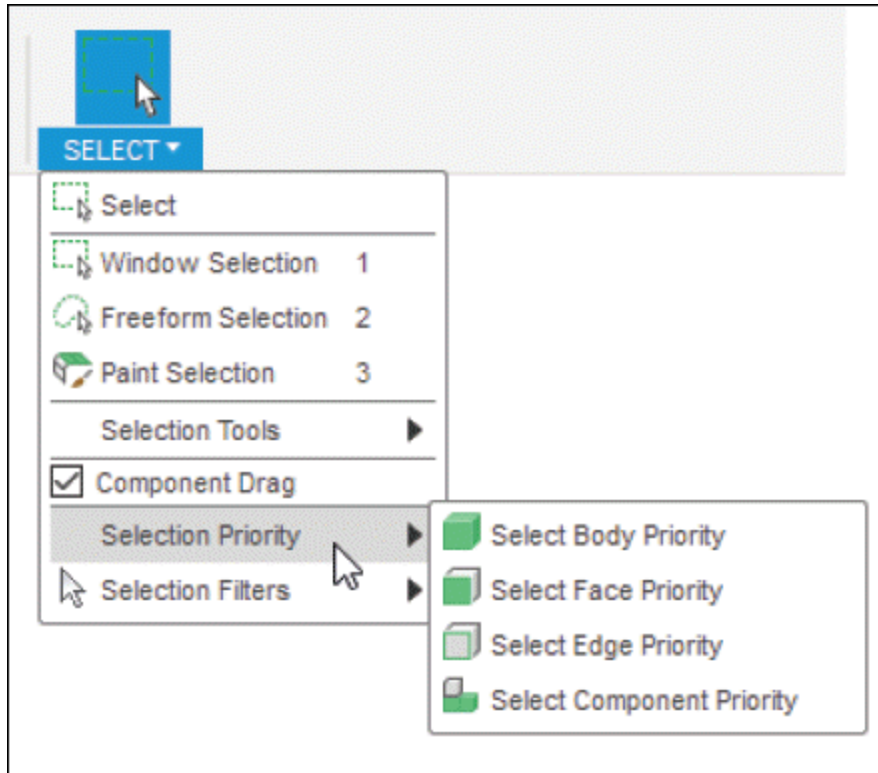


Figure-25. Selection Priority cascading menu

Selection Filters

The check boxes in **Selection Filters** cascading menu are used to define which objects can be selected and which objects cannot be selected; refer to [Figure-26](#). Select desired check boxes to activate selections of respective objects. In default condition, the **Select All** check box is selected.

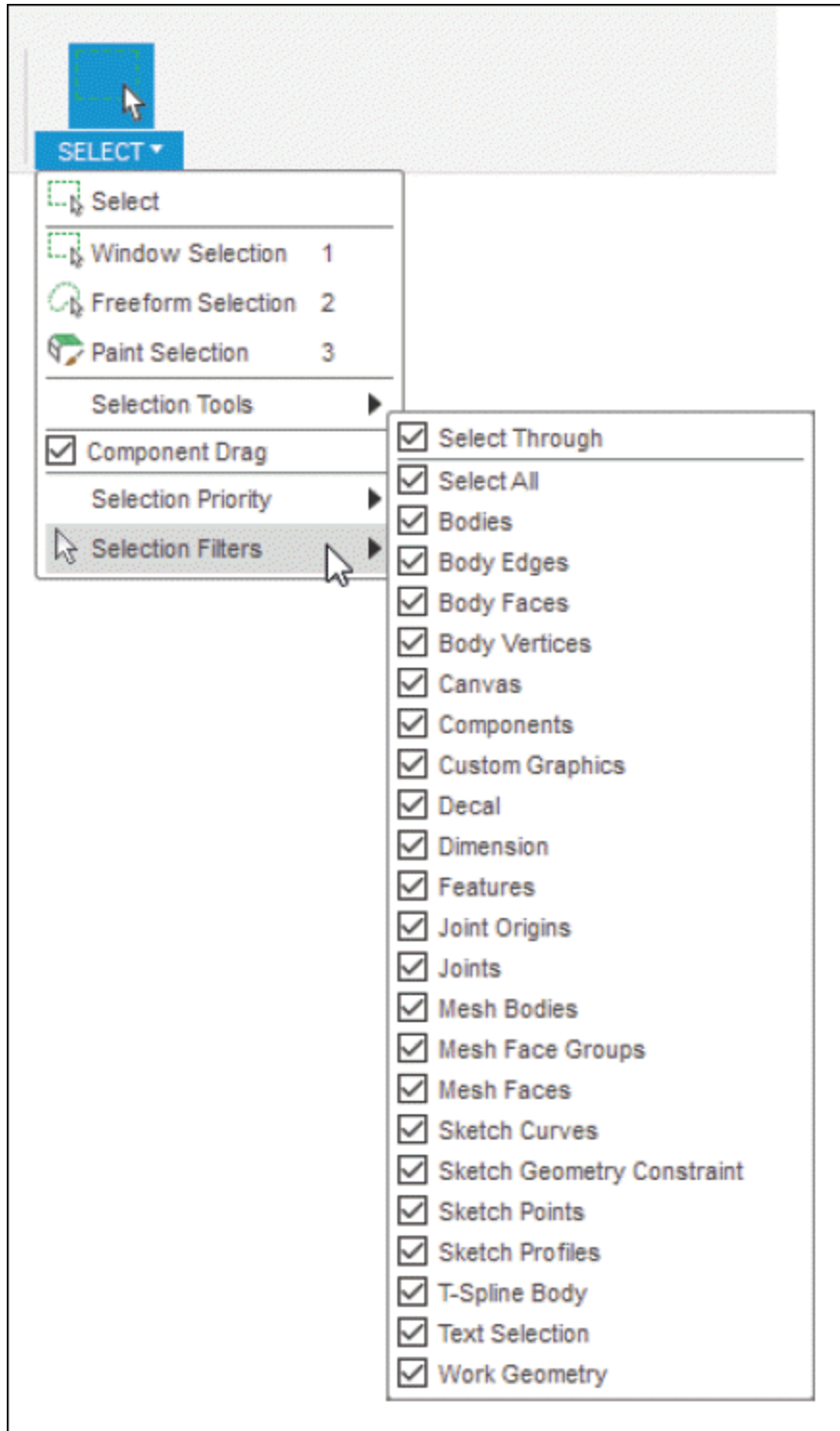


Figure-26. Selection Filters cascading menu

Till this point, we have worked on 2D sketches and have created them on Top or Front plane. We have also created 3D sketches. Now, we will learn the tools for solid modeling. **Solid modeling** is the most advanced method of geometric modeling in three dimensions. Solid modeling is the representation of the solid objects in your computer in digital form. Various tools to perform solid modeling are available in the **SOLID** tab of **Toolbar**. We will learn about various tools and commands used to create solid models in next topic.

CREATE DROP-DOWN

In **Create** drop-down, there are various 3D modeling tools used to create and manage solid objects; refer to [Figure-27](#). These tools are discussed next.

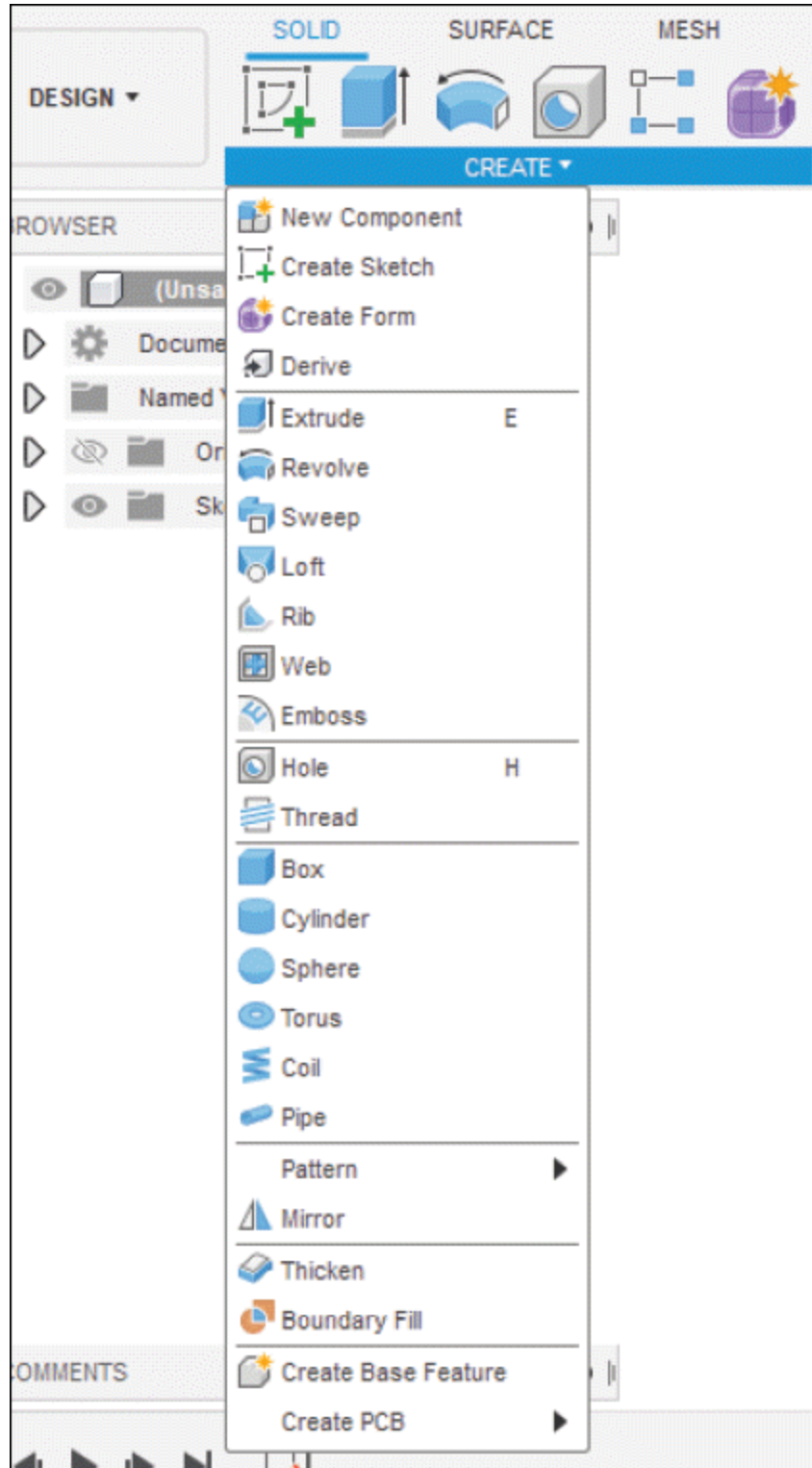


Figure-27. CREATE drop down

New Component

Click on **New Component** tool from **Create** drop-down; refer to [Figure-28](#). A new component will start. You will find use of this tool later in Assembly chapter of the book. (No! we are not skipping this tool, we will discuss it later :-)

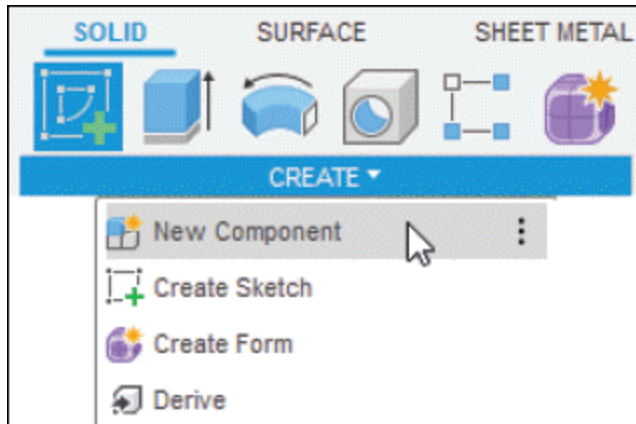


Figure-28. New Component

Extrude

The **Extrude** tool is used to create a solid volume by adding height to the selected sketch. In other words, this tool adds material in the direction perpendicular to the plane of sketch by using the boundaries of sketch. The procedure to use this tool is discussed next.

- Create a closed profile sketch as discussed in previous chapter.
- After creating sketch, exit the sketch environment by selecting the **Finish Sketch** tool from **Toolbar** or **Right-click** outside the sketch and select **Finish Sketch** button from the **Sketch** drop-down in **Marking menu**; refer to [Figure-29](#).

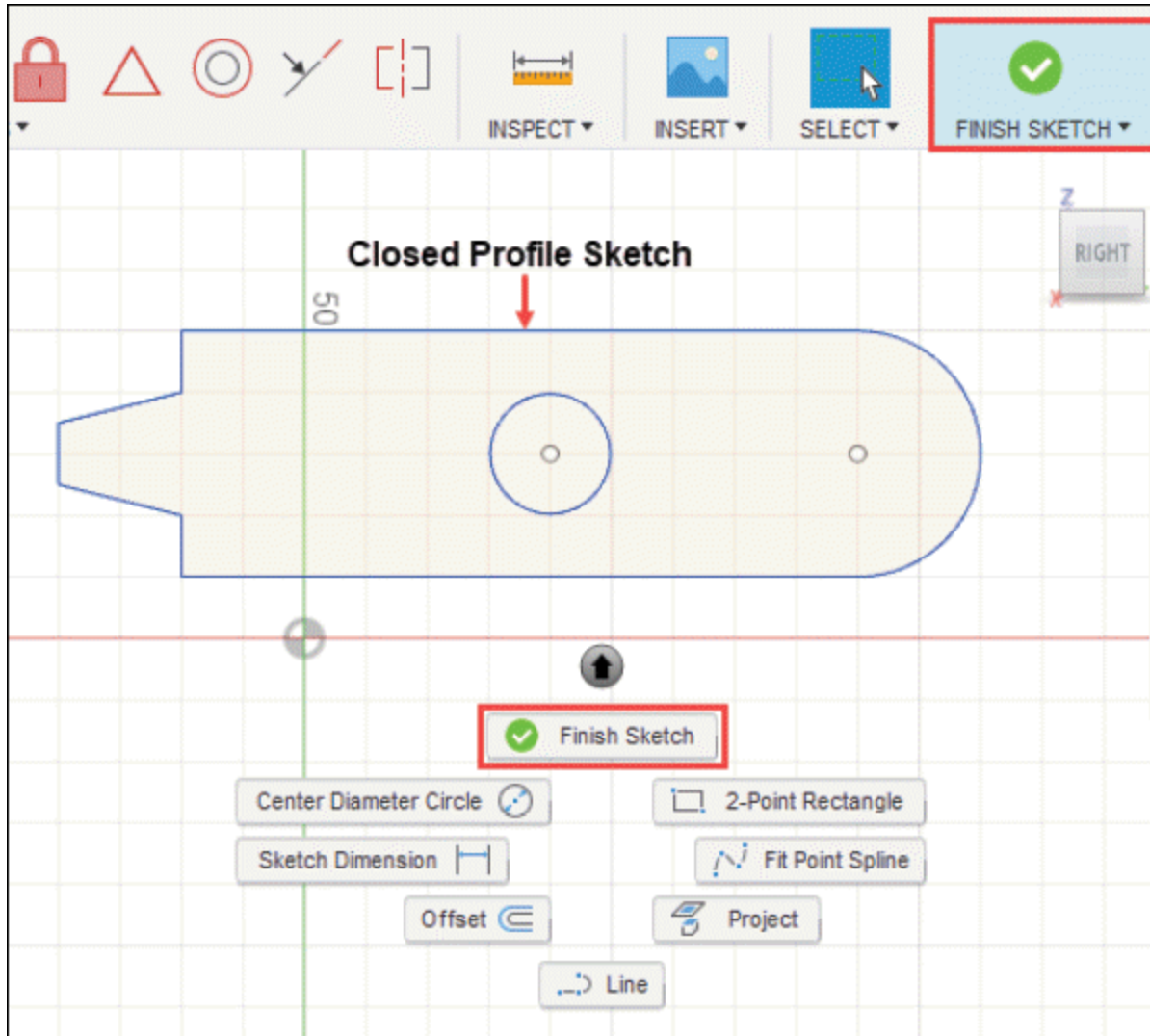



Figure-29. Sketch for extrusion

- Click on **Extrude** tool of **Create** drop-down from **Toolbar**; refer to [Figure-30](#). The **EXTRUDE** dialog box will be displayed; refer to [Figure-31](#).
- Click on the **Presets** drop-down from top of the dialog box and select desired preset to apply or save the current values as a preset to use them. Click on the **New Preset**  button to create new preset for the dialog box.

- Select the type of extrusion you want to create from **Type** section of the dialog box. Select **Extrude** button to extrude entire area within selected closed profiles as solid. Select **Thin Extrude** button to extrude a thin wall along selected closed profiles.

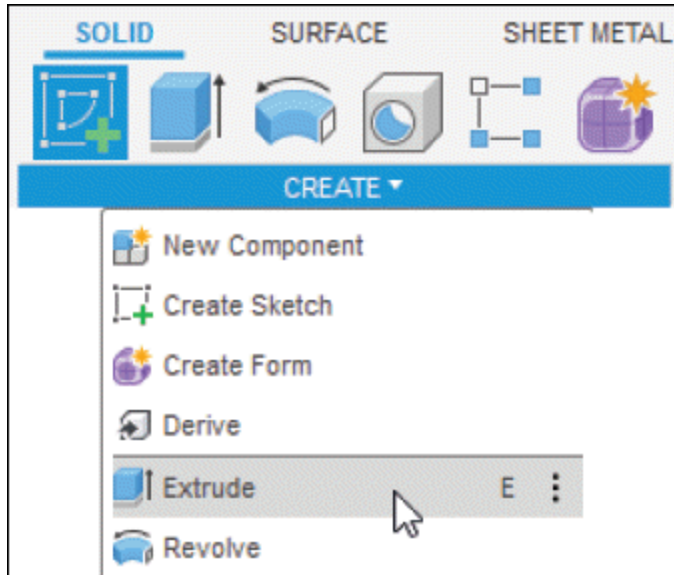


Figure-30. Extrude tool

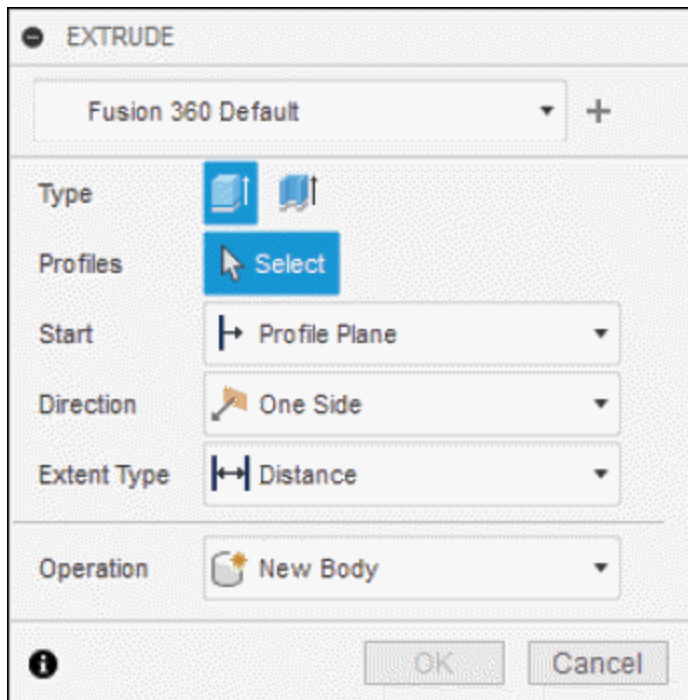


Figure-31. EXTRUDE dialog box

- The **Extrude** option is selected by default in the **Type** section of the dialog box and the profile selection is active by default. Select the sketch to be extruded by clicking on the shaded area in the sketch. If your sketch has multiple closed profiles then make sure to select correct shaded area. For example, in [Figure-29](#), we can select inside the circle, outside the circle, or both to create extrusion. The extruded body will be of the shape selected here.

Start drop-down

- There are three tools in **Start** drop-down; refer to [Figure-32](#).

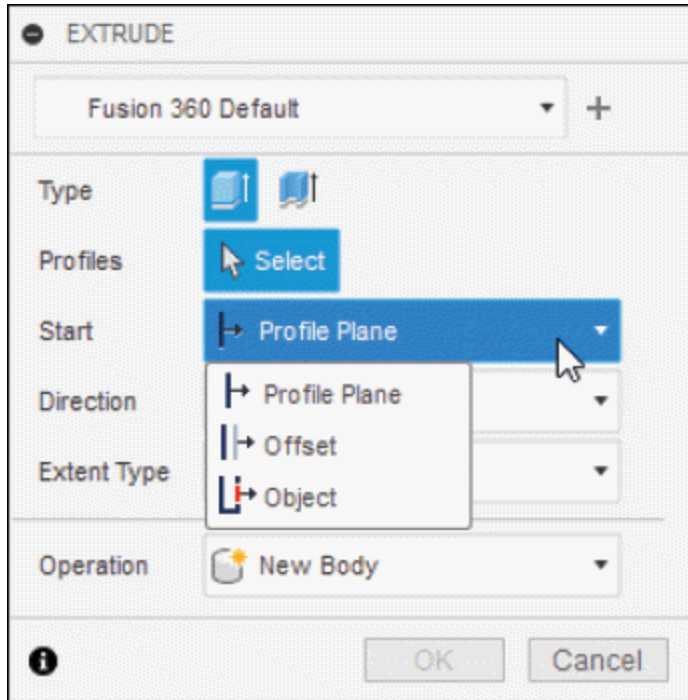


Figure-32. Start drop down

- Select **Profile Plane** option from **Start** drop-down, if you want to extrude the sketch starting from current sketching plane; refer to [Figure-33](#).
- Select **Offset** option if you want to extrude the sketch starting from some offset distance. The updated **EXTRUDE** dialog box will be displayed; refer to [Figure-34](#).

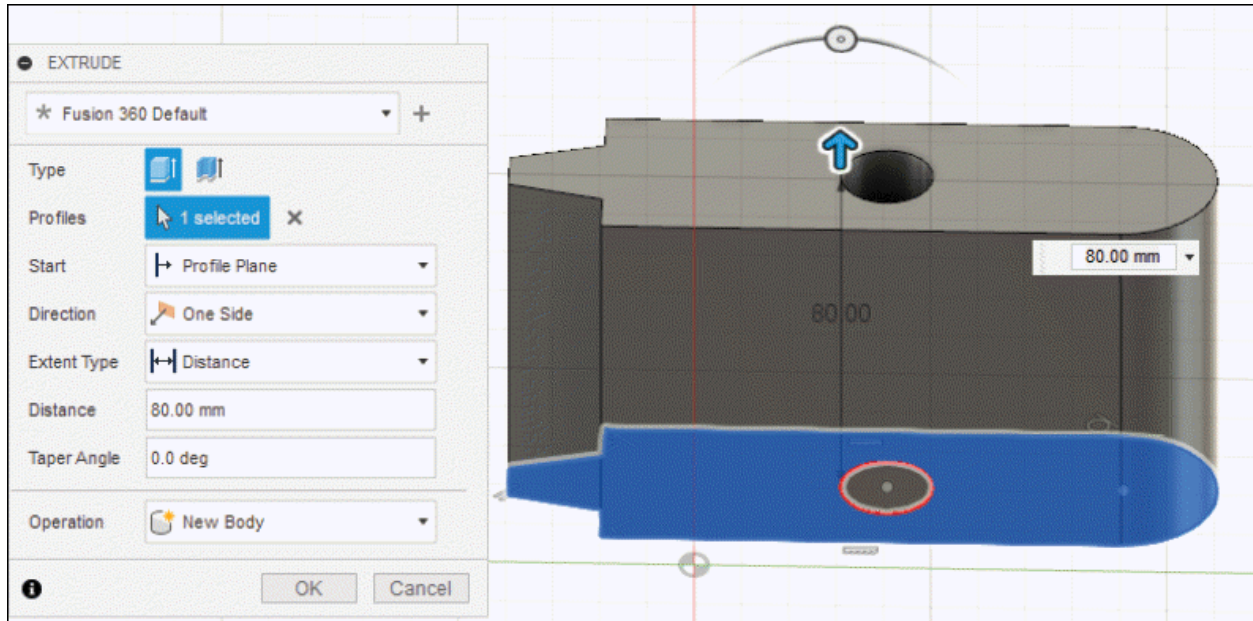


Figure-33. Extrude starting from profile plane

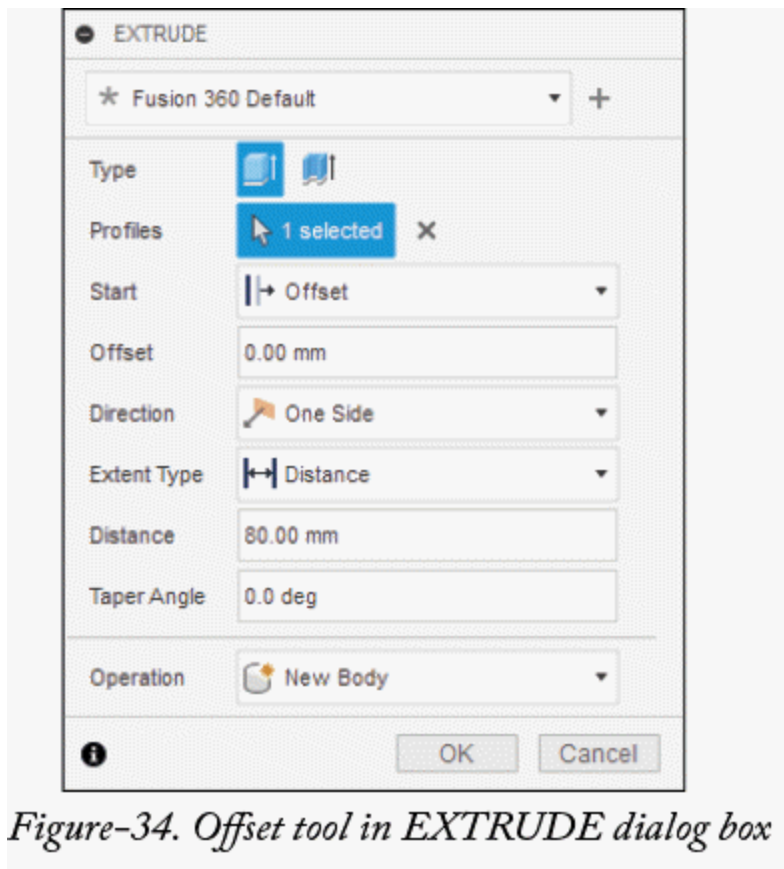


Figure-34. Offset tool in EXTRUDE dialog box

- Click in **Offset** edit box and type desired value of distance

between starting of extrusion and sketch plane; refer to [Figure-35](#).

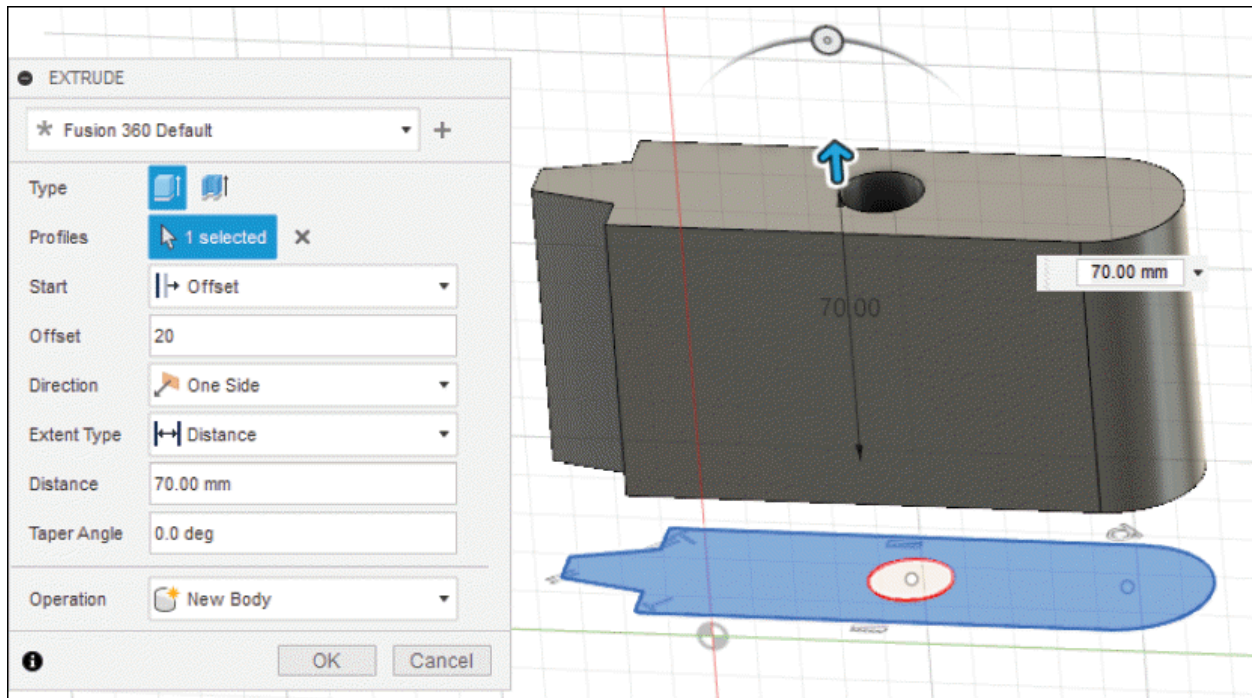


Figure-35. Extruding with offset

- Select the **Object** tool from **Start** drop-down in the dialog box, if you want to extrude the profile starting from selected surface/face (or at a distance from selected face) of another body.
- After selecting **Object** tool, click on **Select** button of **Object** section and select the object from where you are going to start the extrude; refer to [Figure-36](#).

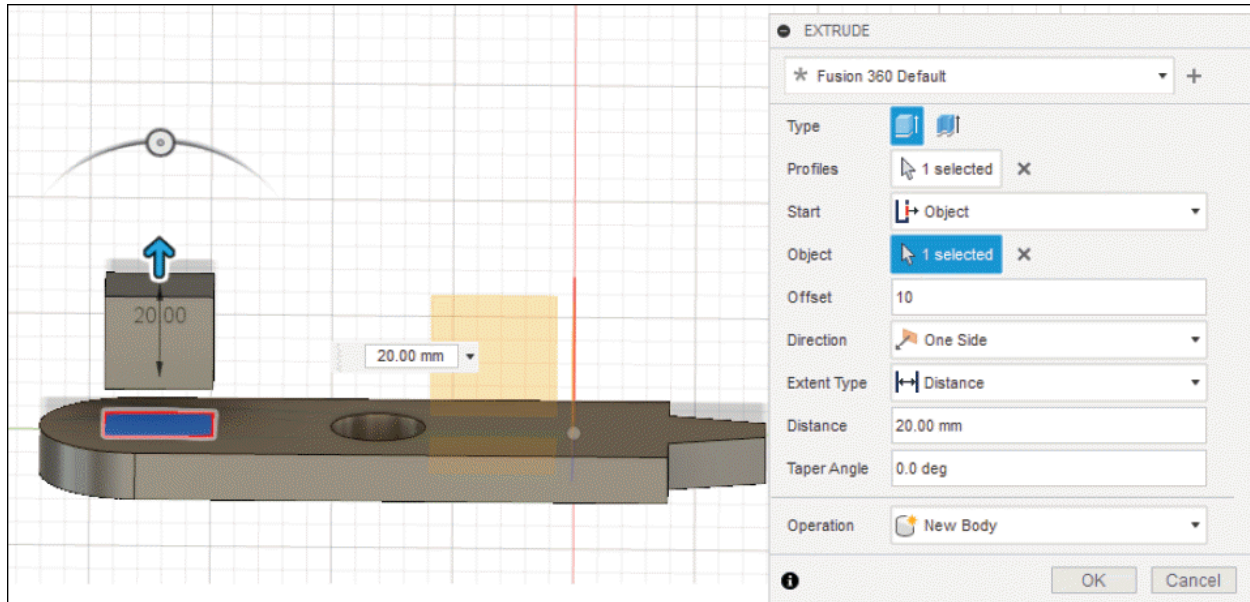


Figure-36. Object tool in EXTRUDE dialog box

- Click in **Distance** edit box and specify the height of extrusion. You can also drag the Arrow handle displayed on the sketch upward to define extrude height.
- If you want to create tapered walls of extrude feature then specify desired angle value in the **Taper Angle** edit box. You can specify both positive and negative values in this edit box. You can also use the rotation handle displayed in the preview to specify taper value. Applying taper angle is also called draft in some software.

Direction drop-down

- There are three tools in **Direction** drop-down of **EXTRUDE** dialog box; refer to [Figure-37](#).

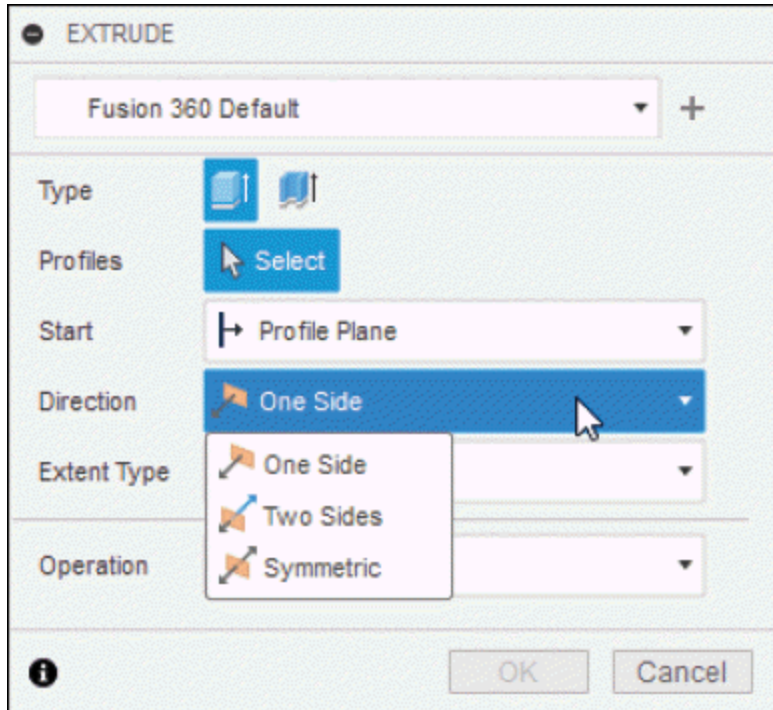


Figure-37. Direction drop down

- Select **One Side** option from **Direction** drop-down if you want to specify the height of extrusion in one direction.
- Select **Two Sides** option from **Direction** drop-down if you want to specify the height of extrusion in both upward and downward directions of sketching plane; refer to [Figure-38](#). The options to specify parameters for side 1 and side 2 will be displayed in the dialog box.

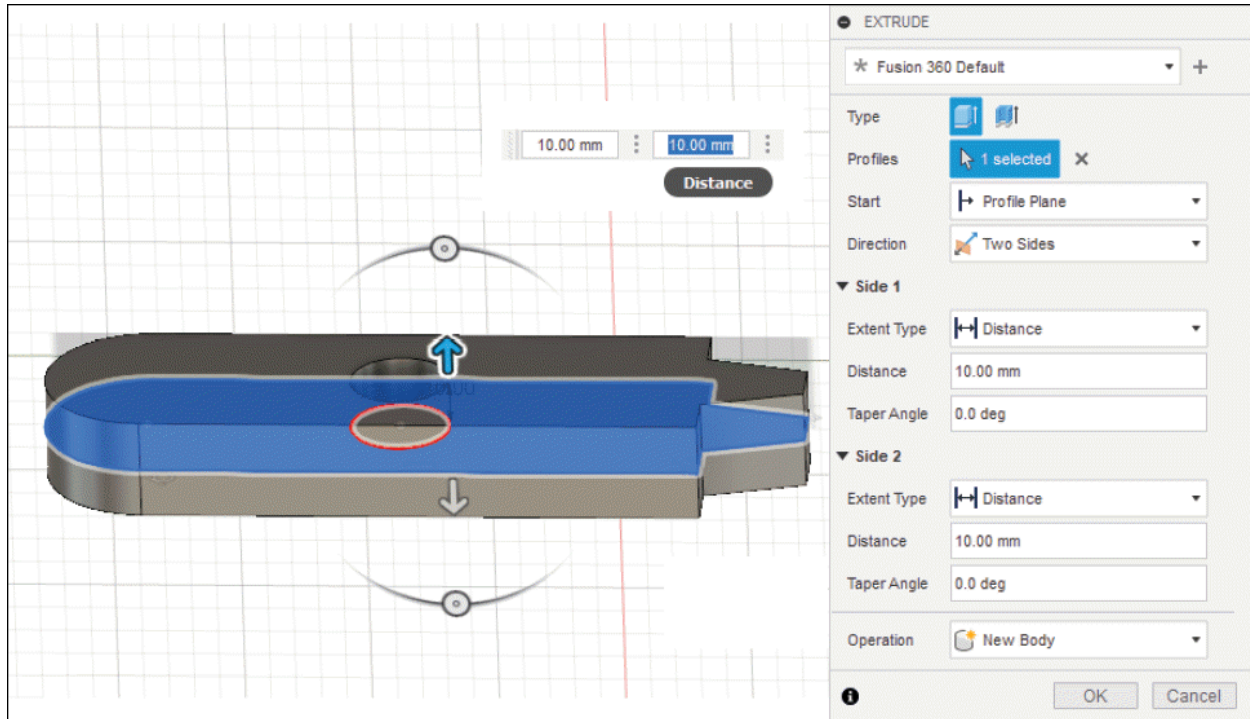


Figure-38. Extrude with two sides

- Select **Symmetric** option from **Direction** drop-down if you want to specify the height of extrusion symmetric in both upward and downward directions of sketching plane. You will be asked to specify height of extrusion for one side and the profile will be extruded symmetrically on the other side with same value. Select the **Half length** or **Whole length** button from **Measurement** section as required; refer to [Figure-39](#).

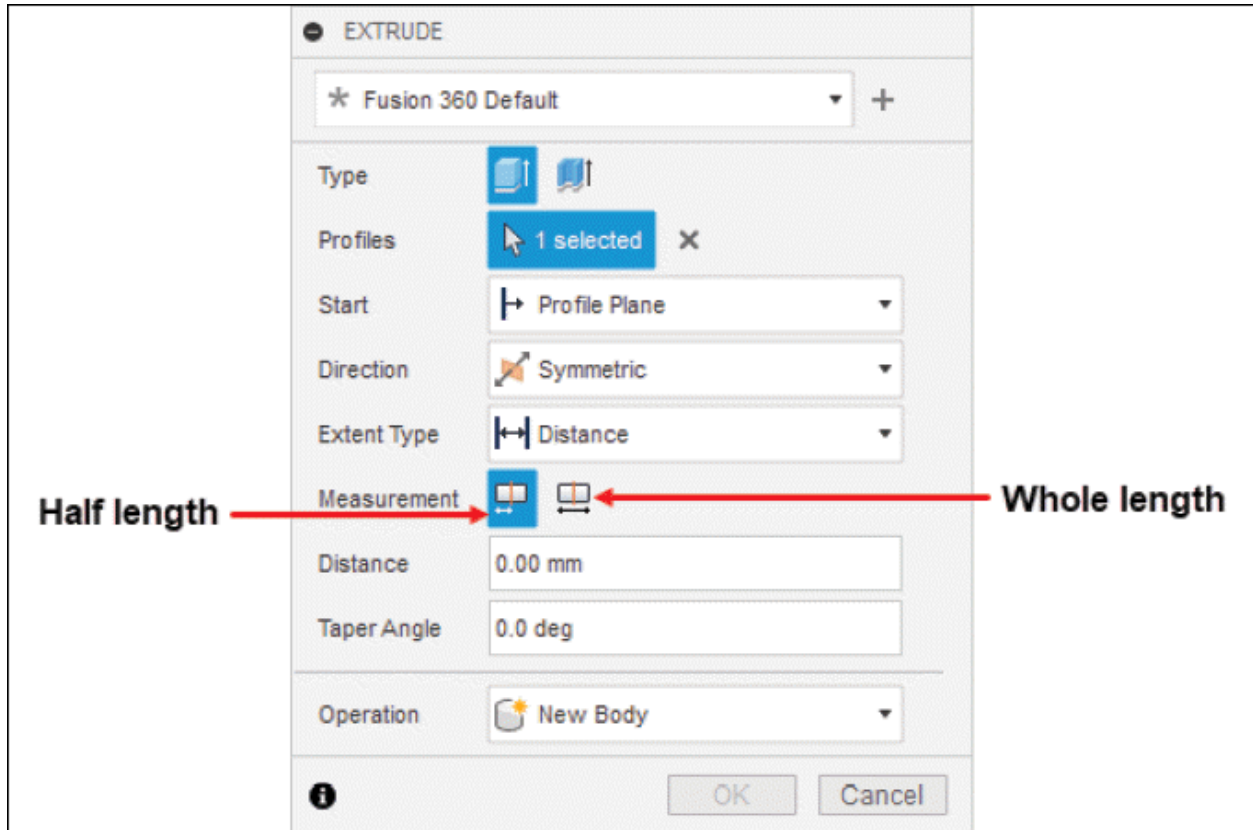


Figure-39. Extrude with symmetric direction

Extent Type drop-down

- Select **Distance** option from **Extent Type** drop-down in the **EXTRUDE** dialog box if you want to set the height of extrusion manually. Click in the **Distance** edit box and specify desired height.
- Select **To Object** option from **Extent Type** drop-down if you want to define a surface/face at which extrusion will end; refer to [Figure-40](#). After selecting this option, click on the face or surface up to which extrusion is to be created.

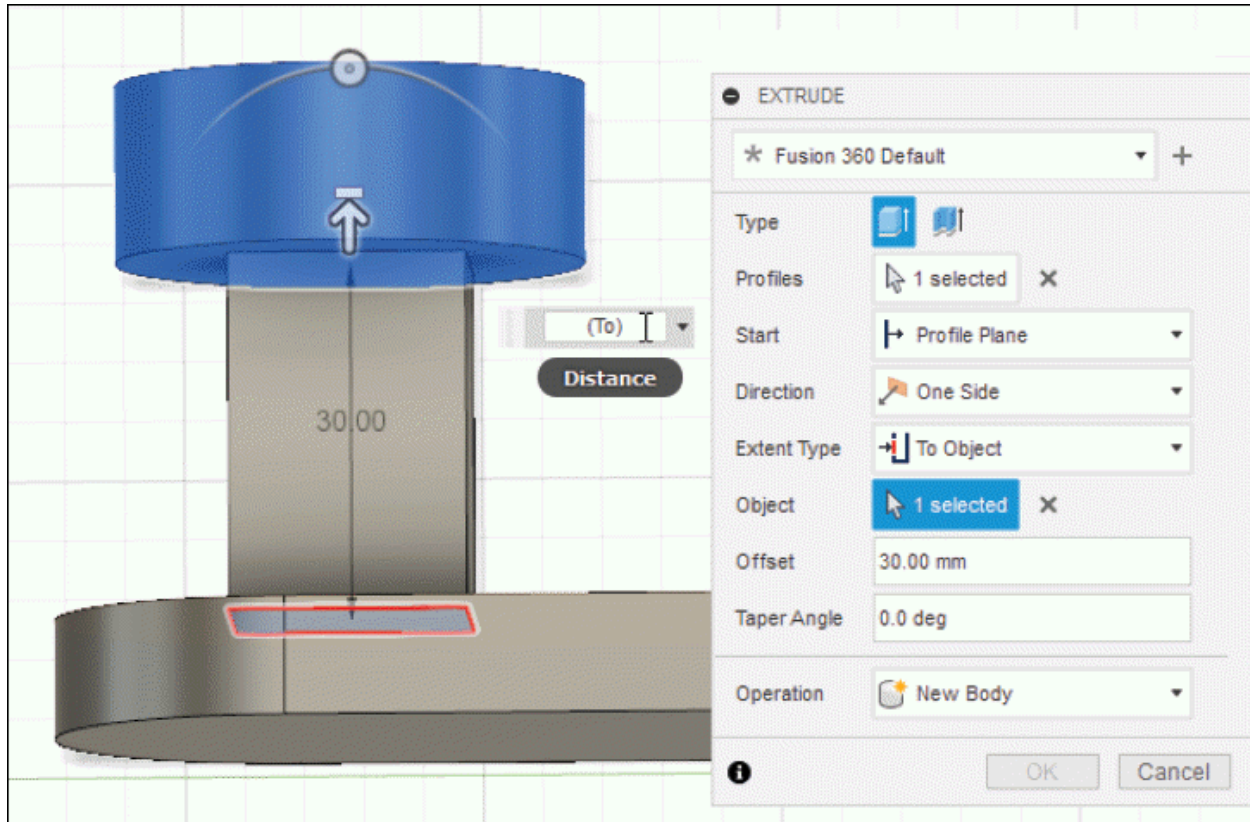


Figure-40. To Object extrude

- Select **All** option from **Extent Type** drop-down, if you want to cut through the model or part from extrusion. The extrusion will go through from the body of model; refer to [Figure-41](#). If you want to add material in place of removing then select the **Join** option from the **Operation** drop-down in the dialog box.

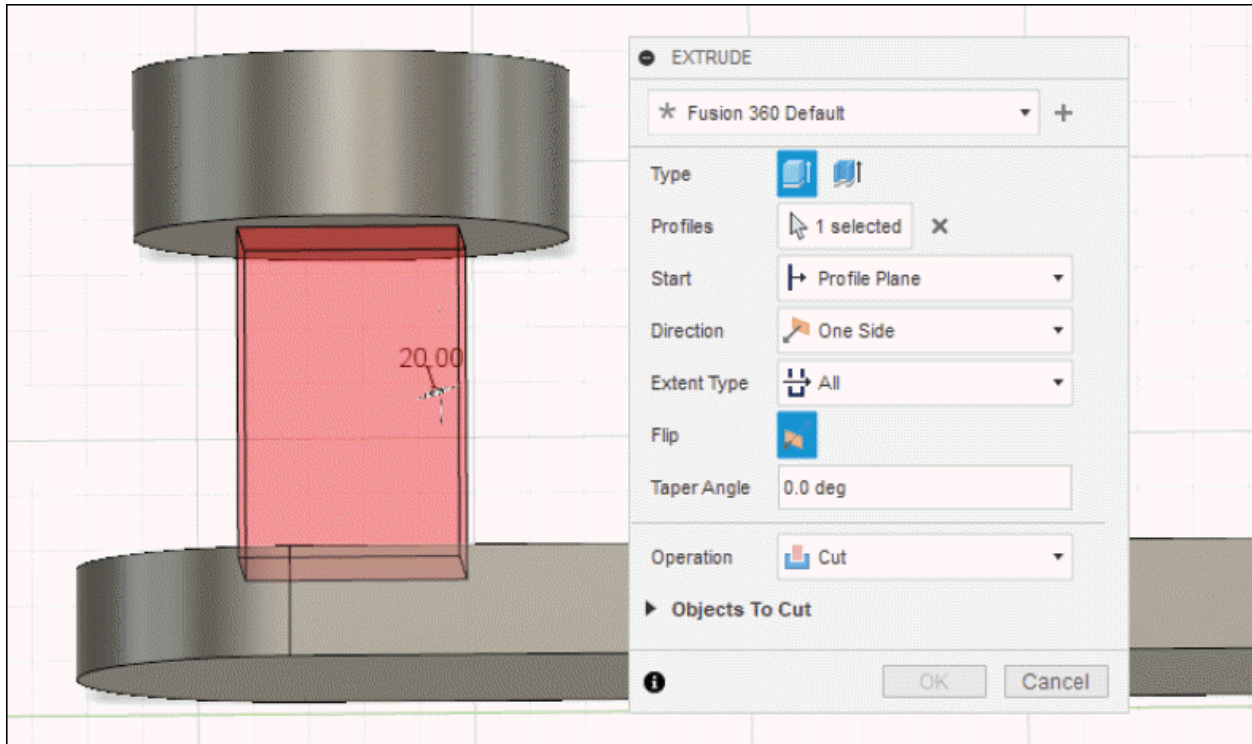


Figure-41. All tool in Extent Type drop down

- Click on the **Flip** button to reverse the direction of extrusion.

Operation drop-down

- There are five options in **Operation** drop-down of **EXTRUDE** dialog box; refer to [Figure-42](#). These options are discussed next.

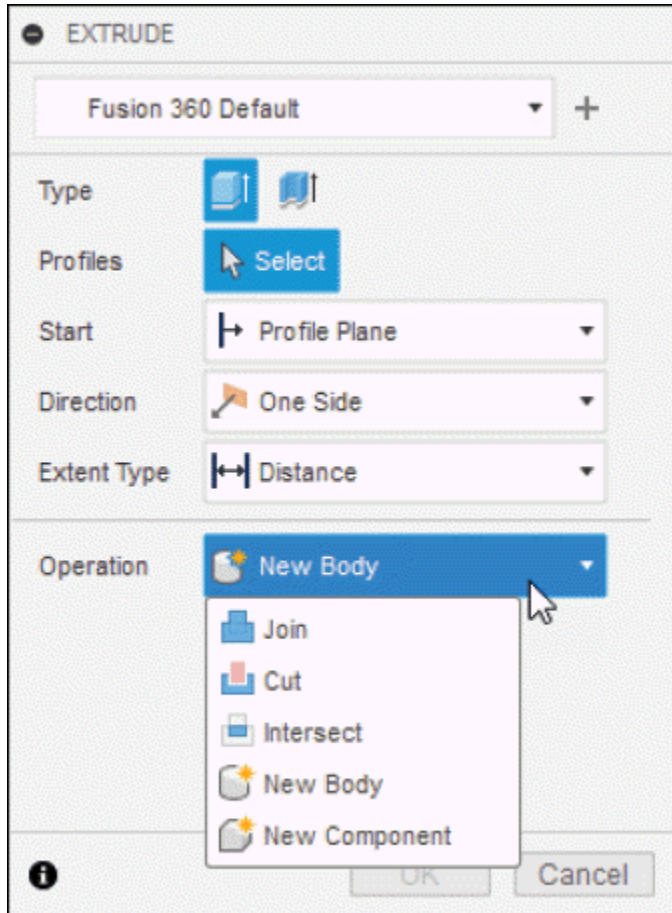


Figure-42. Operation drop down

- Select **Join** option of **Operation** drop-down if you want to combine the extrude feature with intersecting bodies; refer to [Figure-43](#).

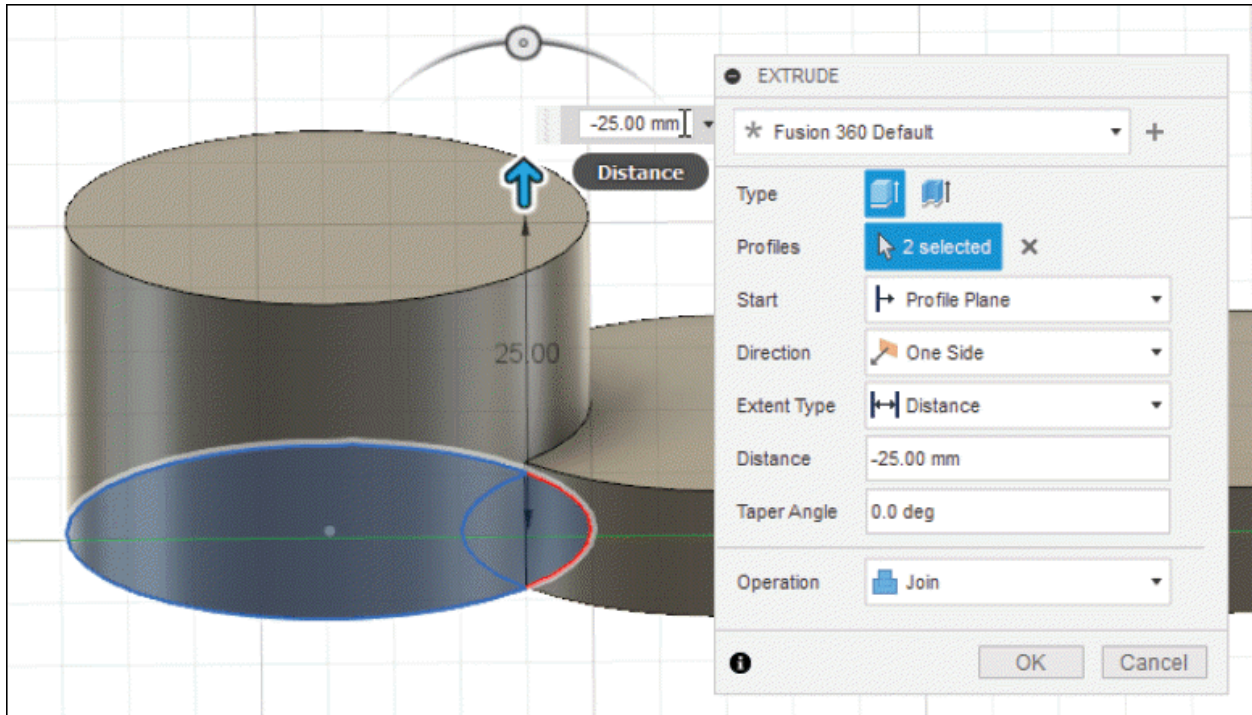


Figure-43. Join tool

- Select **Cut** option if you want to remove material from base body by using current extrude feature; refer to [Figure-48](#).

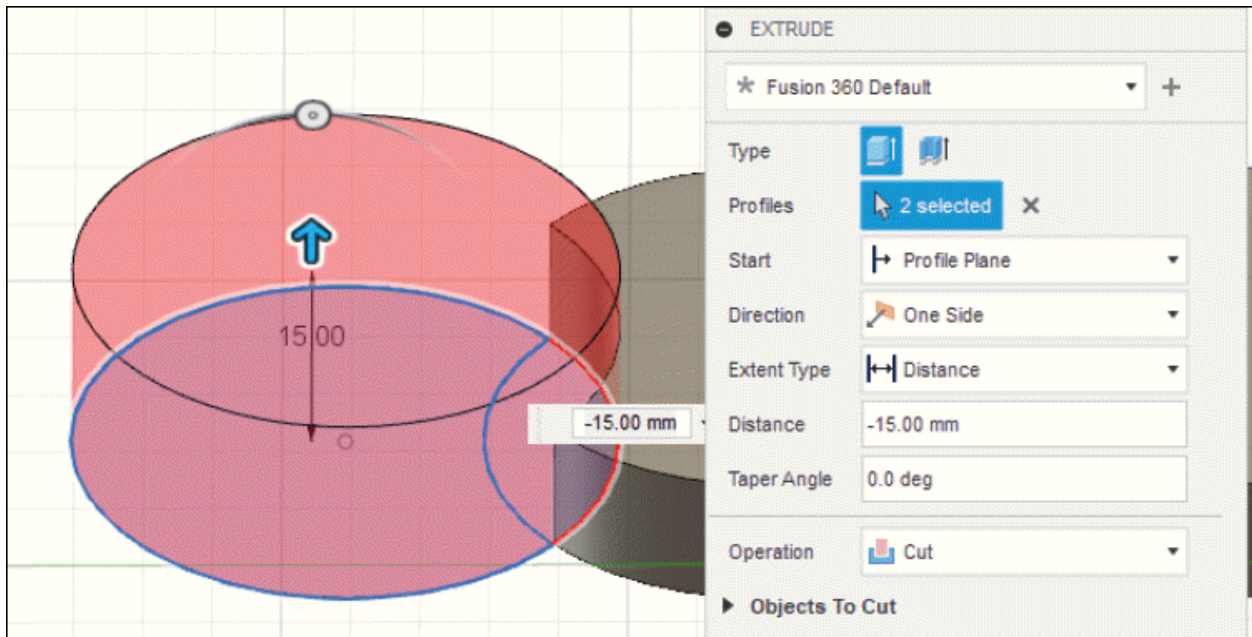


Figure-44. Cut tool

- Select **Intersect** option from **Operation** drop-down if you want to create the region commonly bounded by extrude feature and intersecting solid bodies; refer to [Figure-45](#).

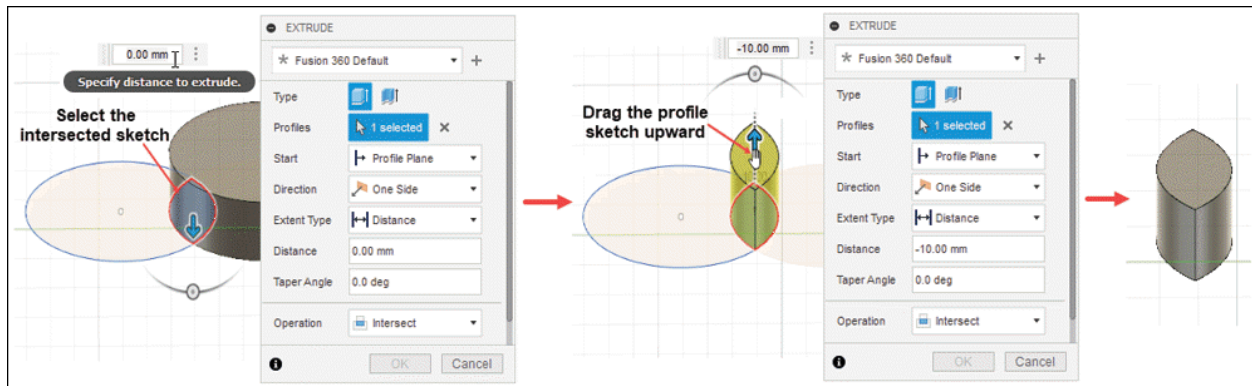


Figure-45. Intersect tool

- Select **New Body** option from **Operation** drop-down if you want to create a new body. A new body creation becomes important when you are creating molds and want all the tool bodies to be individual objects.
- Select the **New Component** option from **Operation** drop-down if you want to create a new component of the current extrude feature. New components are created for assembly. You will learn more about this option later in the book.

Extruding Thin Wall

If you want to extrude a thin wall along selected closed profiles, then select **Thin Extrude** button from **Type** section of the dialog box. The options in the **EXTRUDE** dialog box will be updated; refer to [Figure-46](#).

Wall Location drop-down

There are three options in **Wall Location** drop-down of **EXTRUDE** dialog box; refer to [Figure-47](#). These options are discussed next.

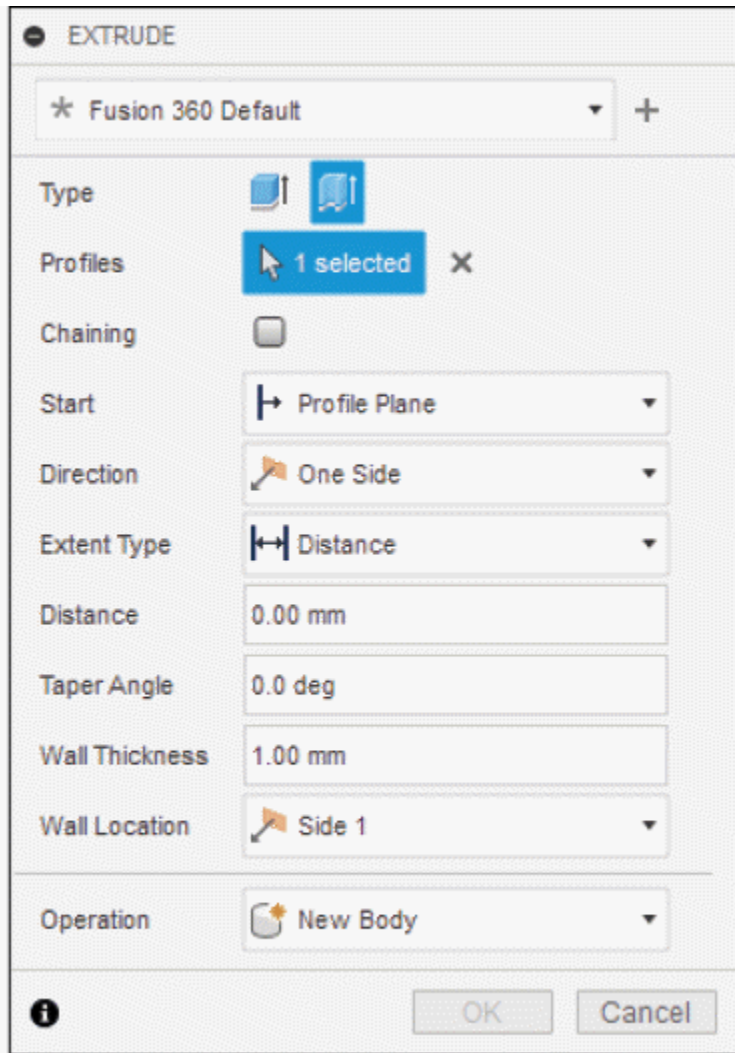


Figure-46. Thin Extrude options in EXTRUDE dialog box

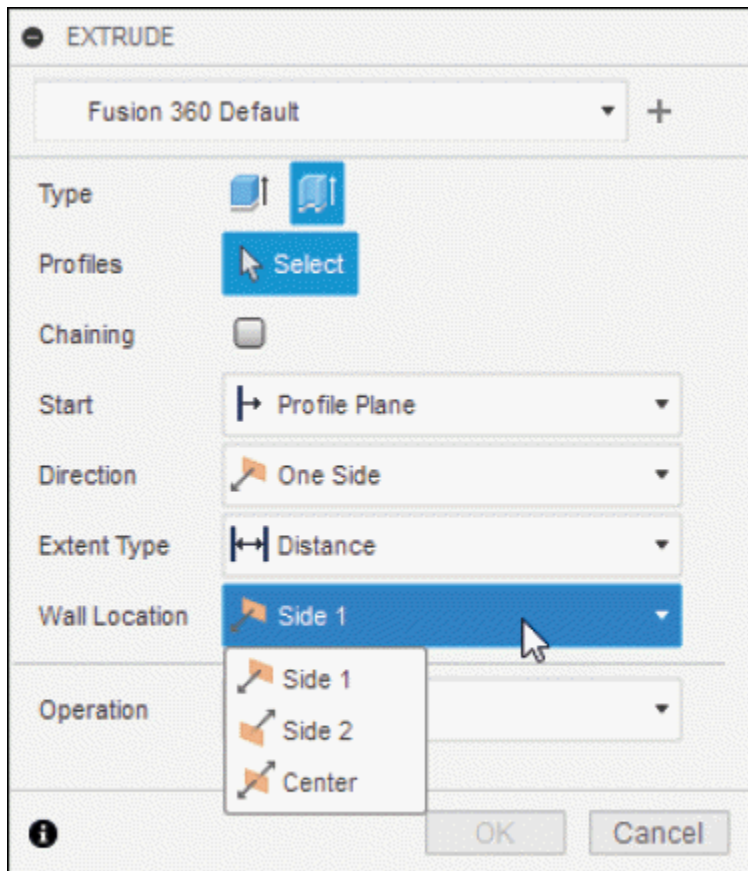


Figure-47. Wall Location drop down

- Select **Side 1** option from the drop-down if you want to offset/move the extrusion to the one side of the profile; refer to [Figure-48](#).

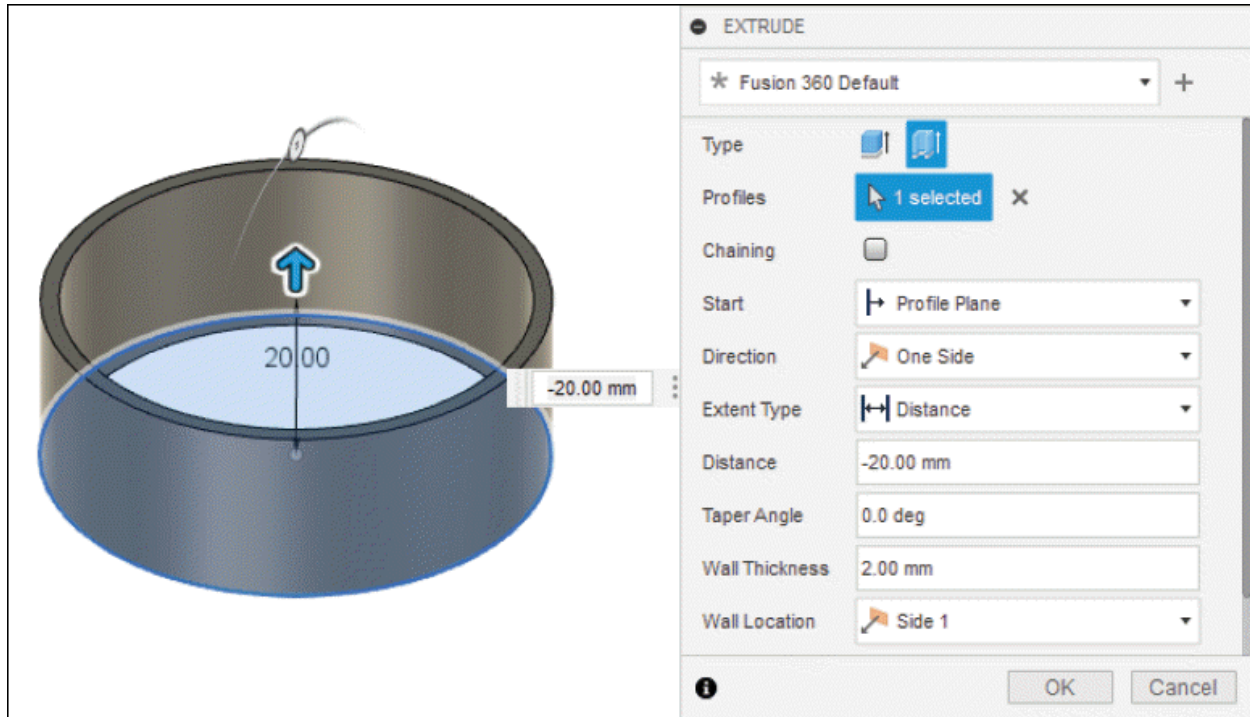


Figure-48. Extrusion inside the profile

- Specify desired thickness of extrusion in the **Wall Thickness** edit box of the dialog box.
- Select the **Chaining** check box to automatically select the connected geometry.
- Specify the parameters in the dialog box as discussed earlier and click on **OK** button. The extrusion inside the profile will be created.
- Similarly, select **Side 2** option from the drop-down to offset/move the extrusion to the other side of the profile and select **Center** option from the drop-down to center the extrusion on the profile.

Revolve

Revolve tool is used to create a solid volume by revolving a

sketch about selected axis. In other words, if you revolve a sketch about an axis then the volume covered by revolving sketch boundary is called revolve feature. The procedure to use this tool is discussed next.

- Create a closed profile sketch with a center line as discussed in previous chapter.
- Click on **Revolve** tool from **CREATE** drop-down of **Toolbar**; refer to [Figure-49](#). The **REVOLVE** dialog box will be displayed; refer to [Figure-50](#).

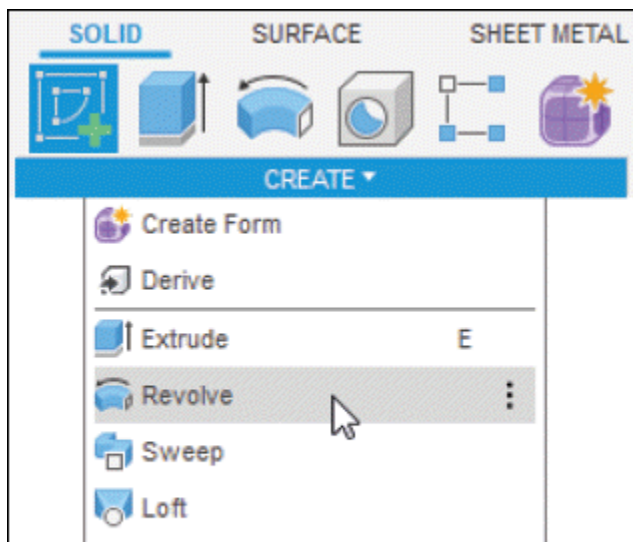


Figure-49. Revolve tool

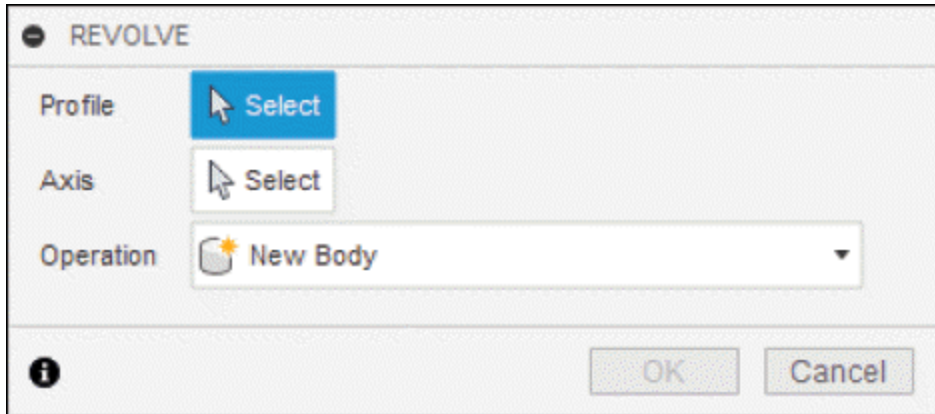


Figure-50. REVOLVE dialog box

- Click on **Select** button for **Profile** selection from **REVOLVE** dialog box and then select the sketch for profile from the screen canvas.
- Click on **Select** button for **Axis** selection from **REVOLVE** dialog box and then select the sketch edge, line, or center line from the screen canvas; refer to [Figure-51](#).

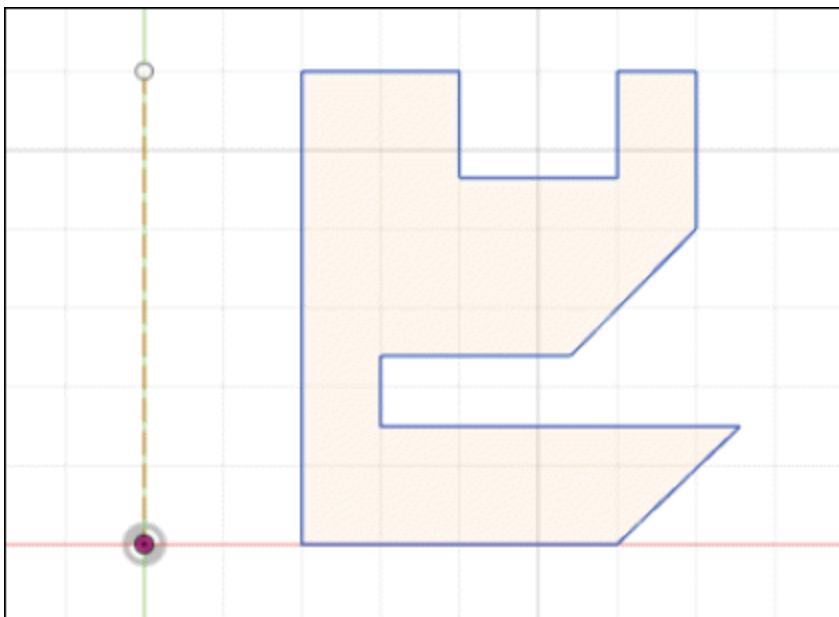


Figure-51. Region selected for revolve

- After selecting the entities, the preview of revolve feature will be displayed; refer to [Figure-52](#).

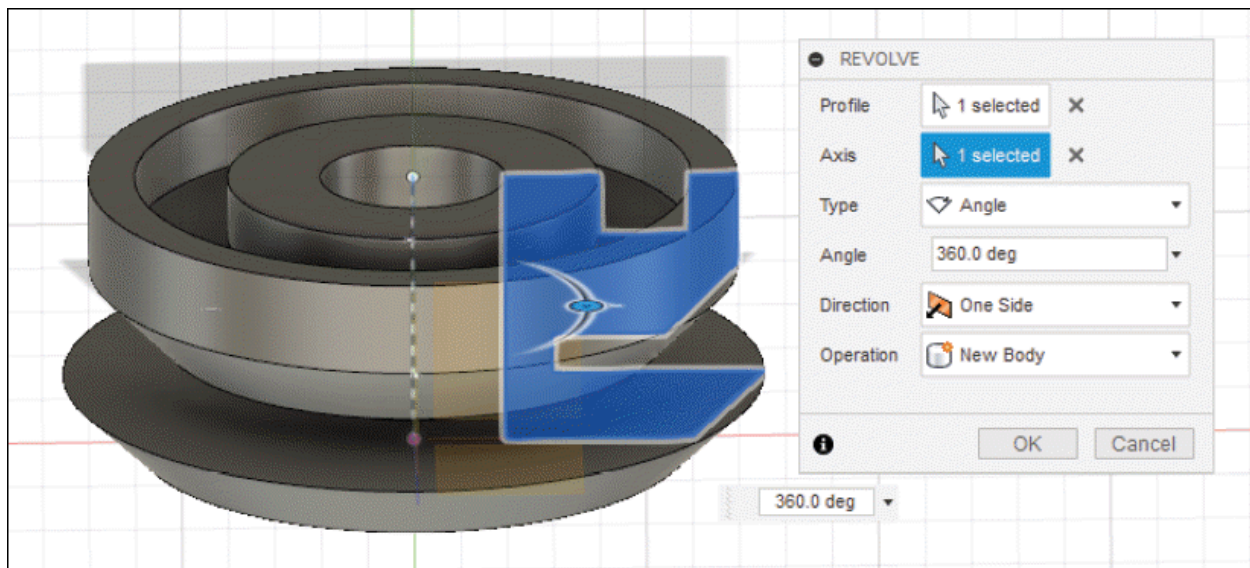


Figure-52. Preview of revolve feature

- Specify desired value of revolve angle in the **Angle** dynamic input box. You can also set the value of angle by moving the drag handle; refer to [Figure-53](#).

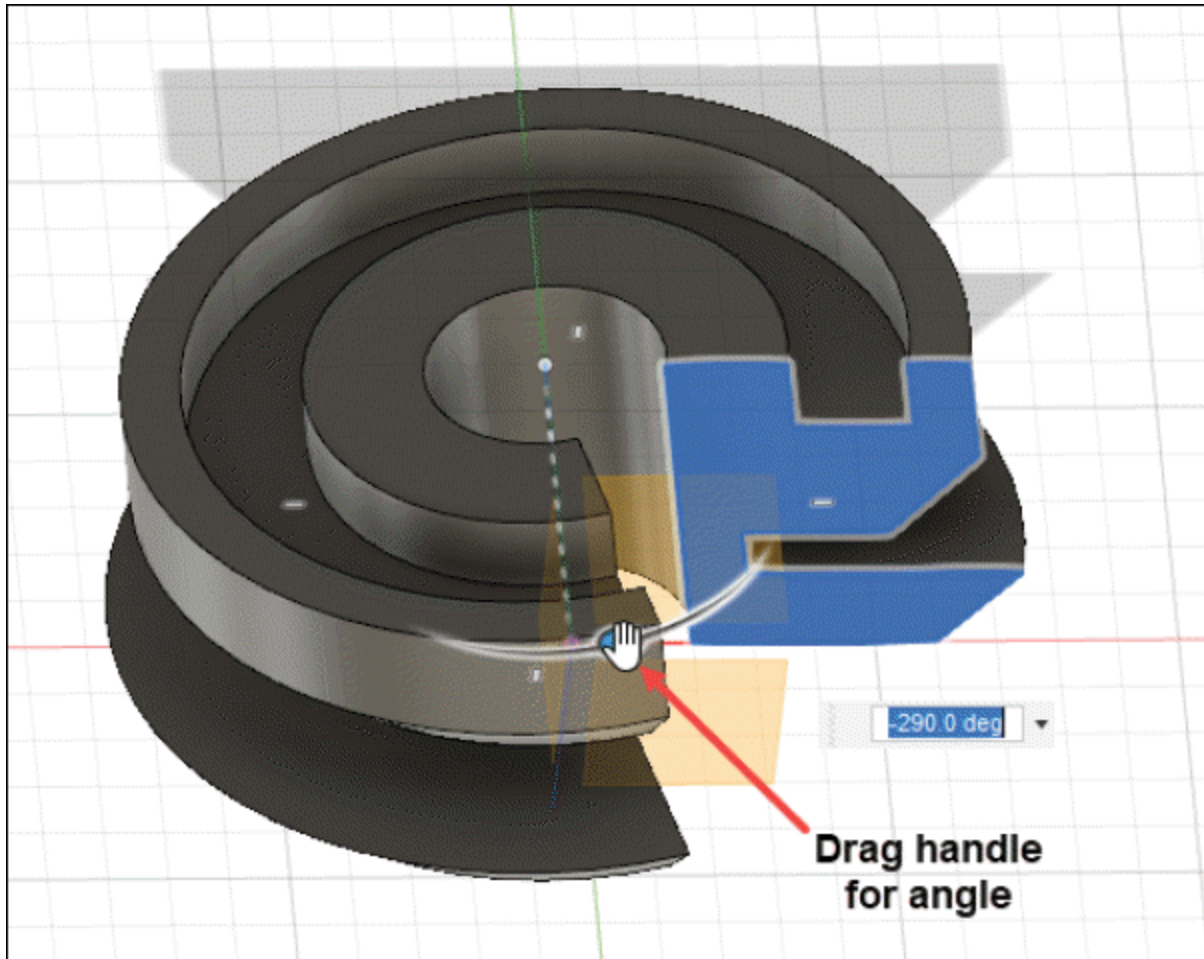


Figure-53. Moving the drag handle

Direction drop-down

The options in **Direction** drop-down are used to define in which direction profile will be revolved.

- Select **One Side** option from **Direction** drop-down to revolve the profile in one direction.
- Select **Two Side** option from **Direction** drop-down if you want to revolve the profile in both sides with different revolution angles; refer to [Figure-54](#).

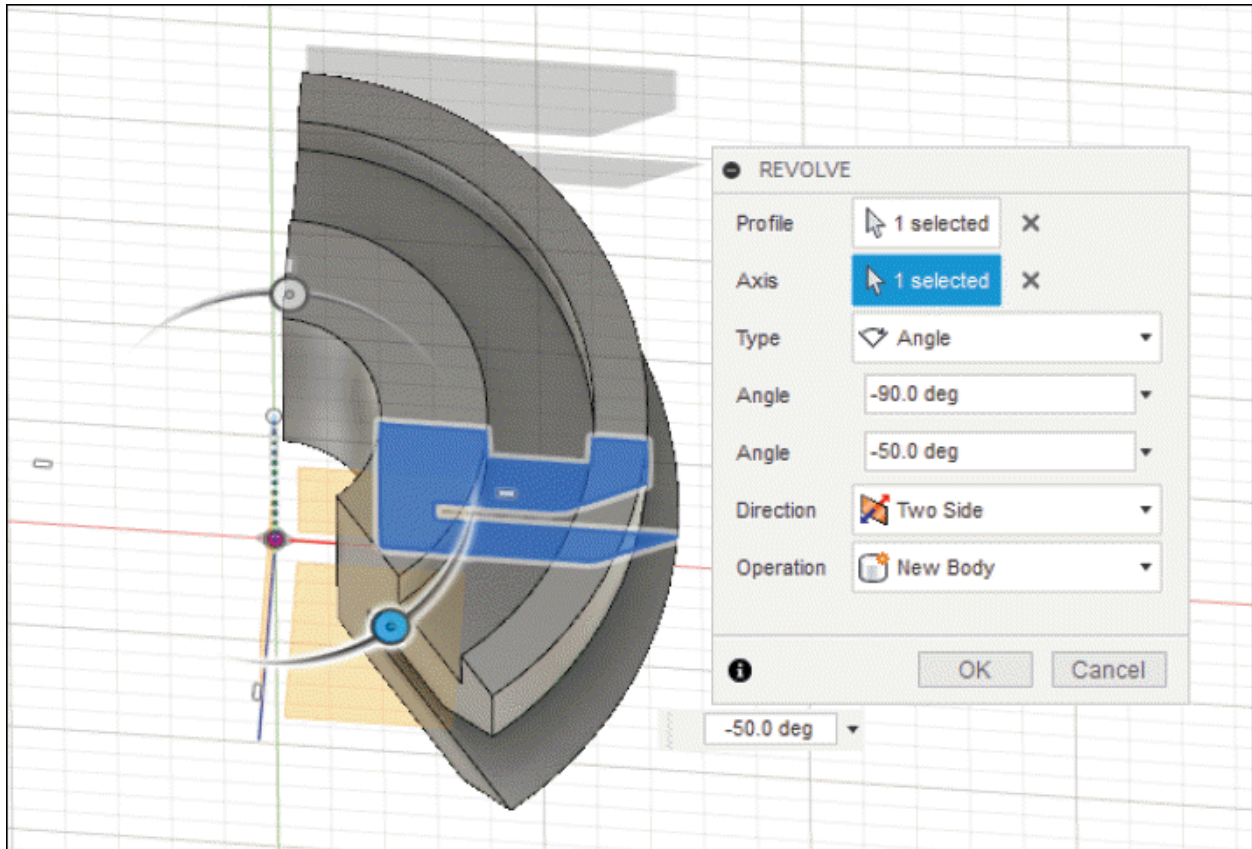


Figure-54. Two side tool

- Select the **Symmetric** option from **Direction** drop-down if you want to revolve the model in both direction with same angle i.e. symmetrically.

Type

- Select the **Angle** option from **Type** drop-down if you want to revolve the profile by specified angle.
- Select the **To** option from **Type** drop-down if you want to revolve the model up to a plane, face, surface, or vertex; refer to [Figure-55](#).

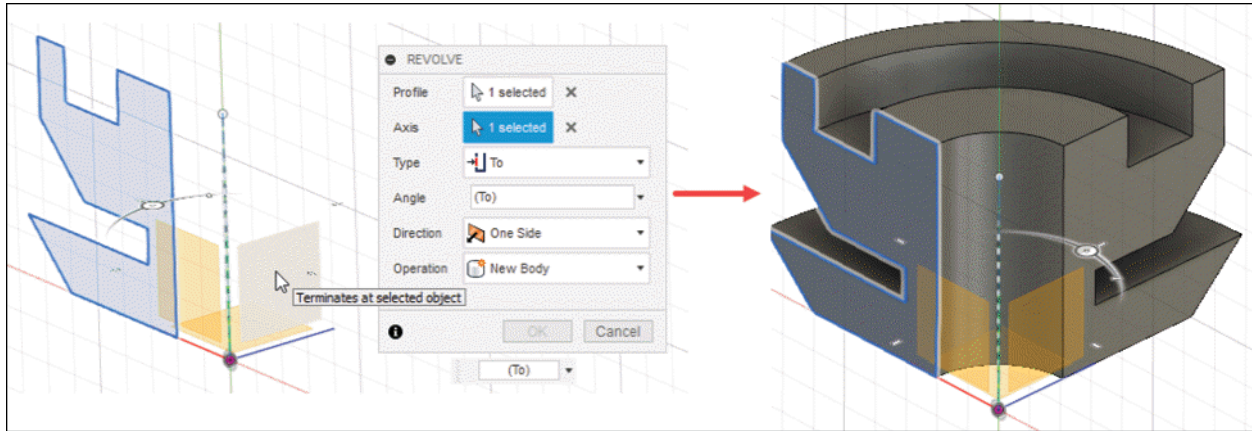


Figure-55. Preview of To option

- Select the **Full** option from **Type** drop-down if you want to revolve the profile to 360 degree angle.
- Set the other options as discussed earlier.
- Click **OK** button from **REVOLVE** dialog box to create the feature.

Sweep

The **Sweep** tool is used to create a solid volume by moving a sketch along the selected path. In other words, if you move a sketch along a path then the volume that is covered by moving sketch boundary is called sweep feature. Note that to create a solid sweep feature, you will need two sketches, one for the profile and another for the path already created in the canvas. The steps to use this tool are discussed next.

- Click on **Sweep** tool from **CREATE** drop-down; refer to [Figure-56](#). The **SWEEP** dialog box will be displayed with **Feature** tab opened by default; refer to [Figure-57](#).

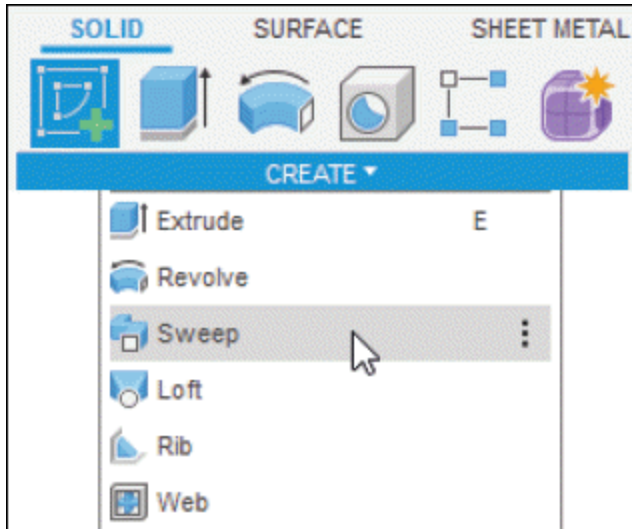


Figure-56. Sweep tool

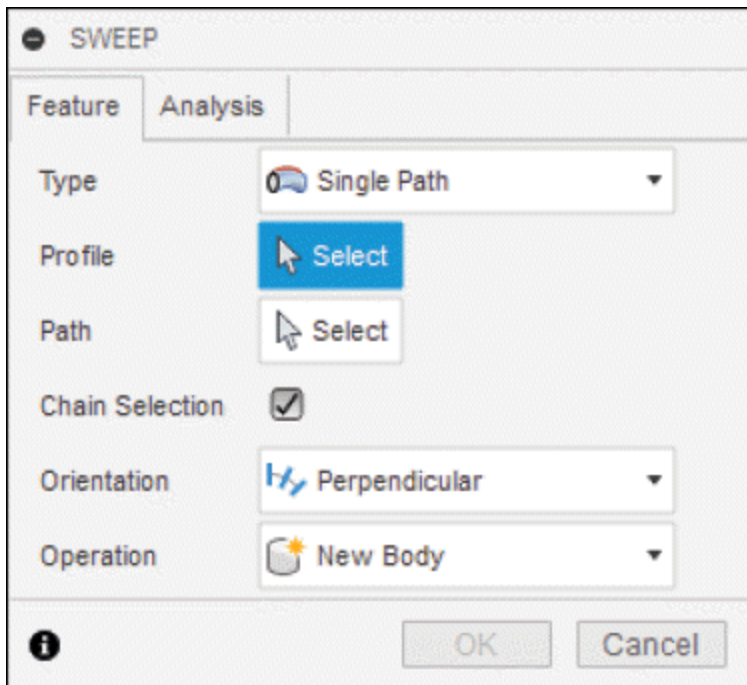


Figure-57. SWEEP dialog box

In this dialog box, there are three options in **Type** drop-down to create sweep feature; **Single Path**, **Path + Guide Rail**, and **Path + Guide Surface**. The procedures to create sweep feature by using these options are discussed next.

Single Path

The **Single Path** tool sweeps the selected profile along a single path.

- Click on the **Select** button of **Profile** section from the **SWEEP** dialog box and select the sketch for profile from the canvas screen.
- Click on the **Select** button of **Path** section from **SWEEP** dialog box and select the path from the canvas screen; refer to [Figure-58](#). Note that few more options are now available in the dialog box.

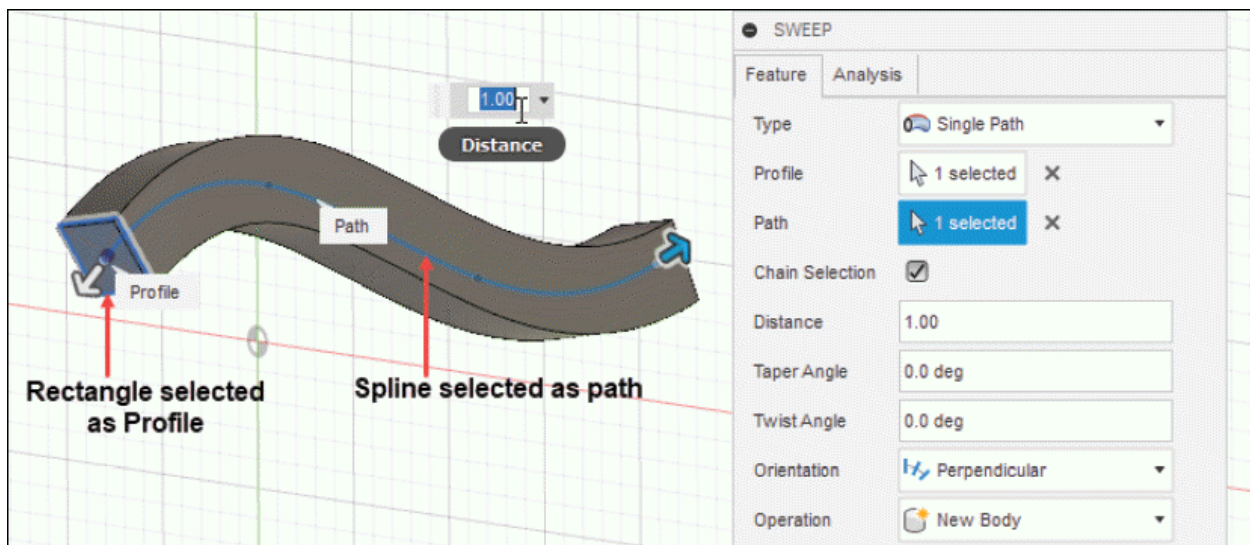


Figure-58. Selection of Sketch And Path

- Select the **Chain Selection** check box if you want to select chain of entities connected with path curve being selected; refer to [Figure-59](#).
- The distance in the **Distance** edit box is not the actual distance value rather it is the percentage of total distance along the path. **1** is the maximum value which is the total

path distance of sweep. Enter **0.5** value in the edit box to create the sweep along only half length of the path. You can also adjust the length of the sweep by entering value in dynamic input box or by moving the drag arrow handles displayed on the model.

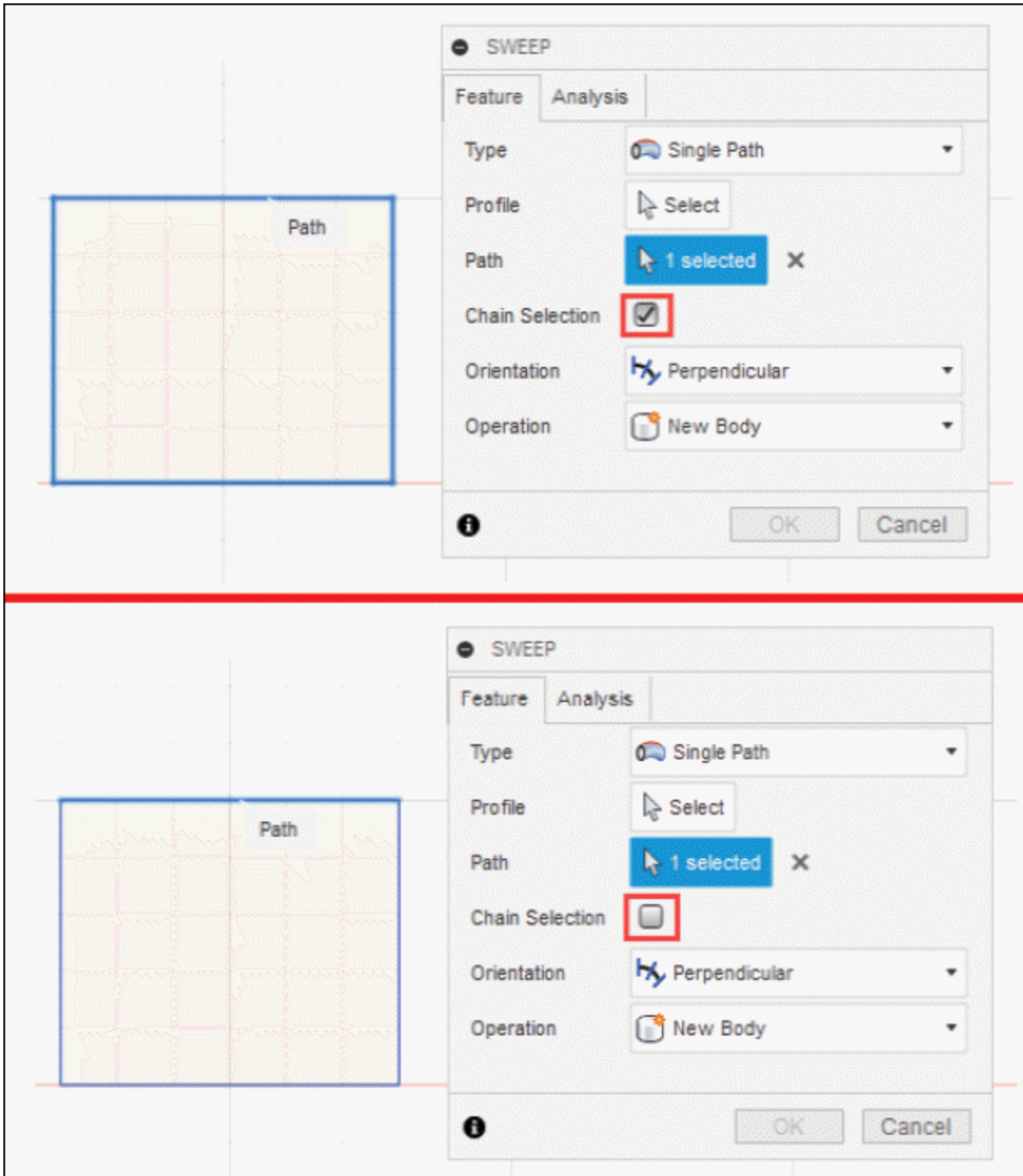


Figure-59. Selecting path with and without chain selection

- Click in the **Taper Angle** edit box and specify desired angle value to apply draft to the faces of feature.
- Click in the **Twist Angle** edit box and specify the angle

value by which you want to twist the profile along path while creating sweep feature; refer to [Figure-60](#).

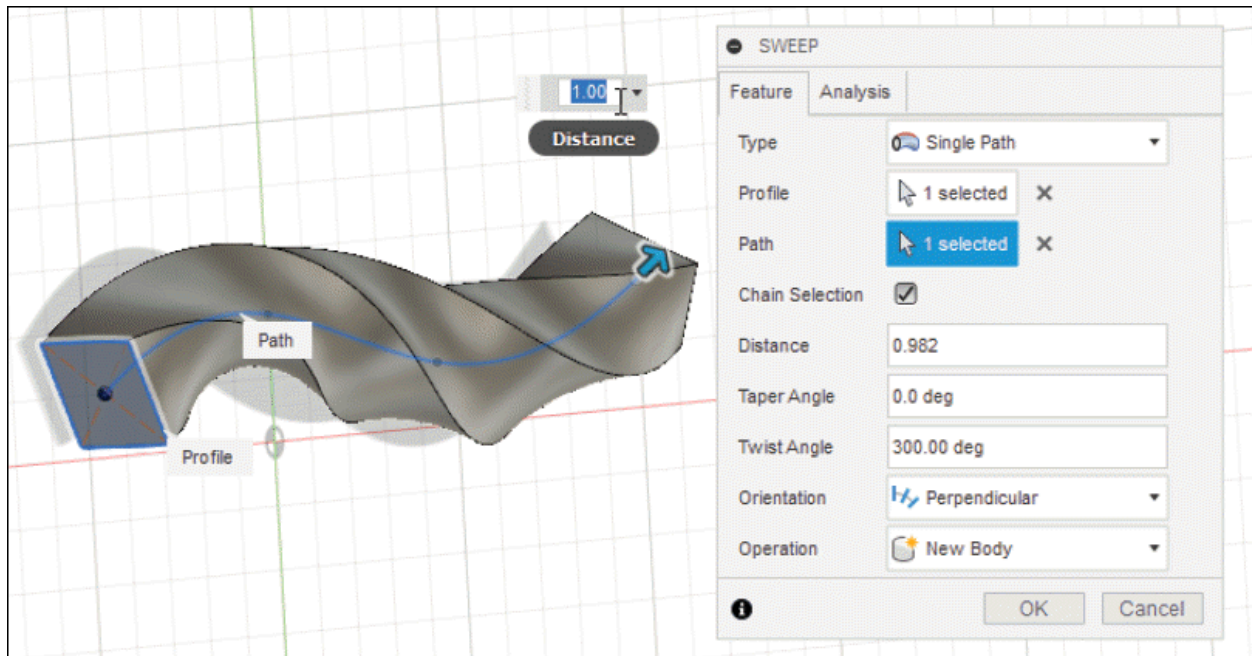


Figure-60. Preview of twisted sweep feature

- The options in the **Orientation** drop-down of **SWEEP** dialog box are used to define how the profile is pulled along the sweep path.
- **Perpendicular** option is the default selection of **Orientation** drop-down. This option creates the sweep feature in such that profile section is always perpendicular to the path. This allow more curvature on the path; refer to [Figure-61](#).

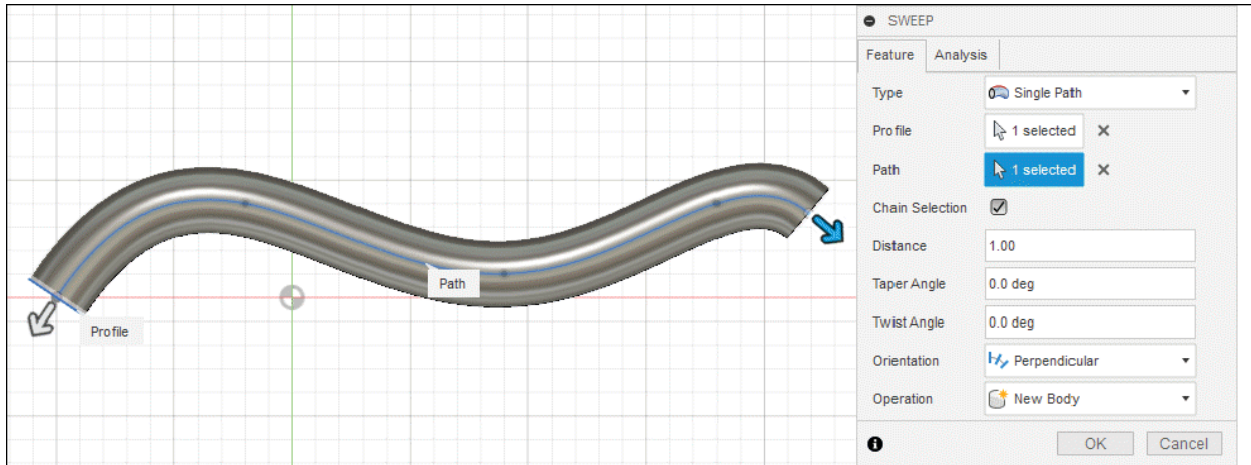


Figure-61. Perpendicular orientation

- **Parallel** option of **Orientation** drop-down creates sweep feature in such a way that the profile section is always parallel to its original orientation; refer to [Figure-62](#).

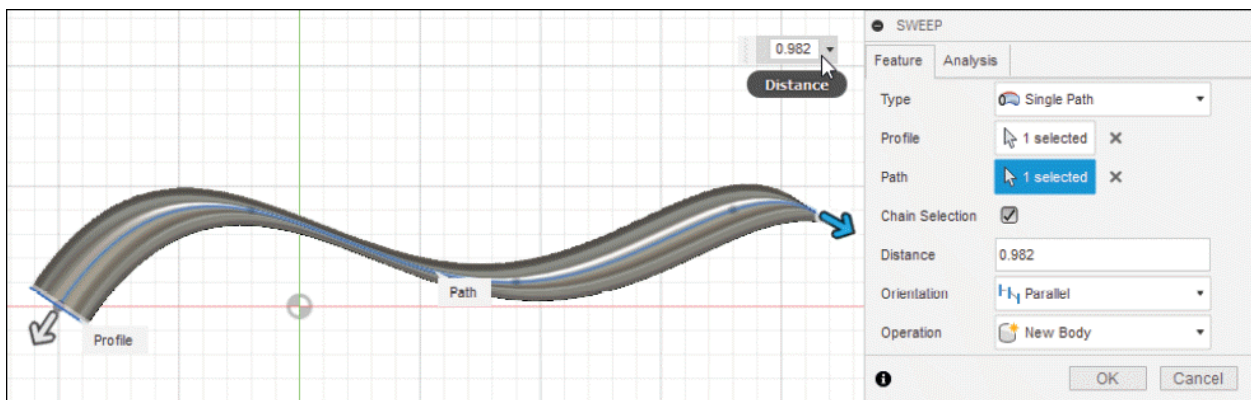


Figure-62. Parallel orientation

- After specifying the parameters, click on **OK** button to create the sweep feature.

Path + Guide Rail

Selecting this option sweeps the selected sketch along the path and uses the guide rail to control the shape. The procedure to use this tool is discussed next.

- Select the **Path + Guide Rail** option from **Type** drop-down

of **SWEEP** dialog box. The options in the dialog box will be displayed as shown in [Figure-63](#).

- Click on the **Select** button for **Profile** from the **SWEEP** dialog box and then select the sketch created for profile from the canvas screen.
- Click on the **Select** button for **Path** from **SWEEP** dialog box and then select the sketch/edge for path from the canvas screen.

Note that you can also select edges of solids/surfaces as path or guide rails and faces of solids as profile section.

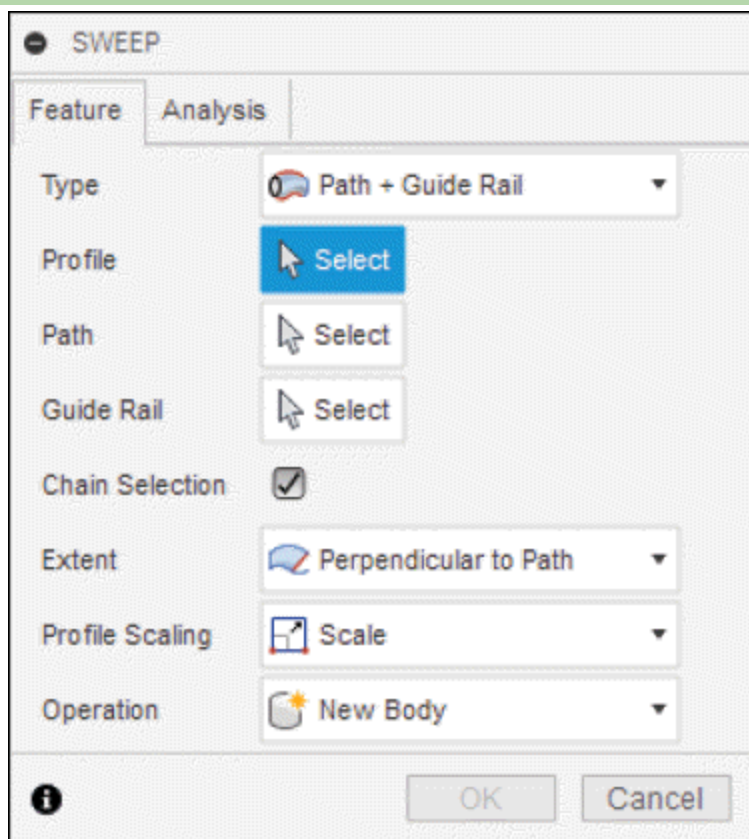


Figure-63. Path + Guide Rail option

- Click on the **Select** button for **Guide Rail** from the

SWEEP dialog box and then select the sketch for guide from the canvas screen; refer to [Figure-64](#). Preview of sweep feature will be displayed and additional options will be available in the **SWEEP** dialog box.

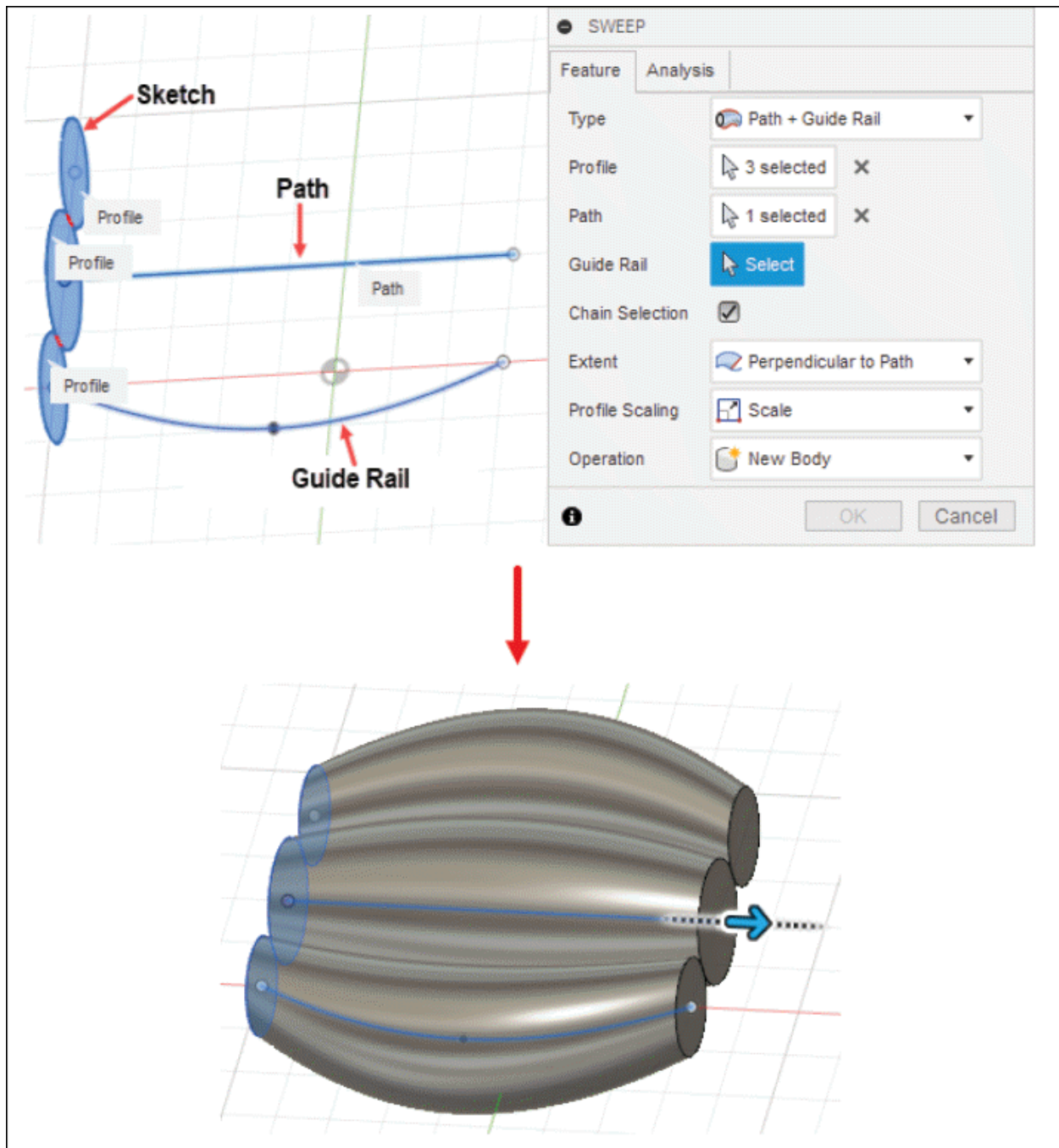


Figure-64. Selection of entities for path + guide rail

- Click in the **Extent** drop-down and select the **Perpendicular to Path** option if you want to create the sweep feature till the profile is perpendicular to path. Select the **Full Extents** option from the drop-down if you want to create sweep till the end point of path.
- Specify desired values for distance in **Path Distance** and **Guide Rail Distance** edit boxes as discussed earlier. Note that the **Guide Rail Distance** edit box will be available only when **Full Extents** option is selected in the **Extents** drop-down.
- Select the **Scale** option from **Profile Scaling** drop-down in **SWEEP** dialog box, if you want to equally scale the profile in both X and Y direction; refer to [Figure-65](#). **The profile will be scaled up depending on the distance between path and guide rail.**
- Select the **Stretch** option from **Profile Scaling** drop-down if you want to scale the profile in X direction only. In this way, profile will keep following guide rail while moving along the path; refer to [Figure-65](#).
- Click on **None** option from **Profile Scaling** drop-down if you don't want to scale the profile along sweep. On selecting this option, guide rail is used as an orientation guide only; refer to [Figure-65](#).

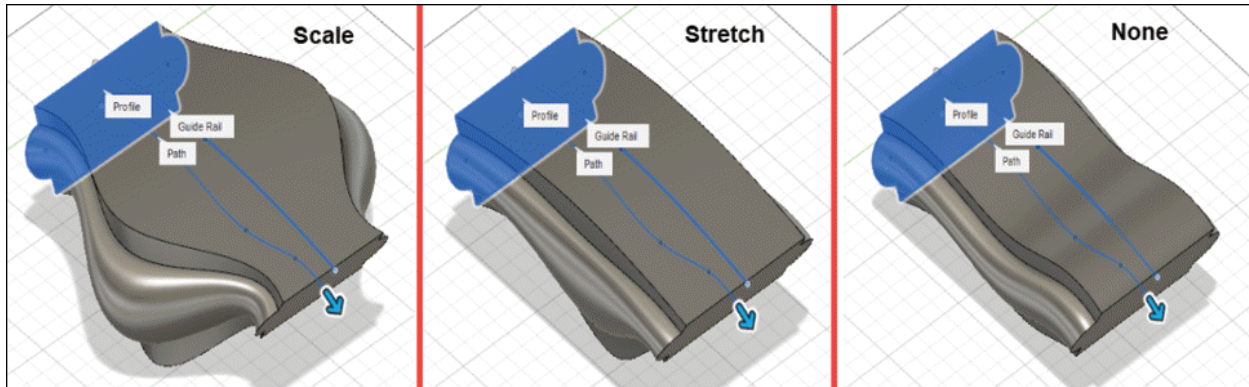


Figure-65. Preview of profile orientation

Note that the profile, path, and guide rail are same in all three cases.

- The tools of **Operation** drop-down are same as discussed earlier in this book. Click on the **OK** button from the dialog box to create the features.

Path + Guide Surface

This option sweeps the selected sketch along the path and uses the guide surface to control the shape. The procedure to use this tool is discussed next.

- Select the **Path + Guide Surface** option of **Type** drop-down from **SWEEP** dialog box. The options in the dialog box will be displayed as shown in [Figure-66](#).
- Select the profile for sweep and click on the **Select** button for **Path**.
- Select the curve/edge for path and click on the **Select** button for **Guide Surface**.
- Select the surface by which you want to control the shape of sweep feature; refer to [Figure-67](#). Preview of feature

will be displayed.

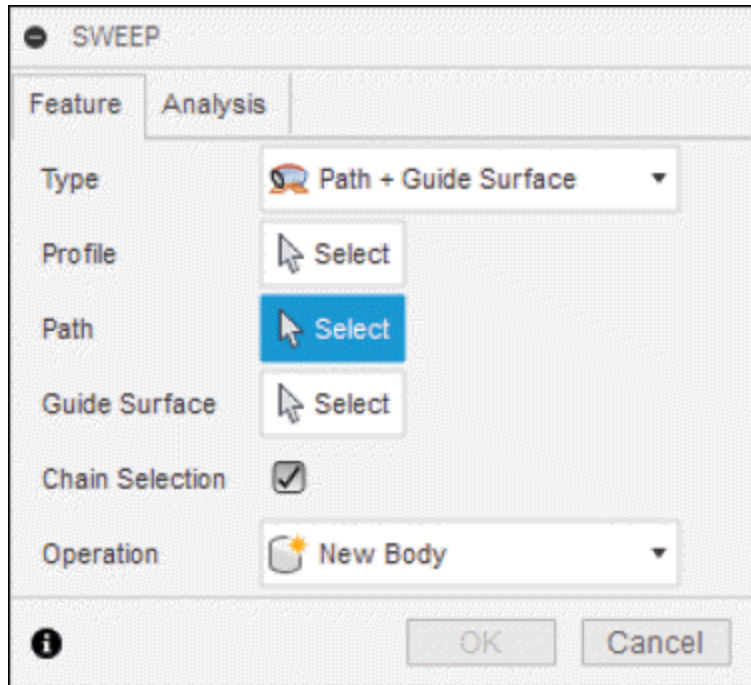


Figure-66. Path + Guide Surface option

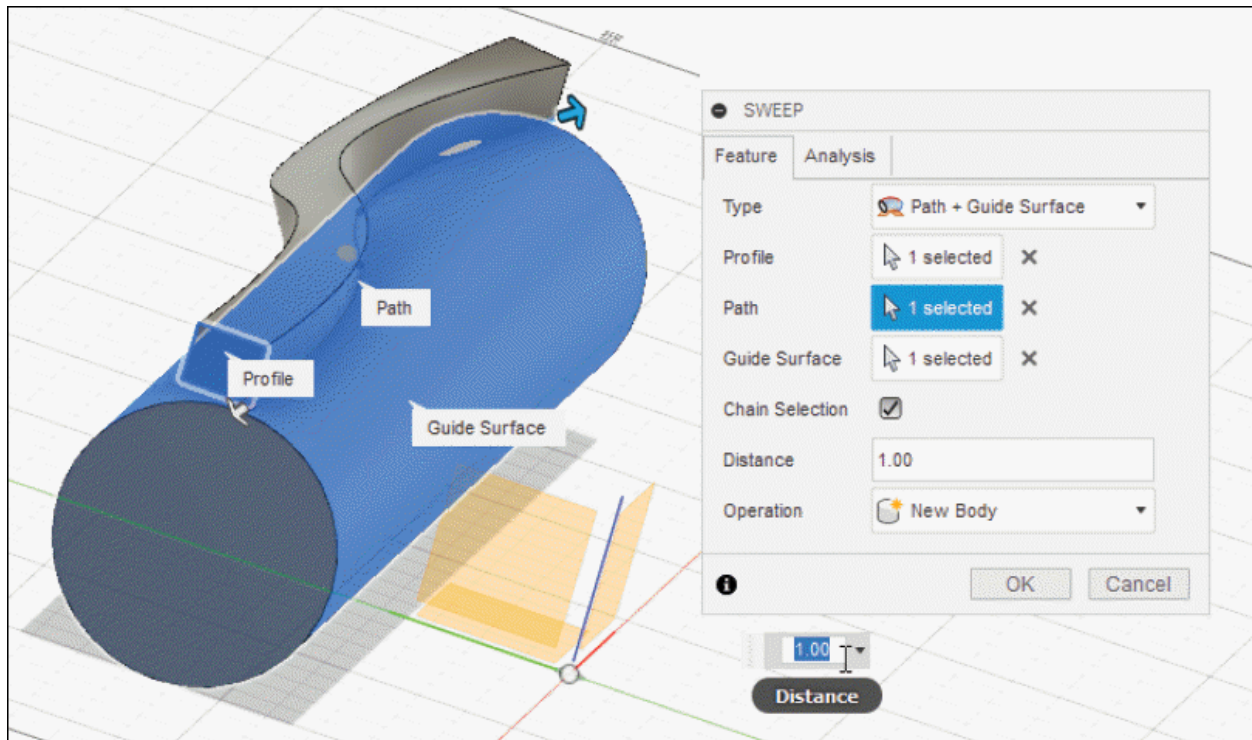


Figure-67. Sweep feature created using guide surface

- Set the parameters as discussed earlier and click on the **OK** button to create the feature.

Part file of this example is available in resources folder of this book. You can try different combinations of sweep for this example.

The **Analysis** tab of **SWEEP** dialog box is used to analyze the previewed result. We will discuss the options of this tab, later in this book.

CONSTRUCTION GEOMETRY

Till now, we have created all the features on default planes but sometimes, we need to create features that cannot be created on default planes. In such cases, we create some construction geometries. Construction geometries are used as references for other features. Some of the well known entities that come under construction geometries are:

- Planes
- Axes
- Points
- Coordinate Systems
- Curves
- Sketches

You have learned about sketches earlier in the book. Now, we will work on other features. All the tools for construction geometry features are available in the **CONSTRUCT** drop-down; refer to [Figure-68](#). These tools are discussed next.

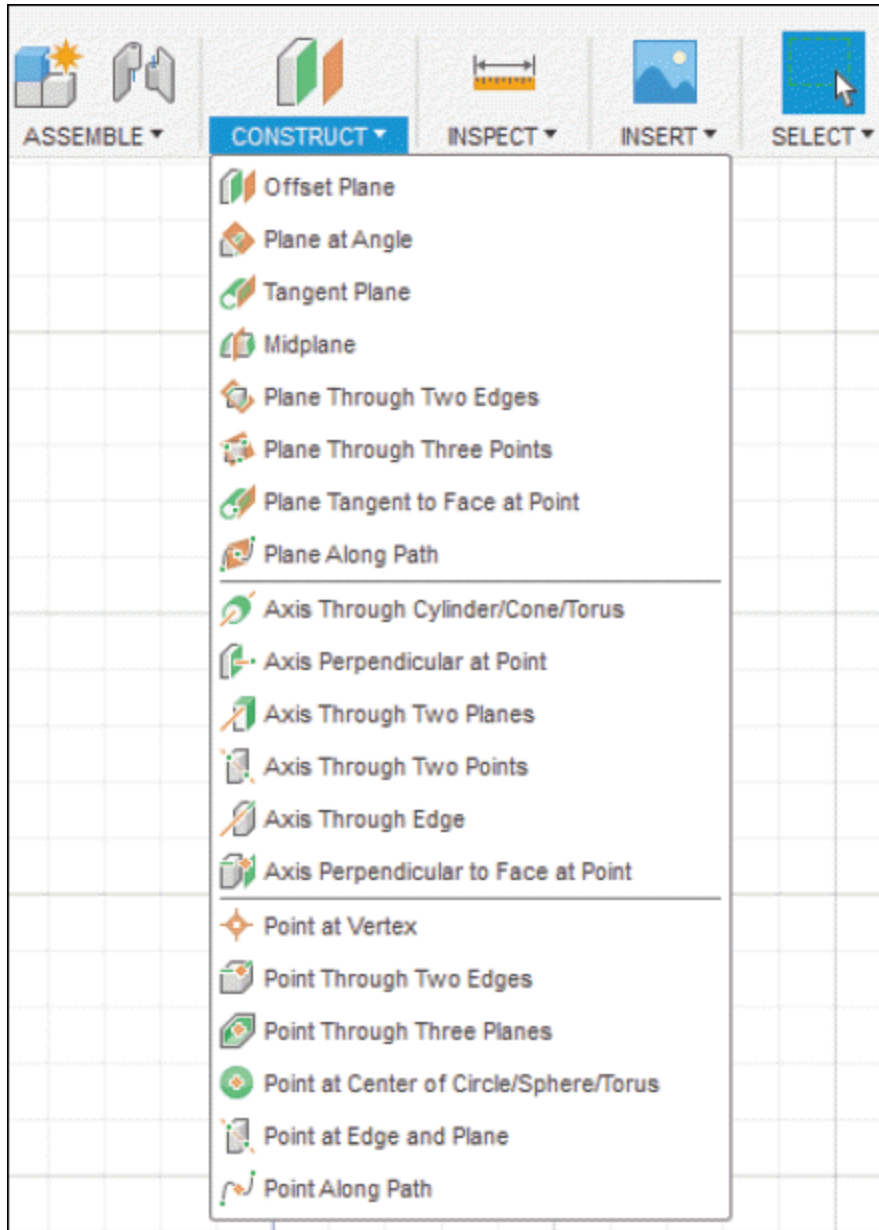


Figure-68. CONSTRUCT panel drop down

Offset Plane

The **Offset Plane** tool is used to create a plane at specified distance from a face/plane reference. The procedure to use this tool is discussed next.

- Click on the **Offset Plane** tool of **CONSTRUCT** drop-down

from **Toolbar**; refer to [Figure-69](#). The **OFFSET PLANE** dialog box will be displayed; refer to [Figure-70](#).

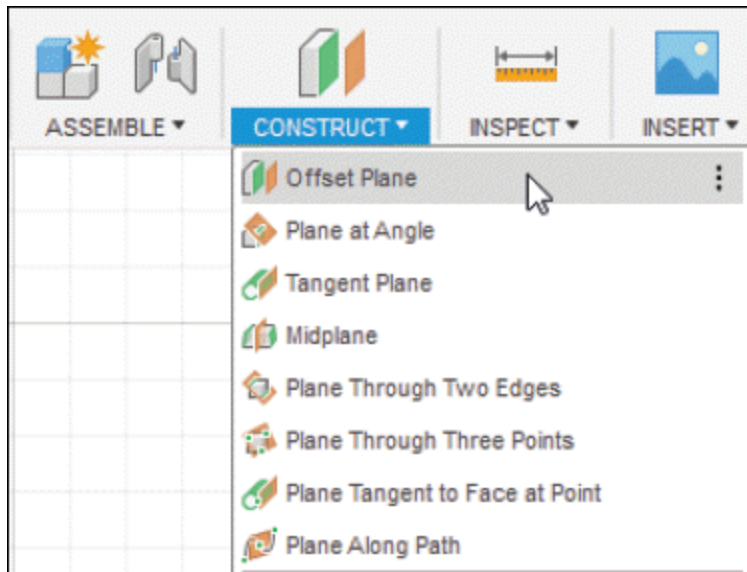


Figure-69. Offset Plane tool

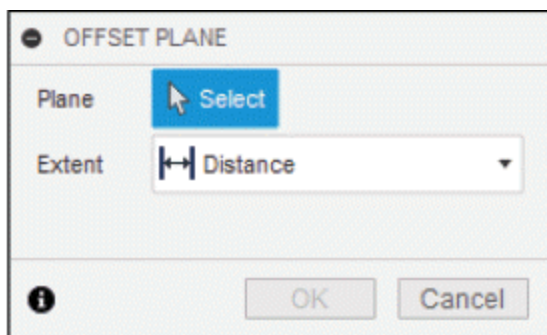


Figure-70. OFFSET PLANE dialog box

- Plane selection is active by default. Click on the plane/face of model to be used as reference for offset plane; refer to [Figure-71](#). The preview for offset plane will be displayed.

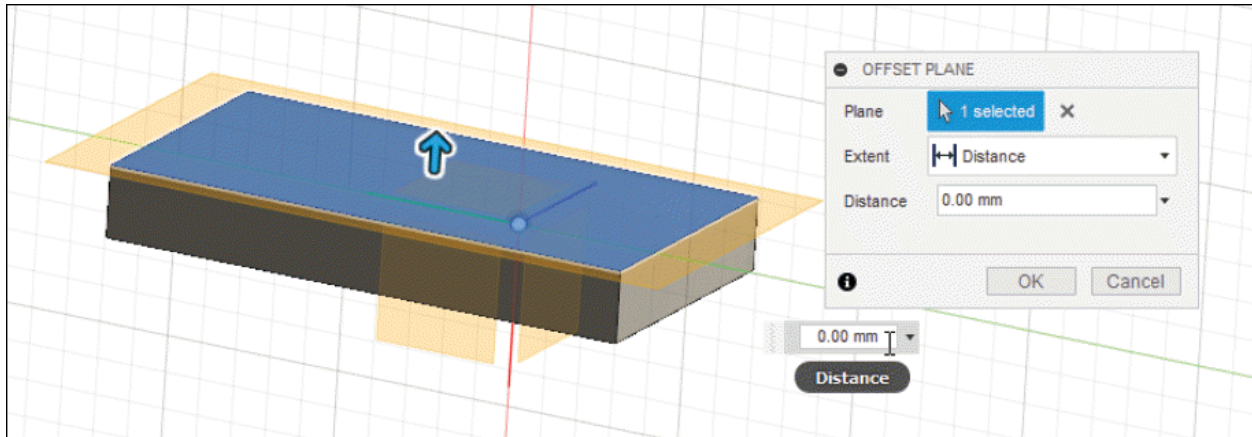


Figure-71. Selection of plane for creating offset plane

- Click in the **Distance** edit box of **OFFSET PLANE** dialog box and enter the distance at which you want to create the plane. You can also set the distance by moving the drag handle.
- After specifying the distance for offset plane, click on the **OK** button from **OFFSET PLANE** dialog box. The plane will be created; refer to [Figure-72](#).

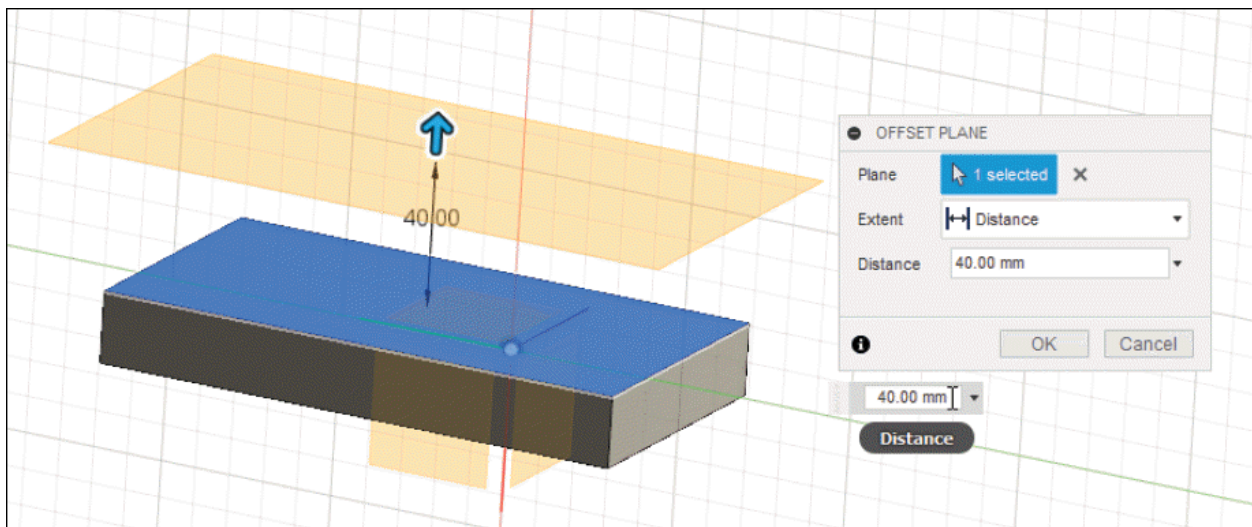


Figure-72. Preview of offset plane

Plane at Angle

The **Plane at Angle** tool is used to create a plane at specified angle. The procedure to use this tool is discussed next.

- Click on the **Plane at Angle** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-73](#). The **PLANE AT ANGLE** dialog box will be displayed; refer to [Figure-74](#).

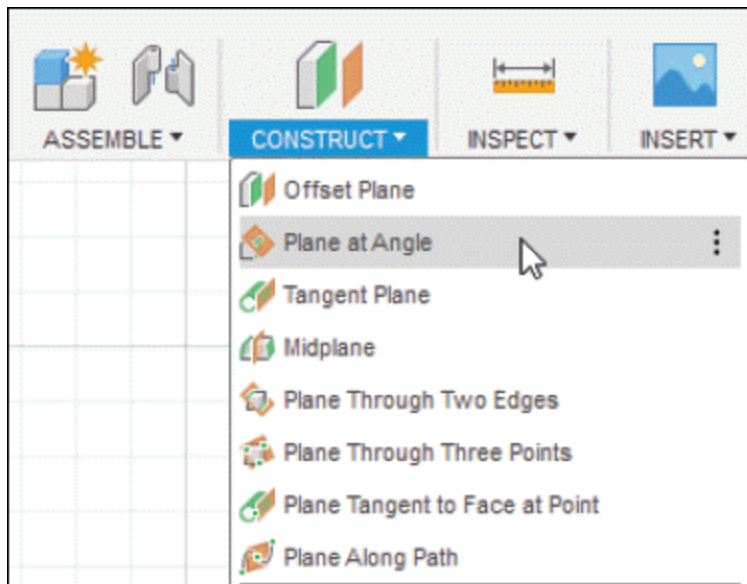


Figure-73. Plane at Angle tool

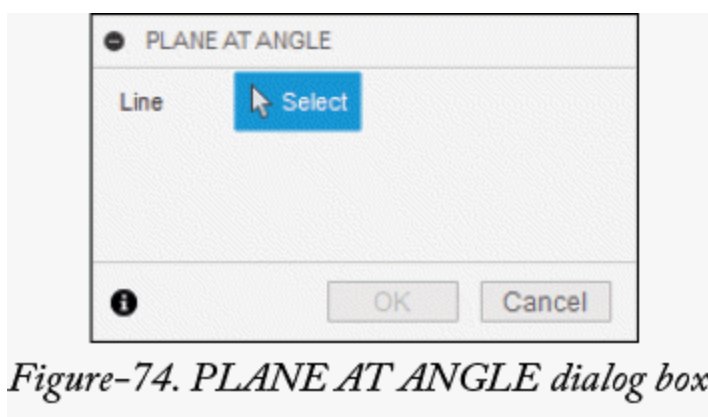


Figure-74. PLANE AT ANGLE dialog box

- **Line** selection of **PLANE AT ANGLE** dialog box is active by default. Click on the line/edge/axis from the model as

reference for creating a plane. The updated **PLANE AT ANGLE** dialog box will be displayed; refer to [Figure-75](#).

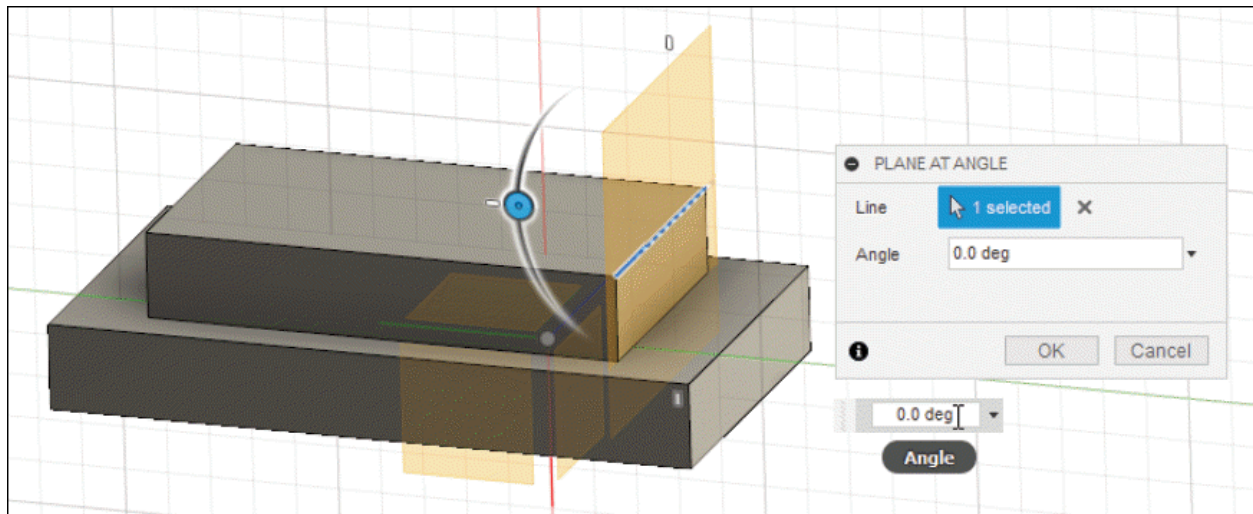


Figure-75. Updated *PLANE AT ANGLE* dialog box

- Specify desired angle value in **Angle** edit box or drag the handle.
- After specifying the angle for plane, click on the **OK** button from **PLANE AT ANGLE** dialog box. The plane will be created; refer to [Figure-76](#).

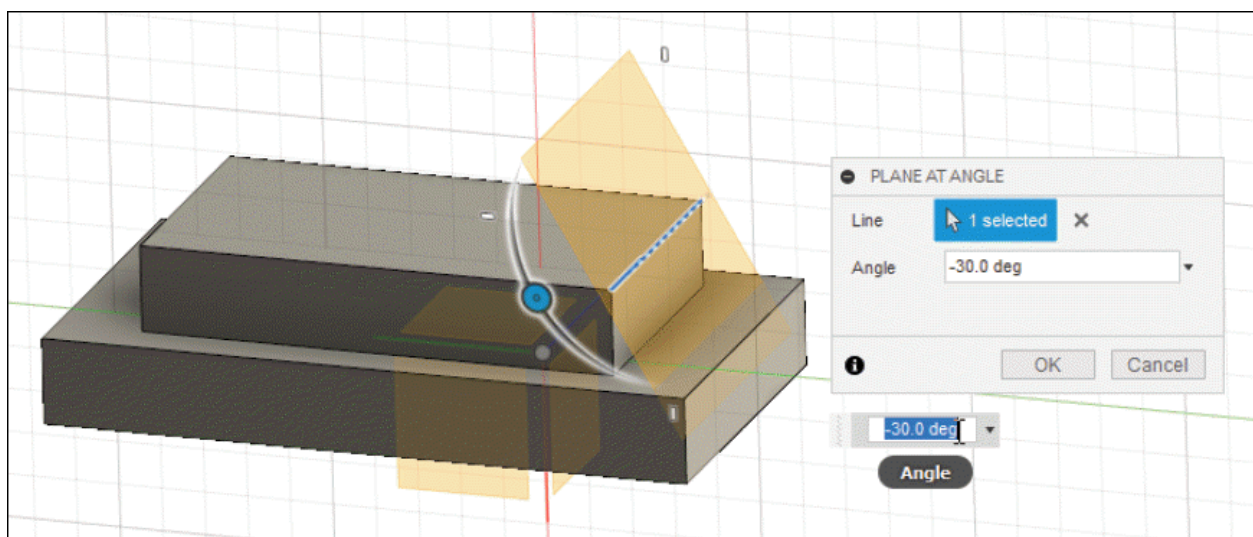


Figure-76. *Preview of plane at angle*

Tangent Plane

The **Tangent Plane** tool is used to create a plane tangent to the selected face/surface. The procedure to use this tool is discussed next.

- Click on the **Tangent Plane** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-77](#). The **TANGENT PLANE** dialog box will be displayed; refer to [Figure-78](#).

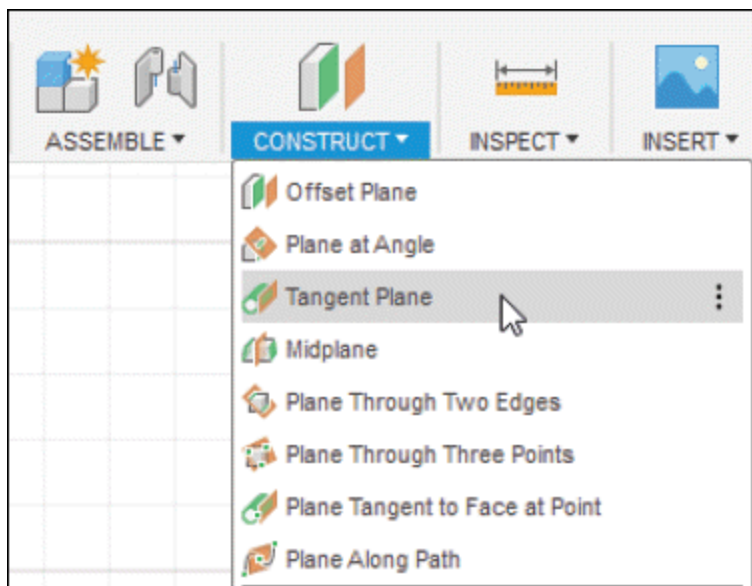


Figure-77. Tangent Plane tool

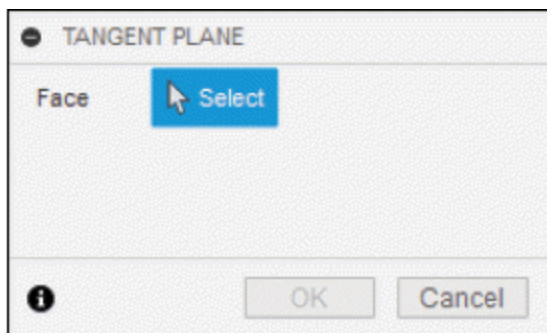


Figure-78. TANGENT PLANE dialog box

- The **Face** selection of **TANGENT PLANE** dialog box is

active by default. Click on the round face of model to which plane should be tangent. The updated dialog box will be displayed; refer to [Figure-79](#).

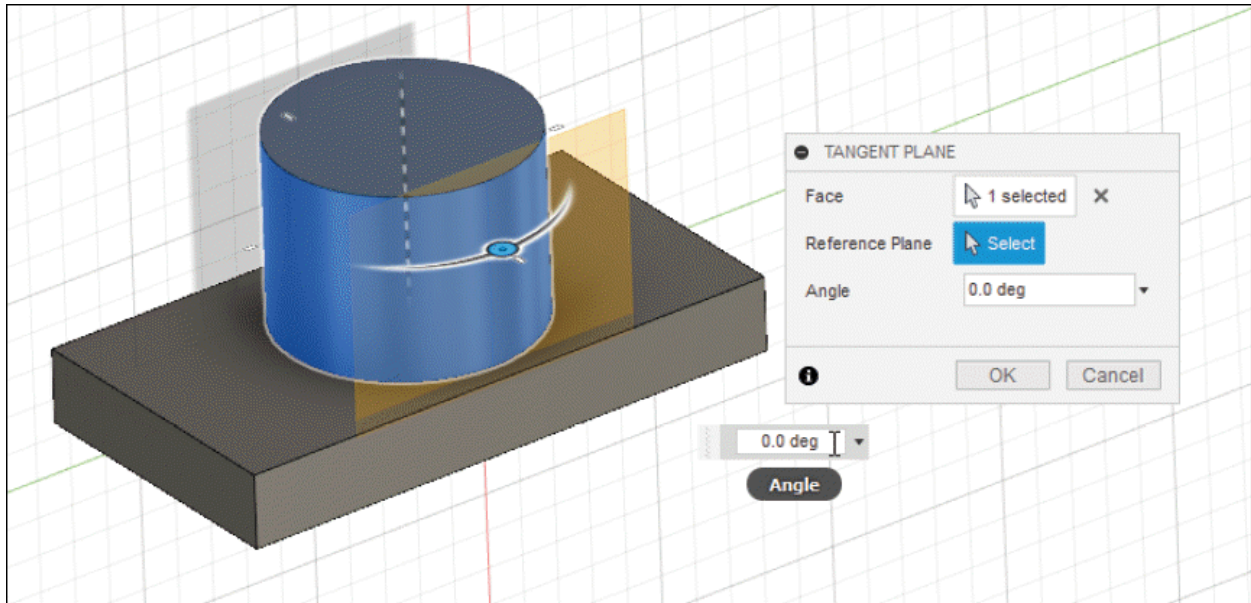


Figure-79. Updated TANGENT PLANE dialog box

- Selection of **Reference Plane** is optional. If you want to select the reference plane to define orientation of the tangent plane then click on the **Select** button of **Reference Plane** section and click on plane/face. The plane will orient accordingly to selected plane.
- Click in the **Angle** edit box and specify the angle of plane with respect to the reference plane. If you have not specified reference plane then angle will be calculated from original position. You can also set the angle by moving the drag handle.
- After specifying the parameters, click on the **OK** button from **TANGENT PLANE** dialog box to create a tangent

plane; refer to [Figure-80](#).

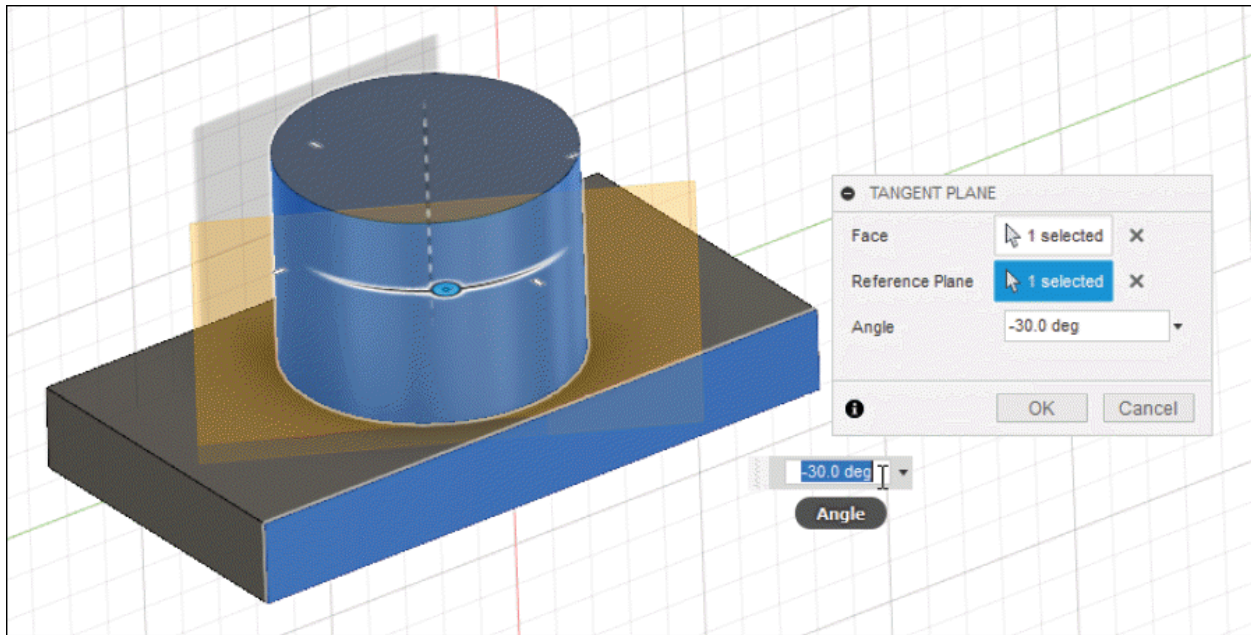


Figure-80. Preview of tangent plane

Midplane

The **Midplane** tool is used to create a plane at the midpoint of two faces/planes. The procedure to use this tool is discussed next.

- Click on the **Midplane** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-81](#). The **MIDPLANE** dialog box will be displayed; refer to [Figure-82](#).

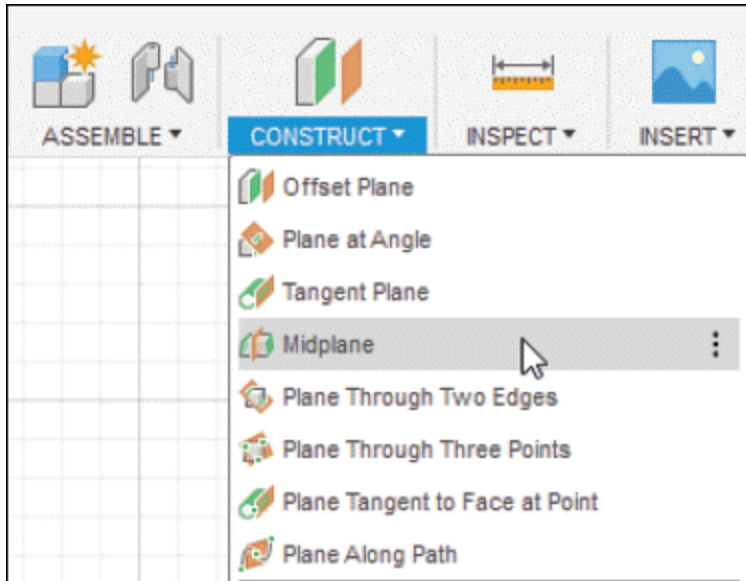


Figure-81. Midplane tool

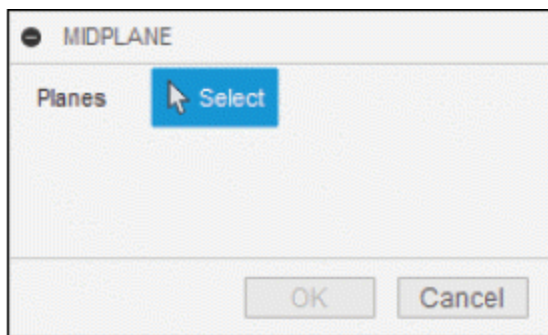


Figure-82. MIDPLANE dialog box

- Now, you need to select two planes or faces to create a mid plane; refer to [Figure-83](#).

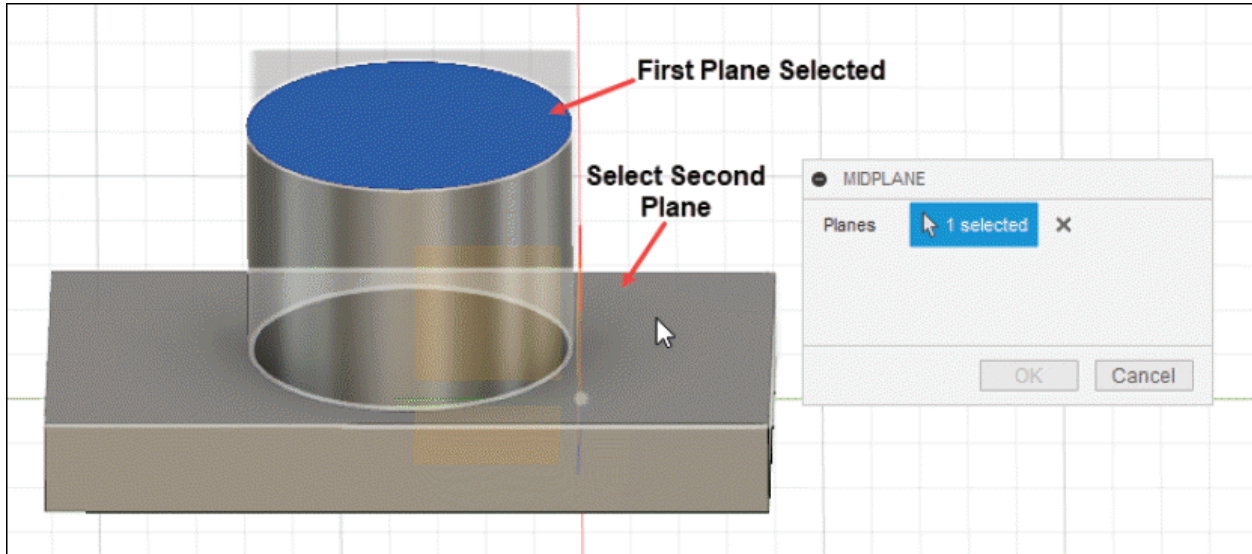


Figure-83. Selection of face to create midplane

- After selecting the second plane or face, preview of plane will be displayed; refer to [Figure-84](#).

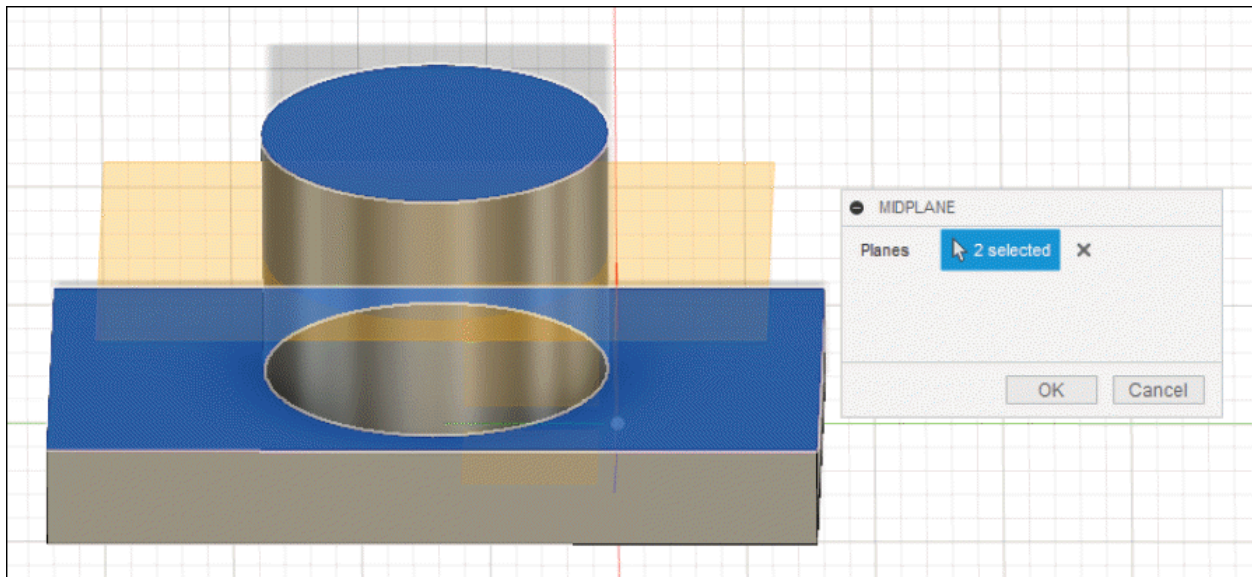


Figure-84. Preview of midplane

- Click on the **OK** button from the dialog box. The plane will be created.

Plane Through Two Edges

The **Plane Through Two Edges** tool is used to create a plane with the reference as two linear edges. The procedure to use this tool is discussed next.

- Click on the **Plane Through Two Edges** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-85](#). The tool will be activated and the **PLANE THROUGH TWO EDGES** dialog box will be displayed; refer to [Figure-86](#).

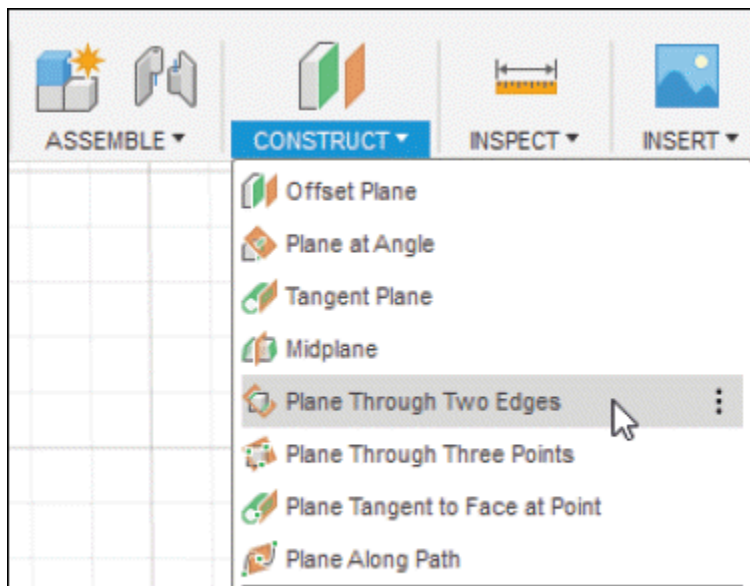


Figure-85. Plane Through Two Edges tool

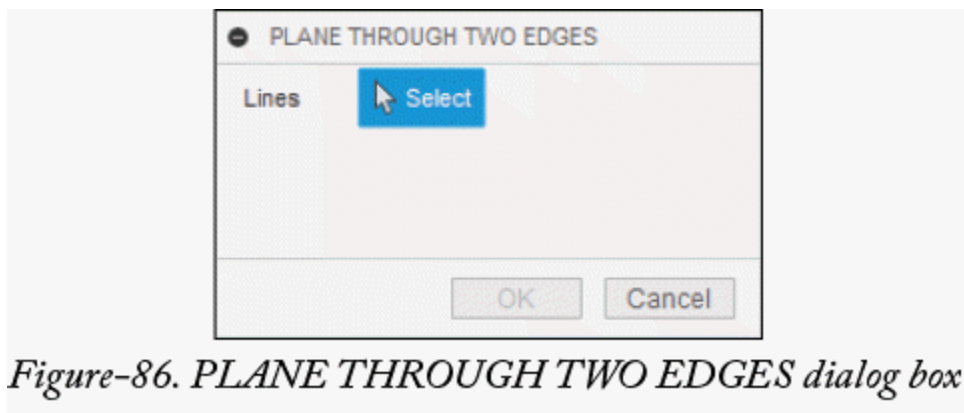


Figure-86. PLANE THROUGH TWO EDGES dialog box

- Click on two straight edges of model to create plane; refer to [Figure-87](#).

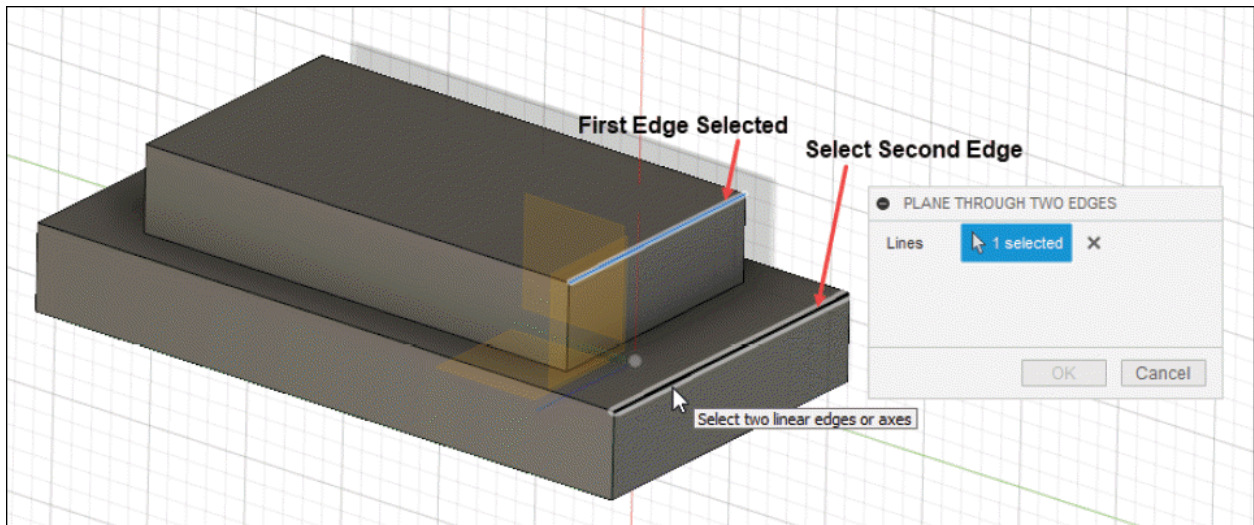


Figure-87. Selection of edges

- On selecting the second edge, the preview of plane will be displayed; refer to [Figure-88](#).

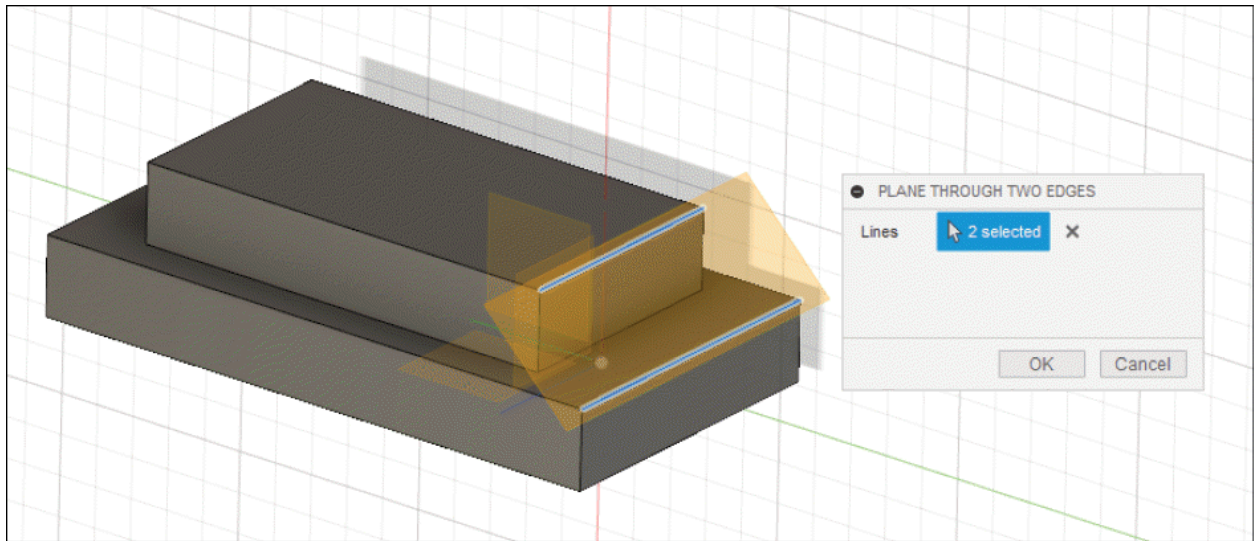


Figure-88. Preview of plane through two edges

- Click on the **OK** button from the dialog box to create the plane.

Plane Through Three Points

The **Plane Through Three Points** tool is used to create a plane with the help of three points as reference. The procedure to use this tool is discussed next.

- Click on the **Plane Through Three Points** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-89](#). The **PLANE THROUGH THREE POINTS** dialog box will be displayed; refer to [Figure-90](#).

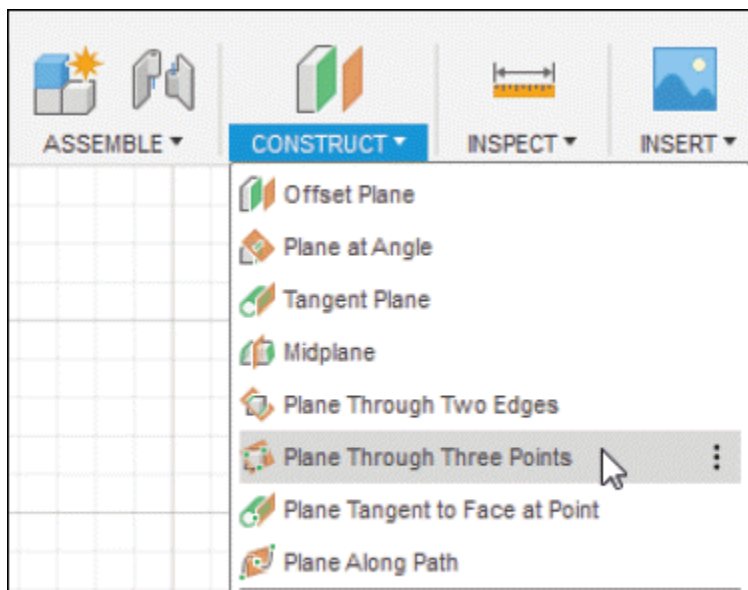


Figure-89. Plane Through Three Points tool

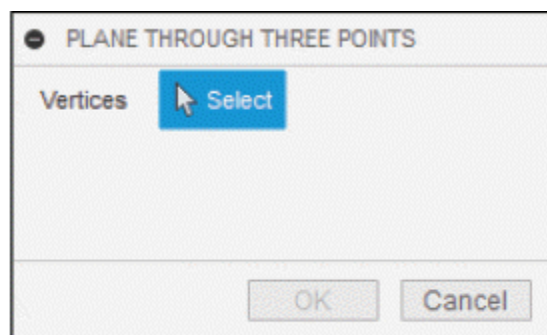


Figure-90. PLANE THROUGH THREE POINTS dialog box

- Select three points from model as reference to create a

plane; refer to [Figure-91](#).

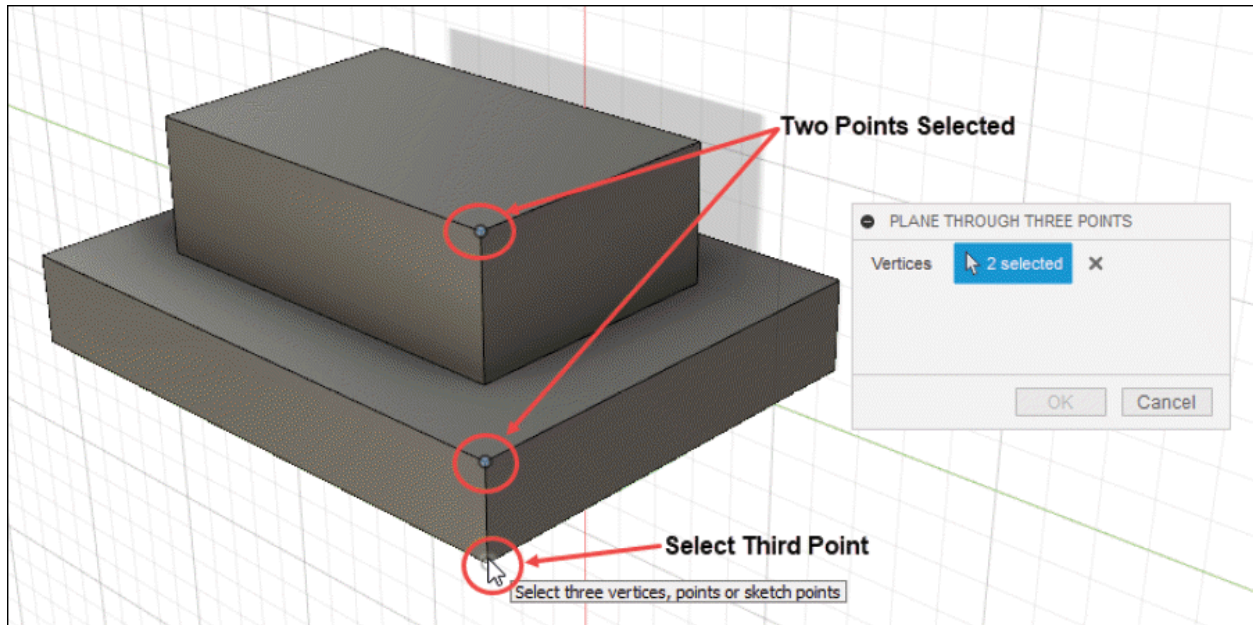


Figure-91. Selection of points for creating plane

- After selecting the third point, the preview of plane will be displayed along with model; refer to [Figure-92](#). Click on the **OK** button from the dialog box.

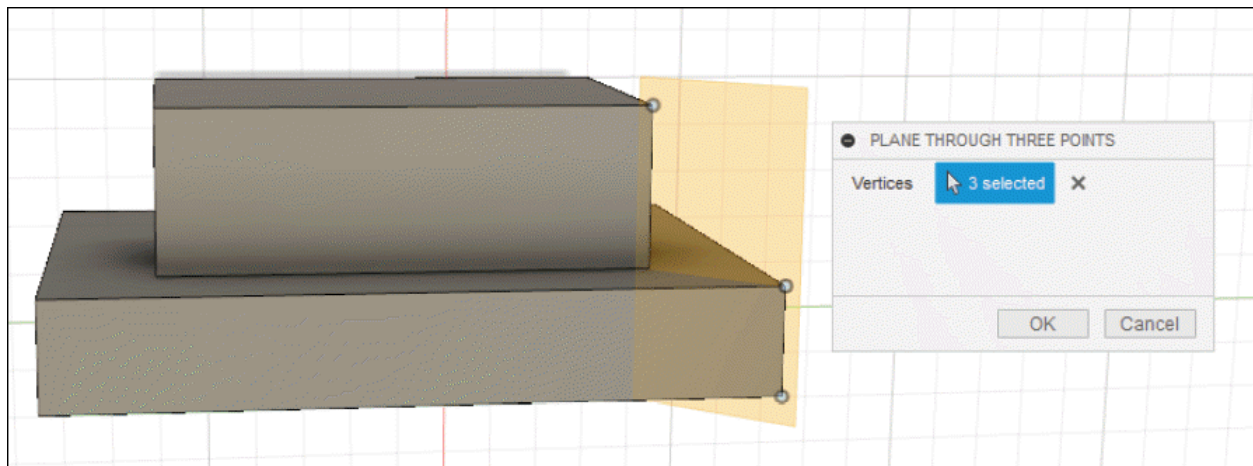


Figure-92. Preview of plane through three points

Plane Tangent to Face at Point

The **Plane Tangent to Face at Point** tool is used to create

a plane which is tangent to selected face and aligned to selected point. The procedure to use this tool is discussed next.

- Click on the **Plane Tangent to Face at Point** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-93](#). The **PLANE TANGENT TO FACE AT POINT** dialog box will be displayed; refer to [Figure-94](#).

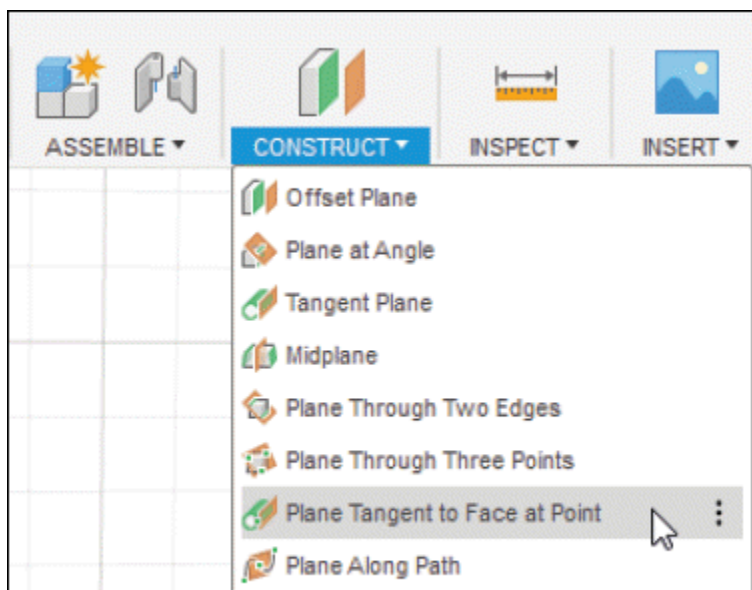


Figure-93. Plane Tangent to Face at Point tool

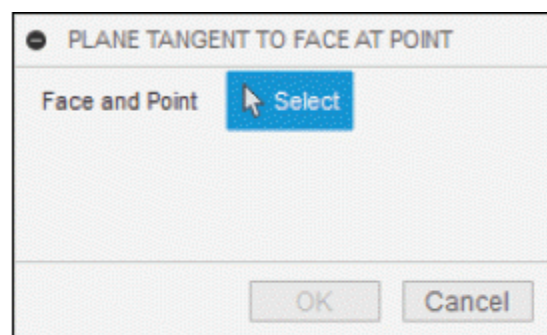


Figure-94. PLANE TANGENT TO FACE AT POINT dialog box

- Click on the model to select a point and face; refer to

[Figure-95](#). You can select round as well as flat faces in this case.

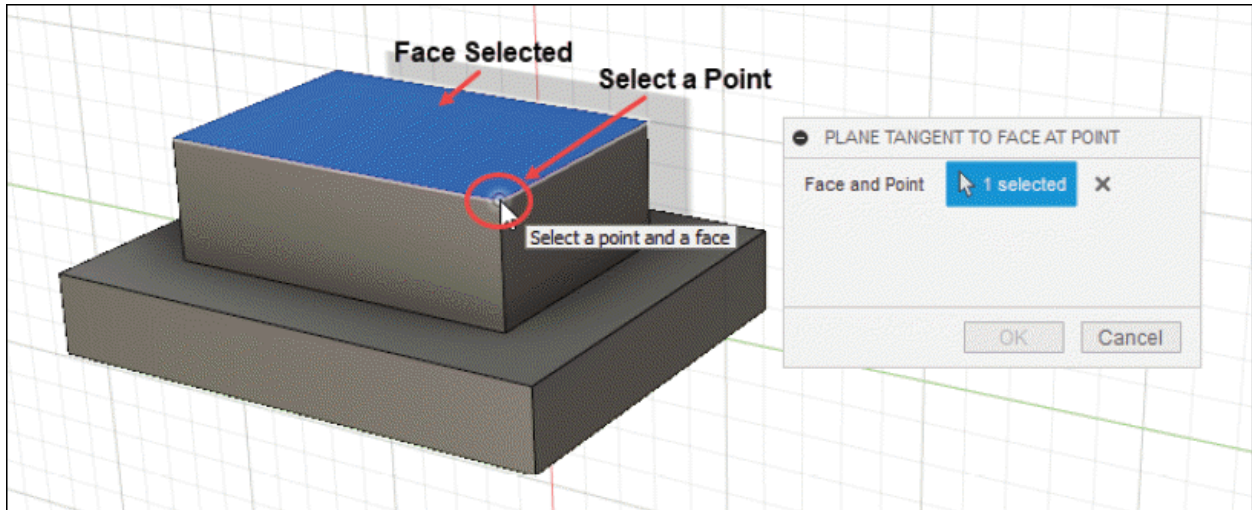


Figure-95. Selection of point and face

- On selecting the second face or point, preview of plane will be displayed; refer to [Figure-96](#). Click on the **OK** button from the dialog box to create the plane.

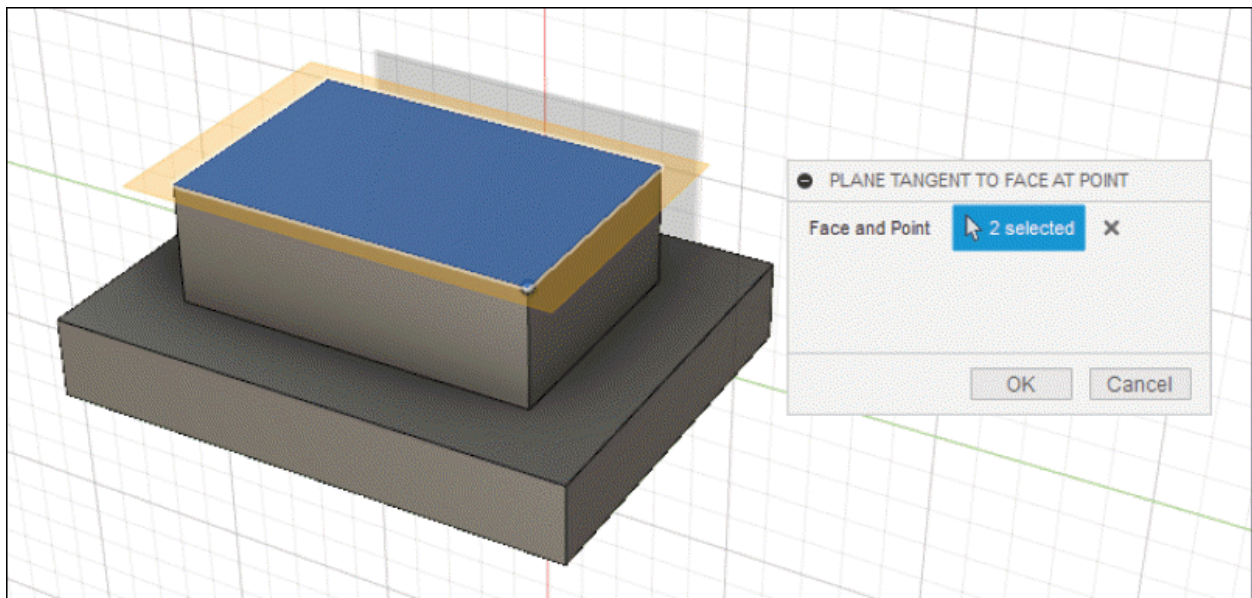


Figure-96. Preview of plane tangent to face at point

Plane Along Path

The **Plane Along Path** tool is used to create a plane along selected path. The plane will be perpendicular to the path at selected point. The procedure to use this tool is discussed next.

- Click on the **Plane Along Path** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-97](#). The **PLANE ALONG PATH** dialog box will be displayed; refer to [Figure-98](#).

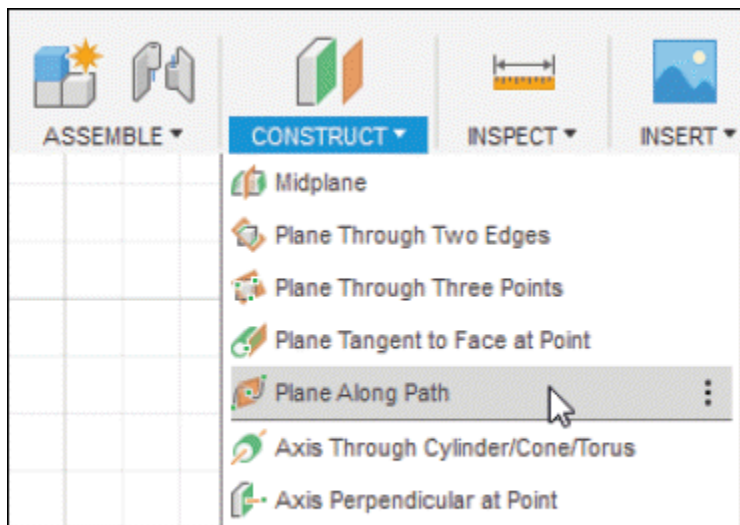


Figure-97. Plane Along Path tool

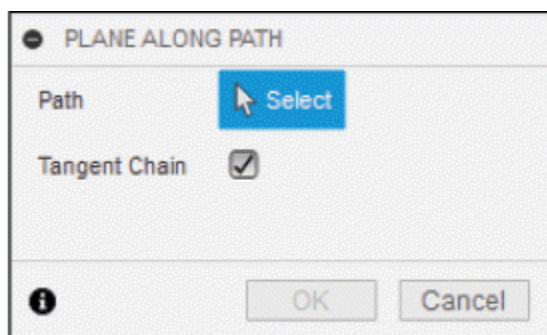


Figure-98. PLANE ALONG PATH dialog box

- **Path** selection of **PLANE ALONG PATH** dialog box is

active by default. Click on the edge/curve from model as reference for plane; refer to [Figure-99](#).

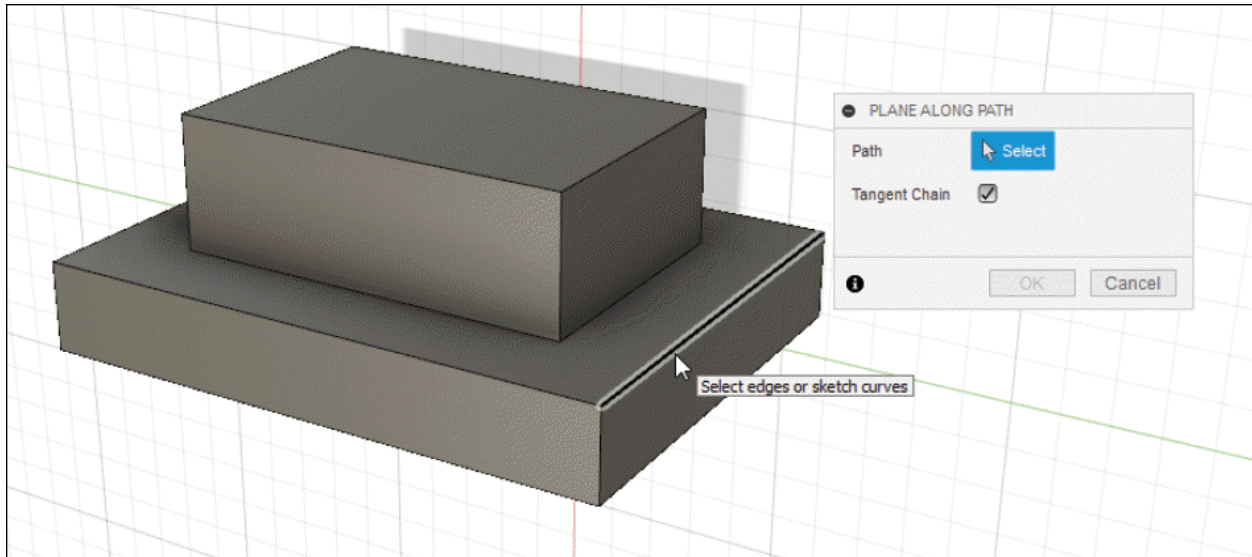


Figure-99. Selection of path

- On selecting the path, the updated **PLANE ALONG PATH** dialog box will be displayed along with the preview of plane; refer to [Figure-100](#).

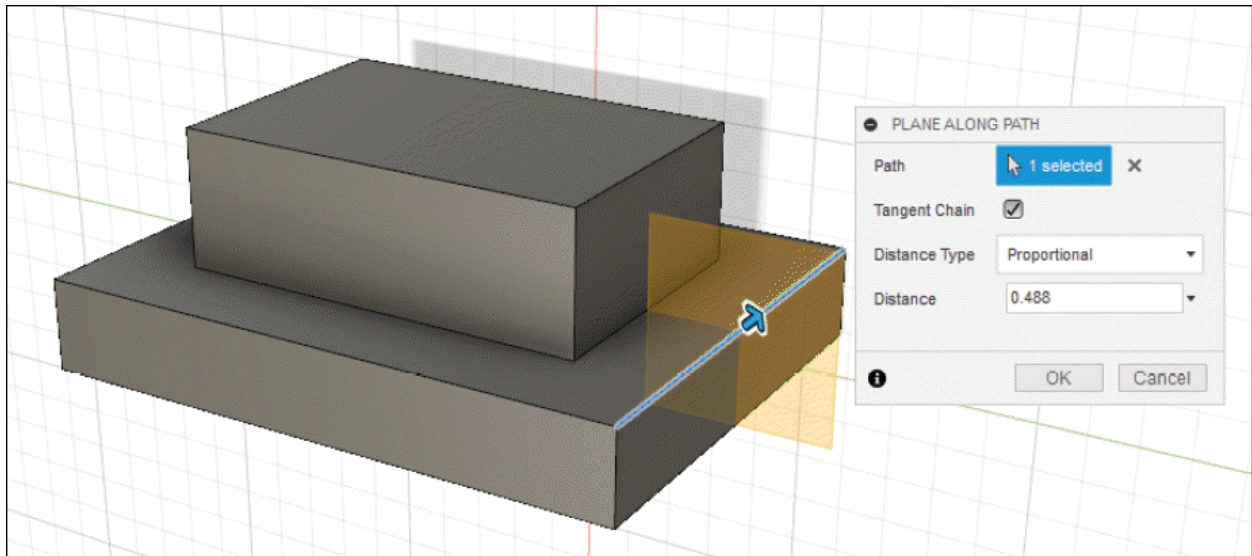


Figure-100. Preview of plane along path

- Click in the **Distance** edit box and specify the distance of

plane in percentage between 0 to 1. You can also adjust the distance by moving the drag handle.

- After specifying the parameters, click on the **OK** button from **PLANE ALONG PATH** dialog box. The plane will be created.

Till now, we have created plane through different references. In next section, we will create axis through different references.

Axis Through Cylinder/Cone/Torus

The **Axis Through Cylinder/Cone/Torus** tool is used to create axis through center of cylinder, cone, or torus body. The procedure to use this tool is discussed next.

- Click on the **Axis Through Cylinder/Cone/Torus** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-101](#). The **AXIS THROUGH CYLINDER/CONE/TORUS** dialog box will be displayed; refer to [Figure-102](#).

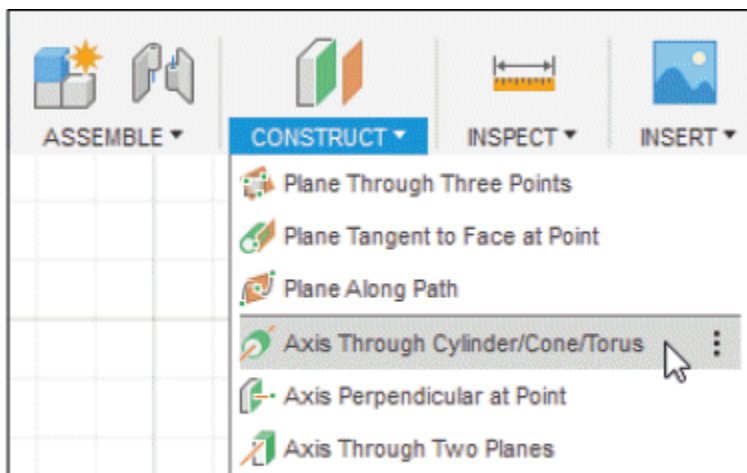


Figure-101. Axis Through Cylinder Cone Torus tool

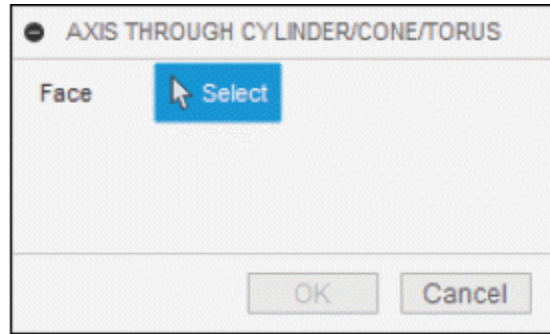


Figure-102. AXIS THROUGH CYLINDER CONE TORUS dialog box

- Click on the round face of cylinder, cone, or torus from model, preview of the axis will be displayed; refer to [Figure-103](#). Click on the **OK** button from the dialog box to create the axis.

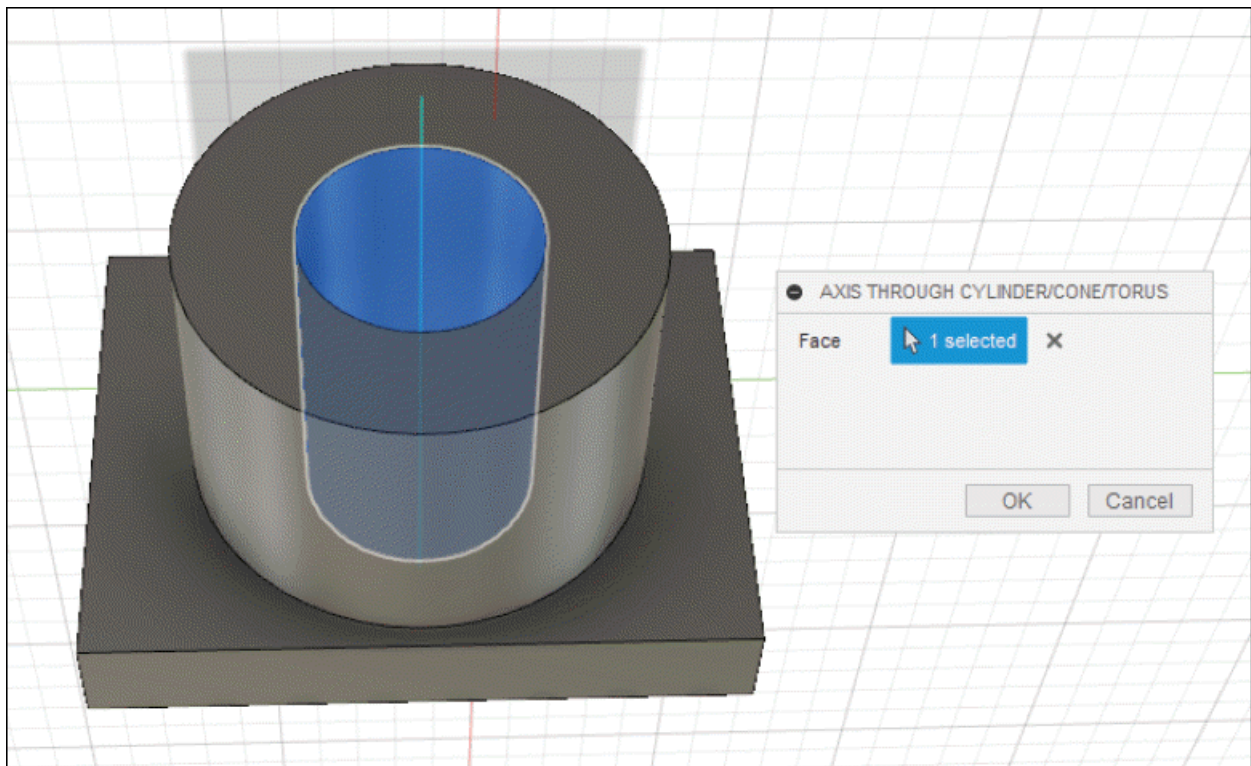


Figure-103. Preview of axis of cylinder

Axis Perpendicular at Point

The **Axis Perpendicular at Point** tool is used to create an

axis perpendicular to the selected face at selected point. The procedure to use this tool is discussed next.

- Click on the **Axis Perpendicular at Point** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-104](#). The tool will be activated and **AXIS PERPENDICULAR AT POINT** dialog box will be displayed; refer to [Figure-105](#).

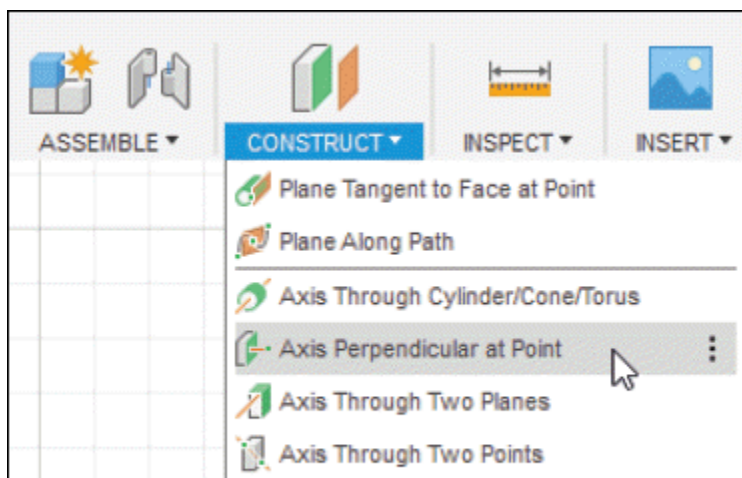


Figure-104. Axis Perpendicular at Point tool

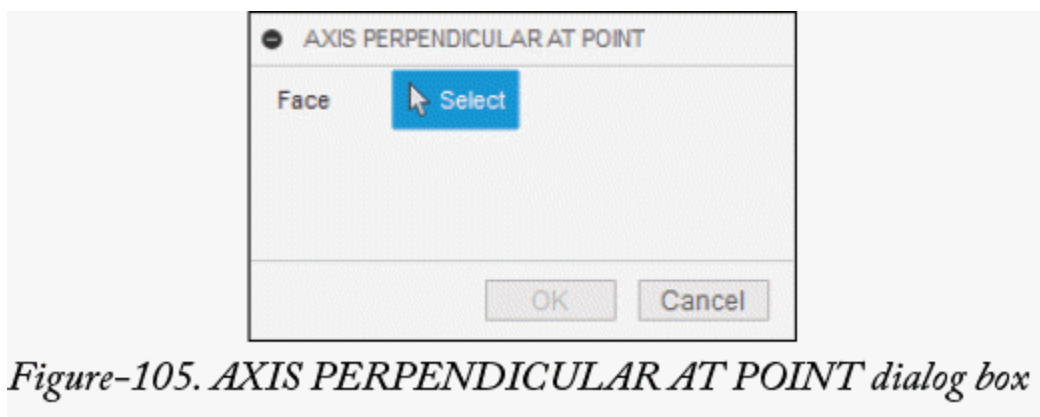


Figure-105. AXIS PERPENDICULAR AT POINT dialog box

- Click at desired location on the face of model; refer to [Figure-106](#).

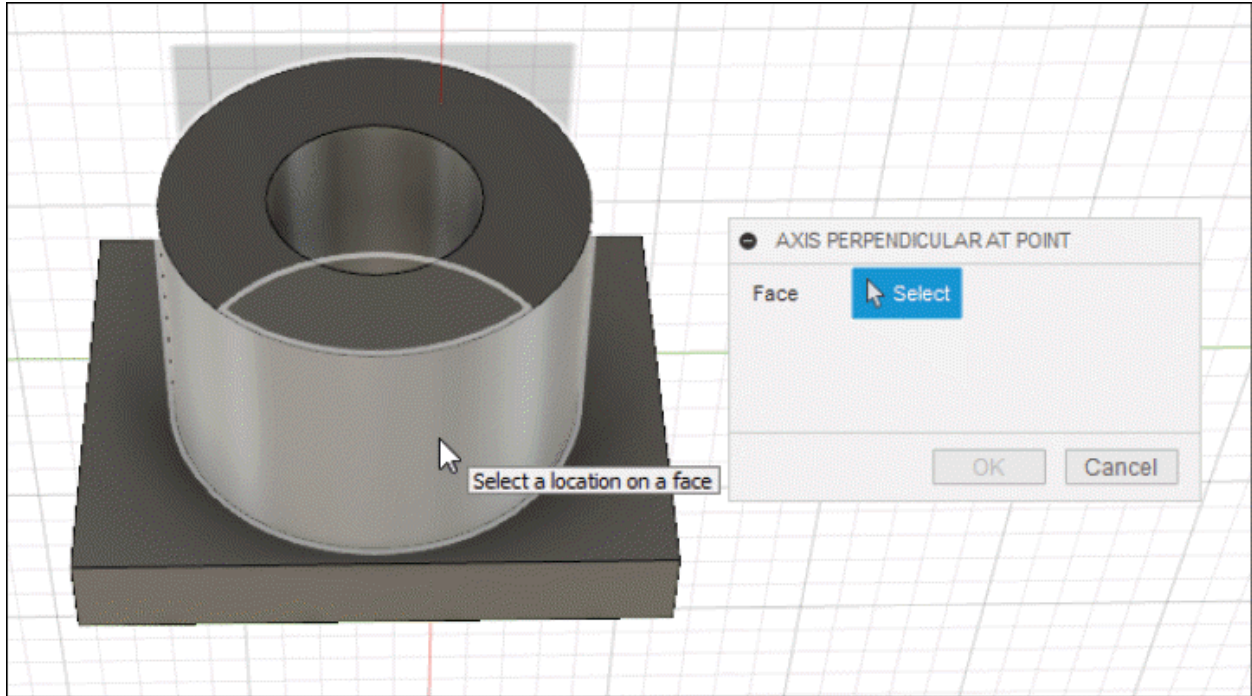


Figure-106. Selection of face for axis

- After selecting the point on any face, preview of axis will be displayed; refer to [Figure-107](#). Click on the **OK** button from the dialog box to create axis.

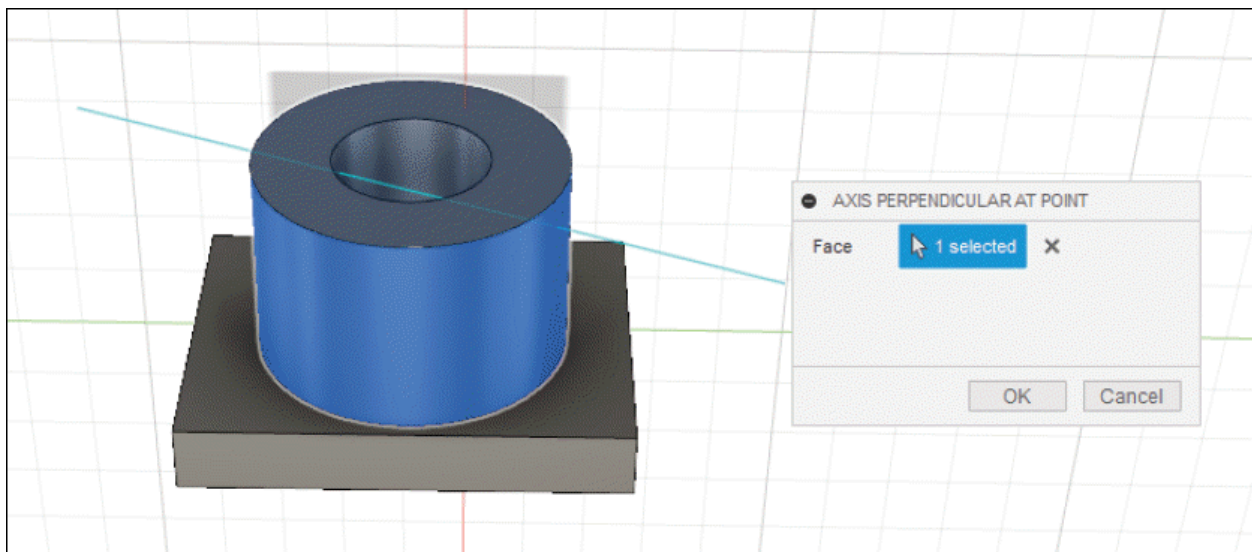


Figure-107. Preview of axis perpendicular at point

Axis Through Two Planes

The **Axis Through Two Planes** tool is used to create an axis at the intersection of two planes or planar faces. The procedure to use this tool is discussed next.

- Click on the **Axis Through Two Planes** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-108](#). The **AXIS THROUGH TWO PLANES** dialog box will be displayed; refer to [Figure-109](#).

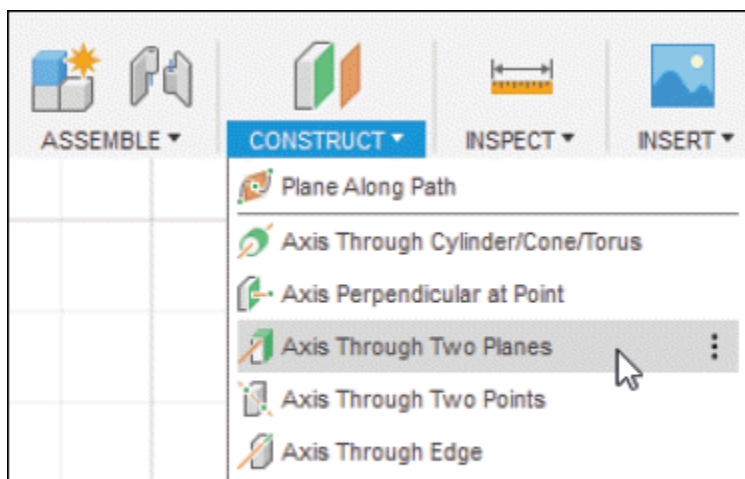


Figure-108. Axis Through Two Planes tool

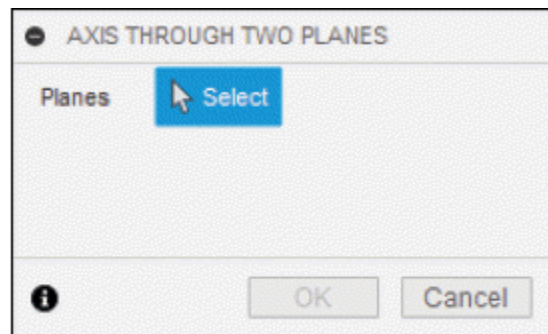


Figure-109. AXIS THROUGH TWO PLANES dialog box

- Planes selection button of **AXIS THROUGH TWO PLANES** dialog box is active by default. You need to select two planes to create an axis; refer to [Figure-110](#).

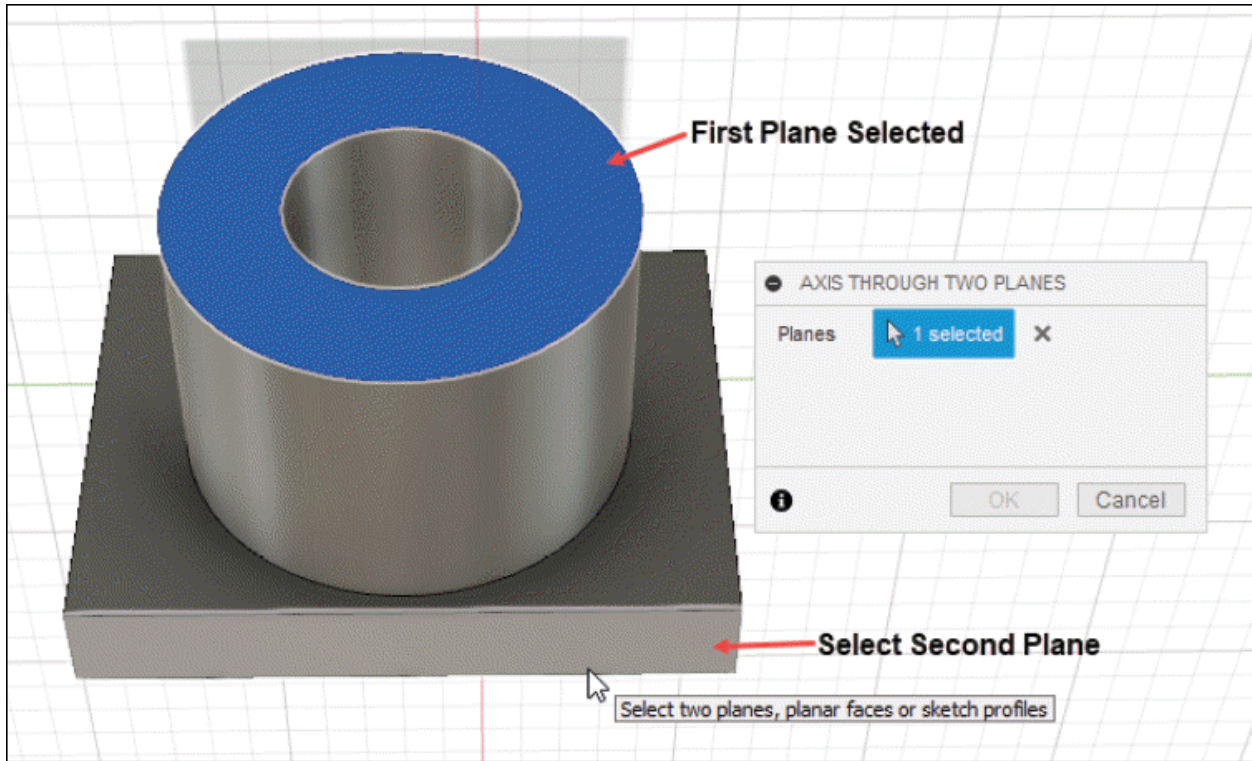


Figure-110. Selection of planes for creating axis

- On selecting the second plane, preview of axis will be displayed; refer to [Figure-111](#).

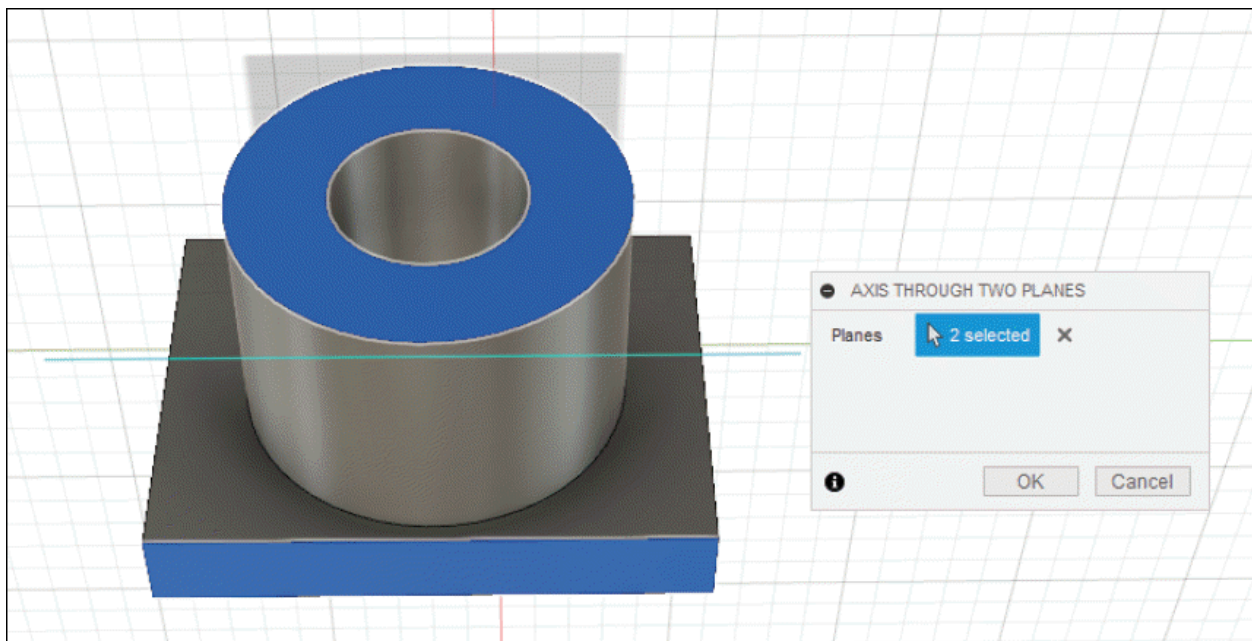


Figure-111. Preview of axis through two planes

- Click on the **OK** button from the dialog box to create the axis.

Axis Through Two Points

The **Axis Through Two Points** tool is used to create an axis passing through two reference points. The procedure to use this tool is discussed next.

- Click on the **Axis Through Two Points** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-112](#). The tool will be activated and **AXIS THROUGH TWO POINTS** dialog box will be displayed; refer to [Figure-113](#).

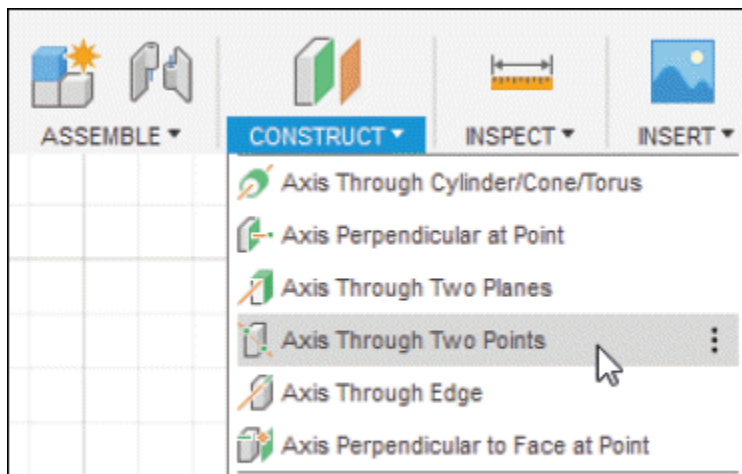


Figure-112. Axis Through Two Points tool

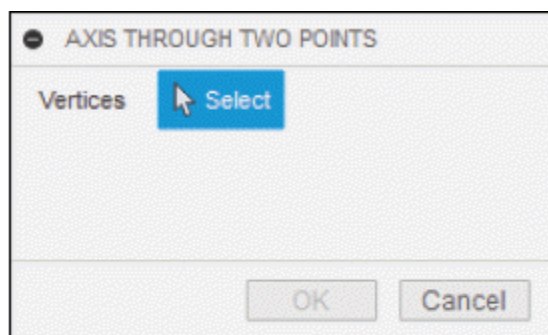


Figure-113. AXIS THROUGH TWO POINTS dialog box

- Select the two points from model; refer to [Figure-114](#). Preview of axis will be displayed. Click on the **OK** button from the dialog box to create the axis.

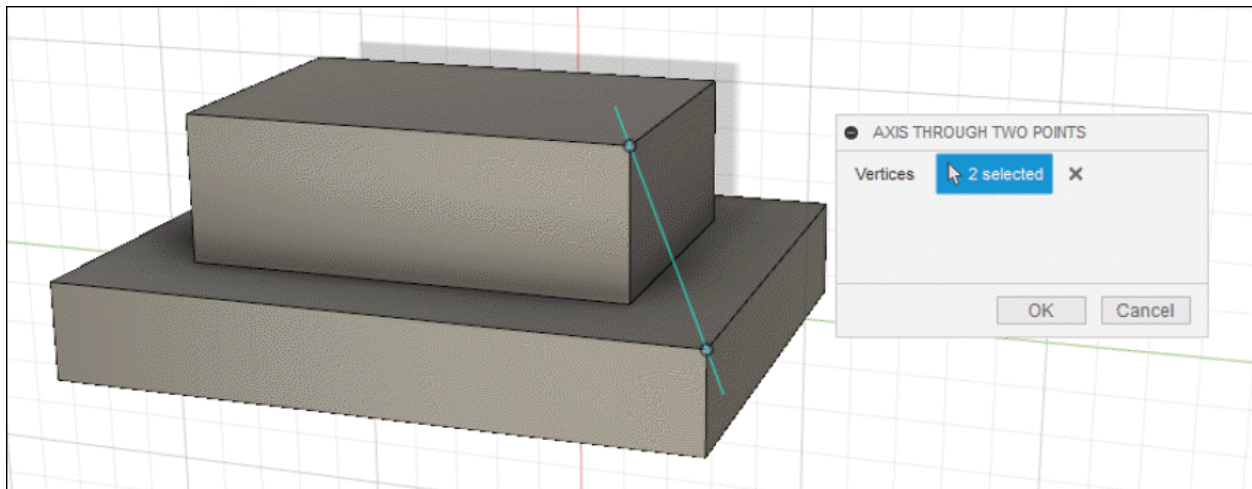


Figure-114. Preview of axis through two points

Axis Through Edge

The **Axis Through Edge** tool is used to create an axis on the selected edge/axis. The procedure to use this tool is discussed next.

- Click on the **Axis Through Edge** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-115](#). The tool will be activated and **AXIS THROUGH EDGE** dialog box will be displayed; refer to [Figure-116](#).

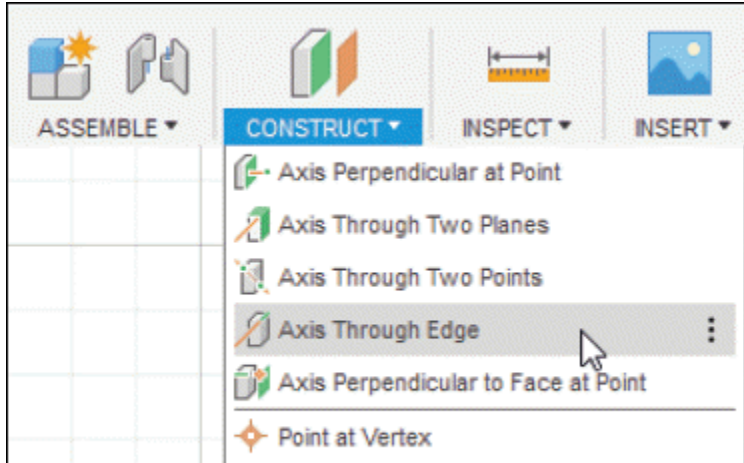


Figure-115. Axis Through Edge tool

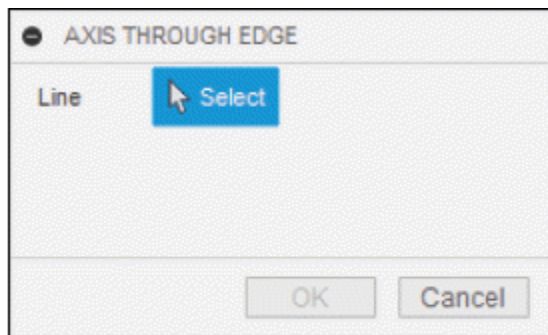


Figure-116. AXIS THROUGH EDGE dialog box

- Click on desired edge of model to create the axis; refer to [Figure-117](#). Preview of axis will be displayed.

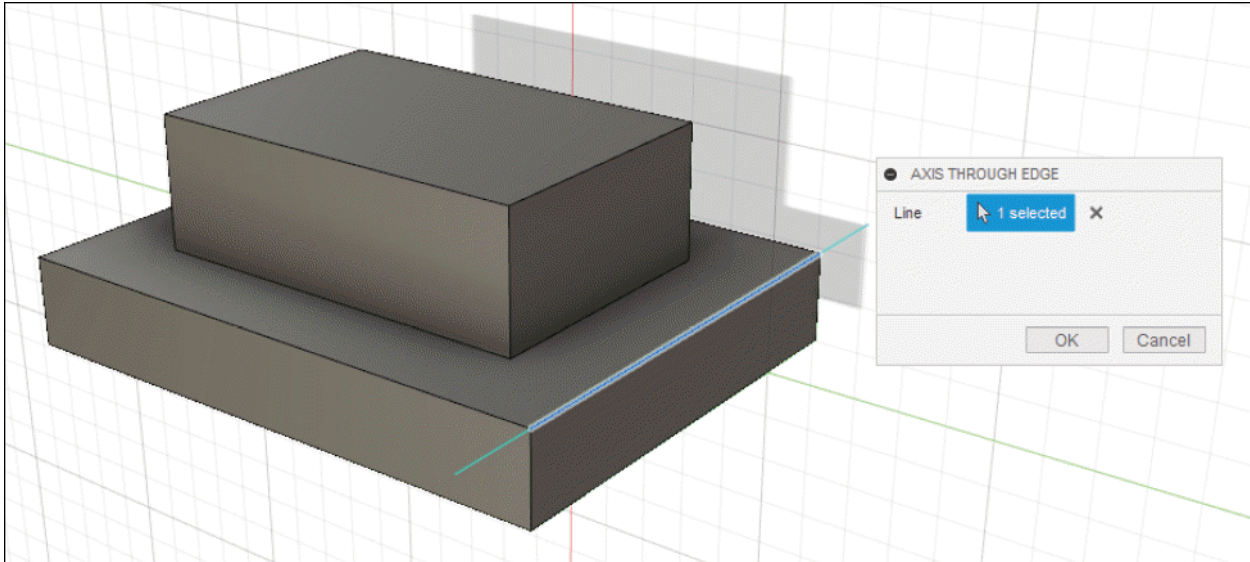


Figure-117. Preview of axis through edge

- Click on the **OK** button from the dialog box to create the axis.

Axis Perpendicular to Face at Point

The **Axis Perpendicular to Face at Point** tool is used to create an axis by using a face and a point as reference. The procedure to use this point is discussed next.

- Click on the **Axis Perpendicular to Face at Point** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-118](#). The tool will be activated and **AXIS PERPENDICULAR TO FACE AT POINT** dialog box will be displayed; refer to [Figure-119](#).

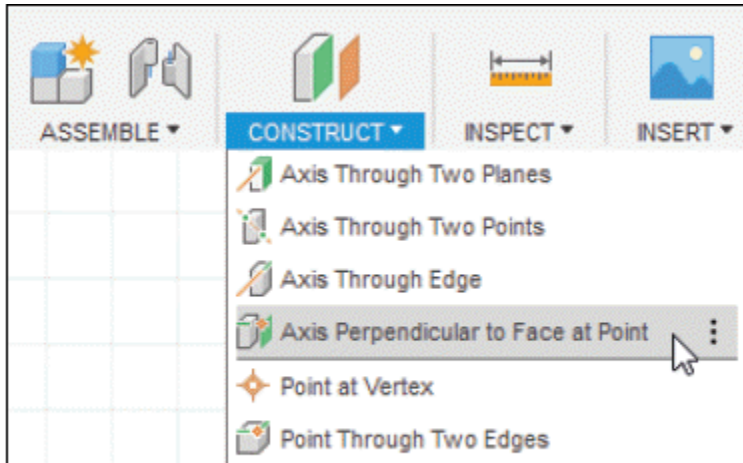


Figure-118. Axis Perpendicular to Face at Point tool

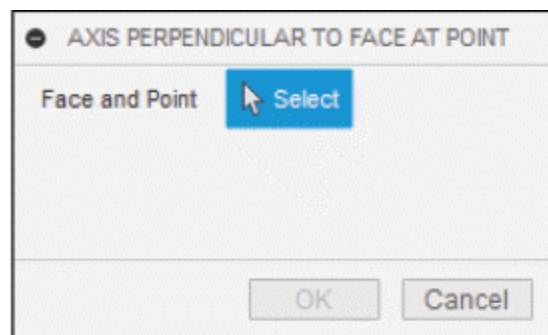


Figure-119. AXIS PERPENDICULAR TO FACE AT POINT dialog box

- Click on the model to select the point and face as a reference for creating plane; refer to [Figure-120](#).

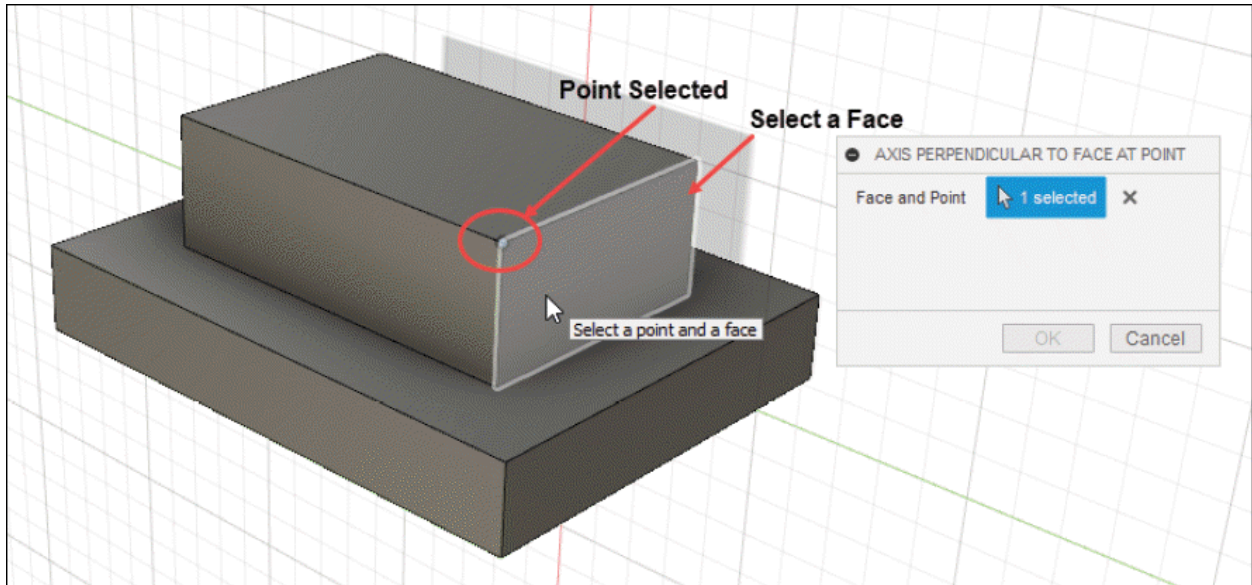


Figure-120. Selection of point and face for creating axis

- On selecting the face, preview of axis will be displayed; refer to [Figure-121](#).

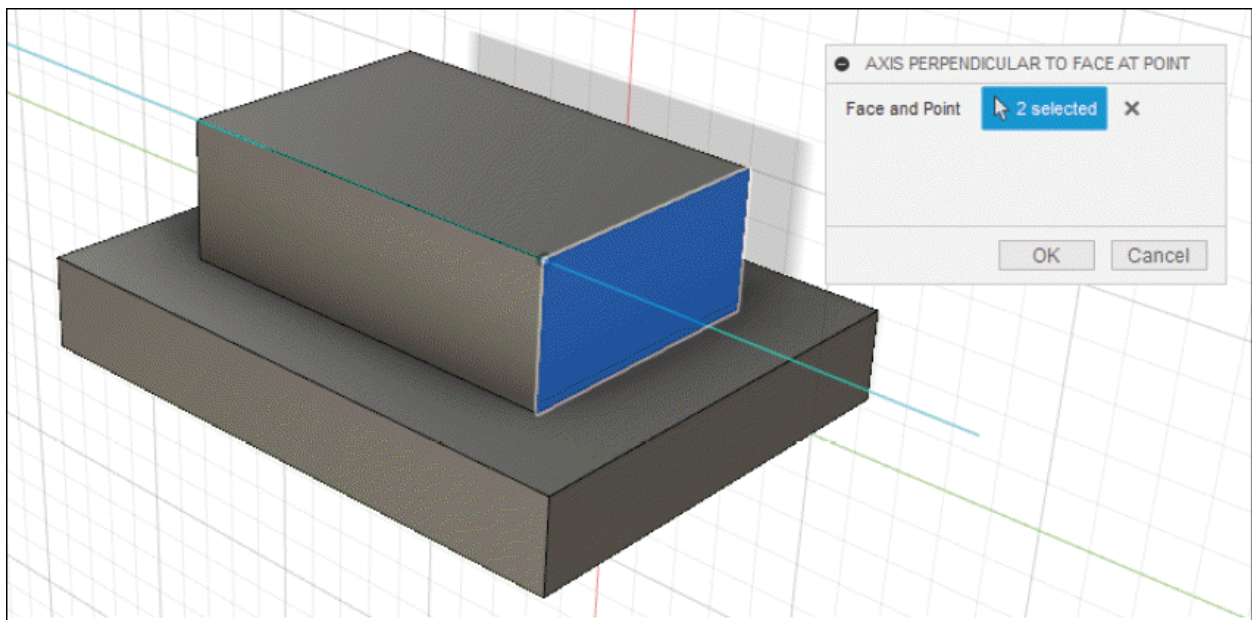


Figure-121. Preview of axis perpendicular to face at point

Note that axis will always be created passing through the point.

Till now, we have learned to create plane and axis. In next topics, we will learn the procedure to create points. Points can be used as origin, coordinate system location, or reference for other features. The tools to create points are discussed next.

Point at Vertex

The **Point at Vertex** tool is used to create a point by selecting any vertex of the model. The procedure to use this tool is discussed next.

- Click on the **Point at Vertex** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-122](#). The tool will be activated and **POINT AT VERTEX** dialog box will be displayed; refer to [Figure-123](#).

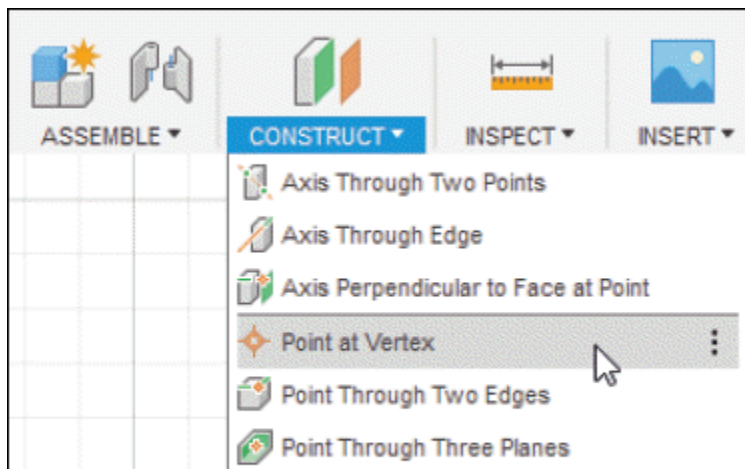


Figure-122. Point at Vertex tool

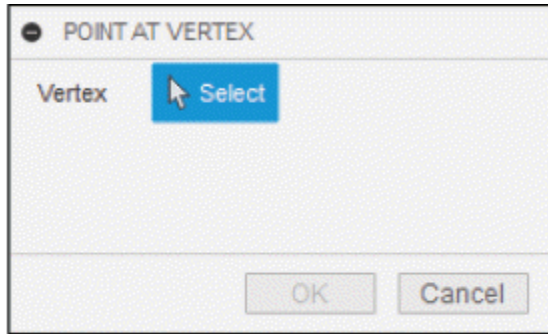


Figure-123. POINT AT VERTEX dialog box

- Click on the vertex of model to create a constructional point; refer to [Figure-124](#).

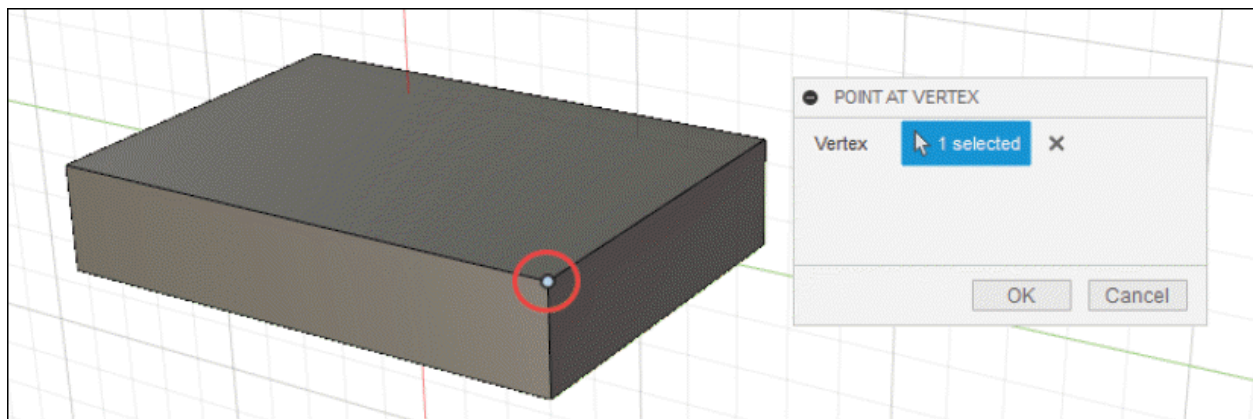


Figure-124. Preview of point at vertex

Note that you can create only one point at a time using this tool.

- Click on the **OK** button from the dialog box to create point.

Point Through Two Edges

The **Point Through Two Edges** tool is used to create a point at intersection of two edges/axes intersecting each other. The procedure to use this tool is discussed next.

- Click on the **Point Through Two Edges** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-](#)

[125](#). The tool is activated and **POINT THROUGH TWO EDGES** dialog box will be displayed; refer to [Figure-126](#).

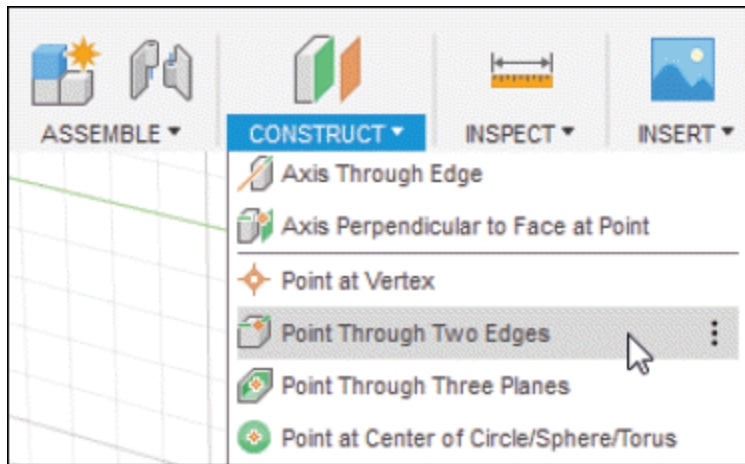


Figure-125. Point Through Two Edges tool

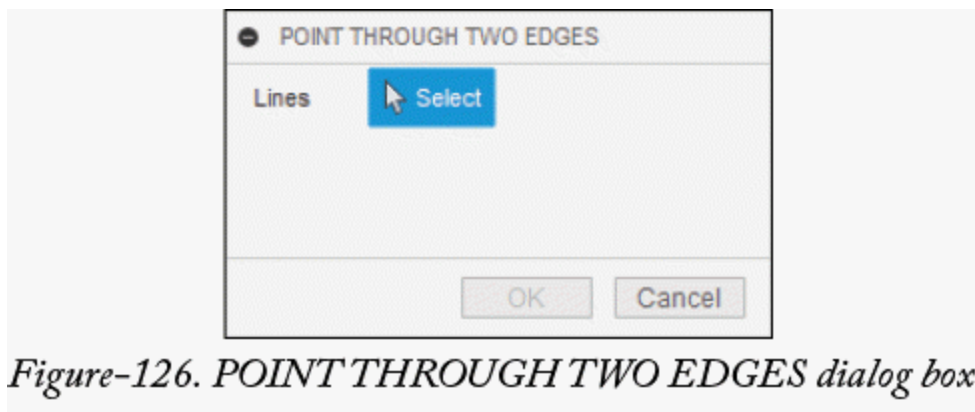


Figure-126. POINT THROUGH TWO EDGES dialog box

- Select the two intersecting lines on the model to create point; refer to [Figure-127](#).

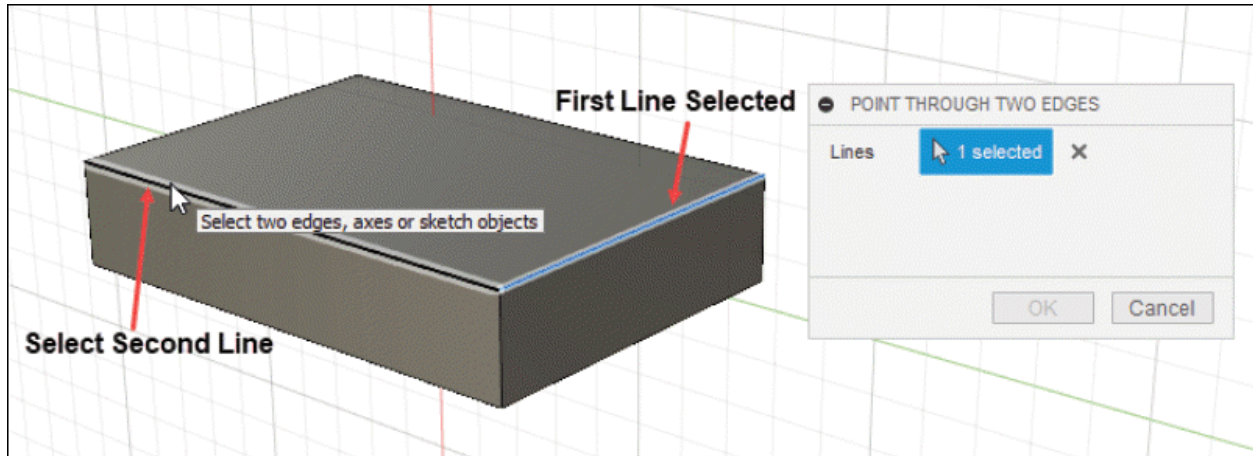


Figure-127. Selection of lines to create point

- On selecting the second line, preview of point will be displayed. Click on the **OK** button from the dialog box to create the point; refer to [Figure-128](#).

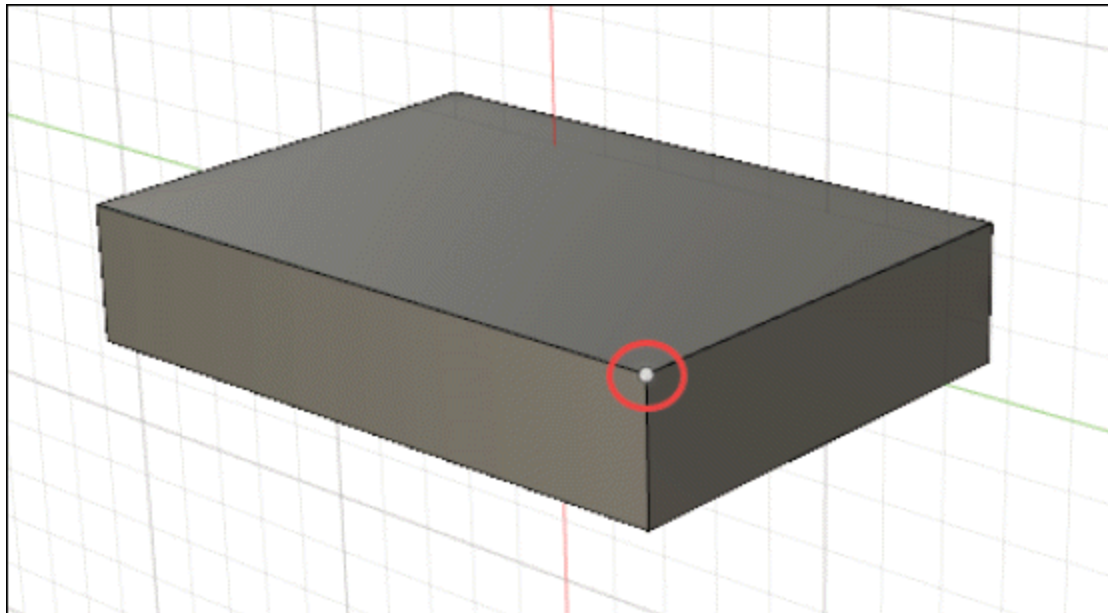


Figure-128. Point created through two edges

Point Through Three Planes

The **Point Through Three Planes** tool is used to create a point at the intersection of three planes/faces. The

procedure to use this tool is discussed next.

- Click on the **Point Through Three Planes** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-129](#). The tool will be activated and **POINT THROUGH THREE PLANES** dialog box will be displayed; refer to [Figure-130](#).

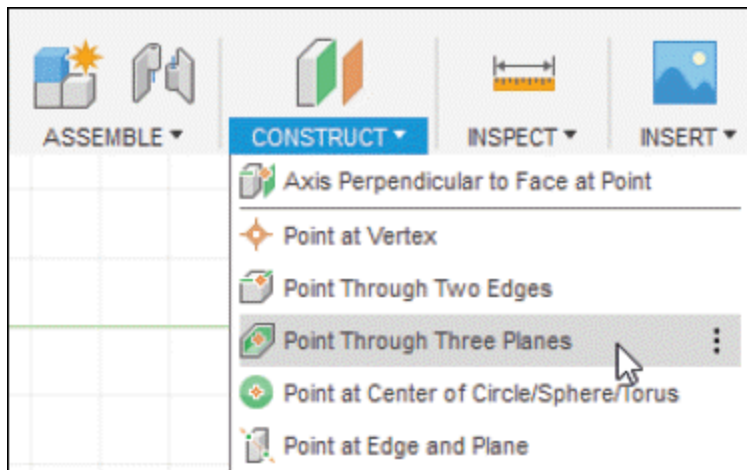


Figure-129. Point Through Three Planes tool

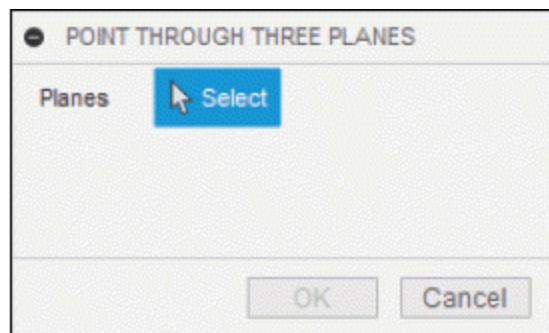


Figure-130. POINT THROUGH THREE PLANES dialog box

- Select the three intersecting planes/faces from the model; refer to [Figure-131](#).

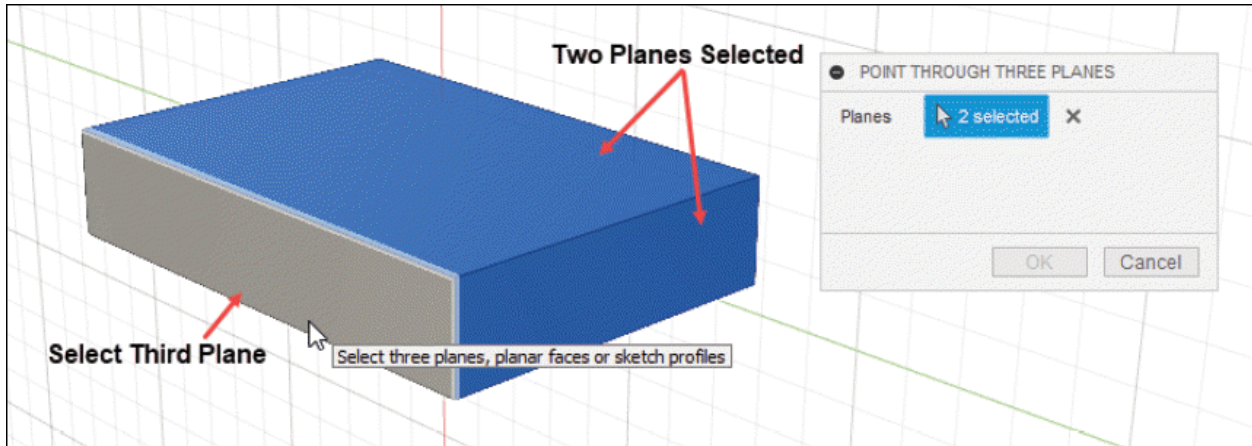


Figure-131. Selection of three planes to create point

- On selecting the third plane, preview of the constructional point will be displayed; refer to [Figure-132](#).

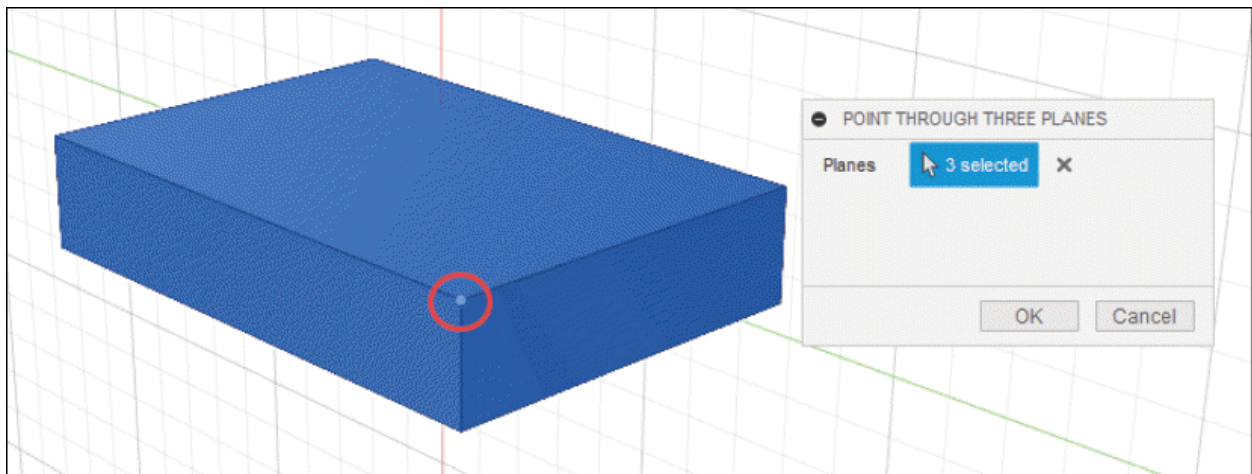


Figure-132. Preview of point through three planes

- Click on the **OK** button from the dialog box to create the point.

Point at Center of Circle/Sphere/Torus

The **Point at Center of Circle/Sphere/Torus** tool is used to create a point at the center of selected circle/sphere/torus. The procedure to use this tool is discussed next.

- Click on the **Point at Center of Circle/Sphere/Torus** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-133](#). The tool will be activated and respective dialog box will be displayed; refer to [Figure-134](#).

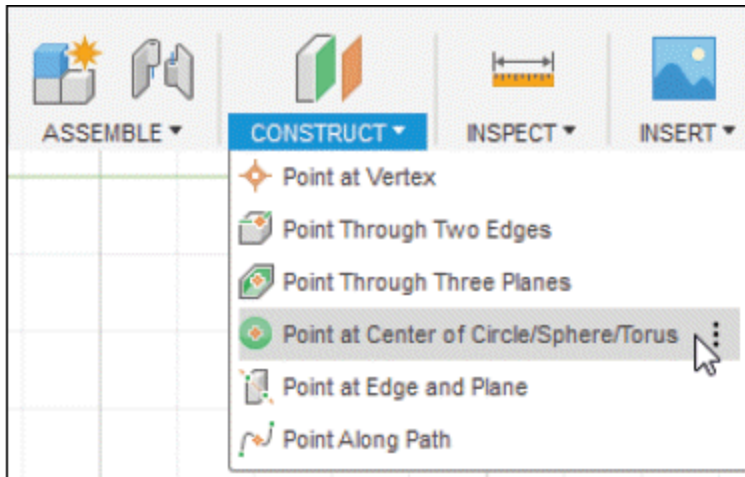


Figure-133. Point at Center of Circle Sphere Torus tool

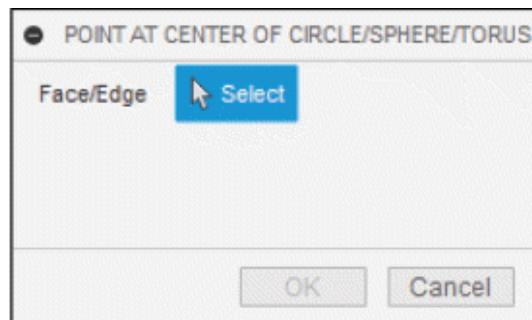


Figure-134. POINT AT CENTER OF CIRCLE SPHERE TORUS dialog box

- Select the circle, or round face of sphere/torus to create the center point. Preview of point will be displayed; refer to [Figure-135](#).

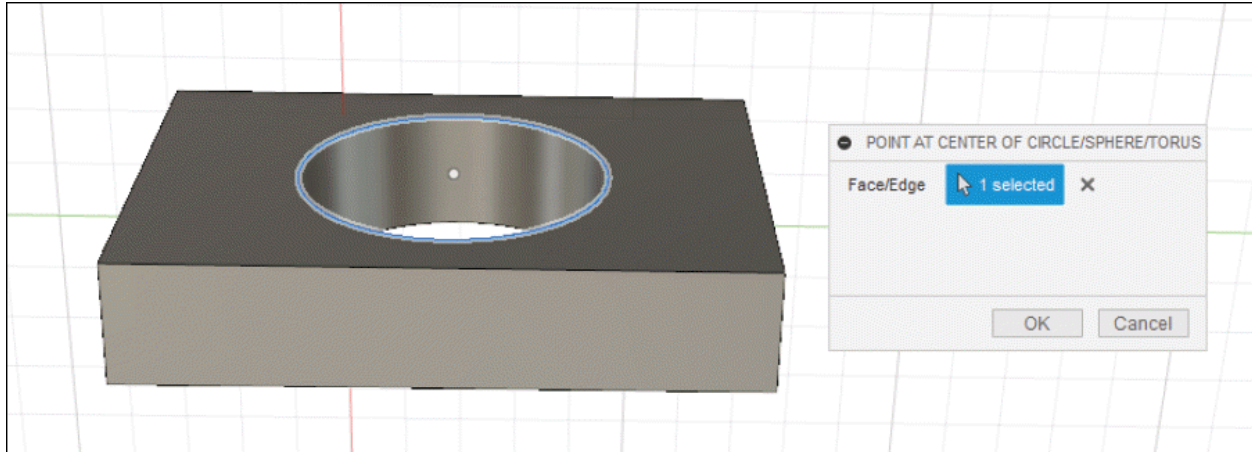


Figure-135. Preview of point at center

- Click on the **OK** button from the dialog box to create the point.

Point at Edge and Plane

The **Point at Edge and Plane** tool is used to create a point with the help of edge and plane as references. The procedure to use this tool is discussed next.

- Click on the **Point at Edge and Plane** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-136](#). The tool will be activated and respective dialog box will be displayed; refer to [Figure-137](#).

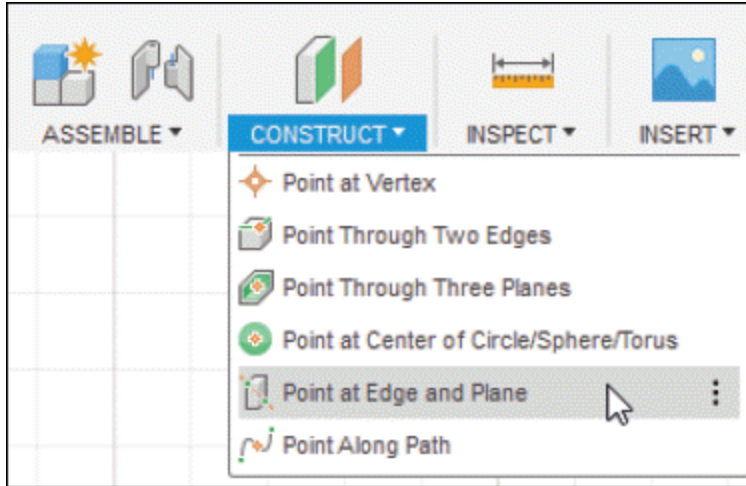


Figure-136. Point at Edge and Plane tool

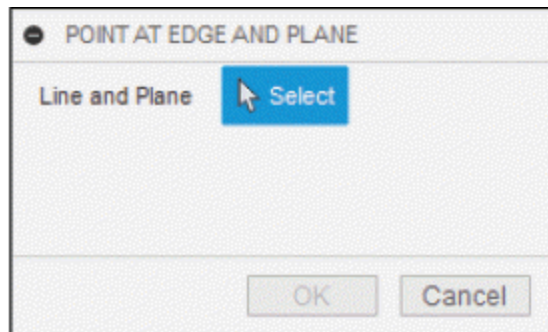


Figure-137. POINT AT EDGE AND PLANE dialog box

- Select the edge and plane/face to create a point at the intersection; refer to [Figure-138](#).

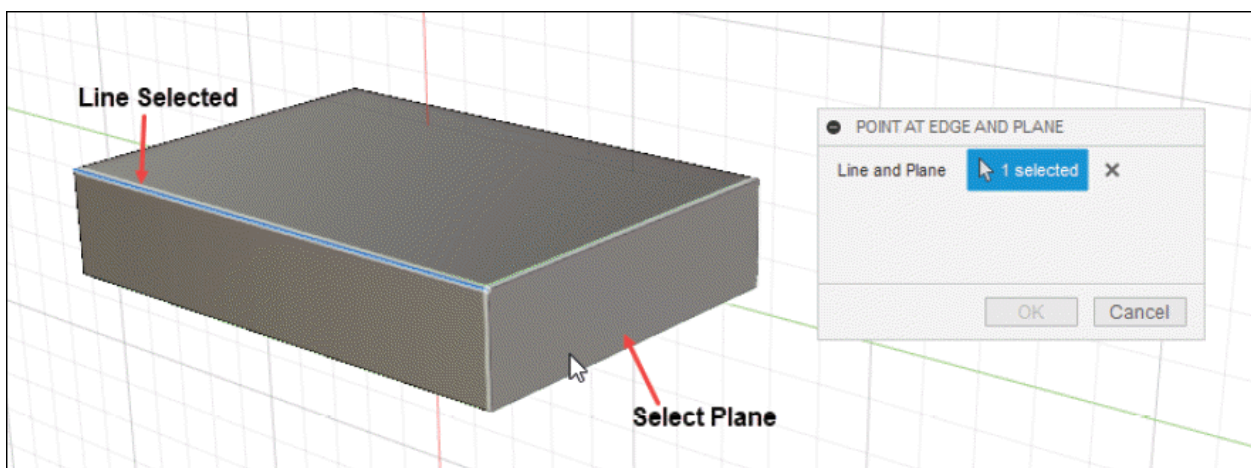


Figure-138. Selection of edge and plane

- On selecting the plane, preview of point will be displayed. Click on the **OK** button from the dialog box to create the point; refer to [Figure-139](#).

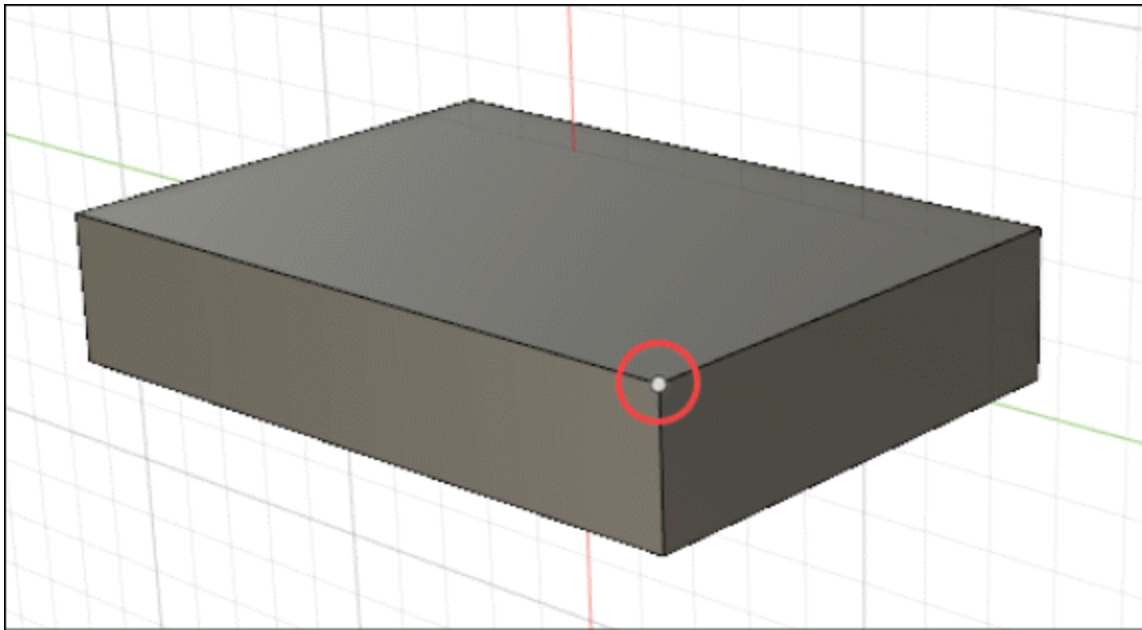


Figure-139. Point at edge and plane created

Point Along Path

The **Point Along Path** tool is used to create a point along selected path. The procedure to use this tool is discussed next.

- Click on the **Point Along Path** tool of **CONSTRUCT** drop-down from **Toolbar**; refer to [Figure-140](#). The **POINT ALONG PATH** dialog box will be displayed; refer to [Figure-141](#).

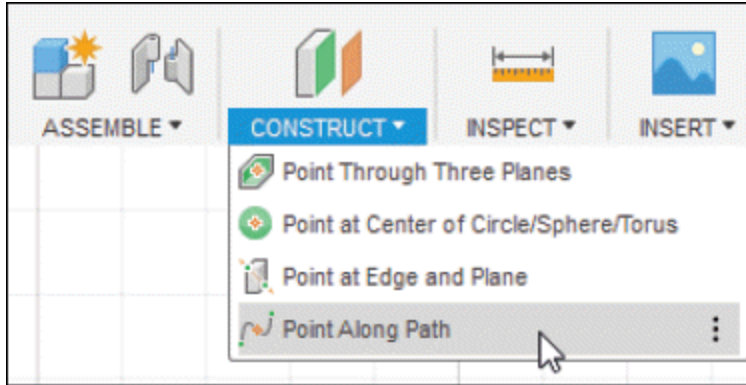


Figure-140. Point Along Path tool

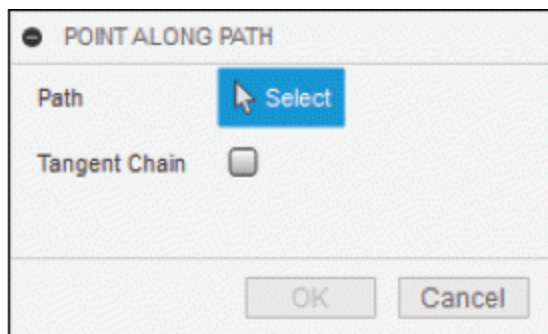


Figure-141. POINT ALONG PATH dialog box

- **Path** selection of **POINT ALONG PATH** dialog box is active by default. Click on the edge/curve from model as reference for point. On selecting the path, the updated **POINT ALONG PATH** dialog box will be displayed; refer to [Figure-142](#).

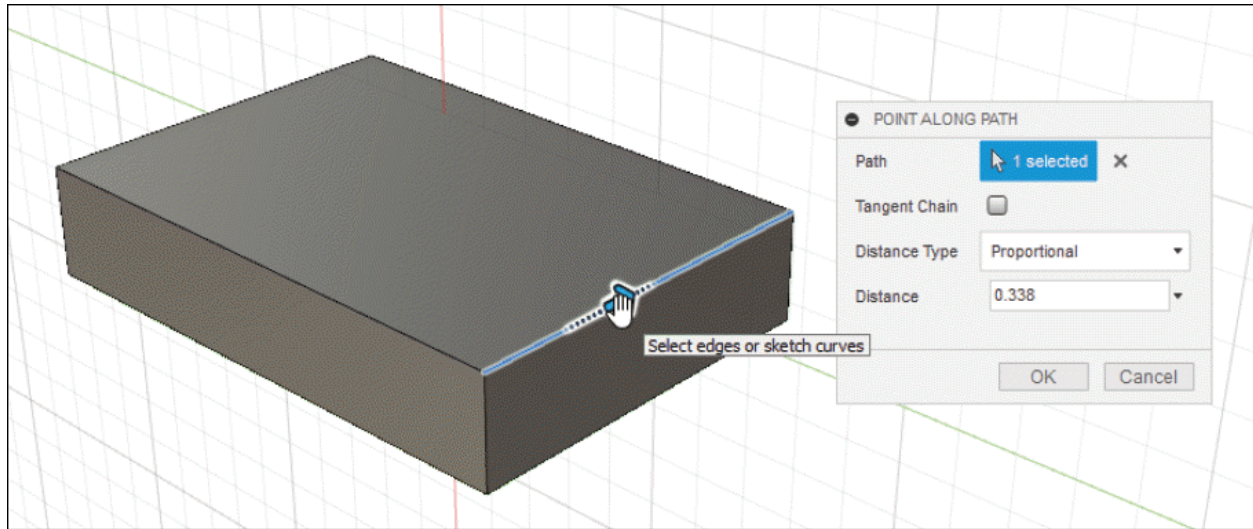


Figure-142. Selection of path to create point

- Click in the **Distance** edit box and specify the distance of point in percentage between 0 to 1. You can also adjust the distance by moving the drag handle.
- After specifying the parameters, click on the **OK** button from **POINT ALONG PATH** dialog box. The point will be created; refer to [Figure-143](#).

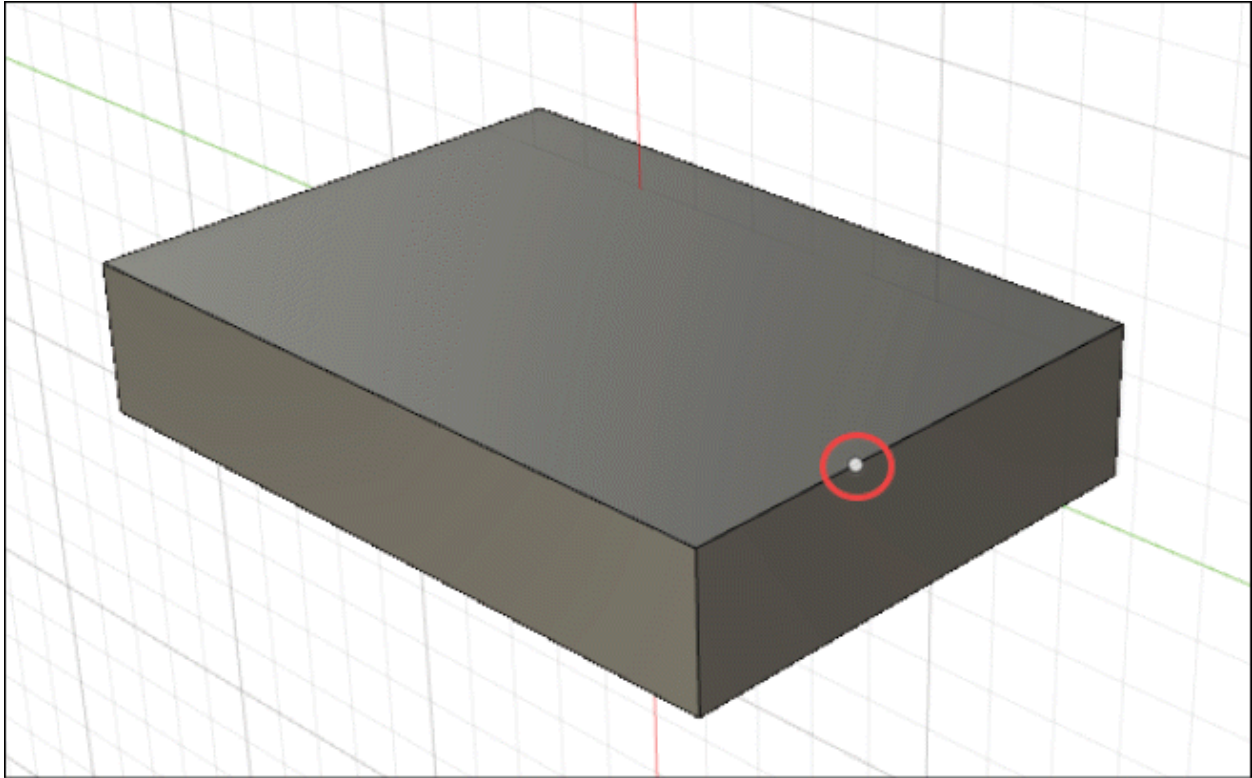


Figure-143. Point along path created

PROJECT/INCLUDE TOOLS

There are various tools in **Project/Include** cascading menu of **CREATE** drop-down in **Toolbar** of **Sketch** mode, which are used to generate sketch entities on current sketching plane or surfaces by using edges, faces, and other geometries of selected objects; refer to [Figure-144](#). You can also generate intersection curves using 3D objects. The tools in this cascading menu are given next.

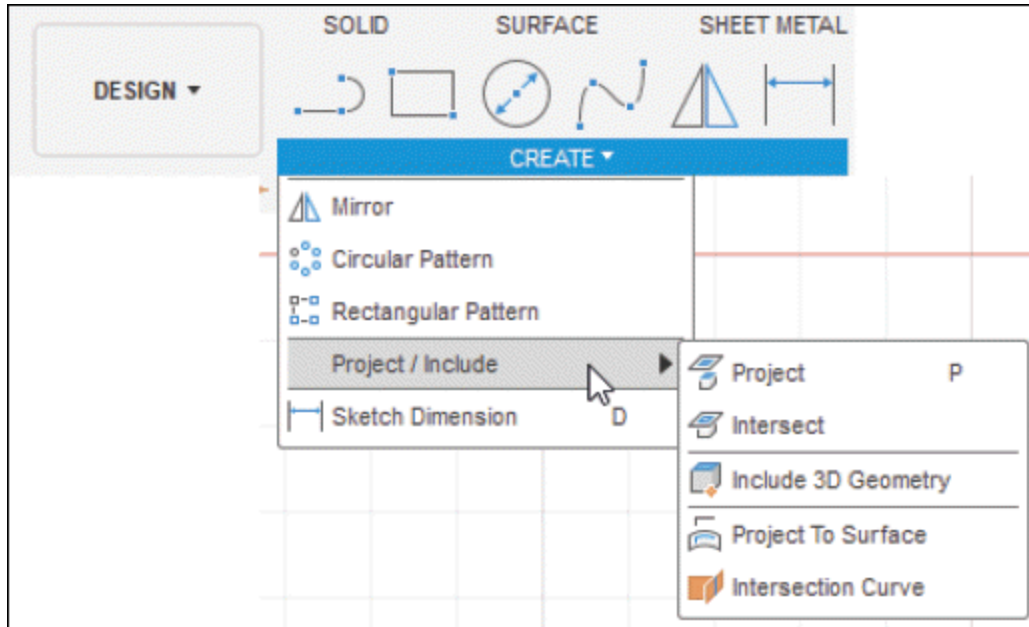


Figure-144. Project Include cascading menu

Project Tool

The **Project** tool is used to project selected surface/face/edge/point/sketch of other features on selected sketching plane. Like, you can project the boundaries of selected face on the sketching plane. The procedure to use this tool is given next.

- Click on the **Project** tool from **Project/Include** cascading menu of **SKETCH** drop-down in the **Toolbar** after activating sketch on desired sketching plane. The **PROJECT** dialog box will be displayed; refer to [Figure-145](#).

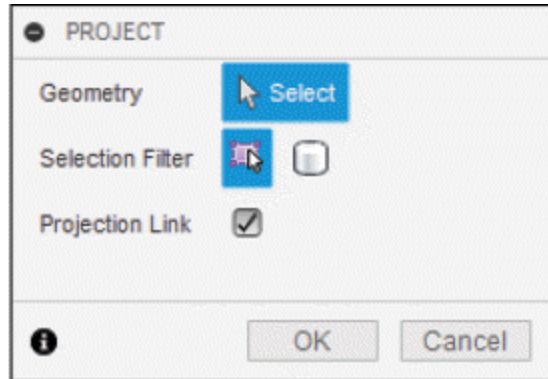


Figure-145. PROJECT dialog box

- Hover the cursor at desired geometry. Preview of project feature will be displayed; refer to [Figure-146](#).
- Click at desired face/edge/point/curve to project it on sketching plane. You can select multiple objects to project them on sketching plane.
- Click on the **OK** button from the dialog box to apply projection.

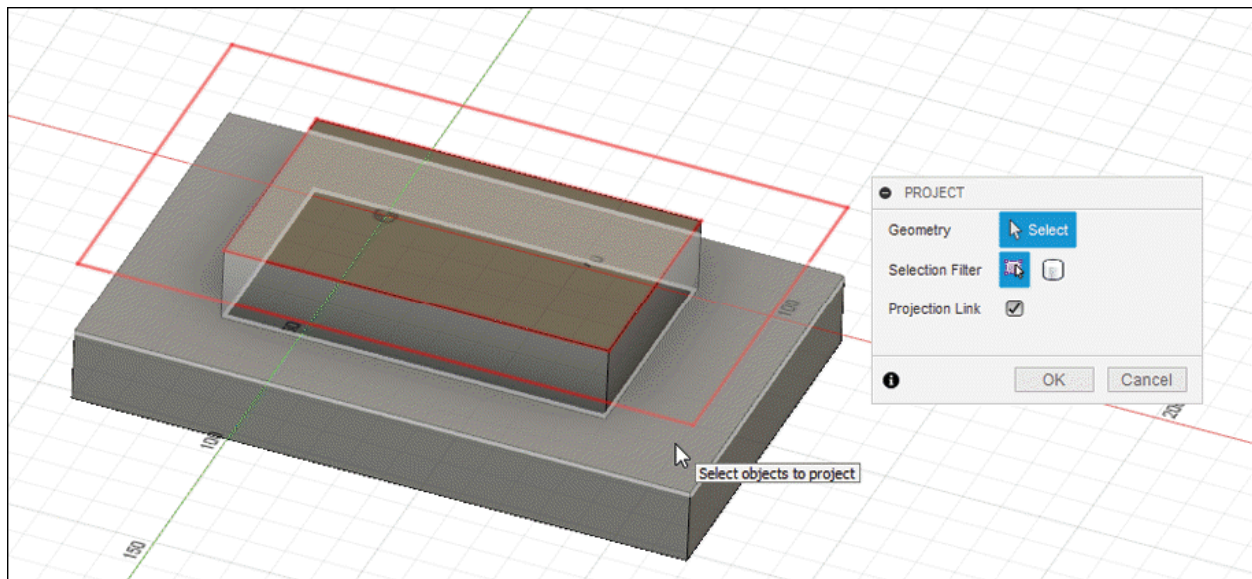


Figure-146. Preview of projected geometry

Intersect Tool

The **Intersect** tool is used when you want to create sketch geometries at the intersection of selected object and intersecting active sketching plane. The procedure to use this tool is given next.

- Start a sketch on desired plane.
- Click on the **Intersect** tool from **Project/Include** cascading menu of the **CREATE** drop-down in the **Toolbar**. The **INTERSECT** dialog box will be displayed; refer to [Figure-147](#) and you will be asked to select the objects that are intersecting with current sketching plane.

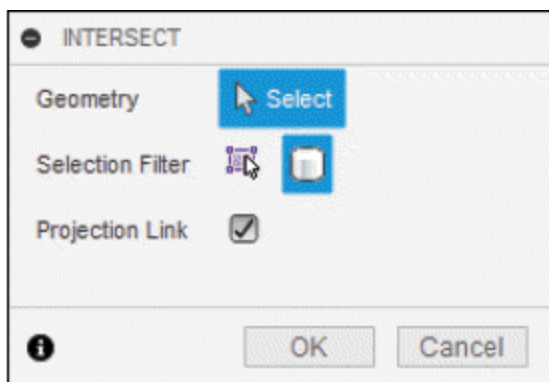


Figure-147. INTERSECT dialog box

- Select the objects that are intersecting with current sketching plane and their intersection curves are to be generated; refer to [Figure-148](#).

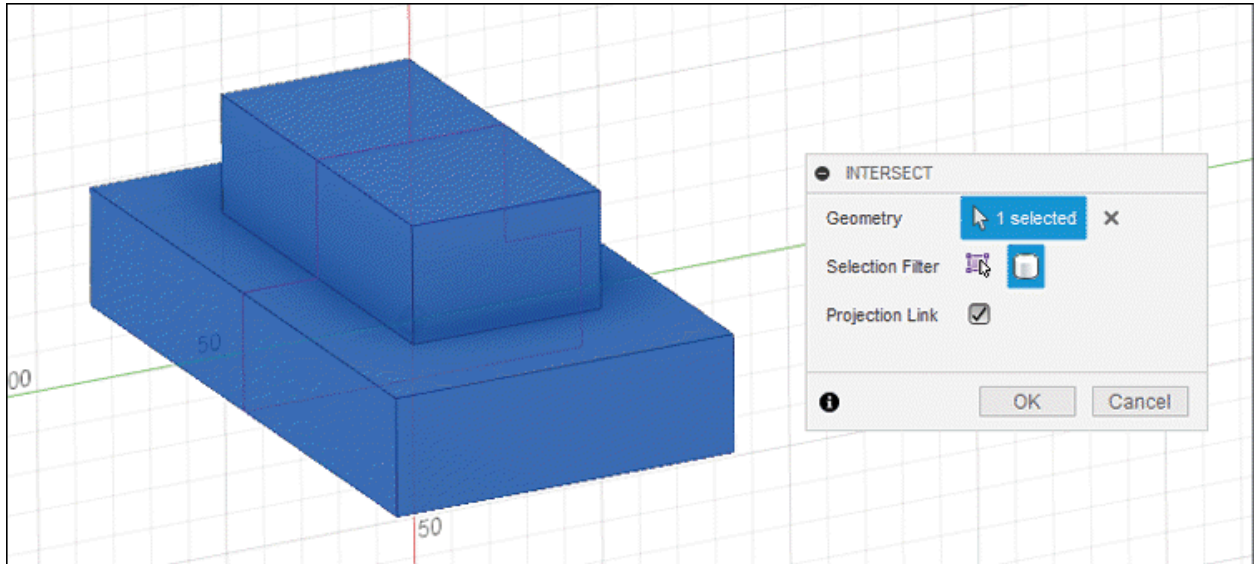


Figure-148. Selection of intersecting objects

- On selecting the objects that are intersecting with current sketching plane, a preview of the intersection curve will be displayed. Click on the **OK** button to create the generated curves; refer to [Figure-149](#).

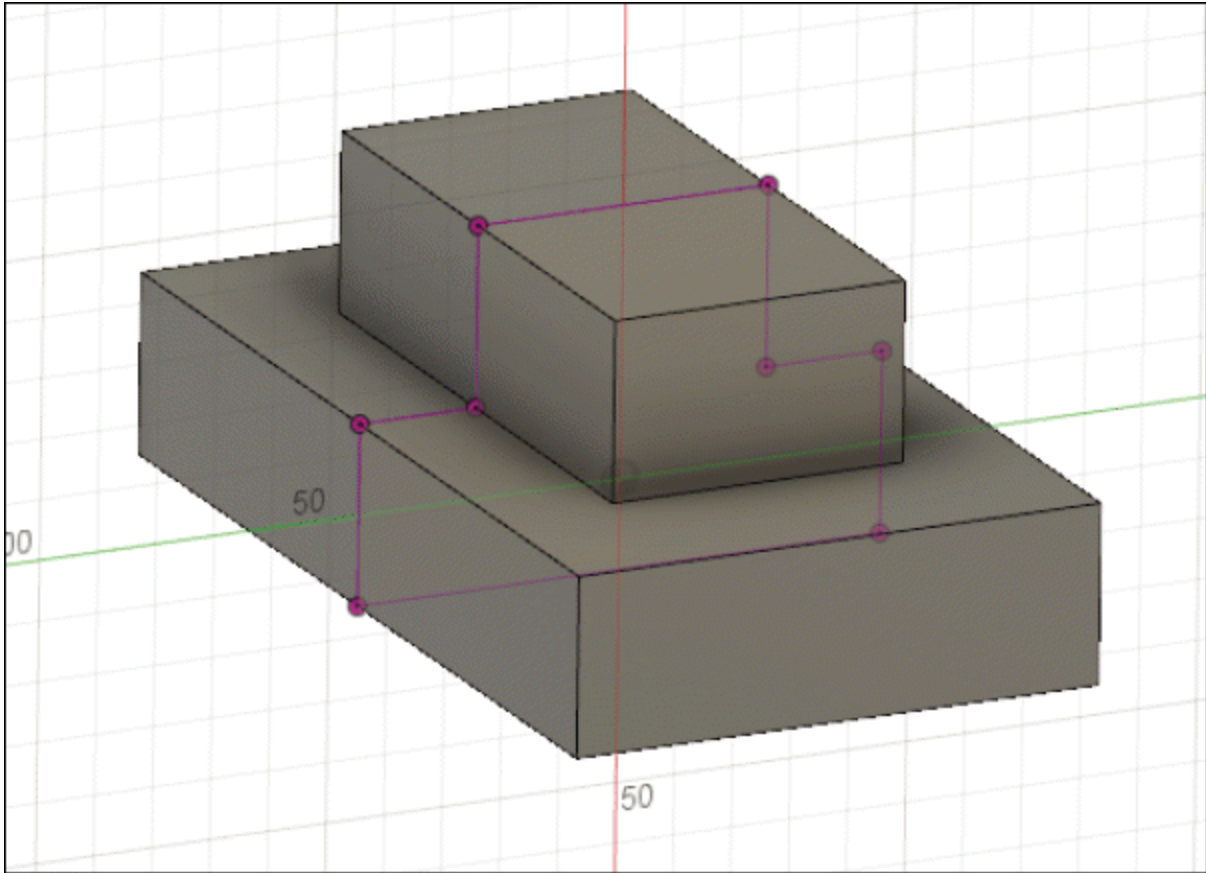


Figure-149. Intersection curves generated

Include 3D Geometry Tool

The **Include 3D Geometry** tool is used to include the boundaries of selected objects in current sketch. Note that this tool will not project the curves on selected sketching plane but you can use the curves for other purposes like as a reference. The procedure to use this tool is given next.

- Start a sketch on desired plane.
- Click on the **Include 3D Geometry** tool from the **Project/Include** cascading menu of the **CREATE** drop-down in **Toolbar**. You will be asked to select the 3D

geometry to be included in current sketch.

- Select desired object. Boundary curves of selected object will be created on itself; refer to [Figure-150](#).

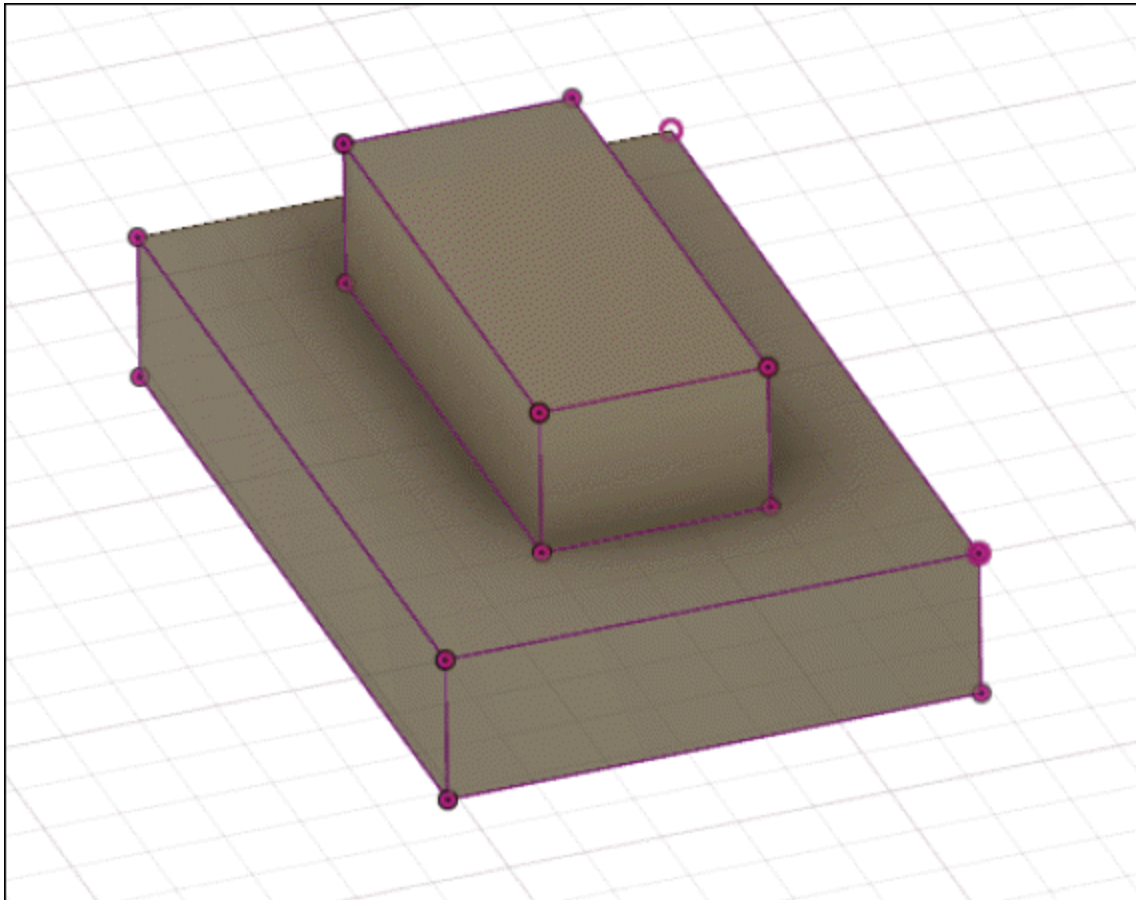


Figure-150. Boundary curves generated

- Press **ESC** to exit the tool.

Project to Surface Tool

The **Project to Surface** tool is used to project selected curves on the selected faces/surfaces. The procedure to use this tool is given next.

- Click on the **Project to Surface** tool from the **Project/Include** cascading menu of the **CREATE** drop-

down in the **Toolbar**. The **PROJECT TO SURFACE** dialog box will be displayed; refer to [Figure-151](#). You will be asked to select a sketching plane.

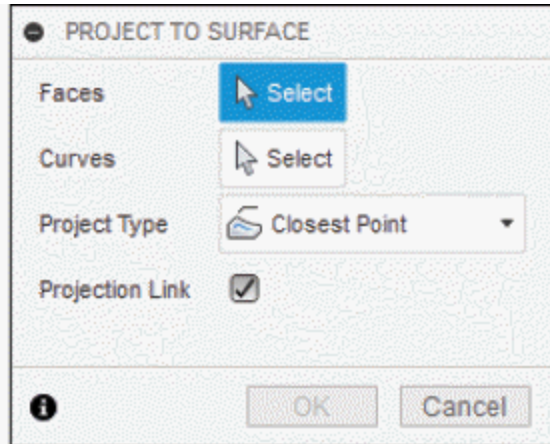


Figure-151. PROJECT TO SURFACE dialog box

- Select the plane which is parallel to face/surface on which sketch is to be projected. The sketching environment will become active. Click on the **Project to Surface** tool, the **PROJECT TO SURFACE** dialog box will be displayed; refer to [Figure-152](#).

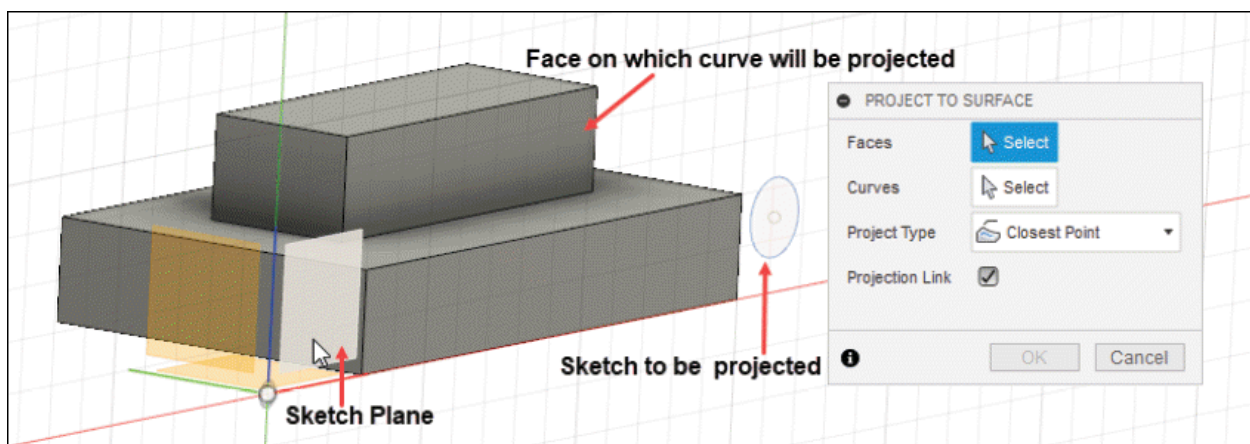


Figure-152. Selection for project to surface tool

- Select the face(s) on which you want to project selected

curves.

- Click on the **Select** button for Curves from the dialog box and select the curves to be projected. Preview of projection will be displayed; refer to [Figure-153](#).

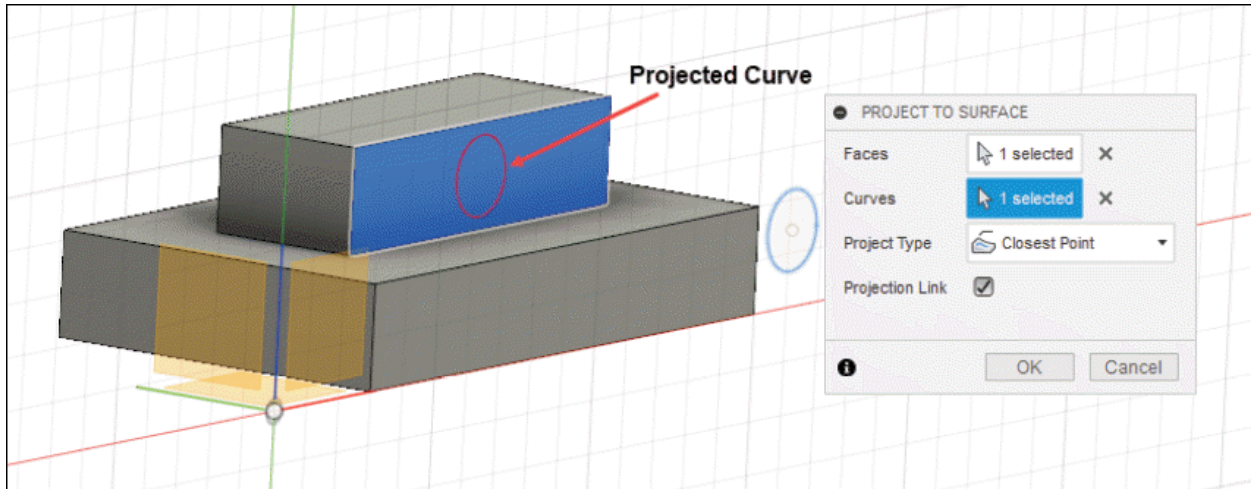


Figure-153. Preview of projected curve

- Select the **Closest Point** option from the **Project Type** drop-down if your object is well within the boundaries of selected face or you want to project along the surface vector to the closest point on the faces; refer to [Figure-153](#). If some of the part of curve is outside the nearest surface then rest of the curve will not be projected; refer to [Figure-154](#). Select the **Along Vector** option from the **Project Type** drop-down if you want to define the direction along which the curve will be projected and select a direction references like edge, axis, face, or plane. Preview of the projection will be displayed; refer to [Figure-155](#).

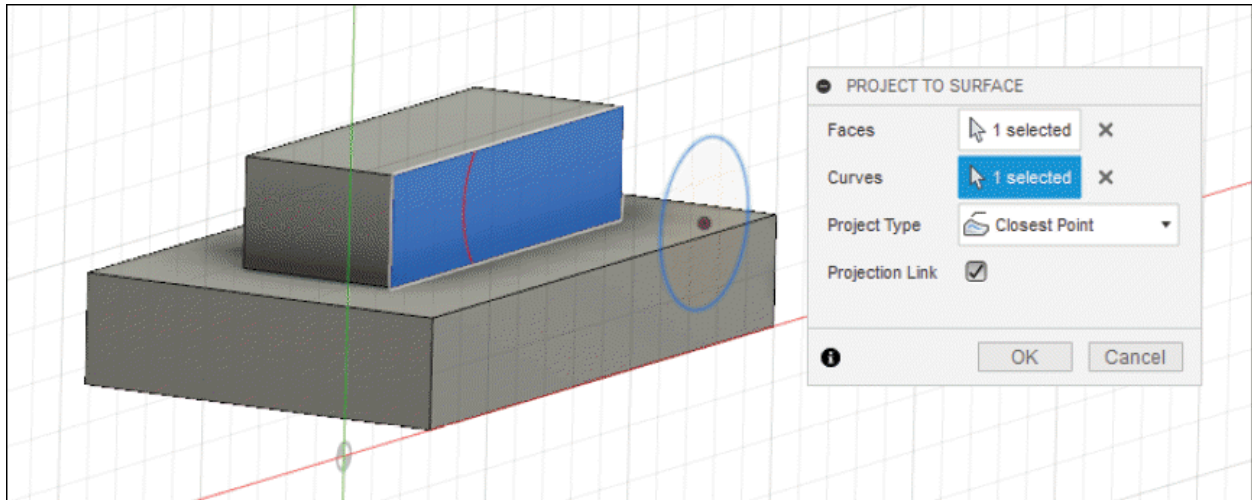


Figure-154. Closest point projection

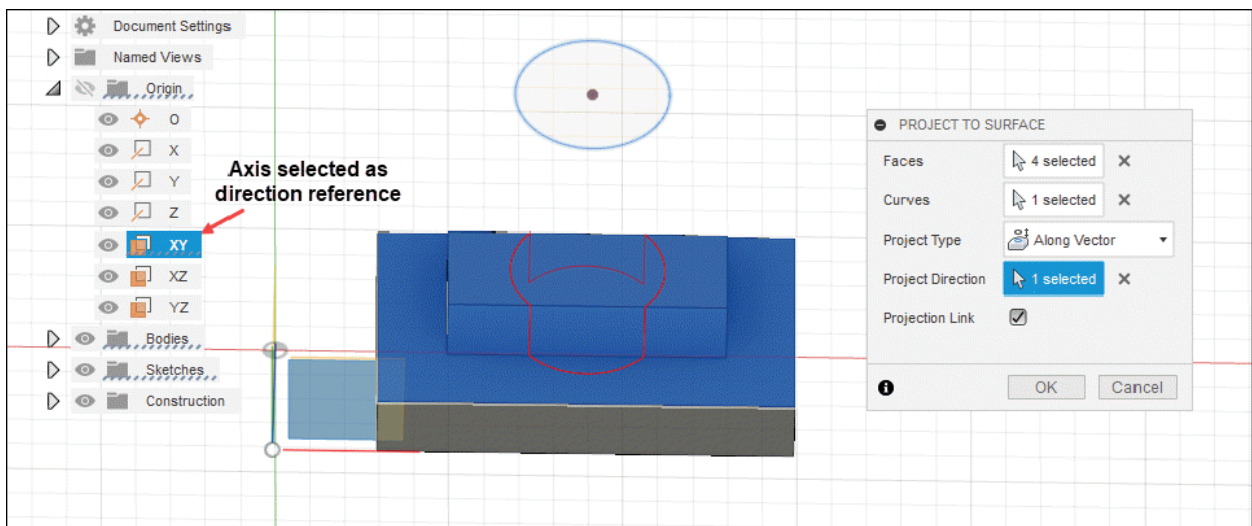


Figure-155. Projection along selected axis

- Click on the **OK** button from the dialog box to create projection.

Intersection Curve Tool

The **Intersection Curve** tool is used to create projection curves generated by selected curve and selected surfaces/faces. The procedure to use this tool is given next.

- Start a new sketch on the plane perpendicular to

projection direction.

- Click on the **Intersection Curve** tool from the **Project/Include** cascading menu of the **CREATE** drop-down in **Toolbar**. The **INTERSECTION CURVE** dialog box will be displayed; refer to [Figure-156](#).

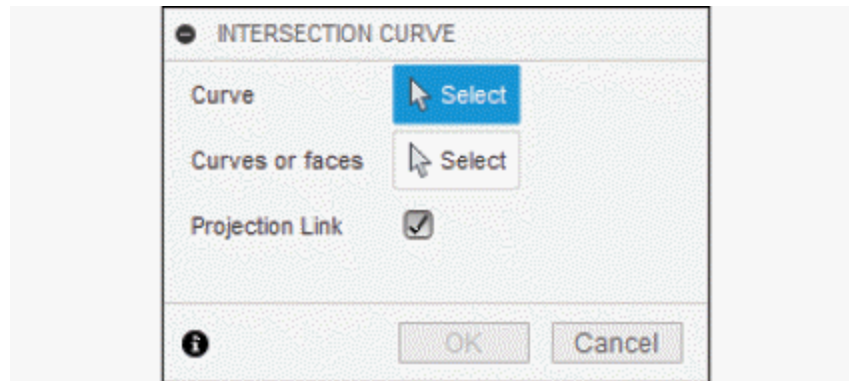


Figure-156. INTERSECTION CURVE dialog box

- Select the curve that you want to be projected and then select the faces/surfaces on which curve is to be projected. Preview of intersection curve will be displayed; refer to [Figure-157](#).

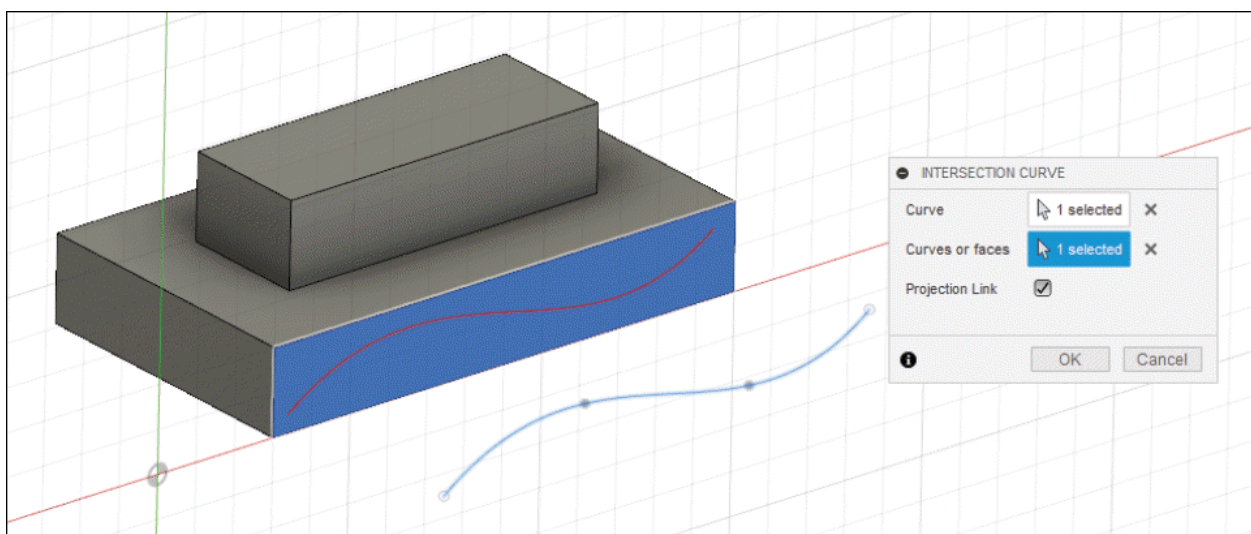


Figure-157. Preview of intersection curve

- Click on the **OK** button from the dialog box to create curves.

SELF ASSESSMENT

Q1. Discuss the procedure of cutting material from a body.

Q2. Which tool is used to combine two bodies?

Q3. Discuss the use of **Sweep** tool with example?

Q4. Discuss the procedure of creating plane using **Offset Plane** tool with example.

Q5. Create a plane tangent to the face of model.

Q6. Create a plane using three points as reference.

Q7. Create a plane using edge of a model as reference.

Q8. Create a point at the center of cylinder.

Chapter 4

Advanced 3D Modeling

Topics Covered

The major topics covered in this chapter are:

- *Introduction*
- *Loft Tool*
- *Rib Tool*
- *Web Tool*
- *Hole Tool*
- *Thread Tool*
- *Box Tool*
- *Cylinder Tool*
- *Sphere Tool*
- *Torus Tool*
- *Coil Tool*
- *Pipe Tool*
- *Pattern Tool*
- *Mirror Tool*
- *Thicken Tool*
- *Boundary Fill Tool*
- *Create Form Tool*
- *Create Base Feature Tool*
- *Create Mesh Tool*
- *Create Mesh Section Sketch Tool*
- *Create 3D PCB Feature Tool*

INTRODUCTION

In the last chapter, you have learned the procedure to use some 3D modeling tools and creating the constructional geometry. In this chapter, you will use the constructional geometry tools with some advanced 3D modeling tools. The tools are available in the panel of **DESIGN** workspace.

LOFT

The **Loft** tool is used to create a solid volume by joining two or more sketches created on different planes; refer to

[Figure-1](#). The procedure to use this tool is given next.

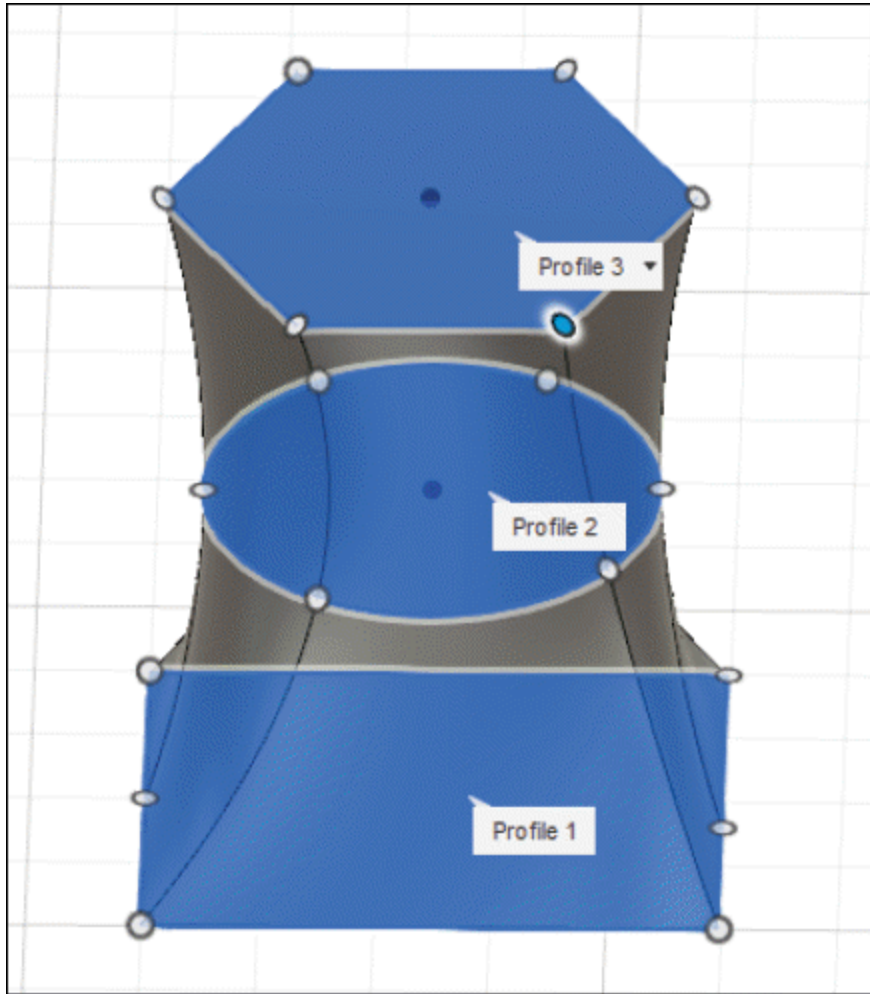


Figure-1. Lofted feature

- Click on the **Loft** tool from **CREATE** drop-down in the **Toolbar**; refer to [Figure-2](#). The **LOFT** dialog box will be displayed as shown in [Figure-3](#).

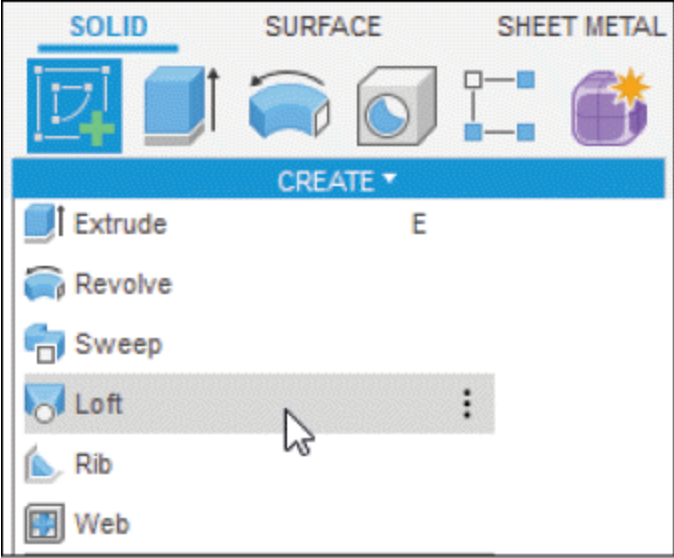


Figure-2. Loft tool

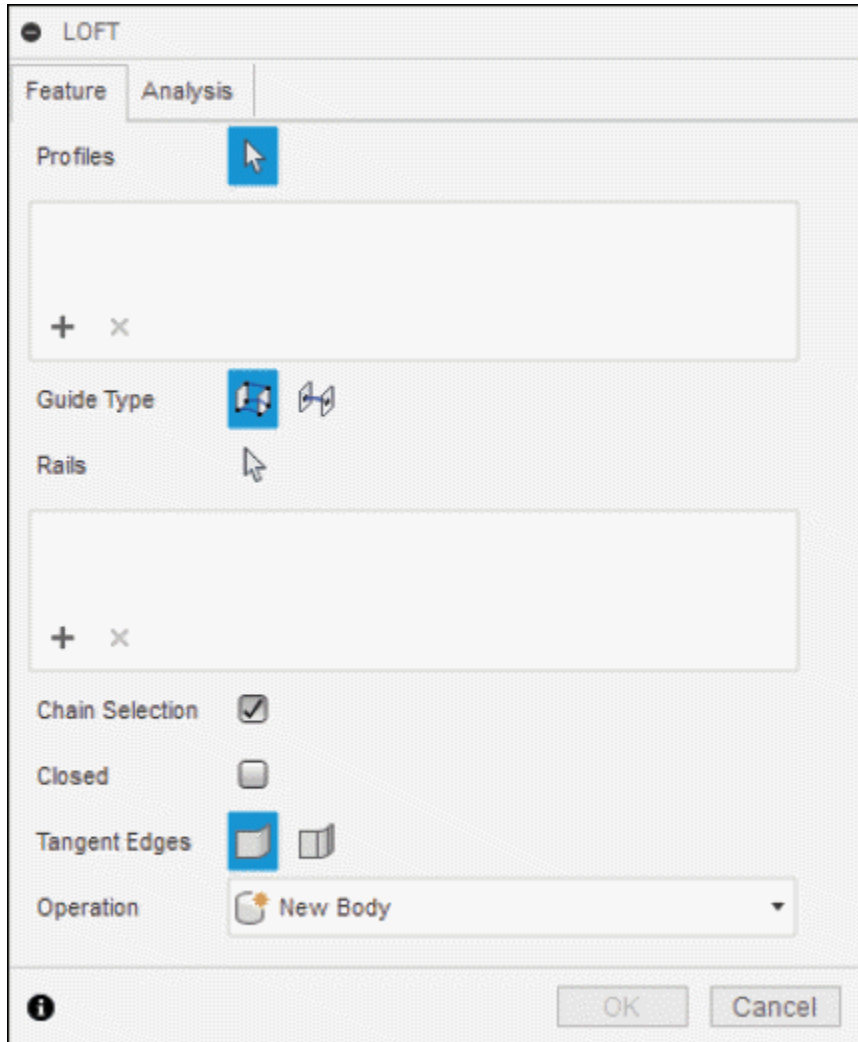


Figure-3. LOFT dialog box

- By default, **Profiles** selection box is active and you are asked to select sketches for loft feature. You can also select faces/surfaces as profile for loft.
- Click one by one on the sketches created at different planes. Note that you need to select the sketches in order by which they can be joined to each other successively. The preview of loft will be displayed; refer to [Figure-4](#).

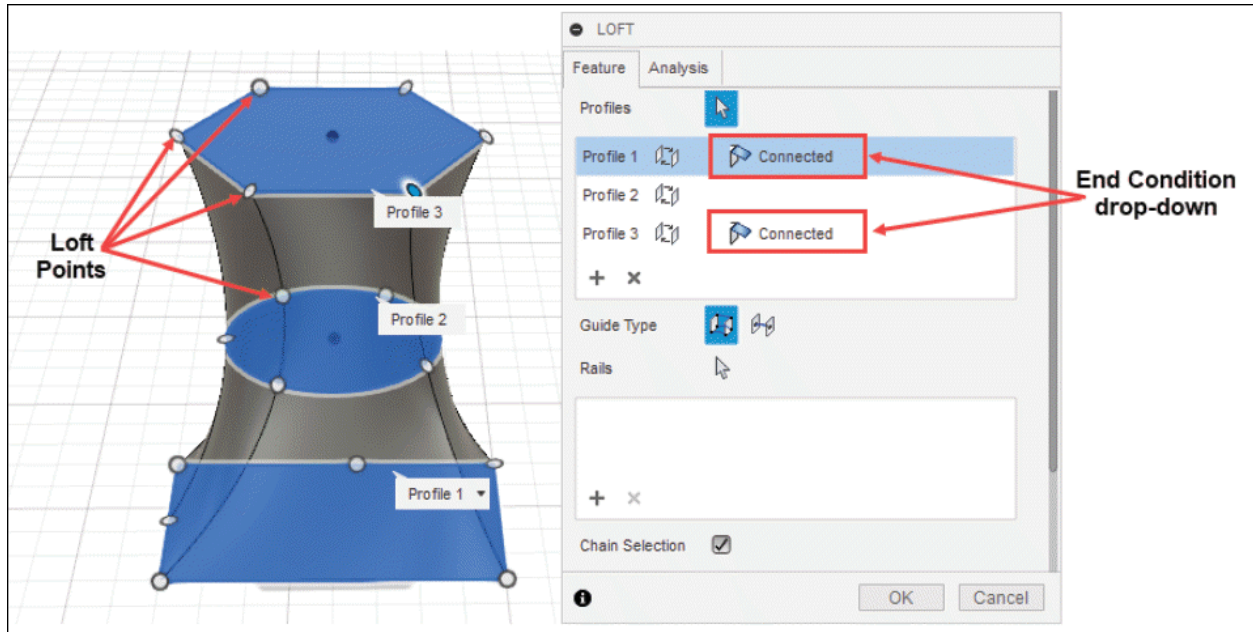


Figure-4. Preview of loft

- To make adjustments to the loft, you can move the loft points along the profile edge.
- Click the **End condition** drop-down of **Profile 1** from **Profiles** selection box. The options will be displayed as per the geometry selected.

If you have selected sketches only as profiles then **Connected** and **Direction** options are available in the drop-down; refer to [Figure-5](#). If you have selected sketches and faces of model or only faces of the model as profiles then **Connected (G0)**, **Tangent(G1)**, and **Curvature(G2)** options are available in the **End condition** drop-down; refer to [Figure-6](#). If you have selected a point as profile at the end then **Sharp** and **Point Tangent** options will be available in the **End condition** drop-down; refer to [Figure-7](#).

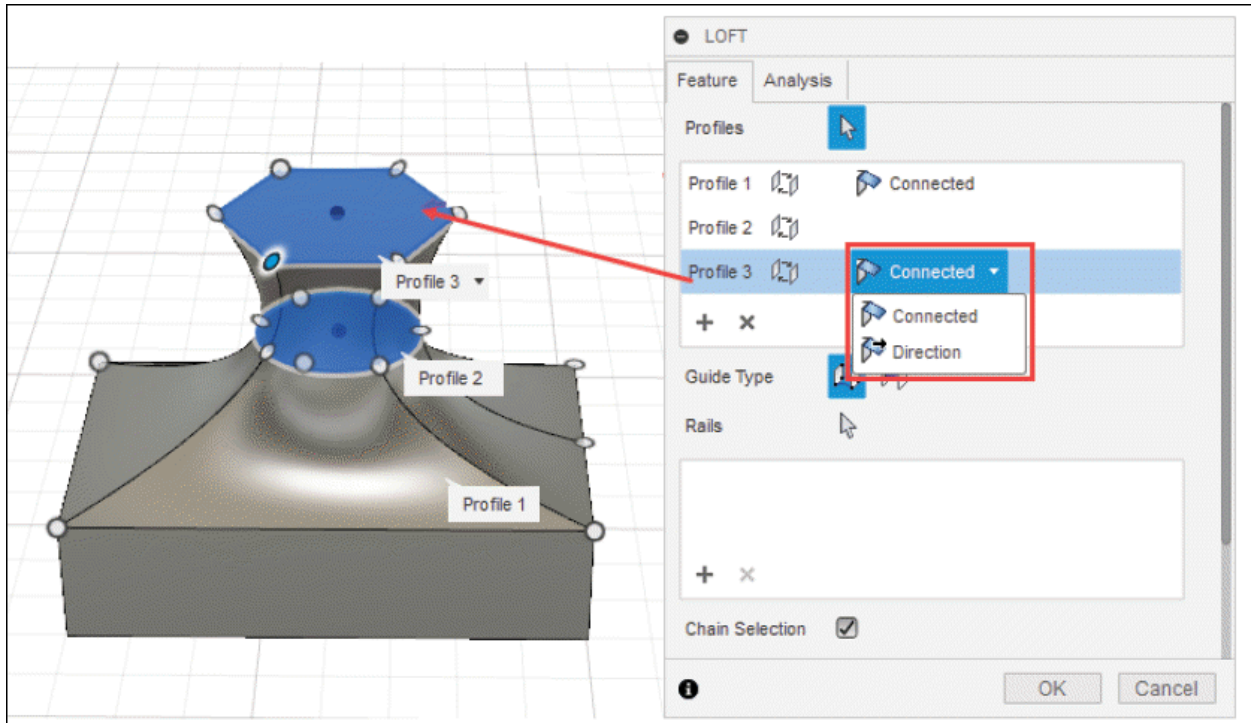


Figure-5. End condition options for sketch profile in loft

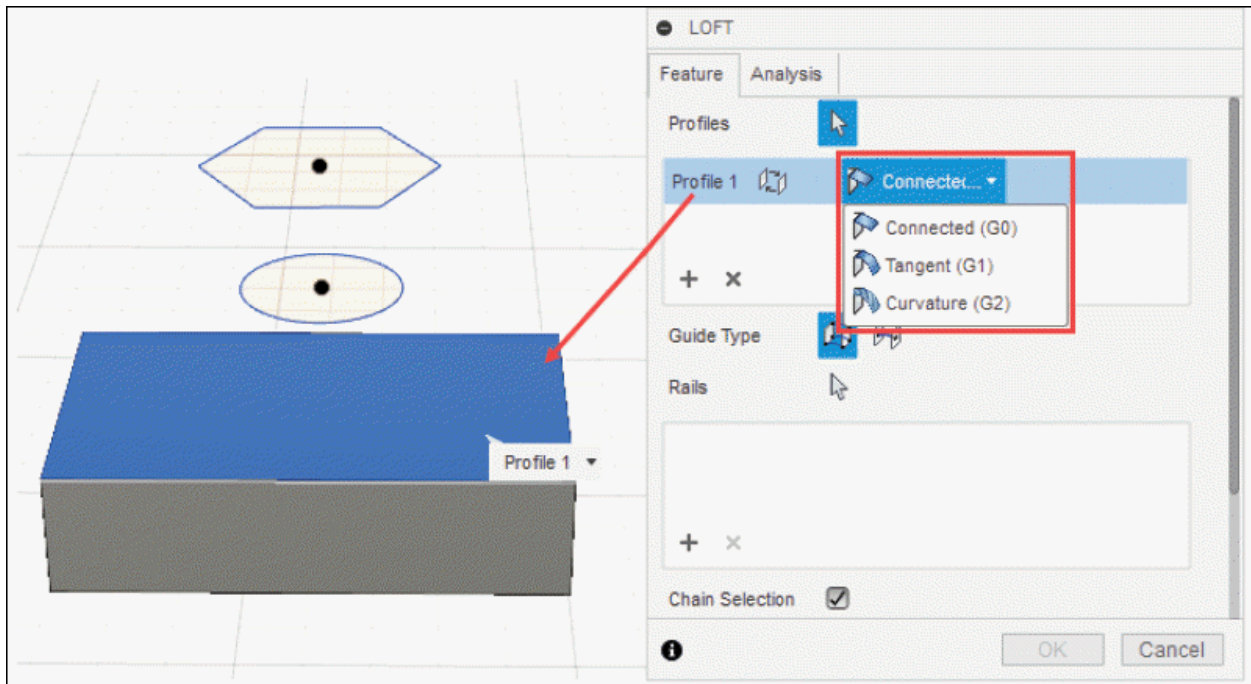


Figure-6. End condition options for face profile in loft

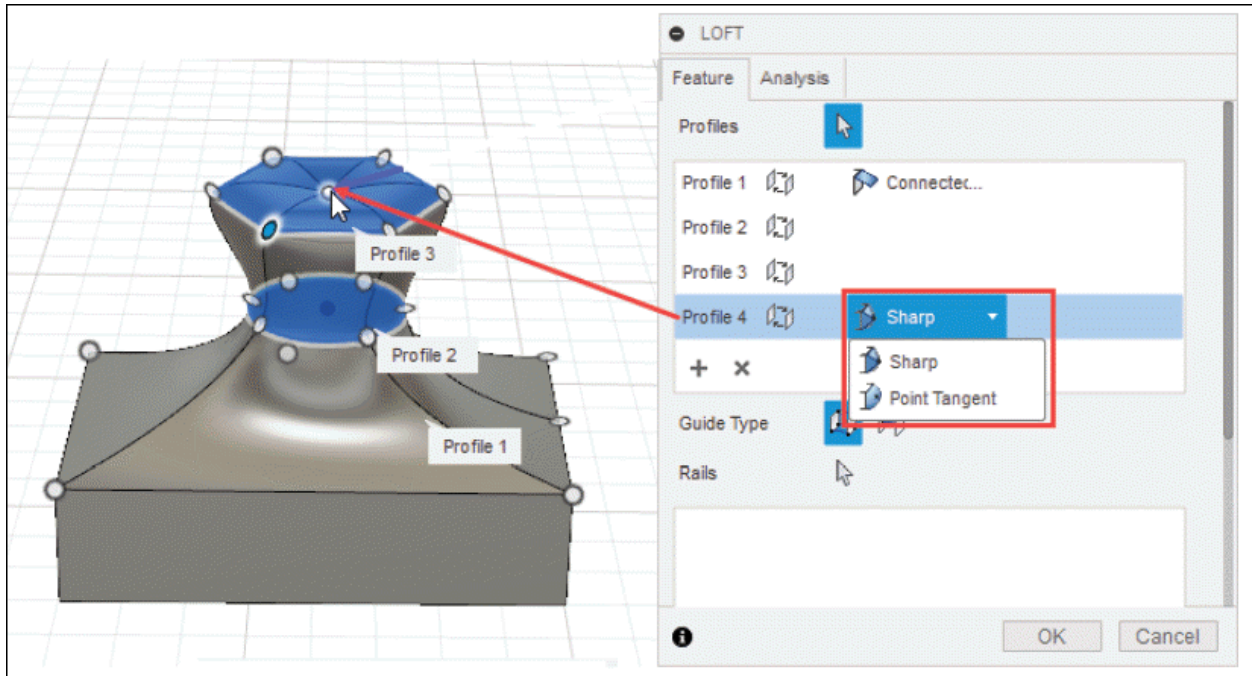


Figure-7. End condition options for point profile in loft

- Select desired end condition option from the **End condition** drop-down. One line explanation of each end condition option is given in [Figure-8](#).

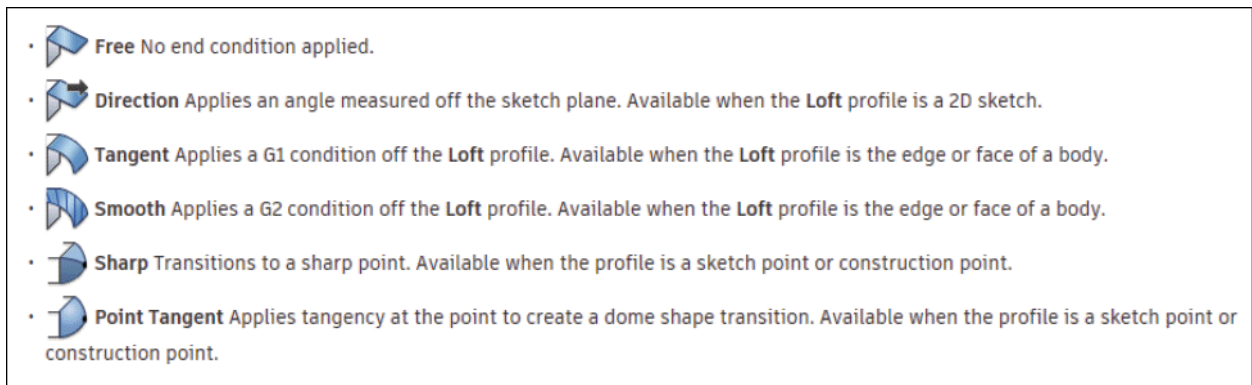


Figure-8. End condition

- Select the **Direction** option from the **End Condition** drop-down. The updated **LOFT** dialog box will be displayed with options to specify **Takeoff Weight** and **Takeoff Angle**; refer to [Figure-9](#). As the name suggests, these options are

used to specify scale and angle at which loft will start or end at selected profile.

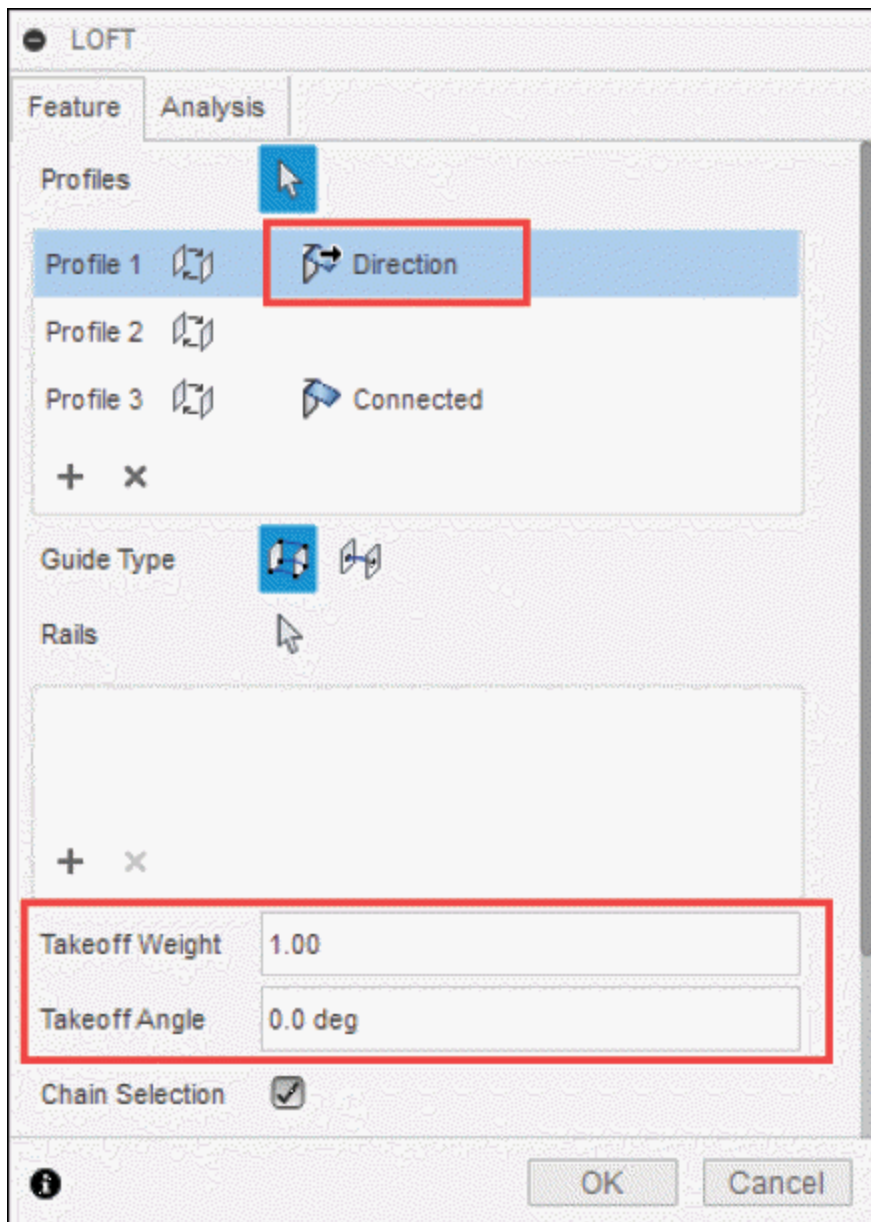


Figure-9. Updated LOFT dialog box

- If you have selected **Tangent**, **Curvature**, or **Point Tangent** option from the **End Condition** drop-down then **Tangency Weight** edit box will be displayed in the dialog

box to specify scale factor of tangency.

- Enter the value as **4** in **Takeoff Weight** edit box and specify the outward angle in **Takeoff Angle** edit box by **20** degree. You will find a bulge in the loft feature; refer to [Figure-10](#).

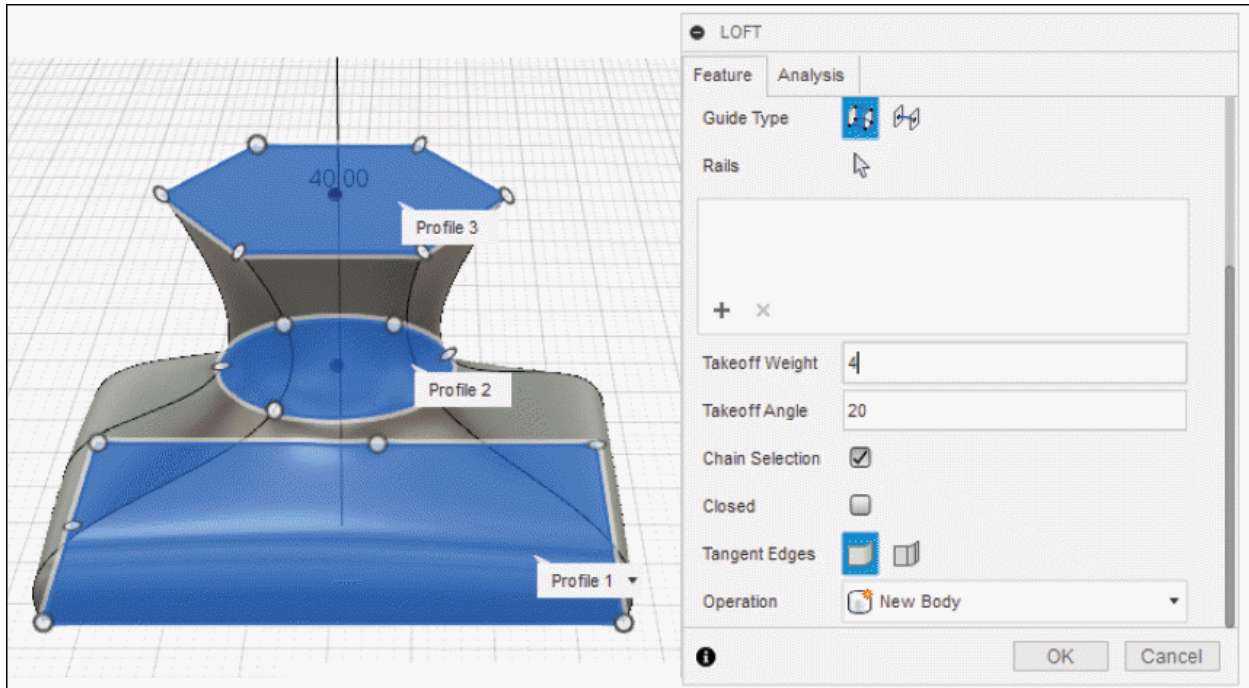


Figure-10. Preview after entering the value

- Select the **Chain Selection** check box if you also want to select the adjacent edges and included as one profile. Clear the check box if you want to select the each edge of profile individually. This check box hardly has any effect in modeling but you will find its uses in surfacing while creating loft feature.
- After setting desired parameters, click **OK** button from **LOFT** dialog box to create the loft feature.

Loft feature can be very useful in joining two faces or

surfaces to create solid volume; refer to [Figure-11](#). You can adjust the shape of loft by using the drag points on the edges.

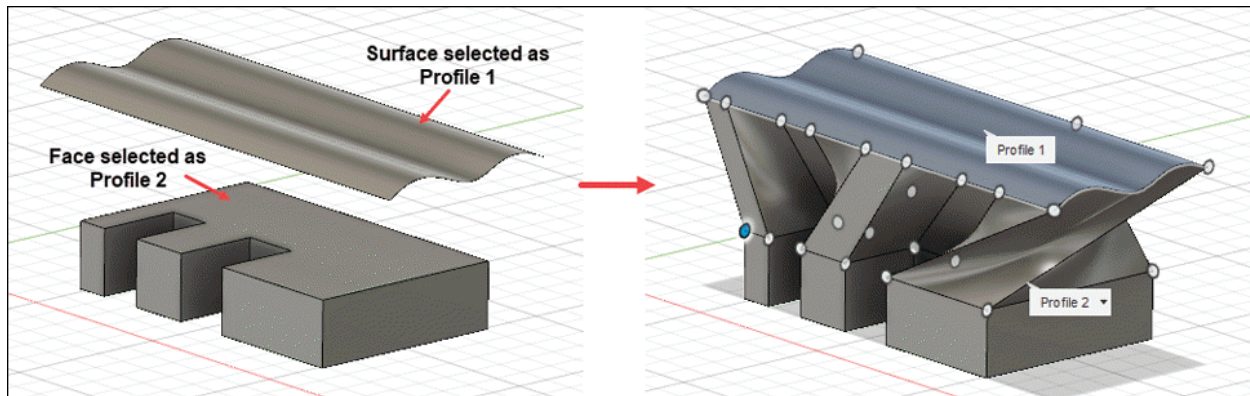


Figure-11. Creating loft using surface and face

Guide Type

There are two ways to control transition between profiles of loft; using **Rails** and using **Centerline**. **Rails** are 2D or 3D curves that affect the loft shape between sections at boundaries. To refine the shape of loft, you need to add number of rails. These rails must intersect each section and terminate on or beyond the first and last sections. **Rails** must be tangentially continuous. A **Centerline** tool is a type of rail to which the loft sections are held normal. It behaves like a sweep path. Centerline lofts maintain a more consistent transition between the cross-sectional areas of selected loft sections. Center lines follow the same criteria as rails, except they need not intersect the sections and only one centerline can be selected. We have discussed two examples next for tent and ring which use rails and centerline, respectively.

Using Rails

- To select rails for the loft, click the **Rails** button in **Guide Type** section; refer to [Figure-12](#).

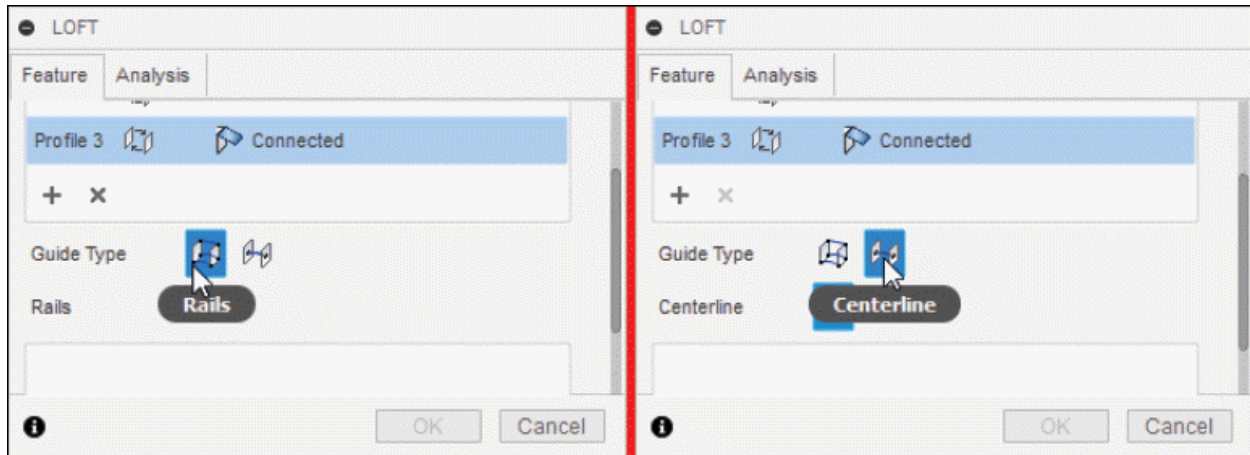


Figure-12. Rails and Centerline tool

- In this case, we have created a sketch of tent like structure using a rectangle and four arcs (to be used as rail) on different planes; refer to [Figure-13](#).

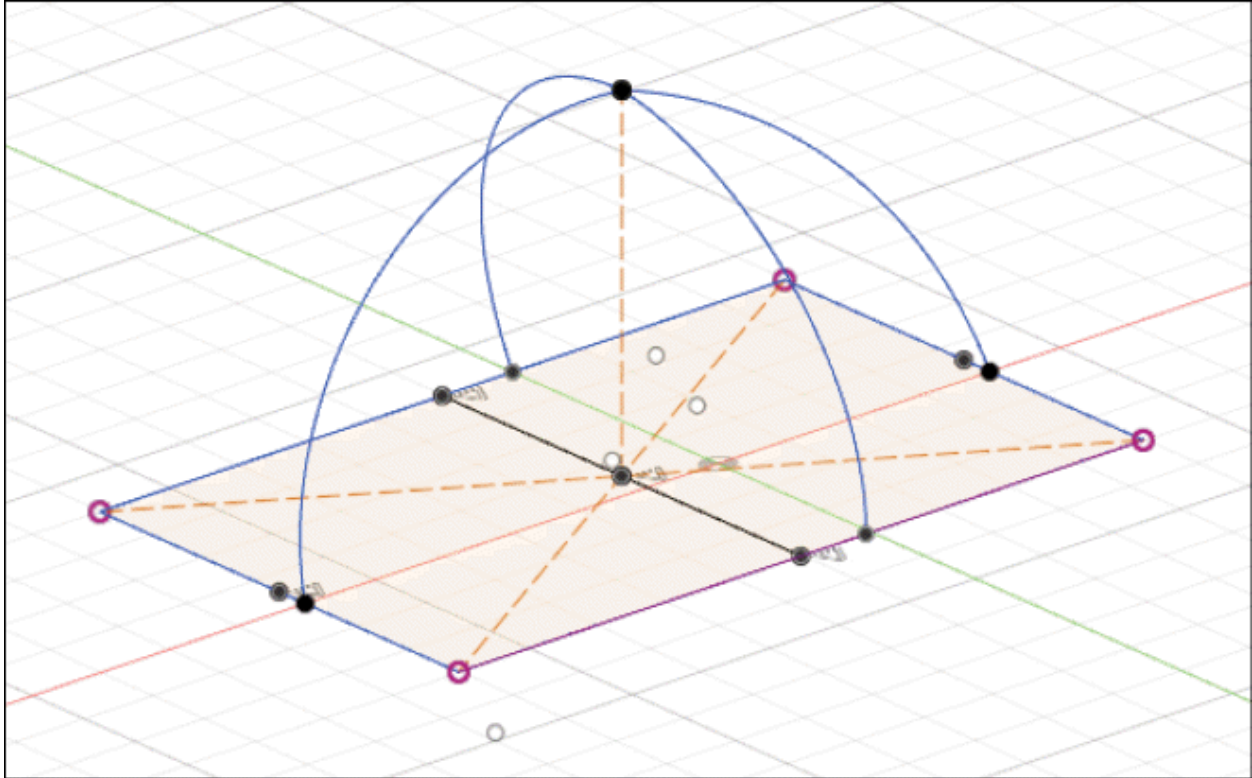


Figure-13. Sketch for using Rails tool

- Select the **Loft** tool from **CREATE** drop-down. The **LOFT** dialog box will be displayed.
- Select the rectangle and point of the sketch in **Profiles** section of **LOFT** dialog box. The preview of model will be shown.
- Click on the **Arrow** selection button of **Rails** from **Guide Type** section box, to select the rails for the loft.
- Select the four arcs of the model; refer to [Figure-14](#).

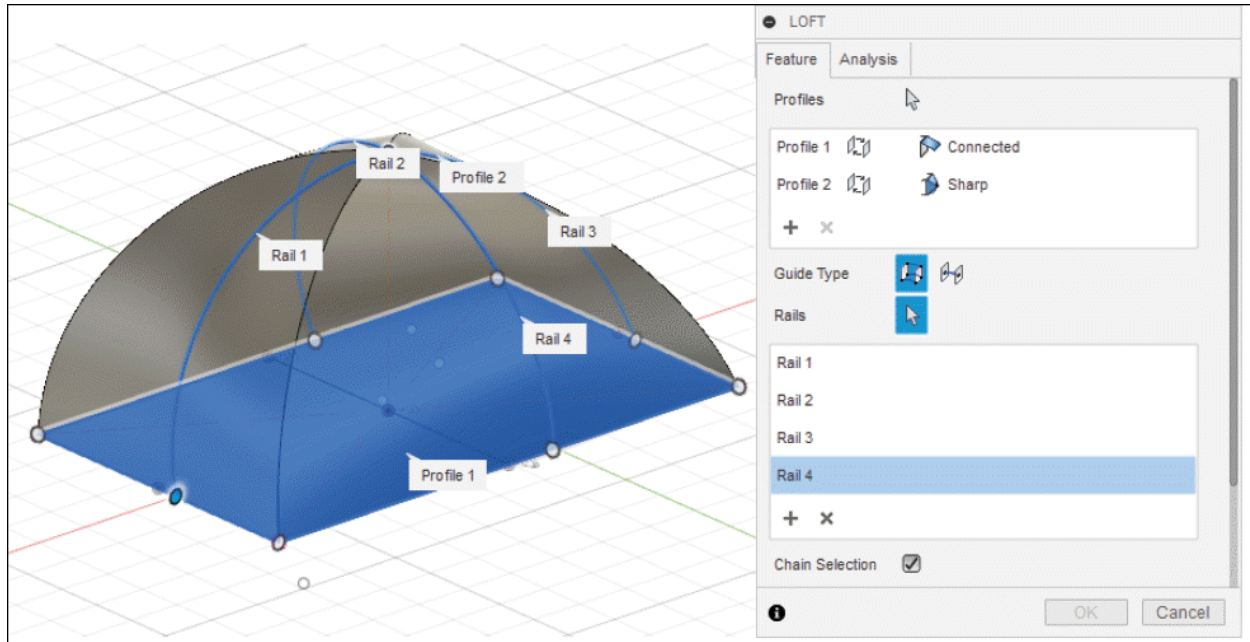


Figure-14. Preview of loft on selecting Rails

- Click **OK** button of **LOFT** dialog box to complete the process of making loft. The shape of loft will be modified according to rails selected.

Using Centerline

- To use the **Centerline** guide type, we have created a sketch of ring like structure with the help of circle and rectangle on different planes; refer to [Figure-15](#).

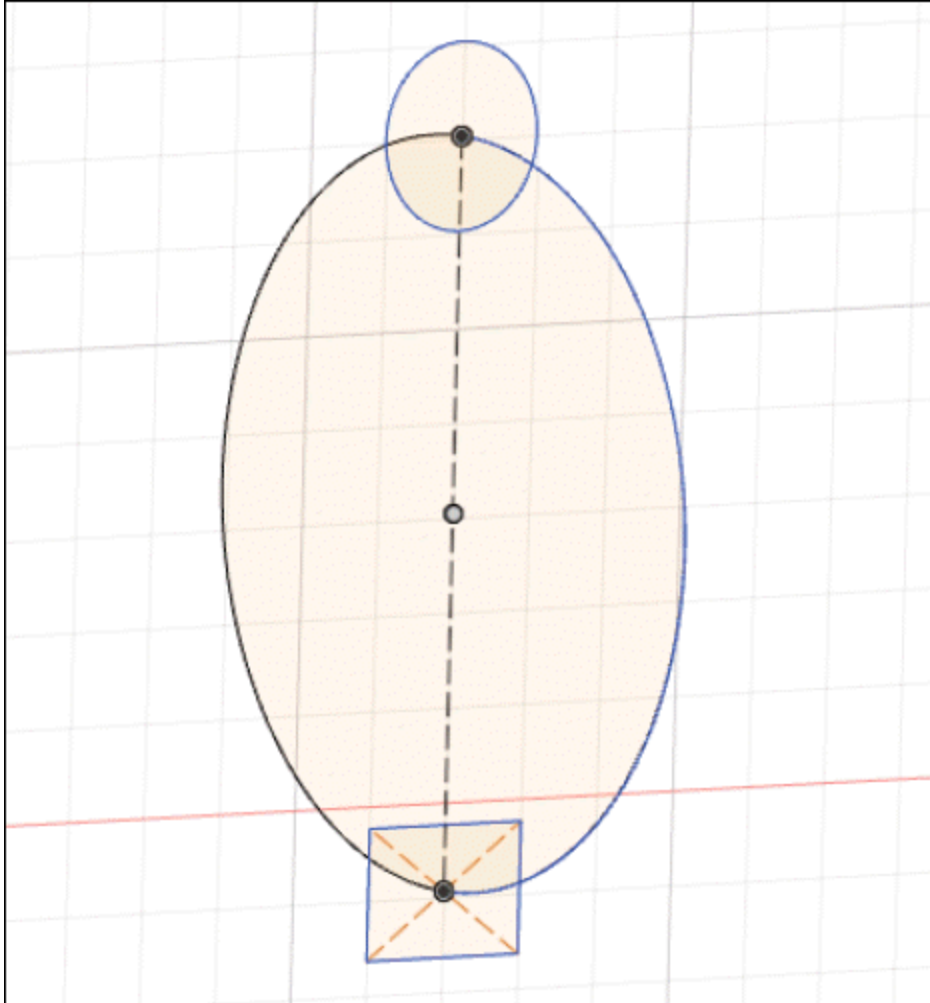


Figure-15. Sketch of ring

- Select the **Loft** tool from **CREATE** drop-down. The **LOFT** dialog box will be displayed.
- Select the circle and rectangle as profile in **Profiles** section of **LOFT** dialog box.
- Since these two profiles are coplanar, you may not get any preview and an error might welcome you.
- Click on **Centerline** button from **Guide Type** section and select the circle to be used as center line. The preview will

be displayed; refer to [Figure-16](#).

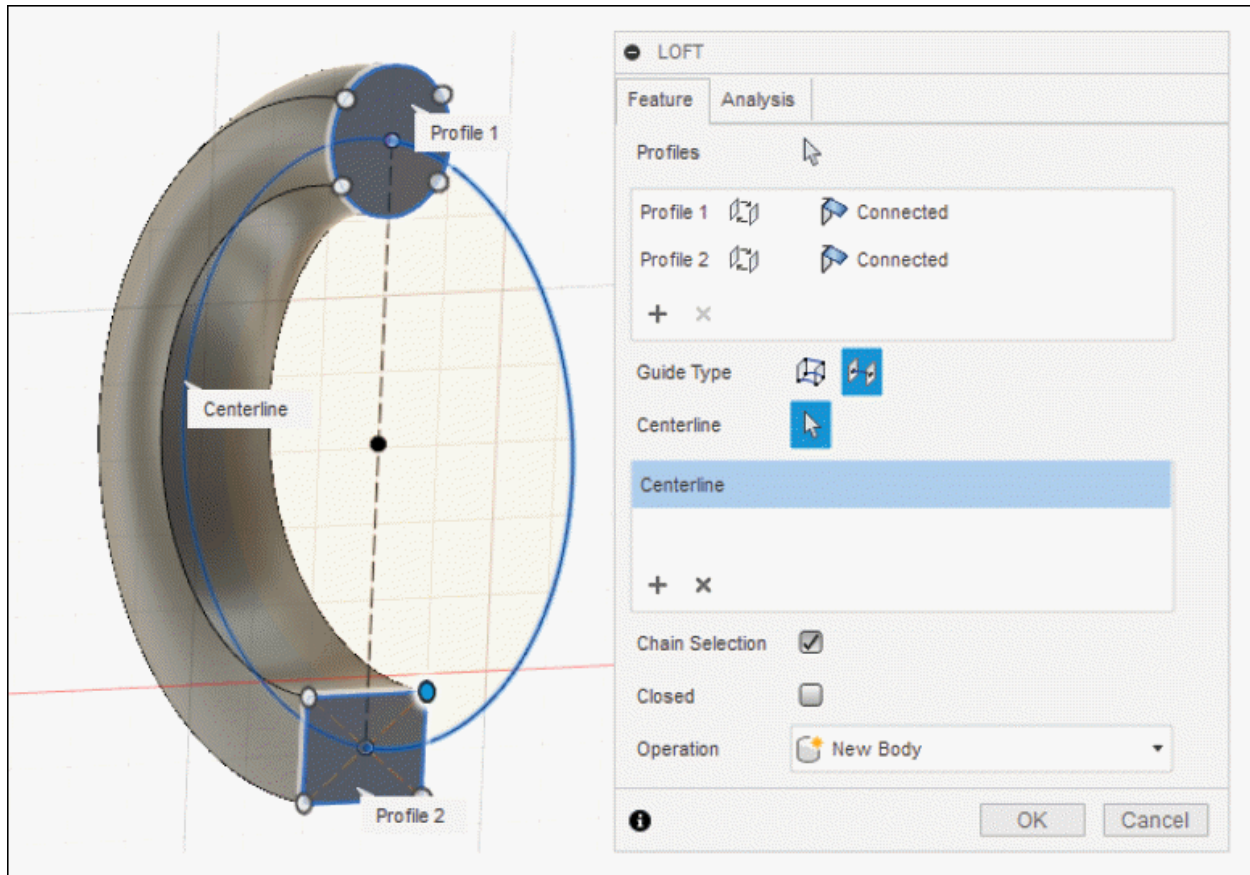


Figure-16. Preview of loft on selecting centerline

- By default, **Closed** check box is clear so you will be getting half of the ring. Select the **Closed** check box to complete the loft. Preview of the loft feature will be modified; refer to [Figure-17](#).

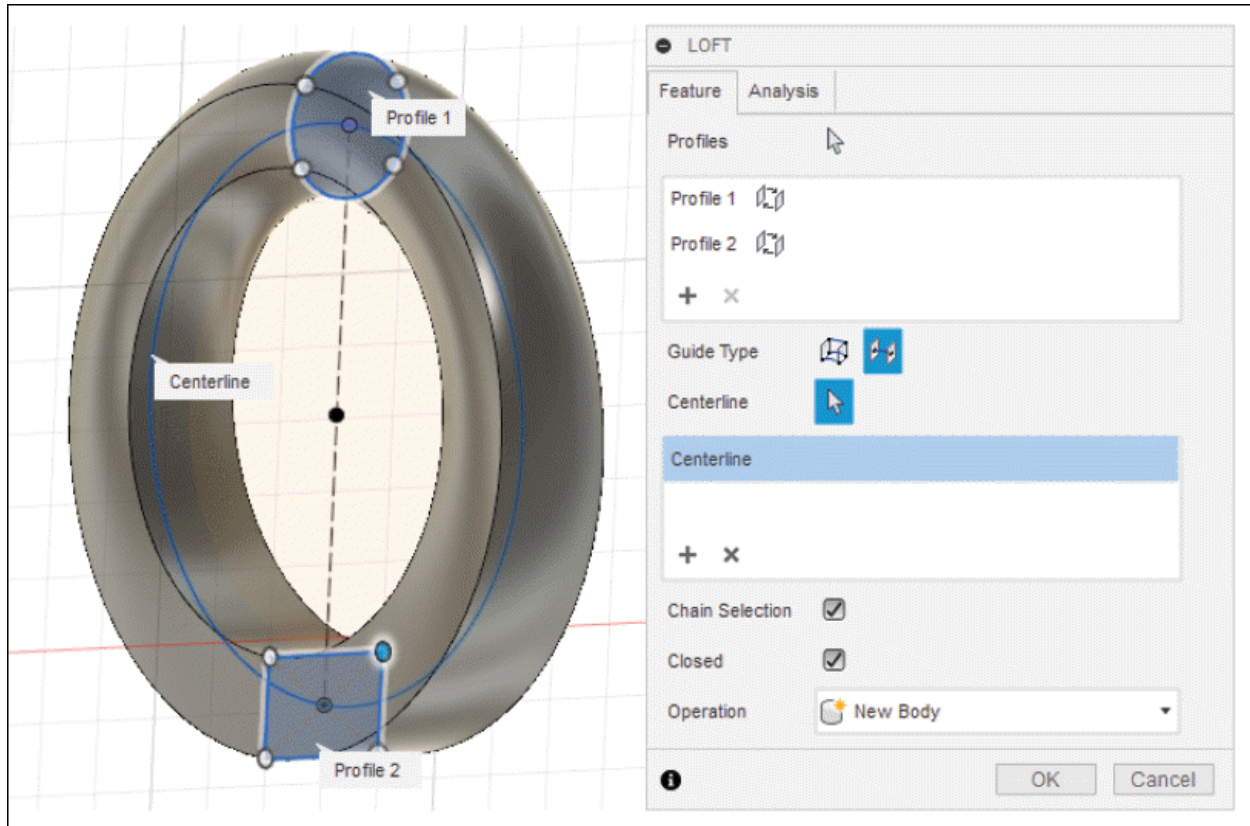


Figure-17. Preview of loft on selecting closed centerline

- Click **OK** button of **LOFT** dialog box to complete the process. The tools of **Operation** drop-down are same as discussed earlier.

RIB

The **Rib** tool is used to create supporting features in the model by using curve(s). The procedure to use this tool is discussed next.

- Click on **Rib** tool from **CREATE** drop-down; refer to [Figure-18](#) or right-click on the screen canvas and select **Rib** from **Marking menu** displayed. The **RIB** dialog box will be displayed; refer to [Figure-19](#). Note that the curve

for rib should be created in such a way that its projection is within solid faces of the model; refer to [Figure-20](#).

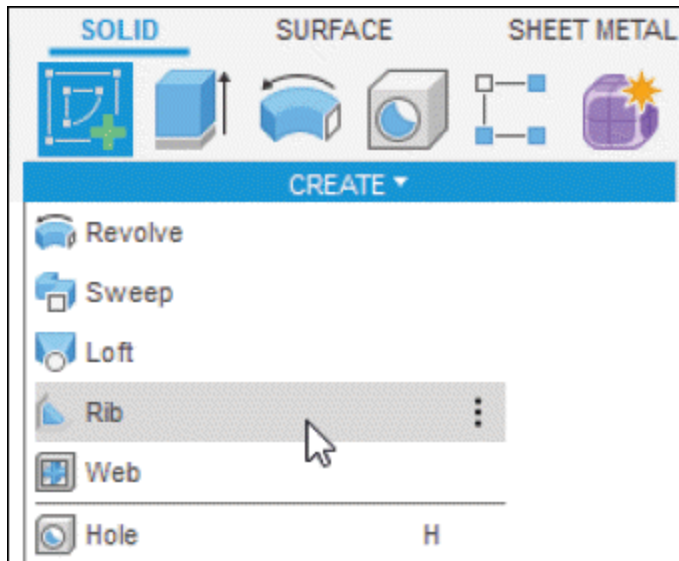


Figure-18. Rib tool

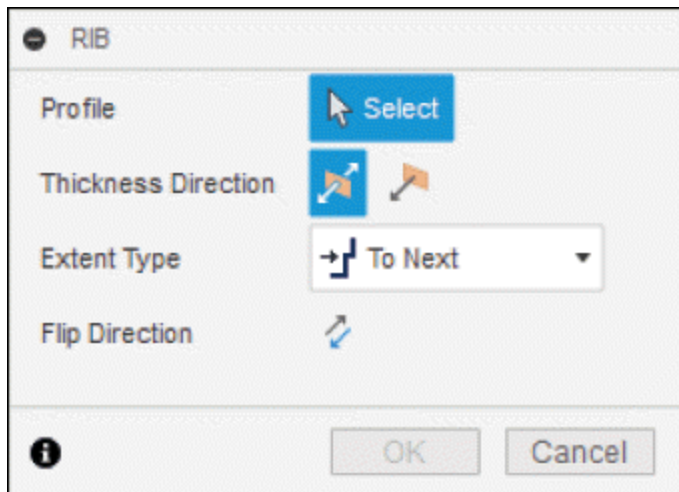


Figure-19. RIB dialog box

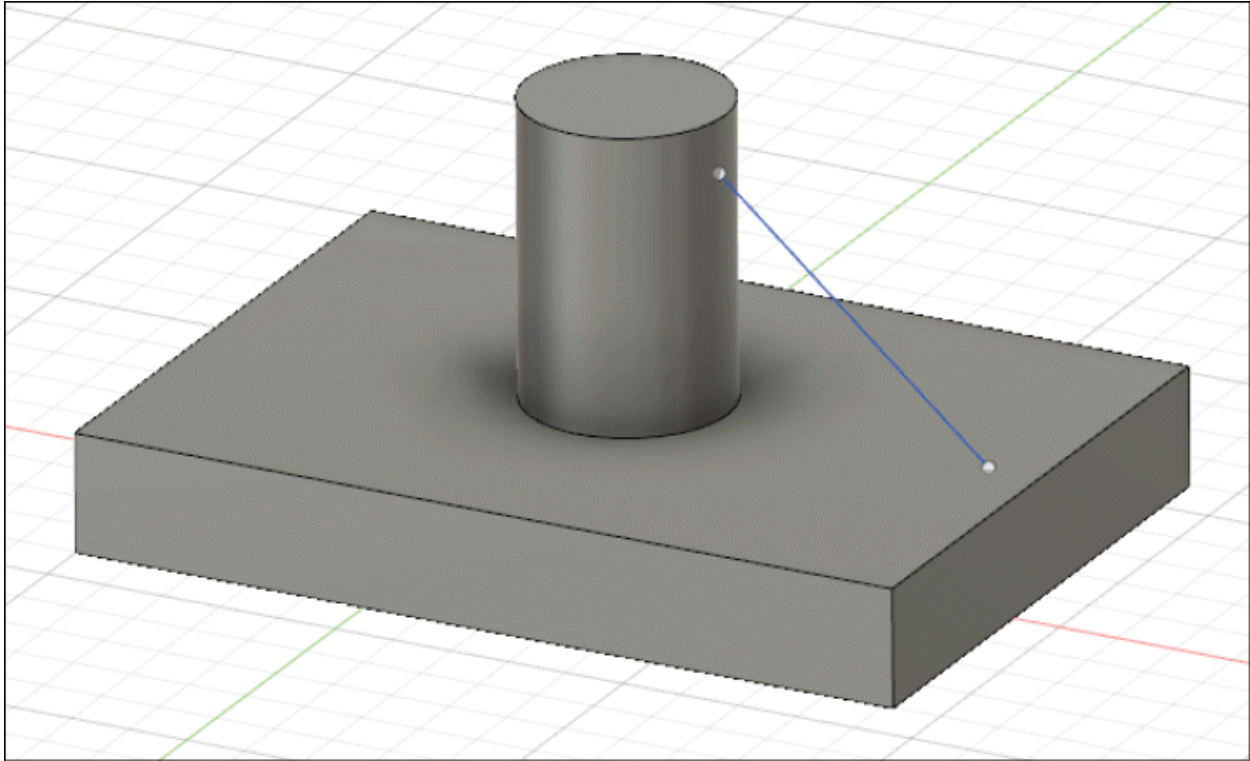


Figure-20. Sketch for rib

- The **Profile** section of **RIB** dialog box is active by default. Select the sketch curve joining edges of solid model to create rib. The updated **RIB** dialog box will be displayed; refer to [Figure-21](#).

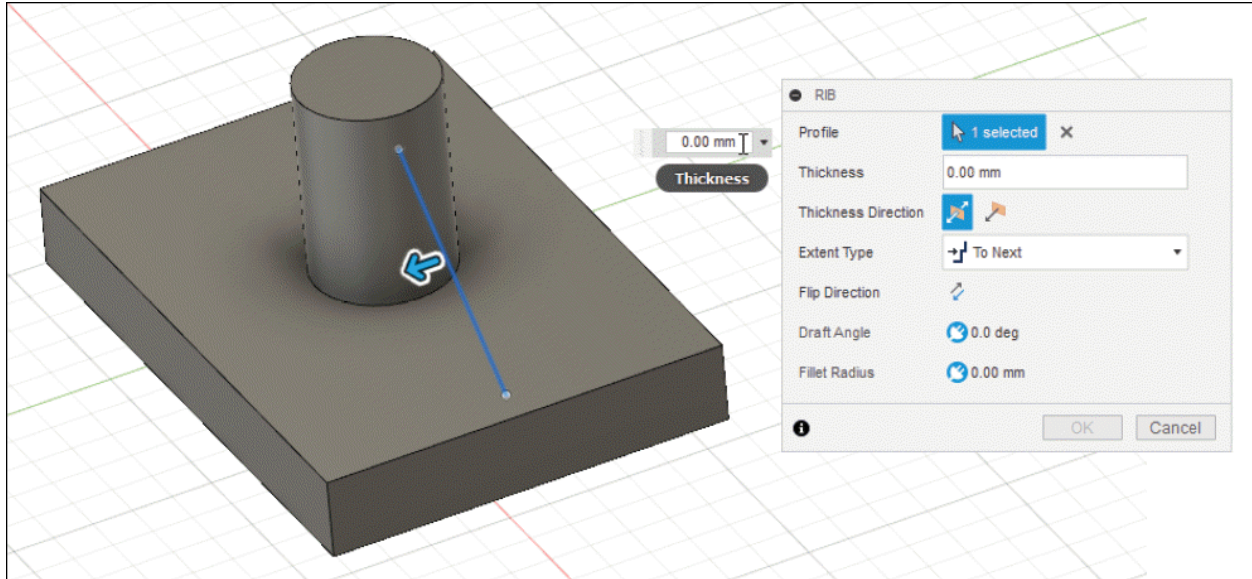


Figure-21. Updated RIB dialog box

- Select the **Symmetric** option from **Thickness Direction** section to set the thickness of rib on both side of curve. Select **One Side** option from **Thickness Direction** section to apply the thickness of rib in one direction or one side.
- Specify desired value of thickness in **Thickness** edit box or floating window.
- Select the **To Next** option from **Extent Type** drop-down from **RIB** dialog box to create the rib up to next surface or model; refer to [Figure-22](#).

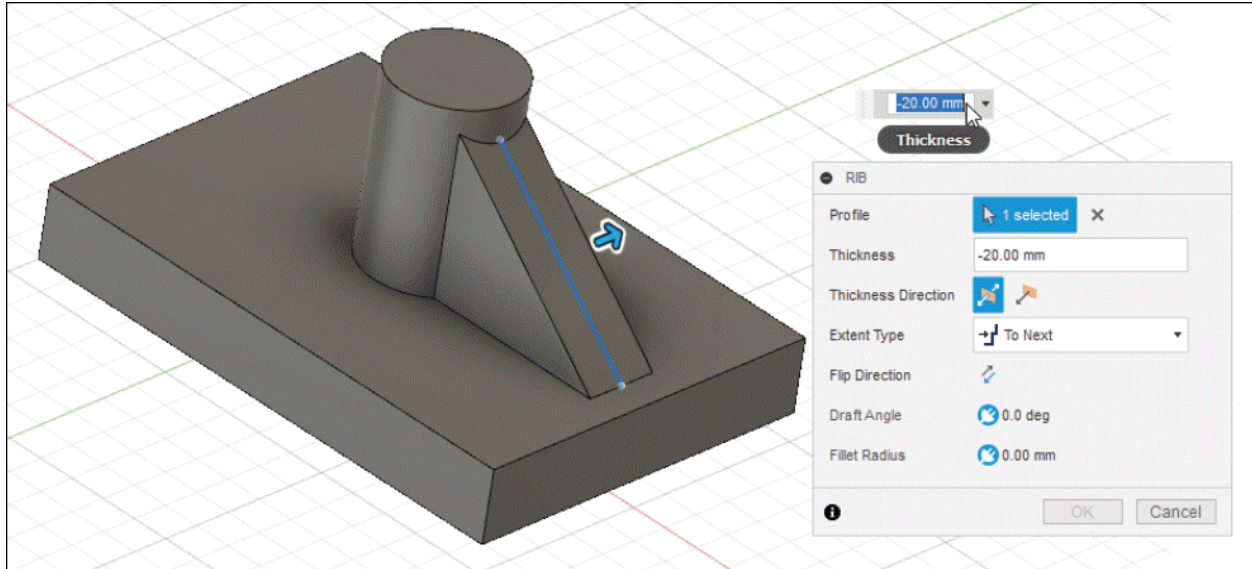


Figure-22. To Next tool in Rib

- Select the **Distance** option from **Extent Type** drop-down to specify the depth of rib. Type the numerical value of depth in **Depth** edit box or adjust the depth by moving the **Drag arrow**.
- You can also enter the value of depth in minus(-) to reverse the direction of depth in **Depth** edit box; refer to [Figure-23](#).

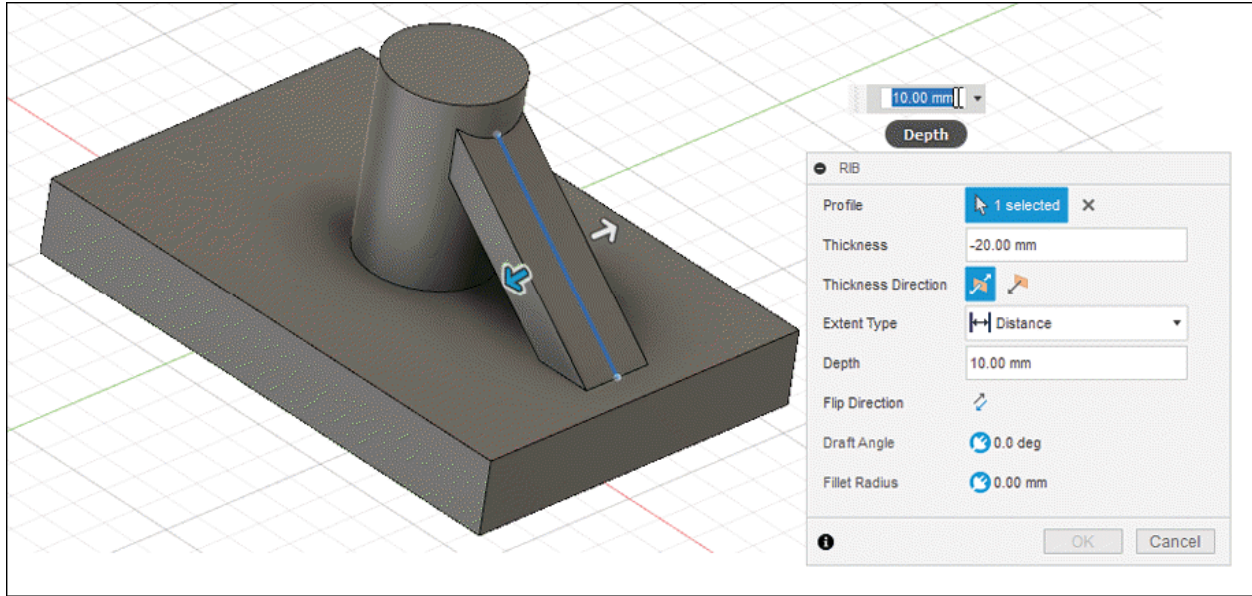


Figure-23. Depth tool in Rib

- Click on the **Flip Direction** button to flip the direction of rib.
- After specifying the parameters, click **OK** button of **RIB** dialog box to complete the process.

The **Draft Angle** and **Fillet Radius** section is inactive at this moment. It is a part of the **Product Design Extension** which is free in student version of this software. To activate these sections, follow the procedure discussed next.

- Click on the **Draft Angle** section in the **RIB** dialog box; refer to [Figure-24](#). A pop-up box will be displayed; refer to [Figure-25](#).

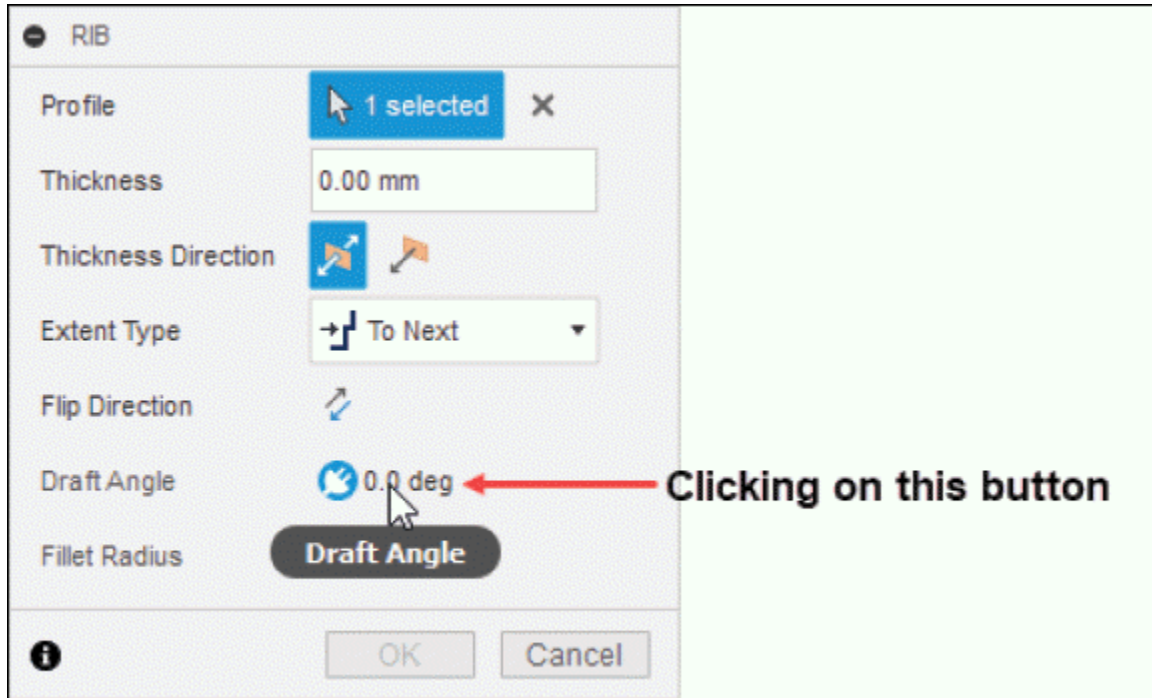


Figure-24. Clicking on the button

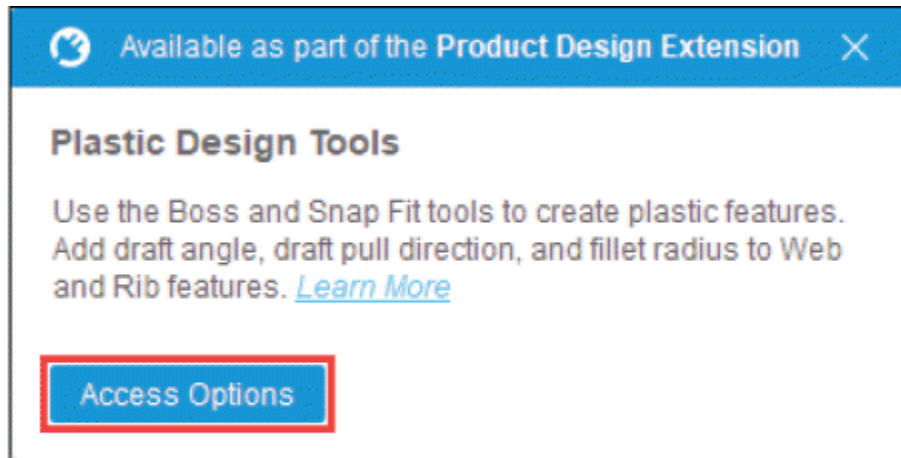


Figure-25. Pop up box on clicking Draft Angle section

- Click on the **Access Options** button highlighted in the pop-up box; refer to [Figure-25](#). The **Extension Manager** dialog box will be displayed; refer to [Figure-26](#).

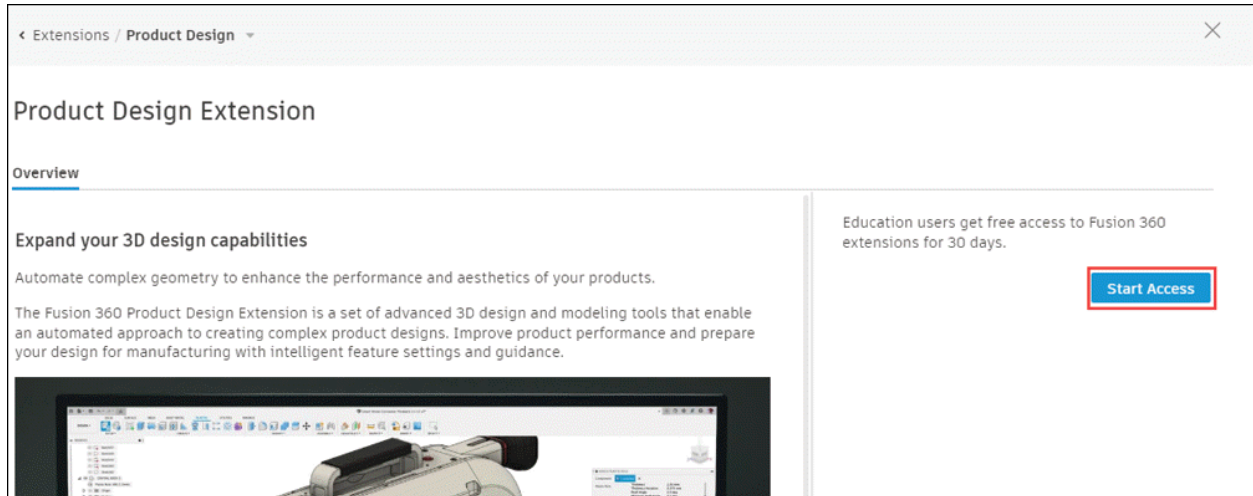


Figure-26. Extension Manager dialog box

- Click on the **Start Access** button highlighted in the dialog box; refer to [Figure-26](#). The **Product Design Extension** access will be enabled successfully; refer to [Figure-27](#).

< Extensions / Product Design ▾

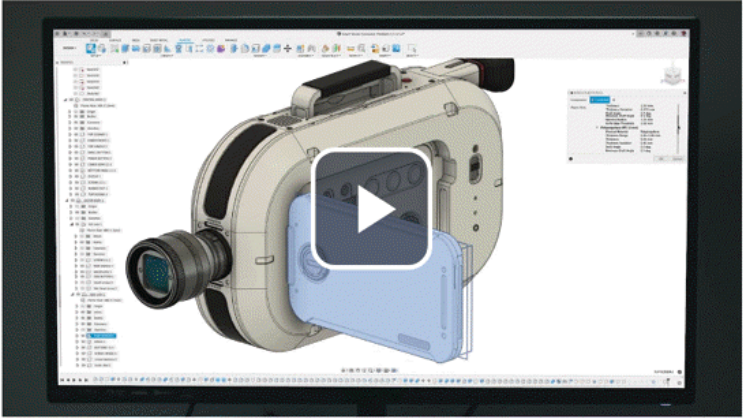
Product Design Extension

Overview


Expand your 3D design capabilities

Automate complex geometry to enhance the performance and aesthetics of your products.

The Fusion 360 Product Design Extension is a set of advanced 3D design and modeling tools that enable an automated approach to creating complex product designs. Improve product performance and prepare your design for manufacturing with intelligent feature settings and guidance.



Key technologies included

-  **Manufacturing aware functionality**
Simplify and speed up the production of milling and drilling programs with intelligent toolpaths capable of safely and efficiently processing entire parts.

Success

Access enabled

Active education customers can reactivate this extension every 30 days.

Figure-27. Product design extension access enabled

- Now, click on the **Rib** tool again. The new **RIB** dialog box will be displayed; refer to [Figure-28](#).

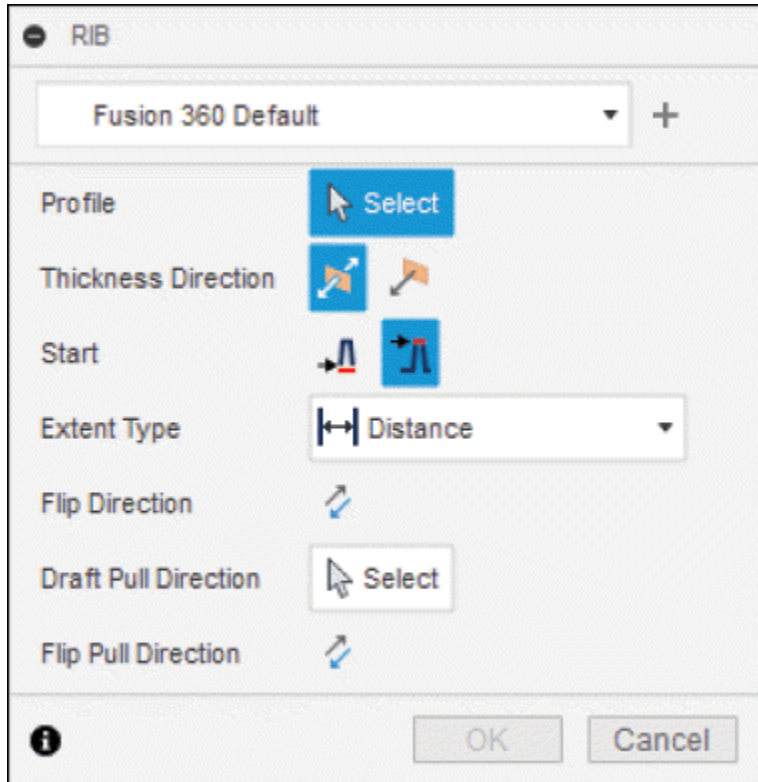


Figure-28. New RIB dialog box

- Select desired preset from **Preset** drop-down to apply or save the current values of the dialog box or create new preset by clicking on the **New Preset** button.
- Select the sketch curve as profile to create rib. The updated **RIB** dialog box will be displayed; refer to [Figure-29](#).

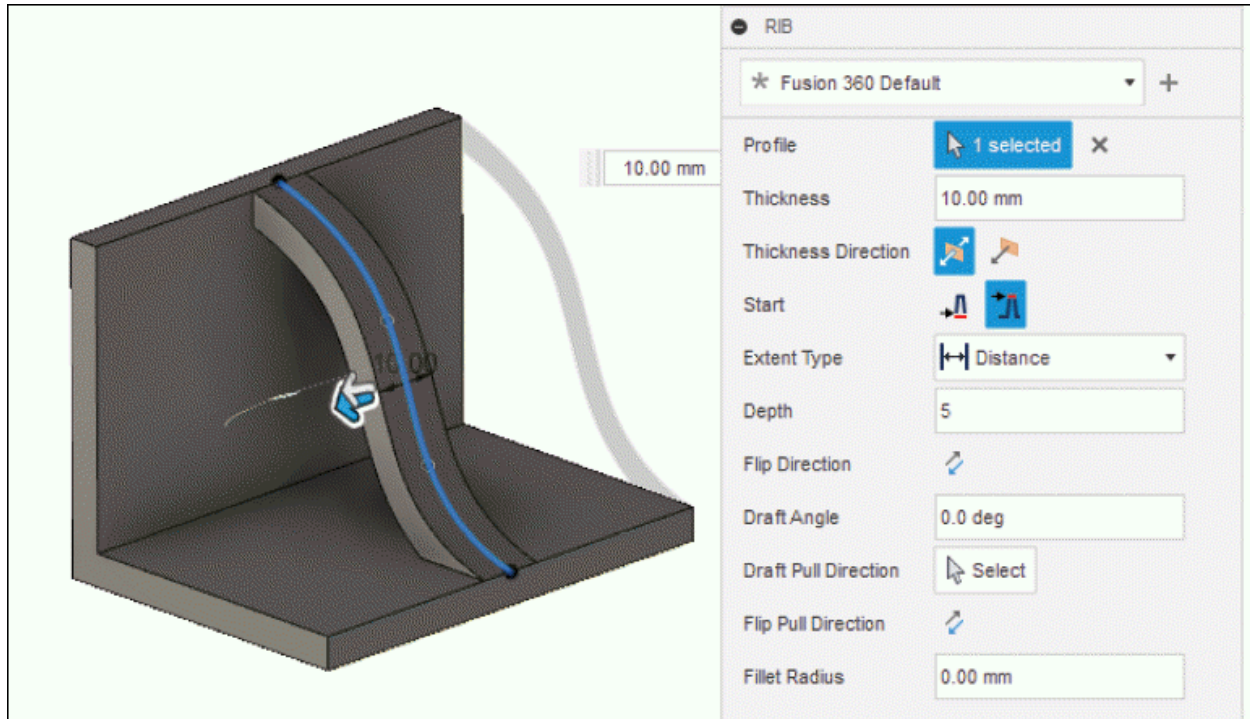


Figure-29. Updated RIB dialog box

- Select **From Bottom** button from **Start** section to measure the thickness starting from the bottom and select **From Top** button to measure the thickness starting from the top.
- Specify desired draft angle and fillet radius value in the **Draft Angle** and **Fillet Radius** edit boxes, respectively.
- Click on the **Select** button of **Draft Pull Direction** section and select desired plane or face to define the pull direction.
- Select the **Flip Pull Direction** button to flip the pull direction for draft.
- Click on the **OK** button to create the rib feature.

The **Web** tool works like **Rib** tool which is used to create geometry from sketch curves that intersect with pre-existing bodies. The **Web** tool uses multiple curves to create several merged ribs at once. The procedure to use this tool is discussed next.

- Click on **Web** tool from **CREATE** drop-down of **SOLID** tab in the **Toolbar**; refer to [Figure-30](#). The **WEB** dialog box will be displayed; refer to [Figure-31](#).

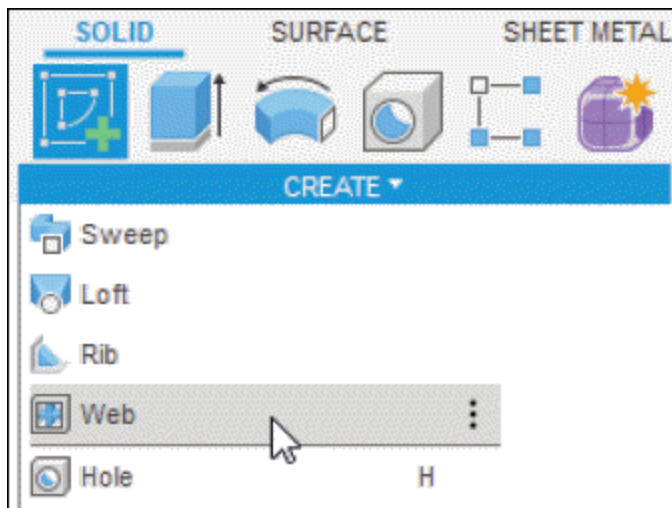


Figure-30. Web tool

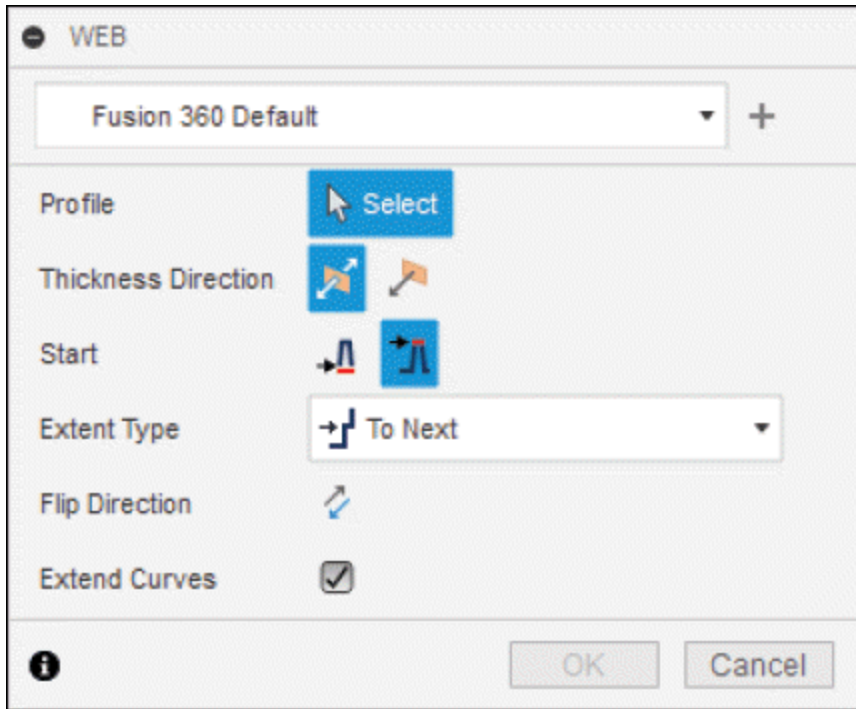


Figure-31. WEB dialog box

- The **Profile** selection is active by default. Select the sketch to create web; refer to [Figure-32](#).

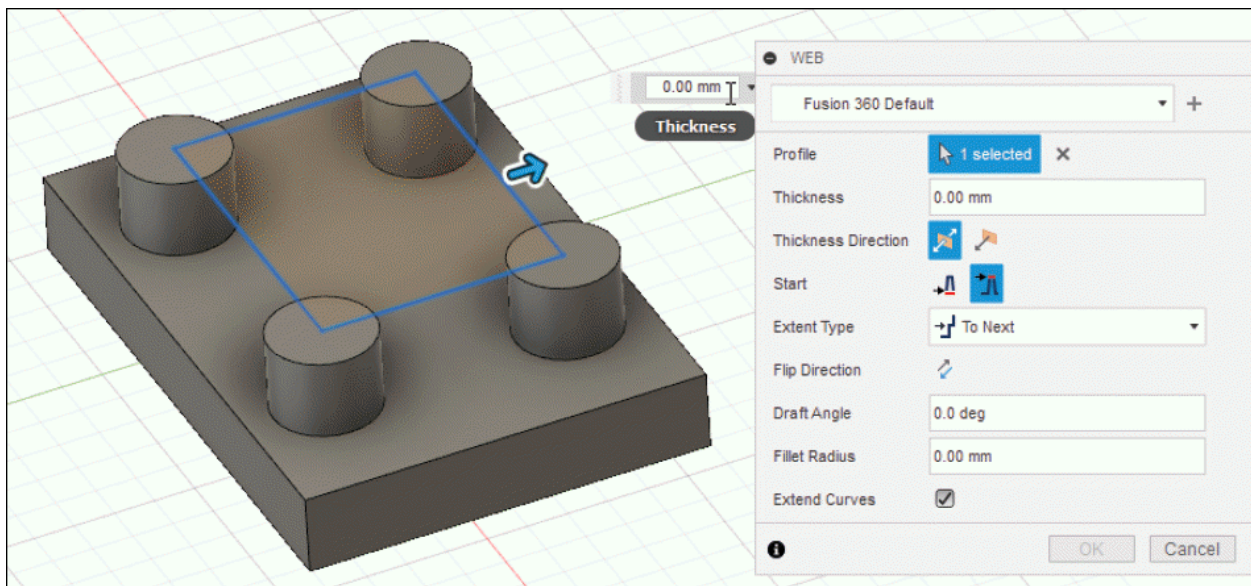


Figure-32. Selecting the sketch

- Enter desired value of thickness in the **Thickness** edit box

or floating window. You can also set the thickness by moving the drag handle.

- The other options in the dialog box are same as discussed earlier. Preview of rib will be displayed according to parameters specified; refer to [Figure-33](#).

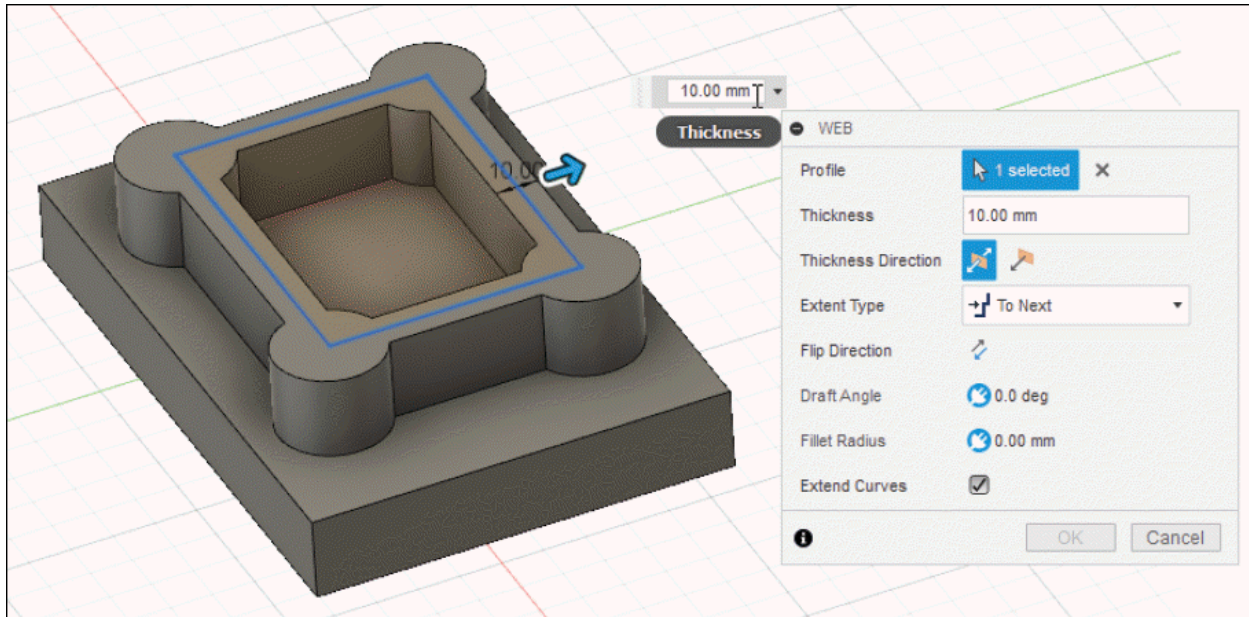


Figure-33. Preview of web

- Click **OK** button from **WEB** dialog box to complete the process.

EMBOSS

The **Emboss** tool is used to raises or recesses a sketch profile relative to selected faces on a solid body by specified depth and direction. The procedure to use this tool is discussed next.

- Click on the **Emboss** tool from **CREATE** drop-down of **SOLID** tab in the **Toolbar**; refer to [Figure-34](#). The **EMBOSS** dialog box will be displayed; refer to [Figure-35](#).

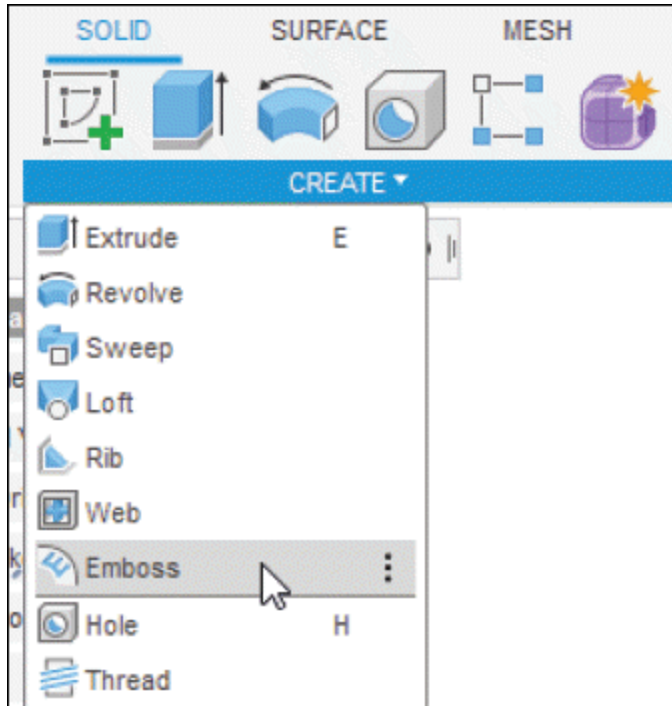


Figure-34. Emboss tool

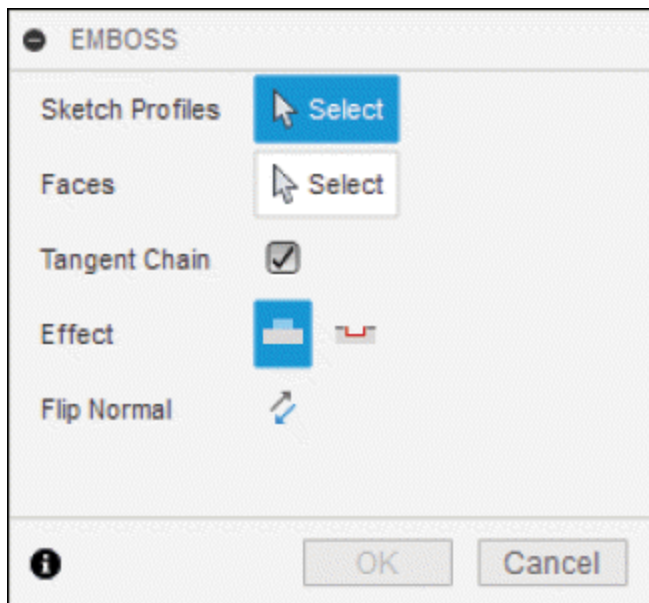


Figure-35. EMBOSS dialog box

- The **Select** button of **Sketch Profiles** section is active by default. Select the sketch profile which you want to emboss on a solid body; refer to [Figure-36](#).

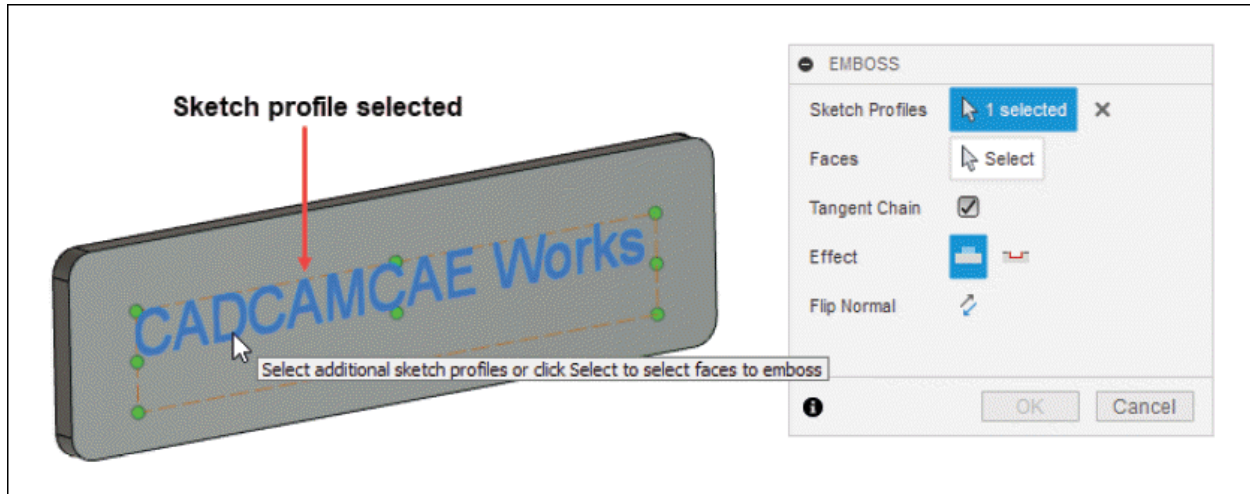


Figure-36. Sketch profile selected

- Click on the **Select** button of **Faces** section and select the face on a solid body on which the sketch profile is to be embossed; refer to [Figure-37](#). The updated **EMBOSS** dialog box along with the preview of emboss will be displayed; refer to [Figure-38](#).

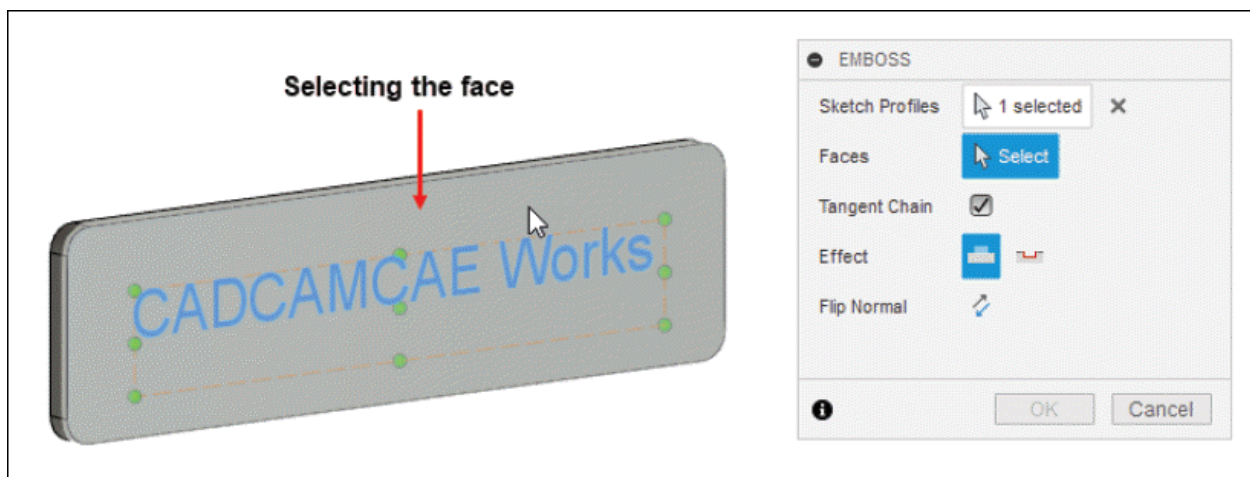


Figure-37. Selecting the face

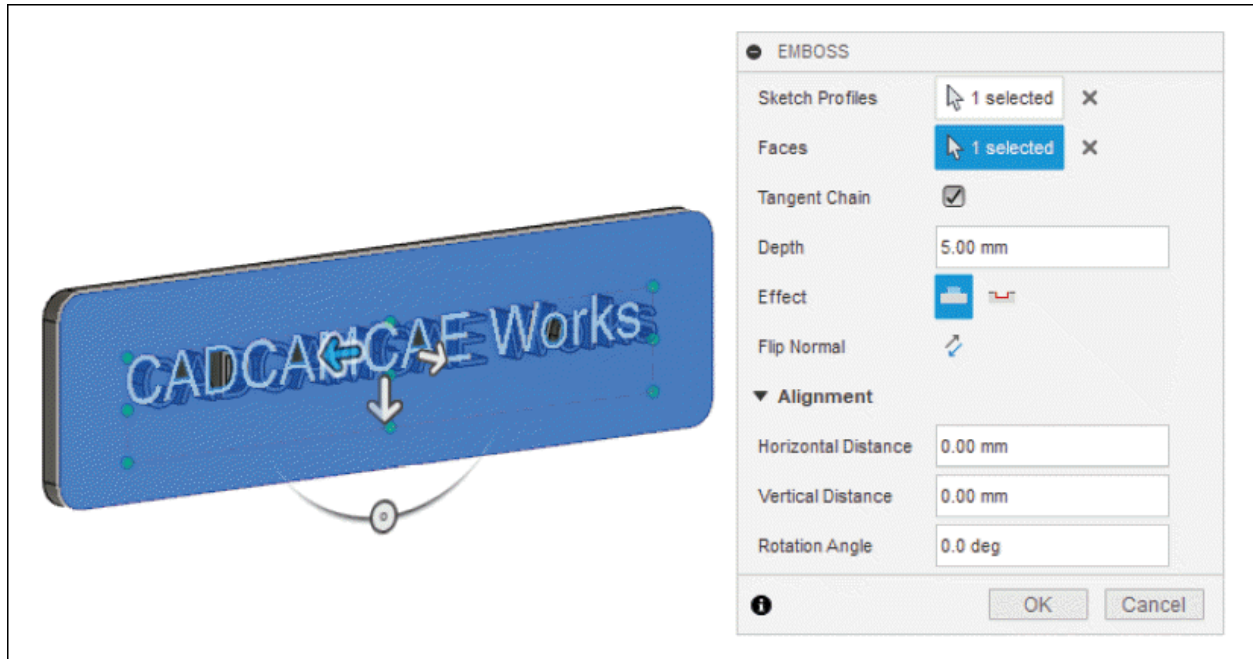


Figure-38. Updated EMBOSS dialog box

- Select **Tangent Chain** check box to automatically select tangent geometry.
- Type desired value in the **Depth** edit box to specify the depth to emboss or deboss.
- Select **Emboss** button from **Effect** section of the dialog box to add material to the body and select **Deboss** button to remove material from the body.
- Select **Flip Normal** button to flip the normal side of sketch profiles.
- Drag the distance and angle manipulator handles or enter desired values in the **Horizontal Distance**, **Vertical Distance**, and **Rotation Angle** edit boxes from **Alignment** section to position the emboss feature on the solid body.

- After specifying desired parameters, click on the **OK** button from the dialog box. The emboss or deboss feature will be created; refer to [Figure-39](#).

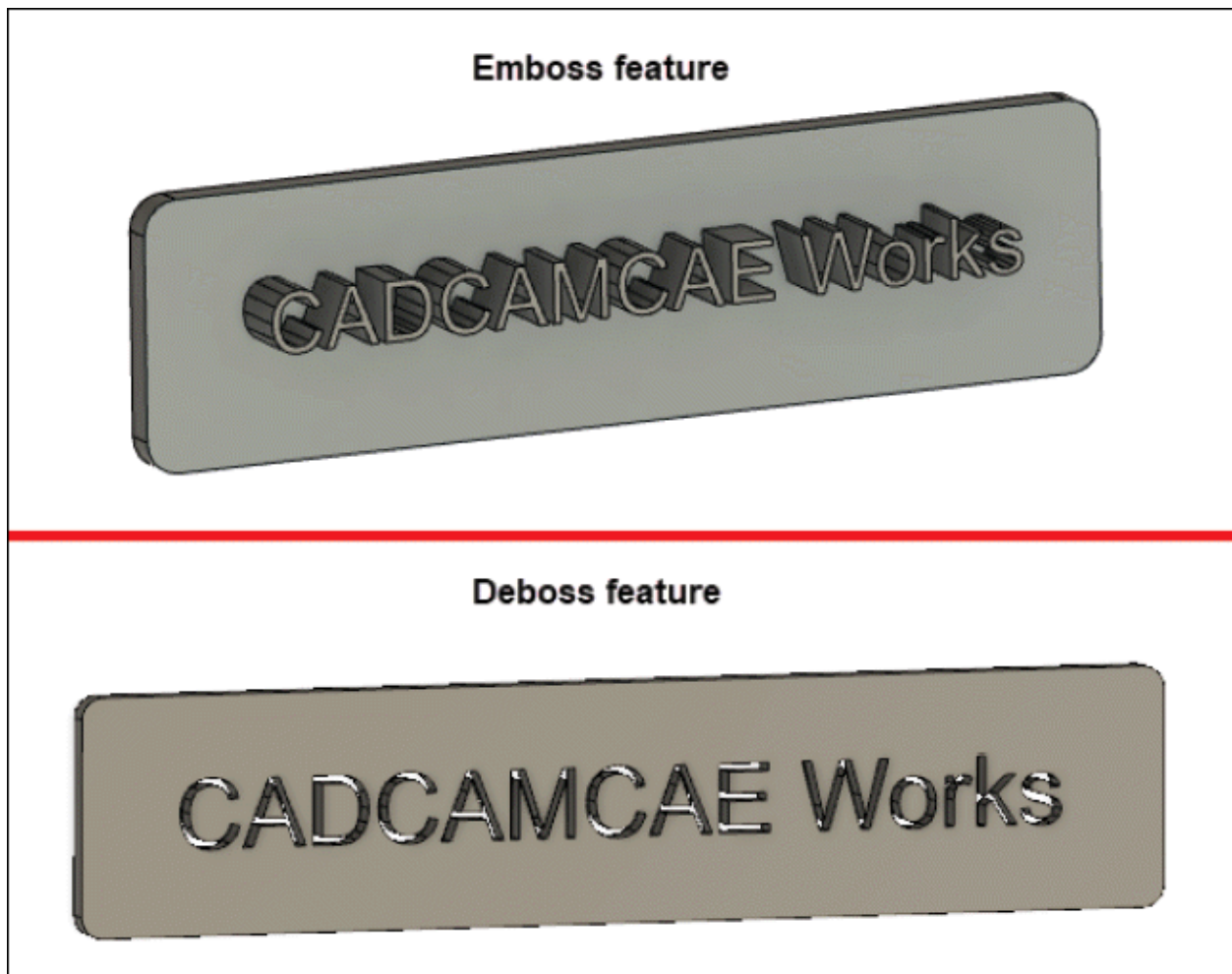


Figure-39. Emboss and Deboss feature created

HOLE

The **Hole** tool is used to create holes that comply with real machining tools. With the help of **Hole** tool, three types of holes are created. The procedure to use this tool is discussed next.

- Click on **Hole** tool from **CREATE** drop-down or right-click

on the screen canvas and select **Hole** from **Marking menu** displayed; refer to [Figure-40](#). The **HOLE** dialog box will be displayed; refer to [Figure-41](#).

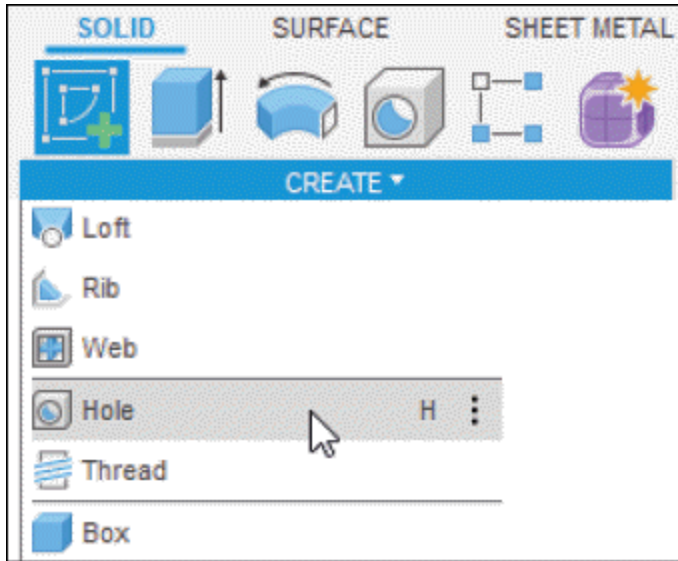


Figure-40. Hole tool

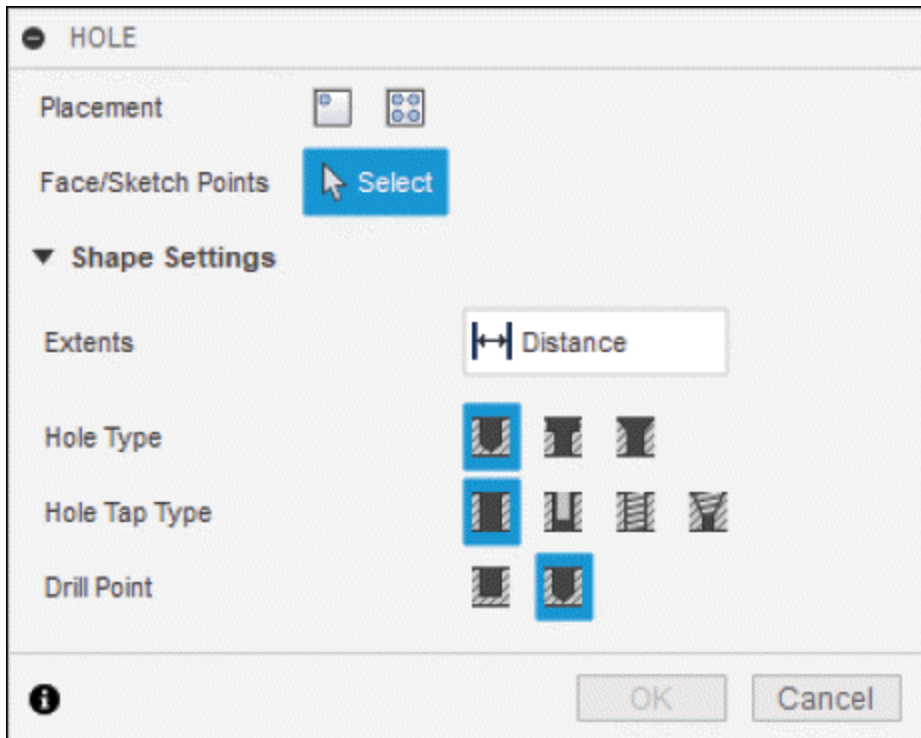


Figure-41. HOLE dialog box

- There are two types of tools in **Placement** section which are discussed next.

Single Hole

- Click on **Single Hole** button from **Placement** section to create one hole. The **Select** button for **Face** section will be activated.
- Select the face/plane on which you want to create hole. The updated **HOLE** dialog box will be displayed with preview of hole; refer to [Figure-42](#).

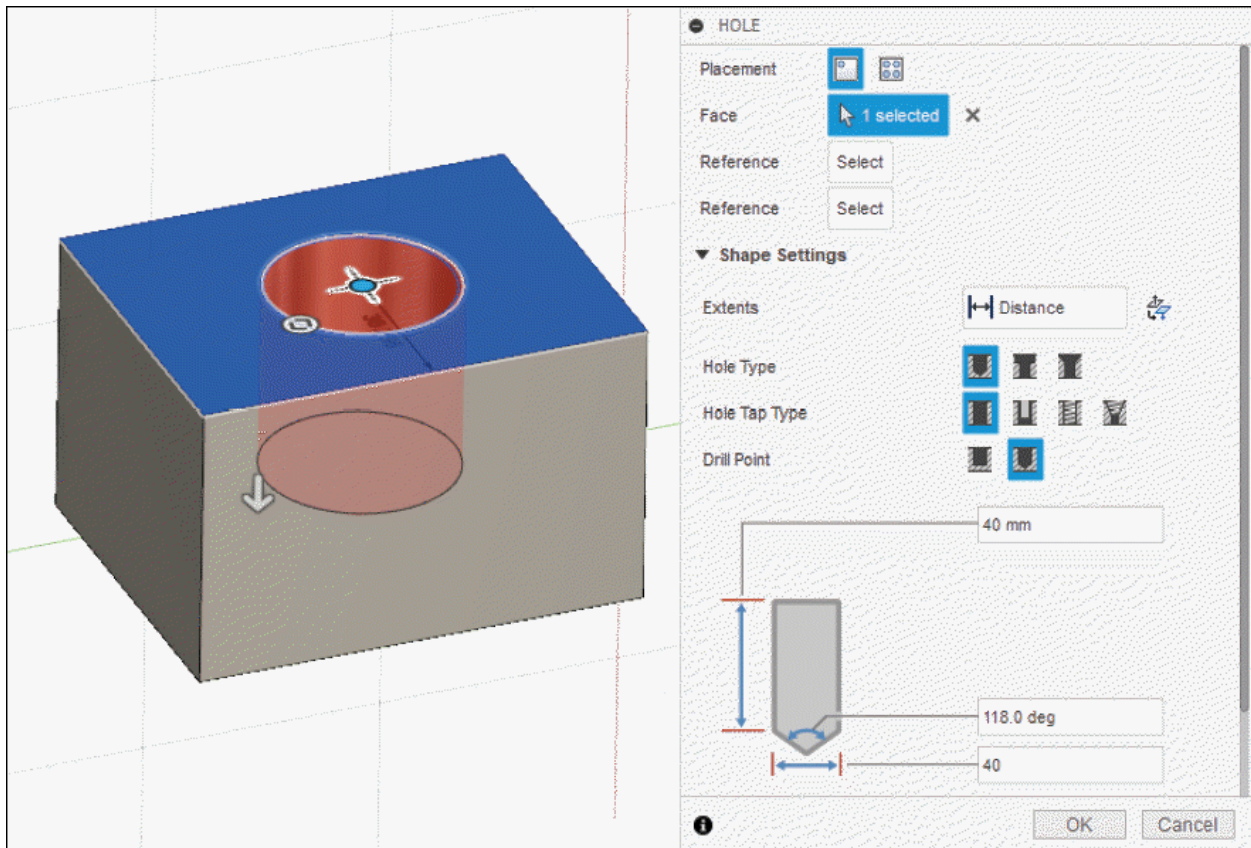


Figure-42. Updated HOLE dialog box

- Click on **Select** button of first **Reference** section from **HOLE** dialog box, click at the center of hole and then click

at the edge from where you want to specify the distance of hole; refer to [Figure-43](#).

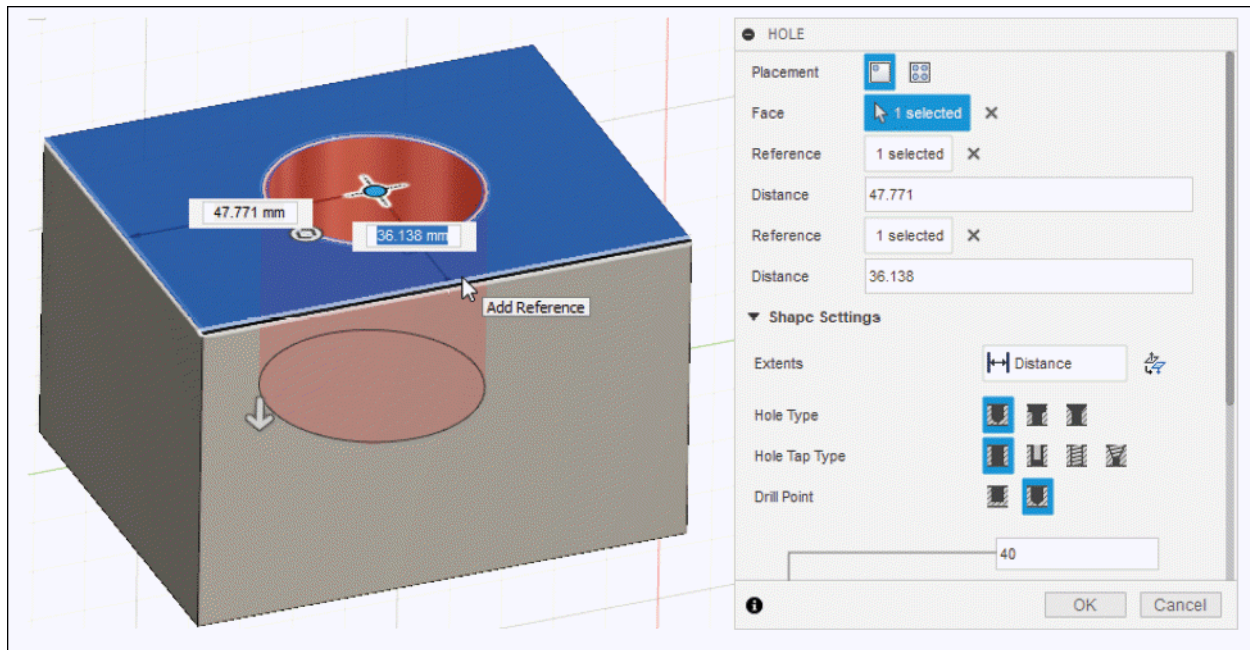


Figure-43. Adding references

- Enter desired value of distance in the dynamic input box or **Distance** edit box in dialog box. Click on another edge to define other reference and set desired distance value.
- Click on **Simple** button in **Hole Type** section to create simple hole of specified dimensions.
- Click on **Counterbore** button in **Hole Type** section to create a counterbore hole and specify the dimension in their respective edit boxes at the bottom in the dialog box; refer to [Figure-44](#).

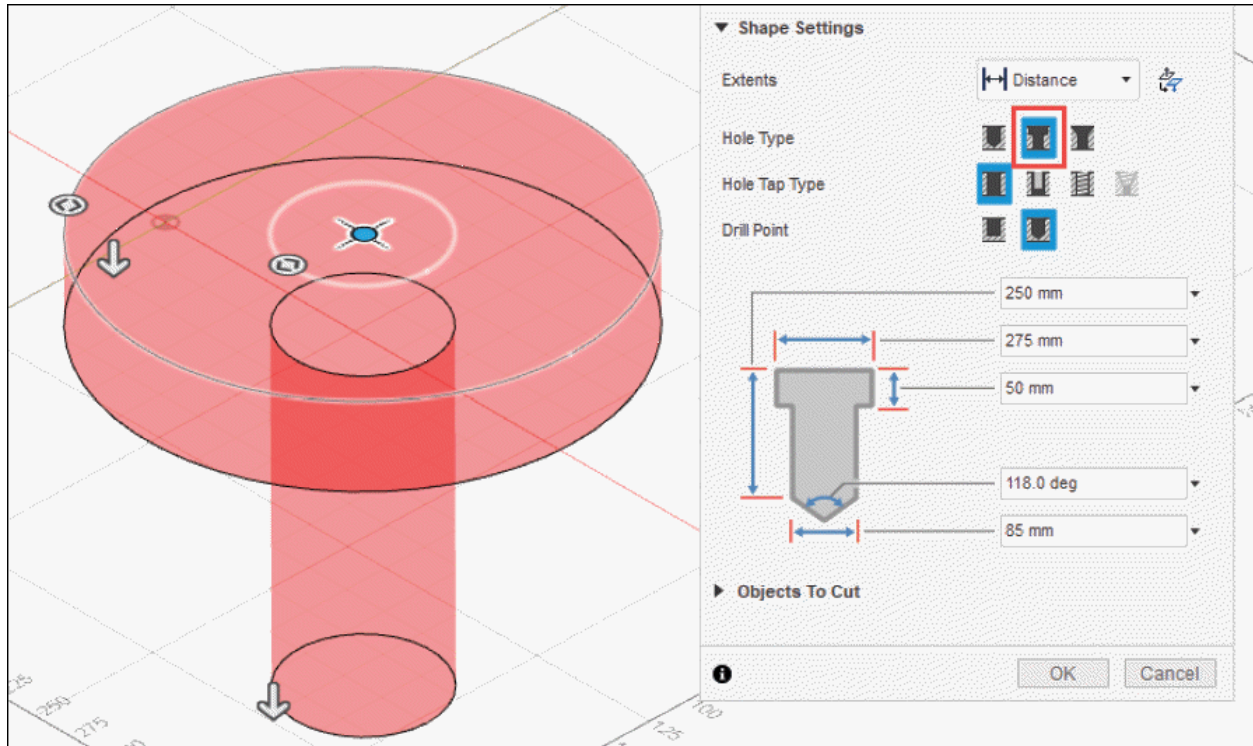


Figure-44. Counterbore hole

- Click on **Countersink** button in **Hole Type** drop-down to create a countersink hole and specify the dimension in their respective edit box at the bottom in the dialog box; refer to [Figure-45](#).

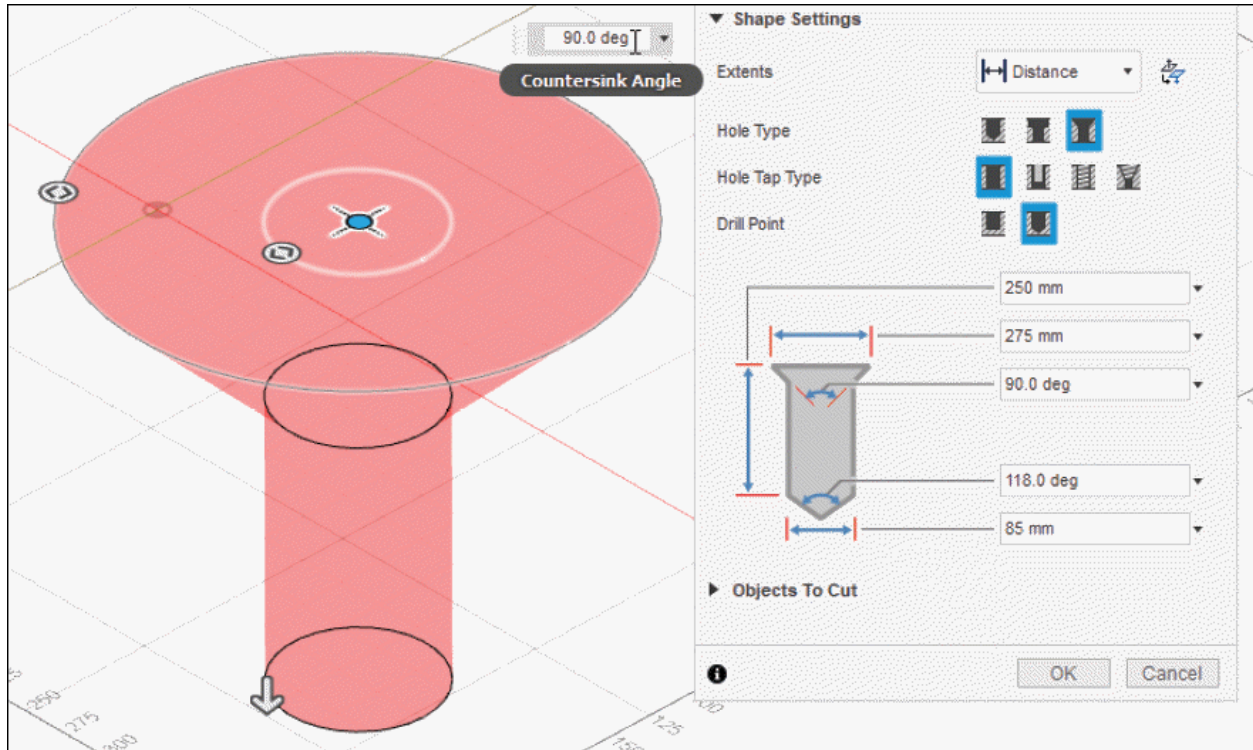


Figure-45. Countersink hole

- Click on desired button in the **Hole Tap Type** section to define tapping inside the hole. Select the **Simple** button to create straight holes without tapping.
- Select the **Clearance** button to create standard holes with defined fit. The options will be displayed accordingly; refer to [Figure-46](#). Select desired options to define size of hole.

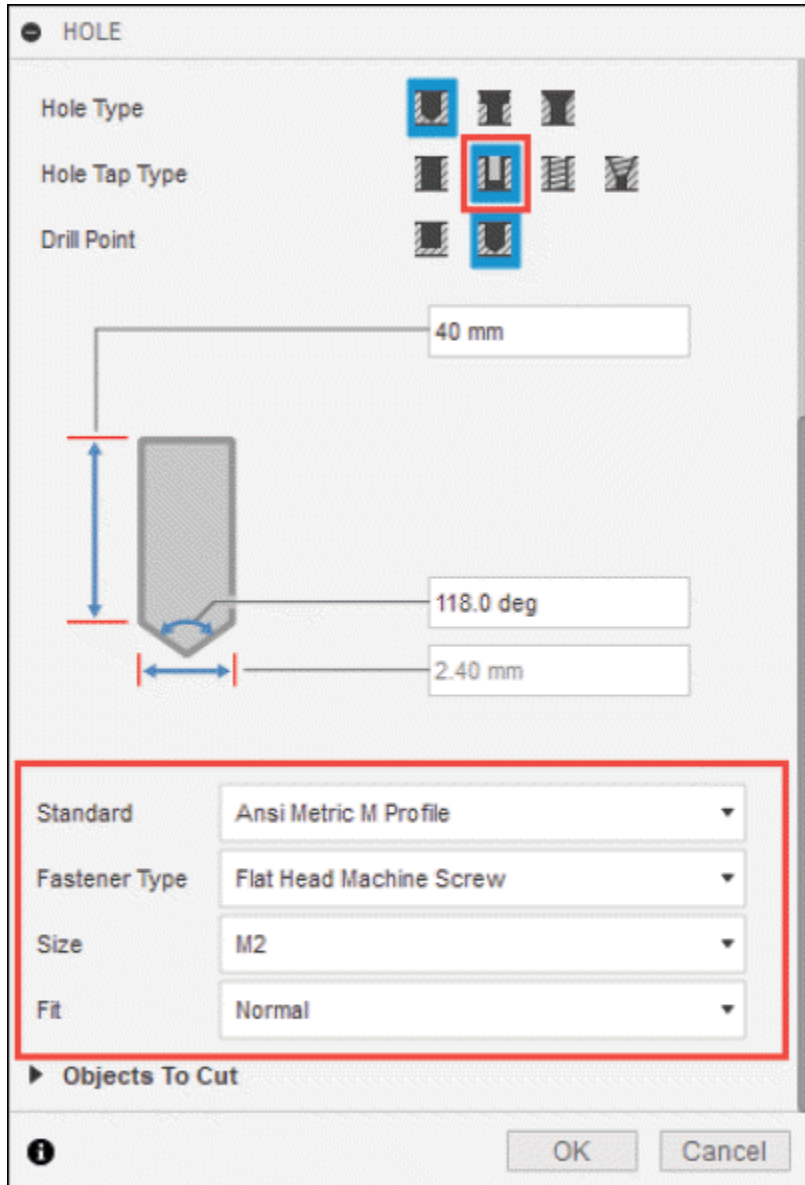


Figure-46. Options for clearance hole tap type

- Select the **Tapped** button to create holes as per their class and designation in standard hole profile database; refer to [Figure-47](#).

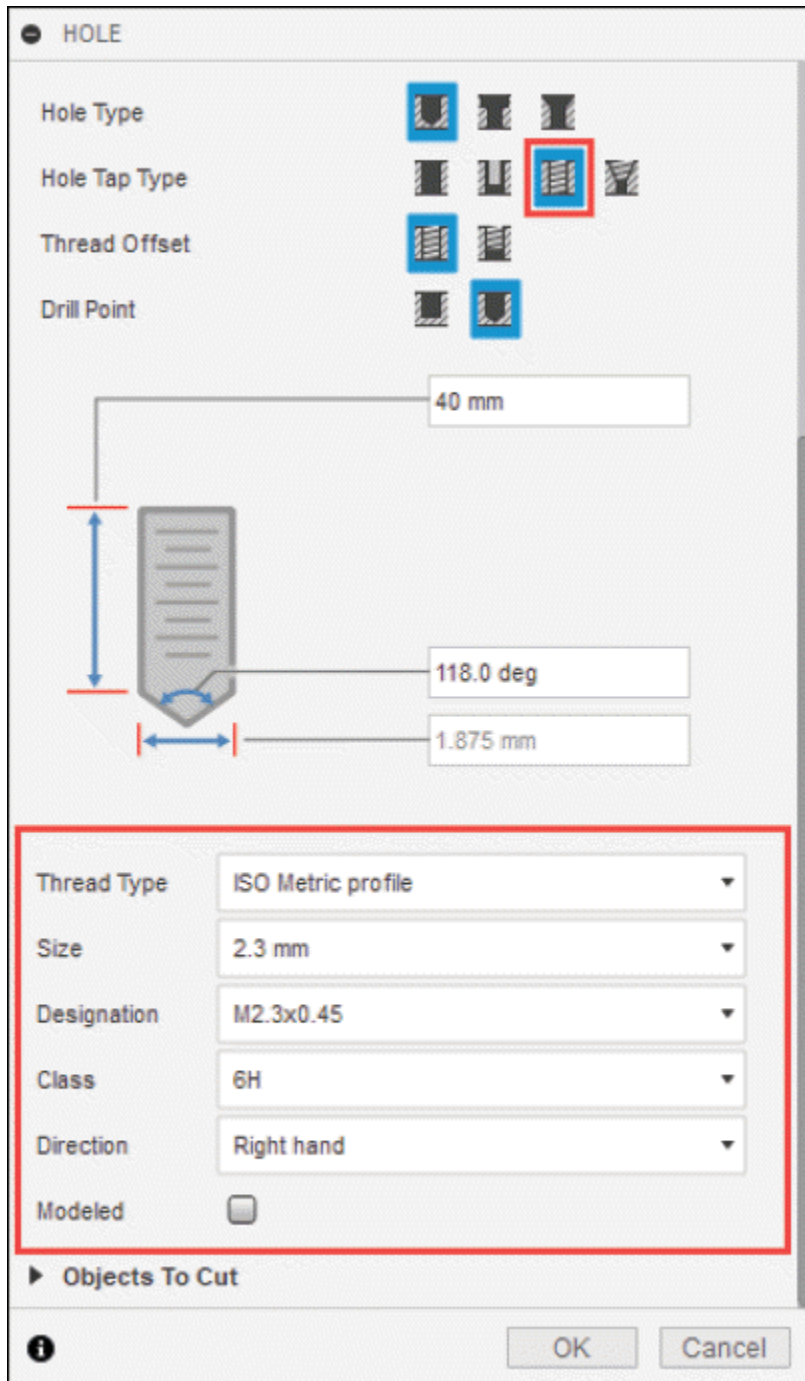


Figure-47. Options for tapped hole tap type

- Select desired options to define shape and size of hole. If you want to model threads created by tapping in the hole then select the **Modeled** check box. Preview of hole will be

displayed with selected thread type; refer to [Figure-48](#).

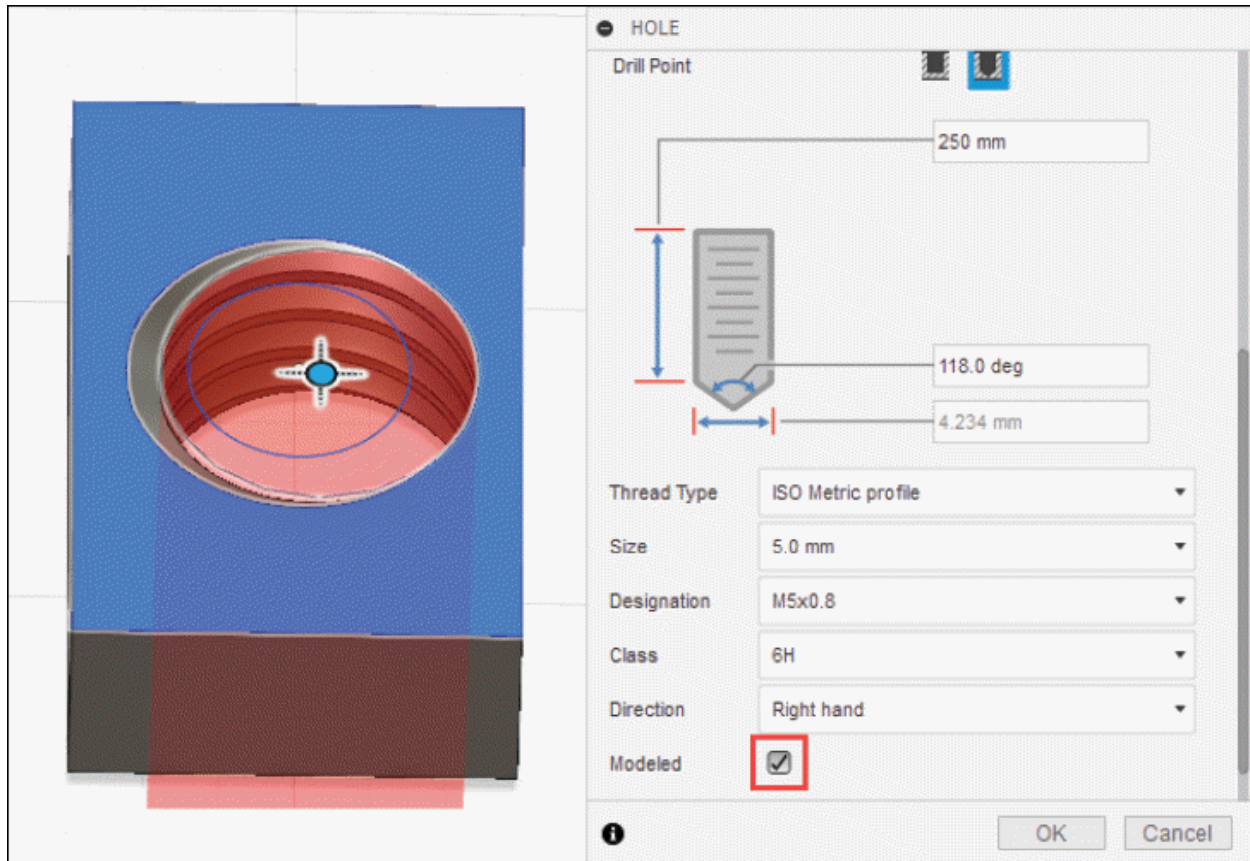


Figure-48. Preview of threads modeled in hole

- Select the **Distance** option from **Extents** drop-down to specify the distance of hole in the model. Select the **To** option from **Extents** drop-down to set the length of hole up to next model or face. Select the **All** option from **Extents** drop-down to create a hole passing through all the features in its path.
- Click on **Flip Direction** button to reverse the direction of hole.
- Expand the **Objects To Cut** node and select check boxes for bodies that you want to be cut by the hole.

- After specifying the parameters, click **OK** button of **HOLE** dialog box to complete the process; refer to [Figure-49](#).

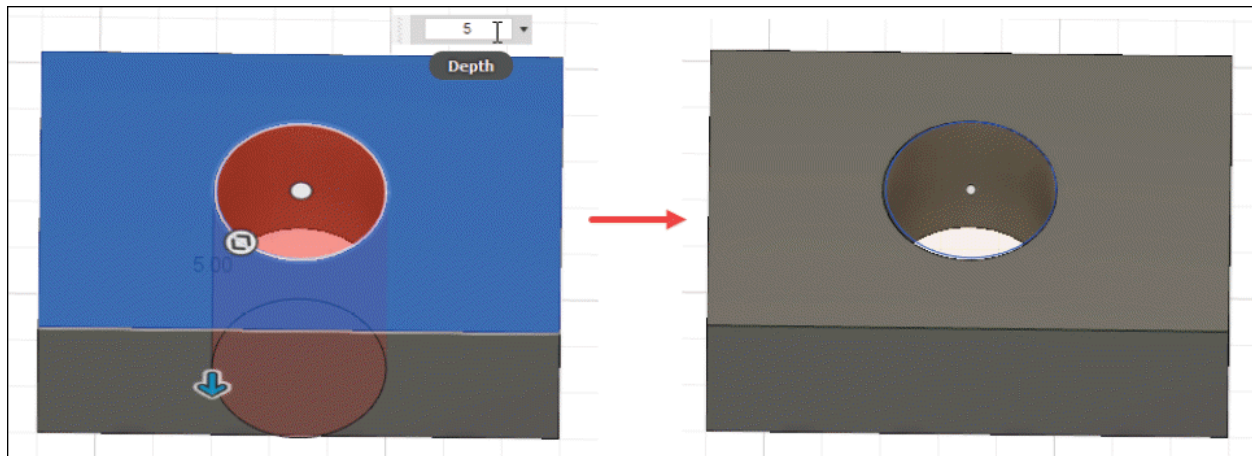


Figure-49. Preview of hole

Multiple Holes

The **Multiple Holes** button in **HOLE** dialog box is used to create multiple similar hole at a time. To use this tool, you must have sketch points already created on the model where you want to cut holes. The procedure to use this tool is discussed next.

- Click on the **Multiple Holes** button in **Placement** section from **HOLE** dialog box. The **Sketch Point** selection tool will be activated. Select the sketch points for defining positions of holes; refer to [Figure-50](#).

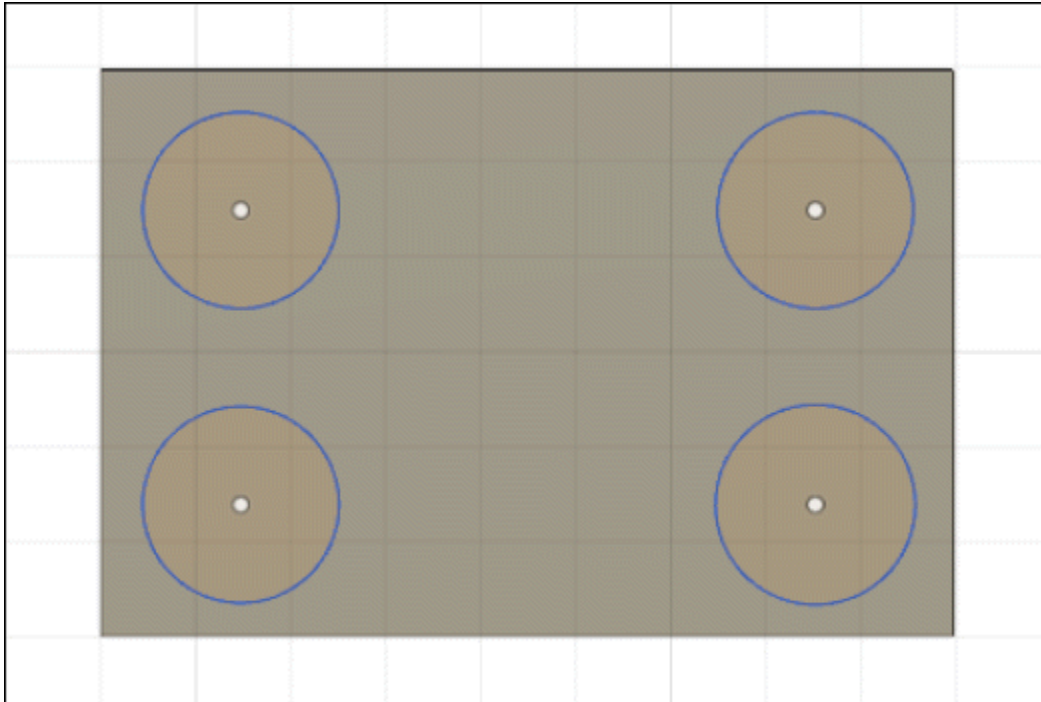


Figure-50. Sketch for multiple holes

- Set the other options as discussed earlier.
- After specifying the parameters, click **OK** button from **HOLE** dialog box to finish the process. The holes will be created; refer to [Figure-51](#).

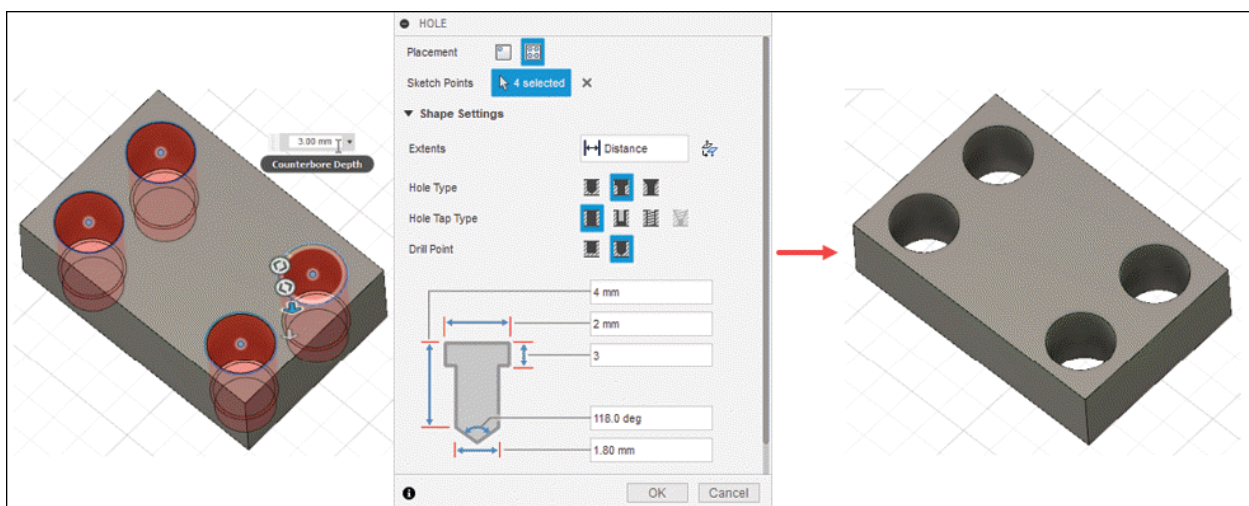


Figure-51. Preview of multiple holes

THREAD

The **Thread** tool is used to cut helical thread on cylindrical faces. Using this tool, you can save the custom threads in library. The procedure to use this tool is discussed next.

- Click on the **Thread** tool from the **CREATE** drop-down in the **Toolbar**; refer to [Figure-52](#). The **THREAD** dialog box will be displayed; refer to [Figure-53](#).

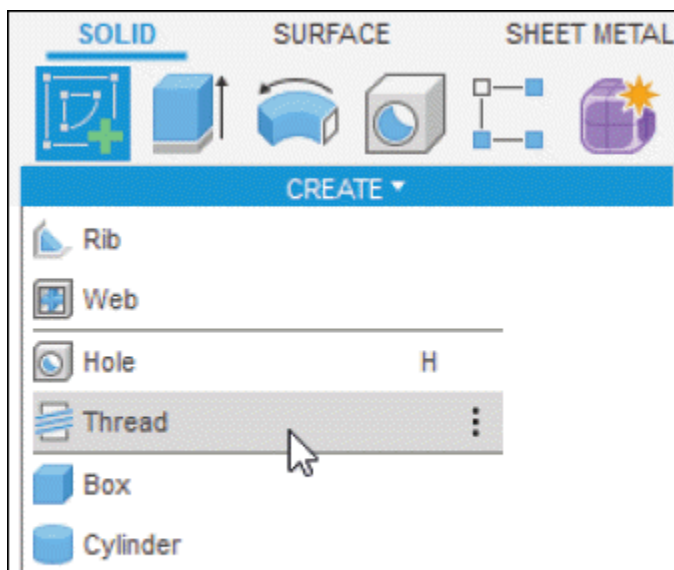


Figure-52. Thread tool

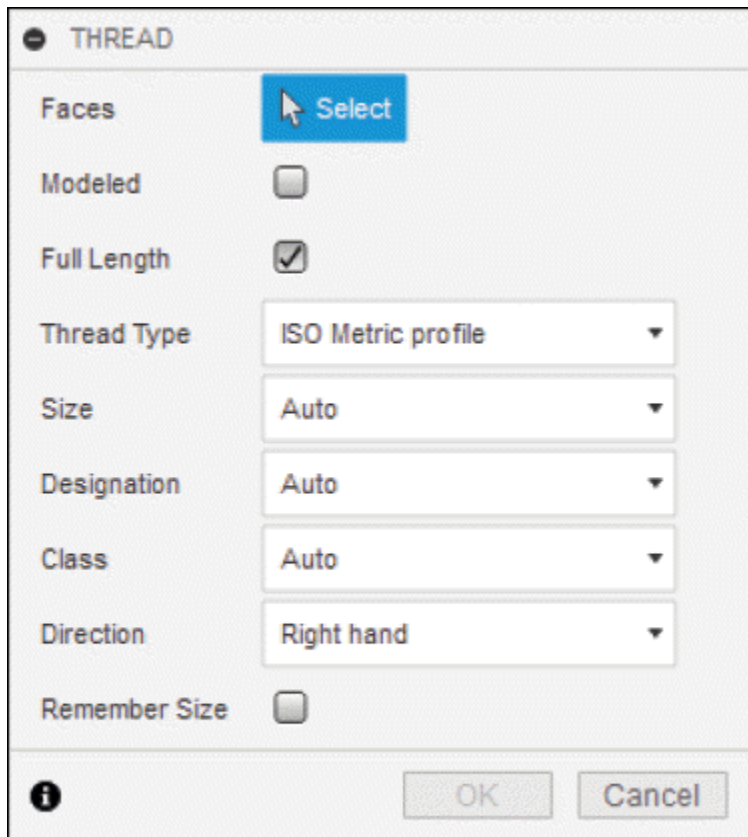


Figure-53. THREAD dialog box

- The **Faces** selection of **THREAD** dialog box is active by default. Select the round faces to which threads are to be applied.
- Select the **Modeled** check box to create the model feature of thread otherwise it will be created as cosmetic feature.
- On selecting **Modeled** check box, preview of thread will be displayed; refer to [Figure-54](#).

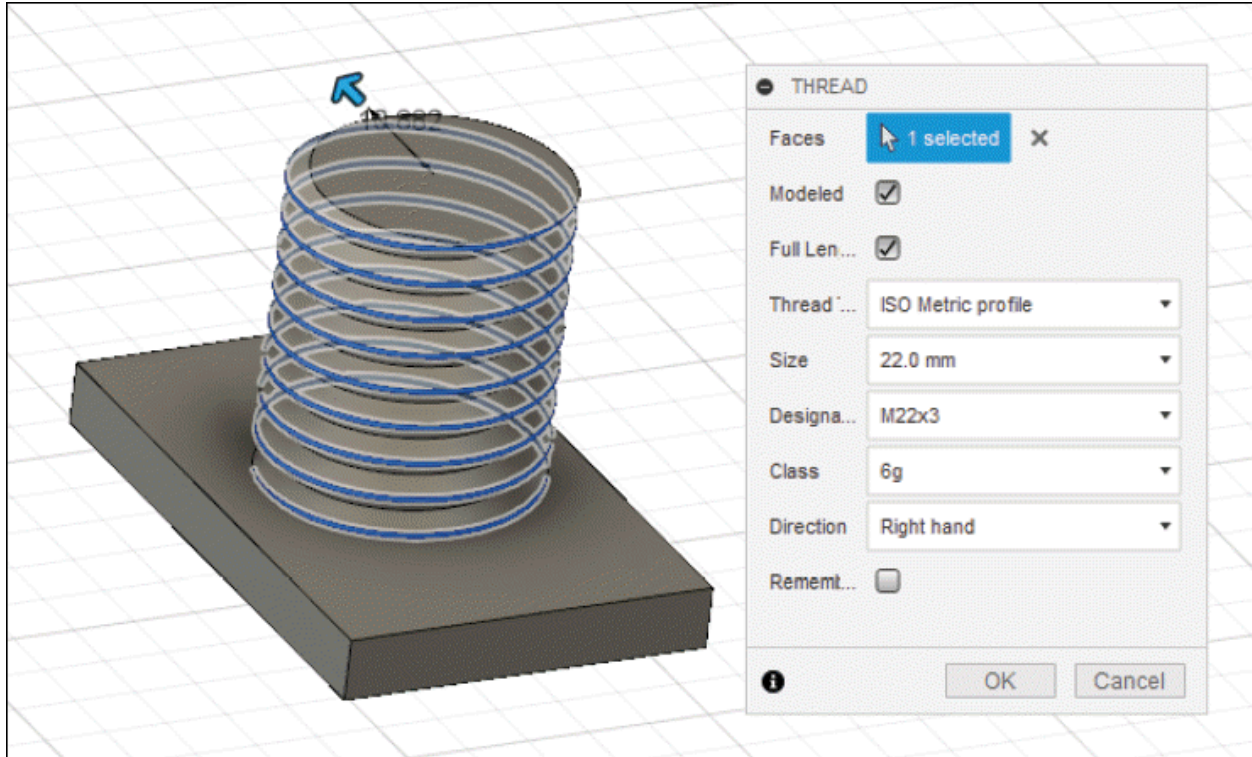


Figure-54. Preview of thread on selecting Modeled check box

- Clear the **Full Length** check box to specify the length of thread on the selected face. Otherwise, thread will be created up to the entire length of selected face.
- Click on the **Thread Type** drop-down to select the type of thread profile; refer to [Figure-55](#). Most of the engineering thread profiles are available in this drop-down.

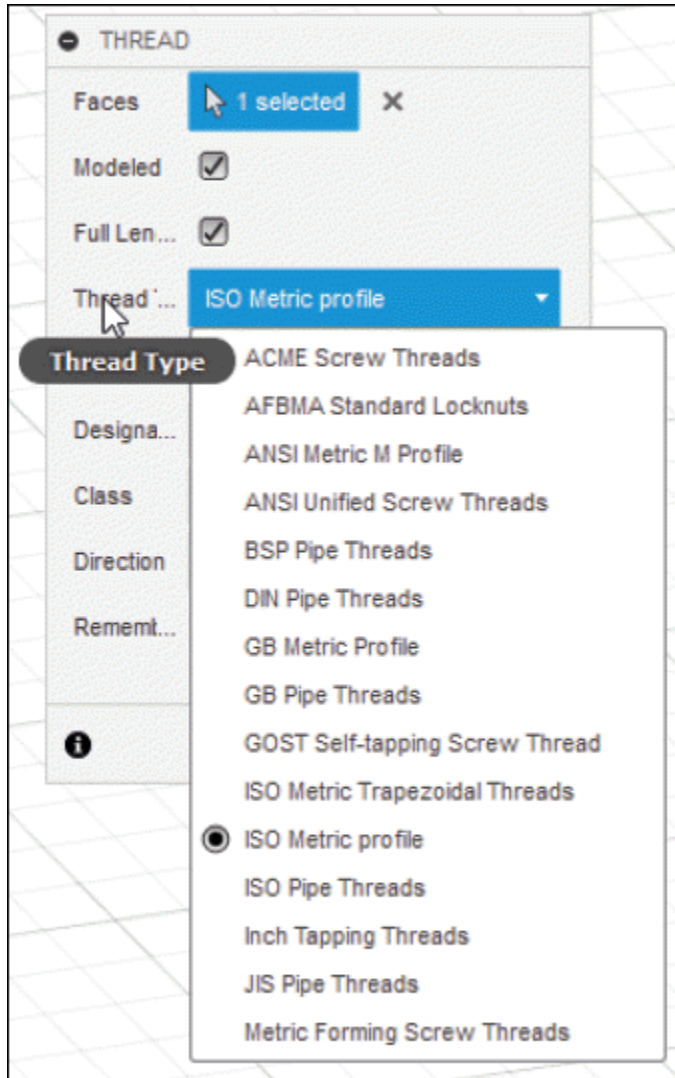


Figure-55. Thread Type drop down

- Click on the **Size** drop-down and select desired size of thread. You can also adjust the size by using the **Drag Handle**.
- Click on the **Designation** drop-down and select the designation of thread; refer to [Figure-56](#).
- Click on the **Class** drop-down and select the class of thread.

- Click on **Direction** drop-down and select the direction for thread; refer to [Figure-57](#).

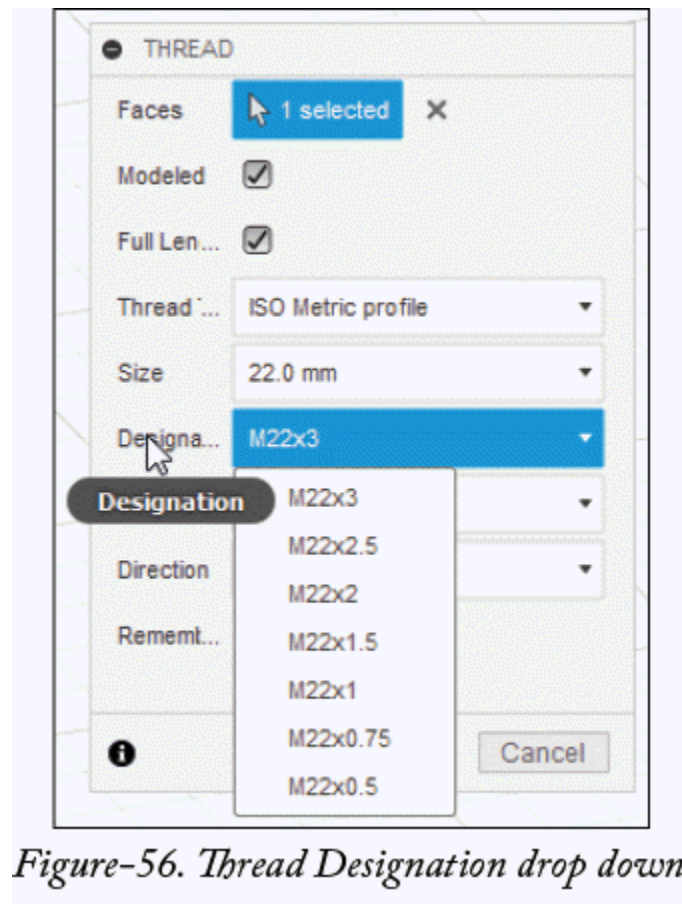


Figure-56. Thread Designation drop down

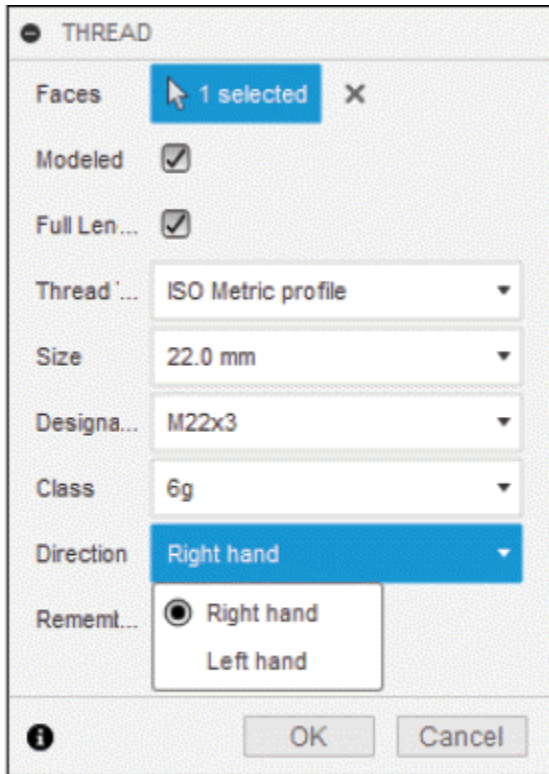


Figure-57. Thread Direction drop down

- Select the **Remember Size** check box to remember these thread setting for the next time, when the **Thread** tool will be invoked.
- Click **OK** button from **THREAD** dialog box to create the thread; refer to [Figure-58](#).

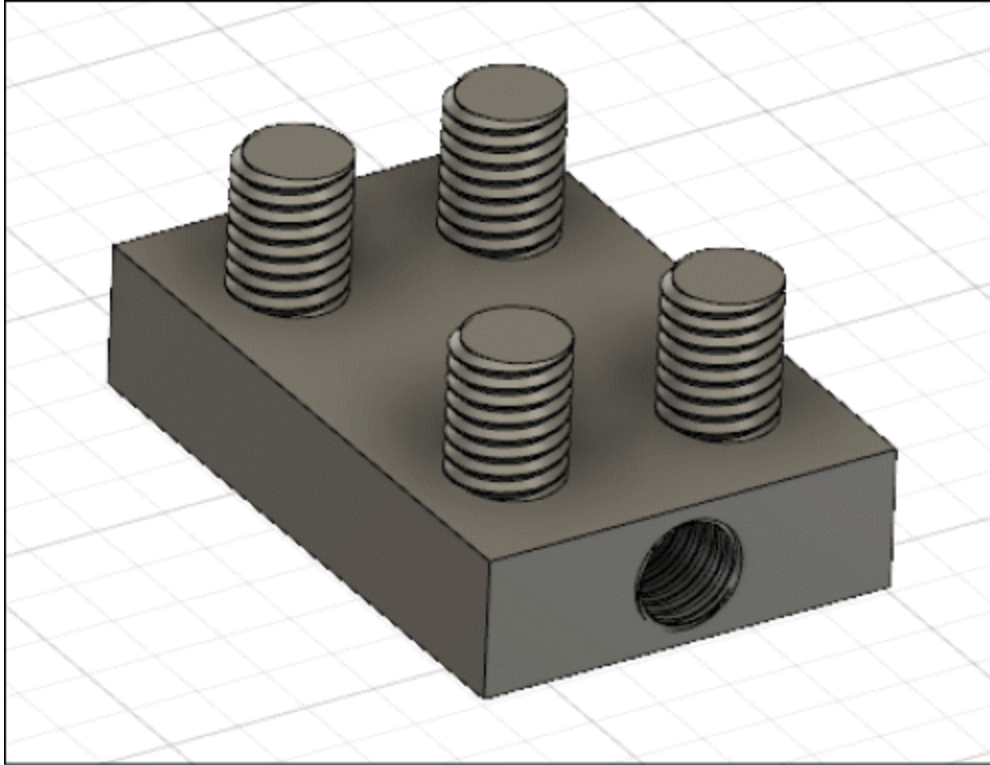


Figure-58. Thread created in solid

BOX

The **Box** tool is used to create a rectangular box on selected work plane. The procedure to use this tool is discussed next.

- Click on **Box** tool in **CREATE** drop-down from **Toolbar**; refer to [Figure-59](#). You will be asked to select the plane for creating the sketch for box.

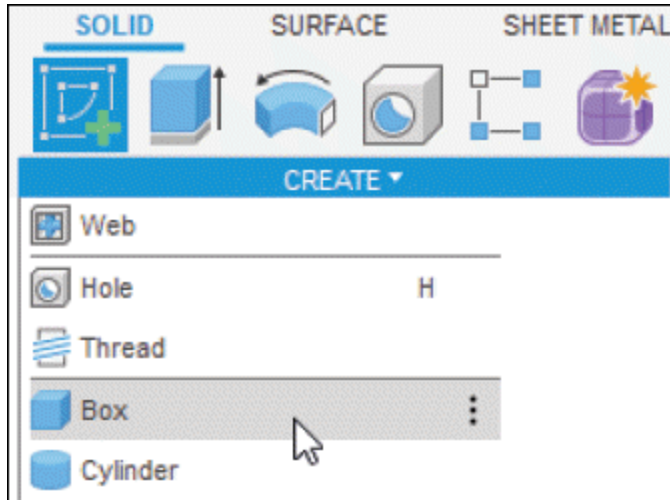


Figure-59. Box tool

- Select **Top** plane or any other desired plane/face.
- Click on the screen to specify the first point of sketch.
- Move the cursor to display preview of rectangle and click at desired location to specify other corner of rectangle. You can also enter the numerical value of **Length** and **Width** in their specific edit box; refer to [Figure-60](#).

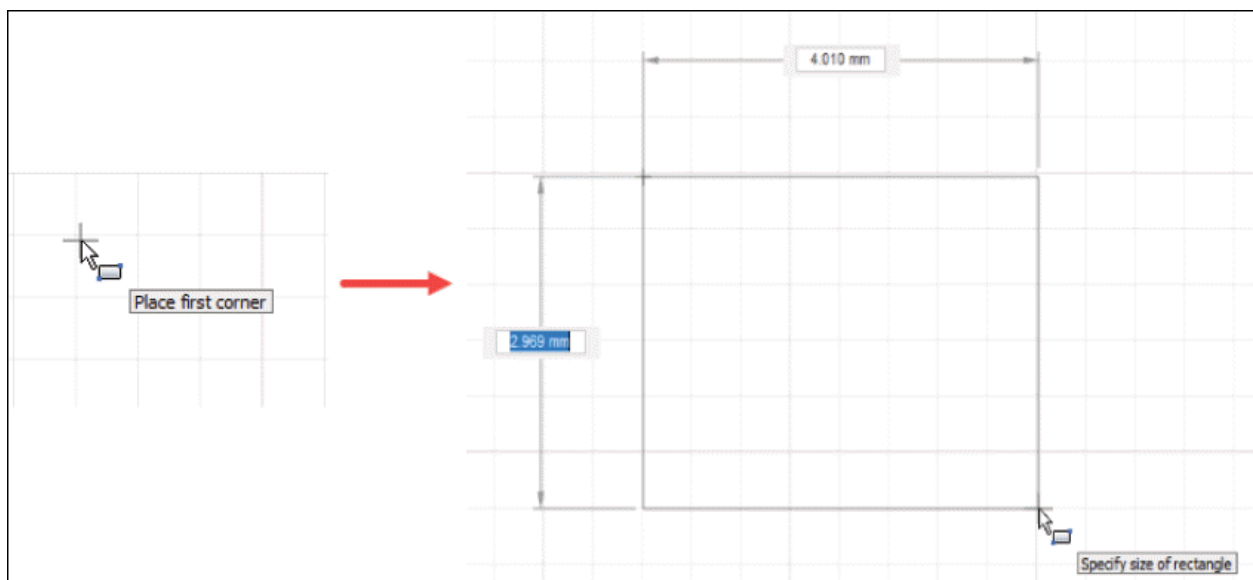


Figure-60. Entering dimension for rectangle

- You can toggle between these two edit boxes by pressing **TAB** key. The **Box** dialog box will be displayed along with the preview of box; refer to [Figure-61](#).

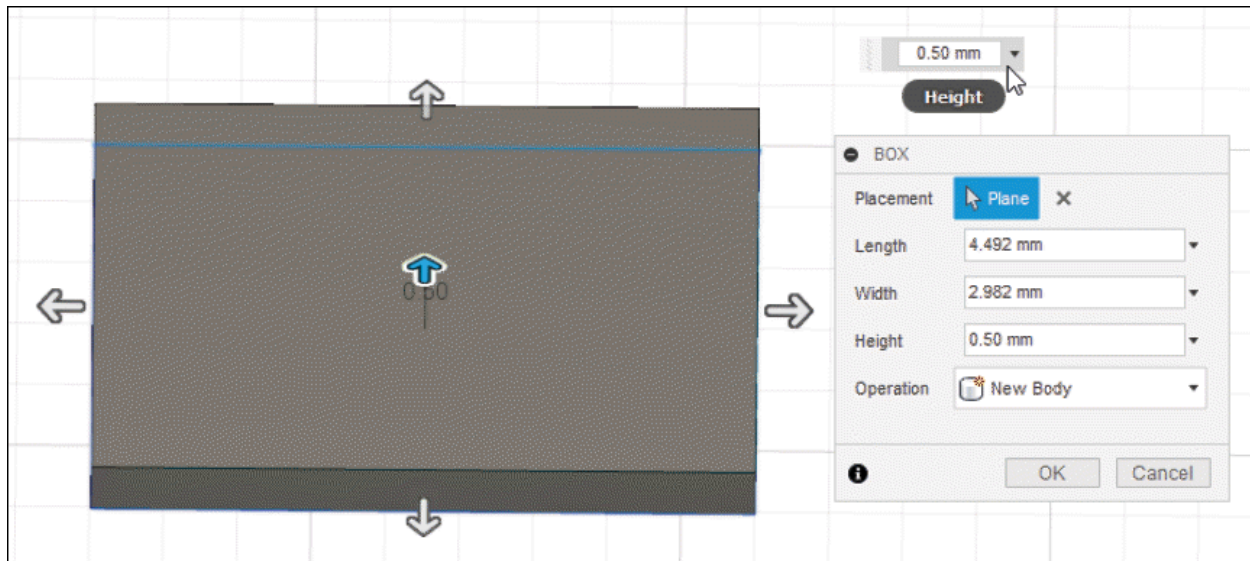


Figure-61. Preview of box with BOX dialog box

- Specify desired value of height in the **Height** edit box of the **BOX** dialog box.
- The options of **Operation** section were discussed earlier.
- Click **OK** button from the **BOX** dialog box to create the box; refer to [Figure-62](#).

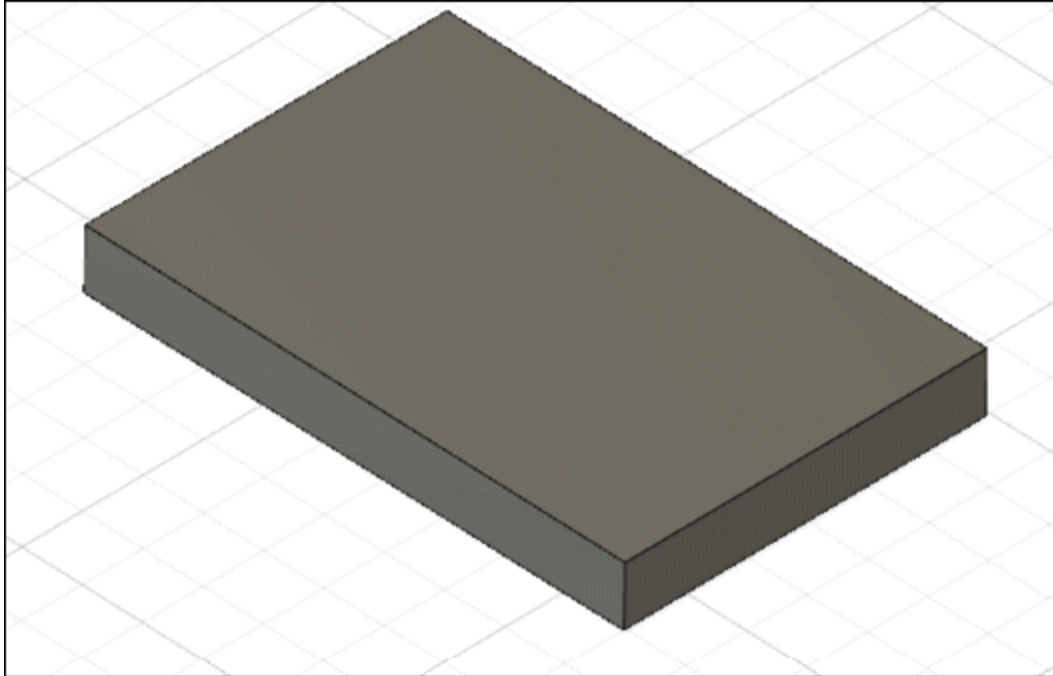


Figure-62. Box created

CYLINDER

This tool is used to create a cylindrical body by adding depth to a circular region. The procedure to use this tool is discussed next.

- Click on **Cylinder** tool in **CREATE** drop-down from **Toolbar**; refer to [Figure-63](#). You are asked to select plane for creating base sketch. Select desired plane/face.

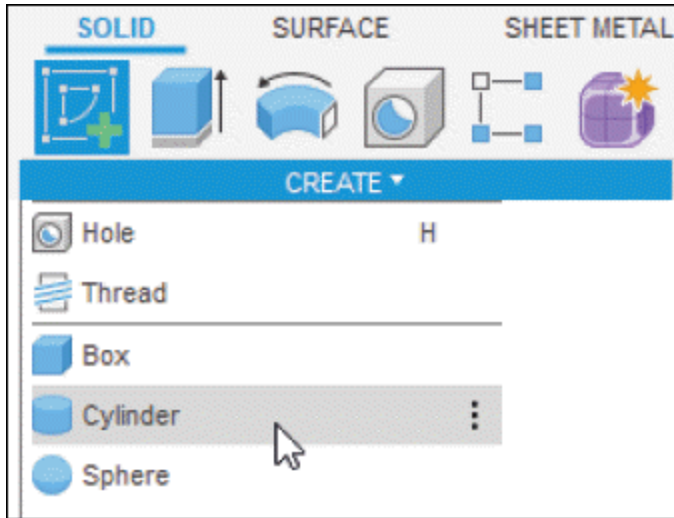


Figure-63. Cylinder tool

- Click on the screen to specify the center point of base circle.
- Type desired value of diameter in the floating window; refer to [Figure-64](#) and press **ENTER** to create a sketch of circle. The **CYLINDER** dialog box will be displayed along with preview of cylinder; refer to [Figure-65](#).

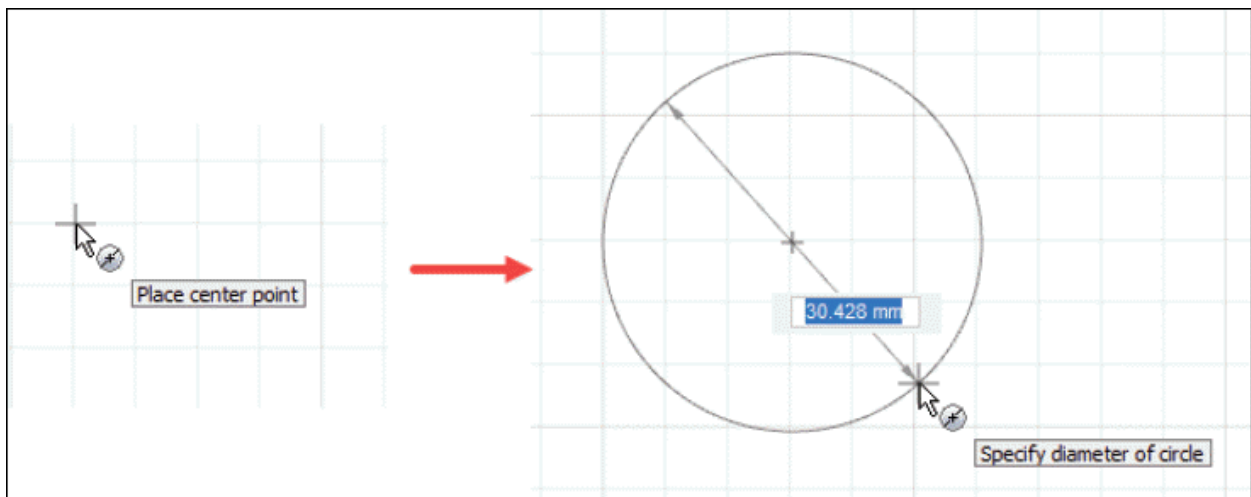


Figure-64. Specifying diameter of circle to create cylinder

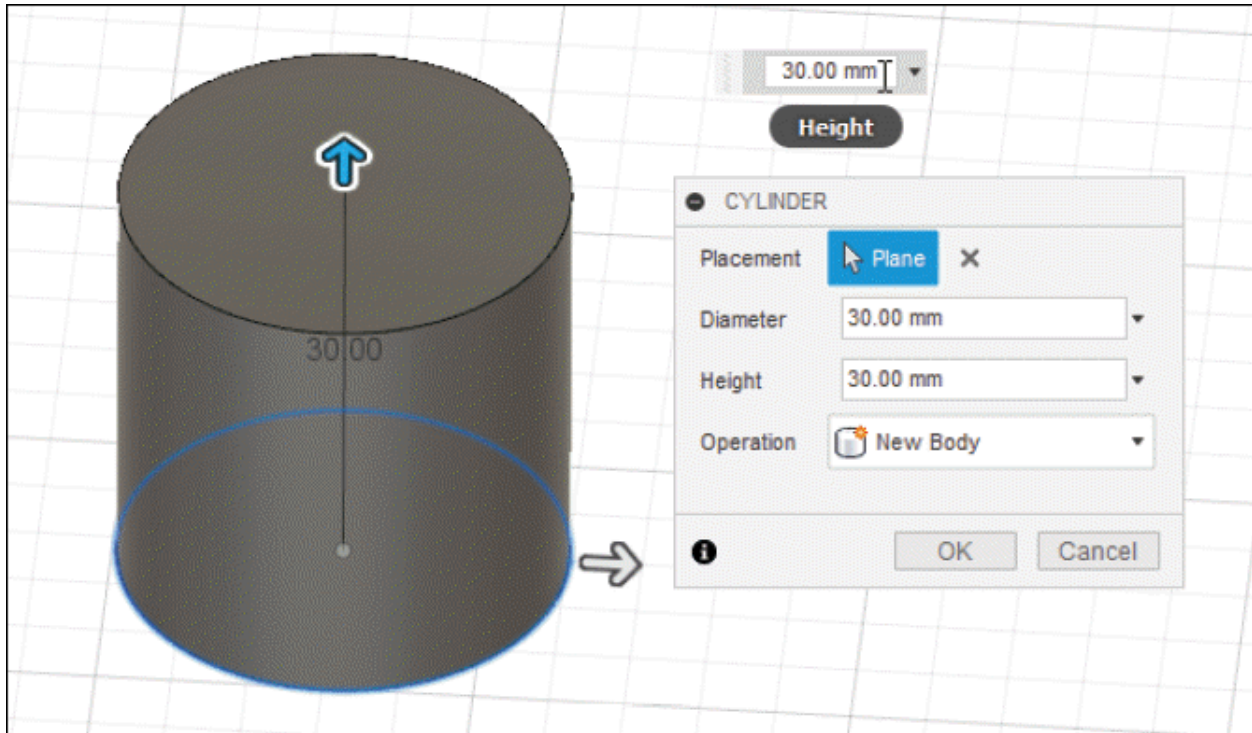


Figure-65. Preview of cylinder with CYLINDER dialog box

- If you want to edit the diameter of cylinder then click in the **Diameter** edit box of **CYLINDER** dialog box and specify desired value.
- Click in the **Height** edit box and specify the height of cylinder.
- Click **OK** button from the **CYLINDER** dialog box to create the cylinder; refer to [Figure-66](#).

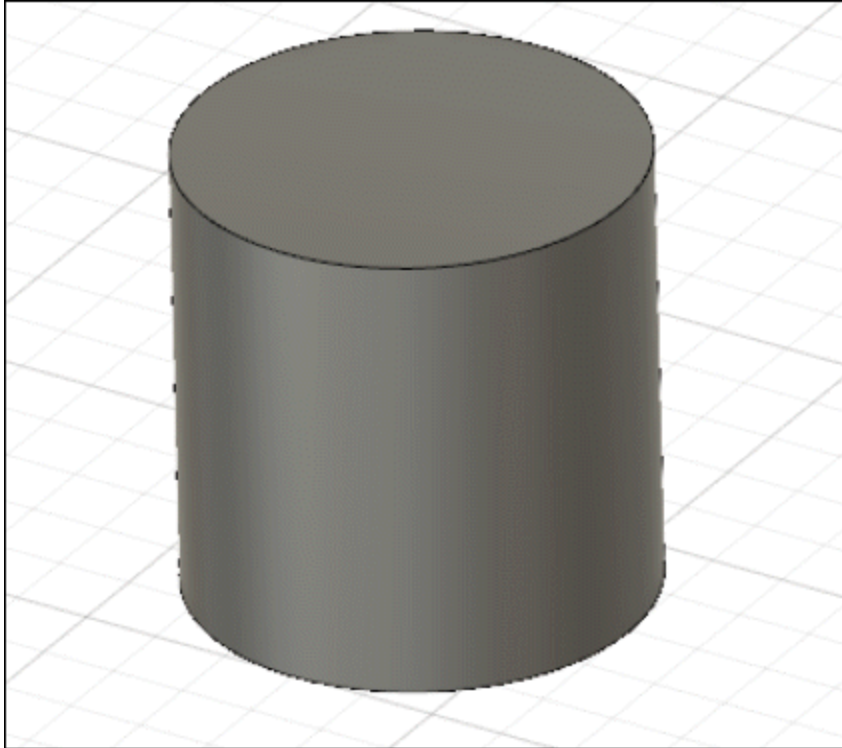


Figure-66. Cylinder created

SPHERE

The **Sphere** tool is used to create a T-spline, solid, or surface sphere on a work plane or face. The procedure to use this tool is discussed next.

- Click on the **Sphere** tool in **CREATE** drop-down from **Toolbar**; refer to [Figure-67](#). You are asked to select the plane for creating the sketch for circle. Select desired plane/face.

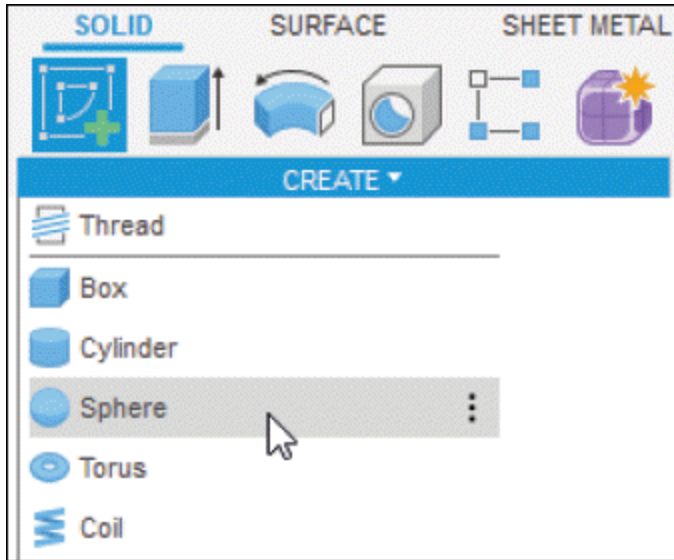


Figure-67. Sphere tool

- Click on the screen at desired location to specify the center point of sphere. The **SPHERE** dialog box will be displayed along with the preview of sphere; refer to [Figure-68](#).

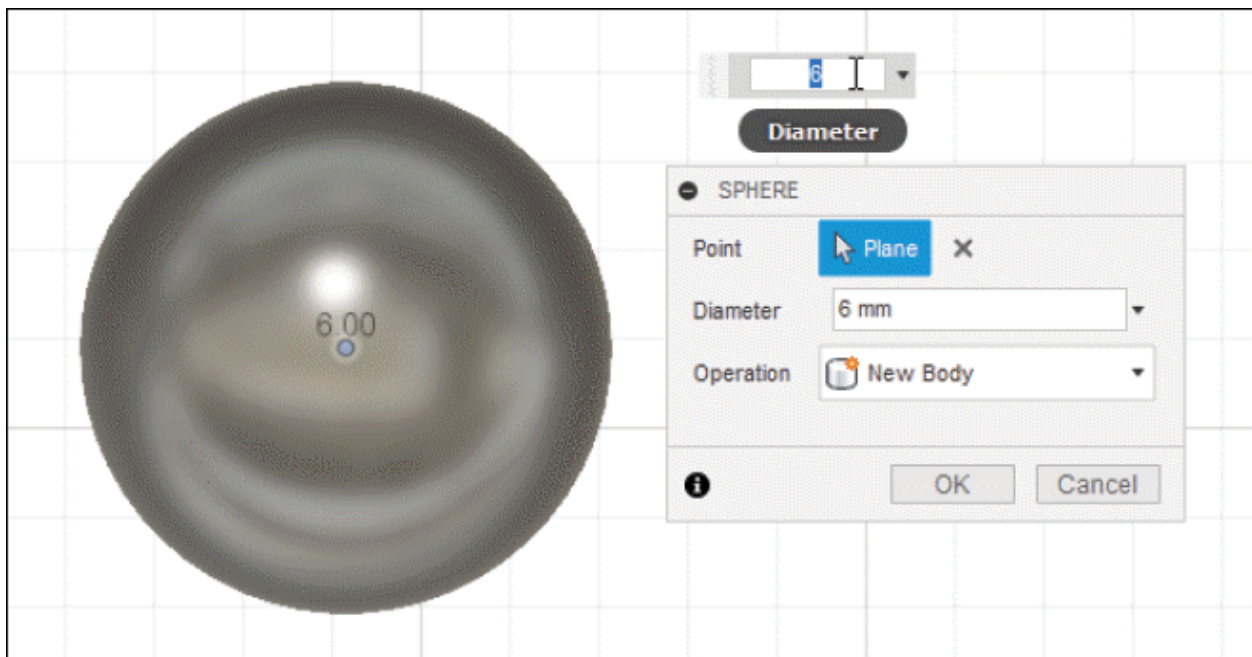


Figure-68. Preview of sphere with SPHERE dialog box

- Click in the **Diameter** edit box and specify desired value of

diameter for sphere.

- The options of **Operation** drop-down are same as discussed earlier.
- Click **OK** button of **SPHERE** dialog box to finish the process; refer to [Figure-69](#).

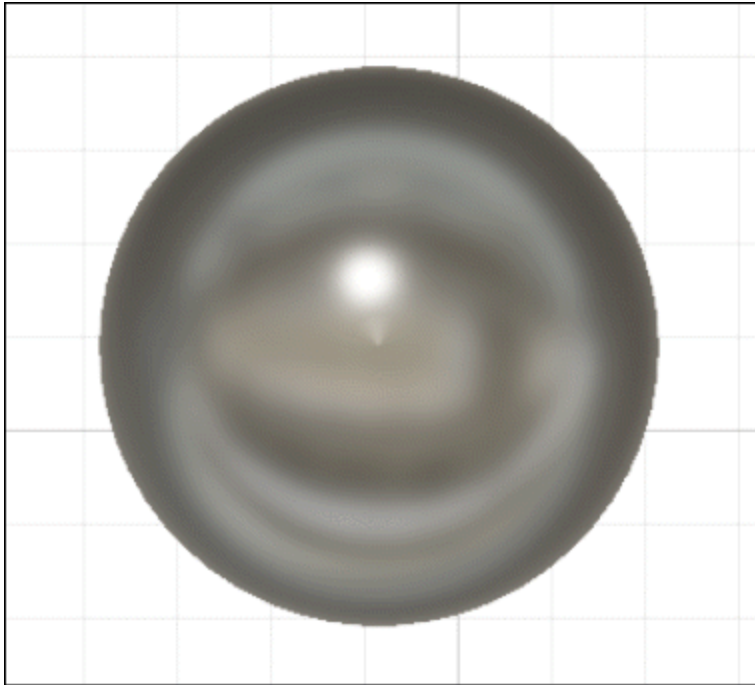


Figure-69. Sphere created

TORUS

The **Torus** tool is used to create T-spline, solid, or surface torus on a work plane or face. The procedure to use this tool is discussed next.

- Click on **Torus** tool in **CREATE** drop-down from **Toolbar**; refer to [Figure-70](#). You are asked to select the plane/face for creating the center circle of torus. Select desired face/plane.

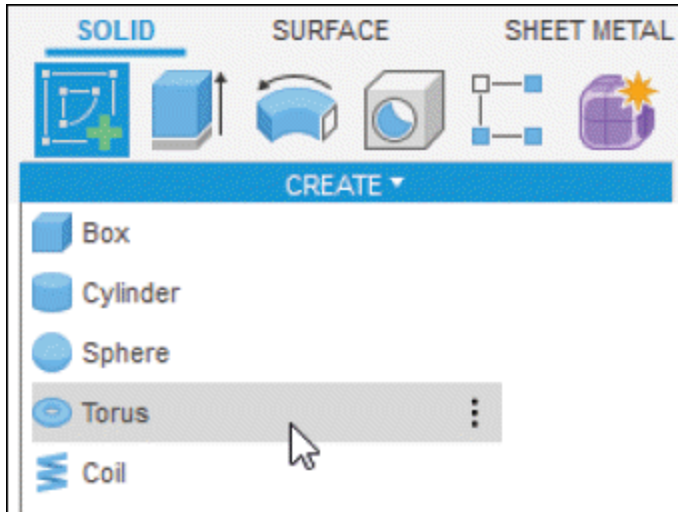


Figure-70. Torus tool

- Click on the screen to specify center point of sketch. Specify the diameter of torus in dynamic input box and press **ENTER**; refer to [Figure-71](#). The **TORUS** dialog box will be displayed along with the preview of torus; refer to [Figure-72](#).

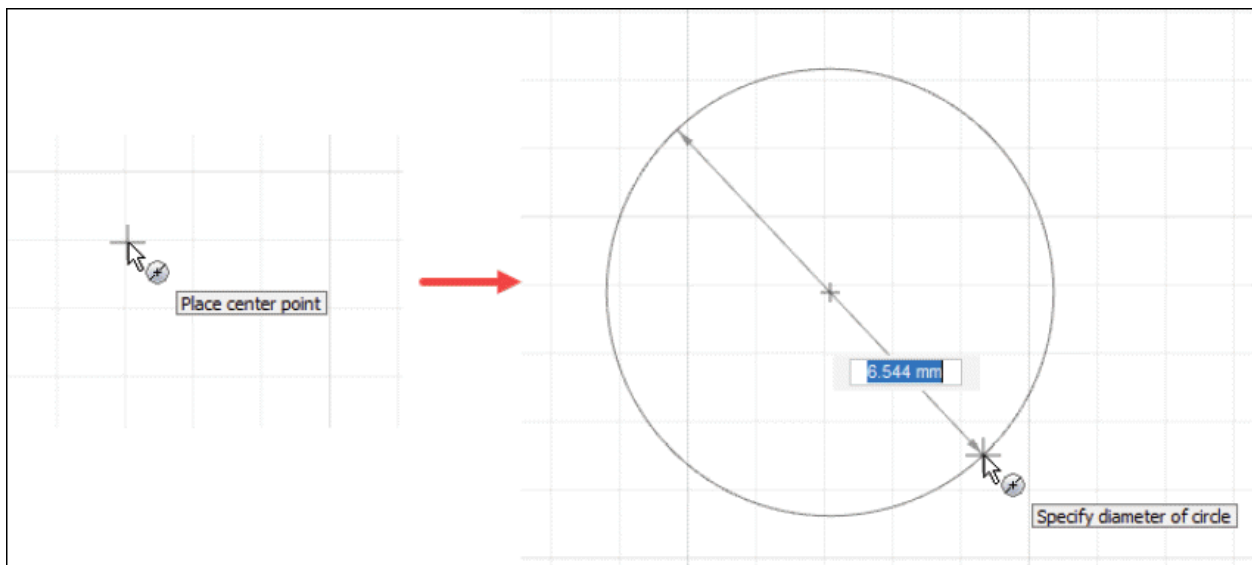


Figure-71. Specifying diameter of circle to create torus

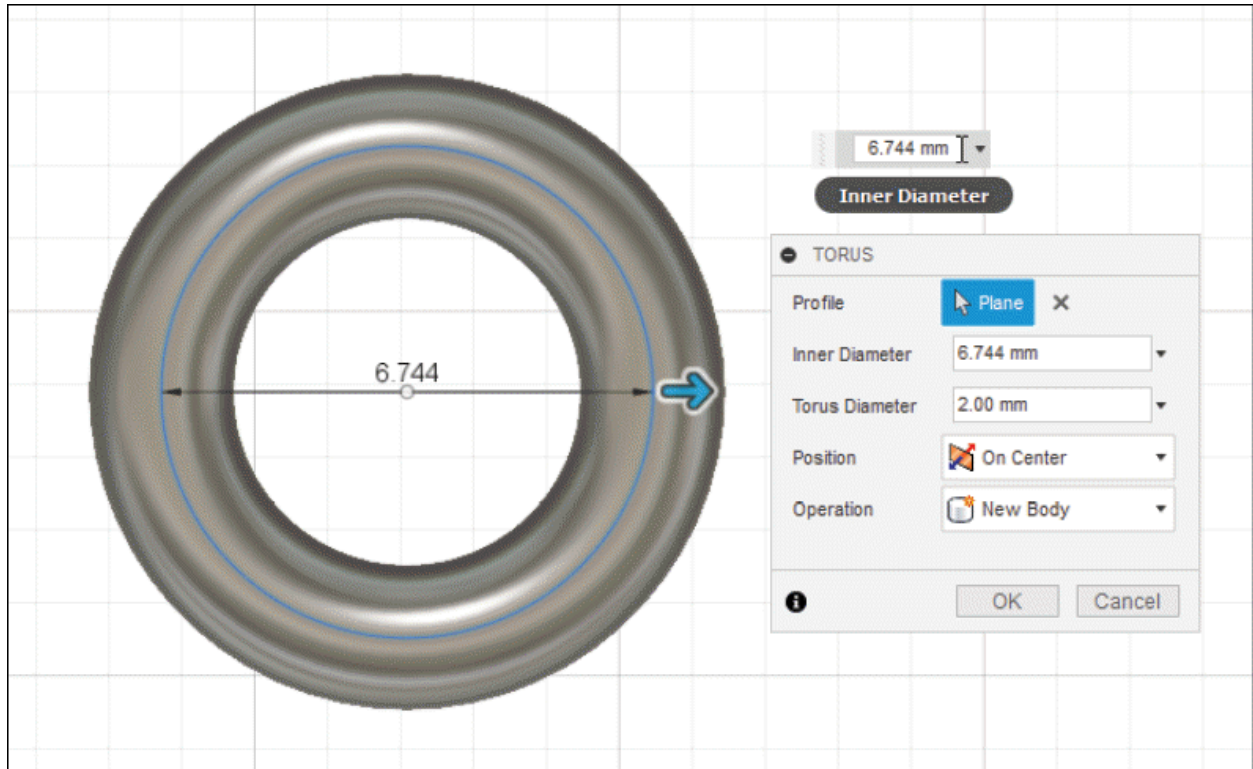


Figure-72. Preview of torus with TORUS dialog box

- If you want to edit the diameter of circle earlier created then click in the **Inner Diameter** edit box and specify desired value.
- Click in the **Torus Diameter** edit box and type the diameter of torus. You can also adjust the diameter of torus by moving the drag handle.
- Select the **On Center** option of **Position** drop-down from **TORUS** dialog box, if you want to use inner circle as center of the torus. Select the **Inside** option of **Position** drop-down if you want to position the torus inside of the inner circle. Select the **Outside** option of **Position** drop-down if you want to position the torus section outside of

the inner circle.

- The options of **Operation** drop-down are same as discussed earlier.
- Click **OK** button of **TORUS** dialog box to finish the process; refer to [Figure-73](#).

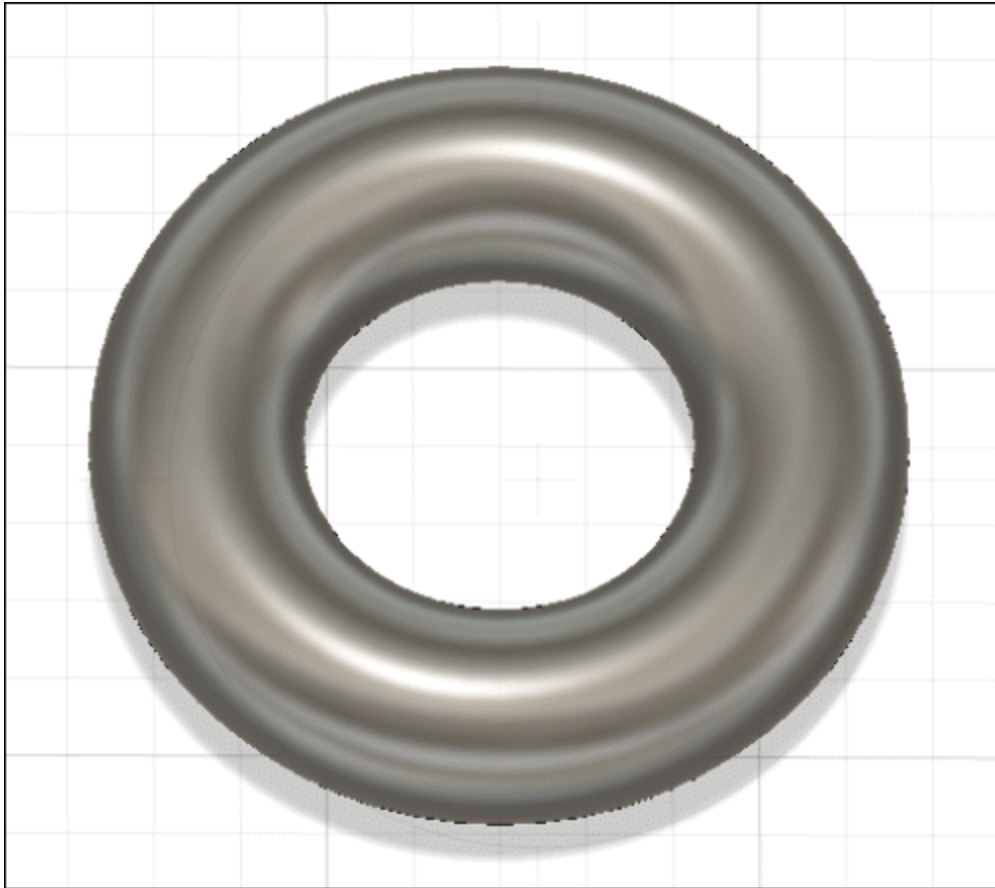


Figure-73. Torus created

COIL

The **Coil** tool is used for creating helix-based features like springs. The procedure to use this tool is discussed next.

- Click on the **Coil** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-74](#). You will be asked to select a

face/plane to create base sketch.

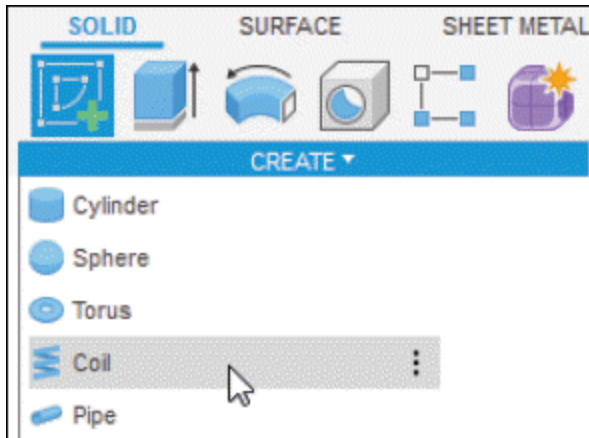


Figure-74. Coil tool

- Click on desired face/plane. You will be asked to specify center of the base circle.
- Click on the screen to specify the center point of sketch. Type desired value of diameter of coil in dynamic input box and press **ENTER**; refer to [Figure-75](#).

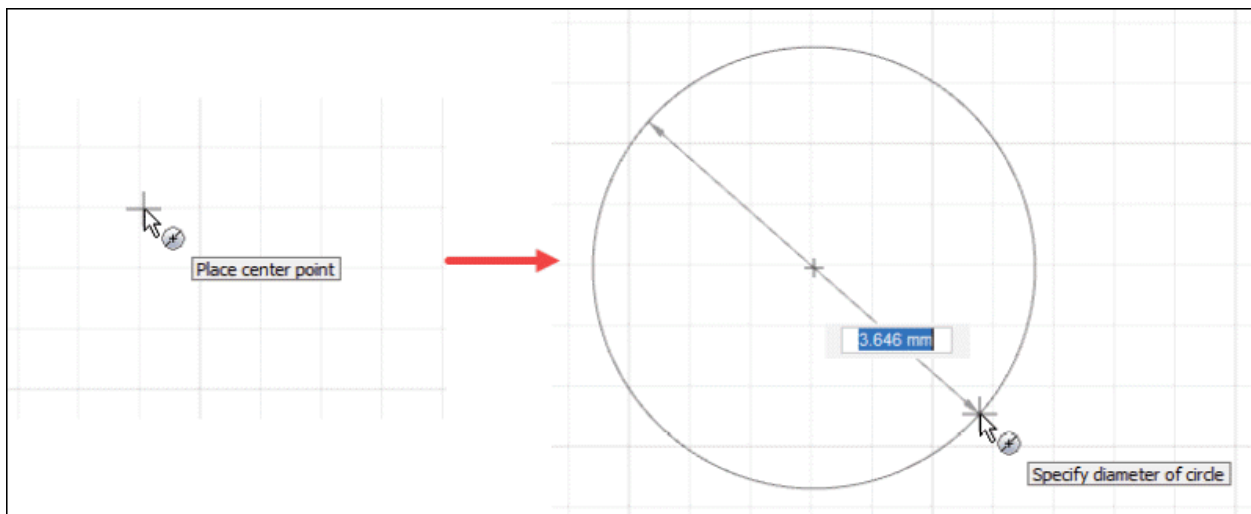


Figure-75. Specifying diameter of circle to create coil

- The **COIL** dialog box will be displayed along with the preview of coil; refer to [Figure-76](#).

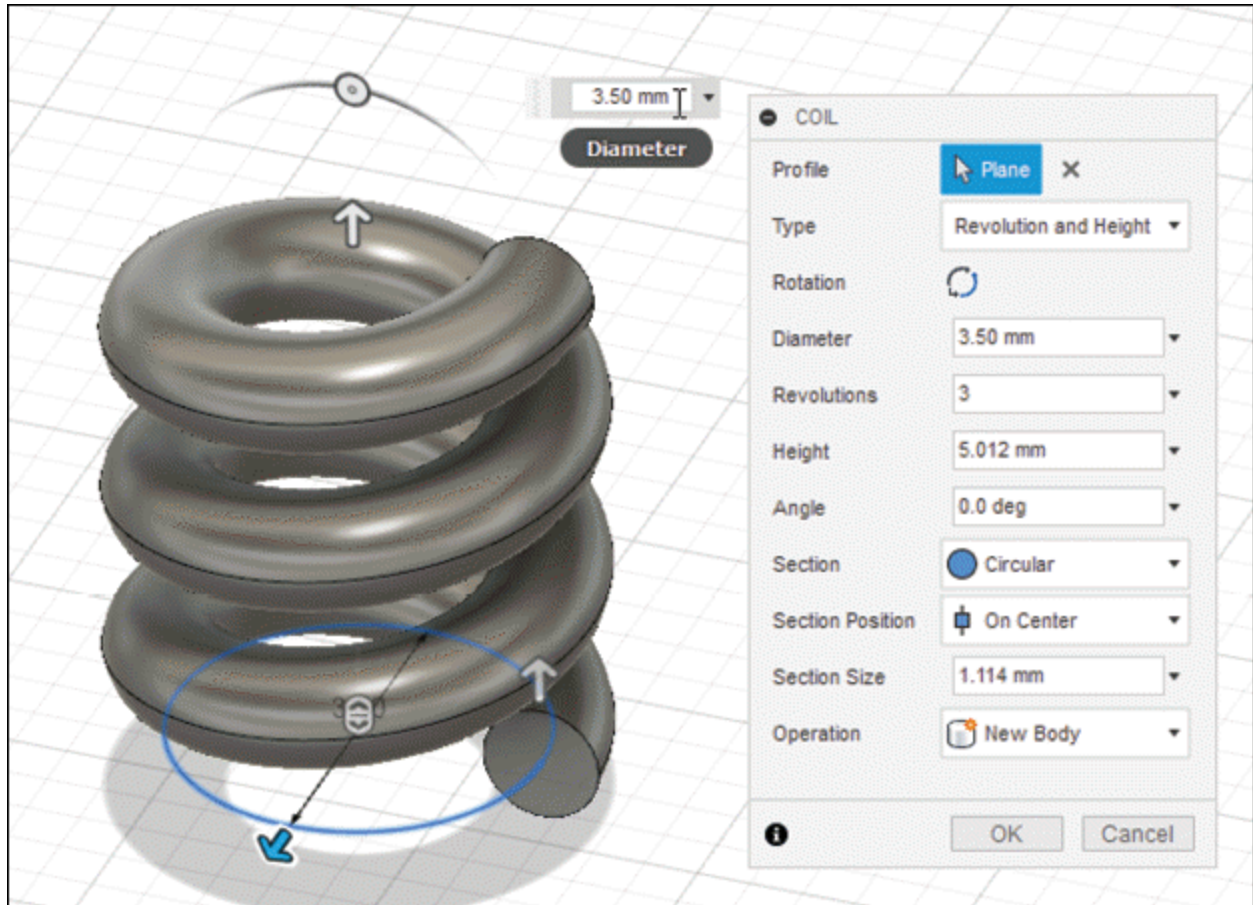


Figure-76. Preview of coil with COIL dialog box

- Select the **Revolution and Height** option from **Type** drop-down if you want to create the coil by specifying number of revolutions in the coil and height of coil. Select the **Revolution and Pitch** option from **Type** drop-down if you want to create the coil by specifying number of revolutions in the coil and distance between two successive turns of the coil. Select the **Height and Pitch** option from **Type** drop-down to specify total height of the coil and distance between two successive turns of the coil. Select the **Spiral** option from **Type** drop-down to create a spiral coil. Note that the options in the dialog box will be modified based on

your selection in **Type** drop-down. In our case, we are selecting the **Revolution and Height** option; refer to [Figure-77](#).

- Click on the **Rotation** button to flip the rotation side of coil to clockwise or counterclockwise direction; refer to [Figure-78](#).

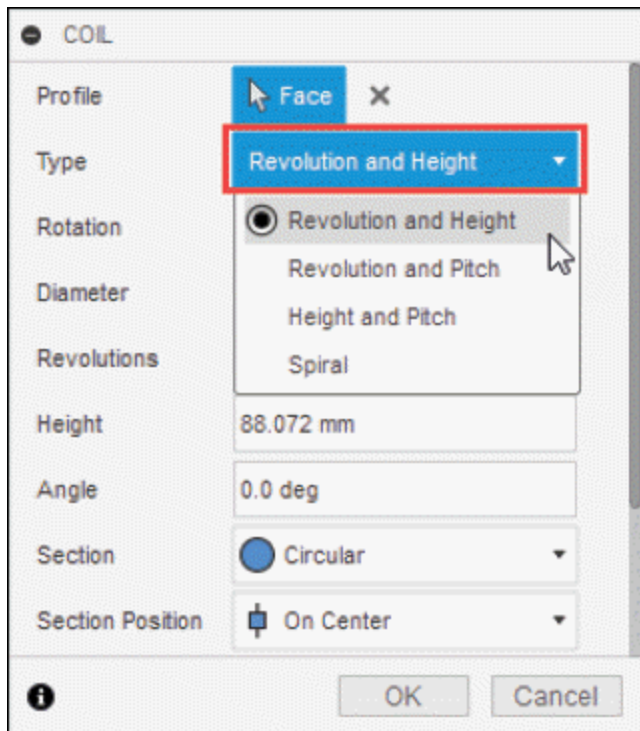


Figure-77. Coil Type drop down

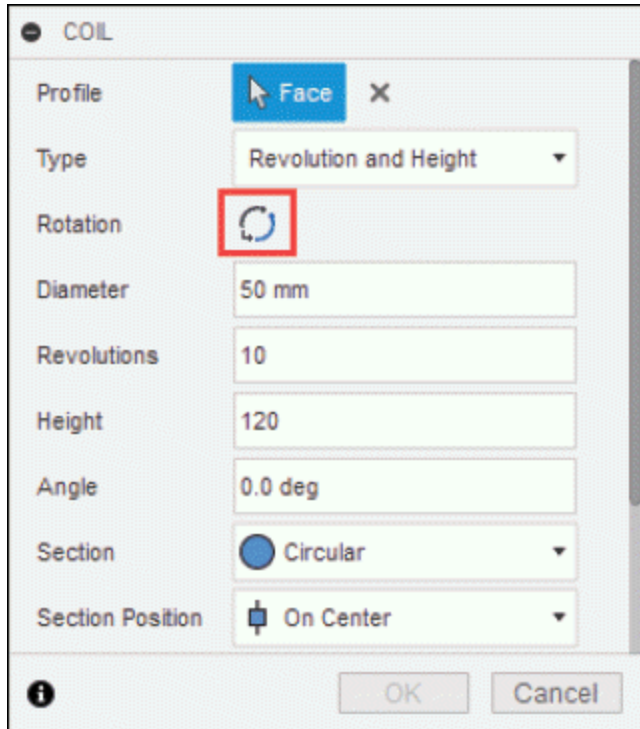


Figure-78. Rotation button

- Click in the **Diameter** edit box and enter desired value to change the diameter of coil.
- Click in the **Revolutions** edit box and enter desired value for number of revolutions of coil.
- Click in the **Height** edit box and specify the total height of coil. You can also set the height of coil by moving the drag handle.
- Click in the **Angle** edit box and specify desired angle value to create taper coil.
- Select **Circular** option from **Section** drop-down of **COIL** dialog box to set the circular profile of coil. Select the **Square** option from **Section** drop-down to set square

profile of coil. Select **Triangular(External)** option from **Section** drop-down to set the external triangular profile of coil. Select **Triangular(Internal)** option of **Section** drop-down to set internal triangular profile of coil; refer to [Figure-79](#).

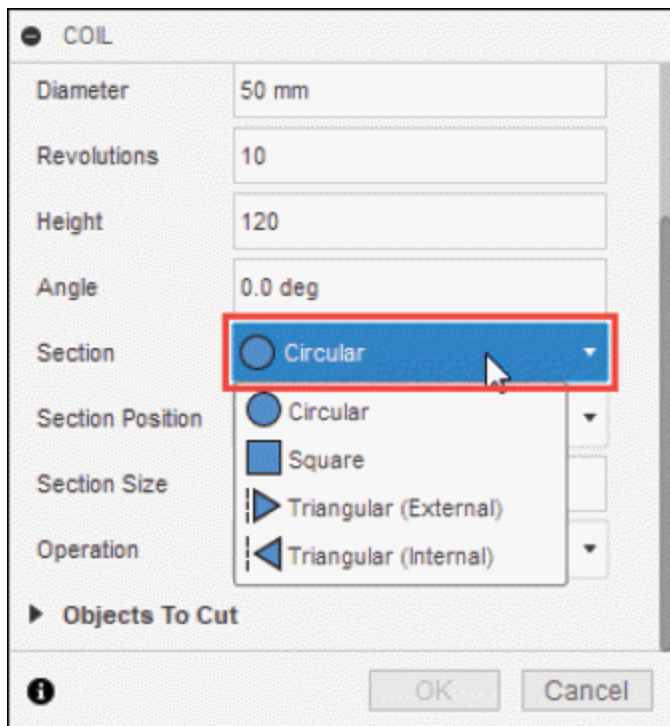


Figure-79. Coil Section drop down

- Select **On Center** option of **Section Position** drop-down from **COIL** dialog box to position the coil in such a way that the base circle is at center of the coil. Click on the **Inside** button of **Section Position** drop-down if you want to position the coil inside the base circle. Click on the **Outside** section of **Section Position** drop-down if you want to position the coil outside the base circle.
- Click in the **Section Size** edit box and specify the size of

coil section. If section is circle then this will be diameter and if section is triangle or square then it will be length of one edge of section.

- The options of **Operations** drop-down were discussed earlier.
- Click **OK** button of **COIL** dialog box to finish the process of creating a coil; refer to [Figure-80](#).

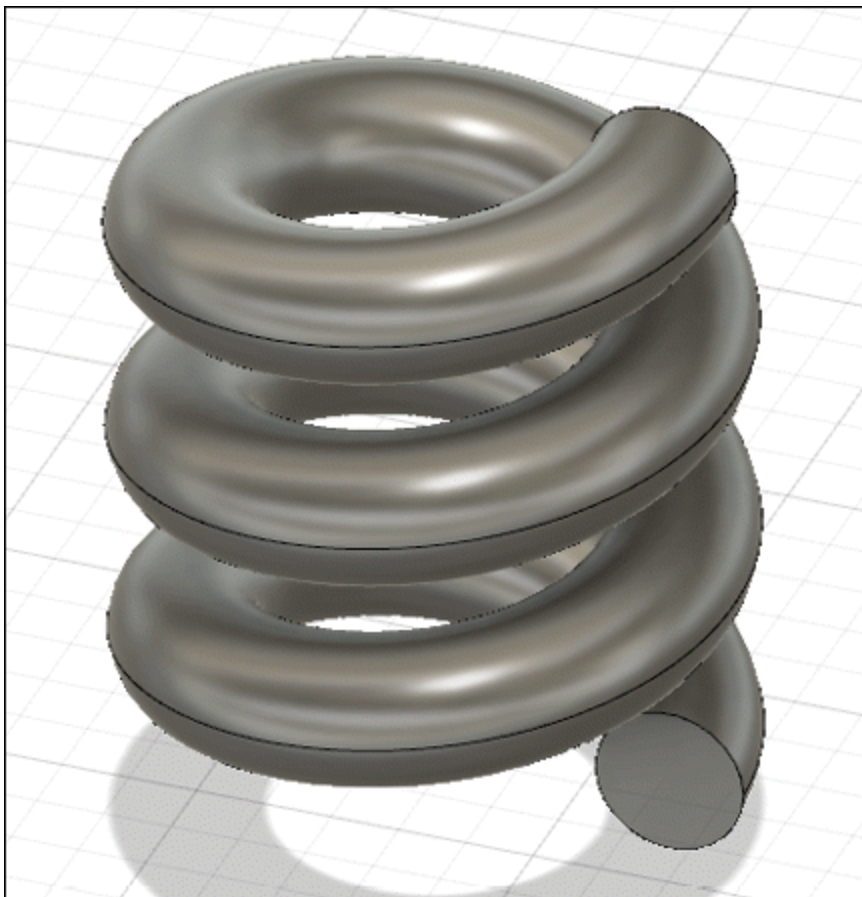


Figure-80. Coil created

PIPE

The **Pipe** tool is used to create pipes of different sections that follow a selected path. The procedure to use this tool is

discussed next.

- Click on **Pipe** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-81](#). The **PIPE** dialog box will be displayed and you will be asked to select a path for pipe; refer to [Figure-82](#).

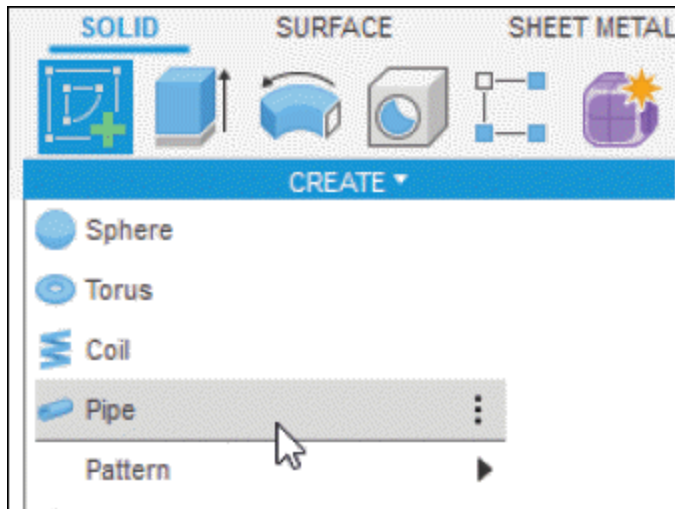


Figure-81. Pipe tool

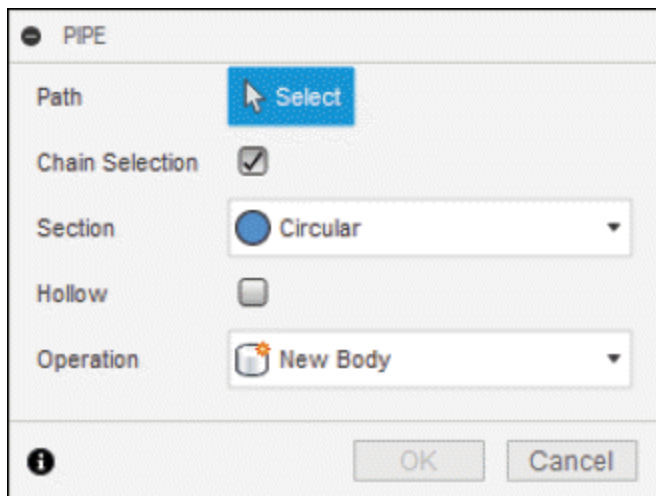


Figure-82. PIPE dialog box

- Select the sketch or edge of a model for path of the pipe; refer to [Figure-83](#).

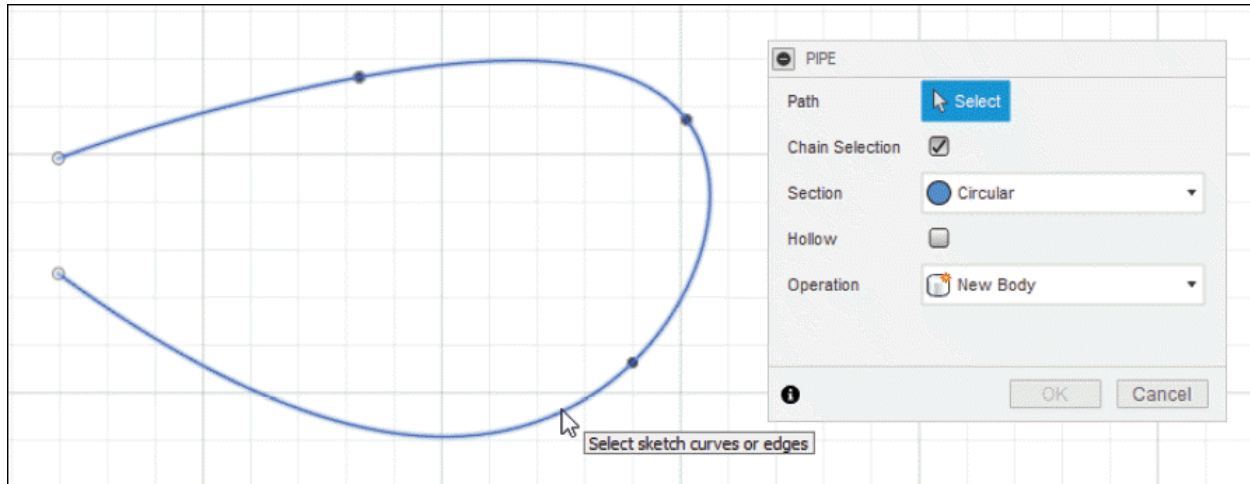


Figure-83. Selection of path to create pipe

- On selecting the path, the updated **PIPE** dialog box will be displayed. Click in the **Distance** edit box and enter desired value. The value of distance lies between 0 to 1. You can also set the distance (length) of pipe on the selected path by moving the drag handle.
- The options of **Section** drop-down were discussed earlier in **Coil** tool.
- Click in the **Section Size** edit box and specify the diameter of pipe.
- Select the **Hollow** check box to create a hollow pipe. The updated **PIPE** dialog box will be displayed along with the preview of pipe; refer to [Figure-84](#).

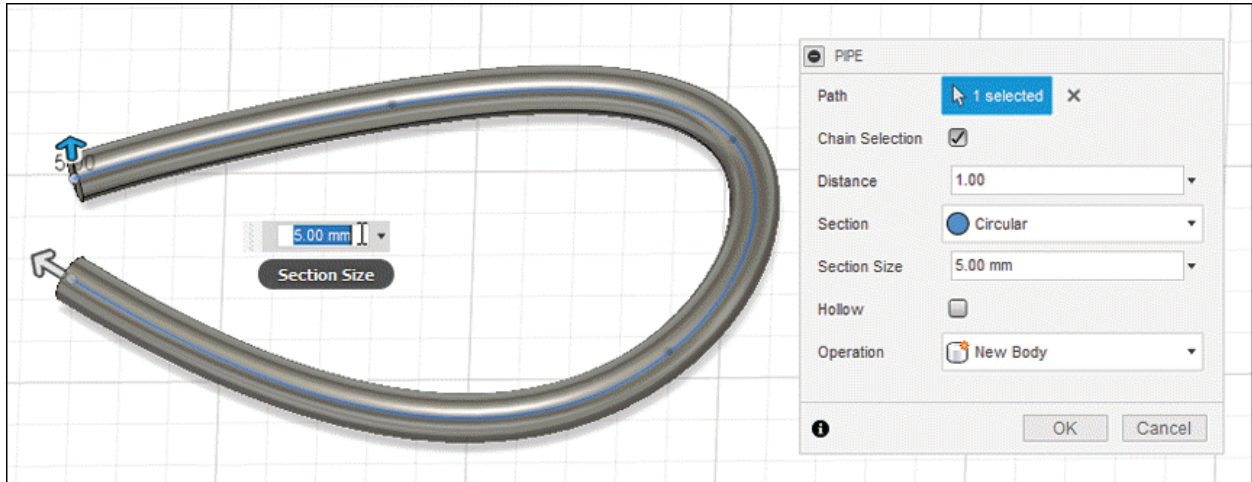


Figure-84. Preview of pipe with PIPE dialog box

- Click in the **Section Thickness** edit box and specify thickness of pipe.
- The options of **Operation** drop-down were discussed earlier in this book.
- Click **OK** button of **PIPE** dialog box to complete the process of creating the pipe; refer to [Figure-85](#).

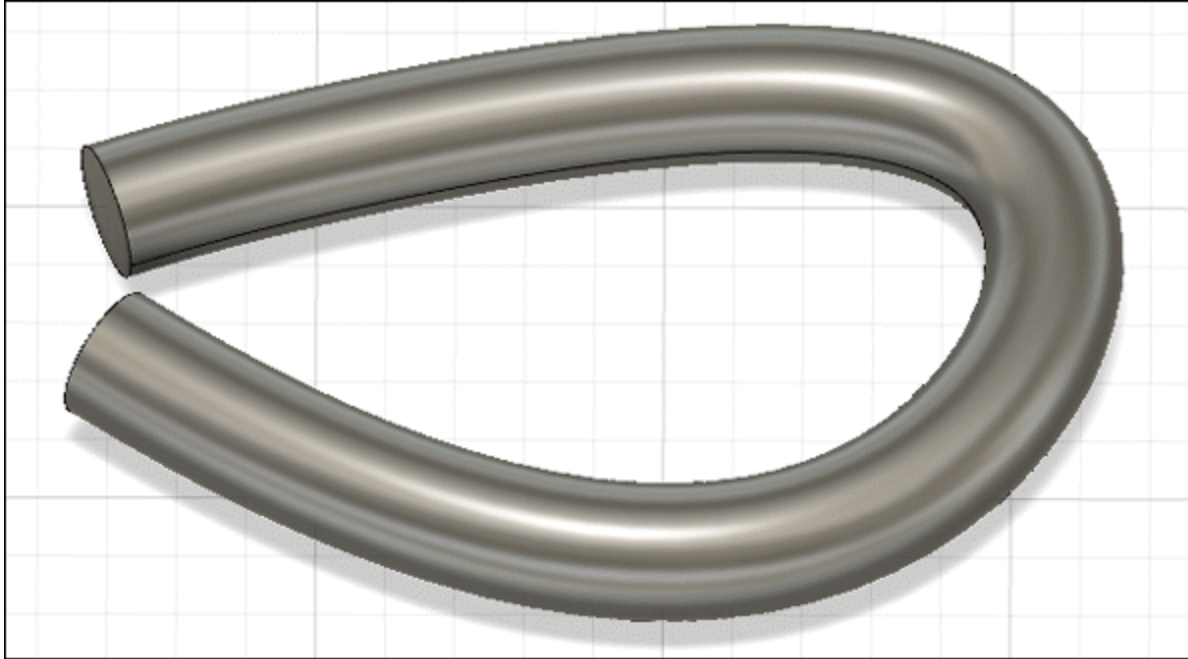


Figure-85. Pipe created

PATTERN

The **Pattern** tool is used to create copies of features or bodies at regular intervals. There are three tools in **Pattern** cascading menu; refer to [Figure-86](#). These tools are discussed next.

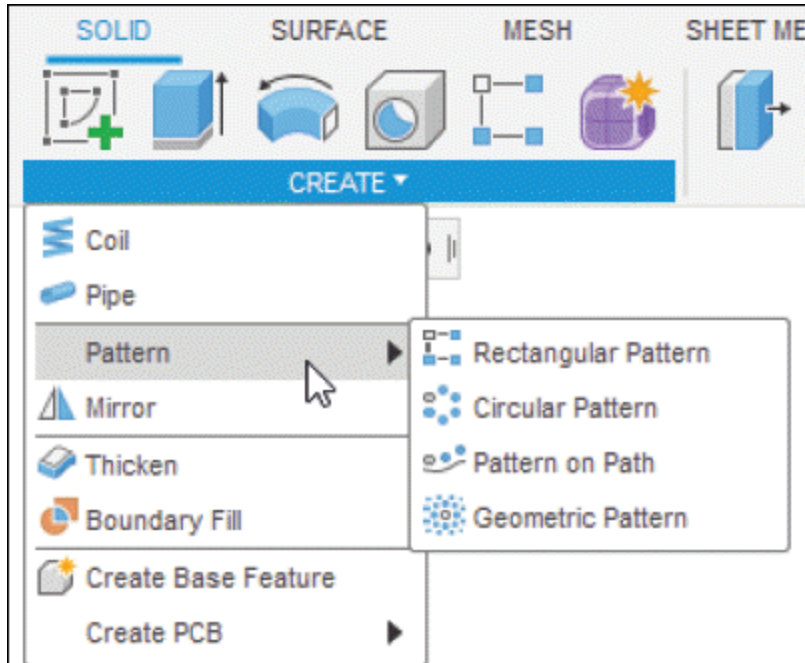


Figure-86. Pattern cascading menu

Rectangular Pattern

The **Rectangular Pattern** tool is used to create copies of objects in one or two directions. The procedure to use this tool are discussed next.

- Click on **Rectangular Pattern** tool of **Pattern** cascading menu from **CREATE** drop-down in the **Toolbar**; refer to [Figure-87](#). The **RECTANGULAR PATTERN** dialog box will be displayed; refer to [Figure-88](#).

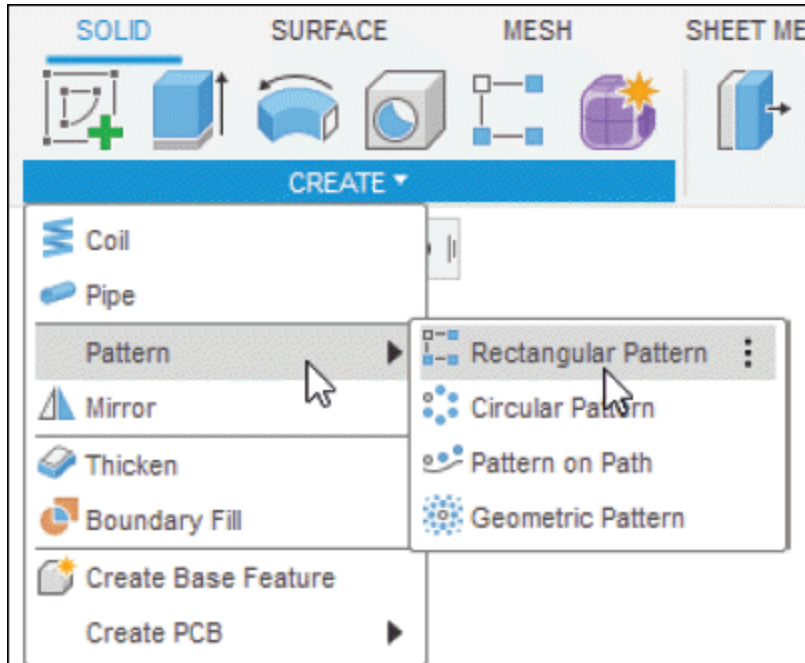


Figure-87. Rectangular Pattern tool in solid

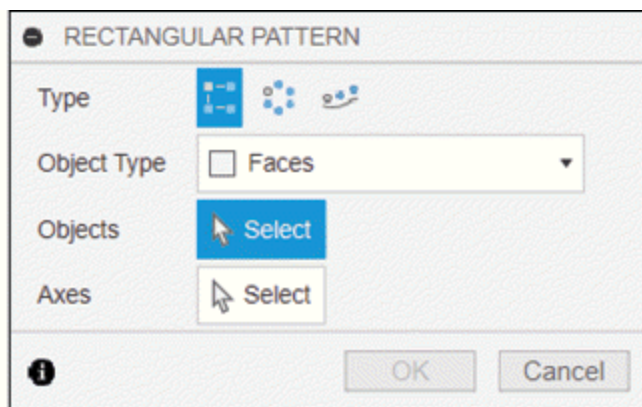


Figure-88. RECTANGULAR PATTERN dialog box in solid

- Select **Faces** option from **Type** drop-down if you want to select faces for pattern. Select **Bodies** option from **Type** drop-down if you want to select bodies for creating copies. Select **Features** option from **Type** drop-down if you want to select features for creating copies. Select **Components** option from **Type** drop-down if you want to select

component for creating copies; refer to [Figure-89](#).

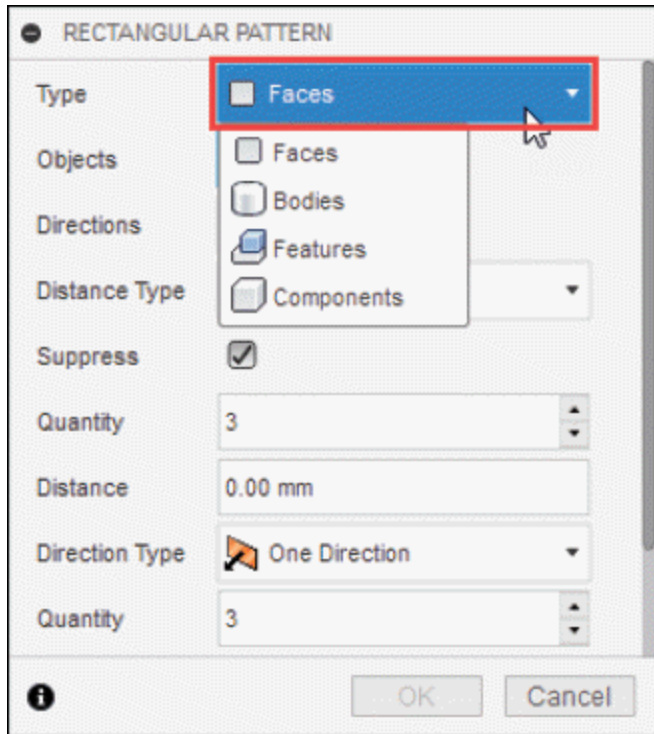


Figure-89. Pattern Type drop down

- We are using **Bodies** option from **Type** drop-down. Select the body for creating its copies; refer to [Figure-90](#).

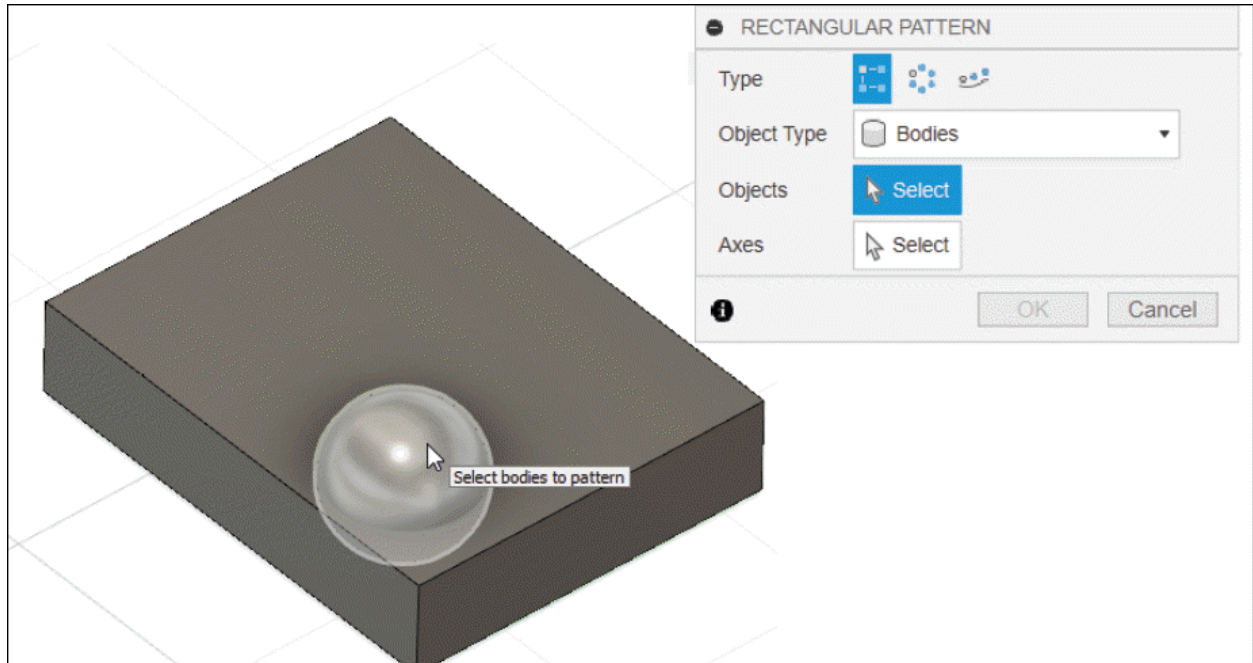


Figure-90. Selection of body for rectangular pattern

- Click on **Select** button of **Axes** option and select the default axes or edges of the model to define direction in which copies will be created; refer to [Figure-91](#).

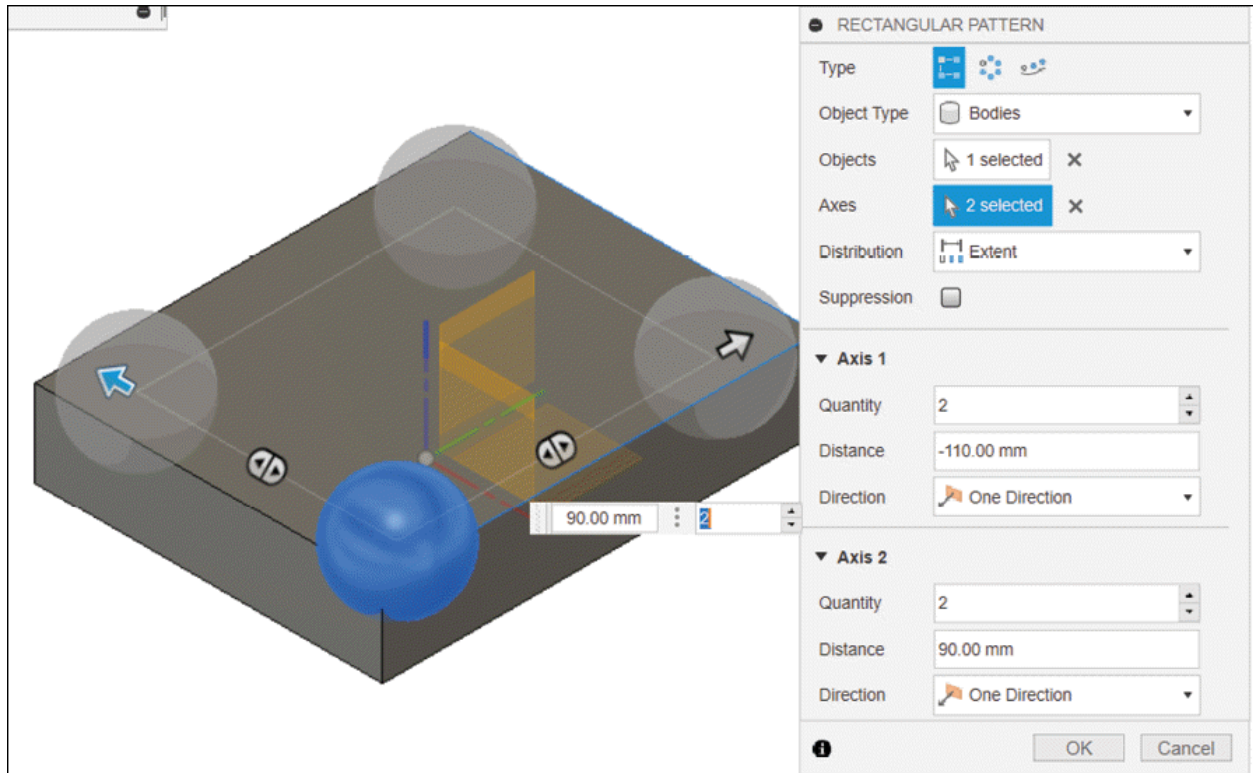


Figure-91. Selection of direction

- Select the **Extent** option from **Distribution** drop-down if you want to specify total length within which copies will be created. Select the **Spacing** option from **Distribution** drop-down to set the distance between two consecutive instances of object. In our case, we have selected **Extent** option.
- Select the **Suppress** check box if you want to suppress any instance of pattern.
- Click in the **Quantity** edit box and specify the number of instances to be created in the pattern along selected direction.
- Click in **Distance** edit box and specify the distance value

within which pattern instances will be created. You can also set the distance by moving the drag handle.

- Select the **One Direction** option from **Direction Type** drop-down if you want to create the copies in one side of selected direction. Click on **Symmetric** option from **Direction Type** drop-down to create the copies in both sides of selected direction.
- Similarly, specify the quantity, distance, and direction type for other direction.
- After specifying the parameter, click **OK** button from **RECTANGULAR PATTERN** dialog box to finish the process; refer to [Figure-92](#).

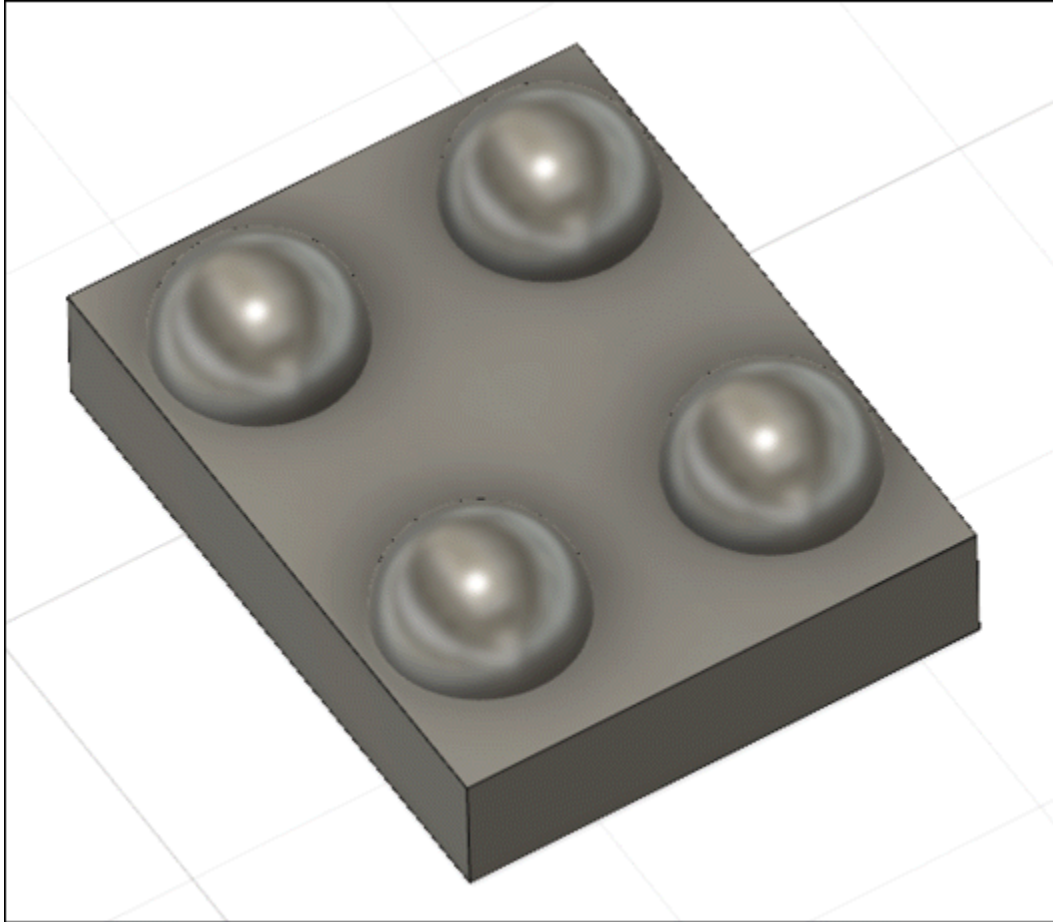


Figure-92. Solid model after applying rectangular pattern tool

Circular Pattern

The **Circular Pattern** tool is used to create copies of selected objects around selected axis/edge. The procedure to use this tool is discussed next.

- Click on **Circular Pattern** tool from **Pattern** cascading menu; refer to [Figure-93](#). The **CIRCULAR PATTERN** dialog box will be displayed; refer to [Figure-94](#).

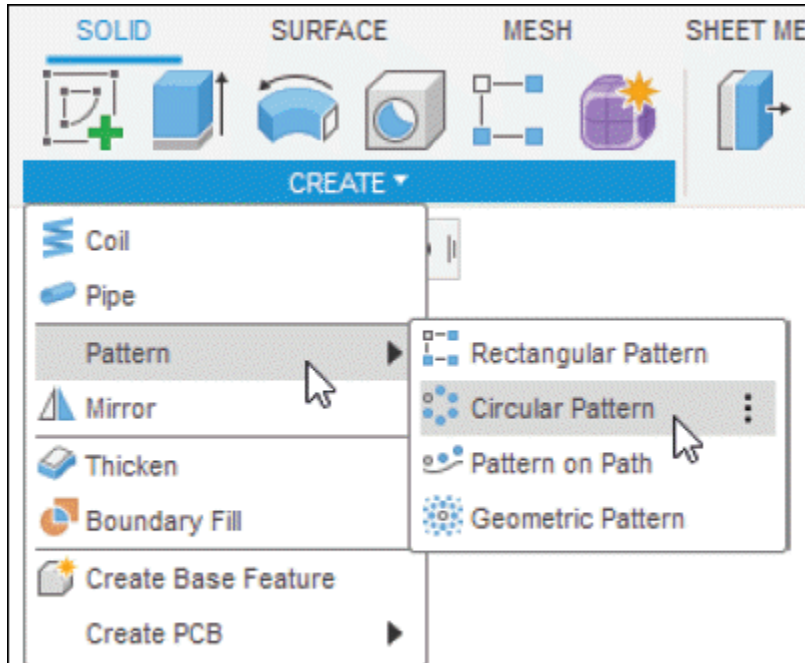


Figure-93. Circular Pattern tool in solid

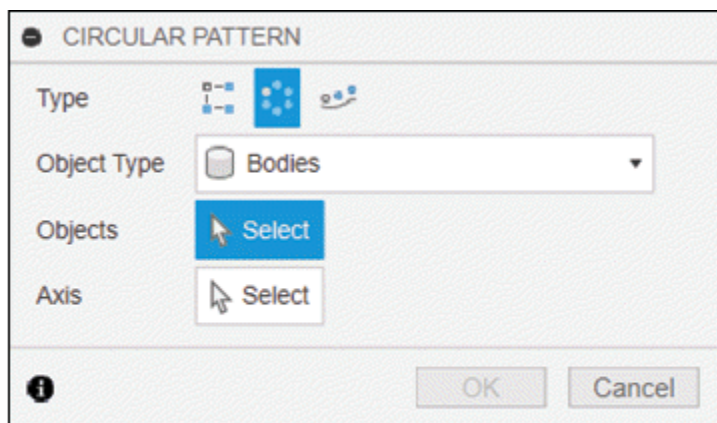


Figure-94. CIRCULAR PATTERN dialog box in solid

- Select desired option from **Type** drop-down. The options of **Type** drop-down are same as discussed in **Rectangular Pattern** tool.
- **Objects** selection is active by default. Select the feature to be patterned; refer to [Figure-95](#).

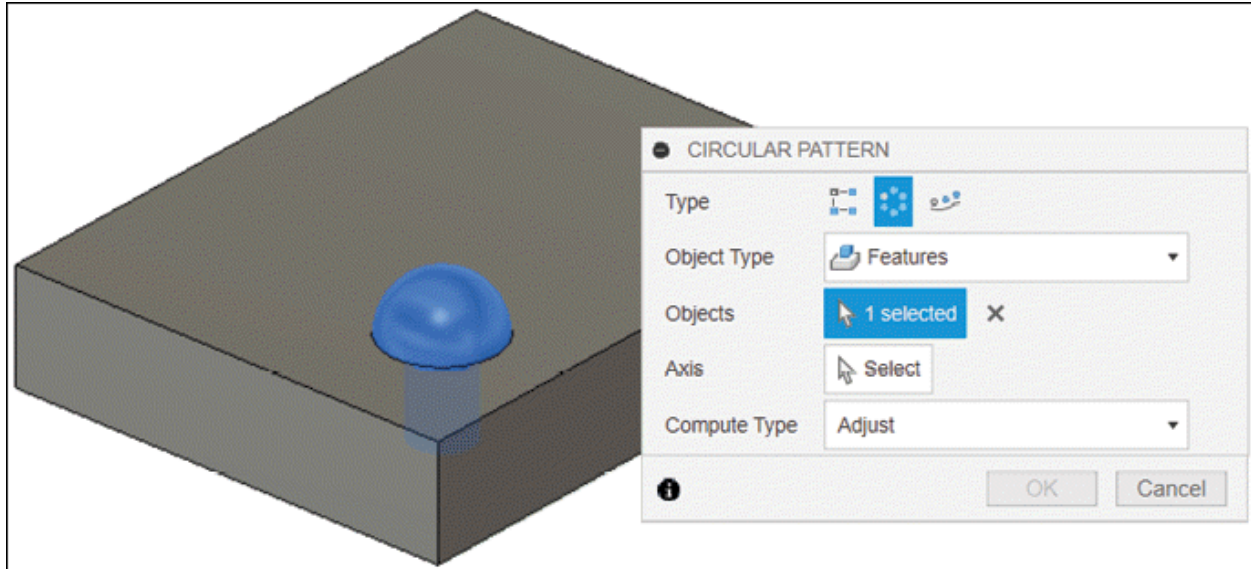


Figure-95. Selection of body for circular pattern

- Click on **Select** button of **Axis** section and select the edge/axis about which the circular pattern will be created; refer to [Figure-96](#).

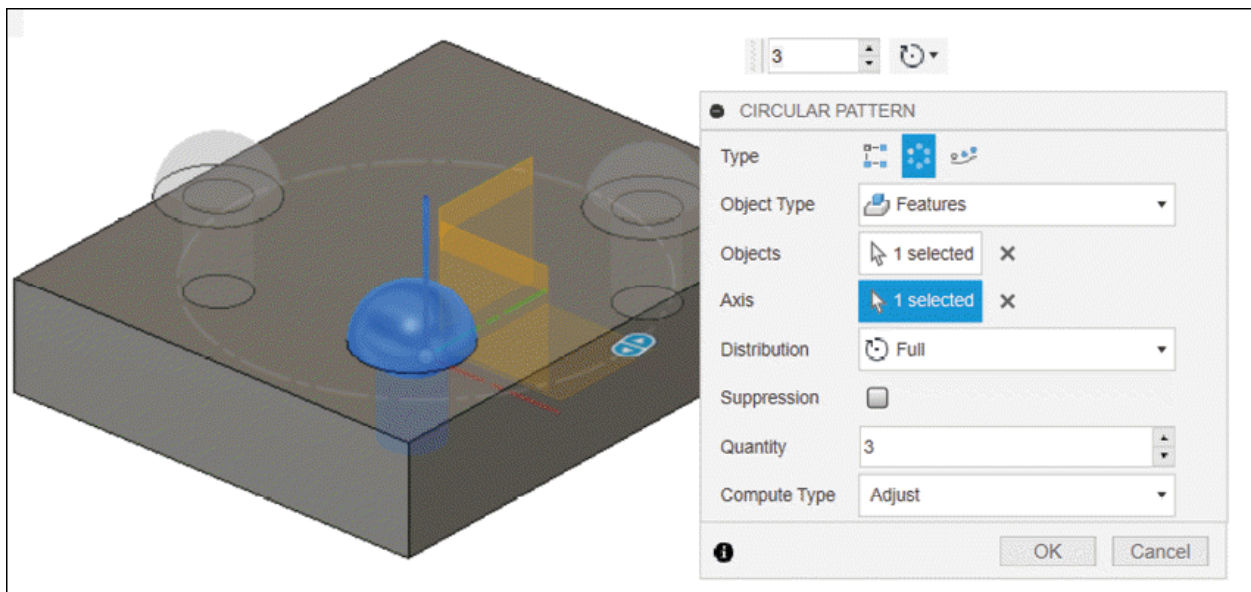


Figure-96. Selection of axis for circular pattern

- Select the **Full** option from **Distribution** drop-down if you want to create the copies of object around 360 degree

path. Select **Angle** option from **Distribution** drop-down if you want to create the copies within specified angle range. Click on **Symmetric** option from **Distribution** drop-down if you want to create copies symmetrically in specified angle range.

- Select the **Suppression** check box if you want to suppress the any instance of pattern.
- Click in the **Quantity** edit box and type the number of copies to be created in pattern.
- Select **Optimized** option from **Compute Option** drop-down to create identical copies by patterning feature faces. Select **Identical** option to create identical copies by replicating results of original features. Select **Adjust** option to create potentially different copies by patterning features and calculating extents or terminations of each instance individually.
- After specifying the parameters, click on the **OK** button from **CIRCULAR PATTERN** dialog box to finish the process; refer to [Figure-97](#).

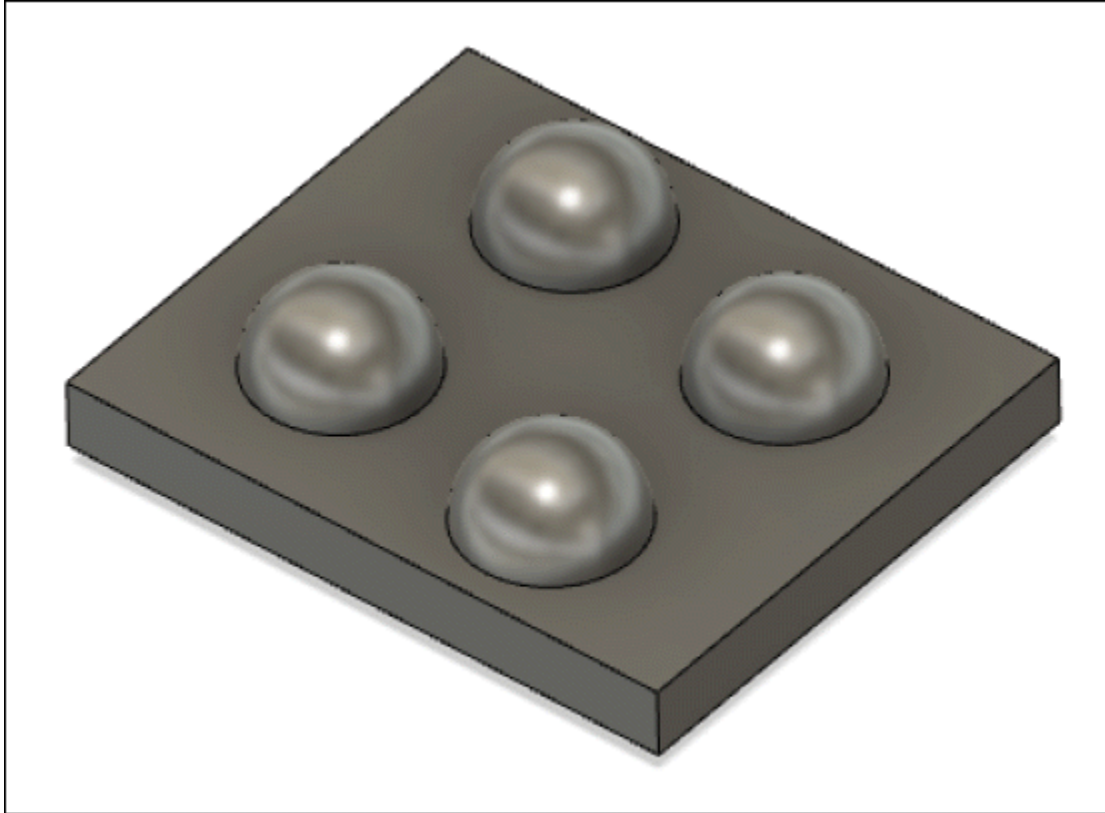


Figure-97. Solid model after applying circular pattern tool

Pattern on Path

The **Pattern on Path** tool is used to create copies of the selected object along selected path. The procedure to use this tool is discussed next.

- Click on the **Pattern on Path** tool of **Pattern** cascading menu from **CREATE** drop-down; refer to [Figure-98](#). The **PATTERN ON PATH** dialog box will be displayed; refer to [Figure-99](#).

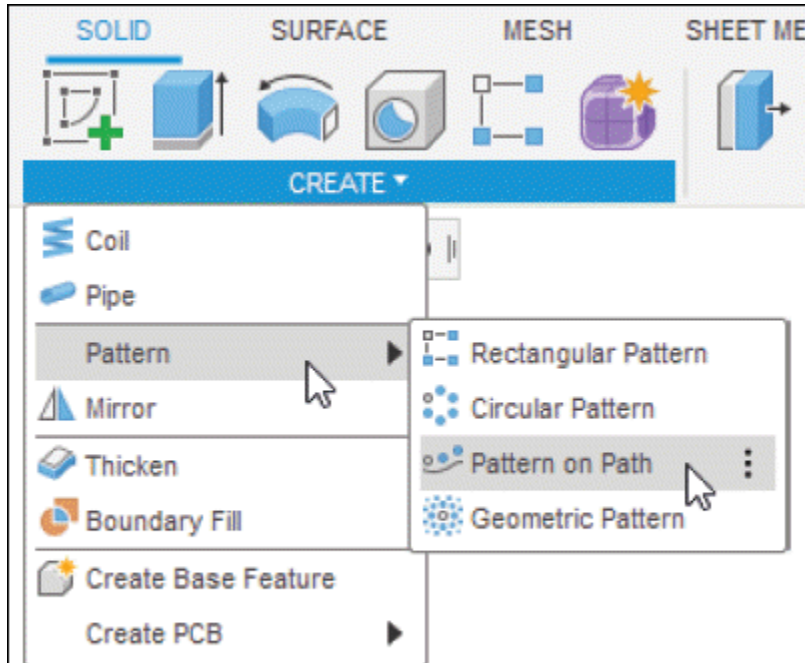


Figure-98. Pattern on Path tool

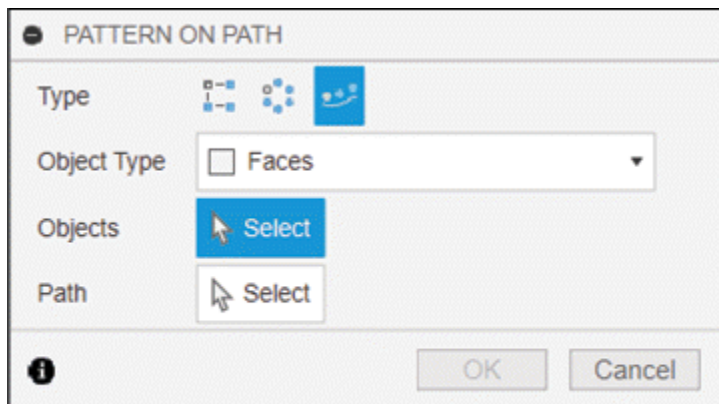


Figure-99. PATTERN ON PATH dialog box

- Click on **Faces** option from **Type** drop-down. Other options of **Type** drop-down are same as discussed earlier.
- **Objects** selection is selected by default. Select the faces for creating the copies; refer to [Figure-100](#).

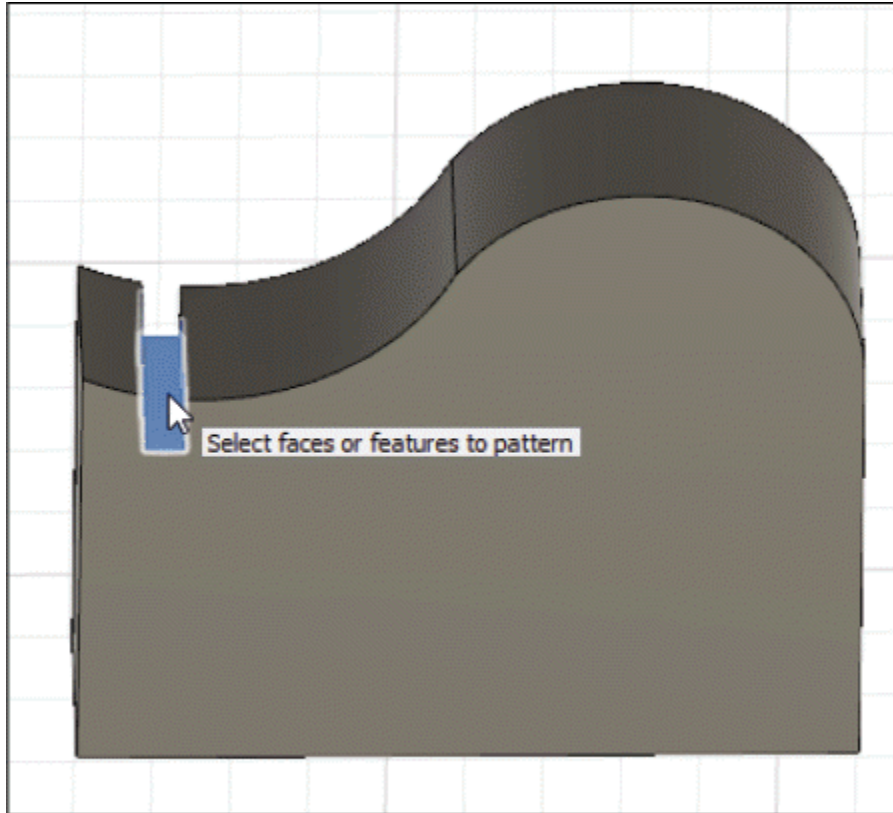


Figure-100. Selection of faces

- Click on **Select** button for **Path** option and select the path for creating copies.
- On selecting path, the updated **PATTERN ON PATH** dialog box will be displayed; refer to [Figure-101](#).

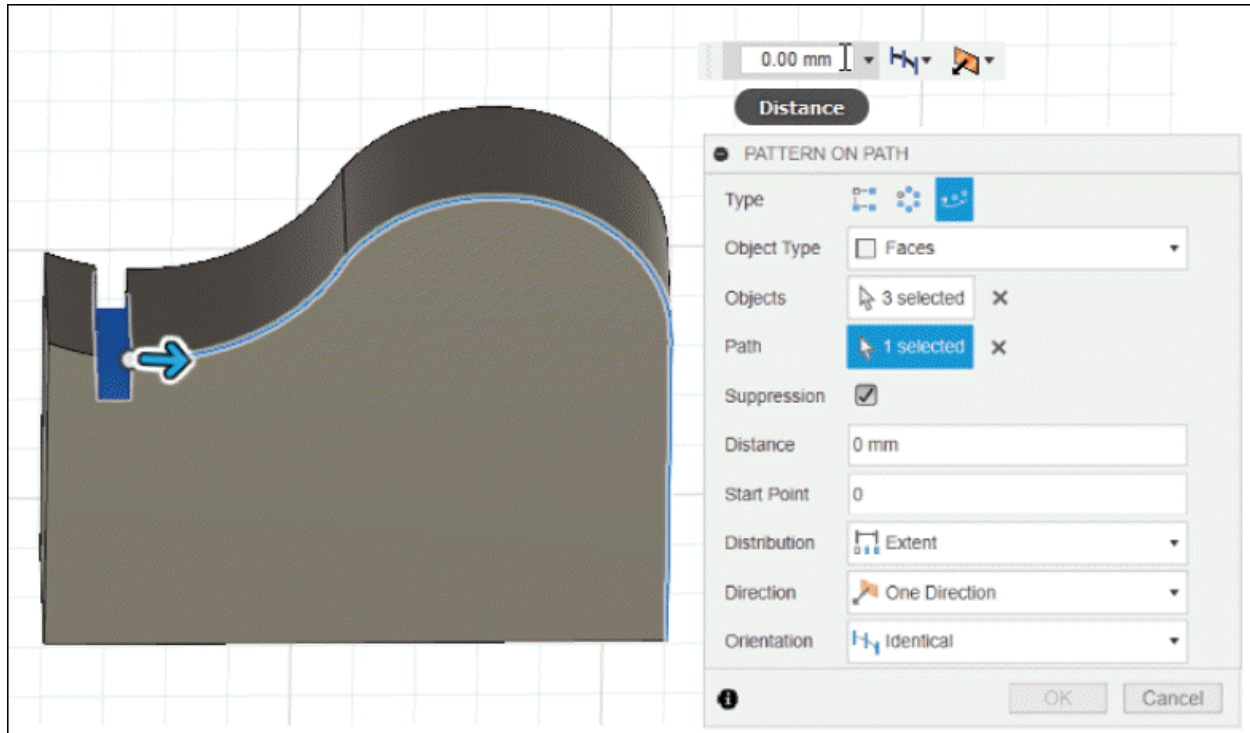


Figure-101. Updated PATTERN ON PATH dialog box on selection of path

- Select the **Suppression** check box if you want to suppress any instance of pattern.
- Click in the **Distance** edit box and specify the distance value along the path.
- Click in the **Quantity** edit box and specify the number of copies to be created.
- The **Start Point** option is used to specify the distance from the start point to place where the first pattern instance will be created. It is shown in the screen canvas as a **white point** and determines the starting point of the selected path. It can be set to any value between zero and one using the **Start Point** edit box, but it is easier to adjust it manually by dragging the start point; refer to [Figure-102](#).

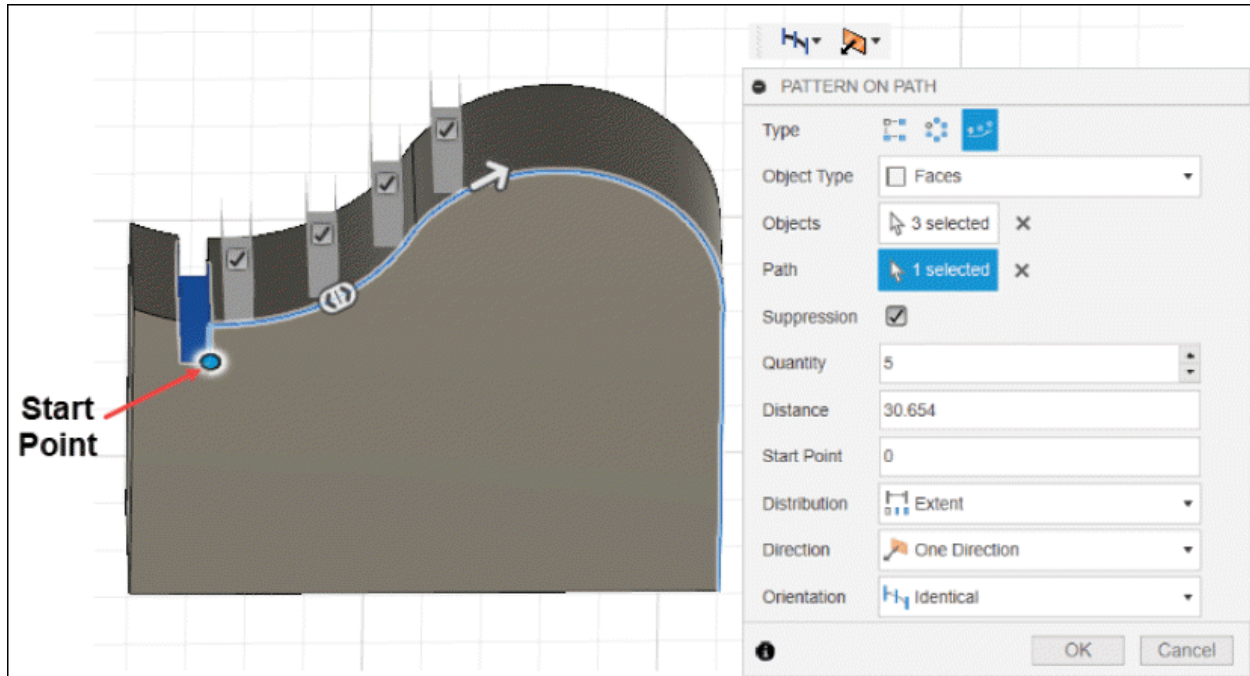


Figure-102. Pattern creation with start point

- The options of **Distance Type** and **Direction** drop-down were discussed earlier in this chapter.
- Select **Path Direction** option from **Orientation** drop-down if you want to rotate each instance according to the pattern path. Select **Identical** option from **Orientation** drop-down to keep the instances in the same orientation as the original object.
- Click **OK** button from **PATTERN ON PATH** dialog box to complete the process of creating the copies; refer to [Figure-103](#).

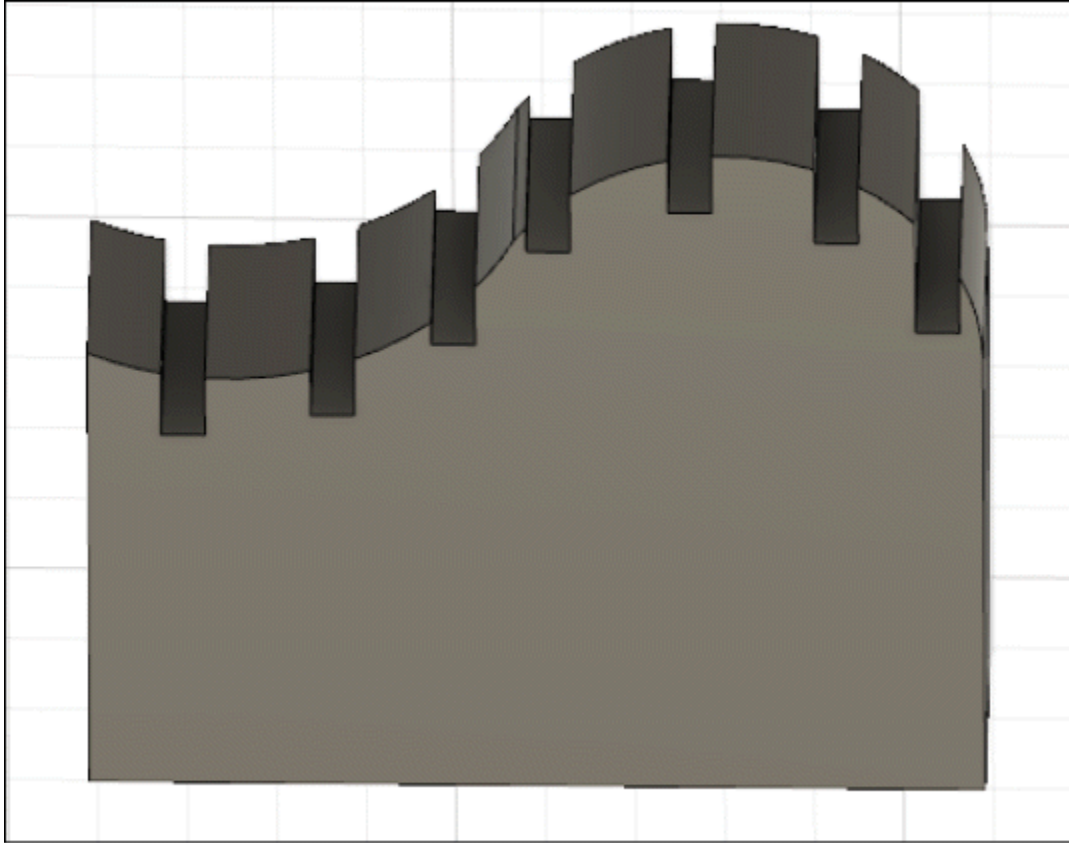


Figure-103. Solid model after applying pattern on path tool

Geometric Pattern

The **Geometric Pattern** tool is used to create a pattern with size and distribution gradients across a face on a solid body. The procedure to use this tool is discussed next.

- Click on the **Geometric Pattern** tool of **Pattern** cascading menu from **CREATE** drop-down; refer to [Figure-104](#). The **GEOMETRIC PATTERN** dialog box will be displayed; refer to [Figure-105](#).

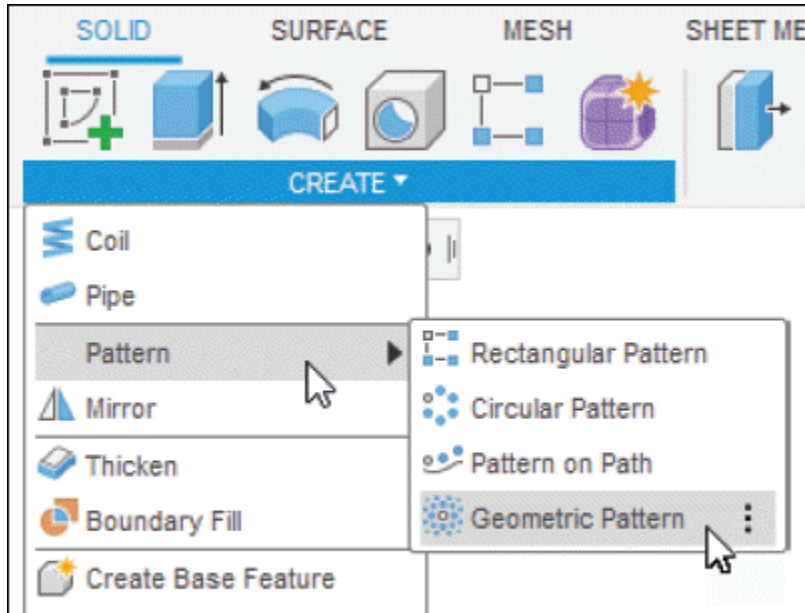


Figure-104. Geometric Pattern tool

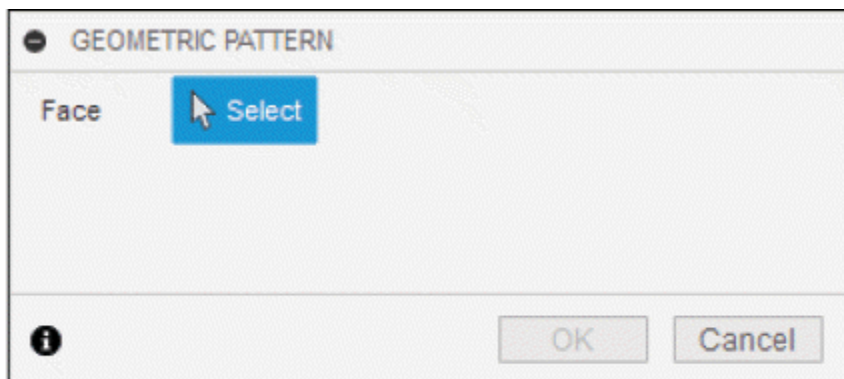


Figure-105. GEOMETRIC PATTERN dialog box

- The **Select** button of **Face** section is active by default. Select desired face on a solid body to create the pattern to it; refer to [Figure-106](#). The updated **Geometric Pattern** dialog box will be displayed; refer to [Figure-107](#).

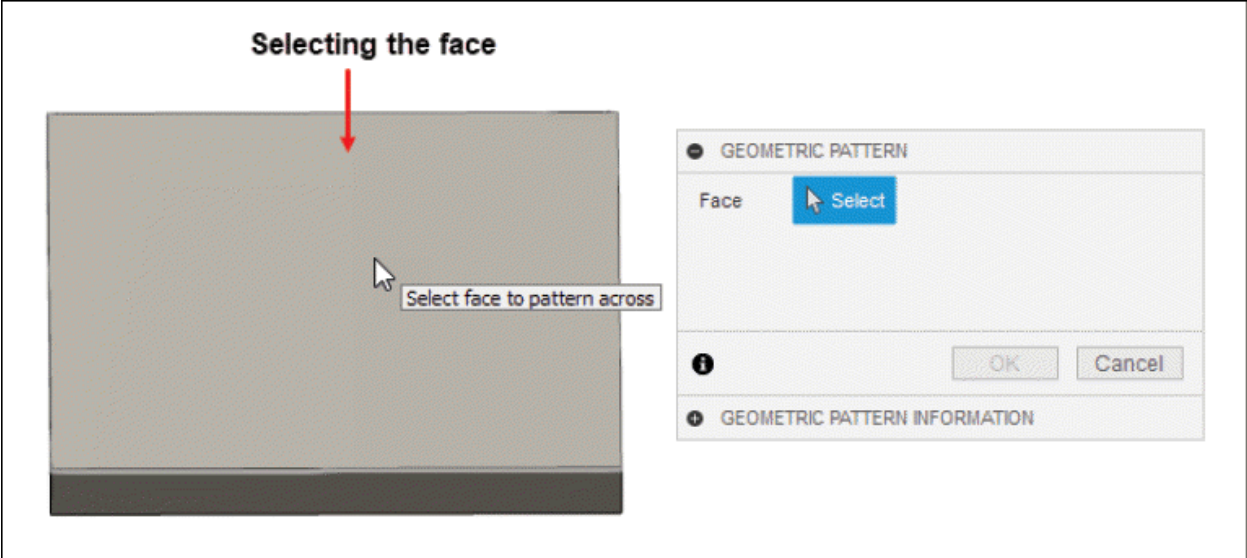


Figure-106. Selecting the face

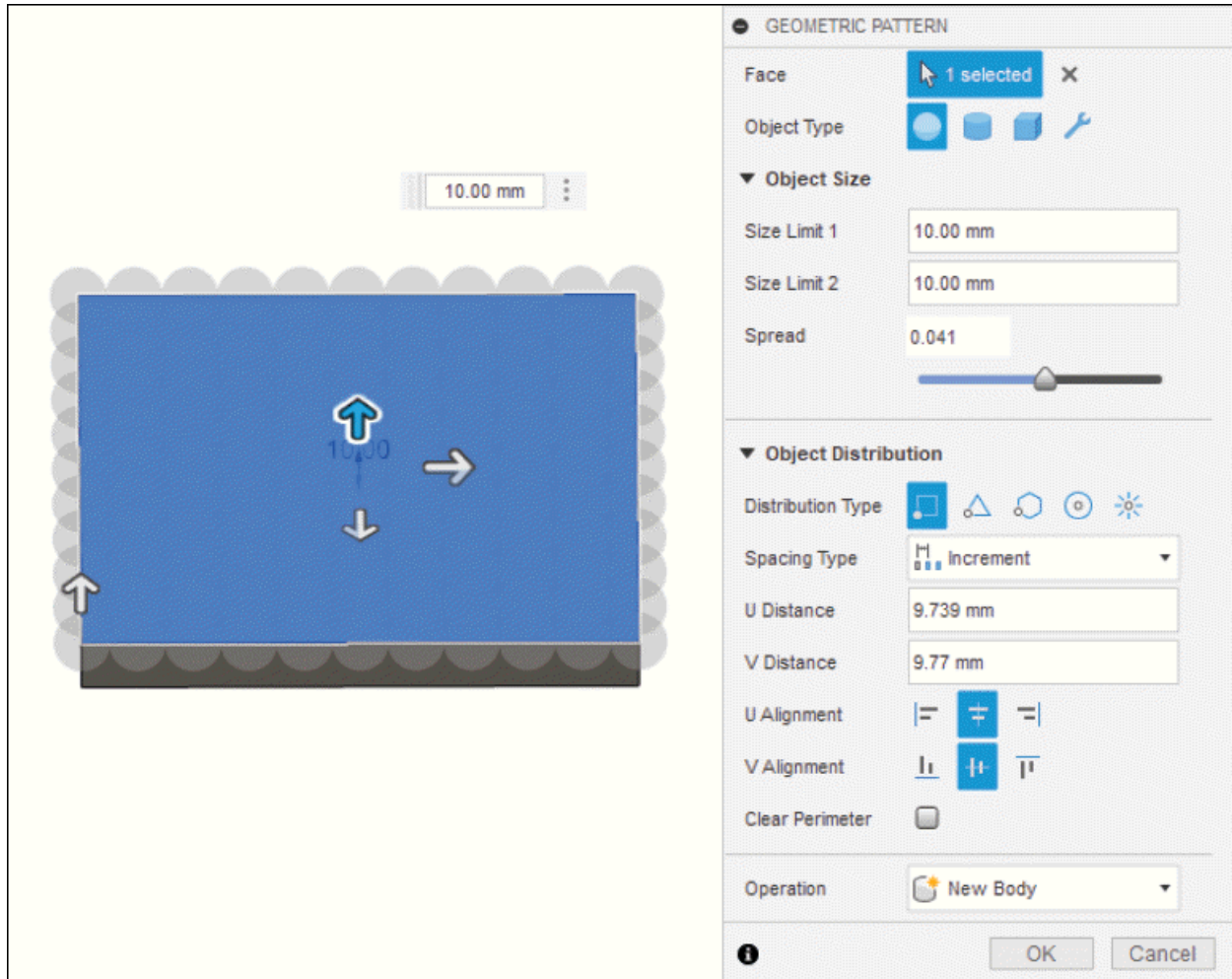


Figure-107. Updated GEOMETRIC PATTERN dialog box

- Select desired option from **Object Type** section to pattern across the selected face. Select **Sphere** option to pattern primitive sphere across the selected face. Select **Cylinder** option to pattern primitive cylinder across the selected face. Select **Box** option to pattern primitive box across the selected face. Select **Custom** option to pattern a custom object across the selected face.
- Specify minimum and maximum sizes for the objects in the pattern in the **Size Limit 1** and **Size Limit 2** edit boxes

from **Object Size** rollout.

- Specify the rate of transition from one size limit to the other in the **Spread** edit box or by sliding the **Spread slider**.
- In the **Object Distribution** rollout, select **Rectangular**, **Triangular**, **Hexagonal**, **Circular**, or **Radial** option from **Distribution Type** section to distribute objects across the faces in a repeating rectangular, triangular, hexagonal, circular, or radial pattern, respectively.
- For the rectangular pattern, select desired option from **Spacing Type** drop-down. Select **Increment** option to space objects in the pattern at specified distance relative to each other. Select **Extent** option to space objects in the pattern evenly over the specified distance.
- For the circular and radial pattern, specify the distance between each concentric circle in the pattern in the **Radial Distance** edit box.
- Specify the distance between each object in the pattern in the **Distance** edit box.
- Select **Align Left**, **Align Center**, or **Align Right** option from **U Alignment** section to align the objects in the pattern to the left of the face, center of the face, or right of the face relative to its U direction, respectively.
- Specify desired distance value to offset the pattern from the edge of the face in the U direction in the **U Offset** edit

box.

- Select **Align Bottom**, **Align Middle**, or **Align Top** option from **V Alignment** section to align the objects in the pattern to the bottom, middle, or top of the face relative to its V direction, respectively.
- Select **Clear Perimeter** check box to remove incomplete objects from the perimeter of the selected faces.
- Specify the distance value to offset the perimeter of the selected faces to adjust the boundary of the pattern in the **Perimeter Offset** edit box.
- After specifying desired parameters, click on the **OK** button from the dialog box. The geometrical pattern will be created; refer to [Figure-108](#).

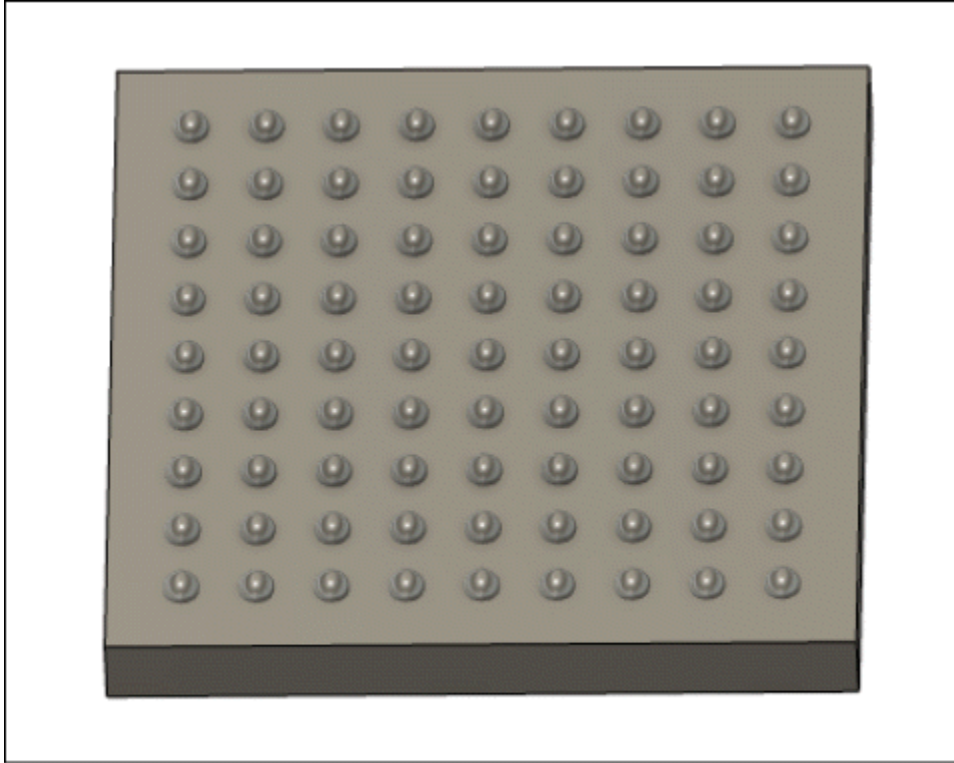


Figure-108. Geometrical pattern created

MIRROR

The **Mirror** tool is used to create mirror copy of selected objects to the opposite side of a selected face or plane. The procedure to use this tool is discussed next.

- Click on **Mirror** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-109](#). The **MIRROR** dialog box will be displayed; refer to [Figure-110](#).

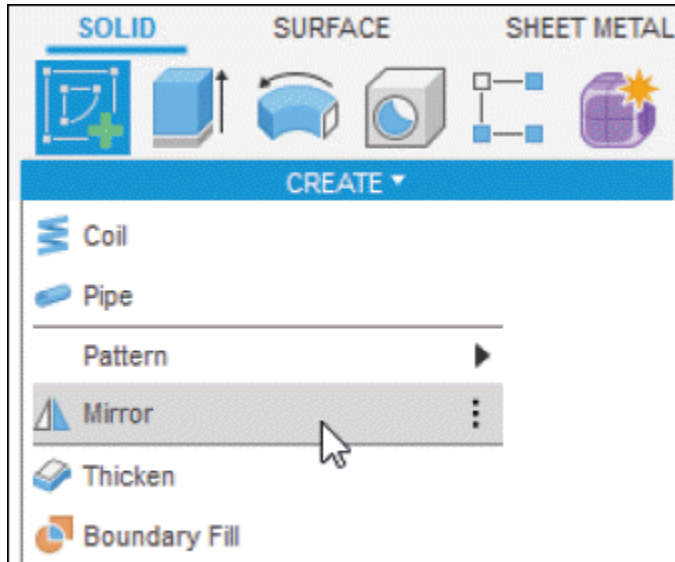


Figure-109. Mirror tool in solid

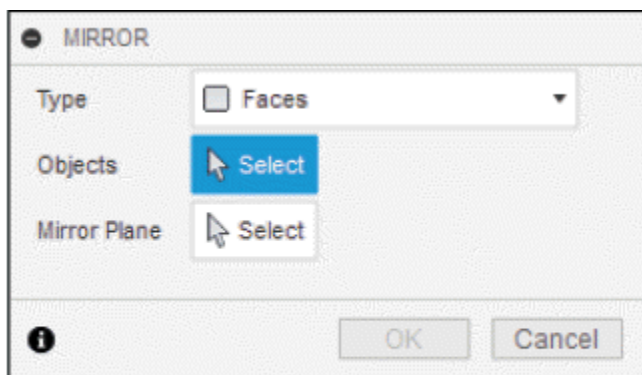


Figure-110. MIRROR dialog box in solid

- Select desired option from **Type** drop-down to select respective objects for mirroring. The other options of **Type** drop-down have already been discussed earlier.
- Select the bodies to be mirrored.
- Click on **Select** button of **Mirror Plane** section and select the plane/face to be used as mirror plane. Preview of mirror copy will be displayed; refer to [Figure-111](#).

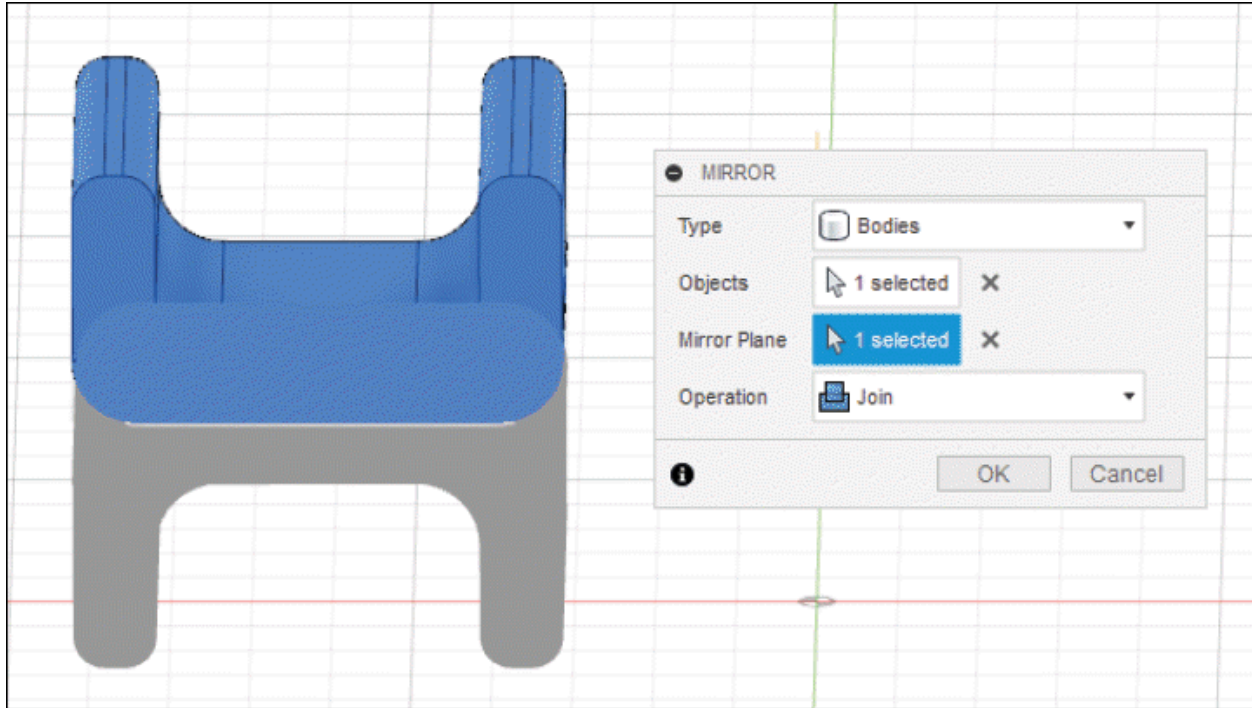


Figure-111. Preview of mirror on selection of plane

- Select desired option from **Operation** drop-down which have been discussed earlier in this chapter.
- If you have selected **Features** in **Type** drop-down then **Compute Option** drop-down will also be displayed in the dialog box; refer to [Figure-112](#). Select the **Optimized** option if you want to create copy of feature with minimum processing time and less flexibility to individually change features later. Select **Identical** option if you want to create copy of feature using same background programming which was used while creating the base feature. Select the **Adjust** option if you want to recompute each instance of the feature as a separate feature.

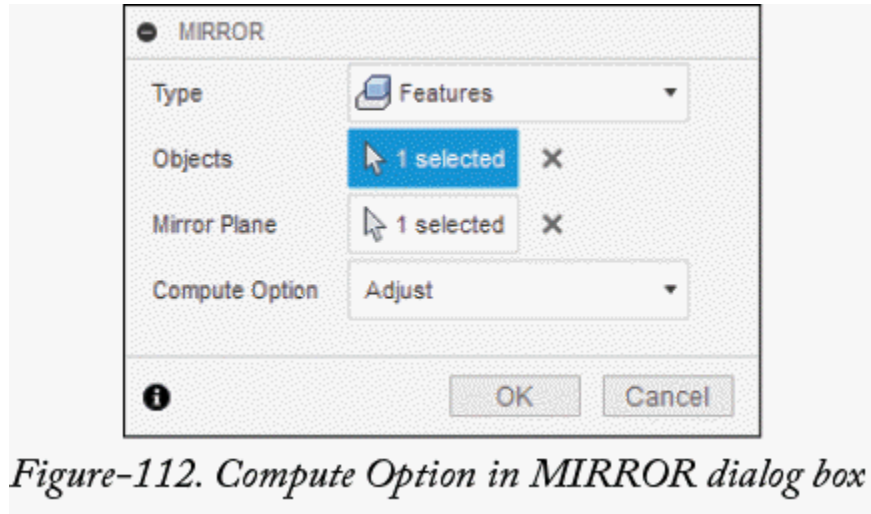


Figure-112. Compute Option in MIRROR dialog box

- Click **OK** button of **MIRROR** dialog box to finish the process of creating mirror copy; refer to [Figure-113](#).

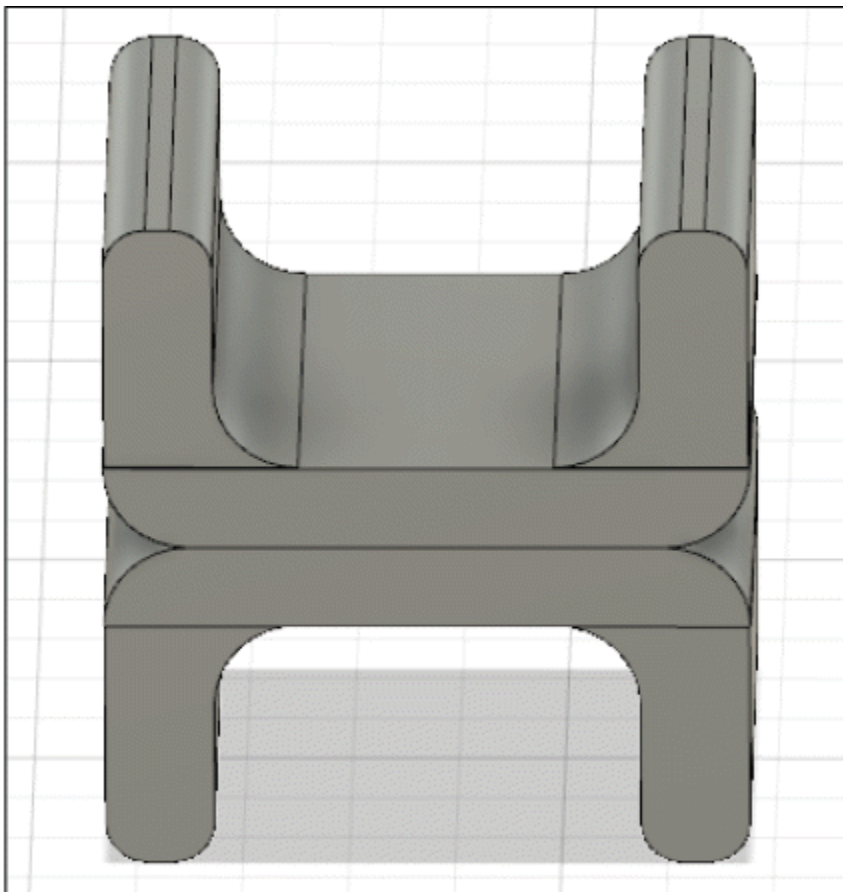


Figure-113. Solid model after applying mirror tool

THICKEN

The **Thicken** tool is used to add thickness to surfaces to make them solid. (You will learn about surfaces in Chapter 9 Surface Modeling of this book.) The procedure to use this tool is discussed next.

- Open any file that has surfaces or create a surface by following basic tools of Chapter 9 of this book.
- Click on **Thicken** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-114](#). A **THICKEN** dialog box will be displayed; refer to [Figure-115](#).

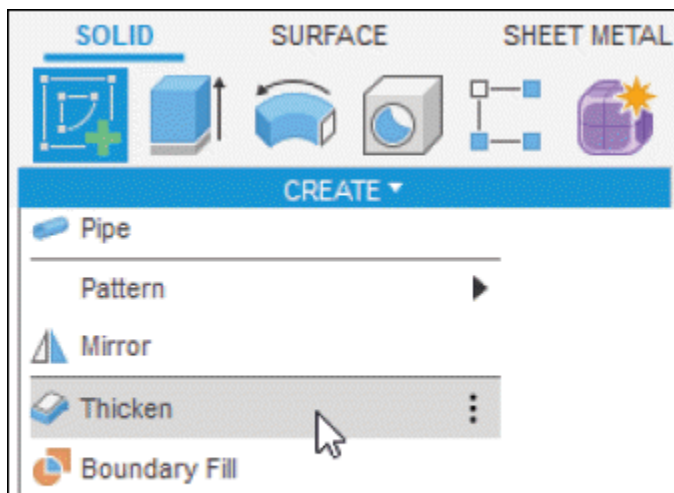


Figure-114. Thicken tool

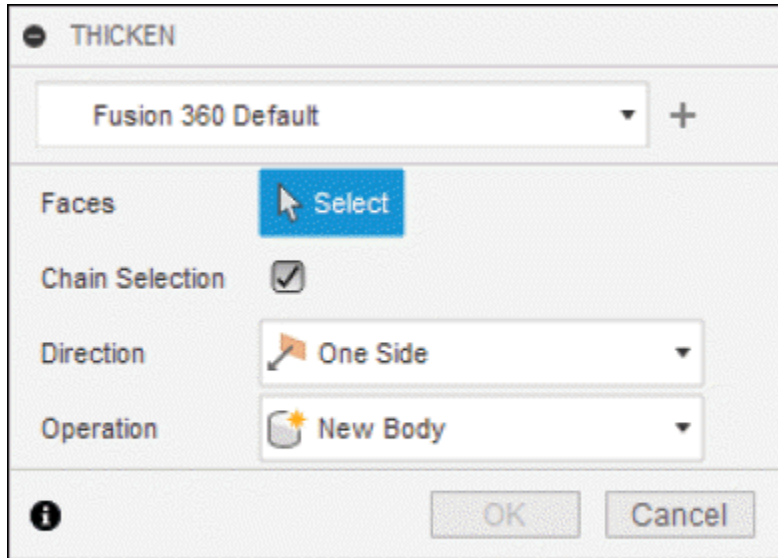


Figure-115. THICKEN dialog box

- **Faces** selection is active by default. Select the surface to add thickness; refer to [Figure-116](#).

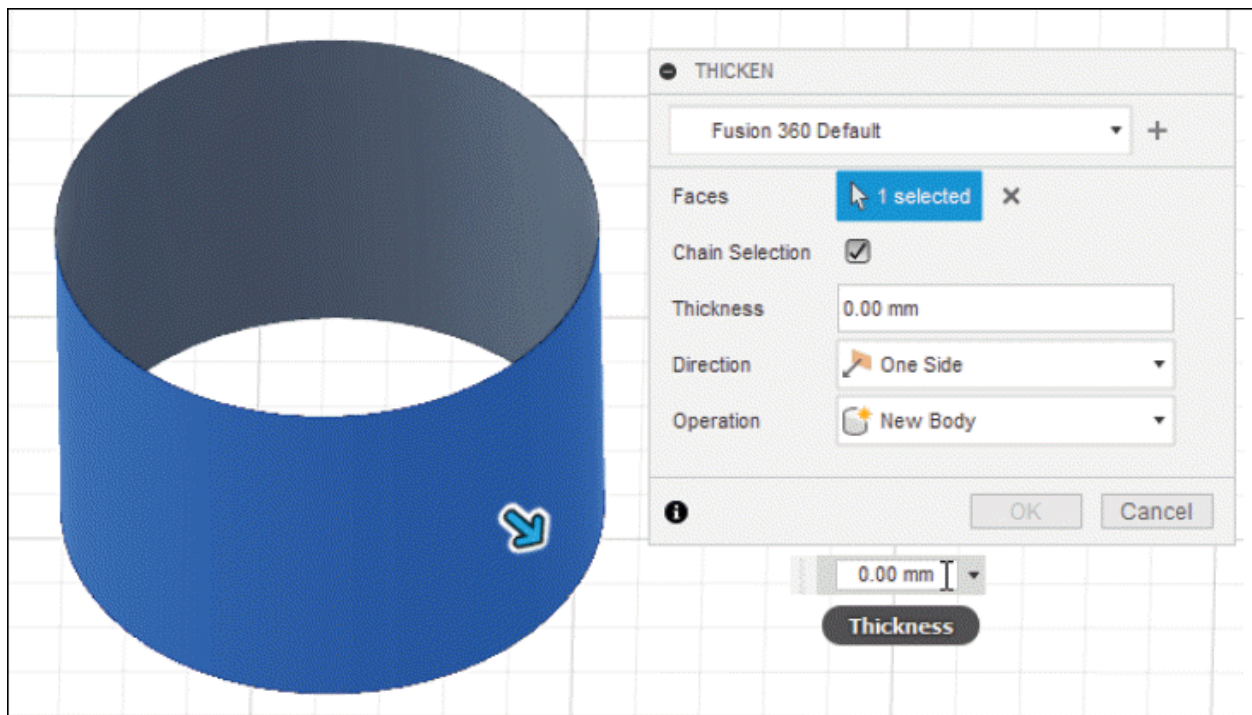


Figure-116. Selection of faces to add thickness

- Click in the **Thickness** edit box of dialog box and specify

desired value of thickness.

- Click on **One Side** option from **Direction** drop-down to apply the thickness to one side of surface. Click on **Symmetric** option from **Direction** drop-down to apply the thickness to both sides of surface equally. In our case, we have selected **Symmetric** option.
- The options of **Operation** drop-down were discussed earlier in this chapter.
- Click **OK** button of **THICKEN** dialog box to finish the process; refer to [Figure-117](#).

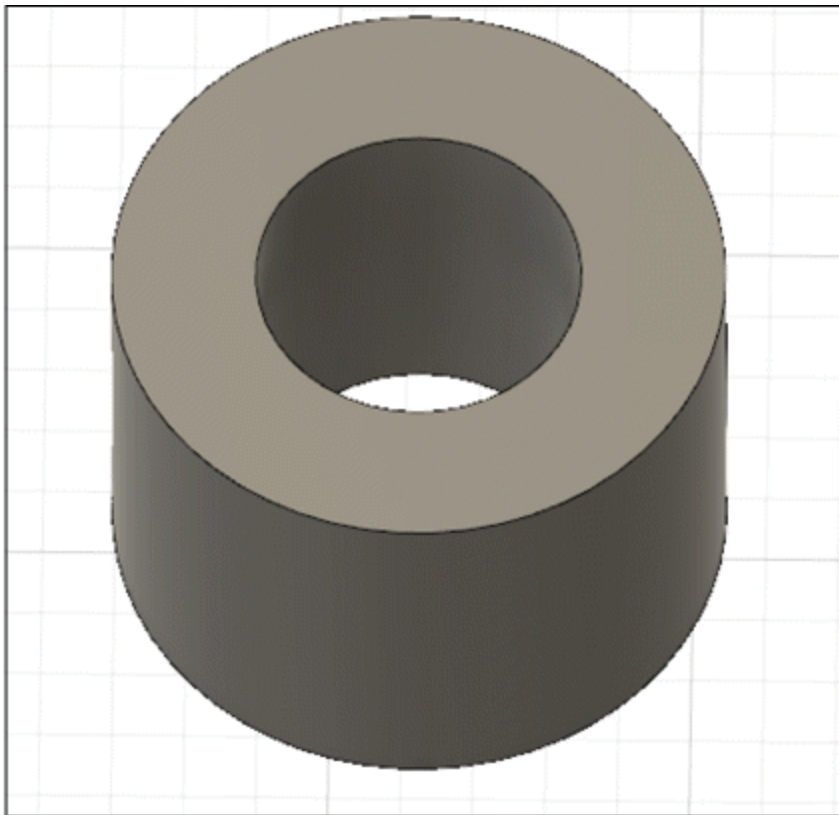


Figure-117. Solid model after applying thicken tool

BOUNDARY FILL

The **Boundary Fill** tool is used to create a volume from bounded by selected planes, bodies, and surfaces. This volume is called cell and can be used to cut, add, or create new body in the model. The procedure to use this tool is discussed next.

- Click on **Boundary Fill** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-118](#). The **BOUNDARY FILL** dialog box will be displayed; refer to [Figure-119](#).

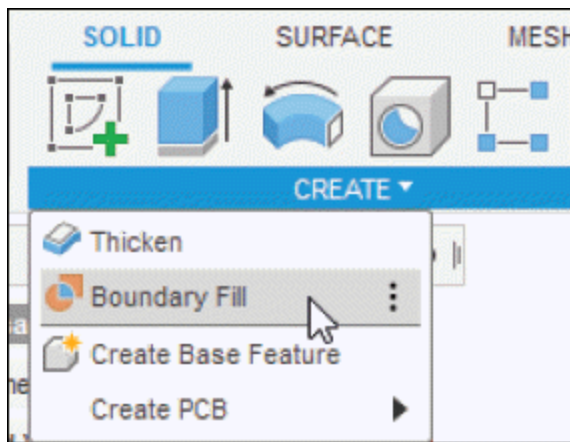


Figure-118. Boundary Fill tool

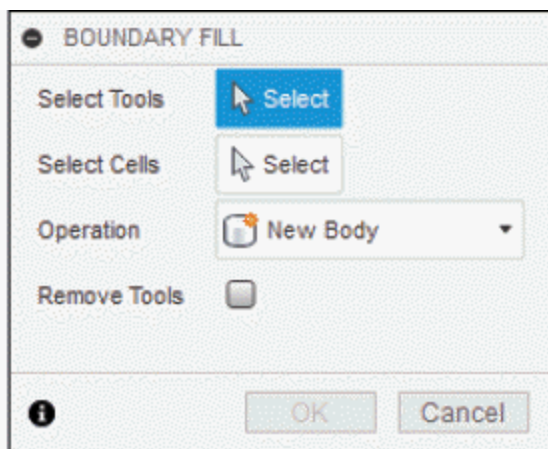


Figure-119. BOUNDARY FILL dialog box

- The **Select Tools** selection is active by default. Select the

bodies/planes/surfaces for boundary fill; refer to [Figure-120](#).

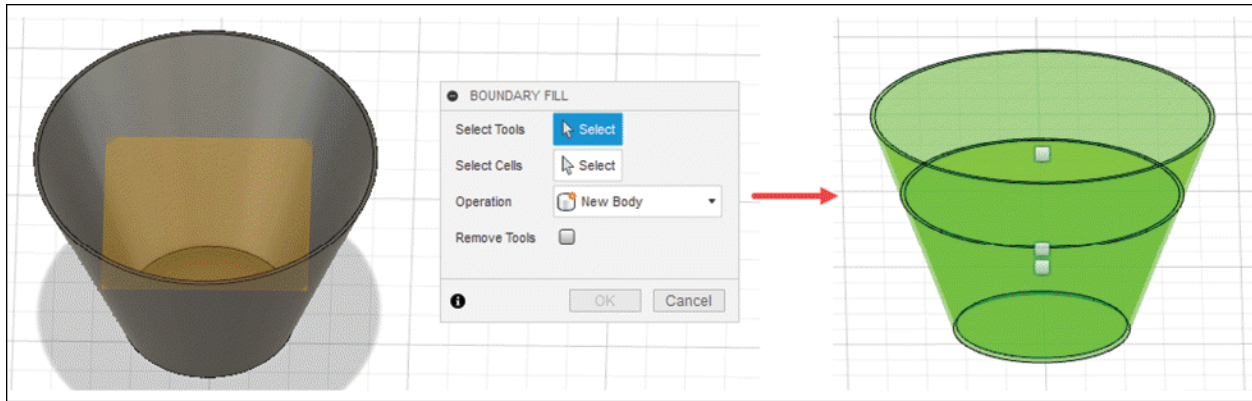


Figure-120. Selection for boundary fill

- Click on **Select** button of **Select Cells** section and select the check boxes for cells to be created.
- The tools of **Operation** section were discussed earlier in this chapter.
- Click **OK** button of **BOUNDARY FILL** dialog box to finish the process; refer to [Figure-121](#). If the parent body is overlapping with new boundary fill bodies then hide the parent body using eye icon in **BROWSER** to check the result.

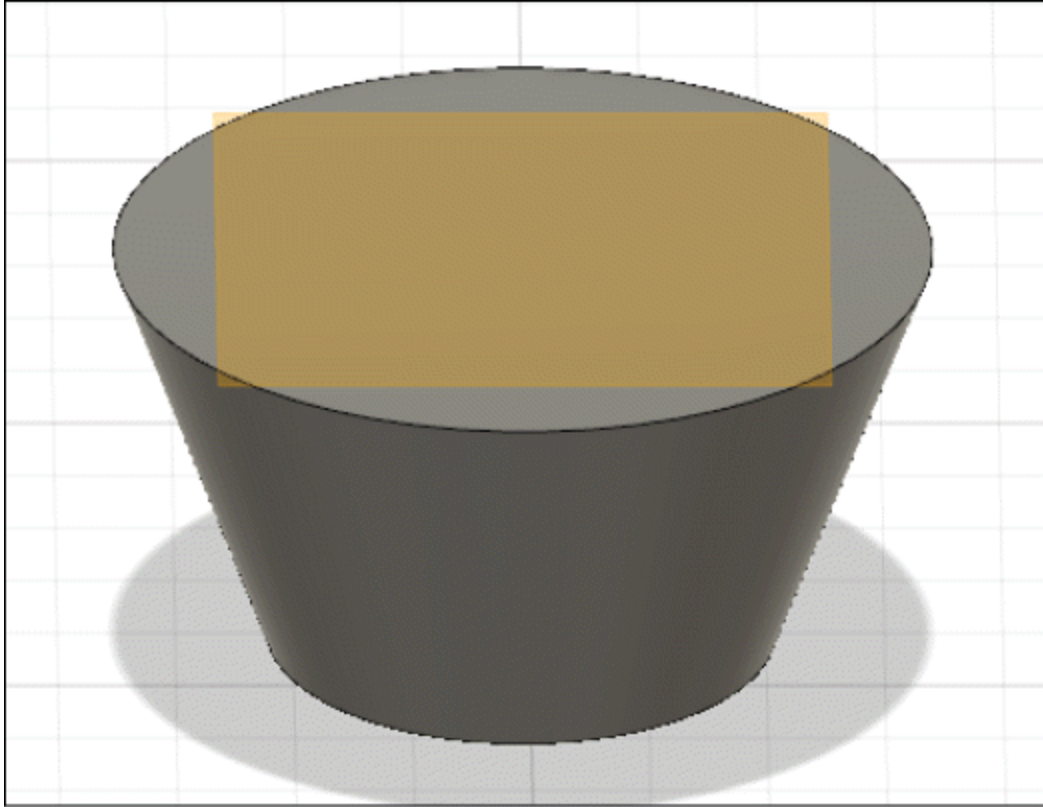


Figure-121. Result of boundary fill after hiding parent body

CREATE FORM

The **Create Form** tool is used to create or modify the form of model using freestyle editing tools. Clicking this tool takes you to **Form** mode. You will learn more about **Form** mode later in Chapter 12 and 13 of this book. The procedure to use this tool is discussed next.

- Click on **Create Form** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-122](#). The **Form mode** will be activated and related toolbar will be displayed; refer to [Figure-123](#).

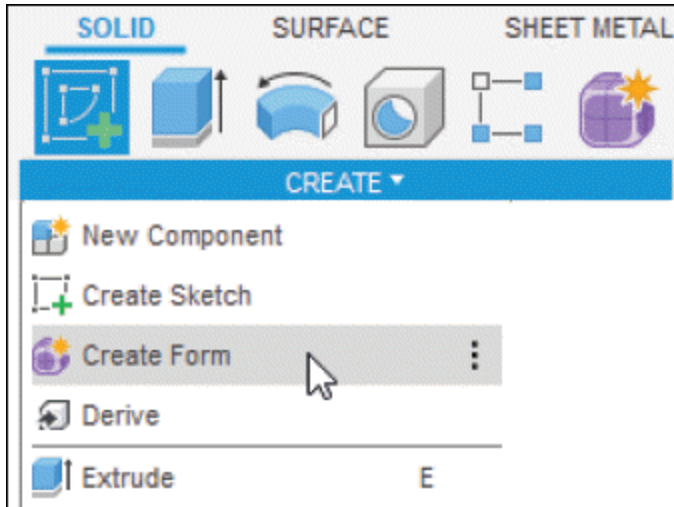


Figure-122. Create Form tool

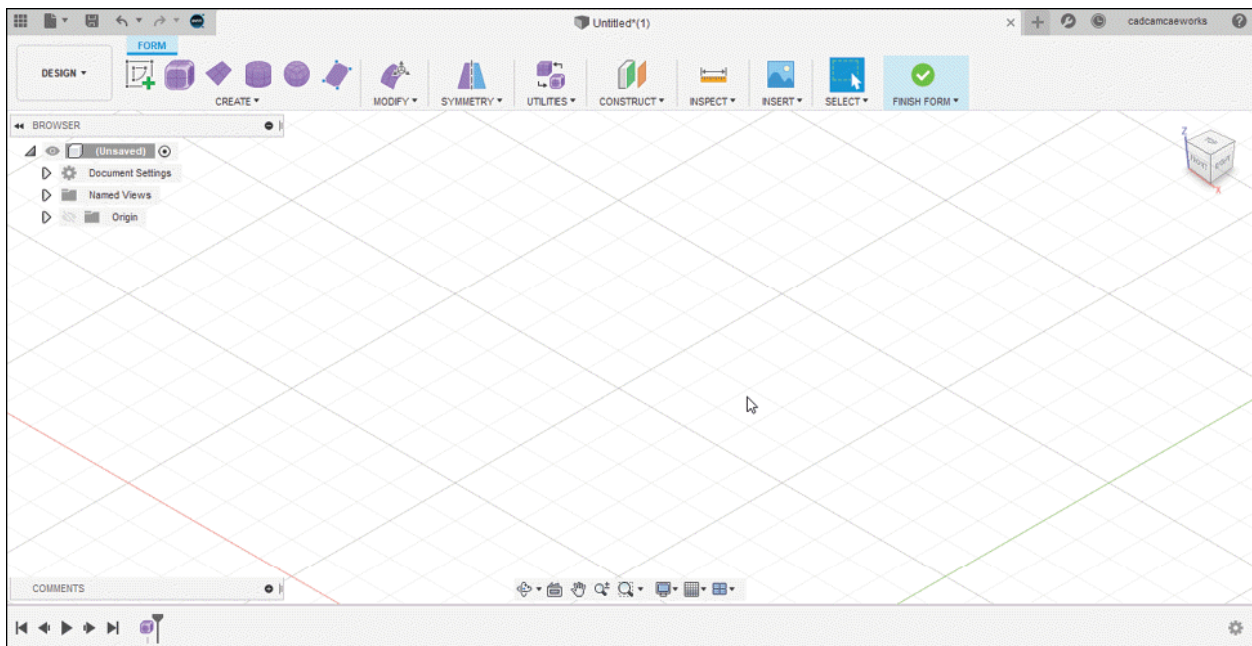


Figure-123. Form mode

- Now, create or modify the model as desired.
- Click **Finish Form** button from **Toolbar** to exit the **Form mode** and return to **Model Workspace**; refer to [Figure-124](#).

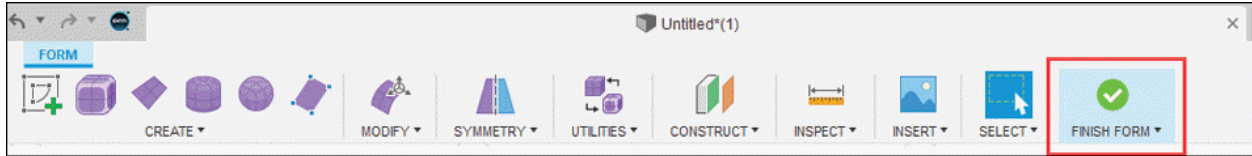


Figure-124. Finish Form button

CREATE BASE FEATURE

The **Create Base Feature** tool inserts a base feature operation in the **Timeline**. Any operations performed on the design while in a base feature are not recorded in the **Timeline**. This tool is used when you want to safeguard some of the operations while sharing the file.

- Click on **Create Base Feature** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-125](#). The tools in the **Toolbar** will be same as for **DESIGN** workspace.

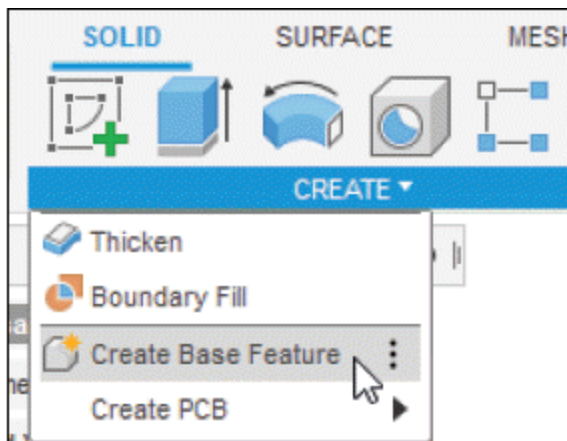


Figure-125. Create Base Feature tool

- Do desired operation that you do not want to record in **Timeline** and then click on **FINISH BASE FEATURE** button from **Toolbar** to exit the mode; refer to [Figure-126](#).

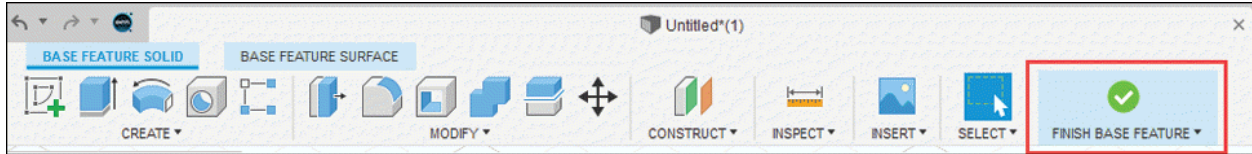


Figure-126. Finish Base Feature button

CREATE PCB FEATURE

The **Create PCB** tools in **CREATE** drop-down is used to create and modify PCB design in collaboration with **EAGLE** users. There are two tools in **Create PCB** cascading menu; refer to [Figure-127](#). The procedures to use these tools are given next.

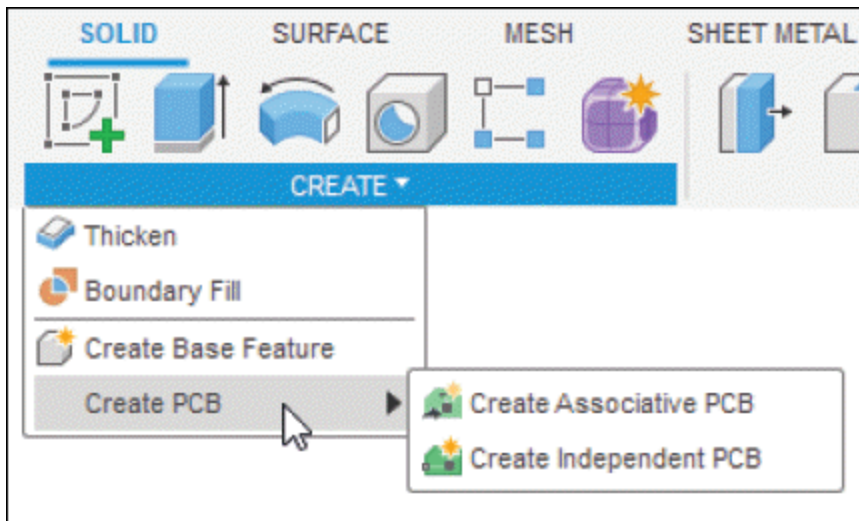


Figure-127. Create PCB cascading menu

Create Associative PCB

- Click on the **Create Associative PCB** tool from **Create PCB** cascading menu; refer to [Figure-127](#). A pop-up window will be displayed asking you to save the current design; refer to [Figure-128](#).

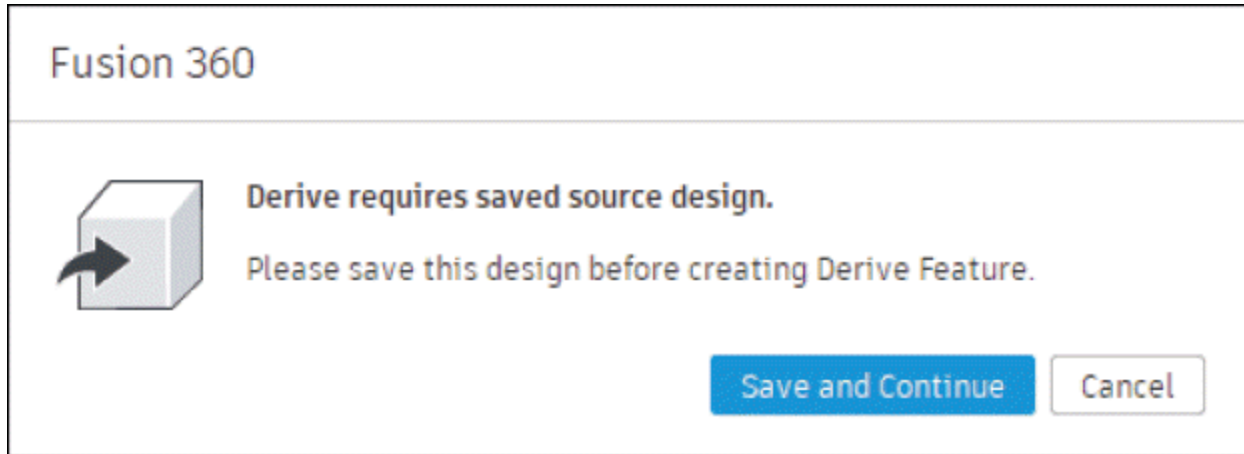


Figure-128. Pop-up window

- Click on the **Save and Continue** button. The **Save** dialog box will be displayed as discussed earlier.
- Specify desired name and location for the file and click on the **Save** button. The **CREATE ASSOCIATIVE PCB** dialog box will be displayed; refer to [Figure-129](#).

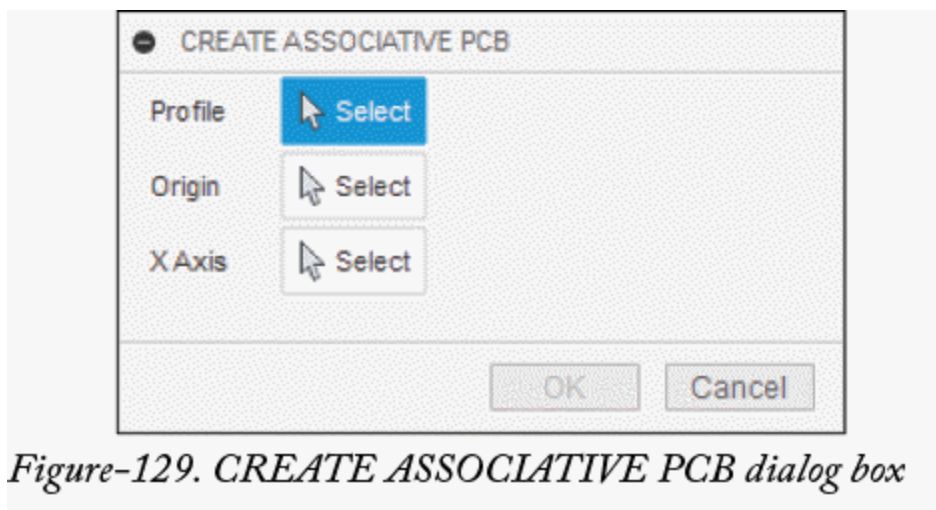


Figure-129. CREATE ASSOCIATIVE PCB dialog box

- Select desired sketch profile from canvas; refer to [Figure-130](#) and specify desired parameters in the dialog box.

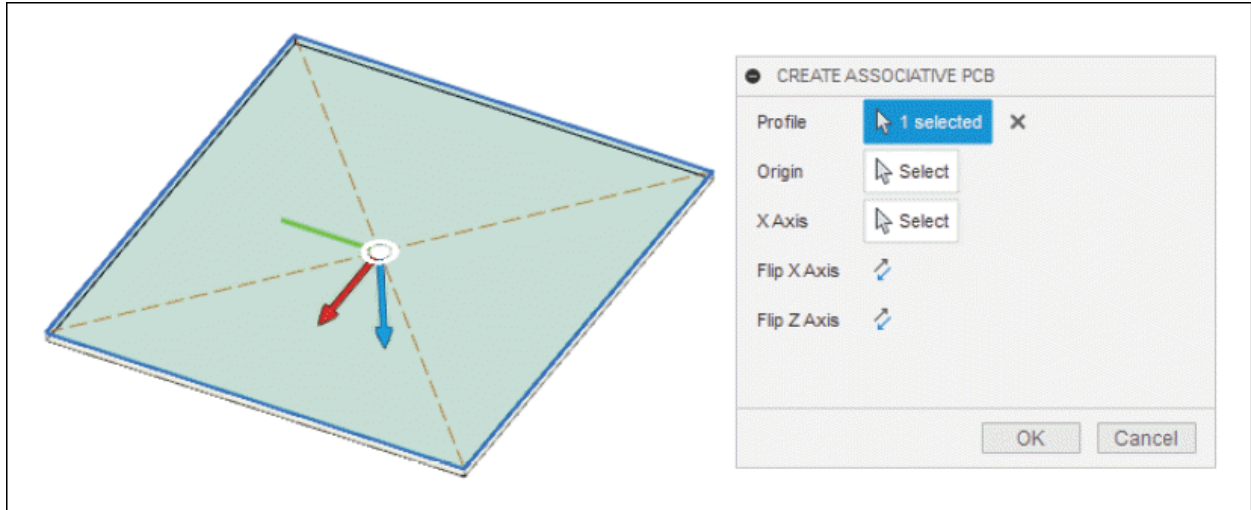


Figure-130. Selecting the profile

- Click on the **OK** button from the dialog box. The **3D PCB Workspace** will be activated and related toolbar will be displayed; refer to [Figure-131](#).

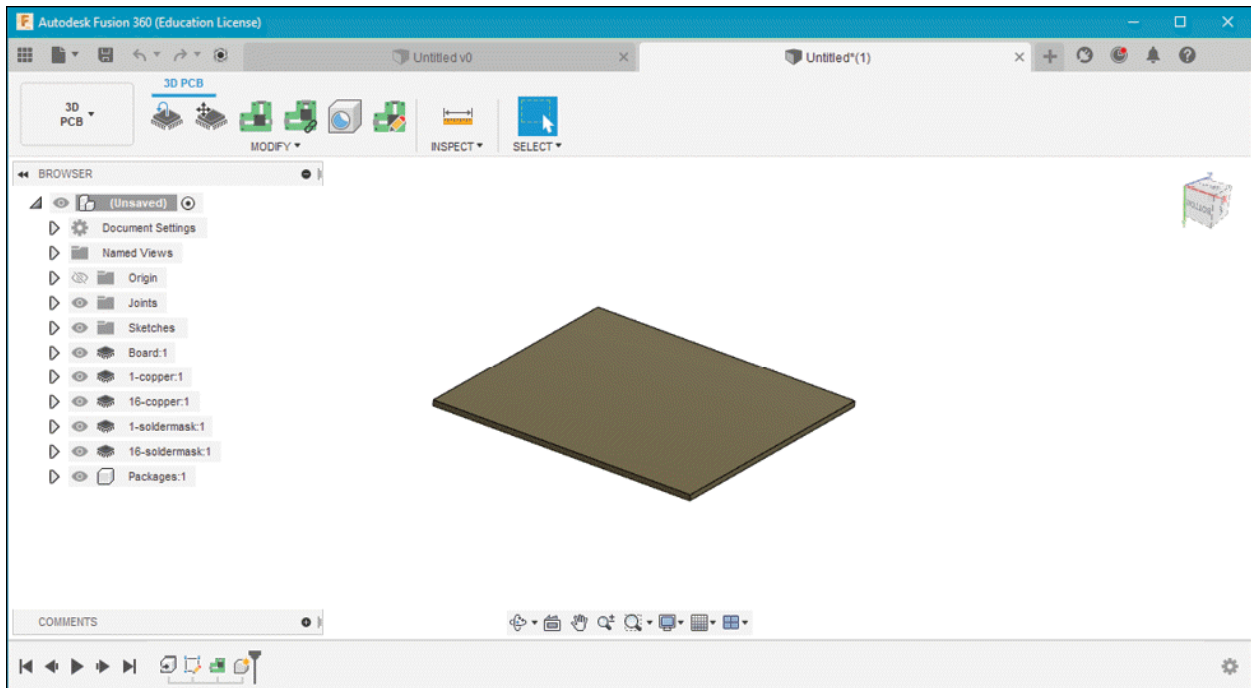


Figure-131. 3D PCB Workspace

Create Independent PCB

- Click on the **Create Independent PCB** tool from **Create PCB** cascading menu; refer to [Figure-127](#). The **CREATE INDEPENDENT PCB** dialog box will be displayed; refer to [Figure-132](#).

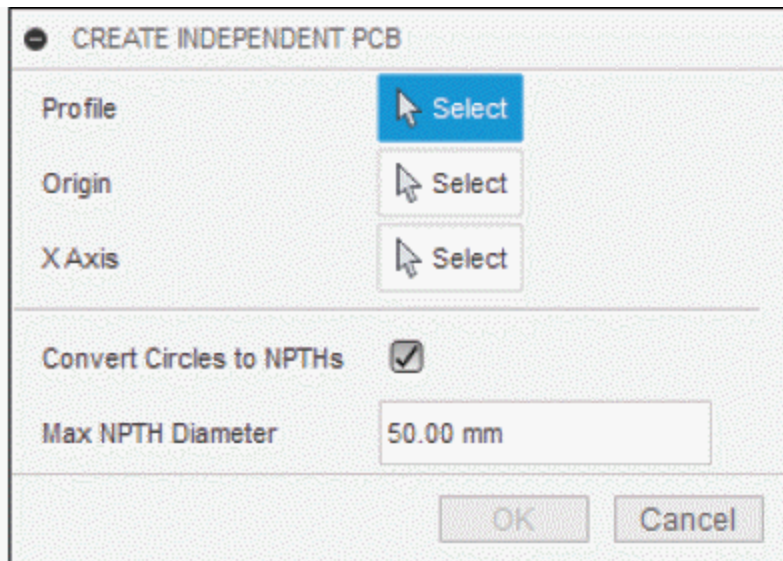


Figure-132. CREATE INDEPENDENT PCB dialog box

- Select desired sketch profile from canvas; refer to [Figure-133](#) and specify desired parameters in the dialog box.

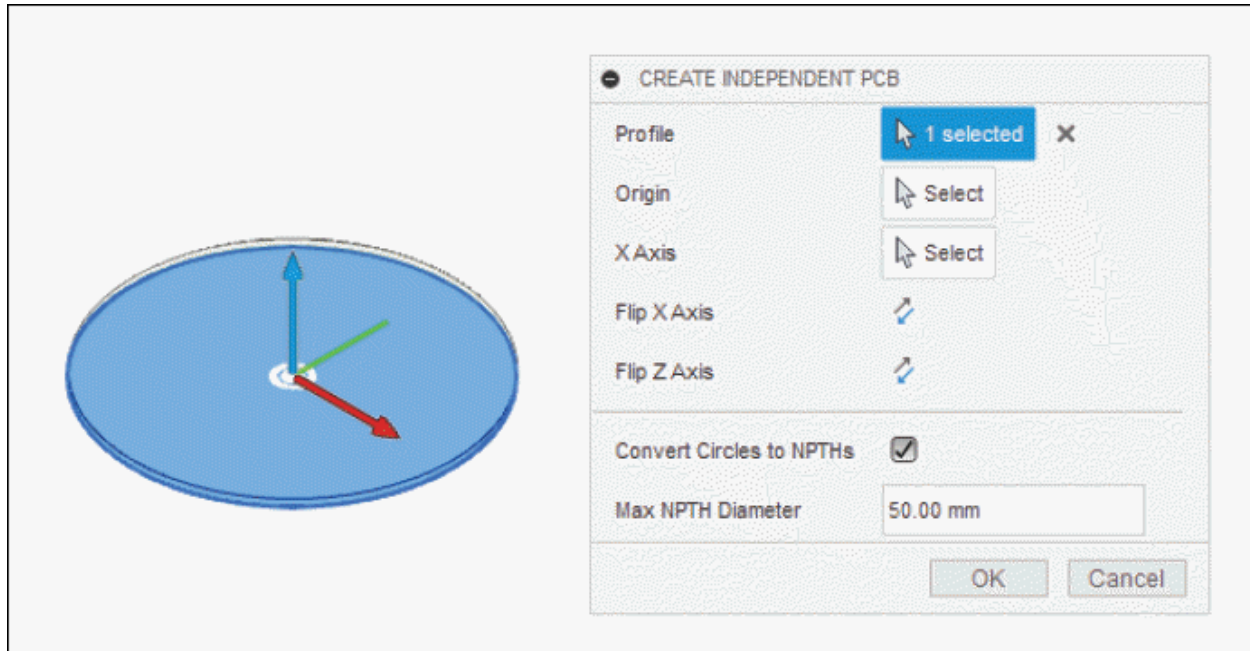


Figure-133. Selecting the profile

- Click on the **OK** button from the dialog box. The **3D PCB Workspace** will be activated and related toolbar will be displayed as discussed earlier.

Adding Holes to PCB

The **PCBHole** tool is used to create holes for adding components to the PCB. The procedure to use this tool is given next.

- Click on the **PCBHole** tool from the **MODIFY** drop-down in the **3D PCB** tab of the **Ribbon**. The **PCBHOLE** dialog box will be displayed.
- Select the face of PCB where you want to place the hole. Preview of hole will be displayed; refer to [Figure-134](#).

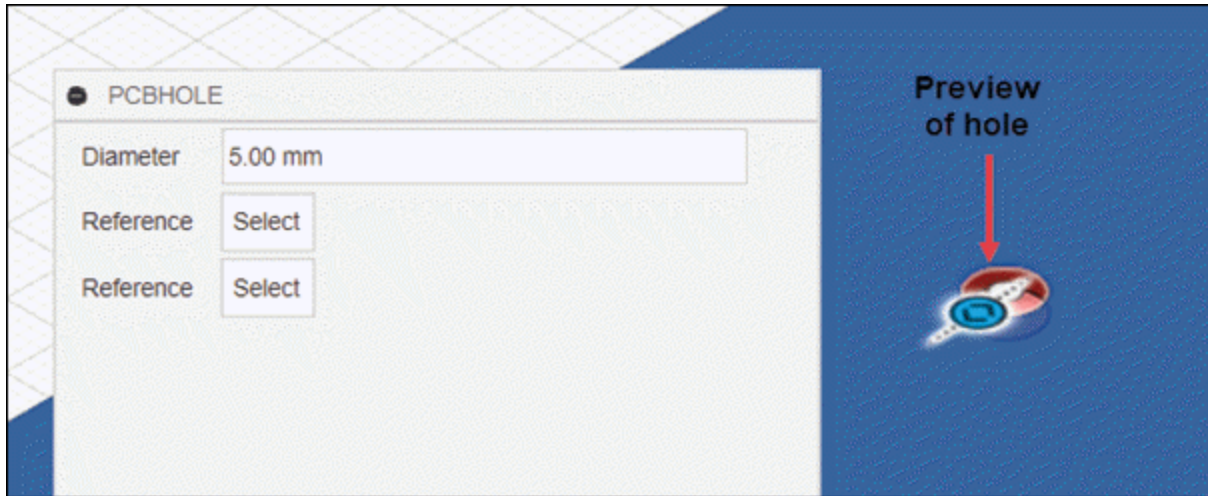


Figure-134. Preview of hole on PCB

- Click at the center of hole and select desired edge of PCB to define first reference. The options to define distance from selected reference will be displayed; refer to [Figure-135](#).

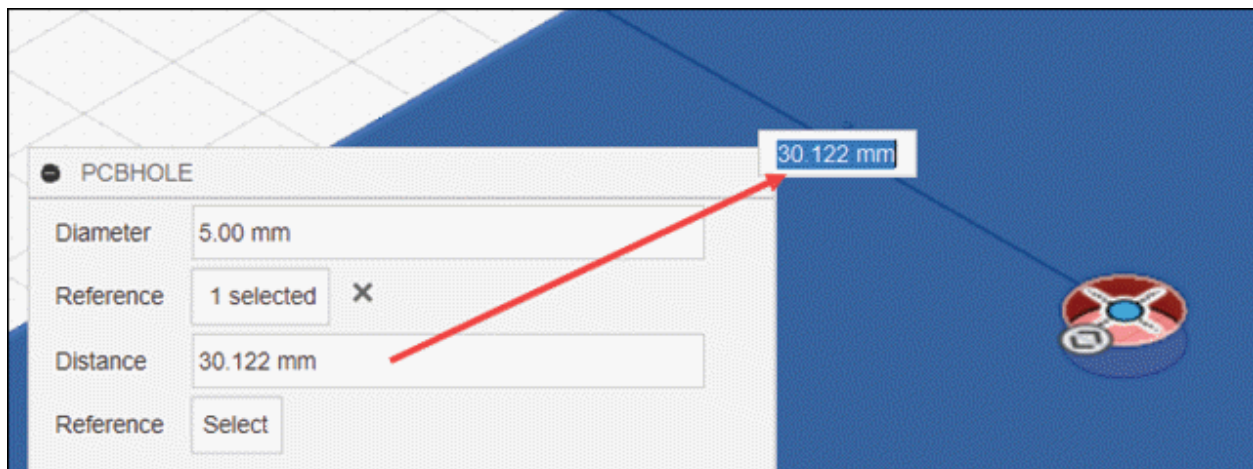


Figure-135. Specifying distance

- Similarly, specify distance for other reference and click on the **OK** button. The hole will be created.

The tools of 3D PCB are beyond the scope of this book. You will learn about PCB and Electronic Design in our another

book.

PLASTIC TOOLS

The Plastic tools are used to assign and manage plastic rules that control certain settings in the components that are assigned to them. The tools are discussed next.

Assign Plastic Rule

The **Assign Plastic Rule** tool is used to assign plastic rule to a component to automatically control the properties of plastic features like physical material and thickness. The procedure to use this tool is discussed next.

- Click on the **Assign Plastic Rule** tool from **SETUP** panel in the **PLASTIC** tab of the **Toolbar**; refer to [Figure-136](#). The **ASSIGN PLASTIC RULE** dialog box will be displayed; refer to [Figure-137](#).

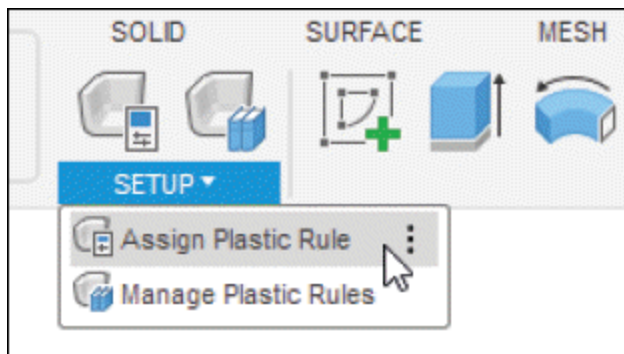


Figure-136. Assign Plastic Rule tool

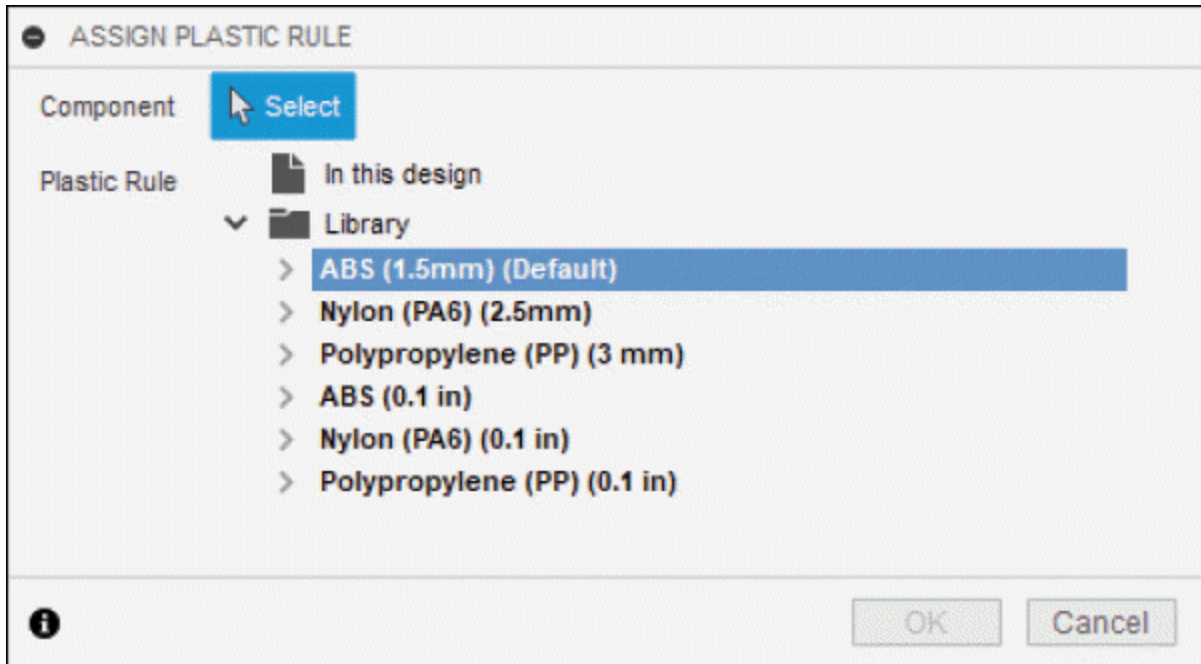


Figure-137. ASSIGN PLASTIC RULE dialog box

- The **Select** button of **Component** section is active by default. Select the component in the Model Space that you want to assign the plastic rule; refer to [Figure-138](#).

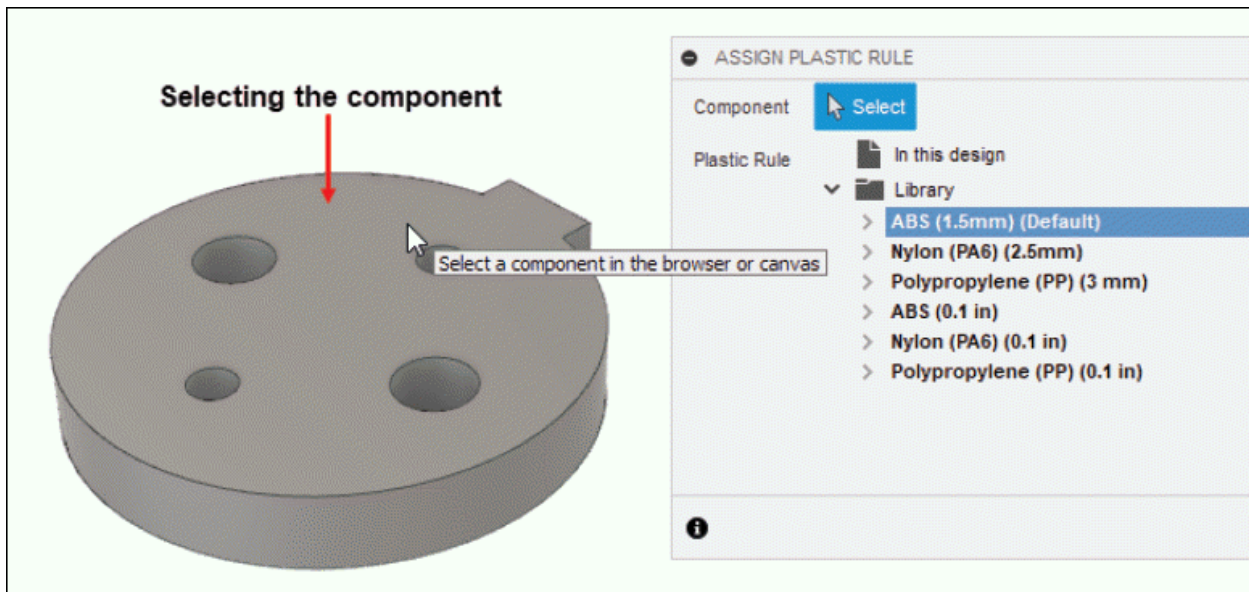


Figure-138. Selecting the component to assign plastic rule

- Select desired plastic rule from the cloud library in the **Library** node of the **Plastic Rule** section in the dialog box.
- After selecting desired plastic rule, click on the **OK** button from the dialog box. The plastic rule will be assigned to the component; refer to [Figure-139](#).

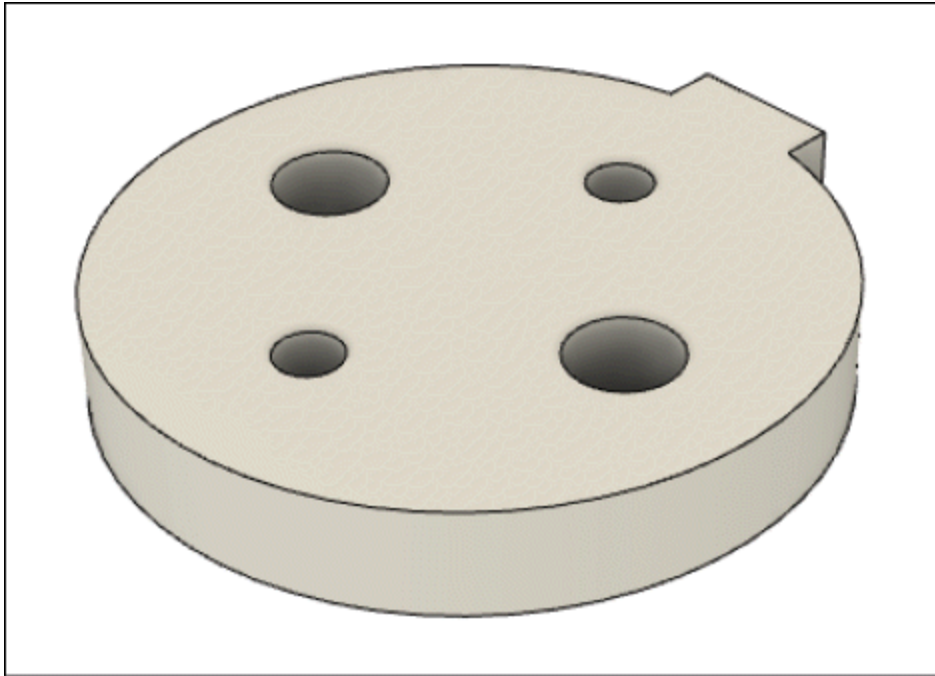


Figure-139. Plastic rule assigned to the component

Manage Plastic Rules

The **Manage Plastic Rules** tool is used to create new rules, edit existing rules, or save rules to a library to use them in multiple designs. The procedure to use this tool is discussed next.

- Click on the **Manage Plastic Rules** tool from **SETUP** panel in the **PLASTIC** tab of the **Toolbar**; refer to [Figure-140](#). The **MANAGE PLASTIC RULES** dialog box will be

displayed; refer to [Figure-141](#).

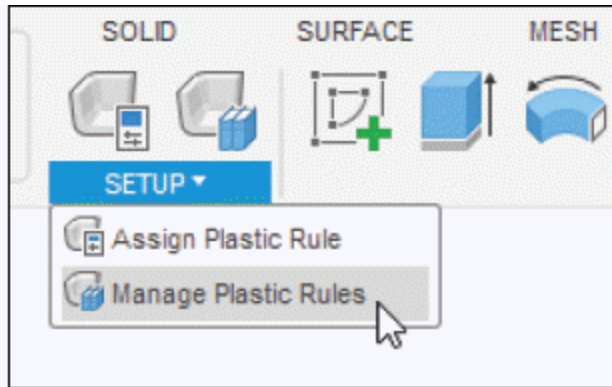


Figure-140. Manage Plastic Rules tool

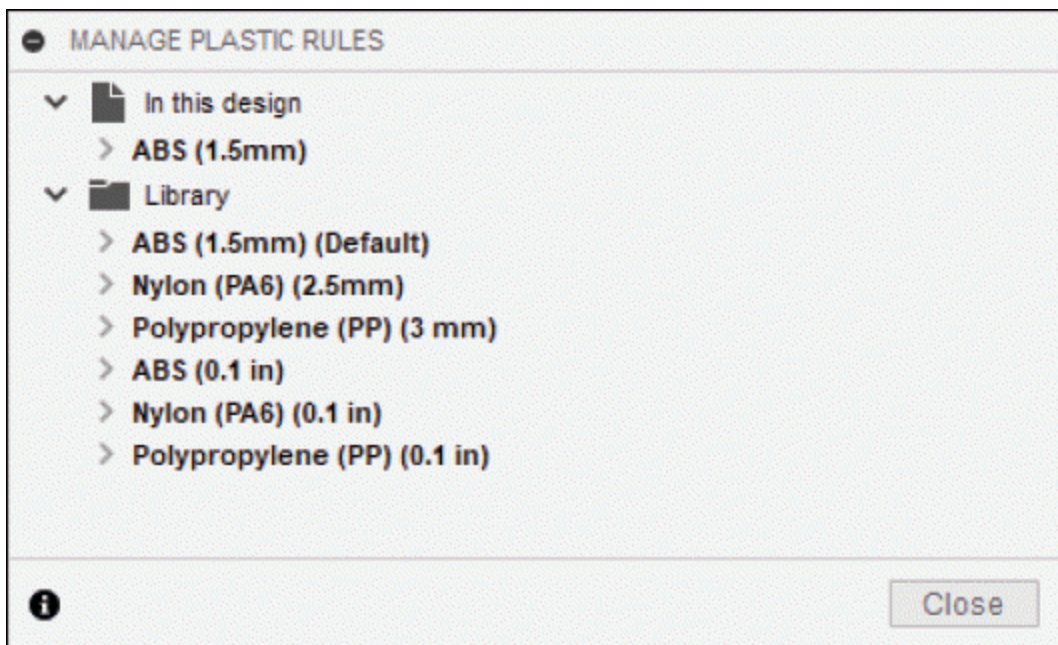


Figure-141. MANAGE PLASTIC RULES dialog box

- If you want to edit the rule, then select the rule you want to edit and click on **Edit Plastic Rule** button or right-click on the rule that you want to edit and select **Edit Plastic Rule** button; refer to [Figure-142](#). The **Edit Plastic Rule** dialog box will be displayed; refer to [Figure-143](#).

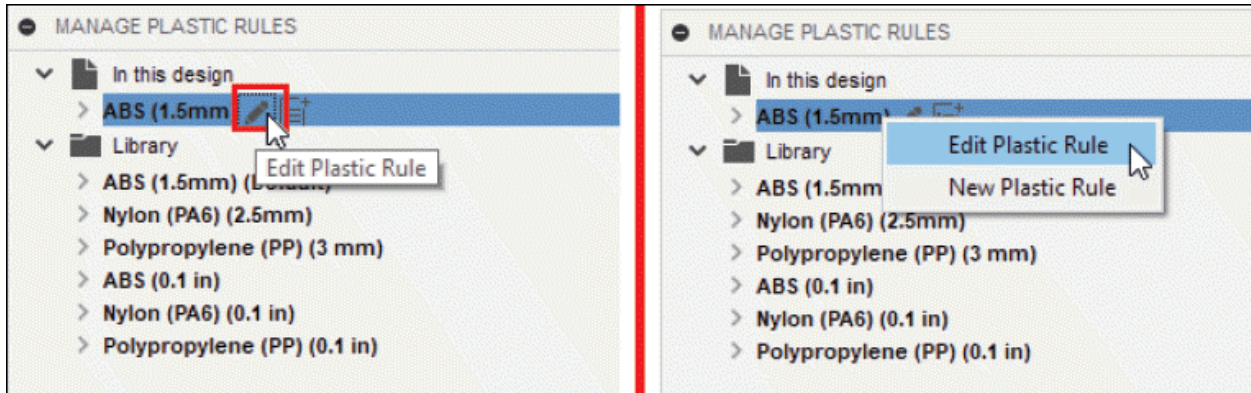


Figure-142. Edit Plastic Rule button

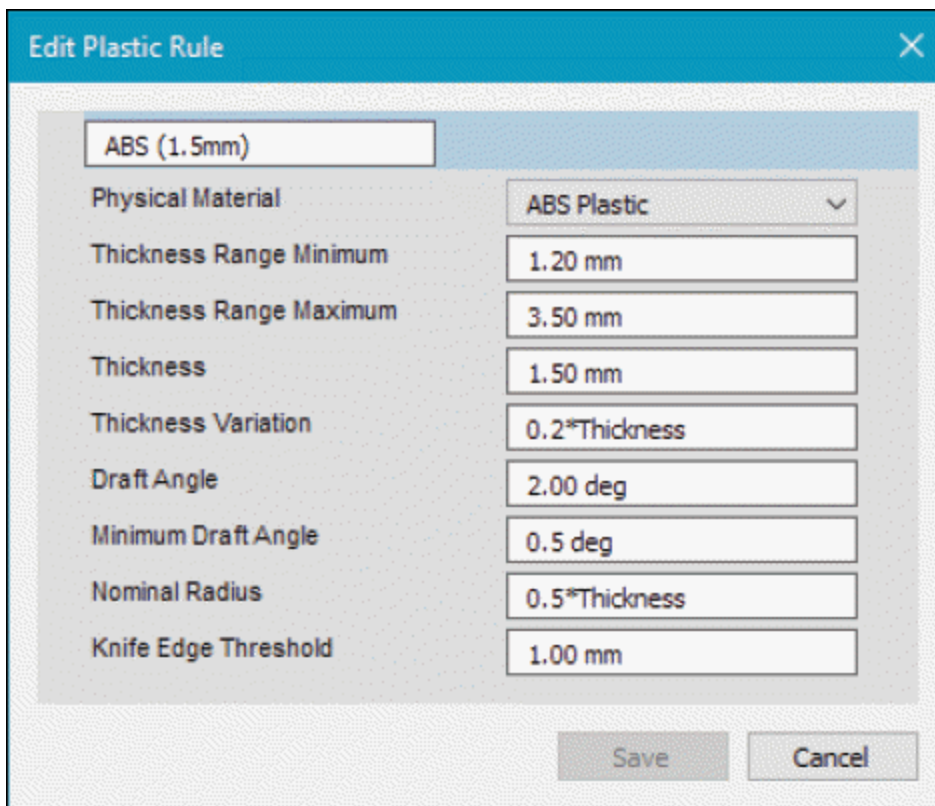


Figure-143. Edit Plastic Rule dialog box

- Edit the parameters of the rule as desired and click on the **Save** button from the dialog box. The plastic rule will be edited.
- If you want to create new plastic rule then select the rule

and click on **New Plastic Rule** button or right-click on the rule and select **New Plastic Rule** button; refer to [Figure-144](#). The **New Plastic Rule** dialog box will be displayed; refer to [Figure-145](#).

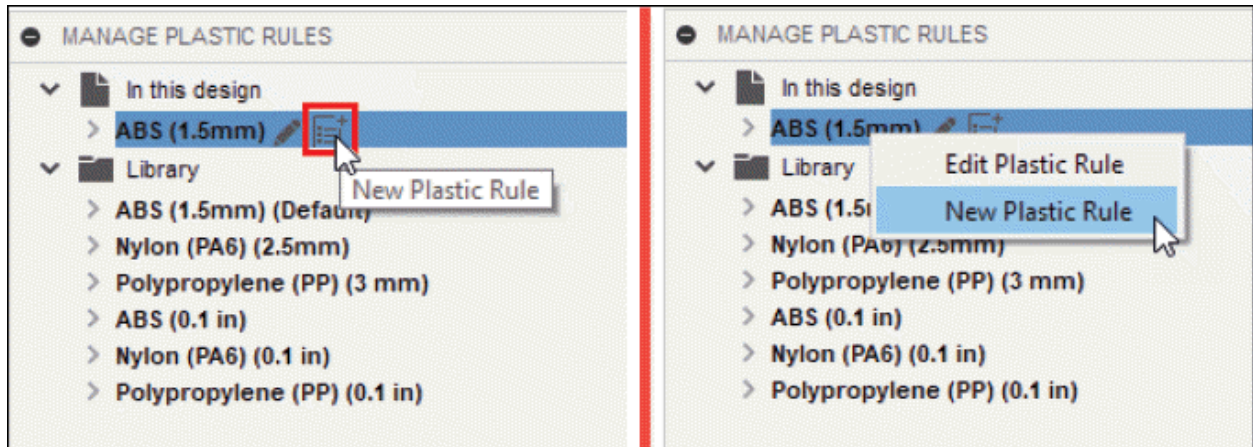


Figure-144. New Plastic Rule button

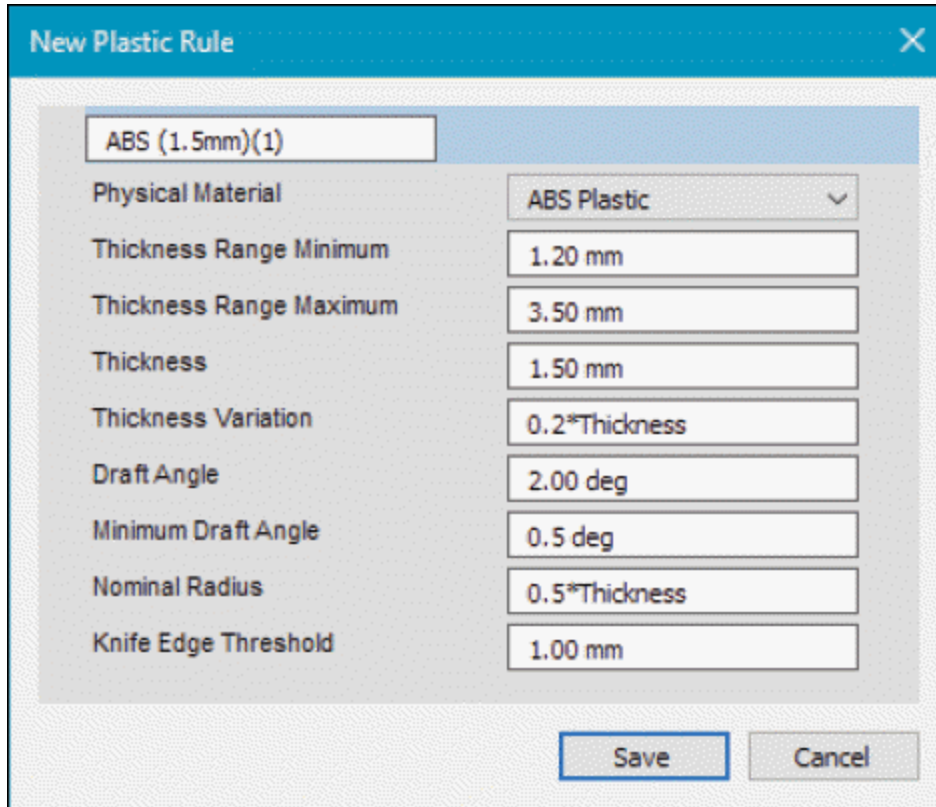


Figure-145. New Plastic Rule dialog box

- Specify desired parameters of the rule in the dialog box and click on the **Save** button. The new plastic rule will be created.

Snap Fit

The **Snap Fit** tool is used to create a cantilever snap fit feature to fasten two solid bodies together in a design. The procedure to use this tool is discussed next.

- Click on the **Snap Fit** tool from **CREATE** panel in the **PLASTIC** tab of the **Toolbar**; refer to [Figure-146](#). The **SNAP FIT** dialog box will be displayed as shown in [Figure-147](#).

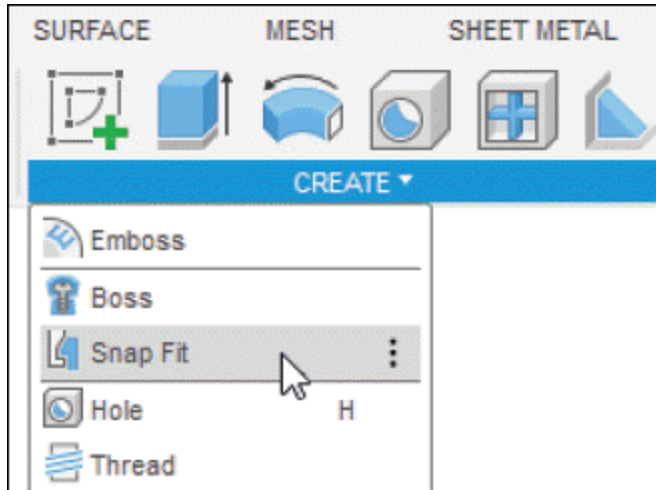


Figure-146. Snap Fit tool

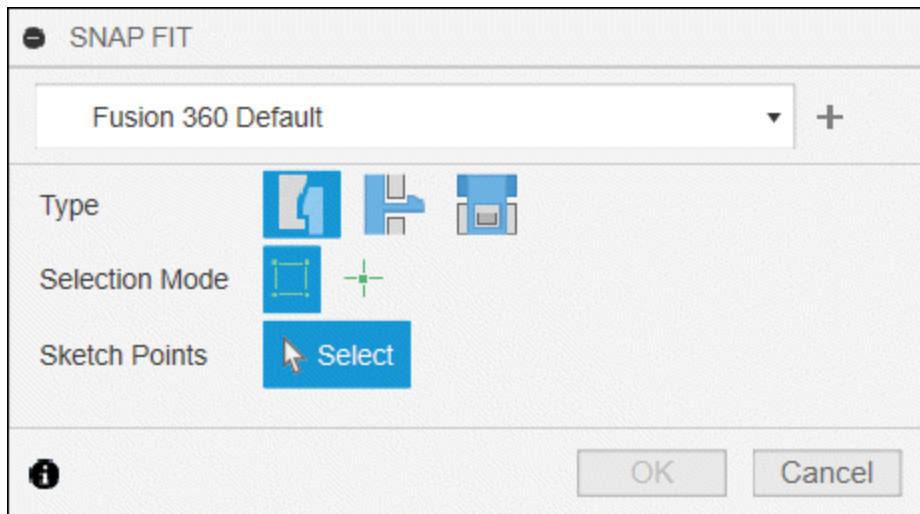


Figure-147. SNAP FIT dialog box

- Select desired option from **Type** section in the dialog box to create the snap fit feature. The **Parallel Hook And Groove** option is selected by default which is used to create a snap fit feature with a hook and groove, parallel to a face on a pair of solid bodies. Select **Perpendicular Hook And Groove** option to create a snap fit feature with a hook and groove, perpendicular to a face on a pair of

solid bodies. Select **Hook And Loop** option to create a snap fit feature with a hook and loop on a pair of solid bodies.

- Select the **Automatic** button from the **Selection Mode** section to automatically select all visible sketch points based on reference points. Select the **Manual** button from the section to select each sketch point for snap fit manually.
- The **Select** button of **Sketch Points** section is active by default. Select the sketch point to position the hook side of each snap fit feature; refer to [Figure-148](#). The preview of snap fit along with updated **SNAP FIT** dialog box will be displayed; refer to [Figure-149](#).

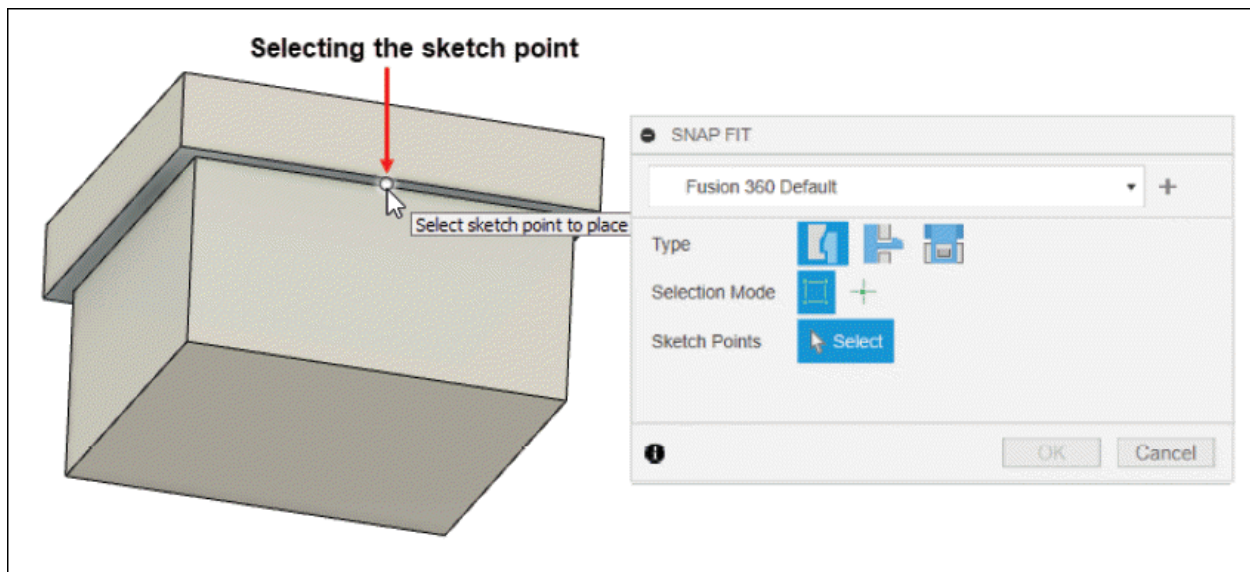


Figure-148. Selecting sketch point for snap fit

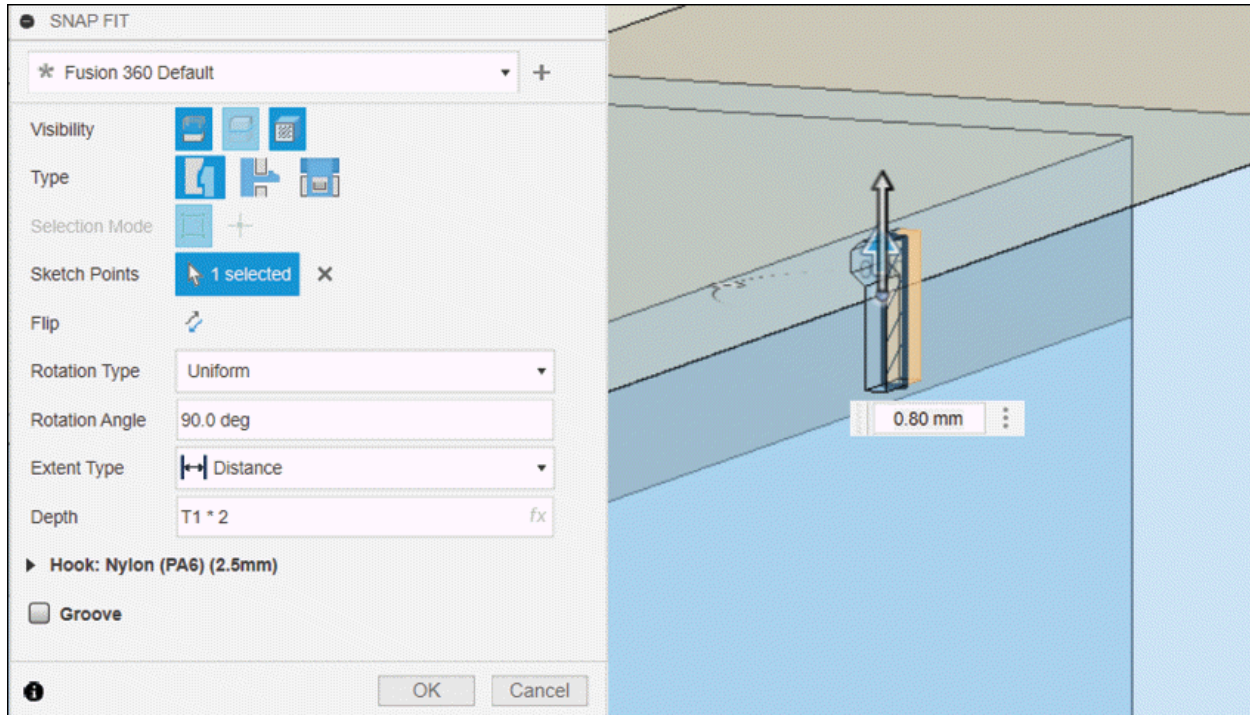


Figure-149. Updated SNAP FIT dialog box

- Select **Flip** button to flip the snap fit feature over the selected sketch point.
- The target body will be selected automatically in the **Body** section on which the snap fit feature will be placed.
- Select desired option from the **Rotation Type** drop-down to define how rotation value will be specified. For example, select the **Aligned** option from drop-down to align the face of snap fit with selected face of the model or select the **Independent** option to specify angle value independent of any model reference.
- Specify desired value in the **Rotation Angle** edit box to rotate the hook around the sketch point or drag the **Rotation manipulator handle**.

- Select **Distance** option from **Extent Type** drop-down to extrude the bottom of the hook from the sketch plane to a specified depth in the **Depth** edit box. Select **To Next** option to extrude the bottom of the hook from the sketch plane to the nearest faces on a solid body.
- Expand the **Hook** node and specify desired values in the edit boxes to adjust the dimensions of the hook side of the snap fit feature.
- Select the **Groove** check box and specify desired values in the edit boxes to create the groove side of the snap fit feature.
- After specifying desired parameters in the dialog box, click on the **OK** button. The snap fit feature will be created; refer to [Figure-150](#).

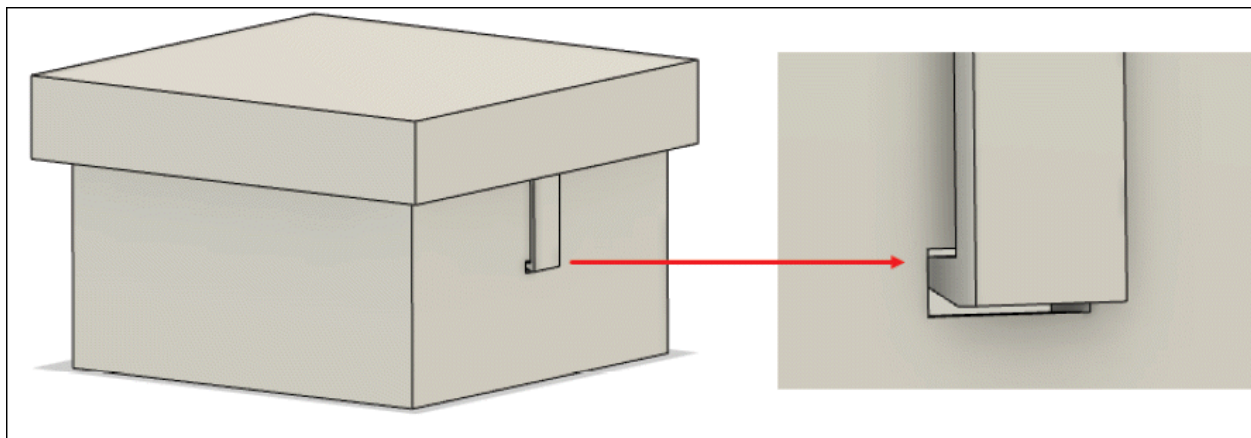


Figure-150. Snap fit feature created

Boss

The **Boss** tool is used to create a boss feature to fasten two solid bodies together and reinforce thin areas of a design.

The procedure to use this tool is discussed next.

- Click on the **Boss** tool from **CREATE** panel in the **PLASTIC** tab of the **Toolbar**; refer to [Figure-151](#). If your model is not saved yet then the **Cannot create fastener** pop-up window will be displayed asking you to save the design to create fastener or continue without creating the fastener; refer to [Figure-152](#).

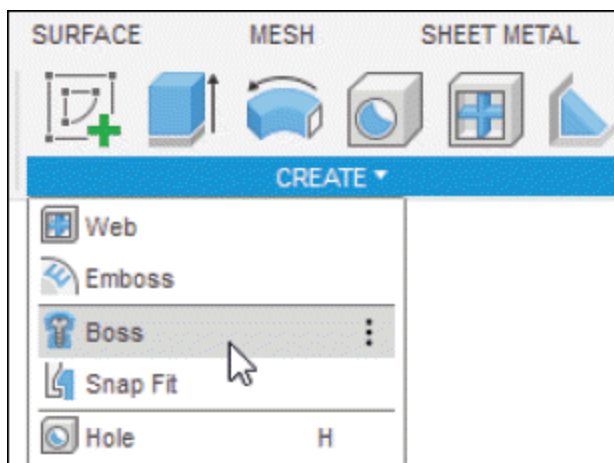


Figure-151. Boss tool

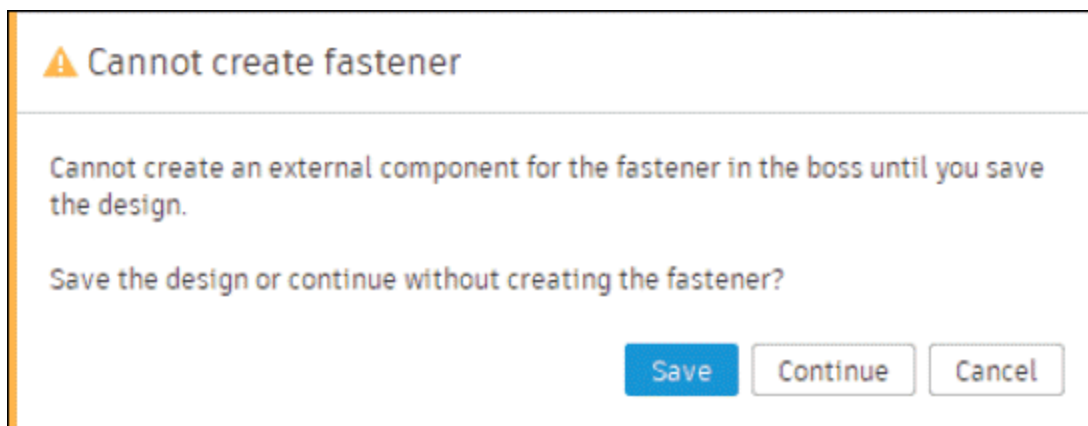


Figure-152. Cannot create fastener pop up window

- Click on the **Save** button to save the design. The **Save As** dialog box will be displayed.

- Specify desired name and location of the design and click on the **Save** button. The design will be saved.
- Click on the **Continue** button to create the boss feature without creating the fastener. The **BOSS** dialog box will be displayed; refer to [Figure-153](#).

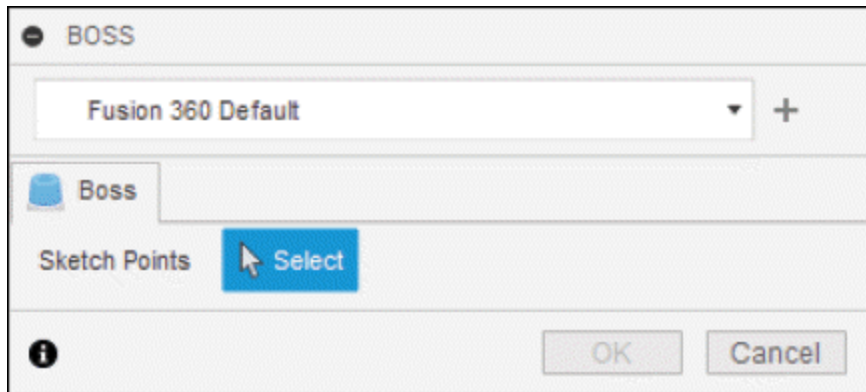


Figure-153. BOSS dialog box

- The **Select** button of **Sketch Points** section is active by default. Select desired sketch point to place the boss; refer to [Figure-154](#). The **No plastic rule assigned** pop-up window will be displayed along with the updated **BOSS** dialog box; refer to [Figure-155](#).

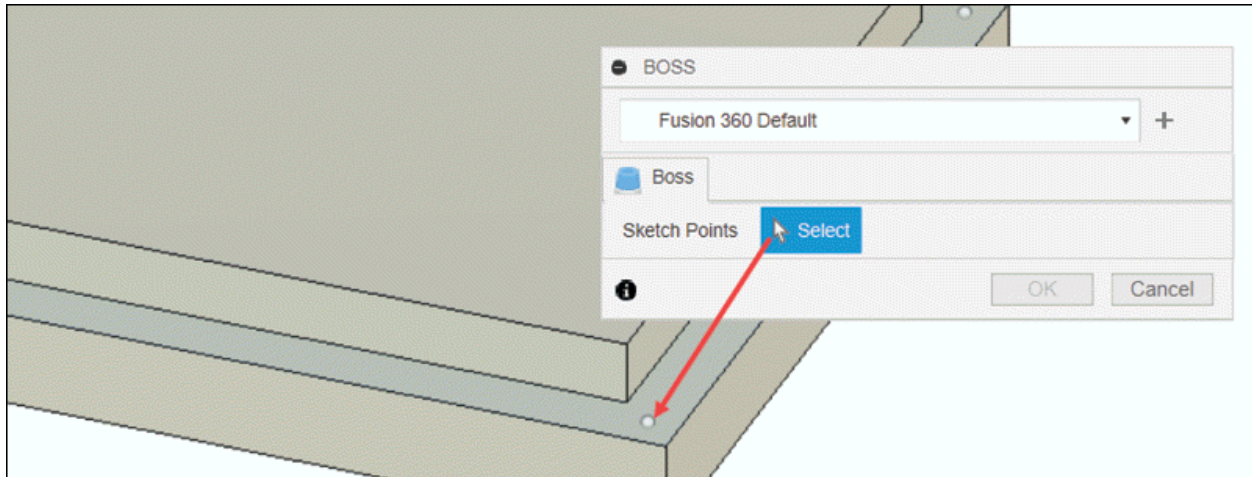


Figure-154. Selecting the sketch point

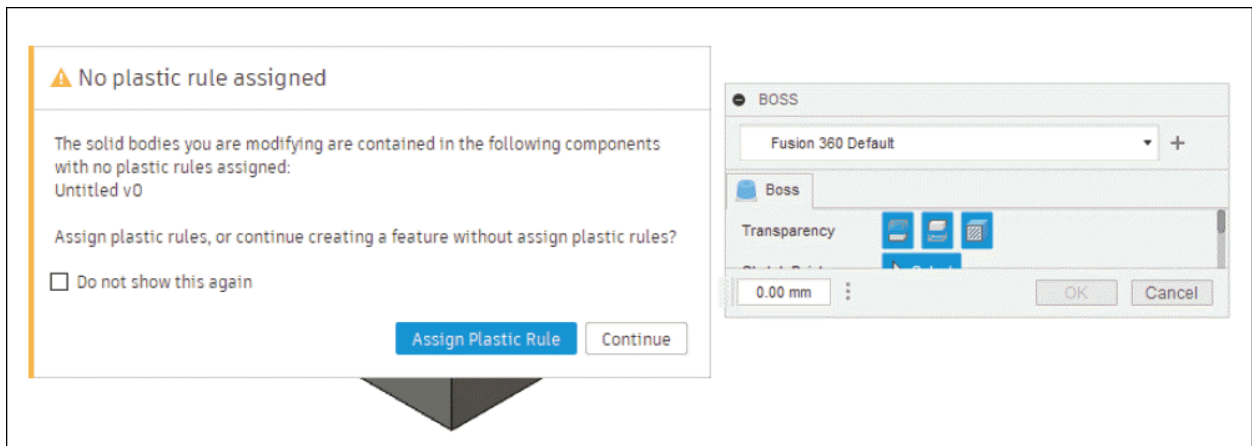


Figure-155. Updated BOSS dialog box

- Click on the **Assign Plastic Rule** button to assign the plastic rule or click on the **Continue** button from the pop-up window to create boss feature without assigning the plastic rule. The preview of boss feature will be displayed; refer to [Figure-156](#). Note that you may need to drag the offset position handle for generating preview of feature.

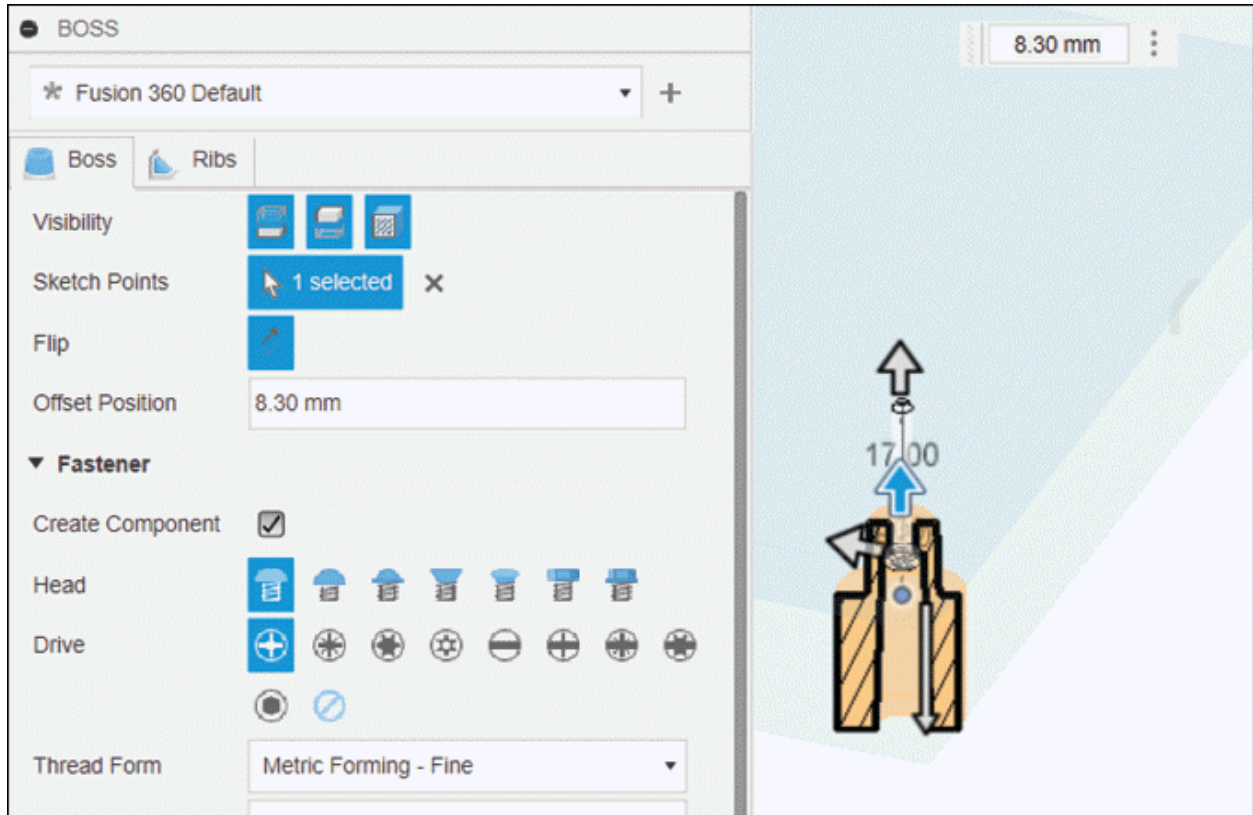


Figure-156. Preview of boss feature

- All buttons are selected by default in the **Visibility** section. The **Side 1 Transparency** button reduces the opacity of the Side 1 body while you edit the boss feature. The **Side 2 Transparency** button reduces the opacity of the Side 2 body while you edit the boss feature. The **Section Analysis** button cuts a section through the center of the boss feature while you edit.
- Select the **Flip** button to flip the boss feature over selected sketch point to switch Side 1 and Side 2 bodies.
- Specify desired distance value in the **Offset Position** edit box to offset the position of the boss feature from the sketch plane.

- Select desired option from **Head** section and **Drive** section of the **Fastener** node to create fastener of desired head type.
- Select desired thread form from **Thread Form** drop-down and thread angle from the **Thread Angle** drop-down to apply to the fastener.
- Select desired diameter from **Thread Diameter** drop-down to apply to the fastener.
- Select desired type of hole type and step from **Hole Type** section and **Step Type** section of **Side** node in the dialog box, respectively to create boss feature.
- Expand the **Advanced** node and specify desired values in the edit boxes to adjust the dimensions of the side of the boss feature.
- Click on the **Ribs** tab to create ribs on sides of boss feature. The related options will be displayed.
- Select the **Side 1** check box to activate rib feature on side 1 of boss feature; refer to [Figure-157](#). Set the parameters as discussed earlier for the rib.

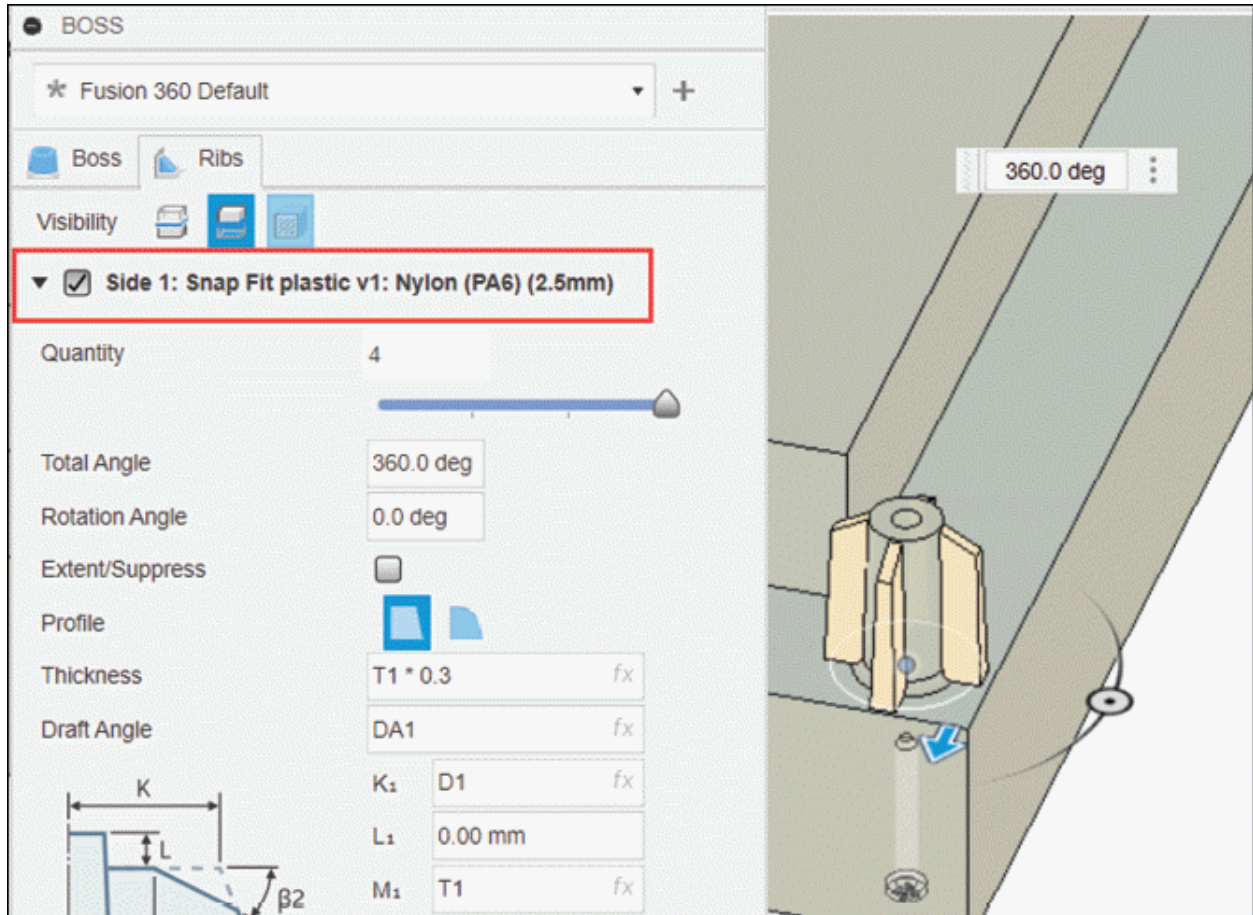


Figure-157. Rib feature for boss

- After specifying desired parameters in the dialog box, click on the **OK** button. The boss feature will be created; refer to [Figure-158](#).

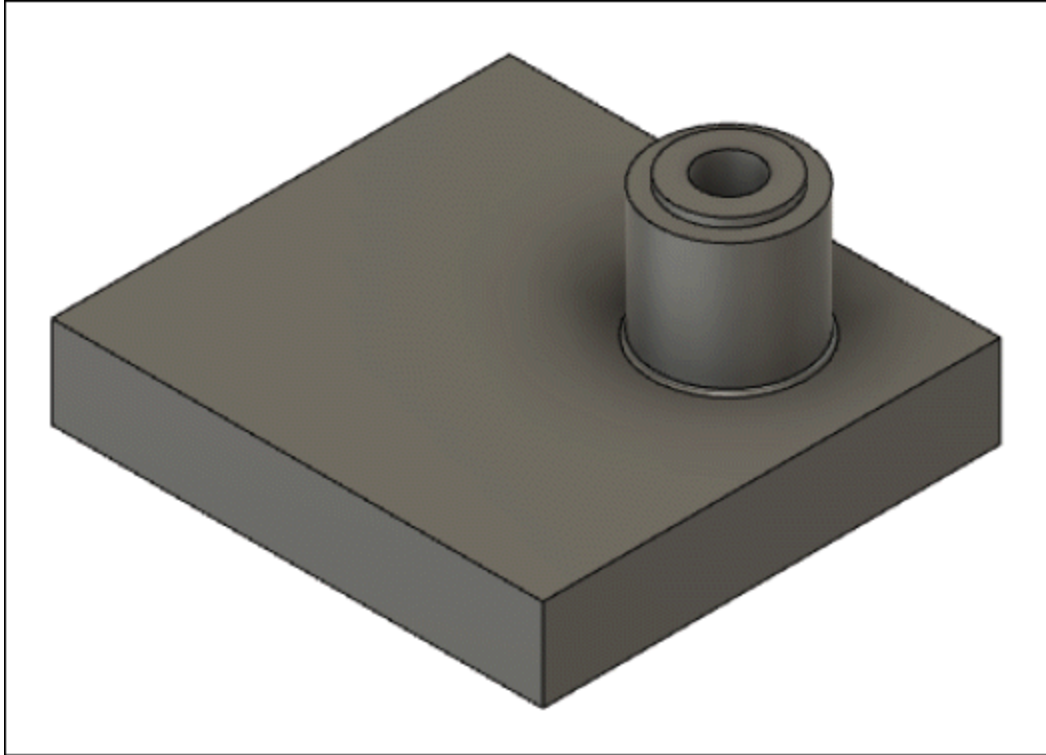


Figure-158. Boss feature created

Creating Rest Feature

The **Rest** tool is used to create a flat rest area in the model based on a closed loop sketch. The procedure to use this tool is given next.

- Click on the **Rest** tool from the **CREATE** drop-down of **PLASTIC** tab in **Toolbar**. The **REST** dialog box will be displayed and you will be asked to select a profile; refer to [Figure-159](#).
- Select the sketch profile from the model. Preview of rest feature will be displayed and options will be displayed as shown in [Figure-160](#).

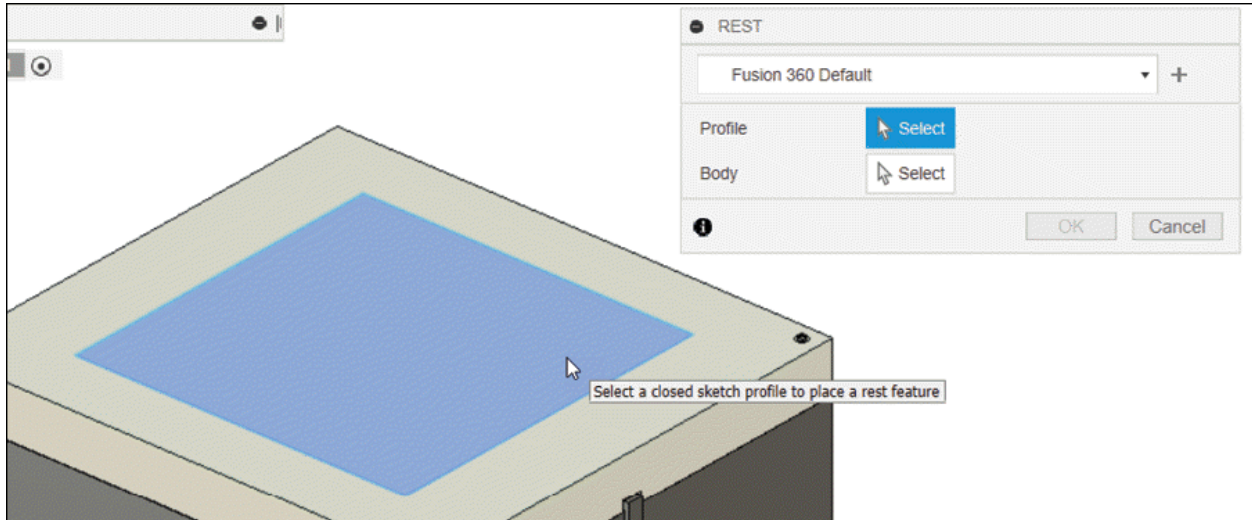


Figure-159. REST dialog box

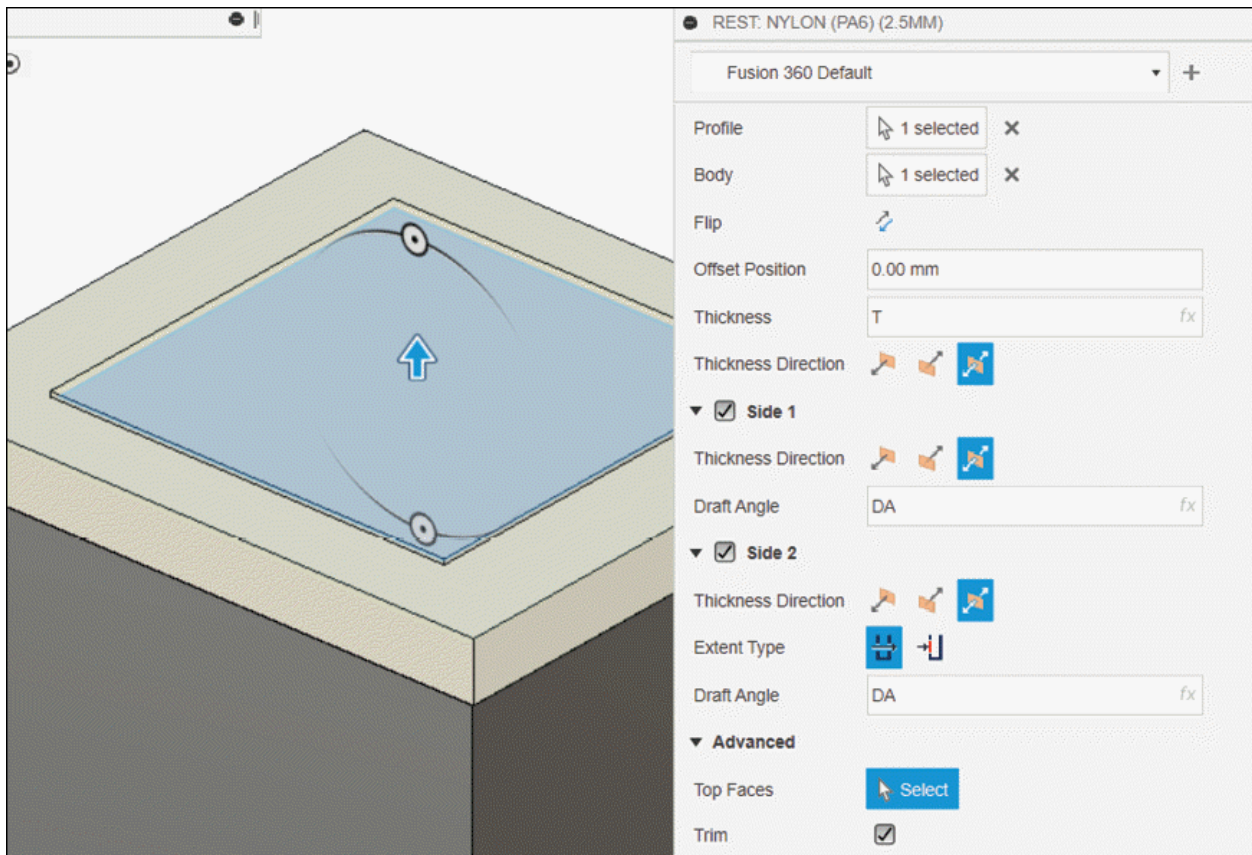


Figure-160. Preview of rest feature

- Set desired parameters in the dialog box and click on the **OK** button.

SELF ASSESSMENT

Q1. Which of the following is not the default plane available in Autodesk Fusion 360?

- a. Front Plane
- b. Right Plane
- c. Top Plane
- d. Left Plane

Q2. Discuss the procedure of creating multiple holes with example.

Q3. Which of the following tool is not available in **CREATE** drop-down of Autodesk Fusion 360?

- a. Thread
- b. Box
- C. Torus
- d. Curve

Q4. **Thicken** tool is used to add thickness and length. (T/F)

Q5. The **Pattern On Path** tool is available in the **CREATE** drop-down. (T/F)

Q6. Discuss the use of **Pipe** tool with examples.

Q7. Discuss the use of **Web** tool with examples.

Q8. Which tool is used to create a solid body connecting two sketch from different planes?

Q9. Discuss the use of **Pattern On Path** tool with examples.

Q10. The tool is used to cut helical thread on cylindrical faces.

Q11. Why plastic rules are assigned to the component? And which tool is used to assign the plastic rule to the component?

Q12. The tool is used to create new rules, edit existing rules, or save rules to a library to use them in multiple designs.

Q13. The **Boss** tool is used to create a boss feature to fasten two solid bodies together and reinforce thin areas of a design. (T/F)

Chapter 5
Practical and Practice

Topics Covered

The major topics covered in this chapter are:

- ***Practical***
- ***Practice***

PRACTICAL 1

Create the model as shown in [Figure-1](#). The views of the model with dimension are given in [Figure-2](#).

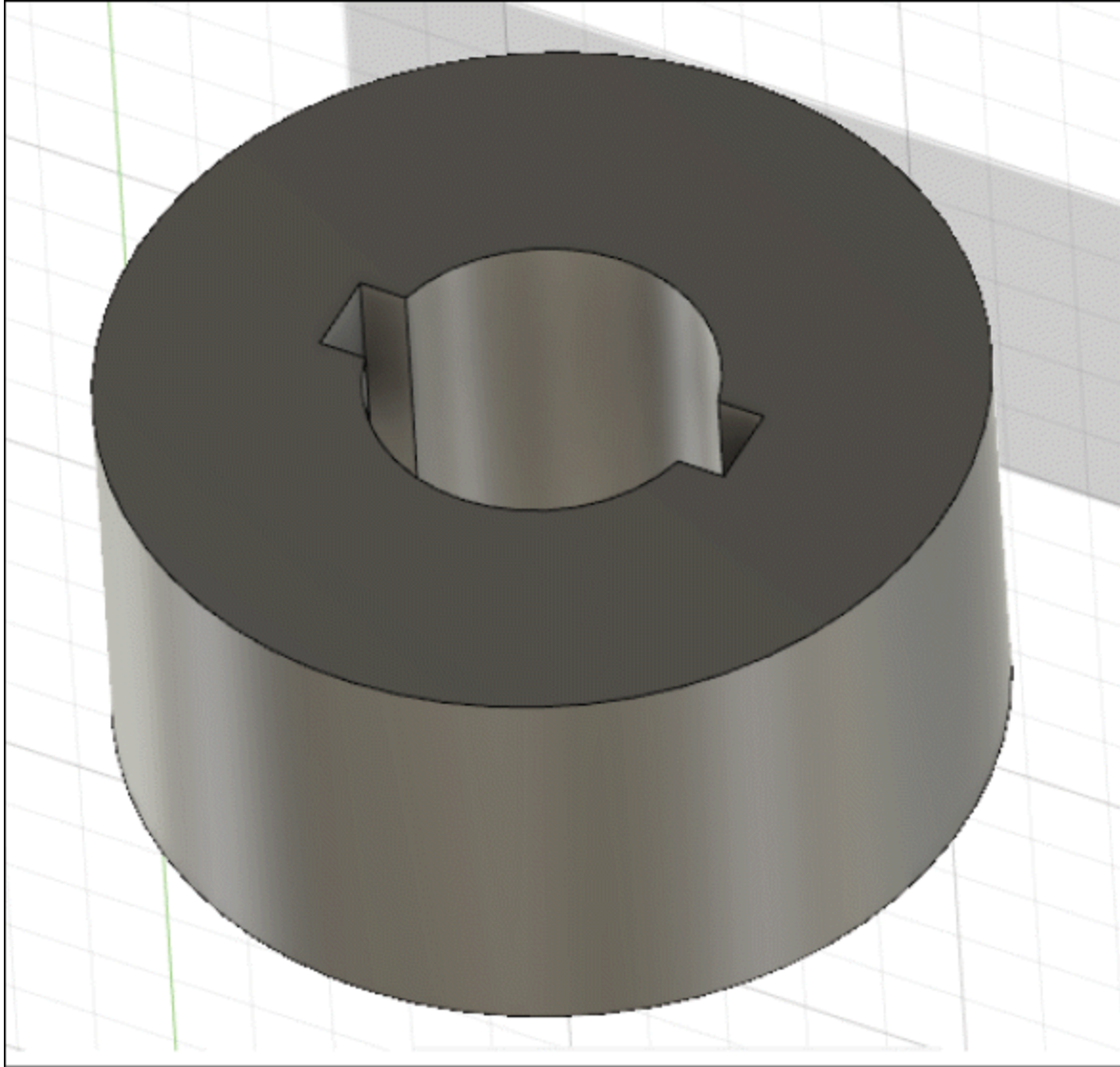


Figure-1. Model for Practical 1

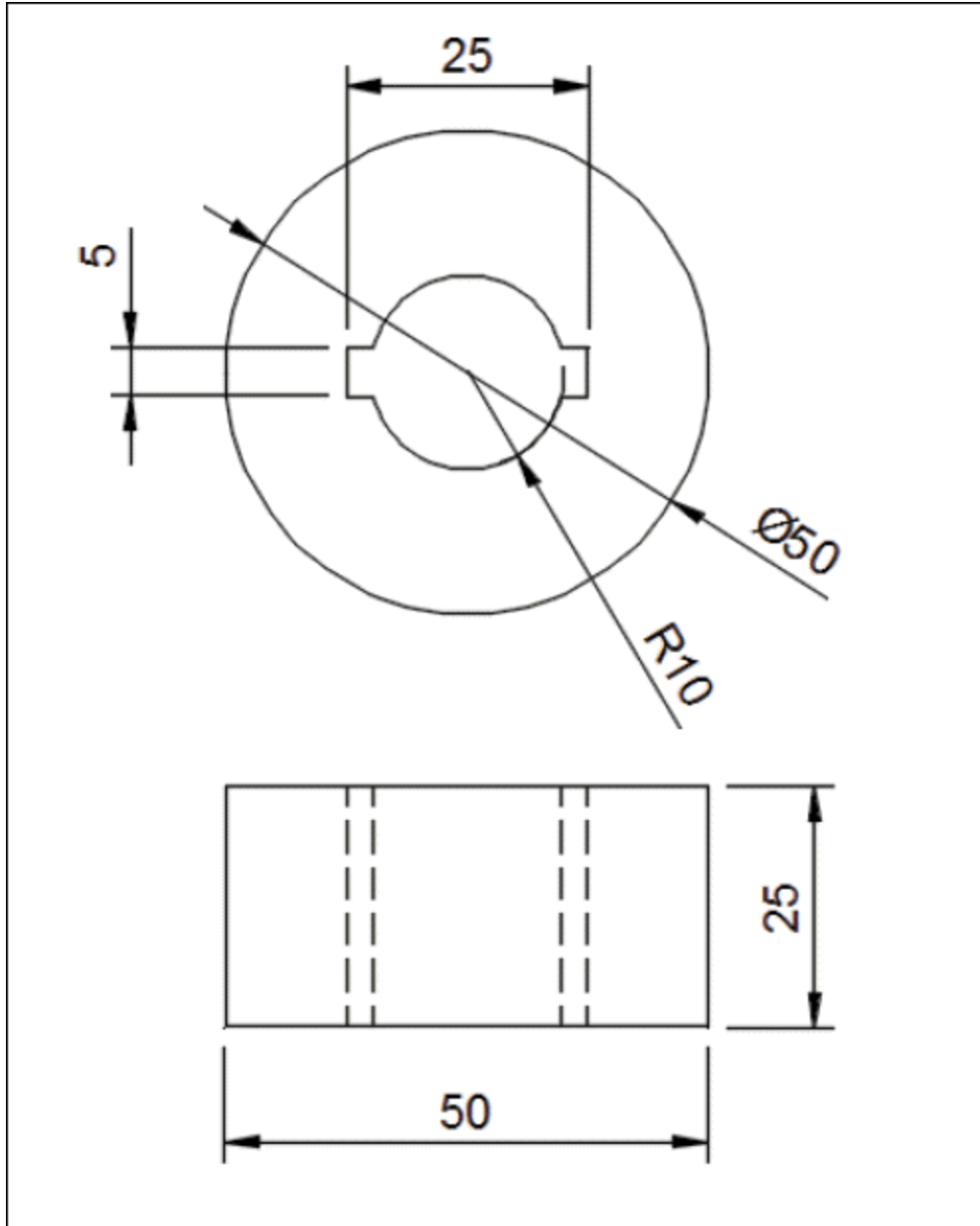


Figure-2. Views of Practical 1

Before we start working on Practical 1, it is important to understand two terms; first angle projection and third angle projection. These are the standards of placing views in the engineering drawings. The views placed in the above figure are using third angle

projection. In first angle projection, the top view of model is placed below the front view and right side view is placed at the left of the front view. You will learn more about projection in chapter related to drafting.

Creating the Sketch

- Click on **Create Sketch** tool from **CREATE** drop-down in **Toolbar**. You will be asked to select a plane for creating sketch.
- Select **Top Plane** from the canvas screen.
- Click on **Center Diameter Circle** tool of **Circle** cascading menu from **CREATE** drop-down in **Toolbar**. You will be asked to place the center point of circle.
- Click at the **Origin** to place the center point of circle. Enter **50** in floating window as diameter of circle and press **ENTER** key.
- Sketch another circle of radius **10** (diameter **20**) taking center point as origin.

How to show radius in place of diameter of circle? Create the circle with diameter value. Right-click on diameter dimension and select **Toggle Radius** button from Marking menu; refer to [Figure-3](#).

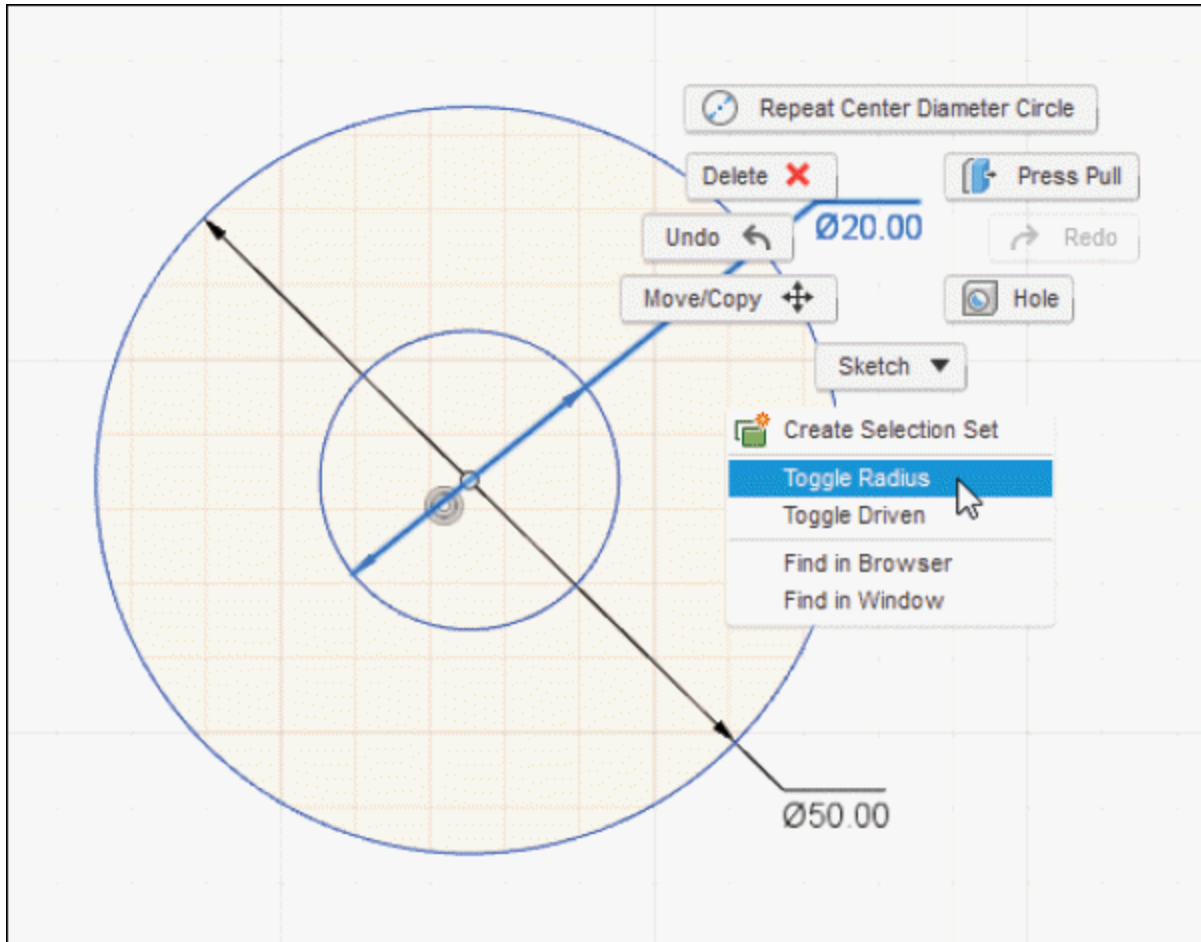


Figure-3. Toggle Radius Option

- Select **Center Rectangle** tool of **Rectangle** cascading menu from **CREATE** drop-down in **Toolbar**.
- Click at **Origin** as center point for rectangle and enter **5** as width & **25** as length in floating window to create a rectangle; refer to [Figure-4](#).

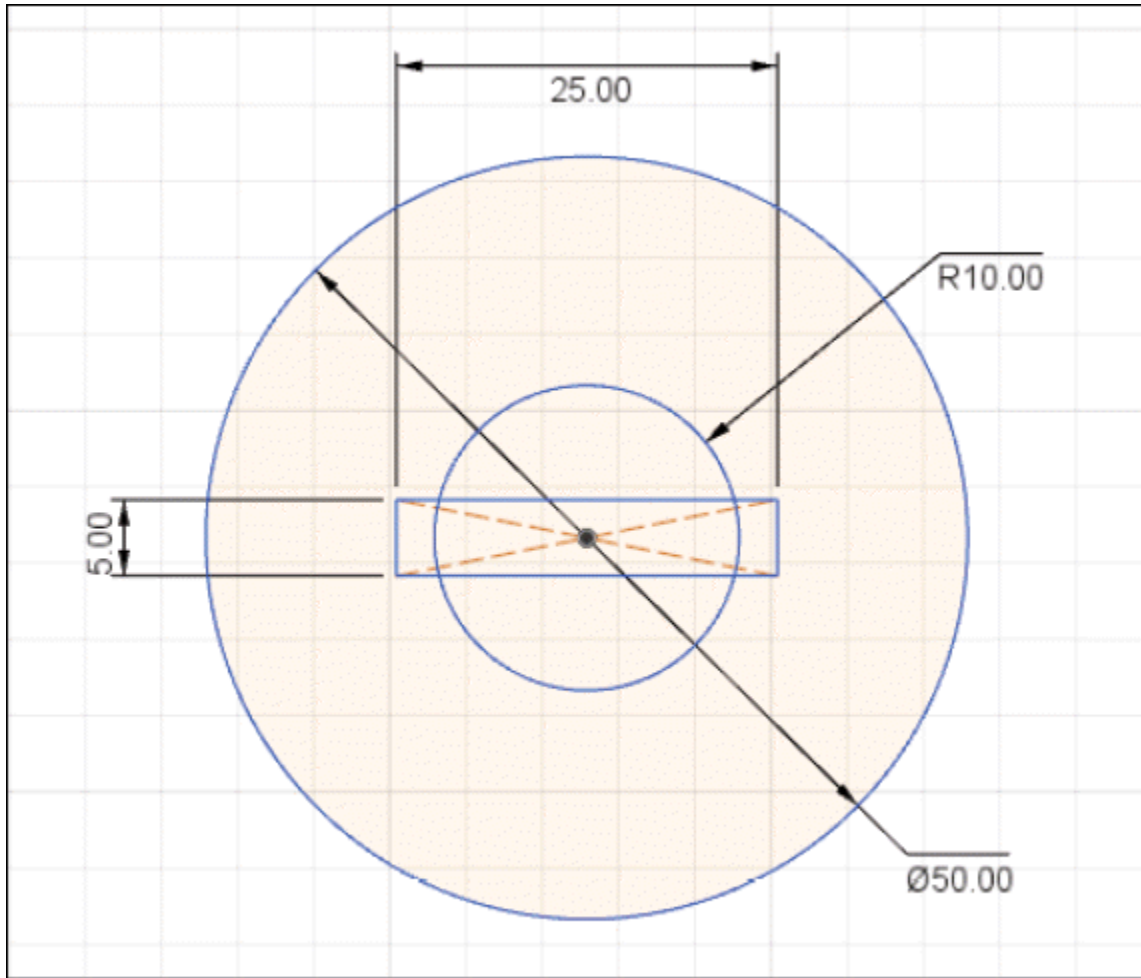


Figure-4. Sketch after creating circle and rectangle

- Select **Trim** tool of **MODIFY** drop-down from **Toolbar** and trim the entities in such a way that the sketch is displayed as shown in [Figure-5](#).

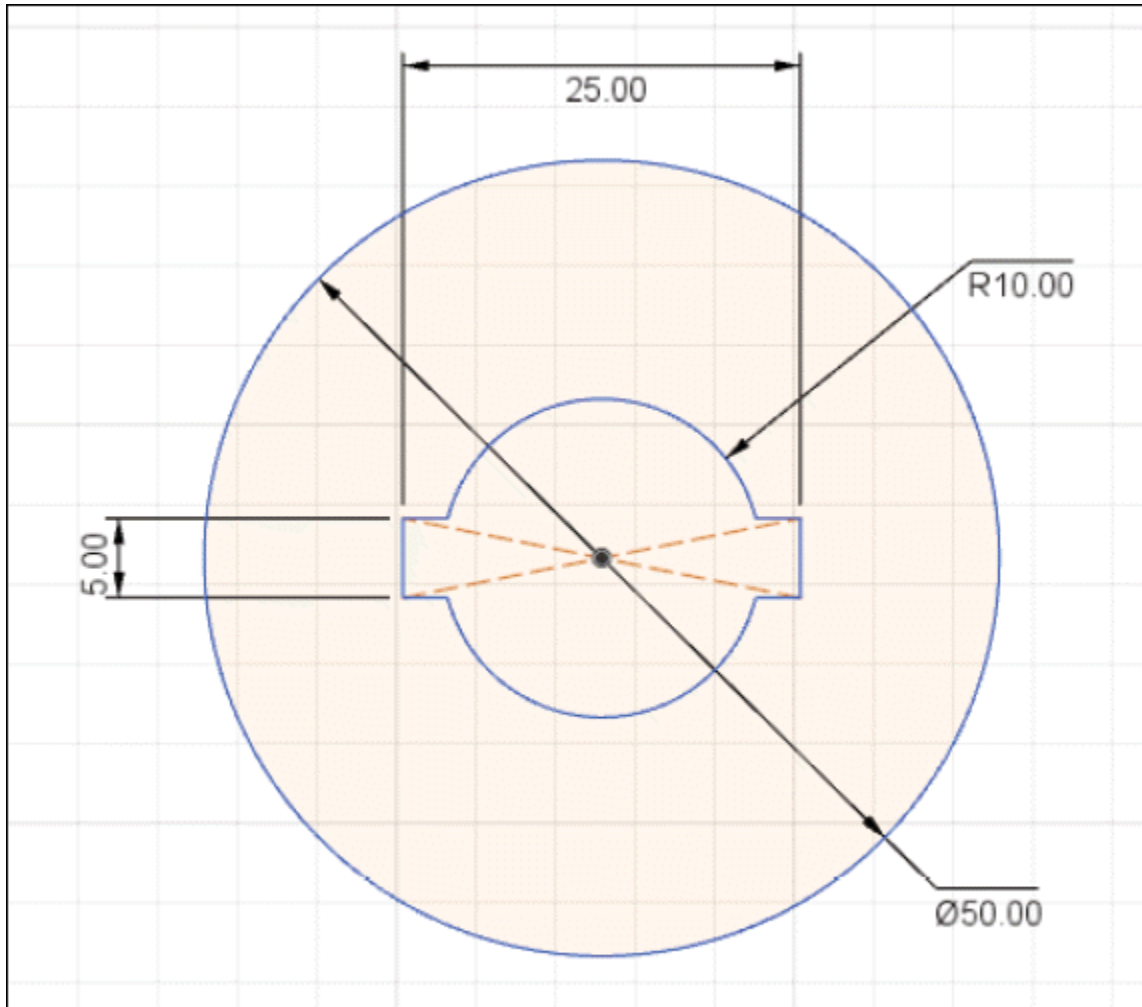


Figure-5. Sketch after trimming

- Click on **Finish Sketch** button from **Toolbar** to exit the sketch mode.

Creating Extrude Feature

- Click on **Extrude** tool of **CREATE** drop-down from **Toolbar** or press **E** key from keyboard to select **Extrude** tool. The **EXTRUDE** dialog box will be displayed.
- Profile selection is active by default. Select the recently created sketch profile. The updated **EXTRUDE** dialog box

will be displayed.

- Specify the height of extrusion as **25** in **Distance** edit box or floating window. The preview of extrusion will be displayed; refer to [Figure-6](#). (You need to rotate the model by using **SHIFT+Middle Mouse Button** to check the preview.)

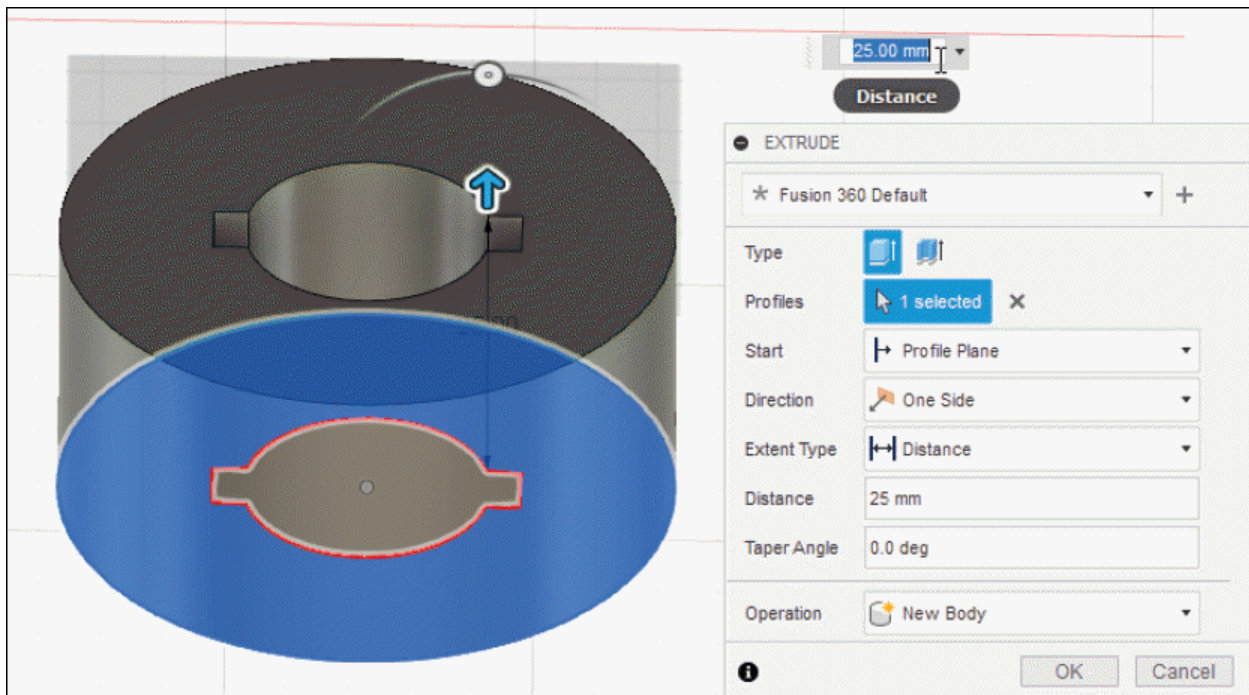


Figure-6. Preview of extrusion

- Click **OK** button from **EXTRUDE** dialog box to complete the process of extrusion. The model will be displayed as shown in [Figure-7](#).
- Save the file and close it.

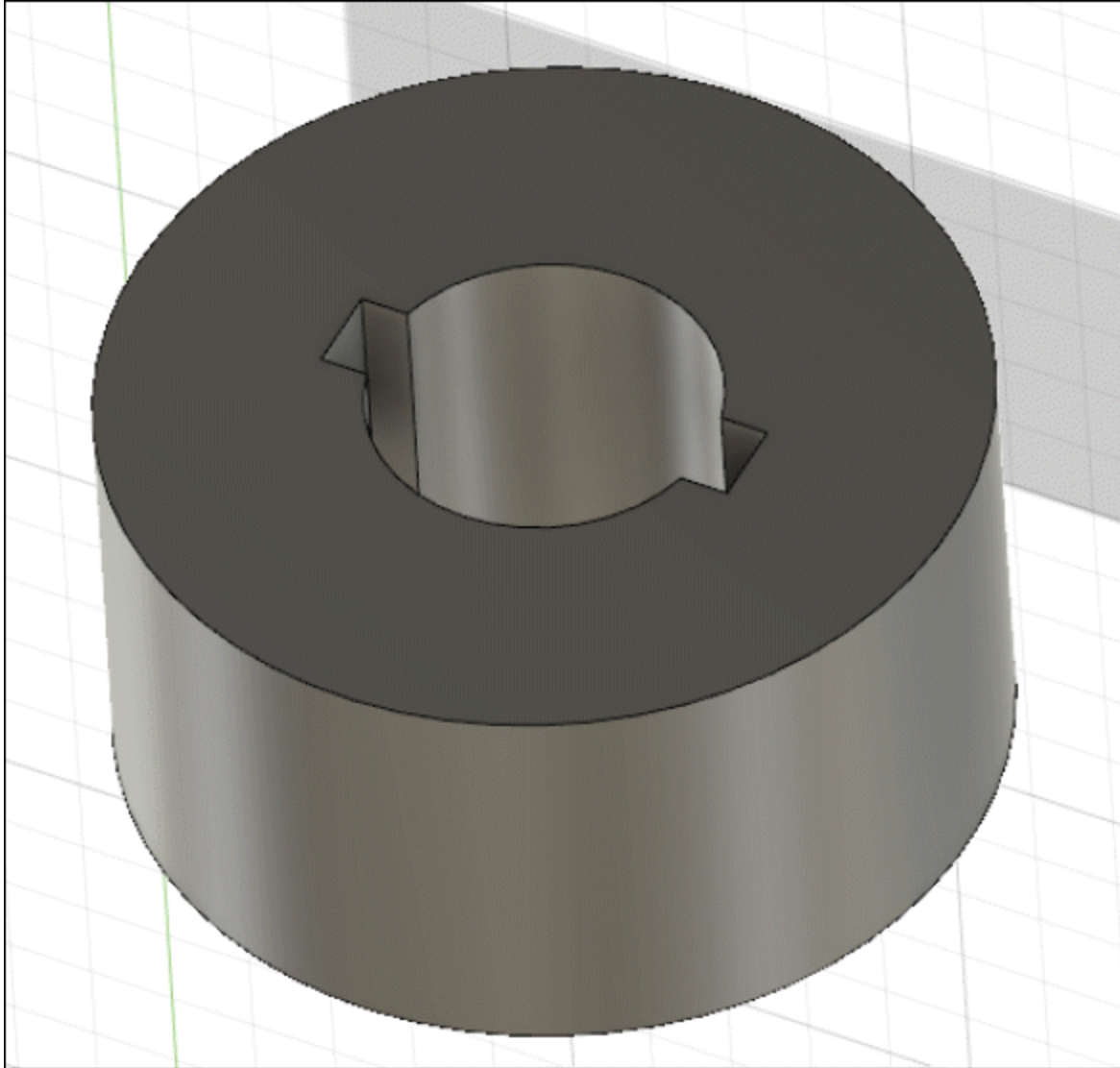


Figure-7. Final model of practical 1

PRACTICAL 2

Create the model as shown in [Figure-8](#). The dimensions are given in [Figure-9](#). Assume the missing dimensions.

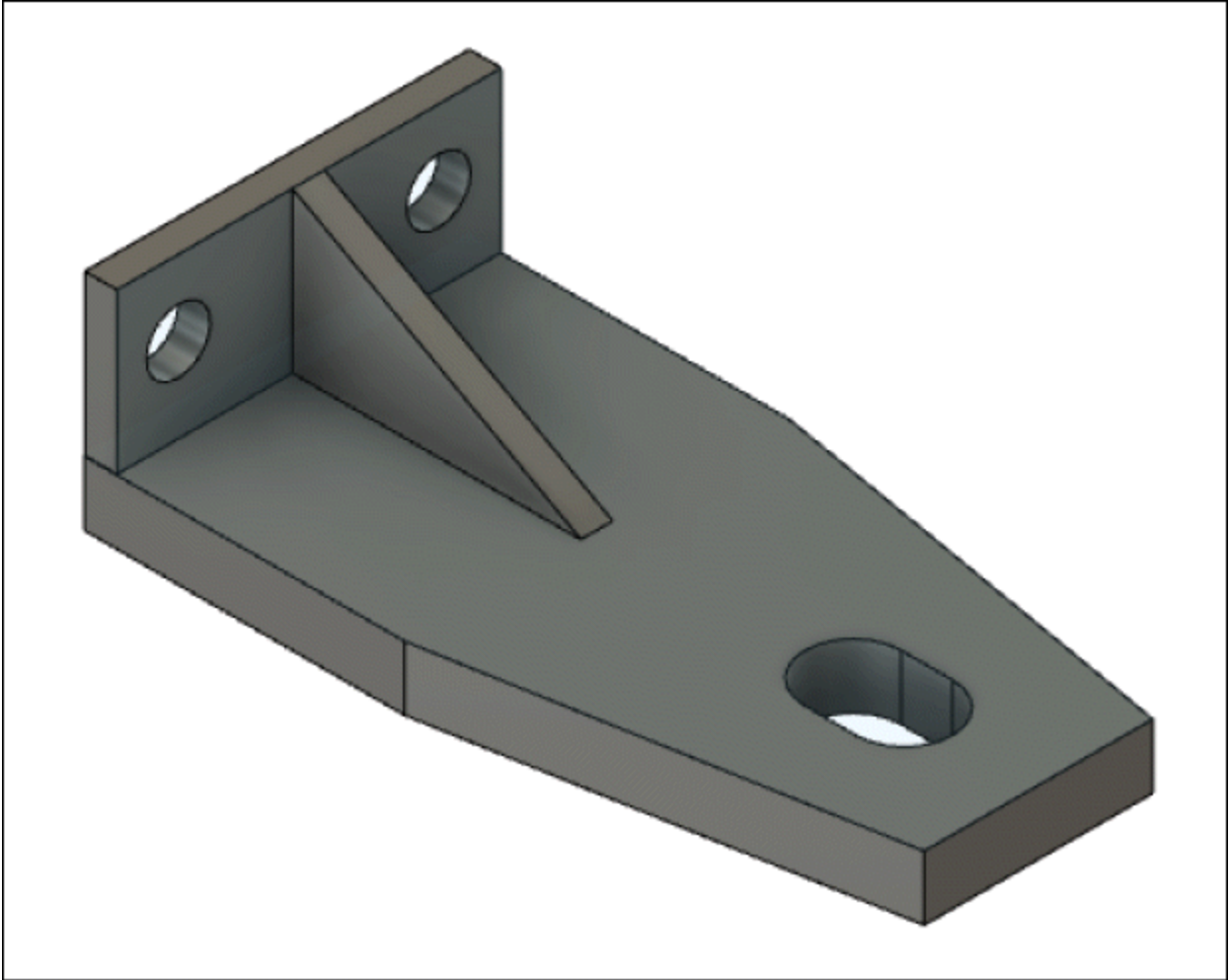


Figure-8. Model for Practical 2

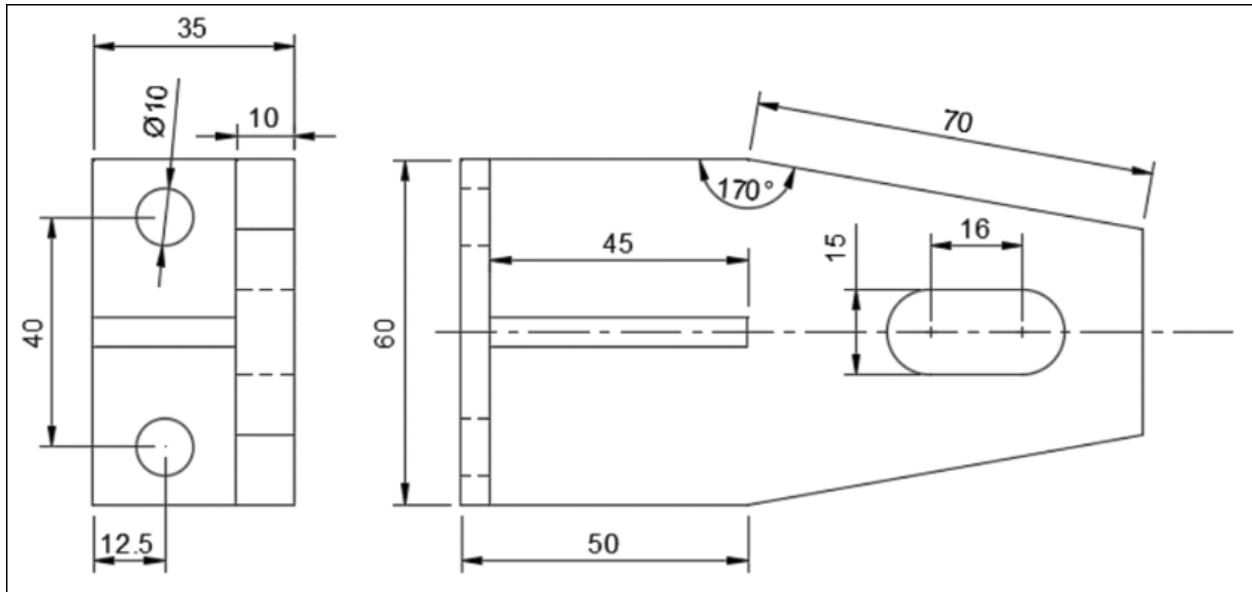


Figure-9. Practical 2 drawing

Creating sketch for model

- Start a new sketch using **Top Plane**.
- Select **Centre Rectangle** tool of **Rectangle** cascading menu from **CREATE** drop-down.
- Click on the origin to specify the centre point of rectangle and specify **60** and **50** in floating window as dimensions of rectangle; refer to [Figure-10](#). Press **TAB** to toggle between edit boxes.

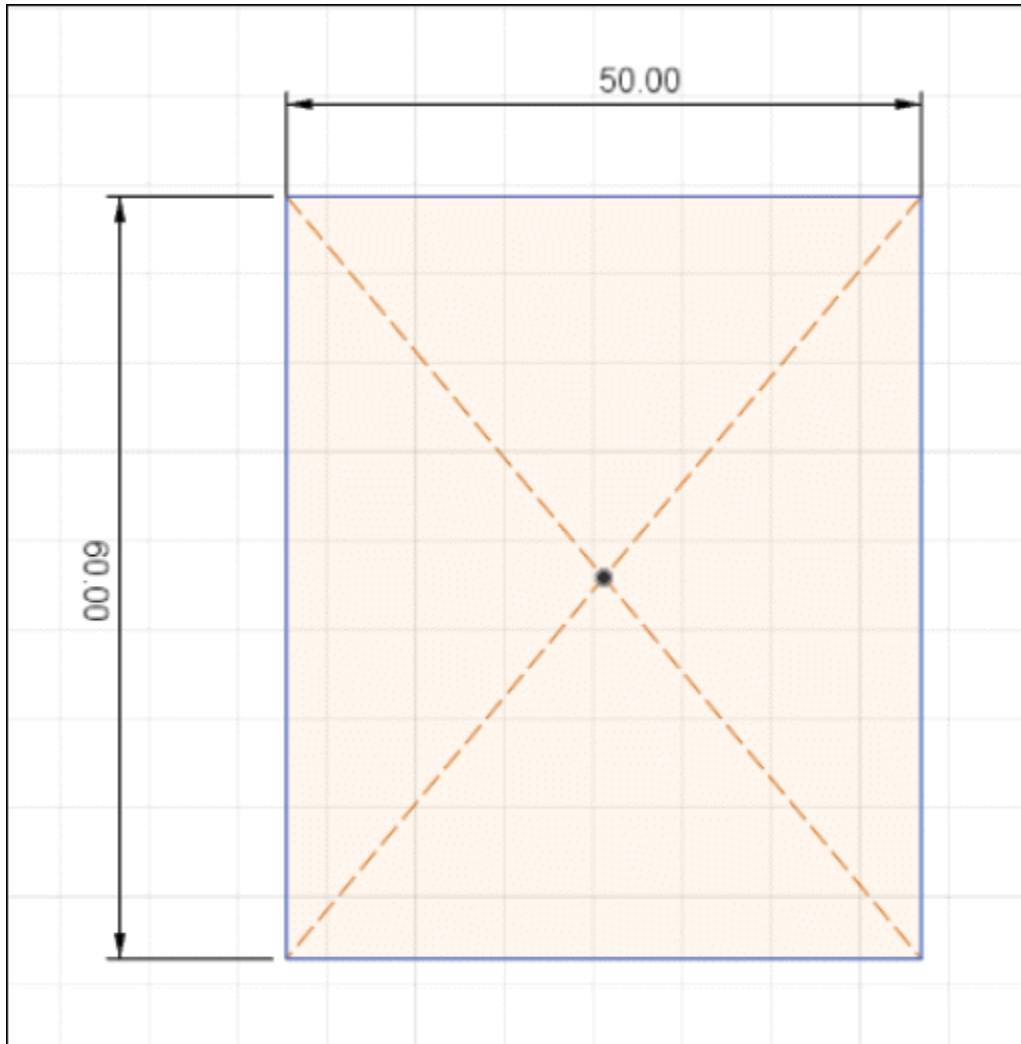


Figure-10. Sketch After Creating Rectangle

- Select **Line** tool from **CREATE** drop-down. Create a line of length as **70** and angle of **80** degree as shown in [Figure-11](#).

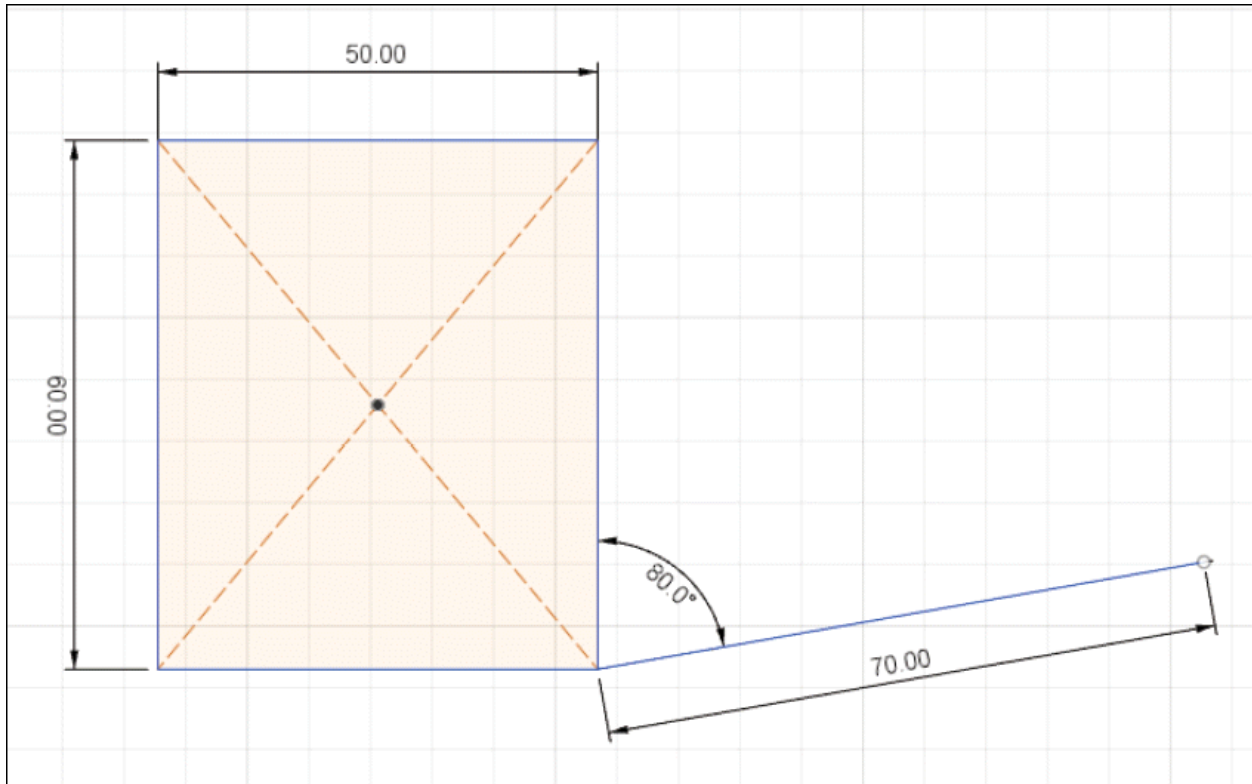


Figure-11. Creating Angular Line

- Select **Mirror** tool of **CREATE** drop-down from **Toolbar** and create a mirror copy of recently created line about centerline as shown in [Figure-12](#).

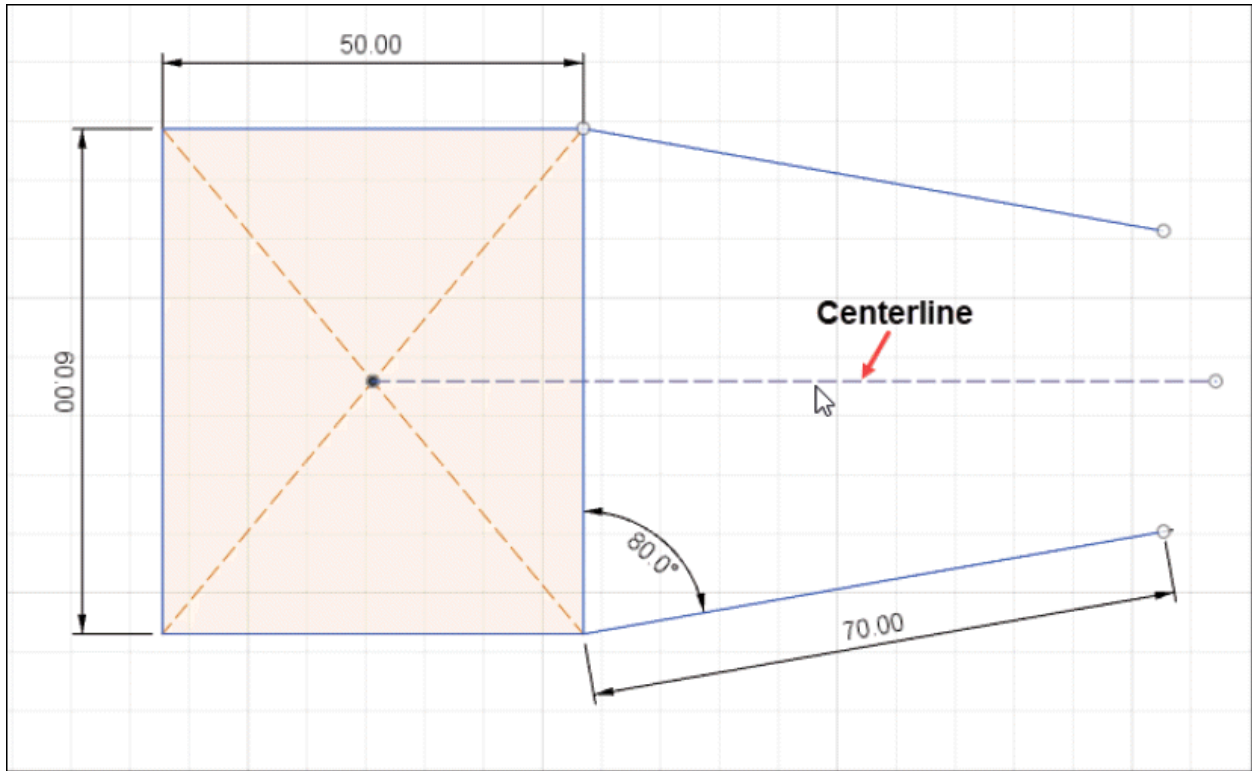


Figure-12. Creating Mirror Copy

- Create a line joining end points of two earlier created lines as shown in [Figure-13](#).

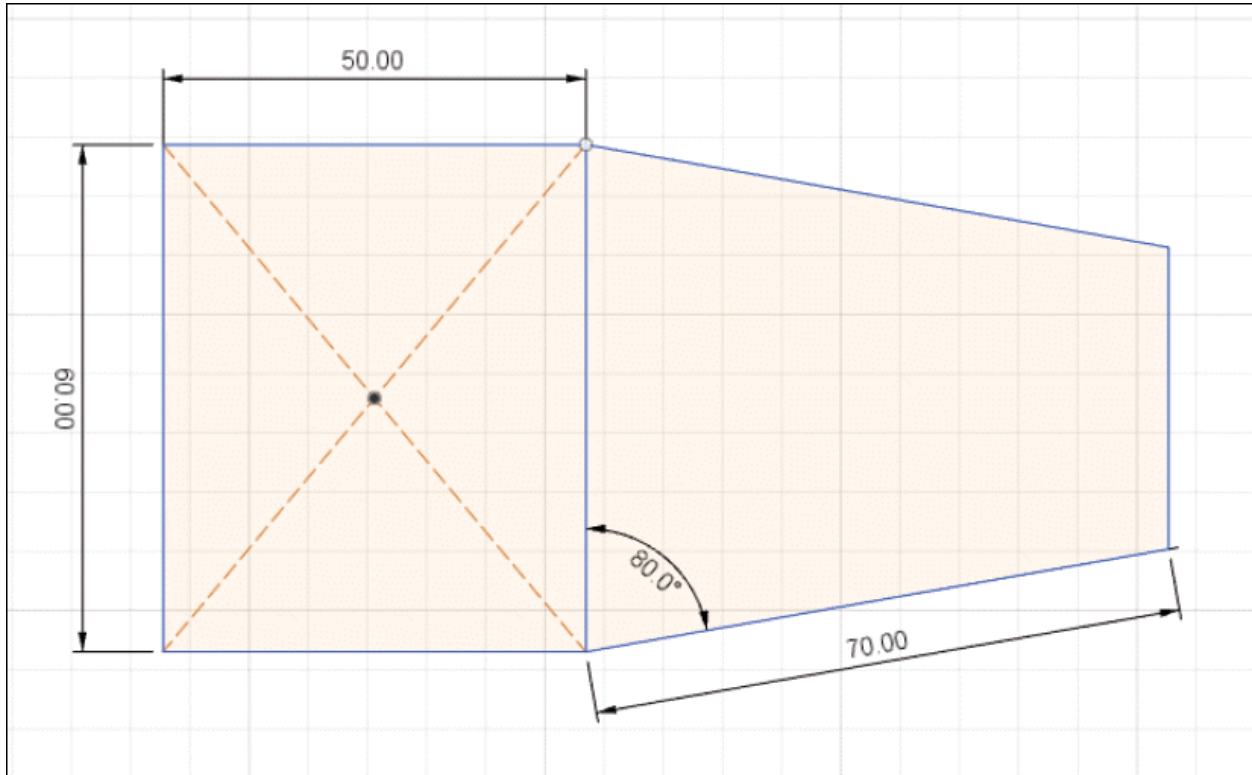


Figure-13. Joining End Points Of Lines

- Select **Center Point Slot** tool of **Slot** cascading menu from **CREATE** drop-down and create a slot of diameter as **15** and length as **8** at a distance of **65** from origin on the centerline; refer to [Figure-14](#).

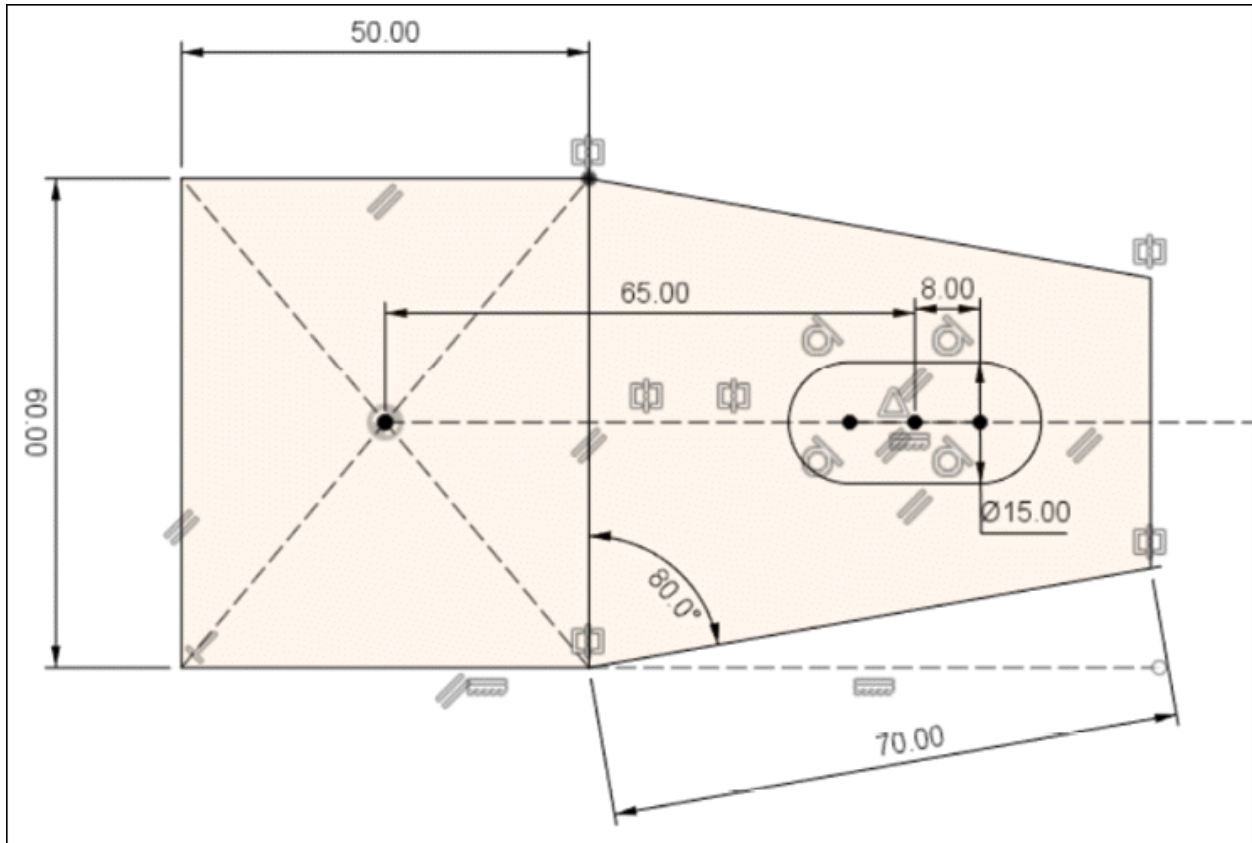


Figure-14. Creating Slot

- Press **T** from keyboard or click on the **Trim** tool from **MODIFY** drop-down in **Toolbar** to activate trimming tool. Select the inner lines of rectangle to remove them; refer to [Figure-15](#).

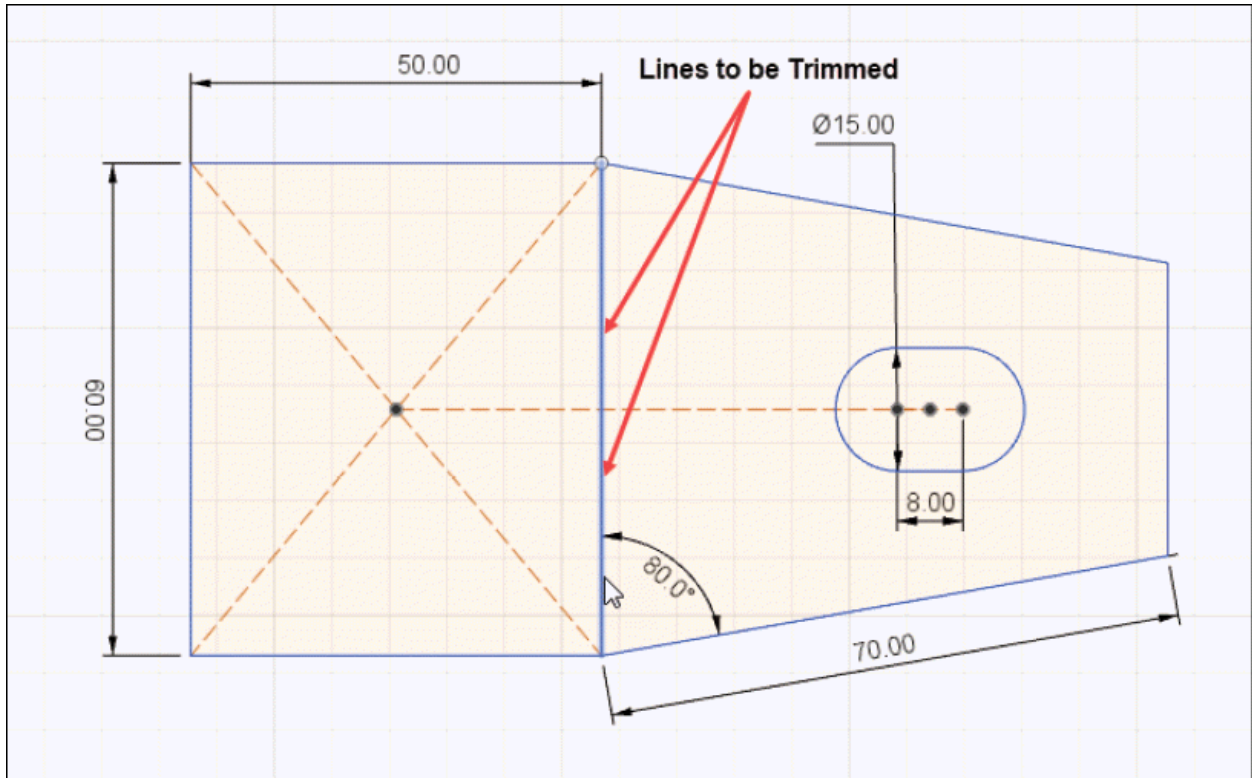


Figure-15. Lines To Be Trimmed

- Click on the **Finish Sketch** button to exit sketch.

Creating Extrude Feature

- Select **Extrude** tool from **CREATE** drop-down or press **E** key. The **EXTRUDE** dialog box will be displayed.
- Profile selection is active by default. Click in the sketch outside the slot.
- Specify **10** as height of extrusion in **Distance** edit box of **EXTRUDE** dialog box; refer to [Figure-16](#). (Rotate the model to get clear view.)

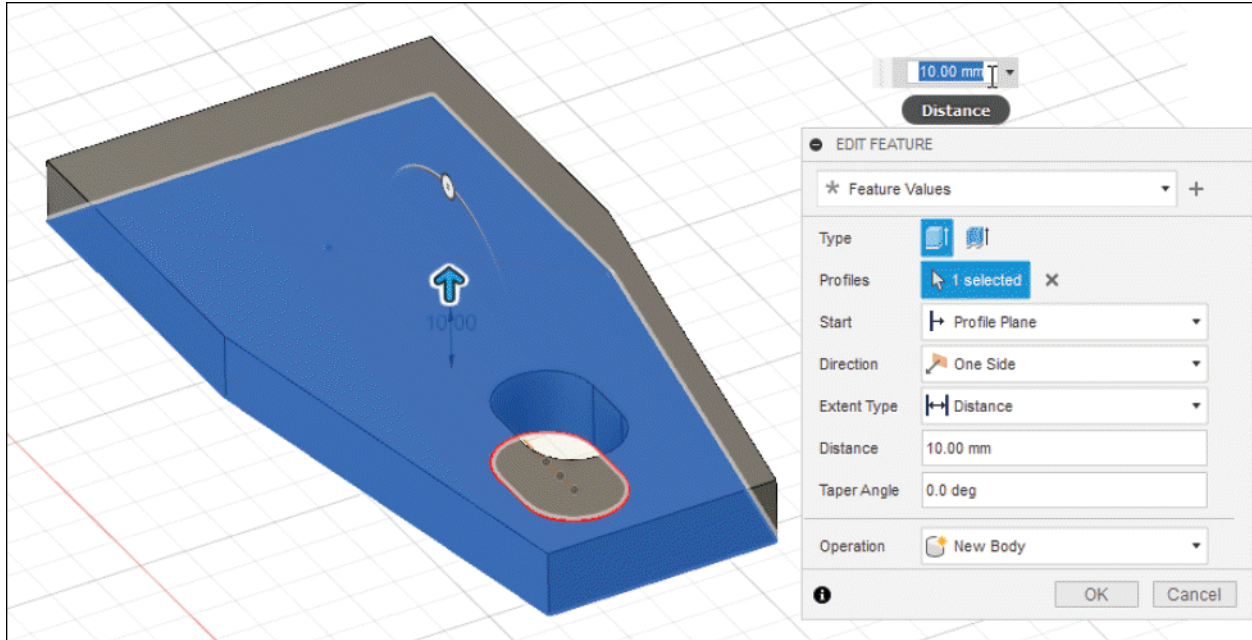


Figure-16. Preview of extrusion

- Click **OK** button from **EXTRUDE** dialog box to complete the process of extrusion.
- Click on the **Create Sketch** tool from **CREATE** drop-down. You are asked to select the plane to create sketch. Select the side face of model as shown in [Figure-17](#).

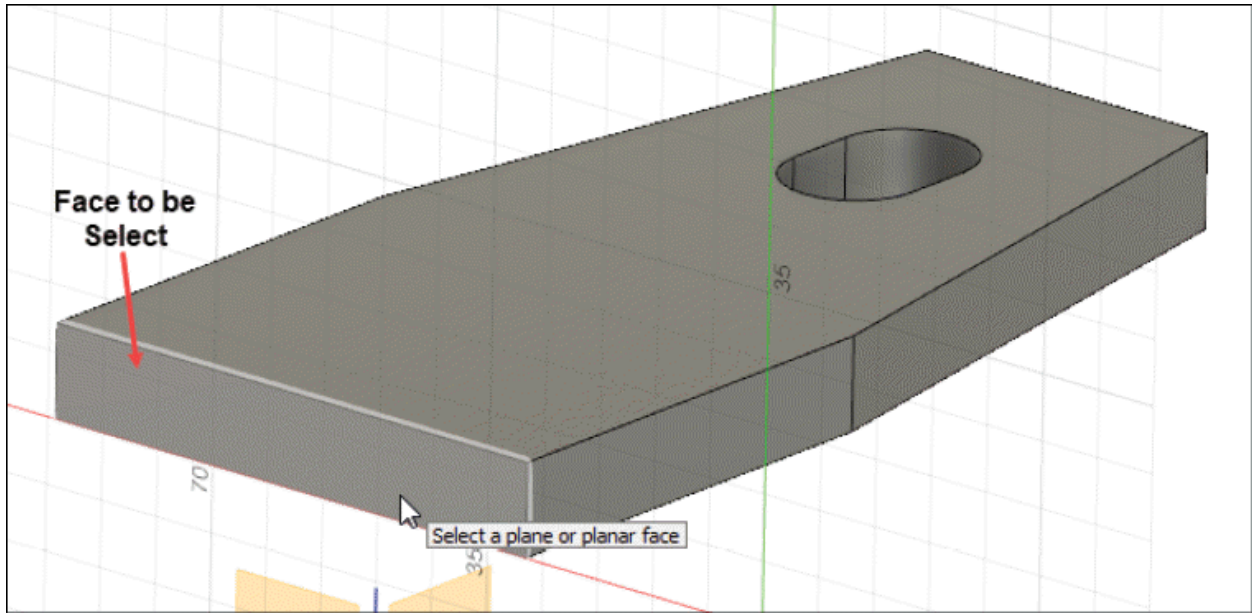


Figure-17. Face to be select for sketching

- Select the **2-Point Rectangle** tool from **Rectangle** cascading menu in the **CREATE** drop-down of **Toolbar**. Create a rectangle of dimensions **25** and **60** with two holes of diameter **10** as shown in [Figure-18](#).
- Click on the **Finish Sketch** button to exit sketching.

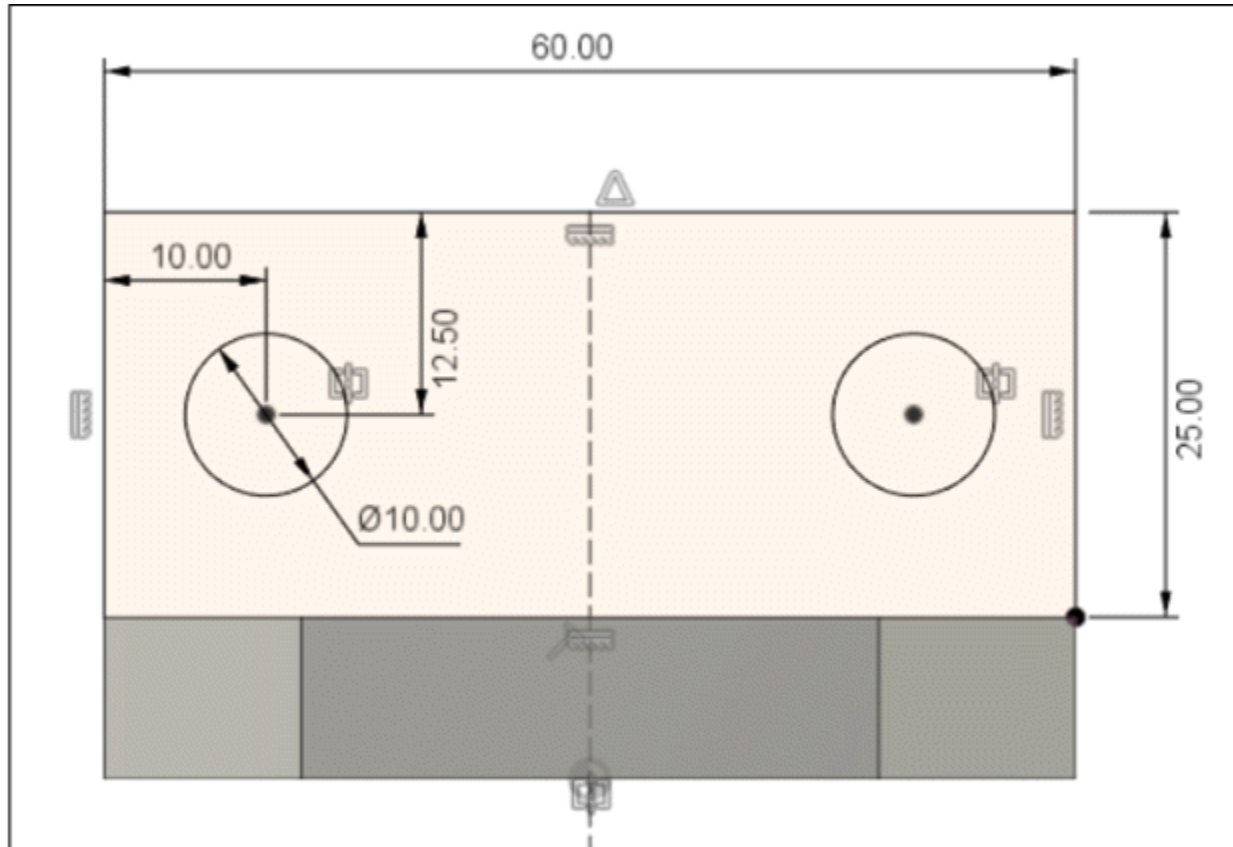


Figure-18. Creating the rectangle with holes

- Select **Extrude** tool from **CREATE** drop-down. The **EXTRUDE** dialog box will be displayed.
- Select the upper portion of recently created sketch in **Profiles** selection and specify **-5** as height of extrusion.
- Select **Join** from **Operation** drop-down of **EXTRUDE** dialog box to join the two extrusions.
- Click **OK** button from **EXTRUDE** dialog box to finish this process; refer to [Figure-19](#).

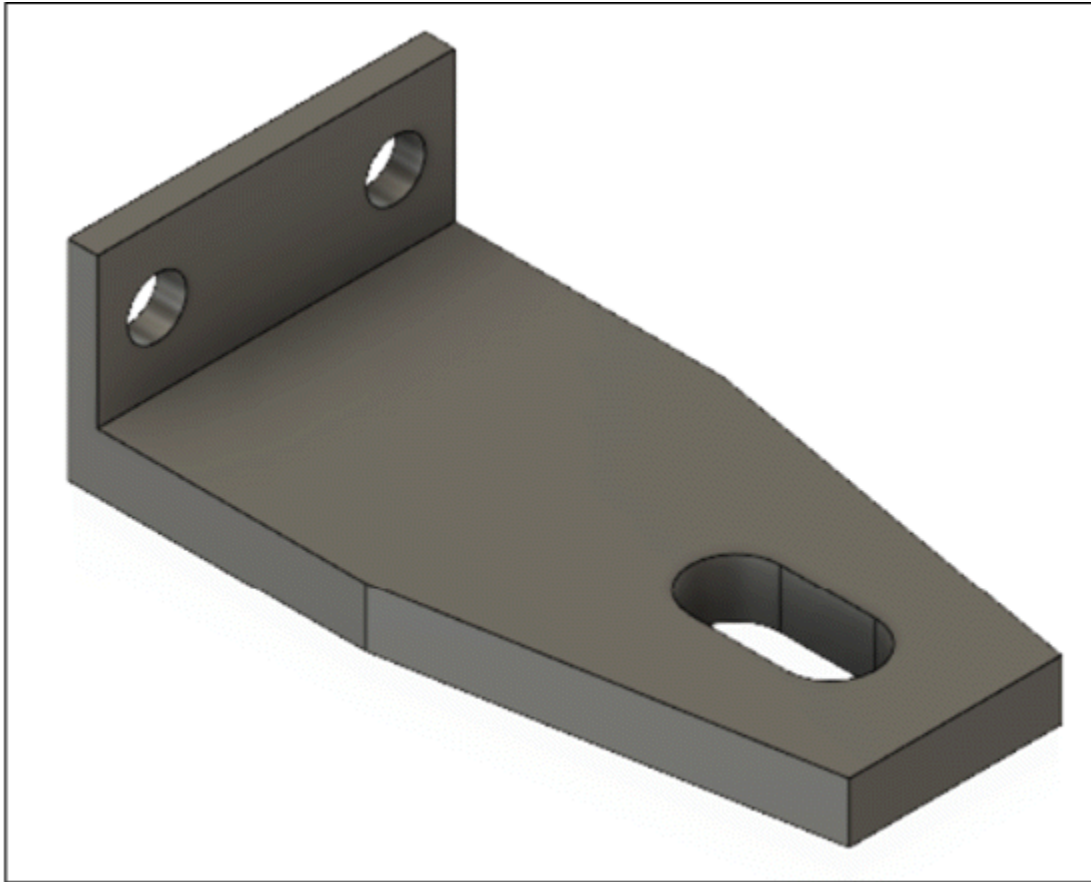


Figure-19. Extrude feature created

Creating Midplane

- Select **Midplane** tool of **CONSTRUCT** drop-down from **Toolbar**. The **MIDPLANE** dialog box will be displayed. You are asked to select the faces between which you want to create a midplane.
- Select both the faces as shown in [Figure-20](#).
- The preview of midplane will be displayed. Click **OK** button from the **MIDPLANE** dialog box to complete the process; refer to [Figure-21](#).

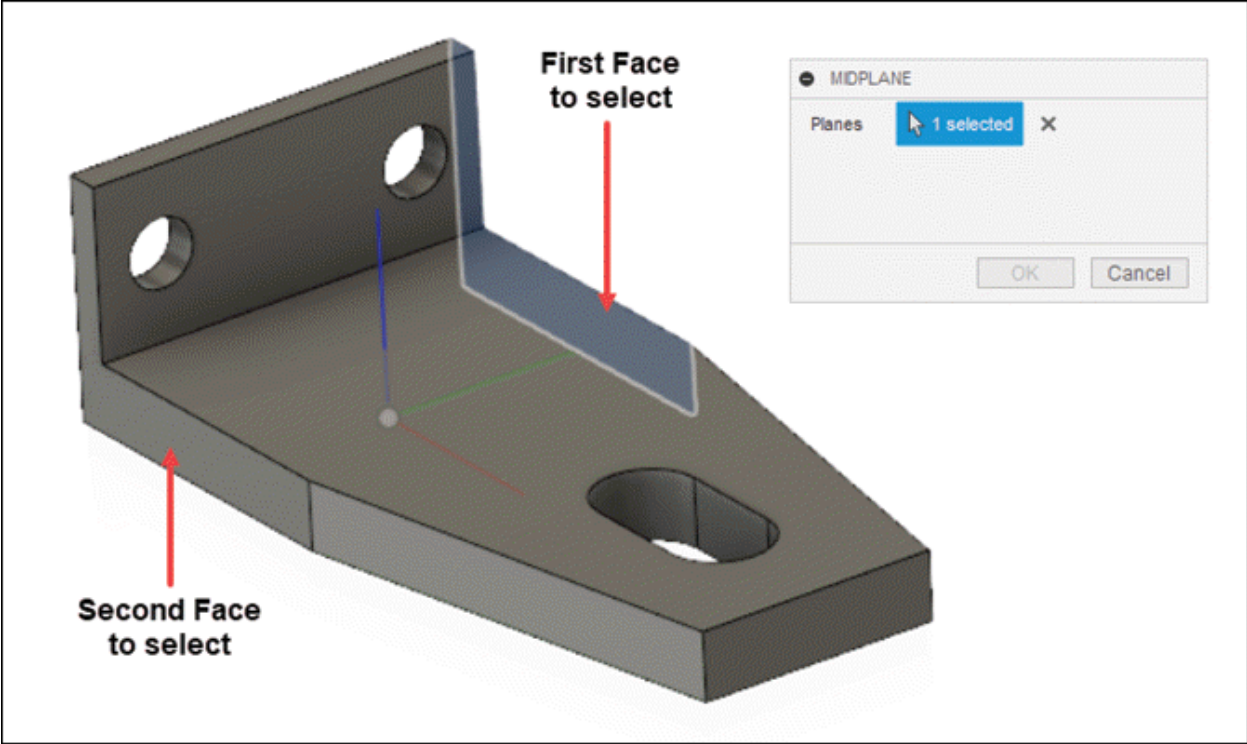


Figure-20. Face selection

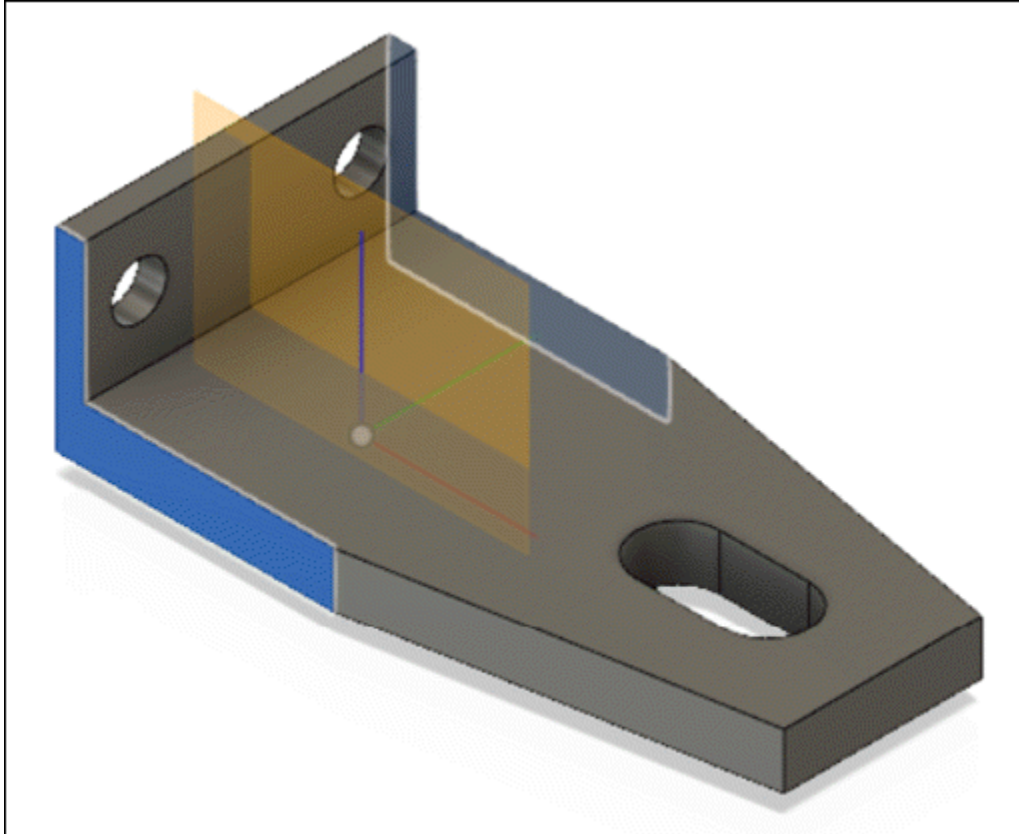


Figure-21. Midplane created

Creating Rib

- Select **Create Sketch** tool from **CREATE** drop-down. You will be asked to select the plane for creating sketch. Select the newly created midplane from **BROWSER**.
- Create a line as shown in [Figure-22](#) with end points coincident to the vertices of model and finish the sketch.

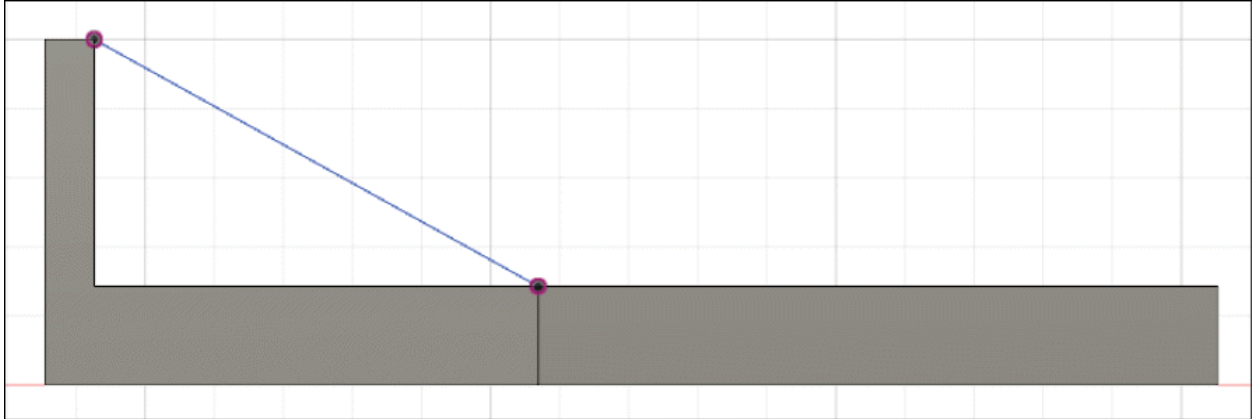


Figure-22. Creating sketch for rib

- Select **Rib** tool from **CREATE** drop-down. The **RIB** dialog box will be displayed.
- Select the recently created line for **Profile** section.
- Select **Symmetric** option from **Thickness Direction** section.
- Select **To Next** option from **Extent Type** drop-down.
- Click in **Thickness** edit box and enter **5** as thickness of rib; refer to [Figure-23](#). (You might need to click on the **Flip Direction** button from the dialog box to get desired preview.)

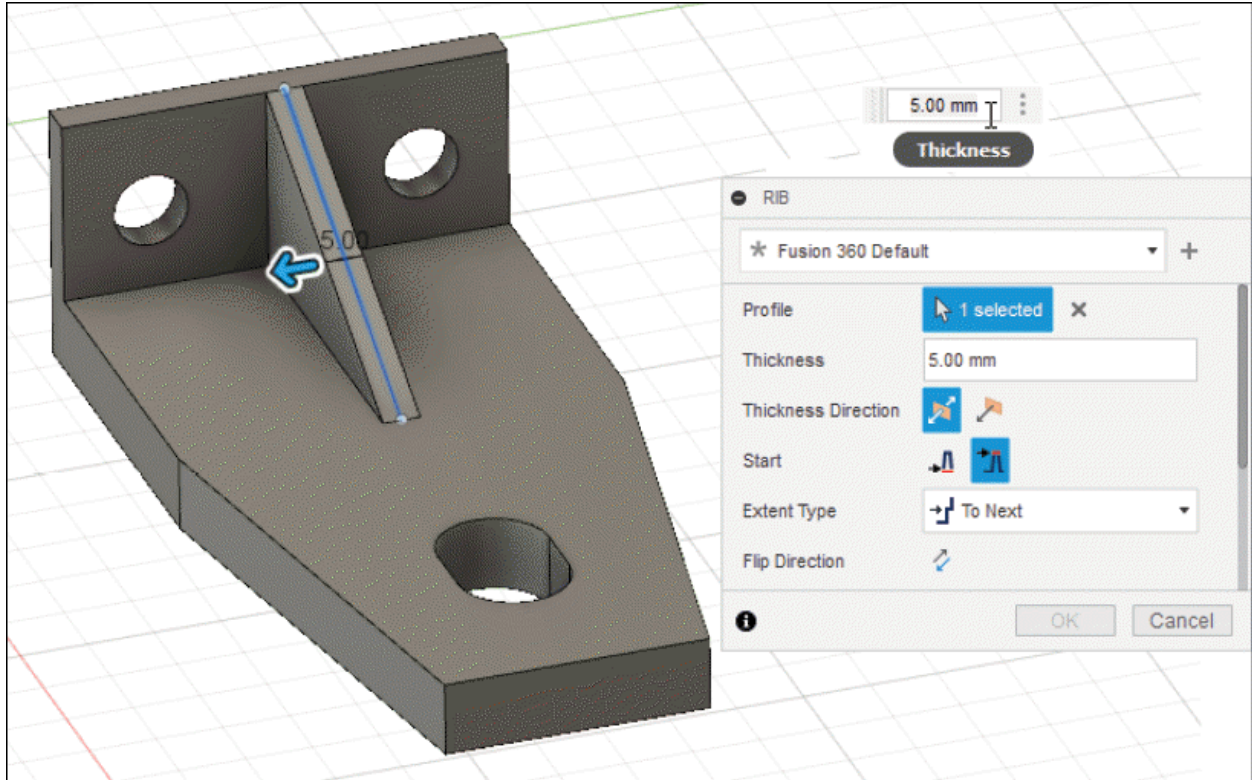


Figure-23. Preview of rib

- Click **OK** button of **RIB** dialog box to finish the process. The model will be displayed as shown in [Figure-24](#).

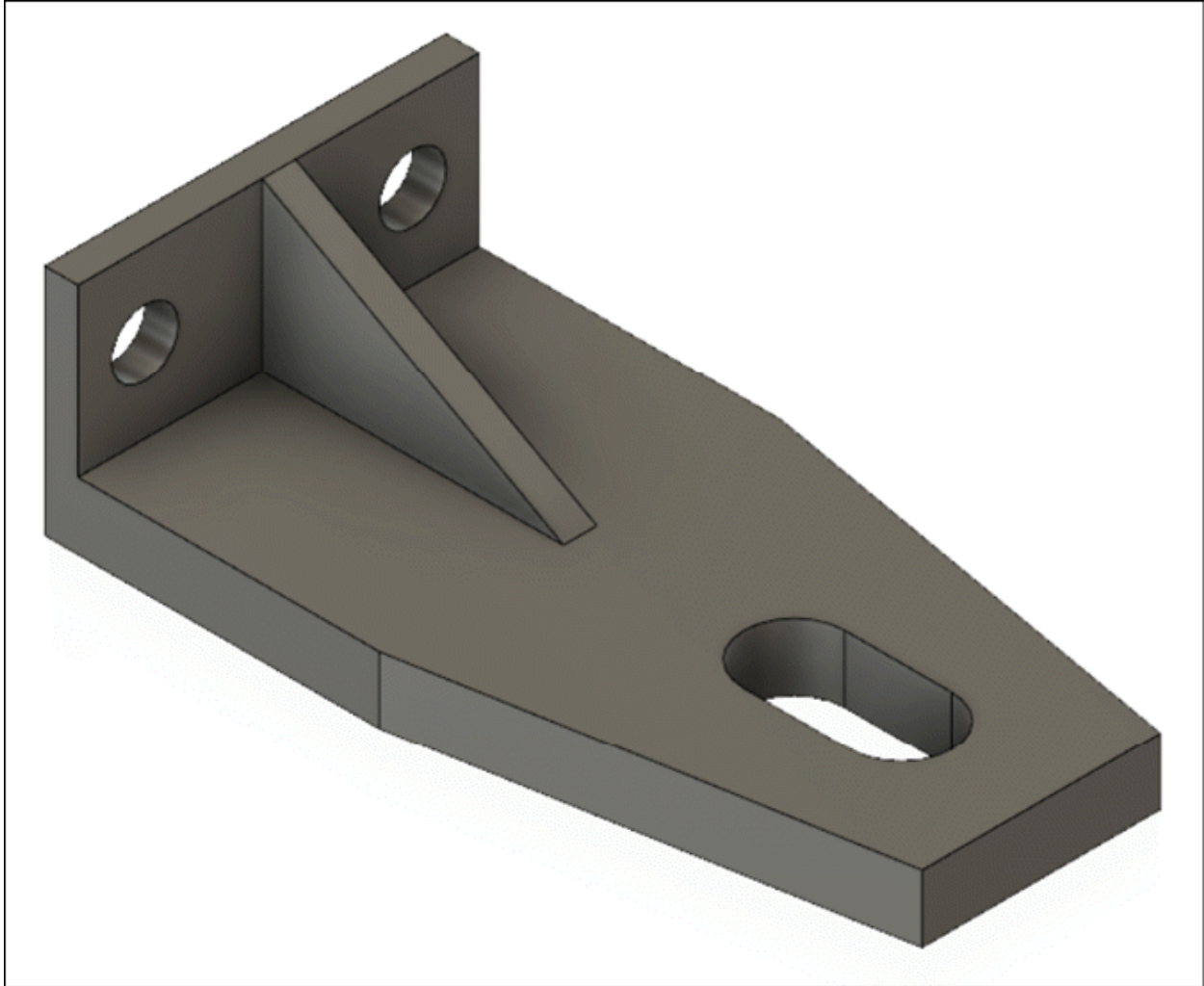


Figure-24. Final model of practical 2

PRACTICAL 3

Create the model as shown in [Figure-25](#). The dimensions of the model are given in [Figure-26](#).

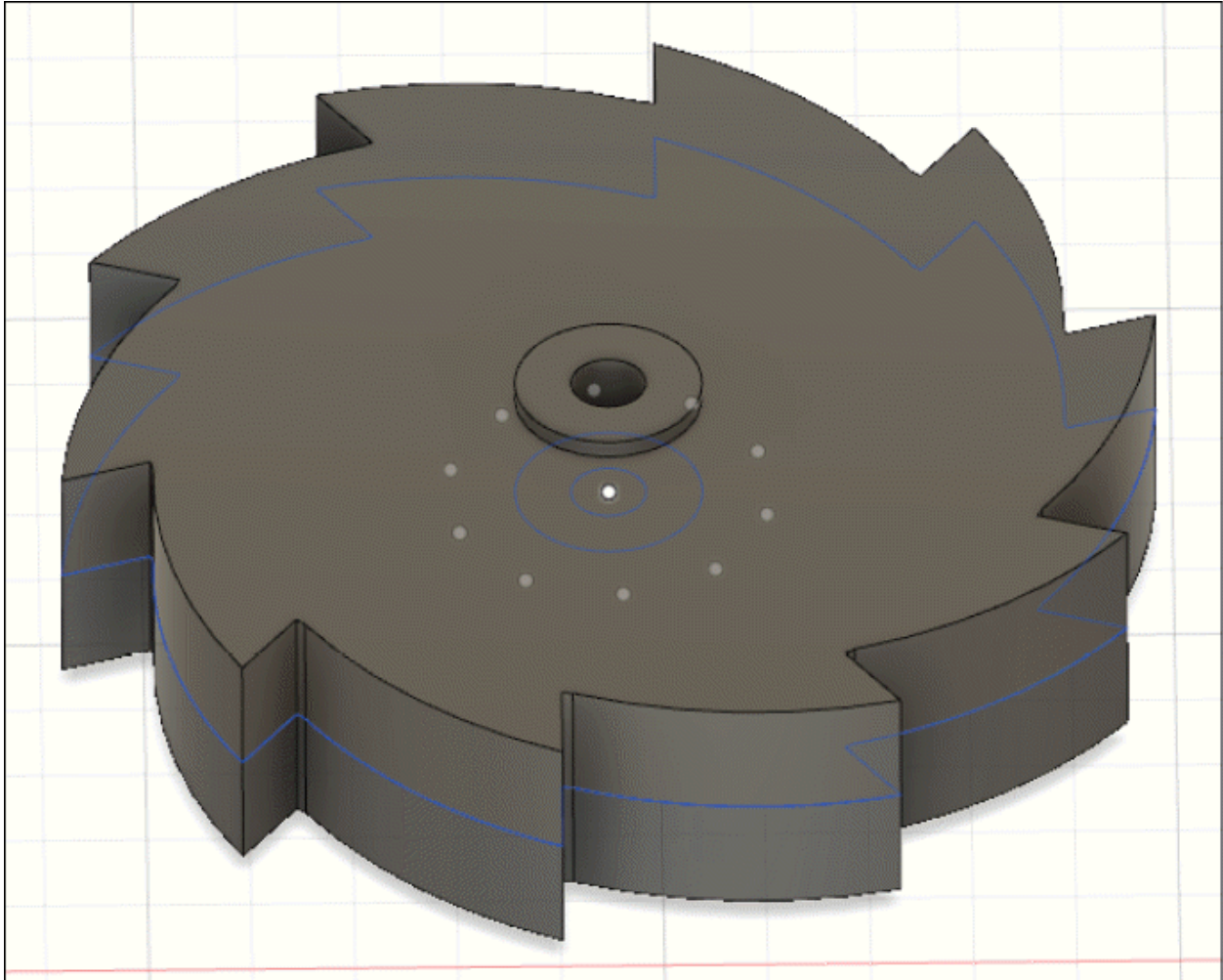


Figure-25. Model for Practical 3

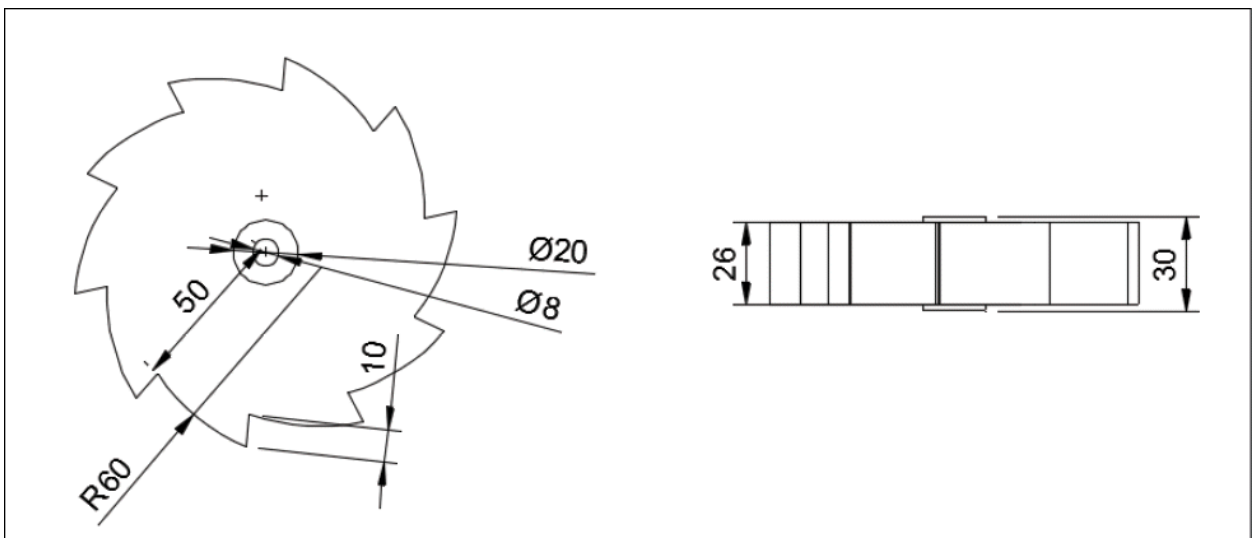


Figure-26. Practical 3 drawing views

Creating sketch for model

- Start a new design file in Autodesk Fusion. Select **Create Sketch** tool from **CREATE** drop-down in the **Toolbar**. You are asked to select sketching plane. Select **Top Plane** to create sketch. Select **Center Diameter Circle** tool of **Circle** cascading menu from **CREATE** drop-down.
- Create three circle of **8**, **20**, and **100** as diameter dimension of circles; refer to [Figure-27](#).

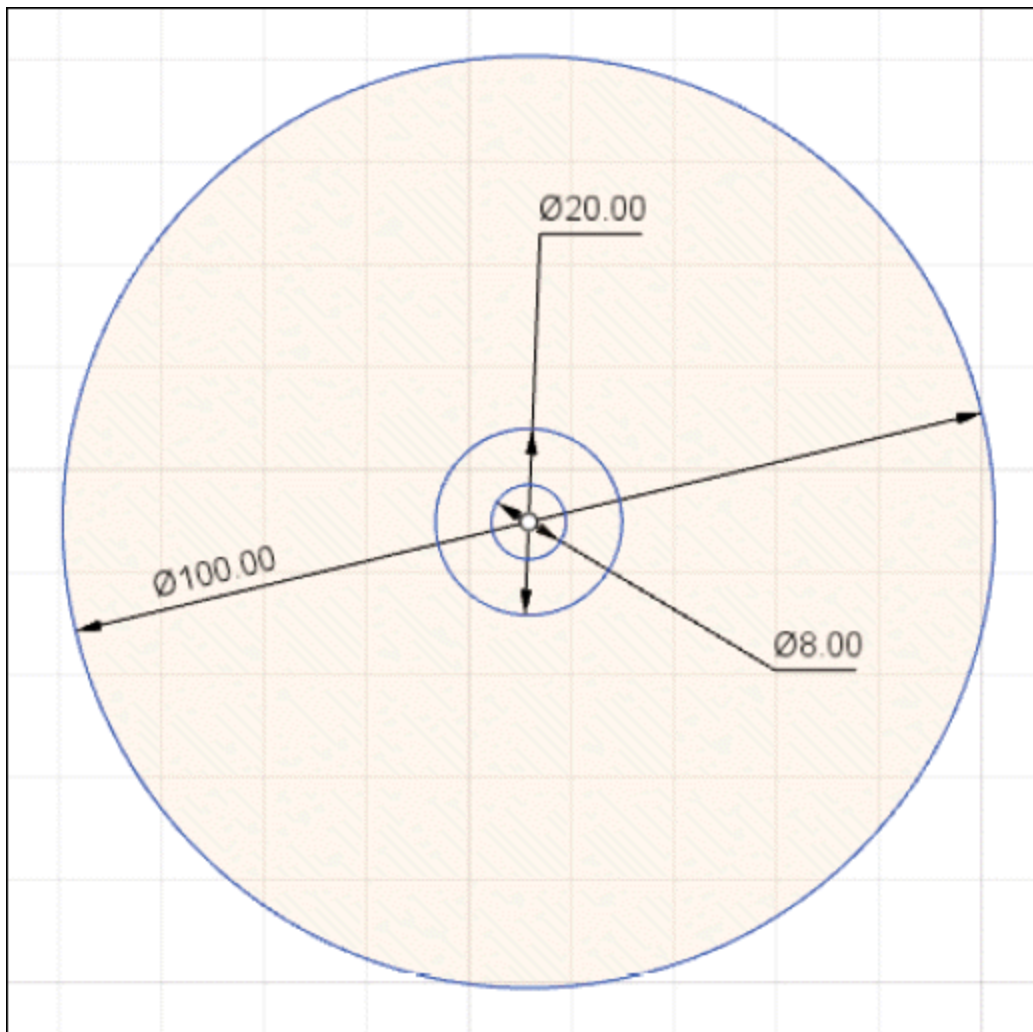


Figure-27. Creating circles

- Press **L** key to select **Line** tool and create a line of length **10**.
- Select **3-Point Arc** tool from **Arc** cascading menu and create an arc of **60** radius and **35** length as shown in [Figure-28](#).

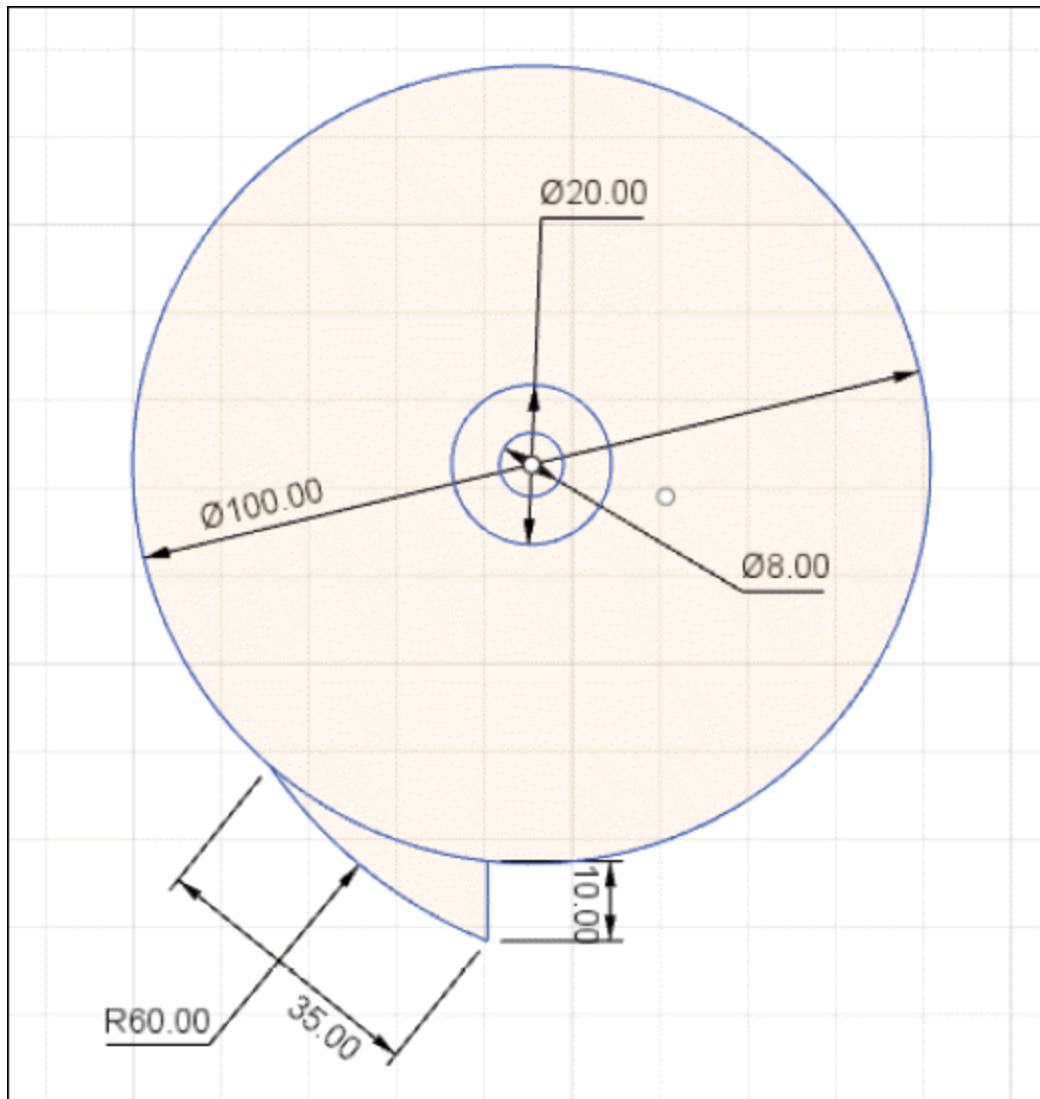


Figure-28. Creating line and arc

Creating Circular Pattern

- Select **Circular Pattern** tool from **CREATE** drop-down.

The **CIRCULAR PATTERN** dialog box will be displayed.

- Objects selection is active by default. Select the recently created line and arc for circular pattern.
- Click on **Select** button of **Center Point** section and select the major circle to define its center as center point for pattern; refer to [Figure-29](#).

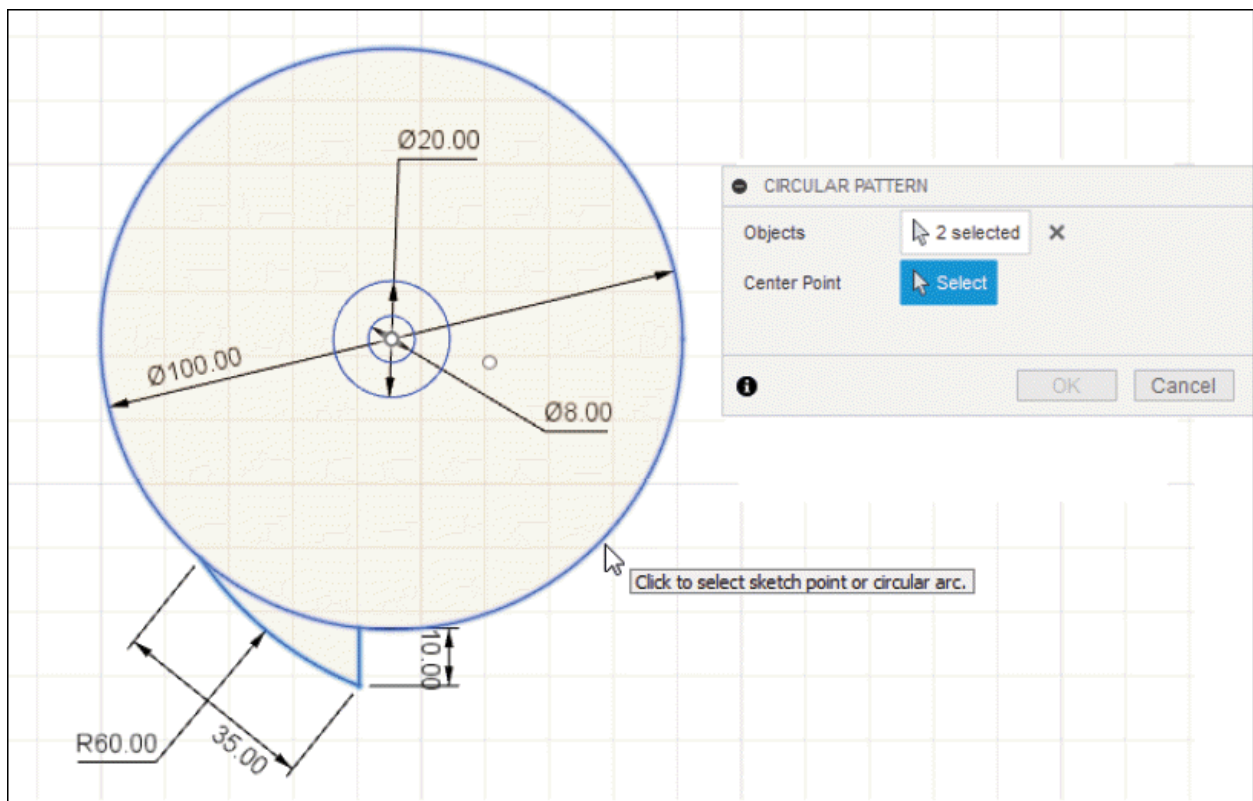


Figure-29. Creating circular pattern

- The updated **CIRCULAR PATTERN** dialog box will be displayed. Click in **Quantity** edit box and specify **10** as quantity of circular pattern.
- Click **OK** button from **CIRCULAR PATTERN** dialog box to complete the process.

Trimming

- Select the **Trim** tool from **MODIFY** drop-down and remove all the extra entities from sketch as shown in [Figure-30](#).

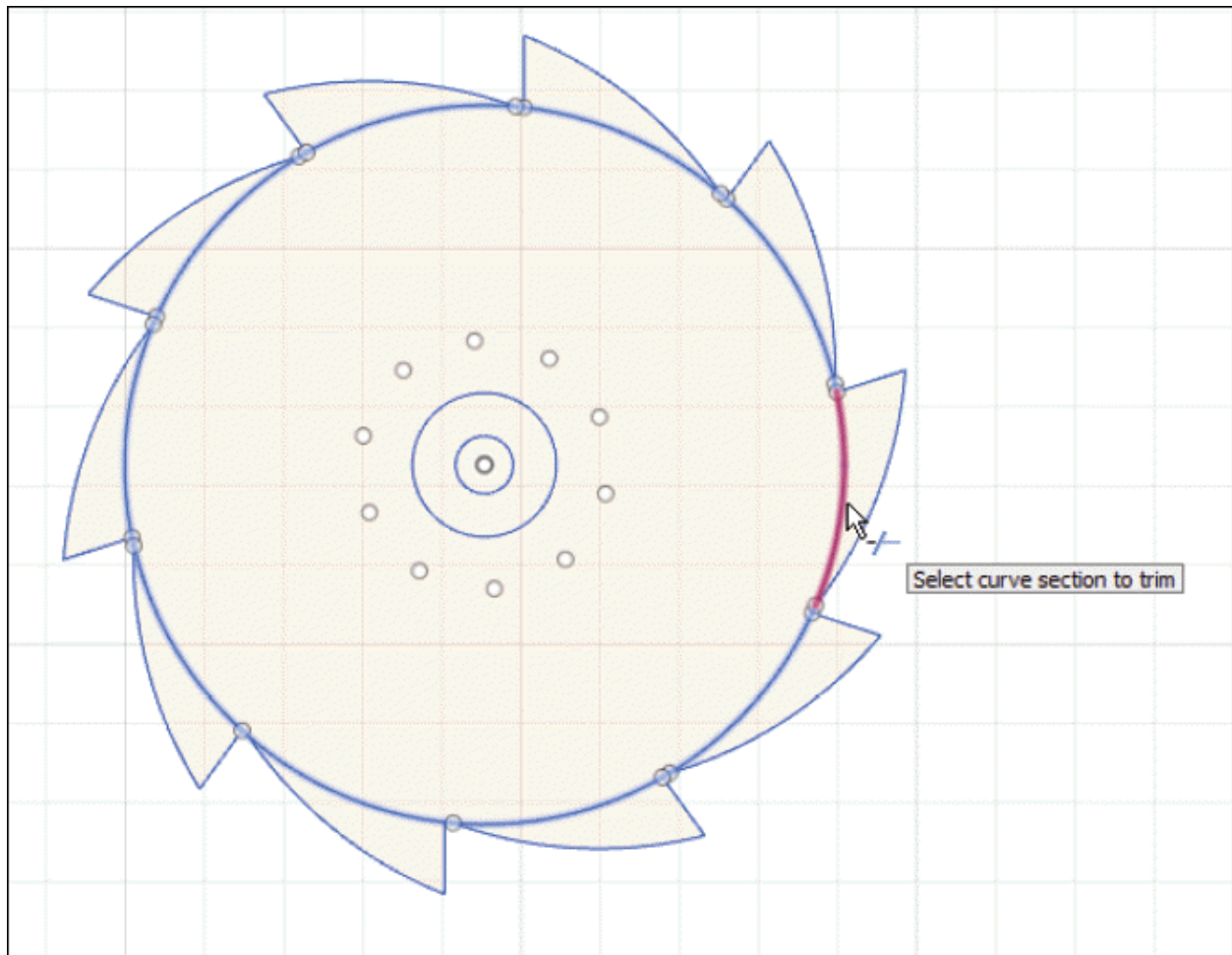


Figure-30. Trimming the extra entities

- After removing the extra entities from sketch, the sketch will be displayed as shown in [Figure-31](#).

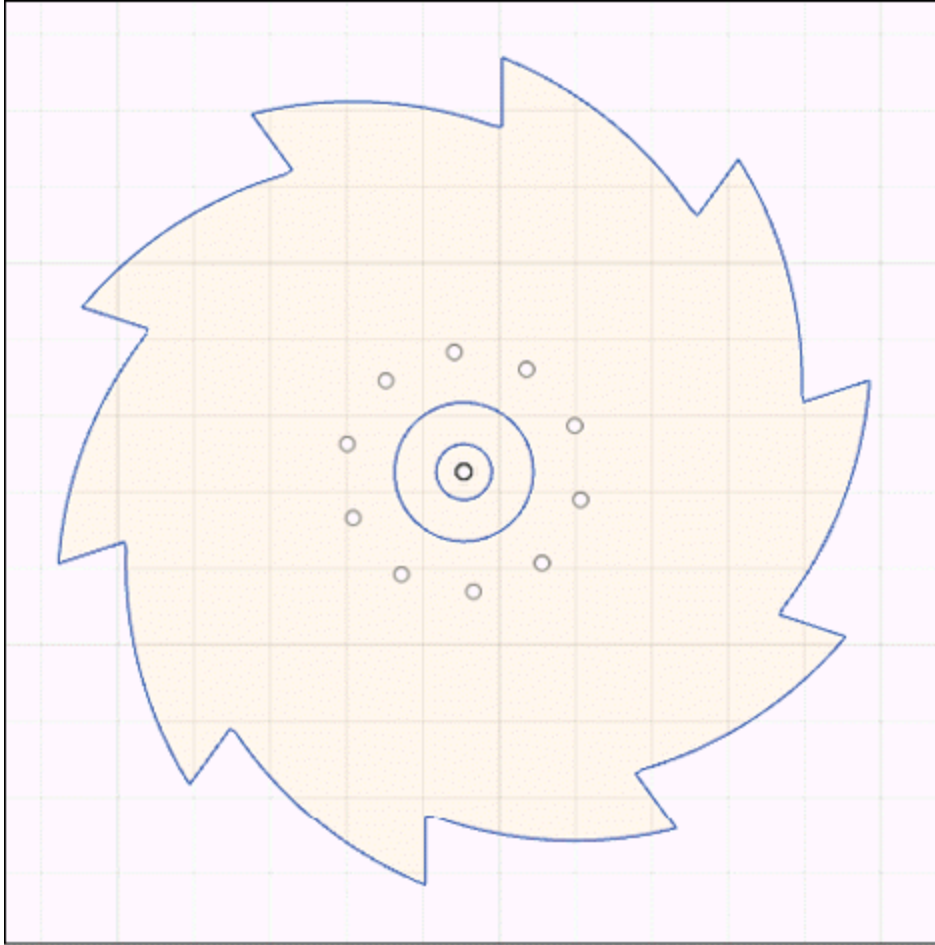


Figure-31. After removing extra entities

Creating Extrude

- Select **Extrude** tool from **CREATE** drop-down or press **E** key. The **EXTRUDE** dialog box will be displayed.
- Profile selection is active by default. Click between **20** diameter and **8** diameter circle to extrude the middle section.
- Select **Symmetric** option of **Direction** drop-down from **EXTRUDE** dialog box.
- Click in **Distance** edit box of **EXTRUDE** dialog box and

specify **15** as height of extrusion. Rotate the model to check preview; refer to [Figure-32](#).

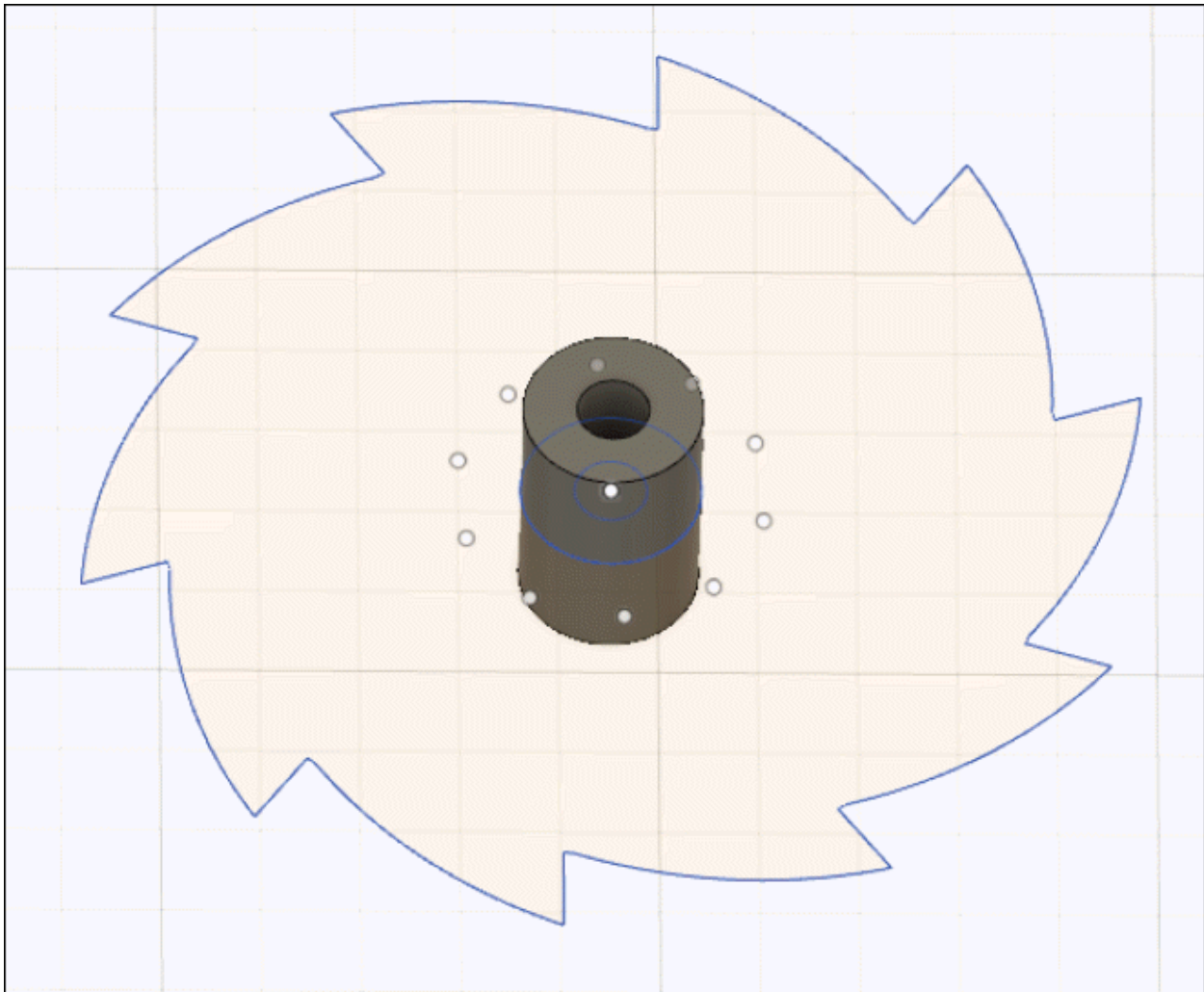


Figure-32. Preview of first extrusion

- Click **OK** button of **EXTRUDE** dialog box to finish this extrusion. The sketch of extrude feature will hide automatically. Click on the crossed eye icon before **Sketch 1** in the **Sketches** node of **BROWSER** to display the sketch again.
- Right-click on the screen and select **Repeat Extrude**

button from **Marking Menu**. The **EXTRUDE** dialog box will be displayed.

- Select the area outside the **20** diameter circle earlier created.
- Select **Symmetric** button of **Direction** drop-down from **EXTRUDE** dialog box.
- Click in **Distance** edit box of **EXTRUDE** dialog box and specify **13** as height of extrusion; refer to [Figure-33](#).

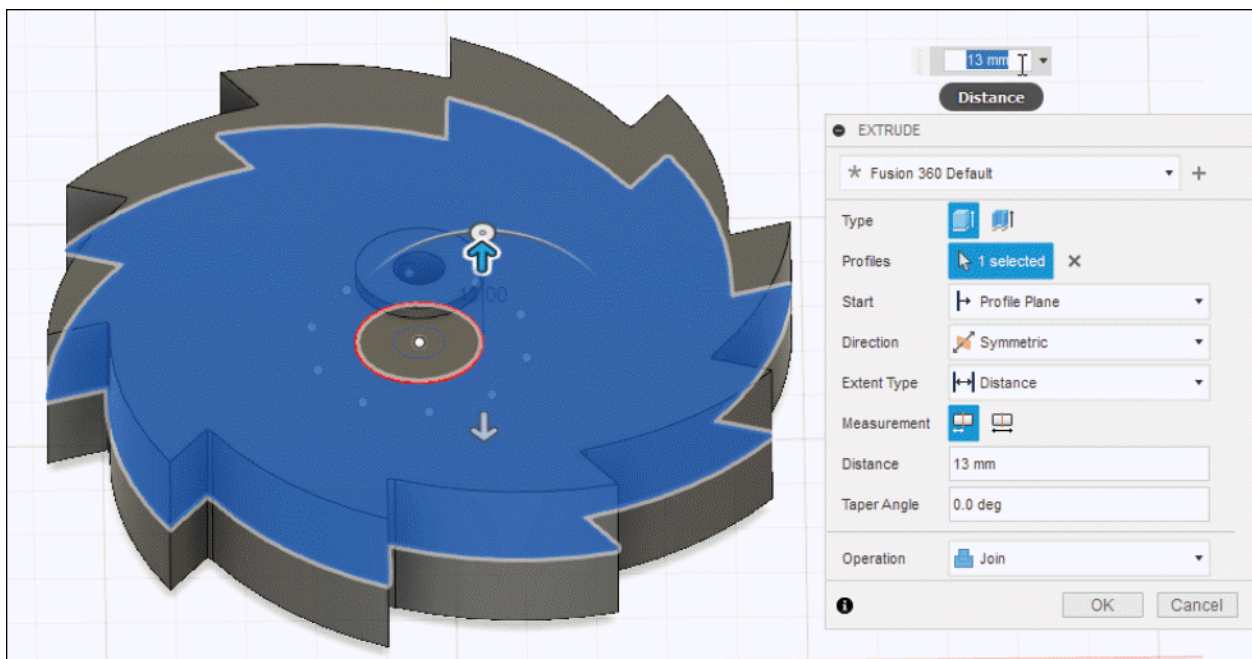


Figure-33. Preview of second extrusion

- Click **OK** button from **EXTRUDE** dialog box to complete the process of extrusion. The model will be displayed as shown in [Figure-34](#).

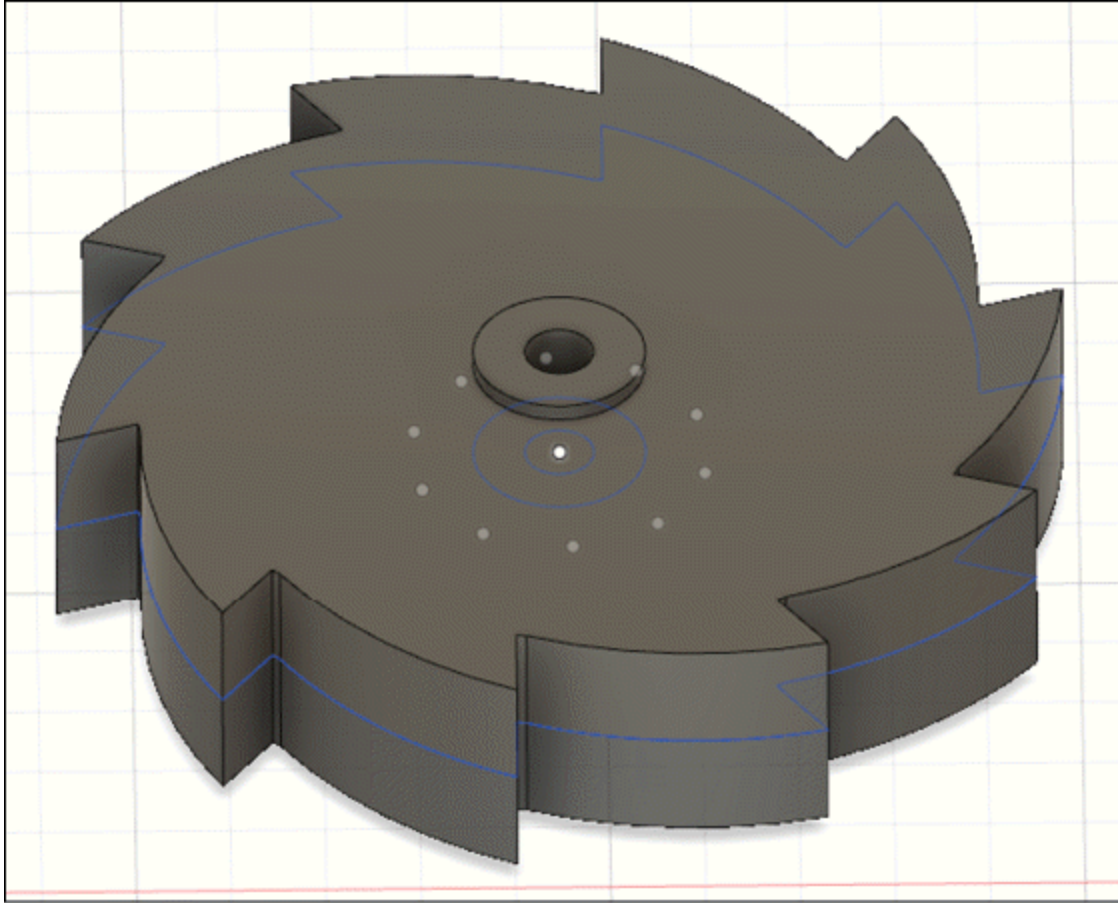


Figure-34. Final model of Practical 3

You can now hide the sketch and save the file.

PRACTICAL 4

Create the model as shown in [Figure-35](#). The dimensions of the model are given in [Figure-36](#).

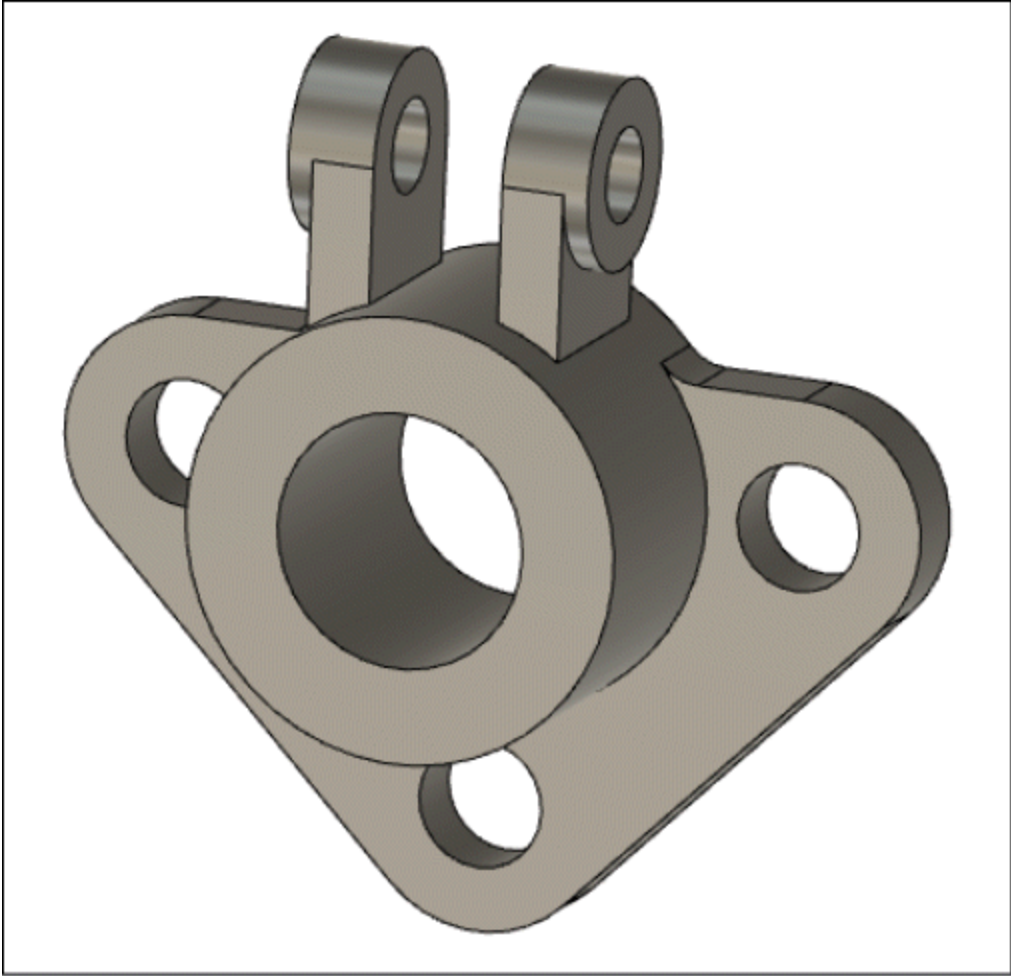


Figure-35. Model for Practical 4

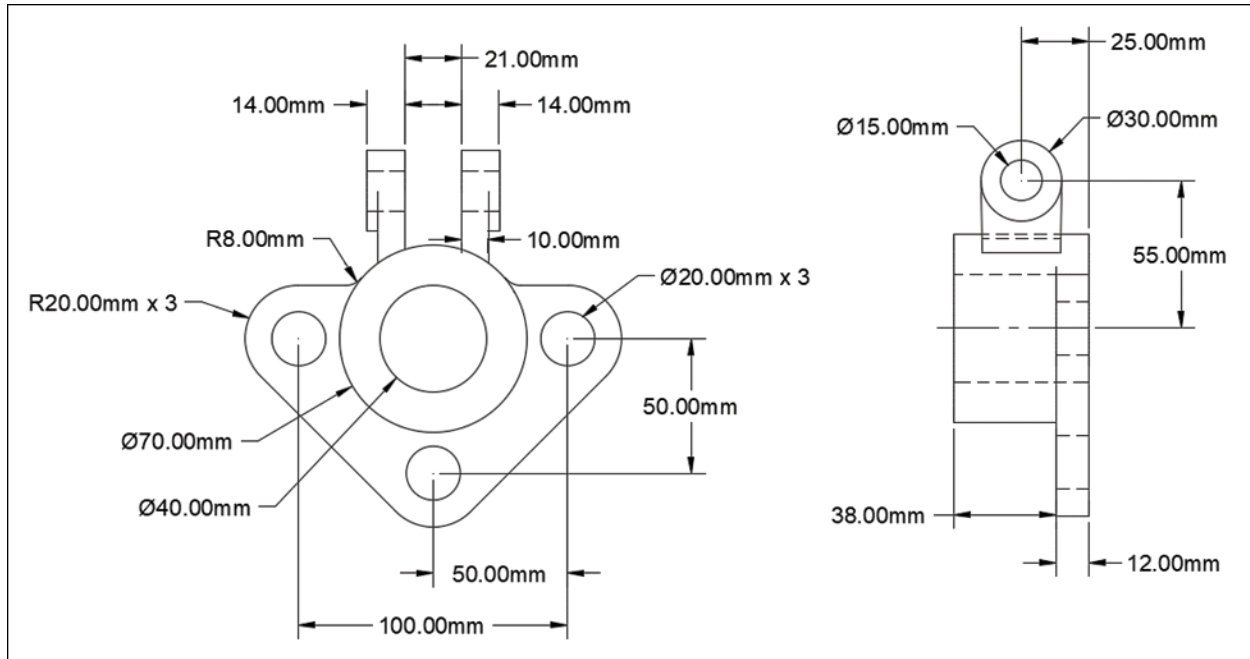


Figure-36. Practical 4 drawing views

Creating sketch for model

- Start a new design file in Autodesk Fusion. Click on the **Create Sketch** tool from **CREATE** drop-down in the **Toolbar**. You are asked to select sketching plane.
- Select **Top Plane** to create sketch. Select **Center Diameter Circle** tool of **Circle** cascading menu from **CREATE** drop-down.
- Create two circles of diameter **70** mm and **40** mm and create three circles of diameter **20** mm each; refer to [Figure-37](#). After creating the circles, apply **Horizontal/Vertical** constraint to the circles.

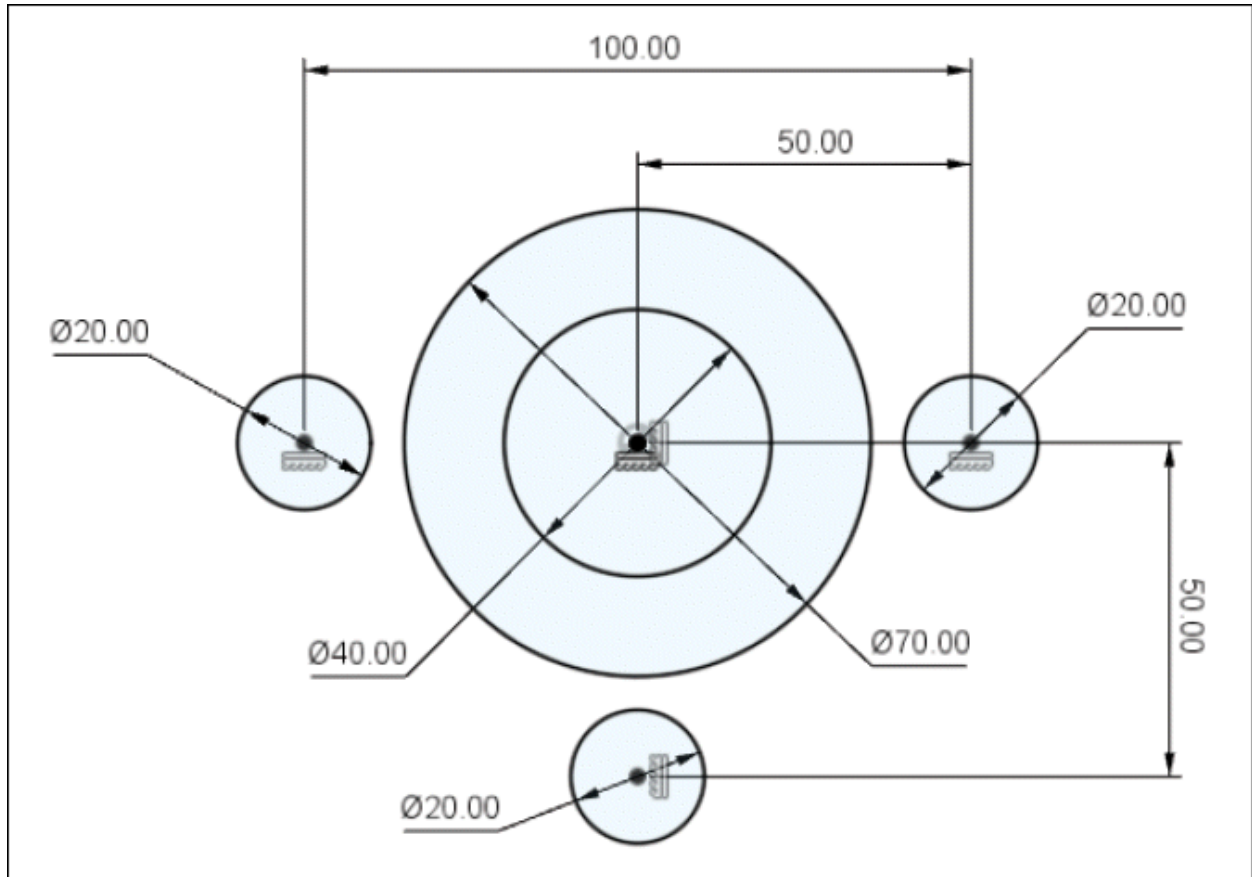


Figure-37. Creating circles

- Select **3-Point Arc** tool of **Arc** cascading menu from **CREATE** drop-down and create three arcs of radius **20** mm each; refer to [Figure-38](#). After creating the arcs, apply **Concentric** constraint to all the arcs.

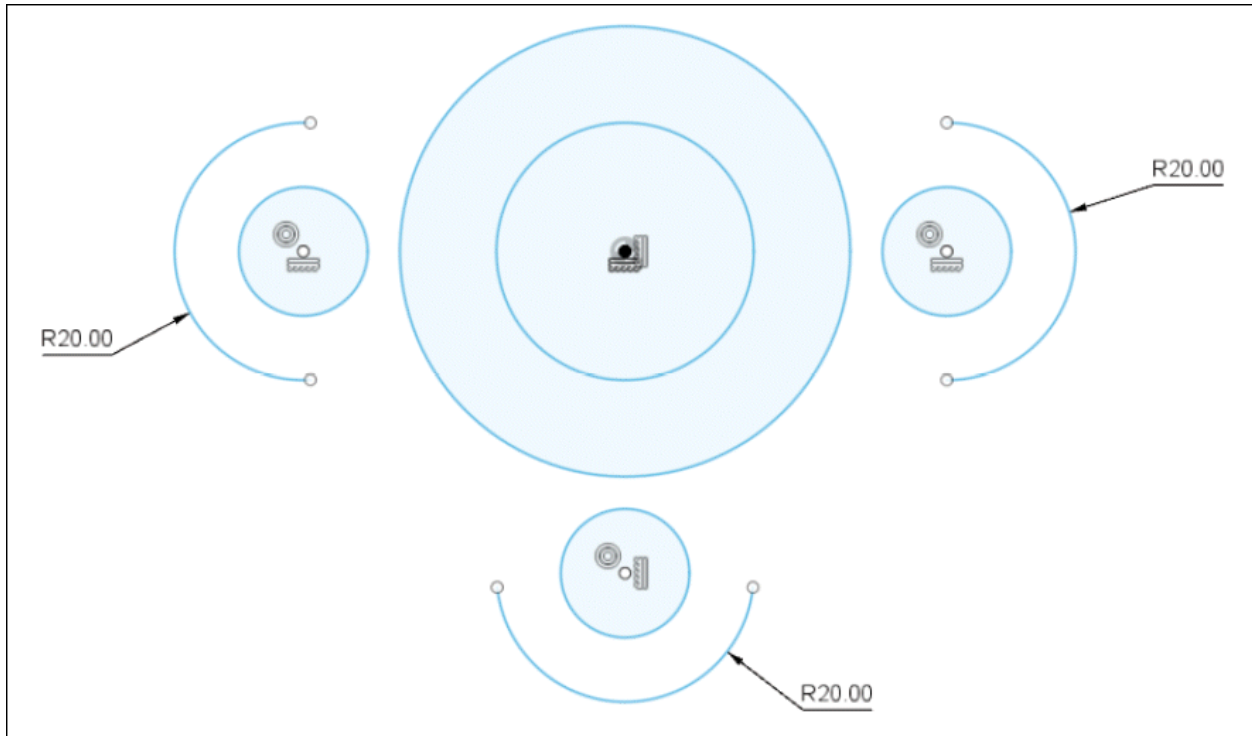


Figure-38. Creating arcs

- Select **Line** tool from **CREATE** drop-down and create three lines joining the arcs; refer to [Figure-39](#). After creating the lines, apply **Tangent** constraint to all the lines.

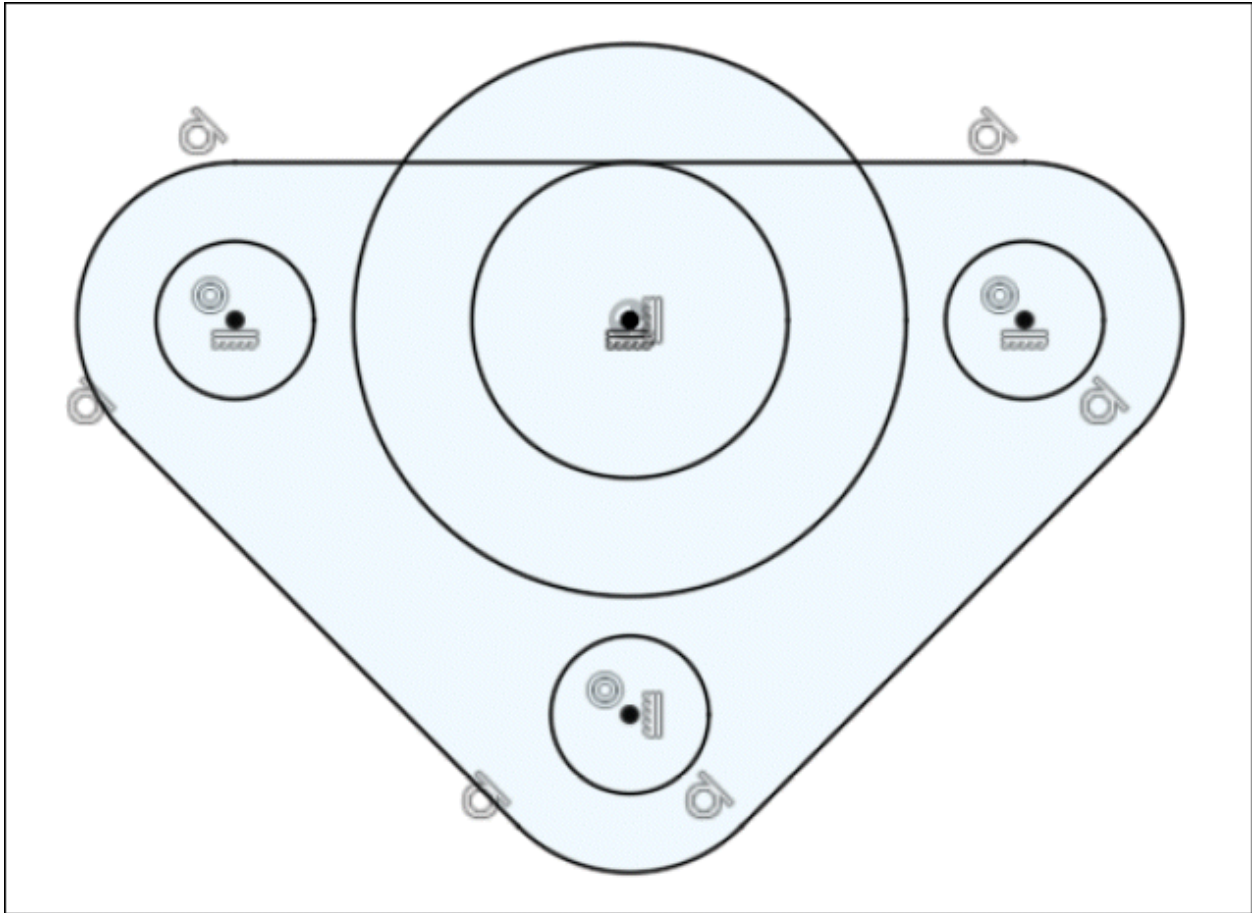


Figure-39. Creating lines joining the arcs

- Select **3-Point Arc** tool of **Arc** cascading menu and create two arcs of radius **8 mm** each; refer to [Figure-40](#). After creating the arcs, apply **Tangent** constraint to the arcs.

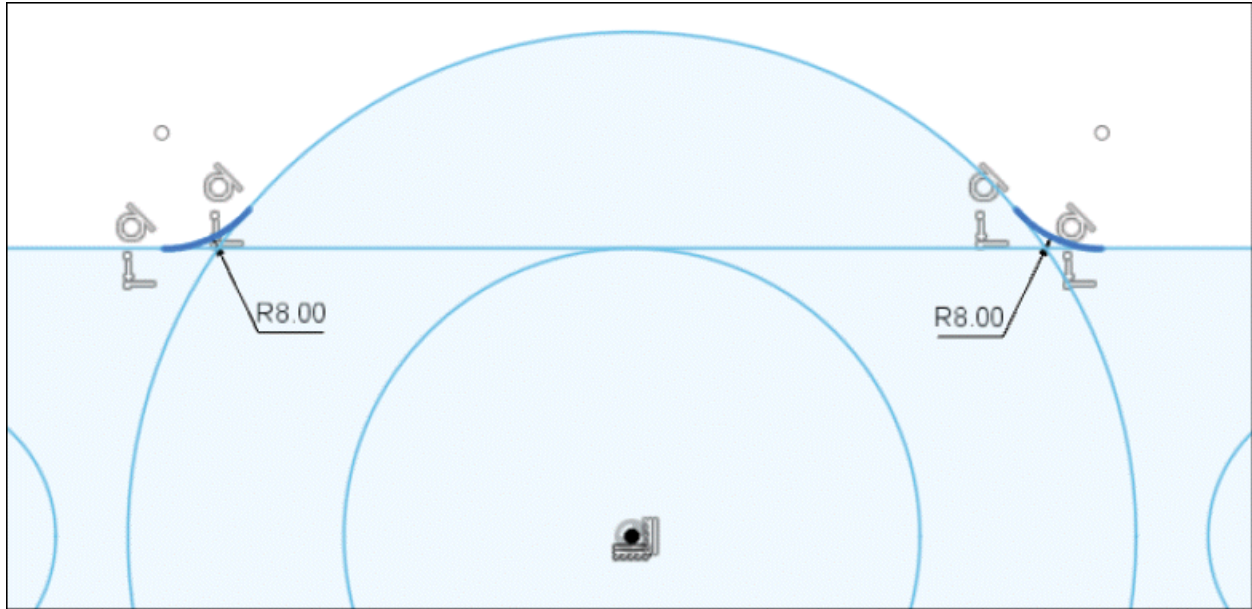


Figure-40. Creating arcs

Trimming

- Select the **Trim** tool from **MODIFY** drop-down and remove extra entities from sketch as shown in [Figure-41](#).

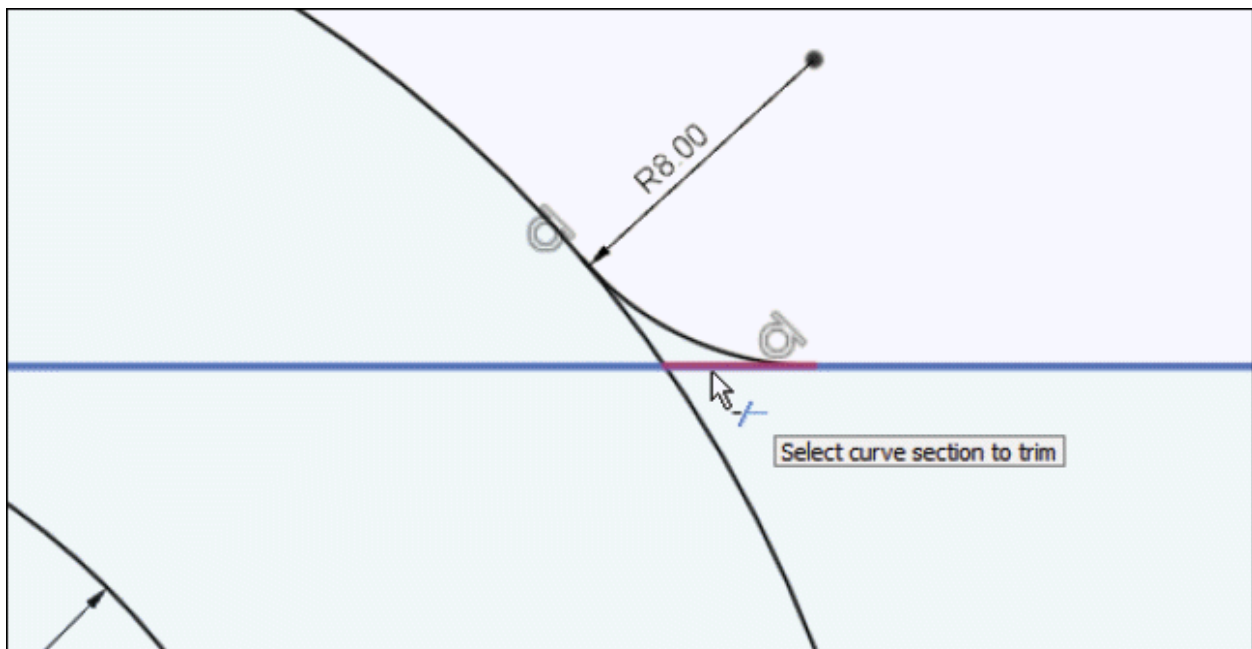


Figure-41. Removing extra entities

- After removing extra entities from sketch, the sketch will be displayed as shown in [Figure-42](#).

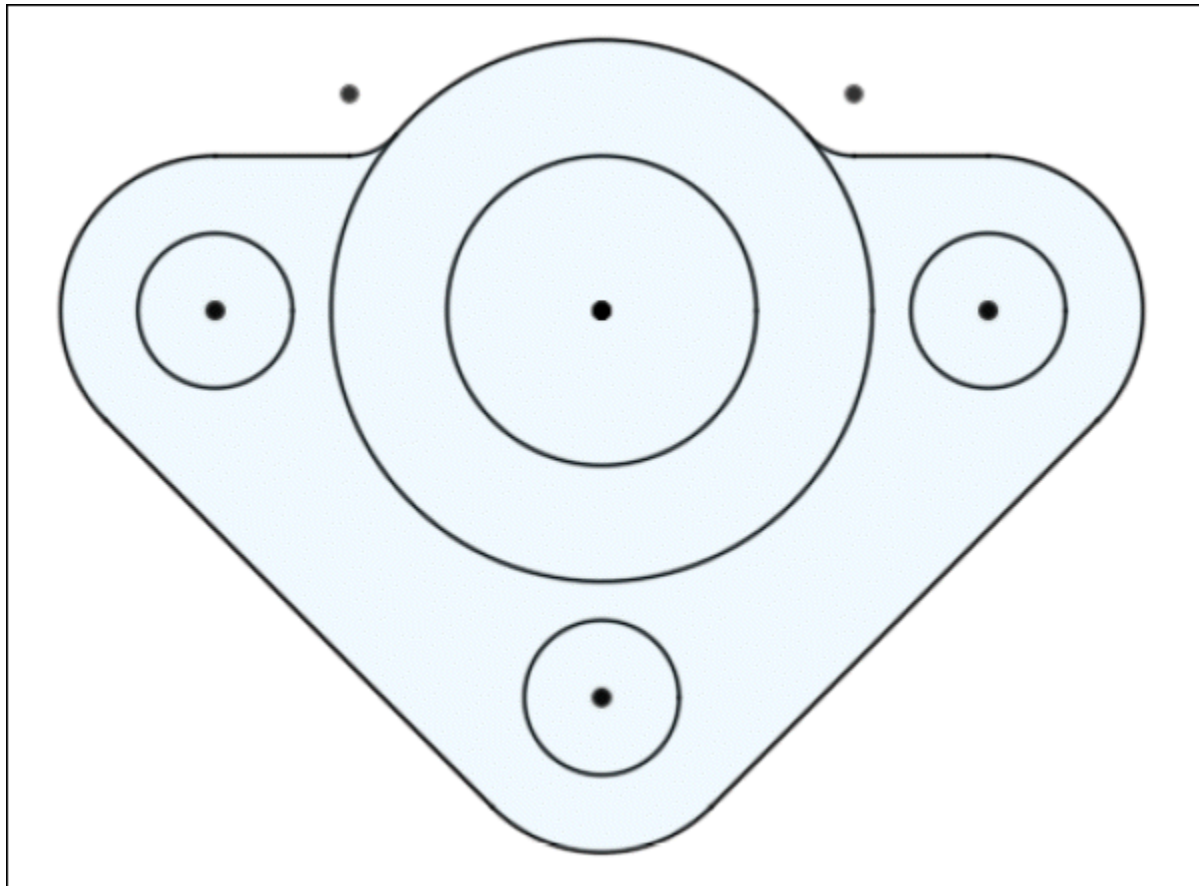


Figure-42. Sketch after trimming

Creating Extrude

- Select **Extrude** tool from **CREATE** drop-down or press **E** key. The **EXTRUDE** dialog box will be displayed.
- Profile selection is active by default. Select the profile from the sketch as shown in [Figure-43](#).

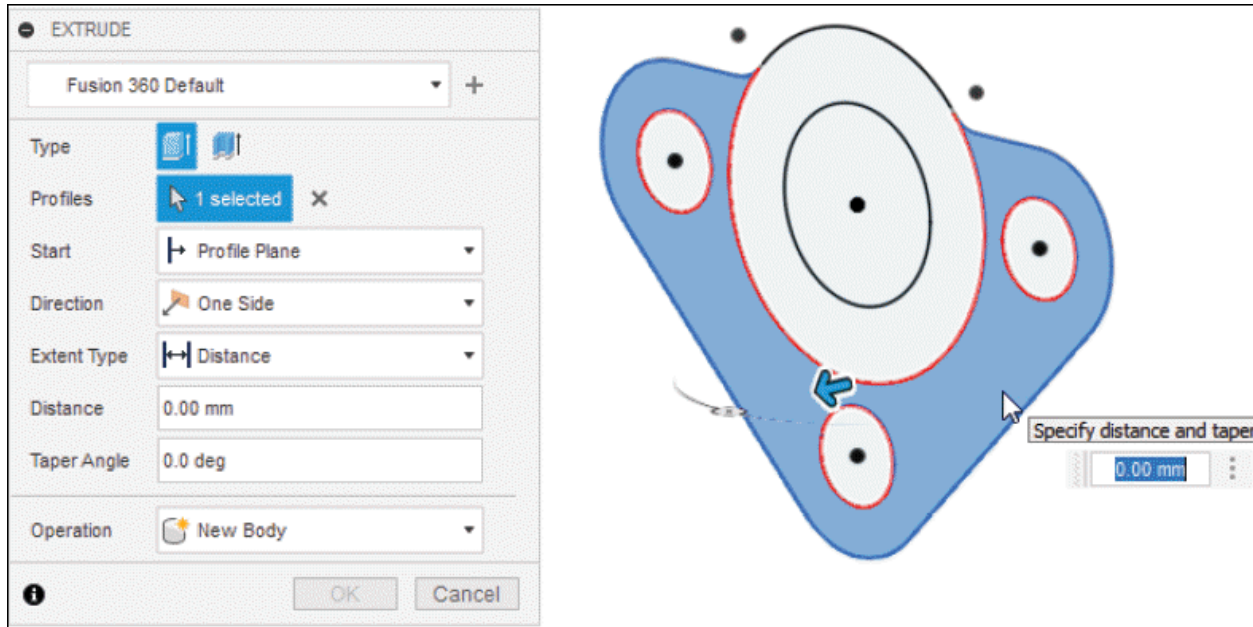


Figure-43. Extruding the sketch

- Select **One Side** option of **Direction** drop-down from **EXTRUDE** dialog box.
- Click in **Distance** edit box of **EXTRUDE** dialog box and specify **12** as distance of extrusion.
- Click **OK** button of **EXTRUDE** dialog box to finish this extrusion. The first sketch will be extruded and another sketch to be extruded will hide automatically; refer to [Figure-44](#).

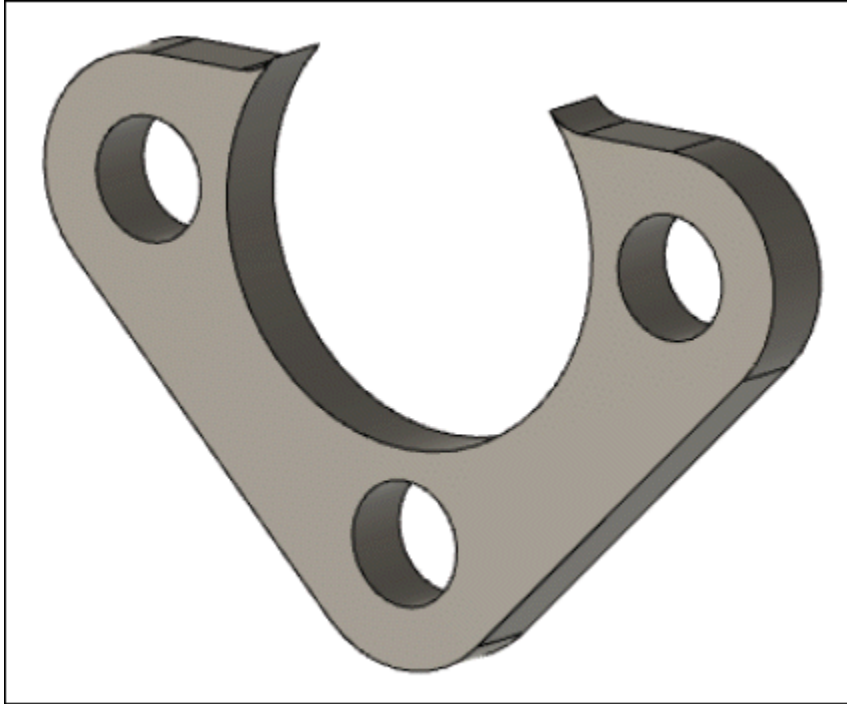


Figure-44. First sketch extruded

- Click on the crossed eye icon before **Sketch 1** in the **Sketches** node of **BROWSER** to display the sketch again.
- Right-click on the screen and select **Repeat Extrude** button from **Marking Menu**. The **EXTRUDE** dialog box will be displayed.
- Select the profile to extrude as shown in [Figure-45](#).

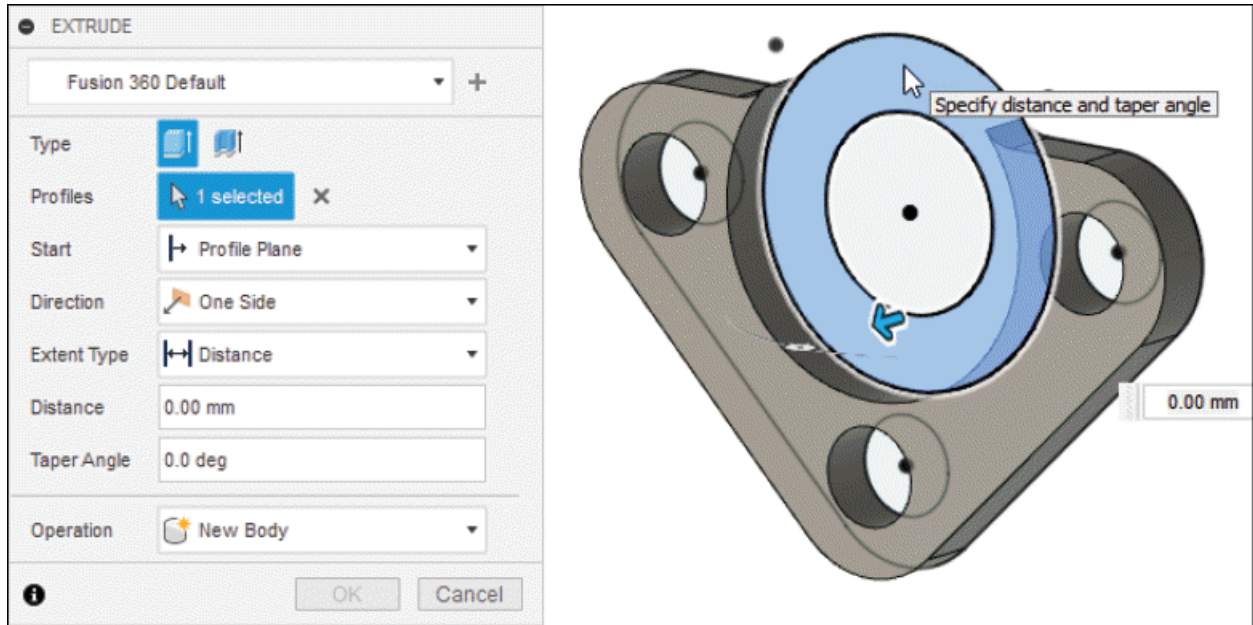


Figure-45. Extruding the second sketch

- Select **One Side** option of **Direction** drop-down from **EXTRUDE** dialog box.
- Click in **Distance** edit box of **EXTRUDE** dialog box and specify **50** as distance of extrusion.
- Click on the **OK** button from **Extrude** dialog box to finish the extrusion. The second sketch will be extruded; refer to [Figure-46](#).

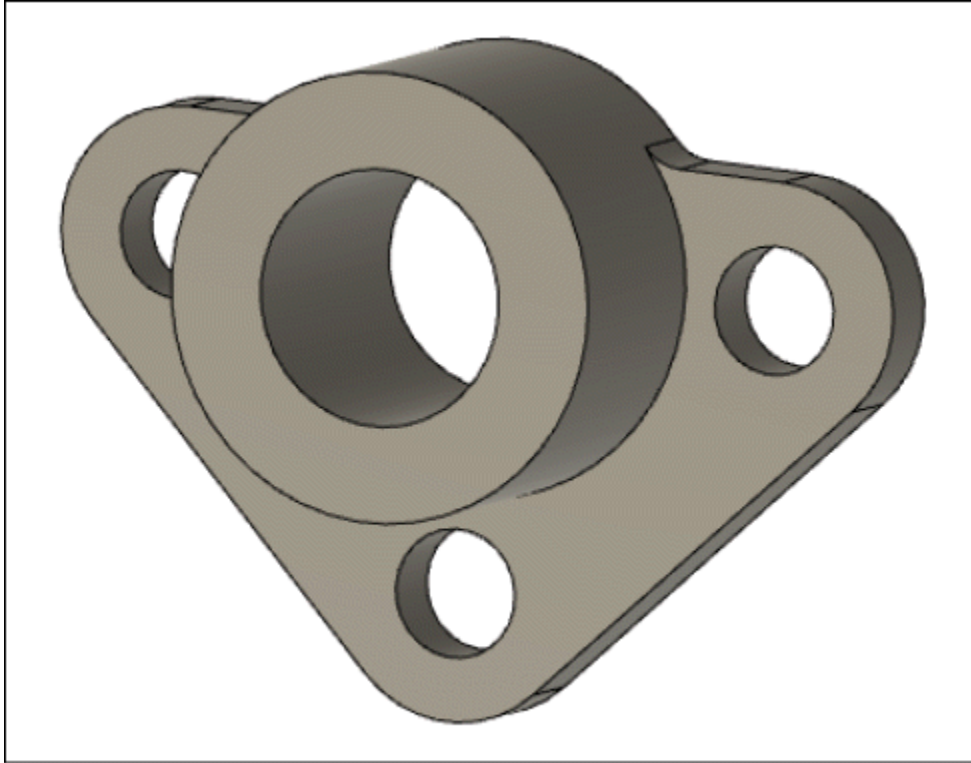


Figure-46. Second sketch extruded

Creating Offset Plane

- Select **Offset Plane** tool from **CONSTRUCT** drop-down. The **OFFSET PLANE** dialog box will be displayed and you are asked to select the plane.
- The **Select** button of **Plane** section is active by default. Select **YZ** plane from **BROWSER**.
- Click in the **Distance** edit box of the dialog box and enter **24.5** as distance value; refer to [Figure-47](#).
- Click on **OK** button from the dialog box to complete the process.

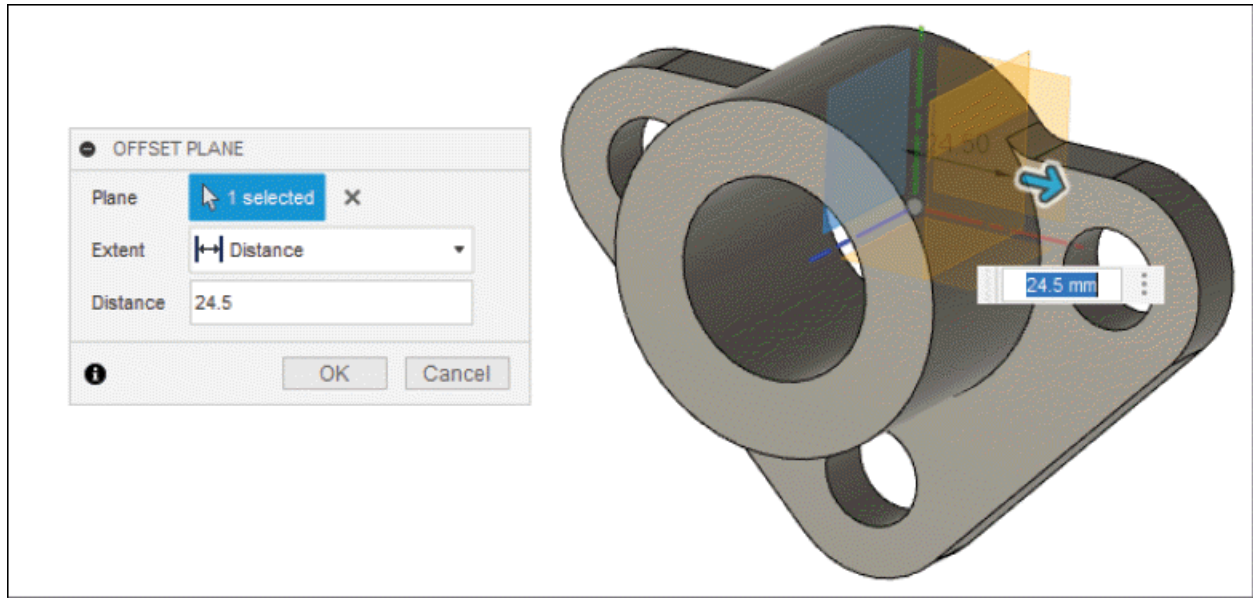


Figure-47. Creating the offset plane

Creating the Sketch

- Select **Create Sketch** tool from **CREATE** drop-down. You are asked to select the plane.
- Select newly created offset plane from **BROWSER** as sketching plane. The sketching environment will be displayed.
- Select **Center Diameter Circle** tool of **Circle** cascading menu from **CREATE** drop-down and create two circles of diameter **15 mm** and **30 mm**; refer to [Figure-48](#).

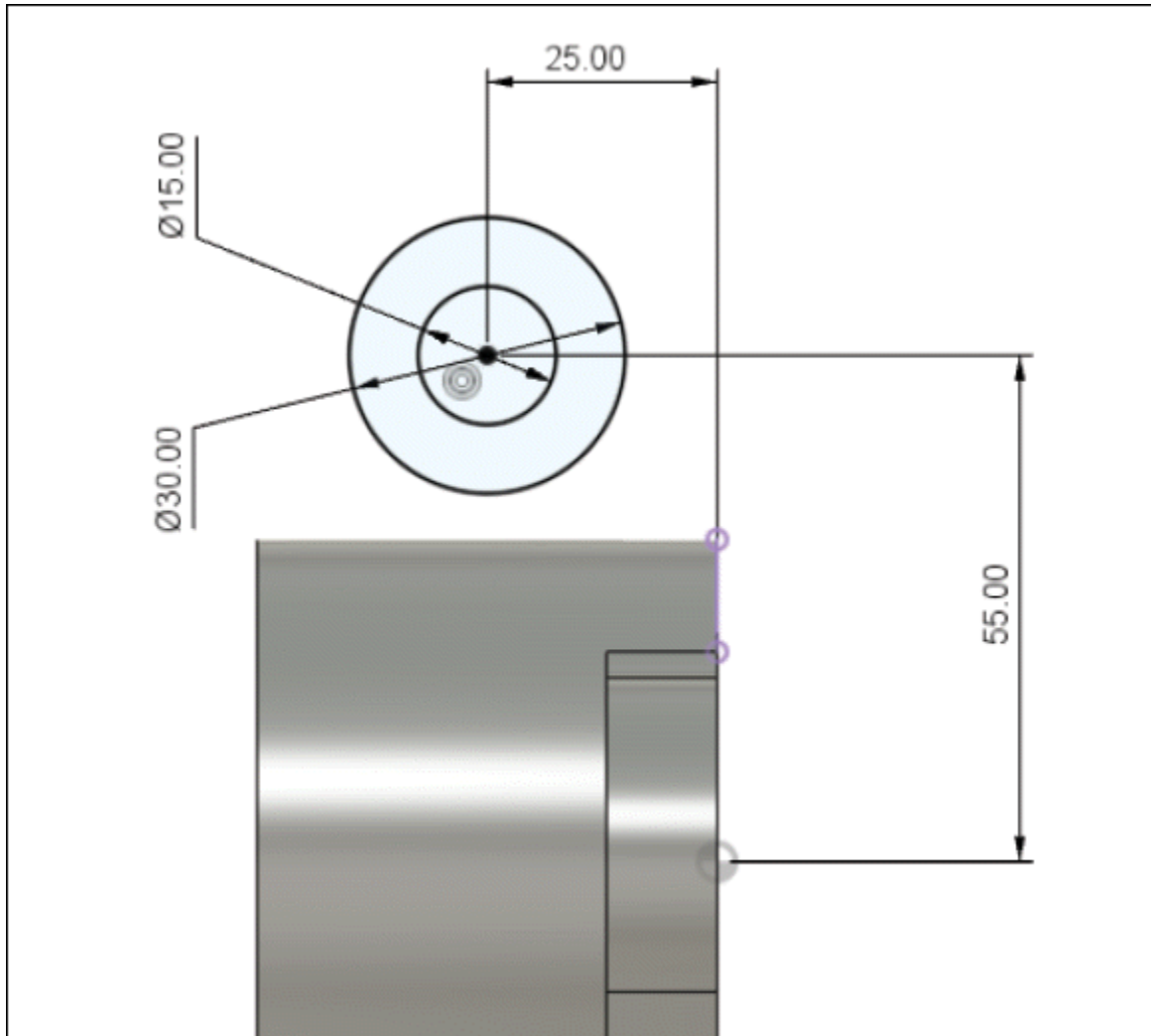


Figure-48. Creating the sketch

Intersecting the Sketch

- Select **Slice** check box from **SKETCH PALETTE** dialog box. The slice of the model will be created; refer to [Figure-49](#).

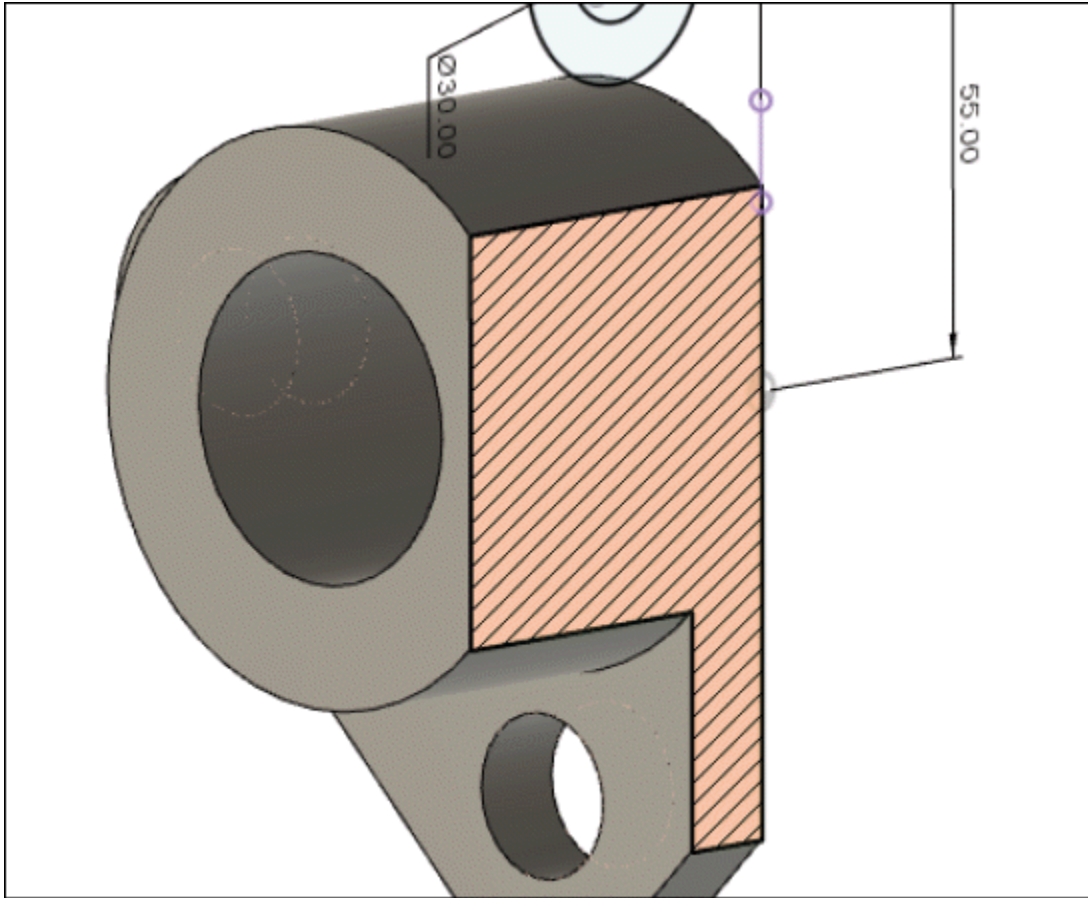


Figure-49. Slice created

- Select **Intersect** tool of **Project/Include** cascading menu from **CREATE** drop-down. The **INTERSECT** dialog box will be displayed. You are asked to select the geometries.
- The **Select** button of **Geometry** section is active by default. Select sliced part of the model as geometry to be intersect; refer to [Figure-50](#).

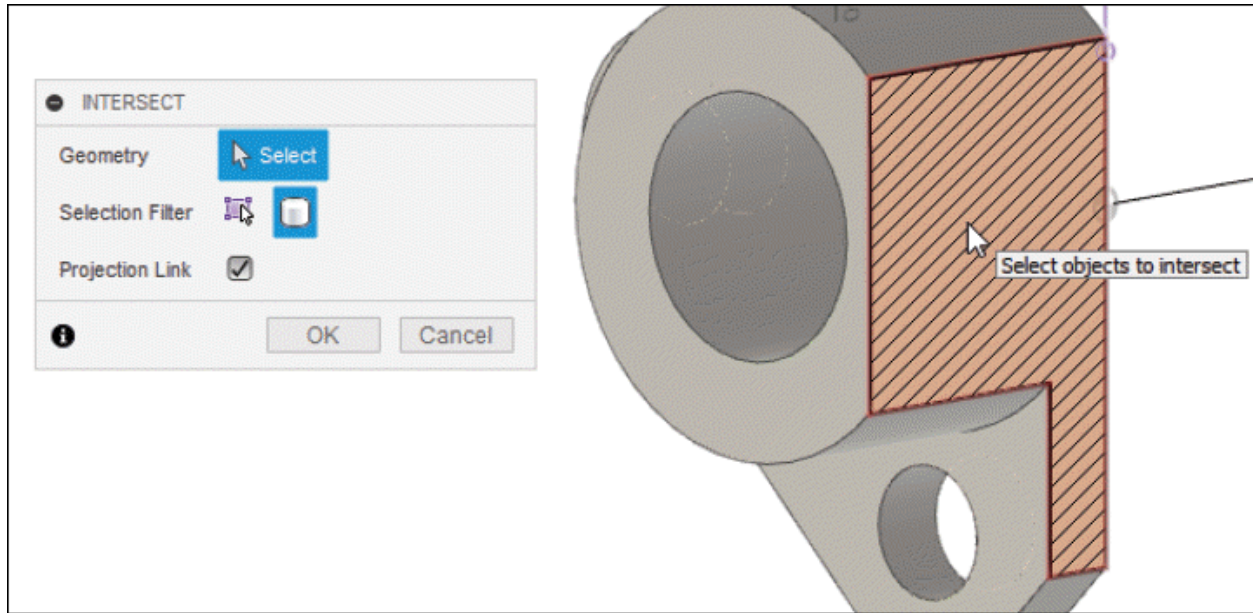


Figure-50. Geometry selected

- Click on **OK** button from **INTERSECT** dialog box to complete the process.
- Hide the body by clicking on eye icon before **Body1** in the **Bodies** node of **BROWSER**. The intersected geometry will be displayed; refer to [Figure-51](#).

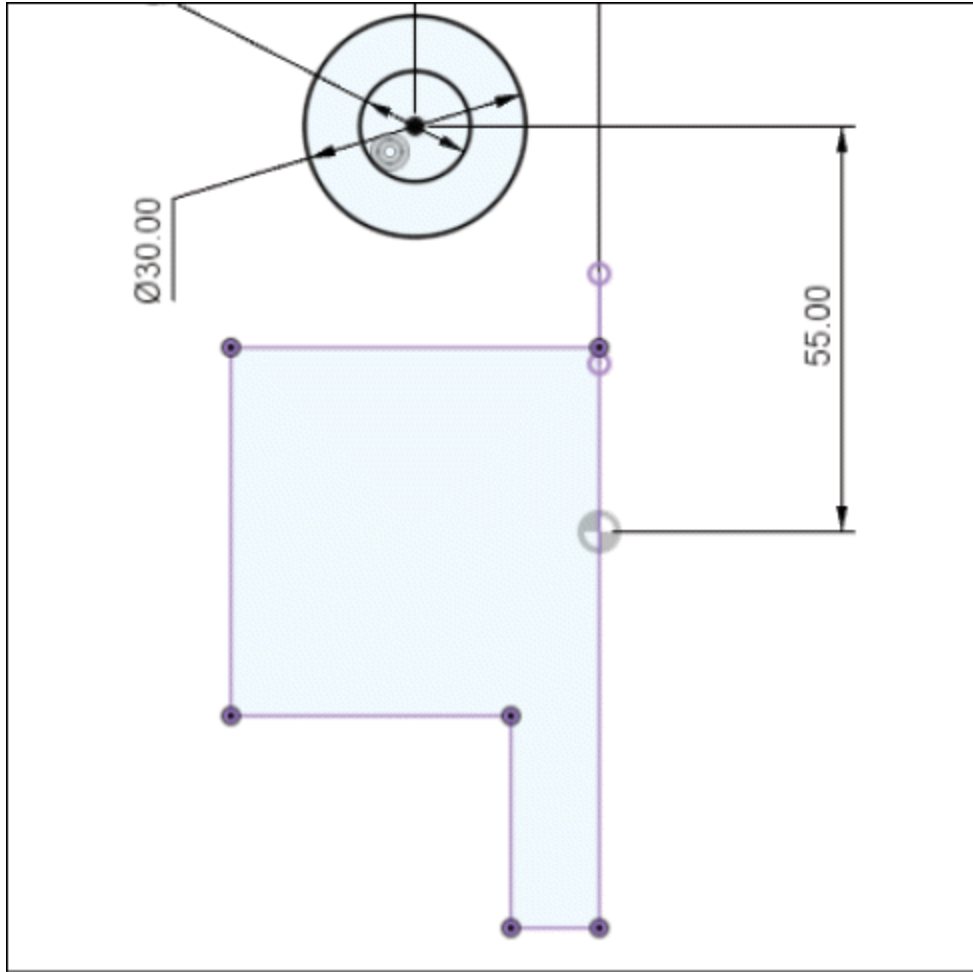


Figure-51. Intersected geometry

- Select **Line** tool from **CREATE** drop-down and create the lines as shown in [Figure-52](#).

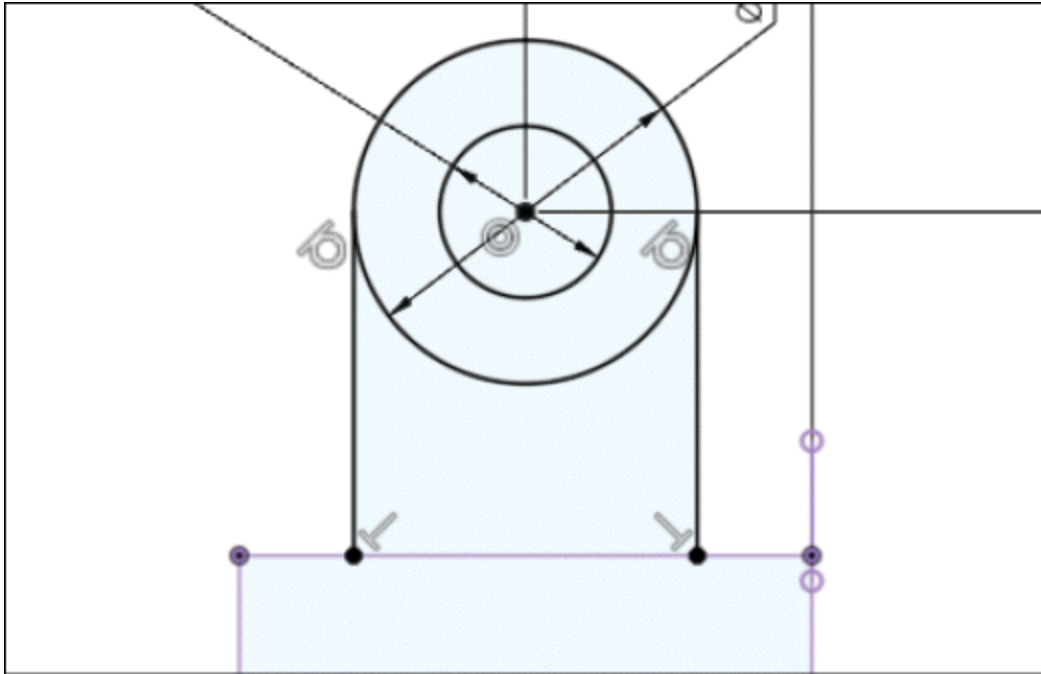


Figure-52. Lines created

- Select the intersected geometry by pressing **CTRL** key and right-click; refer to [Figure-53](#). A **Marking Menu** will be displayed.

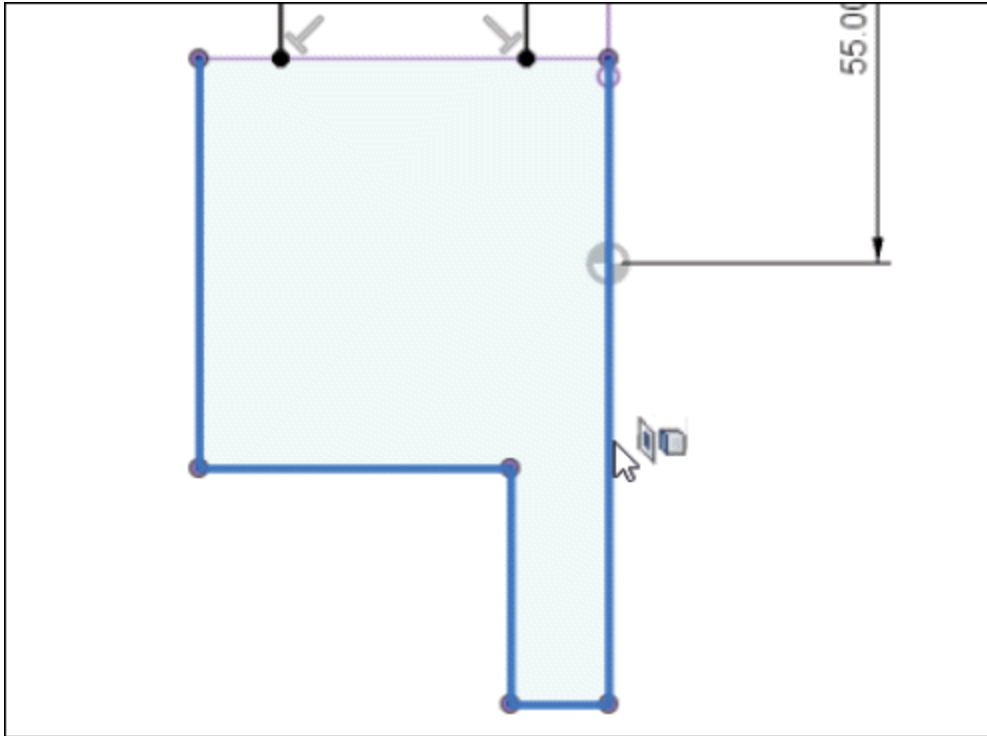


Figure-53. Selecting intersected geometry

- Select **Normal/Construction** button from **Marking Menu**.
- Click on **Finish Sketch** button from **Toolbar**.

Extruding the sketch

- Select **Extrude** tool from **CREATE** drop-down. The **EXTRUDE** dialog box will be displayed. You are asked to select the profile.
- The **Select** button of **Profile** section is active by default. Select the sketch to extrude as shown in [Figure-54](#).

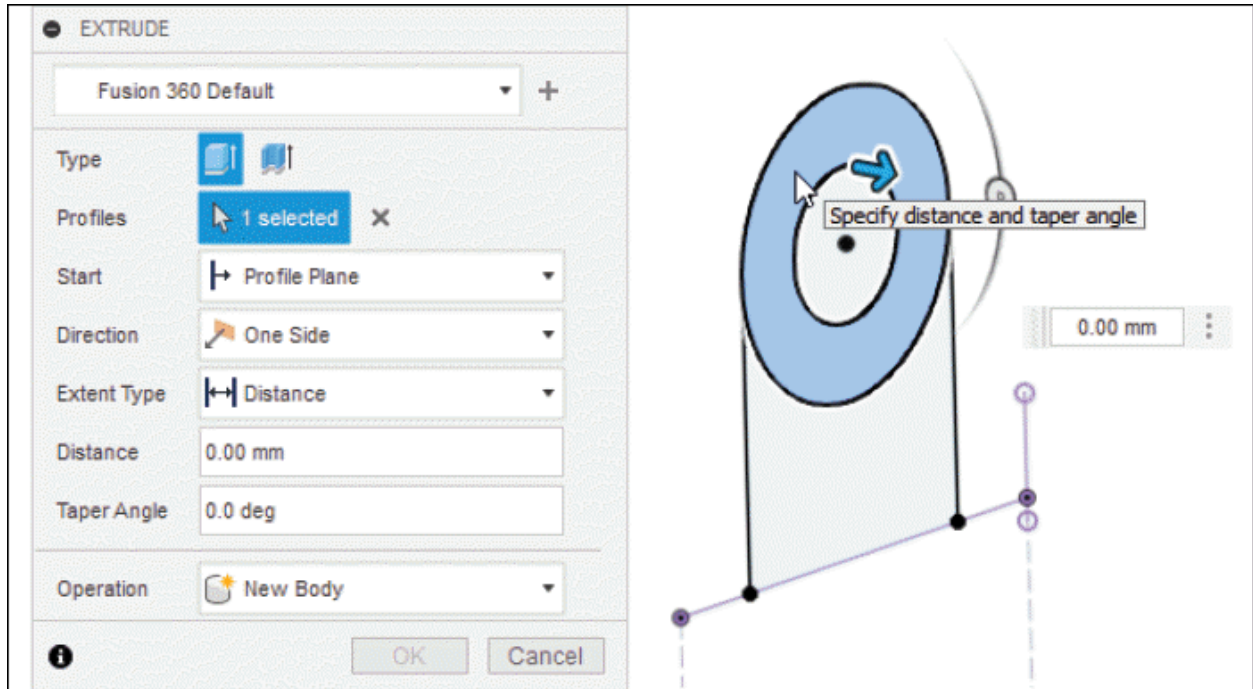


Figure-54. Selecting the sketch to extrude

- Specify the parameters as shown in the [Figure-54](#). Click in the **Distance** edit box and enter **-14** mm as distance of extrusion.
- Click on **OK** button from **EXTRUDE** dialog box to complete the process. The another sketch to be extruded will hide automatically.
- Click on the crossed eye icon before **Sketch 2** in the **Sketches** node of **BROWSER** to display the sketch again.
- Select the sketch and click **RMB**. A **Marking Menu** will be displayed. Click **Repeat Extrude** button from **Marking Menu**. The **EXTRUDE** dialog box will be displayed again along with profile selected for extrusion.
- Specify the parameters as specified for previous extrusion;

refer to [Figure-55](#).

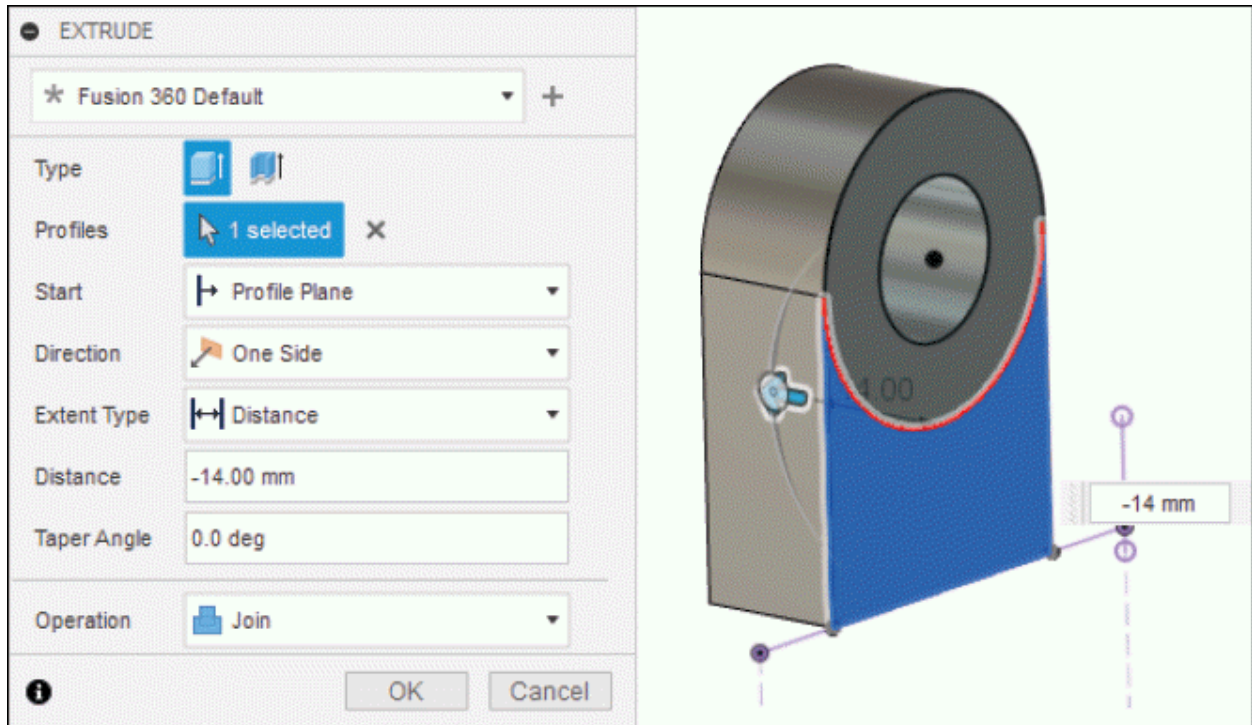


Figure-55. Extruding another sketch

- Click on **OK** button from the dialog box to complete the extrusion.
- Select recently extruded sketch and click **RMB**. A **Marking Menu** will be displayed; refer to [Figure-56](#).

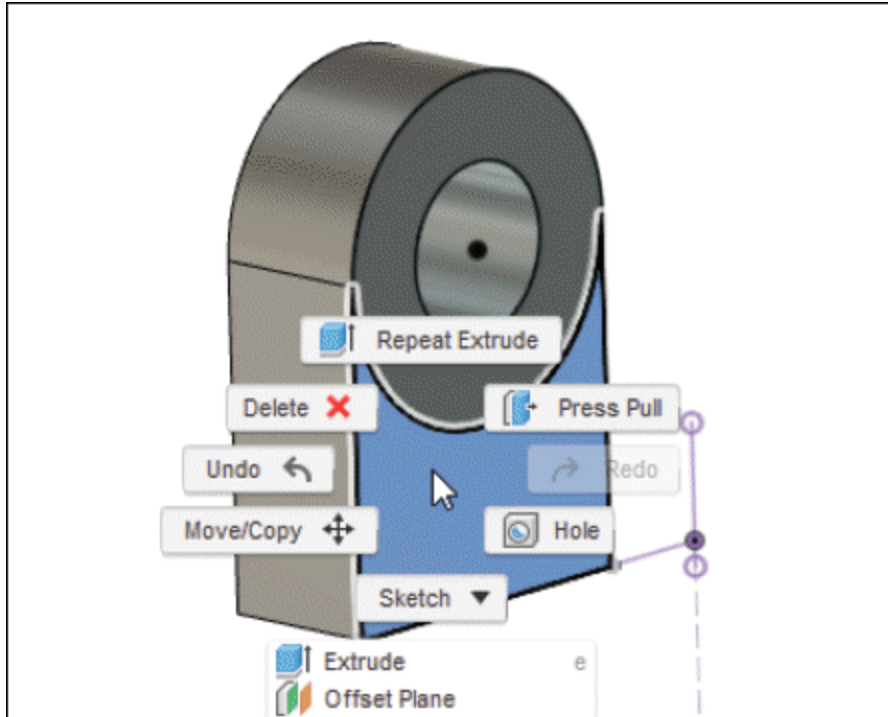


Figure-56. Selecting recently extruded sketch

- Click **Repeat Extrude** button from **Marking Menu**. The **EXTRUDE** dialog box will be displayed along with profile selected for extrusion.
- Click in the **Distance** edit box and enter **-4** as the distance for extrusion.
- Select **Cut** option from **Operation** drop-down. The preview of extrusion will be displayed; refer to [Figure-57](#).

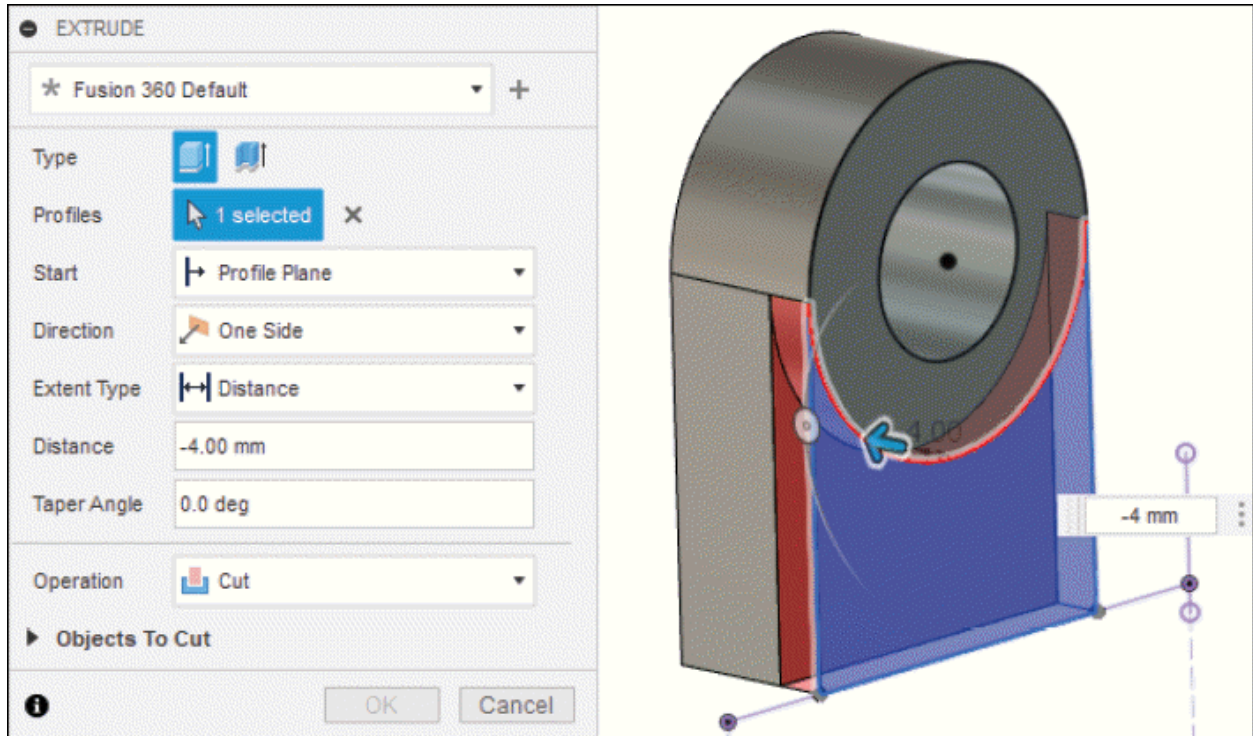


Figure-57. Preview of extrusion

- Click on **OK** button from **EXTRUDE** dialog box to complete the process.

Mirroring the body

- Click on the crossed eye icon before **Body1** in the **Bodies** node of **BROWSER** to display the body.
- Select **Mirror** tool from **CREATE** drop-down. The **MIRROR** dialog box will be displayed. You are asked to select the object.
- Select **Bodies** option from **Type** drop-down of dialog box.
- The **Select** button of **Objects** section is active by default. Select recently created body to be mirror; refer to [Figure-58](#). You are asked to select the mirror plane.

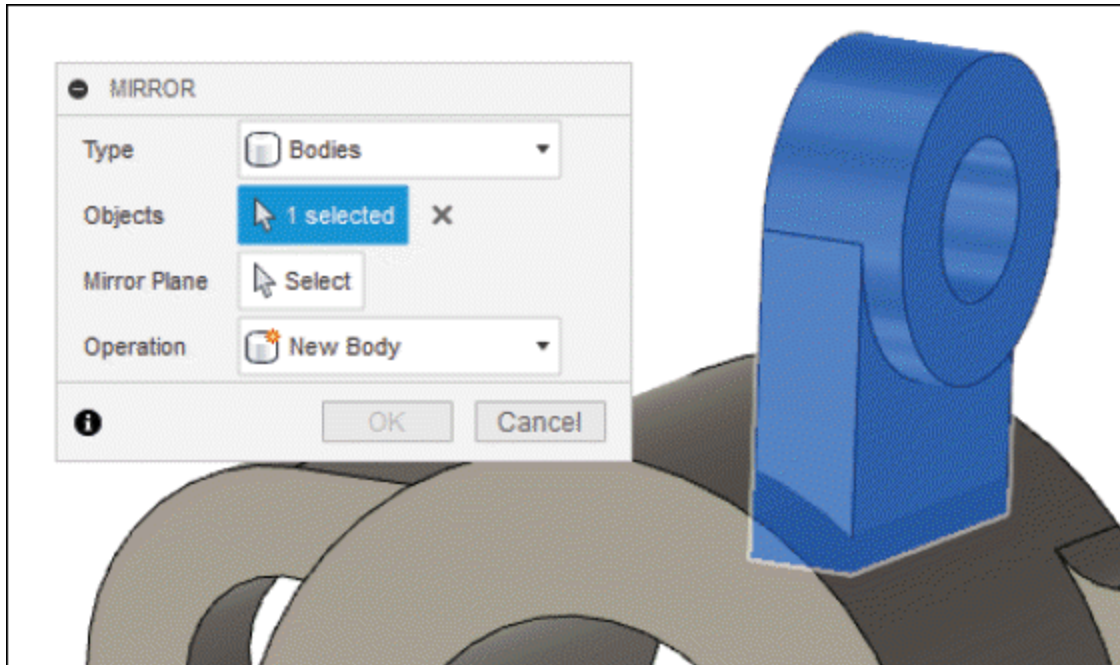


Figure-58. Body selected to be mirror

- Click on the **Select** button of **Mirror Plane** section and select **YZ** plane as mirror plane. The preview of mirror will be displayed; refer to [Figure-59](#).

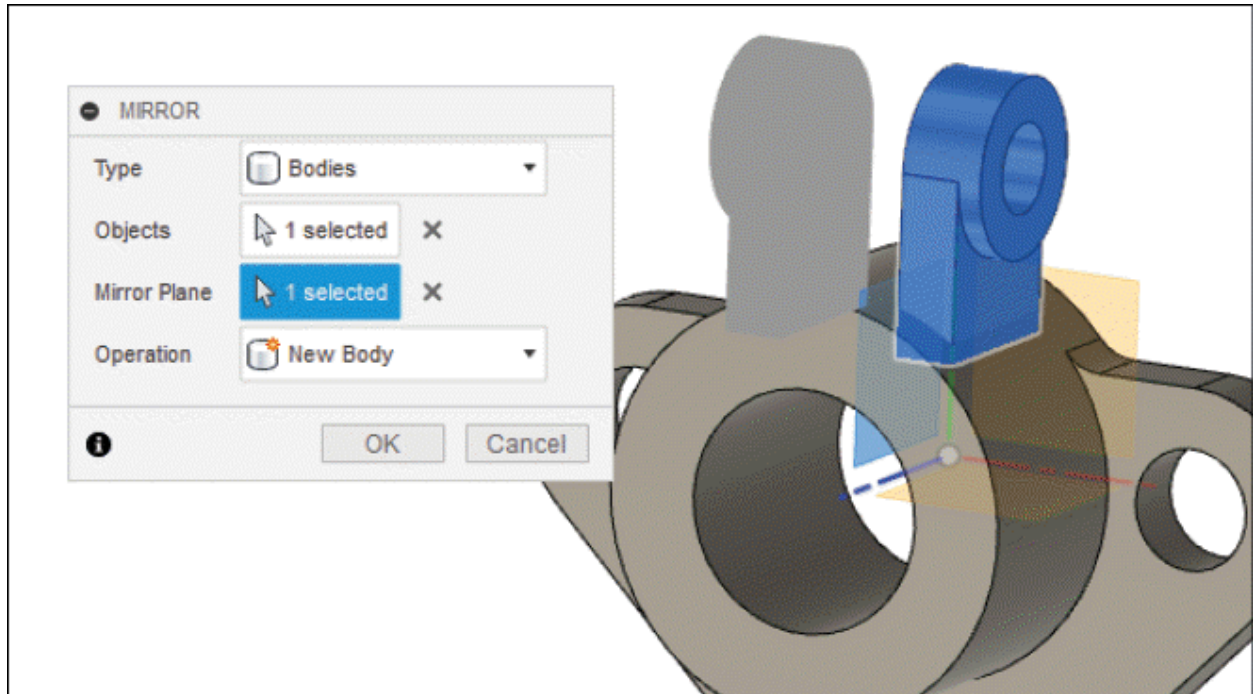


Figure-59. Preview of mirror

- Click on **OK** button from dialog box to complete the process.

Combining the bodies

- Select **Combine** tool from **MODIFY** drop-down. The **COMBINE** dialog box will be displayed. You are asked to select target body.
- The **Select** button of **Target Body** section is active by default. Select the body from the model as shown in [Figure-60](#). You are asked to select the tool body.

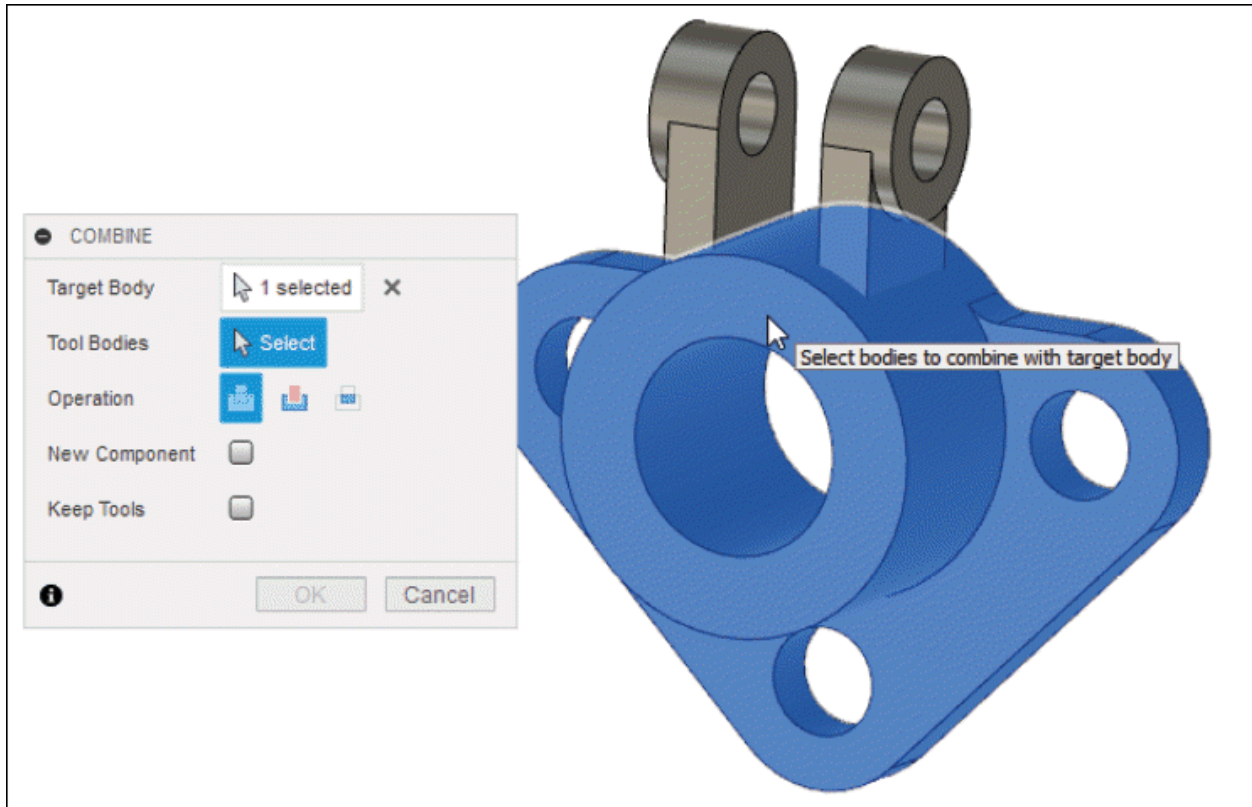


Figure-60. Selecting the target body

- The **Select** button of **Tool Bodies** section is active by default. Select the bodies to combine with target bodies as shown in [Figure-61](#).

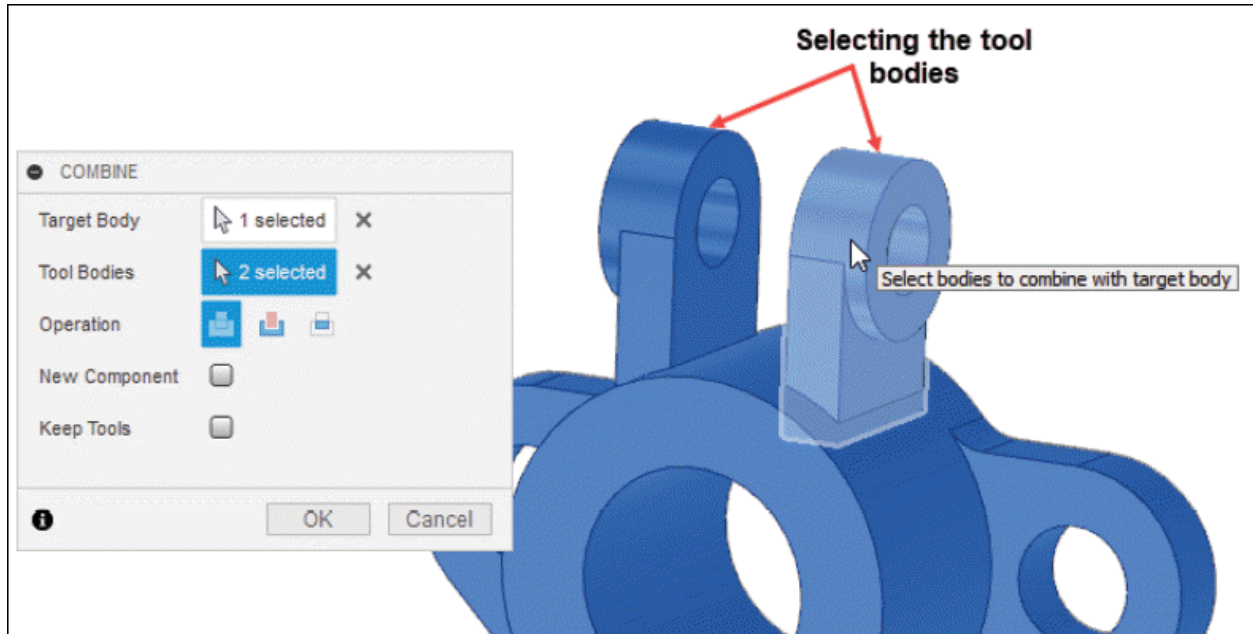


Figure-61. Selecting the tool bodies

- Click on **OK** button from the dialog box to complete the process of combining the bodies. The model will be displayed as shown in [Figure-62](#).

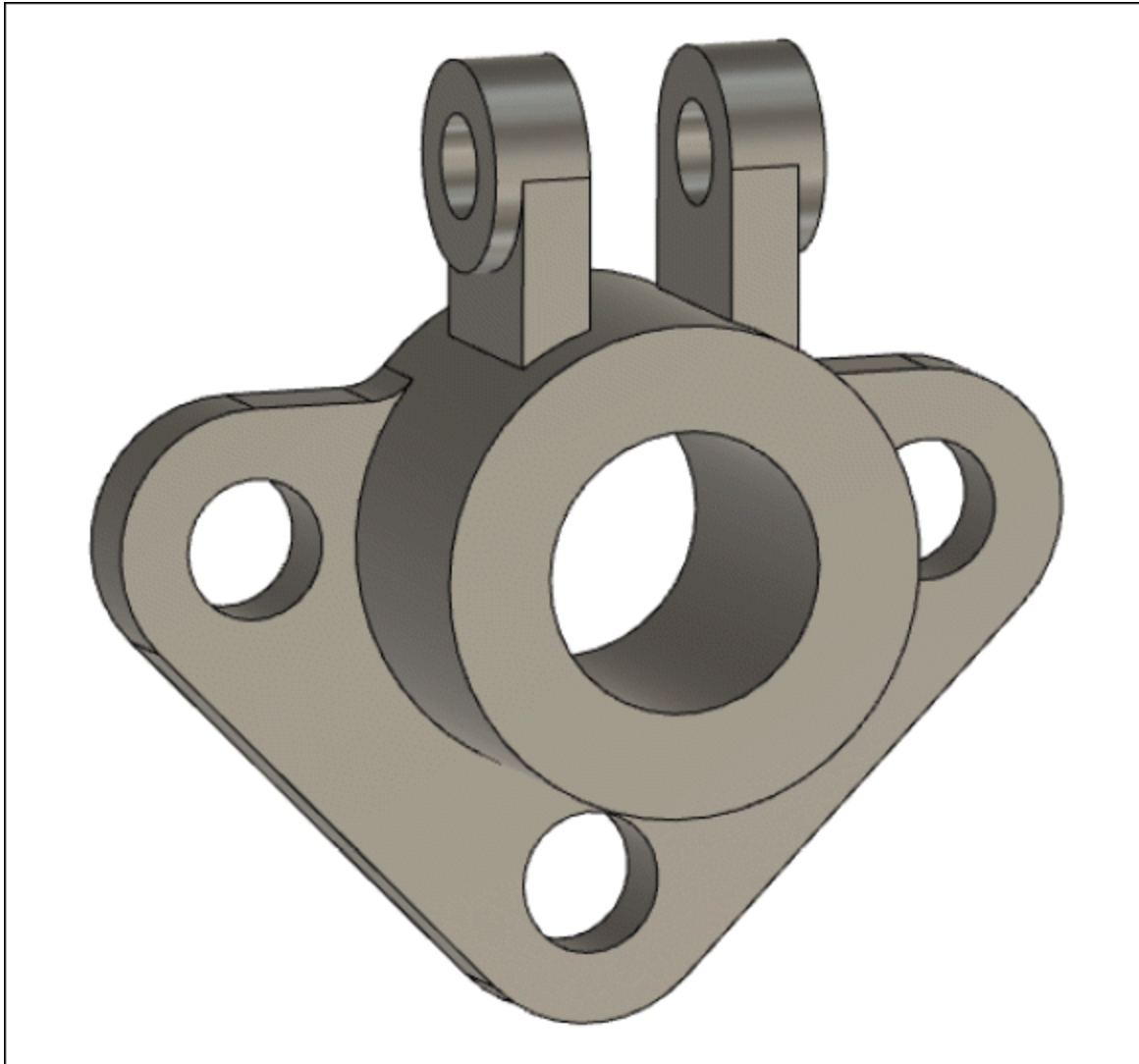


Figure-62. Final model of Practical 4

PRACTICE 1

Create the model as shown in [Figure-63](#). The dimensions are given in [Figure-64](#).

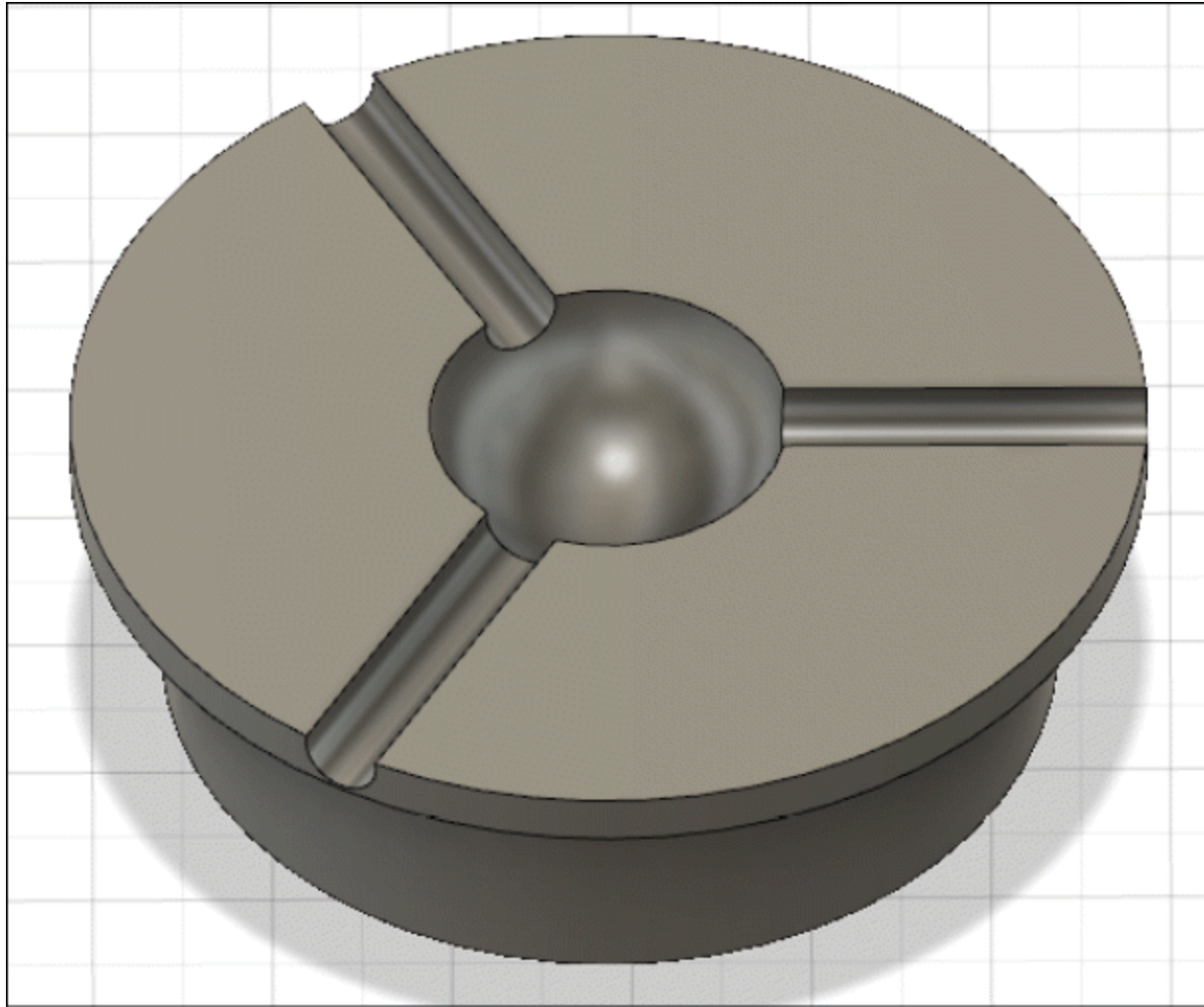


Figure-63. Model for Practice 1

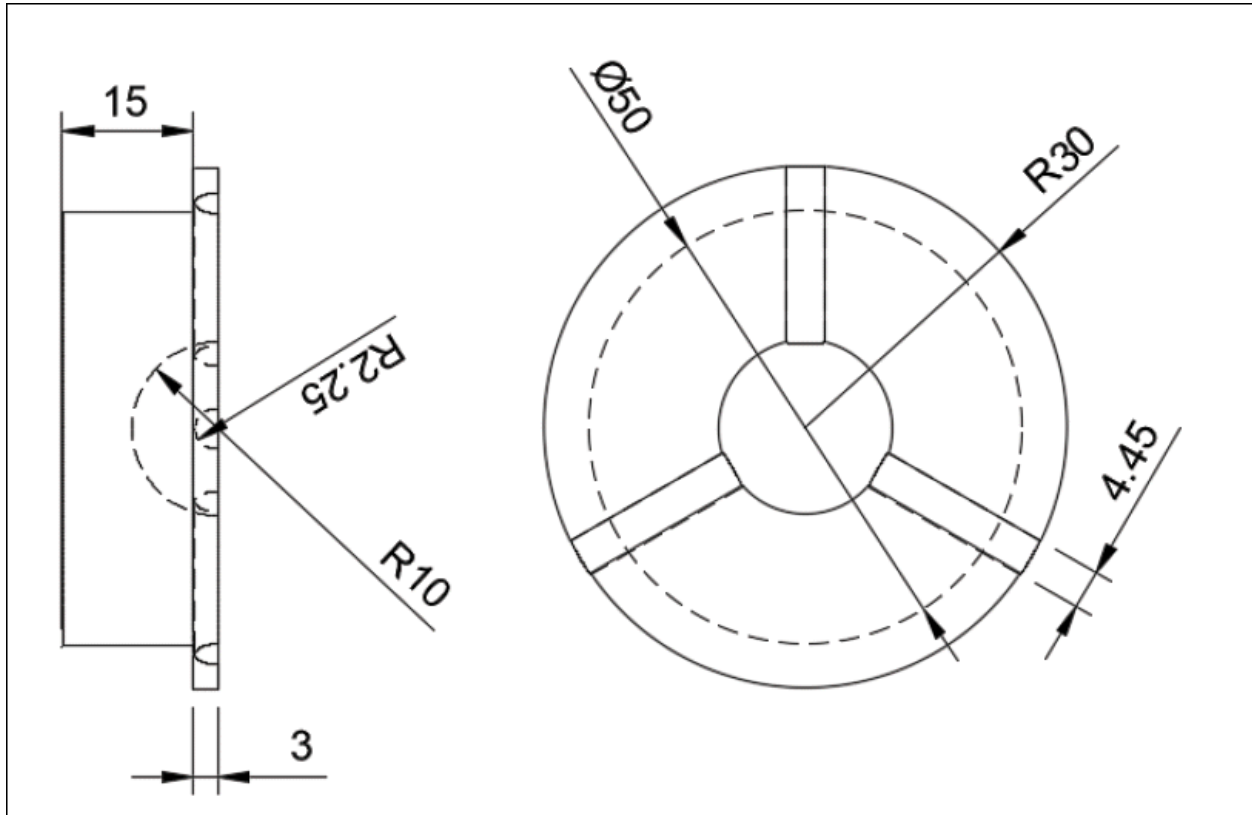


Figure-64. Dimensions for Practice 1

PRACTICE 2

Create the model as shown in [Figure-65](#). The dimensions are given in [Figure-66](#).

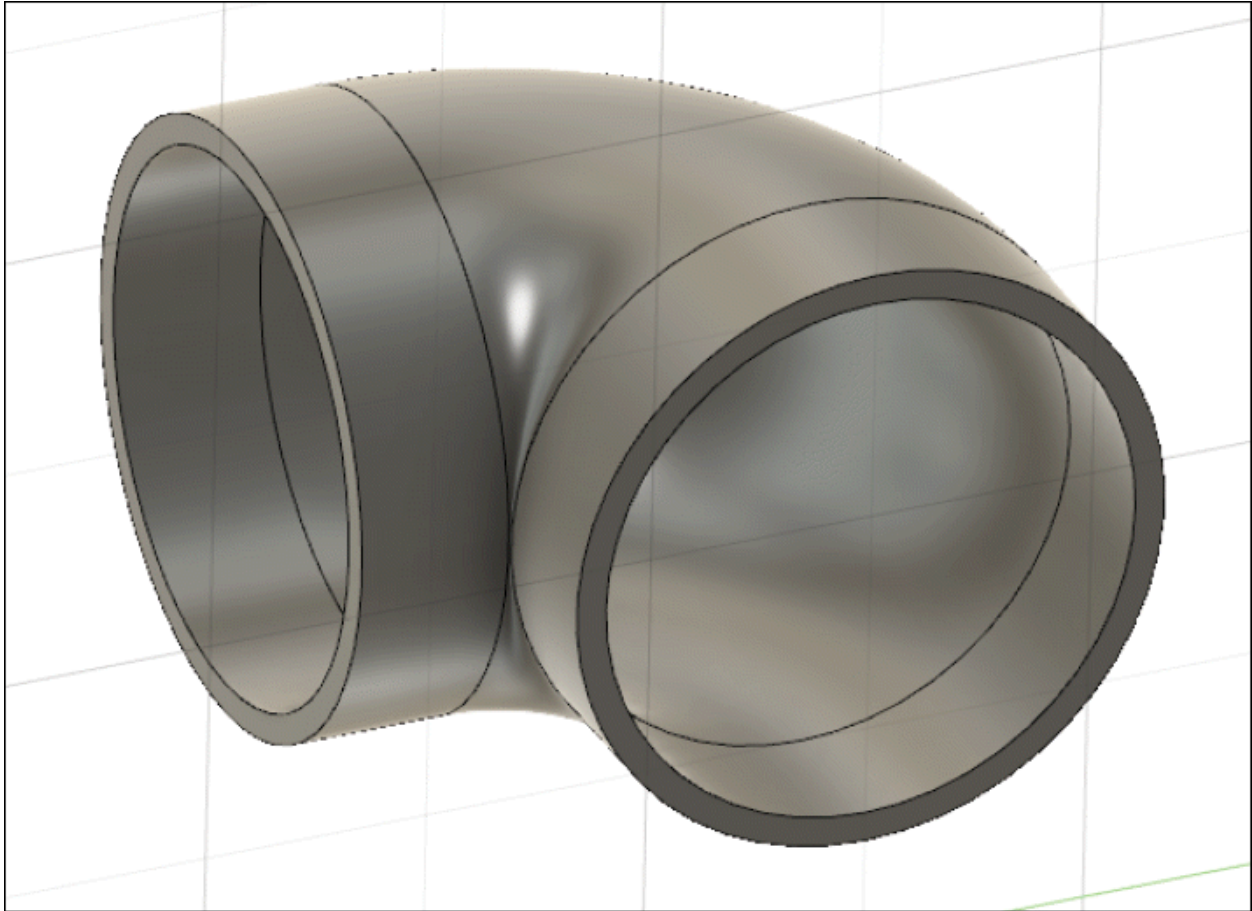


Figure-65. Model for Practice 2

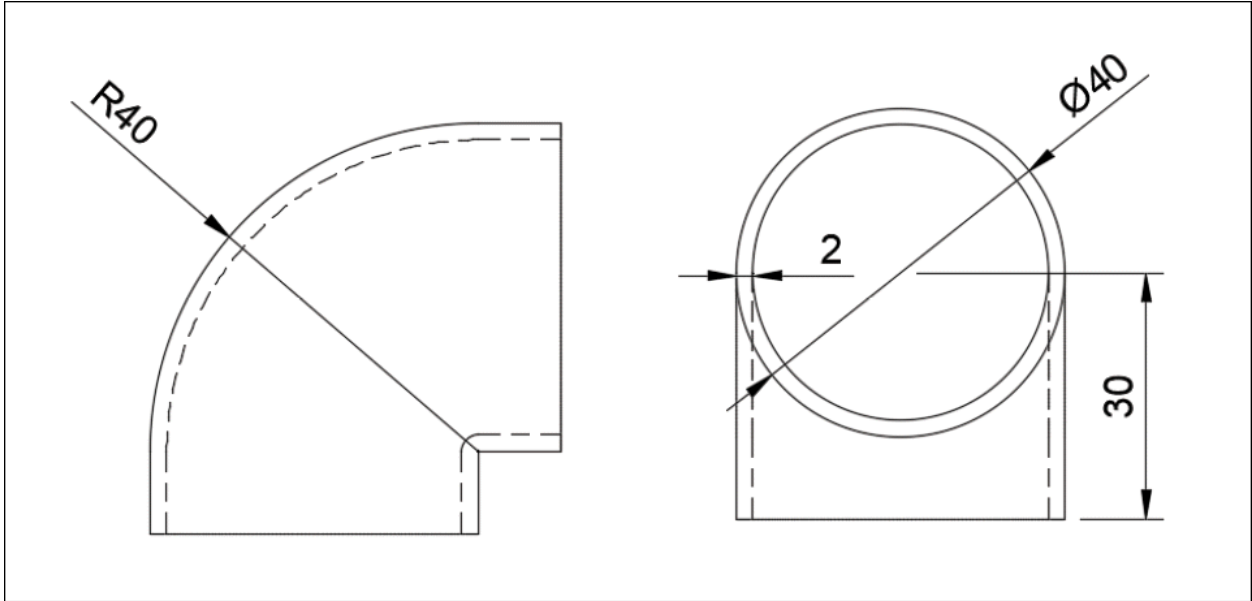


Figure-66. Dimensions for Practice 2

PRACTICE 3

Create the model as shown in [Figure-67](#). The dimensions are given in [Figure-68](#).

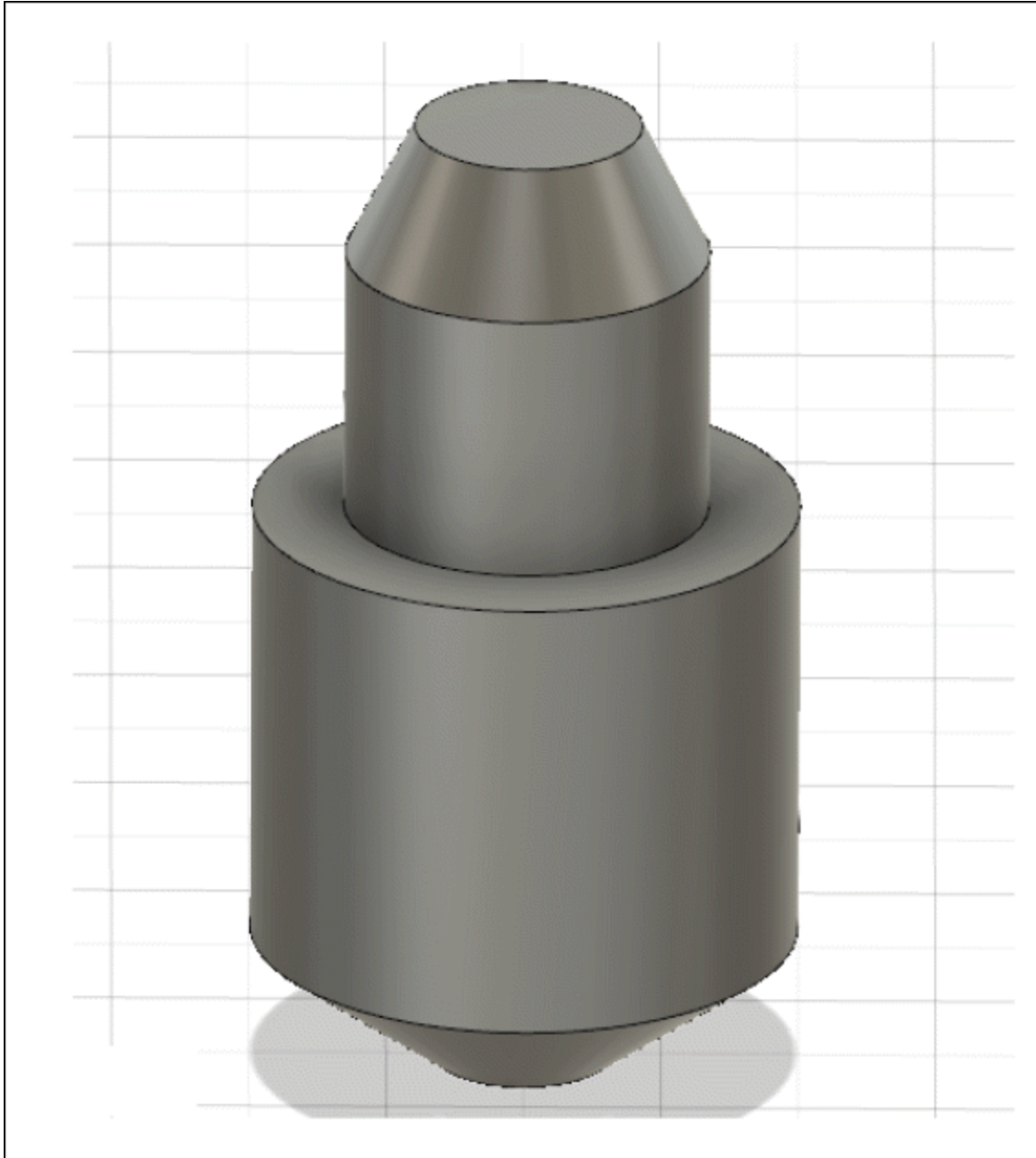


Figure-67. Model for Practice 3

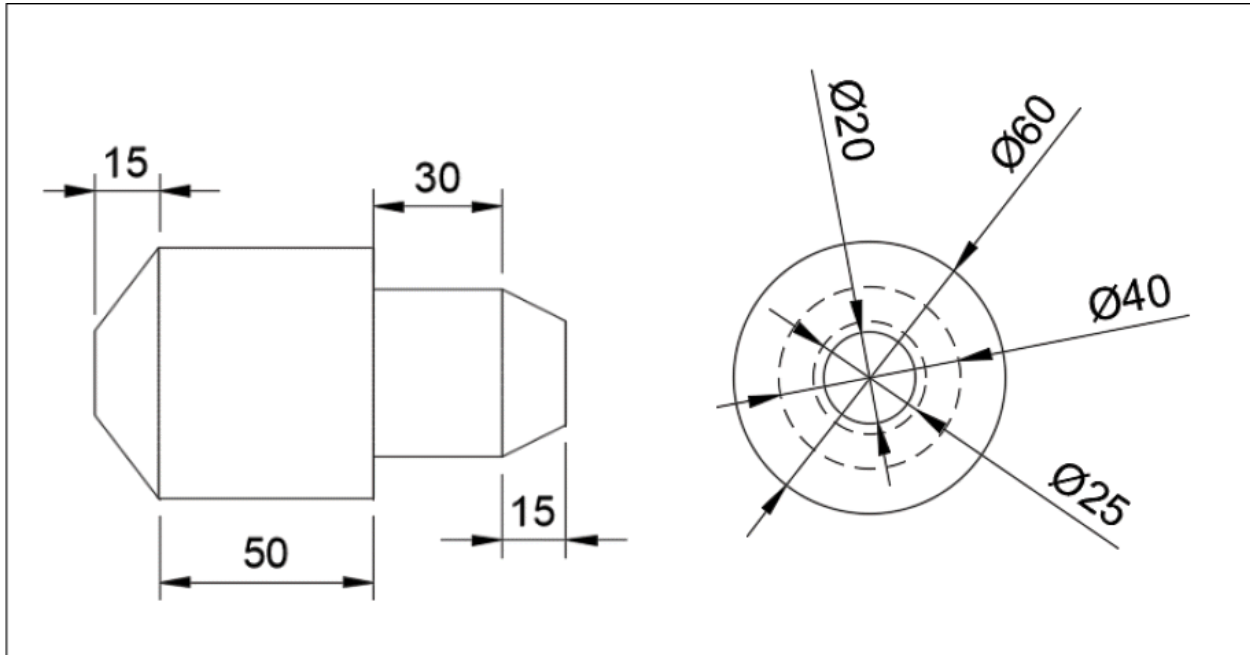


Figure-68. Dimensions for Practice 3

PRACTICE 4

Create the model as shown in [Figure-69](#). The dimensions are given in [Figure-70](#)

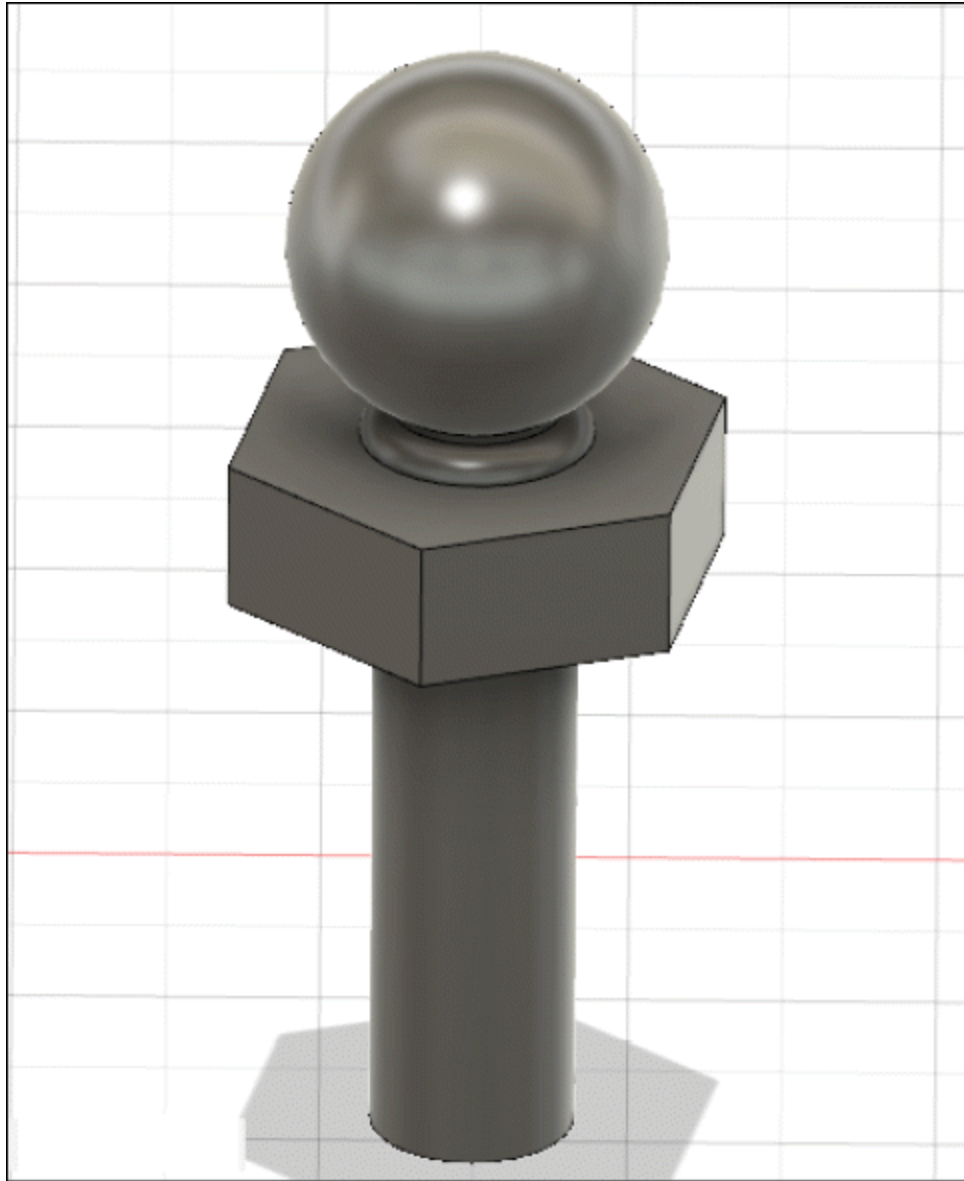


Figure-69. Model for Practice 4

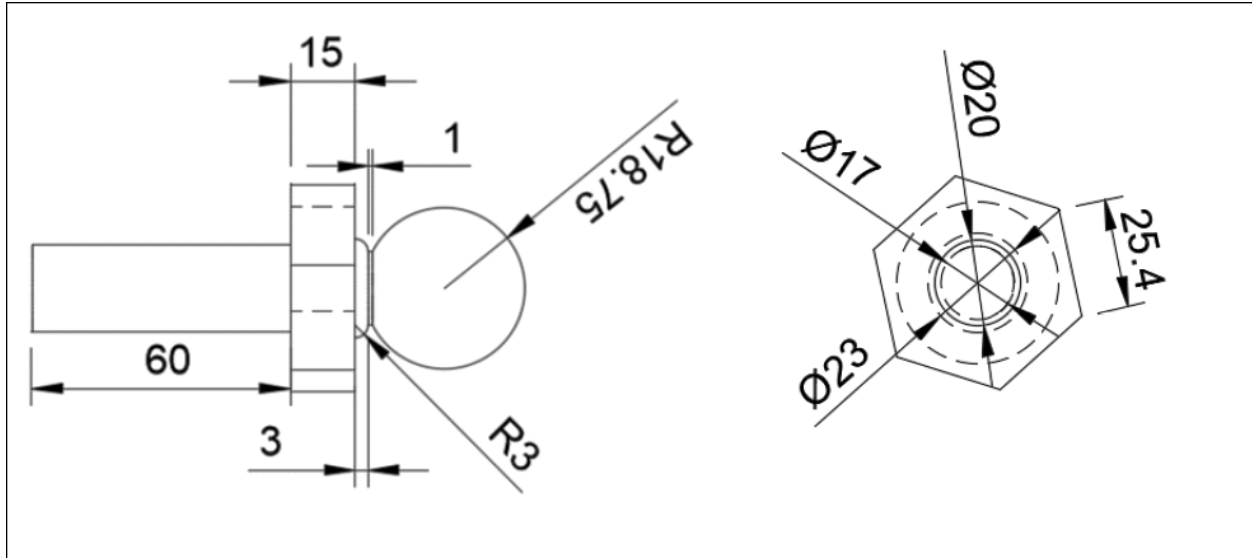


Figure-70. Dimensions for Practice 4

PRACTICE 5

Create the model as shown in [Figure-71](#). The dimensions are given in [Figure-72](#). (Hint: You will need **Rectangular Pattern** tool and **Move/Copy** tool in this model.)

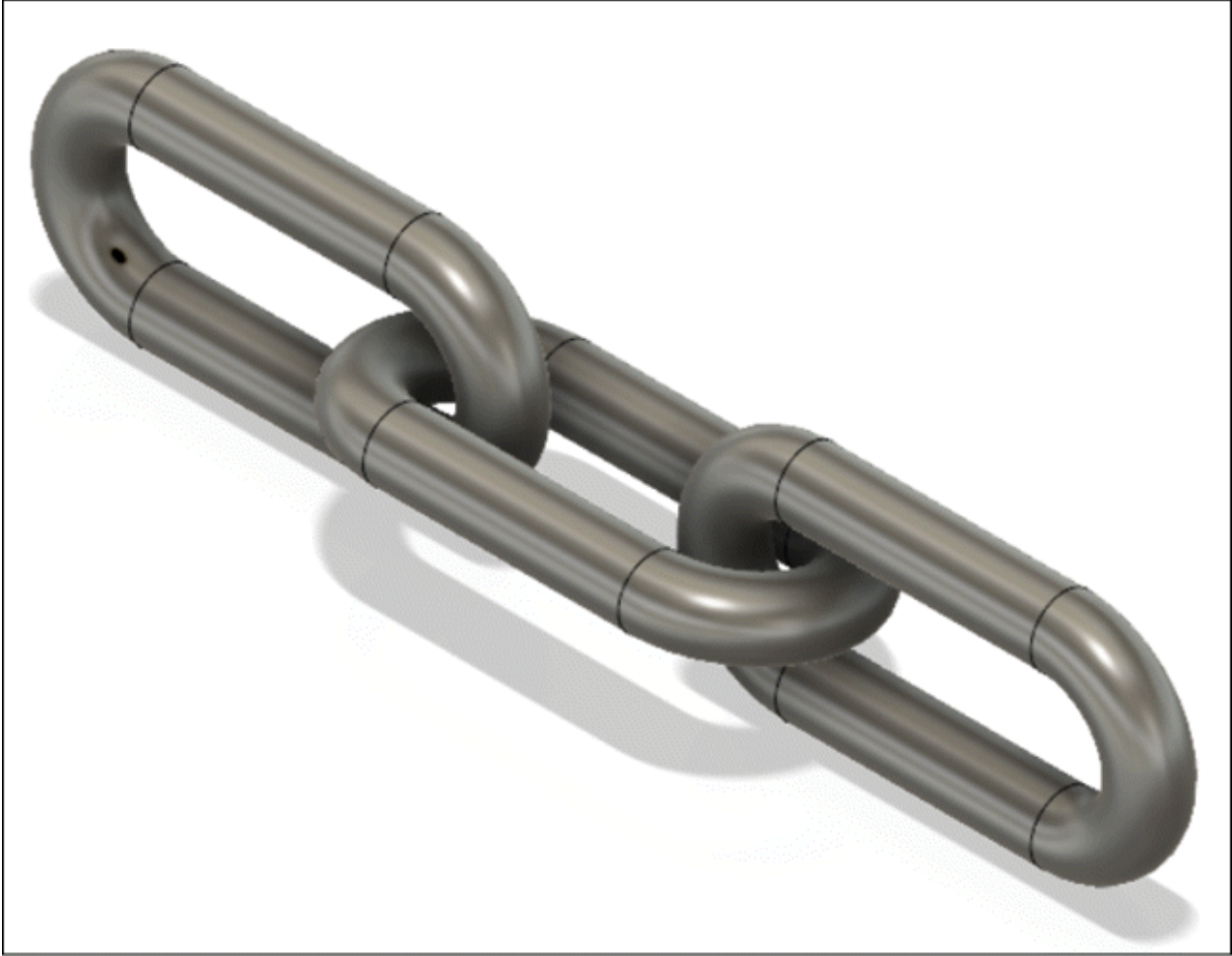


Figure-71. Model for Practice 5

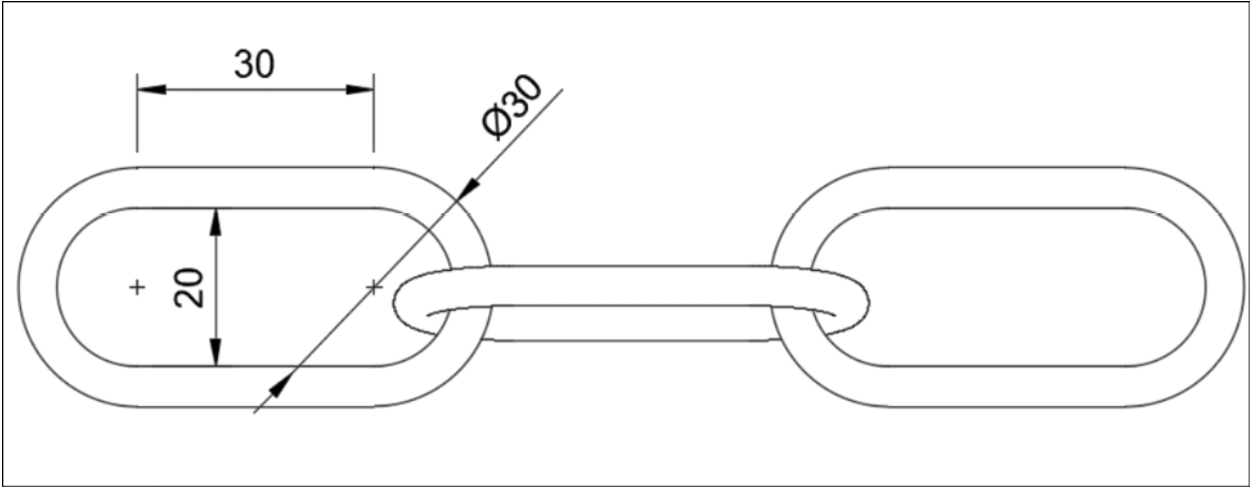


Figure-72. Dimensions for Practice 5

PRACTICE 6

Create the model as shown in [Figure-73](#). The dimensions are given in [Figure-74](#).

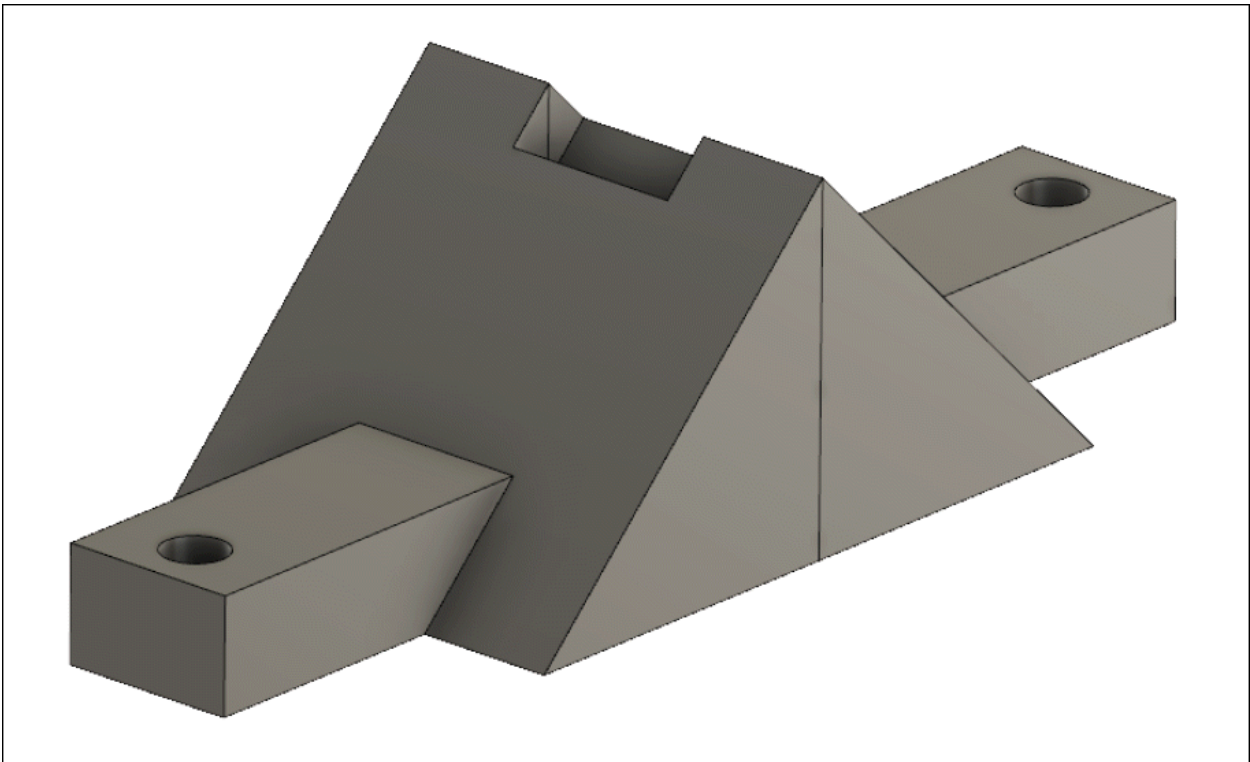


Figure-73. Model for Practice 6

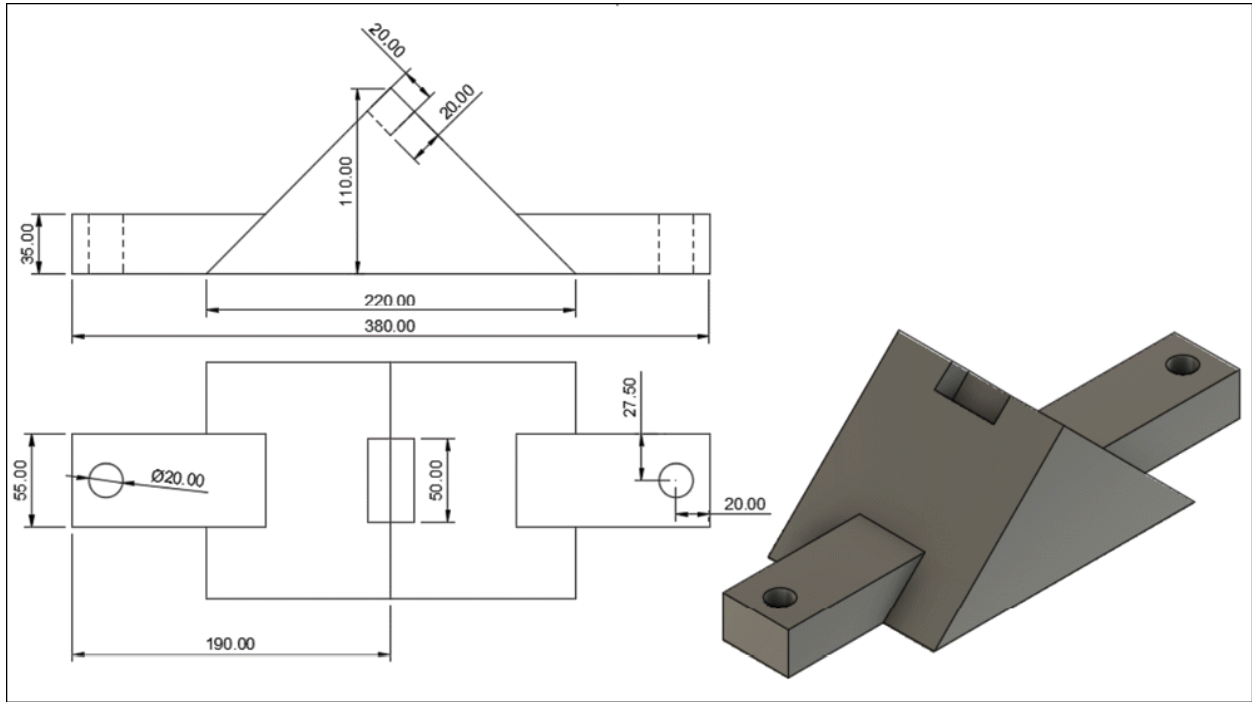


Figure-74. Dimensions for Practice 6

PRACTICE 7

Create the model as shown in [Figure-75](#). The dimensions are given in [Figure-76](#).

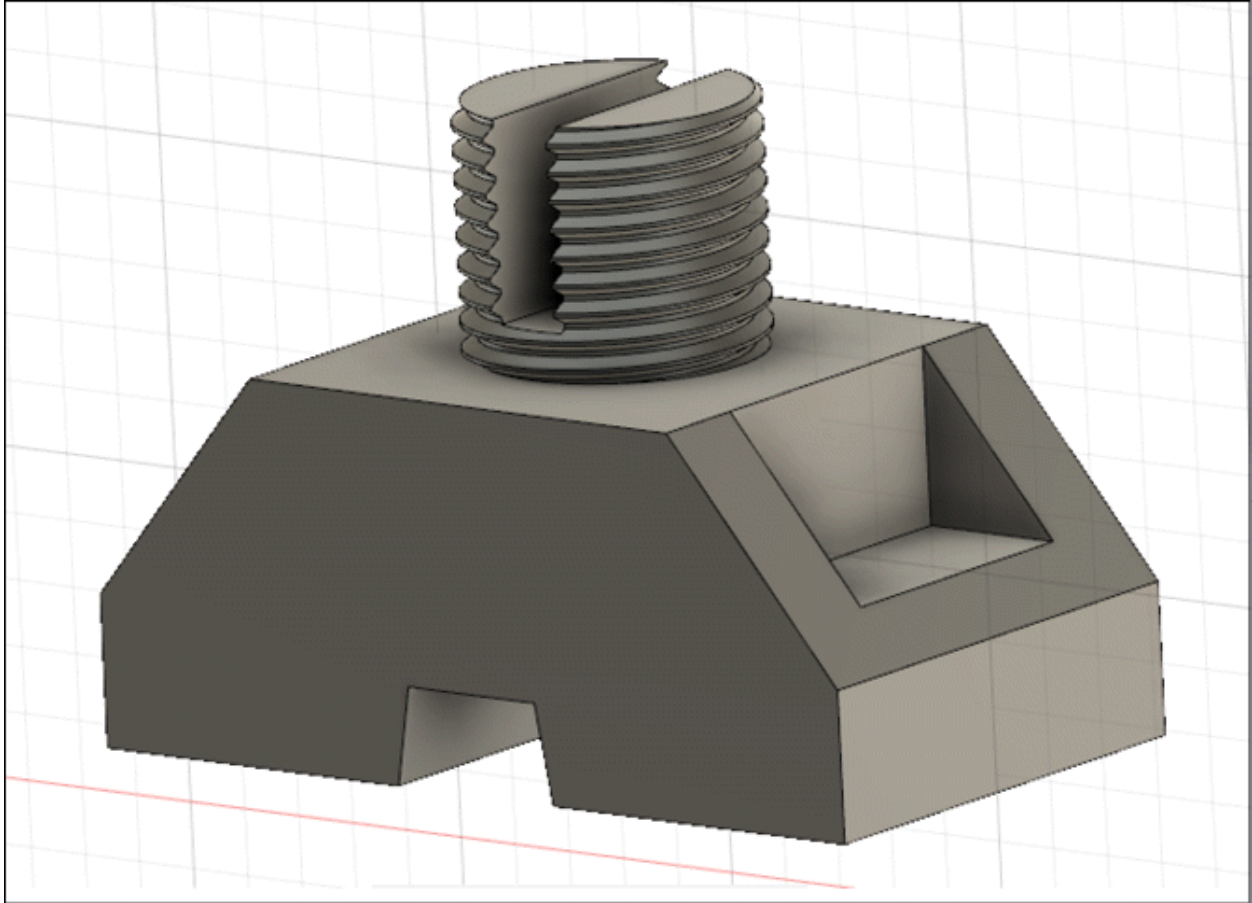
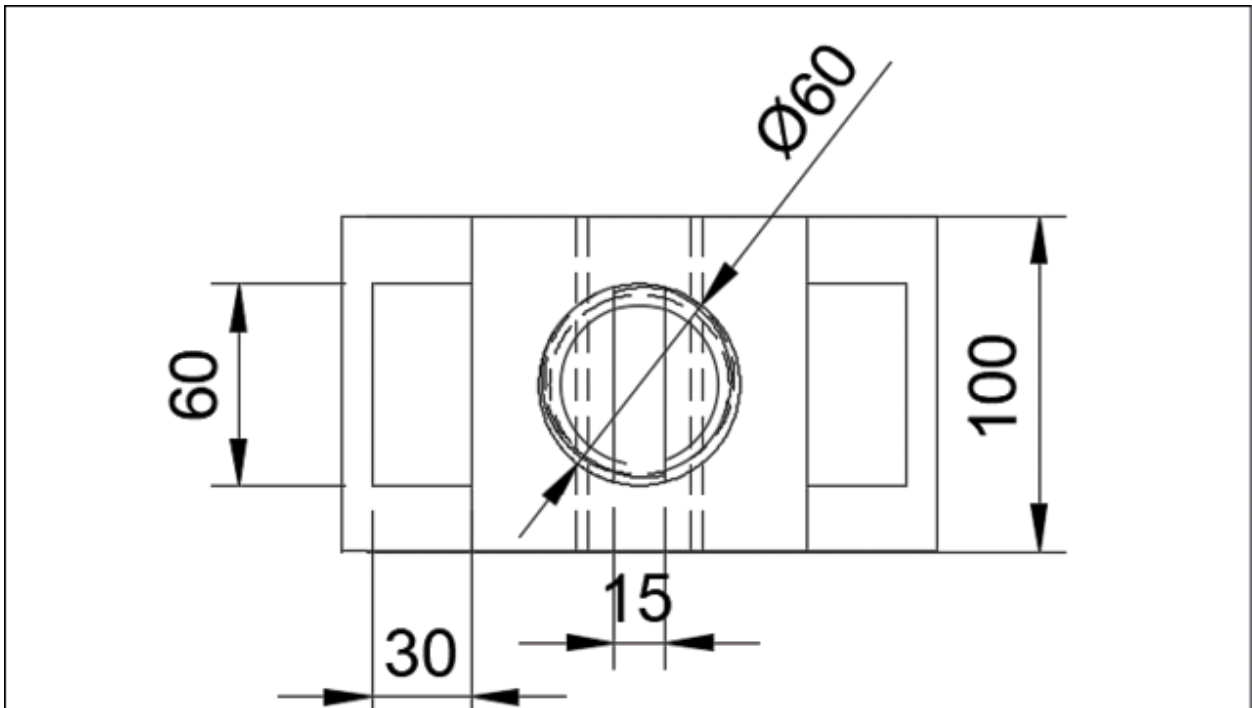


Figure-75. Model for Practice 7



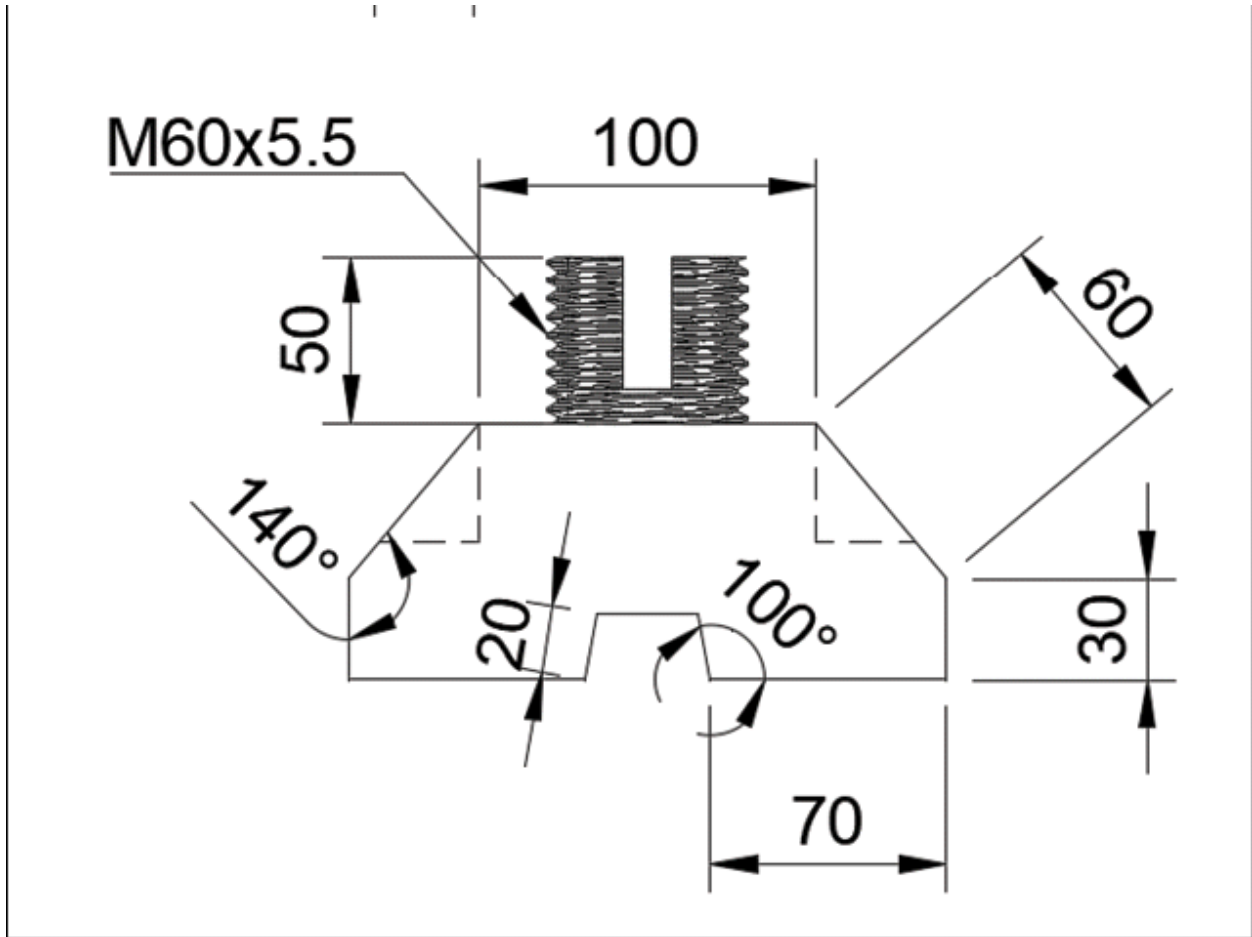


Figure-76. Dimensions for Practice 7

PRACTICE 8

Create the model as shown in [Figure-77](#). The dimensions are given in [Figure-78](#).

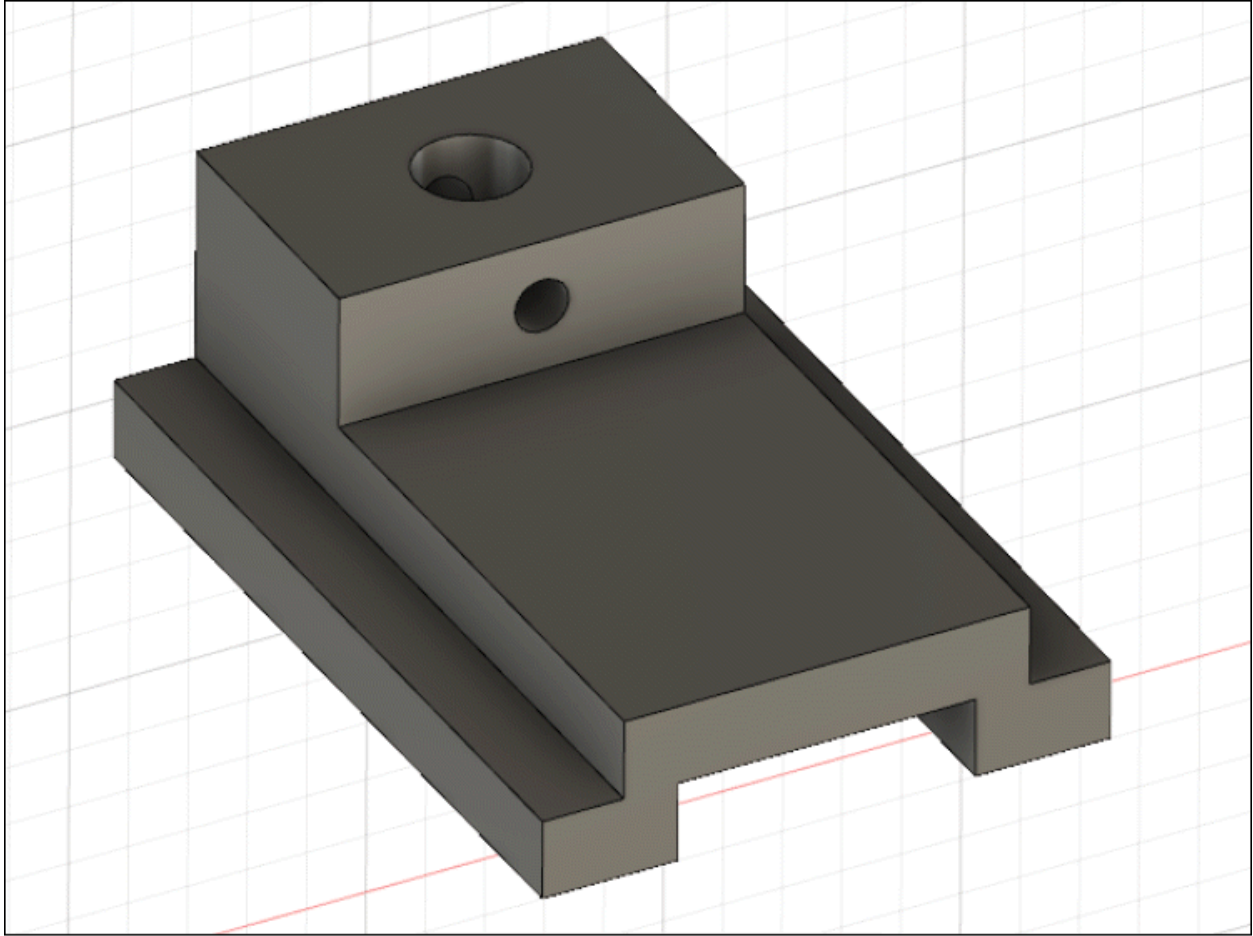


Figure-77. Model for Practice 8

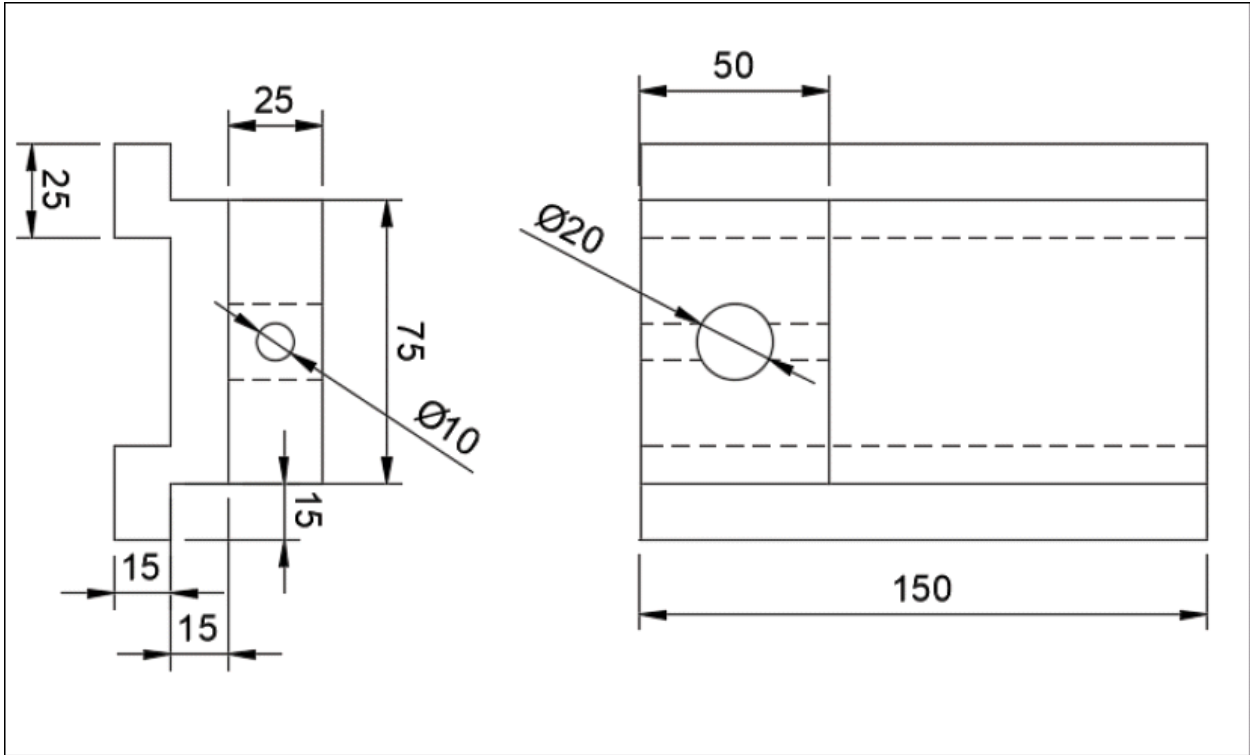


Figure-78. Dimensions for Practice 8

PRACTICE 9

Create the model as shown in [Figure-79](#). The dimensions are given in [Figure-80](#).

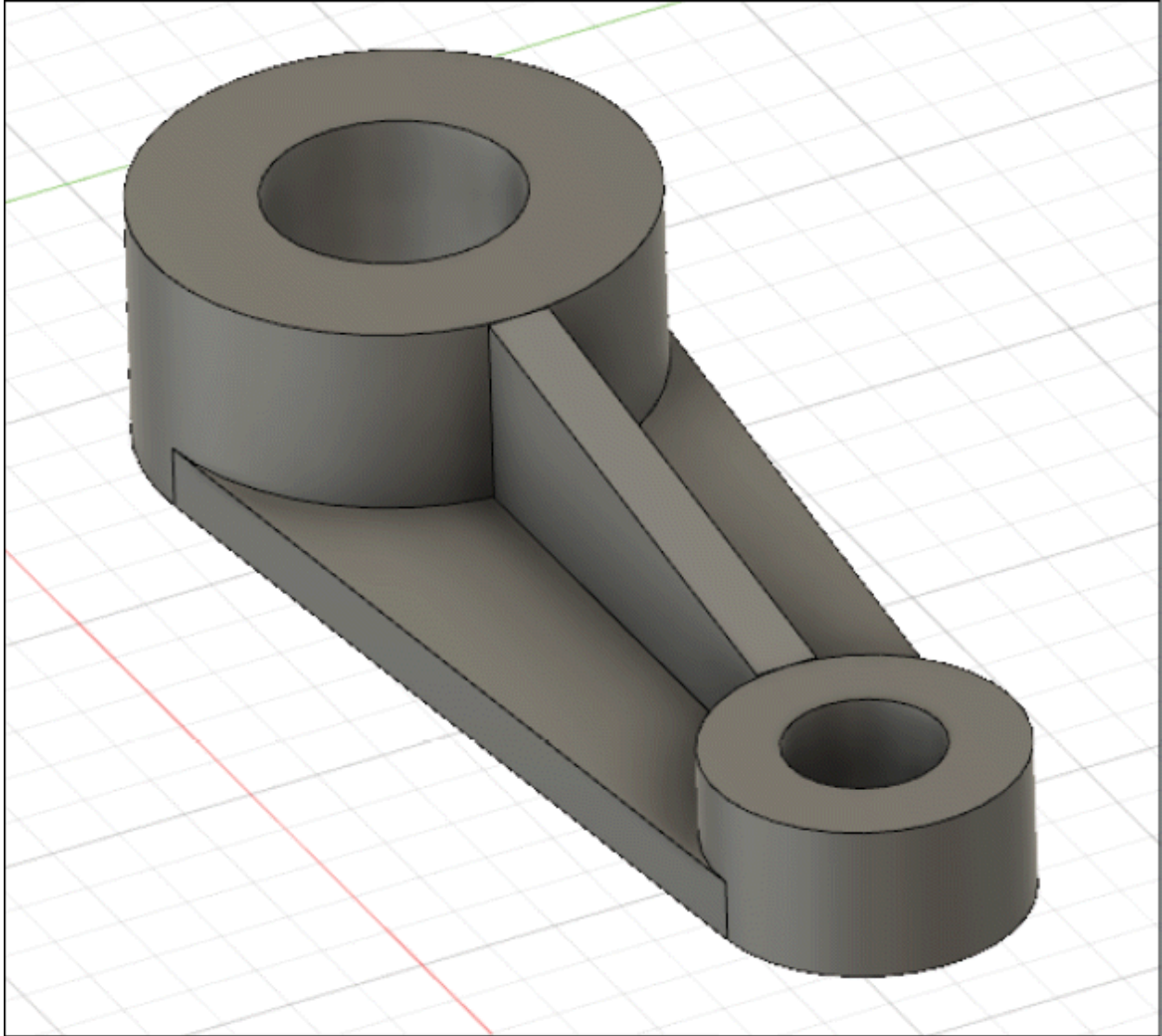


Figure-79. Model for Practice 9

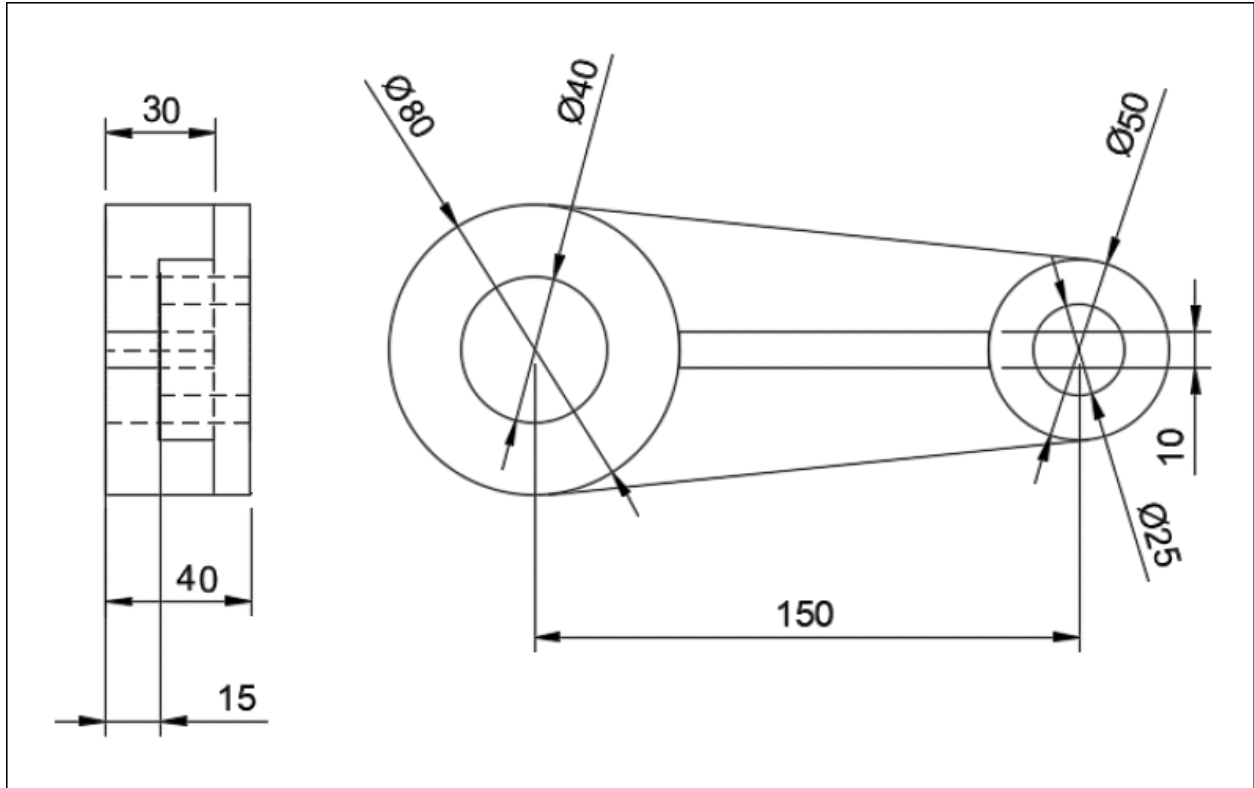


Figure-80. Dimensions for Practice 9

PRACTICE 10

Create the model as shown in [Figure-81](#). The dimensions are given in [Figure-82](#).

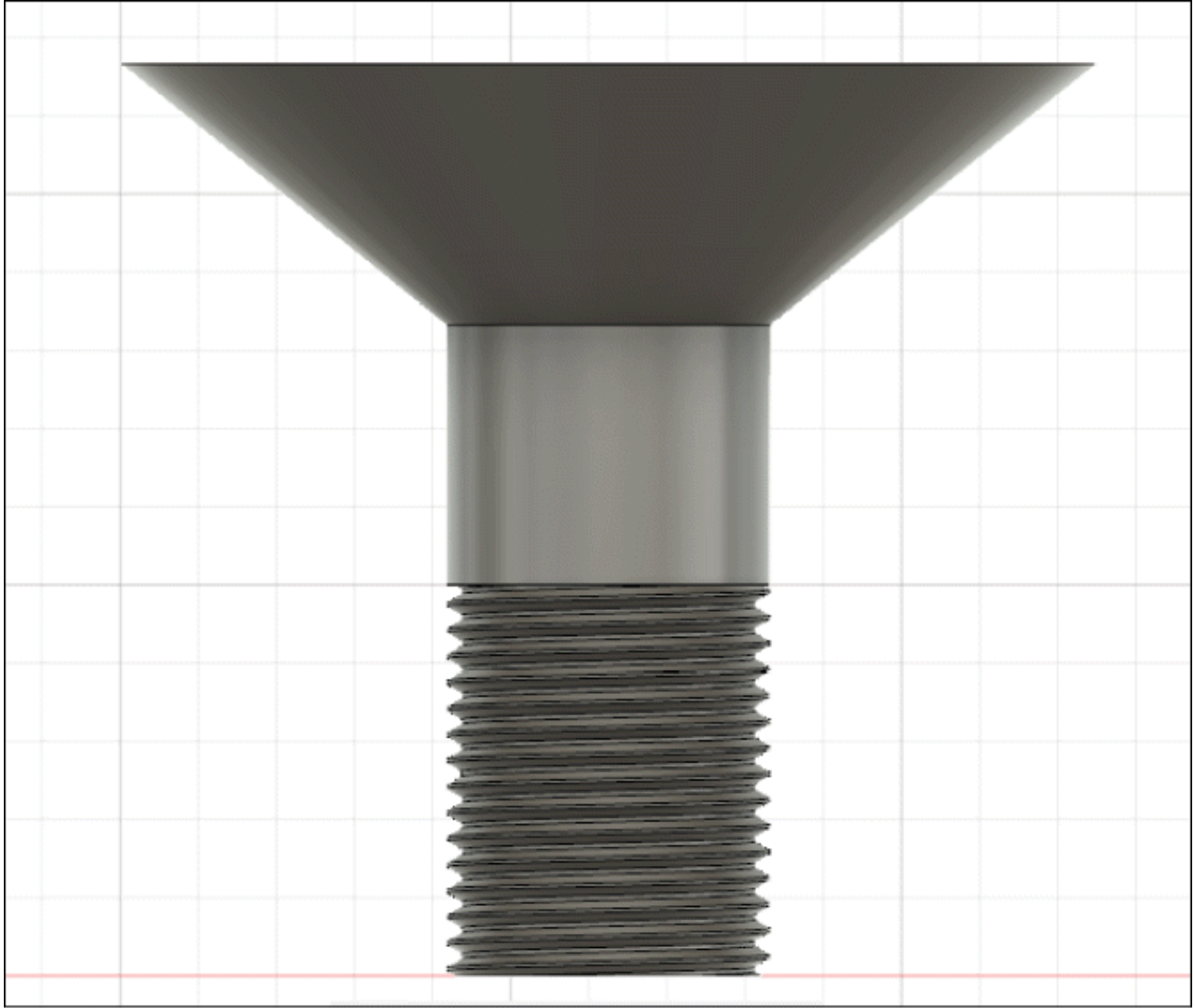


Figure-81. Model for Practice 10

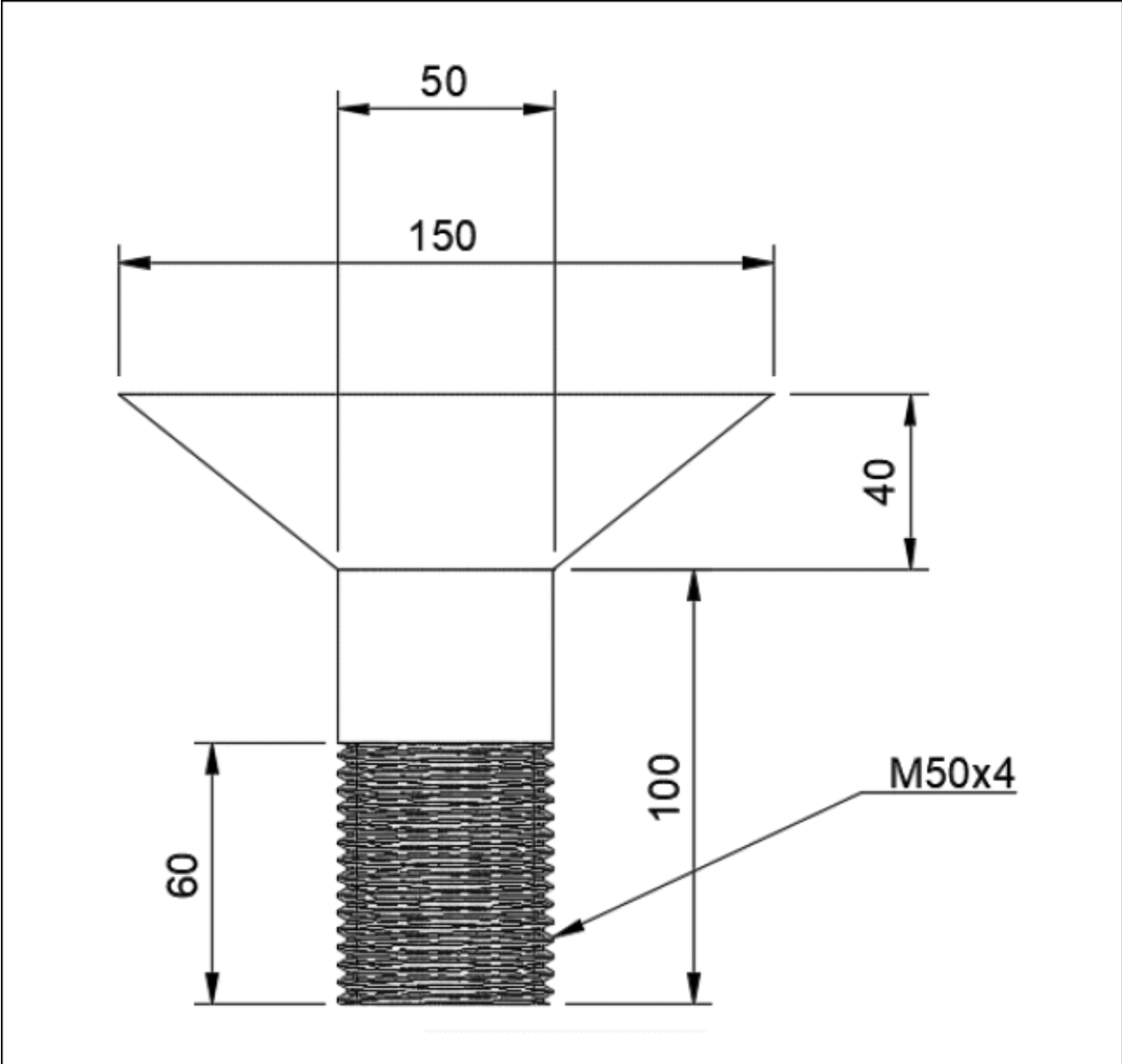


Figure-82. Dimensions for Practice 10

Chapter 6

Solid Editing

Topics Covered

The major topics covered in this chapter are:

- *Solid Editing Tools*
- *Press Pull Tool*
- *Fillet Tool*
- *Chamfer Tool*
- *Shell Tool*
- *Draft Tool*
- *Scale Tool*
- *Combine Tool*
- *Replace Face Tool*
- *Split Face Tool*
- *Split Body Tool*
- *Silhouette Split Tool*
- *Move/Copy Tool*
- *Align Tool*
- *Delete Tool*
- *Physical Material Tool*
- *Appearance Tool*
- *Manage Materials Tool*
- *Change Parameters Tool*
- *Volumetric Lattice Tool*

SOLID EDITING TOOLS

In the previous chapters, we have learned to create solid models and remove materials from them. In this chapter, we will learn to edit the models. Like the other tools, Autodesk Fusion 360 has packed all the editing tools into one drop-down. These tools are available in **MODIFY** drop-down from **Toolbar**; refer to [Figure-1](#). The tools in this drop-down are discussed next.

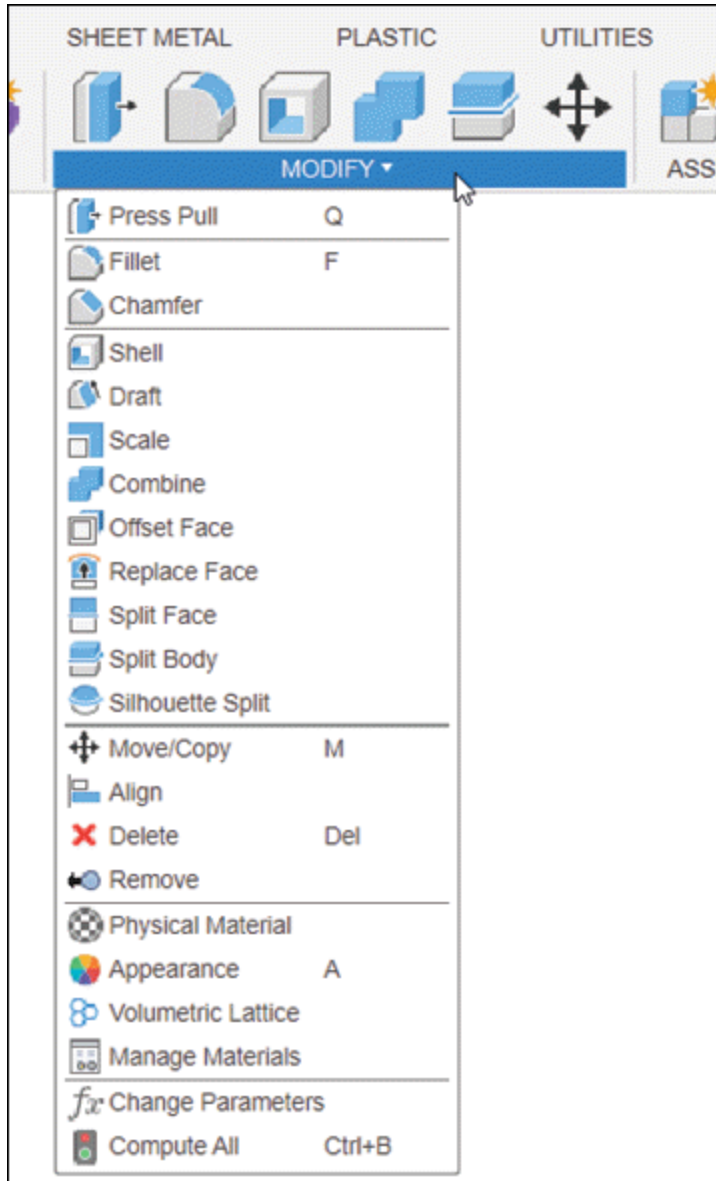


Figure-1. Modify drop down

PRESS PULL

The **Press Pull** tool is used to modify the body by drag and drop methods. The options displayed to perform press pull operations depends on the selected geometry. The procedure to use this tool is discussed next.

- Select **Press Pull** tool of **MODIFY** drop-down from

Toolbar; refer to [Figure-2](#). The **PRESS PULL** dialog box will be displayed; refer to [Figure-3](#).

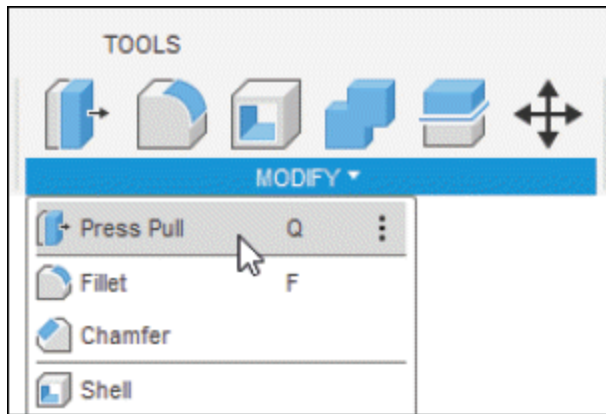


Figure-2. Press Pull tool

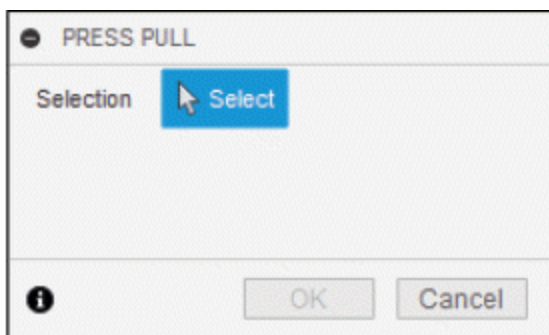


Figure-3. PRESS PULL dialog box

- Select a face, if you want to offset or move the face. The **OFFSET FACE** dialog box will be displayed. The **Offset Face** tool will be discussed later.
- Select an edge, if you want to fillet the edge. The **FILLET** dialog box will be displayed. The **Fillet** tool will be discussed later in this unit.
- Select a sketch, if you want to extrude the profile. The **EXTRUDE** dialog box will be displayed. This tool was discussed earlier in this book.

Offset Face

The **Offset Face** tool is used to offset or move the face. The procedure to use this tool is discussed next.

- Select **Press Pull** tool from **MODIFY** drop-down. The **PRESS PULL** dialog box will be displayed. You can also select **Press Pull** tool by pressing **Q** key.
- Click on the face of the model to offset. The **OFFSET FACE** dialog box will be displayed; refer to [Figure-4](#).

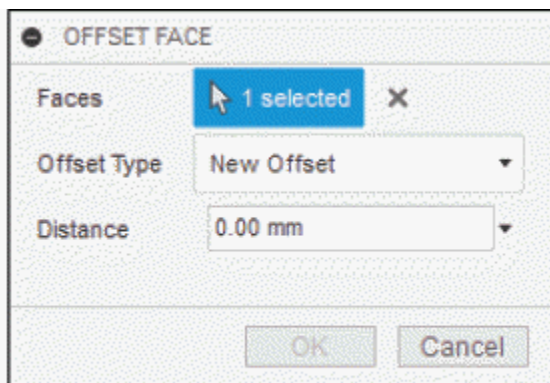


Figure-4. OFFSET FACE dialog box

- Select **Automatic** button from **Offset Type** drop-down if you do not want to update the sketch or base feature on applying the offset. This action is not added to time line.
- Select **Modify Existing Feature** button from **Offset Type** drop-down if you want to update the sketch or base feature on applying offset. This action is not added to time line.
- Select **New Offset** button from **Offset Type** drop-down if you want to create a new face. This action is added to time line.

- Click in **Distance** edit box of **OFFSET FACE** dialog box or floating window from screen and enter desired value as the distance for offset. You can also move drag handle to set the distance. The preview of offset face will be displayed; refer to [Figure-5](#).

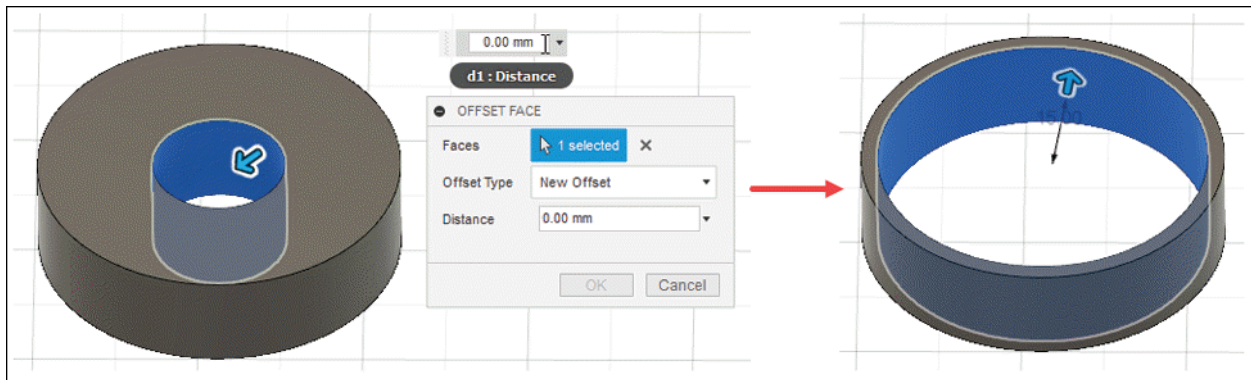


Figure-5. Preview on applying offset face

- Click **OK** button from **OFFSET FACE** dialog box to complete the process.

FILLET

The **Fillet** tool is used to apply round at the edges. This tool works in the same way as the **Sketch Fillet** do. It is recommended that, apply the fillets after creating all the features required in the model because fillets can increase the processing time during modifications. The procedure to use this tool is given next.

- Select **Fillet** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-6](#) or press **Q** and select an edge. The **FILLET** dialog box will be displayed; refer to [Figure-7](#).

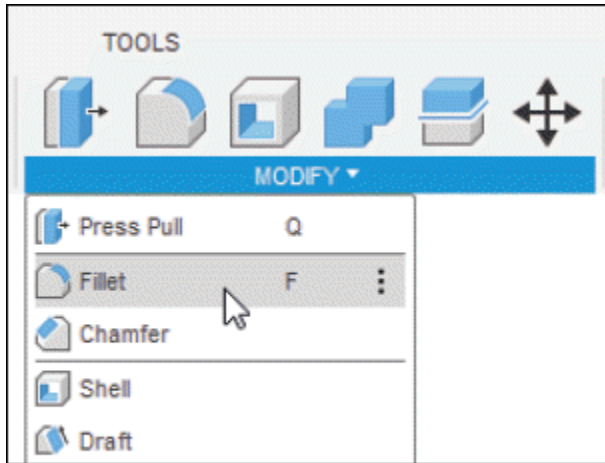


Figure-6. 3D Fillet tool

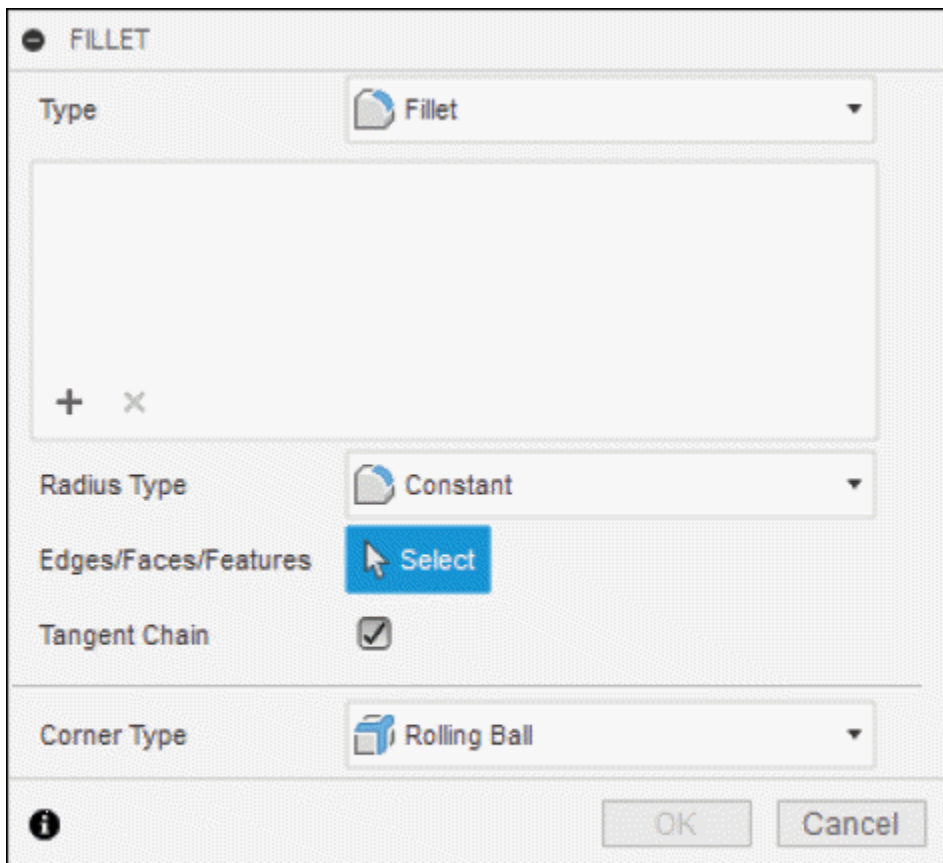


Figure-7. FILLET dialog box

- **Edges** selection is active by default. Click on the edge(s) from the model to apply fillet. You can select multiple

edges also without holding the **CTRL** key. The updated **FILLET** dialog box will be displayed; refer to [Figure-8](#).

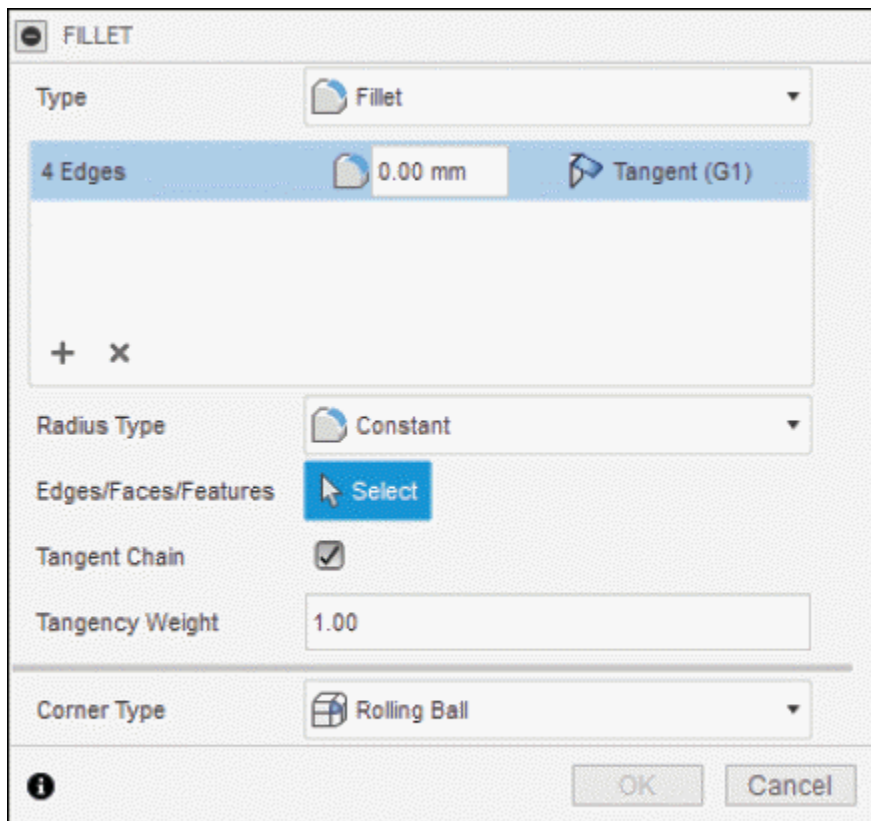


Figure-8. Updated 3D FILLET dialog box

- The options of the updated **FILLET** dialog box are discussed next.

Radius Type Drop-Down

There are three options in **Radius Type** drop-down; refer to [Figure-9](#). These options are used to specify the type of round to be created. These options are discussed next.

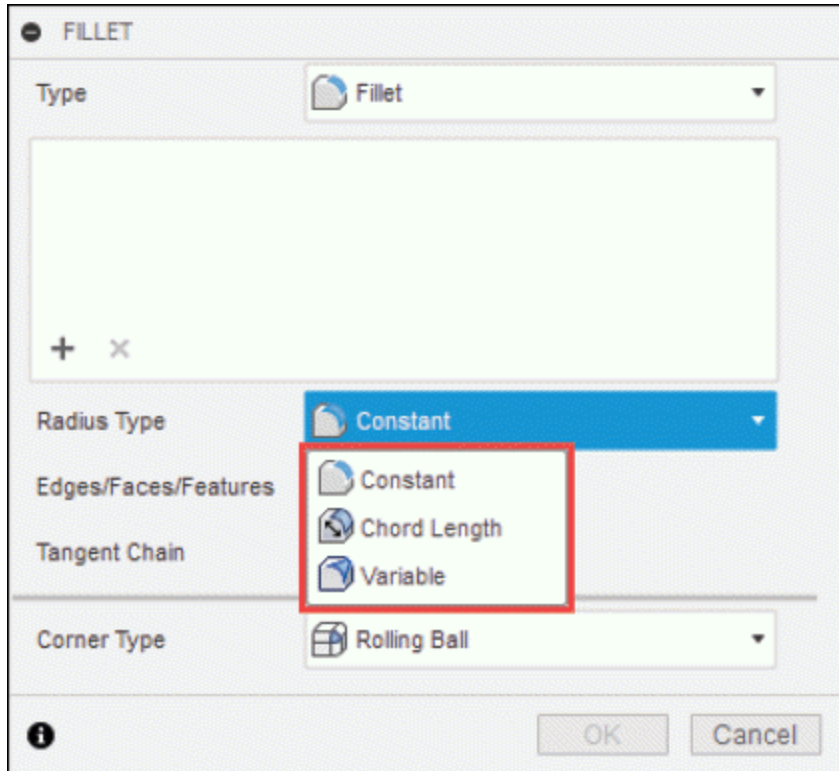


Figure-9. Fillet Radius Type drop down

Constant Radius Type

- Select **Constant** button from **Radius Type** drop-down to apply the fillet of fix size on the selected edge.
- Click in the **Radius** edit box of **FILLET** dialog box and enter **10** as the radius of fillet. You can also move the drag-handle from canvas screen to set the radius of fillet; refer to [Figure-10](#).

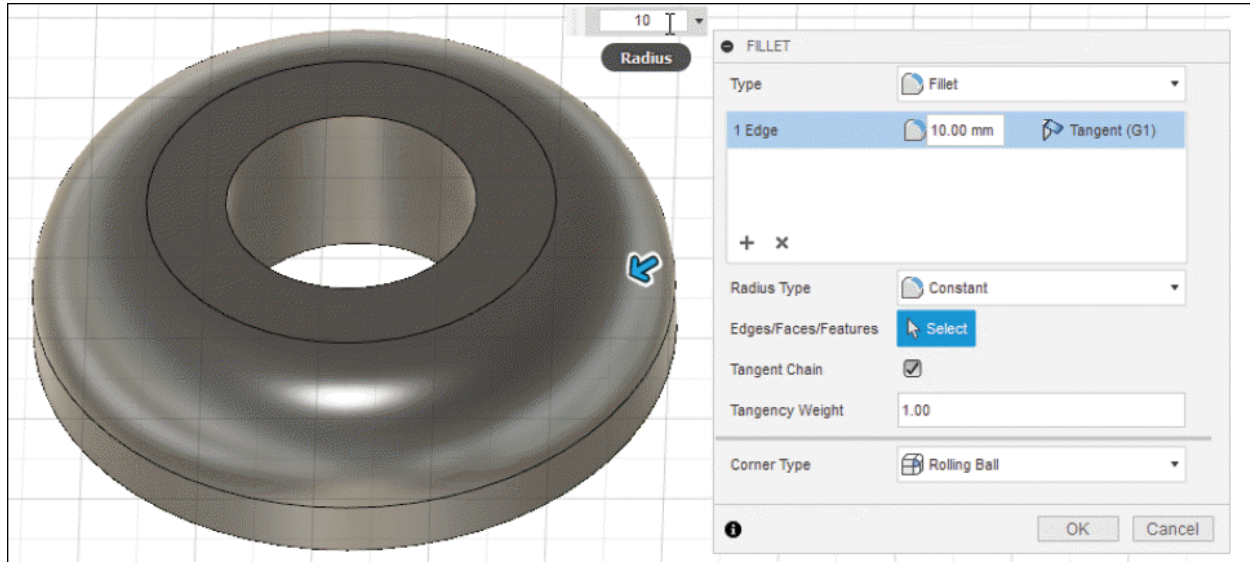


Figure-10. Preview of constant radius type

- Select the **Tangent Chain** check box to include tangentially connected edges in fillet.
- Specify desired value in **Tangency Weight** edit box to increase or decrease the scale of tangency weight.
- Select the **Curvature (G2)** option from **Continuity** drop-down if you want to apply the curvature continuity to the selected fillet.
- Select desired option from **Corner Type** section to define fillet shape at vertex formed by three edges meeting at single point.
- Click **OK** button from **FILLET** dialog box to apply the fillet.

Chord Length Radius Type

- Select **Chord Length** option of **Radius Type** drop-down from **FILLET** dialog box if you want to specify the chord length to control the size for the fillet.

- Click in the **Chord Length** edit box and enter desired value for fillet. You can also set the value by moving the drag handle. The preview will be displayed; refer to [Figure-11](#).

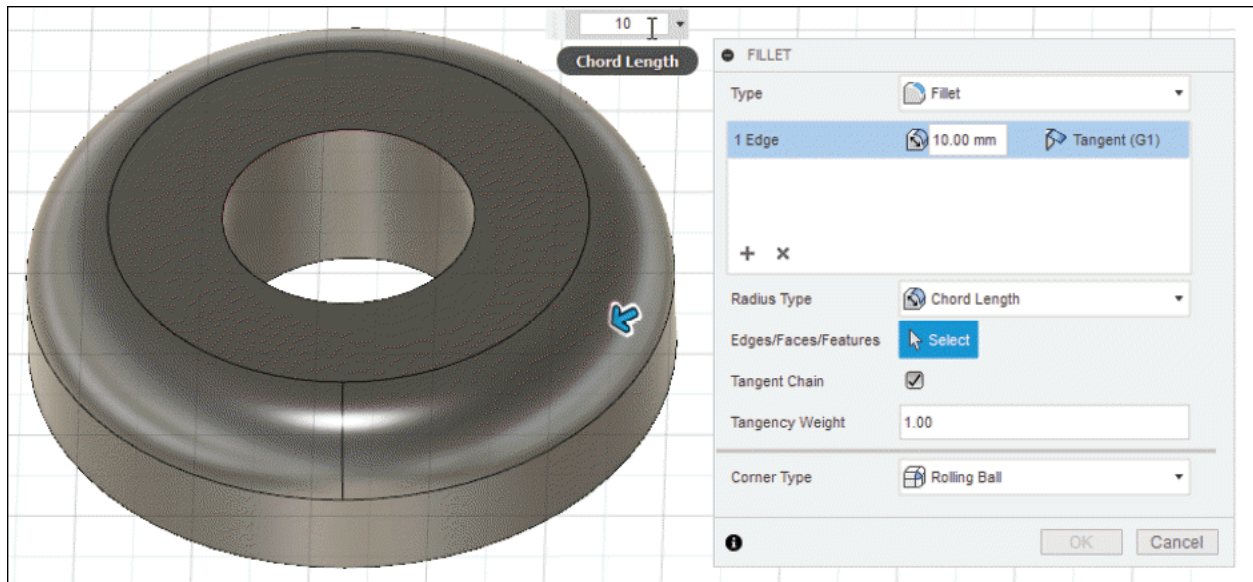


Figure-11. Preview of chord length fillet

- The other parameters are same as discussed in last topic.

Variable Radius Type

- Select **Variable** option from **Radius Type** drop-down of **FILLET** dialog box if you want to specify radii at selected points along the edge. The updated **FILLET** dialog box will be displayed.
- Click on the edge to create points for applying fillet and specify radius for each point in the dialog box or dynamic input box; refer to [Figure-12](#).

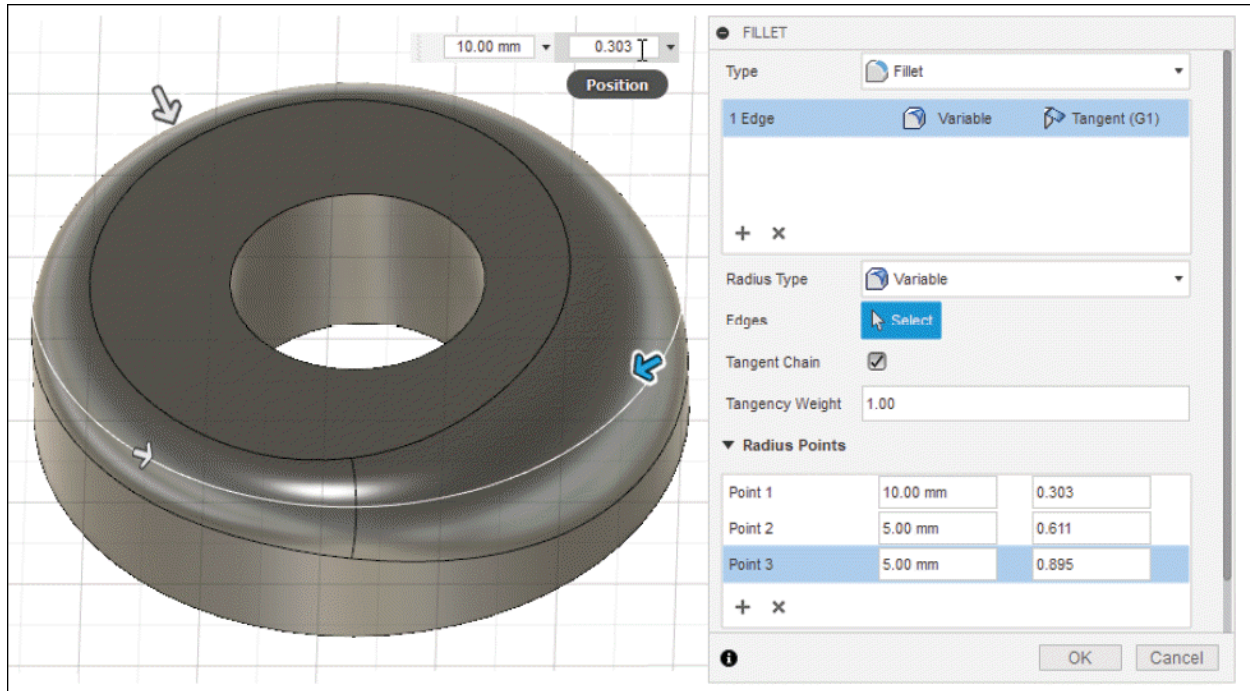


Figure-12. Preview of variable radius type

- To delete the created point, select the specific point from the edge and click on the **Remove selection set** button from the **FILLET** dialog box. Point earlier created to apply fillet will be displayed with arrows.
- The other parameters of this dialog box were discussed earlier.
- After specifying the parameters, click on the **OK** button from **FILLET** dialog box to create the fillet.

RULE FILLET

The **Rule Fillet** option is used to add fillets or rounds to a design based on specified rules. The edges are determined by specified rules rather than selection in the canvas. The procedure to use this tool is discussed next.

- Select **Rule Fillet** option from **Type** drop down in **FILLET** dialog box; refer to [Figure-13](#). The **FILLET** dialog box with rule fillet options will be displayed; refer to [Figure-14](#).

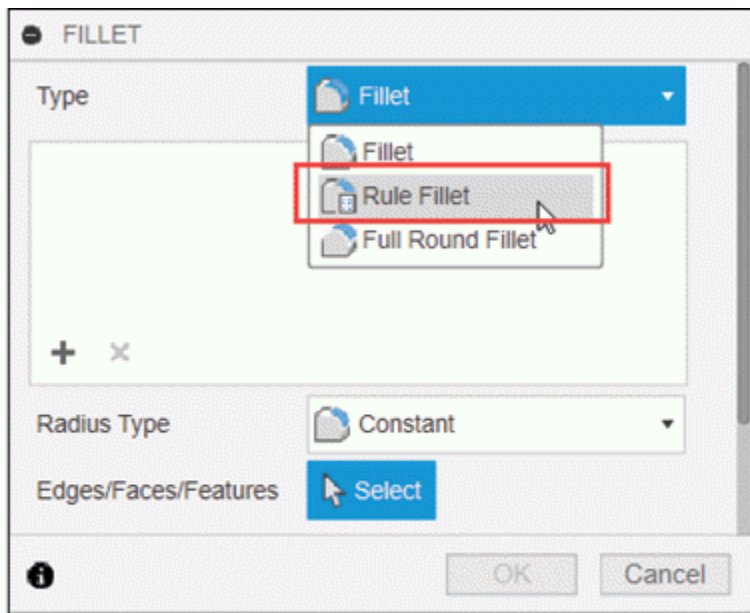


Figure-13. Rule Fillet tool in FILLET dialog box

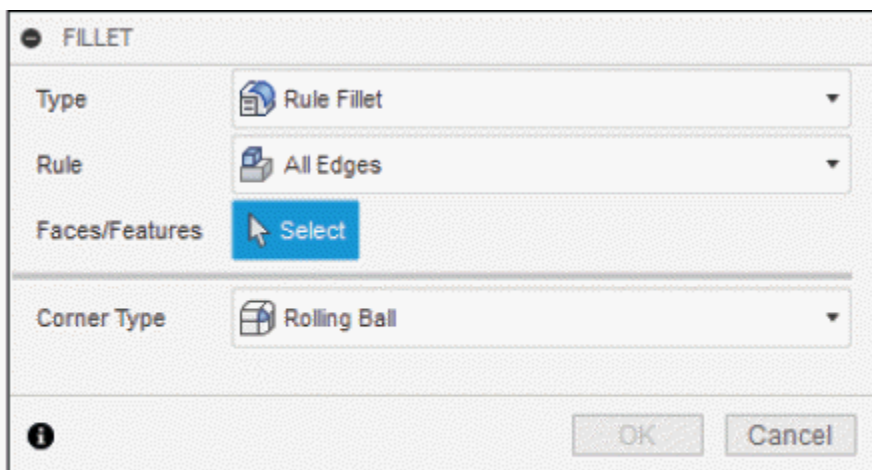


Figure-14. FILLET dialog box with rule fillet options

- The **Faces/Features** selection is active by default. Select desired faces/features from model to apply rule fillet.
- Click in the **Radius** edit box and specify desired value of

radius for fillet. The preview of fillet will be displayed; refer to [Figure-15](#). You can also set the radius by moving the drag handle.

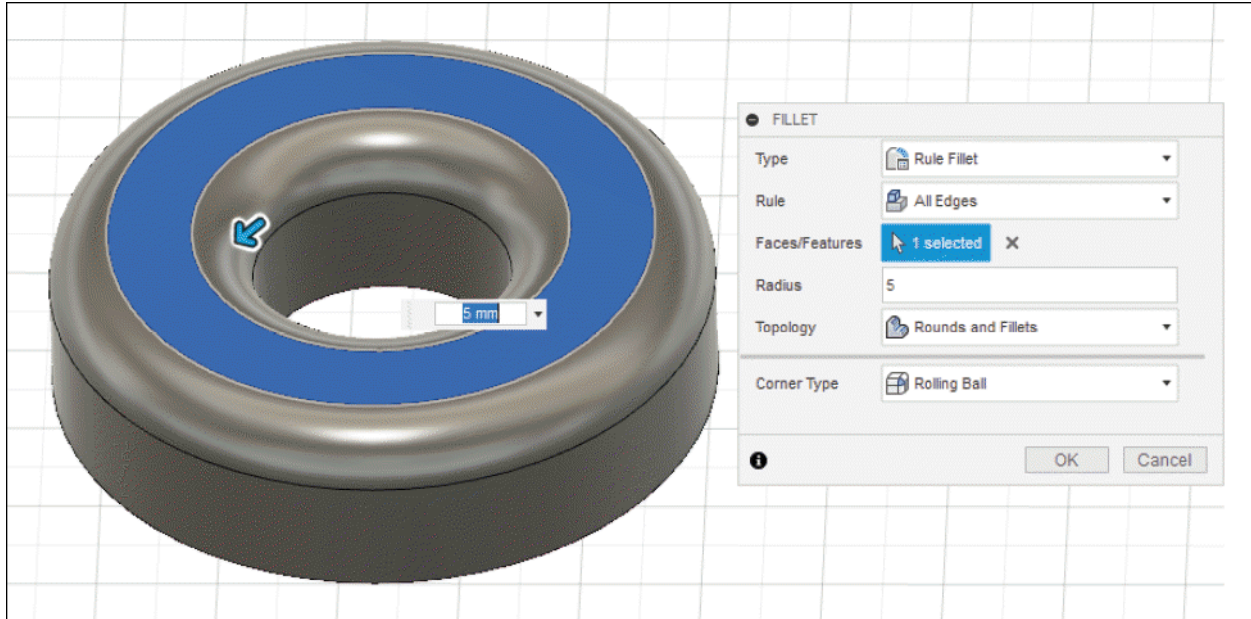


Figure-15. Preview of rule fillet

- Select **All Edges** option from **Rule** drop-down if you want to add the fillets to all edges of the input faces/features and select **Between Faces/Features** option if you want to add the fillets at the intersection of the input faces/features. Note that you can also select features from Time line.
- Select **Rounds Only** option from **Topology** drop-down if you want to create only rounds and select the **Fillets Only** option if you want to create only fillets. Selecting the **Rounds And Fillets** option in this drop-down will create both fillets and rounds.
- Click **OK** button from **FILLET** dialog box to finish this

process.

FULL ROUND FILLET

The **Full Round Fillet** option is used to create full round fillet by selecting a face only and it will automatically find the two opposite edges. The procedure to use this tool is discussed next.

- Select **Full Round Fillet** option from **Type** drop-down in **Fillet** dialog box; refer to [Figure-16](#). The **FILLET** dialog box with full round fillet options will be displayed; refer to [Figure-17](#).

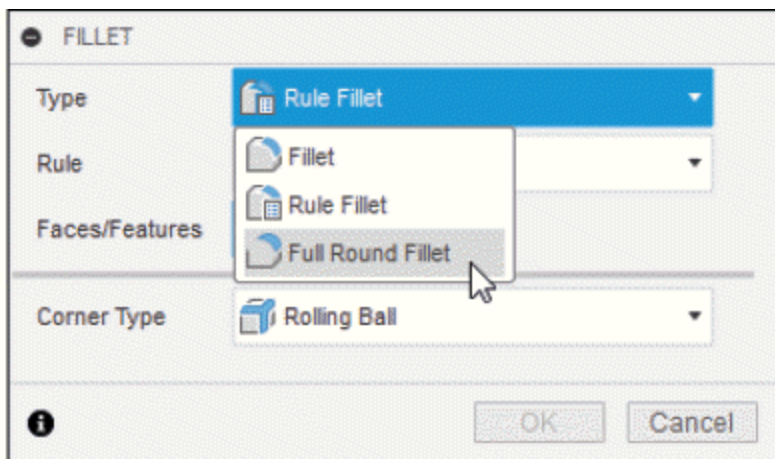


Figure-16. Full Round Fillet tool in FILLET dialog box

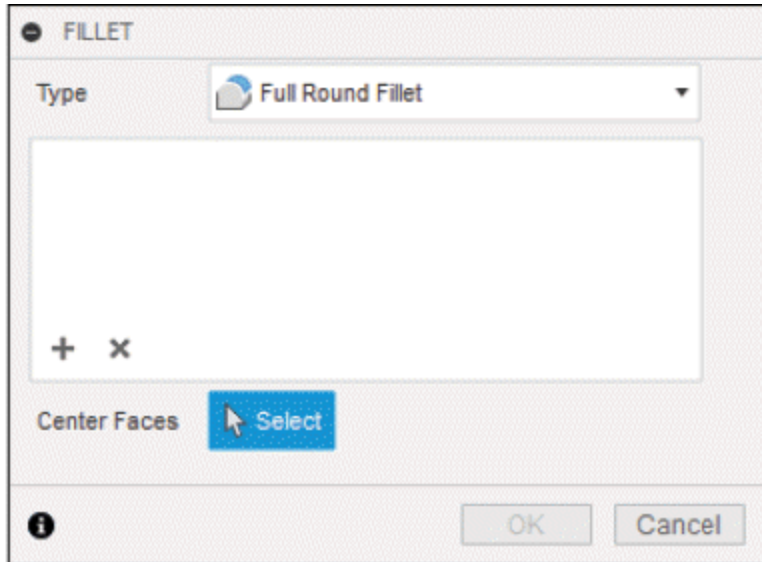


Figure-17. FILLET dialog box with full round fillet options

- The **Select** button of the **Center Faces** section is active by default. Select desired center face of the model to create full round fillet; refer to [Figure-18](#).

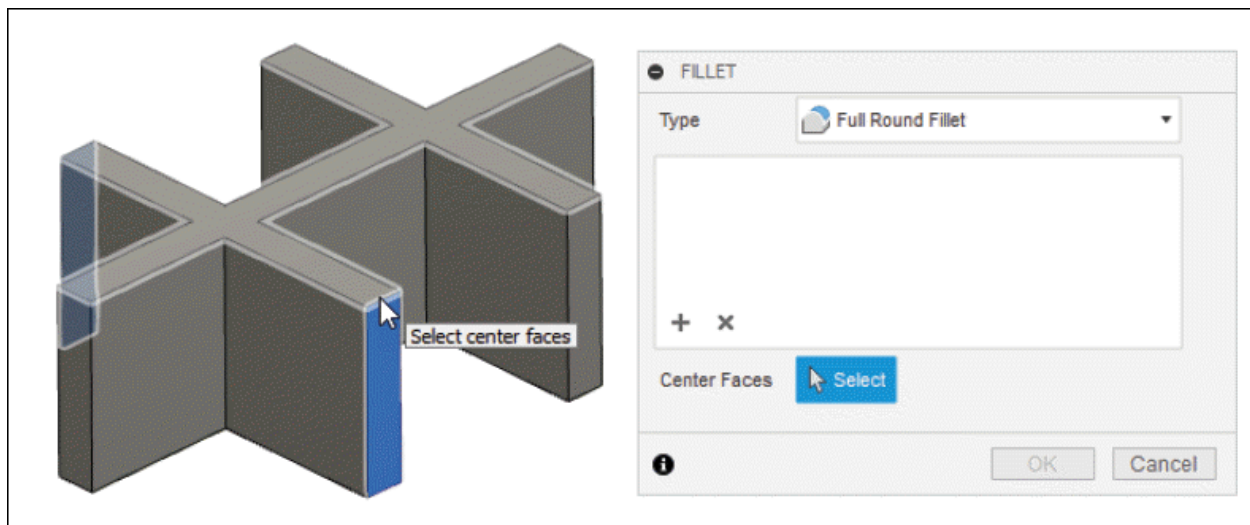


Figure-18. Selecting center face of the model

- On selecting center face of the model, full round fillet of the model will be created and the updated **FILLET** dialog box will be displayed; refer to [Figure-19](#).

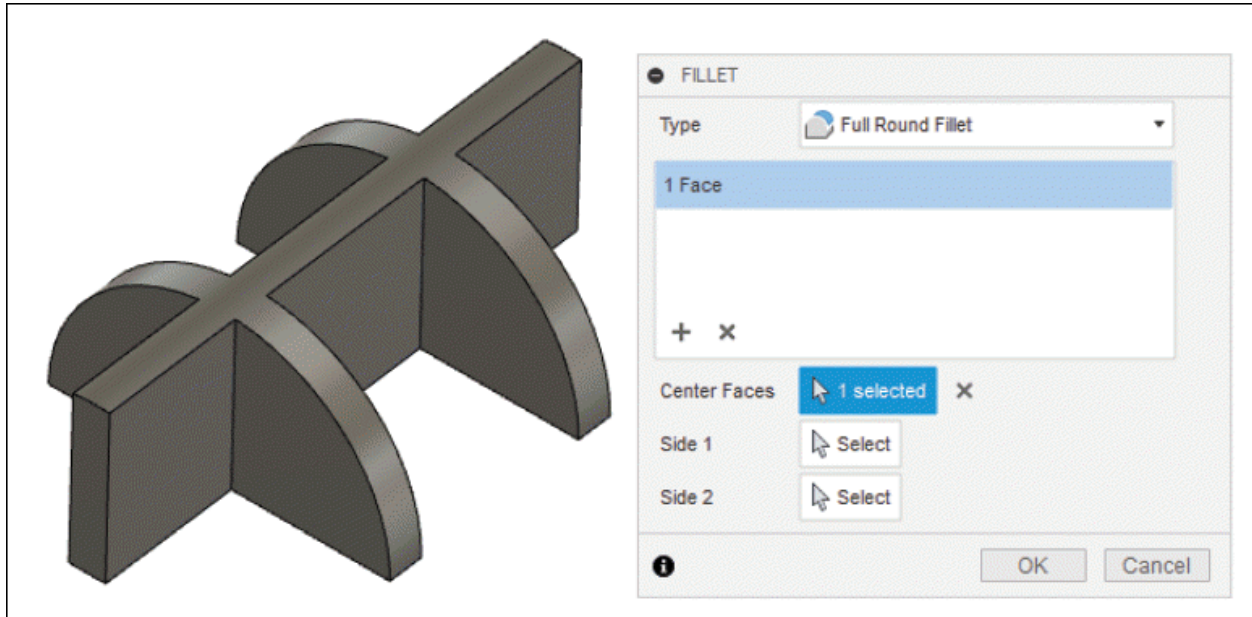


Figure-19. Full round fillet of model created with updated FILLET dialog box

- Click on **Select** button of **Side 1** section and press & hold **CTRL** key to select faces on first side of the full round fillet and click on **Select** button of **Side 2** section and press & hold **CTRL** key to select faces on second side of the full round fillet to modify the fillet.
- After specifying the parameters, click on **OK** button from the dialog box to finish the process.

CHAMFER

The **Chamfer** tool is used to bevel the sharp edges of the model. The procedure to create chamfer by using this tool is given next.

- Click on the **Chamfer** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-20](#). The **CHAMFER** dialog box will be displayed; refer to [Figure-21](#).

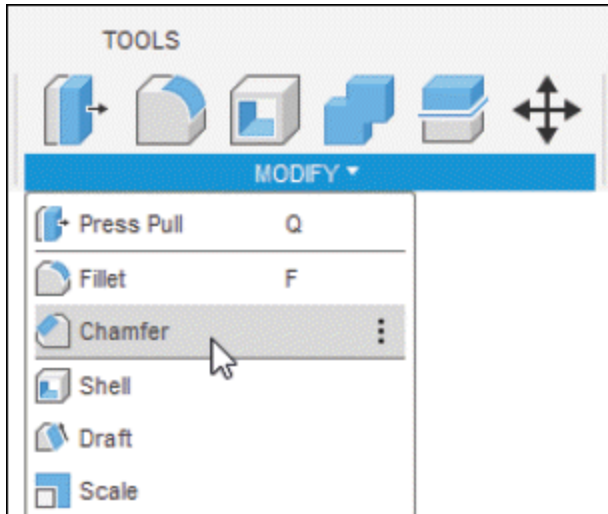


Figure-20. Chamfer tool

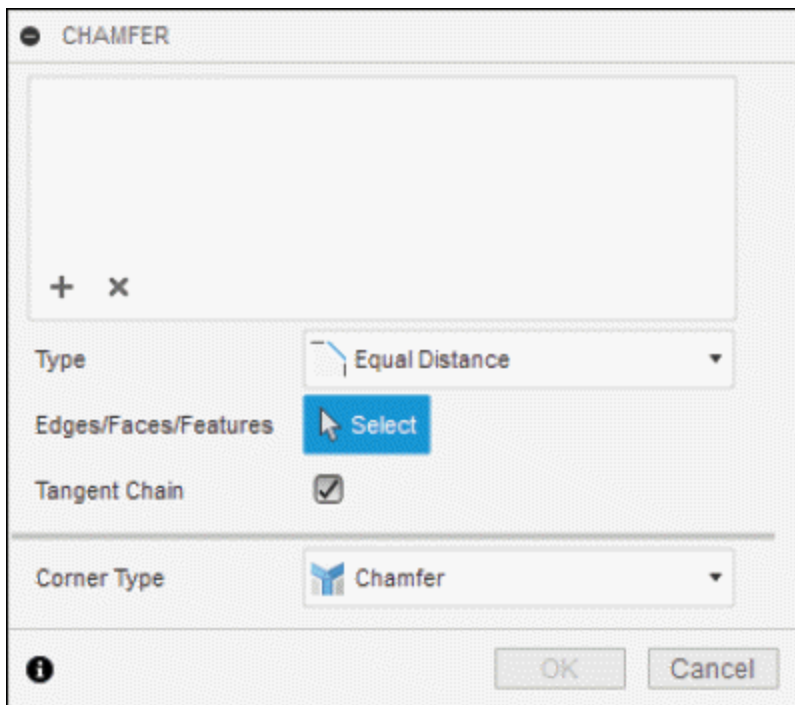


Figure-21. CHAMFER dialog box

- **Edges/Faces/Features** selection is active by default. Click on the edge of model to apply chamfer. The updated **CHAMFER** dialog box will be displayed; refer to [Figure-22](#).

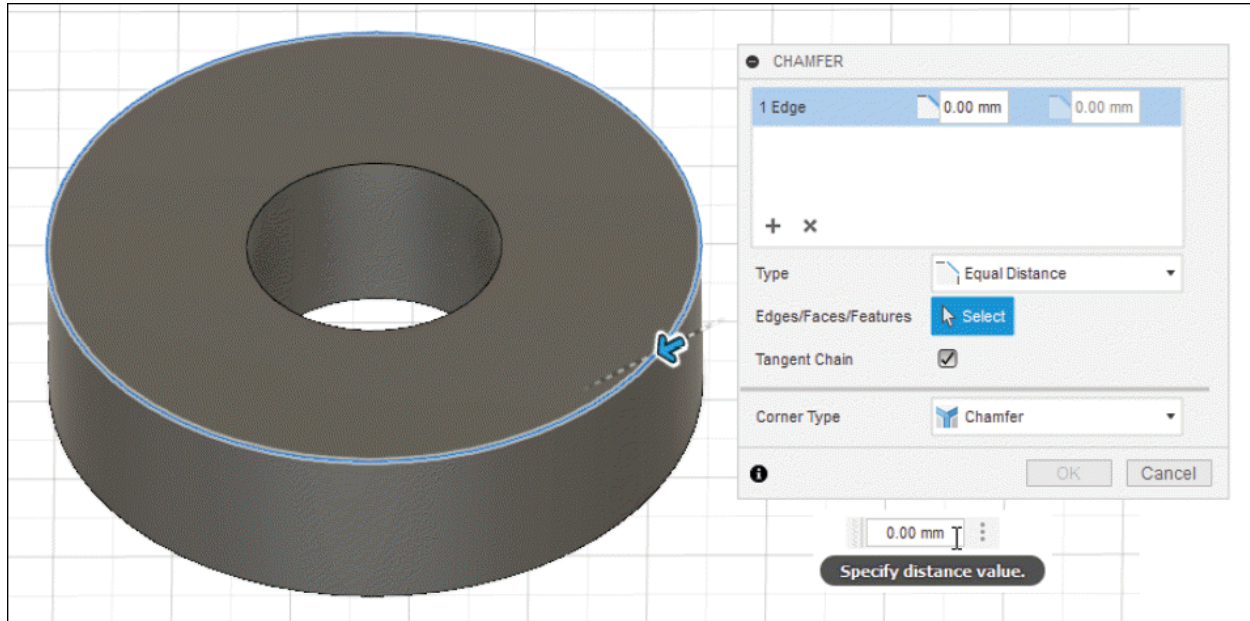


Figure-22. Updated CHAMFER dialog box

- Select the **Tangent Chain** check box to include tangentially connected edges in selection.
- Select **Equal Distance** option from **Type** drop-down of **CHAMFER** dialog box if you want to specify equal distance for both sides of the chamfer.
- Select **Two Distance** option from **Type** drop-down of **CHAMFER** dialog box if you want to specify distance for each side of the chamfer.
- Enter the value of distance for each side in the respective edit box. You can also set the distance by moving the drag handle from canvas screen.
- Select **Distance and Angle** option from **Type** drop-down if you want to specify distance and angle values to create the chamfer.

- Click in the **Specify distance value** edit box and enter desired value for distance for chamfer. The preview of chamfer will be displayed; refer to [Figure-23](#).

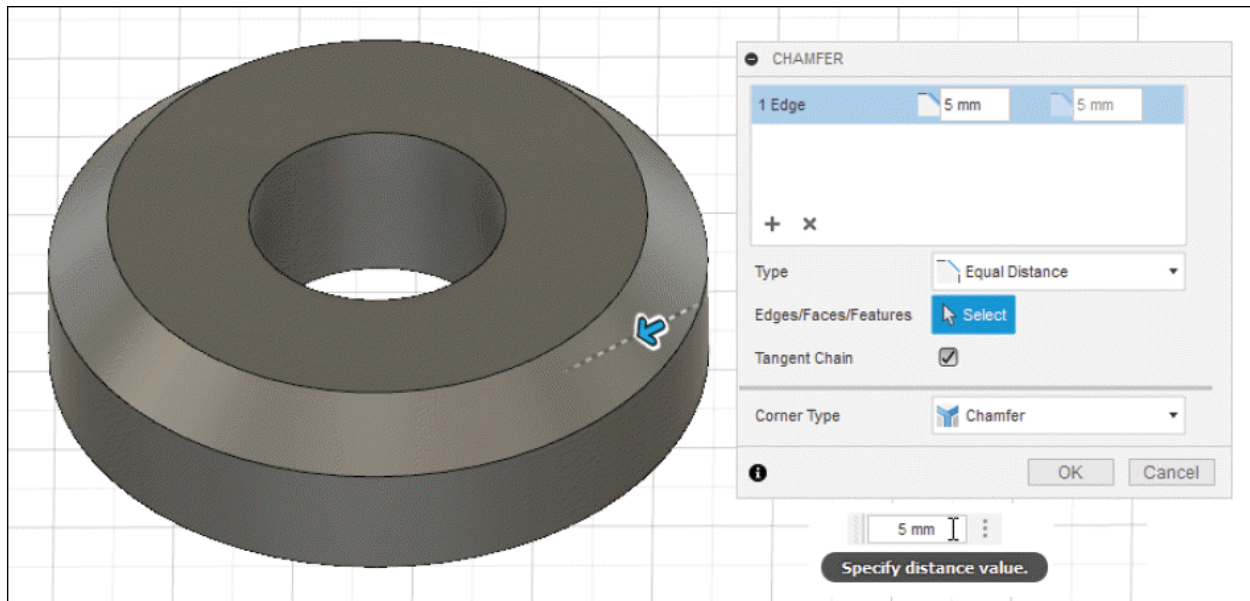


Figure-23. Preview of chamfer

- Select **Chamfer** option from **Corner Type** drop-down to create a chamfer to join beveled edges at the corner.
- Select **Miter** option from **Corner Type** drop-down to merge beveled edges into a mitered corner point.
- Select **Blend** option from **Corner Type** drop-down to blend beveled edges into adjacent faces.
- Click **OK** button of **CHAMFER** dialog box to finish the process.

SHELL

The **Shell** tool is used to make a solid part hollow and remove one or more selected faces. The procedure to use this tool is given next.

- Select **Shell** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-24](#). The **SHELL** dialog box will be displayed; refer to [Figure-25](#).

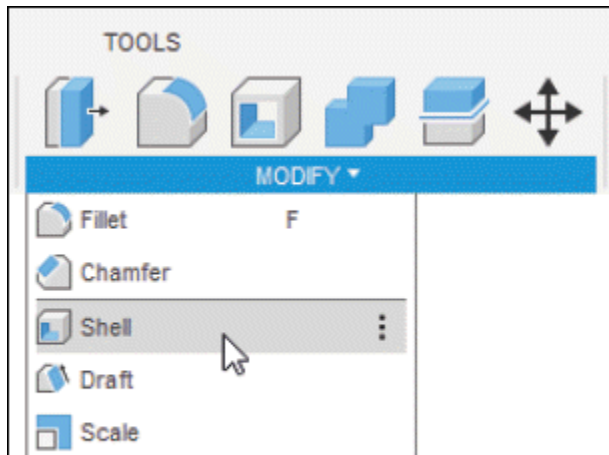


Figure-24. Shell tool

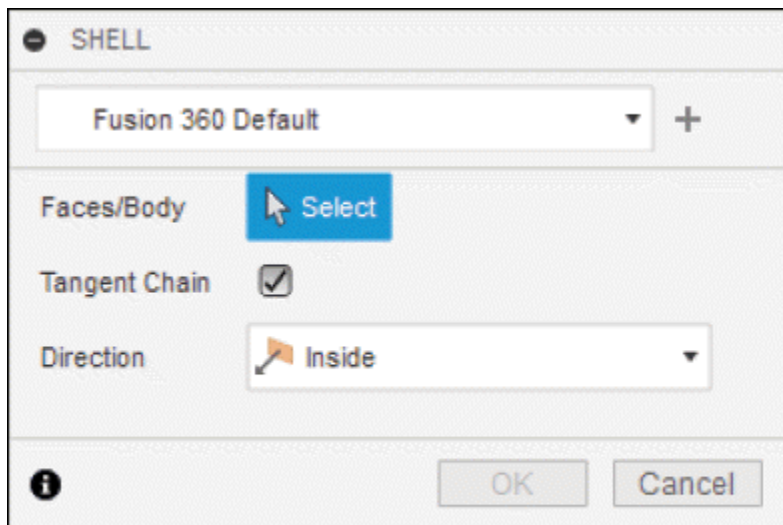


Figure-25. SHELL dialog box

- The **Faces/Body** selection button is active by default. Click on the face/body of the solid to apply shell. The updated **SHELL** dialog box will be displayed; refer to [Figure-26](#).

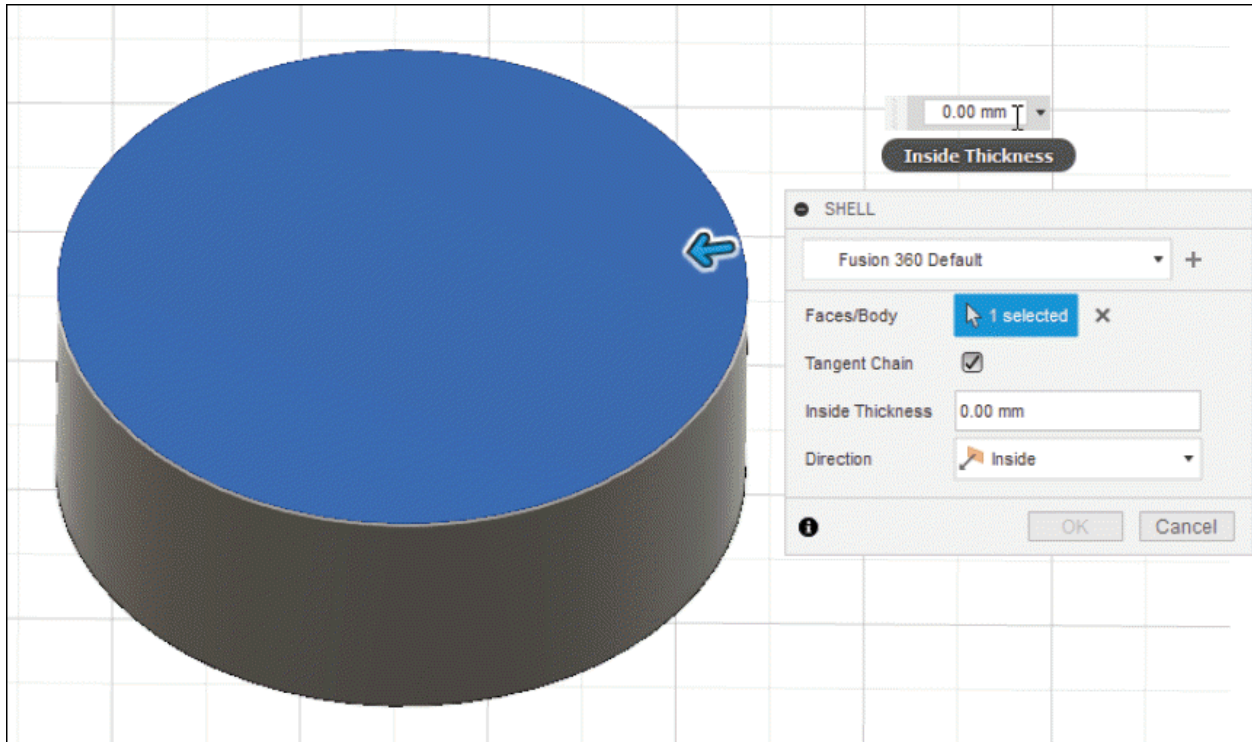


Figure-26. Updated SHELL dialog box

- Select the **Tangent Chain** check box to include tangentially connected edges in selection.
- Click in the **Inside Thickness** edit box and specify desired value of shell thickness. The preview of shell will be displayed; refer to [Figure-27](#).

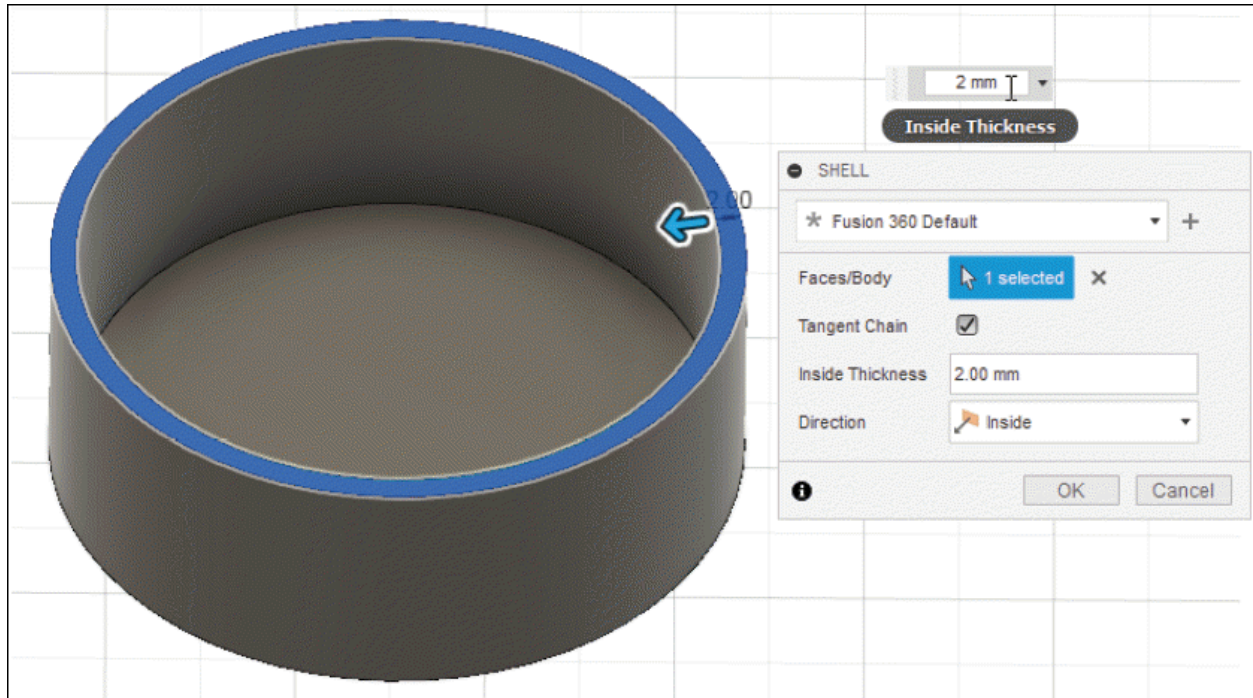


Figure-27. Preview of shell

- Select **Inside** option from **Direction** drop-down if you want to shell the faces towards the interior of the part.
- Select **Outside** option from **Direction** drop-down if you want to shell the faces towards the exterior of the part.
- Select **Both** option from **Direction** drop-down if you want to shell the faces towards the interior and the exterior of the part by equal amount. Click in the **Inside Thickness** and **Outside Thickness** edit boxes and specify desired value to specify thickness for shell.
- Click **OK** button from **SHELL** dialog box to complete the process.

DRAFT

The **Draft** tool is used to apply taper to the faces of a solid

model. This tool is mainly useful when you are designing components for moulding or casting. **Draft** tool applies taper on the face and this taper allows easy & safe ejection of parts from dies. The procedure to use this tool is discussed next.

- Select **Draft** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-28](#). The **DRAFT** dialog box will be displayed; refer to [Figure-29](#).

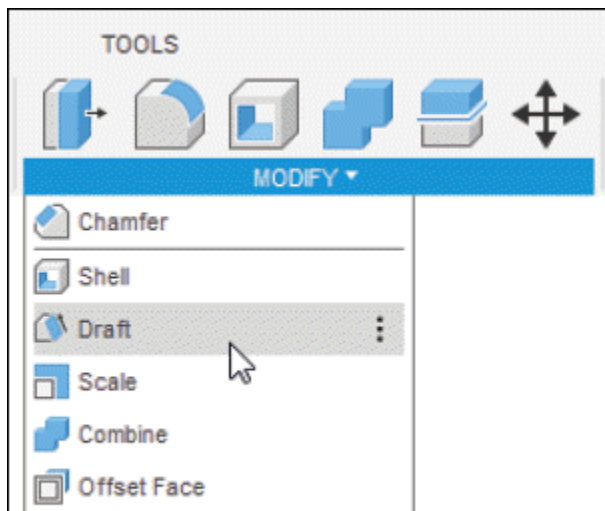


Figure-28. Draft tool

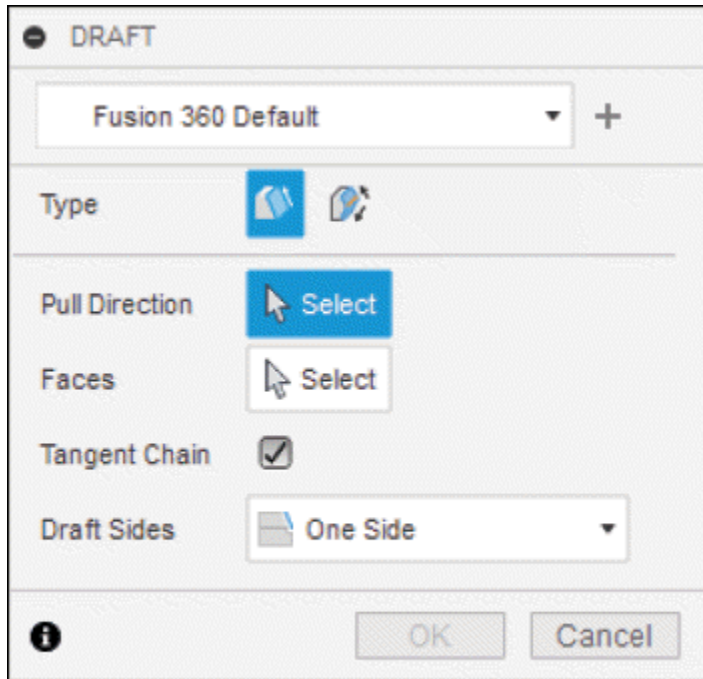


Figure-29. DRAFT dialog box

- The **Fixed Plane** draft type is selected by default in the **Type** section of the dialog box. **Fixed Plane** draft type creates tapered face using angle specified from selected fixed plane.
- The **Pull Direction** selection button is active by default. Click on the face or plane of the model which you want to use as fixed reference.
- Click on **Select** button of **Faces** section and select the faces to be tapered.
- Click on the **Flip Pull Direction** button from **DRAFT** dialog box to flip the direction of draft.
- Click in the **Angle** edit box and specify desired value of angle for draft. The preview of draft will be displayed;

refer to [Figure-30](#). You can also set the angle of draft by moving the drag handle.

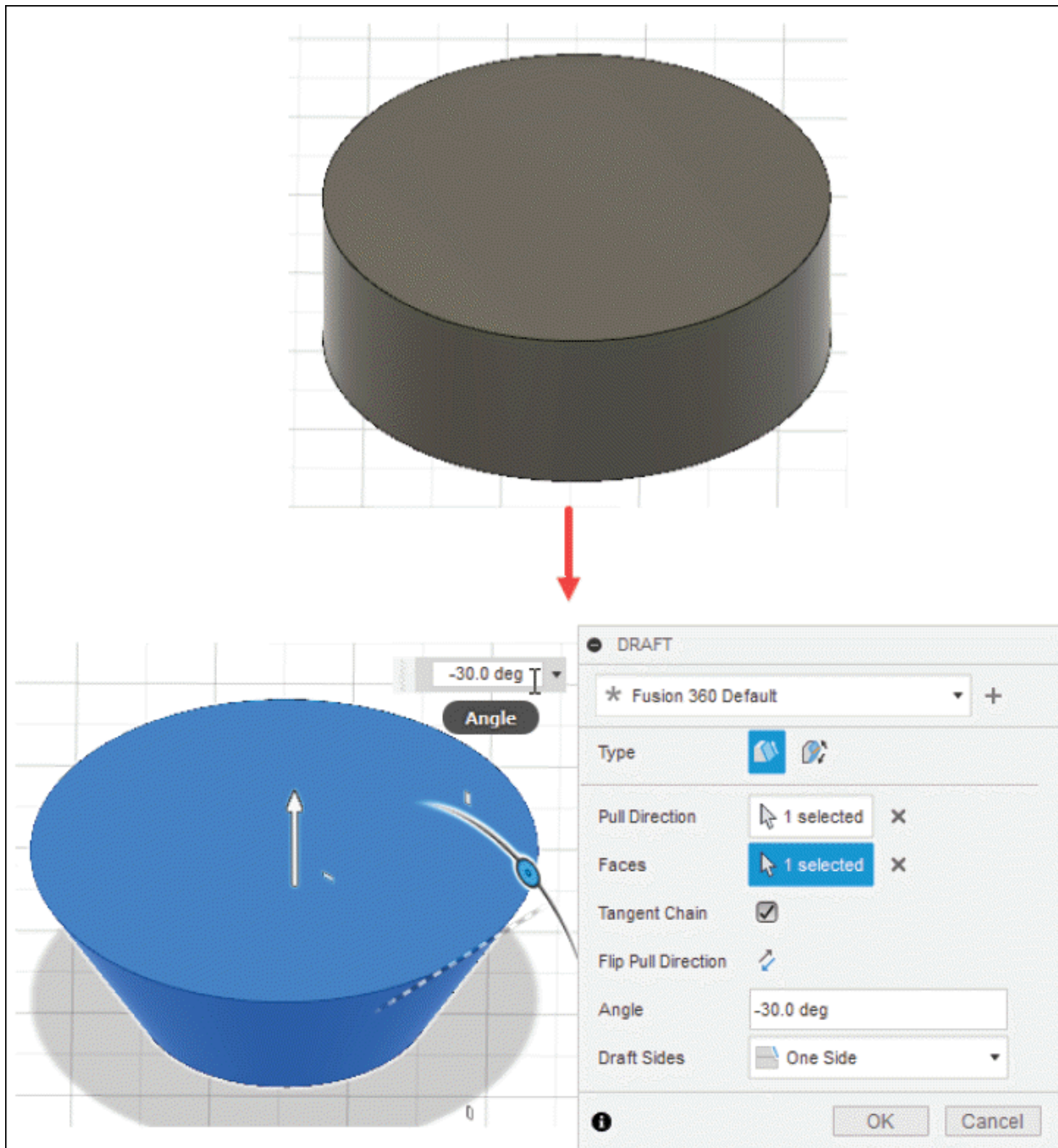


Figure-30. Preview of draft

- The options of **Draft Sides** drop-down were discussed earlier in this book.

- Click **OK** button from **DRAFT** dialog box to finish the process.
- Select **Parting Line** draft type from **Type** section of the dialog box to create draft along a parting line. The updated **DRAFT** dialog box will be displayed; refer to [Figure-31](#).

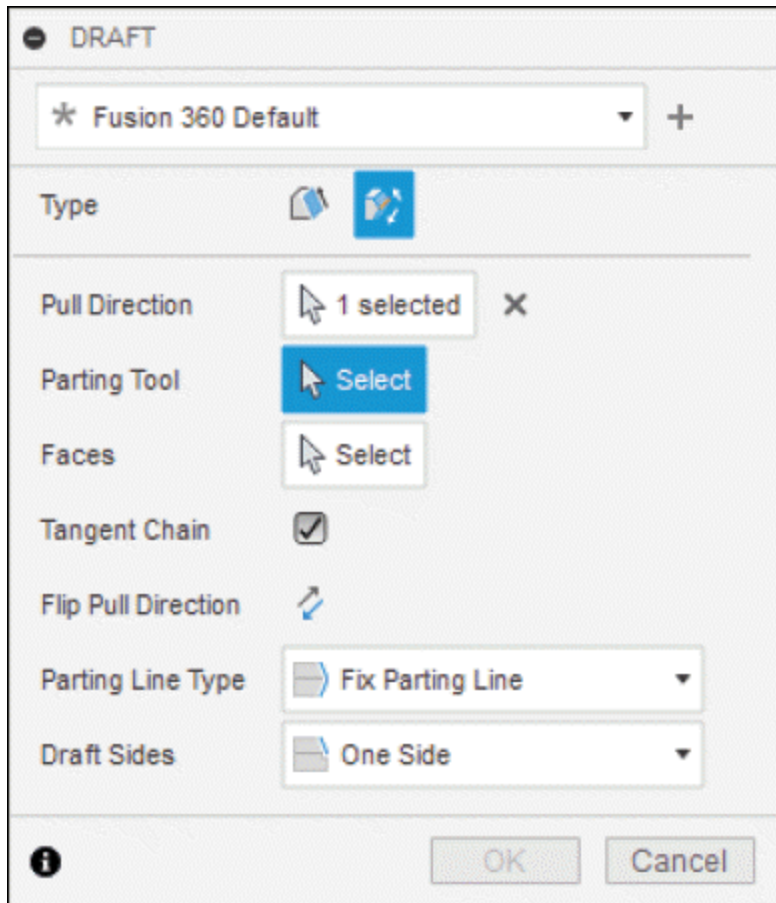


Figure-31. Updated DRAFT dialog box

- Select desired face or plane of the model to define pull direction in **Pull Direction** section.
- Select desired plane, face, edge, or sketch curve on the model to use as parting tool in **Parting Tool** section.
- Select desired faces on the model which you want to draft

in the **Faces** section of the dialog box.

- Specify desired value of angle for draft in the **Angle** edit box. The preview of parting line draft type will be displayed; refer to [Figure-32](#).

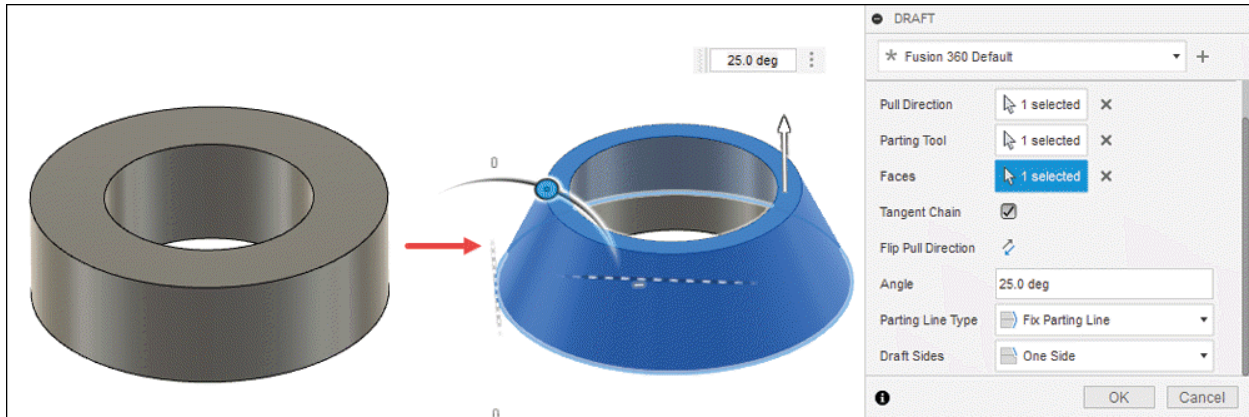


Figure-32. Preview of parting line draft type

- Select **Fix Parting Line** option from **Parting Line Type** drop-down to fix the parting line in place.
- Select desired option from **Draft Sides** drop-down as discussed earlier in this book.
- Select **Move Parting Line** option from **Parting Line Type** drop-down to move the parting line to apply draft.
- Select **Angle Above** option from **Direction** drop-down to maintain angle above parting line.
- Select **Angle Below** option from **Direction** drop-down to maintain angle below parting line.
- Select **Both** option from **Direction** drop-down to maintain angle on both sides of parting line.
- Click on the **Select** button from **Fixed Edges** section of

the dialog box and select edges of the model to prevent deformation when parting line moves.

- The other options in the dialog box have been discussed earlier.
- Click **OK** button from **DRAFT** dialog box to finish the process.

SCALE

The **Scale** tool is used to enlarge or reduce selected bodies, sketches, or components based on specified scale factor. The procedure to use this tool is discussed next.

- Select **Scale** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-33](#). The **SCALE** dialog box will be displayed; refer to [Figure-34](#).

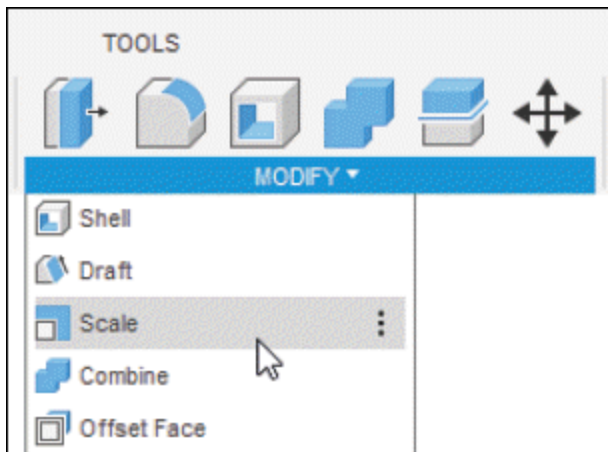


Figure-33. Scale tool

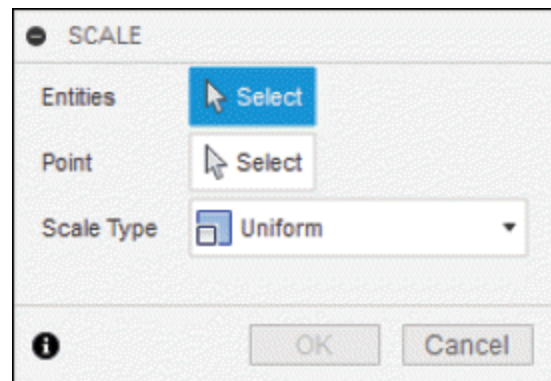


Figure-34. SCALE dialog box

- The **Entities** selection is active by default. Click on the entities to be scaled up or down.
- On selecting the entity for scale, point is automatically

selected in **Point** section. If you want to use a different point as scale reference then click on the **Select** button for **Point** option in the dialog box and select desired point.

- Select **Uniform** option from **Scale Type** drop-down if you want to scale the selected entities uniformly in all the directions.
- Select **Non-Uniform** option from **Scale Type** drop-down if you want to scale the entity non-uniformly in x, y, and z direction.
- Click in the **Scale Factor** edit box and specify desired value for scale. The preview will be displayed; refer to [Figure-35](#).
- Click on **OK** button of **SCALE** dialog box to complete the process.

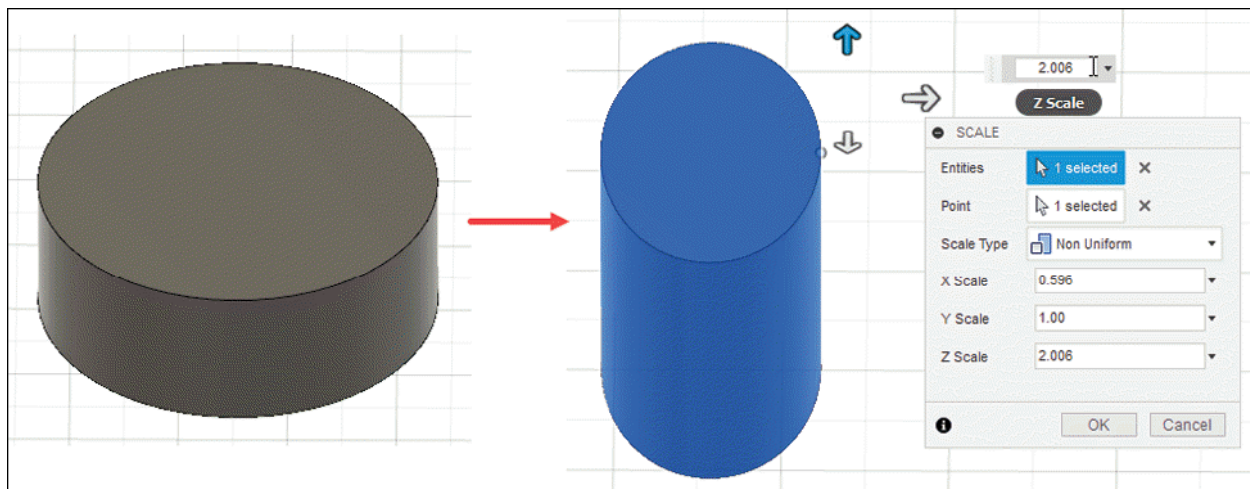


Figure-35. Preview of scale

COMBINE

The **Combine** tool is used to cut, join, and intersect the

selected bodies. The procedure to use this tool is discussed next.

- Select **Combine** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-36](#). The **COMBINE** dialog box will be displayed; refer to [Figure-37](#).

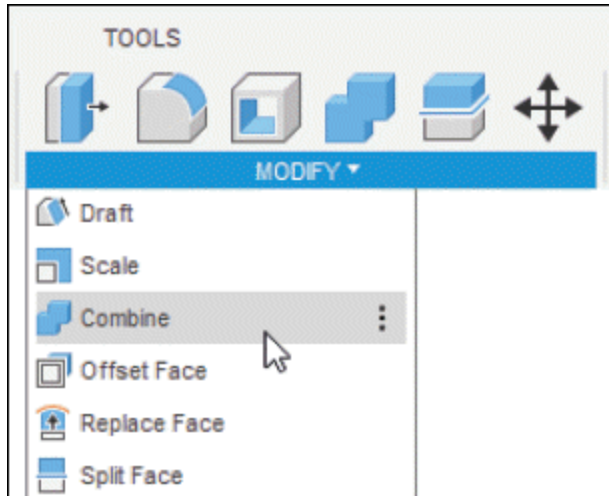


Figure-36. Combine tool

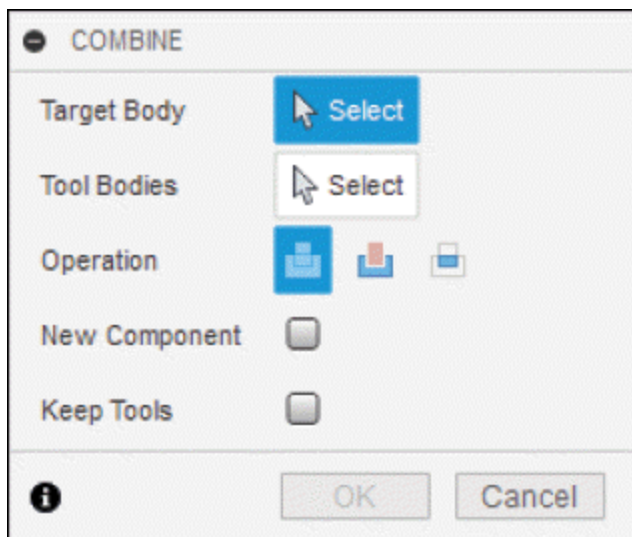


Figure-37. COMBINE dialog box

- The **Target Body** selection button is active by default. Click on the component from model to be used as a target

body.

- Click on other components to select in **Tool Bodies** section. The preview of combine will be displayed; refer to [Figure-38](#).

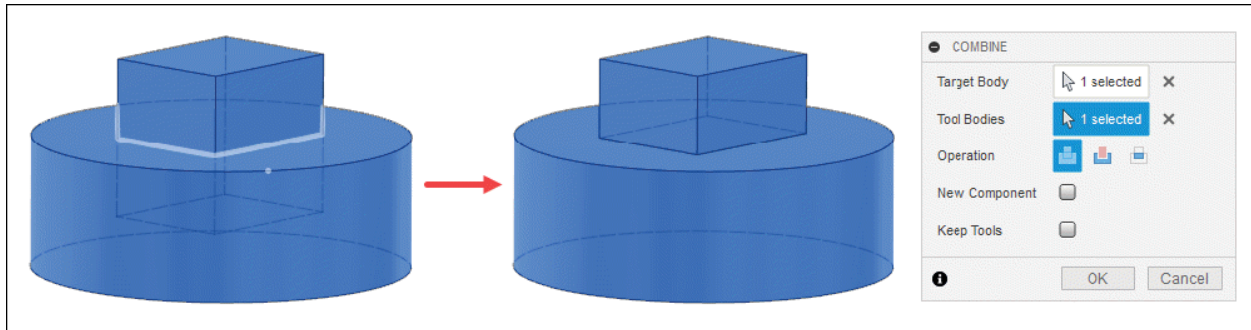


Figure-38. Preview of combine

- The options of **Operations** drop-down have been discussed earlier in this book.
- Select the **New Component** check box to create a new component with results.
- Select the **Keep Tools** check box to retain the tool bodies after the combine result.
- Click **OK** button of **COMBINE** dialog box to finish the process.

REPLACE FACE

The **Replace Face** tool is used to replace an existing face with a surface or work plane. The procedure to use this tool is discussed next.

- Select **Replace Face** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-39](#). The **REPLACE FACE** dialog box will be displayed; refer to [Figure-40](#).

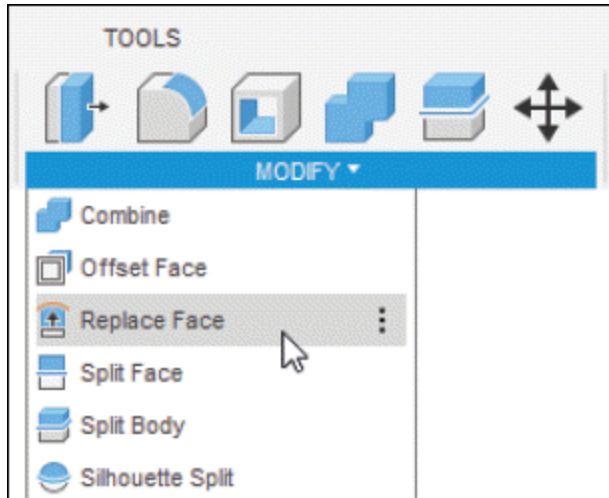


Figure-39. Replace Face tool



Figure-40. REPLACE FACE dialog box

- **Source Faces** selection is active by default. Click on the component face to select the face.
- Select the **Tangent Chain** check box to include tangentially connected faces in selection.
- Click **Select** button of **Target Faces** section and click on the face or plane to apply this tool. The preview will be displayed; refer to [Figure-41](#).

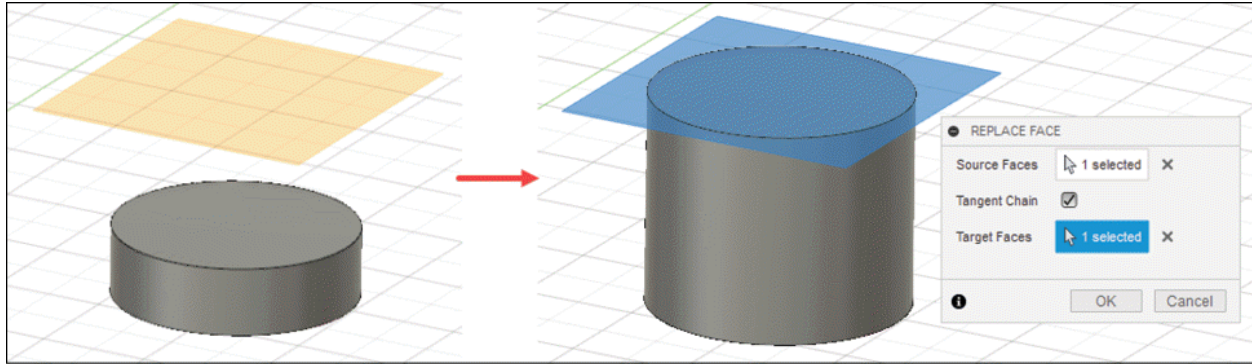


Figure-41. Preview of replace face

- Click **OK** button of **REPLACE FACE** dialog box to finish the process.

SPLIT FACE

The **Split Face** tool is used to divide faces of a surface or solid to add draft, delete an area, or create features. The procedure to use this tool is discussed next.

- Select **Split Face** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-42](#). The **SPLIT FACE** dialog box will be displayed; refer to [Figure-43](#).

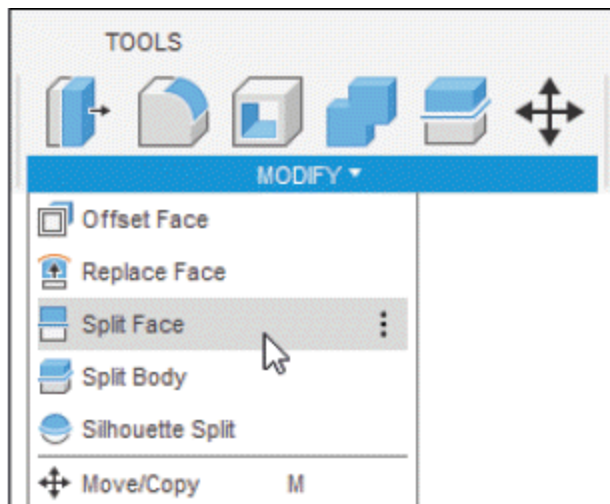


Figure-42. Split Face tool

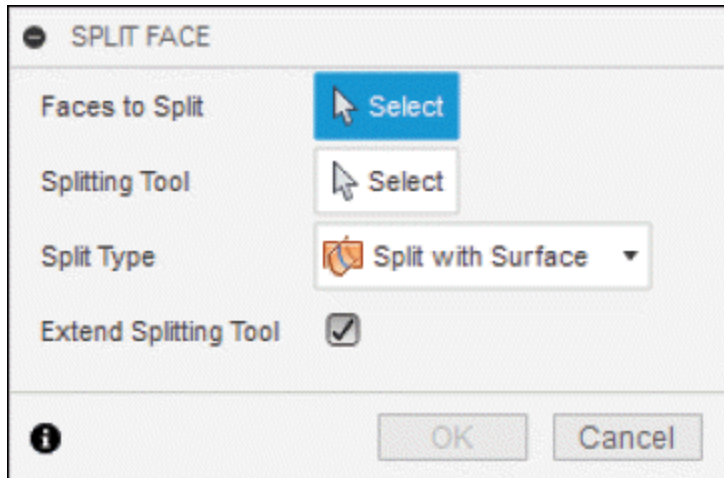


Figure-43. SPLIT FACE dialog box

- The **Faces To Split** option is active by default. Click on the model face to be split. You can select multiple faces by holding the **CTRL** key.
- Click on the **Select** button of **Splitting Tool** option and select the face which will split the other faces.
- Select **Split With Surface** option from **Split Type** drop-down, if you want to project the splitting tool face which you had selected upon the splitting face.
- Select **Along Vector** option from **Split Type** drop-down, if you want to define the projection direction of splitting face by selecting a face, edge, or axis.
- Select **Closest Point** option from **Split Type** drop-down, if you want to define the direction of projection as the closest distance between the splitting body and target faces.
- Select the **Extend Splitting Tool** check box to extend the tool to completely intersect the target faces.

- After specifying the parameters, click on **OK** button from **SPLIT FACE** dialog box to finish the process; refer to [Figure-44](#).

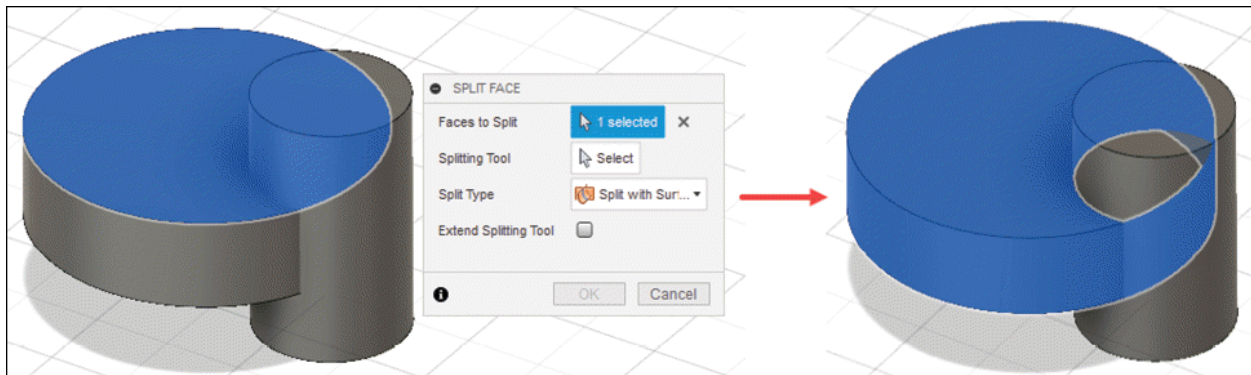


Figure-44. Preview of split face

SPLIT BODY

The **Split Body** tool works similar to **Split Face** tool but in this tool, component or part is selected in place of faces. The procedure to use this tool is discussed next.

- Select **Split Body** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-45](#). The **SPLIT BODY** dialog box will be displayed; refer to [Figure-46](#).

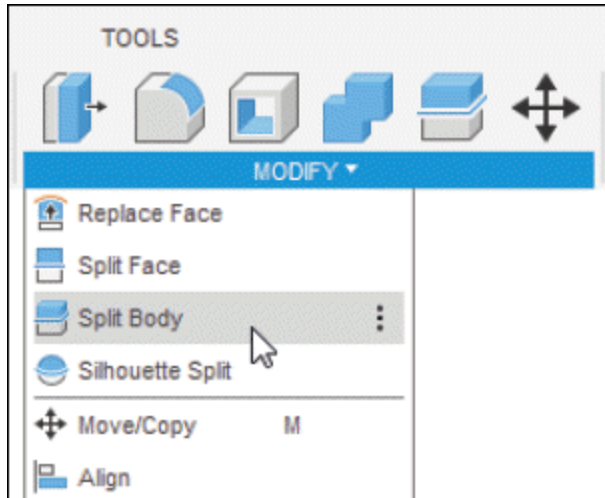


Figure-45. Split Body tool

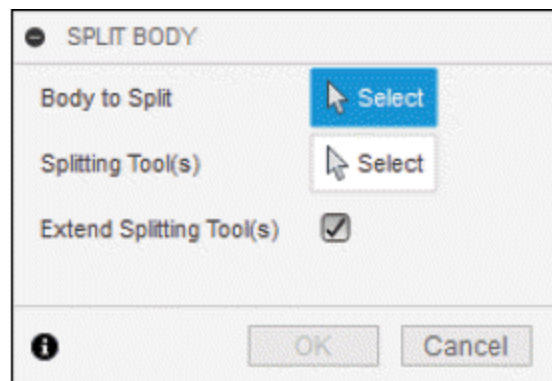


Figure-46. SPLIT BODY dialog box

- The **Body To Split** selection is active by default. Click on the component to be split.
- Click on **Select** button of **Splitting Tool** option and select the body of which splitting will be done.
- Select the **Extend Splitting Tool(s)** check box to extend the tool to completely intersect the target faces.
- After specifying various parameters, click **OK** button of **SPLIT BODY** dialog box to complete the process; refer to [Figure-47](#).

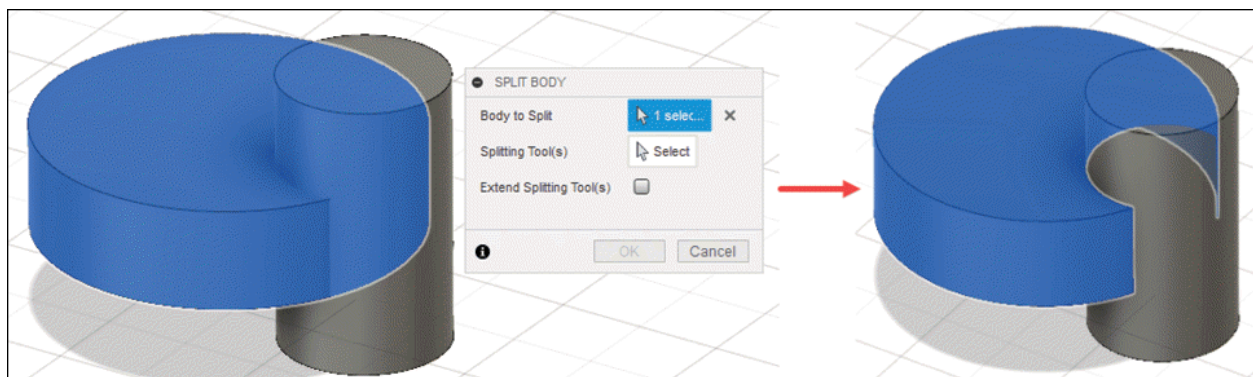


Figure-47. Preview of split body

SILHOUETTE SPLIT

The **Silhouette Split** tool is used to split selected body at the silhouette outline visible from the view direction. The procedure to use this tool is discussed next.

- Select **Silhouette Split** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-48](#). The **SILHOUETTE SPLIT** dialog box will be displayed; refer to [Figure-49](#).

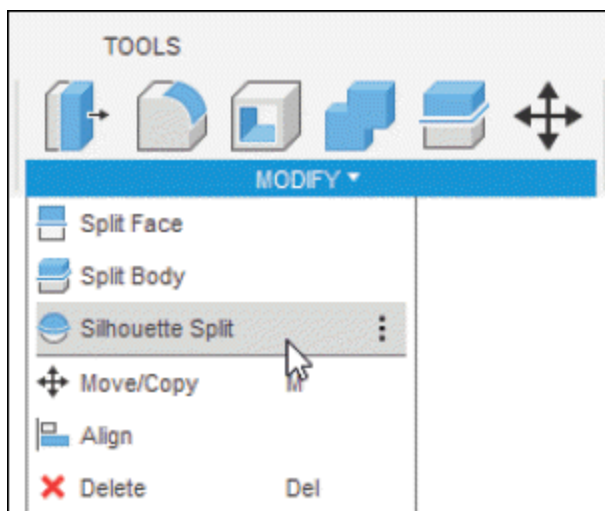


Figure-48. Silhouette Split tool

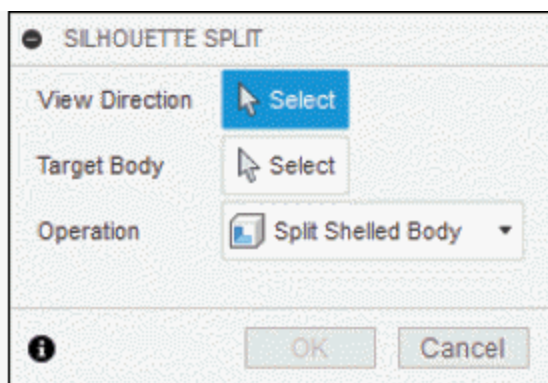


Figure-49. SILHOUETTE SPLIT dialog box

- The **View Direction** selection option is active by default. Click on the plane or face to select the direction to split.

- Click on **Select** button of **Target Body** and select the body which you want to split.
- Select **Split Faces Only** option from **Operation** drop-down, if you want to split the faces of the selected body but keeps it as a single body.
- Select **Split Shelled Body** option from **Operation** drop-down, if you want to split the selected body into two bodies. To use this option , the selected body must be shelled first.
- Select **Split Solid Body** option from **Operation** drop-down, if you want to split the selected body at the parting line. The parting line of the selected body must be planar.
- After specifying the parameters, click **OK** button of **SILHOUETTE SPLIT** dialog box to finish the process; refer to [Figure-50](#).

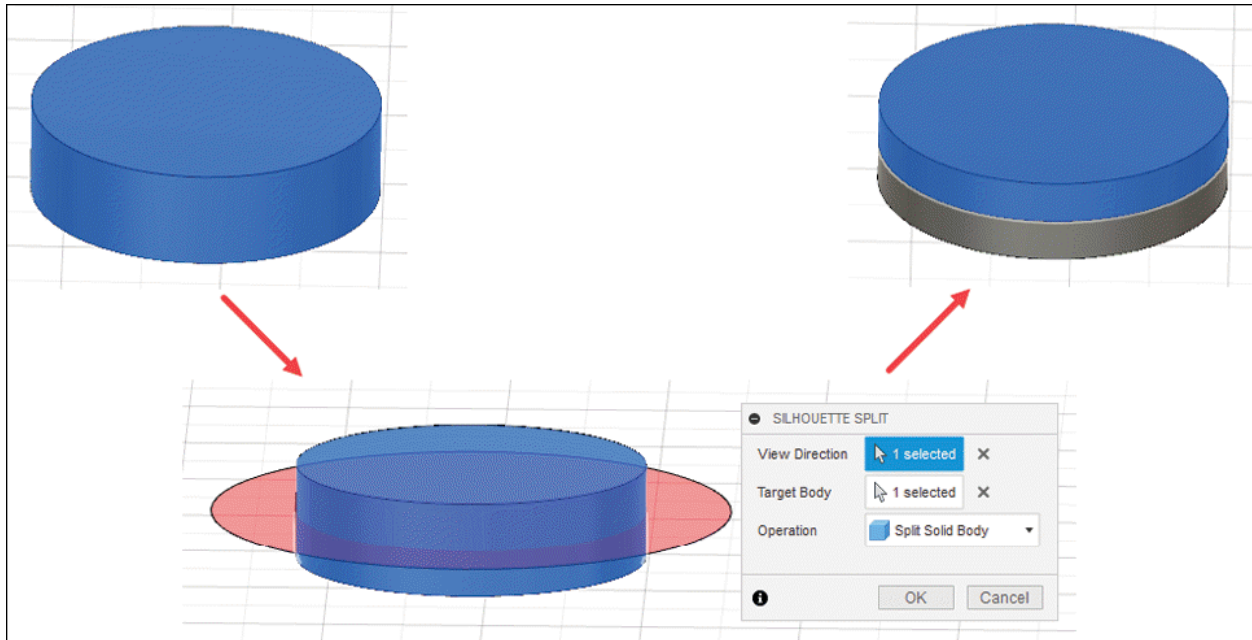


Figure-50. Preview of silhouette split

MOVE/COPY

The **Move/Copy** tool is used to move a face, body, sketch curve, components, or sketch geometry. Using this tool, you can also move the whole geometry. The procedure to use this tool is discussed next.

- Select **Move/Copy** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-51](#). The **MOVE/COPY** dialog box will be displayed; refer to [Figure-52](#).

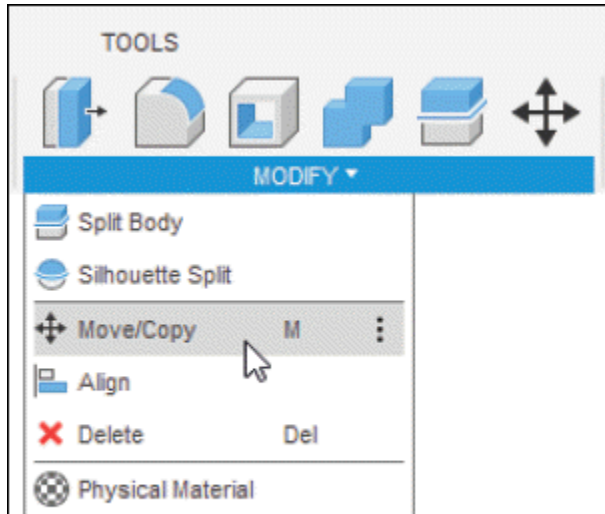


Figure-51. Move Copy tool

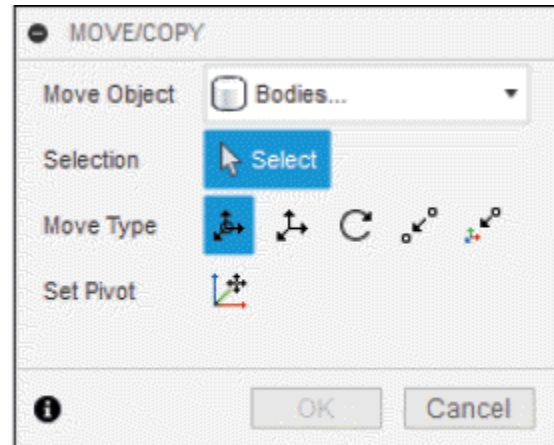


Figure-52. MOVE COPY dialog box

- Select **Component** option from **Move Object** drop-down, if you want to move a component.
- Select **Bodies** option from **Move Object** drop-down, if you want to move a body.
- Select **Faces** option from **Move Object** drop-down, if you want to move a face.
- Select **Sketch Objects** option from **Move Object** drop-down, if you want to move the selected sketch.
- In our case, we are selecting the **Bodies** option from **Move Object** drop-down.
- **Selection** section is active by default. Click on the body to be moved. The updated **MOVE/COPY** dialog box will be displayed.
- Select **Free Move** button from **Move Type** section, if you want to move selected body freely.

- On selecting the **Free Move** button, a Manipulator will be displayed on the selected body. Use the Manipulator to move the body or enter the value in respective field of **MOVE/COPY** dialog box; refer to [Figure-53](#).

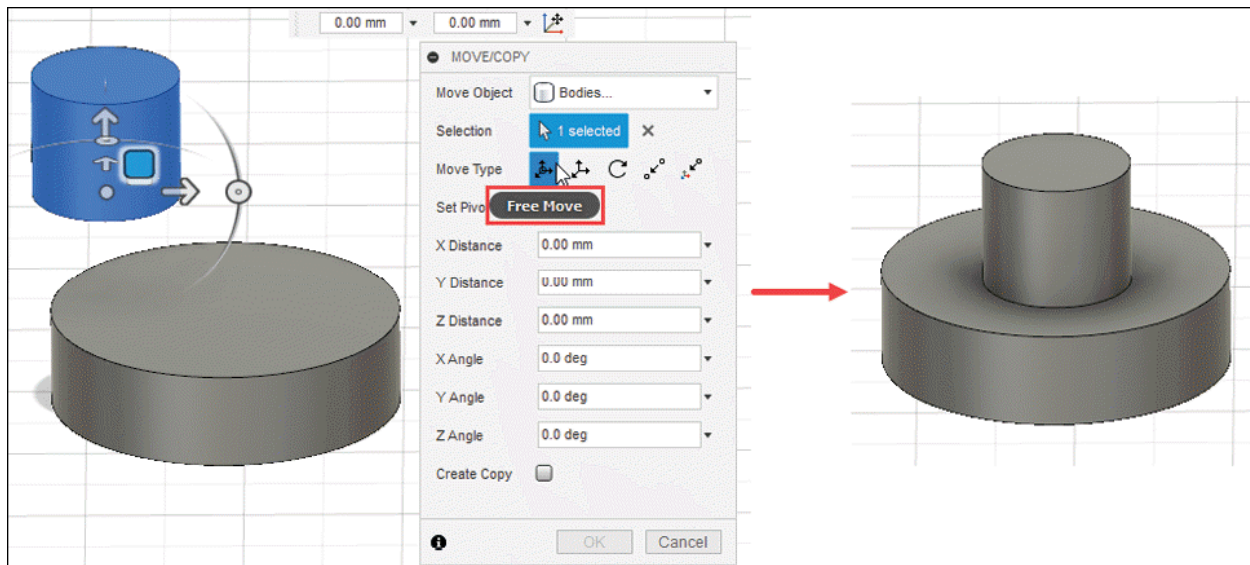


Figure-53. Free Move tool in Move Type section

- Select **Translate** button from **Move Type** section, if you want to move the body along X, Y, and Z directions.
- On selecting the **Translate** button, three arrows will be displayed on the entity. Drag desired arrow to move the body in respective direction. You can also enter the specific value of direction in the respective edit box from **MOVE/COPY** dialog box; refer to [Figure-54](#).

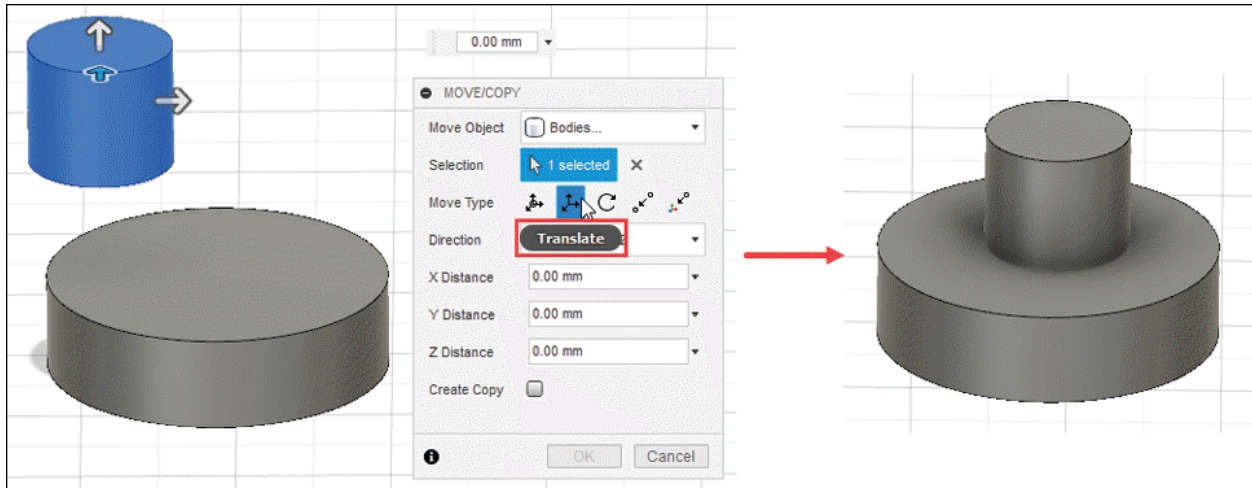


Figure-54. Translate tool in Move Type section

- Select **Rotate** button from **Move Type** section, if you want to rotate the entity.
- On selecting the **Rotate** option, the **Axis** option will be displayed in the **MOVE/COPY** dialog box.
- Click on the **Select** button from **Axis** section of **MOVE/COPY** dialog box and select the axis/edge for rotation of entity. The drag handle will be displayed on the selected face.
- Move the drag handle as required or specify the value in the **Angle** edit box of **MOVE/COPY** dialog box; refer to [Figure-55](#).

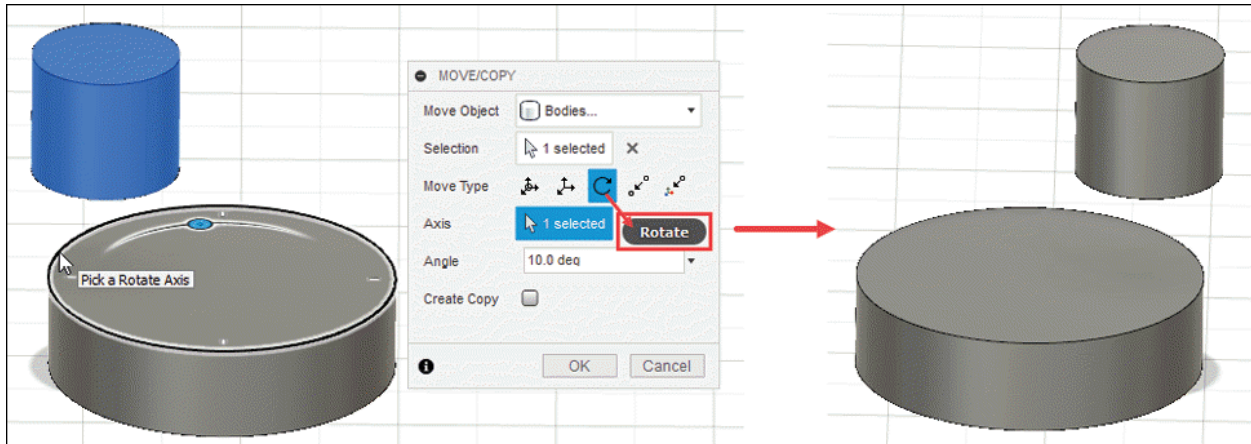


Figure-55. Rotate tool in Move Type section

- Select **Point to Point** button from **Move Type** section, if you want to move the body from one point to another point.
- Click on the **Select** button of **Origin Point** option from **MOVE/COPY** dialog box and select the start point of body from where you want to move the body.
- Click on the **Select** button of **Target Point** option from **MOVE/COPY** dialog box and select the finish point from model.
- After selecting the points, the preview will be displayed.
- If the preview is as required then click on **OK** button from **MOVE/COPY** dialog box to complete the process; refer to [Figure-56](#).

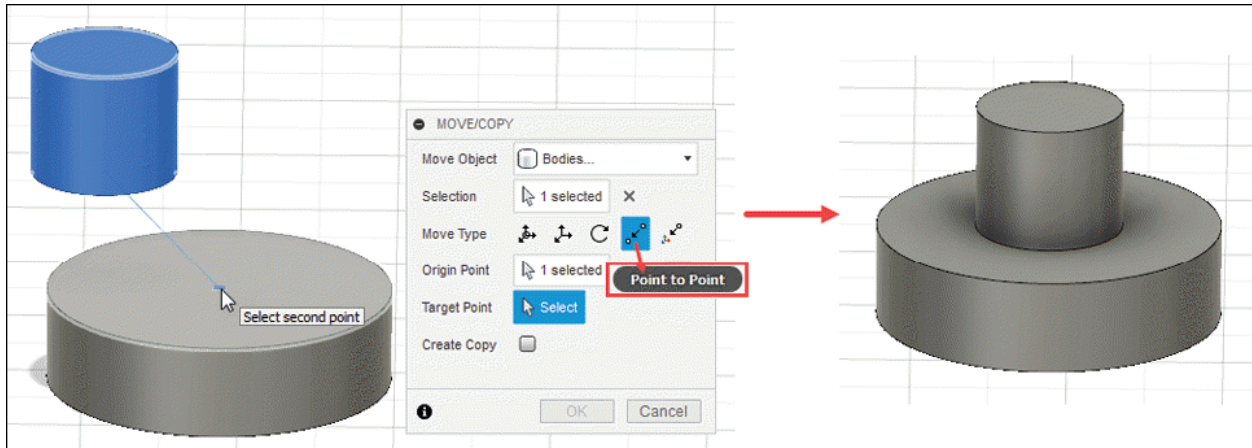


Figure-56. Point to Point tool in Move Type section

- Select **Point to Position** button from **Move Type** section, if you want to move the body by selecting a point and a position from model.
- Select a point of the body. The updated **MOVE/COPY** dialog box will be displayed; refer to [Figure-57](#).

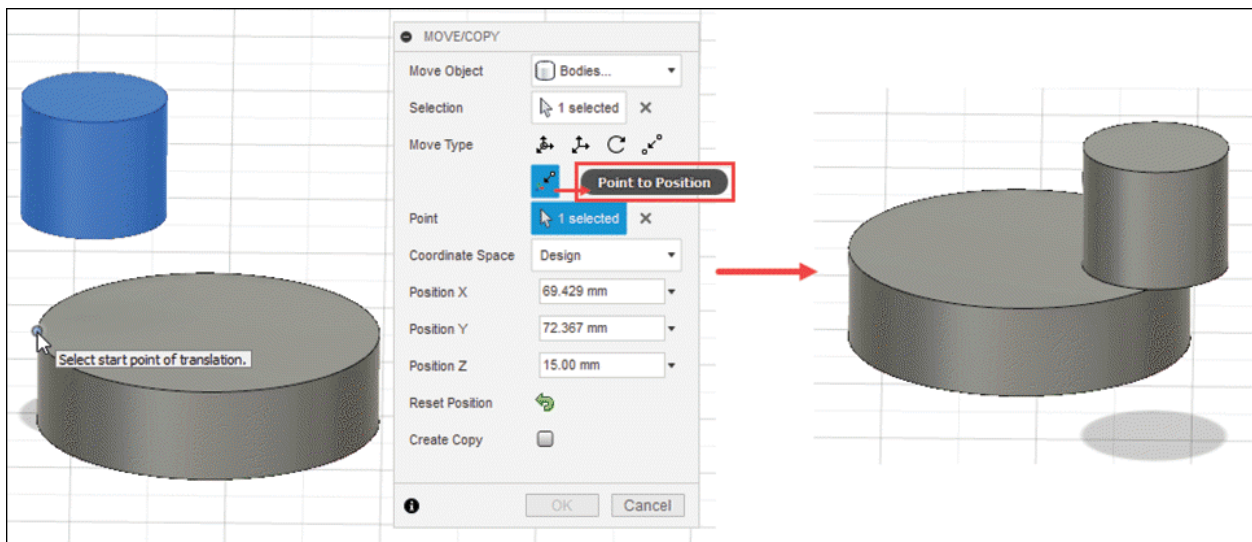


Figure-57. Point to Position tool in Move Type section

- Click in the **Position X** edit box and specify the value. Similarly, specify the value in **Position Y** and **Position Z** edit boxes as required.

- After specifying the parameters, click on the **OK** button from **MOVE/COPY** dialog box to complete the process.

ALIGN

The **Align** tool is used to align selected object to another object. The procedure to use this tool is discussed next.

- Click on the **Align** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-58](#). The **ALIGN** dialog box will be displayed; refer to [Figure-59](#).

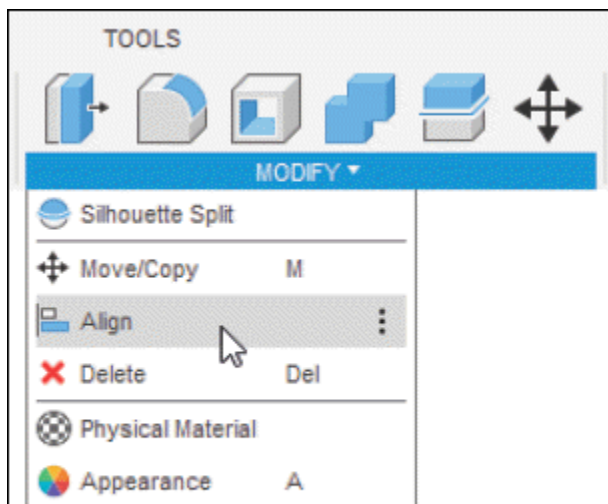


Figure-58. Align tool

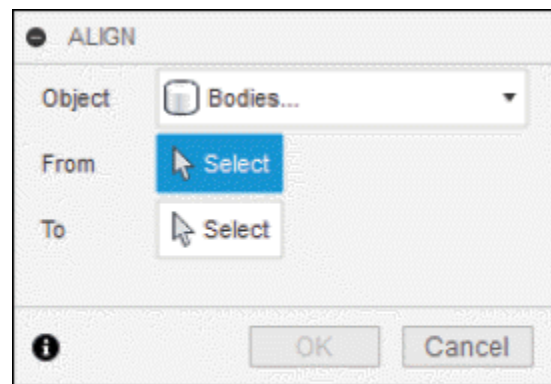


Figure-59. ALIGN dialog box

- Select **Bodies** option from **Object** drop-down of **ALIGN** dialog box if you want to select bodies to align.
- Select **Components** option from **Object** drop-down of **ALIGN** dialog box if you want to select components to align.
- In our case, we are selecting **Bodies** option from **Object** drop-down.

- Click on **Select** button of **From** section of **ALIGN** dialog box and select point on the first body from which you want to align that body.
- Click on **Select** button of **To** section of **ALIGN** dialog box and select point on the second body to which the body to be aligned; refer to [Figure-60](#).

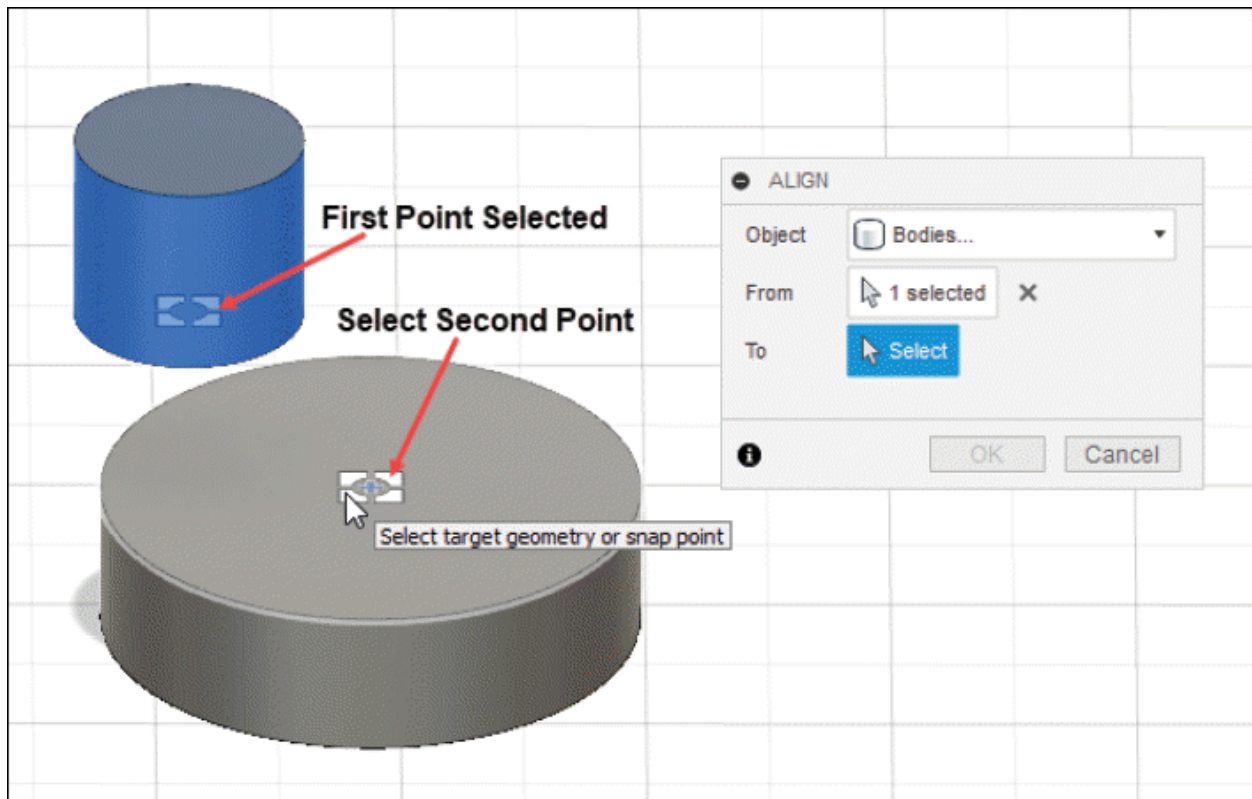


Figure-60. Selection of points to align body

- On selecting the point on second body, the two selected body will align to each other at selected location; refer to [Figure-61](#).

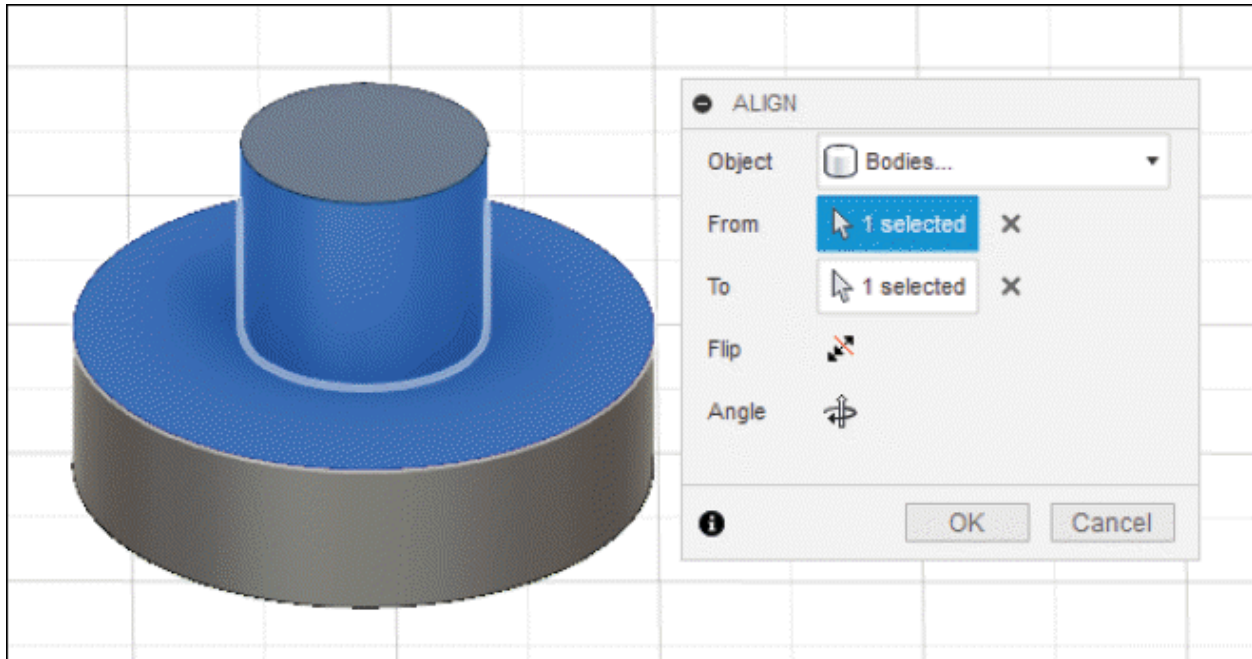


Figure-61. Preview of align

- Click on the **Flip** button, if you want to flip the aligned body.
- Click on **Angle** button, if you want to rotate the aligned body.
- After specifying the parameters, click on the **OK** button from **ALIGN** dialog box to finish the process.

DELETE

The **Delete** tool is used to delete the selected object or component. You can also press **DELETE** from keyboard to do the same. The procedure to use this tool is discussed next.

- Click on the **Delete** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-62](#). The **DELETE** dialog box will

be displayed; refer to [Figure-63](#).

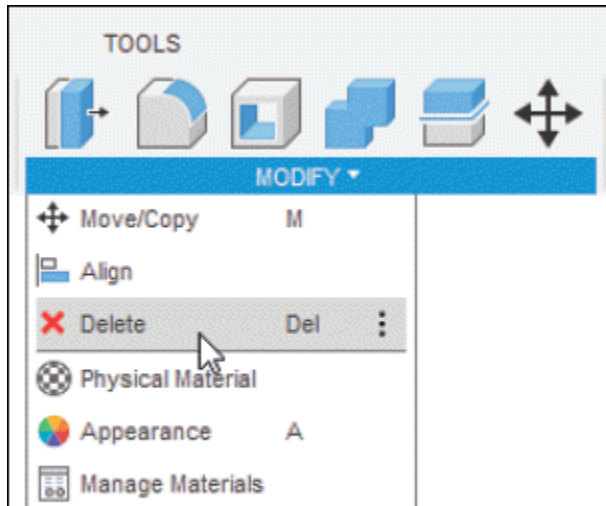


Figure-62. Delete tool

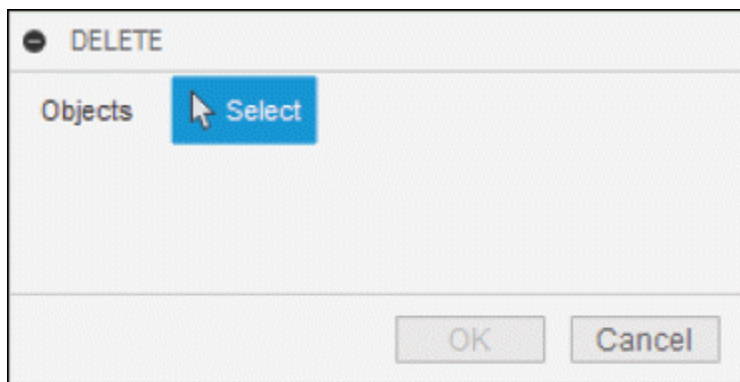


Figure-63. DELETE dialog box

- The **Objects** selection is active by default. Select the object to be deleted from the existing model.
- After selecting desired object, click on the **OK** button from **DELETE** dialog box to delete the selected object; refer to [Figure-64](#).

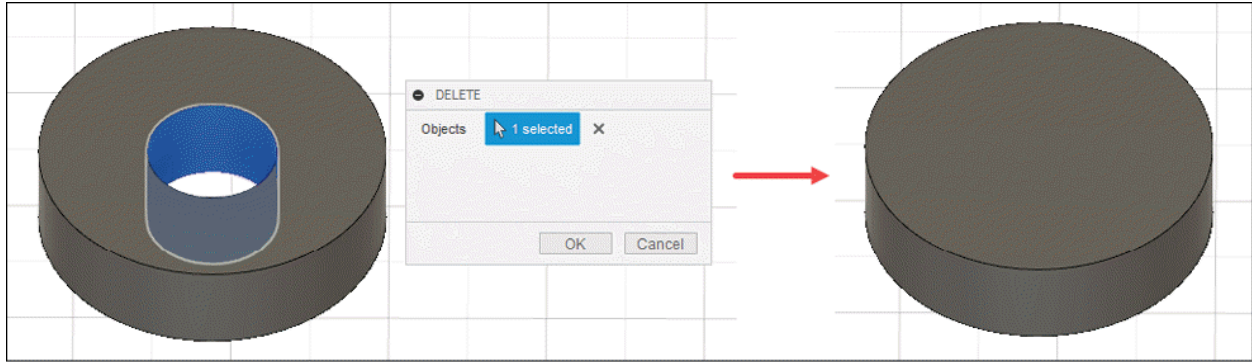


Figure-64. Object deleted

REMOVE TOOL

The **Remove** tool is used to delete selected components and bodies from the design while preserving the operations in the timeline. The procedure to use this tool is given next.

- Click on the **Remove** tool from the **MODIFY** drop-down in the **SOLID** tab of **Ribbon**. The **REMOVE** dialog box will be displayed; refer to [Figure-65](#).

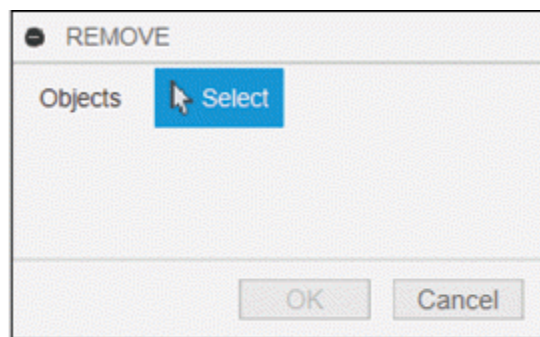


Figure-65. REMOVE dialog box

- Select the body to be removed from the graphics area; refer to [Figure-66](#) and click on the **OK** button. Selected body will be removed from graphics area but still will be available in the timeline bar; refer to [Figure-67](#).

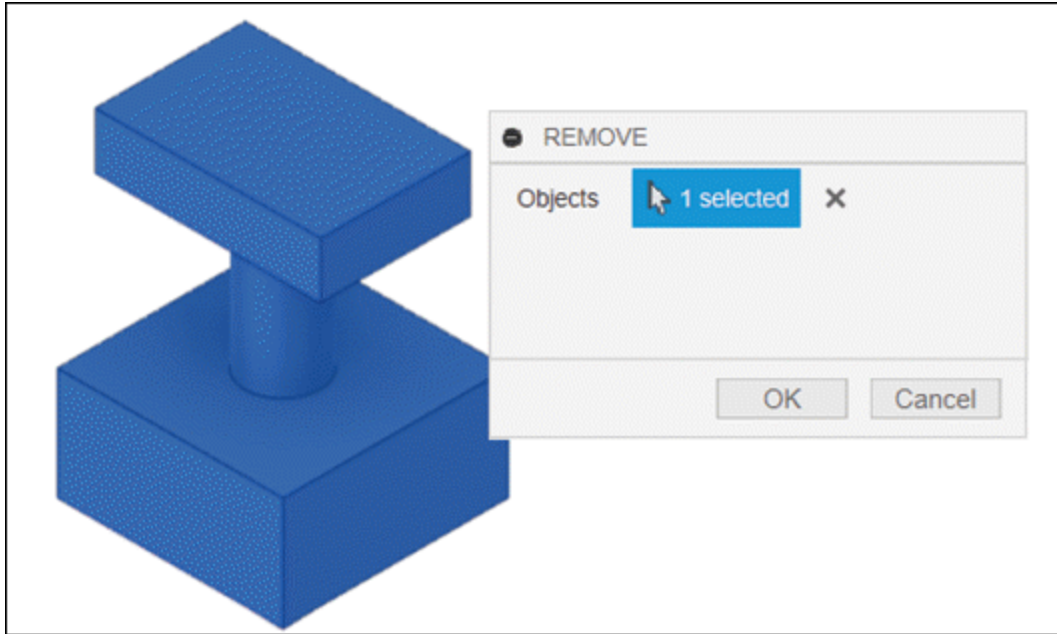


Figure-66. Body selected for removing



Figure-67. Removed body mark in timeline

PHYSICAL MATERIAL

The **Physical Material** tool is used to apply physical and visual material to the component or body. The procedure to use this tool is discussed next.

- Click on the **Physical Material** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-68](#). The **PHYSICAL MATERIAL** dialog box will be displayed; refer to [Figure-69](#).

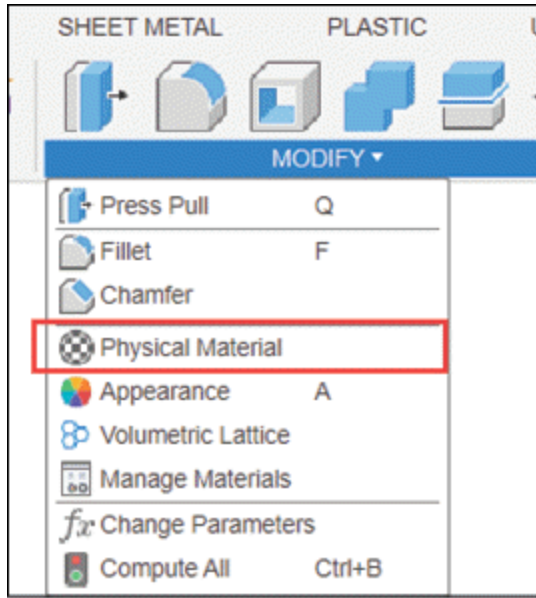


Figure-68. Physical Material tool

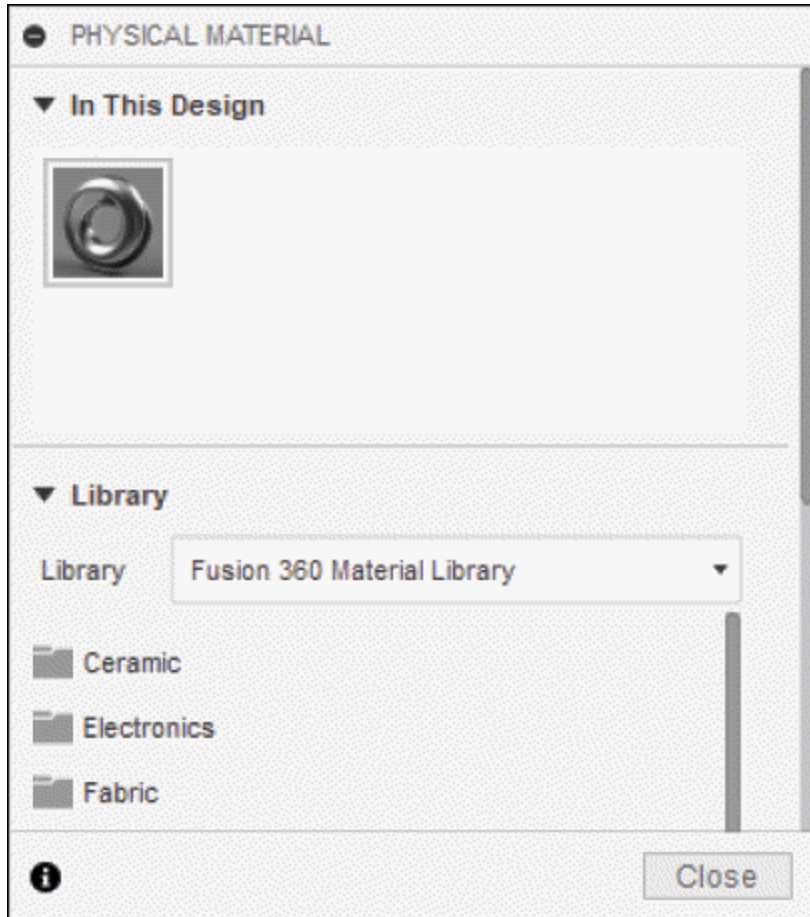


Figure-69. PHYSICAL MATERIAL dialog box

- Select required option from **Library** drop-down.
- Select required physical material from **Library** section of **PHYSICAL MATERIAL** dialog box.
- Now, drag the specific material from the **Library** section and drop it to the body or component. The physical material of body/component will be changed; refer to [Figure-70](#).

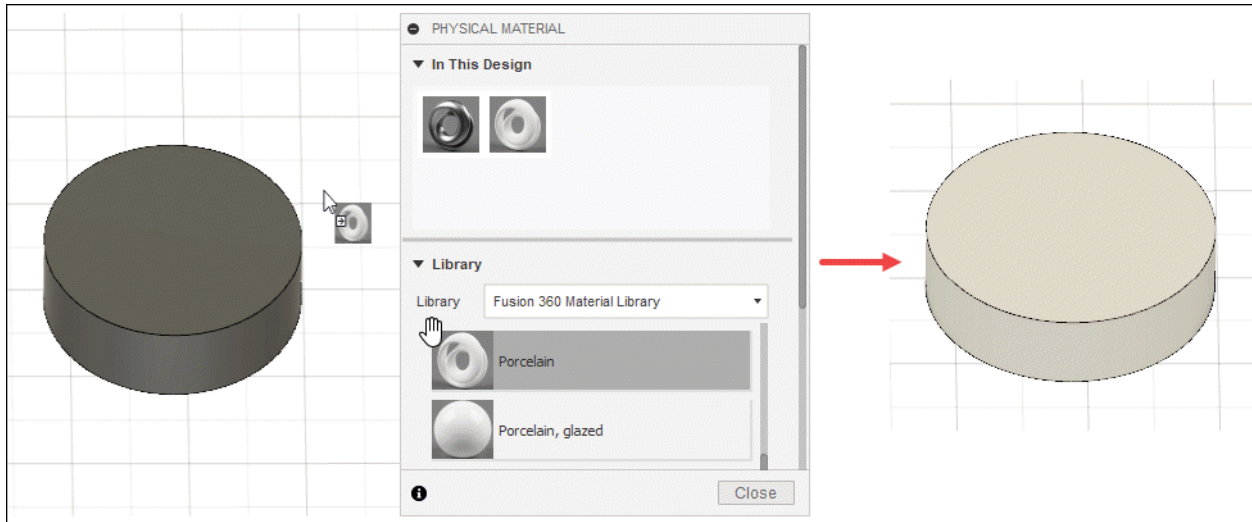


Figure-70. Applying physical material on the body

- After applying material, click on the **Close** button of **PHYSICAL MATERIAL** dialog box to apply changes.

APPEARANCE

The **Appearance** tool is used to change the appearance of body. This tool is generally used to change the color of the body. The procedure to use this tool is discussed next.

- Click on the **Appearance** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-71](#). The **APPEARANCE** dialog box will be displayed; refer to [Figure-72](#).

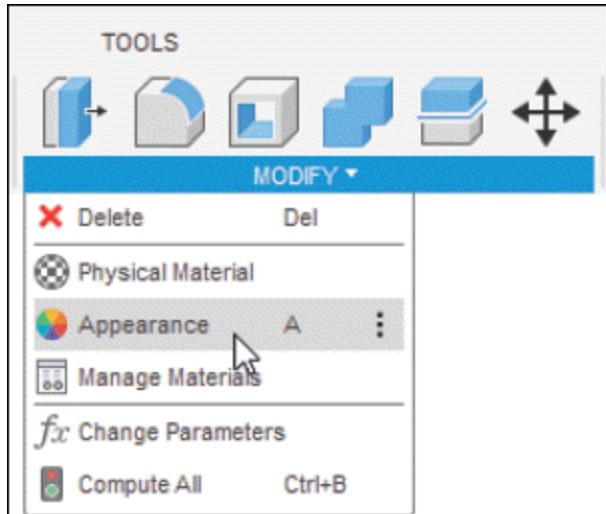


Figure-71. Appearance tool

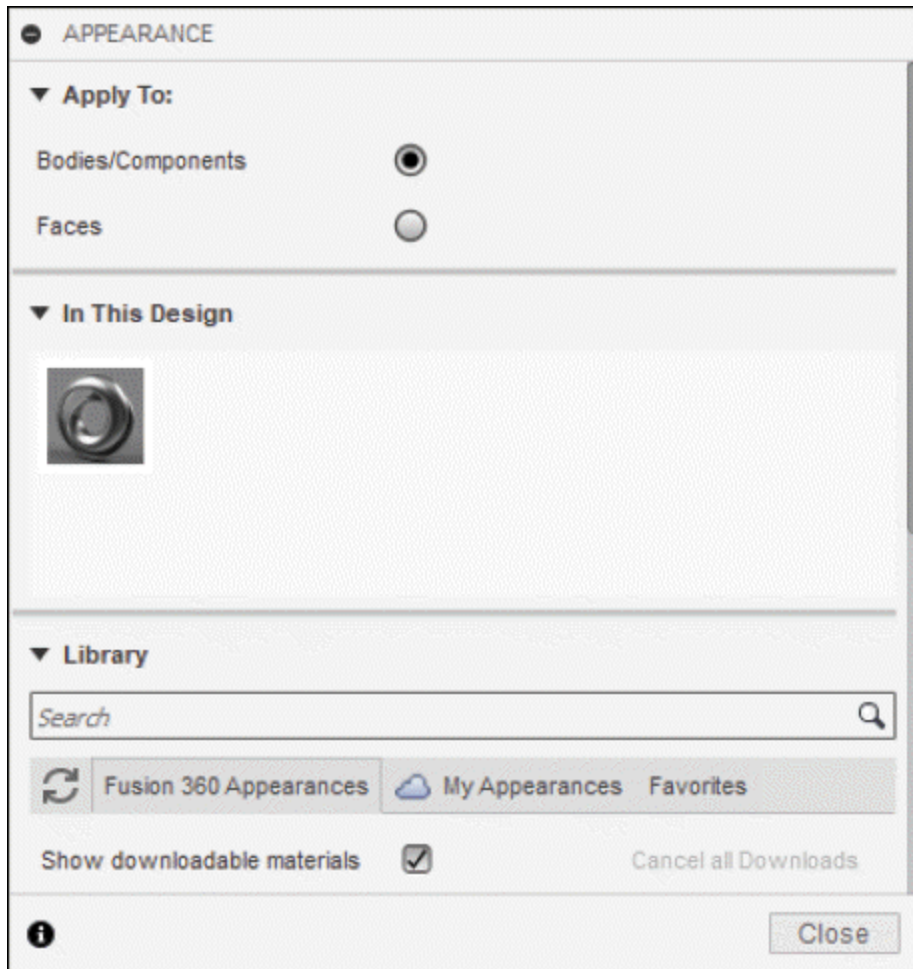


Figure-72. APPEARANCE dialog box

- Select required radio button of **Apply To** section from **APPEARANCE** dialog box to select bodies/components or faces.
- Drag an appearance and drop it to the body or face; refer to [Figure-73](#).

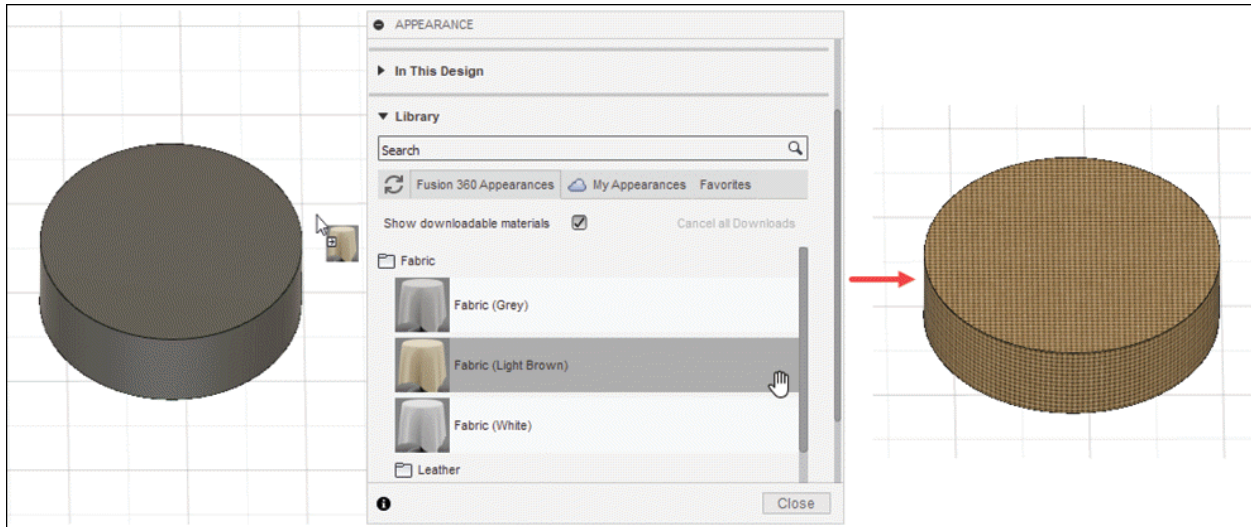


Figure-73. Applying appearance on the body

- After applying the appearance, click on the **Close** button from **APPEARANCE** dialog box to finish the process.

CREATING VOLUMETRIC LATTICE

The **Volumetric Lattice** tool is used to create internal lattice structure within the boundary of selected body. This tool is useful for converting solid body into a 3D printable lattice which can reduce total material consumption. The procedure to use this tool is given next.

- Click on the **Volumetric Lattice** tool from the **MODIFY** panel in the **SOLID** tab of the **Ribbon**. The **VOLUMETRIC LATTICE** dialog box will be displayed; refer to [Figure-74](#).

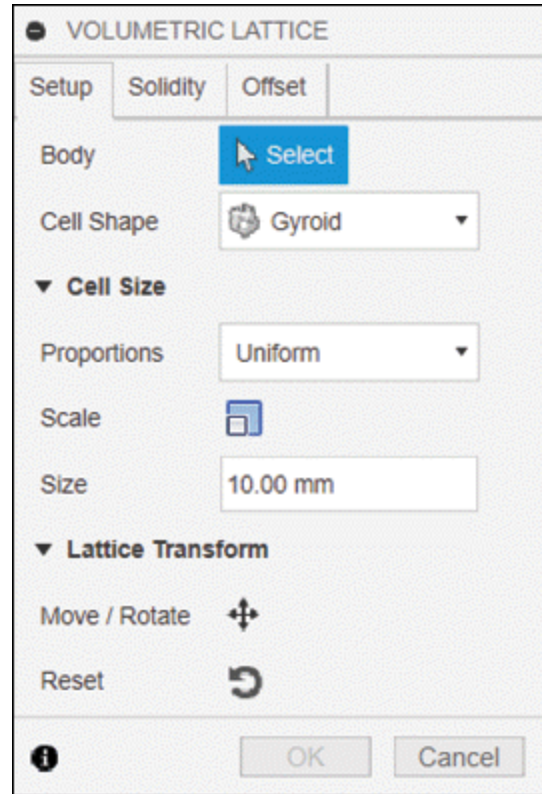


Figure-74. VOLUMETRIC LATTICE dialog box

- Select the body which you want to convert to volumetric lattice. Preview of lattice will be displayed; refer to [Figure-75](#).

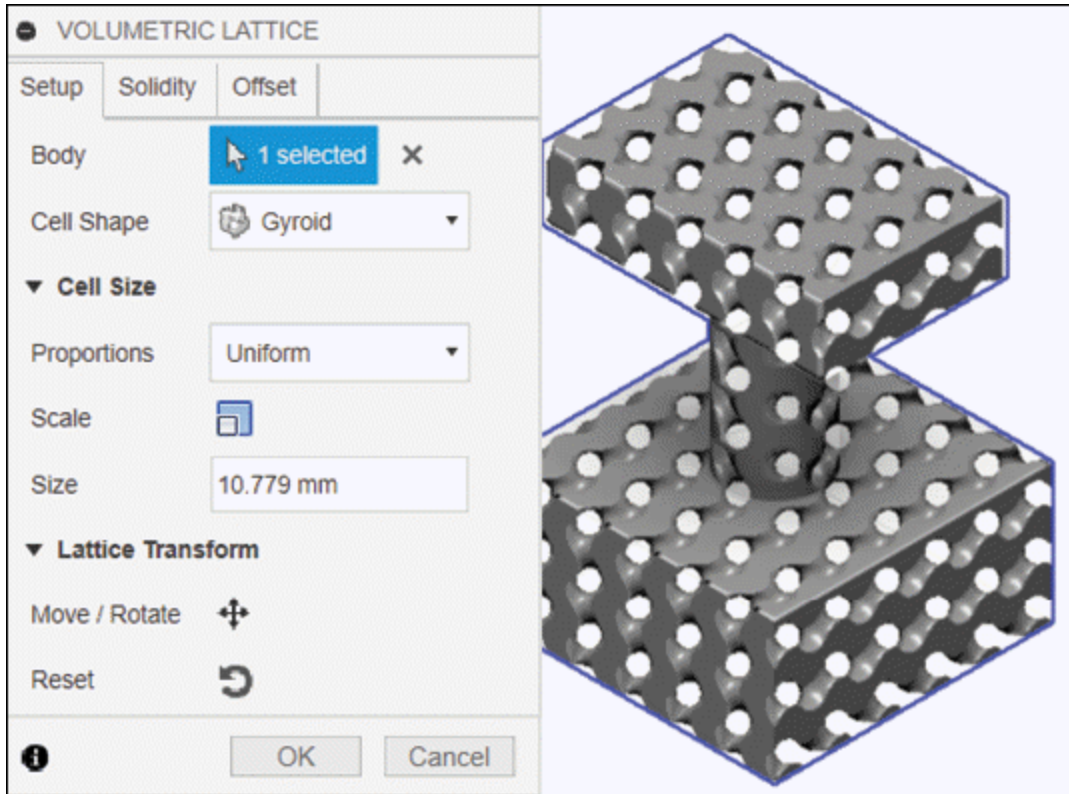


Figure-75. Preview of volumetric lattice

- Select desired shape of lattice element from the **Cell Shape** drop-down; refer to [Figure-76](#).

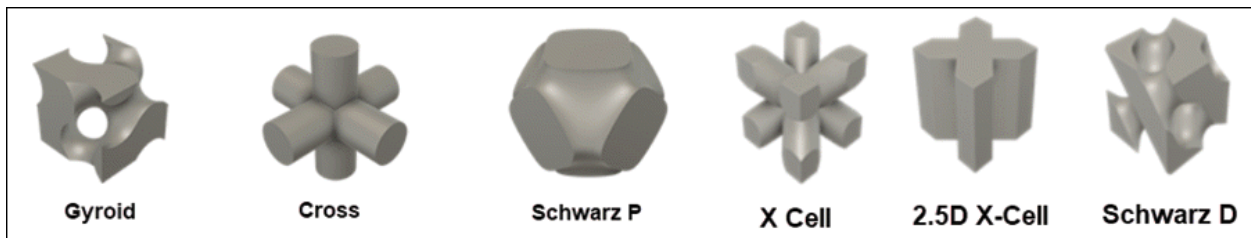


Figure-76. Cell shapes

- Select the **Uniform** option from the **Proportions** drop-down to set the size of cell length, width, and height equal. Specify the size value in the **Size** edit box. You can also set the size of cell using the scale handle displayed on model on selecting the **Scale** toggle button from dialog box; refer

to [Figure-77](#).

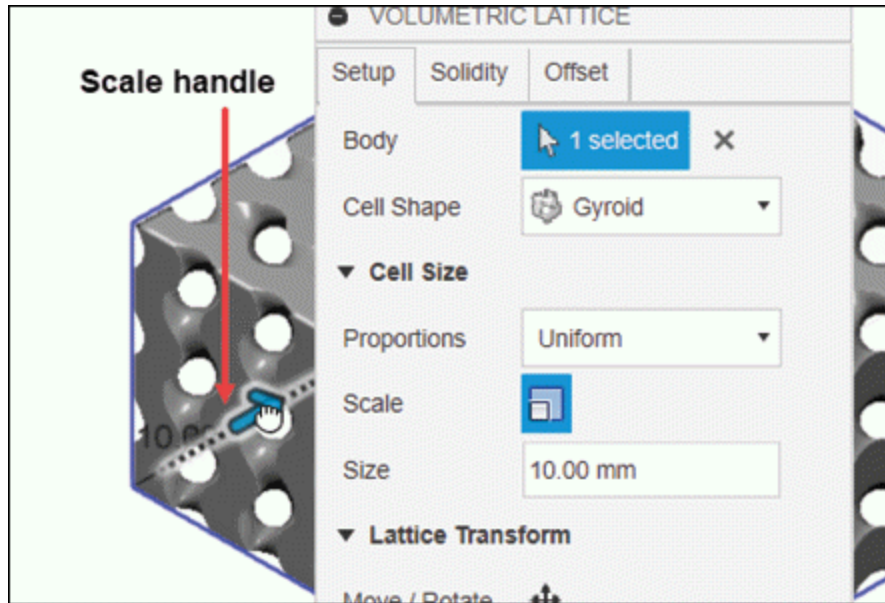


Figure-77. Scale handle

- Select the **Nonuniform** option from the **Proportions** drop-down to specify different sizes of cell shapes in each direction.
- Click on the **Move/Rotate** button from the **Lattice Transform** section of the dialog box and set desired parameters to reorient the lattice cells.
- Click on the **Solidity** tab to define the percentage of model which will be solid after applying lattice; refer to [Figure-78](#).

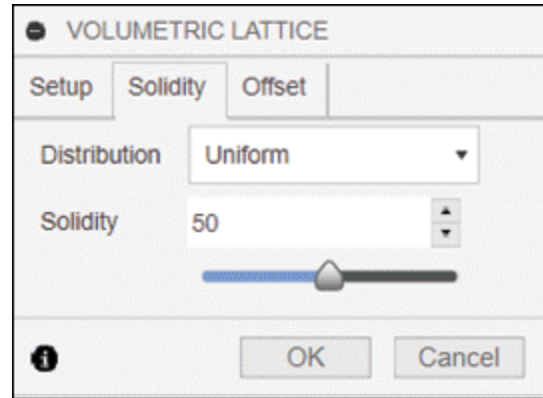


Figure-78. Solidity tab

- Click on the **Offset** tab from the dialog box to set different solidity percentage for selected face; refer to [Figure-79](#). Click on the **Add Objects** button from the dialog box and select face of model to which you want to apply different solidity; refer to [Figure-80](#).

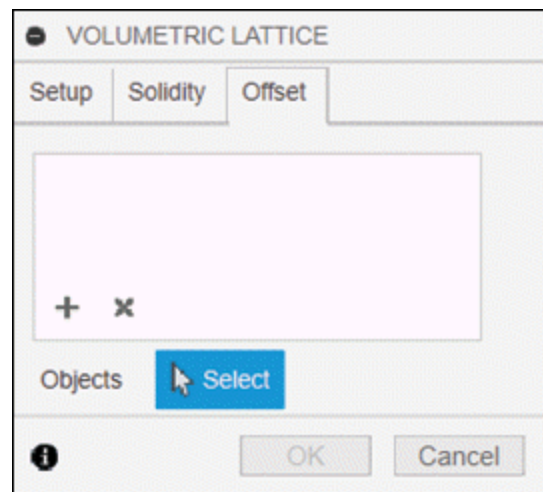


Figure-79. Offset tab

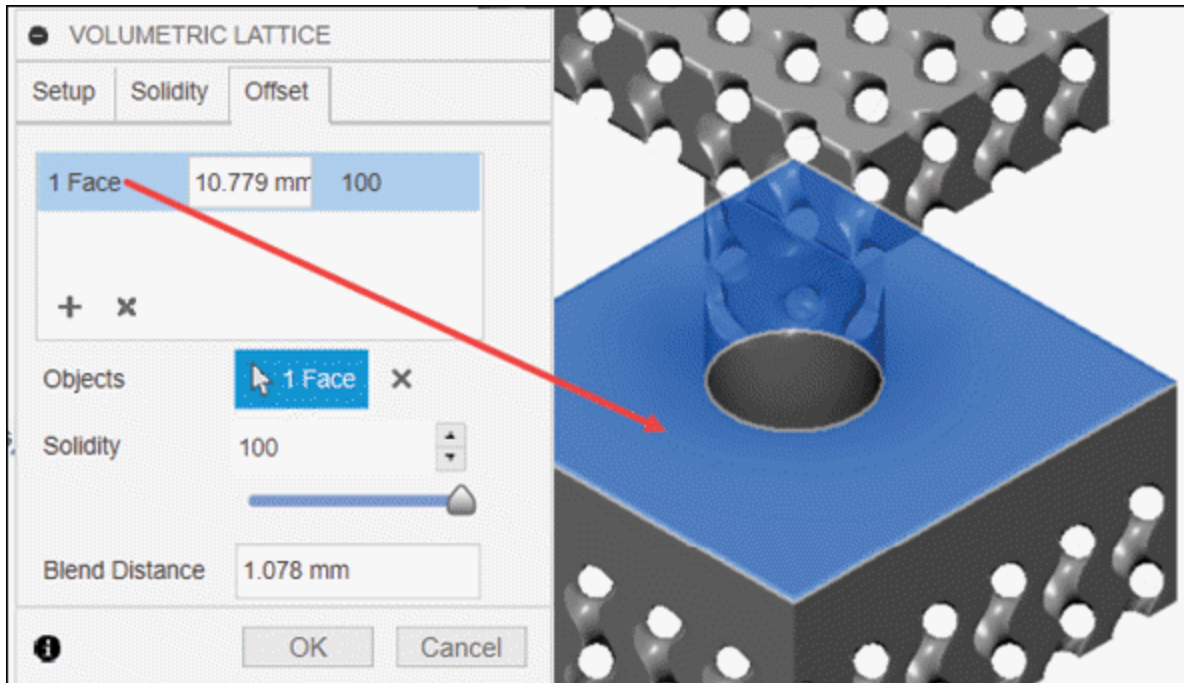


Figure-80. Face selected for offset solidity

- Set the parameters as discussed earlier and click on the **OK** button to create lattice.

MANAGE MATERIALS

The **Manage Materials** tool is used to define the material properties shown in **PHYSICAL MATERIAL** and **APPEARANCE** dialog boxes discussed earlier. The procedure to use this tool is discussed next.

- Click on the **Manage Materials** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-81](#). The **Material Browser** dialog box will be displayed; refer to [Figure-82](#).

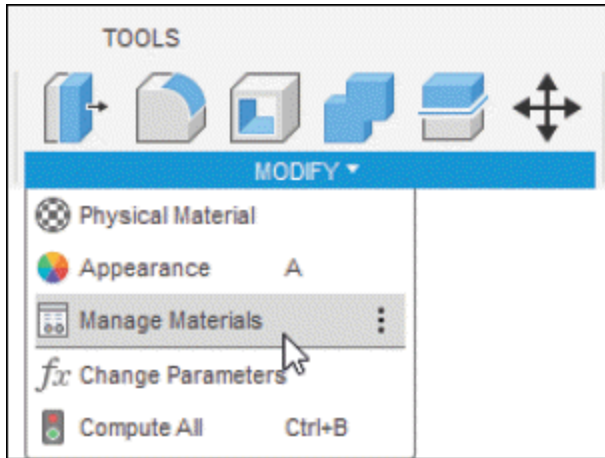


Figure-81. Manage Materials tool

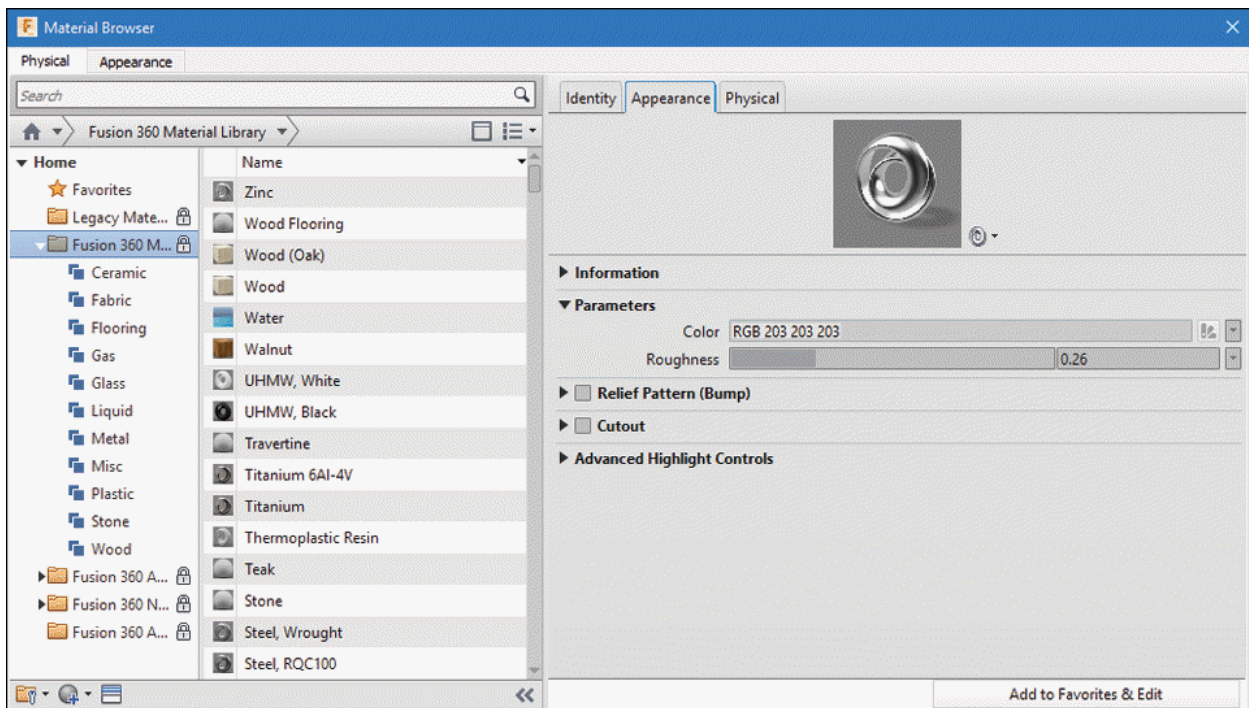


Figure-82. Material Browser dialog box

- Click on any material or appearance from list section to check or modify the property of selected material or appearance.
- To edit properties of any material, select it from the list and then click on the **Add to Favorites & Edit** button at

the bottom-right in the dialog box.

- Change the properties as required using the options displayed in the right area of **Material Browser** dialog box and click on the **OK** button to apply changes.
- Close the dialog box once changes have been applied. Next time, when you will be applying material, make sure to select them from favorites list. You will learn more about modifying material properties in Chapter 20.

CHANGE PARAMETERS

The **Change Parameters** tool is used to change the parameters and apply equations to the model. The procedure to use this tool is discussed next.

- Click on the **Change Parameters** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-83](#). The **Parameters** dialog box will be displayed; refer to [Figure-84](#).

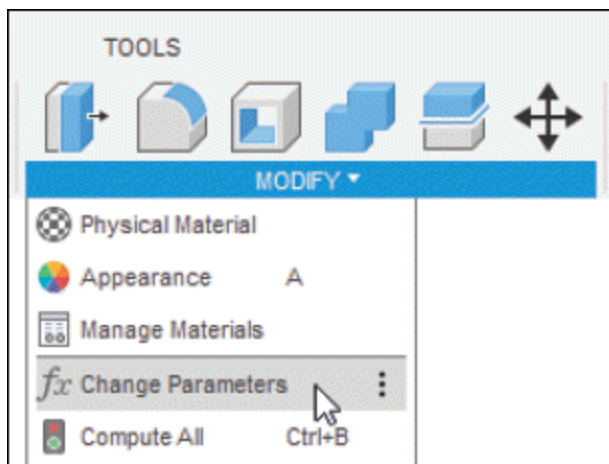


Figure-83. Change Parameters tool

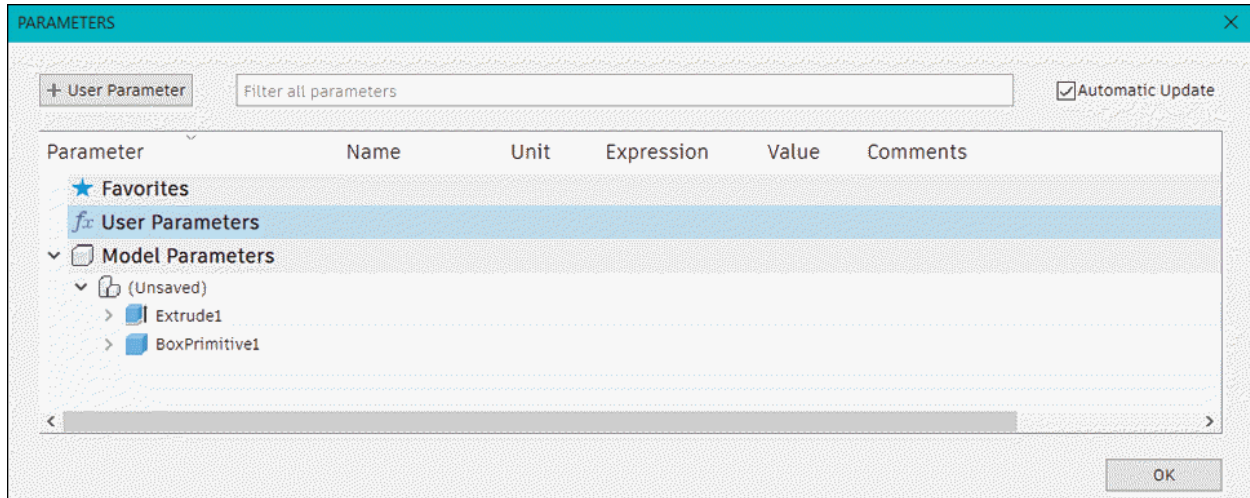


Figure-84. Parameters dialog box

- If you want to create some parameters which are to be used later in equation then click on the **+ User Parameter** button from the **Parameters** dialog box. The **Add User Parameter** dialog box will be displayed; refer to [Figure-85](#).

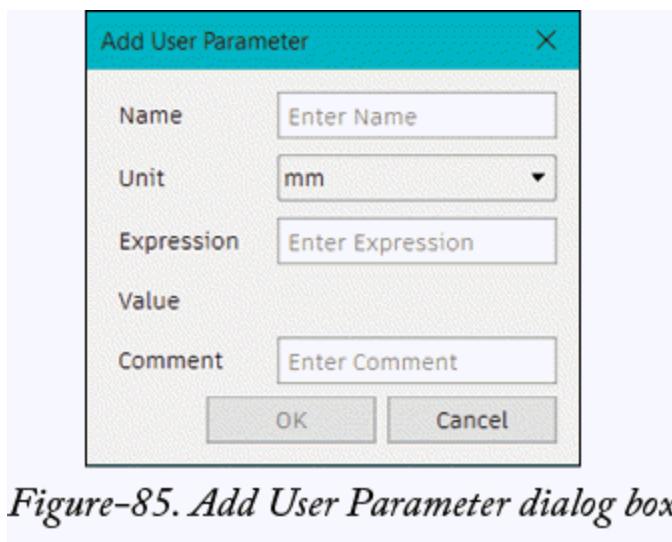


Figure-85. Add User Parameter dialog box

- Click in the **Name** edit box and specify the name of parameter.
- Click on the **Unit** drop-down and select desired unit of the

parameter.

- Click on the **Expression** edit box and enter desired expression like $d1/4$. The **Value** will be displayed.
- Click in the **Comment** edit box and enter required comment.
- After specifying the parameters, click on the **OK** button from **Add User Parameter** dialog box. The user defined parameter will be added in **Parameters** dialog box; refer to [Figure-86](#).

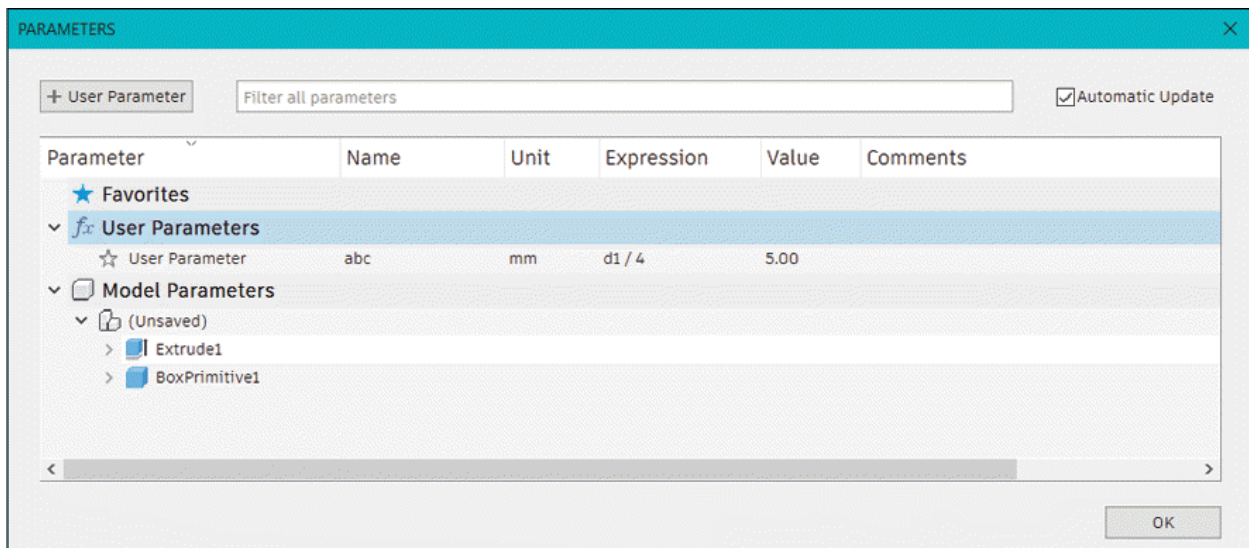


Figure-86. User Parameter added

- Click on the **Model Parameter** node from **Parameters** dialog box, all the model features will be displayed.
- Click on desired model feature node to check its parameters, the parameters applied in that feature will be displayed; refer to [Figure-87](#).

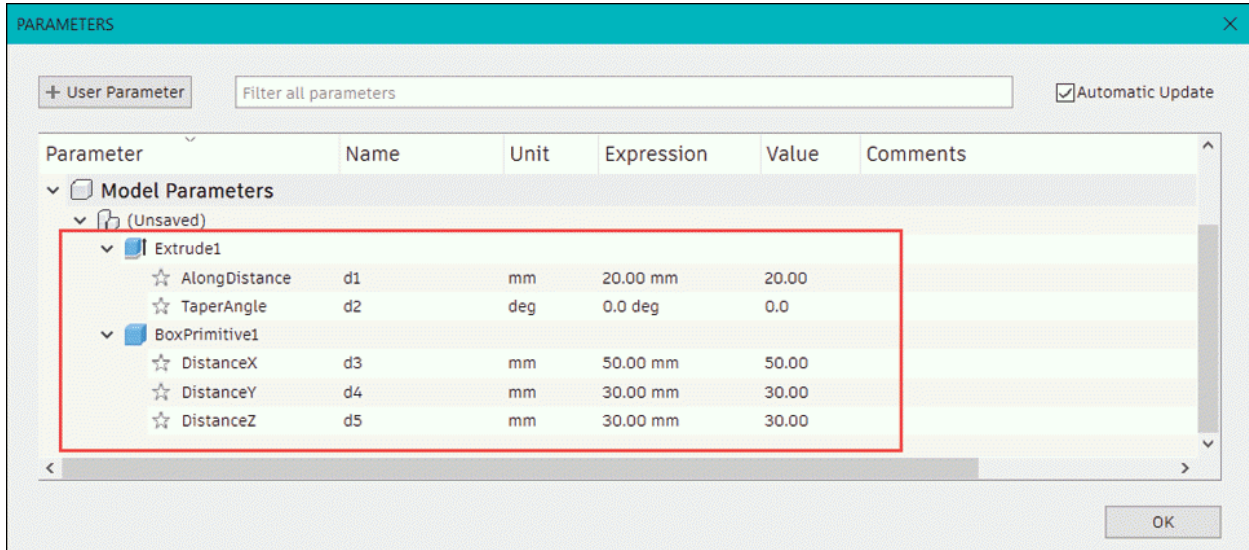


Figure-87. Parameters applied in the model

- If you want to apply equation in parameters then click on the dimension parameter and enter desired equation like $d1*2$; refer to [Figure-88](#). Note that here d1 is dimension number applied automatically to the sketch dimension.

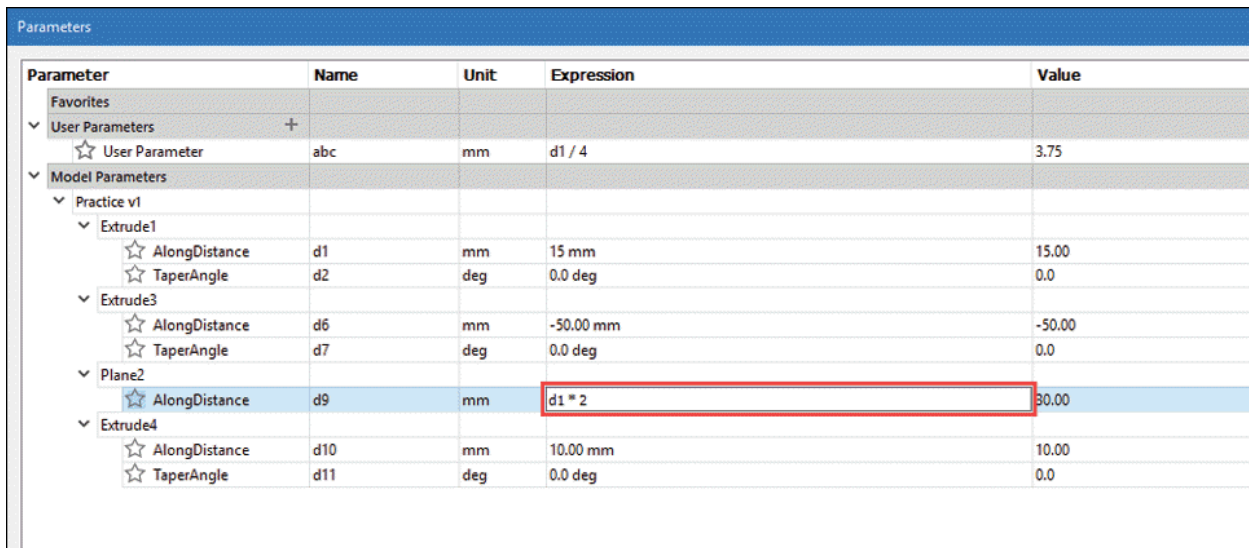


Figure-88. Entering equation

- After specifying the parameters, click on the **OK** button from **Parameters** dialog box

- The equation specified in the **Parameters** dialog box will be applied to the model; refer to [Figure-89](#).

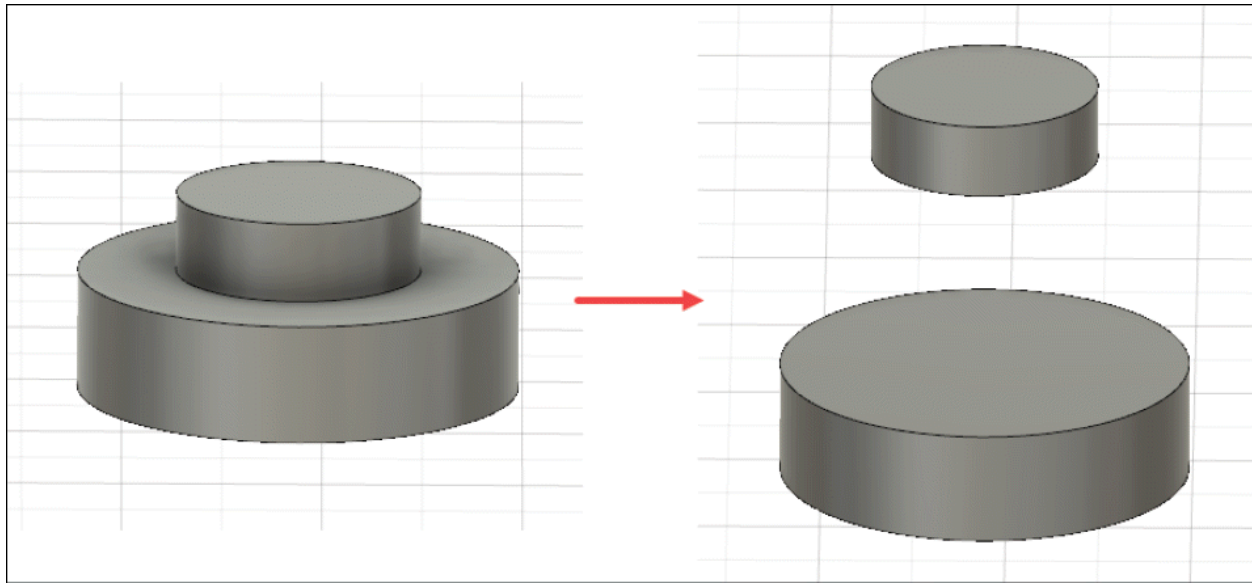


Figure-89. Parameter changed

Click on the **Compute All** tool from the **MODIFY** drop-down to re-calculate all the parameters.

PRACTICAL 1

In this practical, you will create a model of hook as shown in [Figure-90](#). The dimensions of the hook are given in [Figure-91](#).

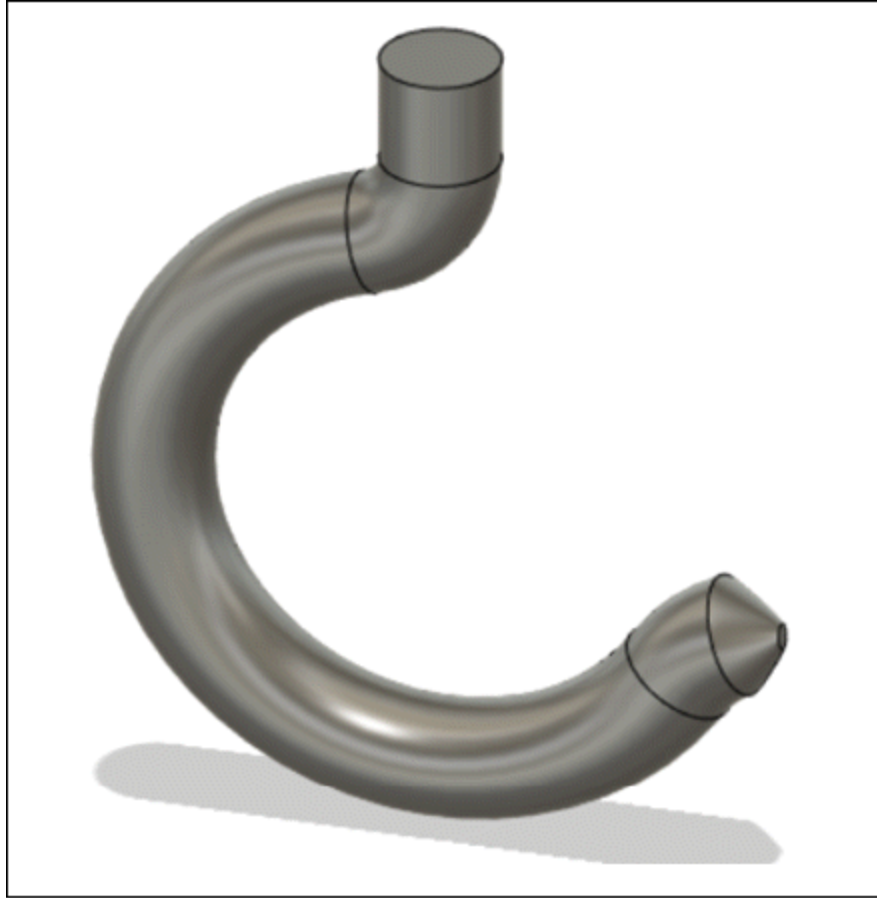


Figure-90. Hook model

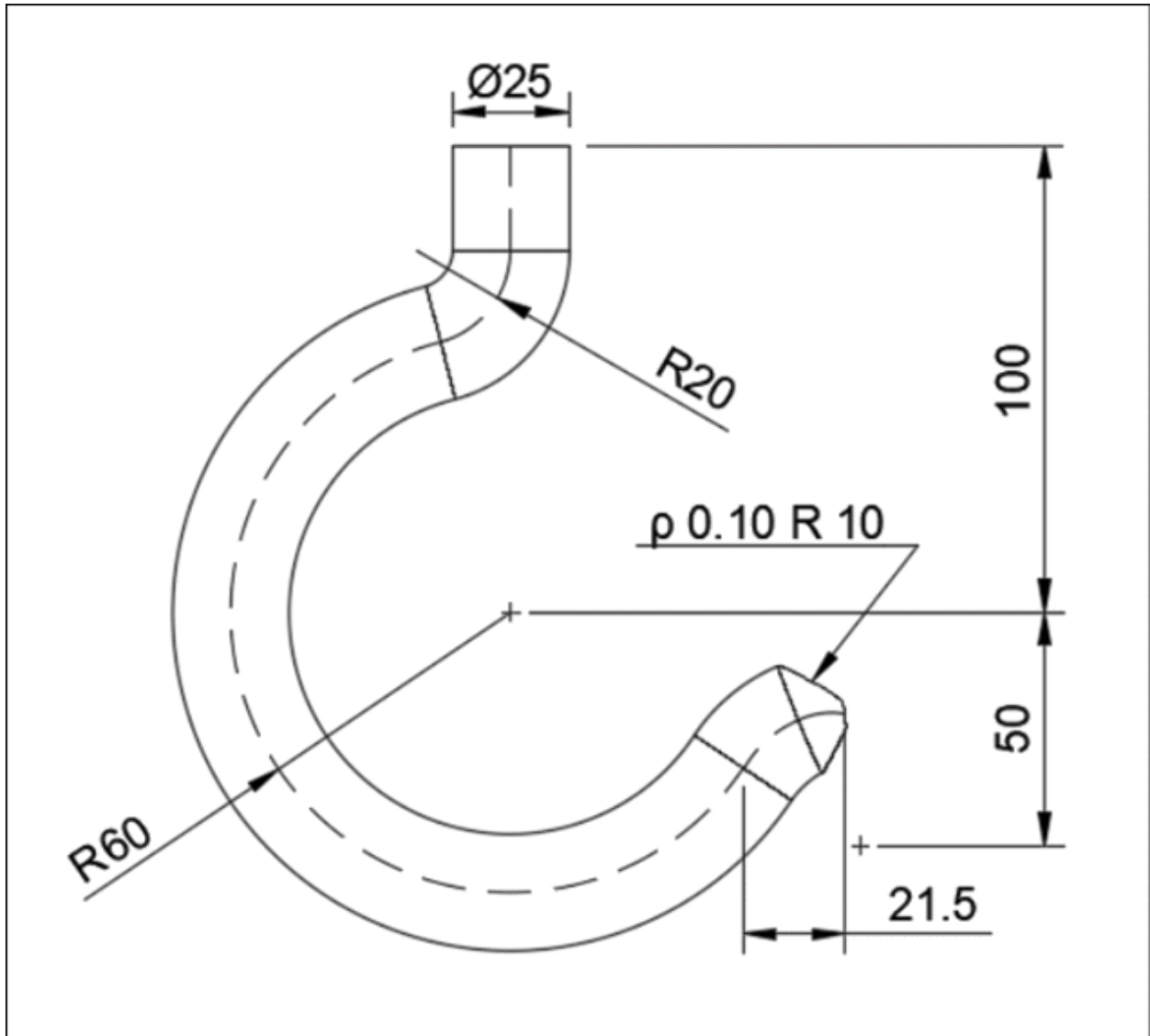


Figure-91. Practical 1 Hook drawing

Creating sketch for model

- Start a new sketch using **Top Plane**.
- Click on the **Circle** tool and create the circle of diameter **120** taking coordinate system as center.
- Draw a straight line starting from top quadrant point and having length of approximately **40**.

- Draw a three point arc in the bottom area of the circle; refer to [Figure-92](#). (It should look like the one given in figure. Accuracy is no required.)
- Click on the **Trim Entities** tool and trim the portion between the straight line and the arc; refer to [Figure-93](#).

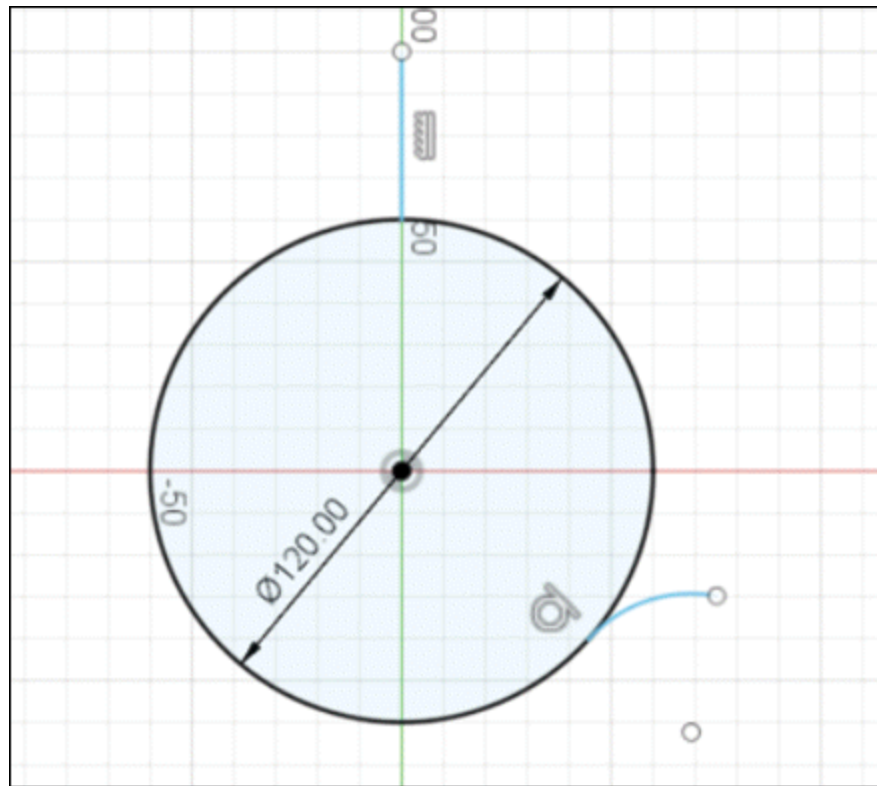


Figure-92. Circle arc and line created

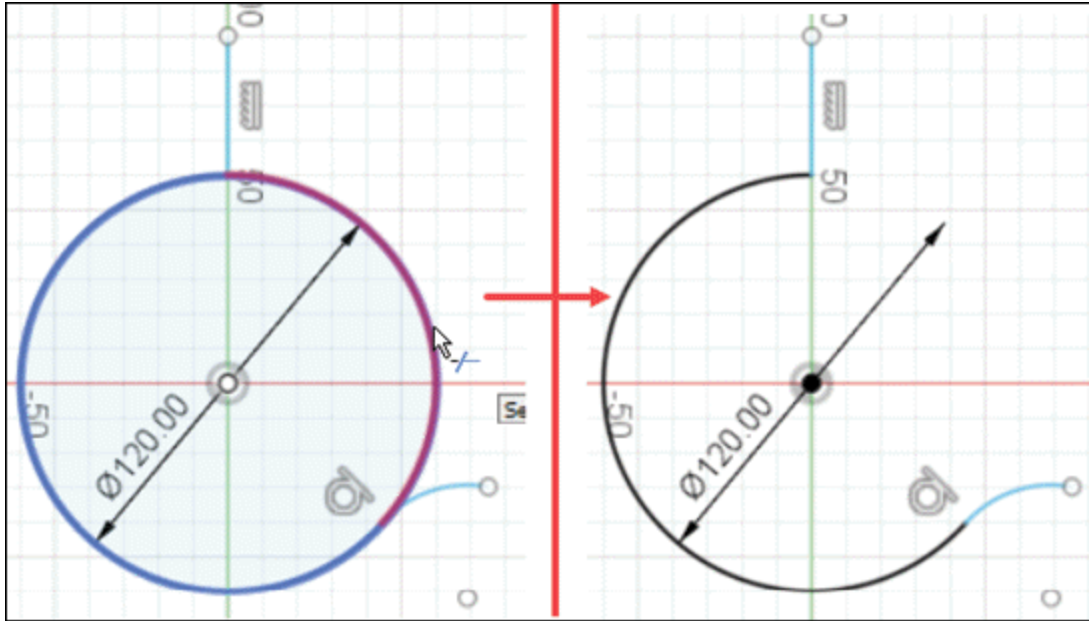


Figure-93. Trimming circle

- Click on the **Fillet** tool from **MODIFY** panel in **Ribbon** and apply a fillet of **20** radius at the intersection of line and circle; refer to [Figure-94](#).
- Click on the **Sketch Dimension** tool from **Ribbon** and apply the dimensions as shown in [Figure-95](#). Note that you can toggle the diameter dimension to radius dimension using shortcut menu.
- Click on the **Finish Sketch** button to exit the sketching environment.

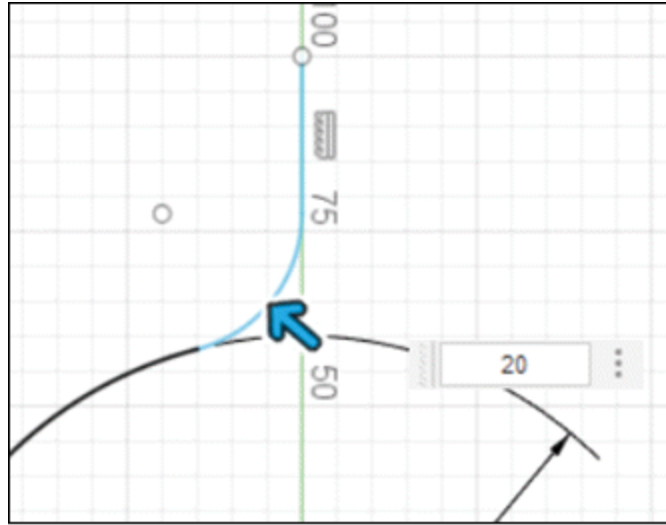


Figure-94. Applying fillet

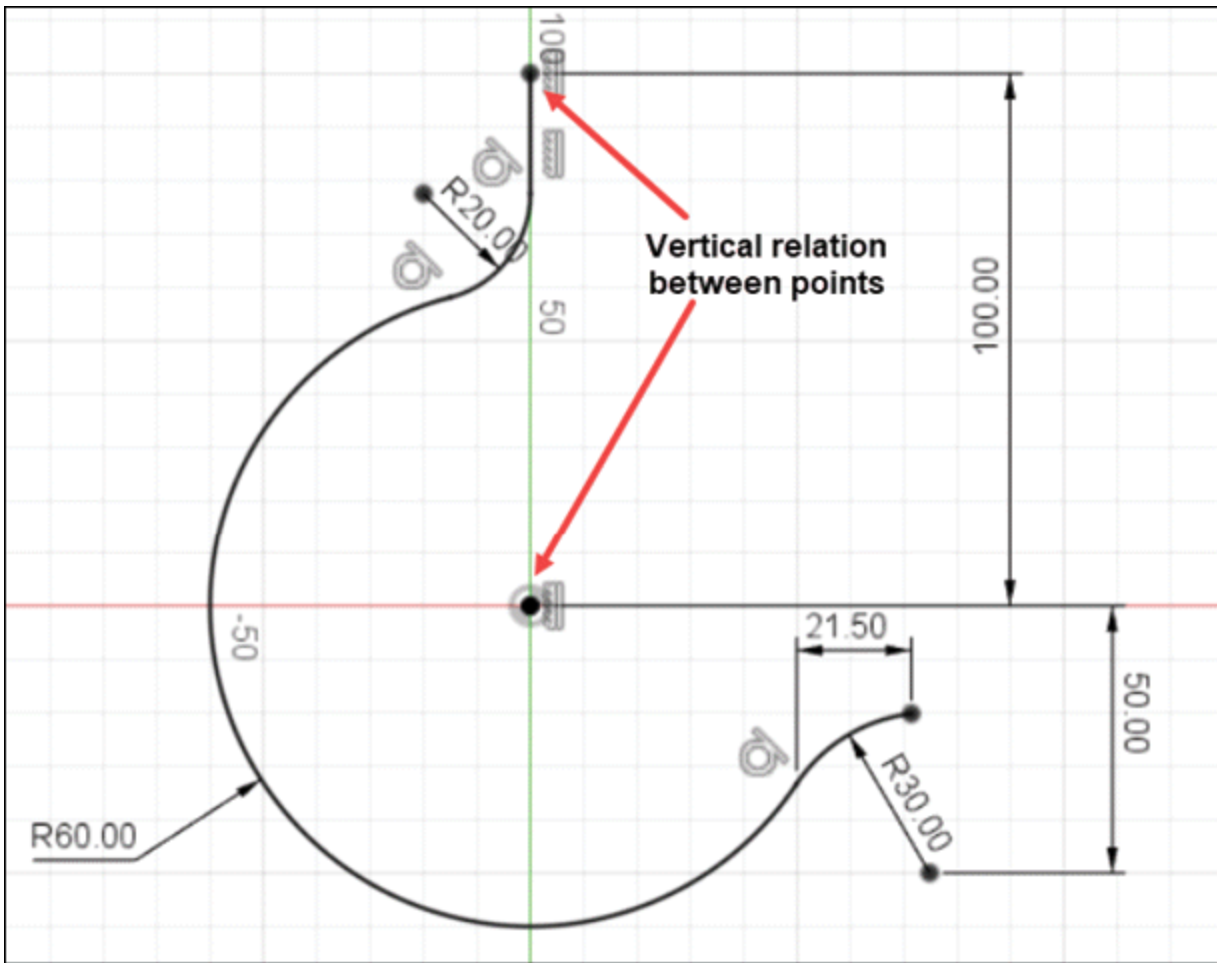


Figure-95. Dimensioning sketch

Creating Plane and Second Sketch

Earlier, the sketch was created for path of sweep feature and now we will create sketch for section.

- Toggle to Isometric view of model using **ViewCube** and click on the **Offset Plane** tool from **CONSTRUCT** drop-down in the **Ribbon**. The **OFFSET PLANE** dialog box will be displayed.
- Create a plane offset of XY plane and distance up to top point of sketch; refer to [Figure-96](#).
- After setting parameters as shown in [Figure-96](#), click on the **OK** button from the dialog box.

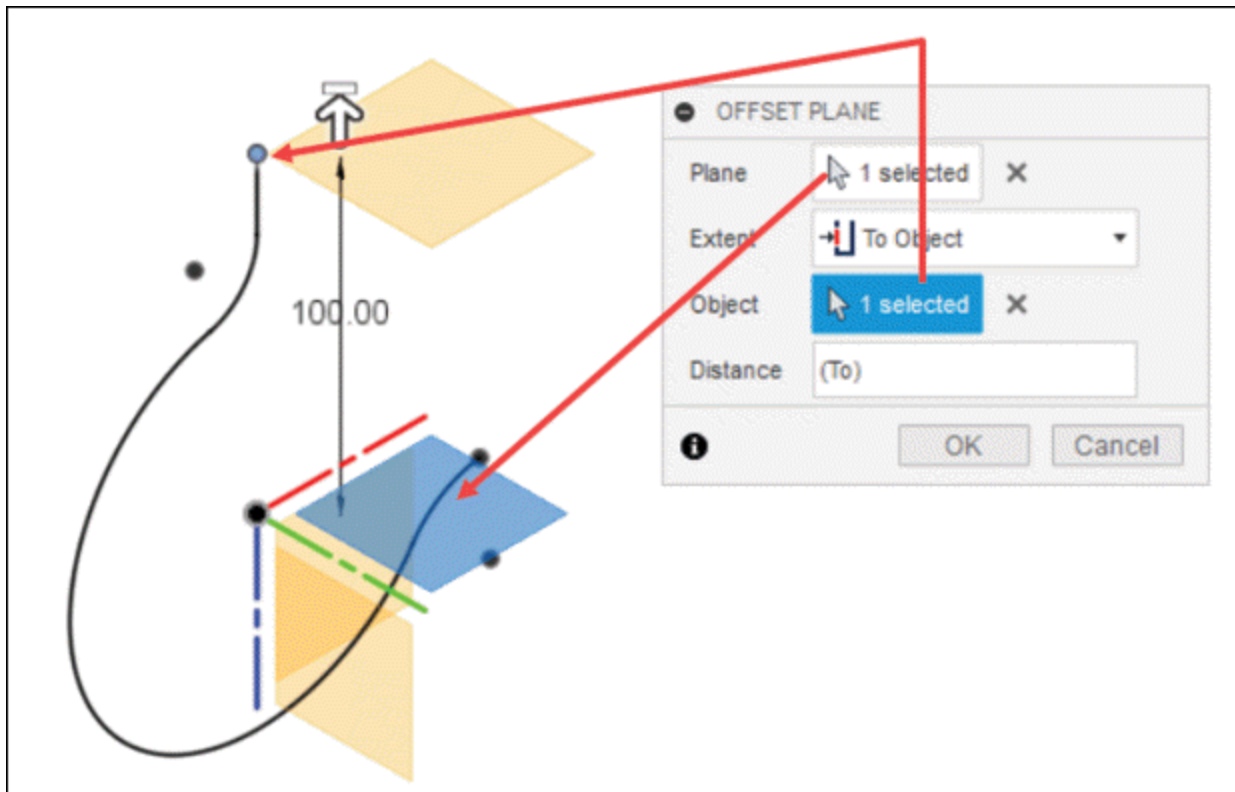


Figure-96. Creating offset plane for section sketch

- Select the new plane and click on the **Create Sketch** tool from **Ribbon**. The sketching environment will be activated.
- Create a circle of diameter **25** at the origin; refer to [Figure-97](#) and click on the **Finish Sketch** tool from the **Ribbon** to exit sketching environment.

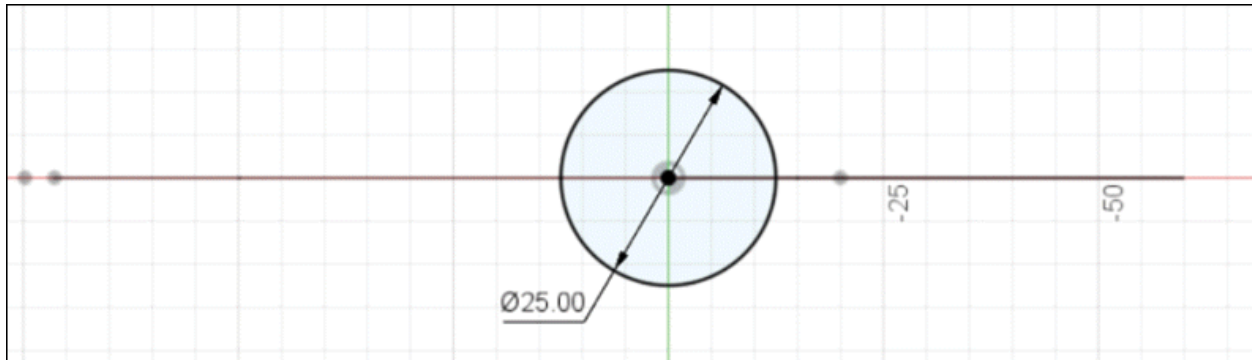


Figure-97. Section sketch circle

Creating Sweep Feature

- Switch to Isometric view using **ViewCube** and click on the **Sweep** tool from **CREATE** drop-down in the **Ribbon**. The **SWEEP** dialog box will be displayed.
- Select profile, path, and other parameters as shown in [Figure-98](#). Click on the **OK** button from the dialog box to create the feature.

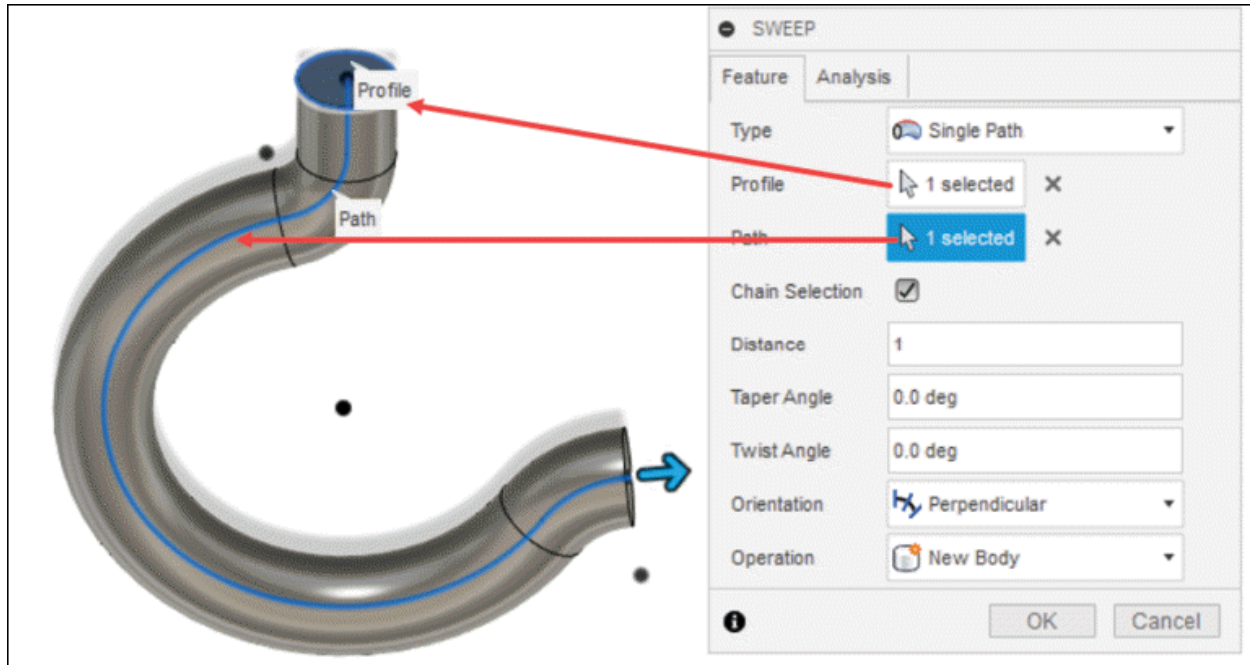


Figure-98. Creating sweep feature

Applying Fillet

- Click on the **Fillet** tool from the **MODIFY** panel in the **Ribbon**. The **FILLET** dialog box will be displayed.
- Select the bottom edge of model and specify the fillet parameters as shown in [Figure-99](#). Click on the **OK** button to create the feature. The final model will be displayed; refer to [Figure-100](#).

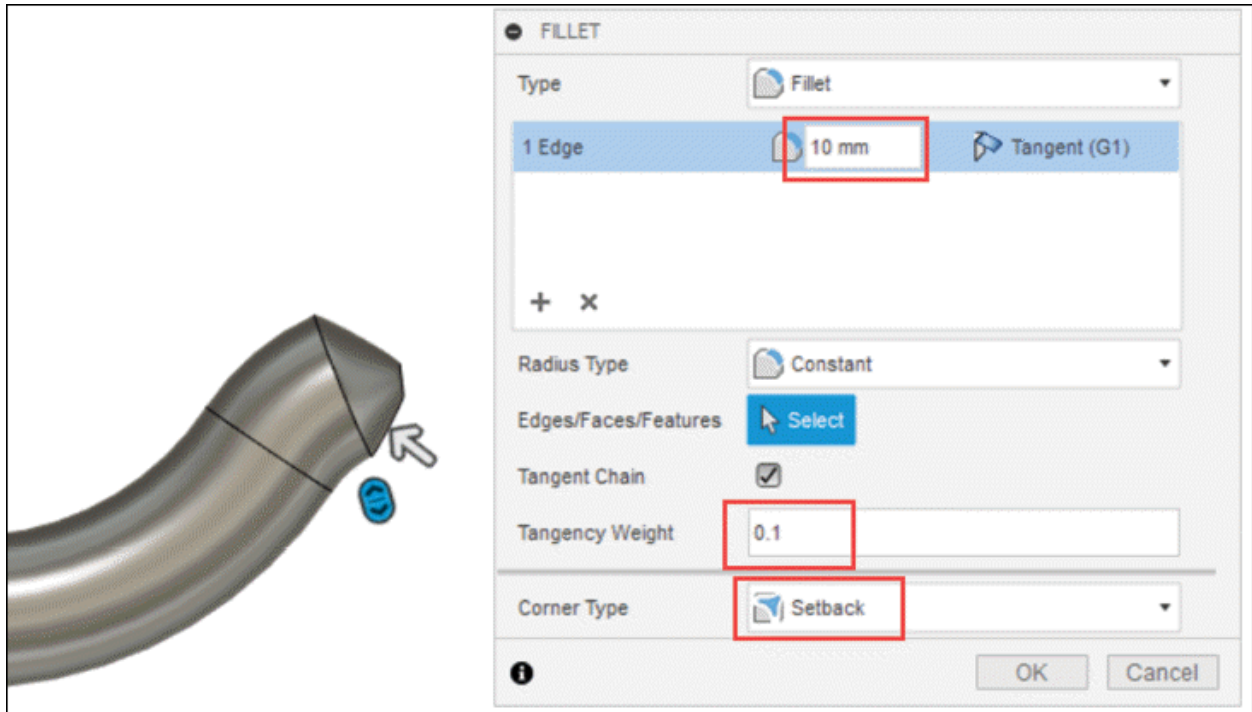


Figure-99. Applying setback corner fillet



Figure-100. Hook model

Note that we have used fillet and round interchangeably because software provides same tool for both features but the difference between two can be seen in [Figure-101](#).

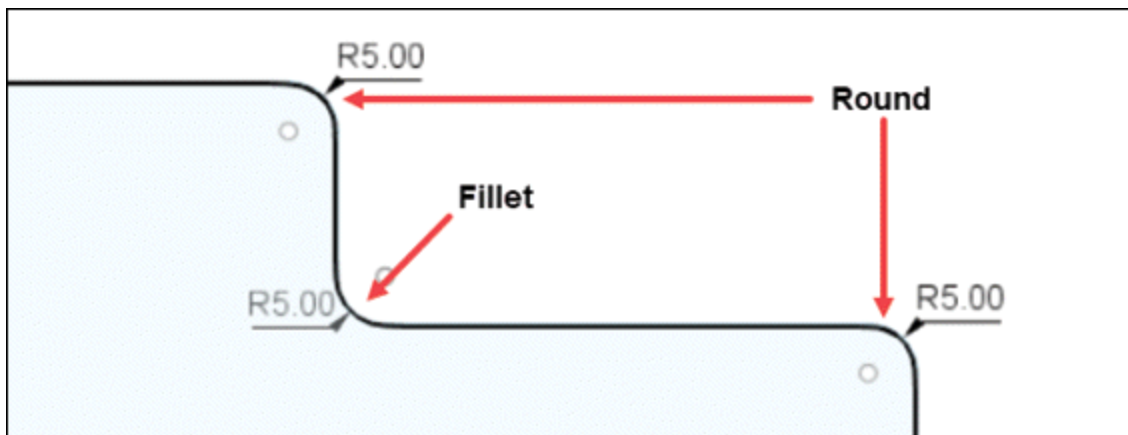


Figure-101. Round vs fillet

PRACTICAL 2

Create the model of wing nut using drawing shown in [Figure-102](#). Use the diameter value **D** as 8, 10, 12, and 14.

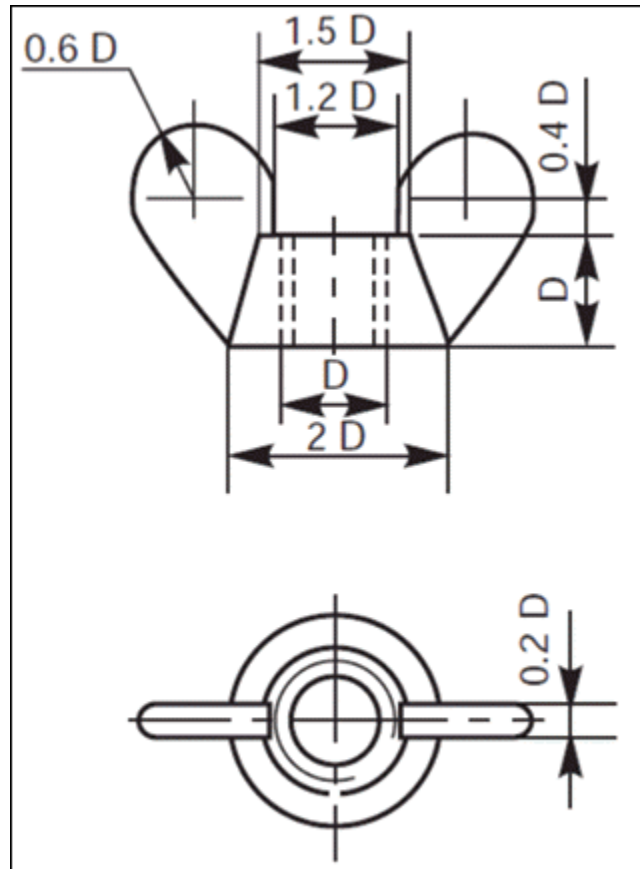


Figure-102. Wing Nut

Creating Taper Extrude Feature

- Start a new file in Autodesk Fusion 360 and start a new sketch using Front plane (XY plane).
- Create the sketch as shown in [Figure-103](#) using **Line** tool. Click on the **Finish Sketch** tool from **Ribbon** after creating sketch.

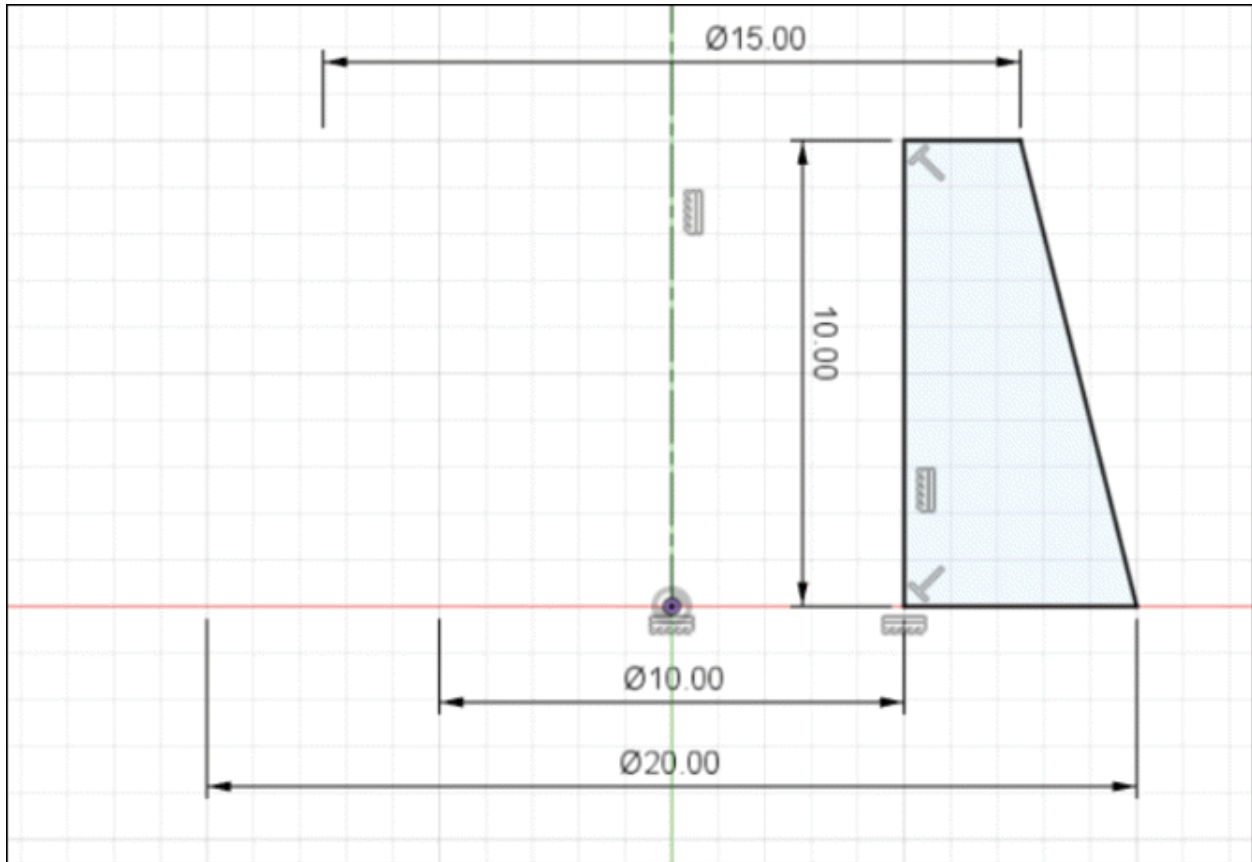


Figure-103. Sketch for wing base

- Click on the **Revolve** tool from **CREATE** panel in the **Ribbon** and specify the parameters as shown in [Figure-104](#). Click on the **OK** button from the **REVOLVE** dialog box to create the feature.

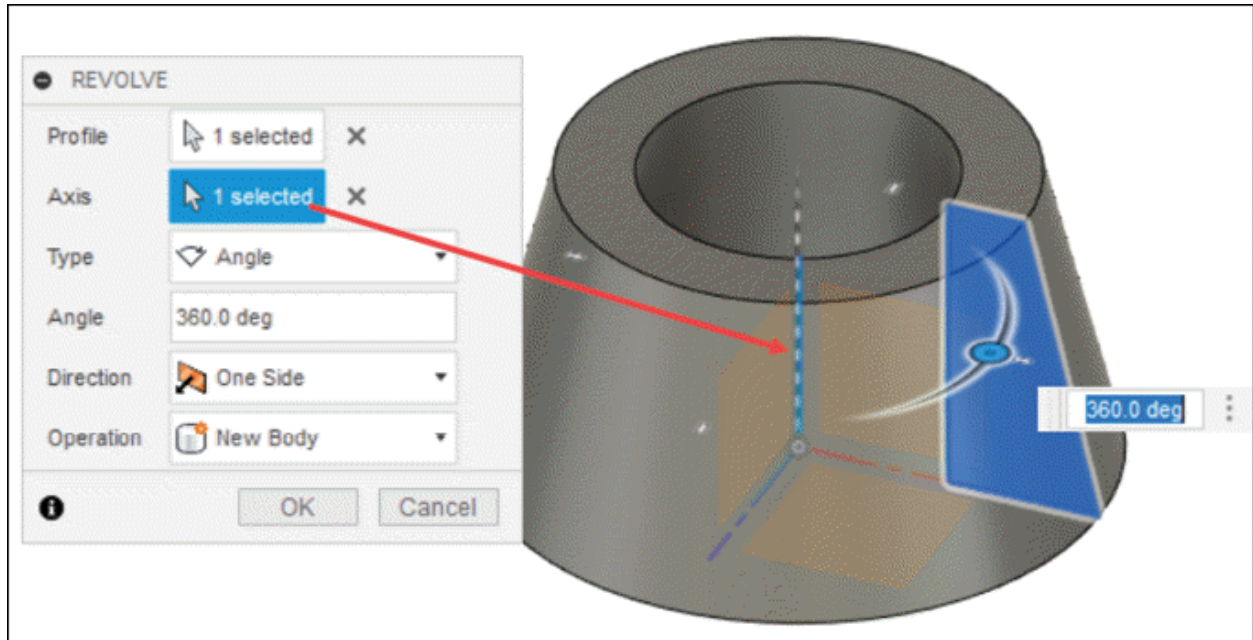


Figure-104. Creating revolve feature

Creating Wings

- Click on the **Create Sketch** tool from **CREATE** panel in the **Ribbon** and select the Front (XY) plane. The sketching environment will be activated.
- Create the sketch as shown in [Figure-105](#). Click on the **Finish Sketch** tool from the **Ribbon** to exit sketching environment.

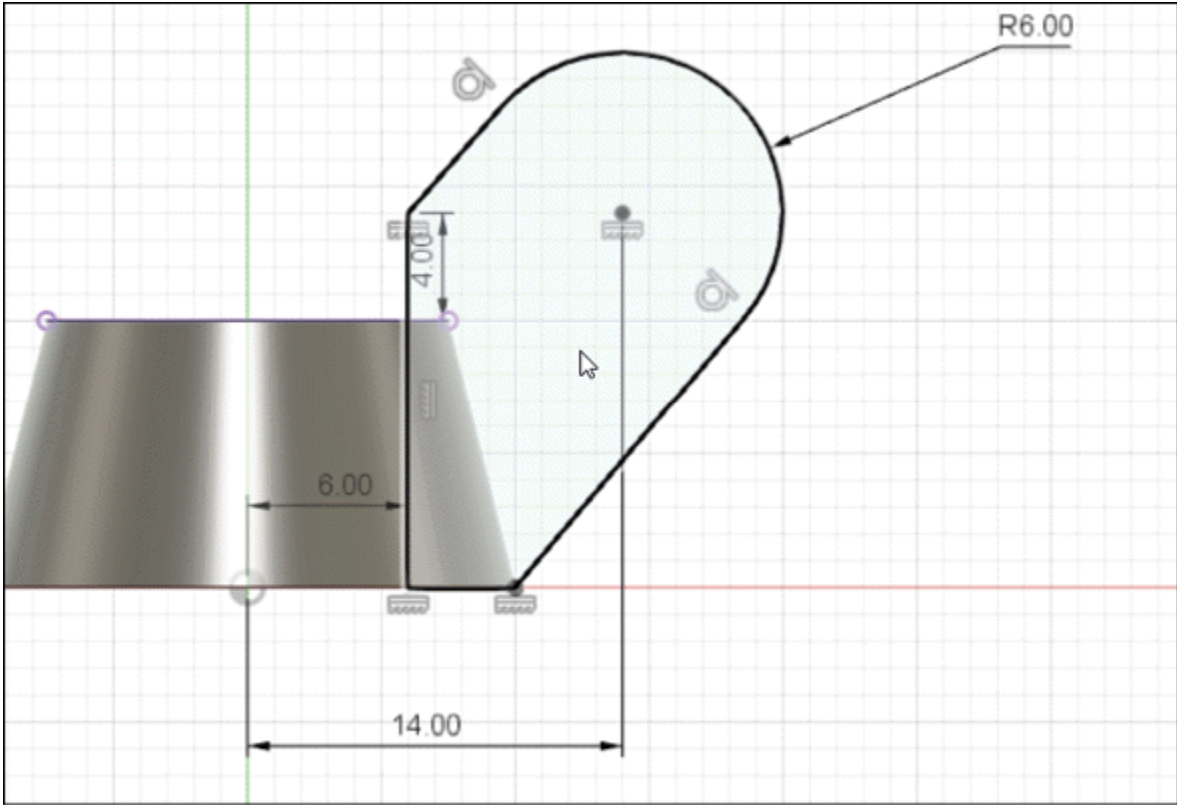


Figure-105. Sketch for nut wing

- Extrude this new sketch to both sides symmetrically by total thickness of **2**; refer to [Figure-106](#).

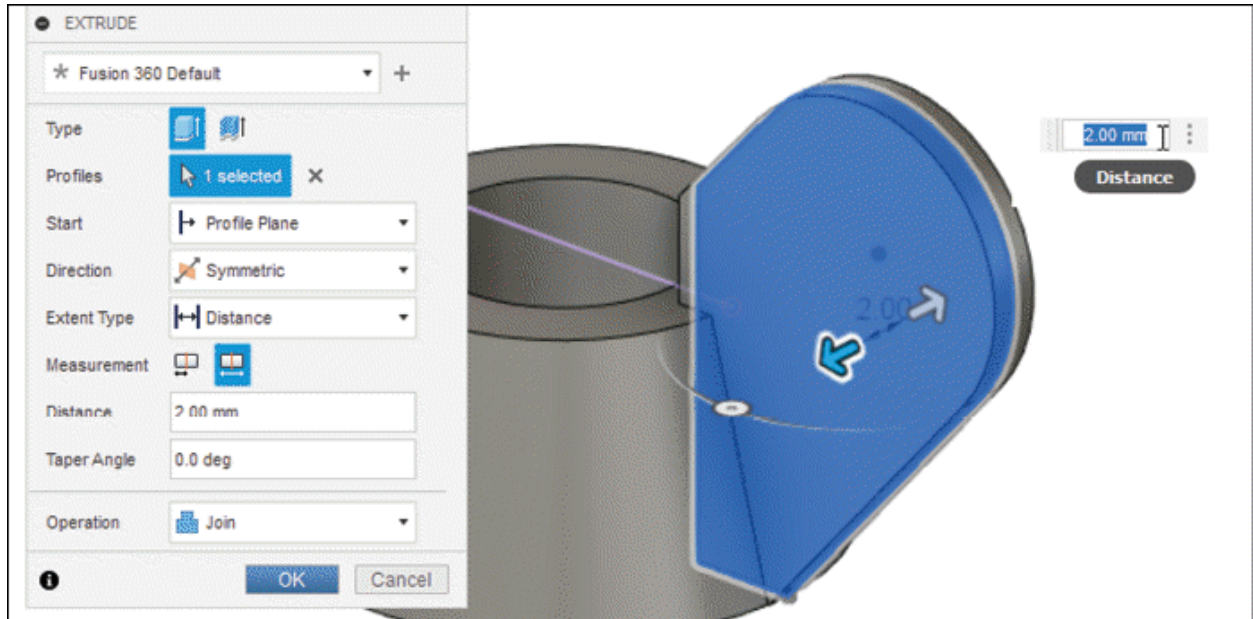


Figure-106. Creating wing for nut

- After creating wing of one side, create its mirror copy on other side of YZ plane; refer to [Figure-107](#).

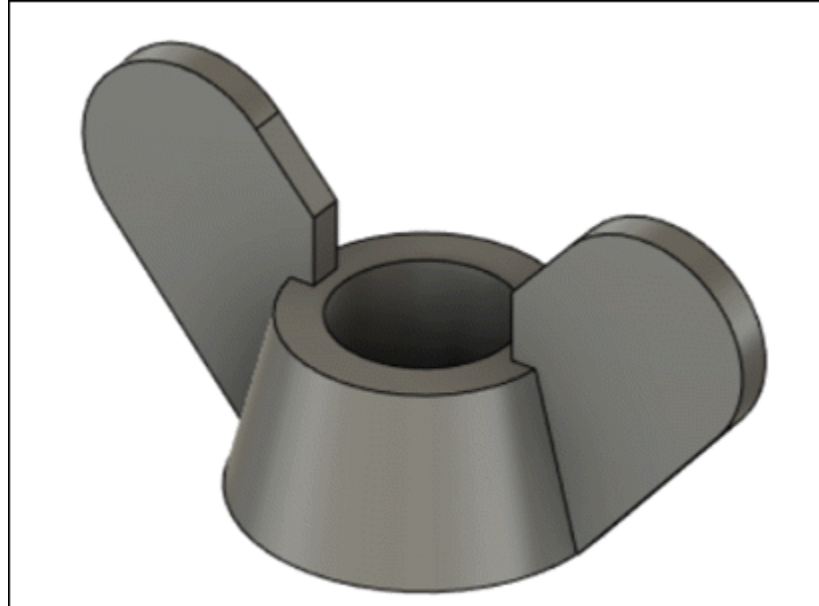


Figure-107. Wing after mirror copy

- Click on the **Thread** tool from **CREATE** drop-down in the **Ribbon** and select internal face of nut to generate threads.

Specify the parameters as shown in [Figure-108](#) and click on the **OK** button create the model.

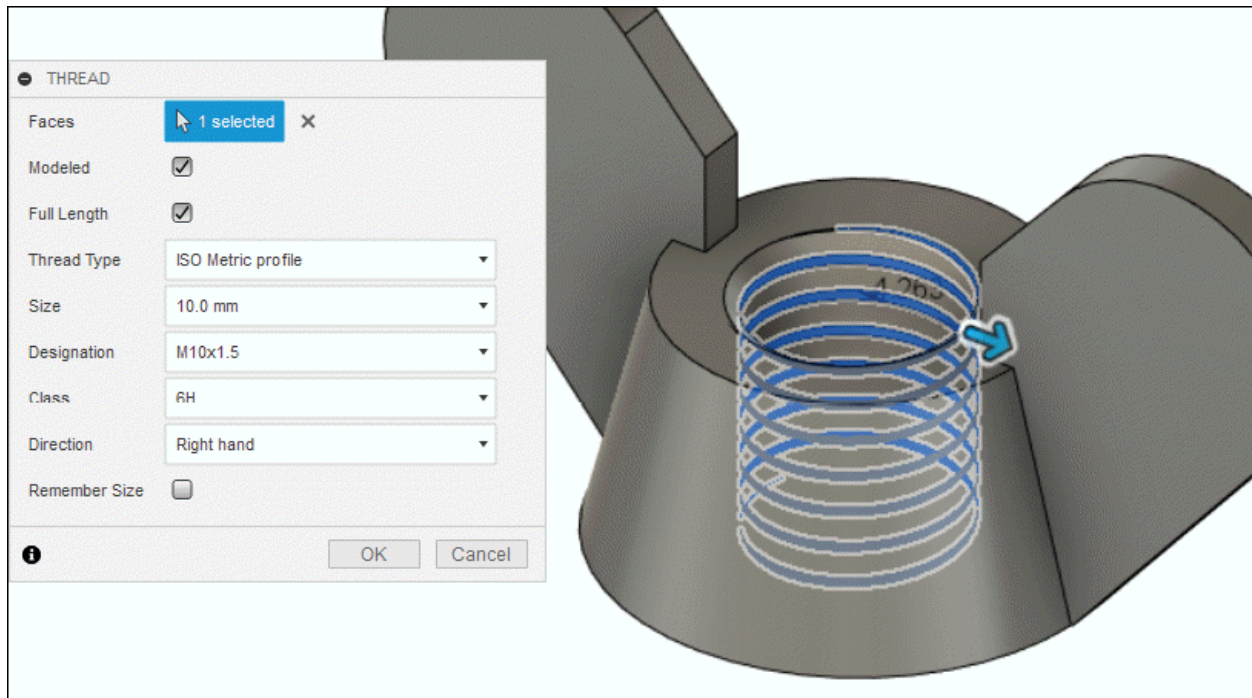


Figure-108. Creating threads

Defining Dimension Relations

Every parameter given in drawing is related to inner diameter of the nut. So, we will not apply relations to other parameters with inner diameter as per the drawing.

- Click on the **Change Parameters** tool from the **MODIFY** drop-down in the **Ribbon**. The **Parameters** dialog box will be displayed.
- Expand all the nodes in **Model Parameters** section. The dialog box should display as shown in [Figure-109](#).

Parameter	Name	Unit	Expression	Value
Favorites				
User Parameters +				
Model Parameters				
(Unsaved)				
Sketch1				
☆ ...	d2	mm	20.00 mm	20.00
☆ ...	d3	mm	10.00 mm	10.00
☆ ...	d4	mm	15.00 mm	15.00
☆ ...	d5	mm	10.00 mm	10.00
Revolve2				
☆ ...	d7	deg	360.0 deg	360.0
Sketch3				
☆ ...	d9	mm	4 mm	4.00
☆ ...	d10	mm	6 mm	6.00
☆ ...	d11	mm	6 mm	6.00
☆ ...	d12	mm	14 mm	14.00
Extrude1				
☆ ...	d13	mm	2.00 mm	2.00
☆ ...	d14	deg	0.0 deg	0.0

Figure-109. Parameters dialog box for wing

- Note that in Sketch1, **d3** is used to specify inner diameter of wing nut which drives all the other equations. If you are wondering how we know that, then this is hit and trial method until Autodesk updates the software to display dimension variables as well in sketch along with values. To find the diameter, we tried changing **d5** and that caused increase in length of nut so there is only one **10 mm** value dimension left which is diameter of nut. So, change the name of **d3** variable to **D** and set the relations as shown in [Figure-110](#).

Parameter	Name	Unit	Expression	Value
Favorites				
User Parameters +				
Model Parameters				
(Unsaved)				
▼	Sketch1			
☆ ...	d2	mm	2 * D	20.00
☆ ...	D	mm	10 mm	10.00
☆ ...	d4	mm	1.5 * D	15.00
☆ ...	d5	mm	D	10.00
▼	Revolve2			
☆ ...	d7	deg	360.0 deg	360.0
▼	Sketch3			
☆ ...	d10	mm	0.6 * D	6.00
☆ ...	d11	mm	d10	6.00
☆ ...	d12	mm	1.4 * D	14.00
☆ ...	d21	mm	0.4 * D	4.00
▼	Extrude1			
☆ ...	d13	mm	0.2 * D	2.00
☆ ...	d14	deg	0.0 deg	0.0

Figure-110. Table after specifying relations

- Now, change the value of **D** in table to get desired size of nut. All the other dimensions will be modified as per relation and model will change accordingly in graphics area. Check what happens when you modify the diameter value in sketch and then exit the Sketching environment!!

PRACTICE 1

Create the model of connecting rod as shown in [Figure-111](#). The dimensions are given in [Figure-112](#).

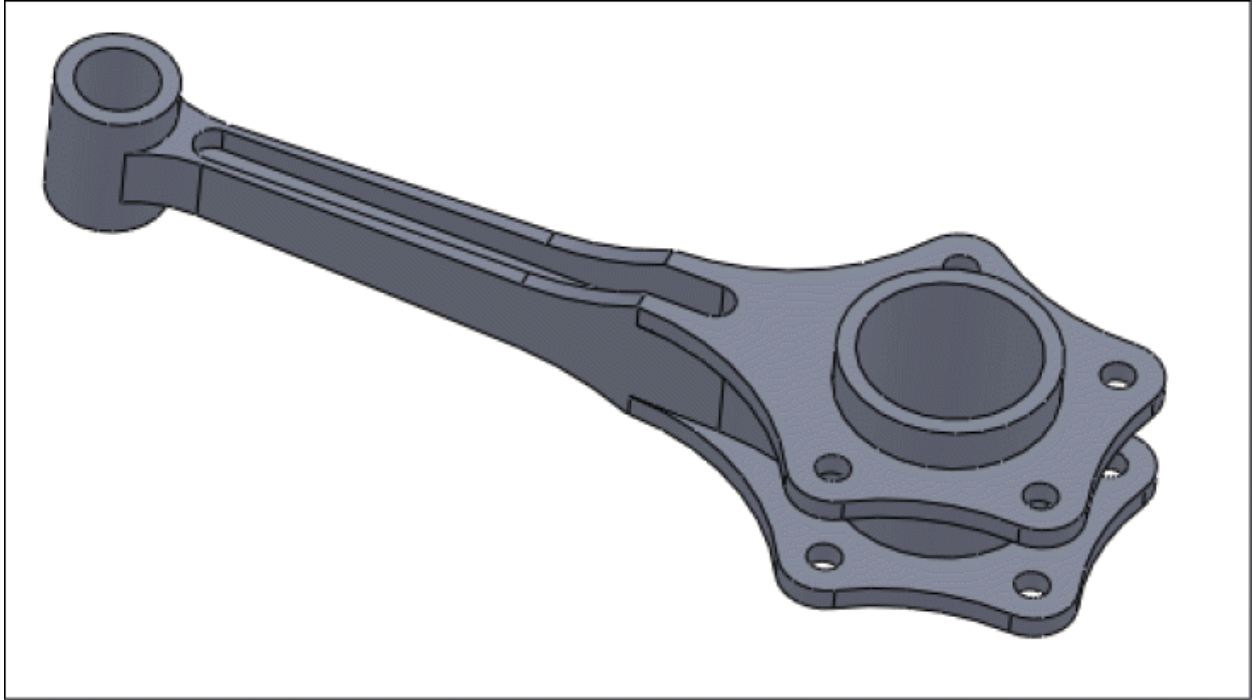


Figure-111. Practice 1

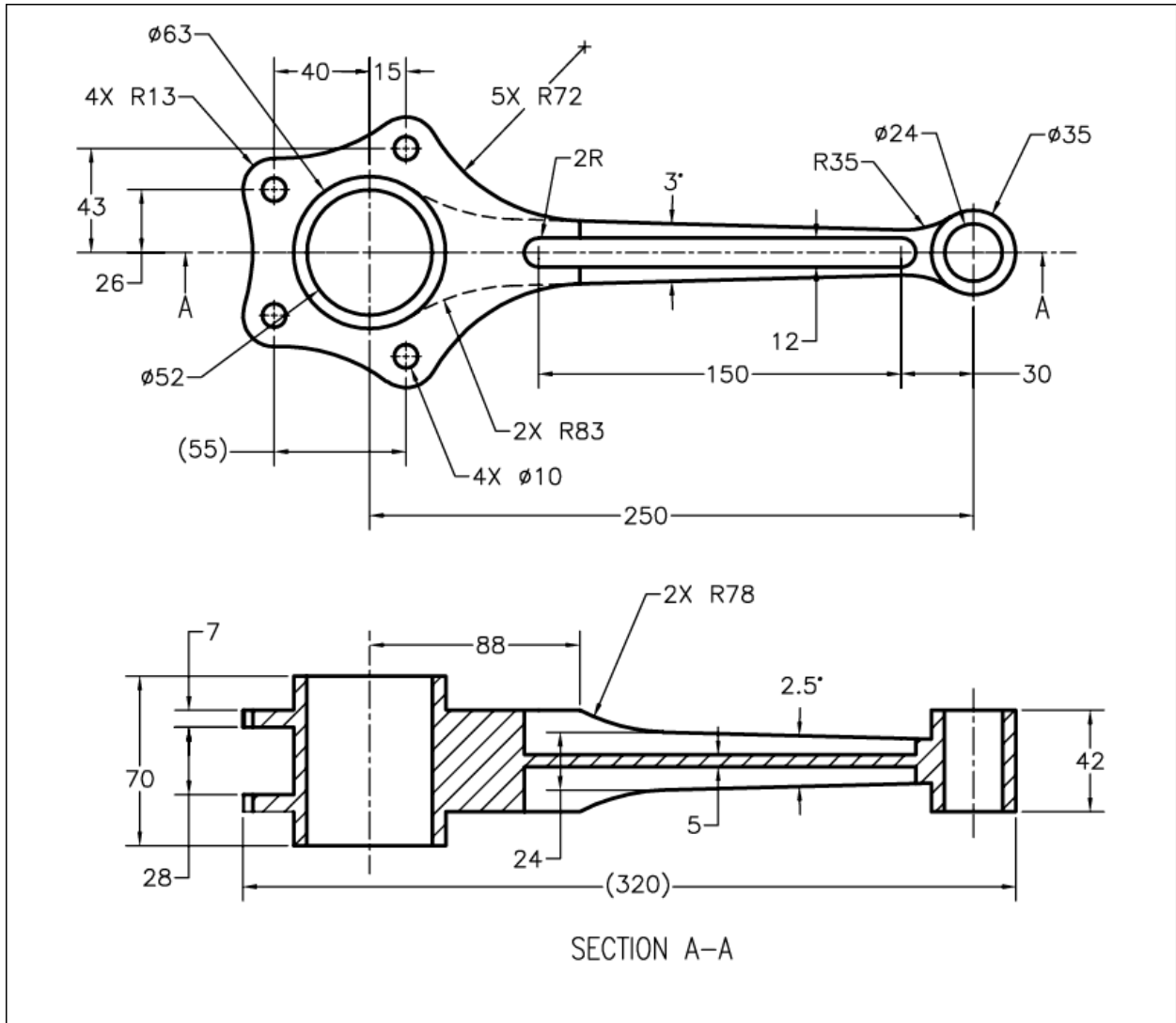


Figure-112. Dimensions of the Practice 1 model

PRACTICE 2

Create a ring nut with value of **D** as **5,6,8**, and **10** using equation and design table. Dimensions are given in [Figure-113](#).

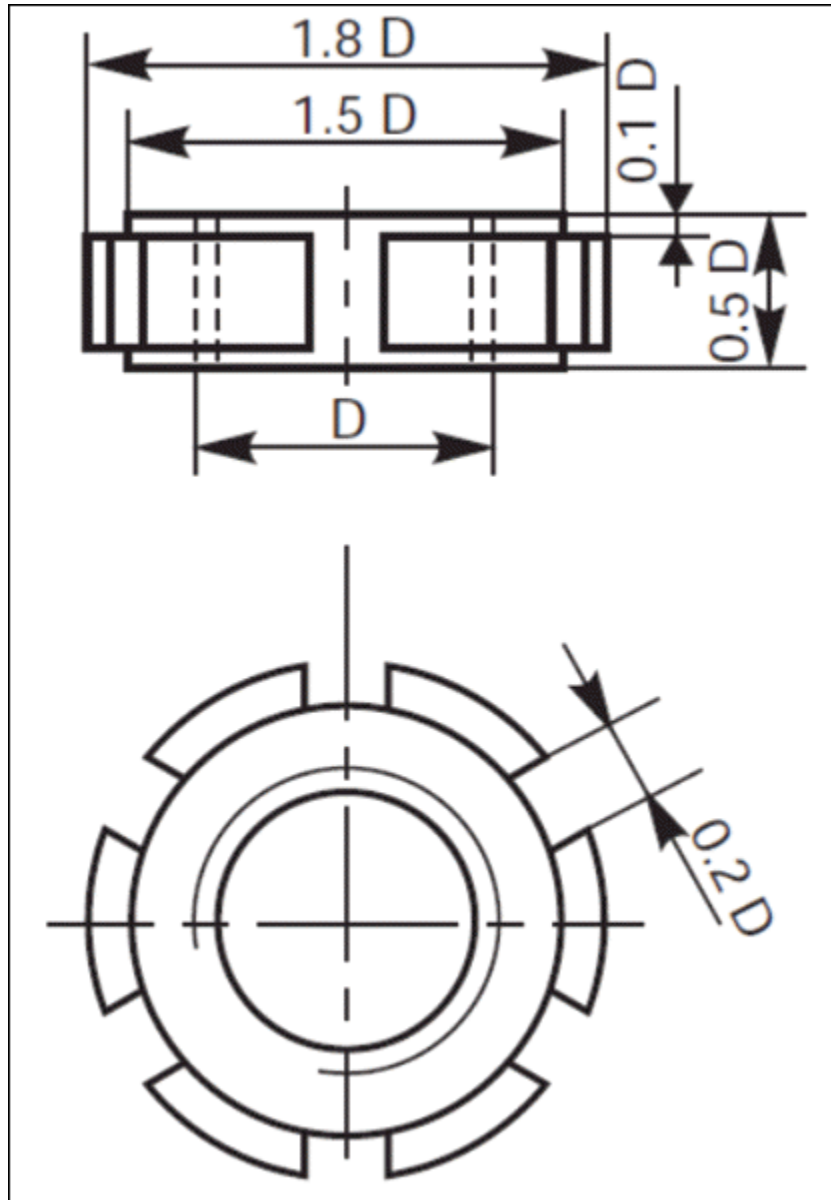


Figure-113. Ring Nut

SELF-ASSESSMENT

Q1. Which of the following tools can be used to dynamically use offset face or fillet function depending on the selection?

- a. Fillet
- b. Offset Face
- c. Press Pull
- d. Combine

Q2. Which of the following tool is used to apply round at the edges?

- a. Fillet b. Chamfer
- c. Draft d. Shell

Q3. Which of the following tool is used by casting mold designer to allow easy extraction of part from the die?

- a. Fillet b. Draft
- c. Shell d. Chamfer

Q4. Which of the following tool is used to create a body using intersection of two bodies?

- a. Combine b. Shell
- c. Split Body d. Split Face

Q5. The **Move/Copy** tool in Autodesk Fusion **Design** workspace can also be used to rotate the objects. (T/F)

Q6. What is the difference between applications of **Physical Material** tool and **Appearance** tool in **Modify** drop-down of **Design** workspace?

Q7. In Autodesk Fusion 360, you can create a fillet with changing radius at specified points. (T/F)

Q8. Which of the following tools is used to remove material from a solid body and convert it into a thin body of specified thickness?

- a. Combine b. Shell
- c. Split Body d. Split Face

Chapter 7

Assembly Design

Topics Covered

The major topics covered in this chapter are:

- ***Introduction***
- ***New Component Tool***
- ***Joint Tool***
- ***As-Built Joint Tool***
- ***Joint Origin Tool***
- ***Rigid Group Tool***
- ***Drive Joints Tool***
- ***Motion Link Tool***
- ***Enable Contact Sets Tool***
- ***Motion Study Tool***

INTRODUCTION

Most of the things you find around you in real-world are assembly of various components; the computer you are working on is an assembly, the automotive you may be driving or sitting in for travelling is an assembly, there are lots of examples. Till this point, we have learned to create

solid parts. But most of the time, we need to assemble the parts to get some use of them. In this chapter, we will work on the assembly design of Autodesk Fusion 360 in which we will be assembling two or more components using some assembly constraints.

NEW COMPONENT

The **New Component** tool is used to create an empty component or components from the selected bodies. Note that we need to convert all parts to components before performing their assembly. The procedure to use this tool is discussed next.

- Click on the **New Component** tool of **ASSEMBLE** drop-down from **Toolbar**; refer to [Figure-1](#). The **NEW COMPONENT** dialog box will be displayed; refer to [Figure-2](#).

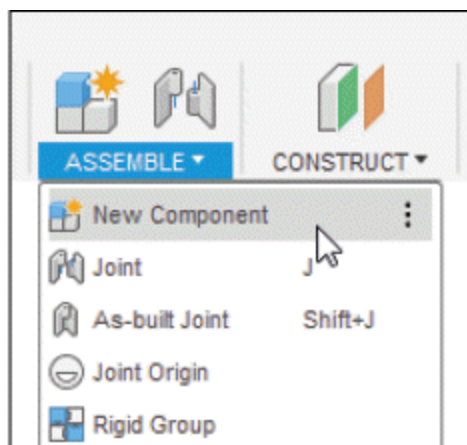


Figure-1. New Component tool

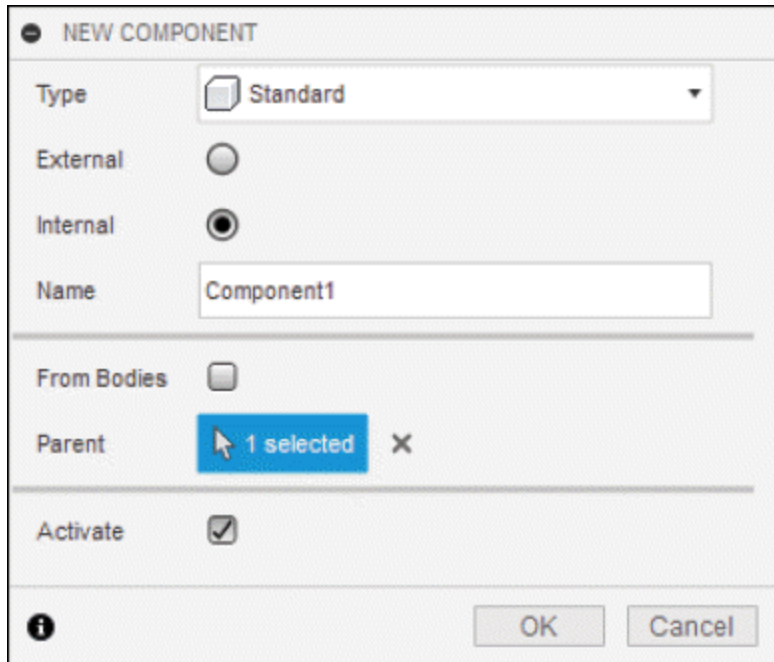


Figure-2. NEW COMPONENT dialog box

- Select **Standard** option from **Type** drop-down in the dialog box, if you want to create a standard component in the assembly structure.
- Select **Sheet Metal** option from **Type** drop-down, if you want to create a sheet metal component in the assembly structure.

Standard Component

In this topic, we will discuss the procedure of creating standard component.

- Select **Standard** option from **Type** drop-down of **NEW COMPONENT** dialog box if not selected by default.
- Select **External** radio button from dialog box to create an external component and referenced into the assembly in

the current design. The options related to **External** radio button will be displayed in the **NEW COMPONENT** dialog box; refer to [Figure-3](#).

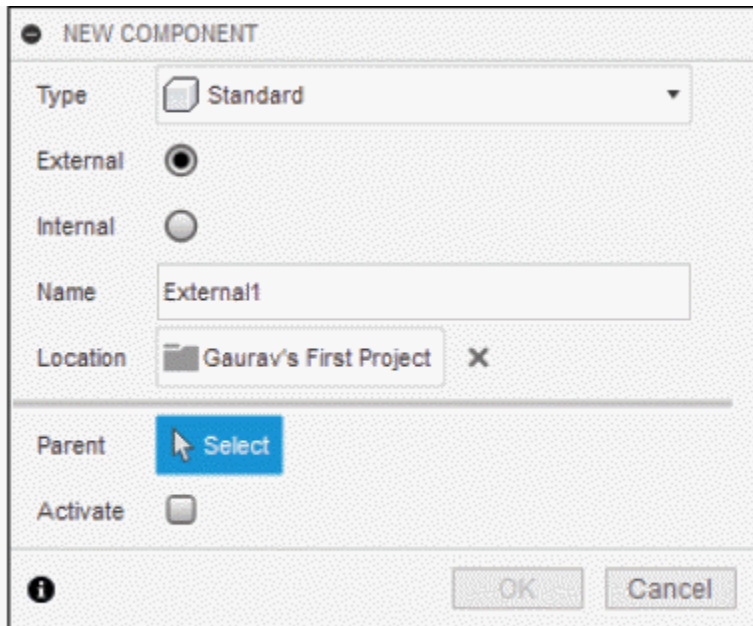


Figure-3. Options on selecting External radio button

- Click in the **Name** edit box of **NEW COMPONENT** dialog box and enter the name of component.
- Specify desired location where you want to save the new design that will contain the new component by clicking on the button from **Location** section of the dialog box.
- Click on **Select** button of **Parent** option and select the parent body under which new component will be created.
- Select the **Activate** check box from **NEW COMPONENT** dialog box to activate the new component. Note that all the modeling operations performed after activating the component will become child feature of that component.

- After specifying the parameters for standard component, click on the **OK** button from **NEW COMPONENT** dialog box. A standard component will be created; refer to [Figure-4](#).

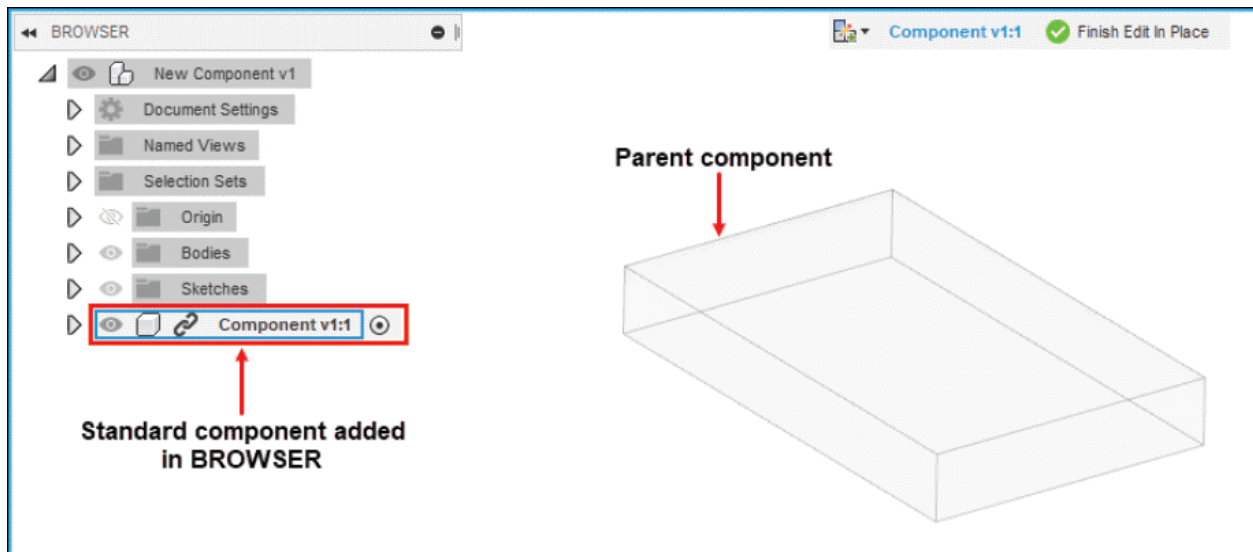


Figure-4. Standard component created

- If you want to create an internal component directly in the current design then select **Internal** radio button from the dialog box. The options related to **Internal** radio button will be displayed in the **NEW COMPONENT** dialog box; refer to [Figure-5](#).

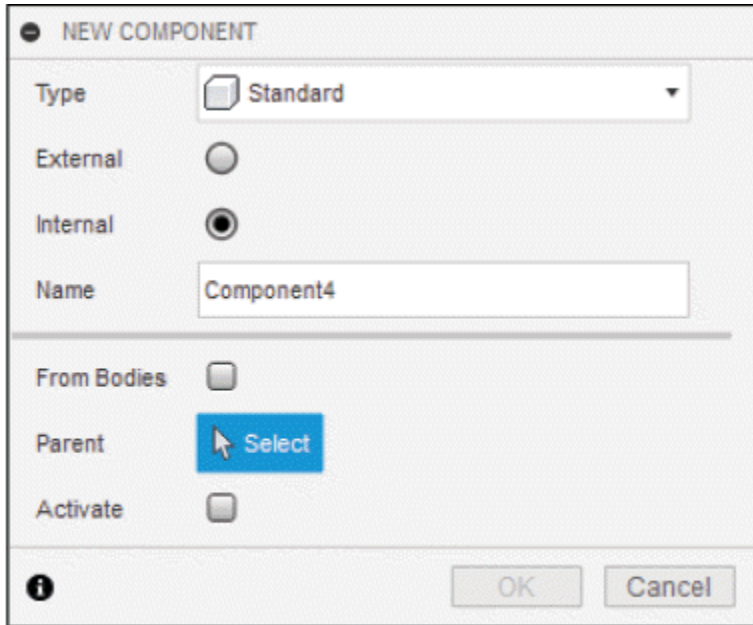


Figure-5. Options on selecting Internal radio button

- Select **From Bodies** check box from **NEW COMPONENT** dialog box to convert existing bodies into individual components. The updated **NEW COMPONENT** dialog box will be displayed; refer to [Figure-6](#).

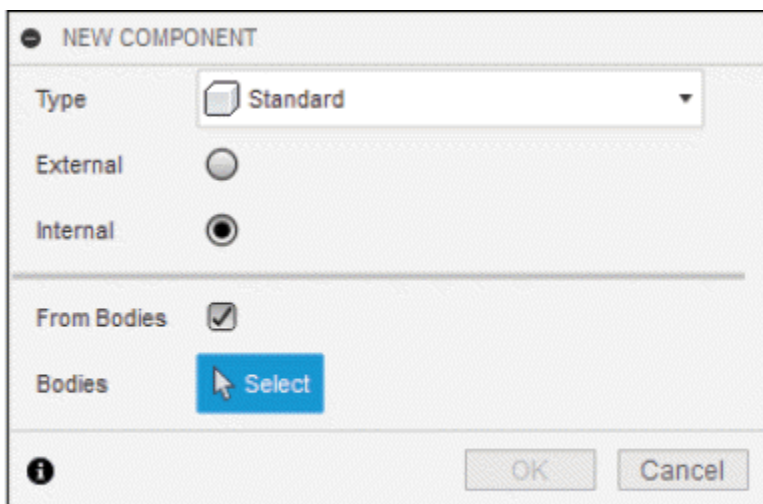


Figure-6. Updated NEW COMPONENT dialog box

- The **Select** button of **Bodies** section is active by default.

Click on the parts/bodies from model to create a new component; refer to [Figure-7](#). You can select multiple bodies by holding the **CTRL** key.

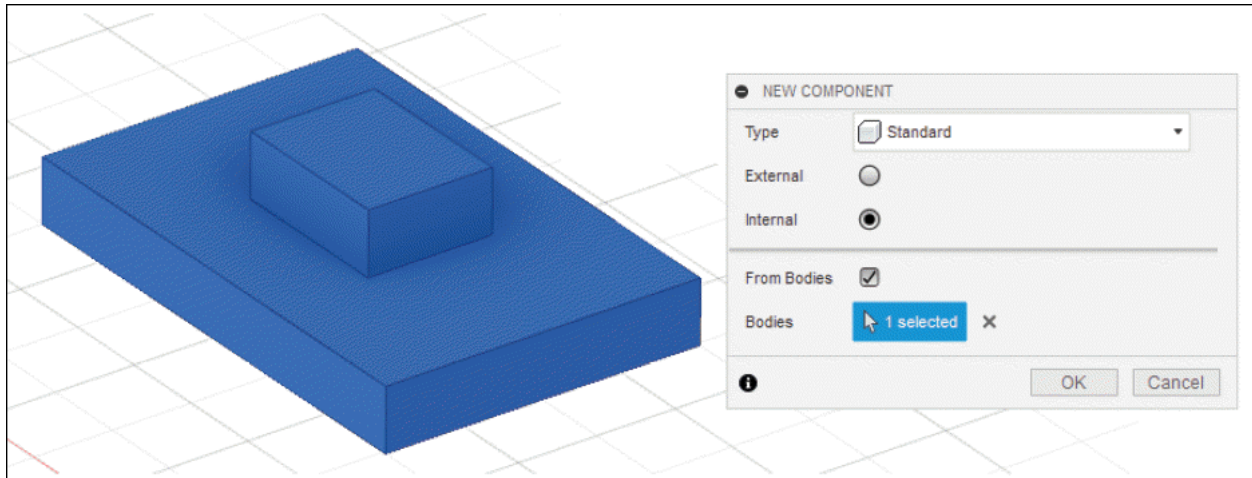


Figure-7. Bodies selected for new component

- After selecting the body, click on the **OK** button from **NEW COMPONENT** dialog box. The new component will be created and action will be displayed in **BROWSER**.

Sheet Metal Component

In this topic, we will discuss the procedure of creating sheet metal component.

- Select the **Sheet Metal** option from **Type** drop-down of **NEW COMPONENT** dialog box. The **NEW COMPONENT** dialog box with sheet metal options will be displayed; refer to [Figure-8](#).

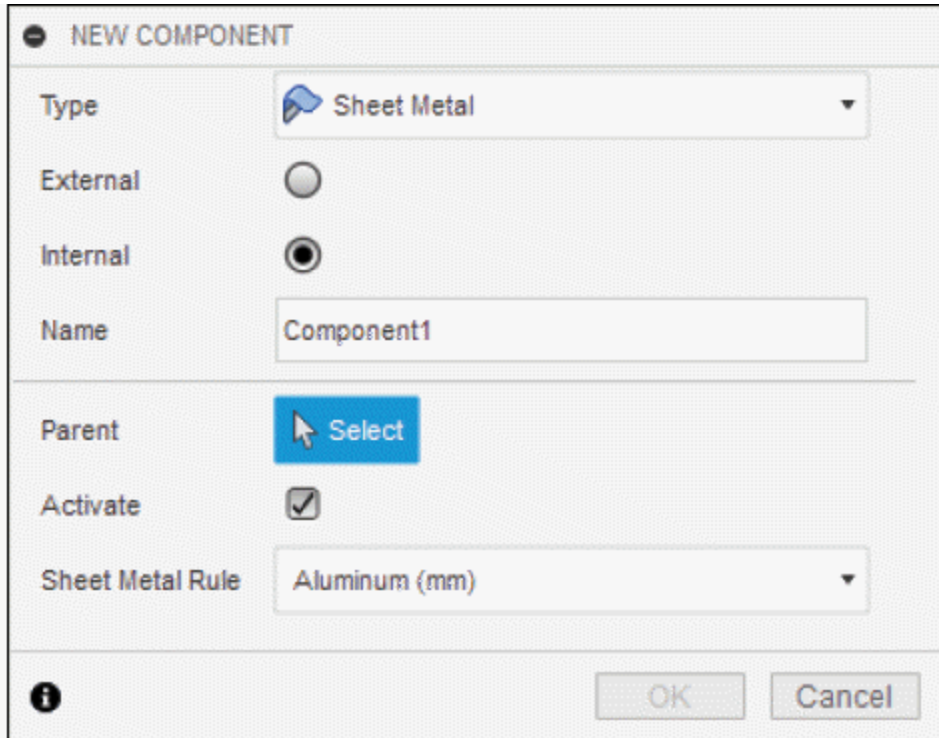


Figure-8. NEW COMPONENT dialog box with sheet metal options

- Select desired sheet metal rule already in use in this design or from the **Sheet Metal Rule** drop-down in the dialog box.
- The other parameters in the **NEW COMPONENT** dialog box have been discussed earlier.
- After specifying parameters for sheet metal component, click on the **OK** button from **NEW COMPONENT** dialog box. A sheet metal component will be created; refer to [Figure-9](#).

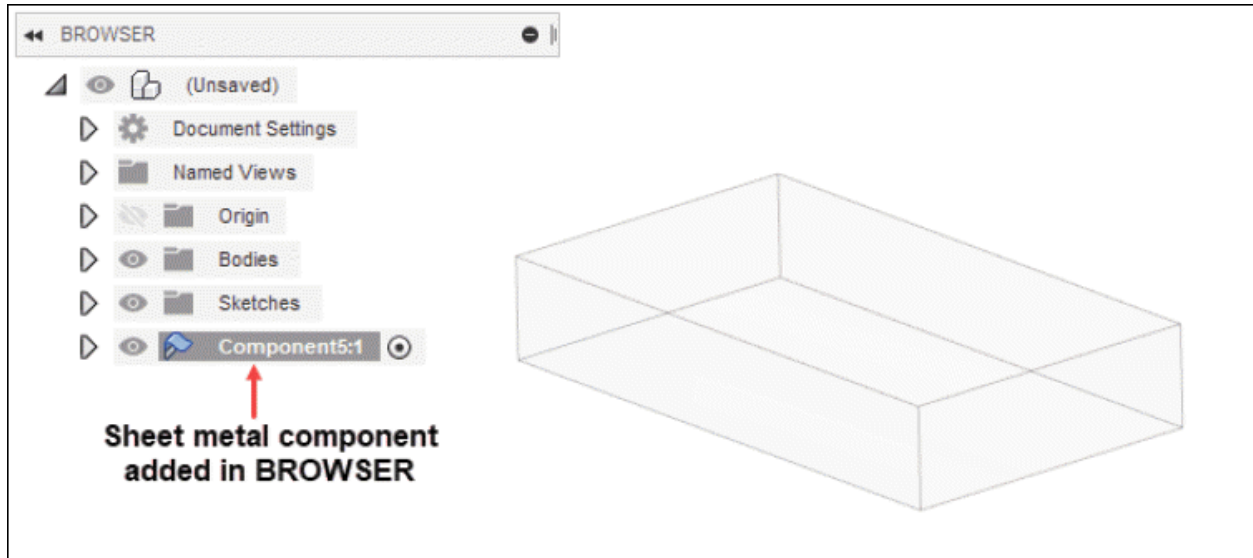


Figure-9. Sheet metal component created

JOINT

The **Joint** tool is used to define joints between components to allow movement of components relative to each other. To apply **Joint** tool, you need to create the component of bodies. The procedure to use this tool is discussed next.

- Click on the **Joint** tool of **ASSEMBLE** drop-down from **Toolbar**; refer to [Figure-10](#). The **JOINT** dialog box will be displayed; refer to [Figure-11](#).

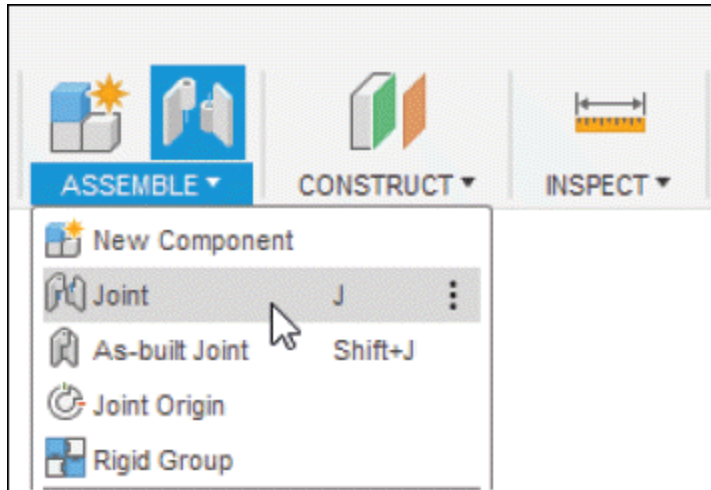


Figure-10. Joint tool

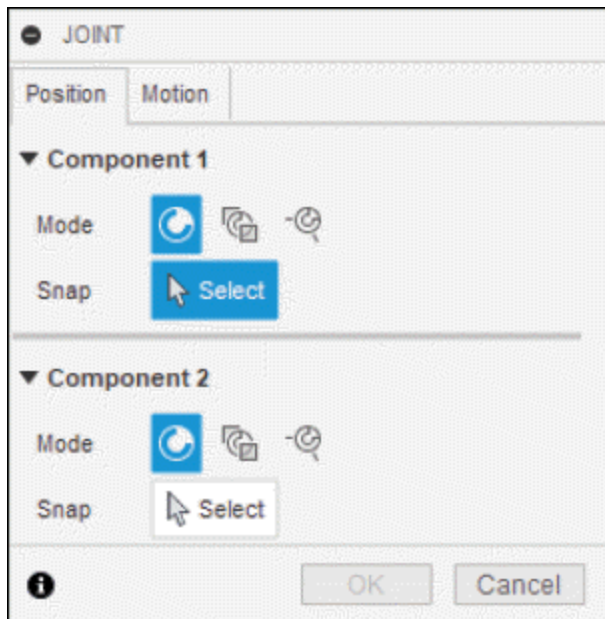


Figure-11. JOINT dialog box

- Select desired option from **Mode** section of **Component 1** area in the **Position** tab of the dialog box. Select **Simple** option from **Mode** section to select a **Snap** point to place the joint origin. Select **Between Two Faces** option from **Mode** section to select **Plane 1** and **Plane 2** to center the joint origin between them then select a **Snap** point. Select

Two Edge Intersection option from **Mode** section to select **Edge 1** and a non-parallel **Edge 2** to locate the joint origin at the extended intersection.

- **Select** button of **Snap** section from **Component 1** area is active by default. Click on the first component at desired location to assemble; refer to [Figure-12](#). You will be asked to select a location on second component.

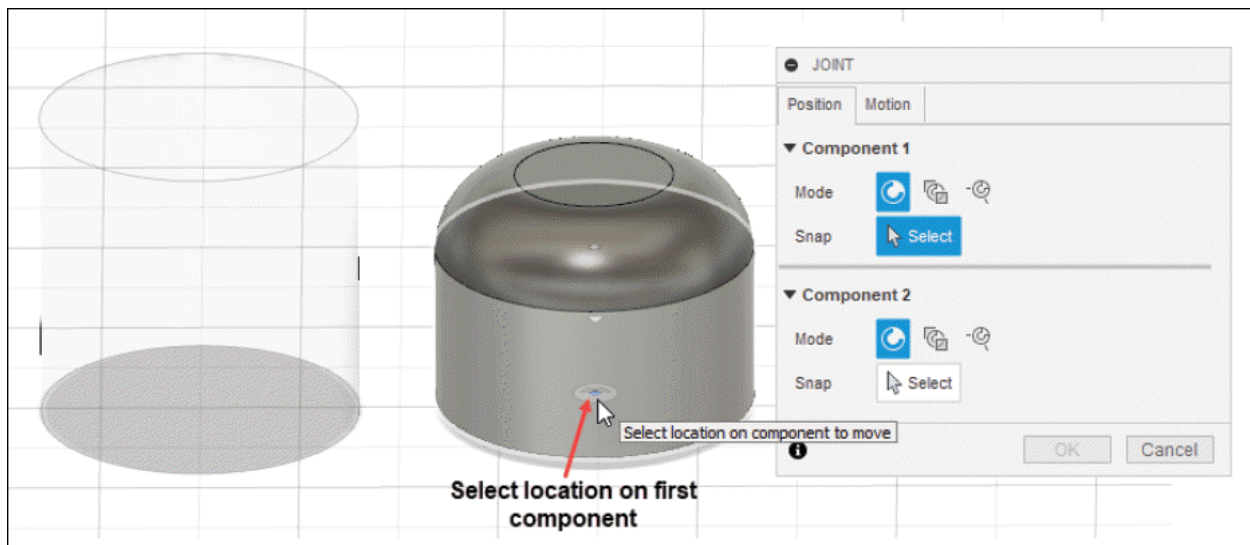


Figure-12. Selection of component 1 location

- Select desired option from **Mode** section of **Component 2** area in the **Position** tab of the dialog box. The options of **Mode** section have been discussed earlier.
- Click on the second component on mating location to select; refer to [Figure-13](#).

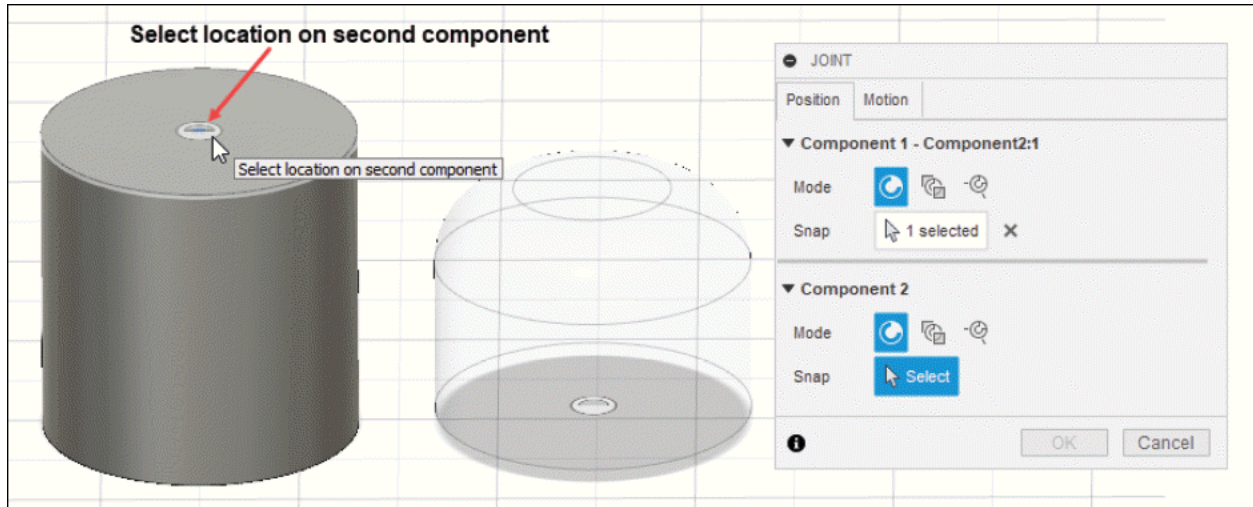


Figure-13. Selection of component 2 location

- On selecting the second component, the selected two component will be joined to each other at the locations selected on them and the updated **JOINT** dialog box will be displayed; refer to [Figure-14](#).

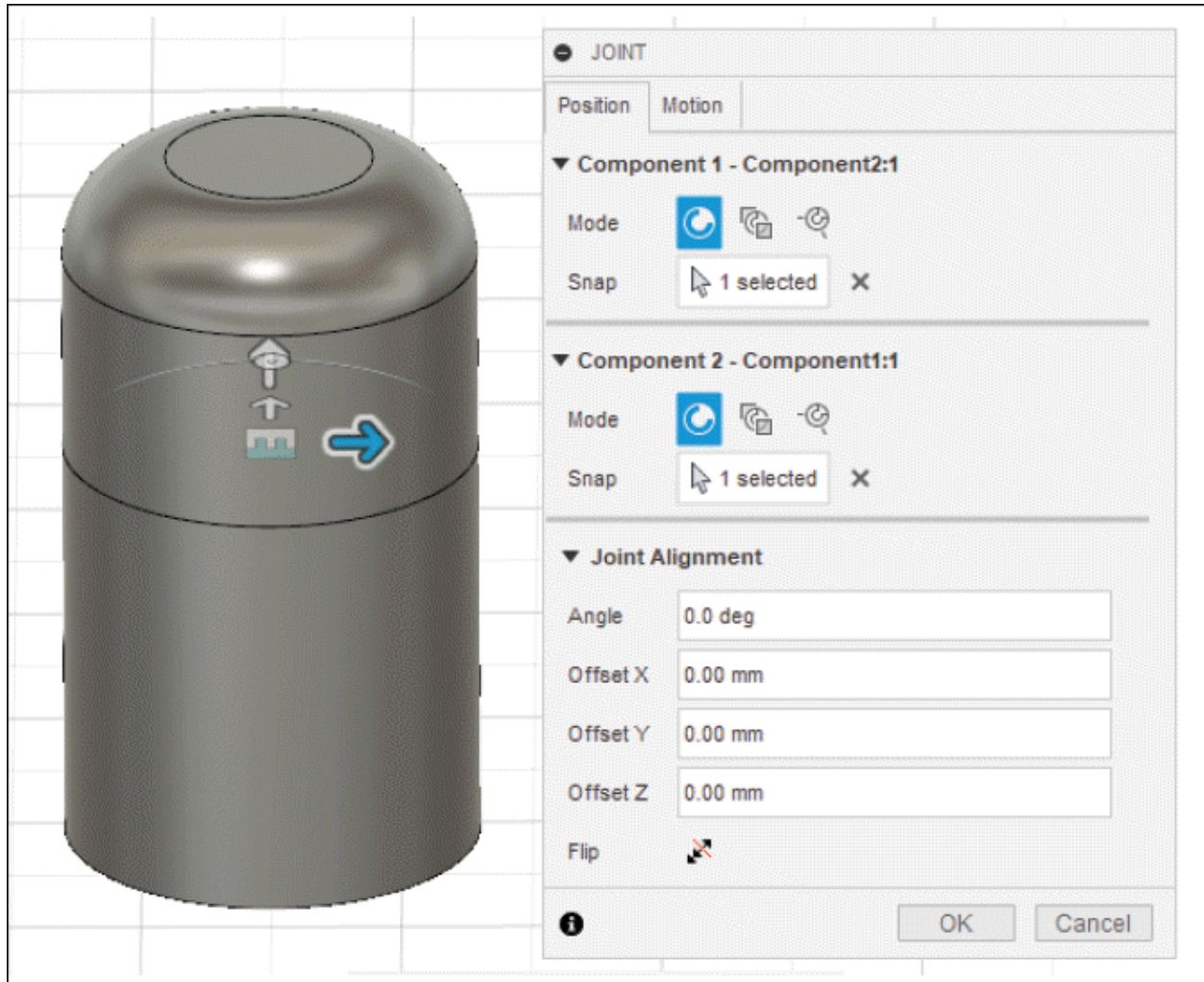


Figure-14. Components joined with updated JOINT dialog box

- Click in the **Angle** edit box from **Joint Alignment** area in **Position** tab of the dialog box and specify the angle of rotation of component.
- If you want to specify offset distance between components then click in the **Offset X** , **Offset Y**, and **Offset Z** edit boxes as required and enter the value. You can also set the offset distance by moving the drag handles.
- Click on the **Flip** button from **Joint Alignment** area of the

dialog box, if you want to flip the direction of second component.

Motion

There are seven motion types in **Motion** tab of **JOINT** dialog box which we can apply to the component as required; refer to [Figure-15](#). These motion types are described next.

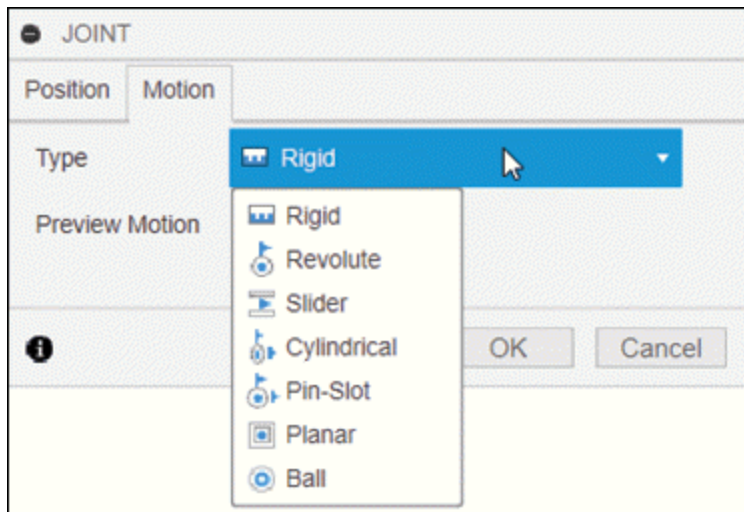


Figure-15. Types of motion in JOINT dialog box

Rigid Motion



- Select the **Rigid** option from **Type** drop-down in the **Motion** tab of the **JOINT** dialog box, if you want the joint of two component to be rigid to each other and there should be no motion between the two joined component.

Revolute Motion



- Select the **Revolute** option from **Type** drop-down in the **Motion** tab of the **JOINT** dialog box, if you want to revolve the component about other at the joint location. On

selecting the **Revolute** option, a revolve drag handle will be displayed. You can also revolve the component by moving the drag handle; refer to [Figure-16](#).

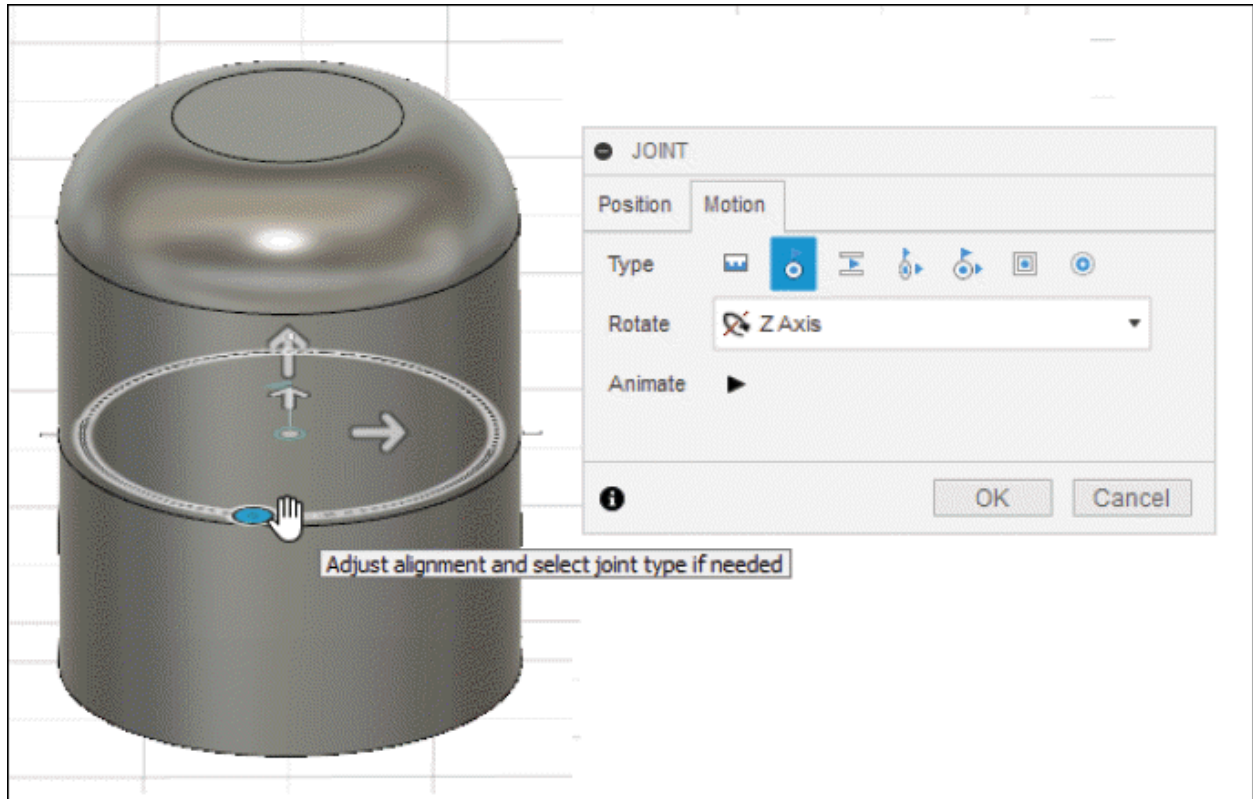


Figure-16. Revolute motion

Slider Motion



- Click on the **Slider** option from **Type** drop-down in the **Motion** tab of the **JOINT** dialog box, if you want to assemble the component in such a way that selected component can slide over the other; refer to [Figure-17](#).

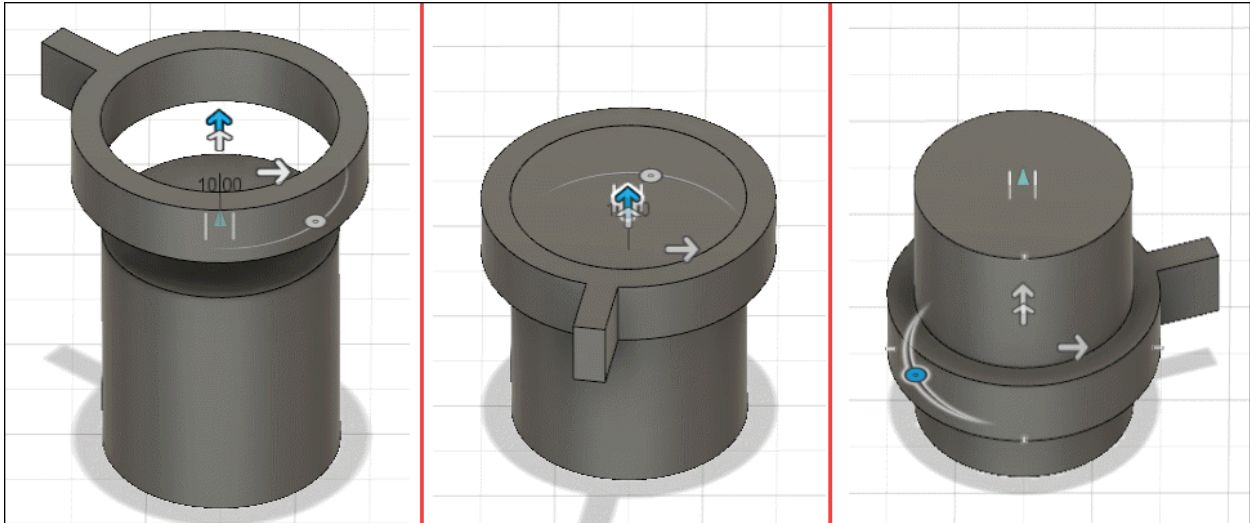


Figure-17. Slider motion

Cylindrical Motion



- Click on the **Cylindrical** option from **Type** drop-down in the **Motion** tab of the **JOINT** dialog box, if you want to assemble the component in such a way that it is free to move along the selected axis and free to rotate about the same selected axis; refer to [Figure-18](#).

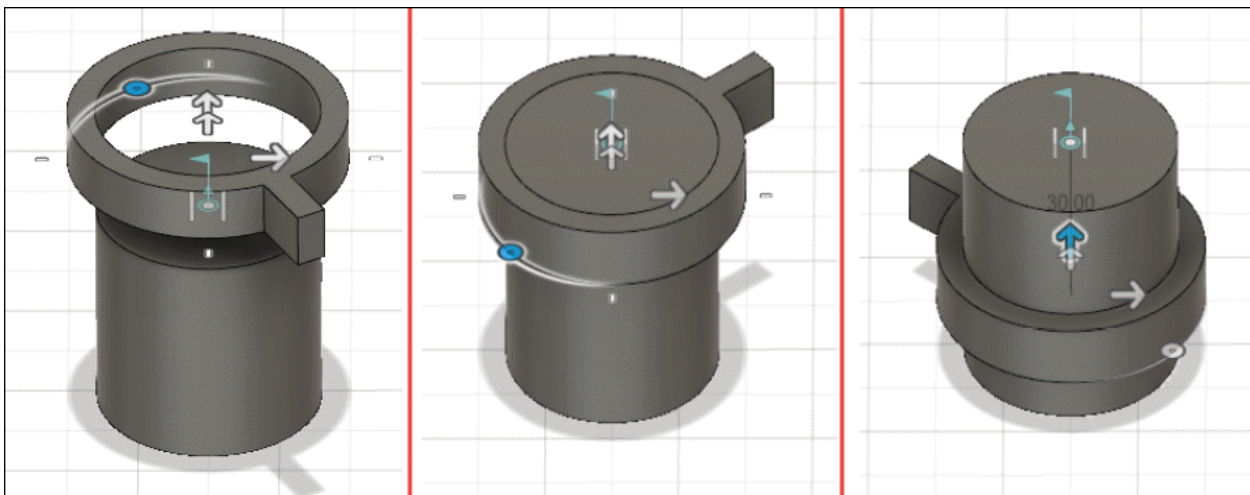


Figure-18. Cylindrical motion

Pin-Slot Motion



- Click on the **Pin-Slot** option from **Type** drop-down in the **Motion** tab of **JOINT** dialog box, if you want to revolve and slide the component together like in nut & bolt locking-unlocking.

Planar Motion



- Click on the **Planar** option from **Type** drop-down in the **Motion** tab of **JOINT** dialog box, if you want to assemble the component in such a way that the component can move over the selected plane only in defined boundary.

Ball Motion



- Click on the **Ball** option from **Type** drop-down in the **Motion** tab of **JOINT** dialog box, if you want to assemble the component in such a way that the component is free to move 360 degree in 3D space pivoted to a point; refer to [Figure-19](#).

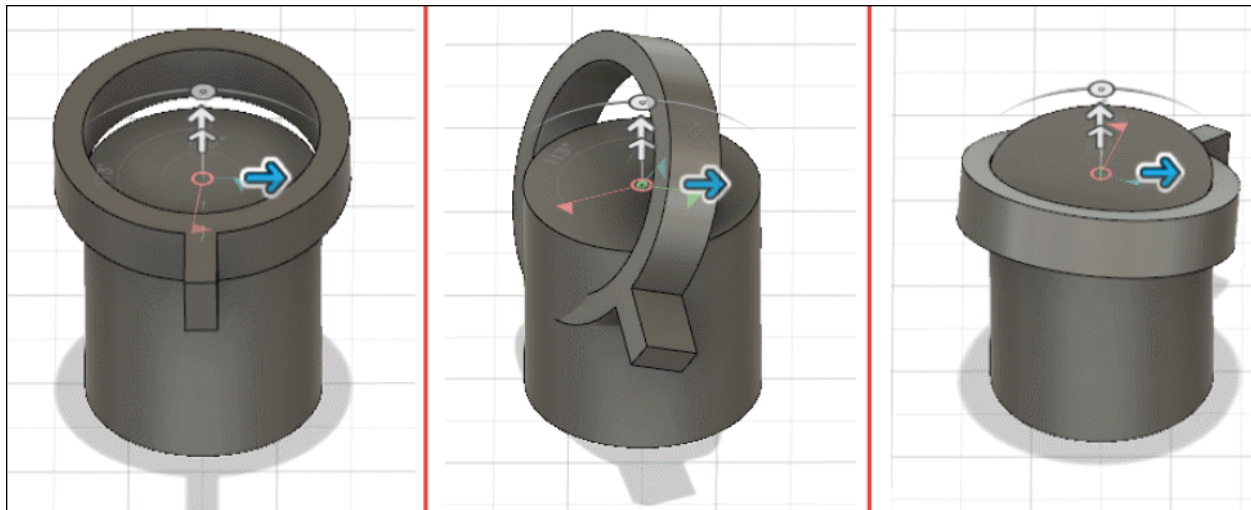


Figure-19. Ball motion

- Click on the **Animate** button from **Motion** tab of **JOINT** dialog box to watch the applied motion by animating **Component 1**. Click the button again to stop the animation.

After specifying the parameters for joint, click on the **OK** button from **JOINT** dialog box to complete the process.

AS-BUILT JOINT

The **As-built Joint** tool allows the joints to be applied to components in their current position as they are built, making it easy to join them in their original orientation. The procedure to use this tool is discussed next.

- Click on the **As-built Joint** tool of **ASSEMBLE** drop-down from **Toolbar**; refer to [Figure-20](#). The **AS-BUILT JOINT** dialog box will be displayed; refer to [Figure-21](#).

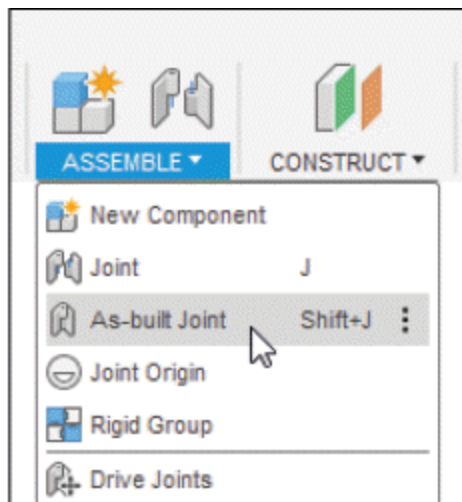


Figure-20. As built Joint tool

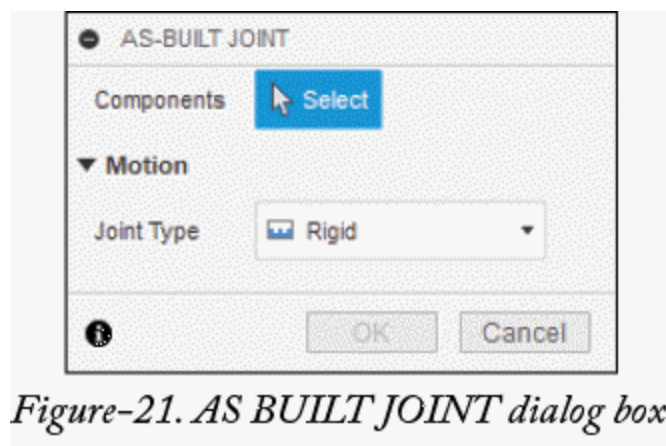


Figure-21. AS BUILT JOINT dialog box

- Select the components from the canvas.
- Select required motion from **Joint Type** drop-down in

Motion area of the dialog box. The updated **AS-BUILT JOINT** dialog box will be displayed; refer to [Figure-22](#).

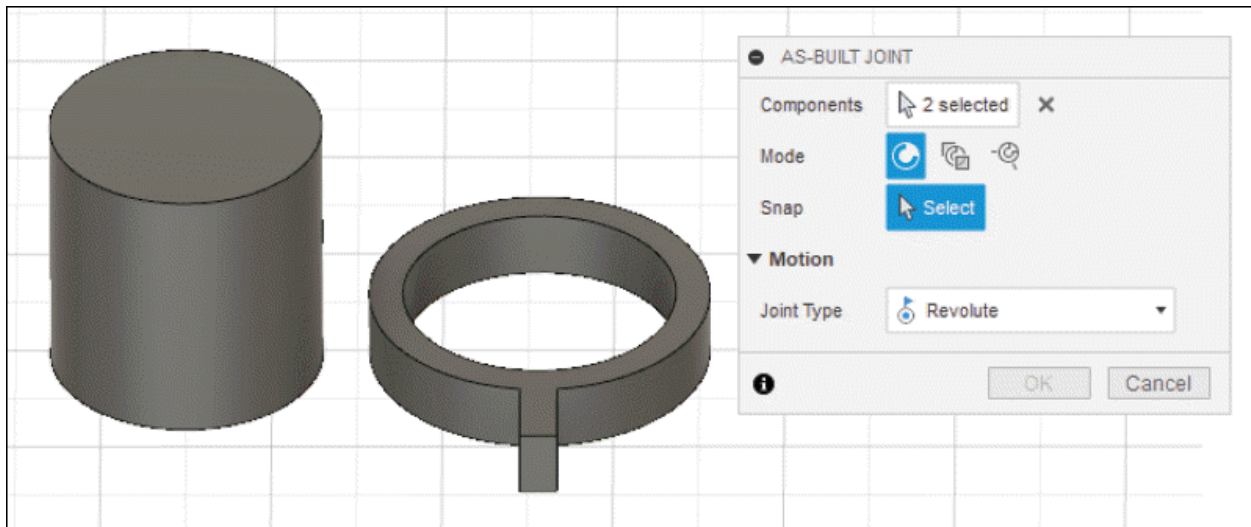


Figure-22. Updated AS BUILT JOINT dialog box

- Select desired option from **Mode** section of the dialog box which have been discussed earlier.
- Select desired references on the components based on the option selected in **Joint Type** drop-down; refer to [Figure-23](#).

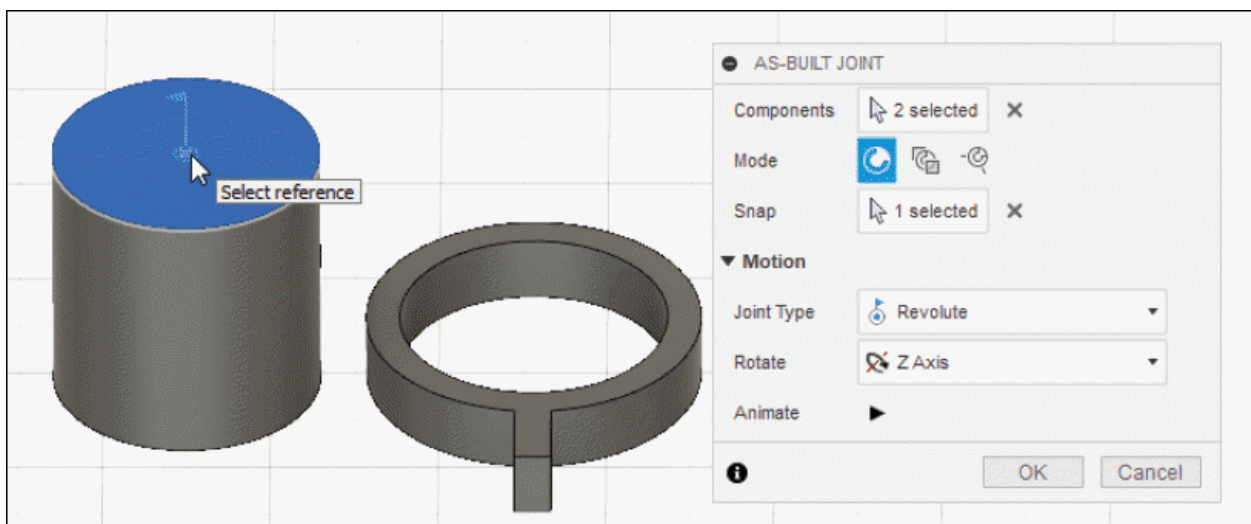


Figure-23. Selecting reference for motion

- Select desired option from **Rotate** drop-down to specify the axis for rotation of the component.
- Click on the **Animate** button from **Motion** area of the dialog box to watch the applied motion by animating the component. Click the button again to stop the animation.
- After specifying the parameters, click on the **OK** button from **AS-BUILT JOINT** dialog box to complete the process.
- If you want to rotate the component after creating joint then double-click on the flag displayed; refer to [Figure-24](#).

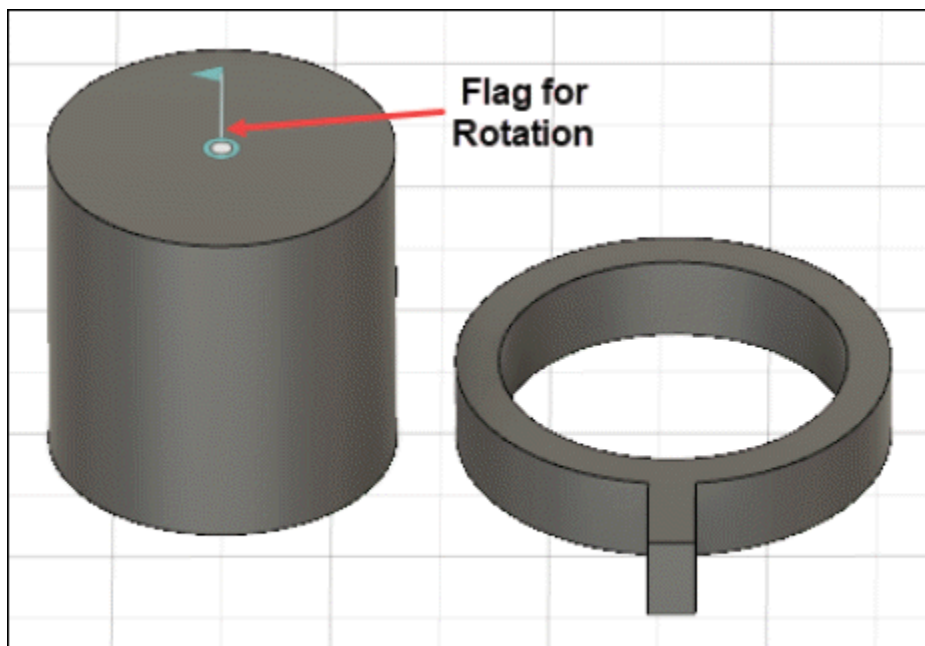


Figure-24. Flag for rotation

- The drag handle will be displayed. Rotate the component as required; refer to [Figure-25](#).

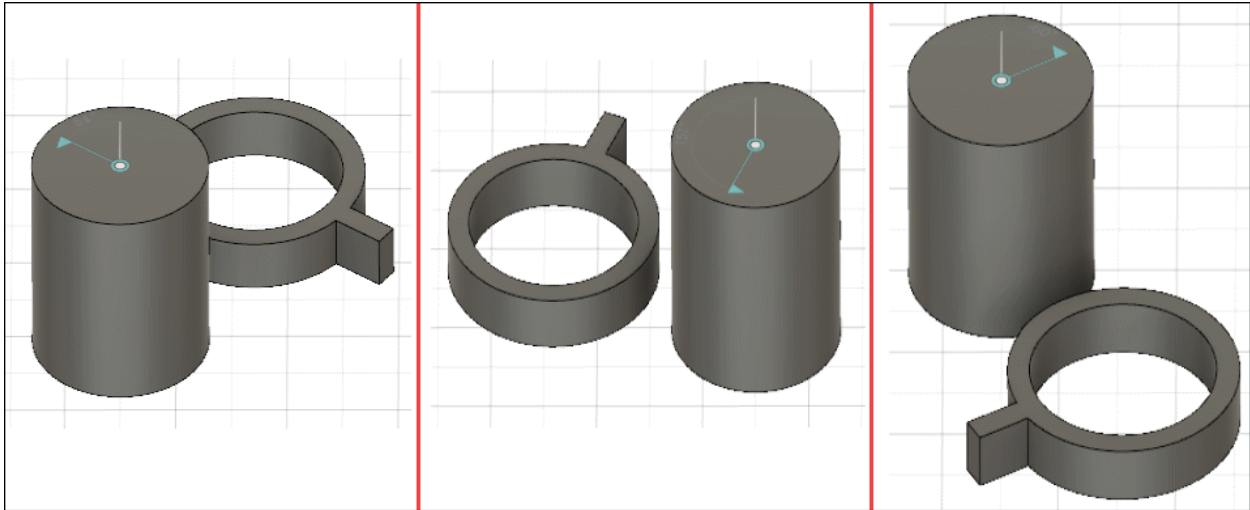


Figure-25. Rotation of component

JOINT ORIGIN

The **Joint Origin** tool is used to define joint origin on a component. The procedure to use this tool is discussed next.

- Click on the **Joint Origin** tool of **ASSEMBLE** drop-down from **Toolbar**; refer to [Figure-26](#). The **JOINT ORIGIN** dialog box will be displayed; refer to [Figure-27](#).



Figure-26. Joint Origin tool Figure-27. JOINT ORIGIN dialog box

- Select **Simple** option from **Mode** section of **JOINT**

ORIGIN dialog box, if you want to place the joint origin on a face.

- Select **Between Two Faces** option from **Mode** section of **JOINT ORIGIN** dialog box if you want to place the joint origin midway between the faces.
- In our case, we are selecting the **Between Two Faces** option from **Mode** section. The updated **JOINT ORIGIN** dialog box will be displayed as shown in [Figure-28](#).

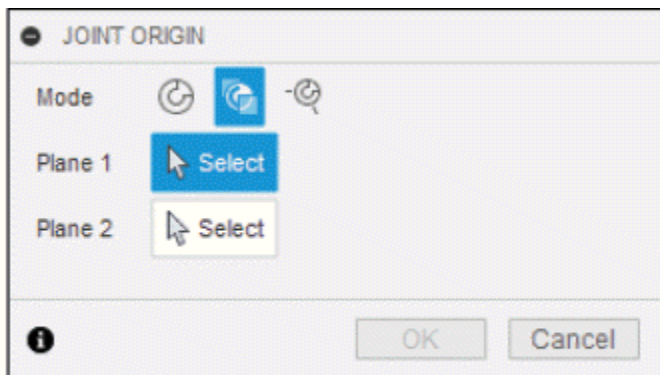


Figure-28. Updated JOINT ORIGIN dialog box

- The **Select** button of **Plane 1** section is active by default. Click on the first face from model to select; refer to [Figure-29](#).

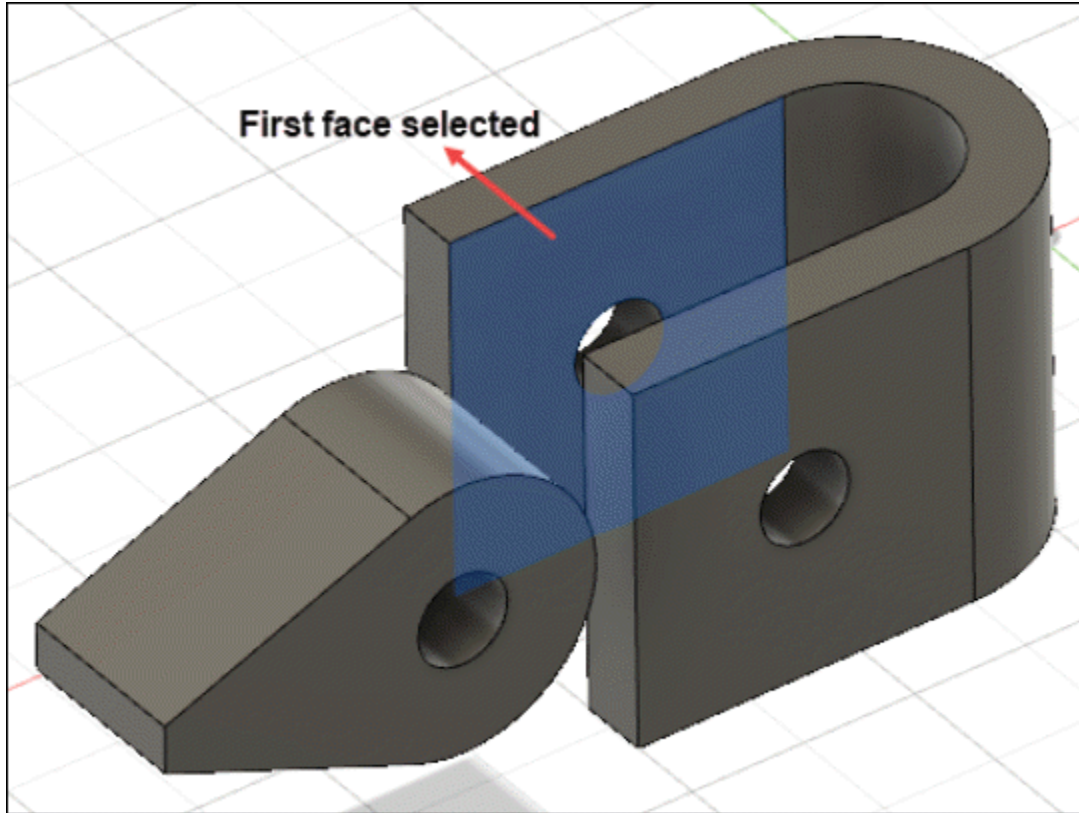


Figure-29. Selection of first face

- On selecting the first face, the **Select** button of **Plane 2** section will be activated. Similarly, select the second face of the model; refer to [Figure-30](#). The updated **JOINT ORIGIN** dialog box will be displayed.

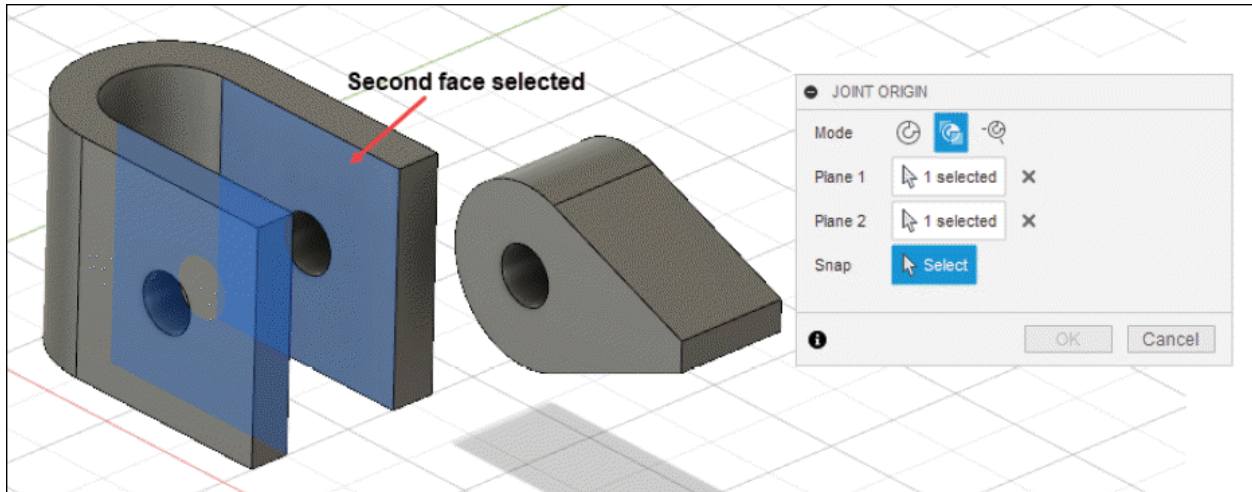


Figure-30. Selection of second face

- The **Select** button of **Snap** option is active by default. Click at desired location to define the snap point; refer to [Figure-31](#).

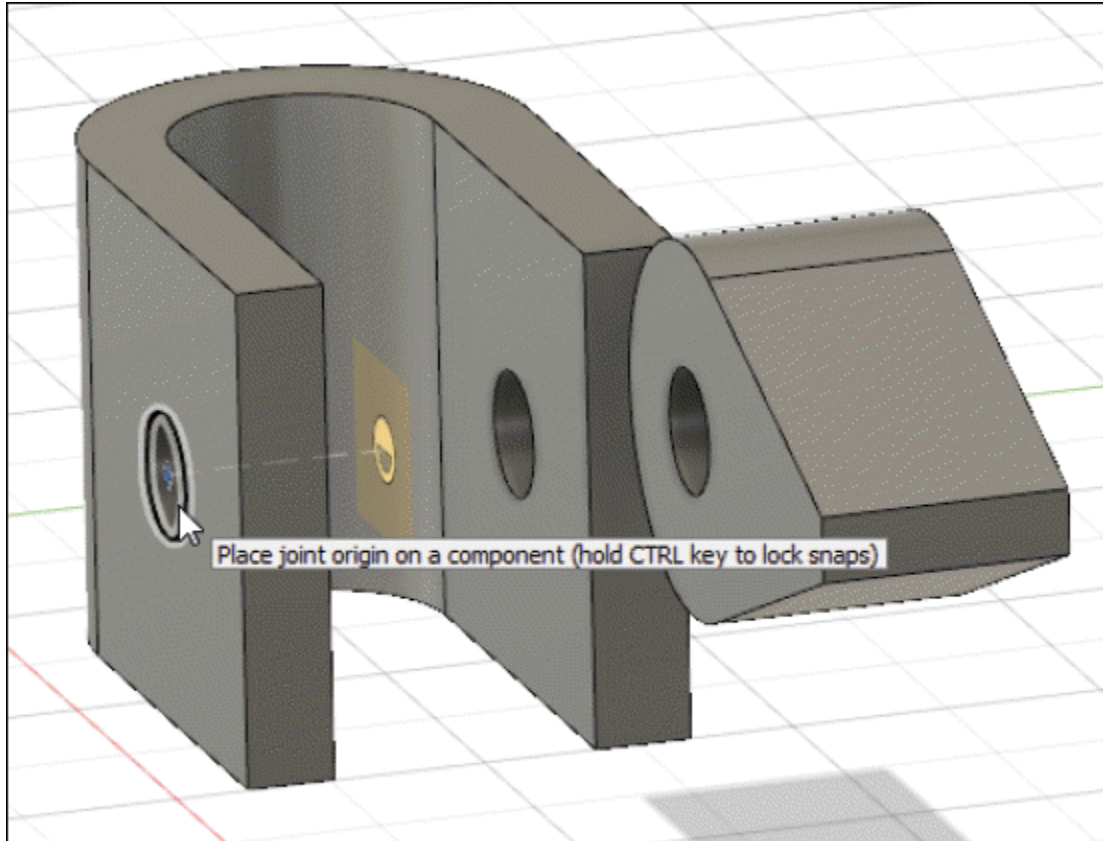


Figure-31. Selection of snap point

- On selecting the snap point, the **JOINT ORIGIN** dialog box will be updated; refer to [Figure-32](#).

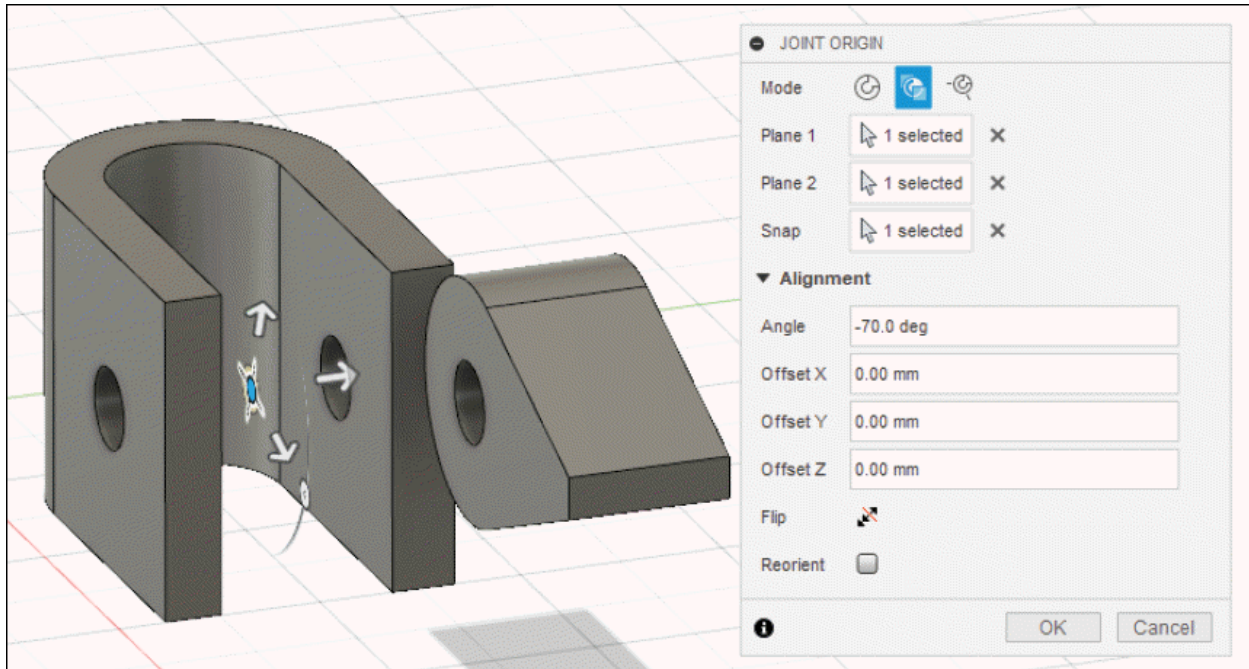


Figure-32. Specifying the location of snap

- If you want to specify the angle of snap then click in the **Angle** edit box of **JOINT ORIGIN** dialog box and specify the angle of rotation of snap.
- If you want to specify the offset distance of snap then click on the **Offset X** edit box and enter the value. You can also set the offset distance by moving the drag handle.
- Similarly, set the offset distance in other edit boxes.
- Select the **Flip** button from **JOINT ORIGIN** dialog box if you want to flip the direction of snap.
- Select the **Reorient** check box from **JOINT ORIGIN** dialog box, if you want to position the axes of origin of geometry.
- After specifying the parameters, click on the **OK** button from **JOINT ORIGIN** dialog box to complete the process.

- Now, you will be able to join the two component with the help of **Joint** tool using newly created joint origin; refer to [Figure-33](#) and [Figure-34](#).

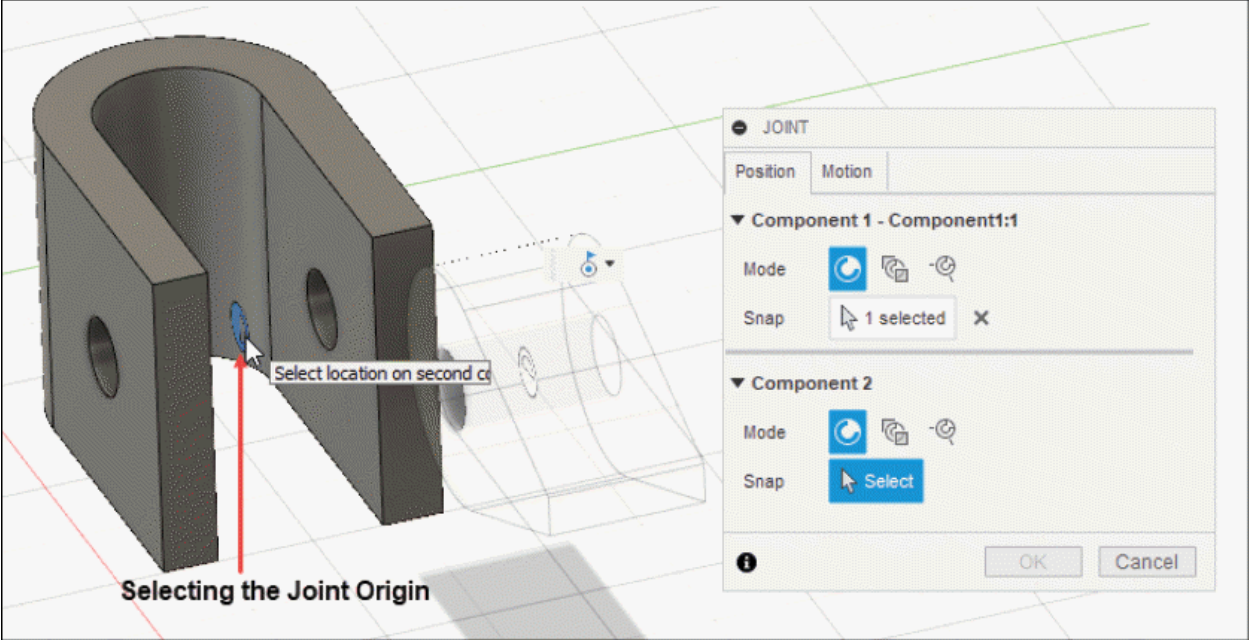


Figure-33. Selection of newly created joint origin

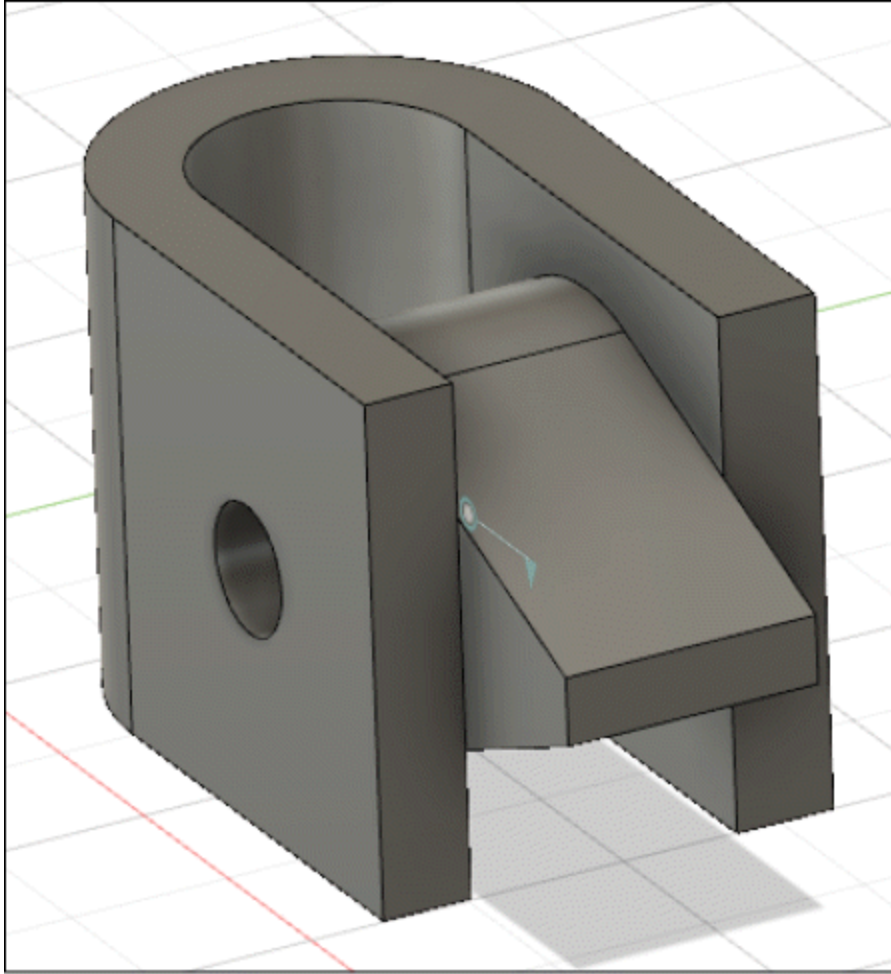


Figure-34. Joint created

RIGID GROUP

The **Rigid Group** tool is used to combine selected components into one group so that the created group act as one object. The procedure to use this tool is discussed next.

- Click on the **Rigid Group** tool of **ASSEMBLE** drop-down from **Toolbar**; refer to [Figure-35](#). The **RIGID GROUP** dialog box will be displayed; refer to [Figure-36](#).

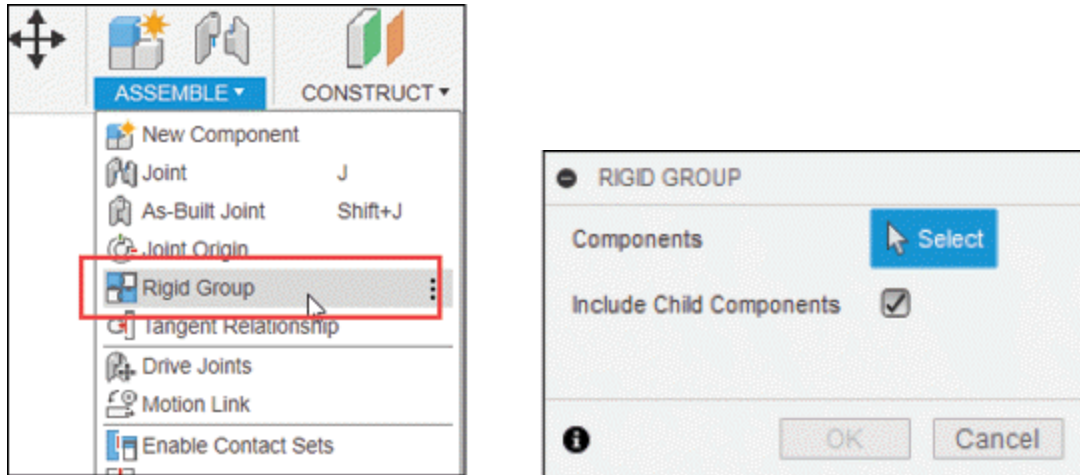


Figure-35. Rigid Group tool Figure-36. RIGID GROUP dialog box

- The **Select** button of **Components** section is active by default. You need to select the components from the model to create a rigid group.
- Select the **Include Child Components** check box if you want to include the child components into the rigid group.
- Click on **OK** button from **RIGID GROUP** dialog box to complete the process.

TANGENT RELATIONSHIP

The **Tangent Relationship** tool is used to create tangent relationship between face on a body in one component and a set of connected faces on body in another component within an assembly in Fusion 360. The procedure to use this tool is discussed next.

- Click on the **Tangent Relationship** tool of **ASSEMBLE** drop-down from **Toolbar**; refer to [Figure-37](#). The **TANGENT RELATIONSHIP** dialog box will be displayed;

refer to [Figure-38](#).

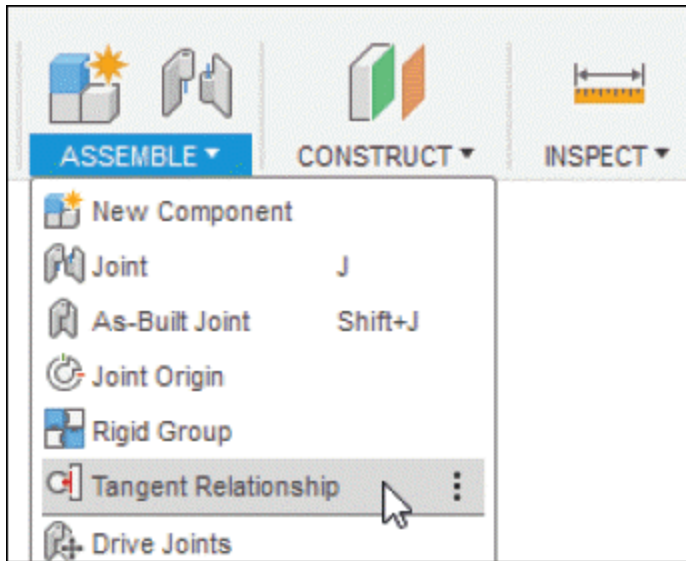


Figure-37. Tangent Relationship tool

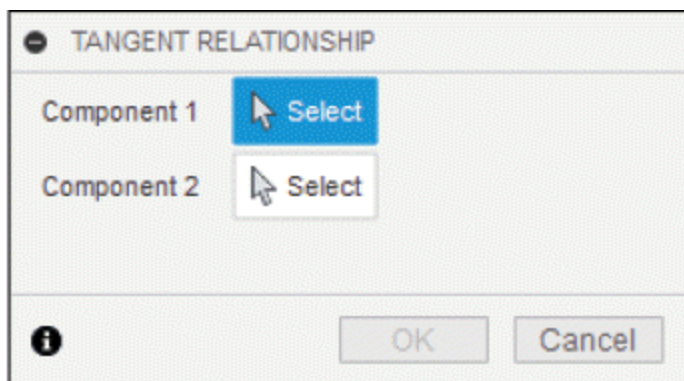


Figure-38. TANGENT RELATIONSHIP dialog box

- The **Component 1** selection button is active by default. Select a face on the body of first component; refer to [Figure-39](#).

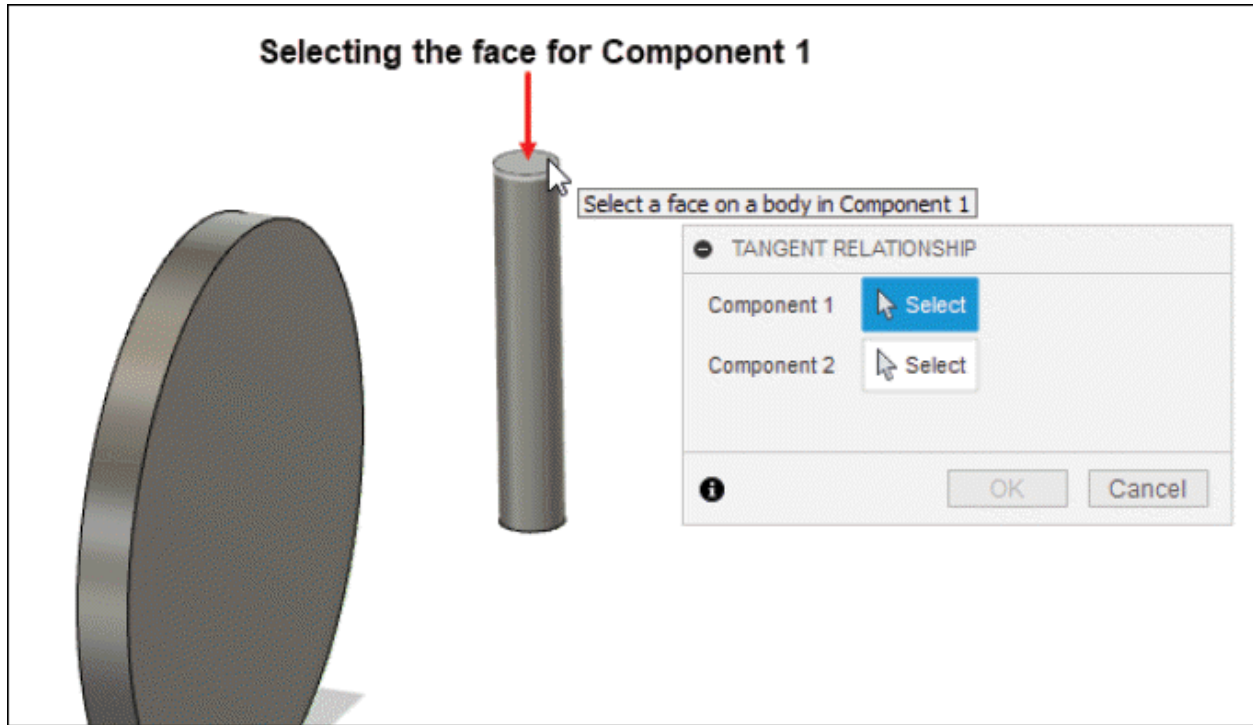


Figure-39. Selecting the face of first component

- After selecting face on a body in the first component, select the face on a body in the second component as shown in [Figure-40](#).

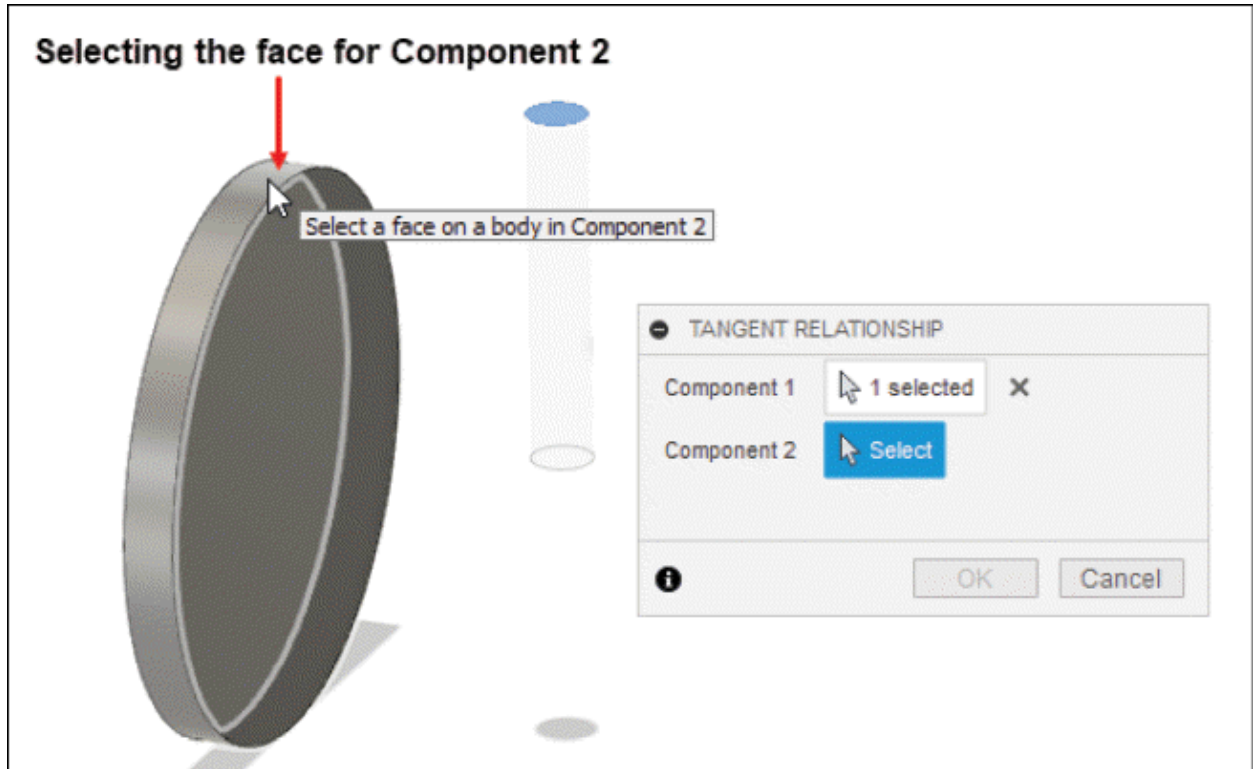


Figure-40. Selecting the face of second component

- After selecting face on a body in the second component, click on the **OK** button from the dialog box. The tangent relationship will be applied between the components; refer to [Figure-41](#).

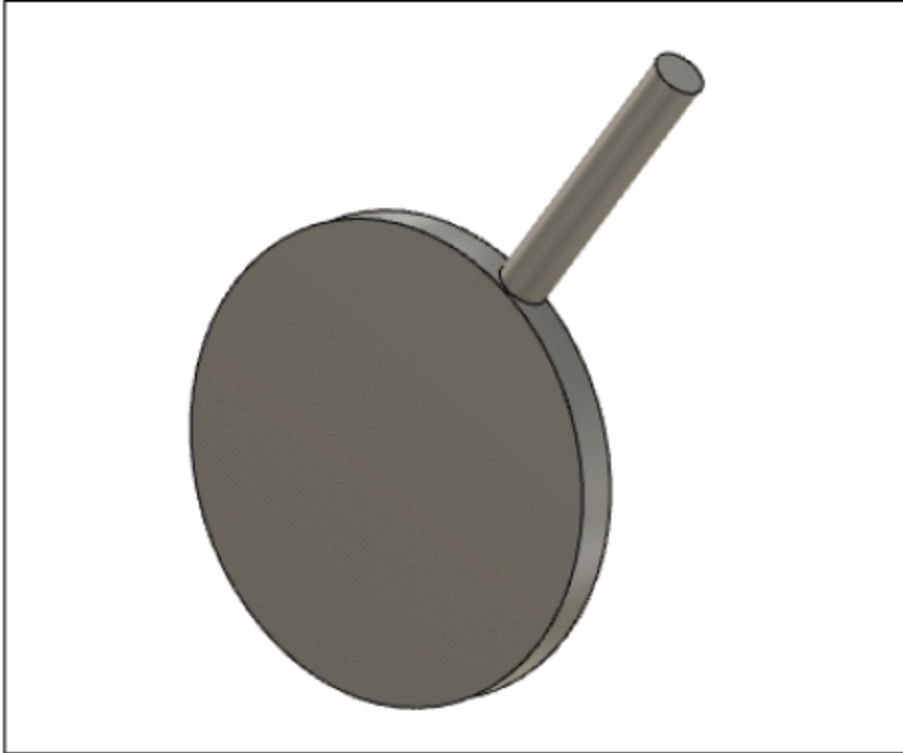


Figure-41. Tangent relationship applied between the components

DRIVE JOINTS

The **Drive Joints** tool is used to specify the rotation angle or distance value based on joint's degrees of freedom. The procedure to use this tool is discussed next.

- Click on the **Drive Joints** tool of **ASSEMBLE** drop-down from **Toolbar**; refer to [Figure-42](#). The **DRIVE JOINTS** dialog box will be displayed; refer to [Figure-43](#).



Figure-42. Drive Joints tool Figure-43. DRIVE JOINTS dialog box

- The **Joint Input** selection is active by default. Click on the joint from model. The updated **DRIVE JOINTS** dialog box will be displayed; refer to [Figure-44](#).

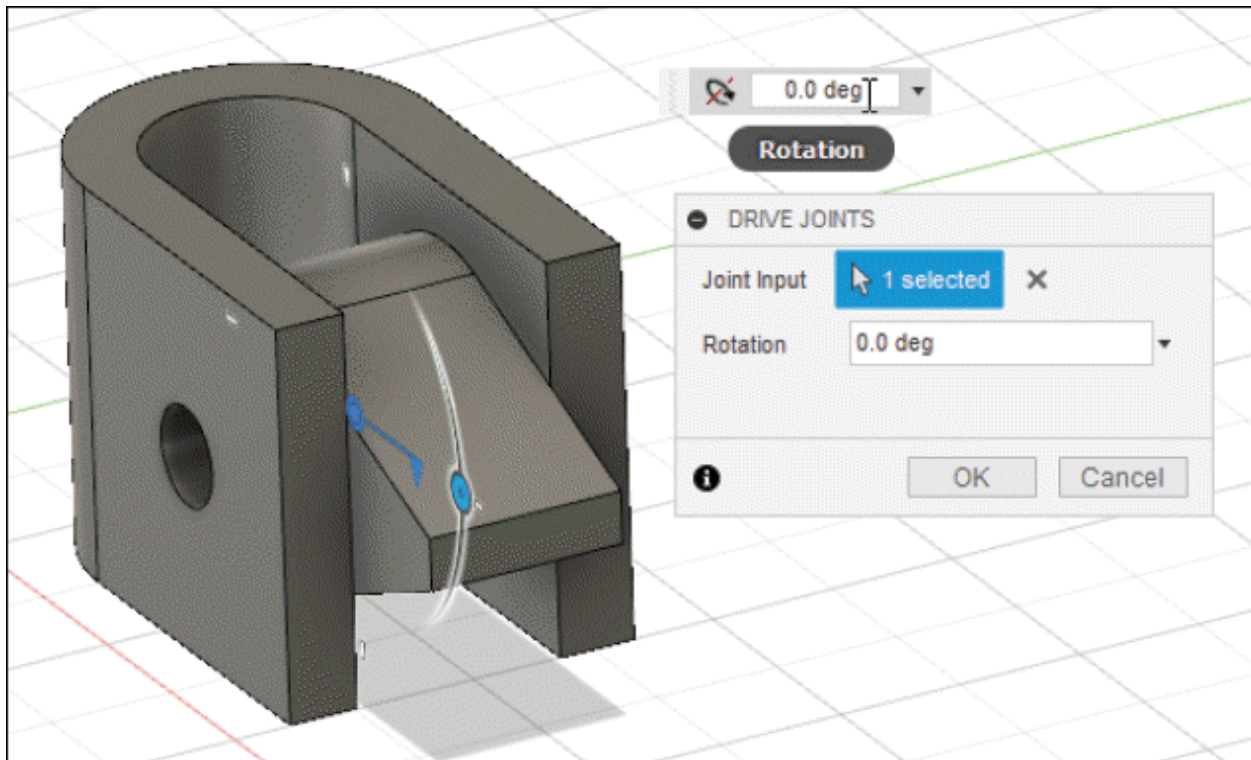


Figure-44. Updated DRIVE JOINTS dialog box

- Click in the **Rotation** edit box and specify the angle of rotation. You can also specify rotation limits by moving the drag handle; refer to [Figure-45](#).

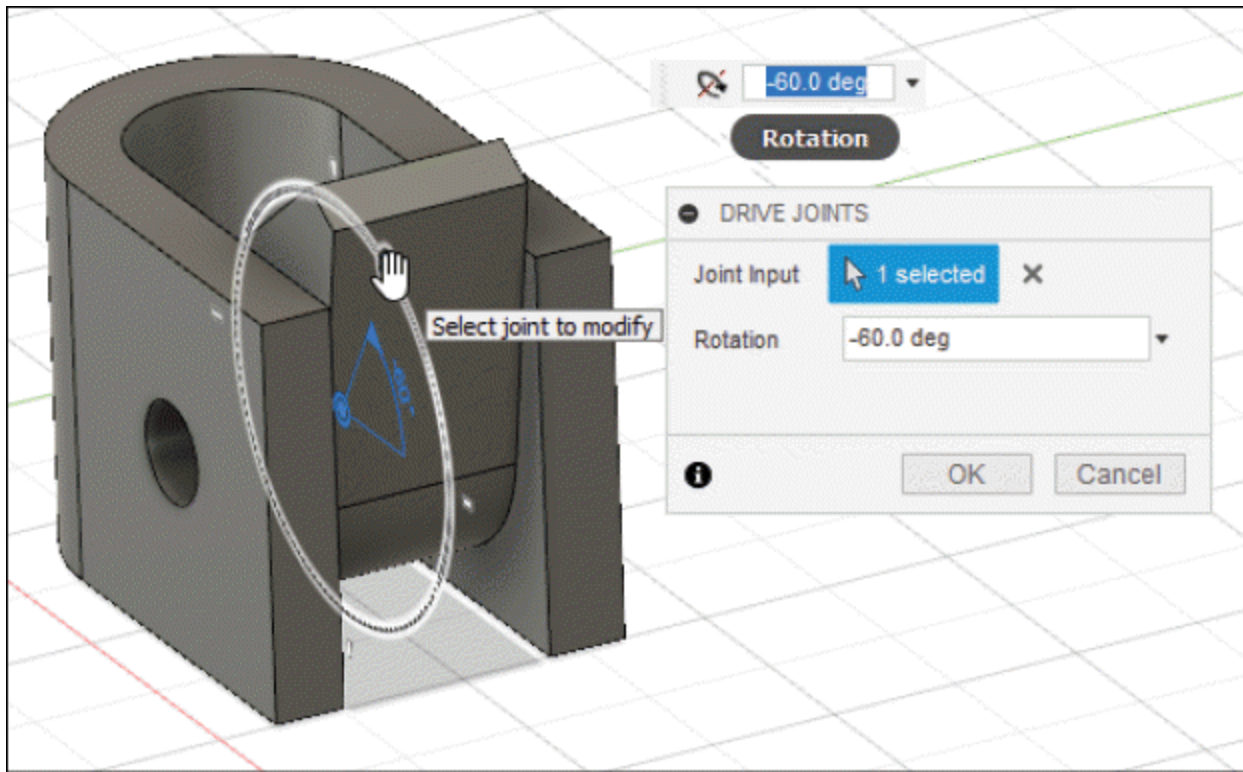


Figure-45. Specifying the rotation limits

- After specifying the parameters, click on the **OK** button from **DRIVE JOINTS** dialog box.

Note that after changing position of joint after driving, the **POSITION** panel is displayed in the **SOLID** tab of **Toolbar**; refer to [Figure-46](#). Select the **Capture Position** tool from the panel if you want to make current position of joint as new start position. Select the **Revert** option from the panel if you want to revert the joint to original position.

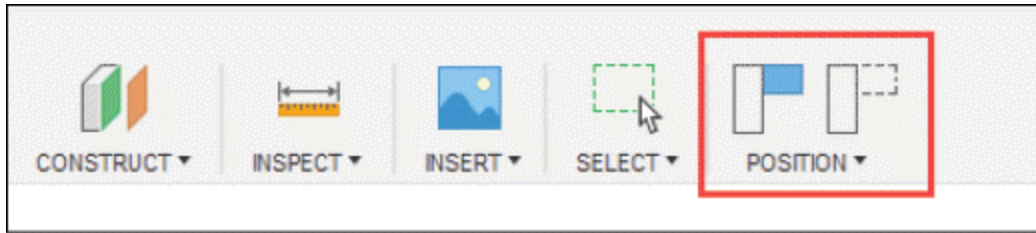


Figure-46. POSITION panel

MOTION LINK

The **Motion Link** tool is used to specify the relationship between motion of two selected joints like relation between two gear rotation or relation between bolt rotation and translation while fastening. The procedure to use this tool is discussed next.

- Click on the **Motion Link** tool of **ASSEMBLE** drop-down from **Toolbar**; refer to [Figure-47](#). The **MOTION LINK** dialog box will be displayed; refer to [Figure-48](#).

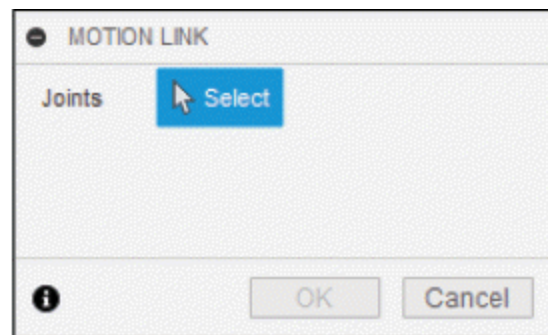
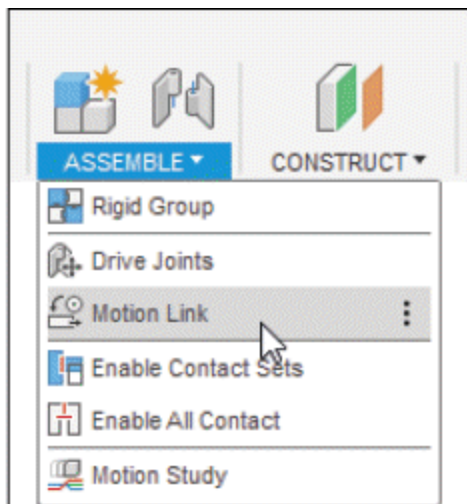


Figure-47. Motion Link tool Figure-48. MOTION LINK dialog box

- **Joints** selection is active by default. Click on desired joints. In our case, we are selecting the slider and revolve

joints to create a motion of vice.

- On selecting the two joints, the animation of vice will start and the dialog box will be updated; refer to [Figure-49](#).

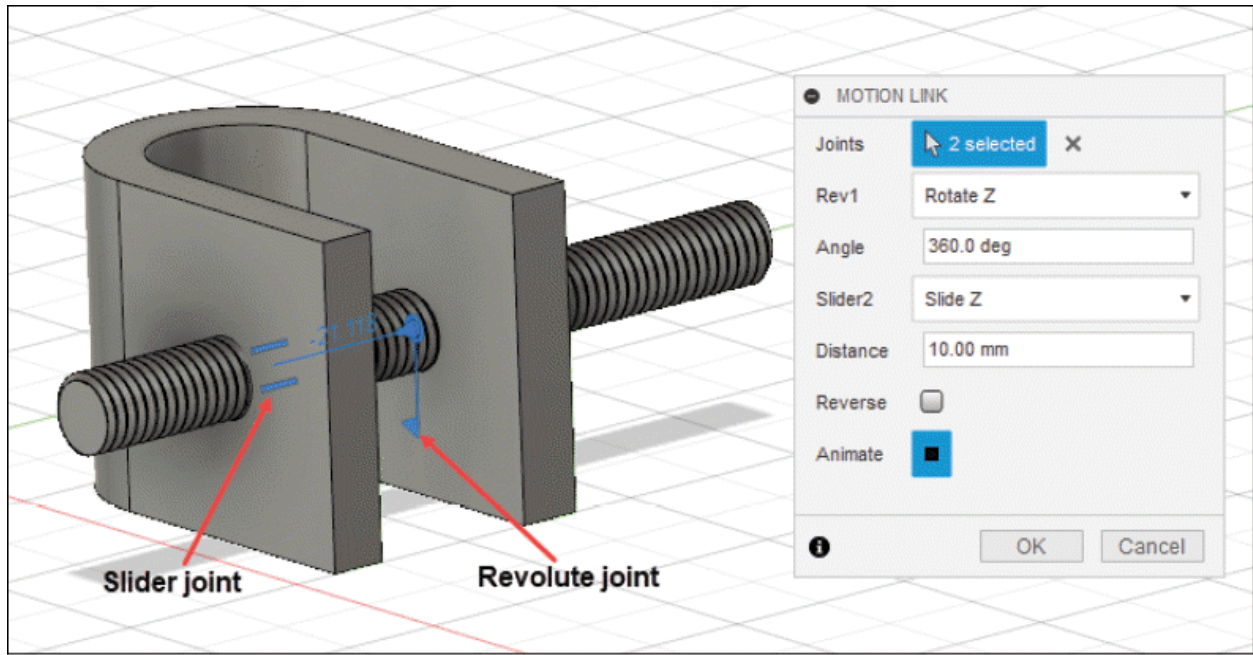


Figure-49. Selection of joints for Motion Link

- Click in the **Distance** edit box of **MOTION LINK** dialog box and specify the value of distance per revolution.
- Click in the **Angle** edit box of **MOTION LINK** dialog box and specify the value of angle of revolution. The combination of distance and angle value define the range of motion like in our case, for one full rotation of **360** degree by handle, the jaw will move **10 mm** of distance.
- Select the **Reverse** check box of **MOTION LINK** dialog box, if you want to reverse the direction of revolution.
- Click on the **Animate** button to watch the animation of joints.

- After specifying the parameters, click on **OK** button of **MOTION LINK** dialog box to complete the process; refer to [Figure-50](#).

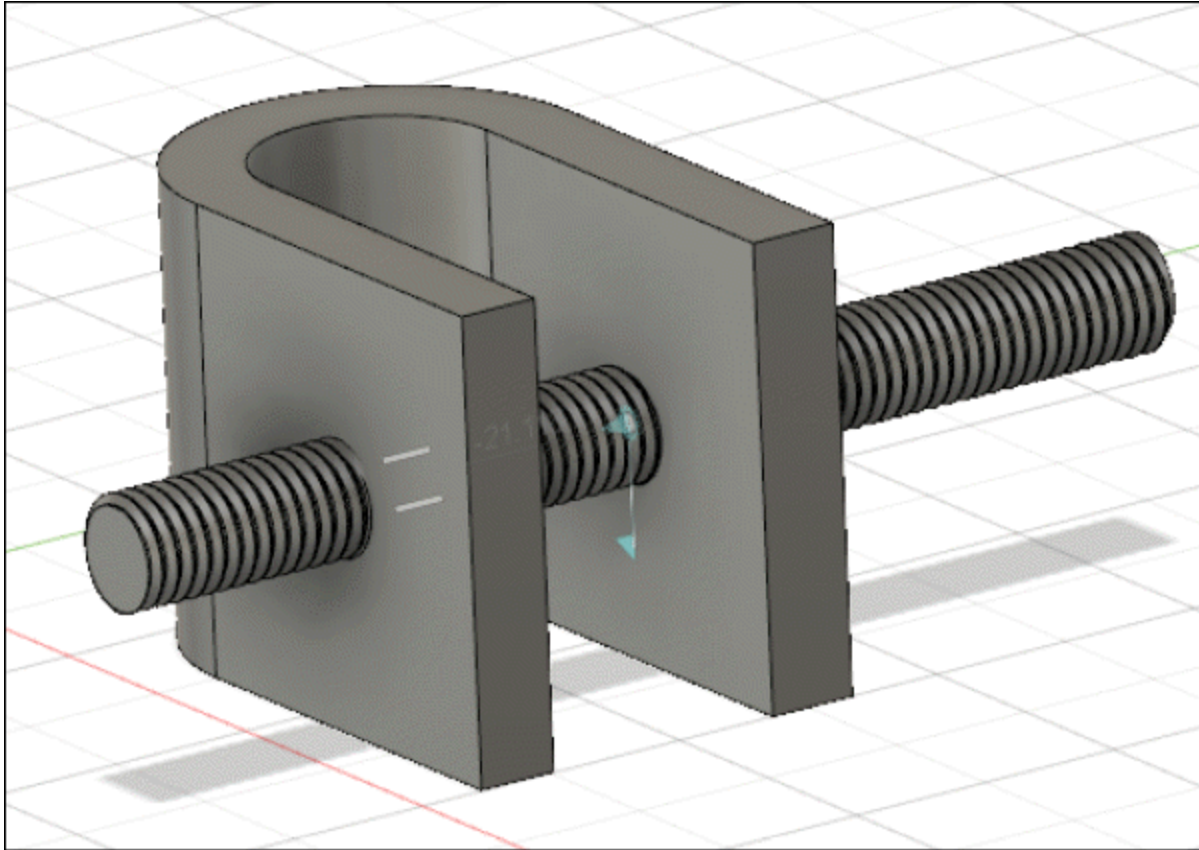


Figure-50. Motion Link created between joints

ENABLE CONTACT SETS

The **Enable Contact Sets** tool is used to enable the physical contact conditions. The procedure to use this tool is discussed next.

- Click on the **Enable Contact Sets** tool of **ASSEMBLE** drop-down from **Toolbar**; refer to [Figure-51](#). The tool will be displayed in **BROWSER**; refer to [Figure-52](#).

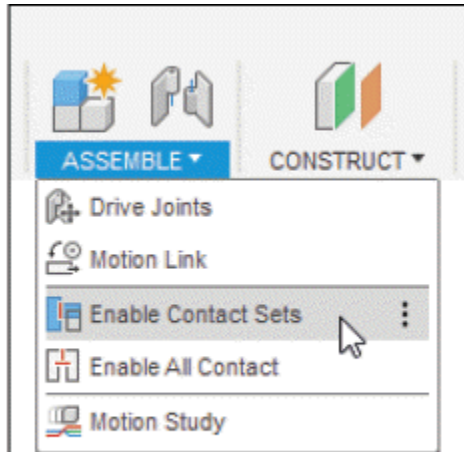


Figure-51. Enable Contact Sets tool

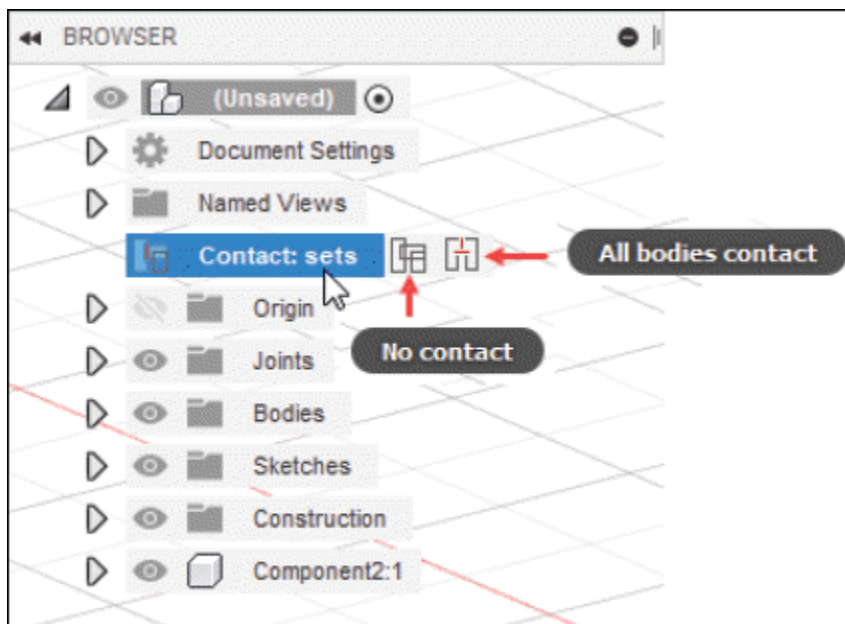


Figure-52. Contact sets

- Click on **All Bodies Contact** button to activate the contact analysis between sets; refer to [Figure-53](#).

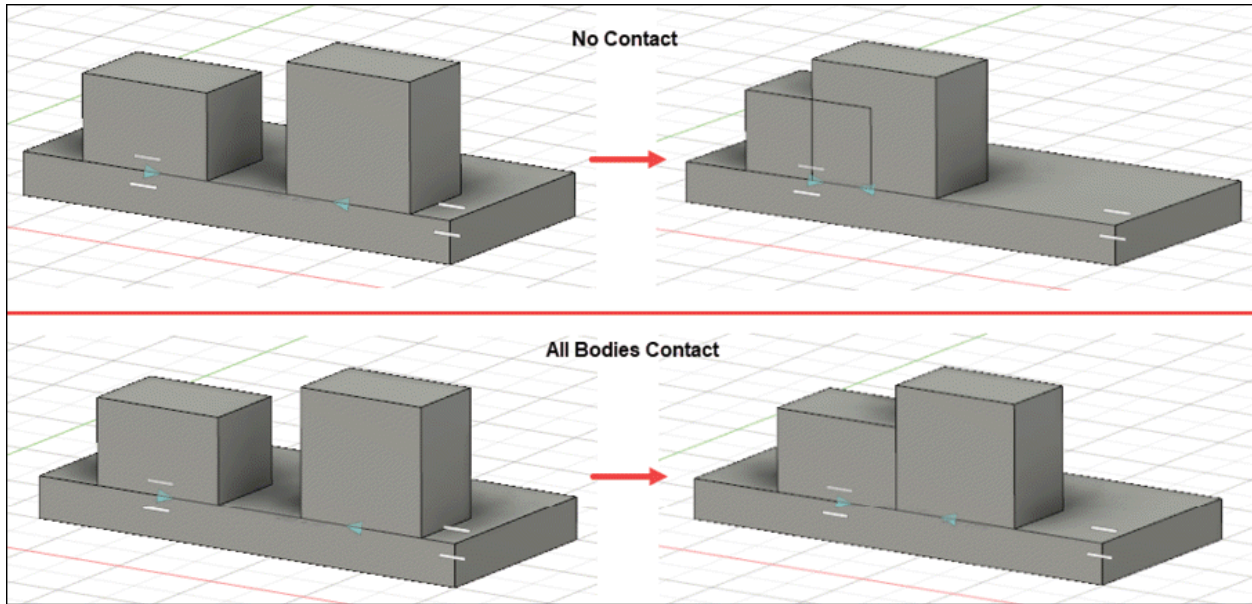


Figure-53. Applying contact sets tool

- Click on **No Contact** button to disable the contact between sets.
- In the above figure, the first case is with contact sets not active and the second case is with contact sets active. So, when contact set is active then physical properties of objects are taken into account like one solid object cannot pass through another solid object.

Enable All Contact

The **Enable All Contact** tool is used to enable all contacts which were applied earlier in the part or model.

- To enable contact, click on the **Enable All Contact** tool of **ASSEMBLE** drop-down from **Toolbar**. All contacts will be enabled and the **ASSEMBLE** drop-down will be updated with two new tools which are discussed next.

Disable Contact

The **Disable Contact** tool is used to disable all contacts which were made earlier in the part or model.

- To disable contact, click on the **Disable Contact** tool of **ASSEMBLE** drop-down from **Toolbar**. All contacts will be disabled.

New Contact Set

The **New Contact Set** tool is used to create new contact set between two selected components. This tool is available when **Enable All Contact** tool is selected. These contact sets prevent parts from passing through one another. The procedure to use this tool is discussed next.

- Click on the **New Contact Set** tool of **ASSEMBLE** drop-down from **Toolbar**; refer to [Figure-54](#). The **NEW CONTACT SET** dialog box will be displayed; refer to [Figure-55](#).

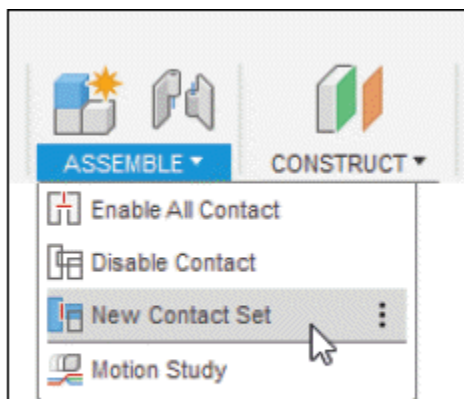


Figure-54. New Contact Set tool

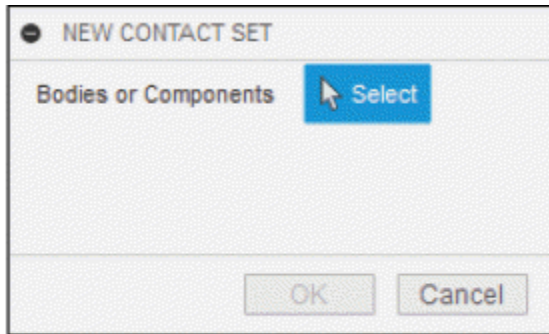


Figure-55. NEW CONTACT SET dialog box

- The **Select** button of **Bodies or Components** section is active by default. Select the two component or bodies to create sets and click on **OK** button from **NEW CONTACT SET** dialog box to complete the process.

MOTION STUDY

The **Motion Study** tool is used to perform kinematic motion analysis based on joints. The procedure to use this tool is discussed next.

- Click on the **Motion Study** tool of **ASSEMBLE** drop-down from **Toolbar**; refer to [Figure-56](#). The **Motion Study** dialog box will be displayed; refer to [Figure-57](#).

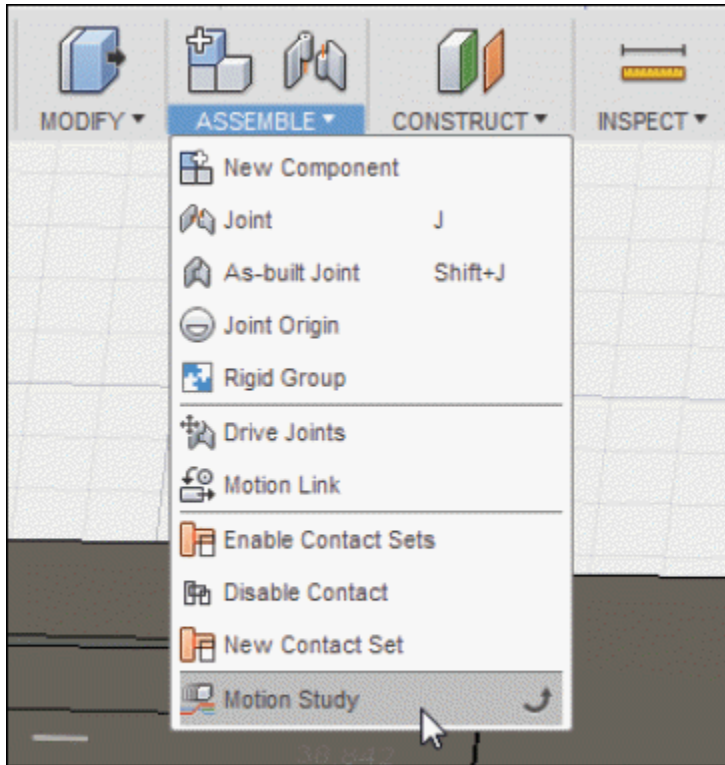


Figure-56. Motion Study tool

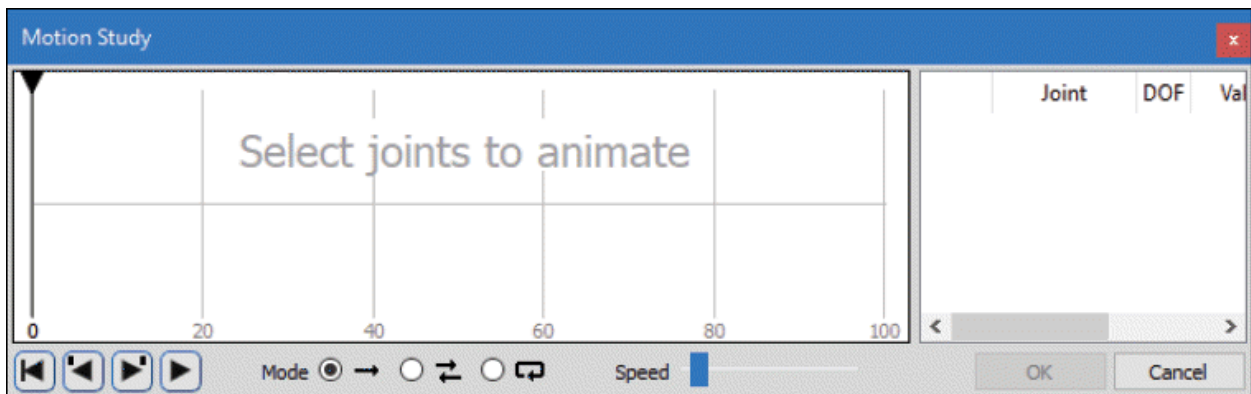


Figure-57. Motion Study dialog box

- You need to select the joint from part or model. Click to select the joint; refer to [Figure-58](#).

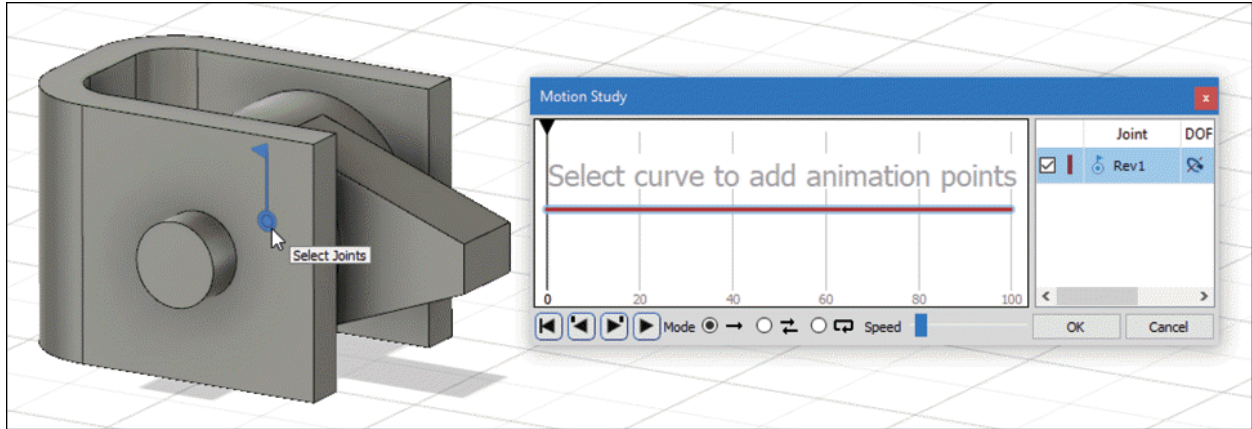


Figure-58. Selecting joint for animation

- After selecting joint, create the points of motion curve as required by clicking in the graph line of motion; refer to [Figure-59](#).

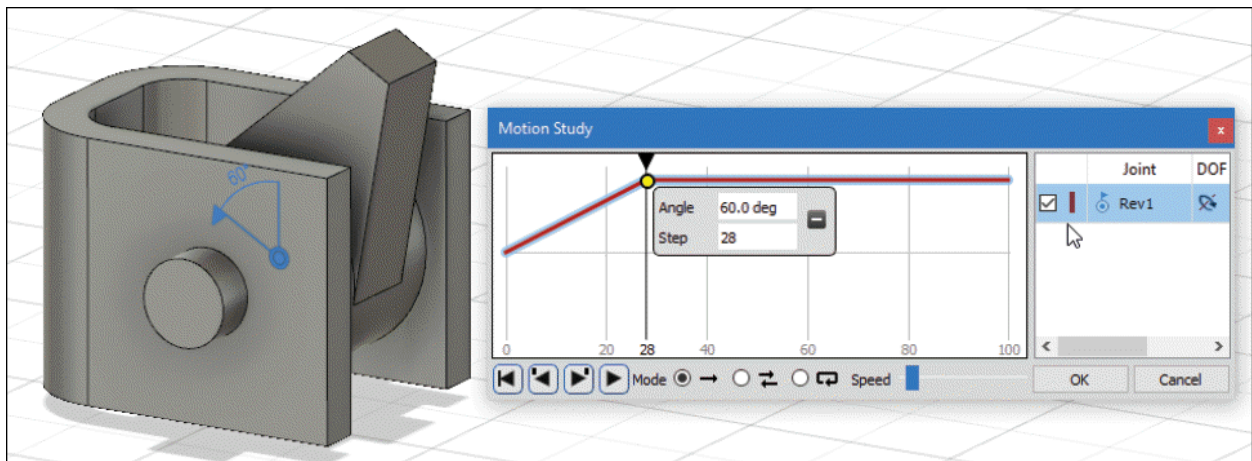


Figure-59. Adjusting points for animation

- Click in the **Angle** edit box and specify required angle for animation point. Note that the options will be displayed based on selected joint.
- After creating the animation points, set the speed of animation from **Speed** bar.
- Select **Play Once** radio button from **Mode** section to play

the animation once.

- Select **Play Forward/Backward** radio button from **Mode** section to play the animation forward and then backward.
- Select **Play Loop** radio button from **Mode** section to play the animation in cycle.
- After specifying the parameters for animation, click on **Play** button to check the animation of selected joint.
- Click on **OK** button from **Motion Study** dialog box to exit the motion study of components.

ARRANGING COMPONENTS IN ENCLOSURES

The **Arrange** tool is used to arrange assembly components in a specified 2D or 3D enclosure utilizing the space optimally. The procedure to use this tool is given next.

- Click on the **Arrange** tool from the **MODIFY** drop-down in the **SOLID** tab of **Toolbar**. The **ARRANGE** dialog box will be displayed; refer to [Figure-60](#) and you will be asked to select components for arranging.

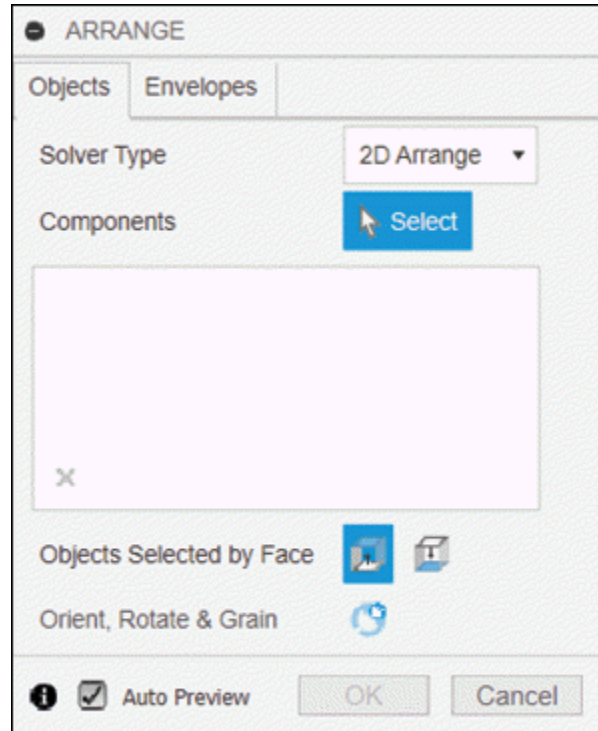


Figure-60. ARRANGE dialog box

- Select all the components from **BROWSER** or graphics area to be arranged.

Arranging in 2 Dimensions

- Select the **2D Arrange** option from the **Solver Type** drop-down to arrange components in 2D plane.
- Select the **Face Up** button from **Objects Selected by Face** section if you want the selected faces of object to point upward when arranging them. Similarly, select the **Face Down** button if you want the faces of selected objects to be pointing downward when arranging them.
- Click on the **Envelopes** tab to define the boundaries for arranging components. The options will be displayed as

shown in [Figure-61](#) and you will be asked to select a plane/face/sketch for defining boundary of envelope.

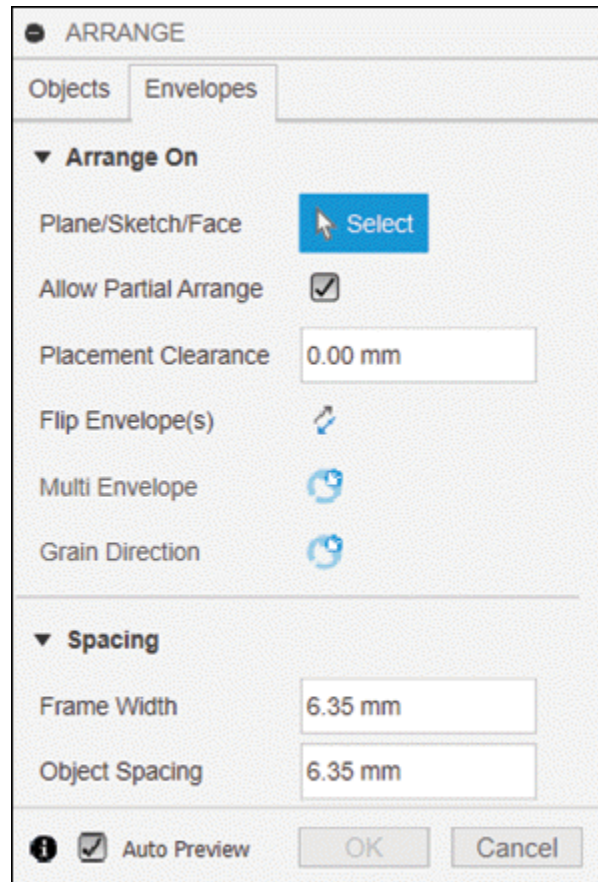


Figure-61. Envelopes tab

- If you have selected a sketch then components will be automatically arranged within sketch boundary and options will be displayed as shown in [Figure-62](#).

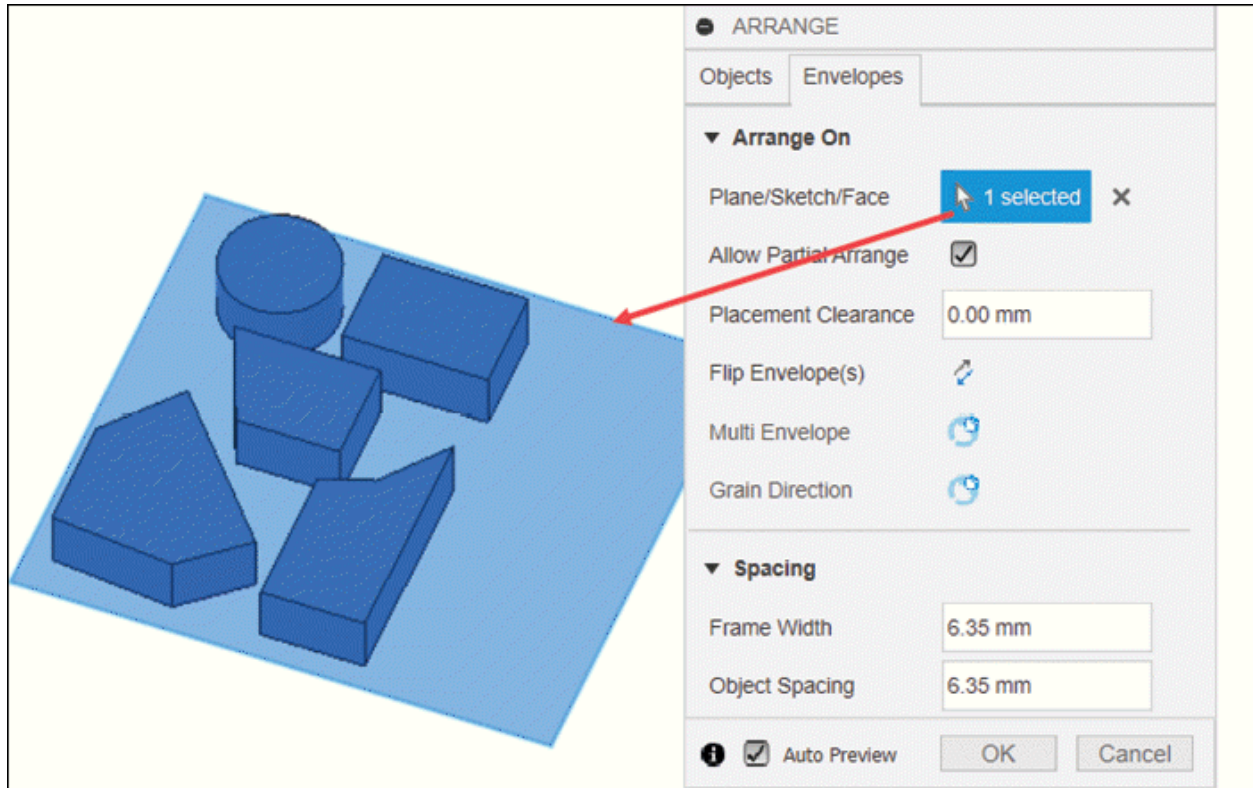


Figure-62. Arranging in 2D sketch

- Select **Allow Partial Arrange** check box if you want to arrange as many components as possible when there is not sufficient space in envelope. Clear this check box if you want to generate an error when there is no sufficient space for all the components in envelope.
- Specify desired value in **Placement Clearance** edit box to raise the components from sketch plane at respective height.
- Click on the **Flip Envelope(s)** button to reverse the arrangement between above and below plane.
- If you have activated the **Nesting & Fabrication Extension** then **Multi Envelope** and **Grain Direction**

options will be available to you and the **Envelopes** tab should display as shown in [Figure-63](#).

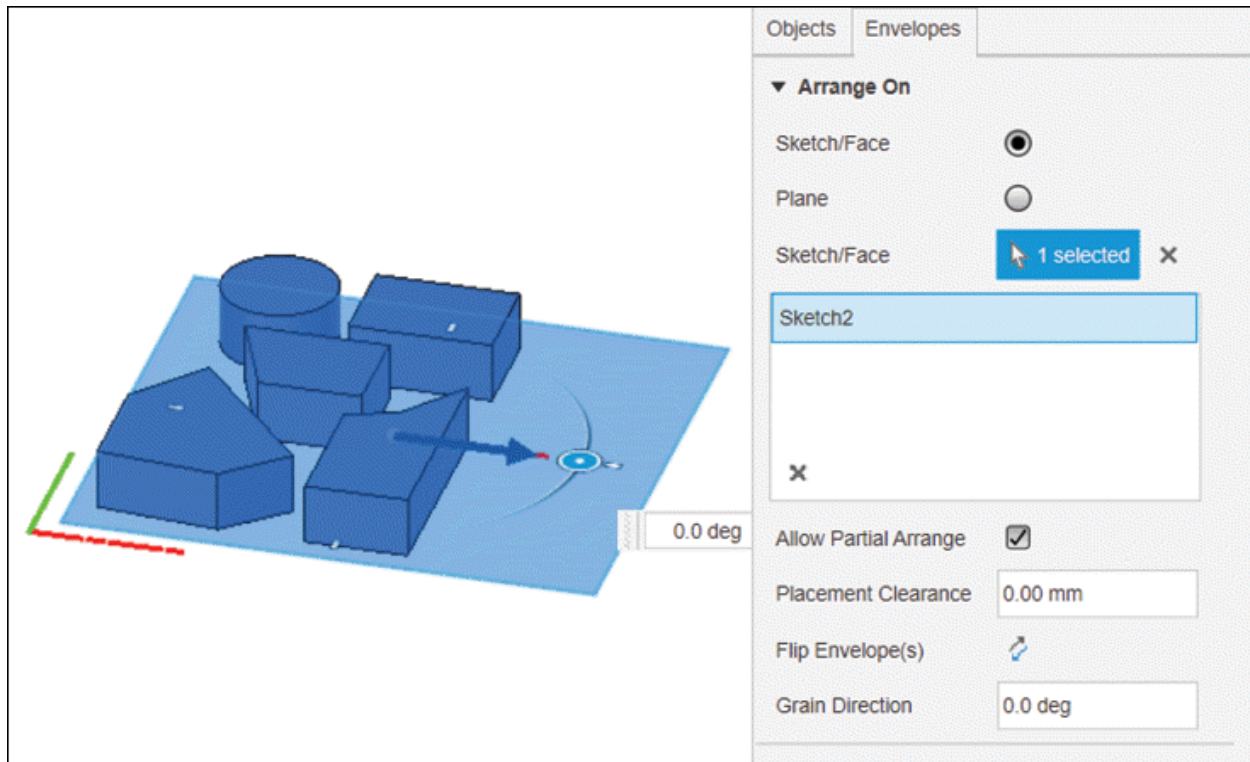


Figure-63. Updated Envelopes tab

- Set desired value in the **Grain Direction** edit box to define angle for orienting components.
- After activating **Multi Envelope** extension, you can now select multiple faces as envelopes for arranging components.

Arranging in 3 Dimensions

- After selecting components for arrangement, select the **3D Arrange** option from the **Solver Type** drop-down in **Objects** tab of dialog box to arrange components in 3 dimensions.

- Click on the **Envelopes** tab from the dialog box. The options will be displayed as shown in [Figure-64](#).
- Select desired plane from graphics area and set desired parameters in the **Length**, **Width**, and **Height** edit boxes to define limits of 3D envelope. The components will be arranged automatically in 3 dimension; refer to [Figure-65](#).

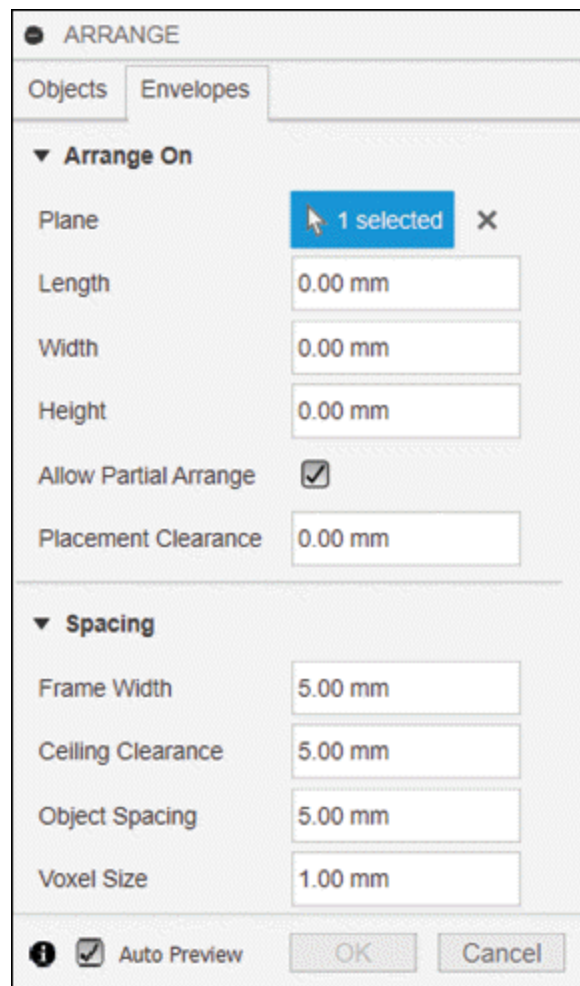


Figure-64. Envelopes tab for 3D

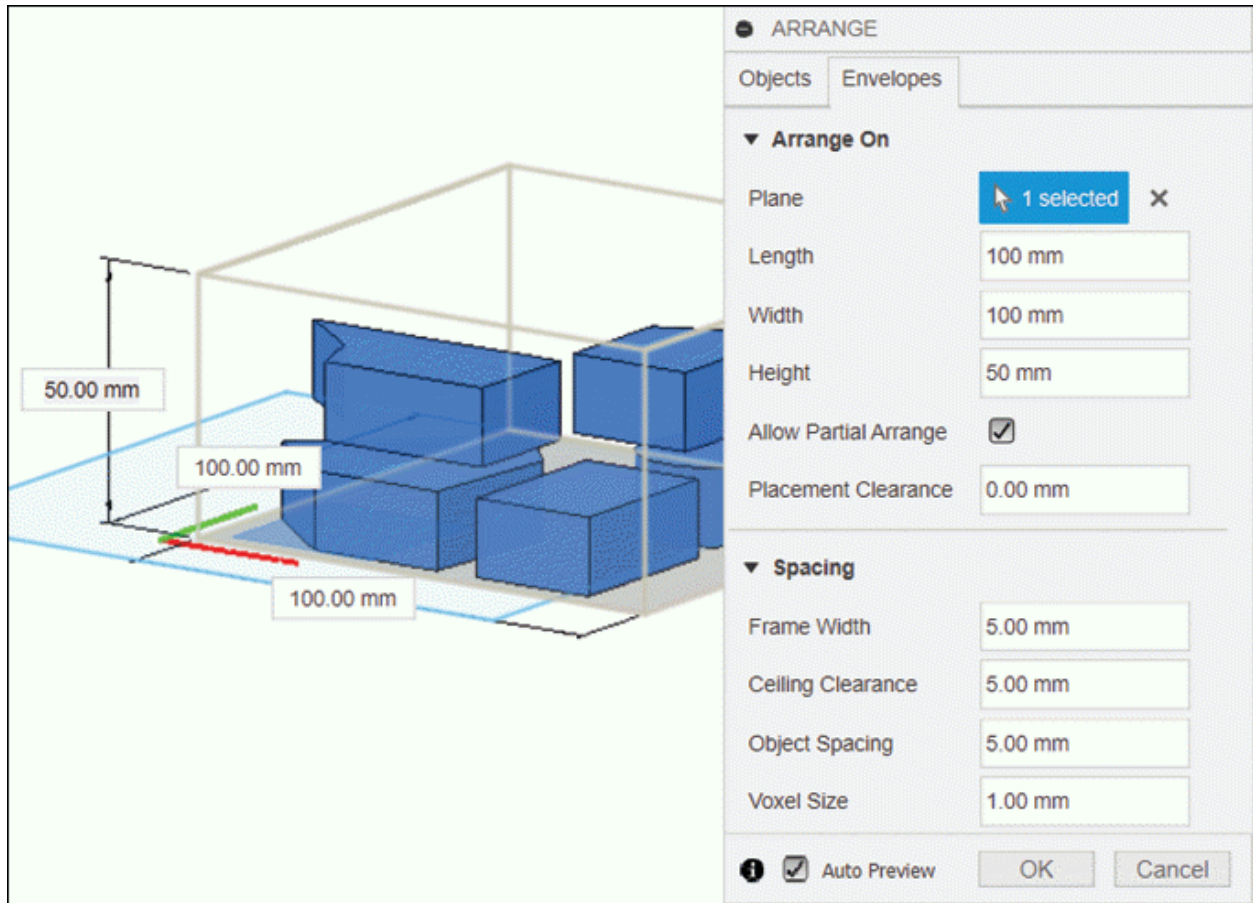


Figure-65. 3D envelope arrangement

- In the **Spacing** section of dialog box, set desired parameters in edit boxes to define spacing of components with respect to frame, ceiling, and consecutive objects. You can also define voxel (square) size to be used as reference for spacing.
- After setting parameters, click on the **OK** button to generate the arrangement.

AUTOMATED MODELING

The **Automated Modeling** tool in **AUTOMATE** drop-down of **Toolbar** is used to automatically generate designs for an

assembly part based on specified connecting faces and avoiding bodies. The tool generates freeform shapes that can be used as base reference for further designing the components. The procedure to use this tool is given next.

- Click on the **Automated Modeling** tool from the **AUTOMATE** drop-down in the **Toolbar**. The **AUTOMATED MODELING** dialog box will be displayed; refer to [Figure-66](#) and you will be asked to select faces to be connected.

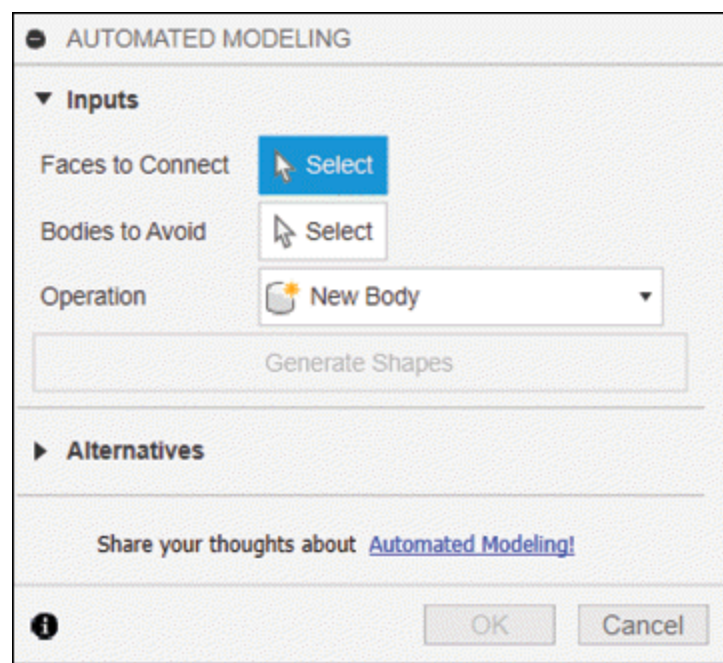


Figure-66. AUTOMATED MODELING dialog box

- Select the faces to be connected by automated modeling and click on the **Select** button for **Bodies to Avoid** section. You will be asked to select the bodies to be avoided.

- Select desired bodies/components to be avoided; refer to [Figure-67](#).

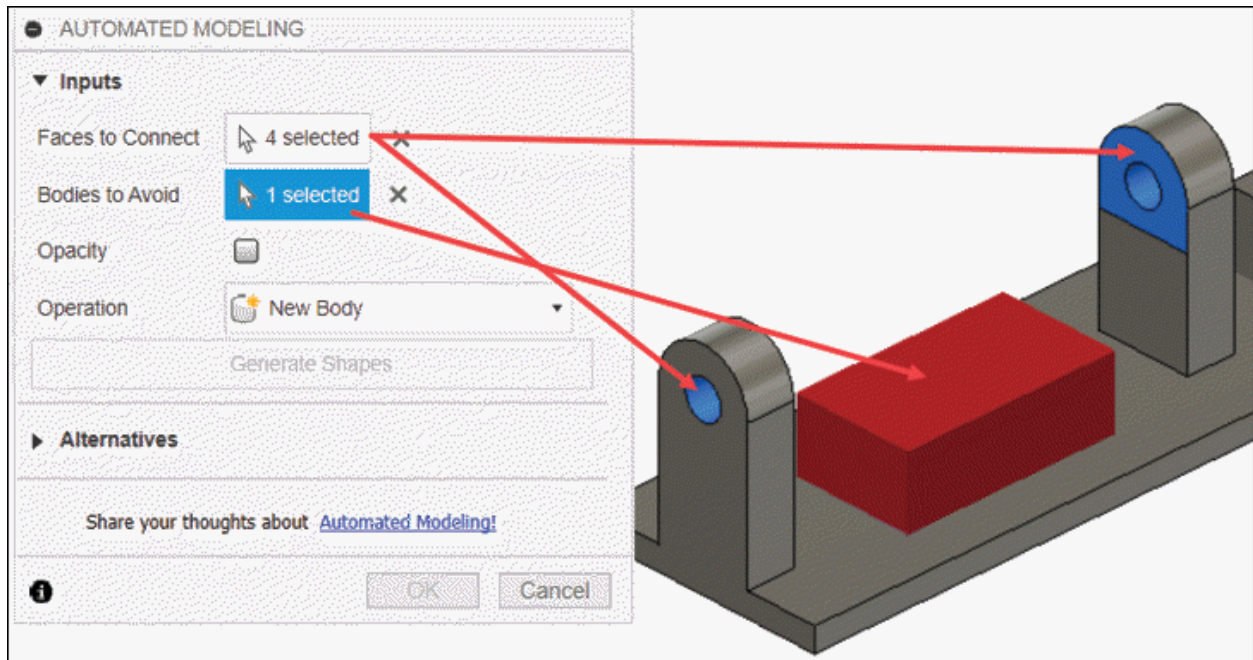


Figure-67. Selection for automated modeling

- Select the **Opacity** check box to make the avoidance bodies transparent.
- Select the **New Body** option from the **Operation** drop-down if you want to create a new body by automated modeling. Select the **New Component** option from the drop-down if you want to create a new component using automated modeling.
- Click on the **Generate Shapes** button from the dialog box to generate the shapes. After calculation, preview of alternatives will be displayed; refer to [Figure-68](#).

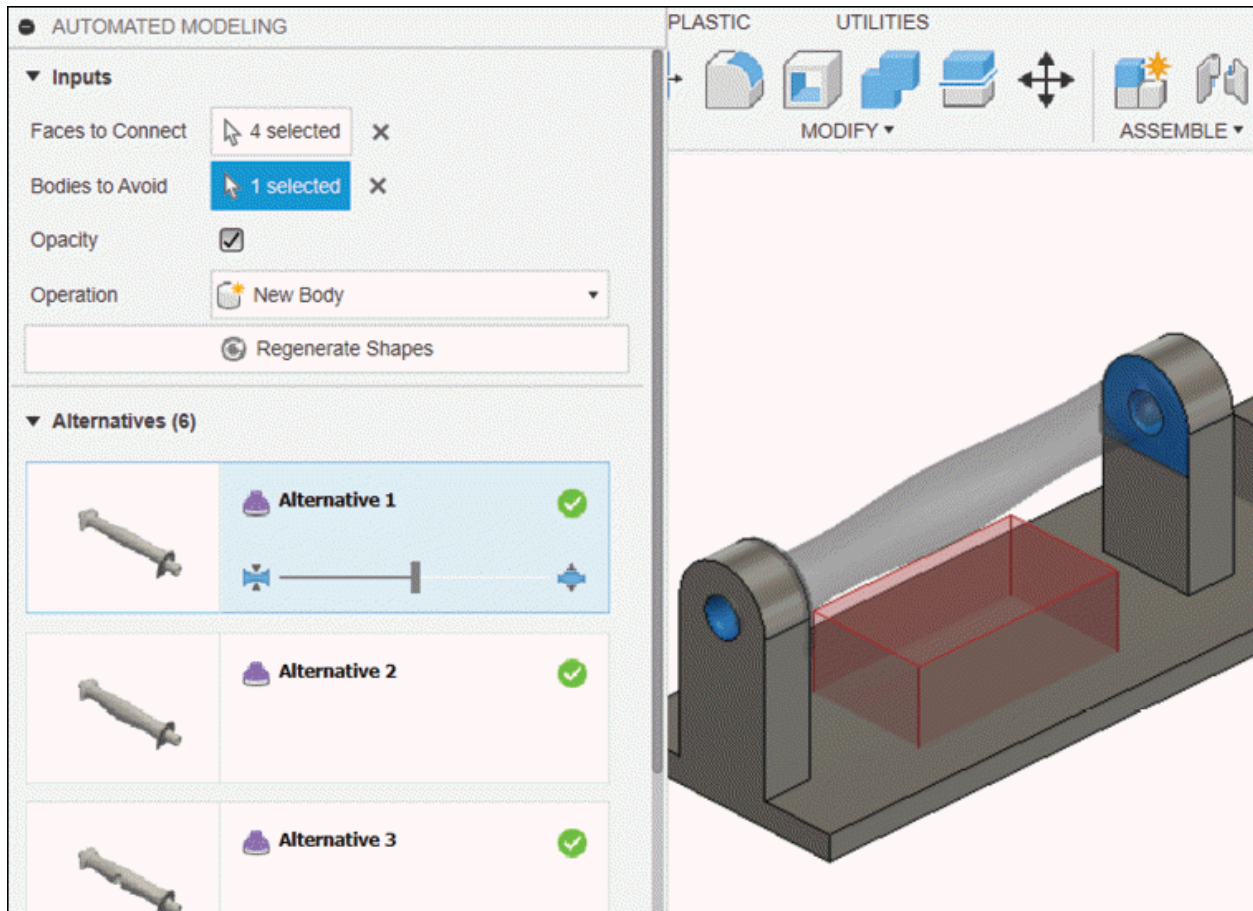


Figure-68. Preview of model alternatives

- Select desired alternative and click on the **OK** button to create the model.

PRACTICAL 1

Assemble the parts of handle as shown in [Figure-69](#). The exploded view of assembly is displayed as shown in [Figure-70](#).

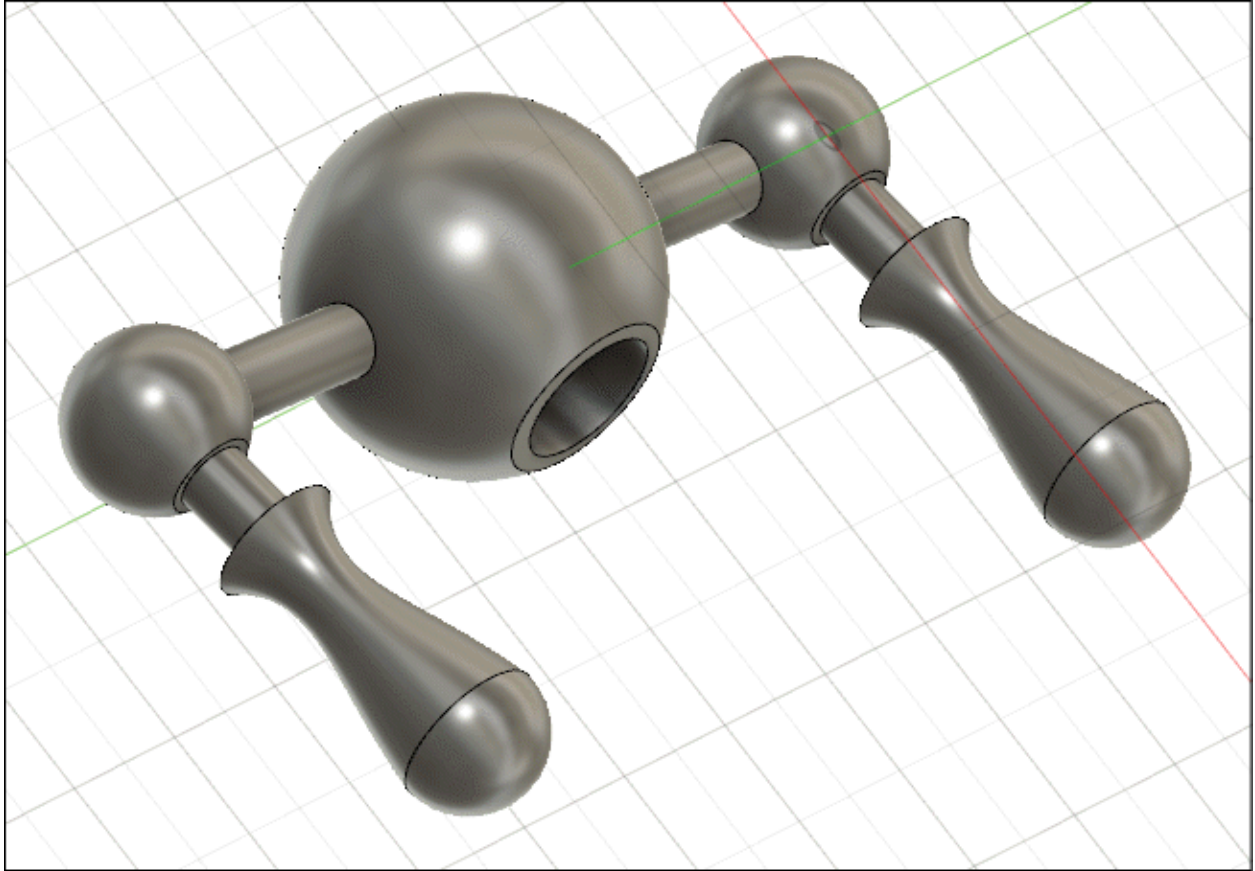


Figure-69. Assembled handle

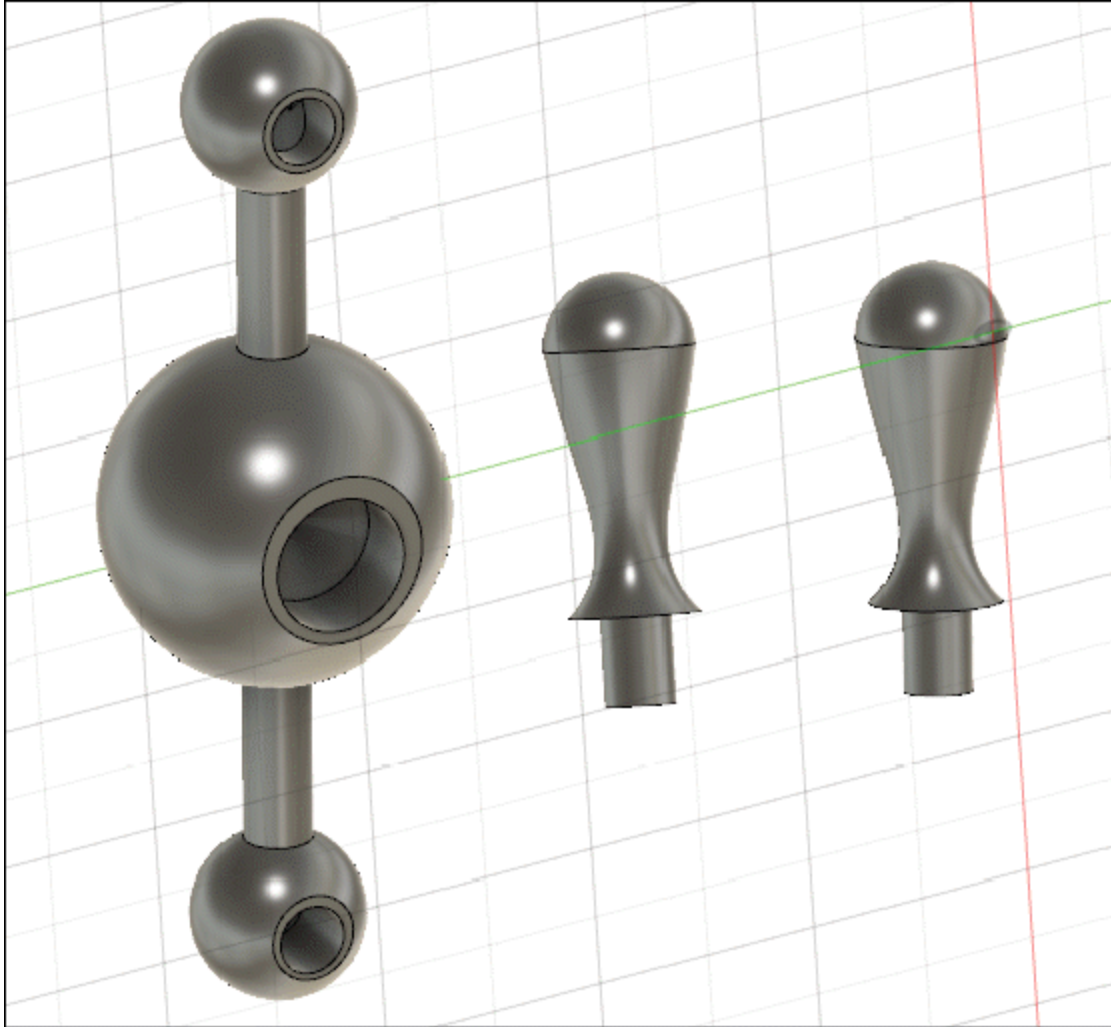


Figure-70. Exploded view of handle

All the part files can be downloaded from resource kit of Fusion 360 software, which is available at **WWW.CADCAMCAEWORKS.COM**.

The steps to assemble these parts are given next.

Adding file into Fusion 360

- After downloading the resource kit from **CADCAMCAEWORKS.COM**, open the **Chapter 7** folder.
- Double-click on **Handle v0.f3d** file. The file will open in

Autodesk Fusion 360 software; refer to [Figure-71](#).

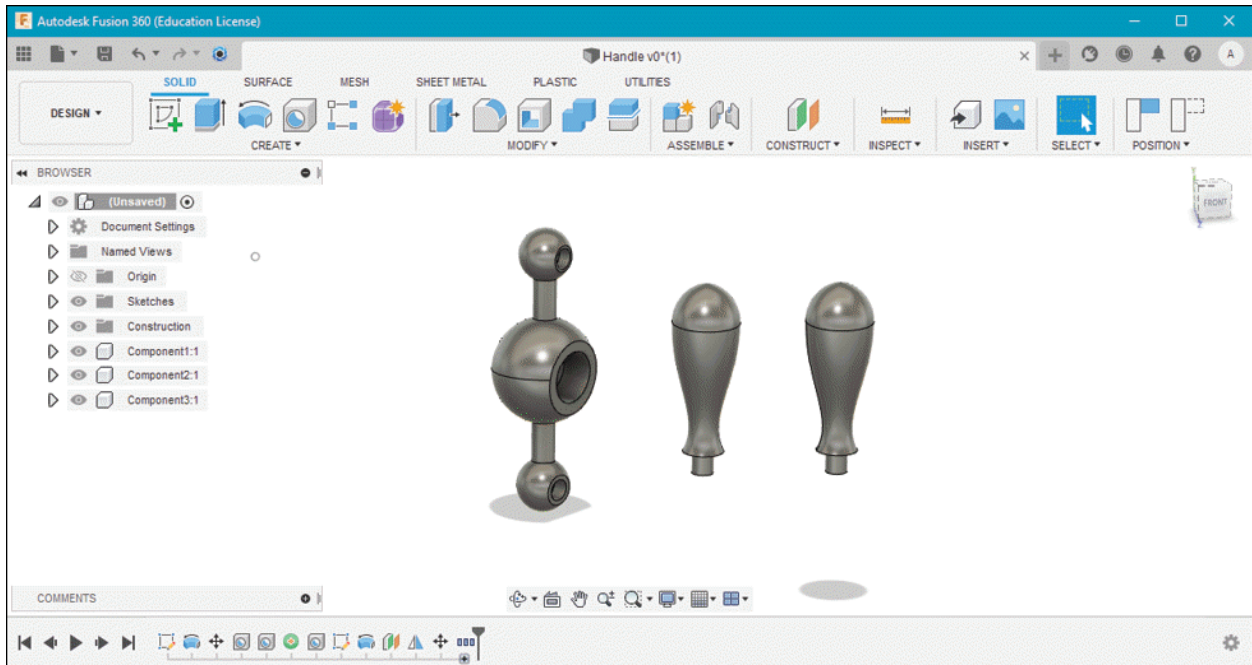


Figure-71. Fusion 360 startup window

Assembling components

- Click on **New Component** tool of **ASSEMBLE** drop-down from **Toolbar**; refer to [Figure-72](#). The **NEW COMPONENT** dialog box will be displayed; refer to [Figure-73](#).

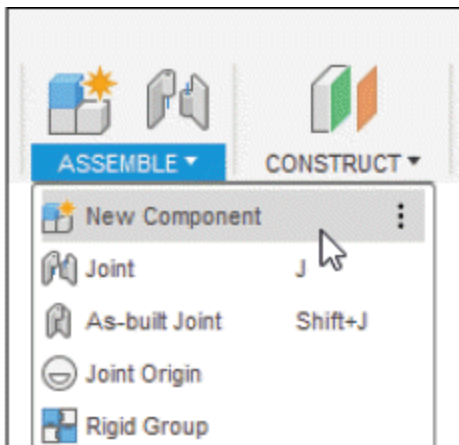


Figure-72. New Component tool

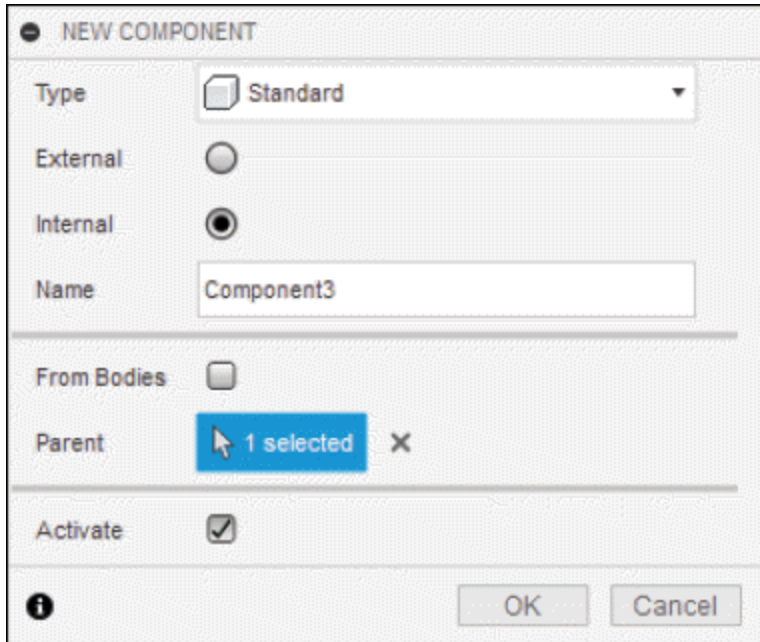


Figure-73. NEW COMPONENT dialog box

- Select the **From Bodies** check box from **NEW COMPONENT** dialog box and select all the bodies to make them component; refer to [Figure-74](#).

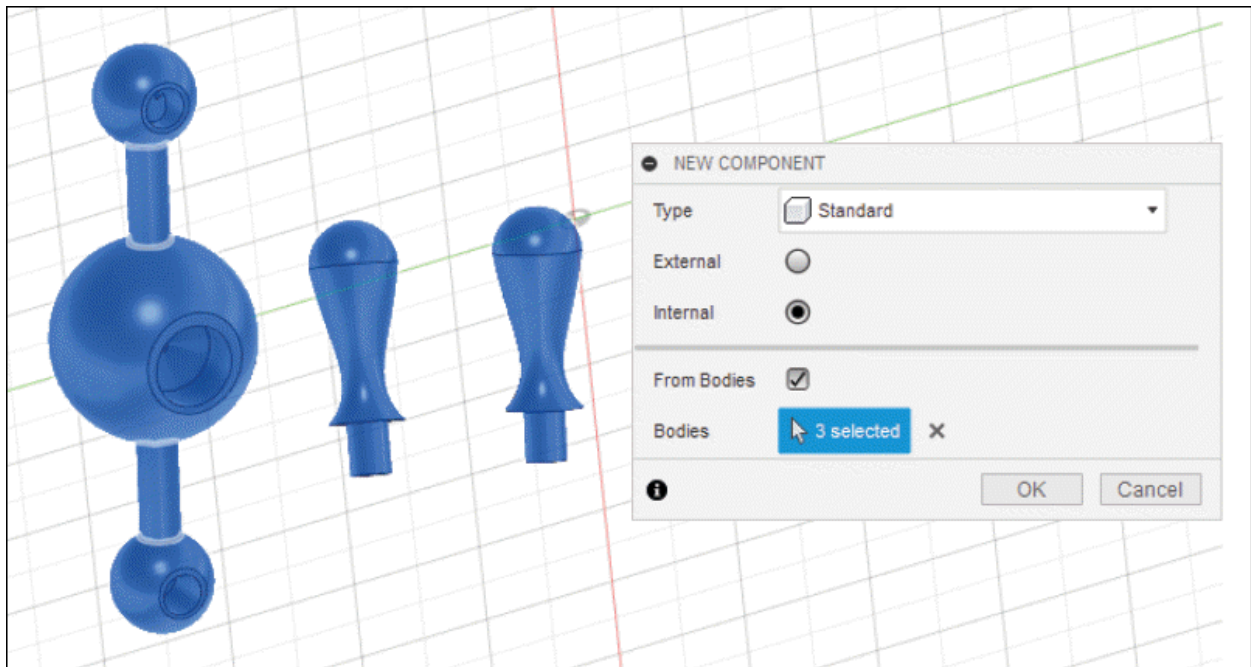


Figure-74. Selecting bodies to create components

- After selecting the bodies, click on **OK** button from **NEW COMPONENT** dialog box. The components will be created and added in **Timeline**.
- Click on **Joint** tool of **ASSEMBLE** drop-down from **Toolbar**. You can also activate the **Joint** tool by pressing **J** key from keyboard. The **JOINT** dialog box will be displayed; refer to [Figure-75](#).

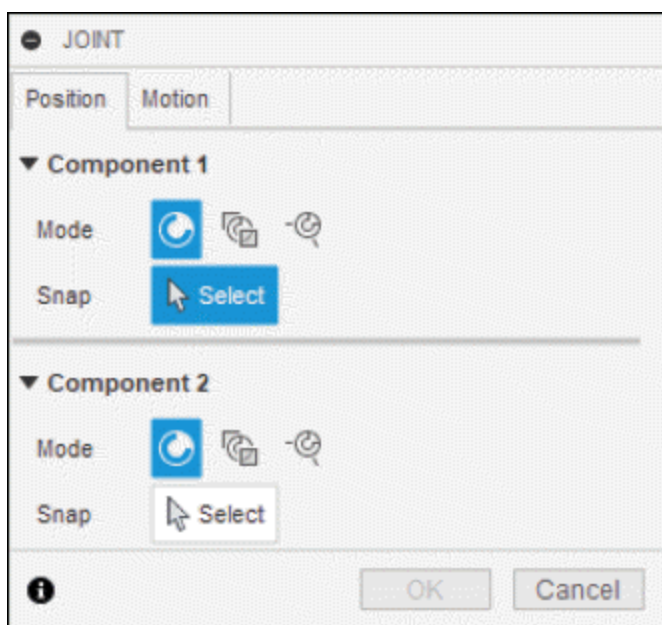


Figure-75. JOINT dialog box

- The **Select** button of **Snap** section from **Component 1** area in **Position** tab is selected by default. Click on the component to join; refer to [Figure-76](#).

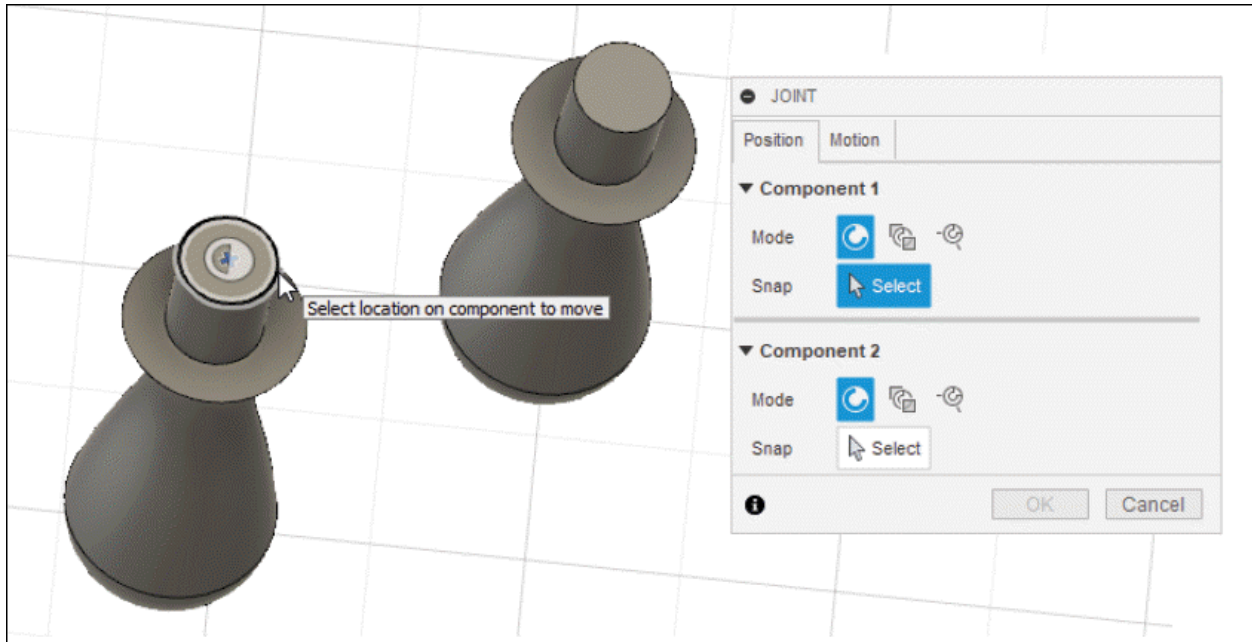


Figure-76. Selection of location on Component 1

- After selection of first component, select the second component; refer to [Figure-77](#).

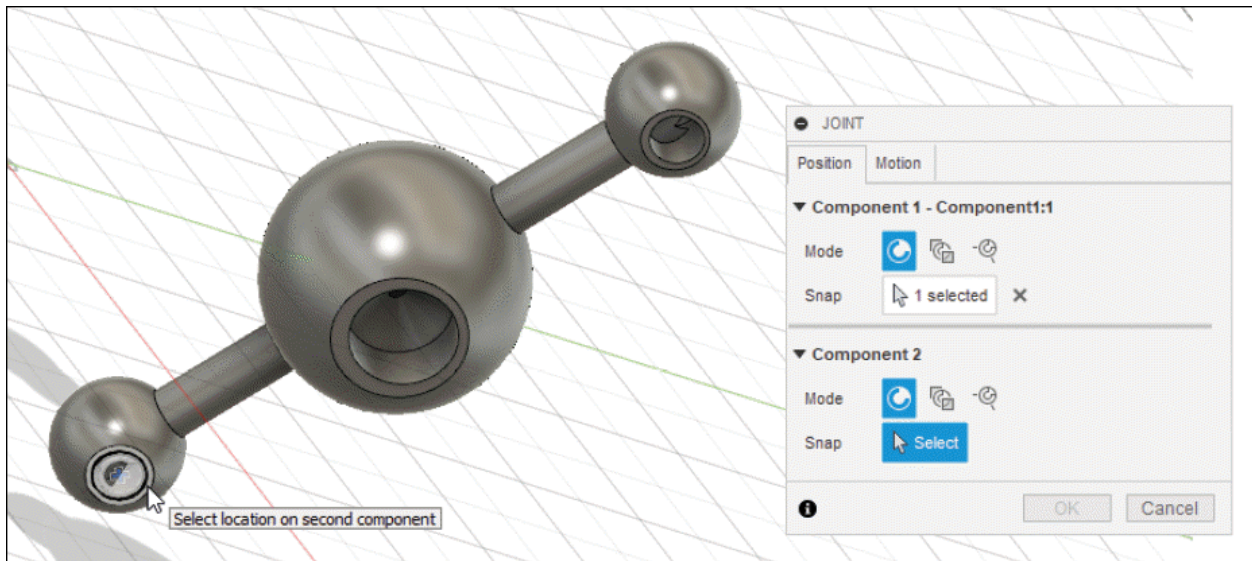


Figure-77. Selection of location on Component 2

- Select **Rigid** option from **Type** section in **Motion** tab of the dialog box to create a rigid joint between two component.

- After specifying the parameters, click on **OK** button from **JOINT** dialog box to create a rigid joint.
- Similarly, create another rigid joint between other component.
- After joining these components, the assembly will be displayed as shown in [Figure-78](#).

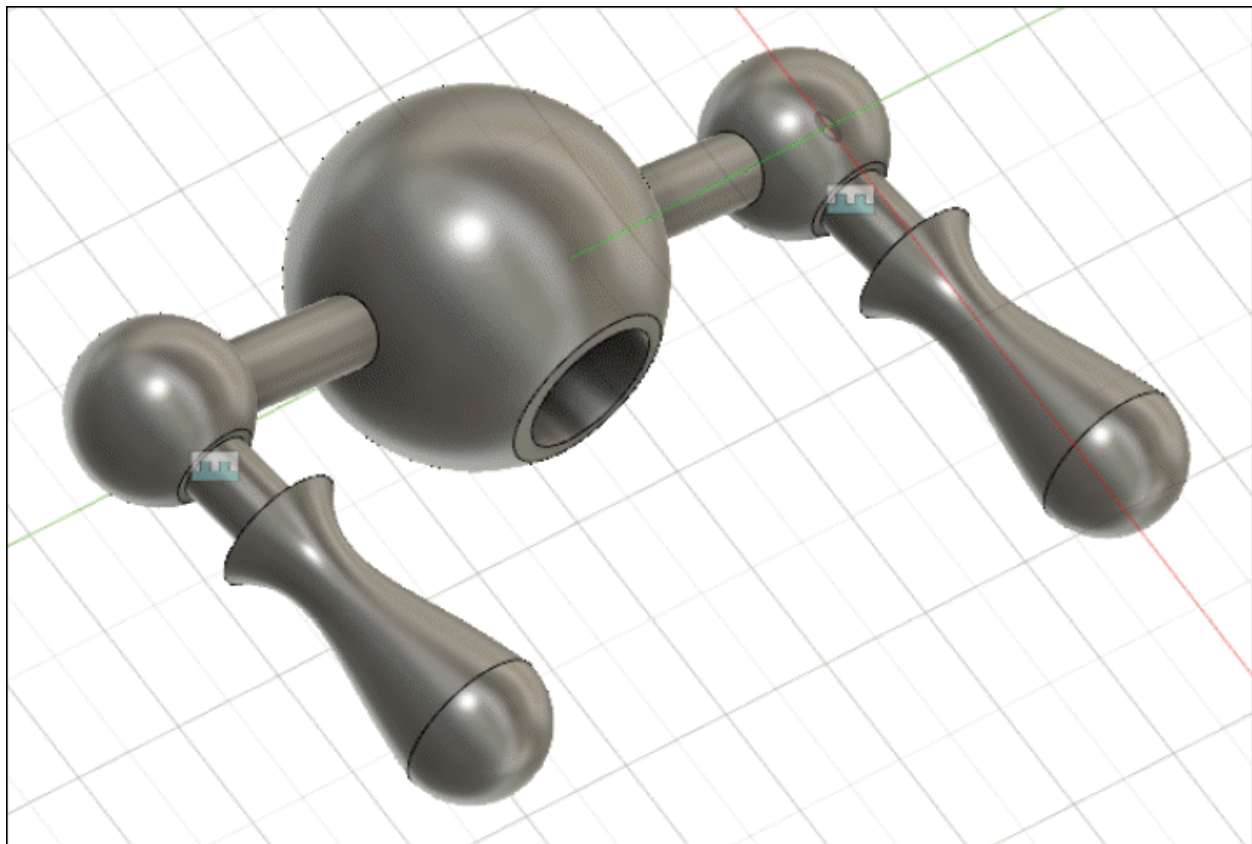


Figure-78. Final assembly of component

PRACTICAL 2

Assemble the parts of **F1 brake balance shifting mechanism** as shown in [Figure-79](#). The exploded view of assembly is displayed as shown in [Figure-80](#).

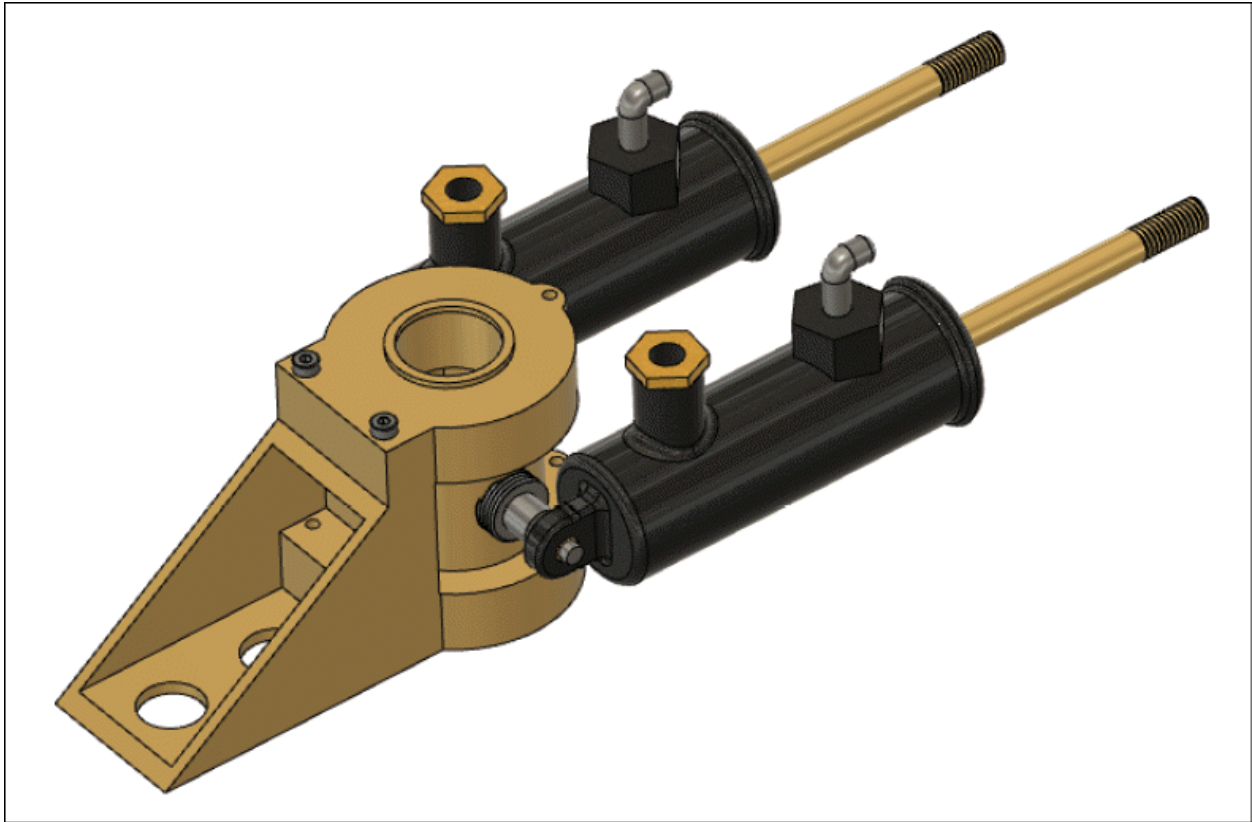


Figure-79. Brake balance shifting mechanism assembly

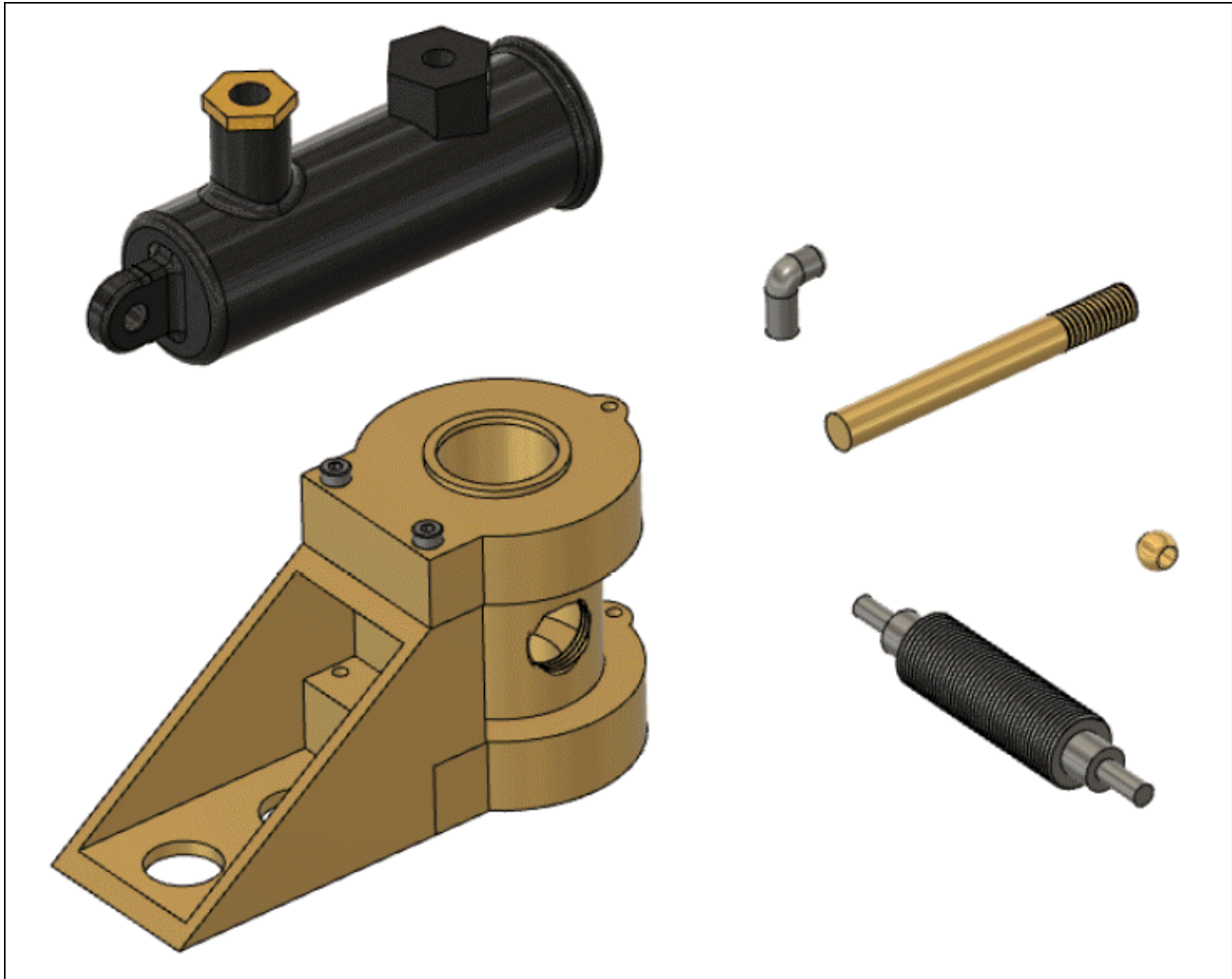


Figure-80. Exploded view of assembly

The steps to assemble these parts are given next.

Adding file into Fusion 360

- After downloading the resource kit from **CADCAMCAEWORKS.COM**, open the **Chapter 7** folder.
- Double-click on **F1 brake balance shifting mechanism.f3d** file. The file will open in **Autodesk Fusion 360** software; refer to [Figure-81](#).

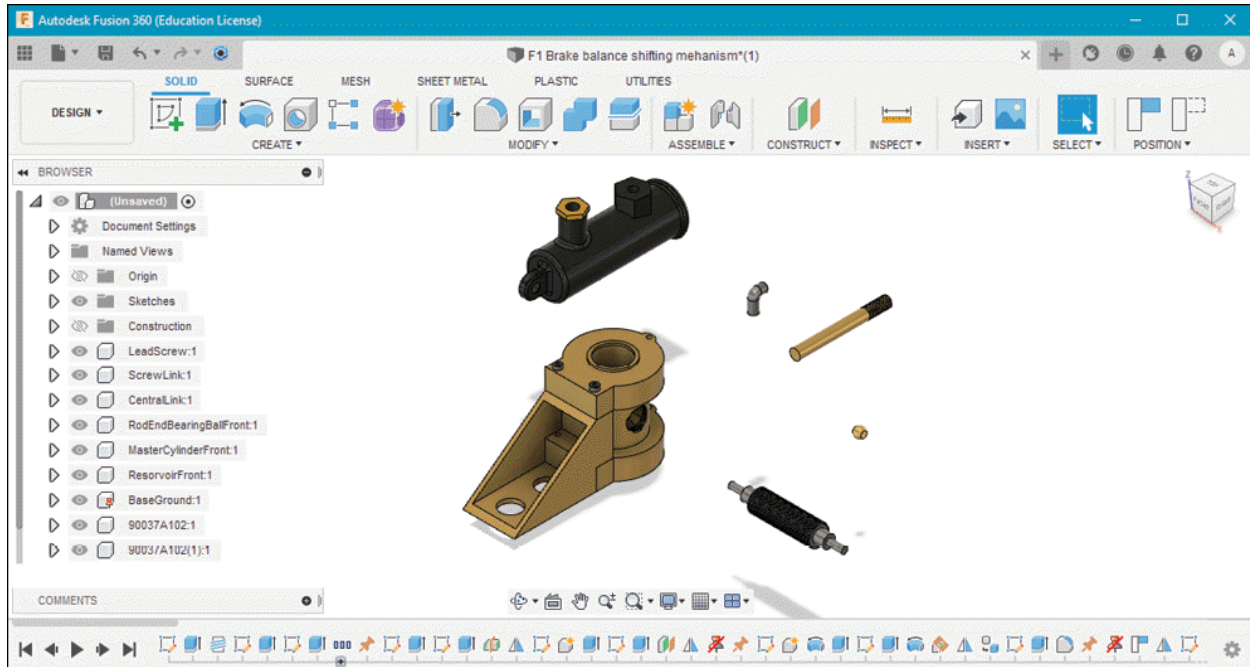


Figure-81. Fusion 360 startup window

Creating New Components

- Select **New Component** tool from **ASSEMBLE** drop-down. The **NEW COMPONENT** dialog box will be displayed.
- Select **From Bodies** check box in the dialog box. You are asked to select the bodies.
- The **Select** button of **Bodies** section is active by default. Select the bodies which you want to convert into components; refer to [Figure-82](#).

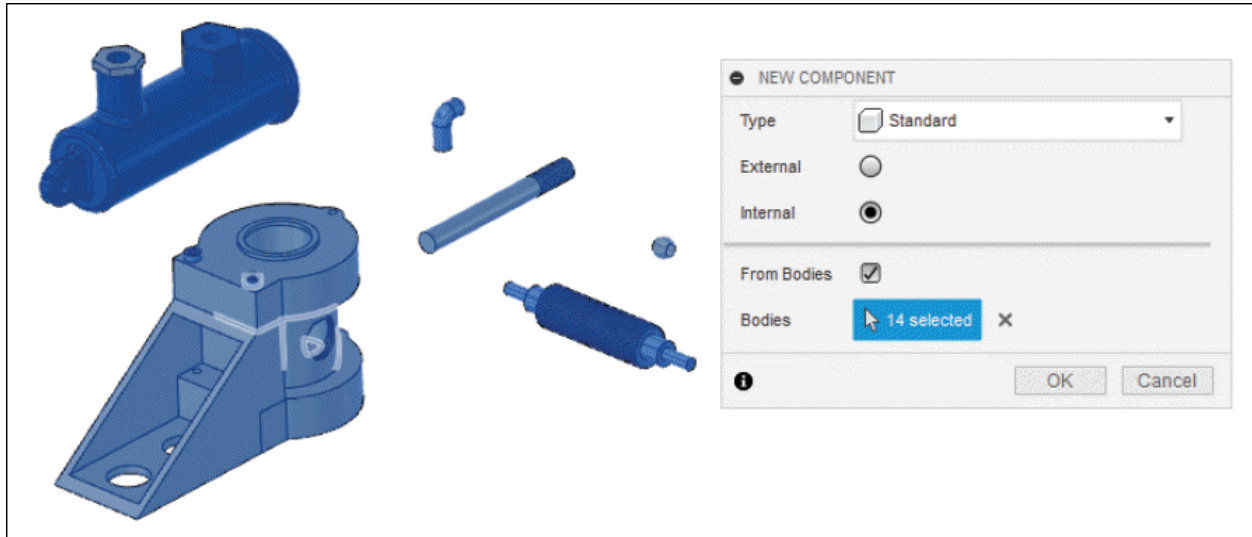


Figure-82. Selecting the bodies to create component

- Click on **OK** button from the dialog box to complete the process.

Assembling the Components

- Select **Joint** tool from **ASSEMBLE** drop-down. The **JOINT** dialog box will be displayed.
- Select **Between Two Faces** option from **Mode** section of **Component 1** area in **Position** tab of the dialog box. You are asked to select the first face.
- The **Select** button of **Plane 1** section is active by default. Select first face of the component as shown in [Figure-83](#). You are asked to select the second face.

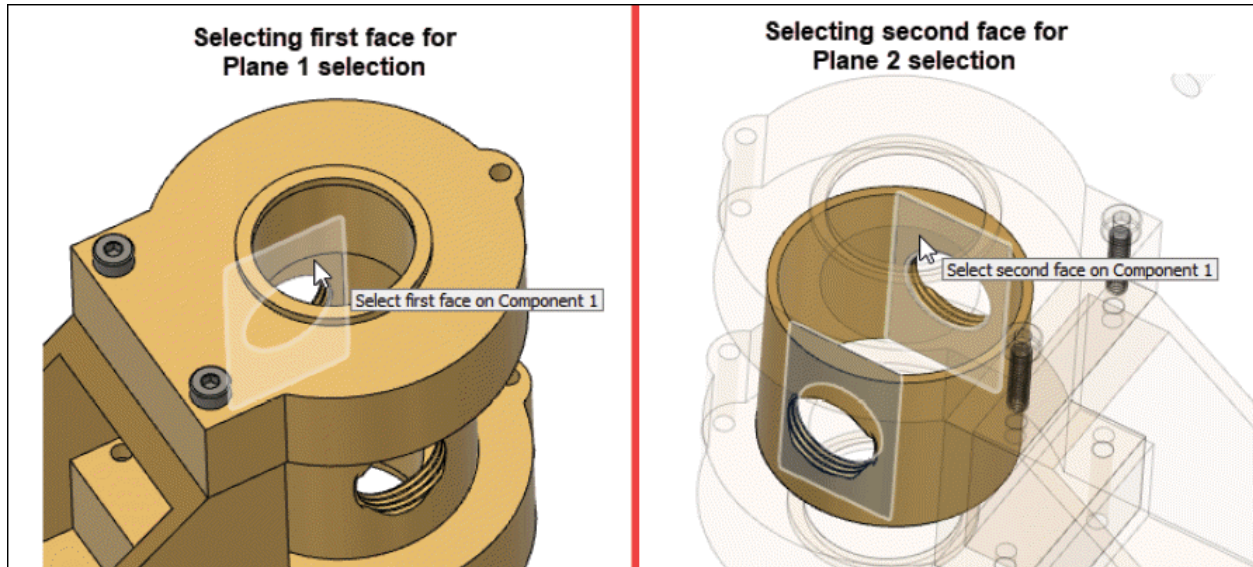


Figure-83. Selecting faces for Component 1

- The **Select** button of **Plane 2** section will become active. Select second face of component as shown in [Figure-83](#). You are asked to select a snap point.
- The **Select** button of **Snap** section will become active. Select the snap point on the component as shown in [Figure-84](#).

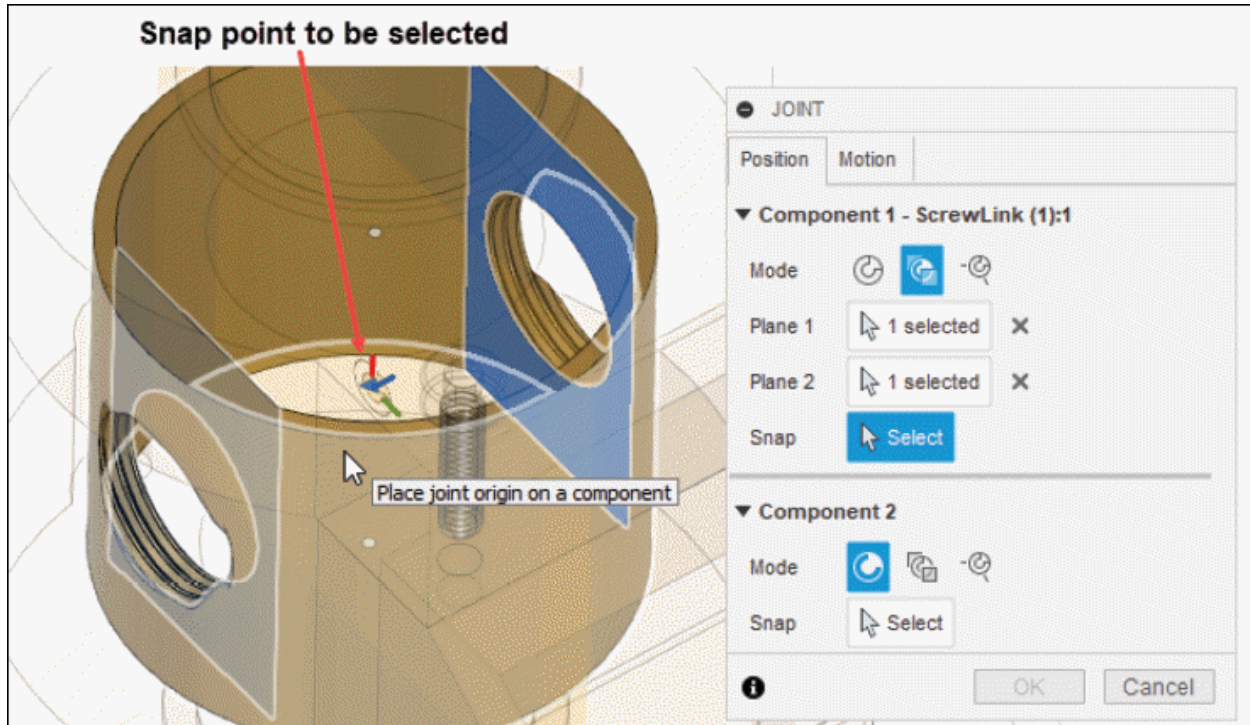


Figure-84. Selecting snap point for Component 1

- Similarly, select **Between Two Faces** option from **Mode** section of **Component 2** area. You are asked to select first and second face of the component.
- Select first face of the component for **Plane 1** selection and select second face of the component for **Plane 2** selection as shown in the [Figure-85](#). You are asked to select the snap point on the component.

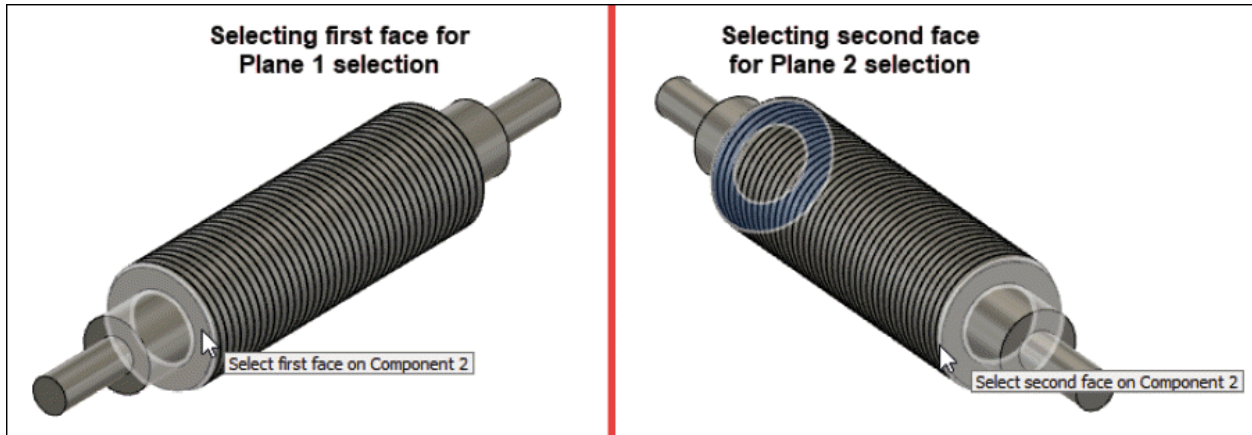


Figure-85. Selecting faces for Component 2

- The **Select** button of **Snap** section will become active. Select the snap point on the component as shown in [Figure-86](#). The components will be joined together; refer to [Figure-87](#).

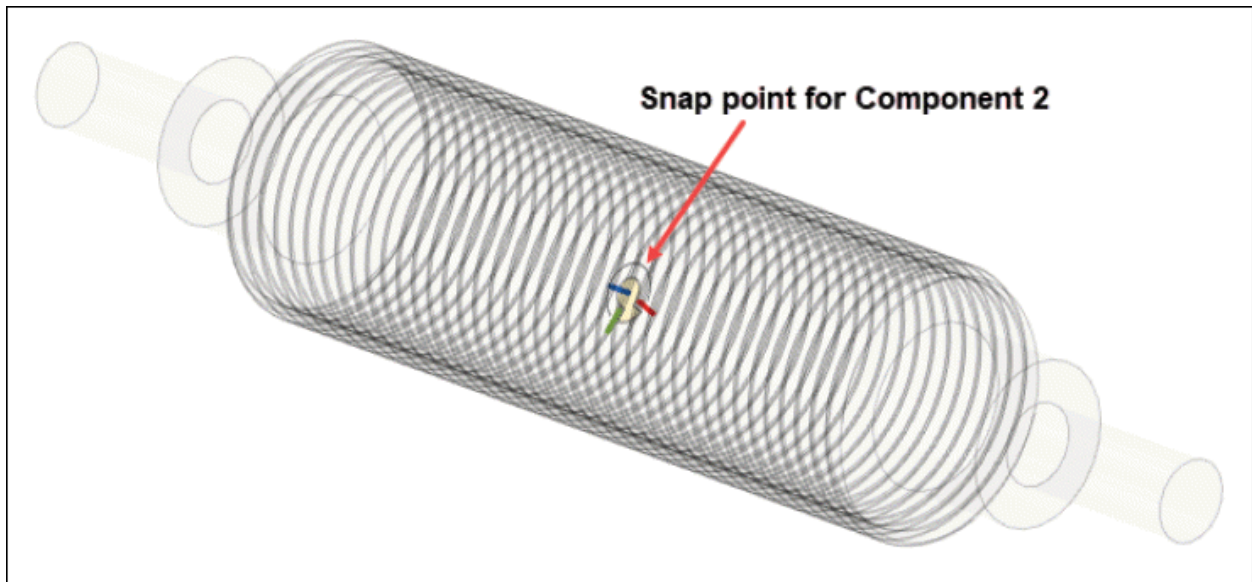


Figure-86. Selecting snap point for Component 2

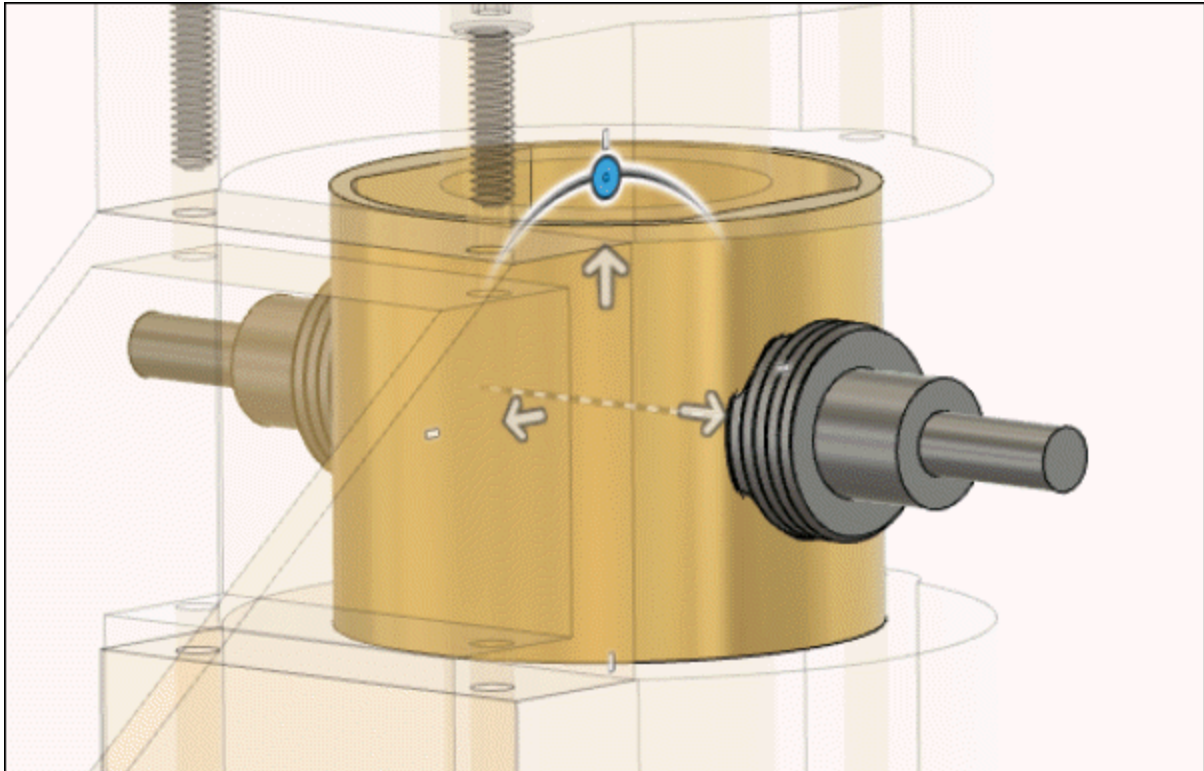


Figure-87. Components joined

- Select **Revolute** option from **Type** section in **Motion** tab of the **JOINT** dialog box.
- After specifying parameters, click on the **OK** button from the dialog box to complete the process.
- Now, select **Joint** tool from **ASSEMBLE** drop-down. The **JOINT** dialog box will be displayed. You are asked to select the snap point on the components.
- Select snap point on both Component 1 and Component 2 as shown in [Figure-88](#).

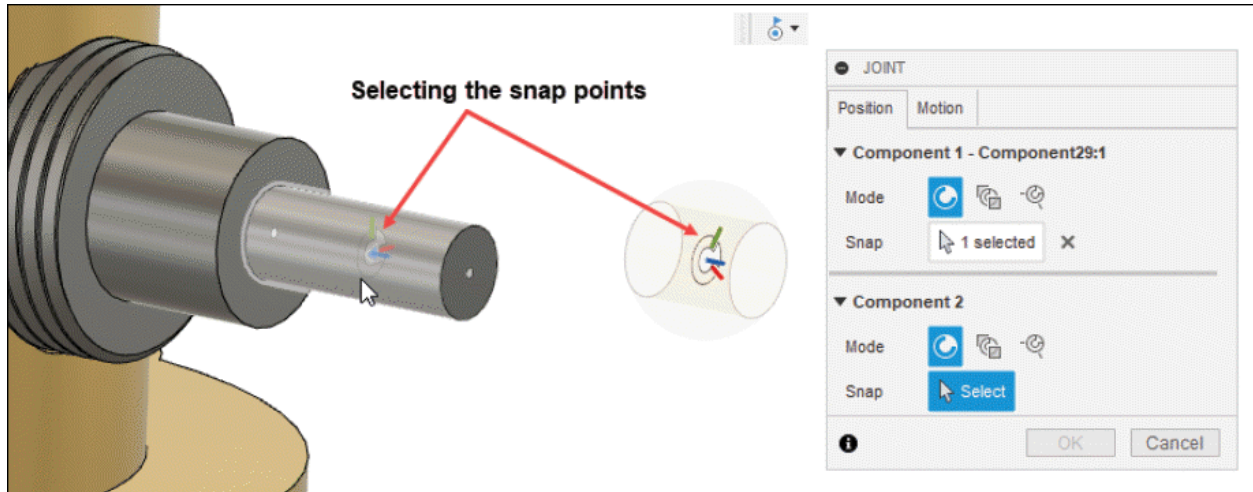


Figure-88. Selecting the snap points

- On selecting the snap point on second component, both the components will be joined; refer to [Figure-89](#).

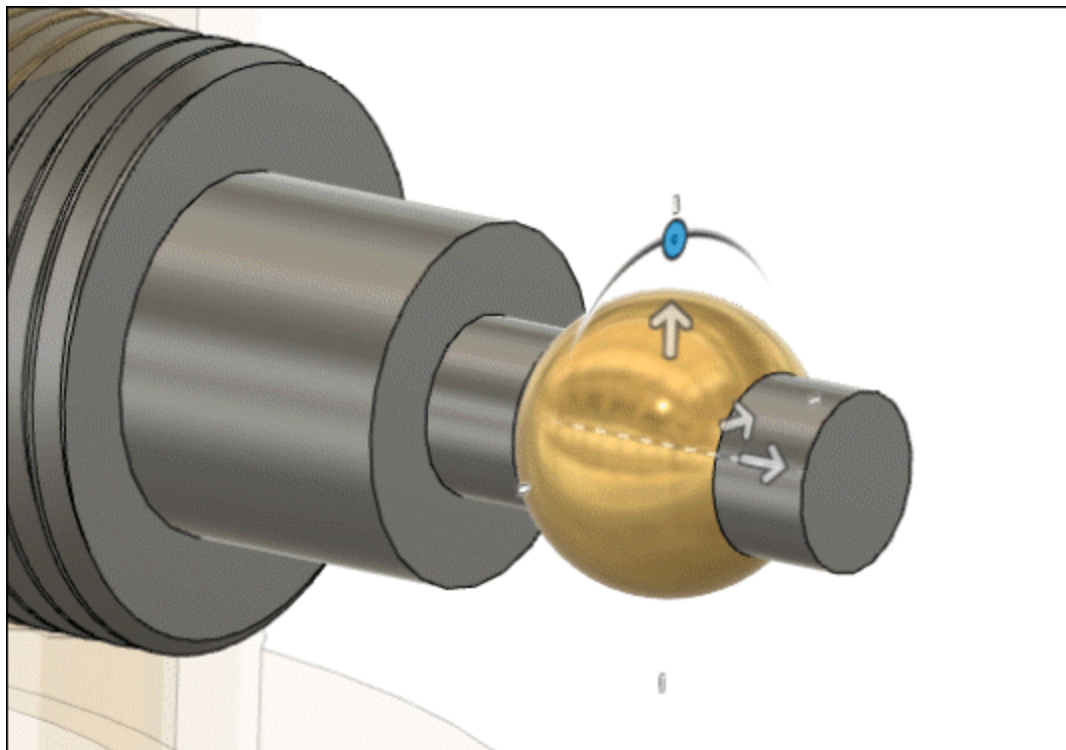


Figure-89. Components joined

- Select **Rigid** option from **Type** section in **Motion** tab of the dialog box.

- After specifying parameters, click on the **OK** button from the dialog box to complete the process.
- Now, select **Joint** tool again from **ASSEMBLE** drop-down. The **Joint** dialog box will be displayed. You are asked to select the snap points on the components.
- Select snap point on both Component 1 and Component 2 as shown in [Figure-90](#).

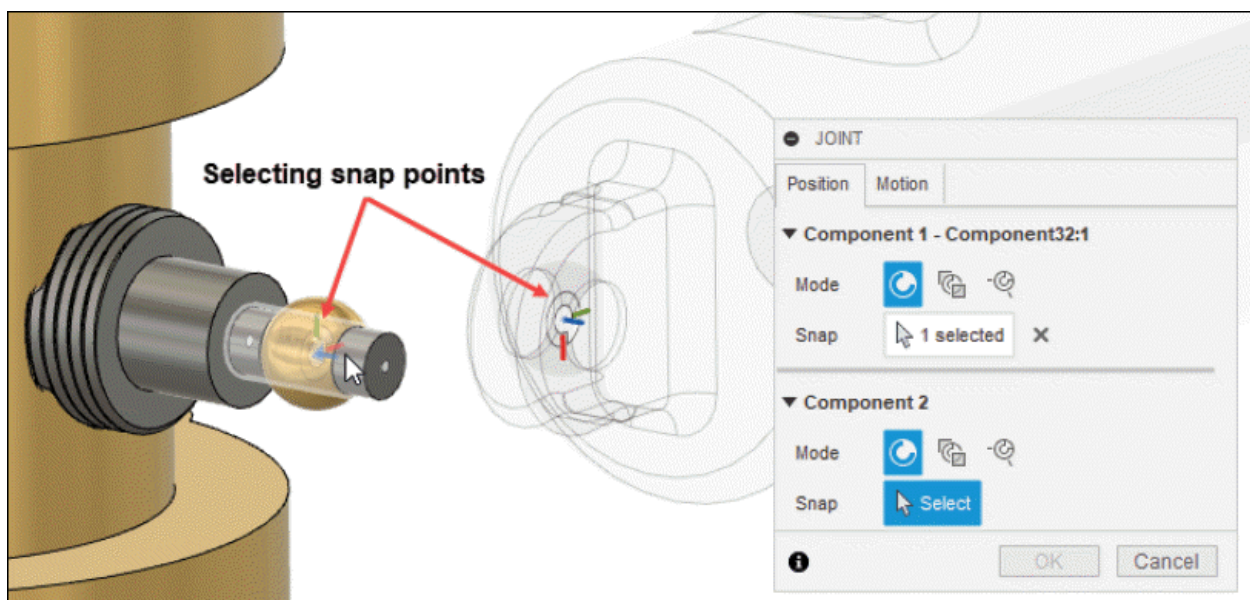


Figure-90. Selecting snap point on the components

- On selecting the snap point on second component, both the components will be joined; refer to [Figure-91](#).

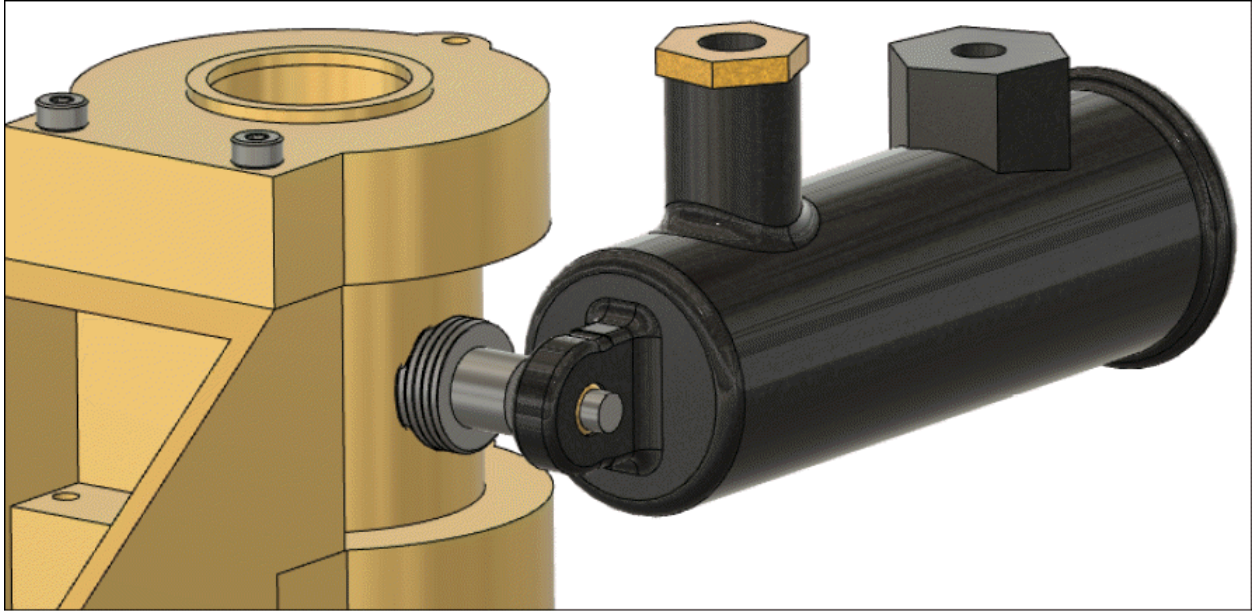


Figure-91. Components joined

- Select **Revolute** option from **Type** section in **Motion** tab of the dialog box.
- After specifying the parameters, click on **OK** button from the dialog box to complete the process.
- Now, select the **Joint** tool from **ASSEMBLE** drop-down. The **JOINT** dialog box will be displayed. You are asked to select the snap point on the components.
- Select snap point on both Component 1 and Component 2 as shown in [Figure-92](#).

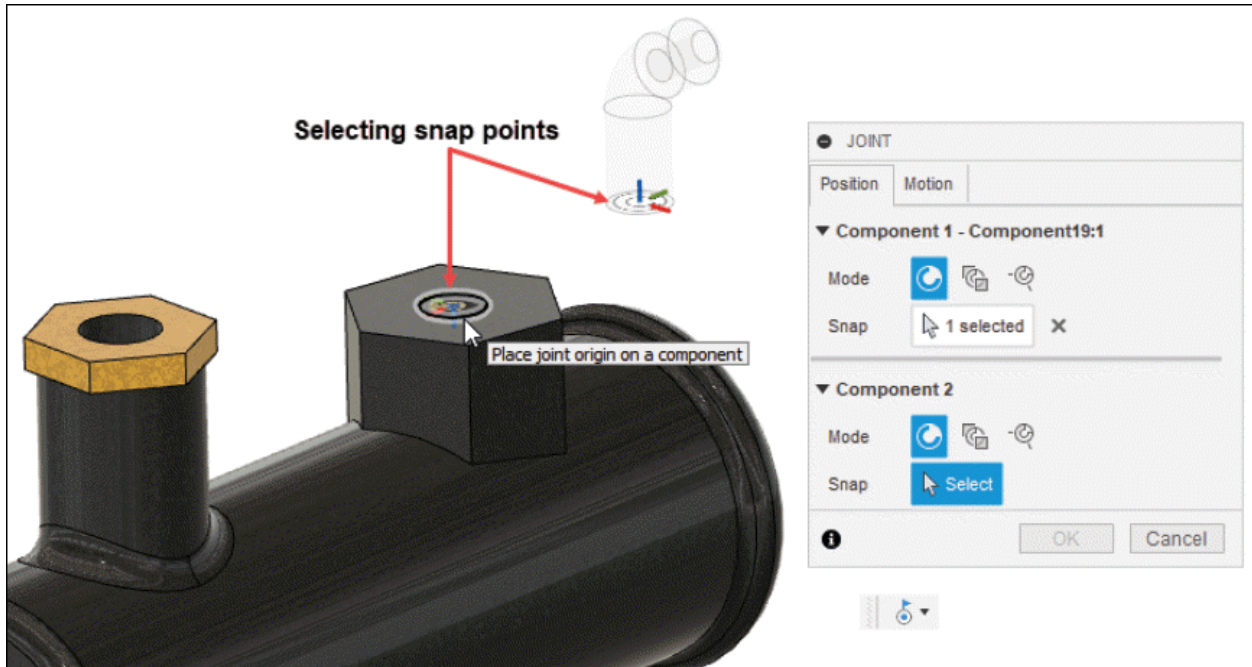


Figure-92. Selecting snap point on the components

- On selecting the snap point on second component, both the components will be joined; refer to [Figure-93](#).

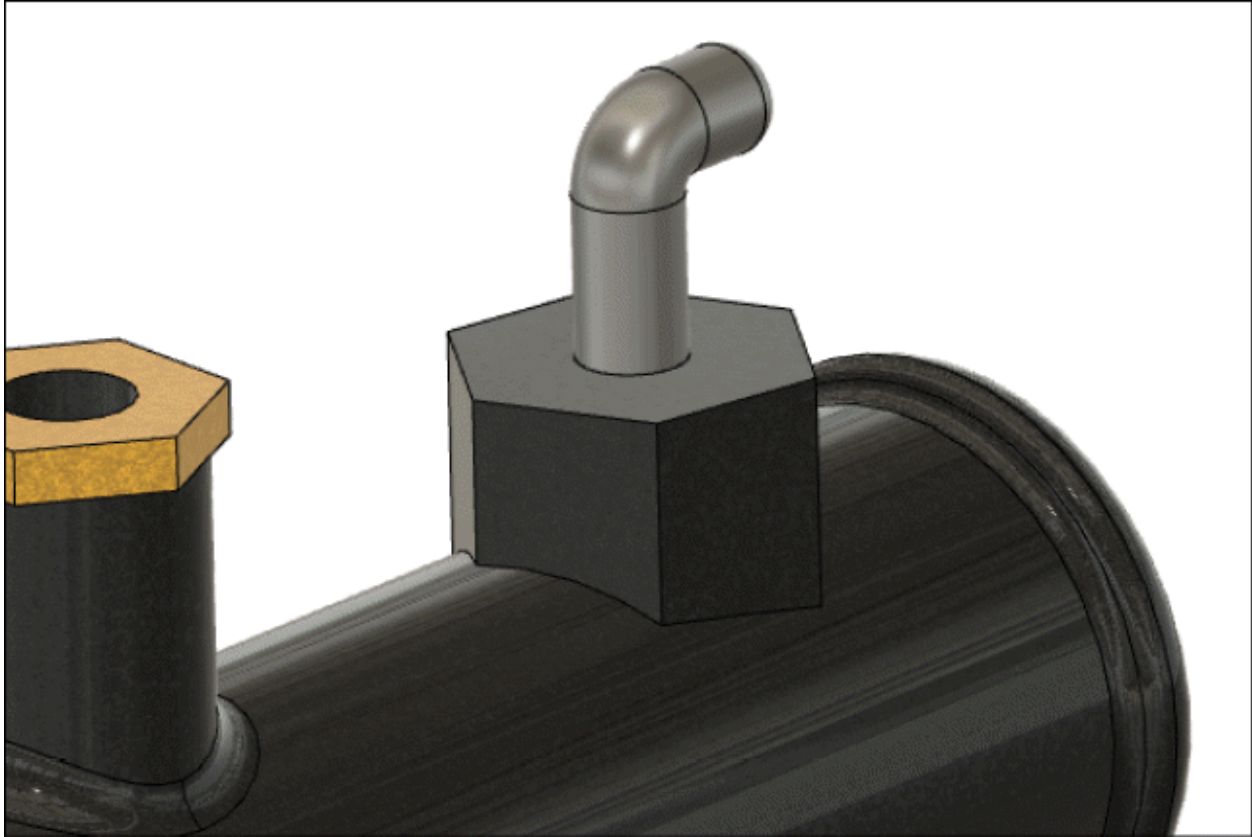


Figure-93. Components joined

- Select **Rigid** option from **Type** section in **Motion** tab of the dialog box.
- After specifying the parameters, click on **OK** button from the dialog box to complete the process.
- Click on **Joint** tool again from **ASSEMBLE** drop-down. The **JOINT** dialog box will be displayed. You are asked to select the snap point on the components.
- Select snap point on both Component 1 and Component 2 as shown in [Figure-94](#).

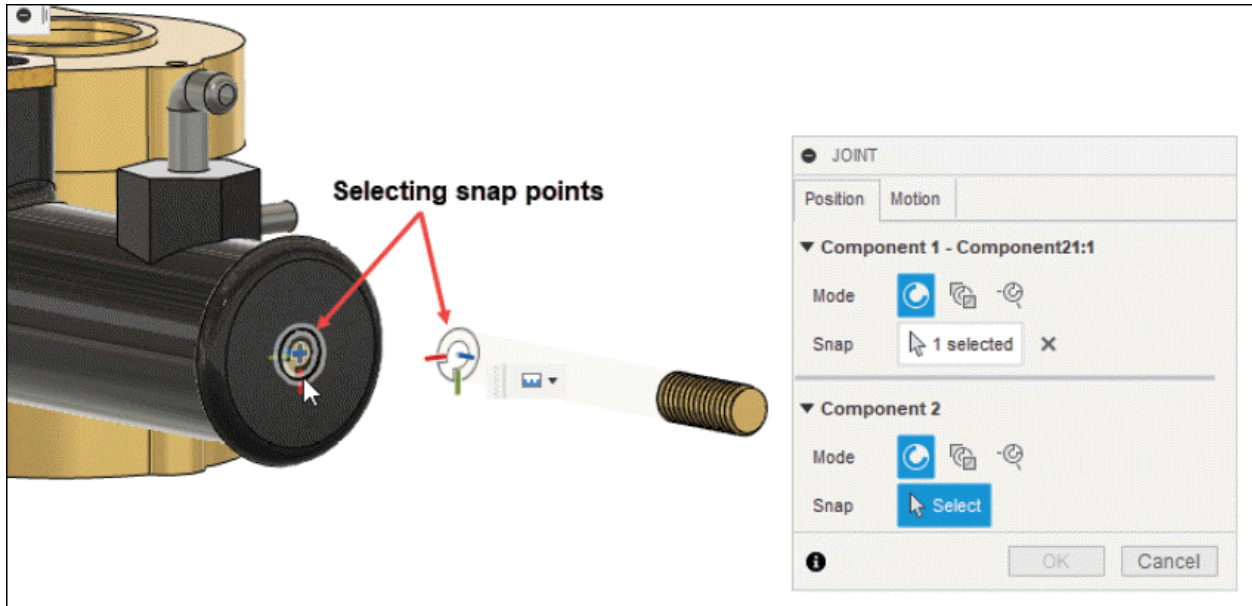


Figure-94. Selecting snap point on components

- On selecting snap point on second component, both the components will be joined; refer to [Figure-95](#).

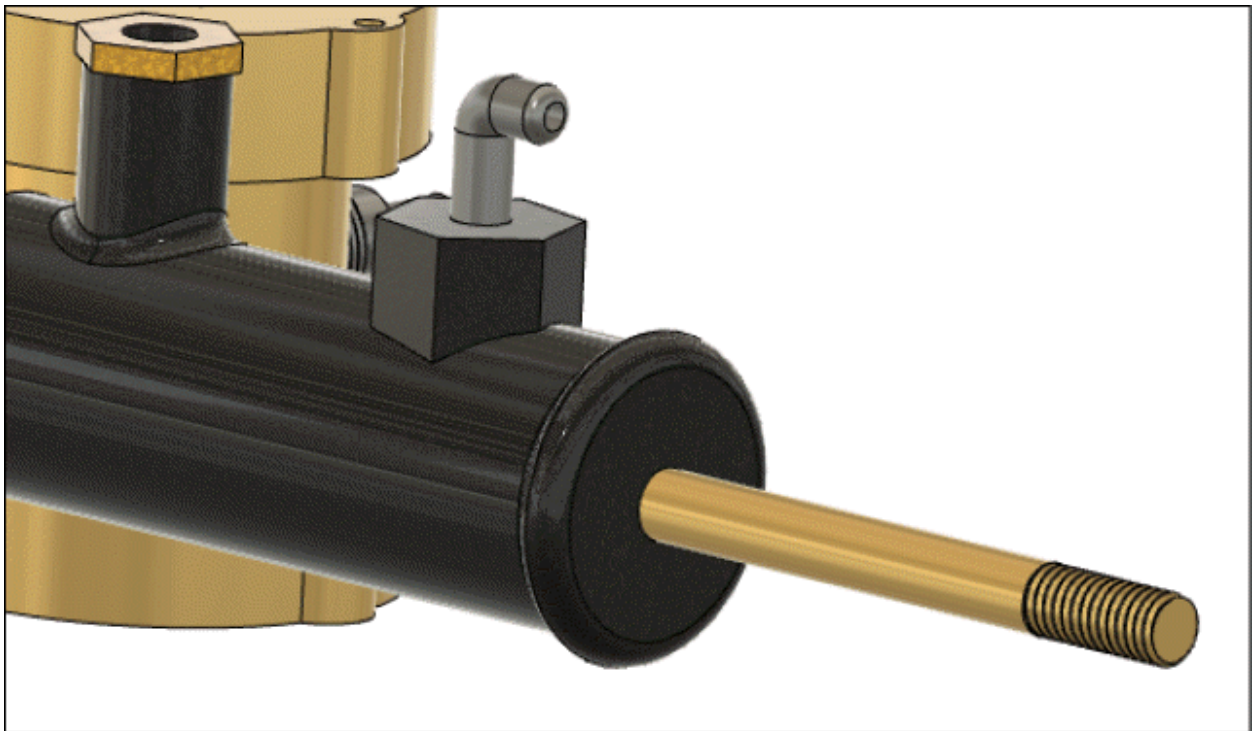


Figure-95. Components joined

- Select **Slider** option from **Type** section in **Motion** tab of the dialog box.
- After specifying the parameters, click on **OK** button from the dialog box to complete the process.

Copying the Components

- Select the components to be copied while holding the **CTRL** key from **BROWSER** as shown in [Figure-96](#). Right-click on any of the selected components, a shortcut menu will be displayed. Select **Copy** option from the shortcut menu.

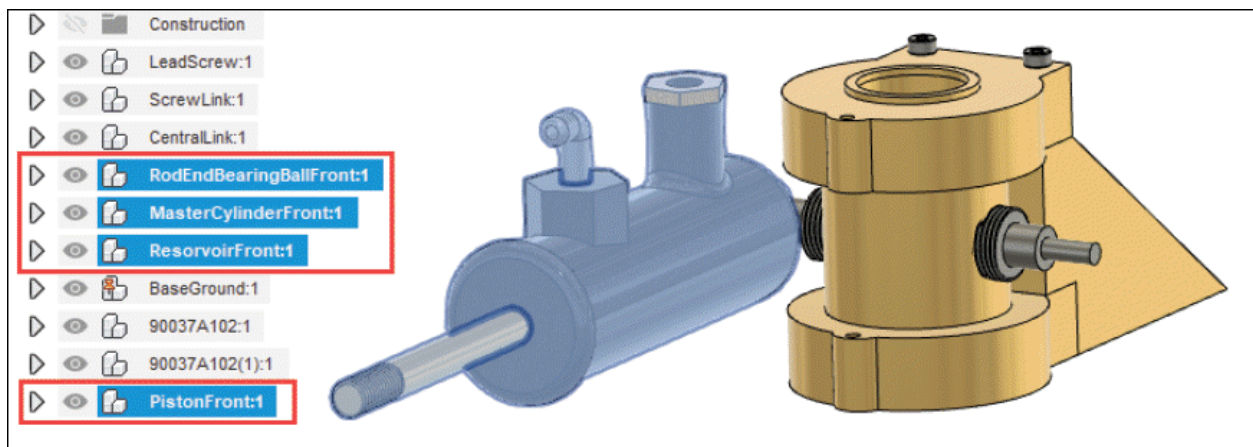


Figure-96. Selecting the components to copy

- Right-click in the empty area of the screen, a **Marking Menu** will be displayed. Select **Paste** option from the **Marking Menu**. The **MOVE/COPY** dialog box will be displayed along with the drag handle; refer to [Figure-97](#).

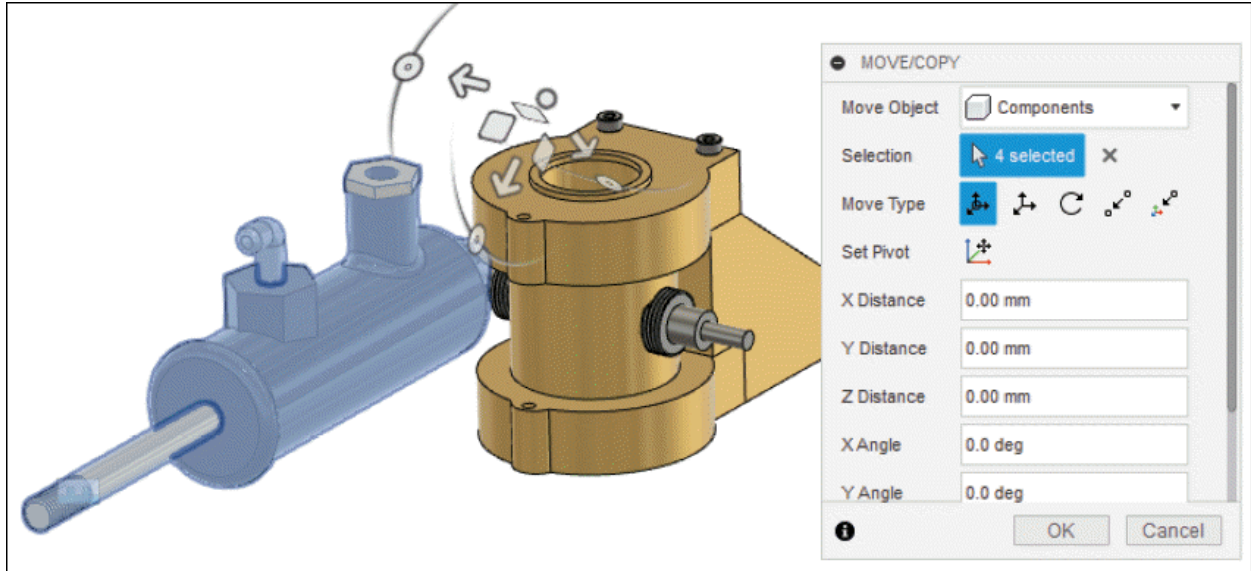


Figure-97. MOVE COPY dialog box along with drag handle

- Drag the copied components with the help of drag handle. The copied components will be displayed; refer to [Figure-98](#).

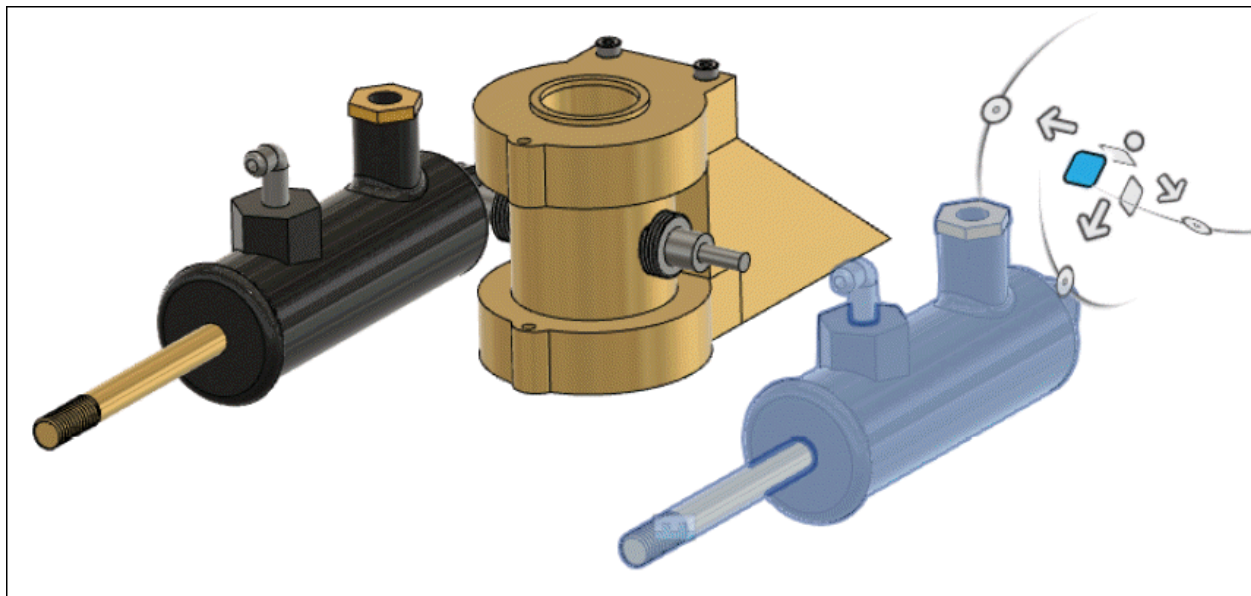


Figure-98. Components copied

- Click on **OK** button from **MOVE/COPY** dialog box to complete the process.

Assembling the Copied Components

- Select **Joint** tool from **ASSEMBLE** drop-down. The **JOINT** dialog box will be displayed and you will be asked to select snap point on components.
- Select snap point on both the Component 1 and Component 2 as shown in [Figure-99](#).

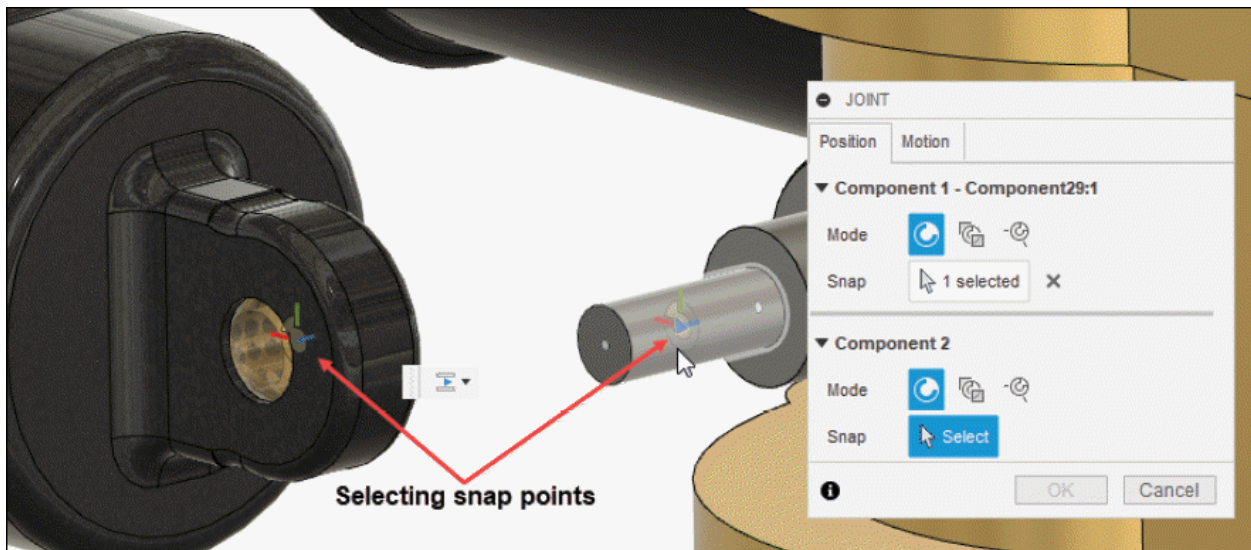


Figure-99. Selecting snap point on components

- On selecting snap point of second component, both the components will be joined.
- Select **Revolute** option from **Type** section in **Motion** tab of the dialog box.
- After specifying the parameters, click on **OK** button from the dialog box to complete the process. The final assembly of components will be displayed; refer to [Figure-100](#).

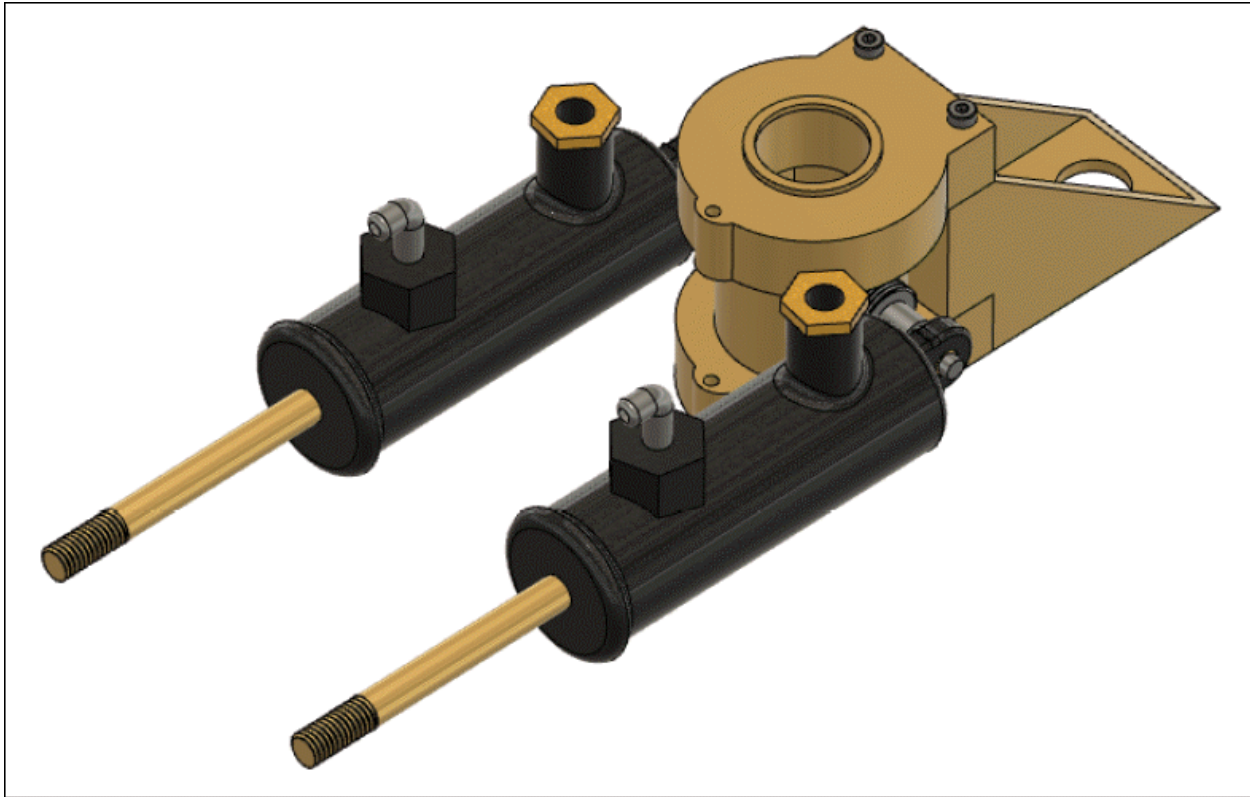


Figure-100. Final assembly of components

SELF ASSESSMENT

Q1. The tool is used to create a standard component or sheet metal component from the selected bodies.

Q2. Which of the following tool is used to define motion type of a component in an assembly?

- a. New Component
- b. Joint
- c. Rigid Group
- d. Drive Joints

Q3. Which of the following motion type is not available to apply joints in assembly of Autodesk Fusion 360?

- a. Slider
- b. Cylindrical

c. Pin-slot d. Symmetry

Q4. Which of the following tools is used to combine multiple components to act as a single component?

a. New Component b. Joint

c. Rigid Group d. Drive Joints

Q5. Which of the following tools is used to define relation between motion of two components of assembly?

a. Motion Link b. Joint

c. Rigid Group d. Drive Joints

PRACTICE 1

Assemble the parts of motor assembly as shown in [Figure-101](#) and [Figure-102](#). The exploded view of assembly is shown in [Figure-103](#). Note that you can open this practice file from the resource kit downloaded for this book.

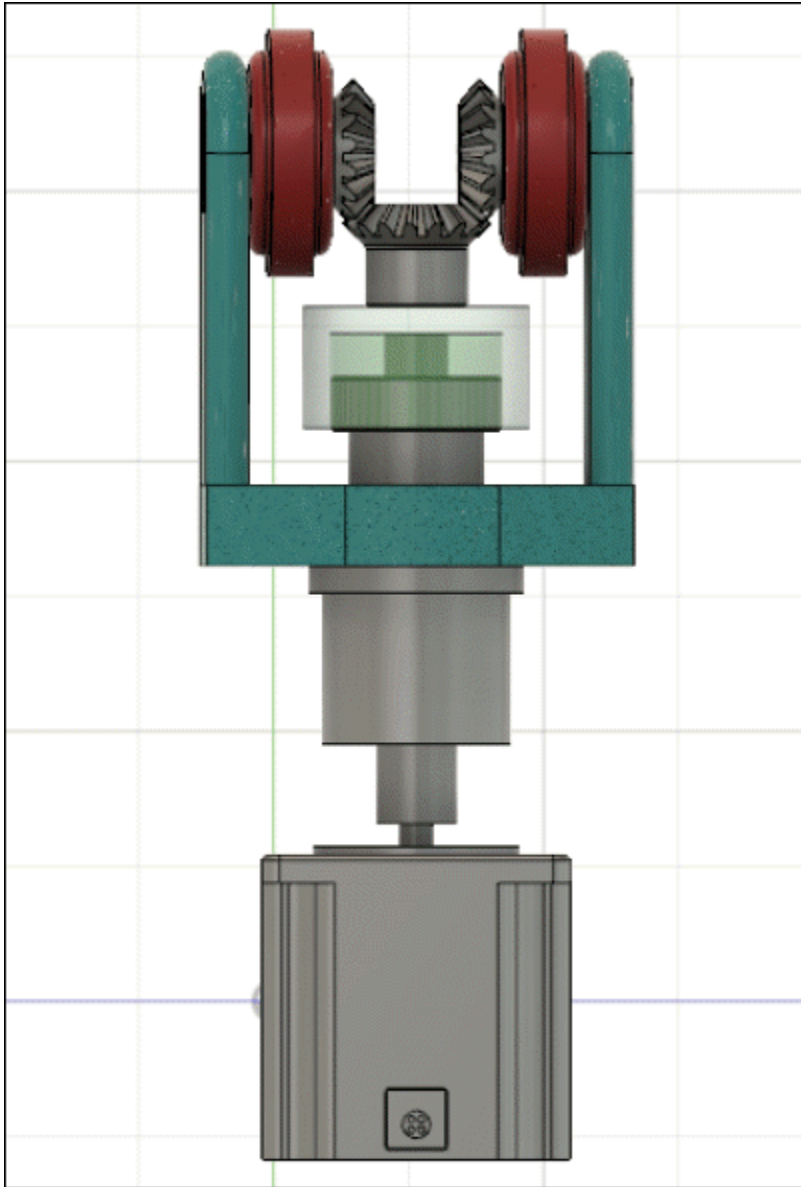


Figure-101. Assembly view 1

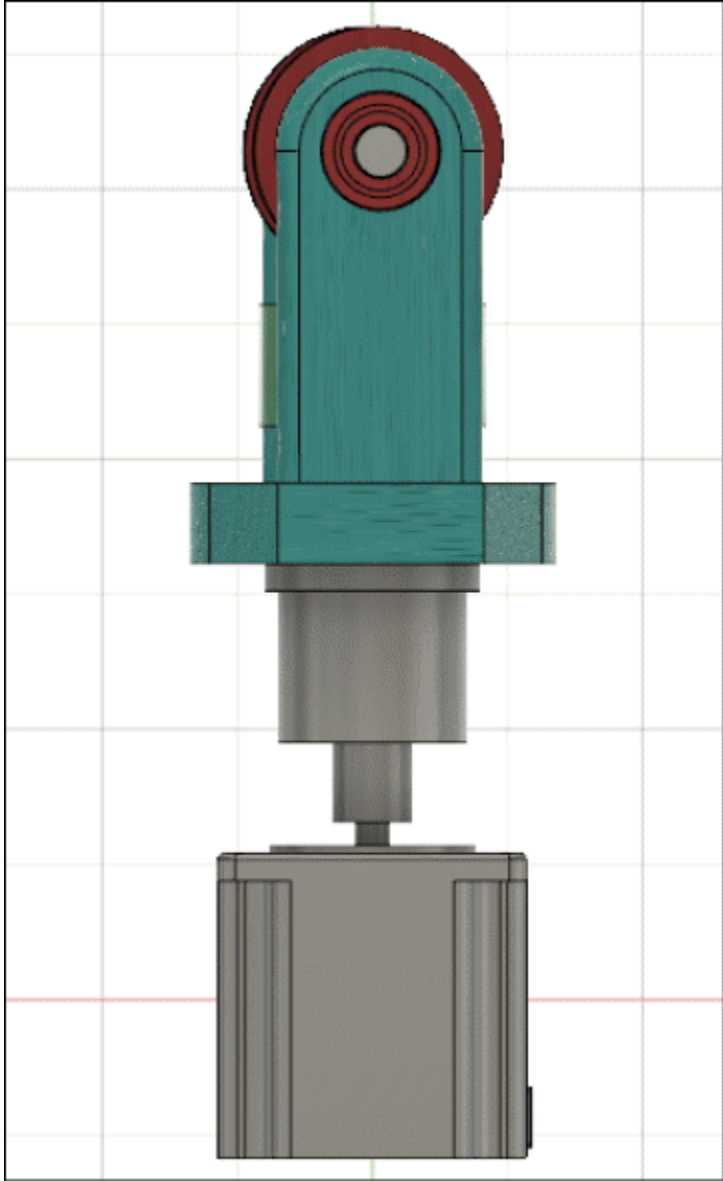


Figure-102. Assembly view 2

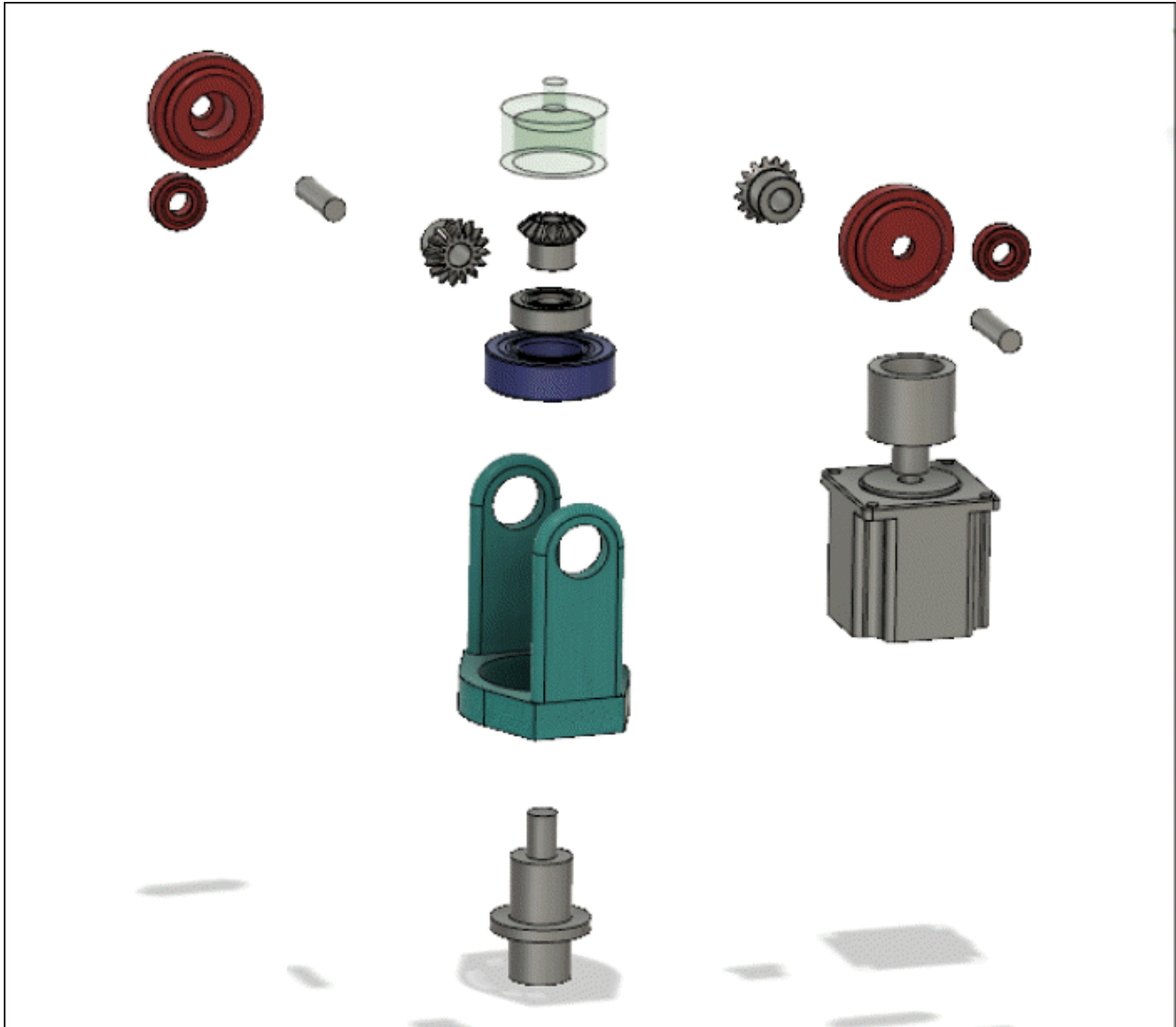


Figure-103. Exploded view of assembly

PRACTICE 2

Assemble the parts of crank assembly as shown in [Figure-104](#). The exploded view in which you will get the model is shown in [Figure-105](#).

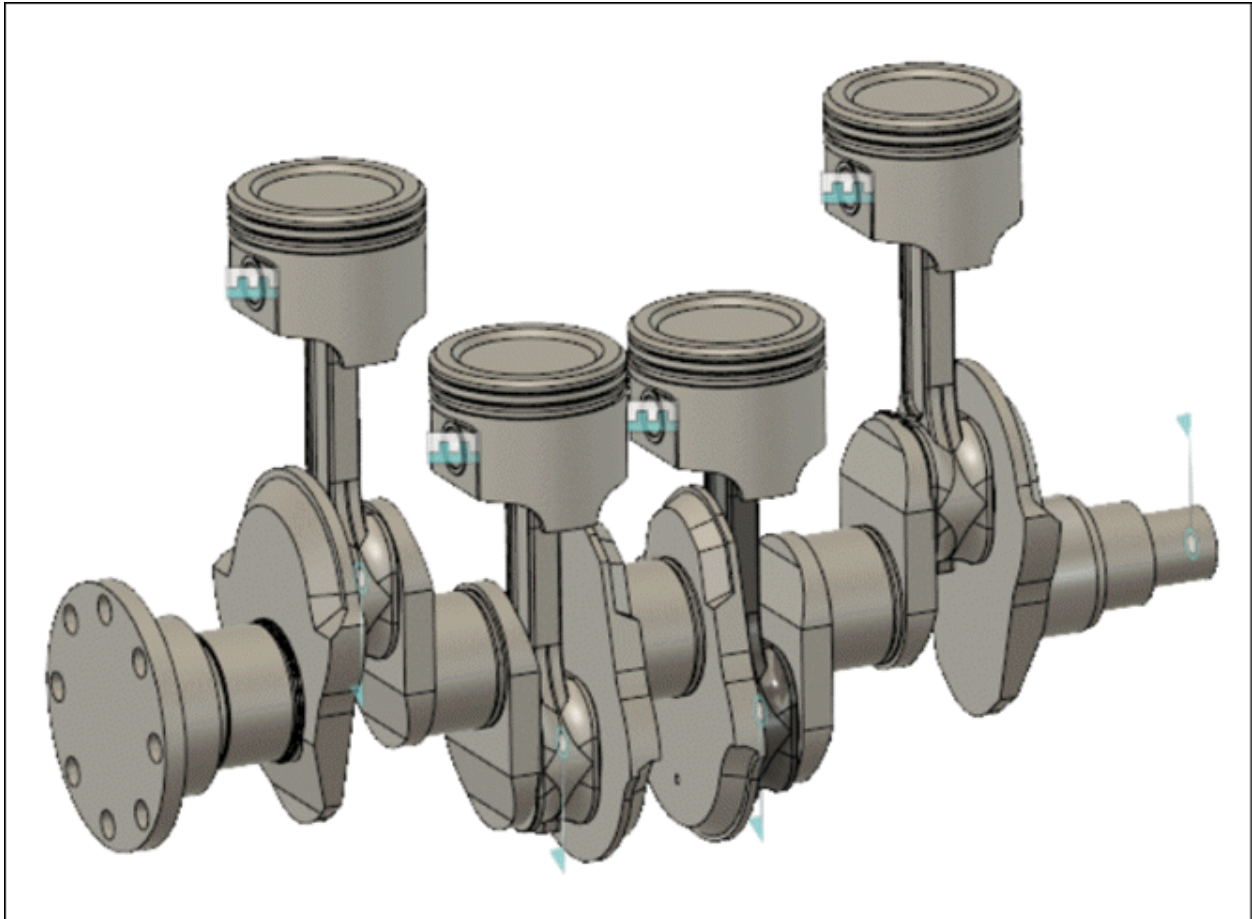


Figure-104. Assembled crank piston model

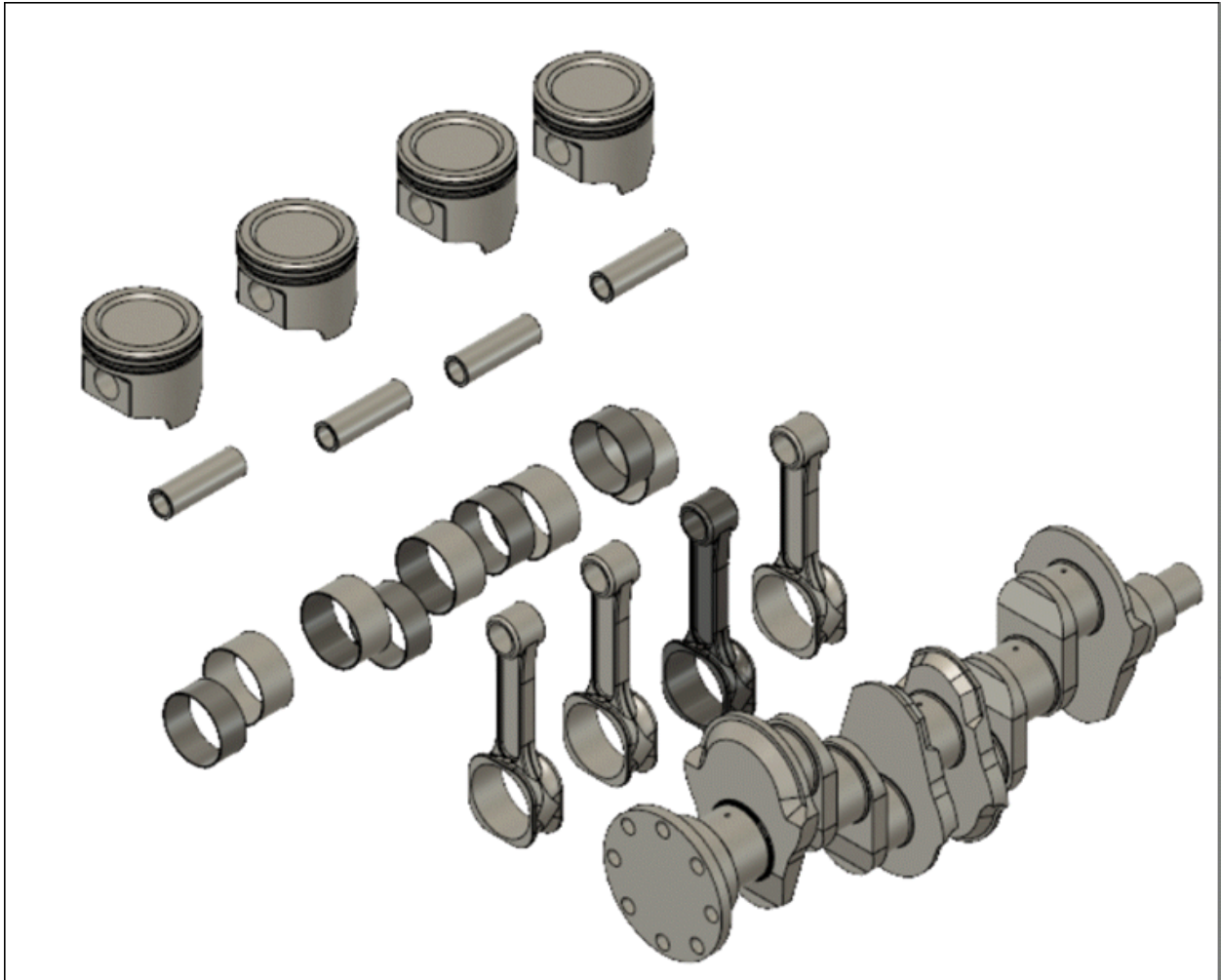


Figure-105. Exploded view of crank assembly

Chapter 8

Importing Files and Inspection

Topics Covered

The major topics covered in this chapter are:

- *Introduction*
- *Insert Derive*
- *Decal tool*
- *Attached Canvas*
- *Insert Mesh*
- *Insert SVG*
- *Insert DXF*
- *Insert McMaster-Carr Component*
- *Inserting a Manufacturing Part*
- *Insert TraceParts Supplier Components*
- *Measure Tool*
- *Interference Tool*
- *Curvature Comb Analysis*
- *Zebra Analysis*
- *Draft Analysis*
- *Curvature Map Analysis*
- *Accessibility Analysis*
- *Minimum Radius Analysis*
- *Section Analysis*
- *Center of Mass Tool*

INTRODUCTION

In the previous chapter, we have learned about the procedure of creating and assembling component. Till now, we have worked on the components that are/have been created in Autodesk Fusion 360. Now, you will learn to insert external model objects like canvas, decal, dxf, and manufacturer parts. In this chapter, you will learn the procedures of inspecting the part and importing files in **Autodesk Fusion 360** software.

INSERTING FILE

In this section, you will learn the procedure to insert various types of files in **Autodesk Fusion 360**.

Insert Derive

The **Insert Derive** tool is used to insert components, sketches, parameters, or bodies from another design file into the current model file. Note that before using this tool, you need to save the current model file. The procedure to use this tool is discussed next.

- Click on the **Insert Derive** tool of **INSERT** drop-down from **Toolbar**; refer to [Figure-1](#). The **Select Source** dialog box will be displayed; refer to [Figure-2](#).

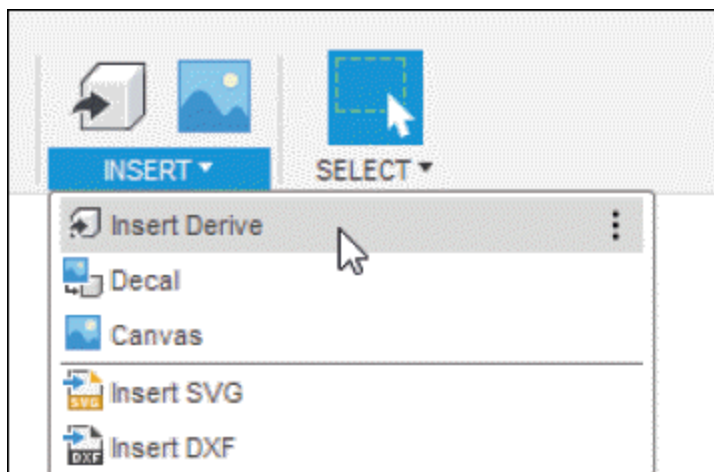


Figure-1. Insert Derive tool

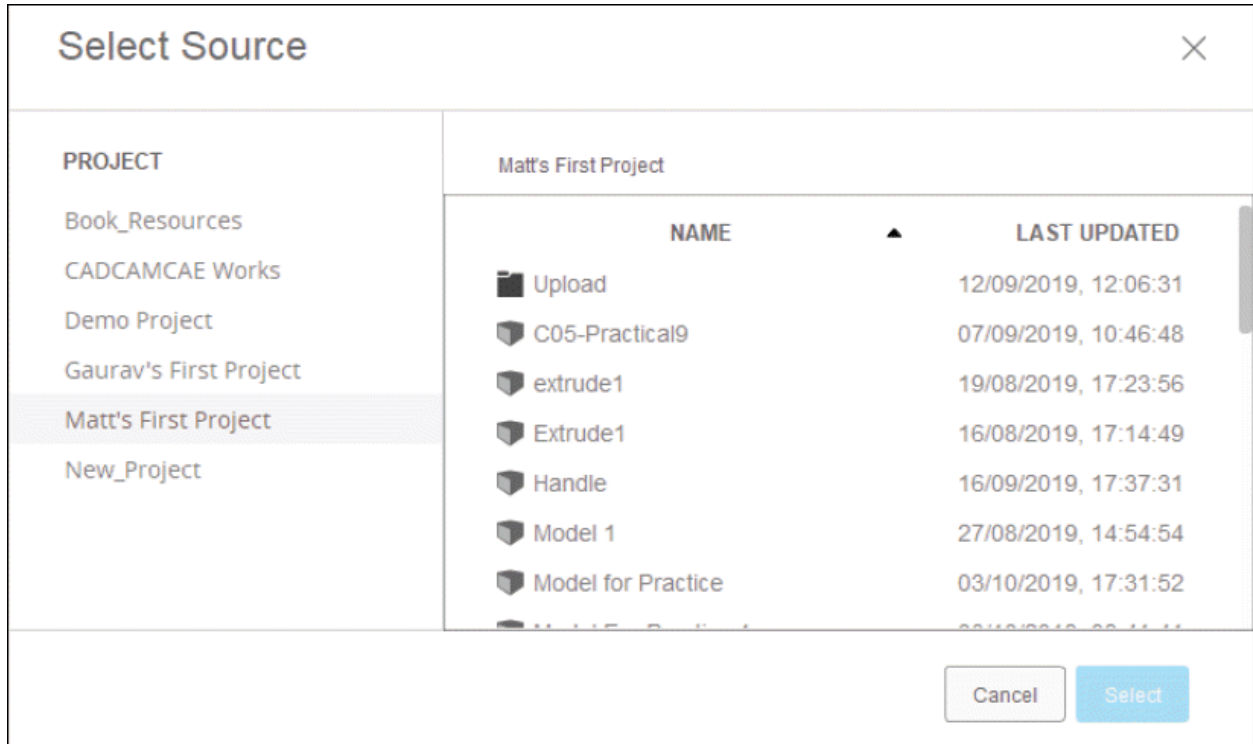


Figure-2. Select Source dialog box

- Select desired source component and click on the **Select** button from **Select Source** dialog box. The selected component will be inserted in the canvas along with the **DERIVE** dialog box; refer to [Figure-3](#).

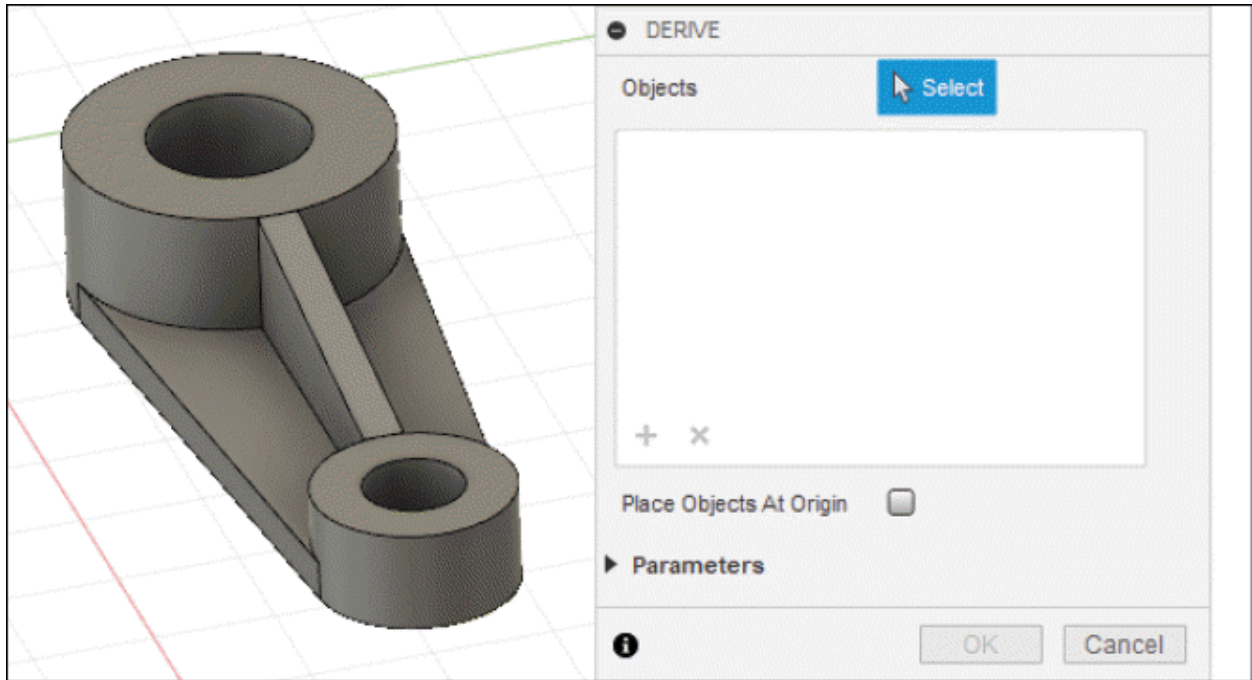


Figure-3. DERIVE dialog box

- The **Select** button of **Model Objects** section is active by default. Click on the inserted component to select the model. The updated **DERIVE** dialog box will be displayed; refer to [Figure-4](#). You can select the sketch and other objects from **BROWSER**.

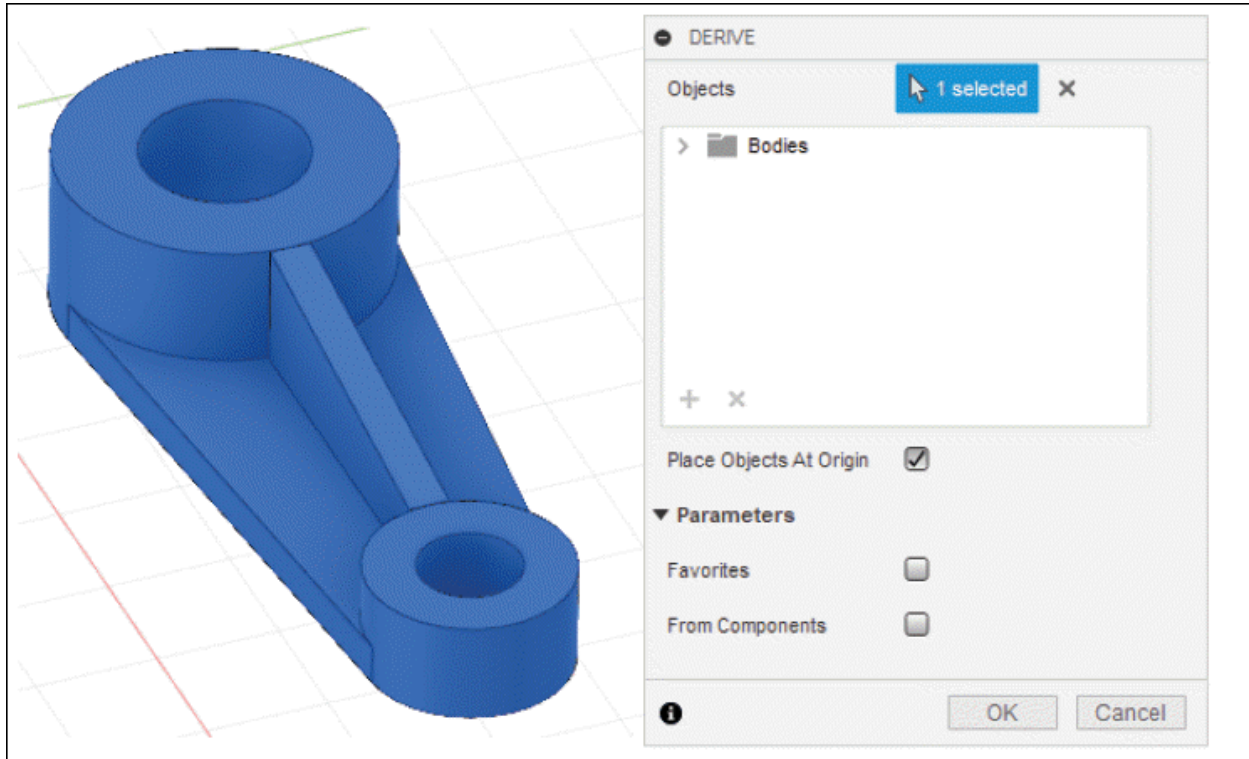


Figure-4. Updated DERIVE dialog box

- Select the **Place Objects At Origin** check box to place the component origin aligned to model origin.
- Expand the **Parameters** node and select desired check boxes to include parameters in desired part.
- After specifying all the parameters, click on **OK** button from the **DERIVE** dialog box to complete the process.

Decal

The **Decal** tool is used to place an image on the selected face/plane. The procedure to use this tool is discussed next.

- Click on the **Decal** tool of **INSERT** drop-down from **Toolbar**; refer to [Figure-5](#). The **Insert** dialog box will be displayed; refer to [Figure-6](#).

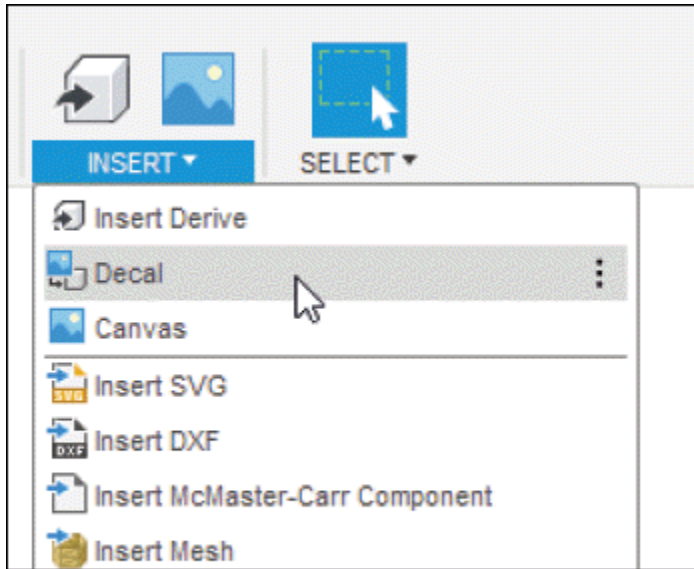


Figure-5. Decal tool

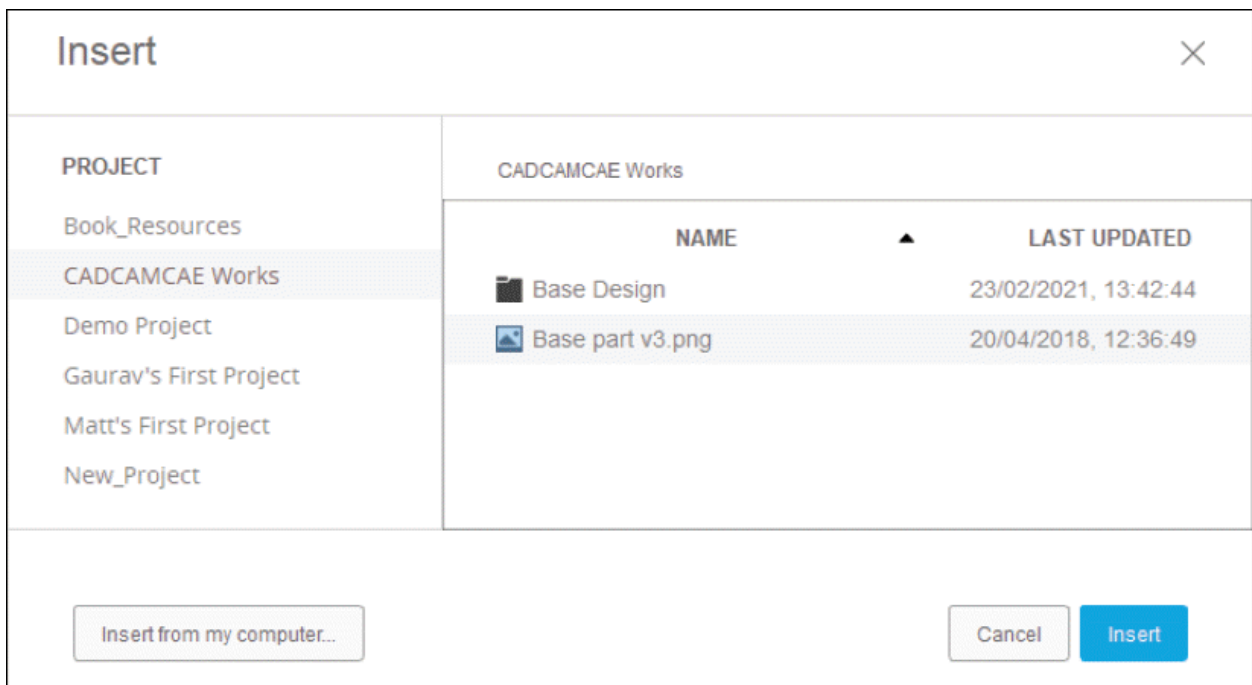


Figure-6. Insert dialog box

- Select required image from **Insert** dialog box and click on **Insert** button from the dialog box. The image will be inserted or click on **Insert from my computer** button from **Insert** dialog box. The **Open** dialog box will be

displayed; refer to [Figure-7](#).

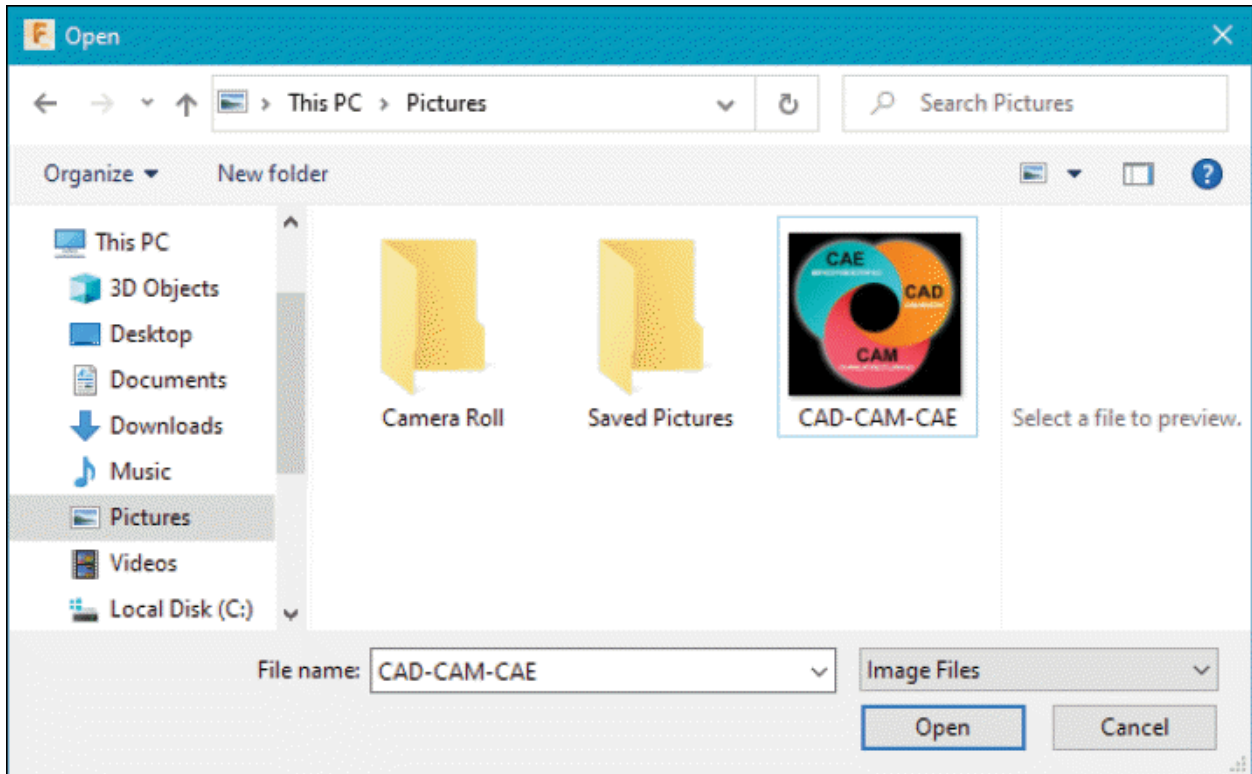


Figure-7. Open dialog box for importing file

- Select desired image which you want to import and click on **Open** button from **Open** dialog box. The **DECAL** dialog box will be displayed with the image inserted; refer to [Figure-8](#).

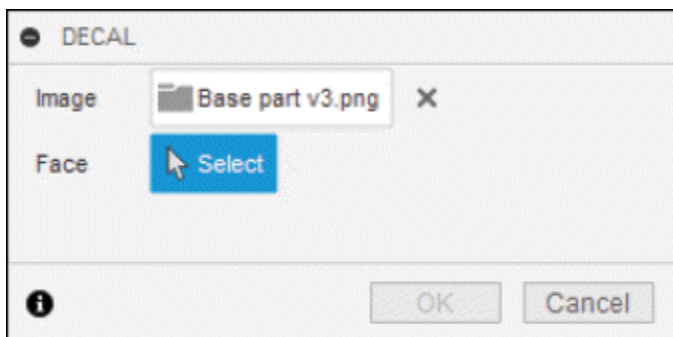


Figure-8. DECAL dialog box

- The **Select** button of **Face** section is active by default. Click on the face of model to which decal is to be applied. The updated **DECAL** dialog box will be displayed; refer to [Figure-9](#).

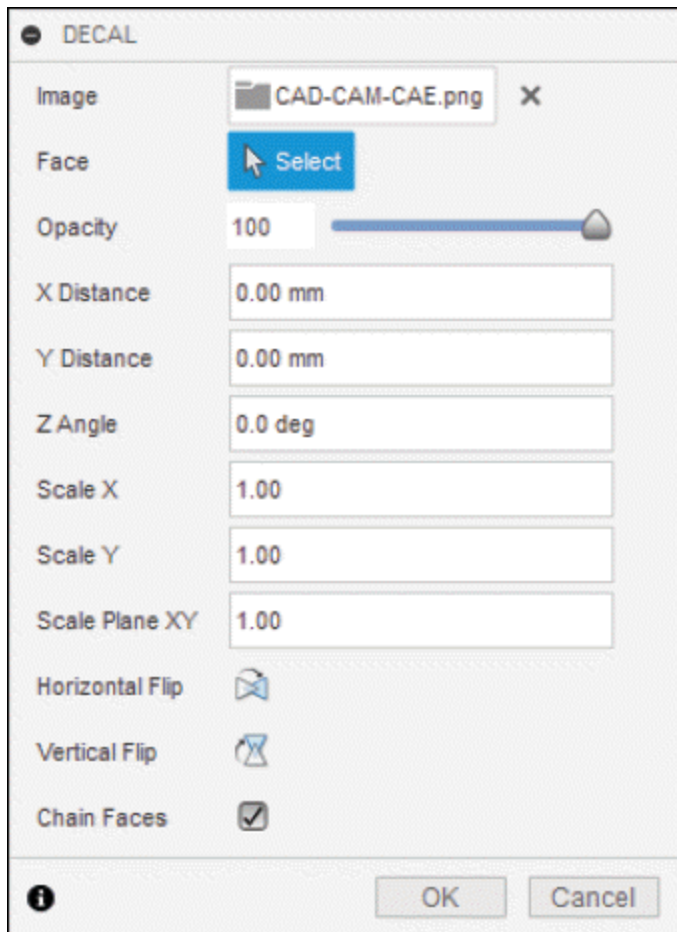


Figure-9. Updated DECAL dialog box

- Set the image on the selected face with the help of drag handle displayed on image. You can also set the image by specifying the values in respective edit boxes in the dialog box.
- Click on the **Horizontal Flip** button to flip the image horizontally.

- Click on the **Vertical Flip** button to flip the image vertically.
- Select the **Chain Faces** check box to include tangent faces in selection.
- After specifying the parameters, the preview of model along with image will be displayed; refer to [Figure-10](#).

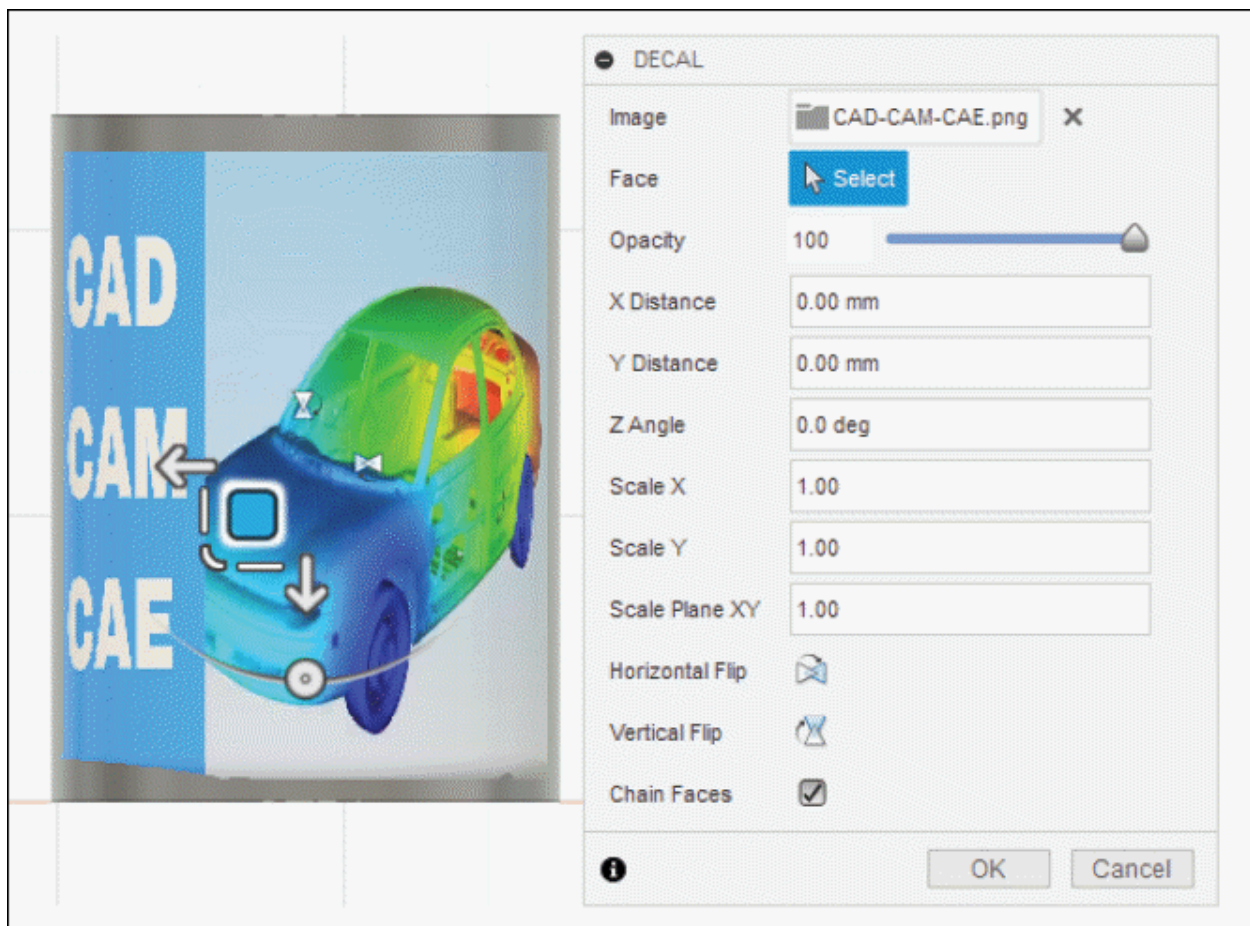


Figure-10. Preview of image on model

If the preview is as desired then click on the **OK** button from the **DECAL** dialog box to complete the process.

Canvas

The **Canvas** tool is used to insert an image on a planer face so that you can create model based on image. The procedure to use this tool is discussed next.

- Click on the **Canvas** tool of **INSERT** drop-down from **Toolbar**; refer to [Figure-11](#). The **Insert** dialog box will be displayed.

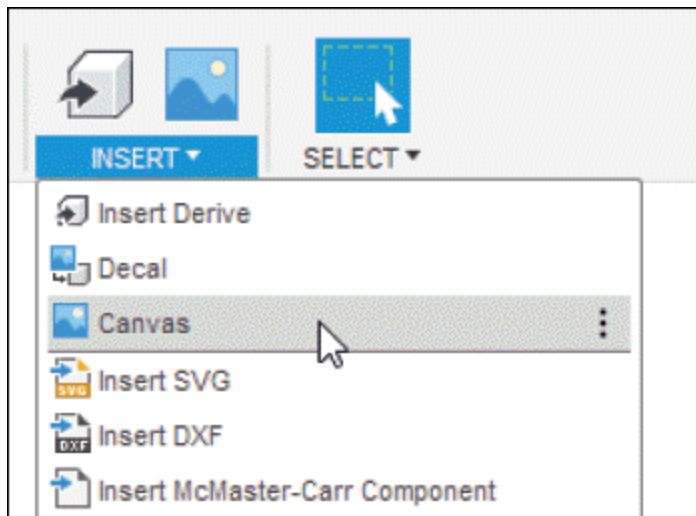


Figure-11. Canvas tool

- Select desired image from the dialog box and import as discussed earlier. The **CANVAS** dialog box will be displayed with the image inserted; refer to [Figure-12](#).

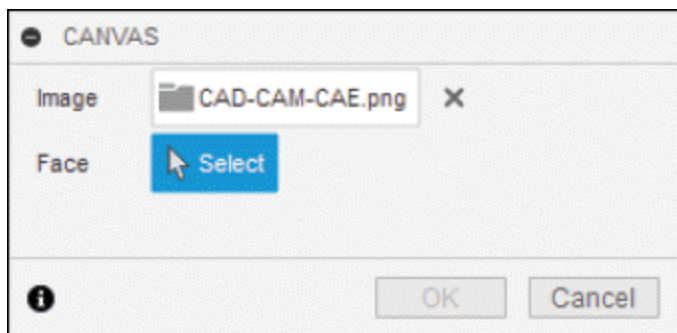


Figure-12. CANVAS dialog box

- The **Select** button of **Face** section is active by default. Click on the face of model/plane where you want to place the canvas image. The updated **CANVAS** dialog box will be displayed; refer to [Figure-13](#).

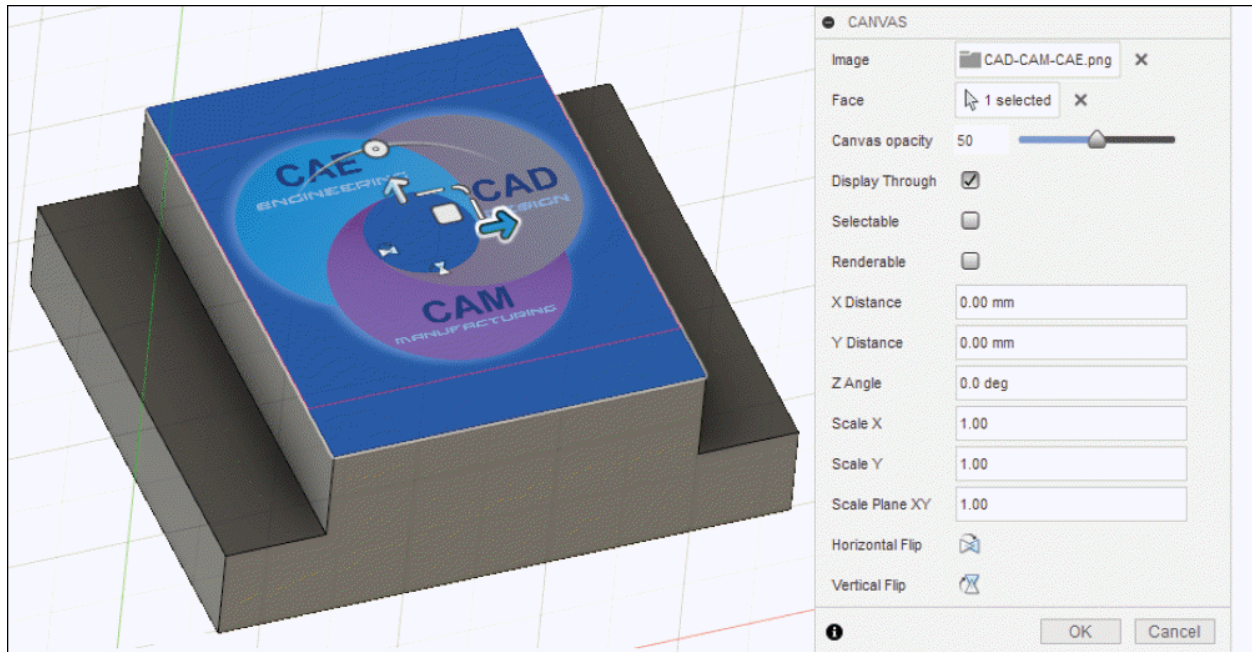


Figure-13. Updated CANVAS dialog box

- Click on **Canvas opacity** edit box and specify the opacity of image. You can also set the opacity by moving the **Canvas opacity** slider.
- Select the **Display Through** check box if you want to make the image visible through the component.
- Select the **Selectable** check box if you want the image to be selectable.
- Select the **Renderable** check box if you want to include the image in renderings of model.
- Click in the **X Distance** edit box and specify a value of

distance to move in X direction.

- Click in the **Y Distance** edit box and specify a value of distance to move in Y direction
- Click in the **Z Angle** edit box and specify a value to rotation around z axis.
- Click in the **Scale X** edit box and specify a scale value in X direction.
- Click in the **Scale Y** edit box and specify a scale value in Y direction.
- Click in the **Scale Plane XY** edit box and specify the value of scale in the XY plane.
- Click on the **Horizontal Flip** button to mirror the image horizontally.
- Click on the **Vertical Flip** button to mirror the image vertically.
- After specifying the parameters, click on the **OK** button from **CANVAS** dialog box to complete the process.

Insert SVG

The **Insert SVG** tool is used to insert an SVG image file into sketch. The procedure to use this tool is discussed next.

- Click on the **Insert SVG** tool of **INSERT** drop-down from **Toolbar**; refer to [Figure-14](#). The **Insert** dialog box will be displayed.

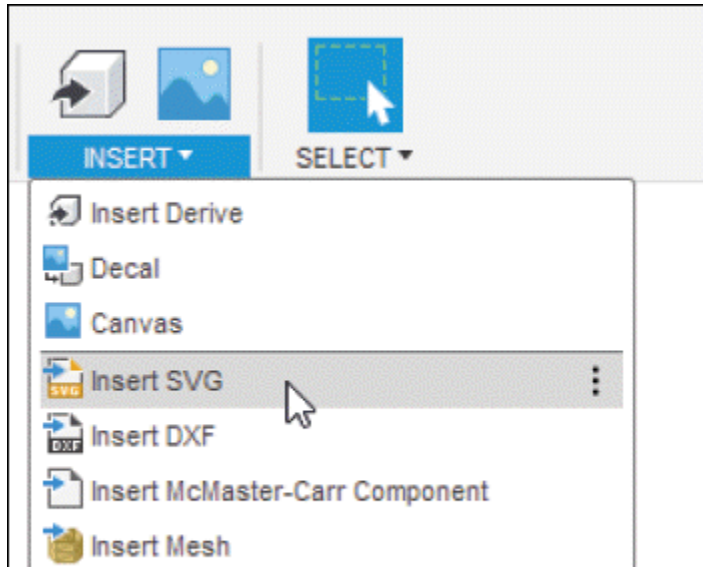


Figure-14. Insert SVG tool

- Select desired file from the dialog box and insert the file as discussed earlier. The **INSERT SVG** dialog box will be displayed with the file selected; refer to [Figure-15](#).

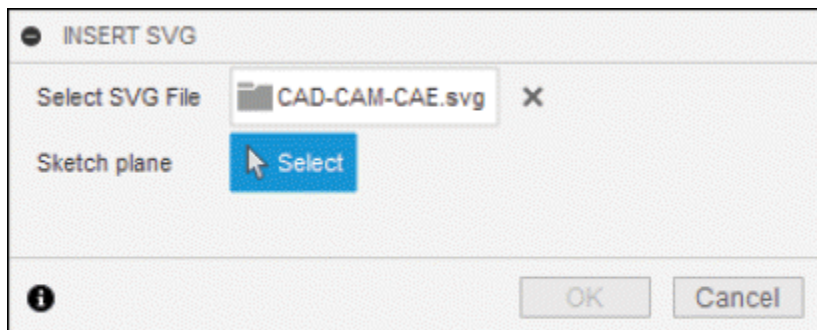


Figure-15. INSERT SVG dialog box

- The **Select** button of **Sketch plane** section is active by default. You need to select the plane for adding SVG file. Click on the plane to select; refer to [Figure-16](#).

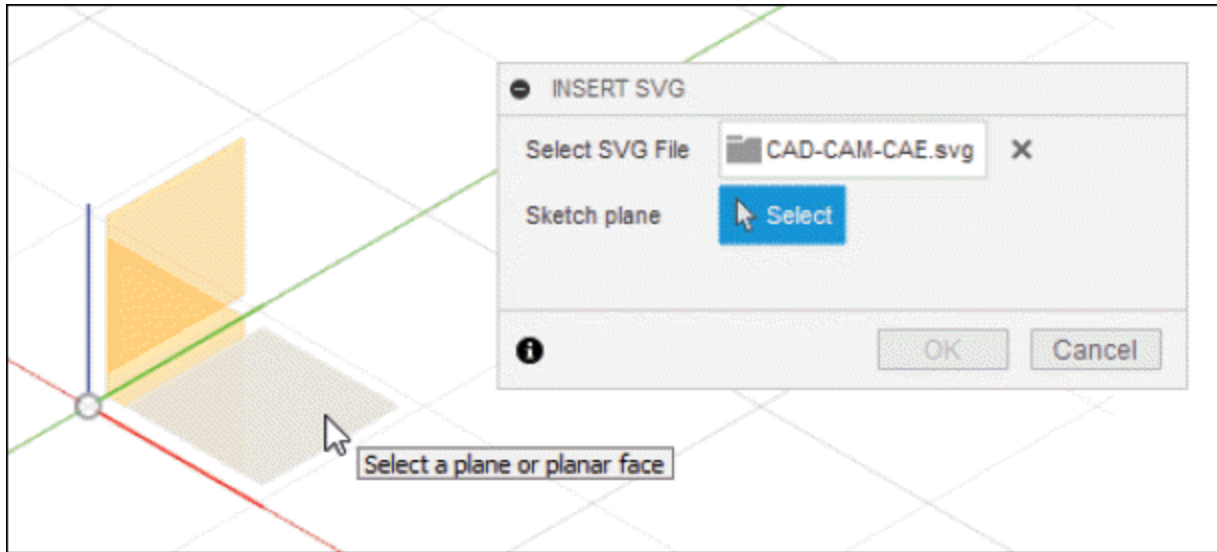


Figure-16. Selecting face or plane

- On selecting the face or plane, the sketching environment will be enabled with updated **INSERT SVG** dialog box; refer to [Figure-17](#).

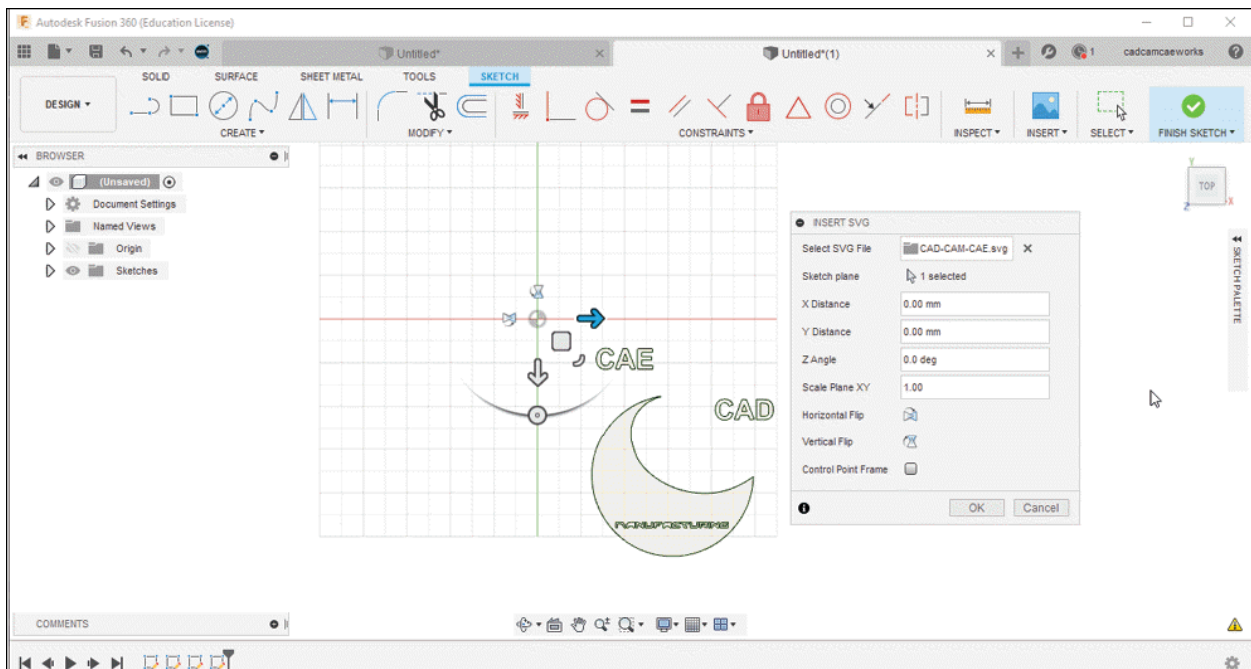


Figure-17. File along with updated INSERT SVG dialog box

- Click in the **X Distance** edit box and specify a value of

distance to move in X direction.

- Click in the **Y Distance** edit box and specify a value of distance to move in Y direction.
- Click in the **Z Angle** edit box and specify a value to rotation around z axis.
- Click in the **Scale Plane XY** edit box and specify the value of scale in the XY plane.
- Click on the **Horizontal Flip** button to mirror the image horizontally.
- Click on the **Vertical Flip** button to mirror the image vertically.
- You can also use the drag handle to move or flip the model.
- After specifying the parameters, click on the **OK** button from **INSERT SVG** dialog box to complete the process.

Insert DXF

The **Insert DXF** tool is used to insert a DXF file into the active design. The procedure to use this tool is discussed next.

- Click on the **Insert DXF** tool of **INSERT** drop-down from **Toolbar**; refer to [Figure-18](#). The **INSERT DXF** dialog box will be displayed; refer to [Figure-19](#).

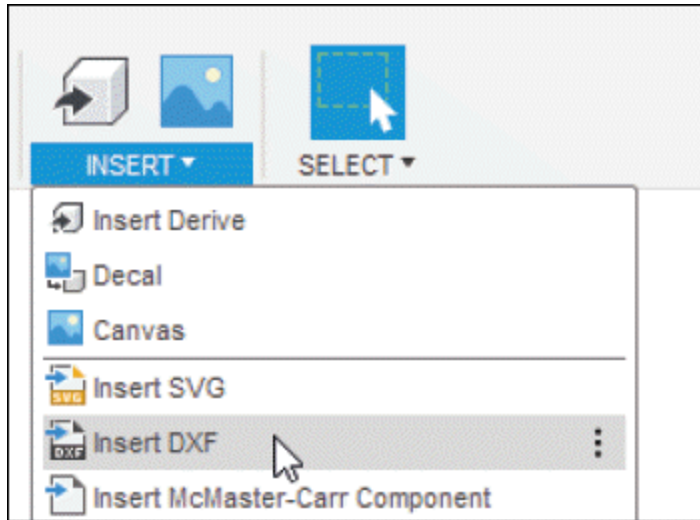


Figure-18. Insert DXF tool

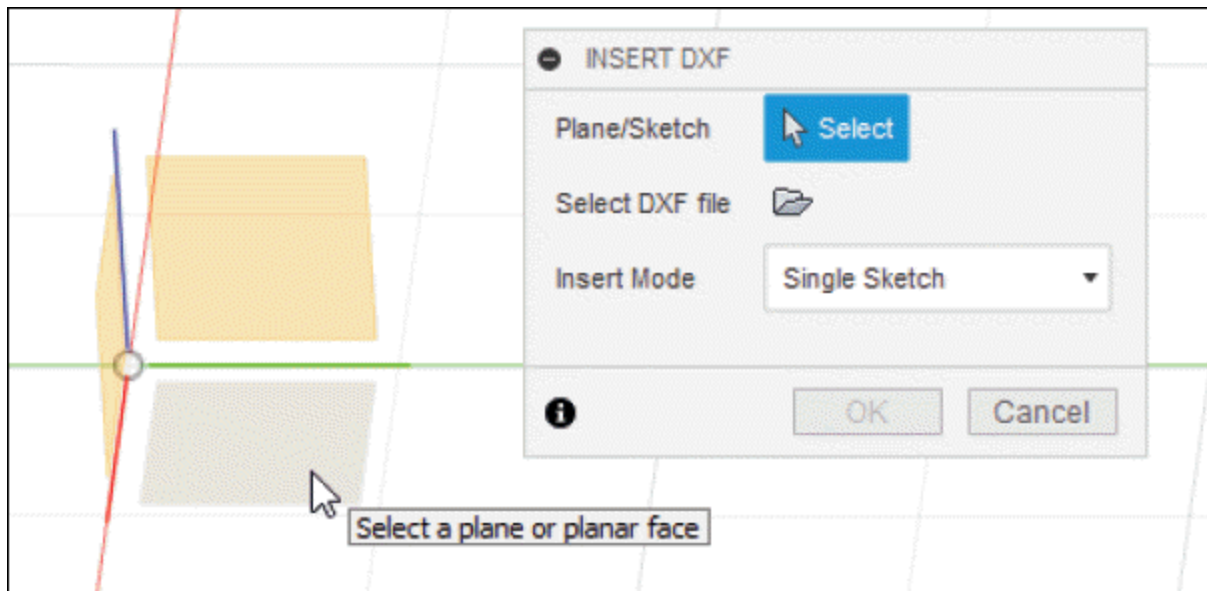


Figure-19. INSERT DXF dialog box

- The **Select** button of **Plane/Sketch** section is active by default. Click on the plane to select.
- Click on **Select DXF File** button from **INSERT DXF** dialog box. The **Open** dialog box will be displayed.
- You need to select the file of **.dxf** format and click on **Open**

button. The preview of DXF file will be displayed along with updated **INSERT DXF** dialog box; refer to [Figure-20](#).

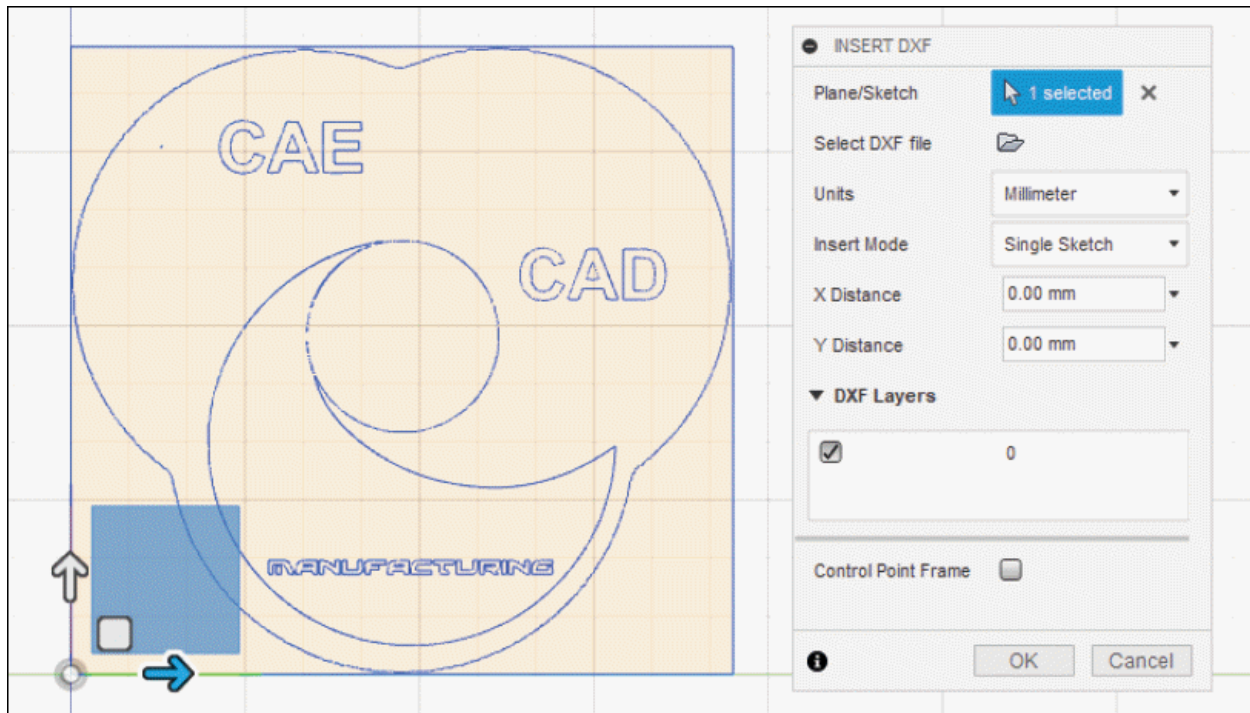


Figure-20. Updated **INSERT DXF** dialog box

- Click on the **Units** drop-down and select required unit.
- Select **Single Sketch** option from **Insert Mode** drop-down to insert the layers of selected .DXF file into one sketch.
- Select **One Sketch per Layer** option from **Insert Mode** drop-down to insert each sketch of DXF file on different layer.
- Click in the **X Distance** edit box and specify a value of distance to move in X direction.
- Click in the **Y Distance** edit box and specify a value of distance to move in Y direction.

- Select the **DXF Layers** check box to list the layers available in the selected DXF file. Select the layers to import.
- After specifying the parameters, click on the **OK** button from **INSERT DXF** dialog box to complete the process.

Insert McMaster-Carr Component

The **Insert McMaster-Carr Component** tool is used to insert 2D or 3D model of various library components provided by McMaster-Carr. The procedure to use this tool is given next.

- Click on the **Insert McMaster-Carr Component** tool from the **INSERT** drop-down in the **Toolbar**; refer to [Figure-21](#). The **INSERT MCMASTER-CARR COMPONENT** window will be displayed; refer to [Figure-22](#).

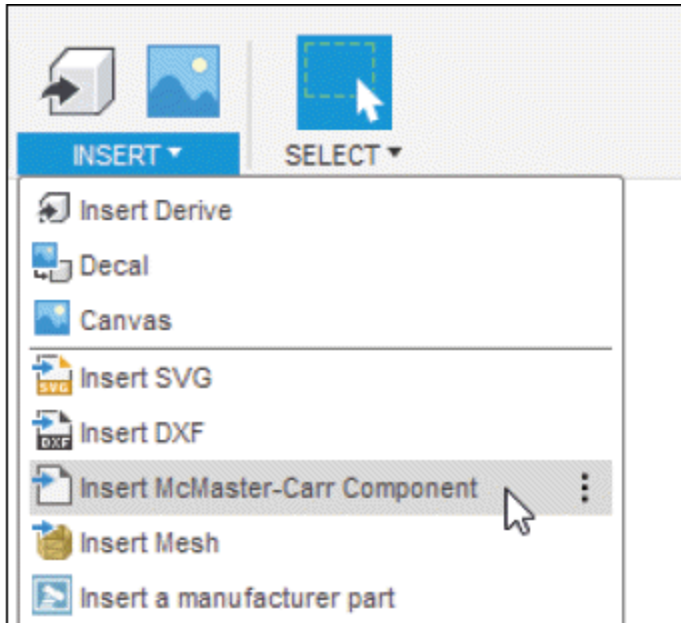


Figure-21. Insert McMaster Carr Component tool

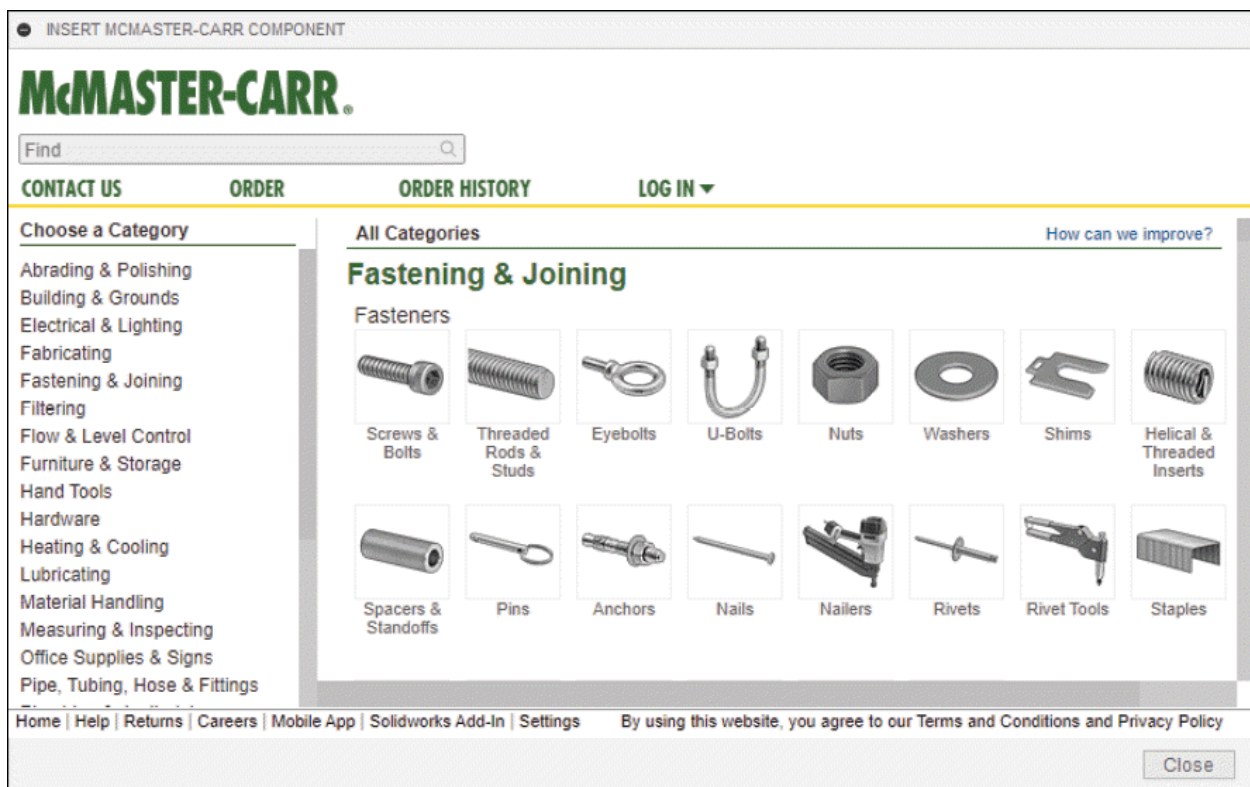


Figure-22. INSERT MCMMASTER CARR COMPONENT window

- Select desired component and its model number from the

window. The page in window will be displayed as shown in [Figure-23](#).

The screenshot shows the McMaster-Carr website interface for selecting a component. The search bar contains "threaded rods". The navigation menu includes "CONTACT US", "ORDER", "ORDER HISTORY", and "LOG IN". The main content area displays "511 Products" for "High-Strength Steel Threaded Rods". A sidebar on the left allows filtering by "System of Measurement" (Inch) and "Thread Size" (8-32, 10-24, 10-32, 1/4"-20, 1/4"-28, 5/16"-18, 5/16"-24, 3/8"-16). The main table lists products with columns for Length (Lg.), Tensile Strength (psi), Hardness, Specifications Met, Part Number, and Price (Each). The selected item is 3313N18, a 1-1/2 inch long rod with 3/8 inch thread size, priced at 8.90. A product detail pop-up is visible for this item, showing a description, "ADD TO ORDER" button, and "In stock" status. Below the table, there are links for "Product Detail", "CAD", and "3-D Solidworks" with a "Download" button. The footer includes "Home | Help | Returns | Careers | Mobile App | Solidworks Add-In | Settings" and "Terms and Conditions and Privacy Policy".

Lg.	Tensile Strength, psi	Hardness	Specifications Met	Part Number	Each
Zinc Yellow-Chromate Plated Steel					
3/8"-16					
1 1/2"	150,000	Rockwell C33	ASTM A354BD	3313N18	8.90
2"	150,000	Rockwell C33	ASTM A354BD	3313N19	9.42
2 1/2"	150				
3"	150				
3 1/2"	150,000	Rockwell C33	ASTM A354BD	3313N23	10.58

Figure-23. Component selected

- Click on the **Product Detail** link button for selected component. The drawing and model of selected component will be displayed at the bottom in the window; refer to [Figure-24](#).

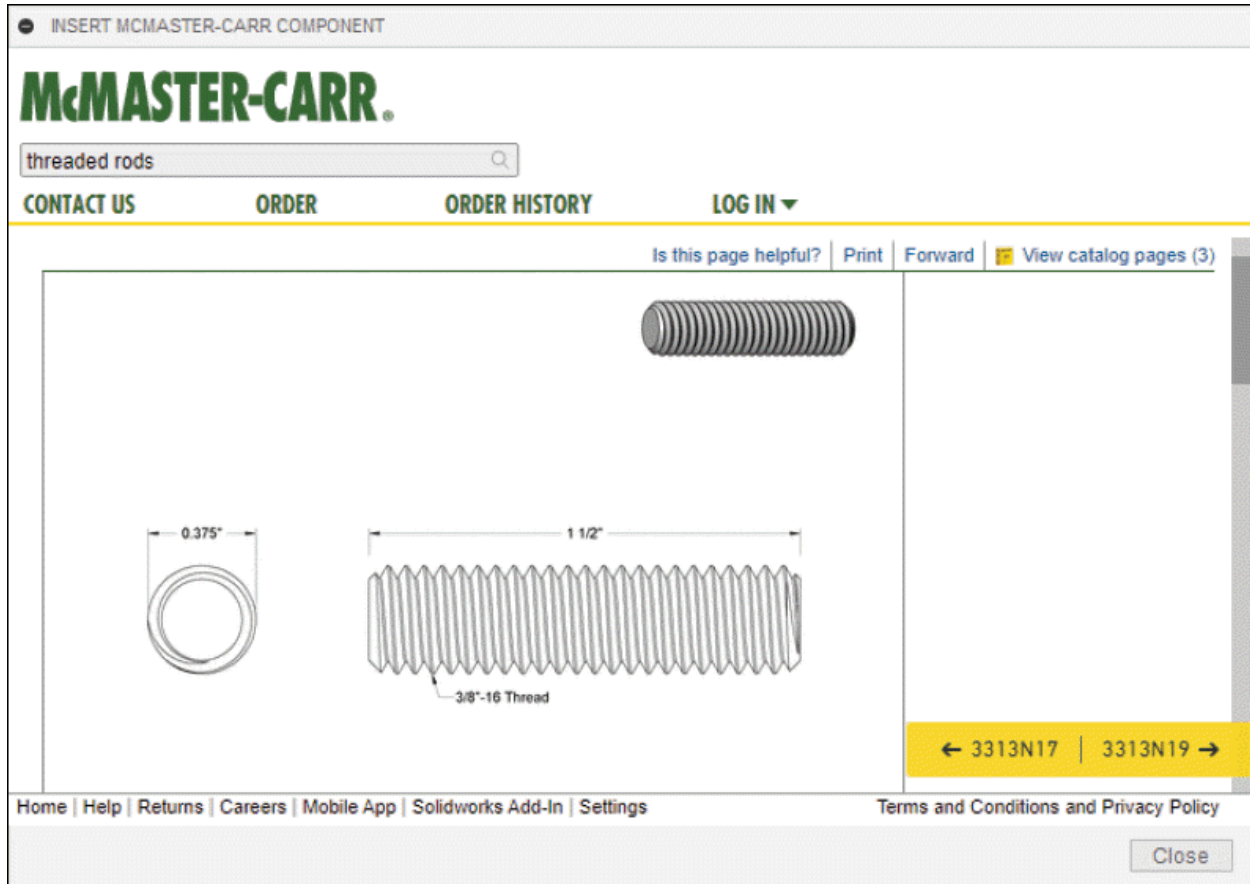


Figure-24. Model for selected component

- Select desired format from drop-down above the drawing and click on the **Save** button to save the model in local drive.
- Close the window by selecting **Close** button.

Insert Mesh

The **Insert Mesh** tool is used to insert a mesh file in current model. The procedure to use this tool is discussed next.

- Click on the **Insert Mesh** tool of **INSERT** drop-down from **Toolbar**; refer to [Figure-25](#). The **Open** dialog box will be displayed; refer to [Figure-26](#).

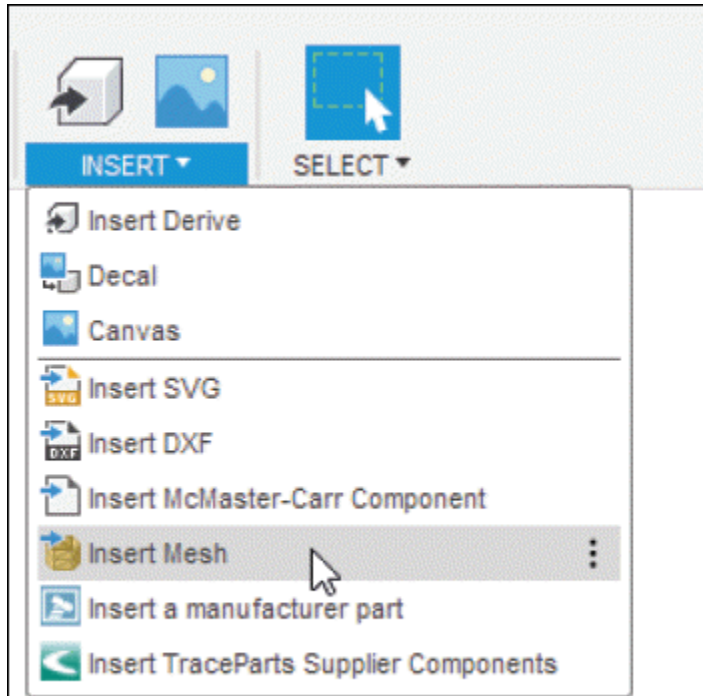


Figure-25. Insert Mesh tool

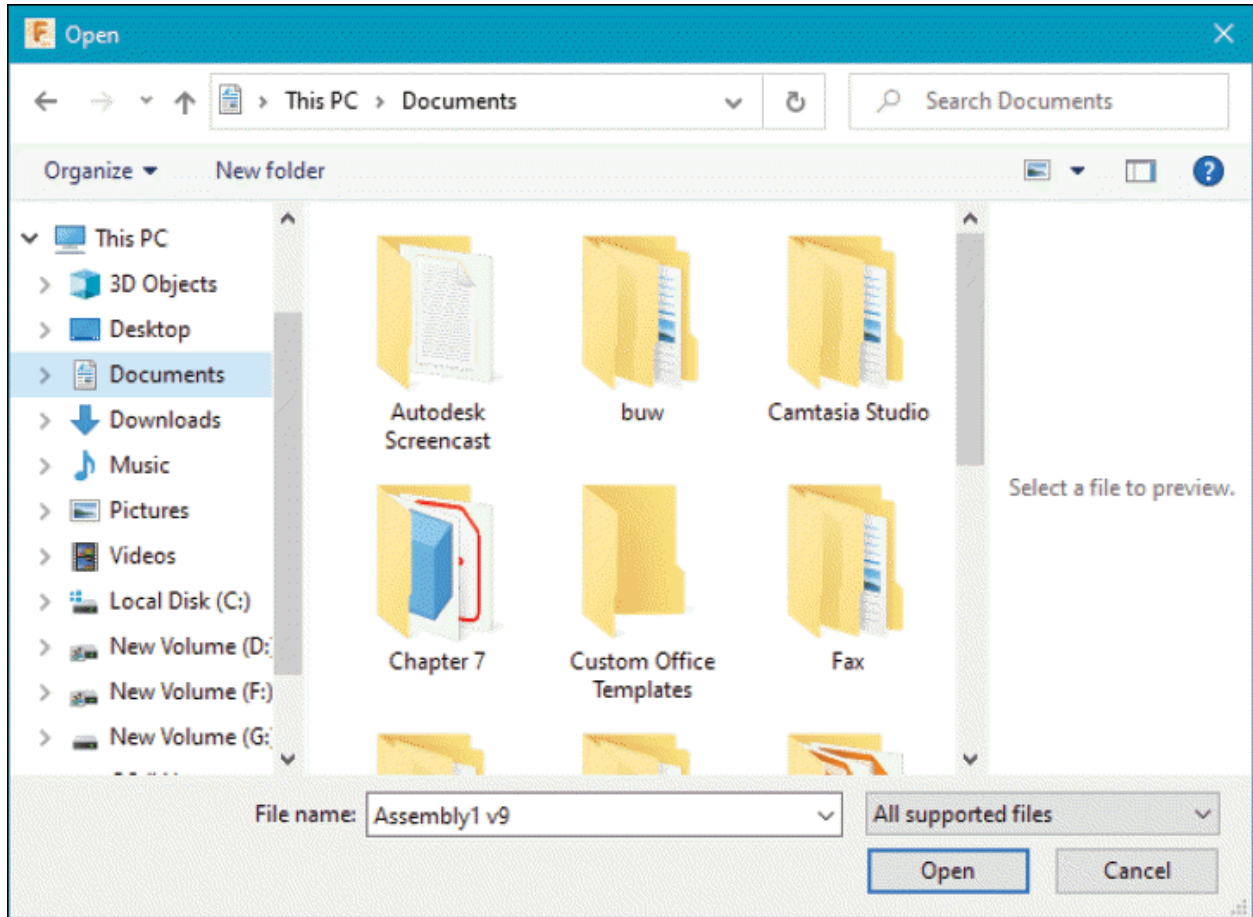


Figure-26. Open dialog box for inserting file

- Select the file and click on **Open** button of **Open** dialog box. The file will be added in application window along with **INSERT MESH** dialog box; refer to [Figure-27](#).

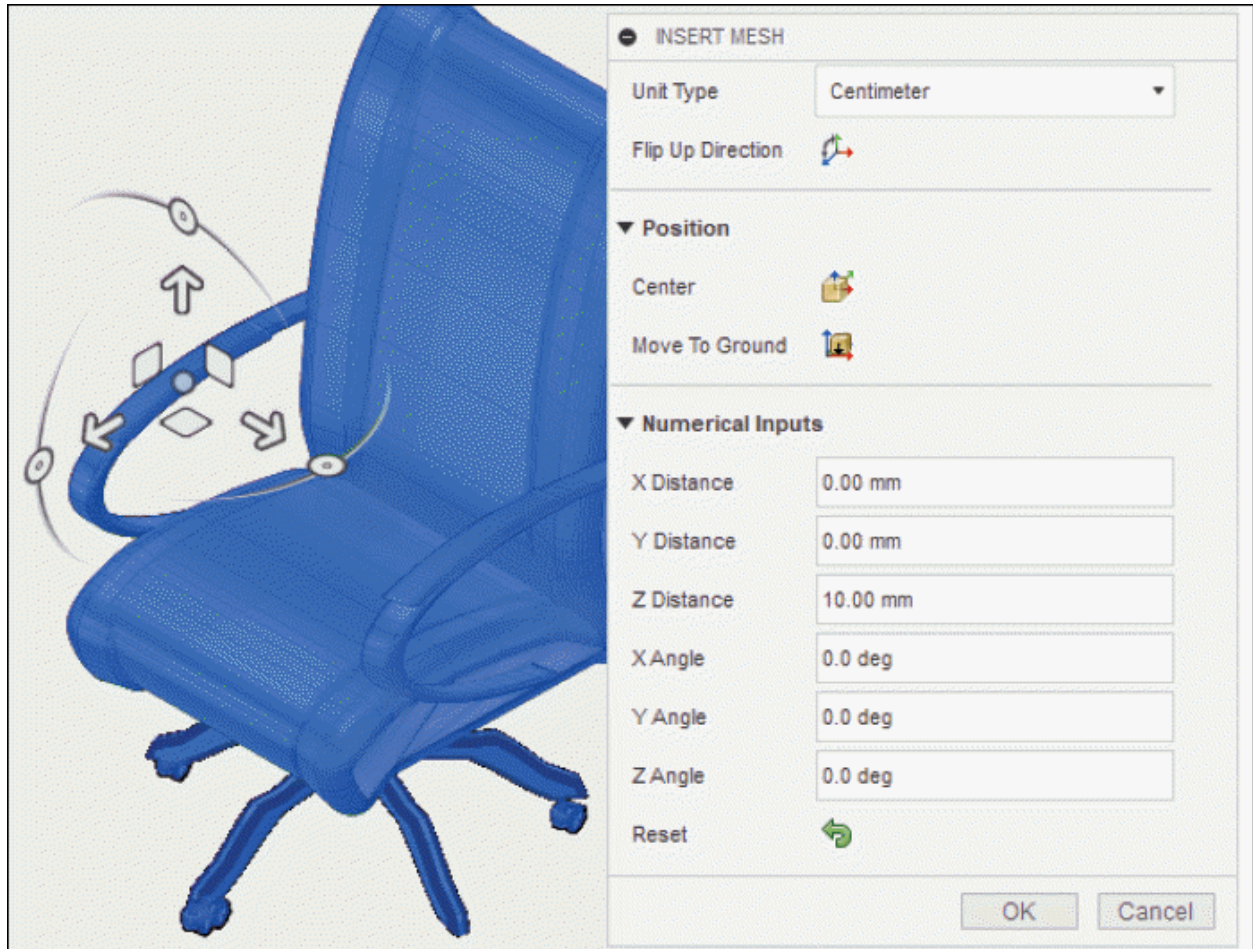


Figure-27. INSERT MESH dialog box

- Click on the **Unit Type** drop-down from **INSERT MESH** dialog box and select required unit to scale the model.
- Click on the **Flip Up Direction** button to flip the direction of model.
- Click on the **Center** button from **Position** section to drag the model to center or coordinate system.
- Click on **Move To Ground** button from **Position** section to move the model to ground plane.
- If you want to enter the numerical value to move the model

then click on the **Numerical Inputs** section and enter desired value. You can also move and rotate the model by using drag handles.

- Click on **Reset** button from **Numerical Inputs** section to reset the model position to the original state.
- After specifying the parameters, click on the **OK** button from **INSERT MESH** dialog box to complete the process.

Insert a manufacturer part

The **Insert a manufacturer part** tool is used for inserting a part of manufacturer or supplier in model through parts4cad. The procedure to use this tool is discussed next.

- Click on the **Insert a manufacturer part** tool of **INSERT** drop-down from **Toolbar**; refer to [Figure-28](#). The Part Community web page will be displayed in default web browser; refer to [Figure-29](#).

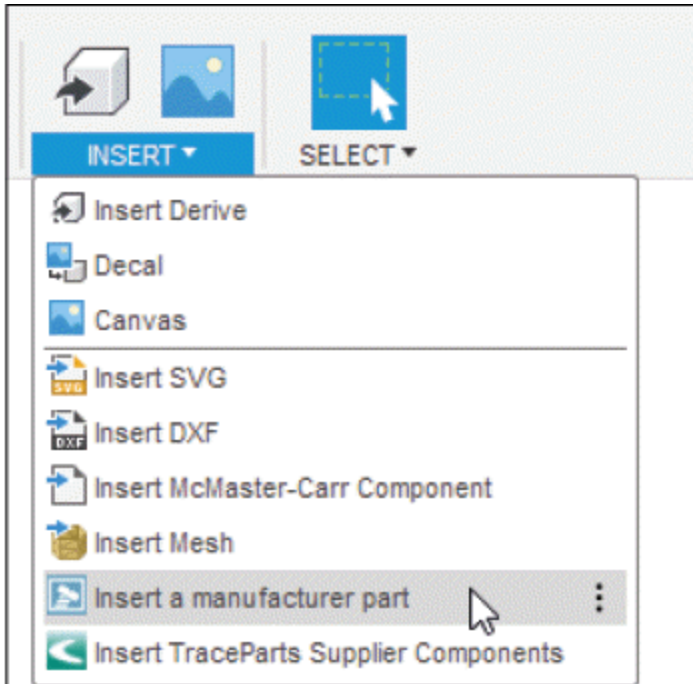


Figure-28. Insert a manufacturer part tool

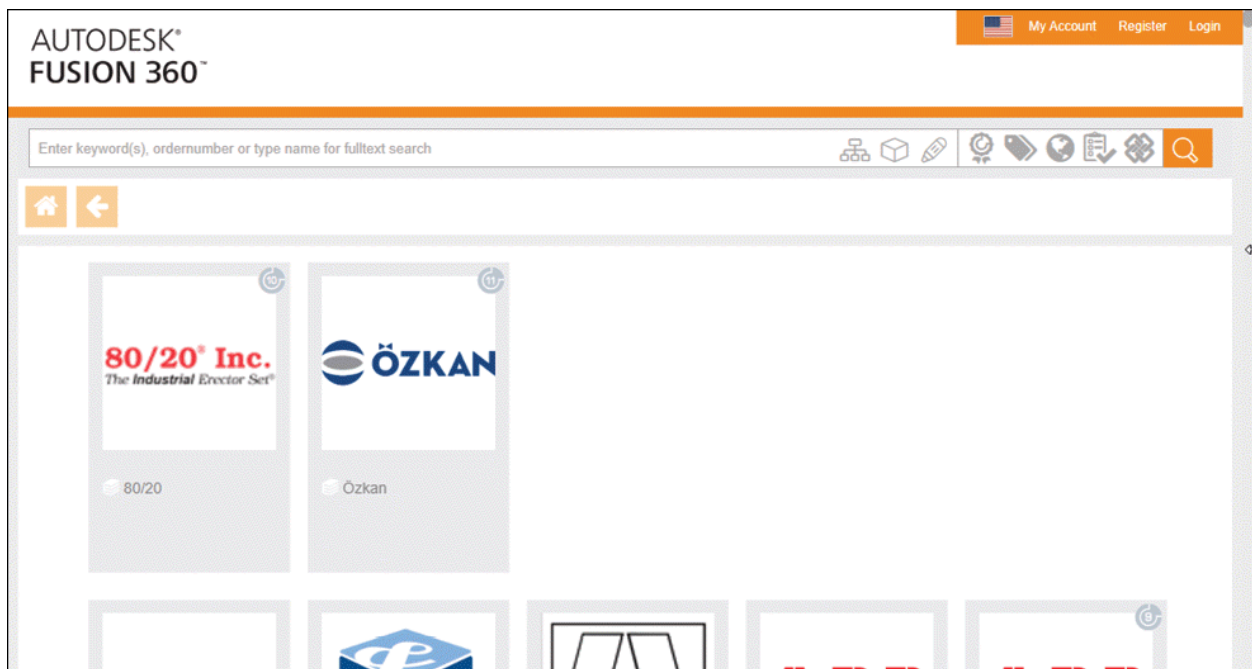


Figure-29. AUTODESK FUSION 360 part community window

- Select the part from manufacturer or supplier and open the file in Autodesk Fusion 360.

Insert TraceParts Supplier Components

The **Insert TraceParts Supplier Components** tool gives you access to millions of supplier components, 3D models, CAD files, and 2D drawings. Simply navigate to what you are looking for, sign-in with your TraceParts account, download, and insert a supplier component directly into Fusion 360 canvas. The procedure to use this tool is discussed next.

- Click on the **Insert TraceParts Supplier Components** tool of **INSERT** drop-down from **Toolbar**; refer to [Figure-30](#). The **AUTODESK FUSION 360 DESIGN LIBRARY** window will be displayed; refer to [Figure-31](#).

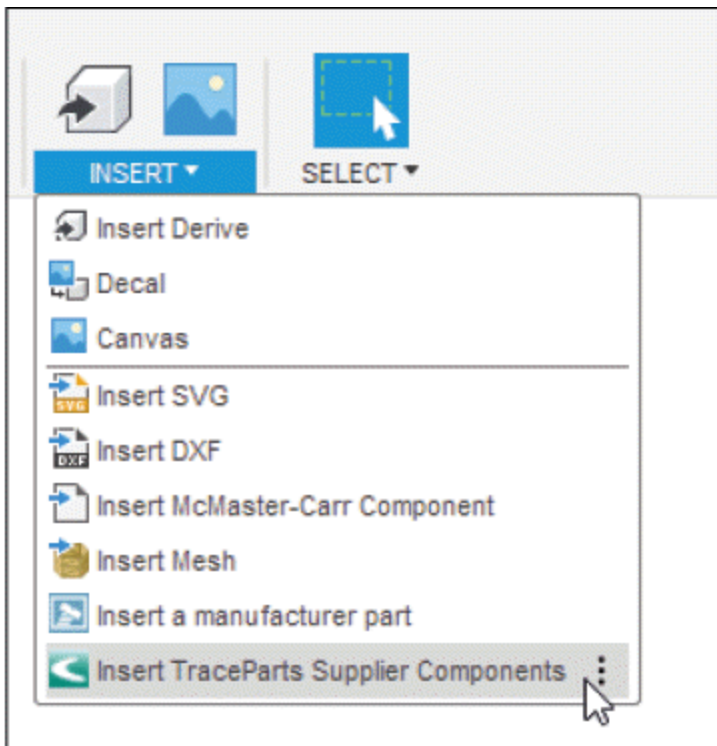


Figure-30. Insert TraceParts Supplier Components tool

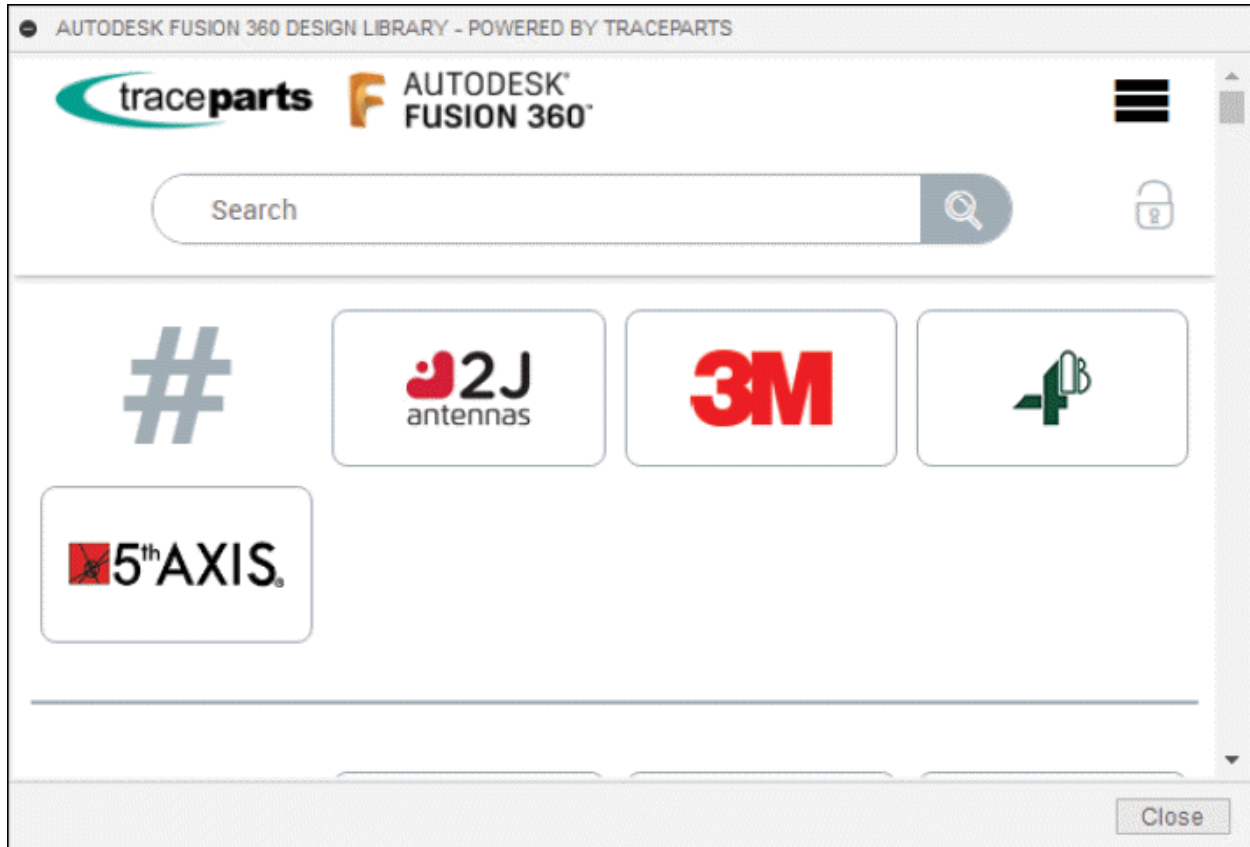


Figure-31. AUTODESK FUSION 360 DESIGN LIBRARY window

- Select the part from supplier and open the file in Autodesk Fusion 360.

INSPECT

Till now, we have discussed various tools to insert the objects in Autodesk Fusion 360. Now, we will discuss about measurement and analysis tools available in Autodesk Fusion 360.

Measure

The **Measure** tool is used to measure selected geometry. The procedure to use this tool is discussed next.

- Click on the **Measure** tool of **INSPECT** drop-down from

Toolbar; refer to [Figure-32](#). The **MEASURE** dialog box will be displayed; refer to [Figure-33](#).

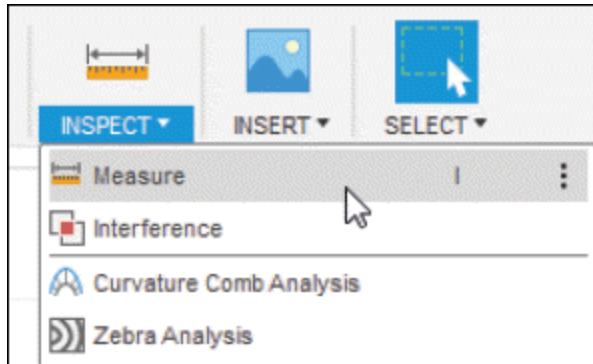


Figure-32. Measure tool

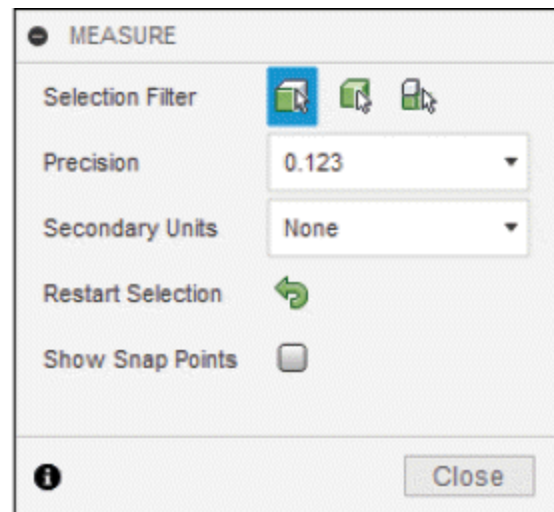


Figure-33. MEASURE dialog box

- Select the **Select Face/Edge/Vertex** button from **Selection Filter** section if you want to select face/edge/vertex for measurement.
- Select the **Select Body** button from **Selection Filter** section if you want to select body for measurement.
- Select the **Select Component** button from **Selection Filter** section if you want to select component for measurement.
- Click on the **Precision** drop-down and select desired precision option for displaying the required number of decimal places in results.
- Select desired secondary unit for the measurement from **Secondary Units** drop-down if you want to display secondary unit as well.

- Click on the **Restart Selection** button to restart the selection process once you have performed measurement.
- In our case, we are selecting **Select Face/Edge/Vertex** button from **Selection Filter** section. Click on the face to measure; refer [Figure-34](#).

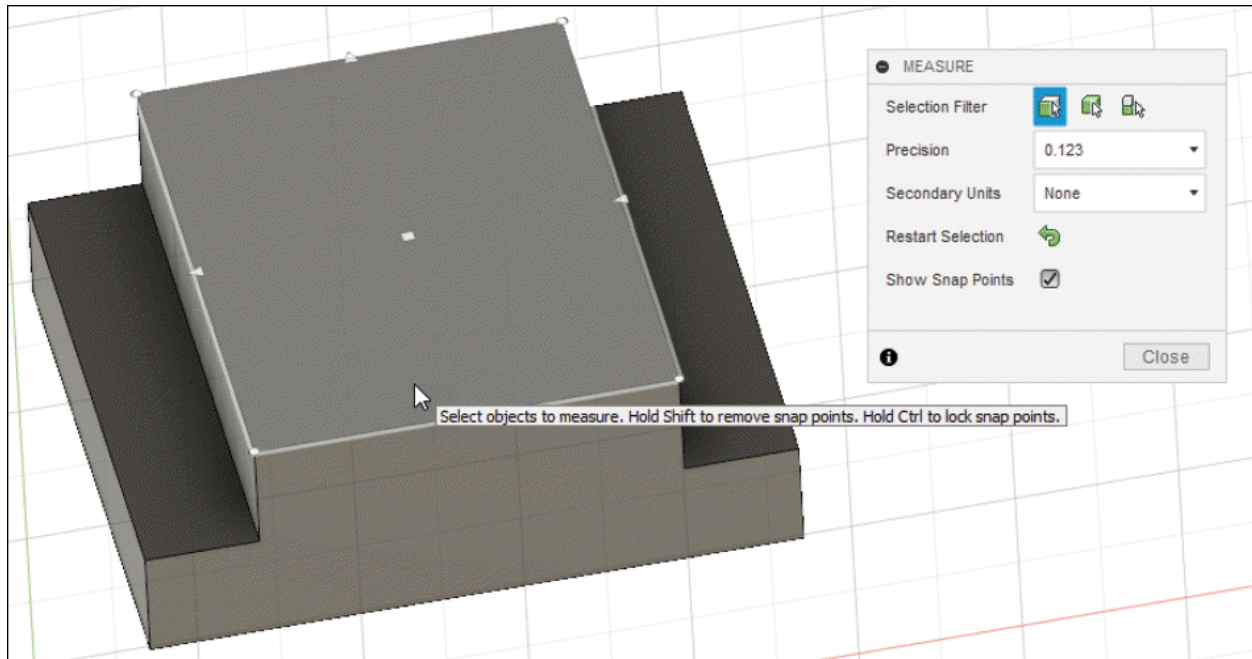


Figure-34. Selection of face for measurement

- On selecting the face, the updated **MEASURE** dialog box will be displayed with the measurement of selected face; refer to [Figure-35](#).

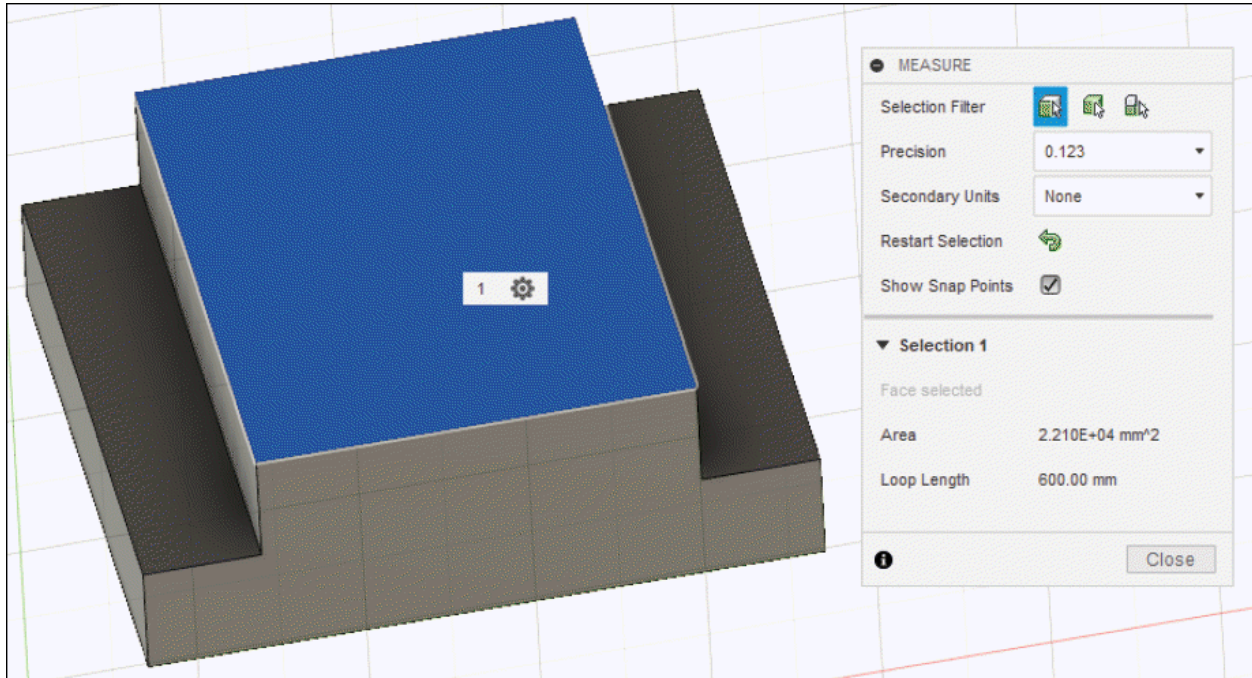


Figure-35. Updated MEASURE dialog box

- If you want to measure the distance between two faces then select the other face.
- Hover the cursor on the respective measurement to copy the value to clipboard; refer to [Figure-36](#).

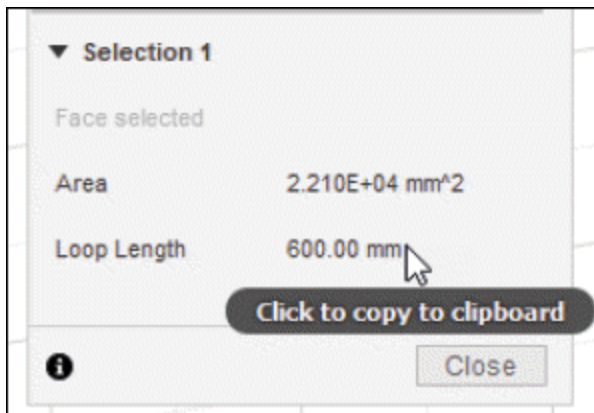


Figure-36. Copying measurement

- Select the **Show Snap Points** check box to display snap points when you hover over your model with the **Measure**

tool activated.

- Click on **Close** button of **MEASURE** dialog box to exit the **Measure** tool.

Interference

The **Interference** tool is used to display interference of selected components or bodies. The procedure to use this tool is discussed next.

- Click on the **Interference** tool of **INSPECT** drop-down from **Toolbar**; refer to [Figure-37](#). The **INTERFERENCE** dialog box will be displayed; refer to [Figure-38](#).

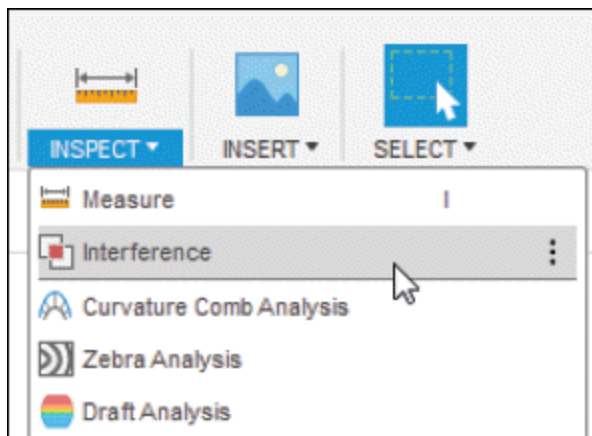


Figure-37. Interference tool

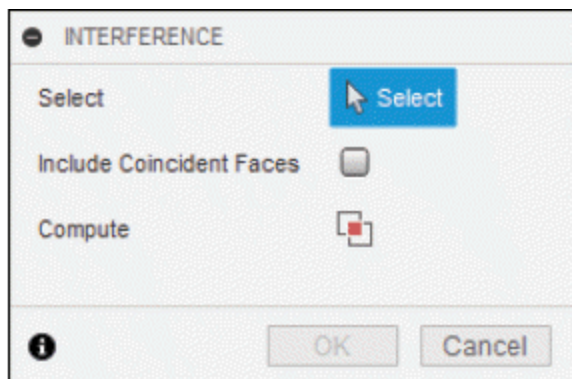


Figure-38. INTERFERENCE dialog box

- The **Select** button of **Select** section is active by default. Click on the model to check interference. You can also use window selection.
- Select the **Include Coincident Faces** check box if you want to include the faces in selection which are touching with each other.
- In our case, we are using window selection to select the component; refer to [Figure-39](#).

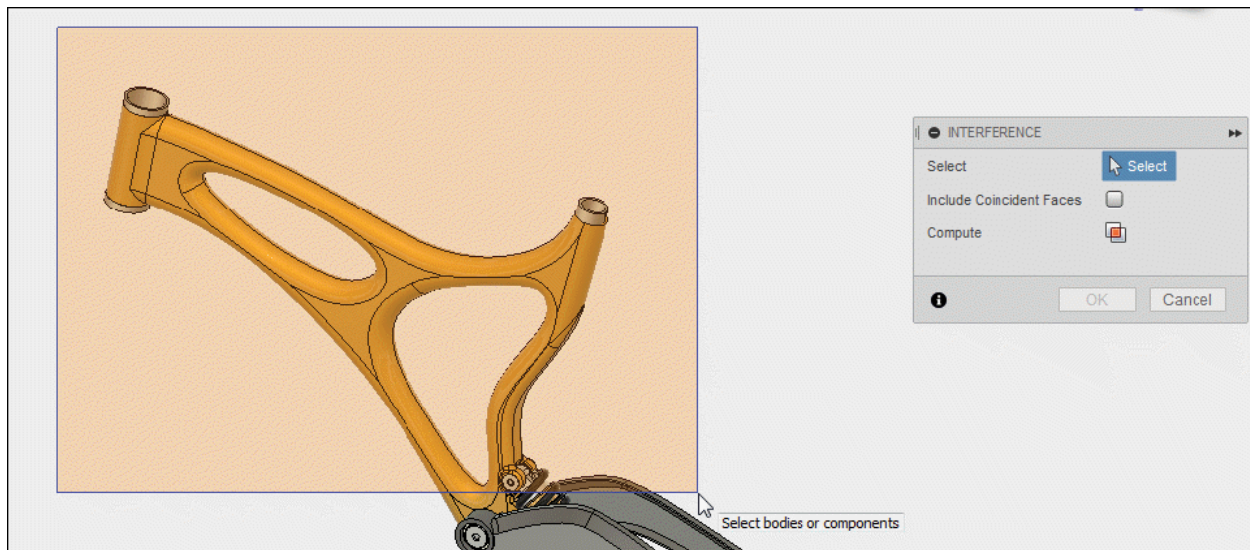


Figure-39. Selection of component for interference

- After selecting the components, click on **Compute** button to calculate the interference. The **Interference Results** dialog box will be displayed; refer to [Figure-40](#).

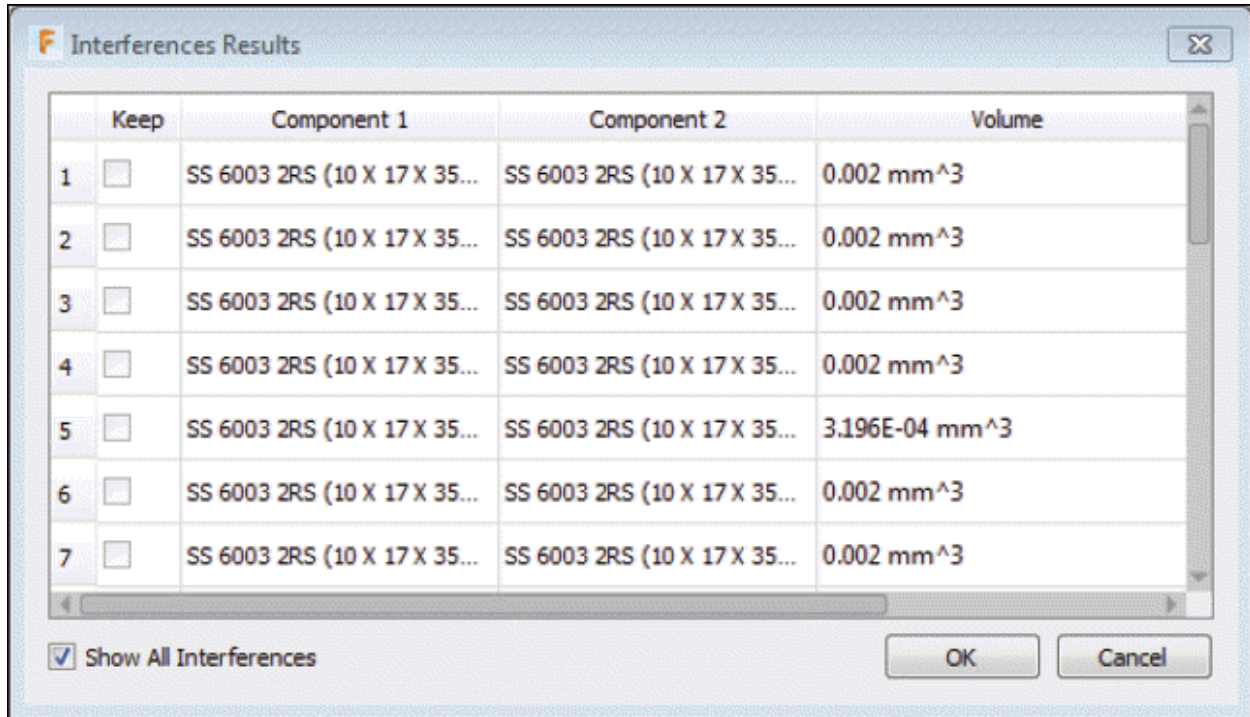


Figure-40. Interference Results dialog box

- Click on the **OK** button from **Interference Results** dialog box to exit the **Interference** tool.

Curvature Comb Analysis

The **Curvature Comb Analysis** tool is used to display a curvature comb on the spline or circular edges to determine the change in curvature. The procedure to use this tool is discussed next.

- Click on the **Curvature Comb Analysis** tool of **INSPECT** drop-down from **Toolbar**; refer to [Figure-41](#). The **CURVATURE COMB ANALYSIS** dialog box will be displayed; refer to [Figure-42](#).

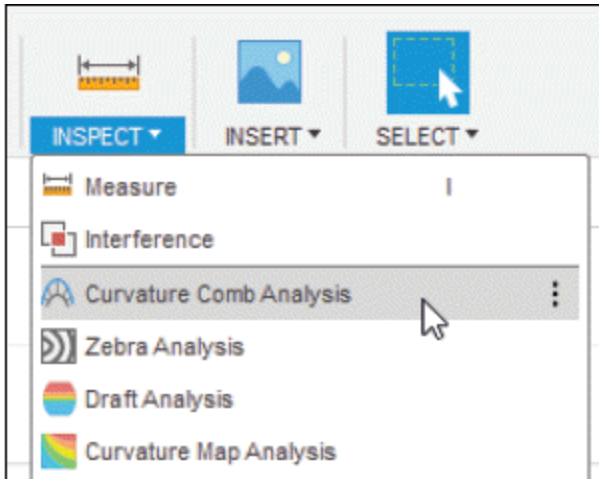


Figure-41. Curvature Comb Analysis tool

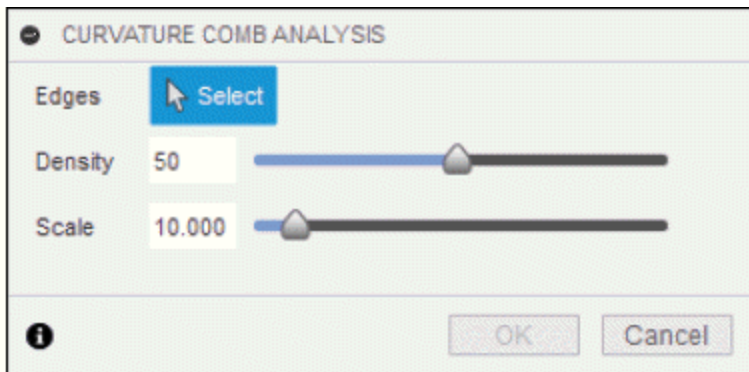


Figure-42. CURVATURE COMB ANALYSIS dialog box

- **Edges** selection of **CURVATURE COMB ANALYSIS** dialog box is active by default. Click on the geometry to select the edge. The preview of comb on edge will be displayed; refer to [Figure-43](#).

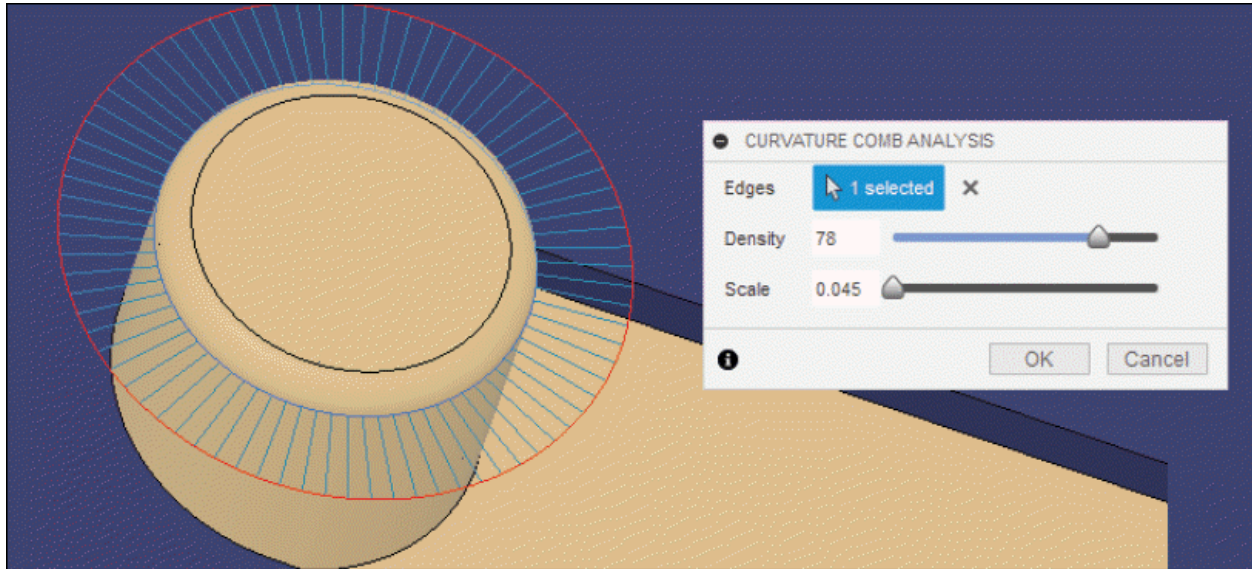


Figure-43. Preview of comb on circular edge

- Click in the **Density** edit box and specify the value. You can also set the comb density by moving the **Density** slider.
- Click in the **Scale** edit box and specify the value. You can also set the comb scale by moving the **Scale** slider.
- After analyzing, click on **OK** button from **CURVATURE COMB ANALYSIS** dialog box to exit the tool.

Zebra Analysis

The **Zebra Analysis** tool is used to analyze continuity between the surfaces of component by displaying stripes on model. The procedure to use this tool is discussed next.

- Click on the **Zebra Analysis** tool of **INSPECT** drop-down from **Toolbar**; refer to [Figure-44](#). The **ZEBRA ANALYSIS** dialog box will be displayed; refer to [Figure-45](#).

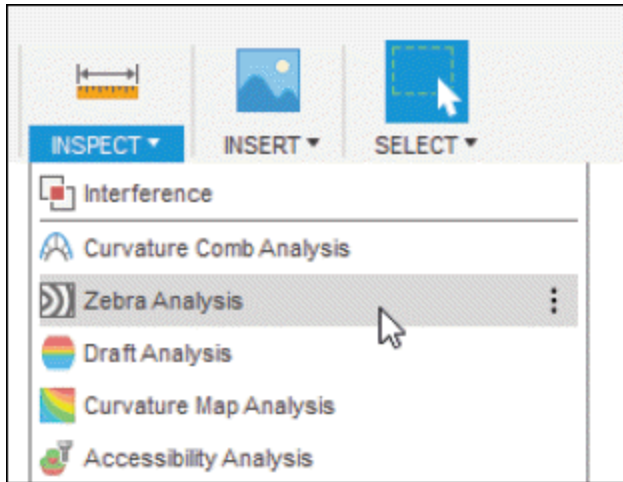


Figure-44. Zebra Analysis tool

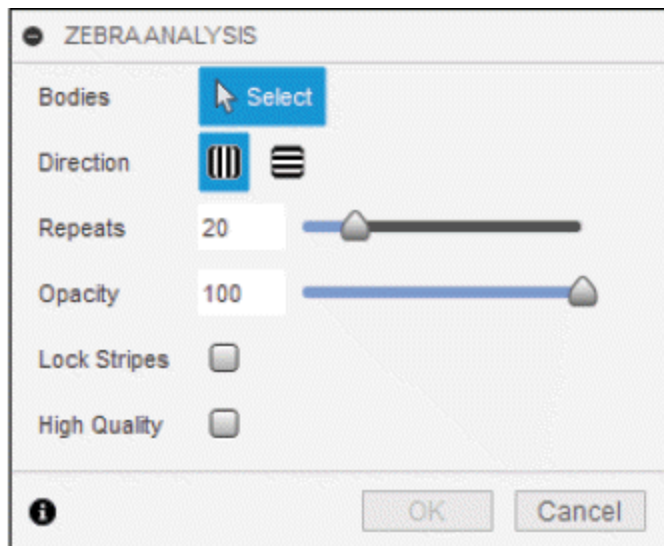


Figure-45. ZEBRA ANALYSIS dialog box

- The **Select** button of **Bodies** section is active by default. Click to select the body for analysis.
- On selecting the body, preview of zebra stripes will be displayed; refer to [Figure-46](#).

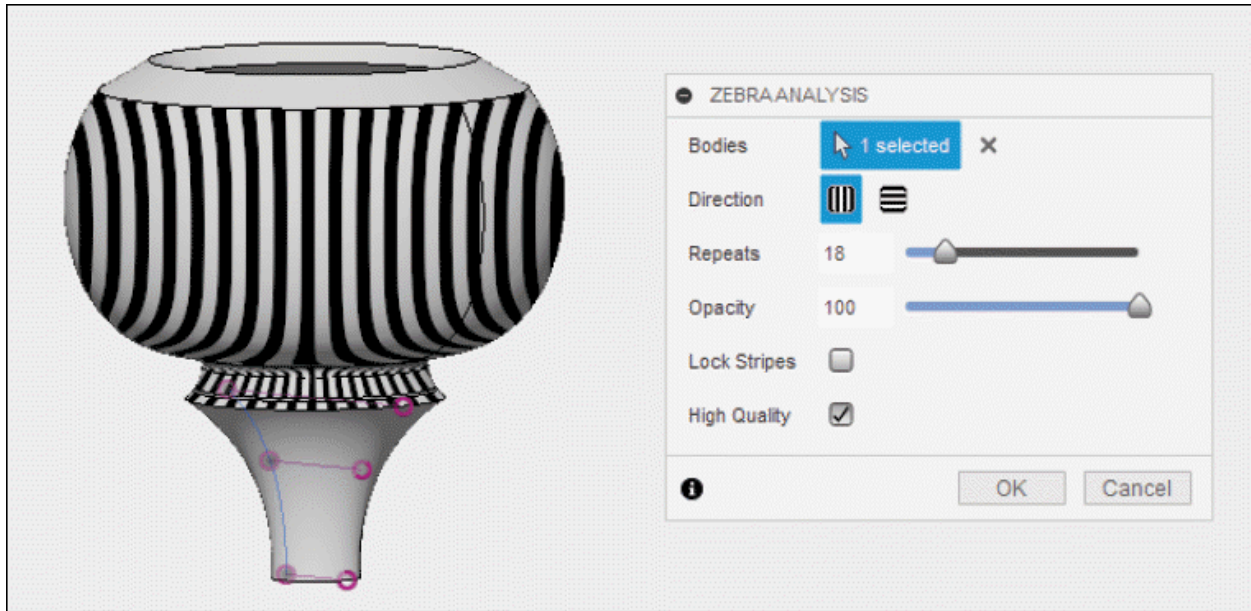


Figure-46. Selection of body for zebra analysis

- Select **Vertical** option from **Direction** section for display of vertical lines on model.
- Select **Horizontal** option from **Direction** section for display of horizontal lines on model.
- Set the number of stripes for model from **Repeats** slider.
- Click on the **Opacity** edit box and enter the value for opacity. You can also set the value of opacity by moving the **Opacity** slider.
- Select the **Lock Stripes** check box of **ZEBRA ANALYSIS** dialog box to freeze the stripes on body.
- Select the **High Quality** check box of **ZEBRA ANALYSIS** dialog box to increase the display quality of stripes.
- After specifying the parameters, click on **OK** button from **ZEBRA ANALYSIS** dialog box. The stripes will be

displayed on model; refer to [Figure-47](#).



Figure-47. Strips on model

Draft Analysis

The **Draft Analysis** tool is used to display a color gradient on the selected body to evaluate draft angle of various faces of the part. The procedure to use this tool is discussed next.

- Click on the **Draft Analysis** tool of **INSPECT** drop-down from **Toolbar**; refer to [Figure-48](#). The **DRAFT ANALYSIS** dialog box will be displayed; refer to [Figure-49](#).

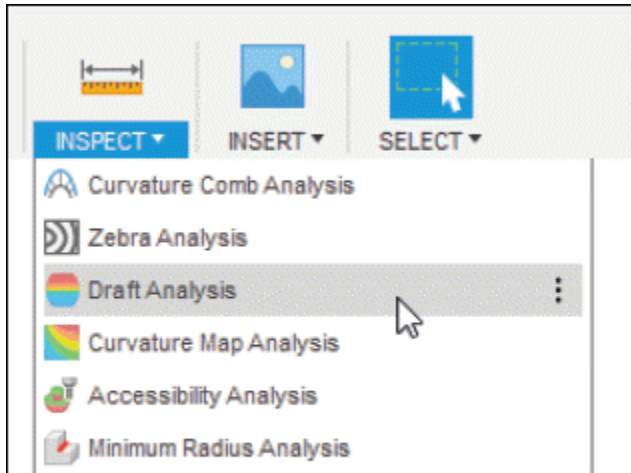


Figure-48. Draft Analysis tool

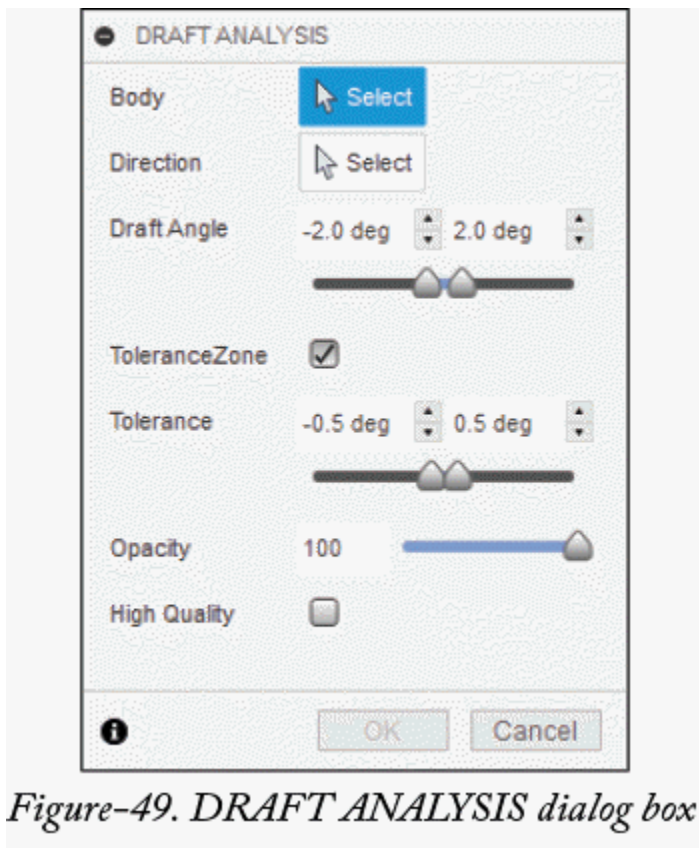


Figure-49. DRAFT ANALYSIS dialog box

- The **Select** button of **Body** section is active by default. Click on the body to perform analysis.
- Click on **Select** button of **Direction** section and select the direction for analysis. On selection, the preview will be

displayed; refer to [Figure-50](#).

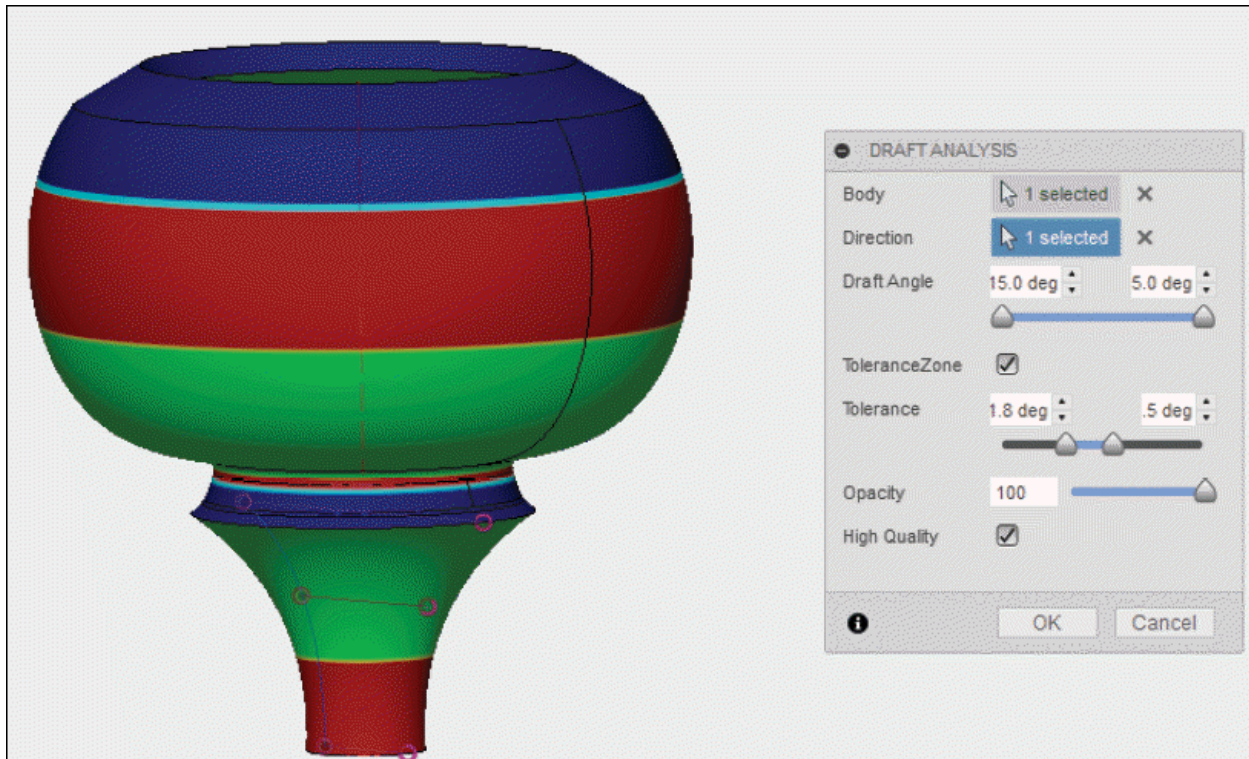


Figure-50. Preview of draft analysis

- Click in **Draft Angle** edit boxes and specify the range of draft angle for which you want to perform analysis. You can also set the draft angle limits by moving the **Draft angle** slider.
- Select the **ToleranceZone** check box if you want to display a tolerance gradient on the results.
- Click on the **Tolerance** edit box and specify the start and stop angles tolerance. You can also set the tolerance angle by moving the slider.
- Click on the **Opacity** edit box and enter the value for opacity for draft colors. You can also set the value of

opacity by moving the **Opacity** slider.

- Select the **High Quality** check box of **DRAFT ANALYSIS** dialog box to increase the display quality of gradient.
- After specifying the parameters for analysis, click on **OK** button from **DRAFT ANALYSIS** dialog box to complete the analysis.

Curvature Map Analysis

The **Curvature Map Analysis** tool is used to displays a color gradient on the faces of selected bodies to evaluate the amount of curvature. The procedure to use this tool is discussed next.

- Click on the **Curvature Map Analysis** tool of **INSPECT** drop-down from **Toolbar**; refer to [Figure-51](#). The **CURVATURE MAP ANALYSIS** dialog box will be displayed; refer to [Figure-52](#).

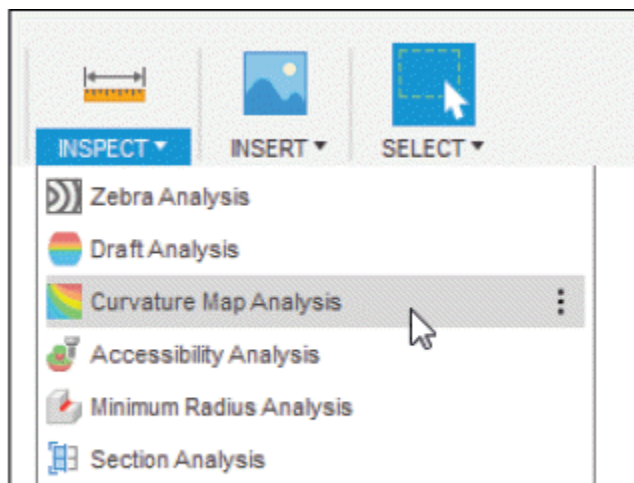


Figure-51. Curvature Map Analysis tool

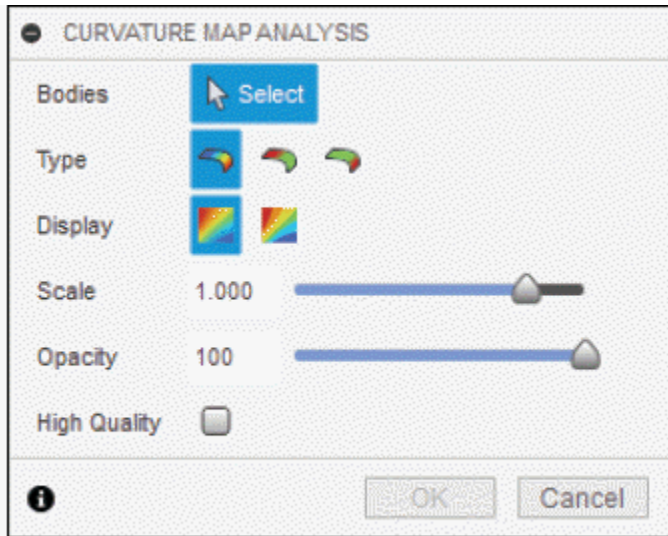


Figure-52. CURVATURE MAP ANALYSIS dialog box

- The **Select** button of **Bodies** section is active by default. Click on the body to select for analysis.
- Select **Gaussian** option from **Type** section if you want to display the gradient as the product of curvature in the U direction and curvature in the V direction.
- Select **Principal Minimum** option from **Type** section if you want to display the gradient highlighting areas of lowest curvature.
- Select **Principal Maximum** option from **Type** section if you want to display the gradient highlighting areas with the highest curvature.
- Select **Smooth** option from **Display** section if you want to display a blend of two colors.
- Select **Bands** option from **Display** section if you want to display no transition between colors.

- Click in the **Scale** edit box of **CURVATURE MAP ANALYSIS** dialog box and enter the value. You can also set the value of scale by moving the **Scale** slider.
- Click on the **Opacity** edit box and enter the value for opacity. You can also set the value of opacity by moving the **Opacity** slider.
- Select the **High Quality** check box of **CURVATURE MAP ANALYSIS** dialog box to increase the display quality of gradient.
- After specifying the parameters for analysis, click on **OK** button from **CURVATURE MAP ANALYSIS** dialog box to exit this analysis; refer to [Figure-53](#).

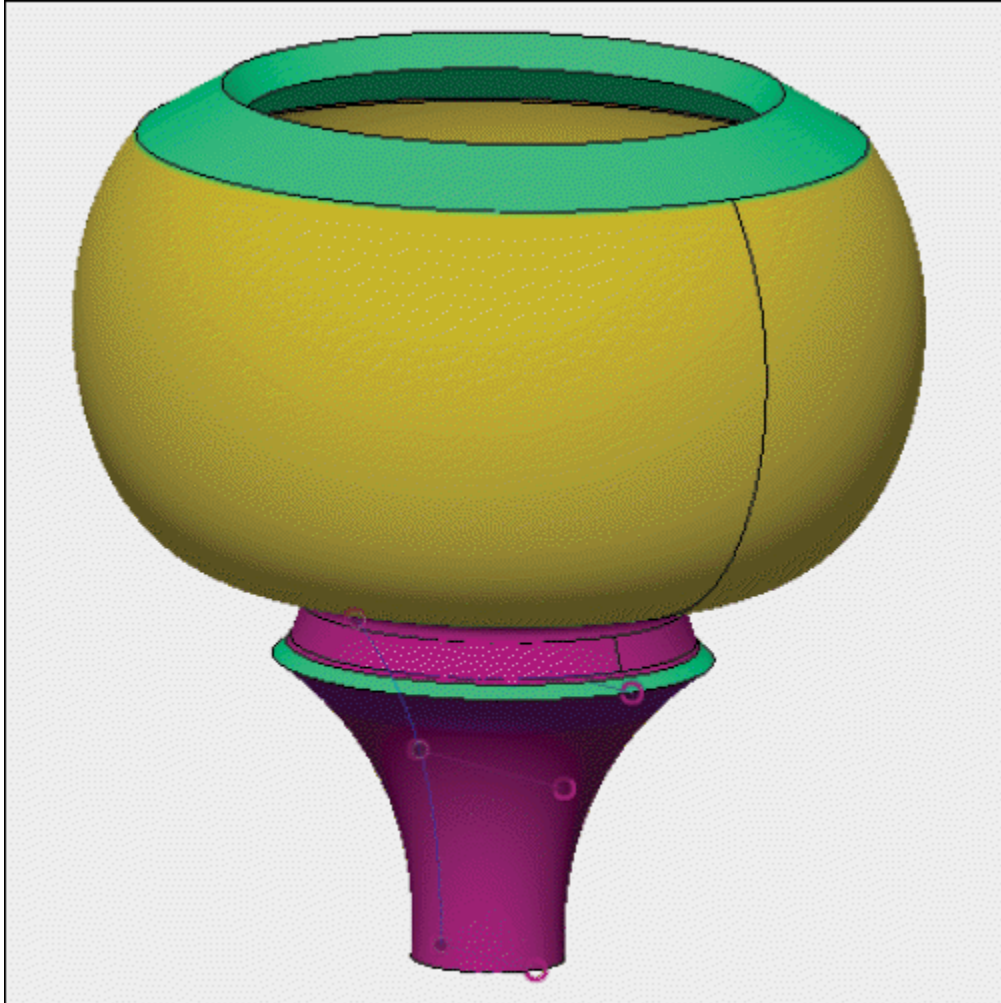


Figure-53. Curvature map analysis of model

Isocurve Analysis

The **Isocurve Analysis** tool is used to analyze the quality of the curvature of a surface by applying UV mapping and curvature combs in Fusion 360. The procedure to use this tool is discussed next.

- Click on the **Isocurve Analysis** tool of **INSPECT** drop-down from **Toolbar**; refer to [Figure-54](#). The **ISOCURVE ANALYSIS** dialog box will be displayed; refer to [Figure-55](#).

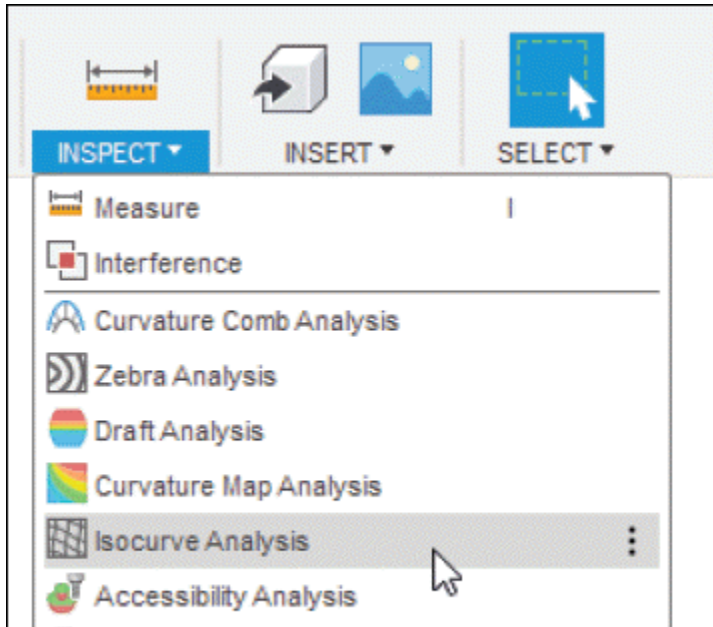


Figure-54. Isocurve Analysis tool

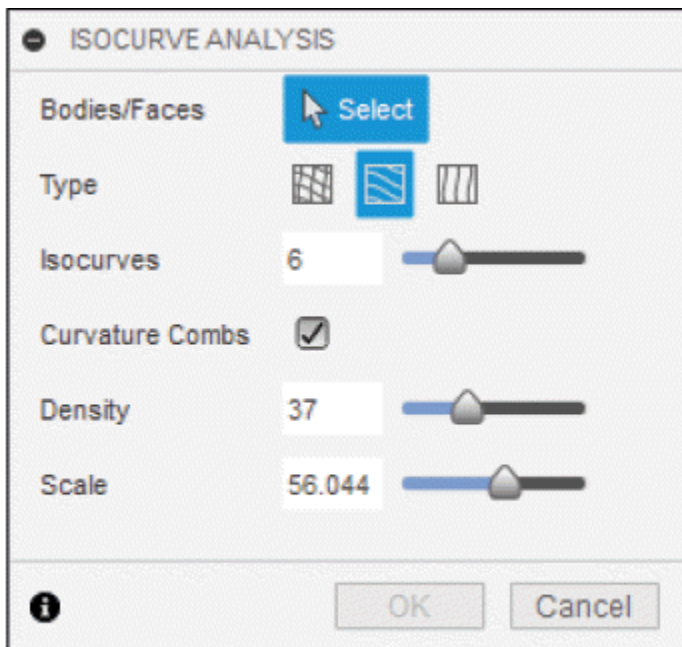


Figure-55. ISOCURVE ANALYSIS dialog box

- The **Bodies/Faces** selection button is active by default. Select the body or face which you want to analyze; refer to [Figure-56](#).

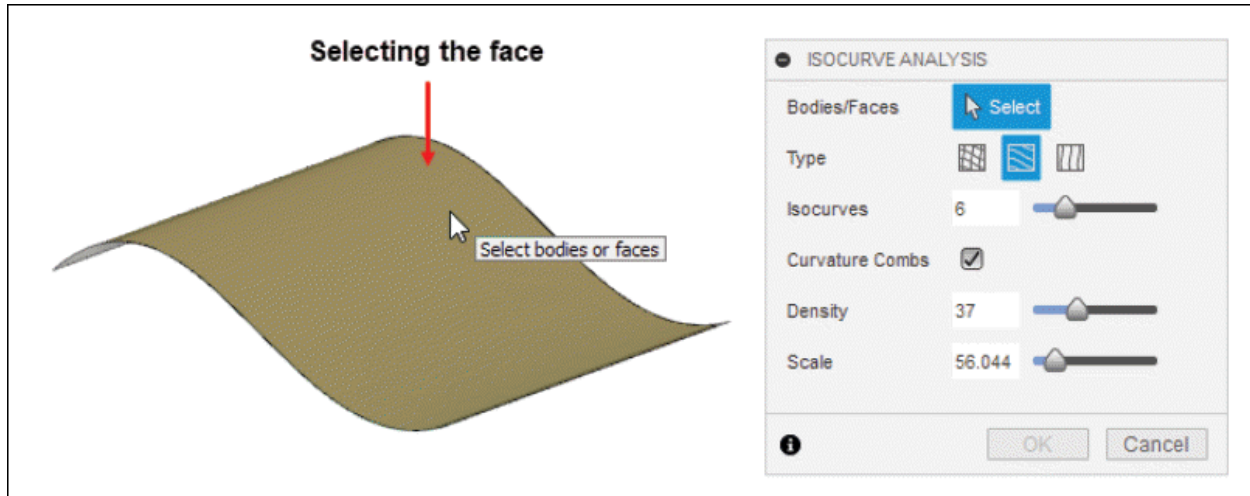


Figure-56. Selecting the face to analyze

- Select desired isocurve type from **Type** section of the dialog box. Select the **U and V** option to display isocurves in U and V directions. Select the **U** option to display isocurves in the U direction. Select the **V** option to display isocurves in the V direction.
- Specify number of isocurves to display on selected bodies or faces in the **Isocurves** edit box. You can also set the number of isocurves by moving the **Isocurves** slider.
- Select **Curvature Combs** check box to display curvature combs on isocurves.
- Specify number of teeth to display on curvature combs in the **Density** edit box. You can also set the number of teeth by moving the **Density** slider.
- Specify scale of curvature combs in the **Scale** edit box. You can also set the value of scale by moving the **Scale** slider.
- After specifying desired parameters for analysis, click on

the **OK** button from the dialog box to exit the analysis; refer to [Figure-57](#).

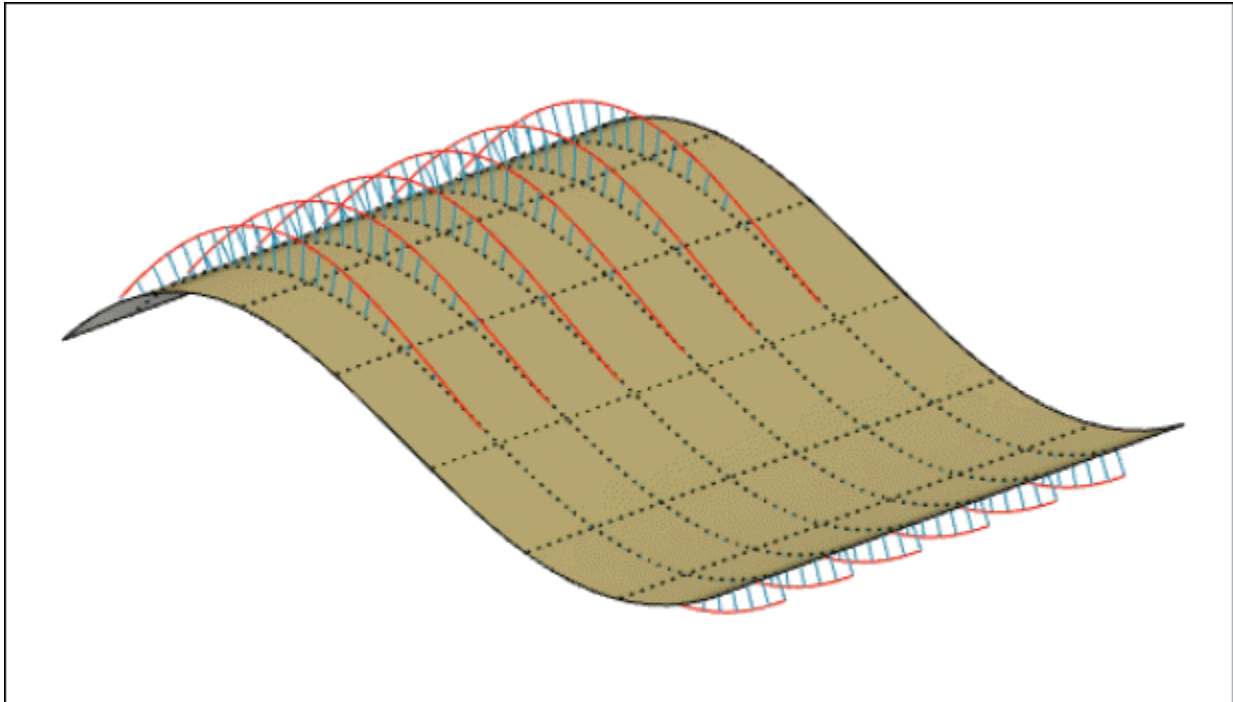


Figure-57. Isocurve analysis of the body

Accessibility Analysis

The **Accessibility Analysis** tool is used to determine whether the area of the model is accessible or inaccessible through selected view of the plane. This analysis is used to perform undercut analysis. The procedure to use this tool is discussed next.

- Click on the **Accessibility Analysis** tool of **INSPECT** drop-down from **Toolbar**; refer to [Figure-58](#). The **ACCESSIBILITY ANALYSIS** dialog box will be displayed; refer to [Figure-59](#).

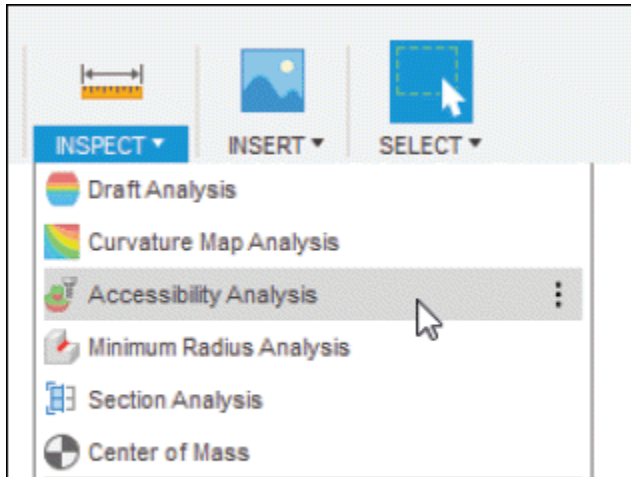


Figure-58. Accessibility Analysis tool

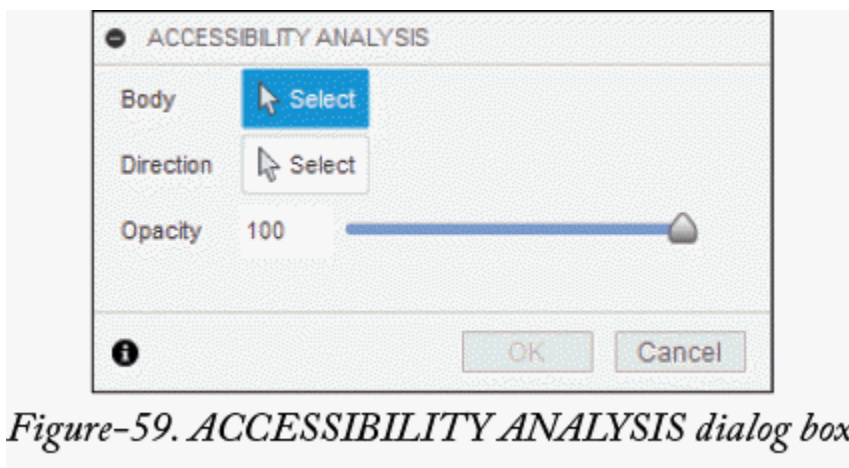


Figure-59. ACCESSIBILITY ANALYSIS dialog box

- The **Select** button of **Body** section is active by default. Click on the face of the body to select; refer to [Figure-60](#). You will be asked to select a direction.

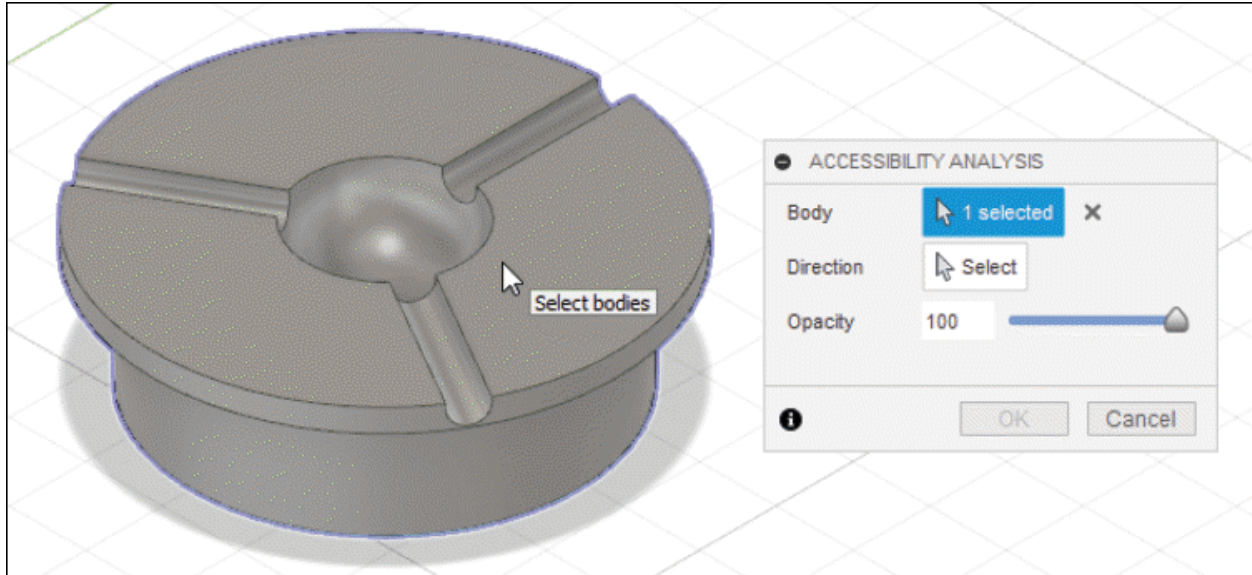


Figure-60. Selecting the body

- Click on the **Select** button of **Direction** section and select the edge, plane, or axis of the model to indicate the direction of analysis. The preview of the accessible areas will be displayed along with updated **ACCESSIBILITY ANALYSIS** dialog box; refer to [Figure-61](#).

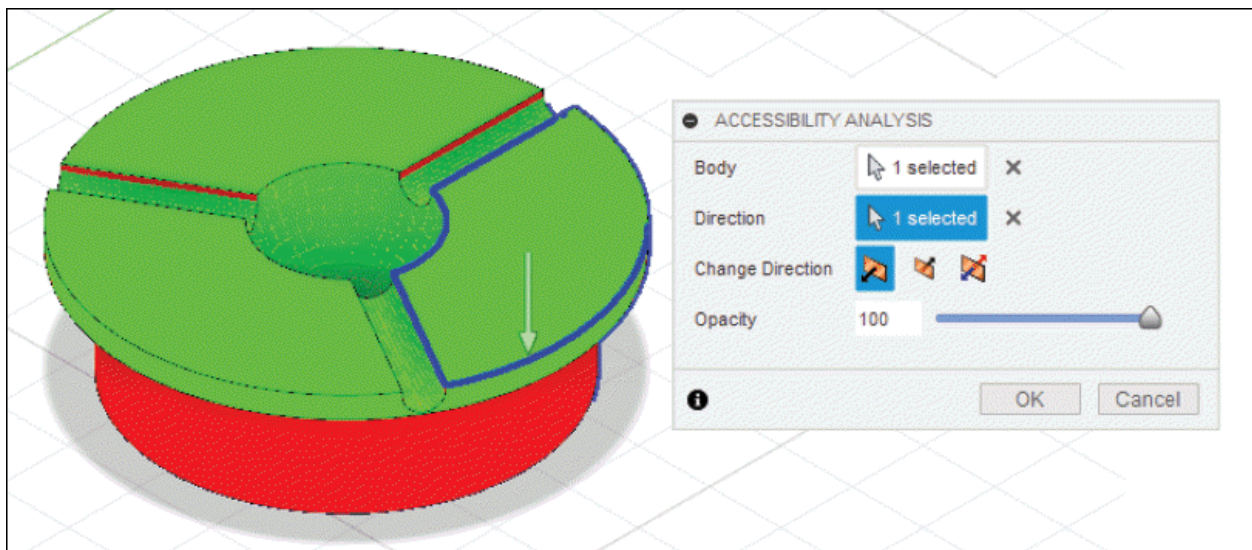


Figure-61. Updated ACCESSIBILITY ANALYSIS dialog box

- On selecting the direction, the areas that are accessible

from selected direction are shaded in green and the areas that are inaccessible are shaded in red.

- Select desired option from **Change Direction** section to change the direction of analysis.
- You can also set the opacity by sliding the slider button or edit the opacity in edit box of **Opacity** section.
- The visibility of this analysis can be controlled by expanding the **Analysis** folder in the **BROWSER** and clicking on the eye button to view or hide the analysis; refer to [Figure-62](#).

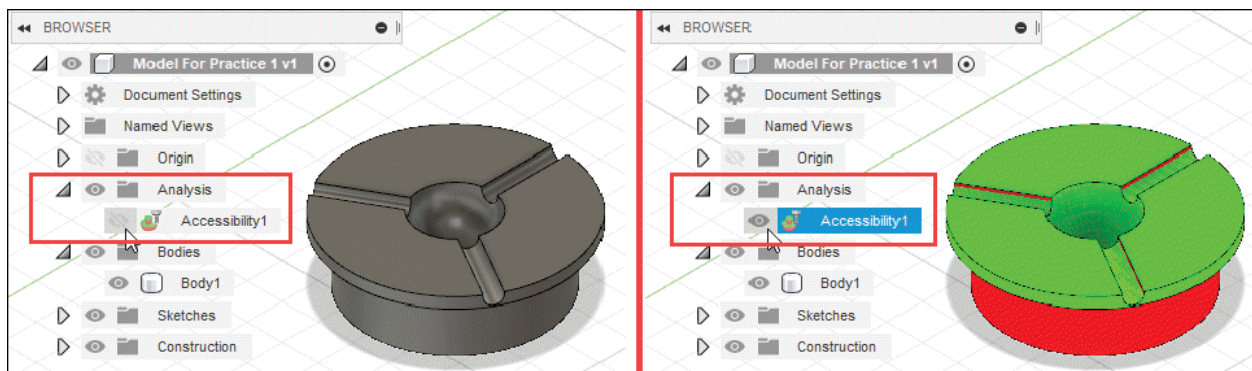


Figure-62. Visibility of analysis

- After specifying the parameters for analysis, click on **OK** button from **ACCESSIBILITY ANALYSIS** dialog box to exit this analysis.

Minimum Radius Analysis

The **Minimum Radius Analysis** tool is used to highlight the areas of the model which are at or below the minimum radius. It also detects the minimum fit for fillets and holes. The procedure to use this tool is discussed next.

- Click on the **Minimum Radius Analysis** tool of **INSPECT** drop-down from **Toolbar**; refer to [Figure-63](#). The **MINIMUM RADIUS ANALYSIS** dialog box will be displayed; refer to [Figure-64](#).

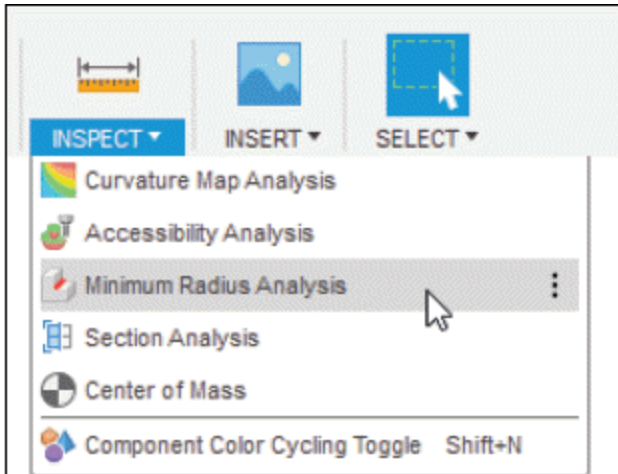


Figure-63. Minimum Radius Analysis tool

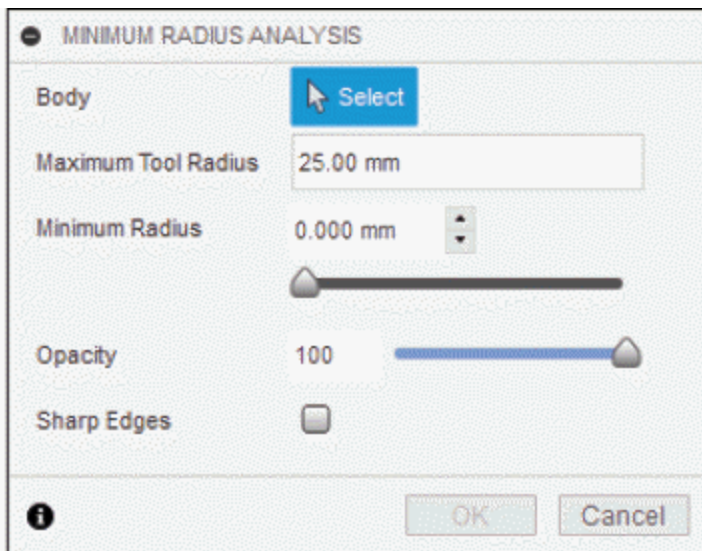


Figure-64. MINIMUM RADIUS ANALYSIS dialog box

- The **Select** button of **Body** section is active by default. Click on the body to be selected; refer to [Figure-65](#).

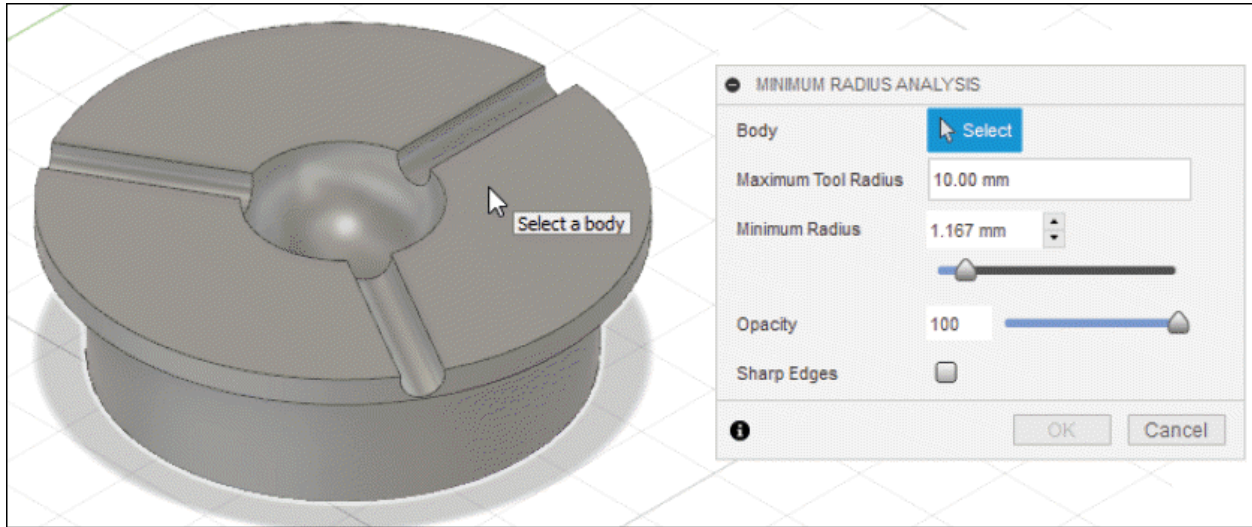


Figure-65. Selection of face of the body

- After selecting the body, set the **Maximum Tool Radius** in the edit box for analysis of the model. For tool of **Minimum Radius 8**, the sections that can be machined are highlighted in green whereas sections with cannot be machined by 8 mm radius tool will be highlighted in red; refer to [Figure-66](#).

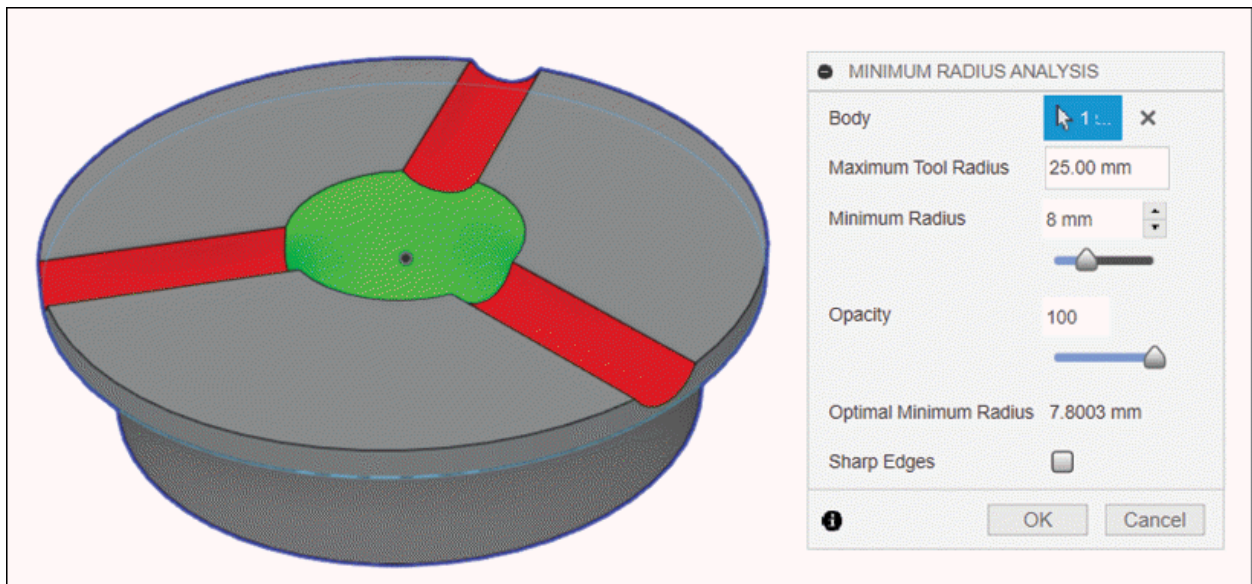


Figure-66. Minimum radius color

- Note that software automatically calculates the optimum minimum radius of tool which can machine most of the sections of model in **Optimal Minimum Radius** field of the dialog box which is 7.8003 mm in our case; refer to [Figure-66](#).
- Set the opacity of the analysis as you did in the **Accessibility Analysis** tool.
- Select the **Sharp Edges** check box to display sharp edges of the model.
- After specifying all the parameters, click on the **OK** button of **MINIMUM RADIUS ANALYSIS** dialog box to exit this analysis.
- The visibility of this analysis can be control by expanding the **Analysis** folder in the **BROWSER** and clicking on the eye button to view or hide the analysis; refer to [Figure-67](#).

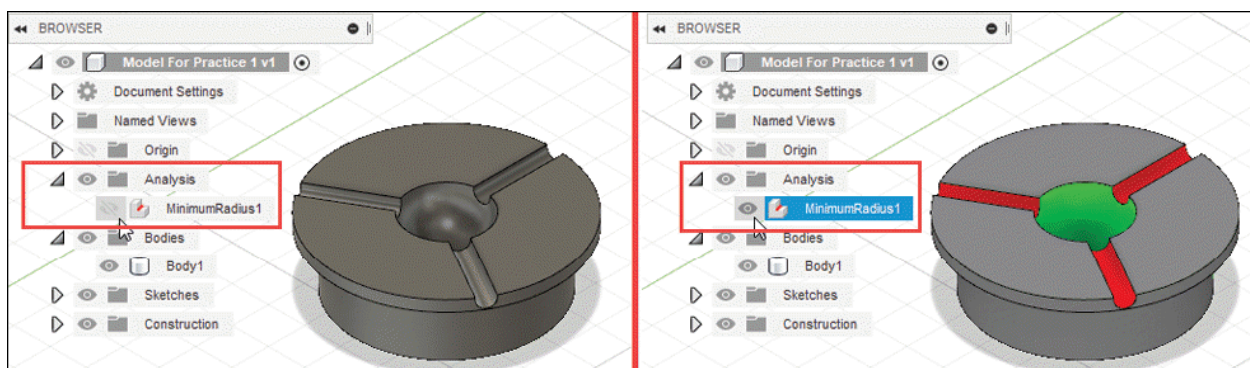


Figure-67. Visibility of minimum radius analysis

Section Analysis

The **Section Analysis** tool is used to cut the model at selected plane reference and check the temporary view. The

procedure to use this tool is discussed next.

- Click on the **Section Analysis** tool of **INSPECT** drop-down from **Toolbar**; refer to [Figure-68](#). The **SECTION ANALYSIS** dialog box will be displayed; refer to [Figure-69](#).

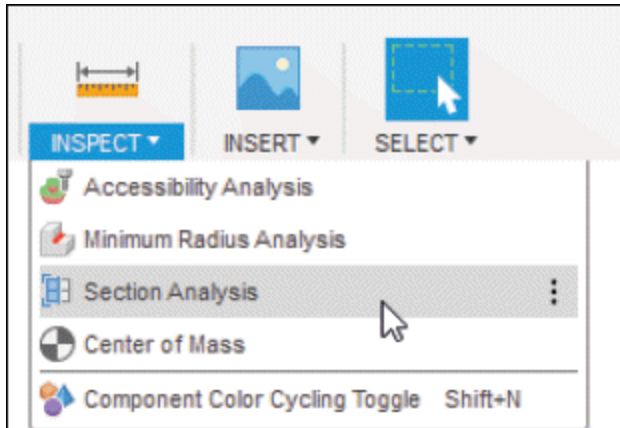


Figure-68. Section Analysis tool

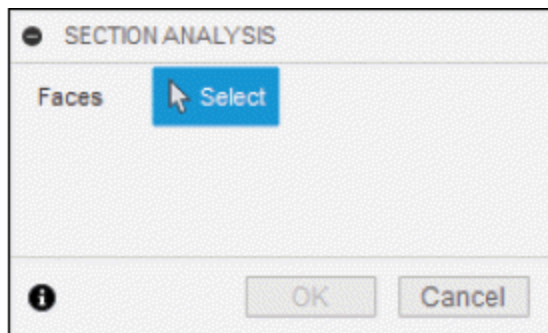


Figure-69. SECTION ANALYSIS dialog box

- The **Select** button of **Faces** section is active by default. Click on the face to select.
- On selection of face, the dialog box will be updated along with the preview of model; refer to [Figure-70](#).

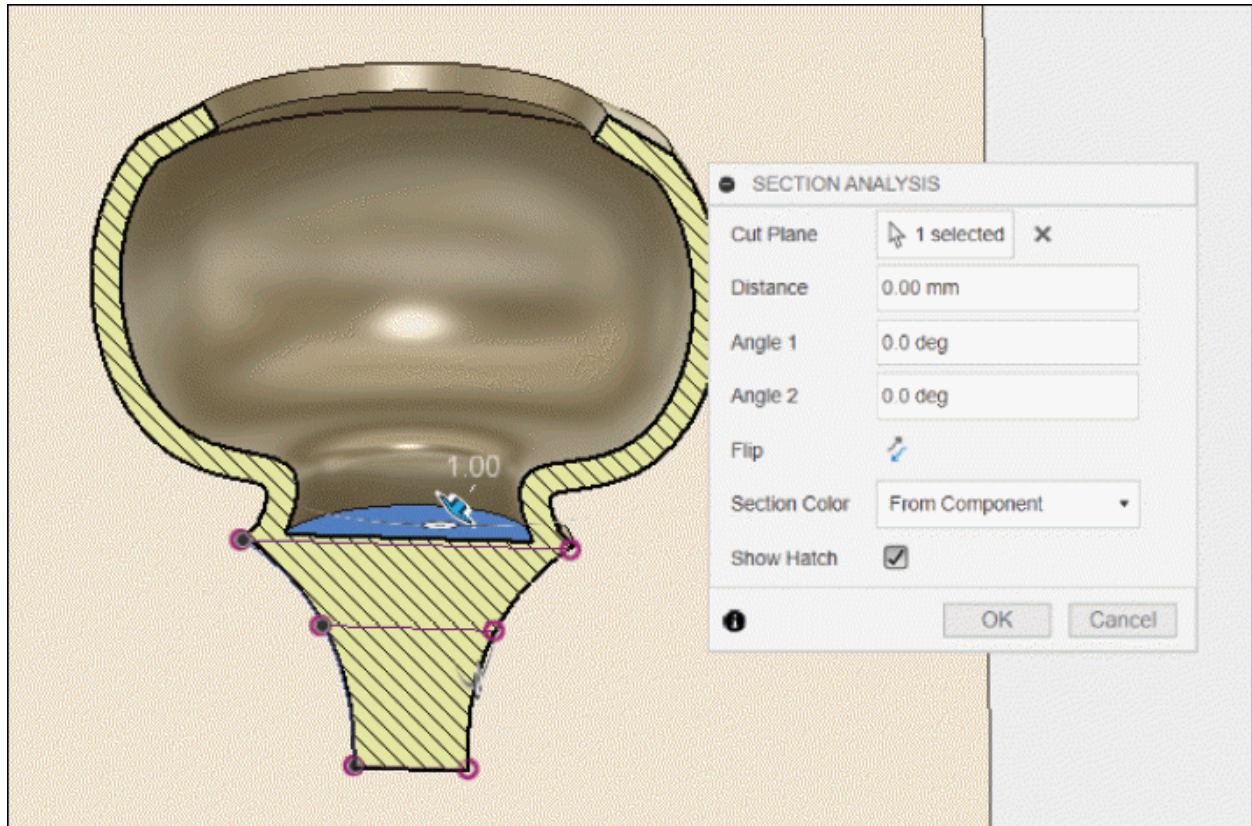


Figure-70. Updated SECTION ANALYSIS dialog box

- Click in the **Distance** edit box and specify the location of the section plane along y direction. You can also set the value by moving the drag handle.
- Click in the **Angle 1** edit box and specify the angle of plane about x direction. You can also set the value by moving the drag handle.
- Click in the **Angle 2** edit box and specify the angle of plane about y direction. You can also set the value by moving the drag handle.
- Click on the **Flip** button to reverse the section side of plane.

- Select **From Component** option from **Section Color** drop-down if you want to use the same color as applied to the model.
- Select **Custom** option from **Section Color** drop-down if you want to use the custom color for the section.
- Select the **Show Hatching** check box if you want to show hatching on the section.
- After specifying the parameters for analysis, click on **OK** button from **SECTION ANALYSIS** dialog box to complete the analysis.

Center of Mass

The **Center of Mass** tool is a center point of body used for representing the mean position of matter in a body. The procedure to use this tool is discussed next.

- Click on the **Center of Mass** tool of **INSPECT** drop-down from **Toolbar**; refer to [Figure-71](#). The **CENTER OF MASS** dialog box will be displayed; refer to [Figure-72](#).

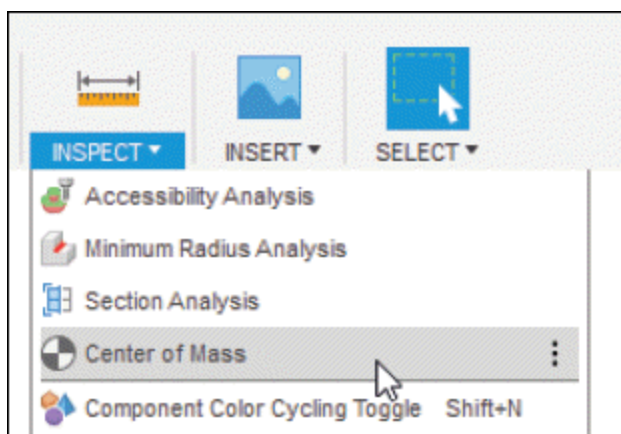


Figure-71. Center of Mass tool

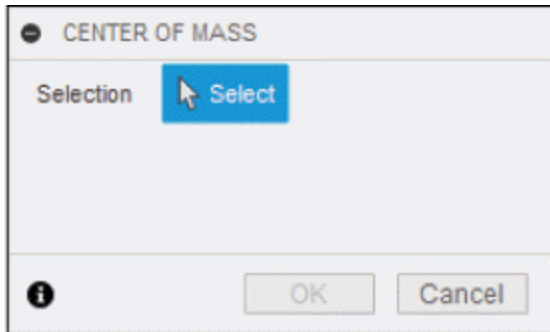


Figure-72. CENTER OF MASS dialog box

- You need to select the body and click on **OK** button from **CENTER OF MASS** dialog box. The center of mass of the body will be displayed; refer to [Figure-73](#).

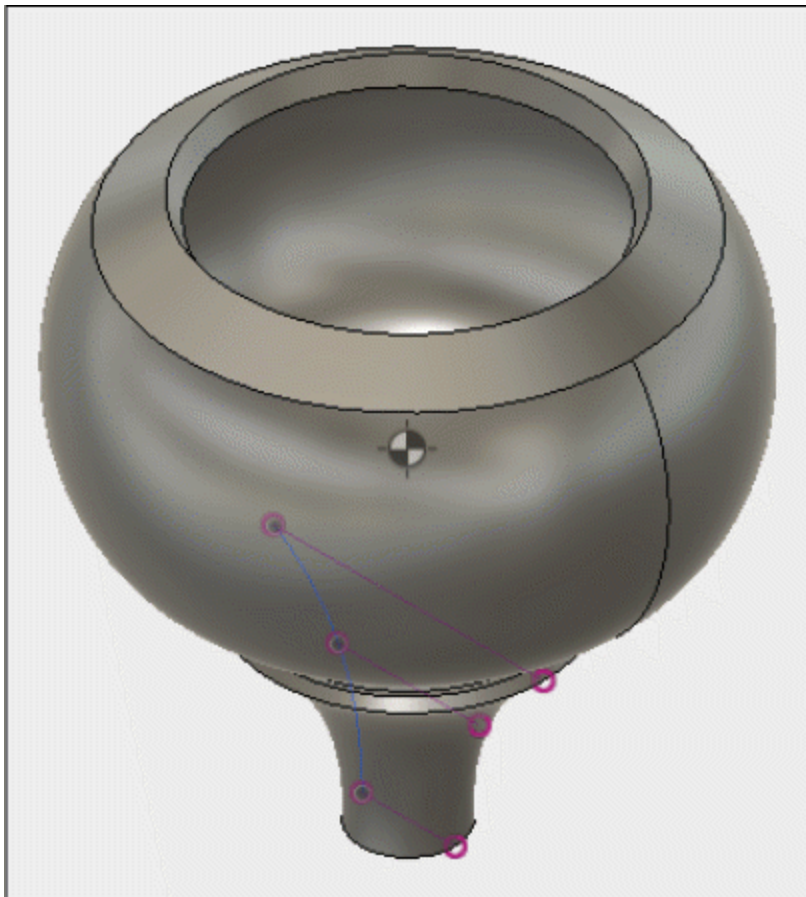


Figure-73. Center of mass of the body

The **Design Advice** tool is used to analyze the part and check its manufacturability using plastic injection modeling process. The procedure to use this tool is given next.

- Open part for which you want to check plastic injection molding recommendations and click on the **Design Advice** tool from the **INSPECT** drop-down of **PLASTIC** tab in the **Toolbar**. The **DESIGN ADVICE** dialog box will be displayed; refer to [Figure-74](#).

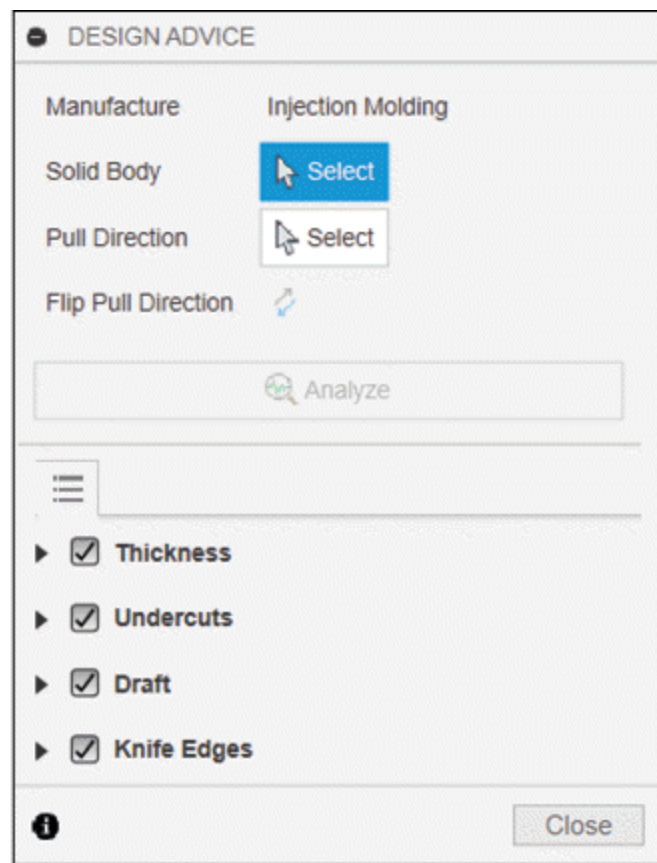


Figure-74. DESIGN ADVICE dialog box

- Select the solid body to be manufactured by injection molding. You will be asked to assign plastic rules if not applied earlier. Assign the rules as discussed earlier.

- Click on the **Select** button for **Pull Direction** and select reference to be used for defining direction of core/cavity pull.
- Click on the **Flip Pull Direction** button to reverse direction of pulling mold plate.
- Select check boxes from the **Summary** section of dialog box and define properties of model to be analyzed for molding; refer to [Figure-75](#).

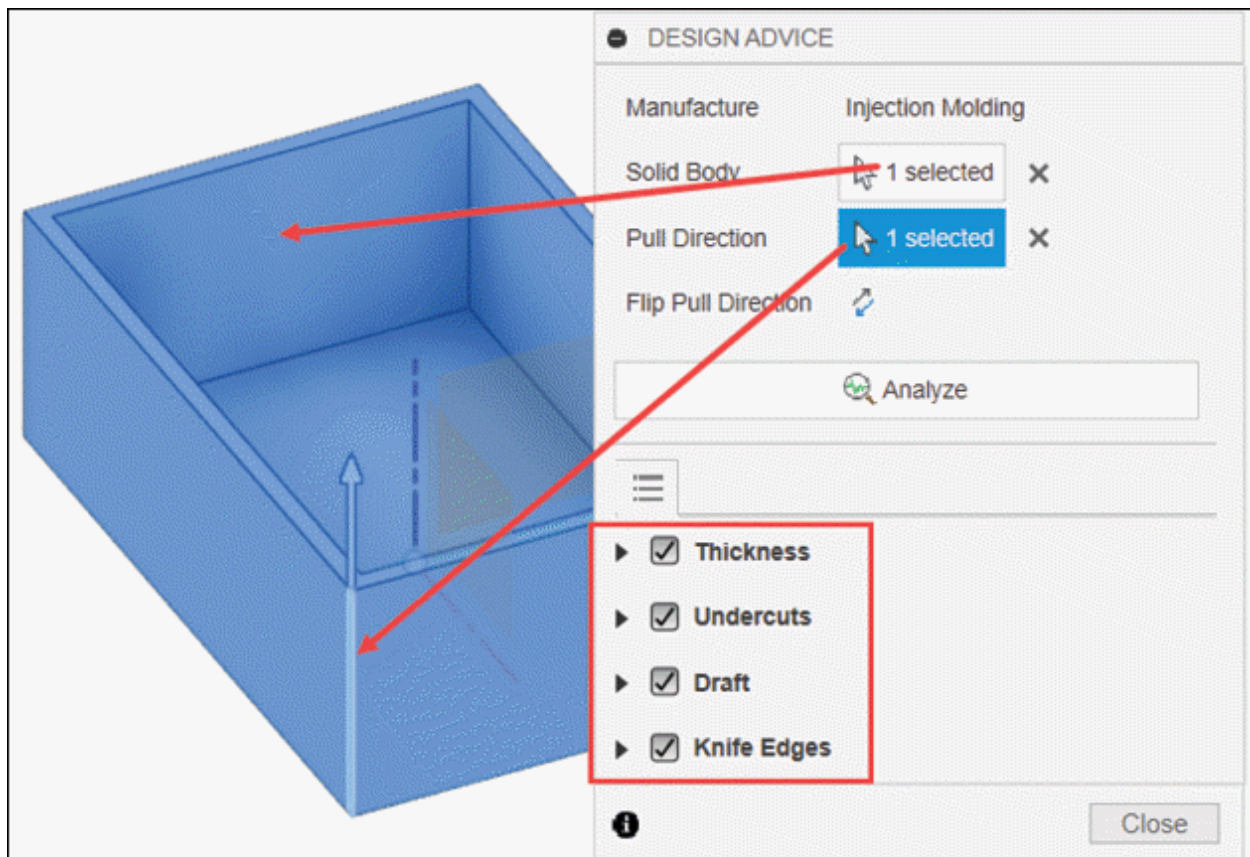


Figure-75. Parameters specified for design advice

- After setting desired parameters, click on the **Analyze** button. The results of analysis will be displayed; refer to [Figure-76](#). Modify the part as per the analysis results and if

there are green tick marks for all the parameters then your part is good to go for manufacturing.

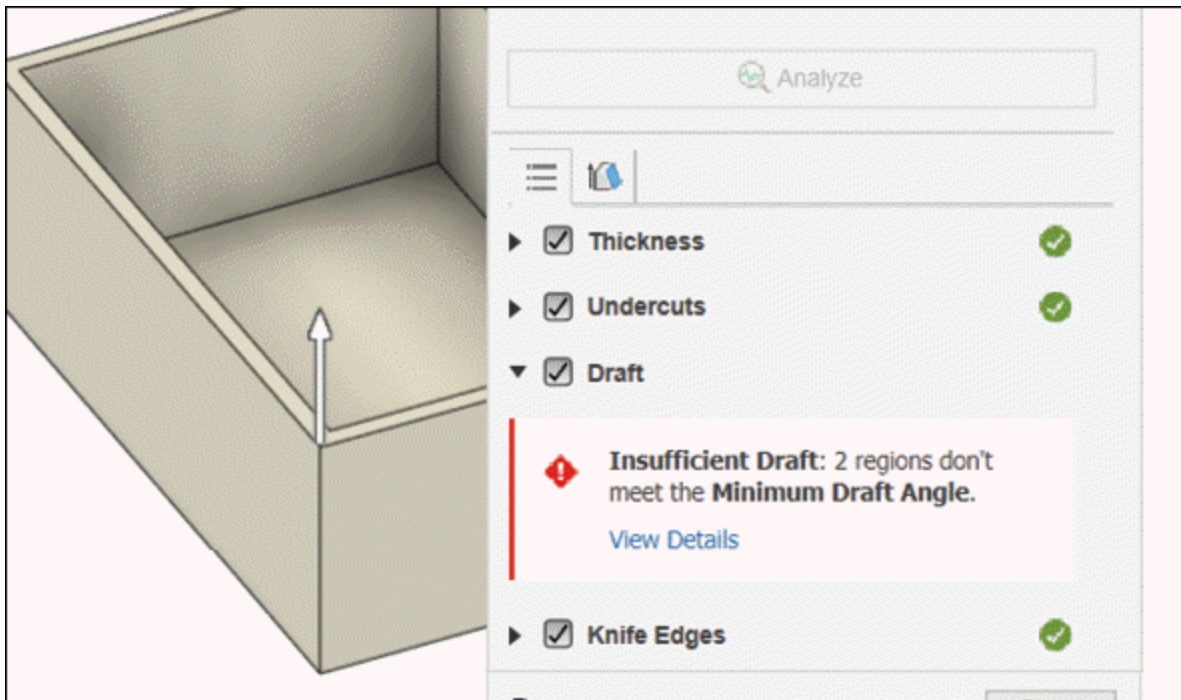


Figure-76. Design advice results

Displaying Component Colors

The **Display Component Colors** tool is used to toggle coloring of various objects in Autodesk Fusion. You can also press **SHIFT+N** to do the same thing.

Similarly, the **Display Mesh Face Groups** tool is used to display mesh face groups with different colors.

SELF ASSESSMENT

Q1. Which of the following tools is used for inserting image on the face of a model?

- a) Insert Mesh
- b) Insert Derive

c) Decal

d) Insert SVG

Q2. What is the use of **Reset Transformation** button in **INSERT MESH** dialog box?

Q3. What is the use of **Insert Mc-Master Carr Component** tool and what is Mc-Master Carr?

Q4. Write the procedure of inserting manufacturer or supplier part in the model.

Q5. Write the procedure to measure the distance between two faces.

Q6. Write the procedure to find interference between two components.

Q7. The **Curvature Comb Analysis** tool is used to display a curvature comb on the spline or circular edges to determine the

Q8. The **Zebra Analysis** tool is used to analyze continuity between the surfaces of component byon model.

Q9. Write the procedure to display zebra stripes on body.

10. The **Draft Analysis** tool is used to display a color gradient on the selected body to of various faces of the part.

11. The tool is used to displays a color gradient on the faces of selected bodies to evaluate the amount of

curvature.

12. What is the use of **Gaussian** option in **Curvature Map Analysis**?

13. The **Section Analysis** tool is used to define the view of component from side.

14. Which of the following analyses uses **ToleranceZone** option to display a tolerance gradient on the results?

- a) Section Analysis
- b) Zebra Analysis
- c) Curvature Comb Analysis
- d) Draft Analysis

Q15. The tool is used to analyze the quality of the curvature of a surface by applying UV mapping and curvature combs in Fusion 360.

Chapter 9

Surface Modeling

Topics Covered

The major topics covered in this chapter are:

- ***Introduction to Surface Designing***
- ***Extrude Tool***
- ***Patch Tool***
- ***Surface Trim and Untrim Tool***
- ***Extend Tool***
- ***Stitch and Unstitch Tool***
- ***Reverse Normal Tool***
- ***Practical***
- ***Practice***

INTRODUCTION

Surface Designing or surfacing is a technique used to create complex shapes. The most general application of surfacing can be seen in the object that are made with good aerodynamics like car body, aeroplanes, ships, and so on. Sometimes, surface modeling is also used to add complex geometry to the models. The models created by surfacing

are called surface models and they do not exist in real world because they have zero thickness. After converting these surfaces to solids, we make real objects.

In Autodesk Fusion 360, there is no separate workspace for creating surface designs. We have all the surfacing tools available in the **SURFACE** tab of **DESIGN** workspace; refer to [Figure-1](#).

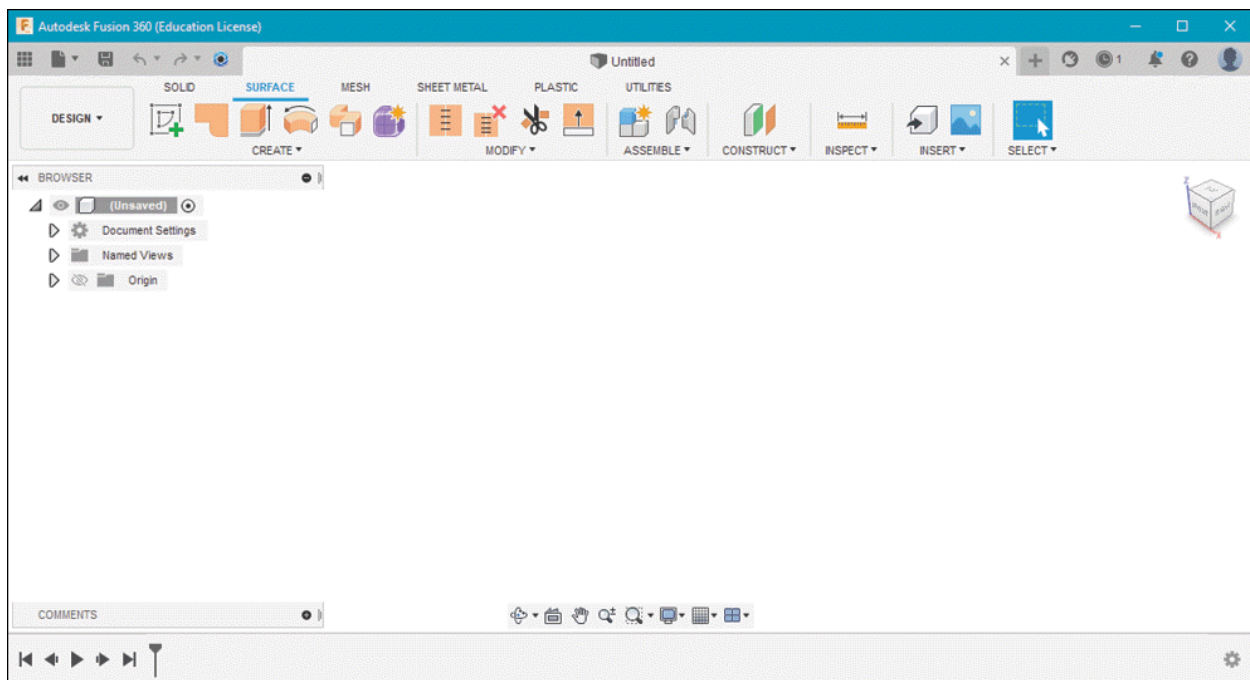


Figure-1. SURFACE modeling tools

Note that we have already used many tools of surface design earlier while creating solid models. Various surface modeling tools are discussed next.

Extrude

The **Extrude** tool can also be used to create the extruded surfaces. The procedure to use this tool is discussed next.

- Click on the **Extrude** tool of **CREATE** drop-down from

Toolbar in **SURFACE** tab; refer to [Figure-2](#). The **EXTRUDE** dialog box will be displayed; refer to [Figure-3](#).

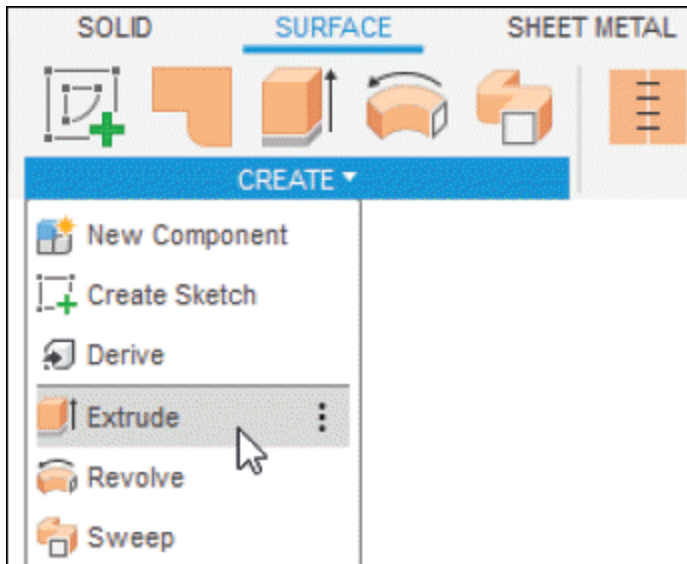


Figure-2. Extrude tool

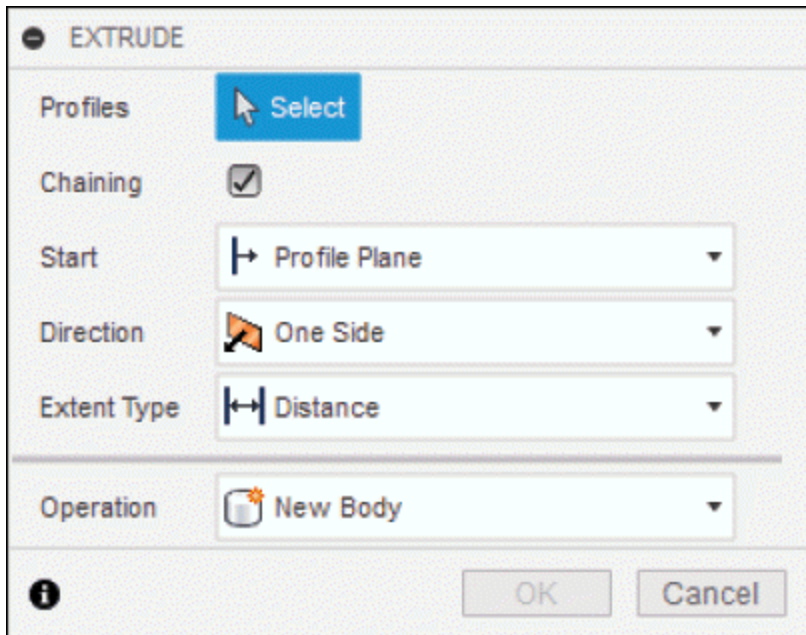


Figure-3. EXTRUDE dialog box

- The **Select** button of **Profiles** section is active by default. Click on the profile to select. The updated **EXTRUDE**

dialog box will be displayed; refer to [Figure-4](#).

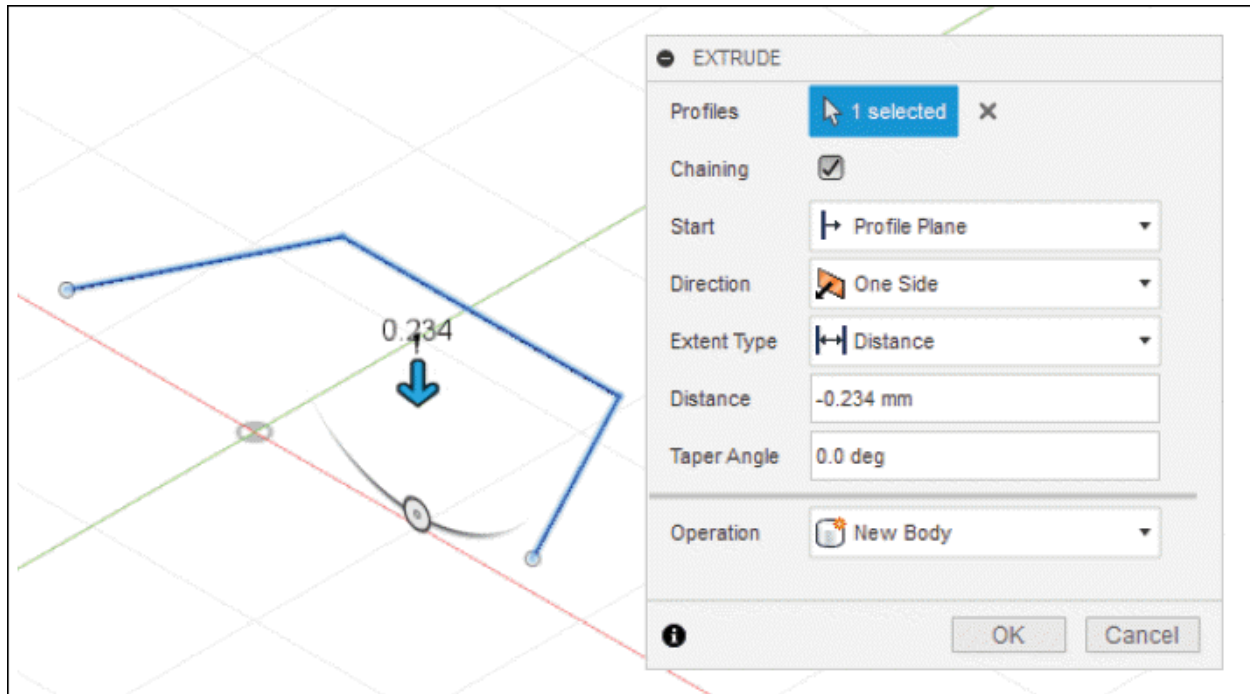


Figure-4. Updated EXTRUDE dialog box

Note that you can select open as well as closed sketch for profile selection to extrude.

- Select the **Chaining** check box if you want to select all the sketch entities in chain.
- The other options of **EXTRUDE** dialog box are same as discussed earlier for solid modeling.
- After specifying the parameters, preview of extrude will be displayed; refer to [Figure-5](#).

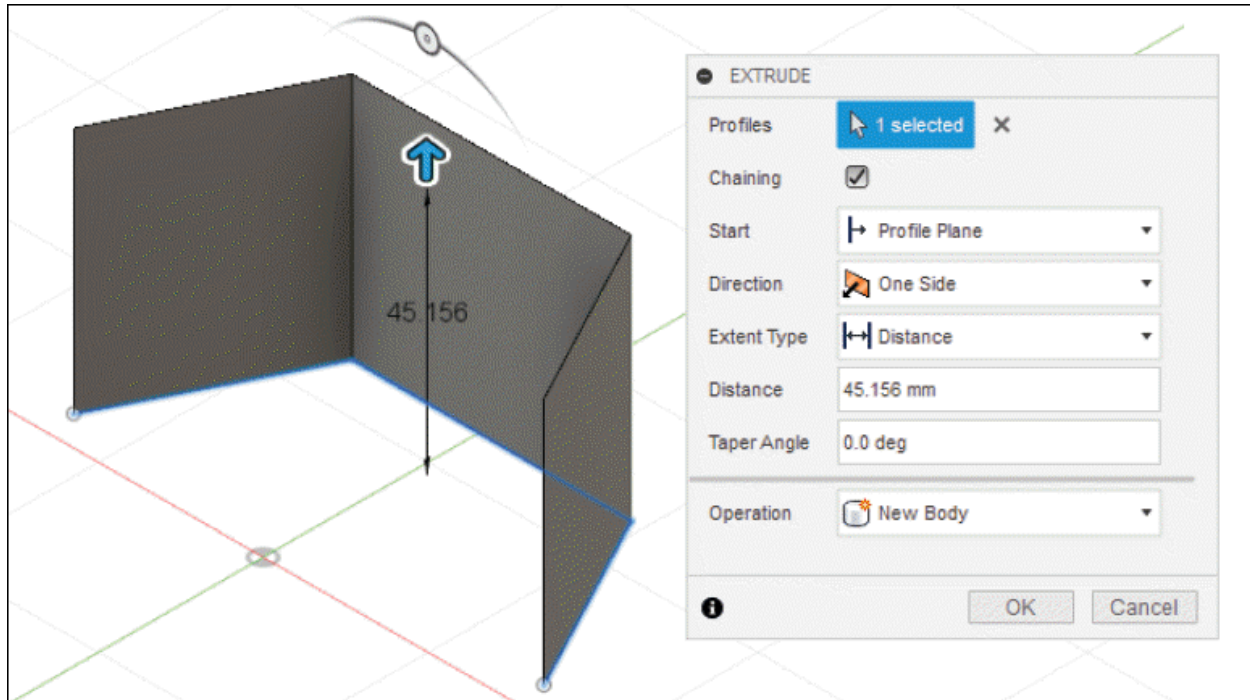


Figure-5. Preview of extrude

- If the preview of extrude is as required then click on **OK** button from **EXTRUDE** dialog box. The extruded surface will be created.

You can use the **Revolve**, **Sweep**, and **Loft** tools in the same way as discussed in previous chapters.

Patch

The **Patch** tool is used to create a surface by connecting edges/curves. The procedure to use this tool is discussed next.

- Click on the **Patch** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-6](#). The **PATCH** dialog box will be displayed; refer to [Figure-7](#).

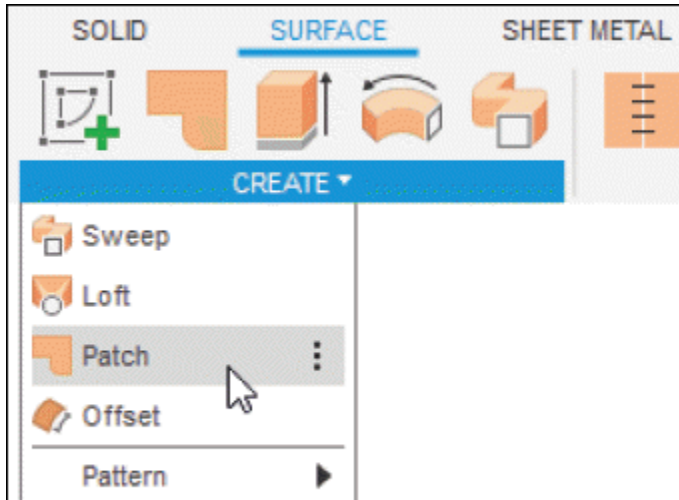


Figure-6. Patch tool

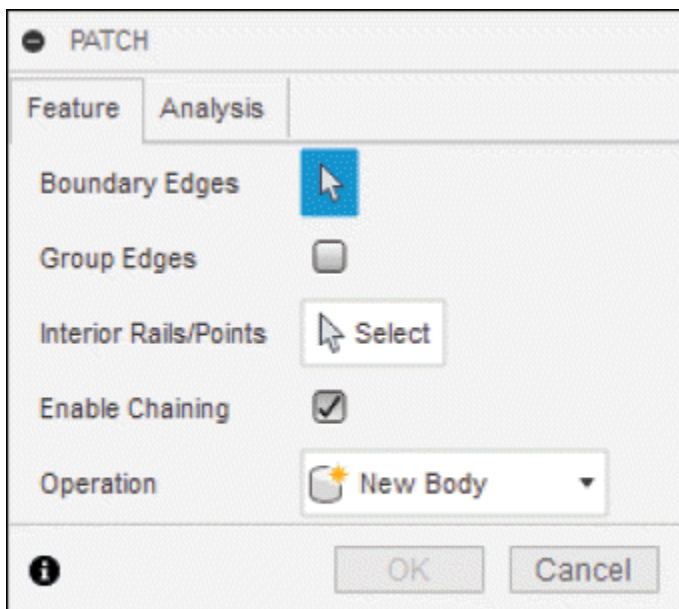


Figure-7. PATCH dialog box

- The **Select** button of **Boundary Edges** section is active by default. Click on the edges/curves forming close loop.
- Select the **Group Edges** check box if you want to select all the edges in a group.
- Select the **Enable Chaining** check box if you want to

select all the adjacent edges when clicking on one edge.

- Select **Connected** option from **Continuity** drop-down if you want to create a surface with G0 edges (connected at an angle).
- Select **Tangent** option from **Continuity** drop-down if you want to create a surface with G1 edges (tangential).
- Select **Curvature** option from **Continuity** drop-down, if you want to create a surface with G2 edges (blended with continuous curvature).
- Click on the **Select** button for **Interior Rails/Points** selection and select the curves/points to be used as guide rails/points.
- After specifying the parameters, the preview of patch will be displayed; refer to [Figure-8](#).

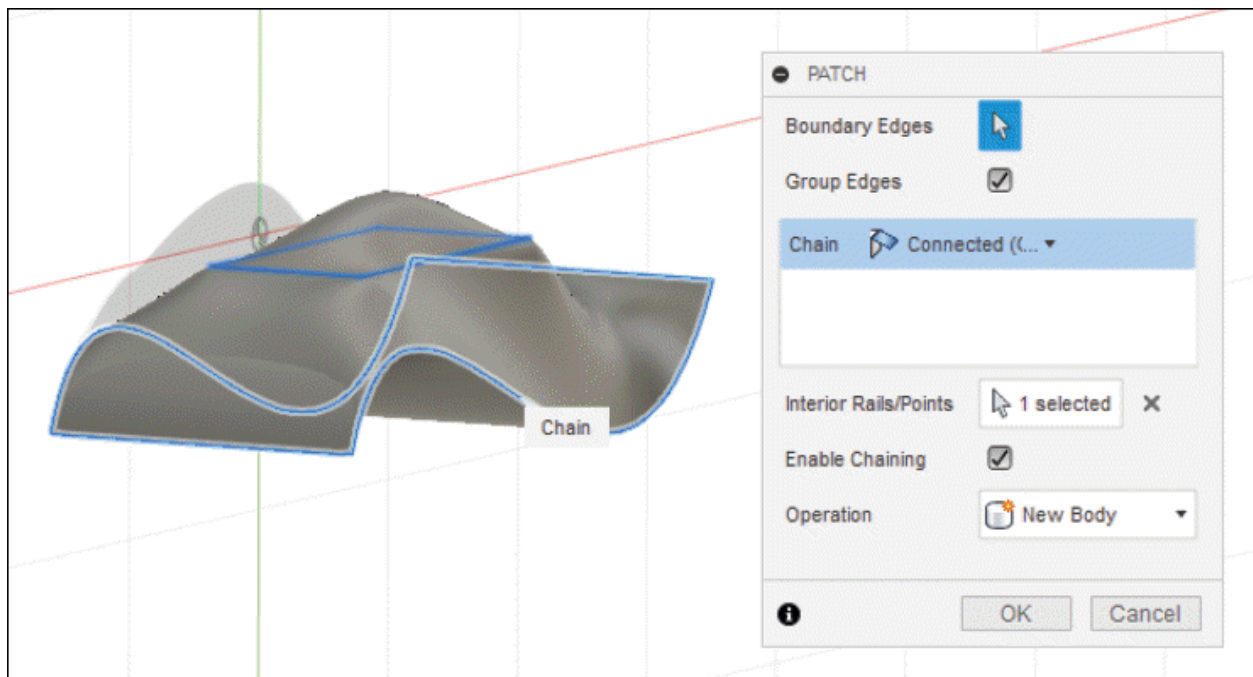


Figure-8. Preview of Patch

- Click on **OK** button from **PATCH** dialog box to complete the process.

Ruled

The **Ruled** tool is used to create a ruled surface at a specified distance and angle from selected edges. The procedure to use this tool is discussed next.

- Click on the **Ruled** tool of **CREATE** drop-down from **Toolbar**; refer to [Figure-9](#). The **RULED** dialog box will be displayed; refer to [Figure-10](#).

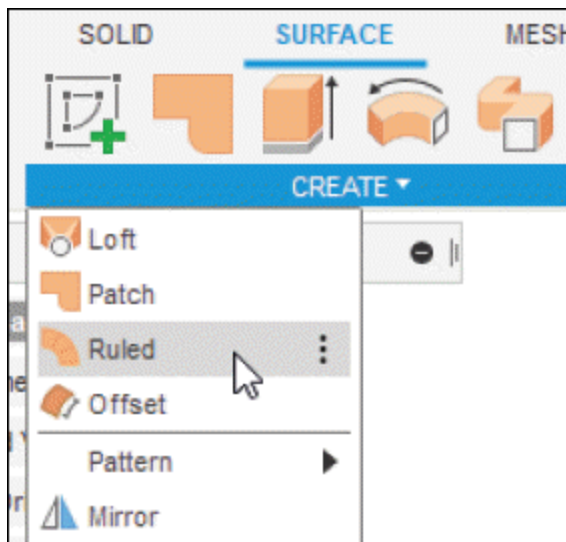


Figure-9. Ruled tool

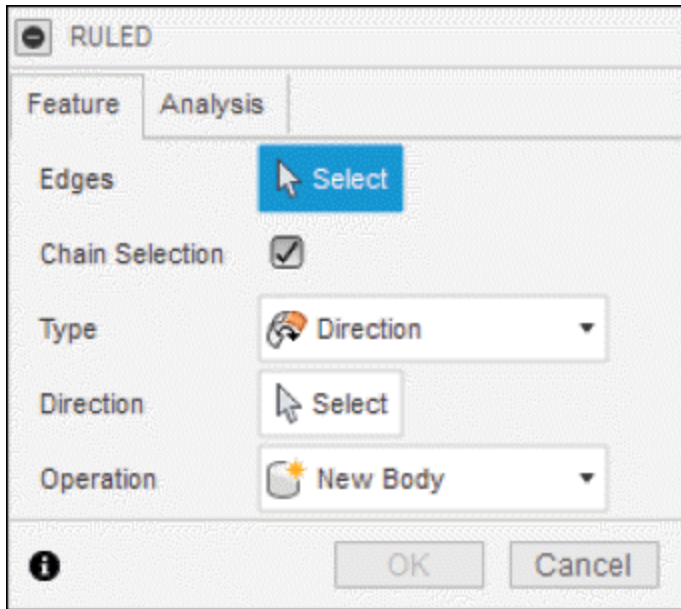


Figure-10. RULED dialog box

- The **Edges** selection button of **Feature** tab is active by default. Select edges on a solid body, surface body, or sketch to extend the ruled surface from; refer to [Figure-11](#). The updated **RULED** dialog box will be displayed; refer to [Figure-12](#).

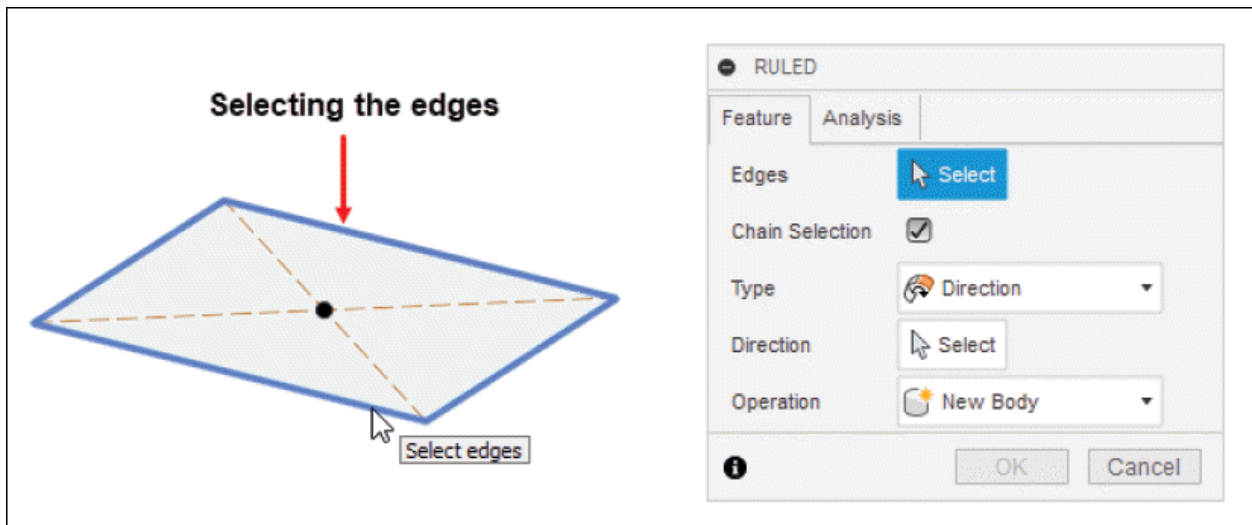


Figure-11. Selecting the edges

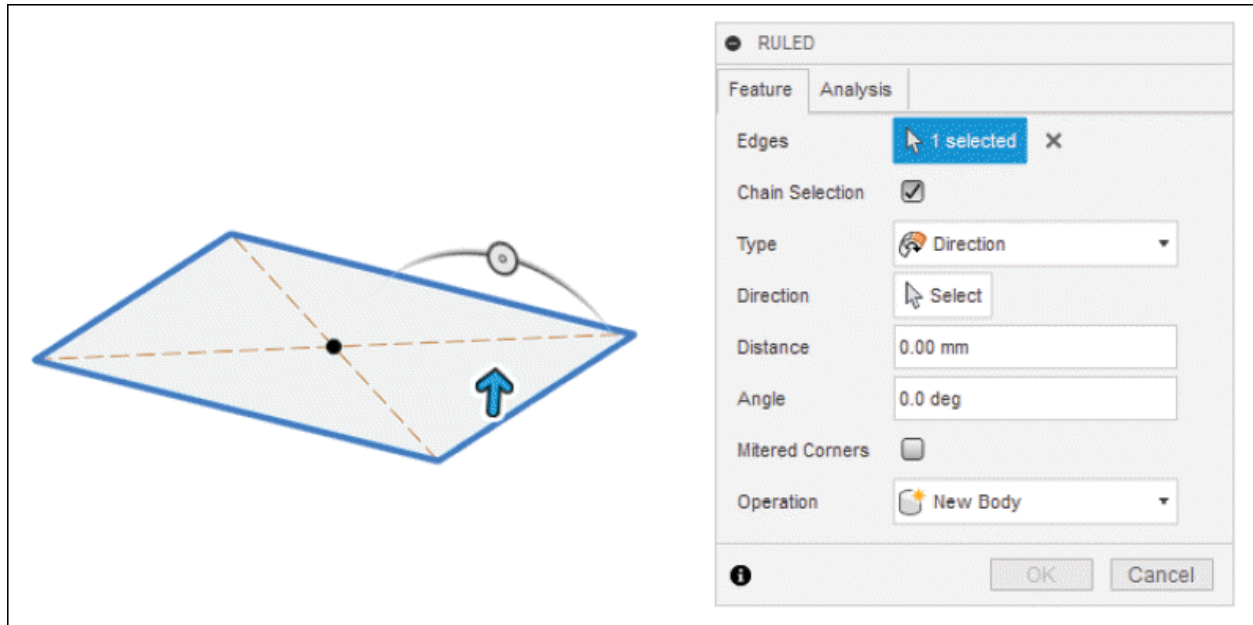


Figure-12. Updated RULED dialog box

- Select **Chain Selection** check box to automatically select the connected geometry.
- Select desired ruled surface type from **Type** drop-down. Select **Normal** option to create a ruled surface normal to the face of the selected edges. Select **Tangent** option to create a ruled surface tangent to the face of the selected edges. Select **Direction** option to create a ruled surface in a direction that you select.
- Click on the **Select** button of **Direction** section and select a linear edge, plane, or axis.
- Specify desired values in the **Distance** and **Angle** edit boxes to extend the surface. You can also use the distance and angle manipulator handles to specify the distance and angle, respectively.

- Select **Mitered Corners** check box to create sharp corners instead of rounded corners.
- After specifying desired parameters in the dialog box, click on the **OK** button. The ruled surface body will be created; refer to [Figure-13](#).

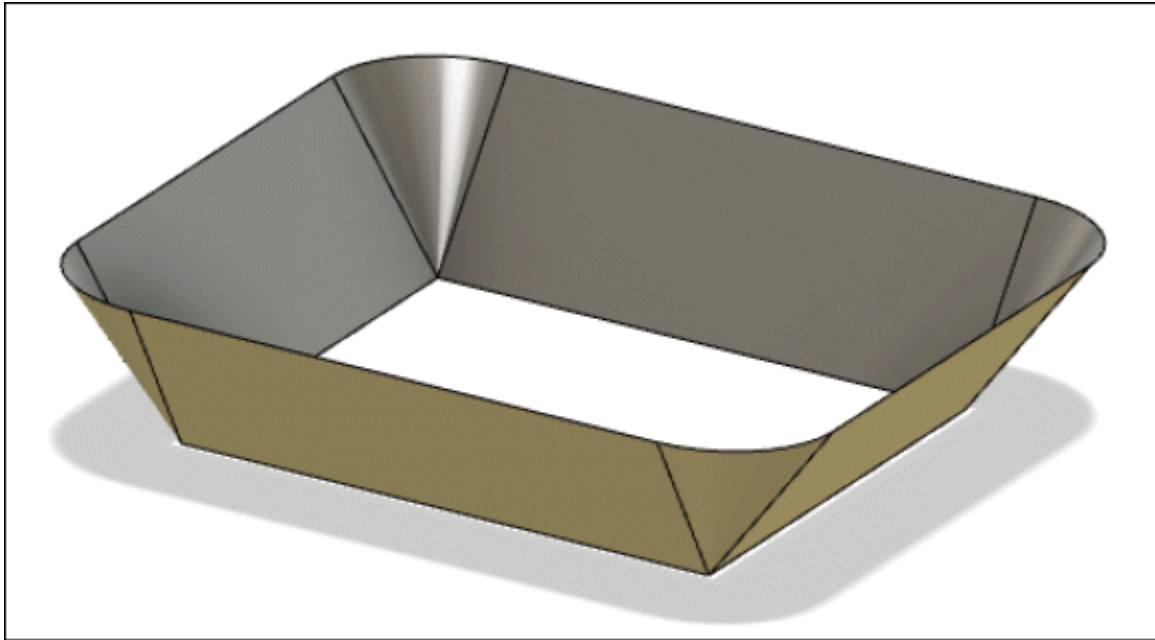


Figure-13. Ruled surface body created

Other Solid Modeling Tools For Surfacing

The other tools in the **CREATE** drop-down can be used in the same way as discussed in **Chapter 4: Advanced 3D Modeling** of the book. You can also create surfaces by using **Offset**, **Pattern**, **Mirror**, **Thicken**, and **Boundary Fill** tools as discussed earlier.

EDITING TOOLS

In the previous section, we have learned about different surface creation tools used in surface modeling. In this

section, we will discuss the surface editing tools.

Trim

The **Trim** tool is used to trim the surface using selected trimming object as cutting tool. The procedure to use this tool is discussed next.

- Click on the **Trim** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-14](#). The **TRIM** dialog box will be displayed; refer to [Figure-15](#).

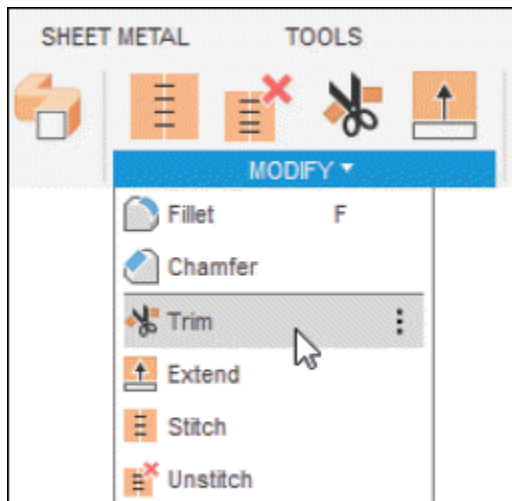


Figure-14. Trim tool

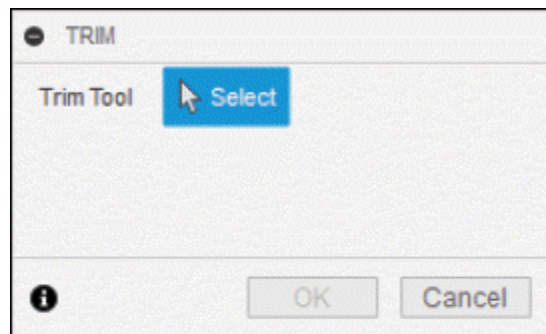


Figure-15. TRIM dialog box

- The **Select** button of **Trim Tool** section is active by default. You need to select an existing sketch to trim the surface. You can also select a plane/face intersecting with the surface as trim tool; refer to [Figure-16](#).

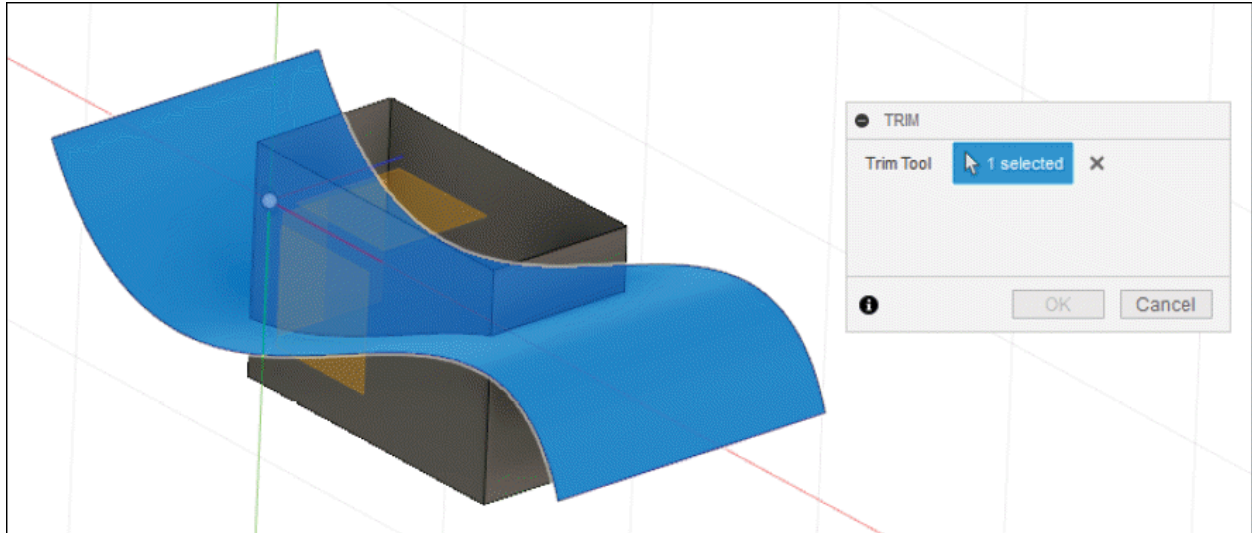


Figure-16. Selection of face as trim tool

- Now, you need to select the area to be removed. Click on desired area of sketch to trim; refer to [Figure-17](#).

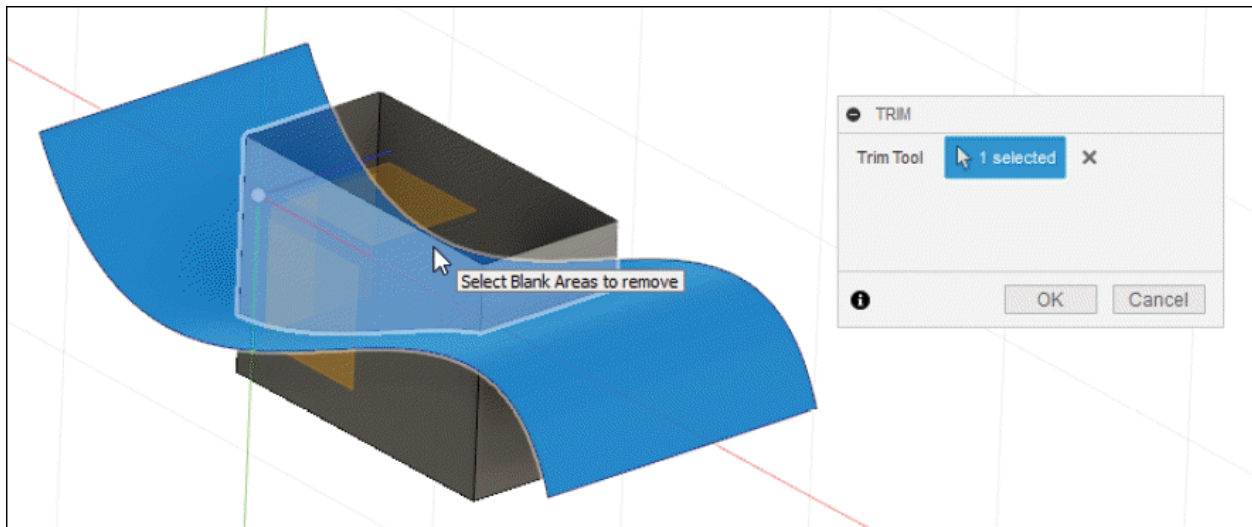


Figure-17. Selection of area to trim

- Click on the **OK** button from **TRIM** dialog box to trim the selected area; refer to [Figure-18](#).

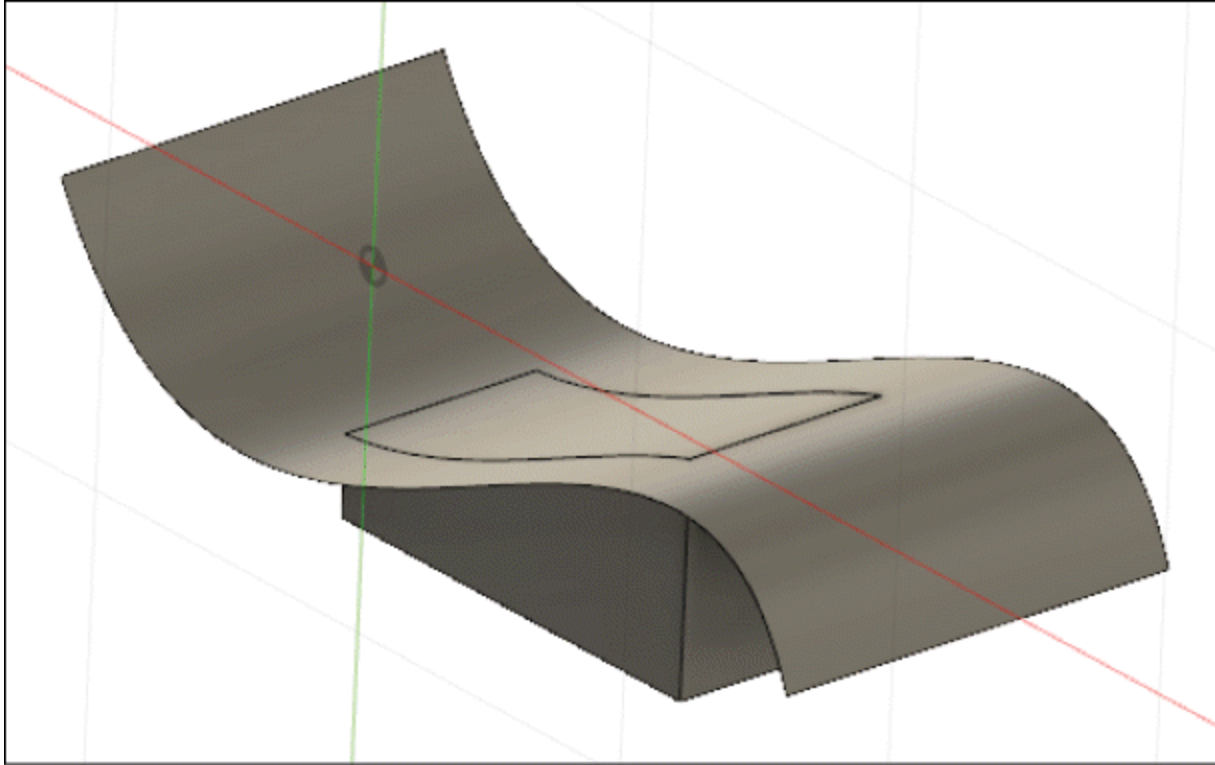


Figure-18. Selected area trimmed

Untrim

The **Untrim** tool is used to extend trimmed surfaces and fill gaps, holes, or empty regions of a surface. The procedure to use this tool is discussed next.

- Click on the **Untrim** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-19](#). The **UNTRIM** dialog box will be displayed; refer to [Figure-20](#).

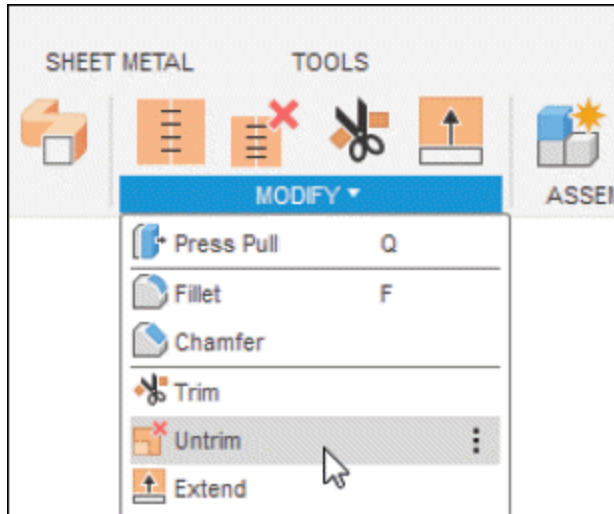


Figure-19. Untrim tool

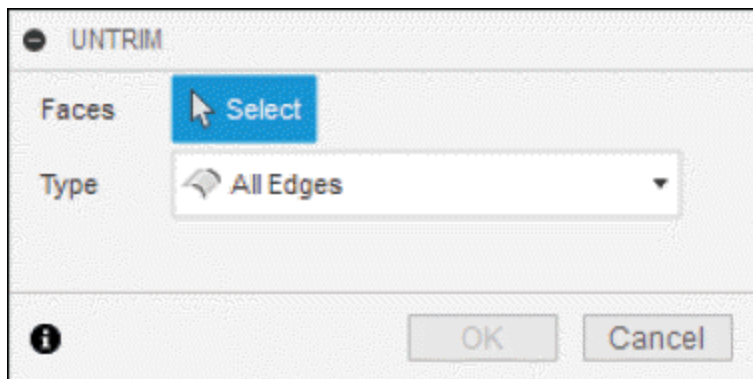


Figure-20. UNTRIM dialog box

- The **Select** button of **Faces** section is active by default. You need to select the faces to be untrimmed.
- Select desired face of the model which you want to be untrimmed; refer to [Figure-21](#). The updated **UNTRIM** dialog box will be displayed with the face untrimmed; refer to [Figure-22](#).

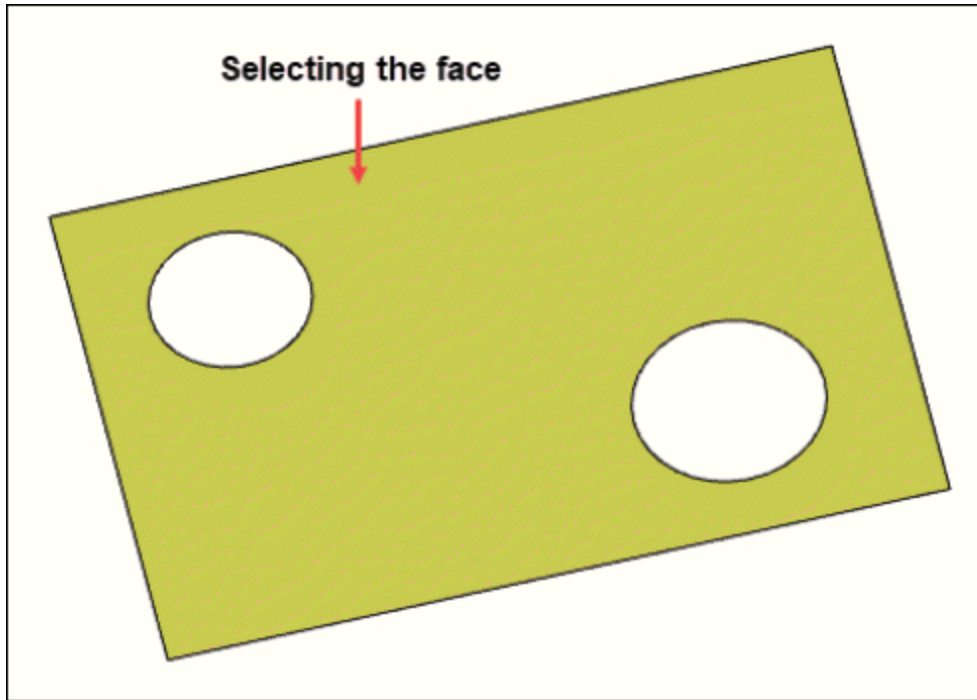


Figure-21. Selecting the face to be untrimmed

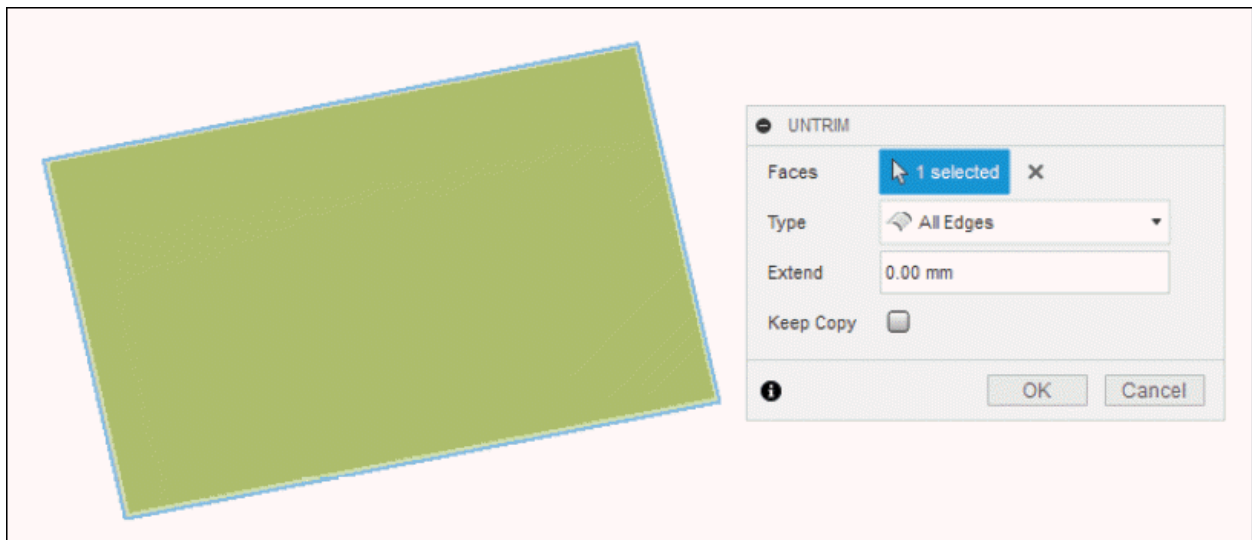


Figure-22. Updated UNTRIM dialog box

- Select **All Edges** option from **Type** drop-down to untrim all internal and external edges of a surface.
- Select **Internal Edges** option from **Type** drop-down to untrim internal edges only.

- Select **External Edges** option from **Type** drop-down to untrim external edges only.
- Select **Manual** option from **Type** drop-down to untrim manually selected edges of a face. Hold **CTRL** button to suppress command preview after selecting first edge and then select multiple edges.
- Specify desired value in **Extend** edit box to extend untrimmed surface.
- Select **Keep Copy** check box to create a copy of original trimmed surface.
- After specifying parameters, click on **OK** button from **UNTRIM** dialog box.

Extend

The **Extend** tool is used to extend the surface edge. The procedure to use this tool is discussed next.

- Click on the **Extend** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-23](#). The **EXTEND** dialog box will be displayed; refer to [Figure-24](#).

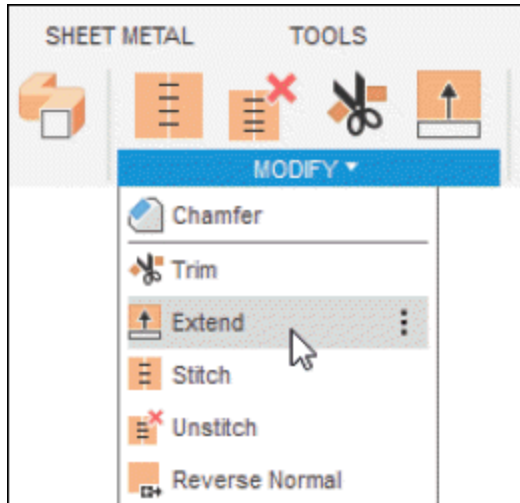


Figure-23. Extend tool

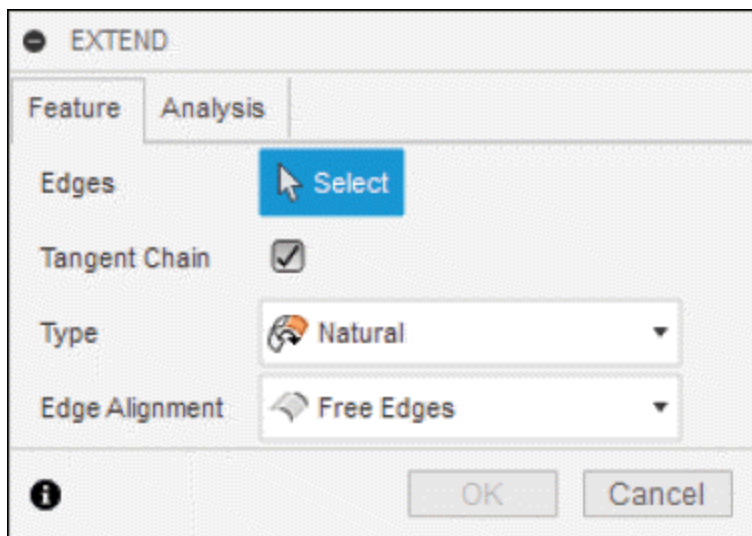


Figure-24. EXTEND dialog box

- The **Select** button of **Edges** section is active by default. You need to select the edge to extend. The updated **EXTEND** dialog box will be displayed; refer to [Figure-25](#).

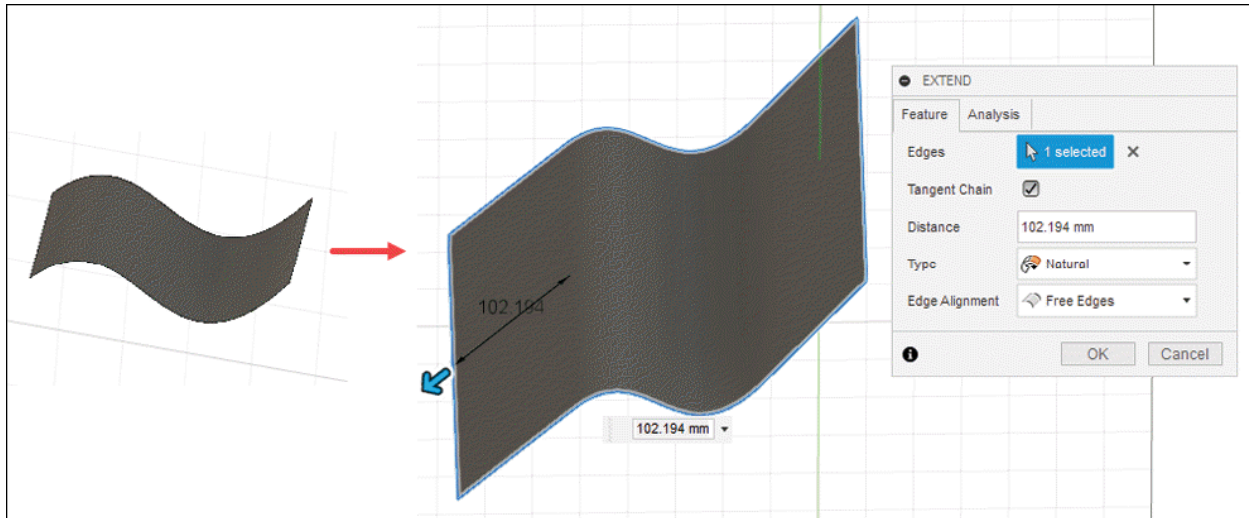


Figure-25. Preview of extend

- Select the **Target Chain** check box if you want to select all tangent or adjacent faces on selection of single face.
- Click in the **Distance** edit box and specify the distance for extend. You can also set the distance by moving the manipulator.
- Select **Natural** option from **Type** drop-down if you want to extend the surface naturally.
- Select **Perpendicular** option from **Type** drop-down if you want to extend the selected edge perpendicularly.
- Select **Tangent** option from **Type** drop-down if you want to extend the selected edge tangentially.
- Select **Free Edges** option from **Edge Alignment** drop-down to apply G0 point continuity to align edges of extended surface.
- Select **Align Edges** option from **Edge Alignment** drop-down to apply G1 tangent continuity to align edges of

extended surface.

- After specifying the parameters, click on **OK** button from **EXTEND** dialog box.

Stitch

The **Stitch** tool is used to combine the surface into one body. The procedure to use this tool is discussed next.

- Click on the **Stitch** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-26](#). The **STITCH** dialog box will be displayed; refer to [Figure-27](#).

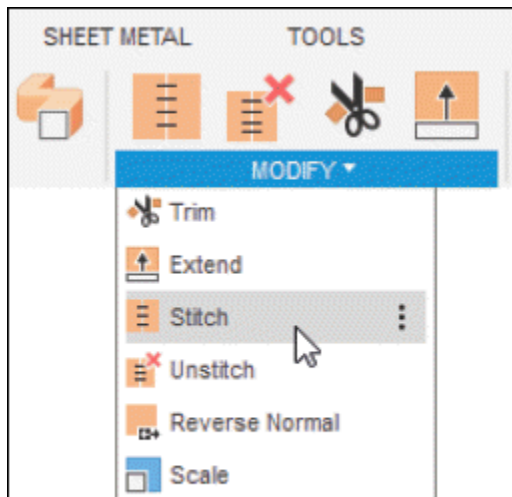


Figure-26. Stitch tool

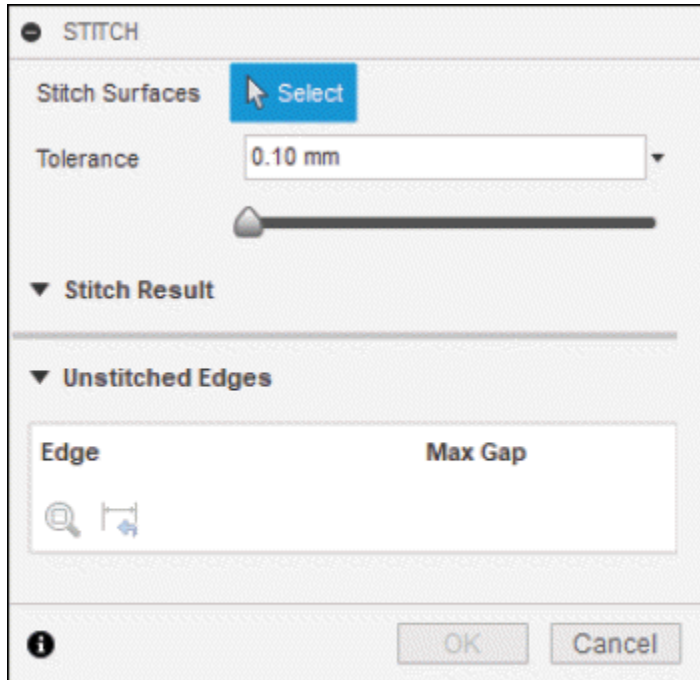


Figure-27. STITCH dialog box

- The **Select** button of **Stitch Surfaces** section is active by default. You need to select the surfaces of model to be stitched; refer to [Figure-28](#).

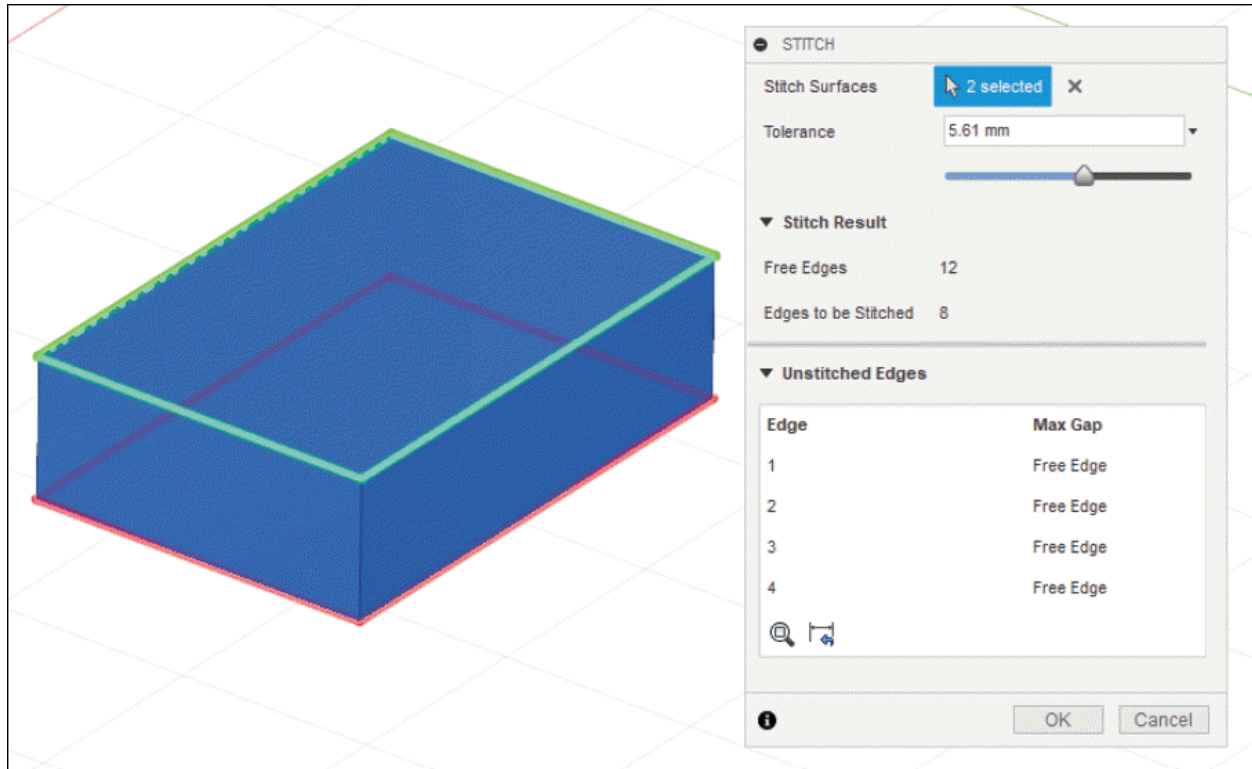


Figure-28. Selection of face to stitch

- The green line on the model shows the stitched surface and red line shows the loose surface connection.
- You need to increase the tolerance from **Tolerance** edit box or **Tolerance** slider to stitch the remaining surfaces which have higher gap between consecutive edges.
- You can also check the result of stitched and unstitched edges from **Stitch Result** section.
- After specifying the parameters, click on the **OK** button from **STITCH** dialog box to complete the process.

Unstitch

The **Unstitch** tool is used to unstitch the earlier stitched surface from model. The procedure to use this tool is

discussed next.

- Click on the **Unstitch** tool of **MODIFY** drop-down from **Toolbar**; refer to [Figure-29](#). The **UNSTITCH** dialog box will be displayed; refer to [Figure-30](#).

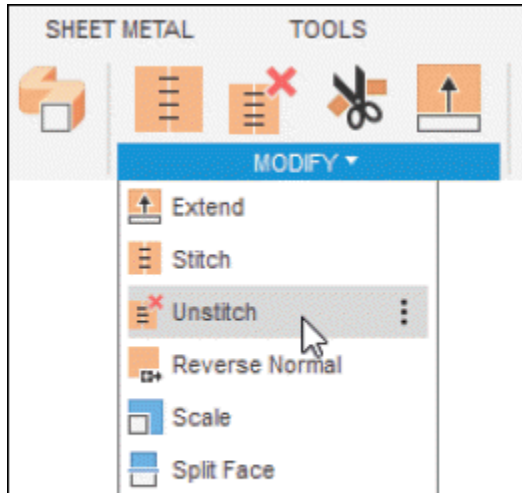


Figure-29. Unstitch tool

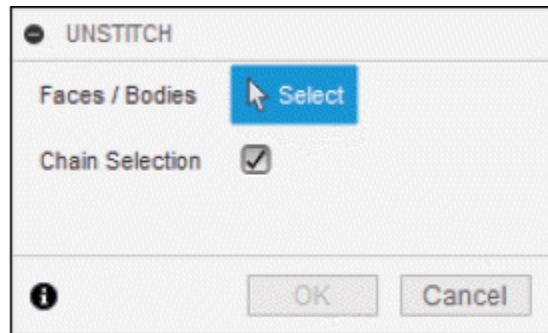


Figure-30. UNSTITCH dialog box

- The **Select** button of **Faces/Bodies** section is active by default. You need to select the surface or body to be unstitched.
- Select the **Chain Selection** check box to select the adjacent surface of the selected surface.
- After selecting the surfaces, click on **OK** button from **UNSTITCH** dialog box to unstitch the selected surface.

Reverse Normal

The **Reverse Normal** tool is used to flip the normal direction of selected surfaces. When modifying the body or model, some of the surfaces of model may have unexpected normal directions. To reverse their normal direction, this

tool is used. The procedure to use this tool is discussed next.

- Click on the **Reverse Normal** tool of **Modify** drop-down from **Toolbar**; refer to [Figure-31](#). The **REVERSE NORMAL** dialog box will be displayed; refer to [Figure-32](#).

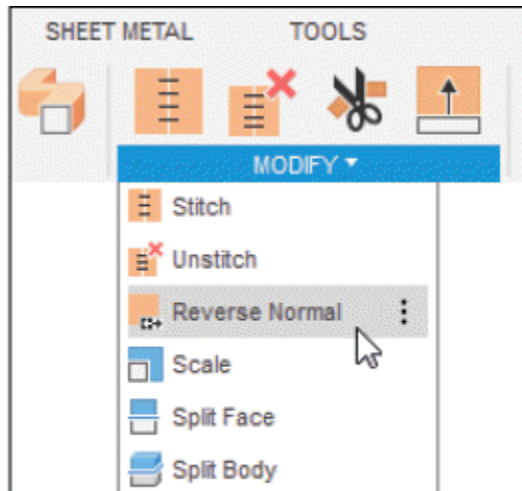


Figure-31. Reverse Normal tool

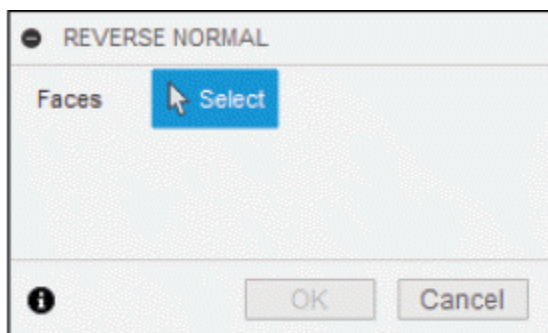


Figure-32. REVERSE NORMAL dialog box

- The **Select** button of **Faces** section is active by default. You need to select the face to flip; refer to [Figure-33](#).

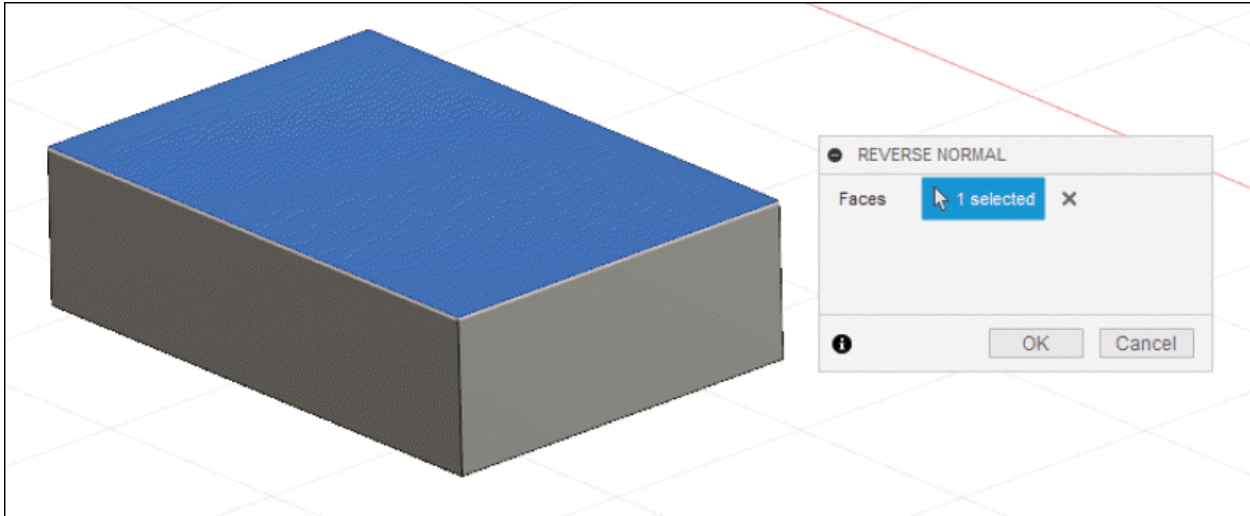


Figure-33. Selection of surface to flip

- After selection of surfaces, click on the **OK** button from **REVERSE NORMAL** dialog box. The selected surfaces will be flipped; refer to [Figure-34](#).

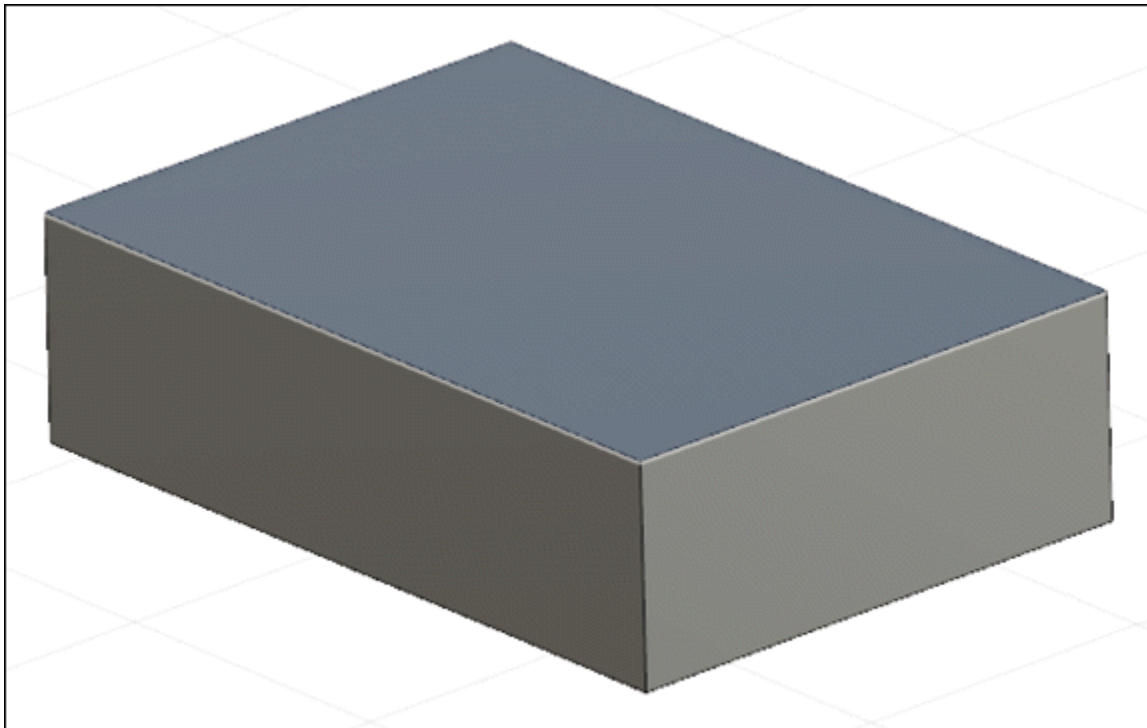


Figure-34. Flipped surface

The other tools of **SURFACE** tab have been discussed earlier

in this book.

PRACTICAL 1

Create the model of aircraft toy as shown in [Figure-35](#). (This is an open dimension problem, you need to get the shape of an aircraft using surfaces.)

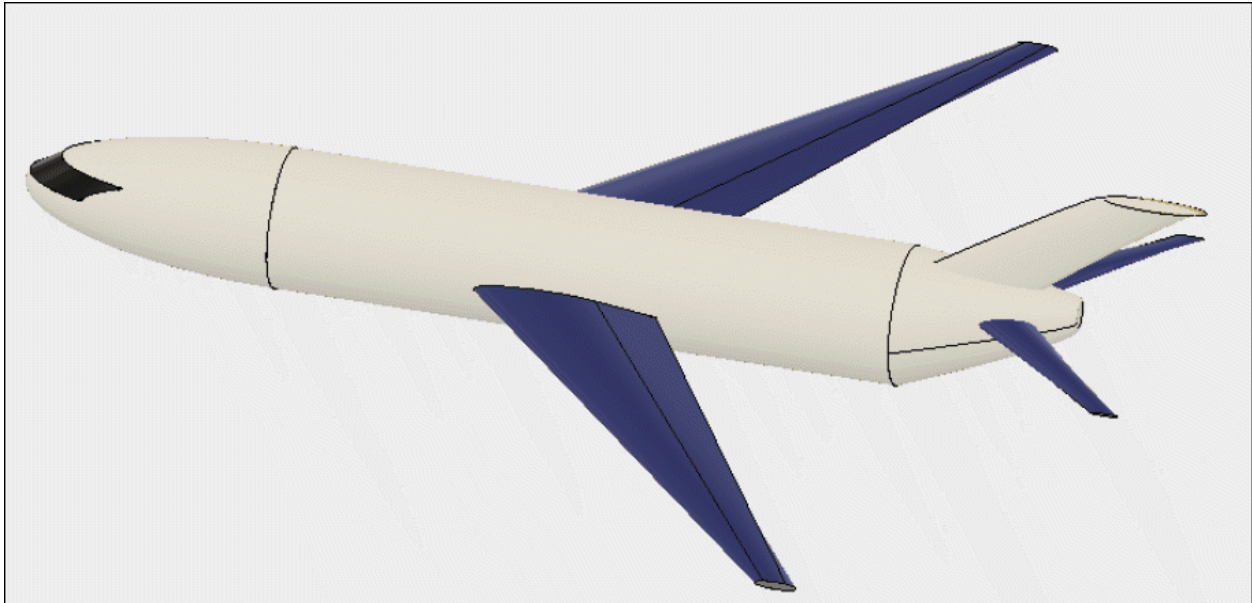


Figure-35. Practical 1 model

Creating first sketch

- Click on the **SURFACE** tab from **Toolbar**; refer to [Figure-36](#). The tools to create surface will be activated.

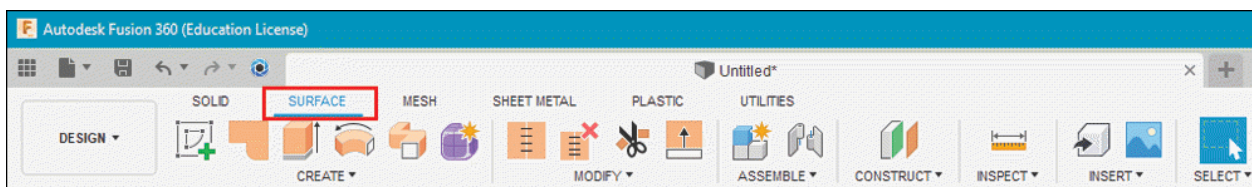


Figure-36. Tools to create surface

- Create the sketch on **Top** plane using **Ellipse**, **Line**, and **Trim** tools; refer to [Figure-37](#).

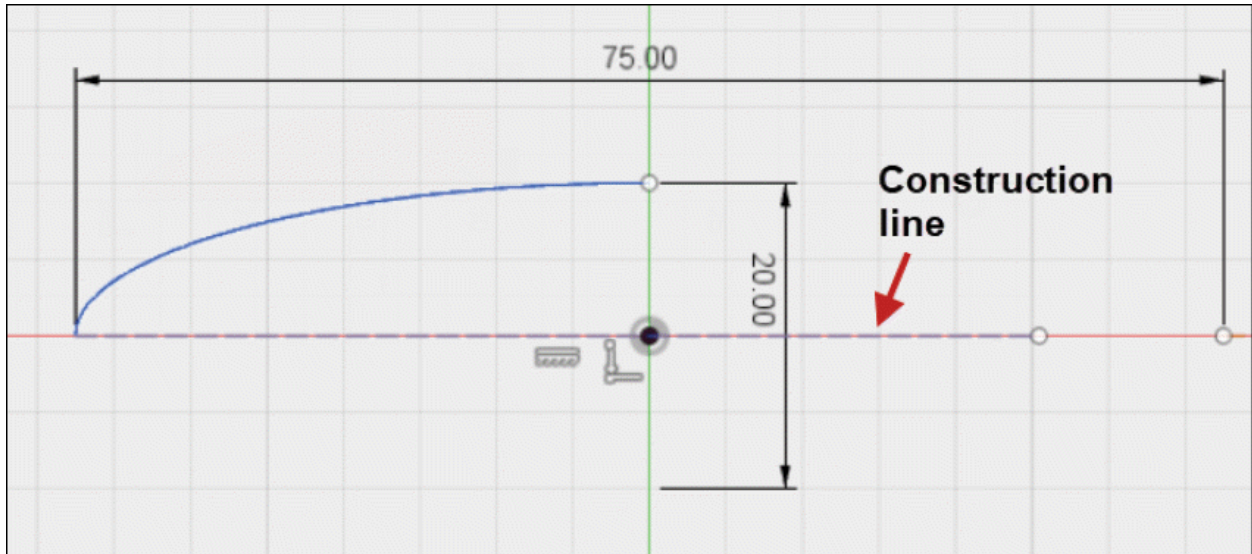


Figure-37. First sketch

Creating Surface

- Click on **Revolve** tool of **CREATE** drop-down from **Toolbar**. The **REVOLVE** dialog box will be displayed; refer to [Figure-38](#).

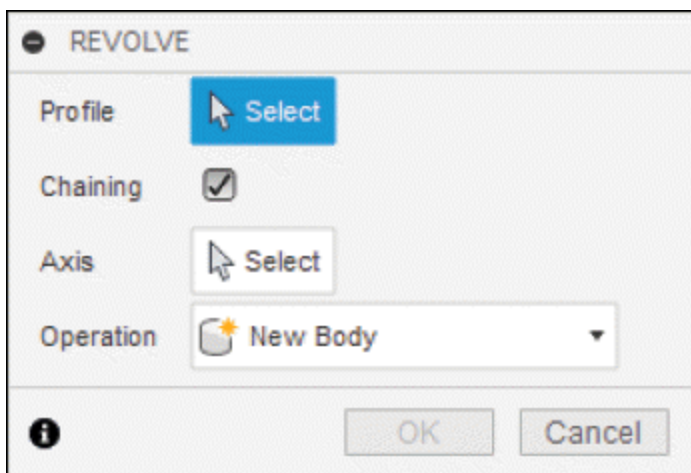


Figure-38. Revolve dialog box

- You need to select the sketch and axis of sketch to apply the revolve feature; refer to [Figure-39](#).

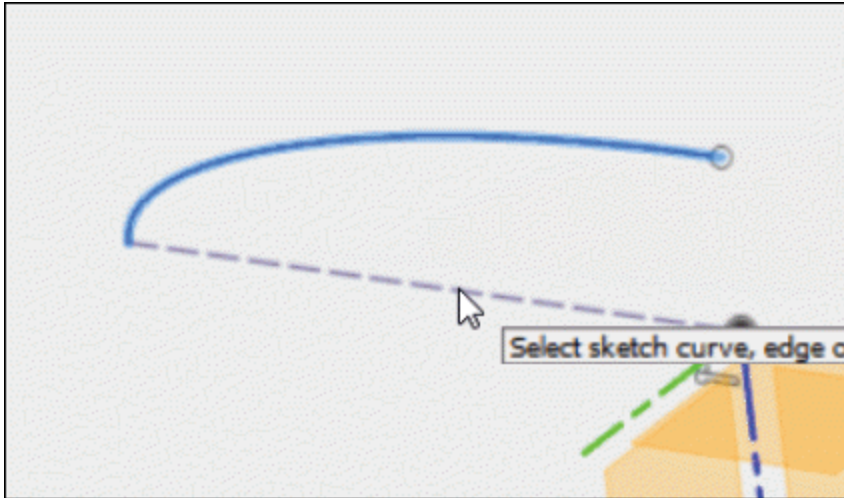


Figure-39. Selecting sketch and axis

- Revolve the sketch to 360 degree to create the nose of aircraft.
- Start a new sketch and project the round edge of nose on **Right** plane as shown in [Figure-40](#). Exit the sketching environment.

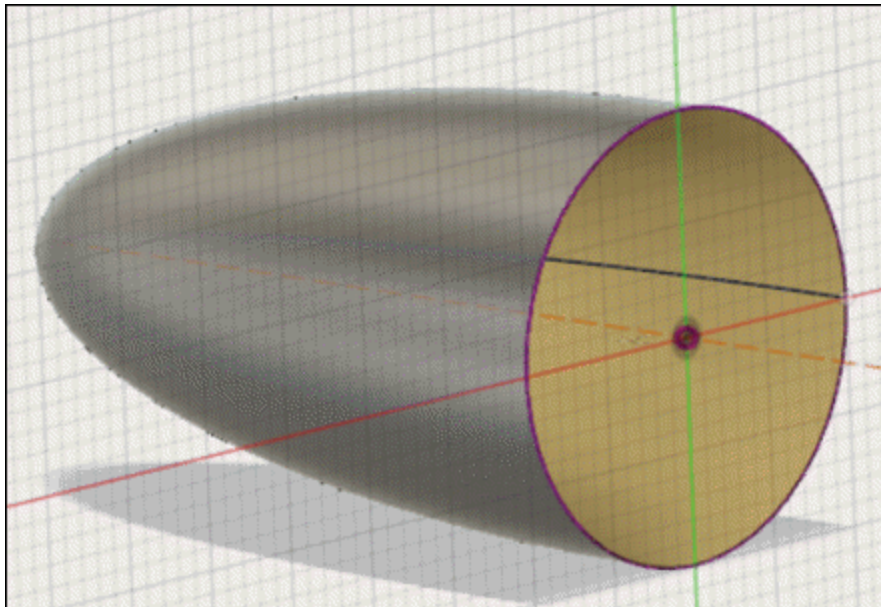


Figure-40. Edge projected on plane

- Click on **Extrude** tool of **CREATE** drop-down from **Toolbar**. The **EXTRUDE** dialog box will be displayed.
- Select the sketch as displayed to extrude; refer to [Figure-41](#).

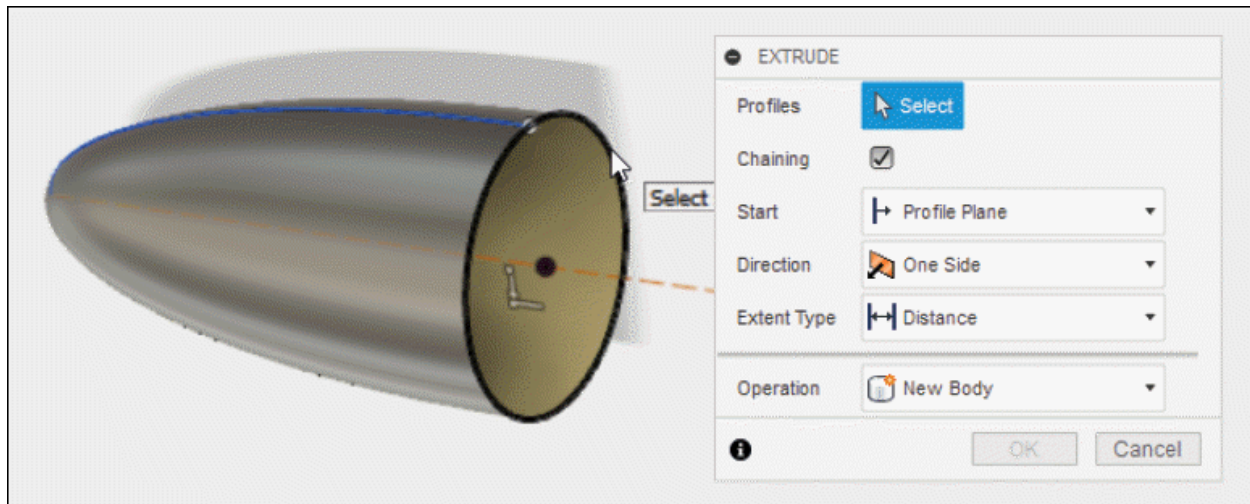


Figure-41. Selecting sketch to extrude

- Click in the **Distance** edit box and enter the value as **90**.
- After specifying the parameters, click on **OK** button from **EXTRUDE** dialog box to complete the extrusion process.
- Click on **Offset Plane** tool of **CONSTRUCT** drop-down from **Toolbar**. The **OFFSET PLANE** dialog box will be displayed.
- Select the **YZ** plane as shown in [Figure-42](#).

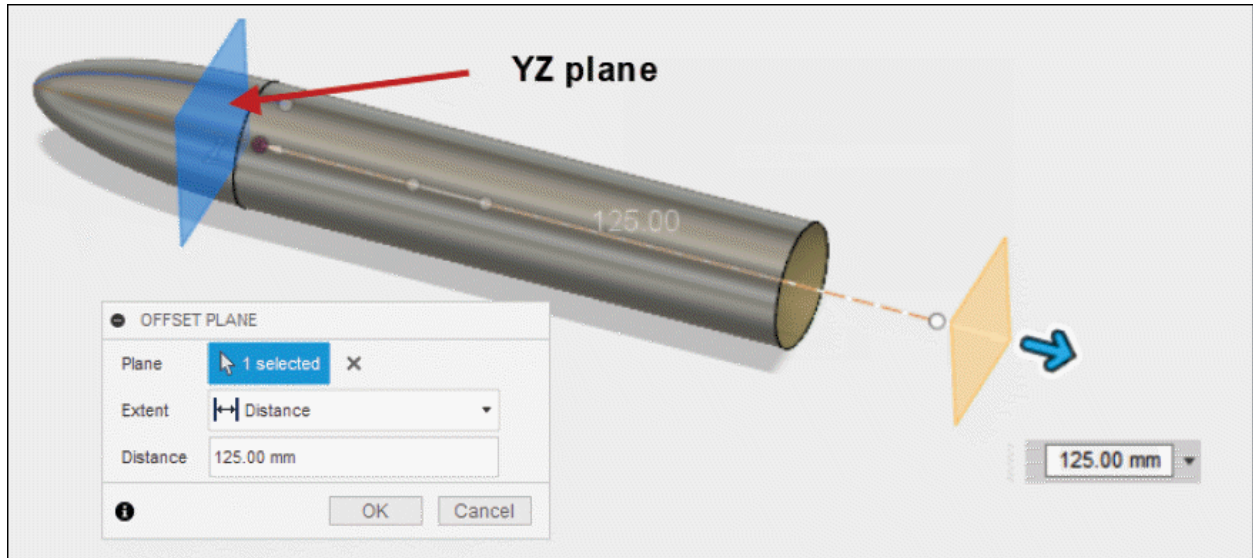


Figure-42. Creating offset plane

- Click in the **Distance** edit box of **OFFSET PLANE** dialog box and specify the value as **125**.
- After specifying the parameters, click on **OK** button from **OFFSET PLANE** dialog box to create the plane.
- Start a new sketch using newly created plane and click on the **Center Diameter Circle** tool from **CREATE** drop-down.
- Create the circle as shown in [Figure-43](#).

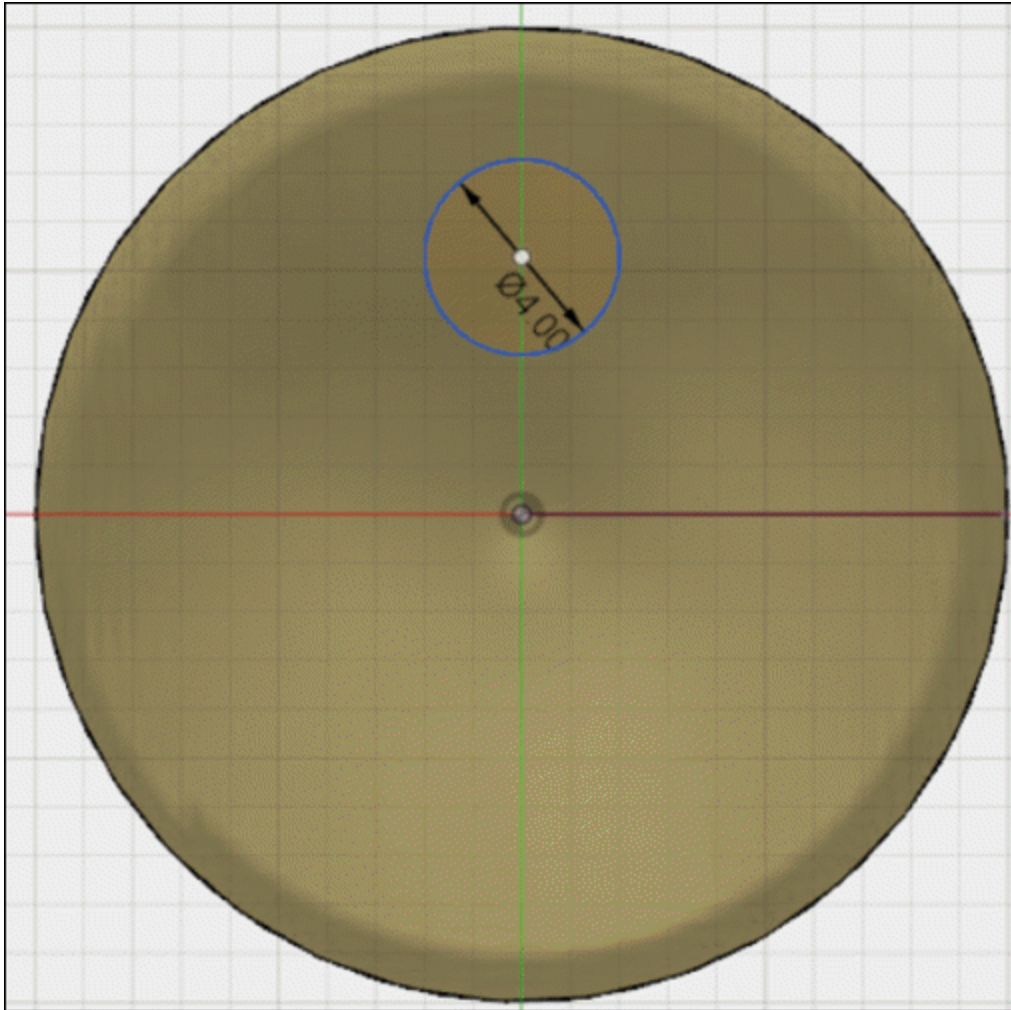


Figure-43. Creating circle

- After creating the sketch, click on **Finish Sketch** button from **Toolbar**.
- Click on **Loft** tool of **CREATE** drop-down from **Toolbar**. The **LOFT** dialog box will be displayed.
- You need to select the two sketch to create loft feature; refer to [Figure-44](#).

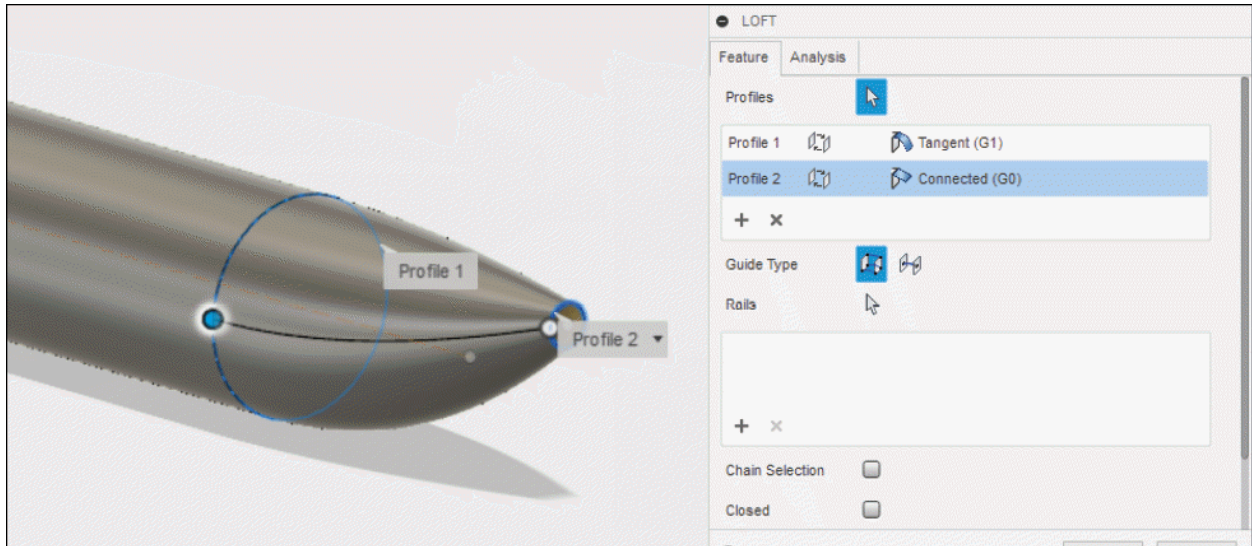


Figure-44. Selecting sketch for loft feature

- Specify the parameters as desired and click on **OK** button from **LOFT** dialog box to complete the process.
- Click on the **Create Sketch** tool and select the **XZ** plane as sketching plane. Create the sketch with the help of **Ellipse** and **Line** tool as shown in [Figure-45](#).

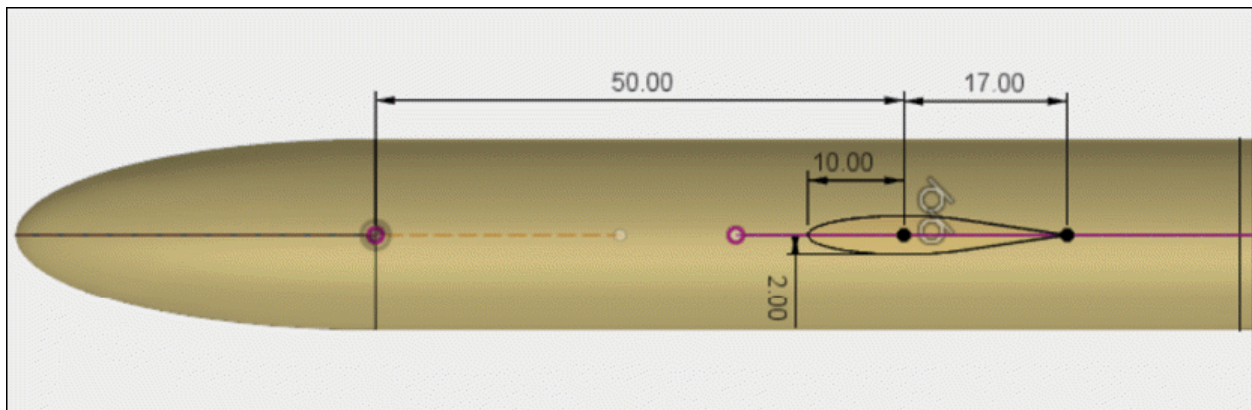


Figure-45. Sketch for wing

- After creating the sketch, click on **Finish Sketch** button from **Toolbar**.
- Now, create the offset plane at a distance of **75** from the

XZ plane; refer to [Figure-46](#).

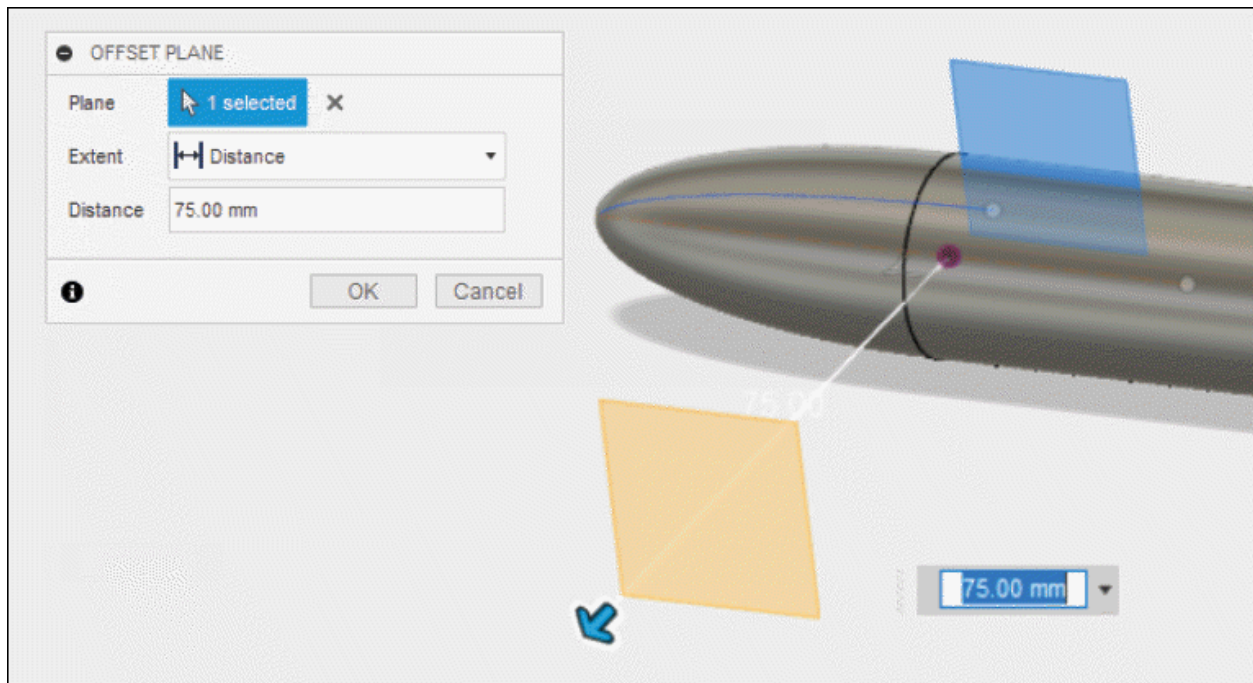


Figure-46. Creating third offset plane

- Click on the **Ellipse** tool from **CREATE** drop-down and select the recently created plane.
- Create the ellipse on current plane as shown in [Figure-47](#).

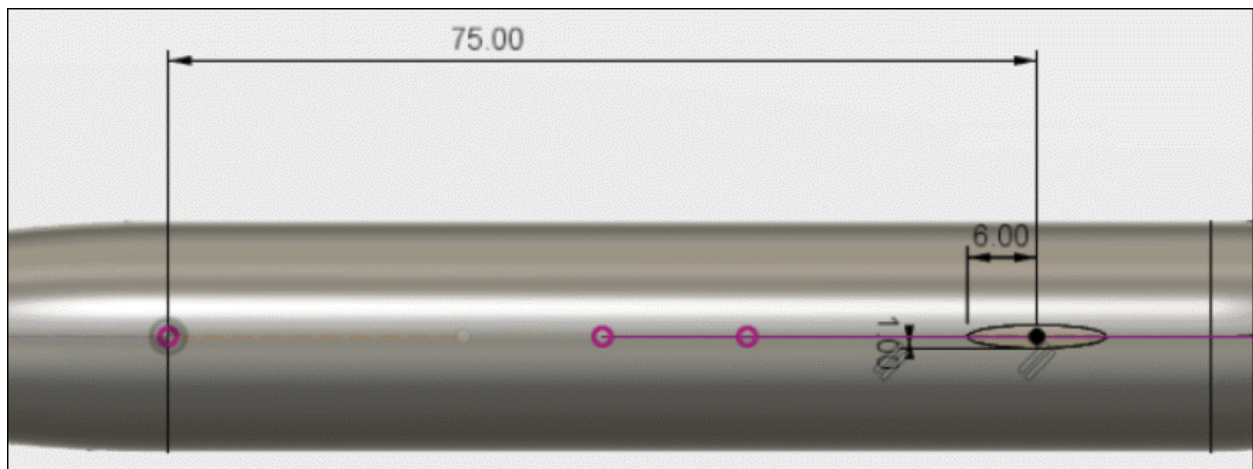


Figure-47. Creating ellipse

- Click on the **Finish Sketch** tool from **Toolbar**.

- Click on the **Loft** tool from **CREATE** drop-down. The **LOFT** dialog box will be displayed.
- Select the recently created sketches (ellipse forms) and create the loft; refer to [Figure-48](#). Make sure the **Chain Selection** check box is selected.

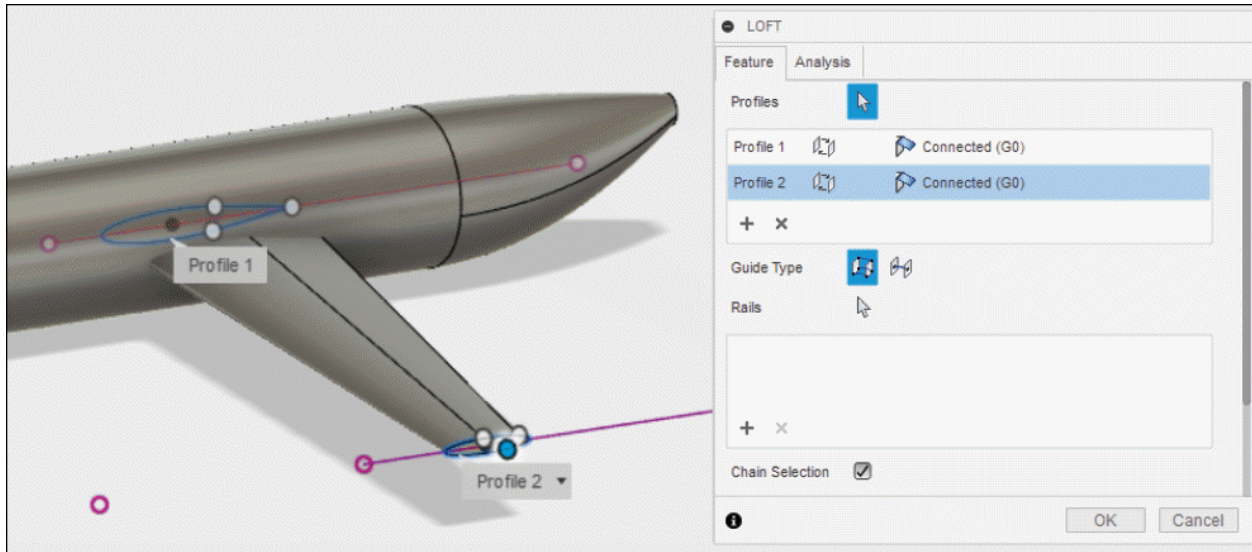


Figure-48. Creating wing

- After selection of profiles for loft, click on the **OK** button from **LOFT** dialog box to complete the loft process.
- Click on **Mirror** tool of **CREATE** drop-down from **Toolbar**. The **MIRROR** dialog box will be displayed; refer to [Figure-49](#).

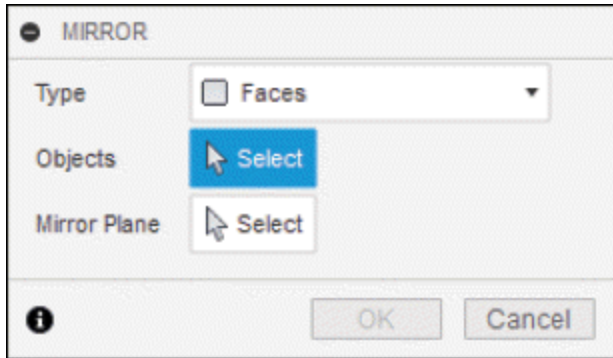


Figure-49. MIRROR dialog box

- Click on **Type** drop-down from **MIRROR** dialog box and select **Bodies** option.
- Click on **Select** button of **Objects** section and select the recently created wing.
- Click on **Select** button of **Mirror Plane** section and select the **XZ** plane; refer to [Figure-50](#).

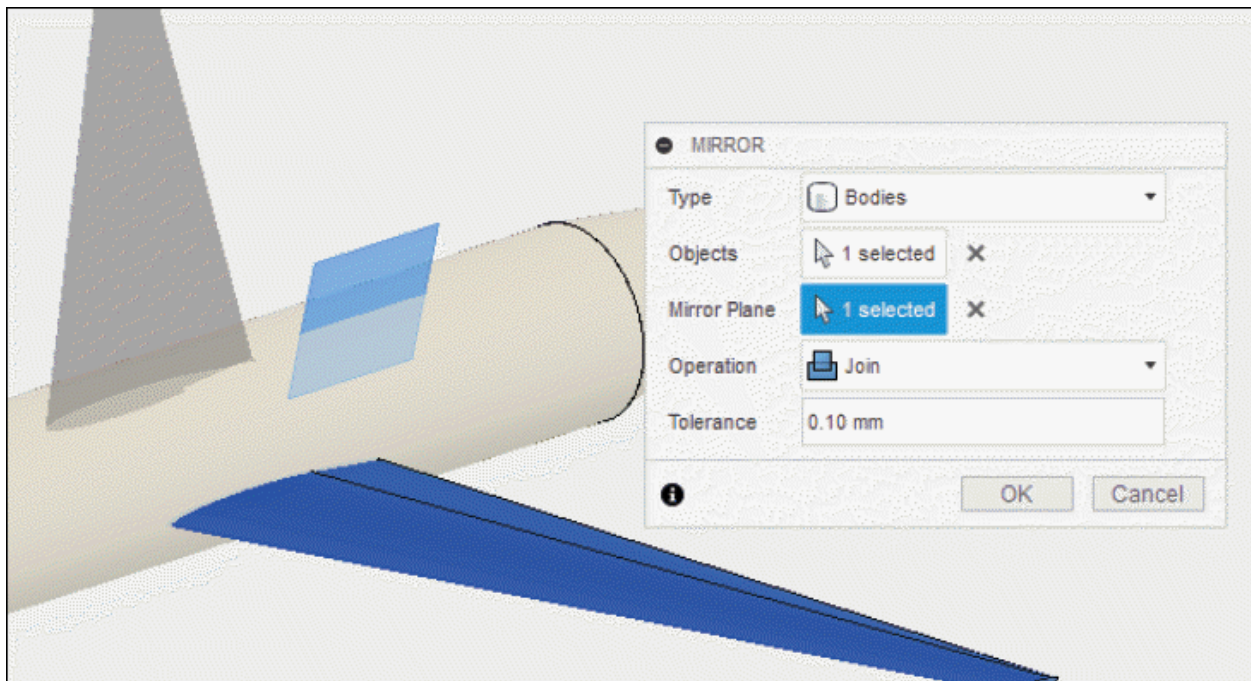


Figure-50. Creating Mirror copy of wing

- Click on **OK** button from **MIRROR** dialog box to complete the process.
- Click on the **Offset Plane** tool of **CONSTRUCT** drop-down from **Toolbar**. The **OFFSET PLANE** dialog box will be displayed.
- Create an offset plane at distance of **23** mm from **XZ** plane as shown in [Figure-51](#).

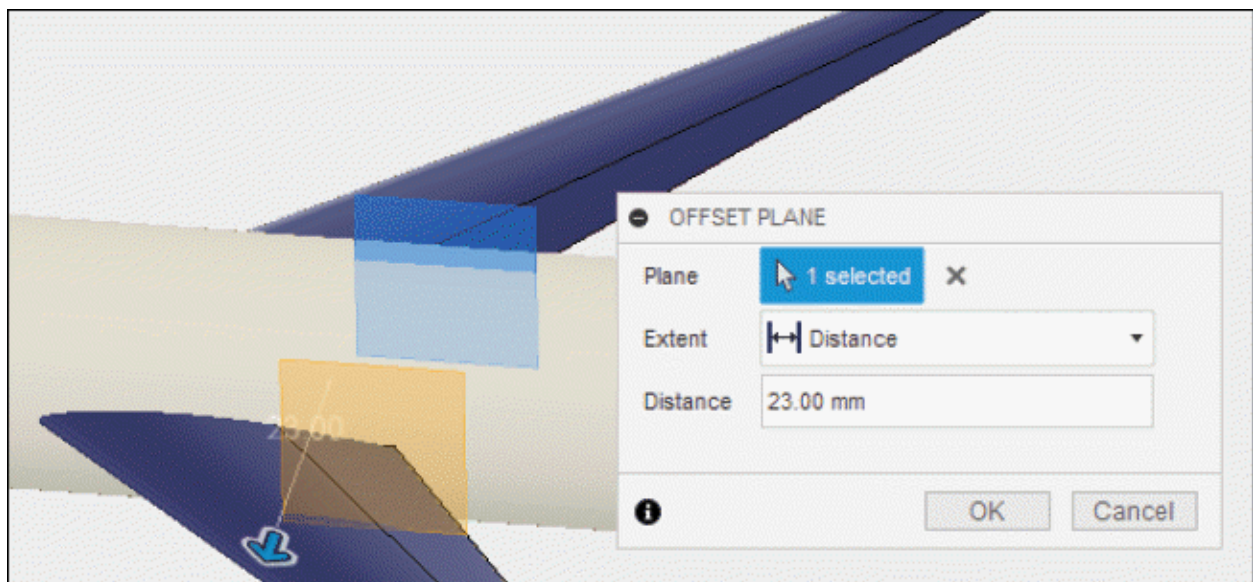


Figure-51. Creating fourth offset plane

- Create an ellipse on the **XZ** plane using **Ellipse** tool as shown in [Figure-52](#).

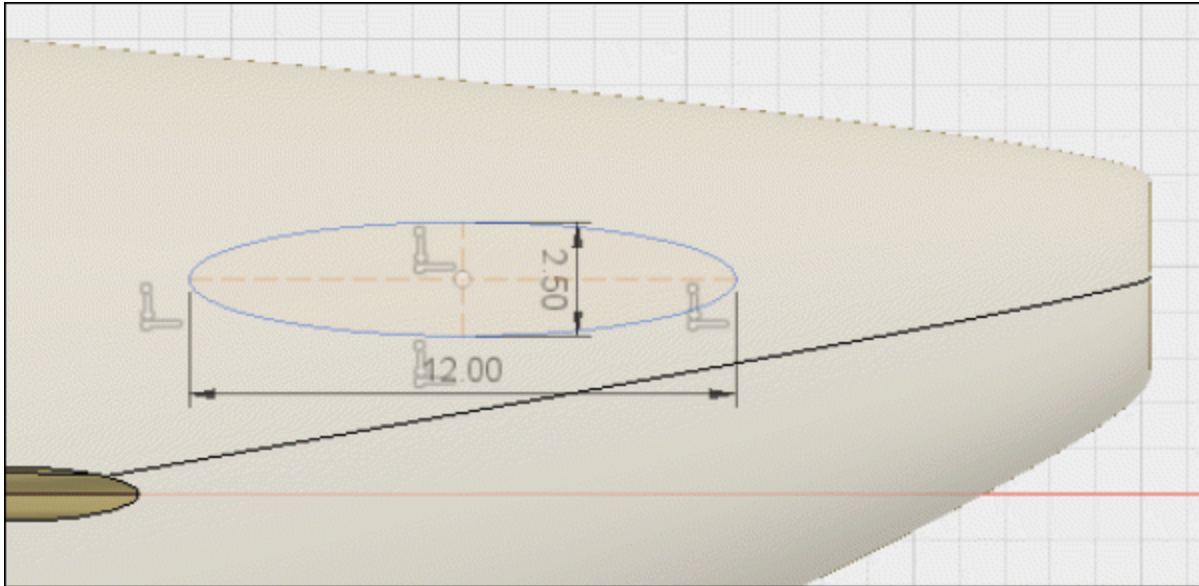


Figure-52. Creating sketch for rear wing

- After creating sketch, click on **Finish Sketch** button from **Toolbar**.
- Create another ellipse on the newly created offset plane as shown in [Figure-53](#).

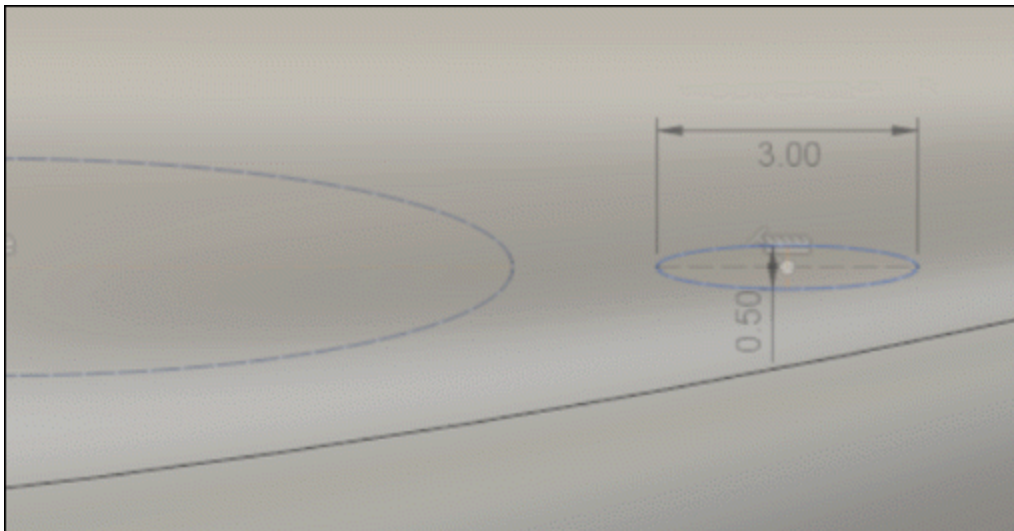


Figure-53. Sketch for rear wing

- Click on **Finish Sketch** button from **Toolbar** to exit the

sketch.

- Click on **Loft** tool of **CREATE** drop-down from **Toolbar**. The **LOFT** dialog box will be displayed.
- You need to select two recently created sketch in **Profiles** section of **LOFT** dialog box; refer to [Figure-54](#).

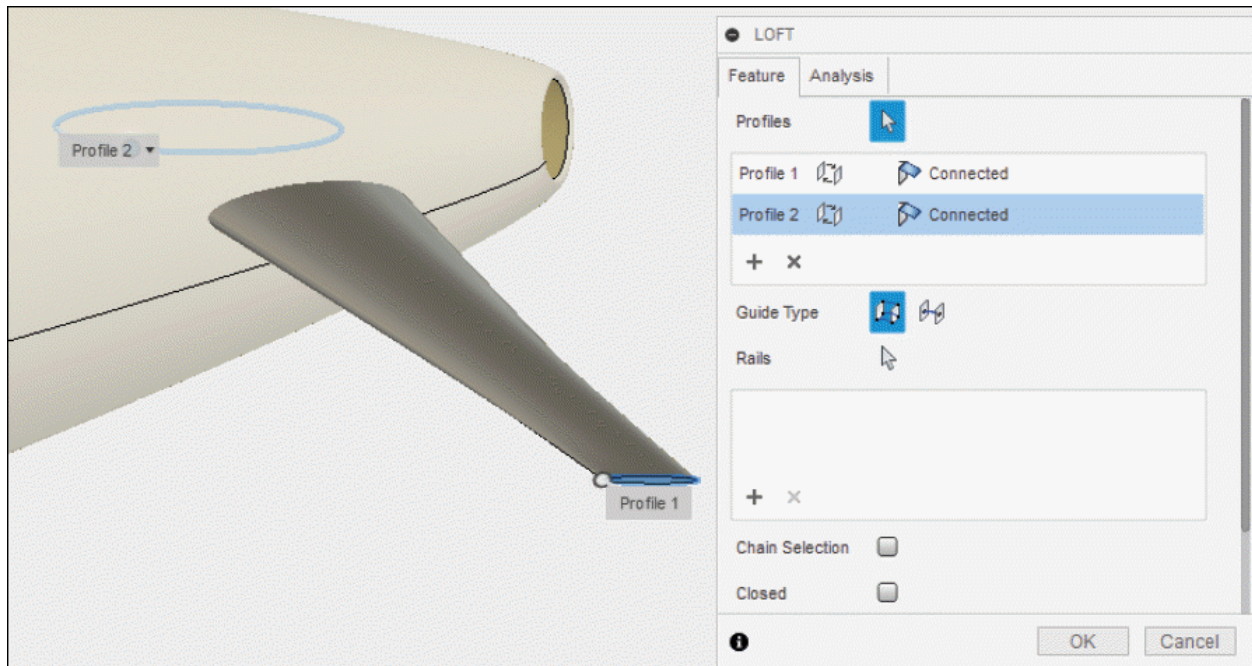


Figure-54. Applying loft tool for rear wing

- After specifying the parameters, click on **OK** button from **LOFT** dialog box to complete the process.
- To fill the open ends of wings, we need to apply the **Patch** tool.
- Click on the **Patch** tool of **CREATE** drop-down from **Toolbar**. The **PATCH** dialog box will be displayed; refer to [Figure-55](#).

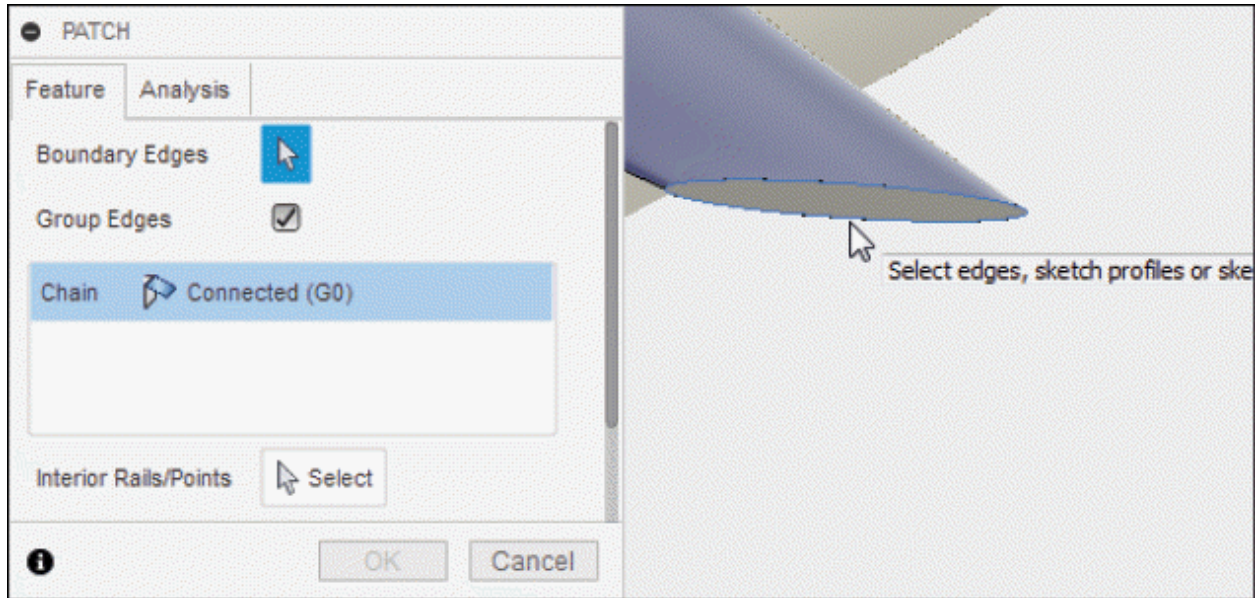


Figure-55. Applying Patch tool

- The **Select** button of **Boundary Edges** section is active by default. Click on the area displayed in the above figure.
- Click on the **Continuity** drop-down and select **Connected** option.
- After specifying the parameters, click on the **OK** button from **PATCH** dialog box to complete the process.
- To create other rear wing, click on **Mirror** tool of **CREATE** drop-down from **Toolbar**. The **MIRROR** dialog box will be displayed.
- Select the recently created wing in **Objects** section of **MIRROR** dialog box.
- Click on **Select** button of **Mirror Plane** section of **MIRROR** dialog box and select the **XY** plane as reference; refer to [Figure-56](#).

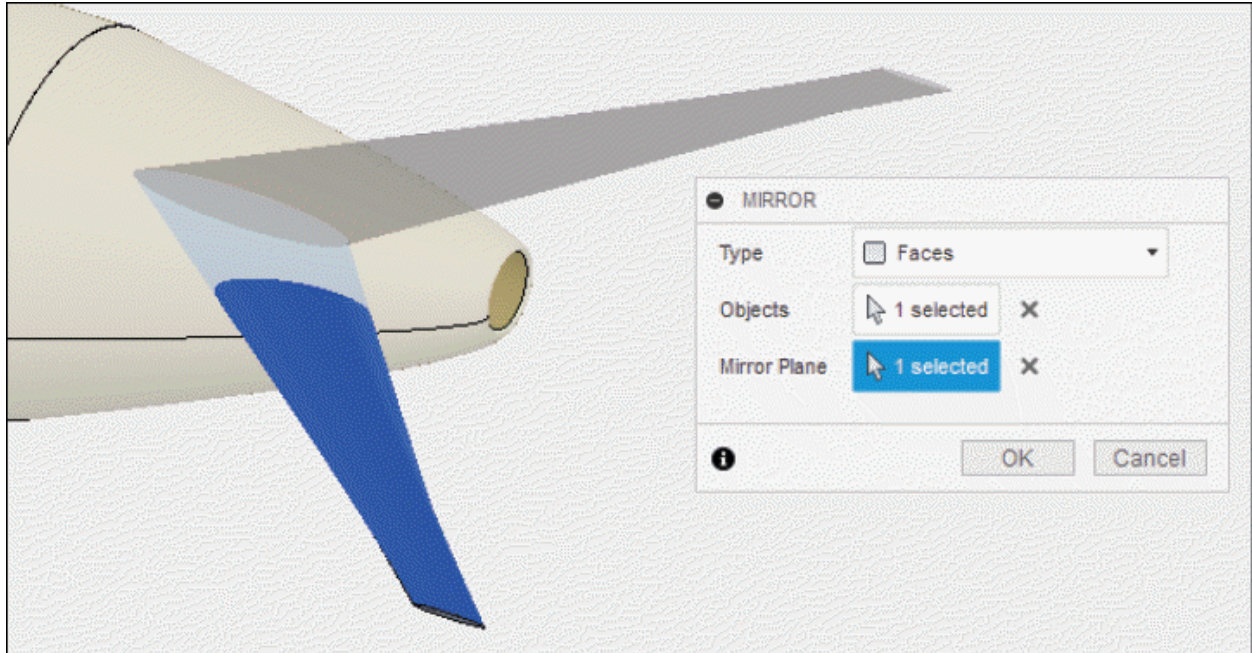


Figure-56. Creating other rear wing

- Click on **OK** button from **MIRROR** dialog box to create the copy.
- Click on **Create Sketch** tool of **CREATE** drop-down and select the **XY** plane for creation of sketch. Create the ellipse as shown in [Figure-57](#).

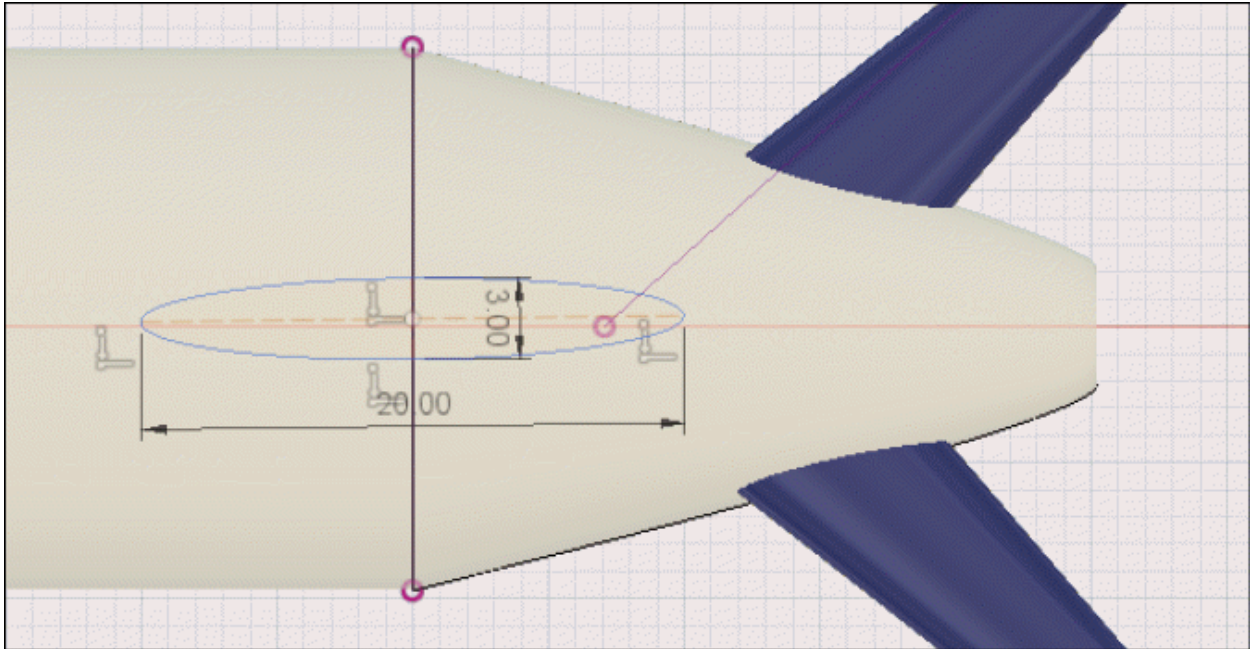


Figure-57. Creating sketch for third wing

- After creating the sketch, click on **Finish Sketch** button from **Toolbar**.
- Click on **Offset Plane** tool from **CONSTRUCT** drop-down and create a plane with **XY** plane as reference; refer to [Figure-58](#).

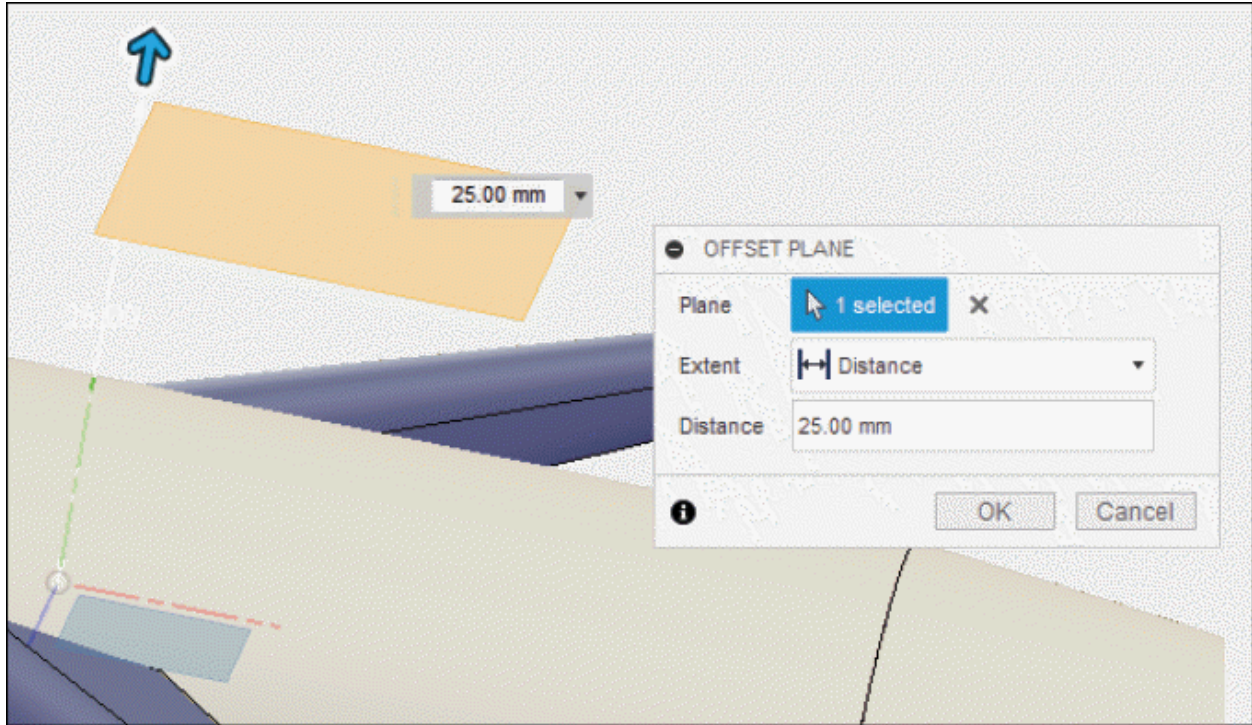


Figure-58. Creating offset plane for third rear wing

- Click on the **Create Sketch** tool of **CREATE** drop-down and select the recently created plane for sketch.
- Create the sketch as shown in [Figure-59](#).

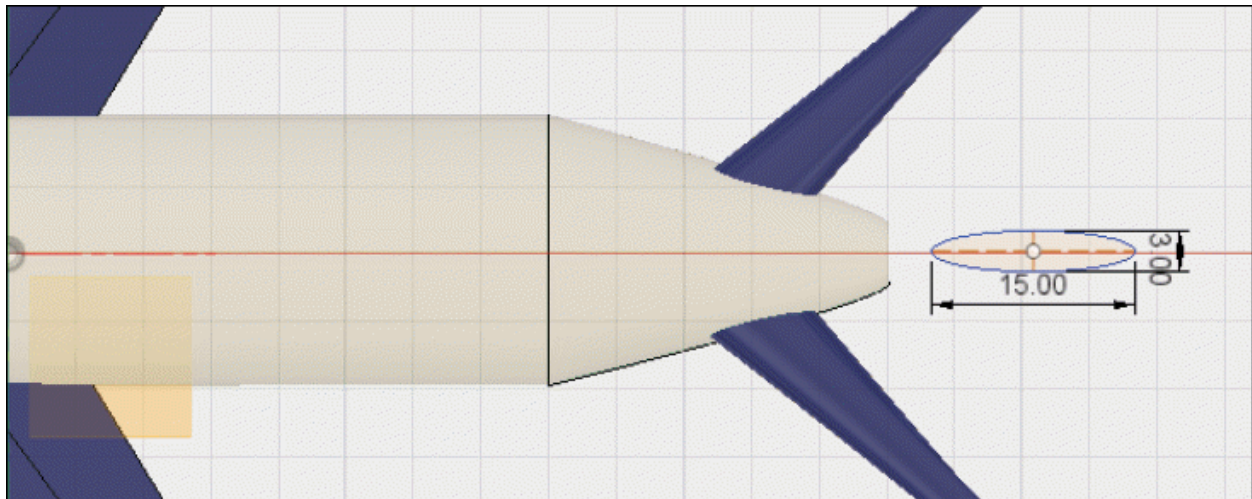


Figure-59. Creating another sketch for rear wing

- Click on **Finish Sketch** button from **Toolbar** to exit the

sketch.

- Click on **Loft** tool from **CREATE** drop-down and select the recently created sketch under **Profiles** section; refer to [Figure-60](#).

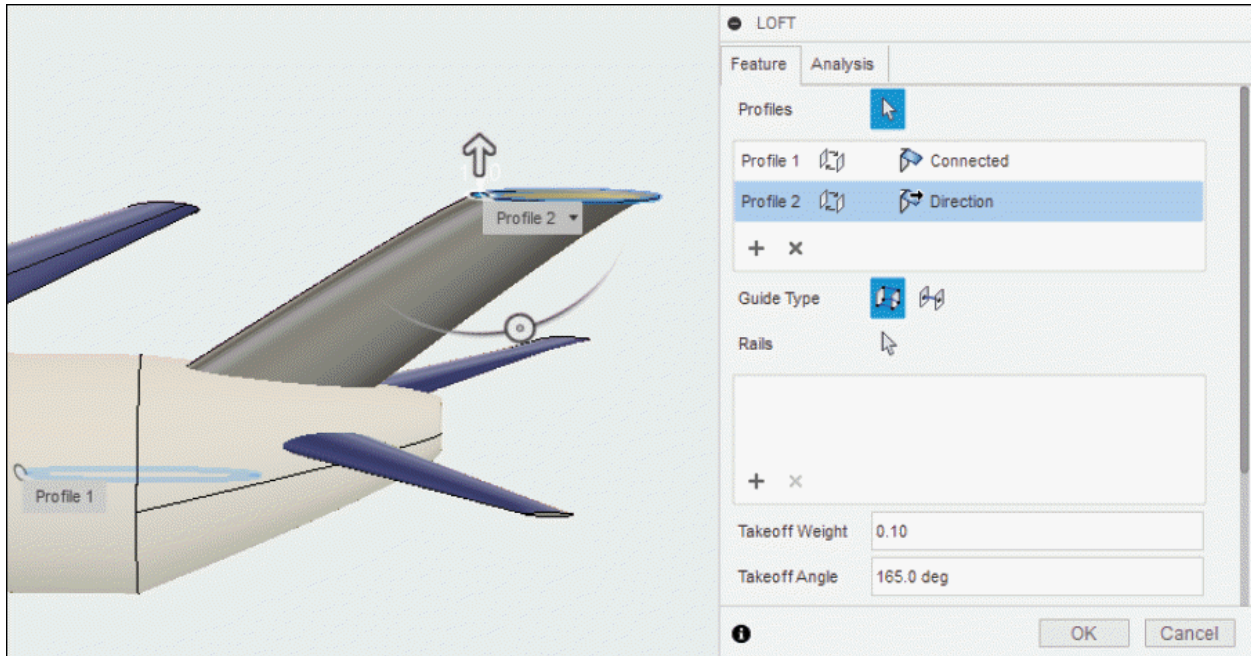


Figure-60. Applying Loft feature

- Click in the **Takeoff Weight** edit box and enter the value as **0.10**.
- Click in the **Takeoff Angle** edit box and enter the value as **165**.
- After specifying the parameters for loft feature, click on the **OK** button from **LOFT** dialog box. The third arm of aircraft will be created.
- Click on **Patch** tool of **CREATE** drop-down in **SURFACE** tab from **Toolbar**. The **PATCH** dialog box will be displayed; refer to [Figure-61](#).

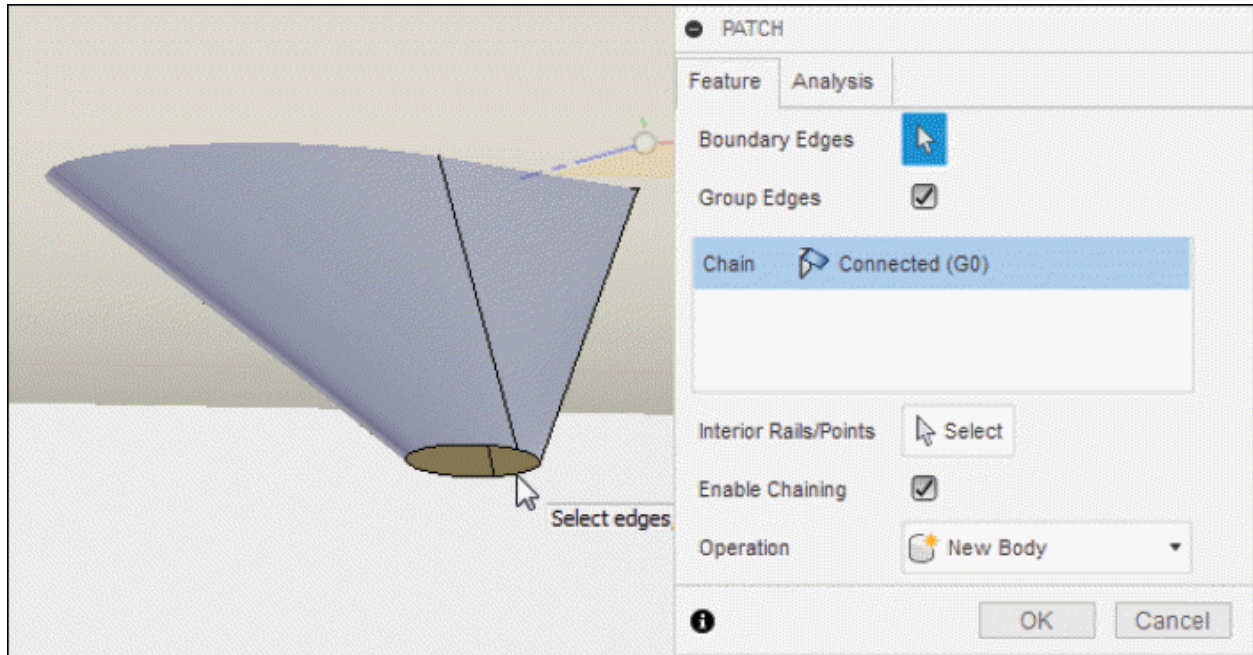


Figure-61. Applying patch on wing

- Select the open edge and click on **OK** button from **PATCH** dialog box to complete the process.
- Similarly, apply the patch tool to all wings and back of aircraft body.
- Click on **Reverse Normal** tool of **MODIFY** drop-down from **Toolbar**. The **REVERSE NORMAL** dialog box will be displayed.
- The **Select** button of **Faces** section is active by default. Click on the faces with wrong normal; refer to [Figure-62](#).

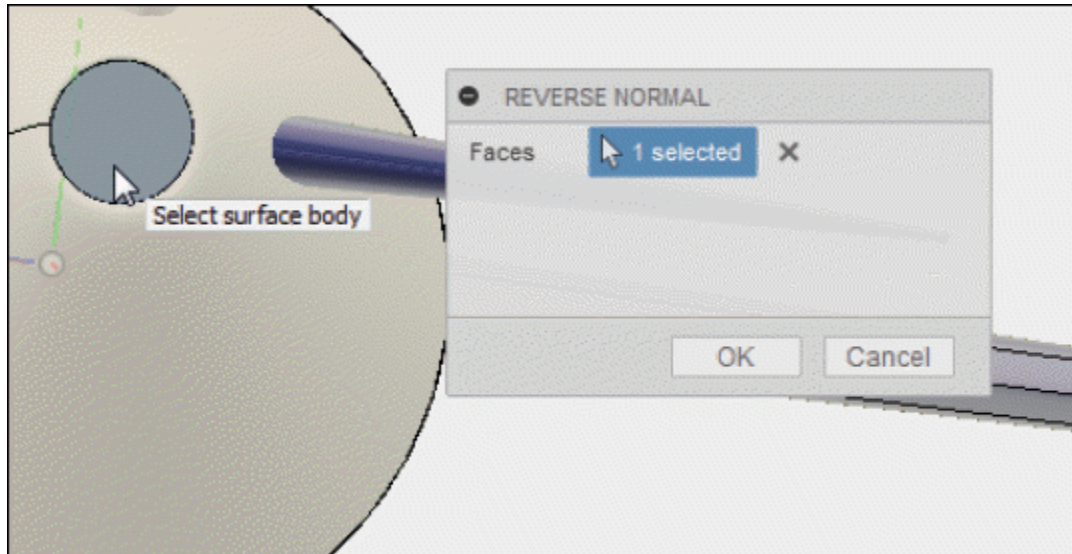


Figure-62. Applying Reverse Normal tool

- Click on the **OK** button from **Reverse Normal** dialog box to complete the process.
- Click on **Center Rectangle** tool of **CREATE** drop-down and select the **YZ** plane for sketch; refer to [Figure-63](#).

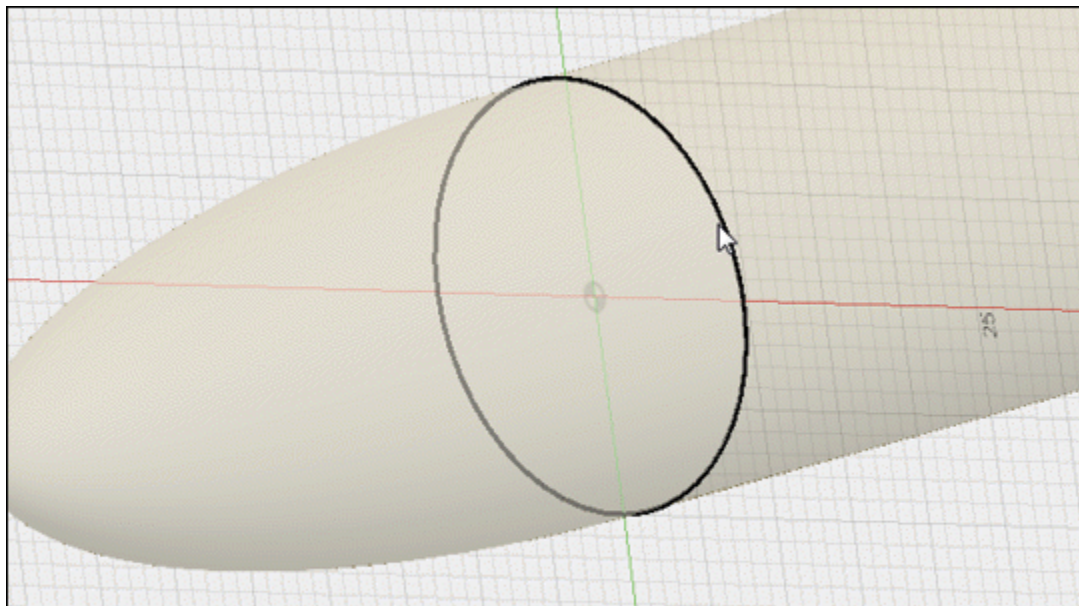


Figure-63. Selecting plane for sketch

- Create a rectangle on the selected plane as shown in [Figure-64](#).

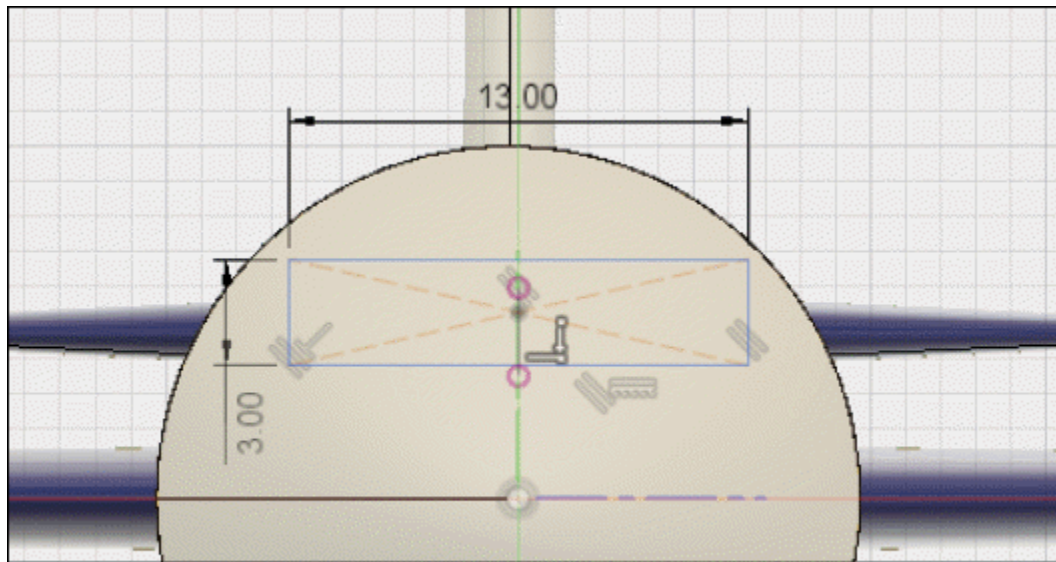


Figure-64. Sketch for windshield

- Click on **Finish Sketch** button from **Toolbar** to exit the sketch.
- Click on **Trim** tool of **MODIFY** drop-down from **Toolbar**. The **TRIM** dialog box will be displayed.
- The **Select** button of **Trim Tool** section is active by default. Select the recently created rectangular sketch and click on the inner area of trim preview; refer to [Figure-65](#).

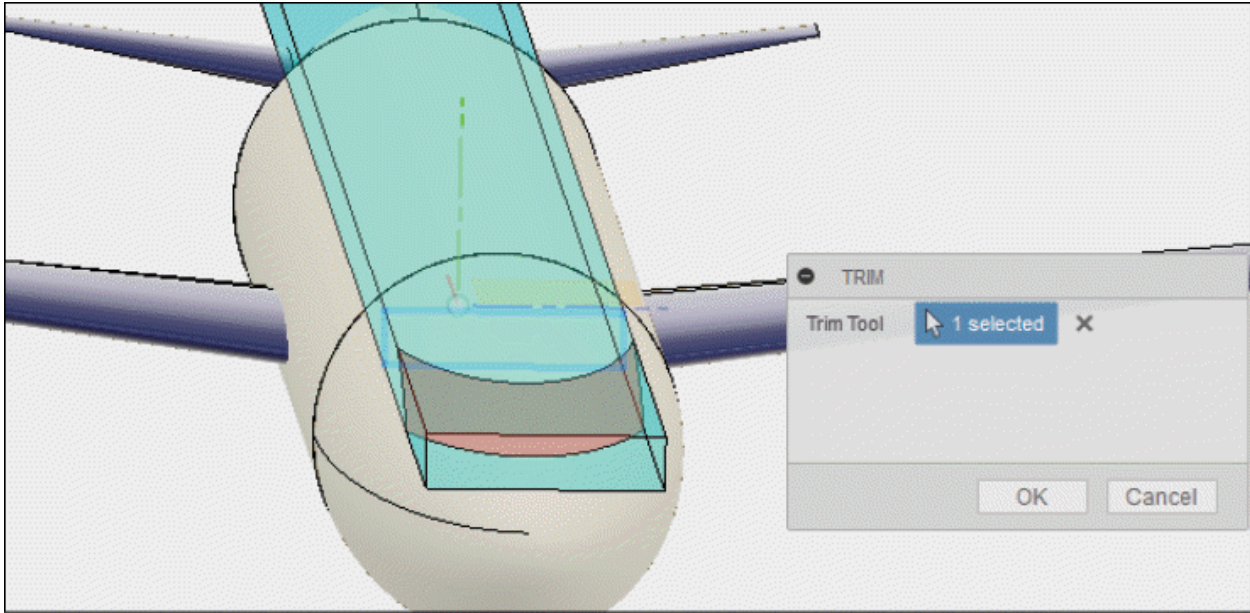


Figure-65. Selection of area for trim

- After selecting, click on the **OK** button of **TRIM** dialog box to exit the **Trim** tool.
- Click on **Loft** tool from **CREATE** drop-down. The **LOFT** dialog box will be displayed.
- Select the area in **Profiles** section as displayed in [Figure-66](#).

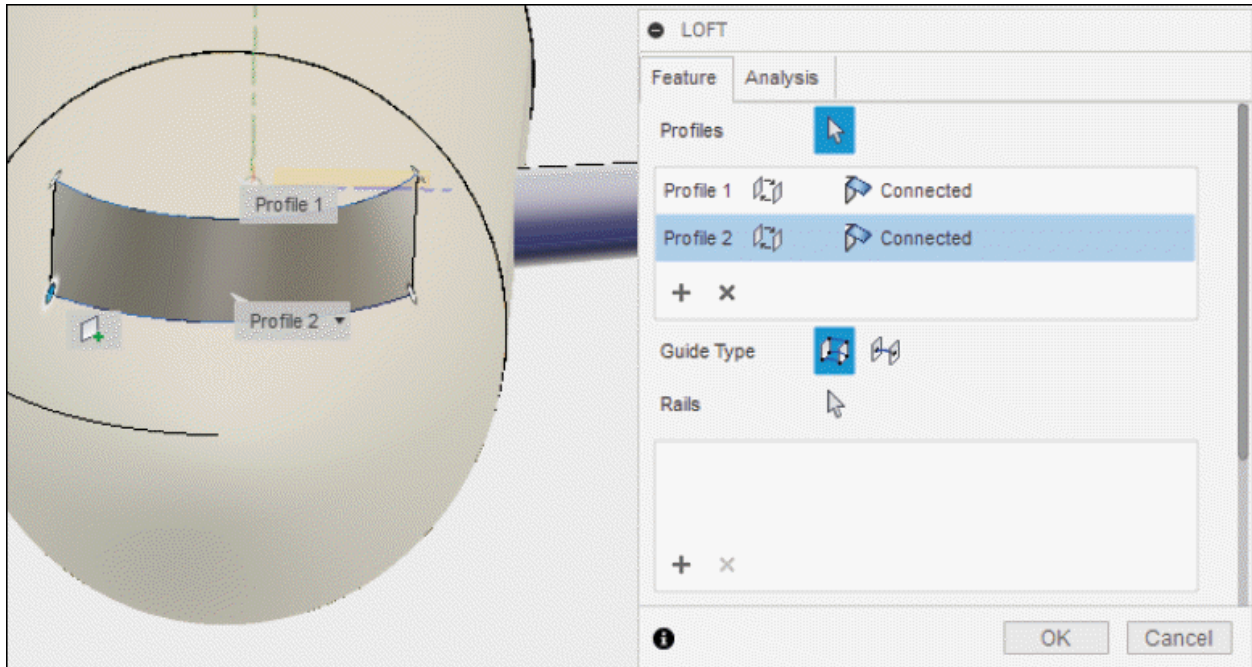


Figure-66. Creating windshield

- Click on the **OK** button from **LOFT** dialog box to complete the process.

Removing extra material

- Click on the **Trim** tool from **MODIFY** drop-down. The **TRIM** dialog box will be displayed; refer to [Figure-67](#).

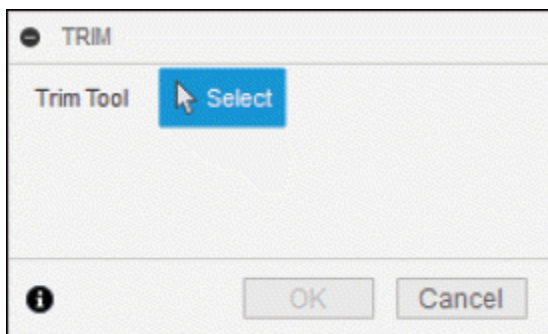


Figure-67. TRIM dialog box

- The **Select** button of **Trim Tool** section is active by default. Click on the extra material to remove; refer to

[Figure-68.](#)

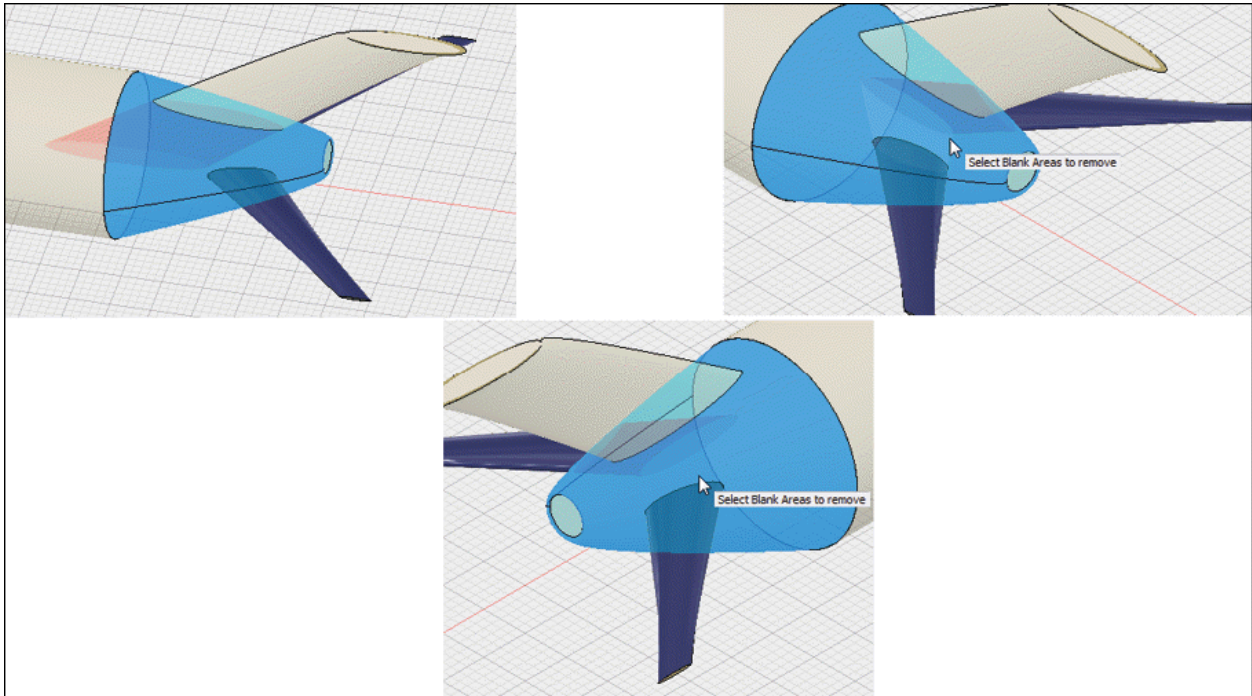


Figure-68. Removing extra material

- After selection of material to remove, click on the **OK** button from **TRIM** dialog box.
- After applying all the tools from above section. The aircraft will be created completely; refer to [Figure-69](#).

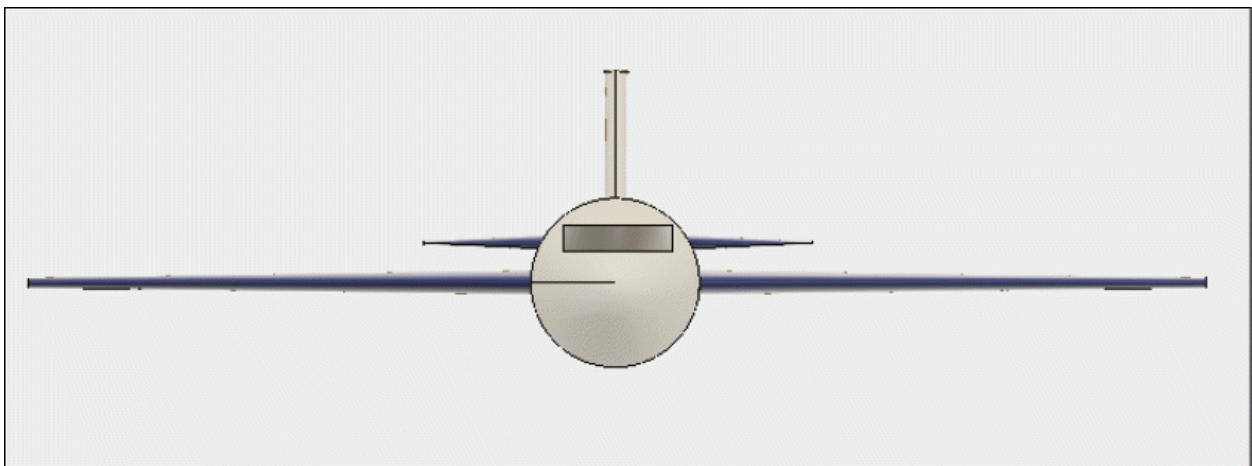


Figure-69. Front view of aircraft

PRACTICAL 2

Create the model of Jug as shown in [Figure-70](#) and [Figure-71](#).



Figure-70. Model of Jug



Figure-71. Another view of Jug

Starting Autodesk Fusion 360

- Double-click on the **Autodesk Fusion 360** software icon from desktop if the software has not started yet.
- Click on the **SURFACE** tab from **Toolbar** in **DESIGN** workspace. The tools to create surface will be activated.

Creating Sketch

The first step to create jug is to create sketches. The procedure is discussed next.

- Click on the **Create Sketch** tool of **CREATE** drop-down from **Toolbar** and select the **Top** plane as reference for

sketch.

- Create the sketch as shown in [Figure-72](#). Click on the **Finish Sketch** tool to exit.

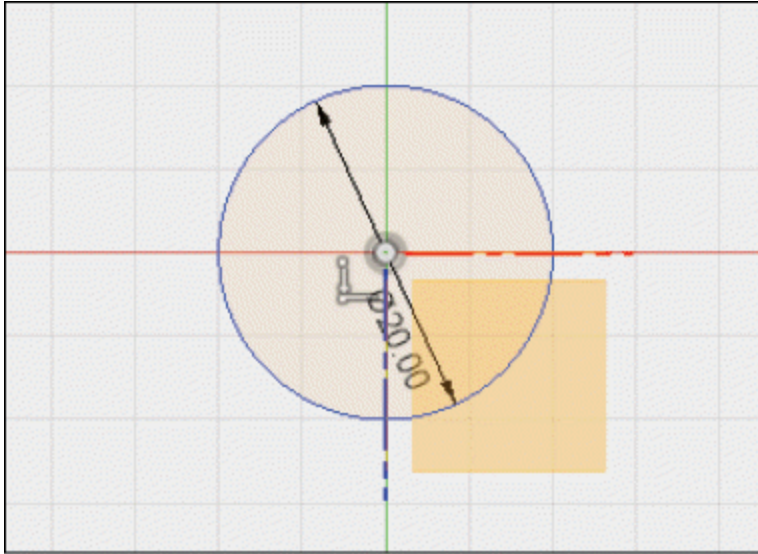


Figure-72. Creating base sketch

- Click on **Offset Plane** tool of **CONSTRUCT** drop-down from **Toolbar**. The **OFFSET PLANE** dialog box will be displayed.
- The **Select** button of **Plane** section is active by default. Select the **Top** plane as reference.
- Click in the **Distance** edit box and enter the value as **2**; refer to [Figure-73](#).

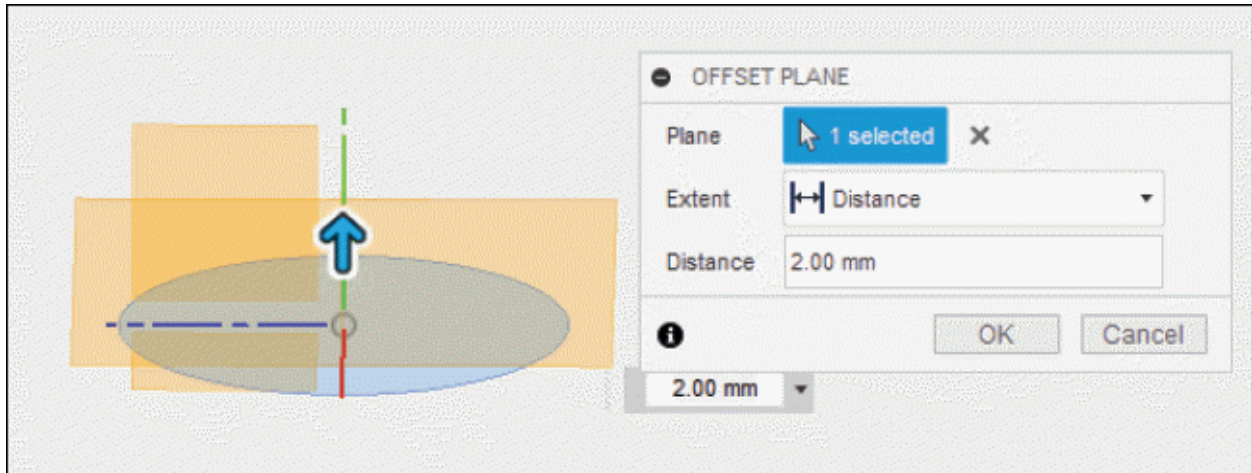


Figure-73. Creating first offset plane

- After specifying the parameters, click on the **OK** button from **OFFSET PLANE** dialog box to complete the process of creating plane.
- Create a center diameter circle on recently created offset plane; refer to [Figure-74](#).

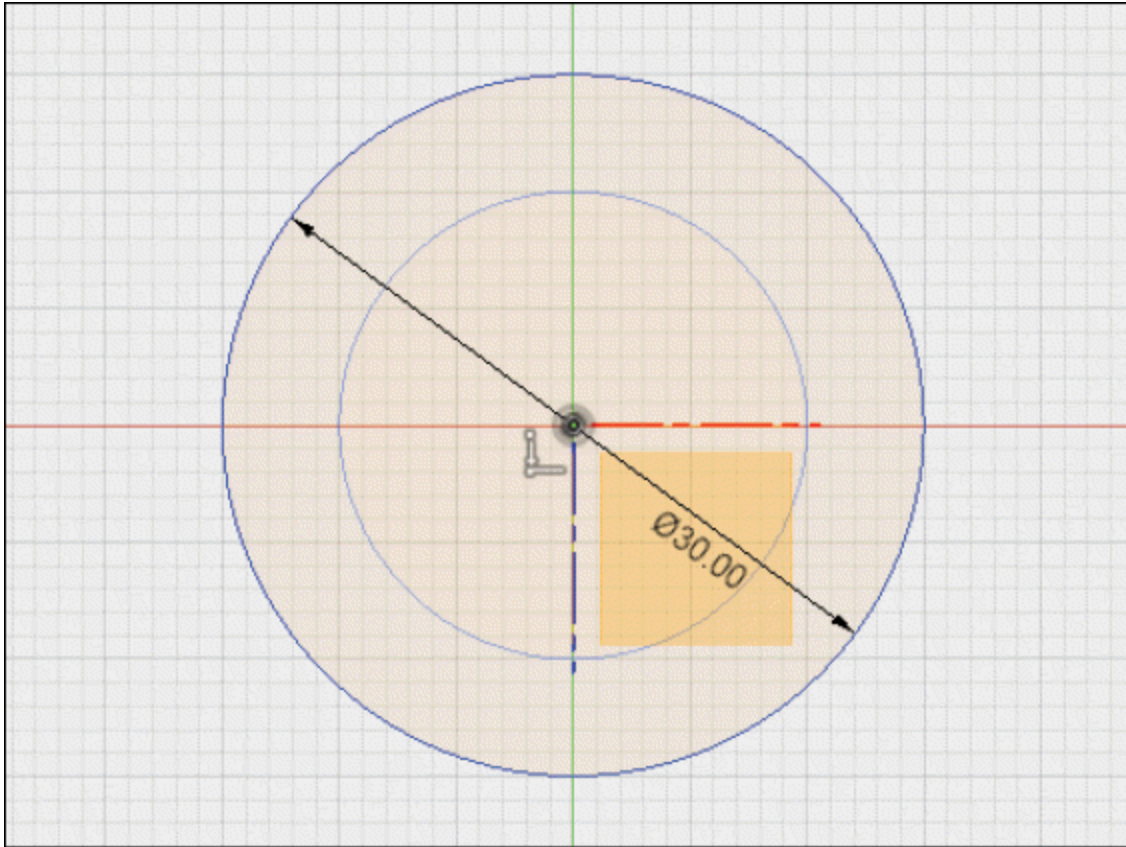


Figure-74. Creating second sketch

- Click on the **Patch** tool from **CREATE** drop-down. The **PATCH** dialog box will be displayed.
- The **Select** button of **Boundary Edges** section from **PATCH** dialog box is active by default. You need to select the first sketch to patch.
- After specifying the parameters, click on the **OK** button from **PATCH** dialog box to complete the process; refer to [Figure-75](#).

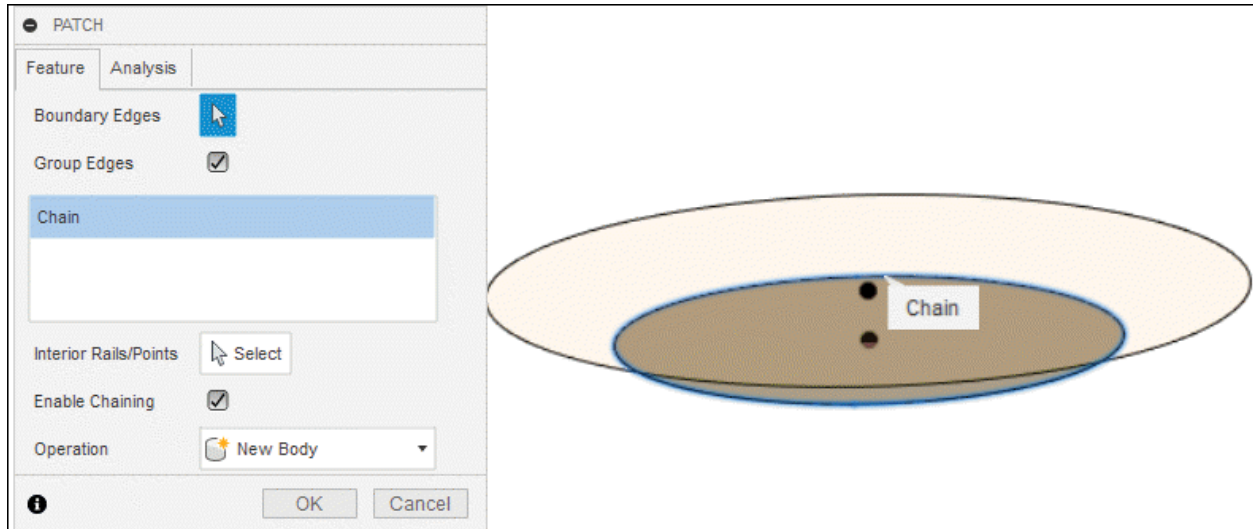


Figure-75. Creating Patch of first sketch

- Click on the **Offset Plane** tool of **CONSTRUCT** drop-down from **Toolbar**. The **OFFSET PLANE** dialog box will be displayed.
- The **Select** button of **Plane** section is active by default. Select the first offset plane as reference.
- Click in the **Distance** edit box and enter the value as **22**; refer to [Figure-76](#).

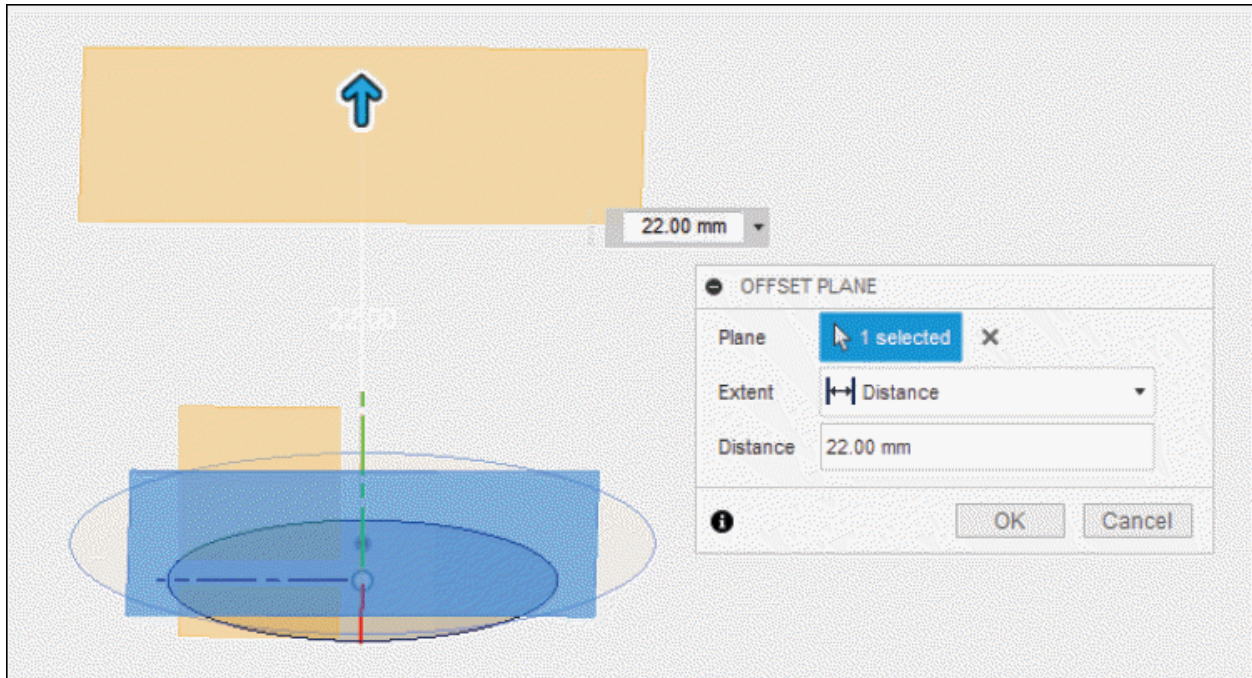


Figure-76. Creating third offset plane

- After specifying the parameter, click on the **OK** button from **OFFSET PLANE** dialog box to create the plane.
- Create the sketch as shown in [Figure-77](#) on newly created offset plane.

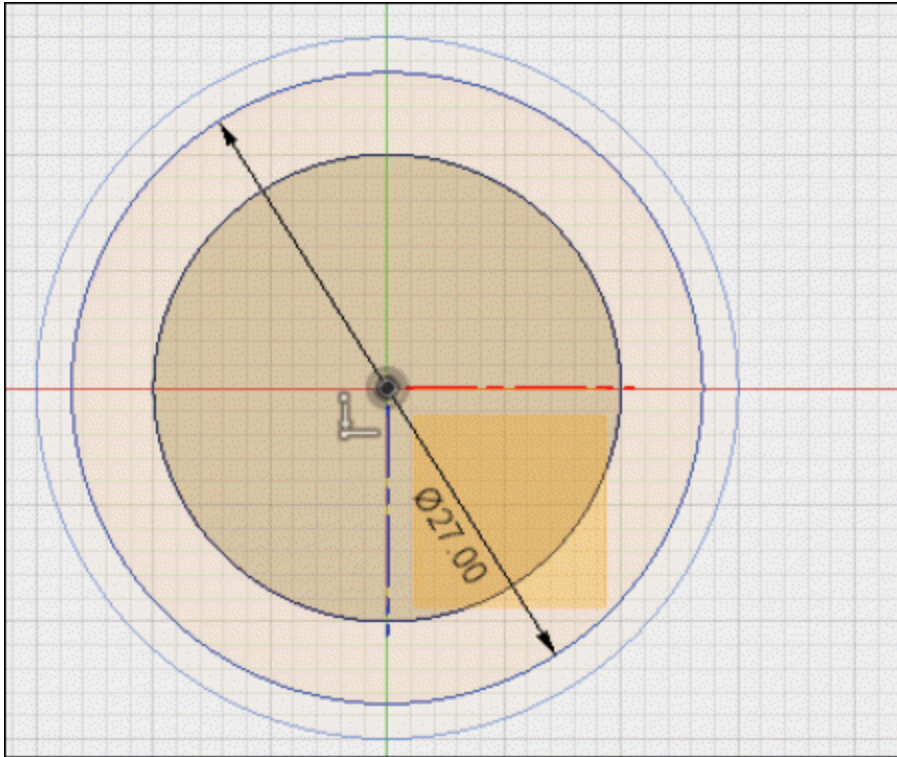


Figure-77. Creating third sketch

- Click on **Finish Sketch** button from **Toolbar** to exit the sketch.
- Click on the **Offset Plane** tool of **CONSTRUCT** drop-down from **Toolbar**. The **OFFSET PLANE** dialog box will be displayed.
- The **Select** button of **Plane** section is active by default. Select the second offset plane as reference.
- Click in the **Distance** edit box and enter the value as **7**; refer to [Figure-78](#). You can also set the distance by moving the drag handle.

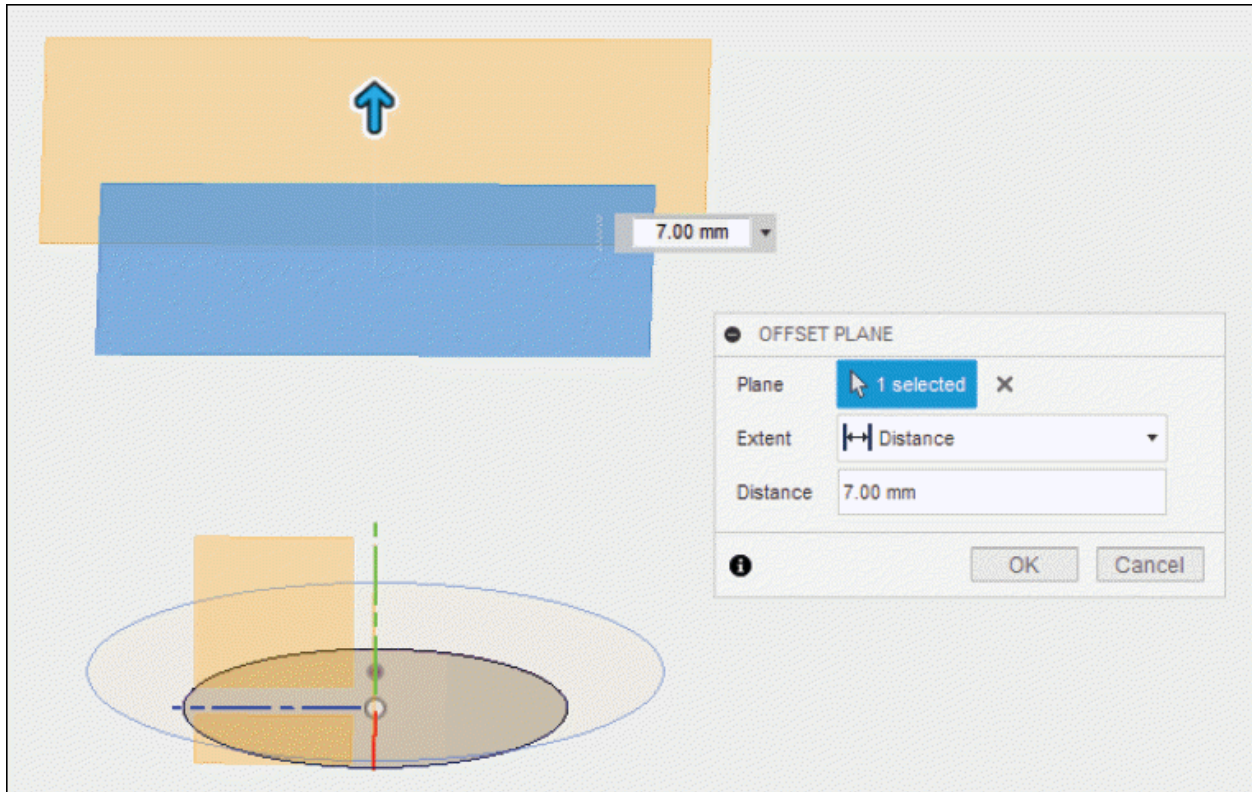


Figure-78. Creation of third offset plane

- After specifying the parameter, click on the **OK** button from **OFFSET PLANE** dialog box to create the plane.
- Create the sketch as shown in [Figure-79](#) on newly created offset plane.

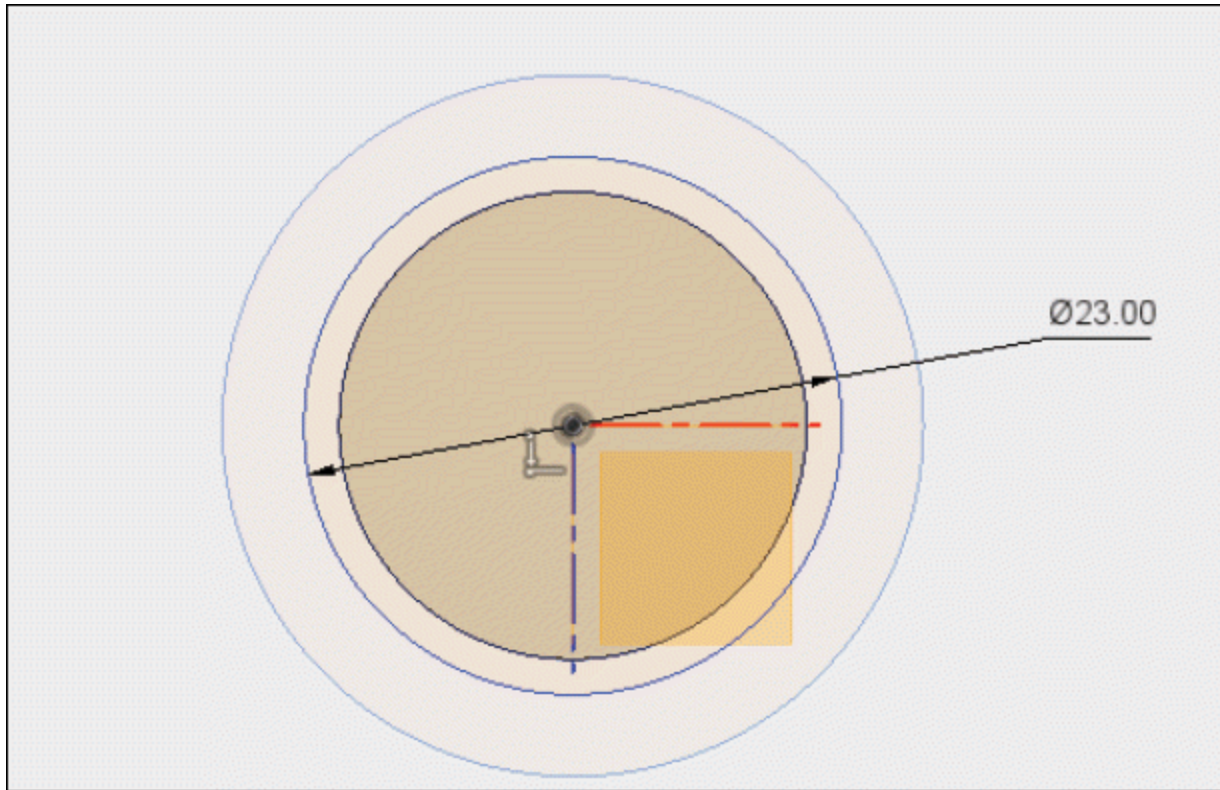


Figure-79. Creating third sketch of jug

- Click on the **Offset Plane** tool of **CONSTRUCT** drop-down from **Toolbar**. The **OFFSET PLANE** dialog box will be displayed.
- The **Select** button of **Plane** section is active by default. Select the third offset plane as reference.
- Click in the **Distance** edit box and specify the value as **8**; refer to [Figure-80](#). You can also set the distance by moving the drag handle.

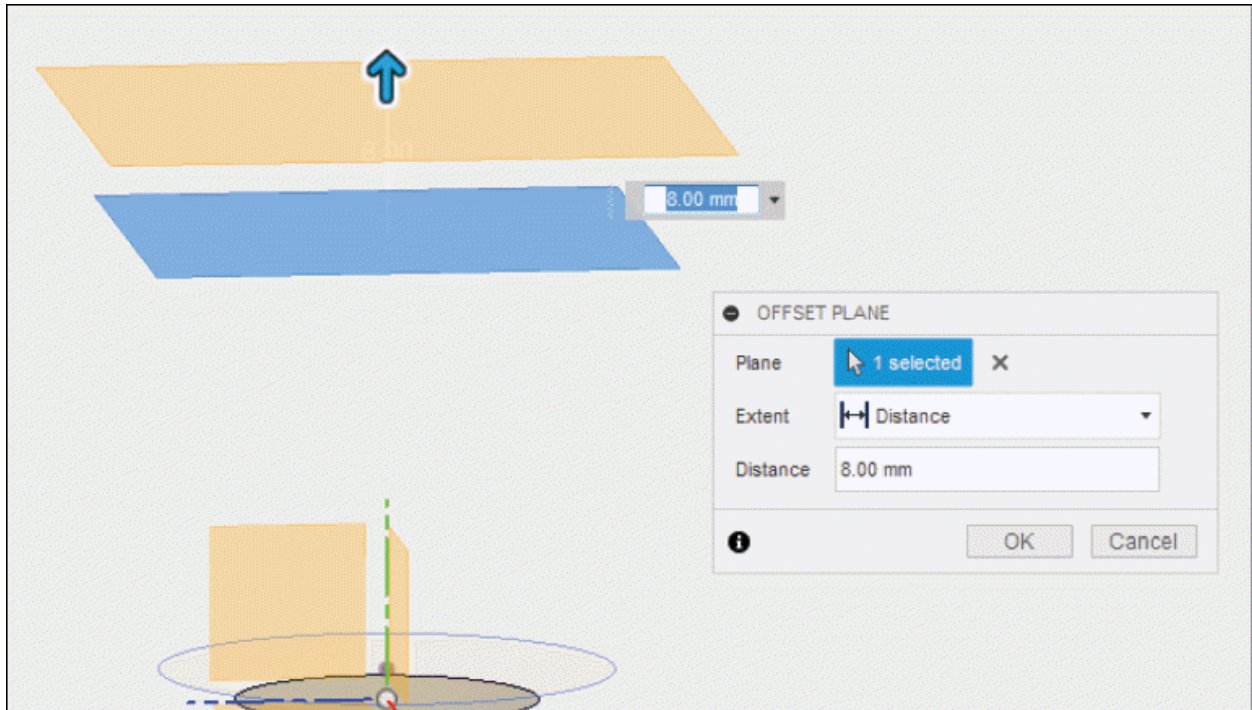


Figure-80. Creating fourth offset plane for jug

- After specifying the parameter, click on the **OK** button from **OFFSET PLANE** dialog box to create the plane.
- Create the sketch as shown in [Figure-81](#) on newly created offset plane.

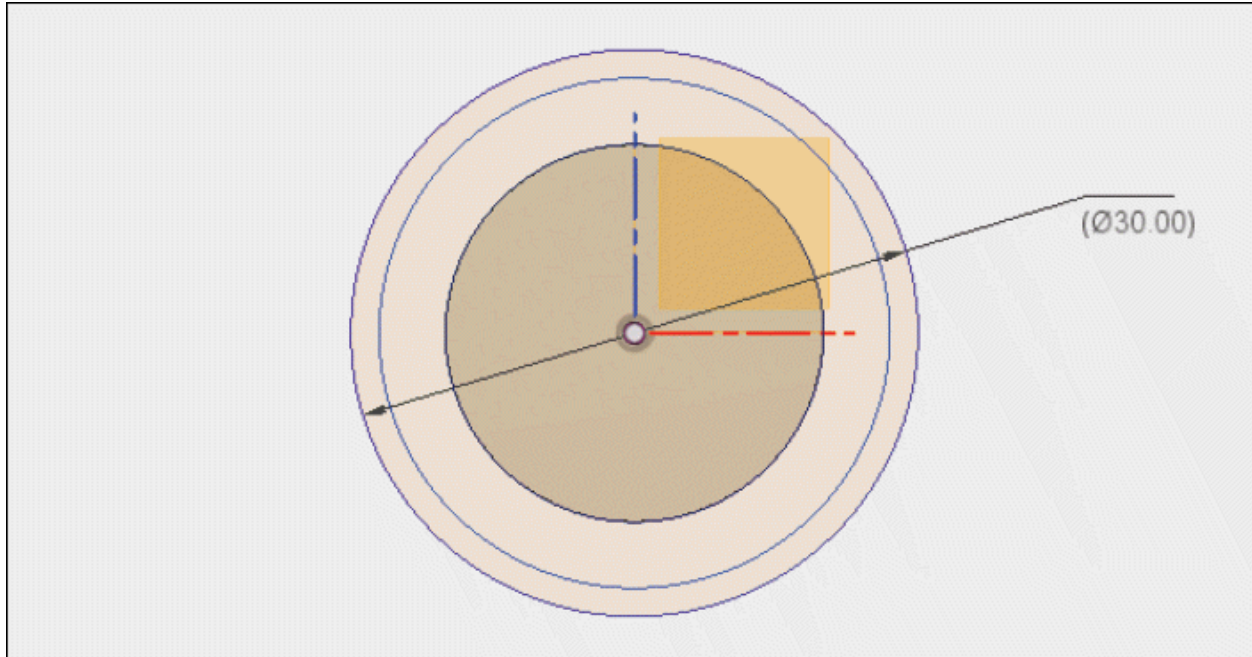


Figure-81. Creating fifth sketch for Jug

- Click on the **Offset Plane** tool of **CONSTRUCT** drop-down from **Toolbar**. The **OFFSET PLANE** dialog box will be displayed.
- The **Select** button of **Plane** option is active by default. Select the third offset plane as reference.
- Click in the **Distance** edit box and enter the value as **8**; refer to [Figure-82](#). You can also set the distance by moving the drag handle.

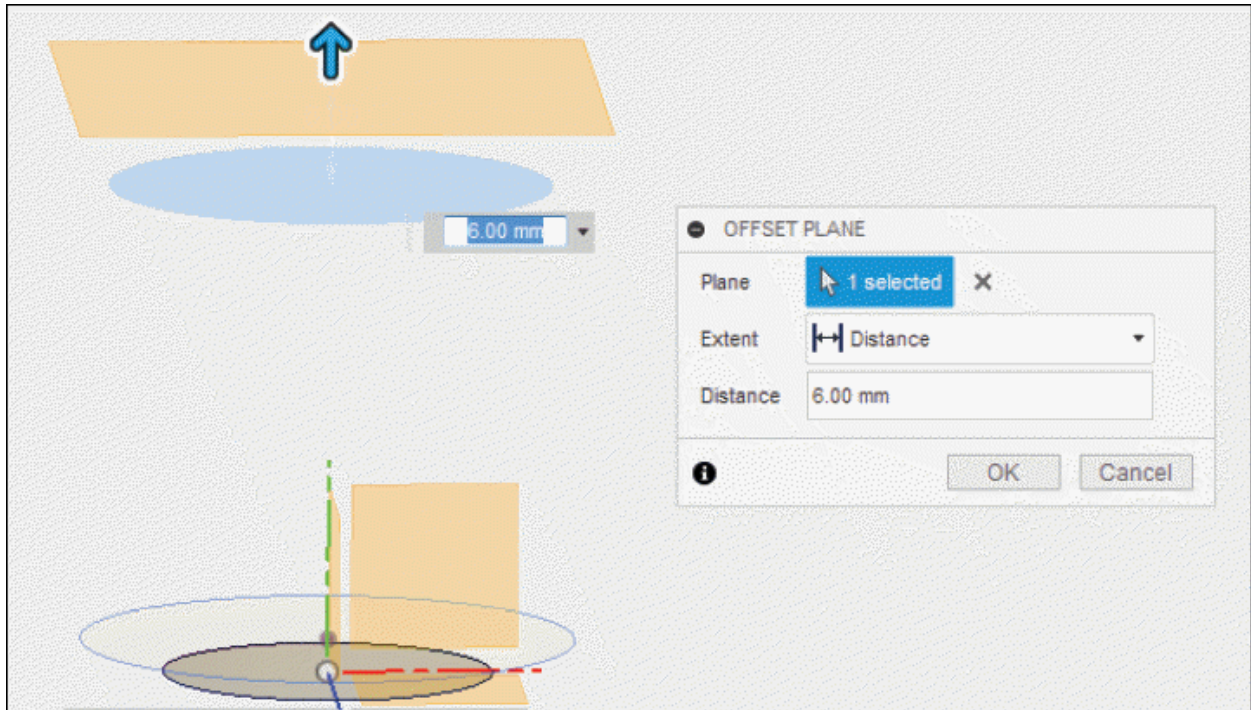


Figure-82. Creating fifth offset plane

- After specifying the parameter, click on the **OK** button from **OFFSET PLANE** dialog box to create the plane.
- Click on the **Circle** tool of **CREATE** drop-down from **Toolbar** and select the recently created plane as reference.
- Create the sketch as shown in [Figure-83](#) with the help of **Circle**, **Trim**, and **Arc** tool.

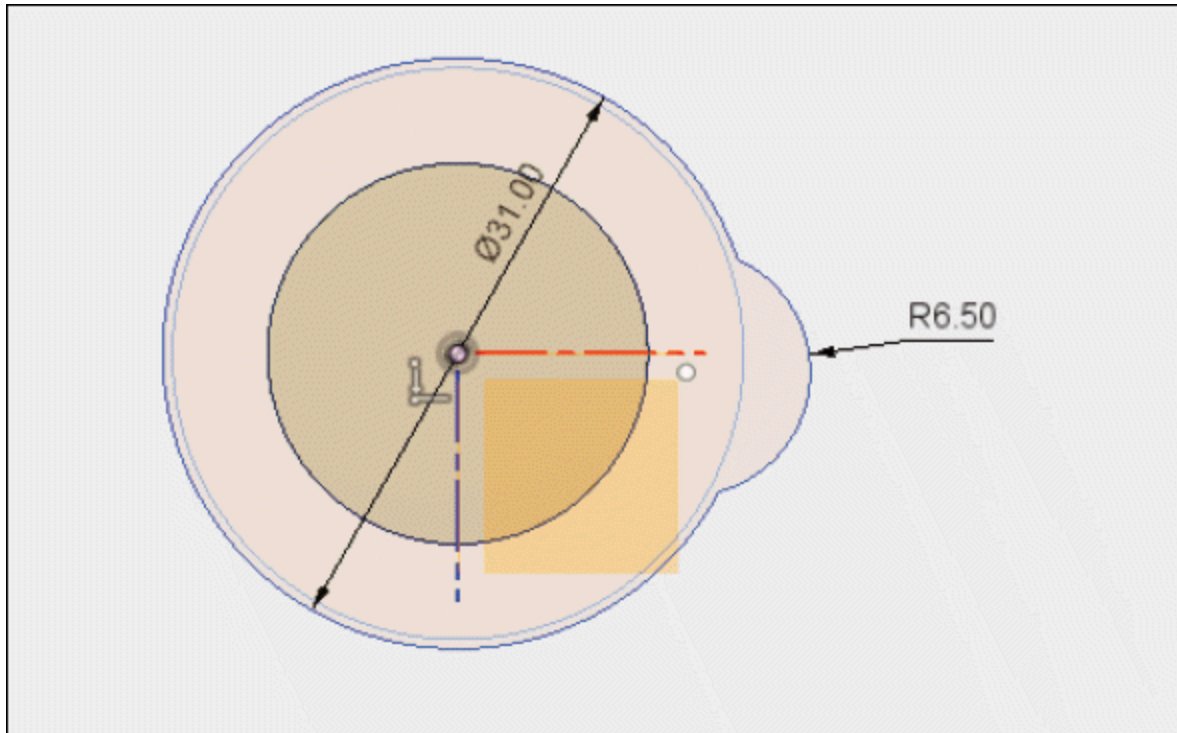


Figure-83. Creating sixth sketch

Creating Loft feature

Till now, we have created the sketch for jug. Now, we will apply the loft feature to join all the sketches. The procedure is discussed next.

- Click on the **Loft** tool of **CREATE** drop-down from **Toolbar**. The **LOFT** dialog box will be displayed.
- You need to select all the sketch from bottom to top in **Profiles** section of **LOFT** dialog box; refer to [Figure-84](#).

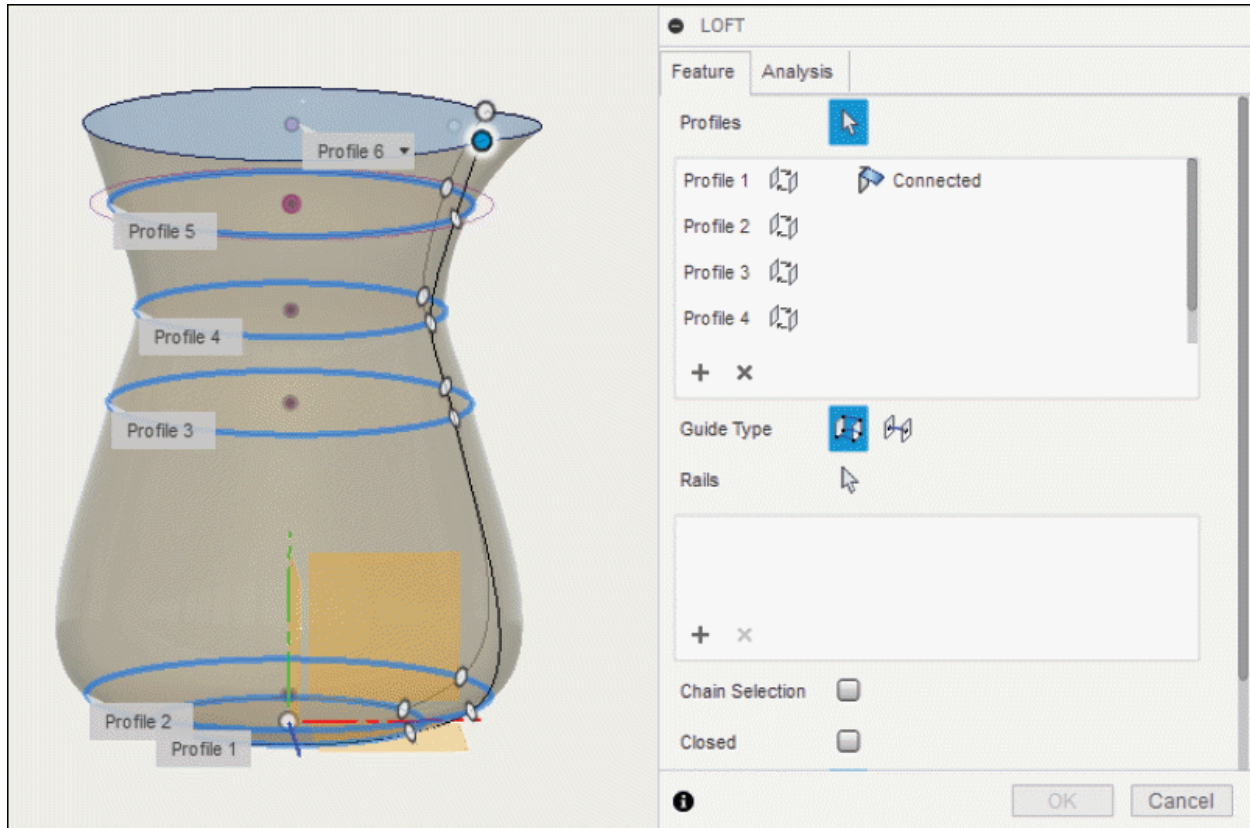


Figure-84. Creating loft feature of jug

- After selecting the profiles, click on the **OK** button from **LOFT** dialog box to complete the loft feature.

Here loft feature is completed. Now, we will create the handle of jug.

- Click on the **Offset Plane** tool of **CONSTRUCT** drop-down from **Toolbar**. The **OFFSET PLANE** dialog box will be displayed.
- The **Select** button of **Plane** section is active by default. Select the **YZ** plane as reference.
- Click in the **Distance** edit box and specify the value as **-10**; refer to [Figure-85](#).

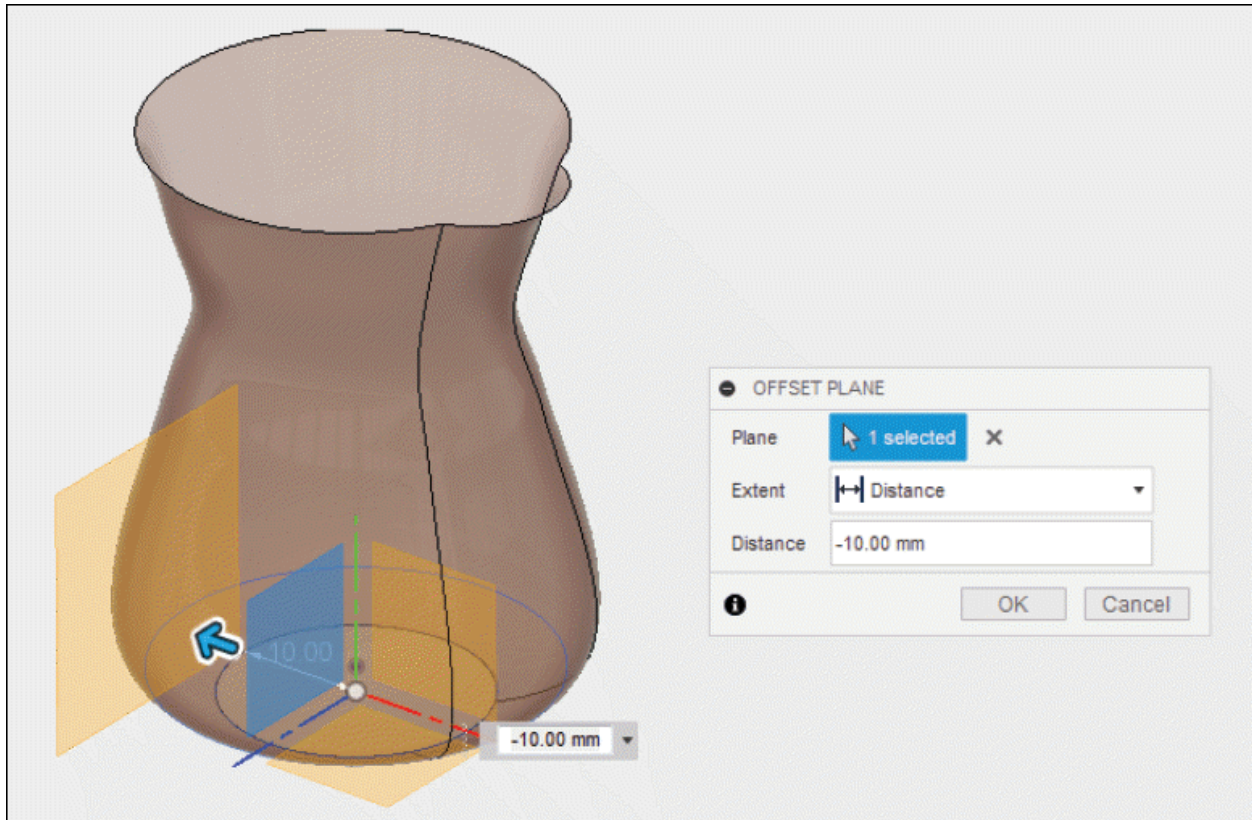


Figure-85. Creating sixth offset plane

- After specifying the parameter, click on the **OK** button from **OFFSET PLANE** dialog box to create the plane.
- Create the sketch as shown in [Figure-86](#) on newly offset plane.

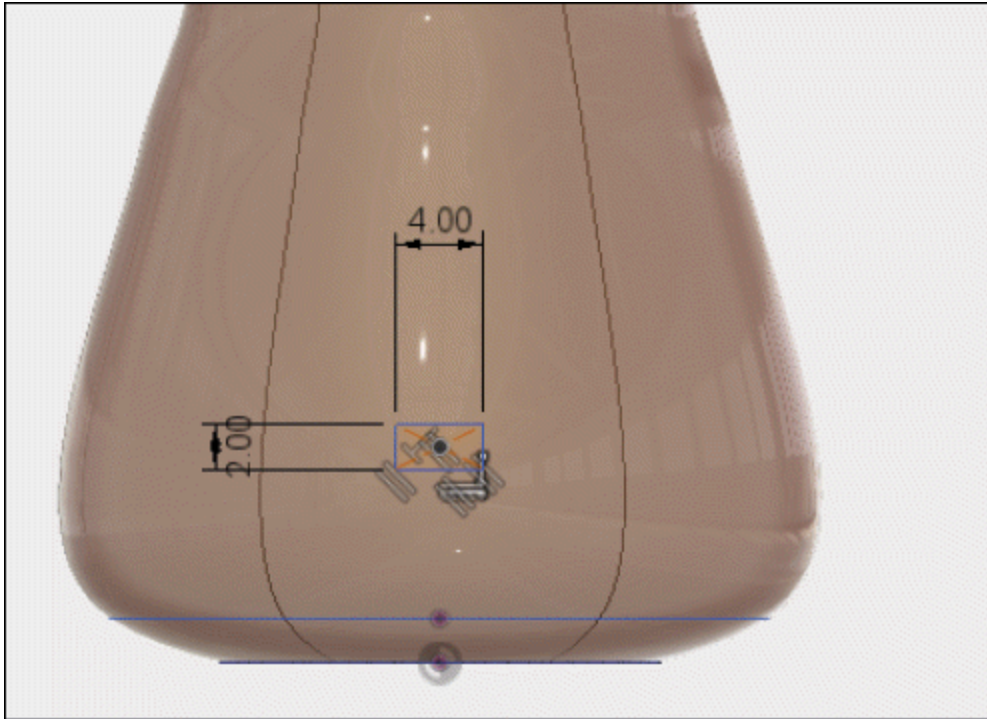


Figure-86. Creating seventh sketch

- Click on the **Create Sketch** tool from **CREATE** drop-down and select the **XZ** plane.
- Create the sketch as shown in [Figure-87](#).

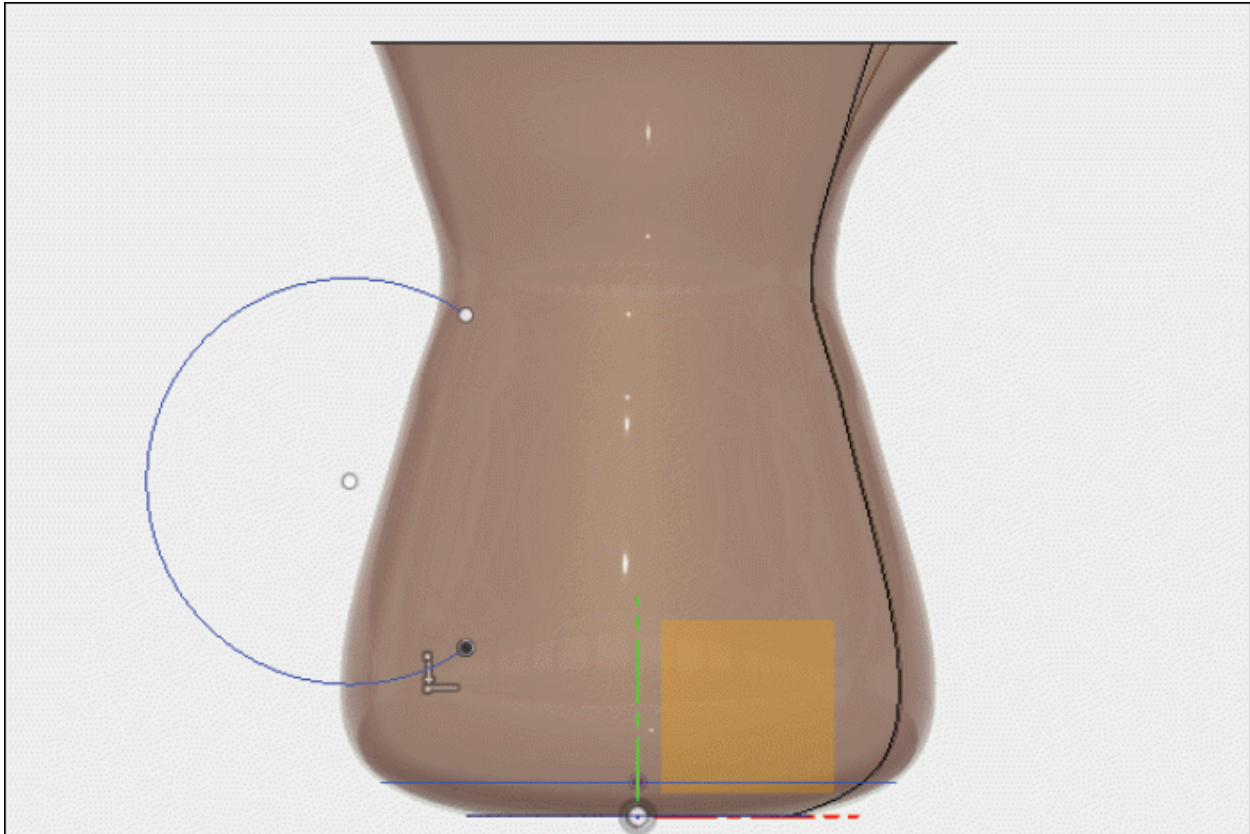


Figure-87. Creating eighth sketch

Creating Sweep feature

- Click on the **Sweep** tool from **CREATE** drop-down. The **SWEEP** dialog box will be displayed; refer to [Figure-88](#).

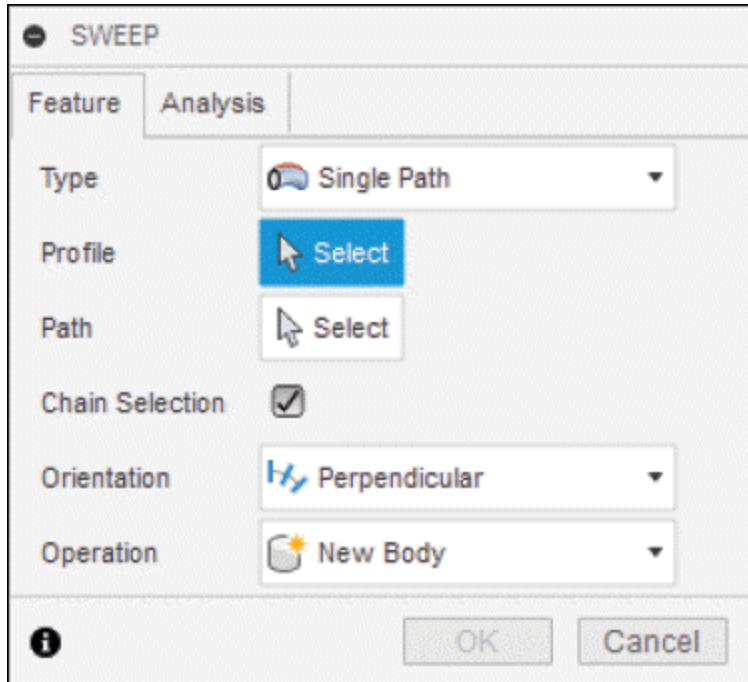


Figure-88. SWEEP dialog box

- The **Select** button of **Profile** section is active by default. You need to select the recently created rectangular sketch.
- Click on the **Select** button of **Path** section and select the recently created arc; refer to [Figure-89](#).

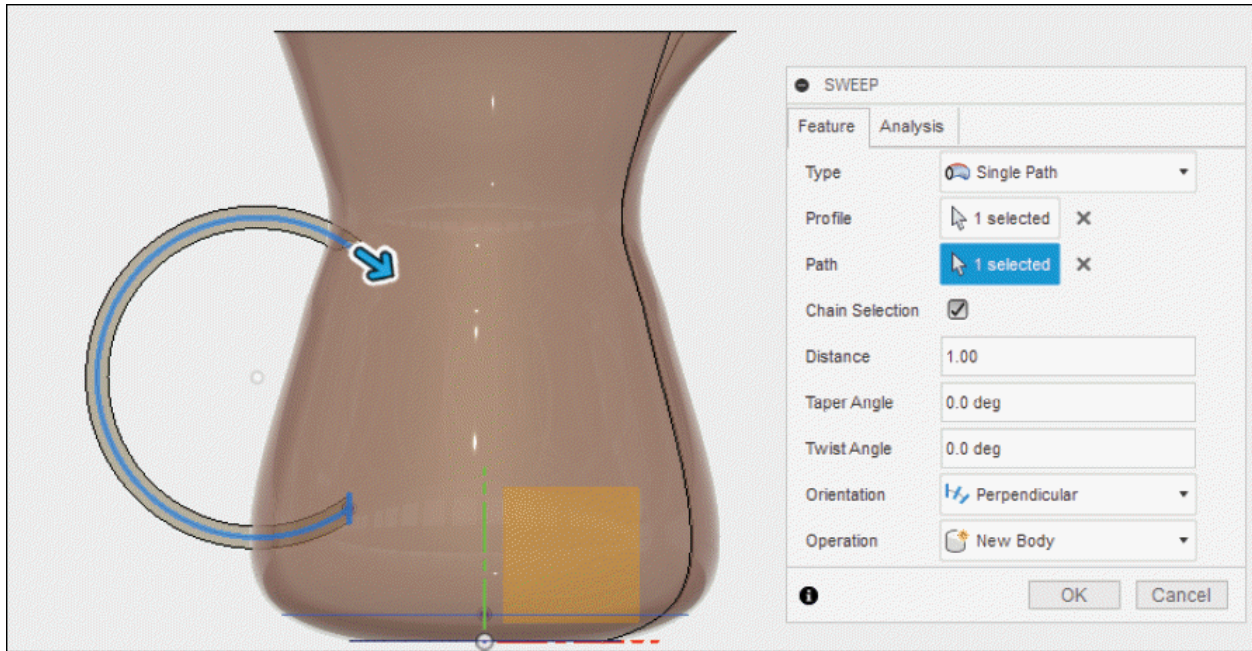


Figure-89. Creating sweep feature for jug

- Click on the **Orientation** drop-down of **SWEEP** dialog box and select the **Perpendicular** option.
- After specifying the parameters, click on the **OK** button from **SWEEP** dialog box to complete the feature.

Till this point, we have created the model of jug but we need to apply the trim tool on the jug to remove the unwanted parts.

Trimming Extra Entities

- Click on the **Trim** tool of **MODIFY** drop-down from **Toolbar**. The **TRIM** dialog box will be displayed.
- The **Select** button of **Trim Tool** section is active by default. Click on the interior of jug and select the unwanted parts; refer to [Figure-90](#).

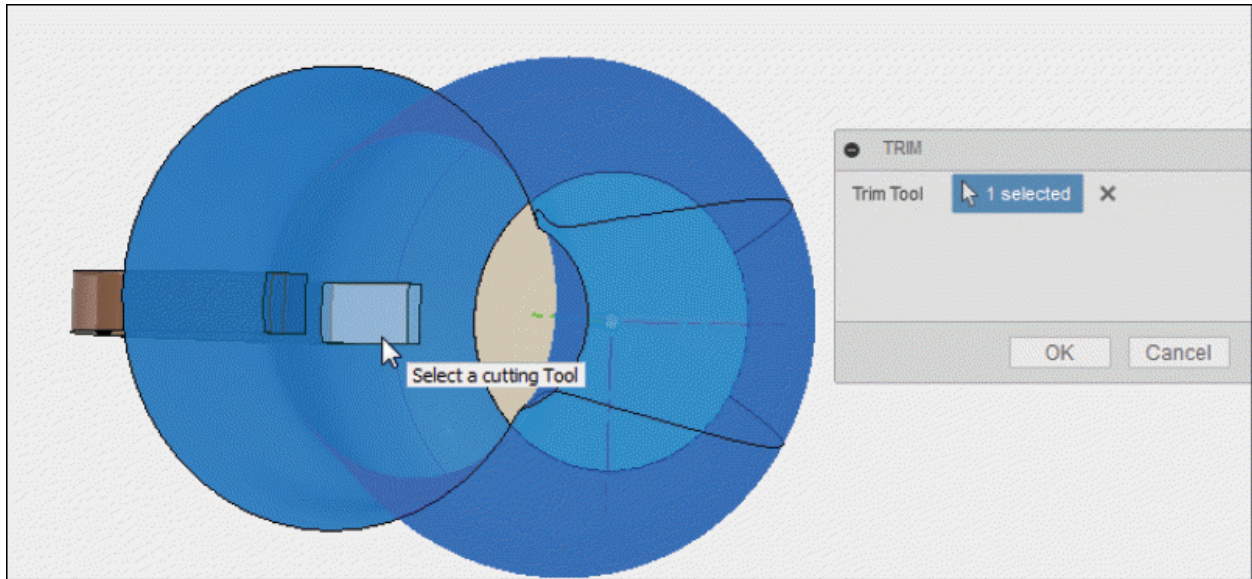


Figure-90. Applying the Trim tool

- After selecting, click on the **OK** button from **Trim** dialog box to trim the material.
- The model of jug is created successfully; refer to [Figure-91](#).

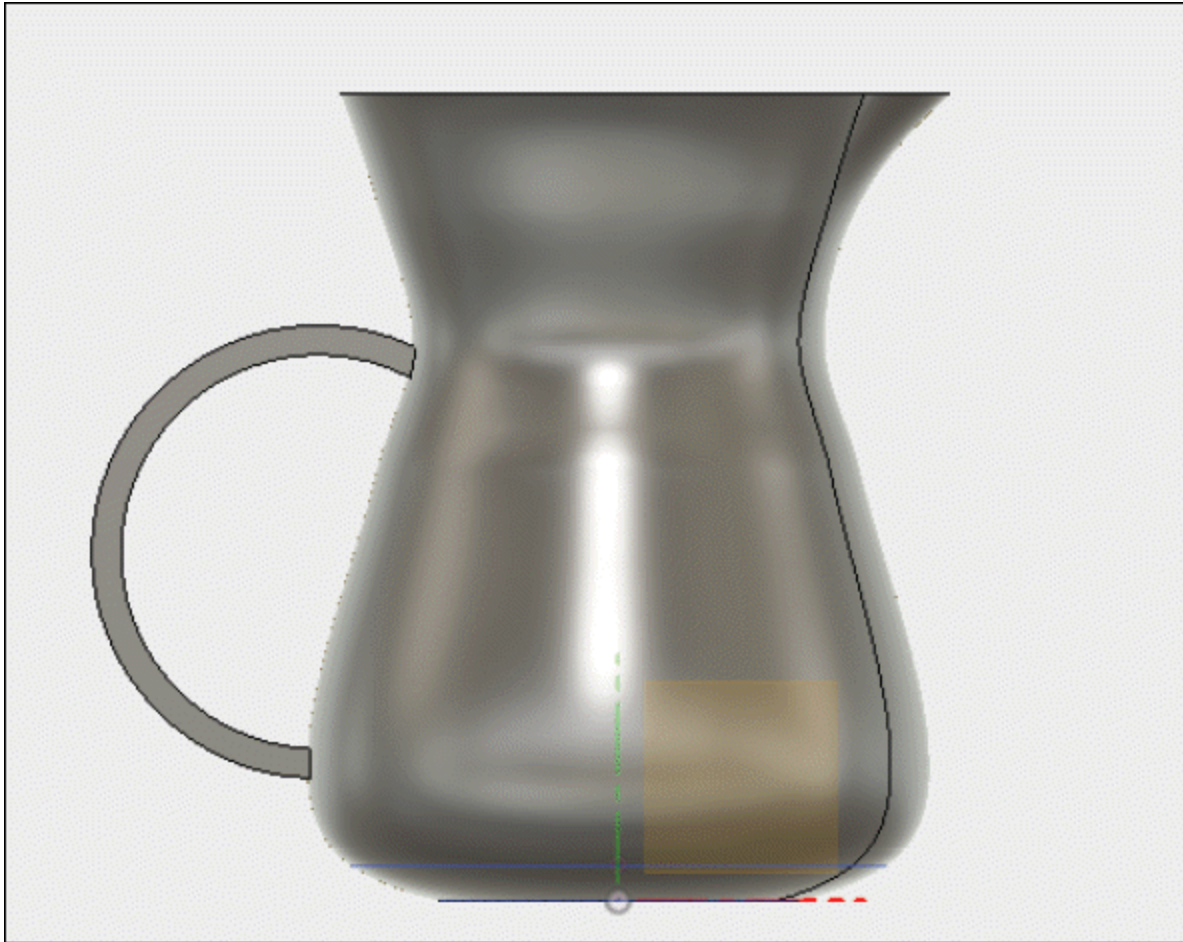


Figure-91. Model of Jug created

- You can apply the physical material or appearance as required to the jug.
- You can also create the model of jug with the help of **Revolve** tool in place of **Loft** tool.

SELF ASSESSMENT

Q1. What is Surfacing? Explain in brief.

Q2. Why surface models can not exist in real world and how they can be made real?

Q3. Explain the creation of surface with G0 edges, G1 edges,

and G2 edges.

Q4. How can you perform an extrude protrusion of an open sketch?

Q5. What is the use of **Reverse Normal** tool?

Q6. The tool is used to combine the surfaces into one body.

PRACTICE 1

Create the model of mouse as shown in [Figure-92](#) and [Figure-93](#). Assume appropriate dimensions. If you are working on your system using a computer mouse then you can use that as reference for creating model.

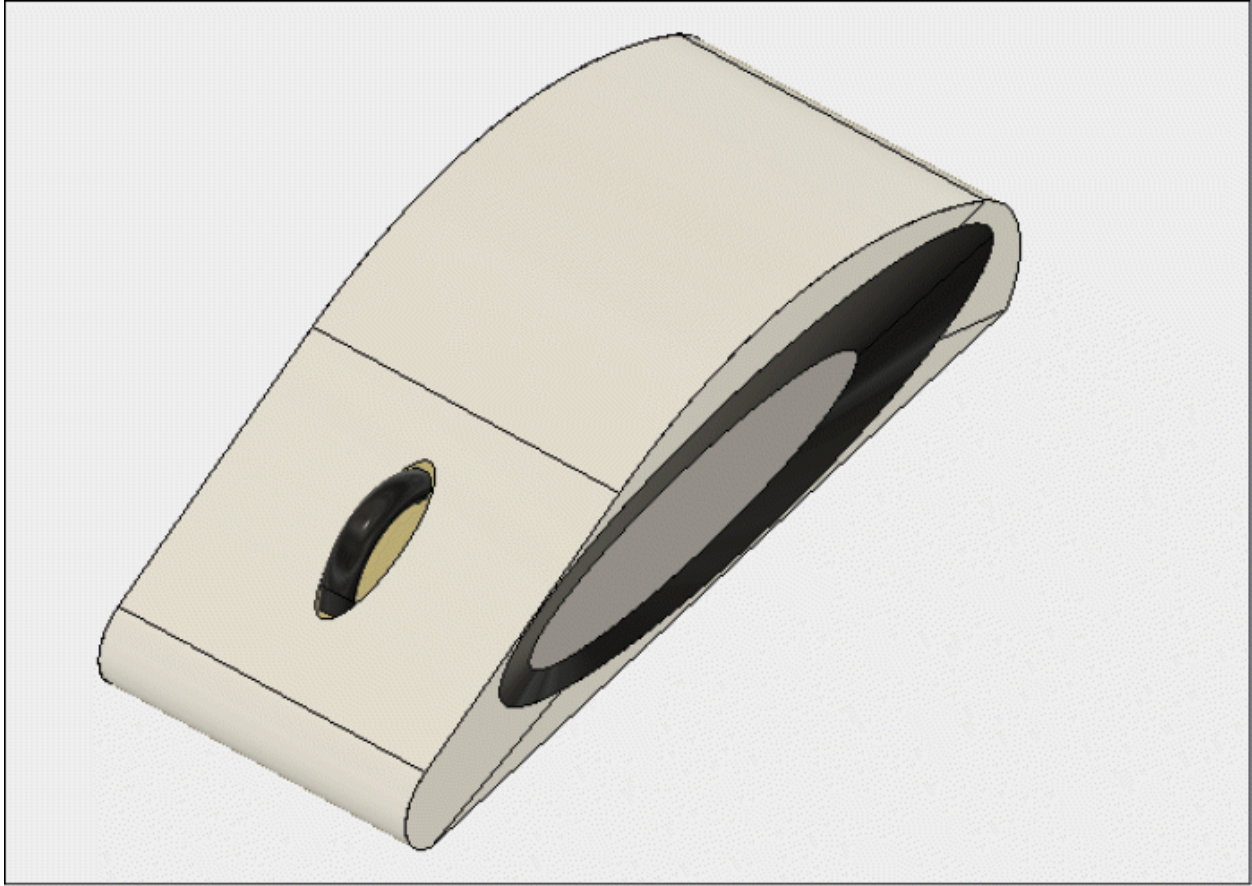


Figure-92. Model of mouse

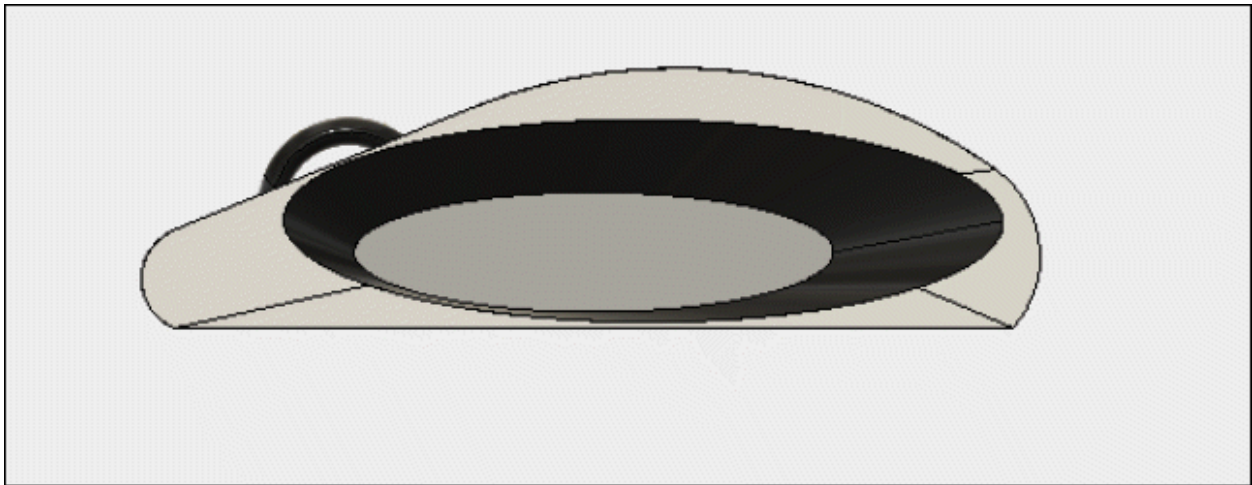


Figure-93. Right view of mouse

Create the model of water bottle as shown in [Figure-94](#) and [Figure-95](#). The drawing view of water bottle is shown in [Figure-96](#). (You are free to apply your artistic mind for improving design of bottle and when at it, why not start a competition in your class for designing bottle.)

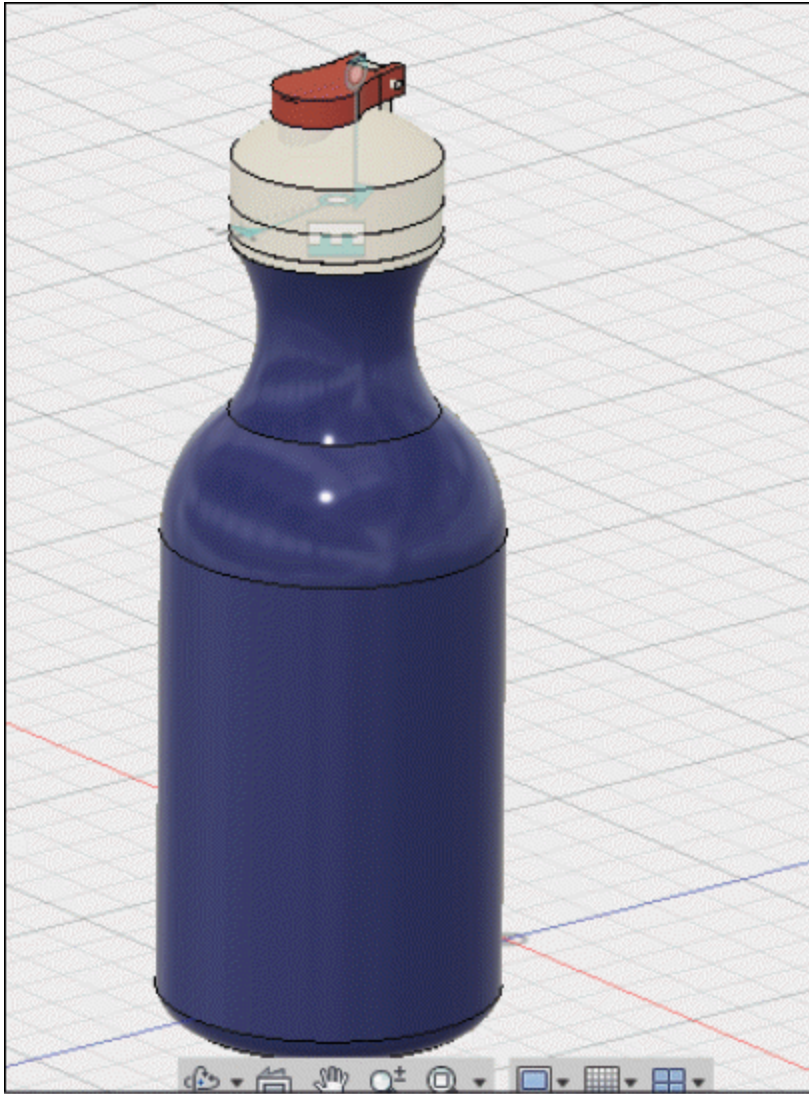


Figure-94. Model of Water Bottle



Figure-95. Top view of water bottle

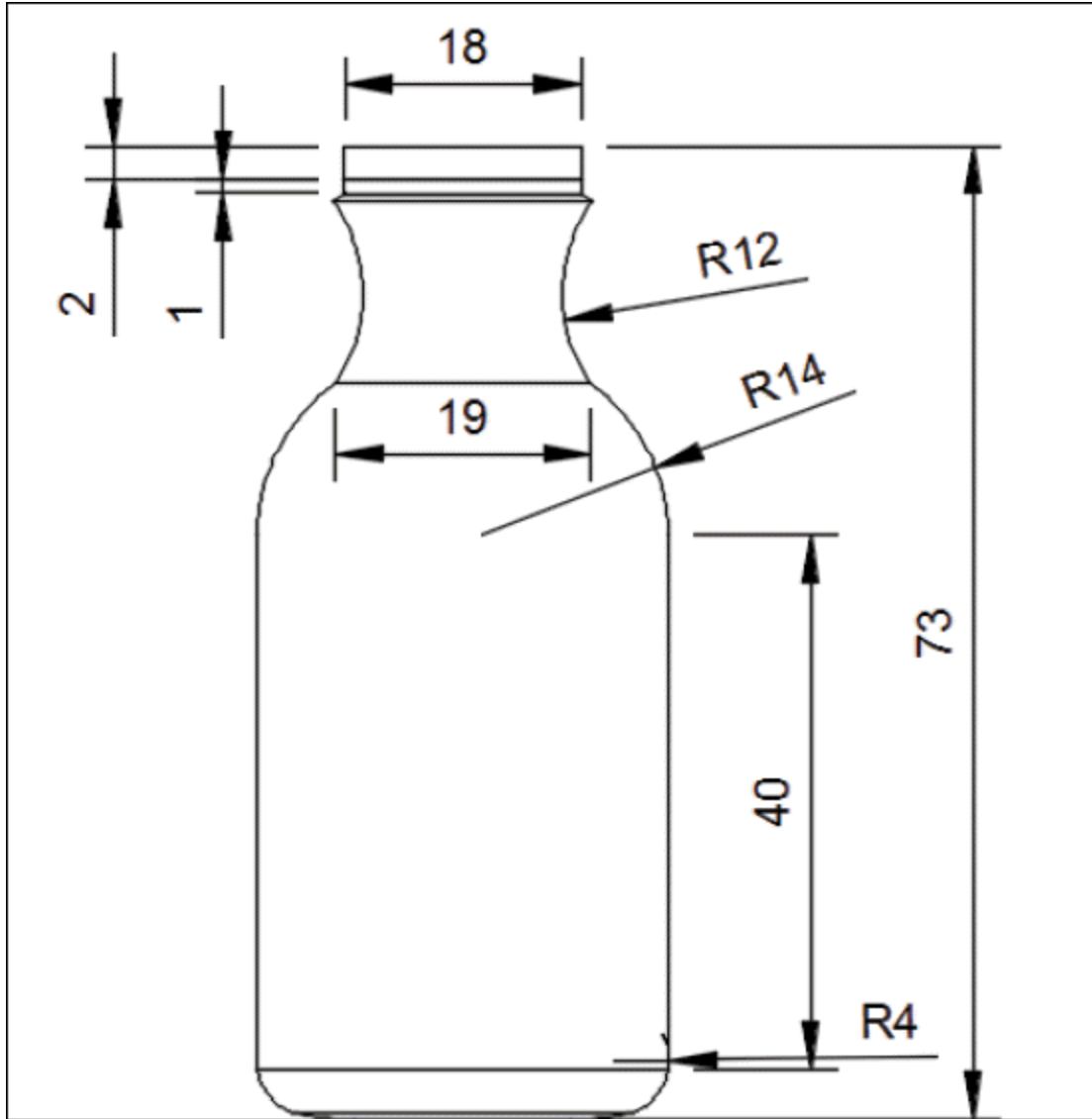


Figure-96. Dimensions of Water Bottle

Chapter 10

Rendering and Animation

Topics Covered

The major topics covered in this chapter are:

- *Introduction to Rendering*
- *Appearance Tool*
- *Scene Settings Tool*
- *Environment Library Tab*
- *Texture Map Control*
- *In-Canvas Render Setting*
- *Capture Image Tool*
- *Render*
- *Introduction to Animation Workspace*
- *New Storyboard*
- *Transform Components*
- *Restore Home Tool*
- *Auto Explode : One Level*
- *Auto Explode : All Levels*
- *Manual Explode*
- *Show/Hide Tool*
- *Create Callout Tool*
- *View Tool*
- *Publish Video Tool*

INTRODUCTION

When a designer is working on a 3D Model, the model he/she creates is generally a mathematical representation of points and surfaces in three-dimensions. Sometimes, a realistic 2D image of 3D model is required for presentation to the client. The conversion of 3D model with specified environmental scenario through mathematical approximation to a finalized 2D image is known as Rendering. During this process, the entire scene's textural and lighting information are combined to determine the color value of each pixel in the finalized image. To perform

rendering, select the **RENDER** option from the **Change Workspace** drop-down in the **Ribbon**.

INTRODUCTION TO RENDER WORKSPACE

The tools in **Render Workspace** are used to create photo-realistic image from a scene of model. There are various tools used for rendering process which are discussed next.

Appearance Tool

The **Appearance** tool is used to apply different materials to selected bodies/components/faces. The procedure to use this tool has been discussed earlier in the book. So, we are not repeating the tool here.

Scene Settings

The **Scene Settings** tool is used to control the lighting, background color, ground effects, and camera rendering. The procedure to use this tool is discussed next.

- Click on the **Scene Settings** tool of **SETUP** panel from **Toolbar**; refer to [Figure-1](#). The **SCENE SETTINGS** dialog box will be displayed; refer to [Figure-2](#).

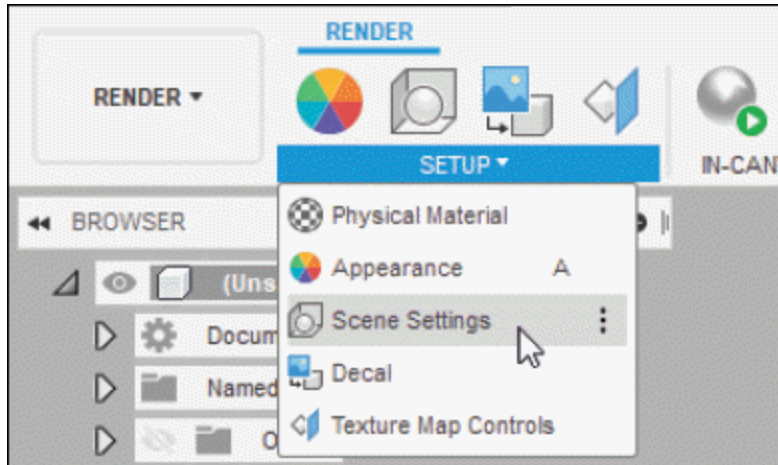


Figure-1. Scene Settings tool

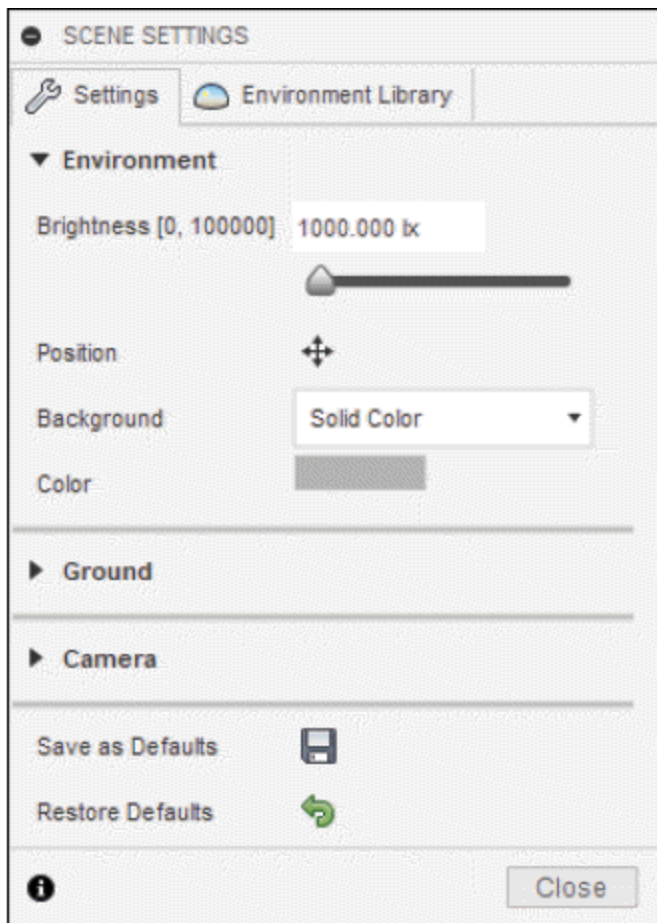


Figure-2. SCENE SETTINGS dialog box

Environment section

- Click on the **Brightness** edit box and specify the value of brightness for model. You can also move the **Brightness** slider to adjust the brightness for the model.
- If you want to control the position and rotation of lights then click on the **Position** button of **Environment** section from **Settings** tab. The drag handle along with some tools will be displayed on the canvas screen; refer to [Figure-3](#).

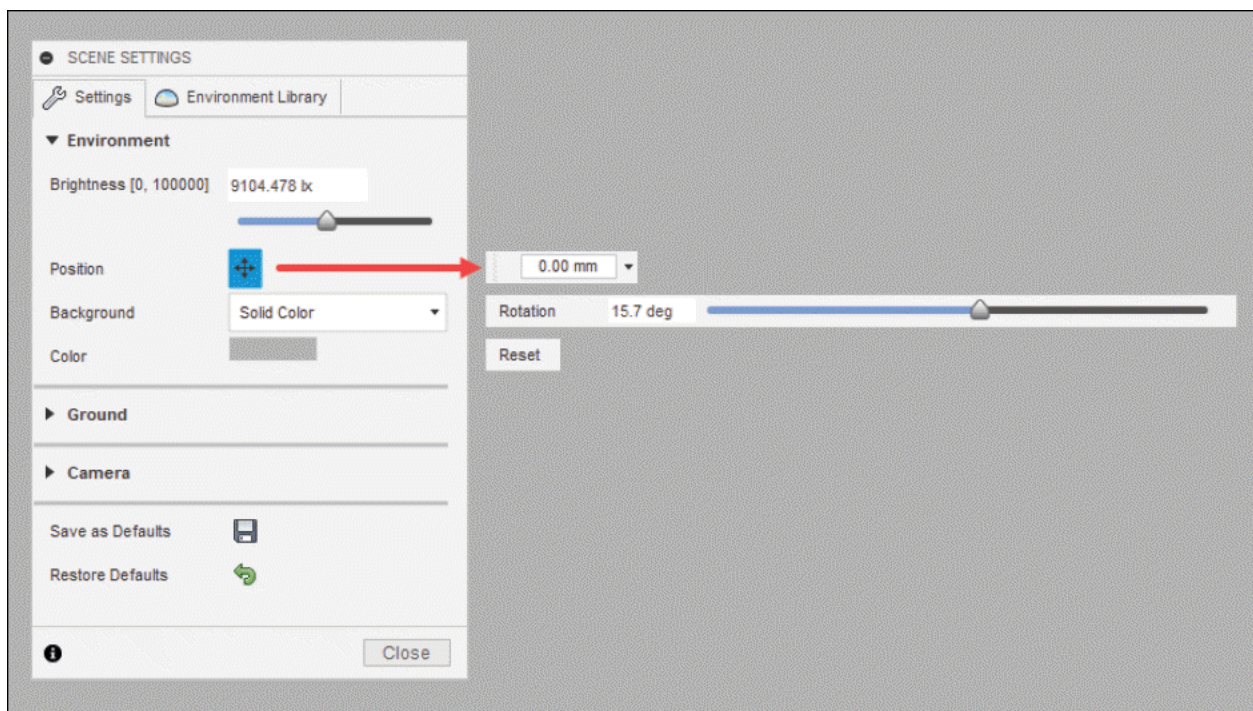


Figure-3. Tools of Position button

- Click in the floating window and specify the position of lights. You can also set the position by moving the drag handle from model.
- Click in the **Rotation** edit box and specify the value of rotation of lights. You can also adjust the value by moving the **Rotation** slider.

- If you want to reset all the changed settings, then click on the **Reset** button from rotation tools.
- Select **Environment** option from **Background** drop-down if you want to use environment image. The environments are available in **Environment Library** tab of this dialog box.
- Select **Solid Color** option from **Background** drop-down if you want to set a particular color to the background.
- To select a solid color, click on the **Color** button. A color box will be displayed; refer to [Figure-4](#).

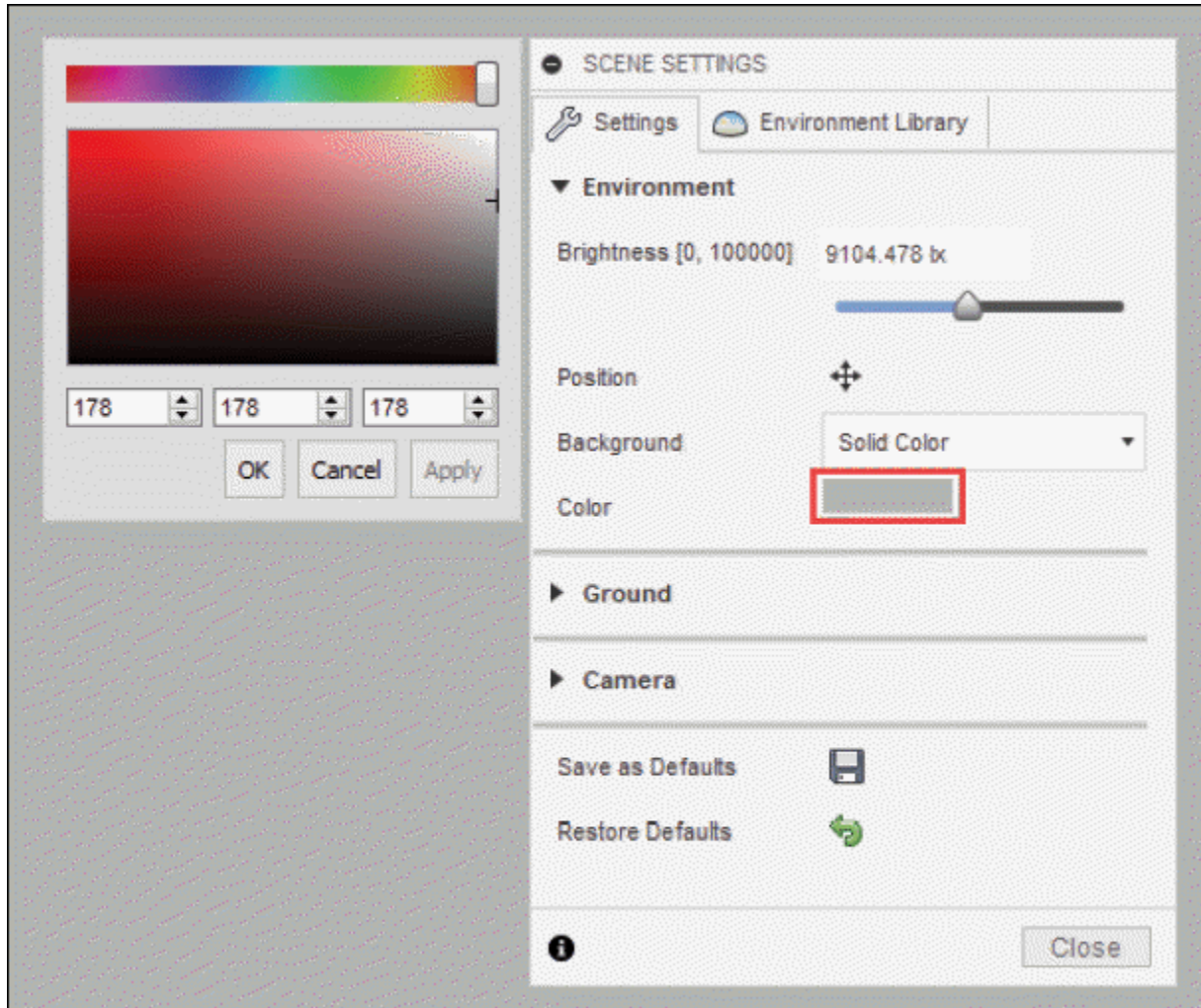


Figure-4. Color box

- Select required color from color box and click on the **Apply** button to set the selected color as background color.

Ground section

- Select the **Ground Plane** check box of **Ground** section from **Settings** tab if you want to display the shadow and reflection of model on the ground canvas.
- Select the **Flatten Ground** check box of **Ground** section from **Settings** tab if you want to enable a “textured”

ground plane where the environment image is mapped as a texture.

- Select the **Reflections** check box of **Ground** section from **Settings** tab if you want to display the reflection of model on the ground.
- Click in the **Roughness** edit box and specify the value of sharpness of reflection. You can also set the sharpness of reflection by moving the **Roughness** slider.

Camera section

- Select the **Orthographic** option of **Camera** drop-down from **Camera** section if you want to set the orthographic view of model.
- Select the **Perspective** option of **Camera** drop-down from **Camera** section if you want to set the perspective view of model.
- Select the **Perspective with Ortho Faces** option from **Camera** drop-down if you want to set the perspective view of model with ortho faces.
- Click in the **Exposure** edit box of **Camera** section from **Settings** tab and enter the value of camera exposure. You can also set the camera exposure by moving the **Exposure** slider.
- Select the **Depth of Field** check box if you want to apply depth of field effect. Depth of field is used to focus on specific area of the model. This effect is only enabled when

Ray Tracing is enabled. On selecting the **Depth of Field** check box, the dialog box will be updated; refer to [Figure-5](#).

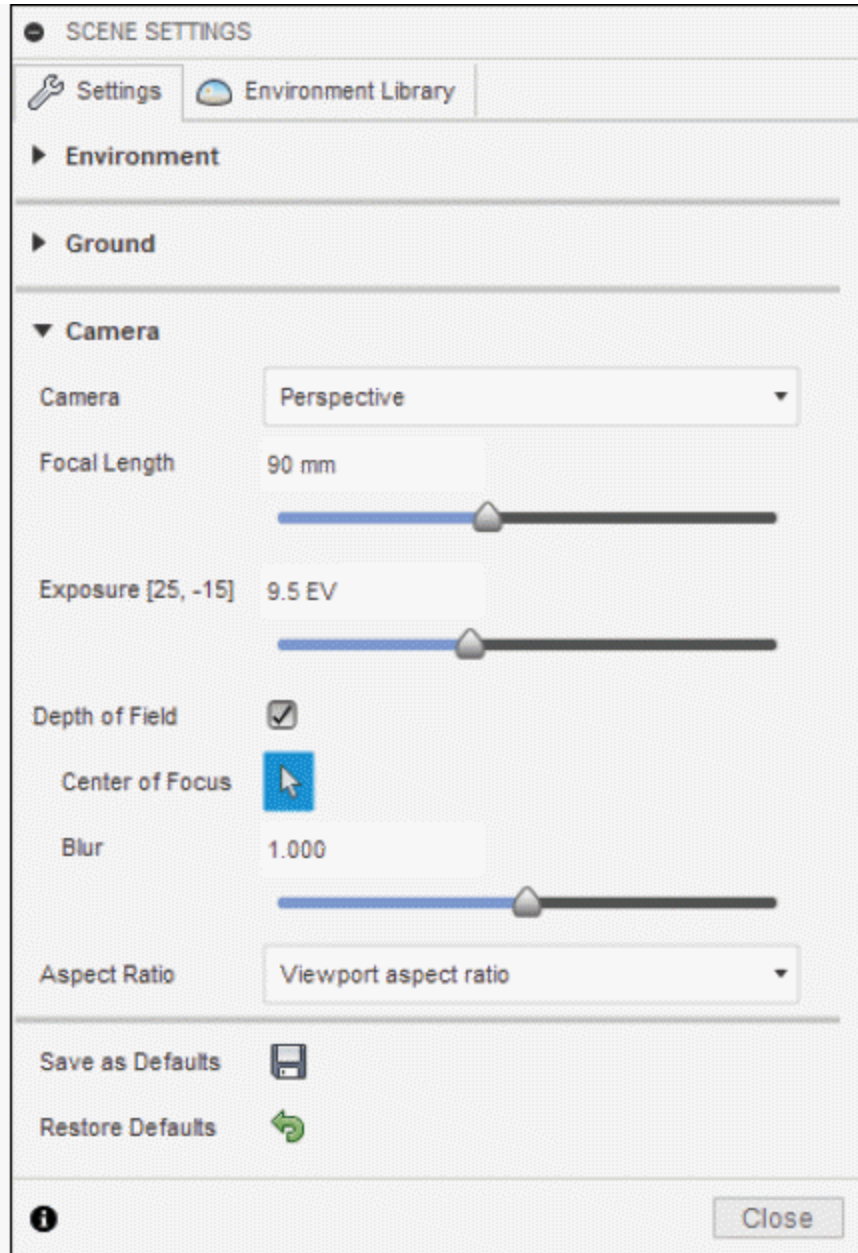


Figure-5. Updated dialog box on selecting Depth of Field check box

- The **Center of Focus** selection button is active by default. You need to select a location on model for focusing.

- Click in the **Blur** edit box and enter the value of blur the model except the selected center of focus. You can also set the value of blur by moving the **Blur** slider.
- Click on the **Aspect Ratio** drop-down and select required aspect ratio of canvas screen.
- Click on **Save as Defaults** button to save the setting as default setting.
- Click on the **Restore Defaults** button to restore the changed settings to default.

Environment Library tab

- Click on the **Environment Library** tab from the **SCENE SETTINGS** dialog box. The tab will be displayed; refer to [Figure-6](#).

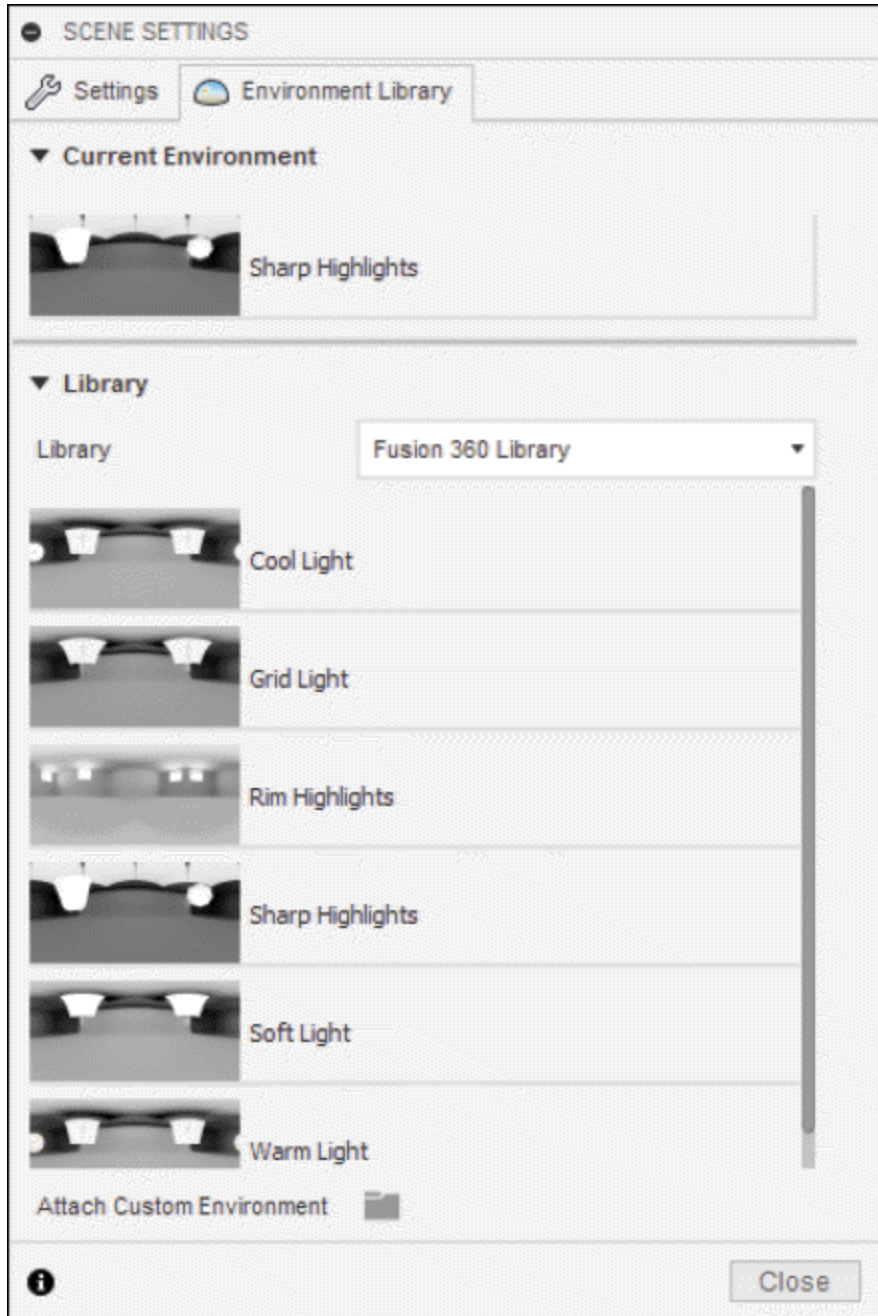


Figure-6. Environment Library tab

- The **Current Environment** section shows the applied environment on the current model.
- If you want to change the environment of the model then drag a specific environment from **Library** section and drop

it to the current model or in the **Current Environment** section. You can download the environments in library by using the **Download** button next to them in the list.

- If you want to apply custom environment for model then click on the **Attach Custom Environment** button of **Environment Library** tab from **SCENE SETTINGS** dialog box. The **Open** dialog box will be displayed; refer to [Figure-7](#).

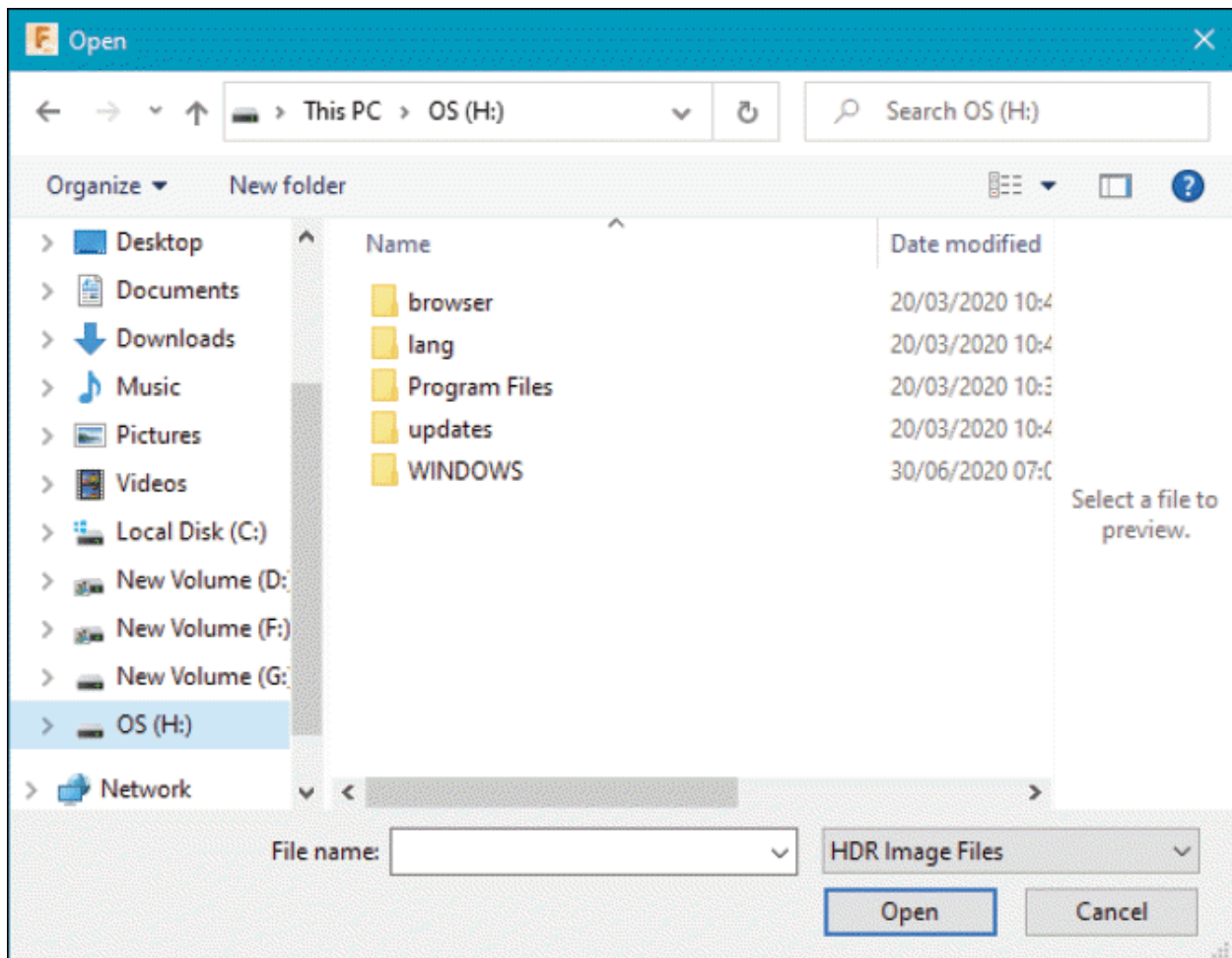


Figure-7. Open dialog box for attaching custom environment

- Select required file and click on **Open** button. The

environment will be applied to model.

- After selecting required environment from **Environment Library** tab, click on **Close** button to exit the **Scene Settings** tool.

Note- There is no **Timeline** in the **Render** workspace. The tools applied in the **RENDER** workspace will not be recorded.

Decal tool has been discussed earlier in the book.

Texture Map Controls

The **Texture Map Controls** tool is used to set the orientation of the texture applied to the face or model. The procedure to use this tool is discussed next.

- Click on the **Texture Map Controls** tool of **SETUP** panel from **Toolbar**; refer to [Figure-8](#). The **TEXTURE MAP CONTROLS** dialog box will be displayed; refer to [Figure-9](#).

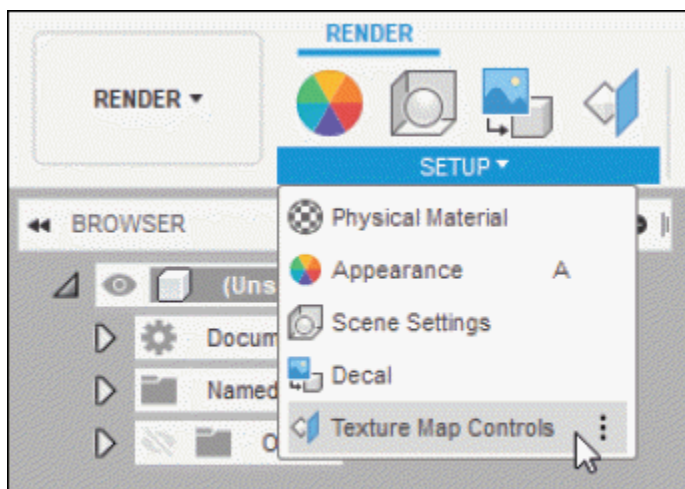


Figure-8. Texture Map Controls tool

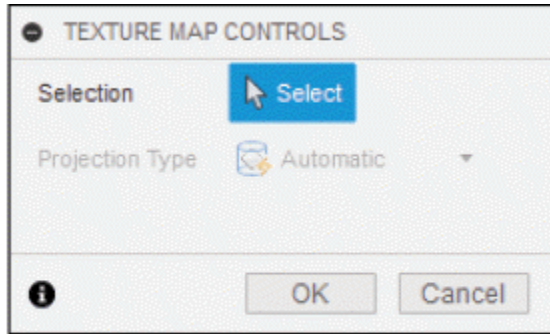


Figure-9. TEXTURE MAP CONTROLS dialog box

- The **Select** button of **Selection** option is selected by default. You need to select the texture of model. The updated **TEXTURE MAP CONTROLS** dialog box will be displayed; refer to [Figure-10](#).

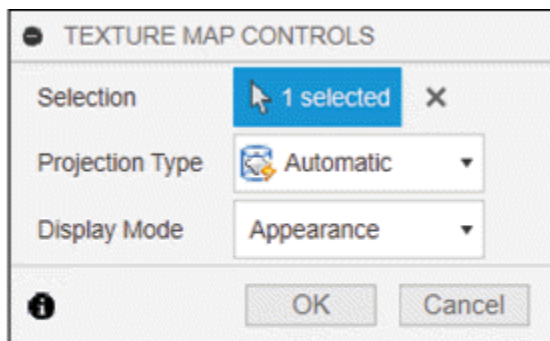


Figure-10. Updated TEXTURE MAP CONTROLS dialog box

Automatic

- Select the **Automatic** option from **Projection Type** drop-down if you want to adjust the texture on the model using model topology.

Planar

- Select the **Planar** option from **Projection Type** drop-down if you want to map the image or texture on the model as projection.

- On selecting the **Planar** option, you need to select the axis for adjusting the texture. On selecting the texture, the dialog box will be updated; refer to [Figure-11](#).

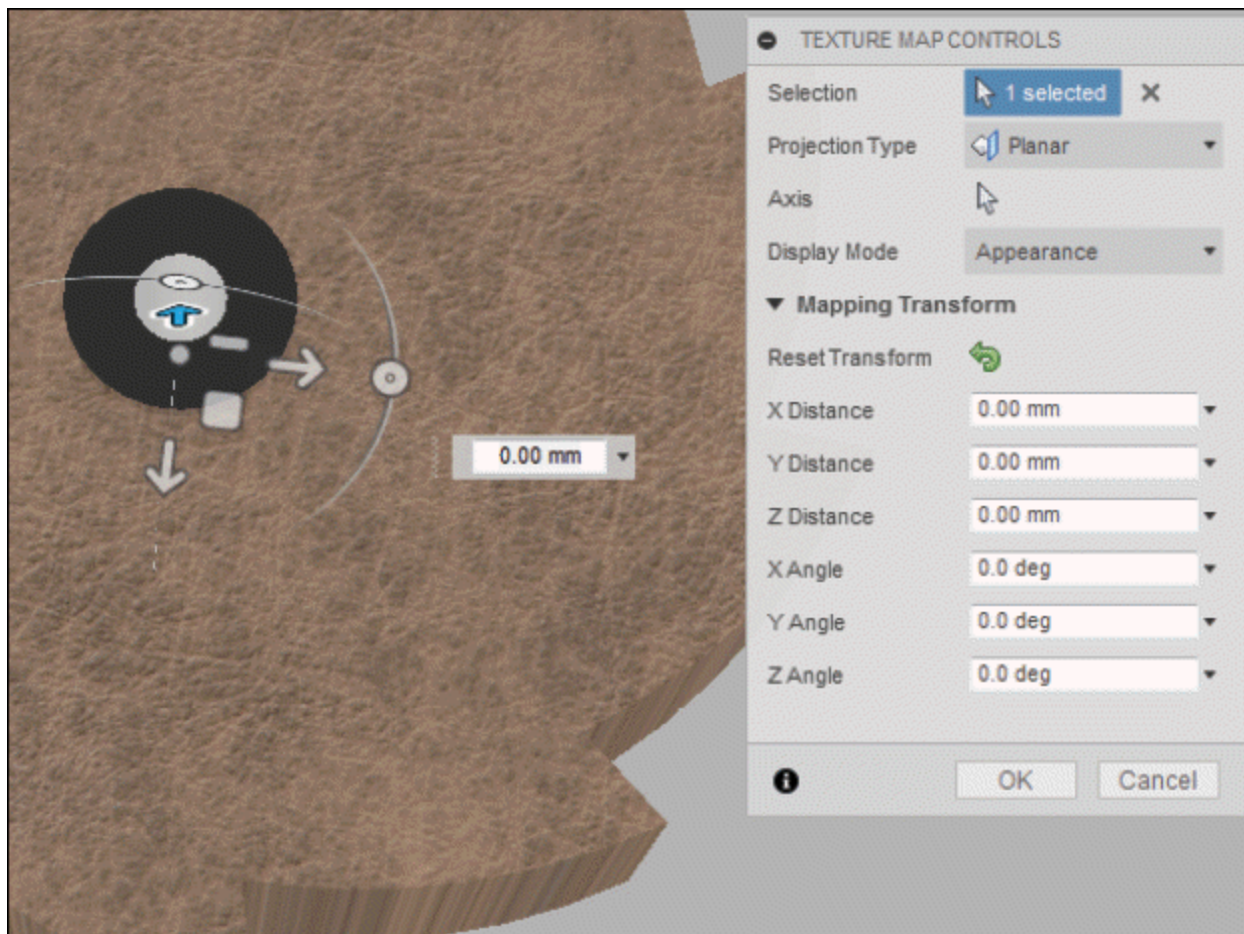


Figure-11. Updated dialog box after selecting axis

- Adjust the image or texture by using the manipulator or by specifying the values in related edit boxes in **TEXTURE MAP CONTROLS** dialog box.
- After adjusting the texture, click on the **OK** button from **TEXTURE MAP CONTROLS** dialog box.

Box

- Click on the **Box** option from **Projection Type** drop-down if you want to map the image on model into box like objects.
- On selecting the **Box** option, the manipulator and updated dialog box will be displayed; refer to [Figure-12](#).

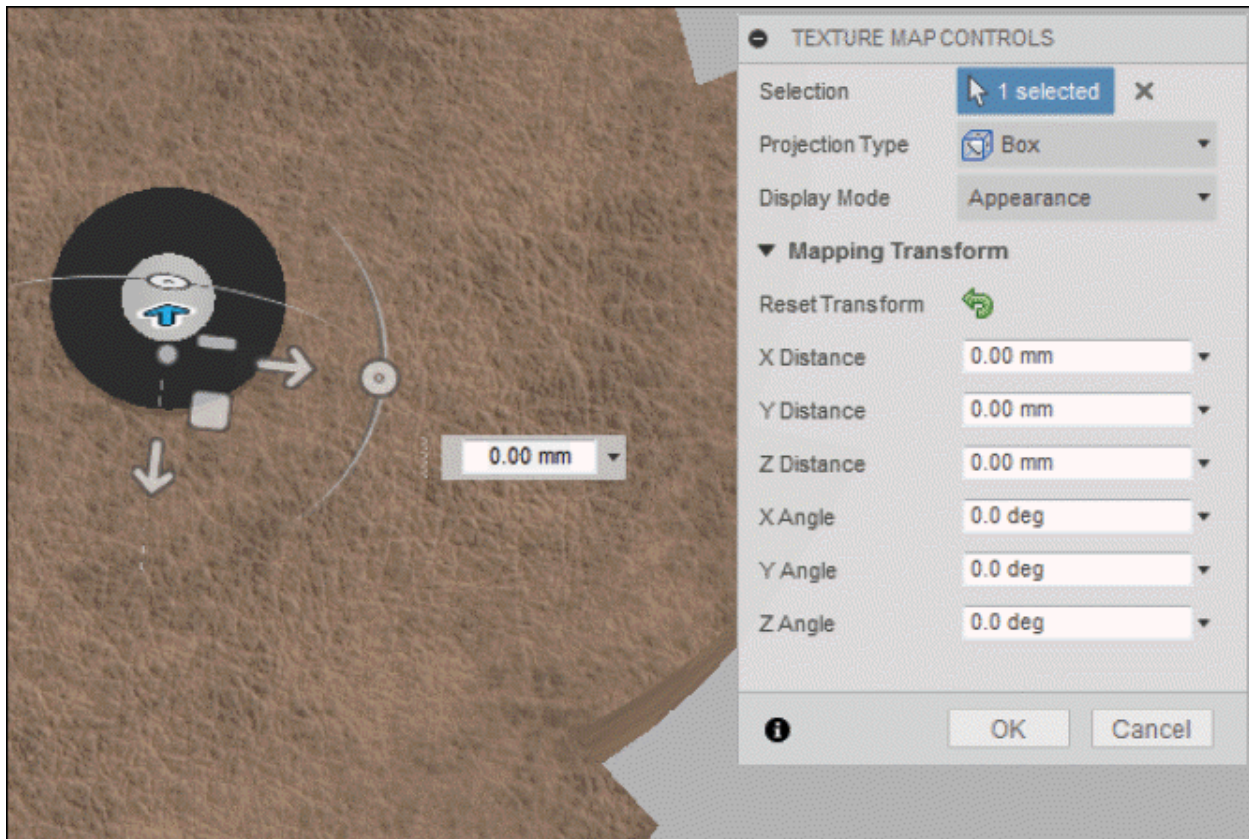


Figure-12. Dialog box on selecting Box-option

- Click in the specific edit box and enter desired value. You can also set the value by moving the arrow of manipulator.
- If you want to reset the values of **Mapping Transform** or manipulator then click on the **Reset Transform** button.

Spherical

- Select the **Spherical** option from **Projection Type** drop-

down if you want to map texture into a spherical object.

- On selecting the **Spherical** option, the manipulator along with updated dialog box will be displayed.
- The options were discussed in last topic.

Cylindrical

- Select the **Cylindrical** option from **Projection Type** drop-down if you want to map texture into a cylindrical object.
- On selecting the **Cylindrical** option, the manipulator along with updated dialog box will be displayed.
- The options were discussed in last topic. After specifying the parameters, click on the **OK** button from **TEXTURE MAP CONTROLS** dialog box to exit the tool.

In-canvas Render

The **In-canvas Render** button is used to start and stop the In-canvas rendering process. The procedure to use this tool is discussed next.

- Click on the **In-canvas Render** button from **IN-CANVAS RENDER** drop-down; refer to [Figure-13](#). The render quality slider will be displayed to display at the bottom right area of canvas screen; refer to [Figure-14](#).

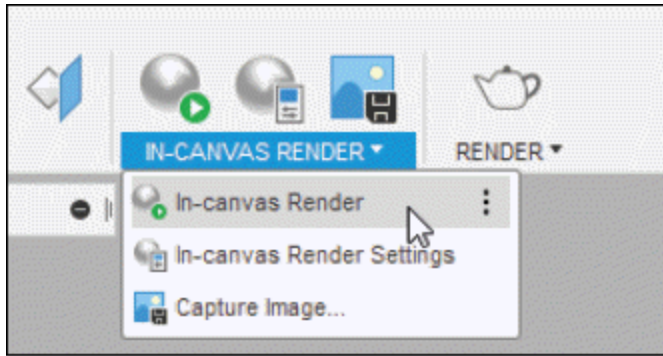


Figure-13. In-canvas Render tool

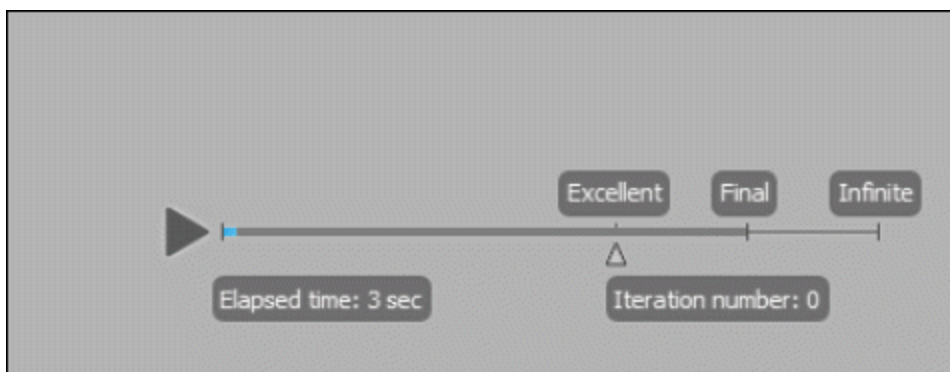


Figure-14. Render quality slider

- The current scene of the model will be changed into the photo-realistic 2D image.
- Clicking again on the **In-canvas Render Stop** button will stop the rendering process.

In-Canvas Render Setting

The **In-canvas Render Setting** tool is used to change the setting of rendering process. The procedure to use this tool is discussed next.

- Click on the **In-Canvas Render Settings** tool from **IN-CANVAS RENDER** drop-down; refer to [Figure-15](#). The **IN-CANVAS RENDER SETTINGS** dialog box will be

displayed; refer to [Figure-16](#).

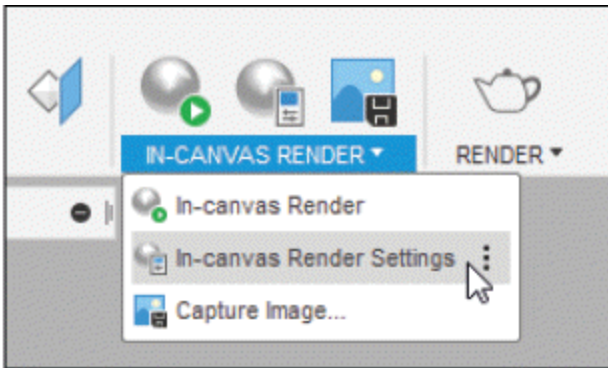


Figure-15. In-canvas Render Settings tool

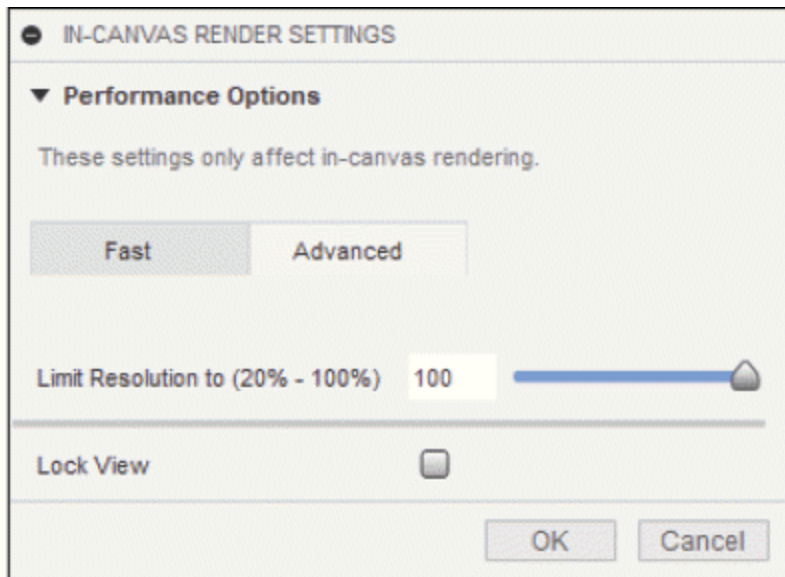


Figure-16. IN-CANVAS RENDER SETTINGS dialog box

- Set the quality of rendering by adjusting the **Limit Resolution to** slider. The higher quality rendering takes more time to calculate rendering process and lower quality rendering takes less time to calculate the rendering process.
- Select the **Lock View** check box to lock the current view of model for rendering. If this check box is selected then you

can not reorient the model during rendering process.

- After specifying the parameters, click on the **OK** button from **IN-CANVAS RENDER SETTINGS** dialog box.

Capture Image

The **Capture Image** tool is used to capture the current canvas as an image. The procedure to use this tool is discussed next.

- Click on the **Capture Image** tool of **IN-CANVAS RENDER** drop-down in **Toolbar**; refer to [Figure-17](#). The **Image Options** dialog box will be displayed; refer to [Figure-18](#).

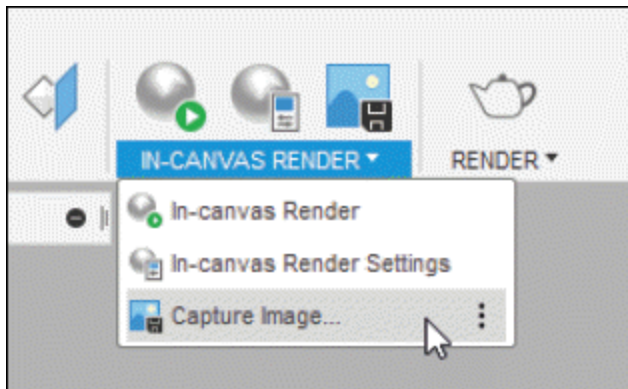


Figure-17. Capture Image tool

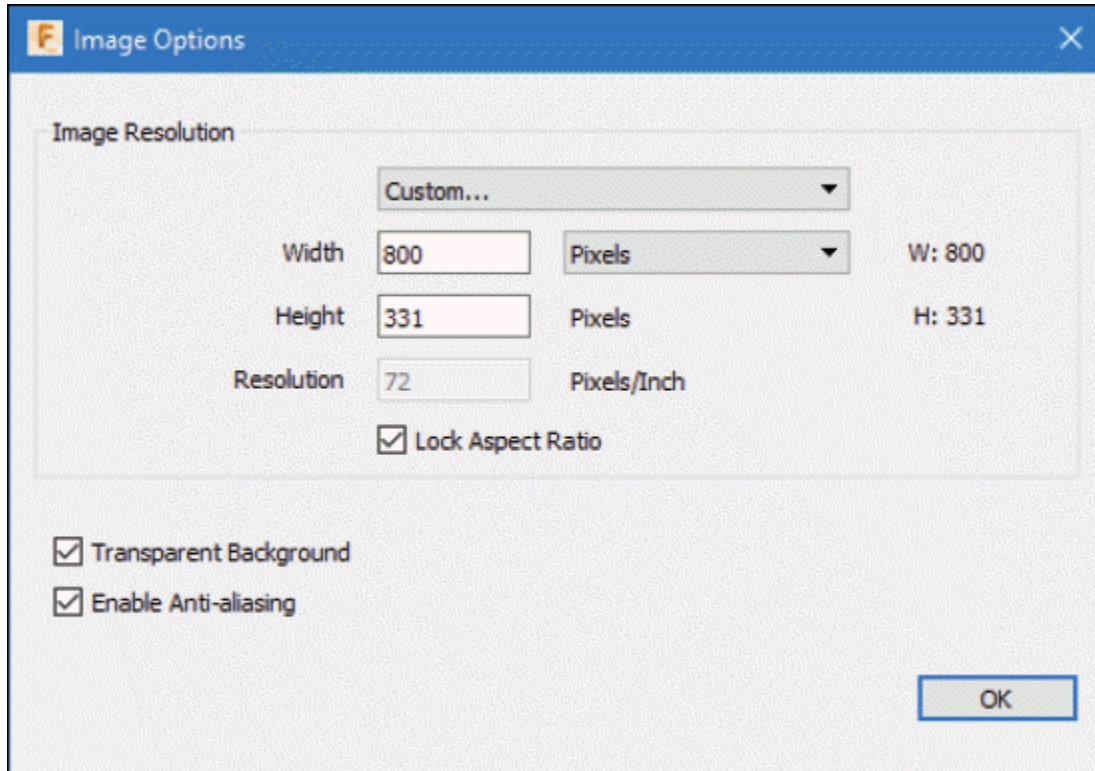


Figure-18. Image Options dialog box

- Click on the **Image Resolution** drop-down and select required resolution of image.
- Select the **Transparent Background** check box if you want to set the transparent background of model.
- Select the **Enable Anti-aliasing** check box if you want to activate the anti-aliasing property.
- After specifying the parameters, click on the **OK** button from **Image Options** dialog box. The **Save As** dialog box will be displayed; refer to [Figure-19](#).

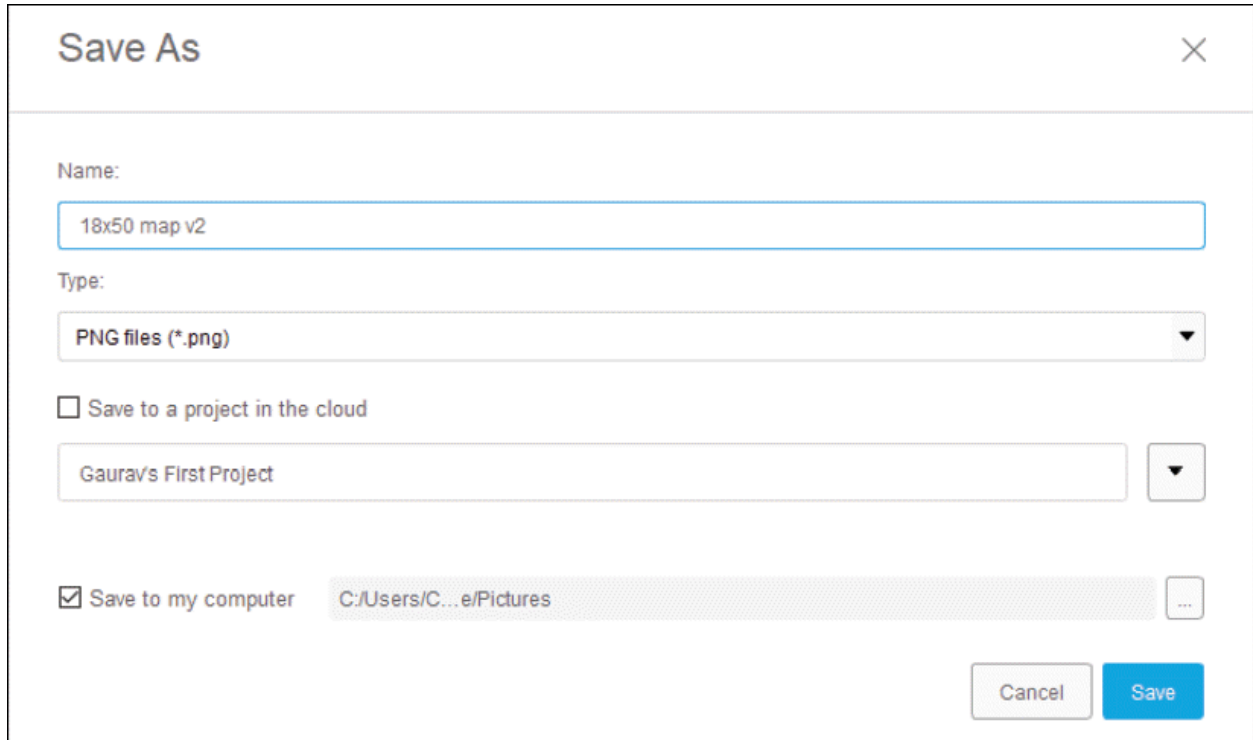


Figure-19. Save As dialog box

- Click in the **Name** edit box of **Save As** dialog box and enter the specific name for image.
- Click on the **Type** drop-down and select required format of image file.
- Click on the **Location** drop-down button and select the location of saving image.
- Select the **Save to my computer** check box if you want to save the file to computer.
- Select the **Save to a project in the cloud** check box to save the image file on cloud.
- After specifying the parameters for image, click on the **Save** button from **Save As** dialog box. The image will be

saved to the specified location.

Render

The **Render** tool is used to create a high quality rendered image of the current scene. The procedure to use this tool is discussed next.

- Click on the **Render** tool from **Toolbar**; refer to [Figure-20](#). The **RENDER SETTINGS** dialog box will be displayed; refer to [Figure-21](#).

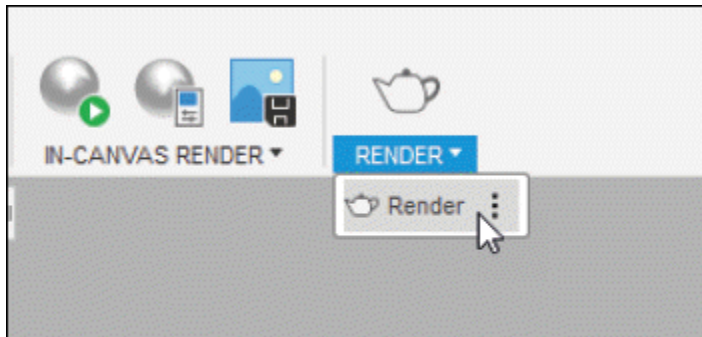


Figure-20. Render tool

RENDER SETTINGS

WEB
 MOBILE
 PRINT
 VIDEO
 CUSTOM

Image Size: Custom
 Aspect Ratio: Custom Aspect Ratio
 Exposure: Native

Width: 1366 px
 Height: 566 px

RENDER WITH

Cloud Renderer
 Local Renderer

RENDER QUALITY

Standard
 Final

[CLOUD CREDITS](#) | [FAQ](#) | [*EDUCATION TERMS OF USE](#)

1 Required	16 Max per request	Unlimited* Available
----------------------	------------------------------	--------------------------------

RENDER QUEUE TIME: < 20 minutes

Render

Figure-21. RENDER SETTINGS dialog box

WEB

- Click on the **WEB** tab of **RENDER SETTINGS** dialog box. Various options under **WEB** tab will be displayed; refer to [Figure-22](#).

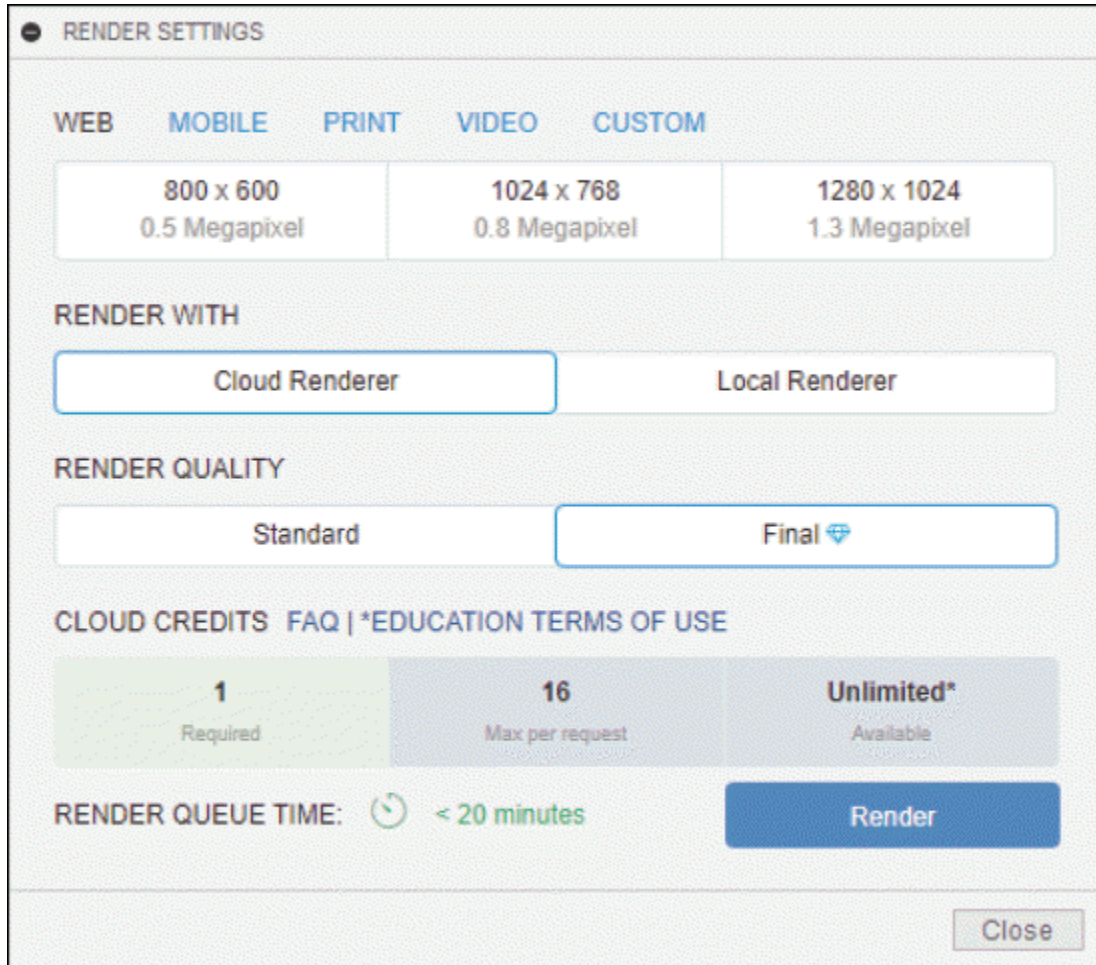


Figure-22. WEB tab

- Three highlighted options are the size of image in which your rendered file will be saved. Click on required size.
- Click on the **Cloud Renderer** option from **RENDER WITH** section, if you want to render the file from Autodesk cloud. On selecting the **Cloud Renderer** button, the dialog box will be updated; refer to [Figure-23](#).

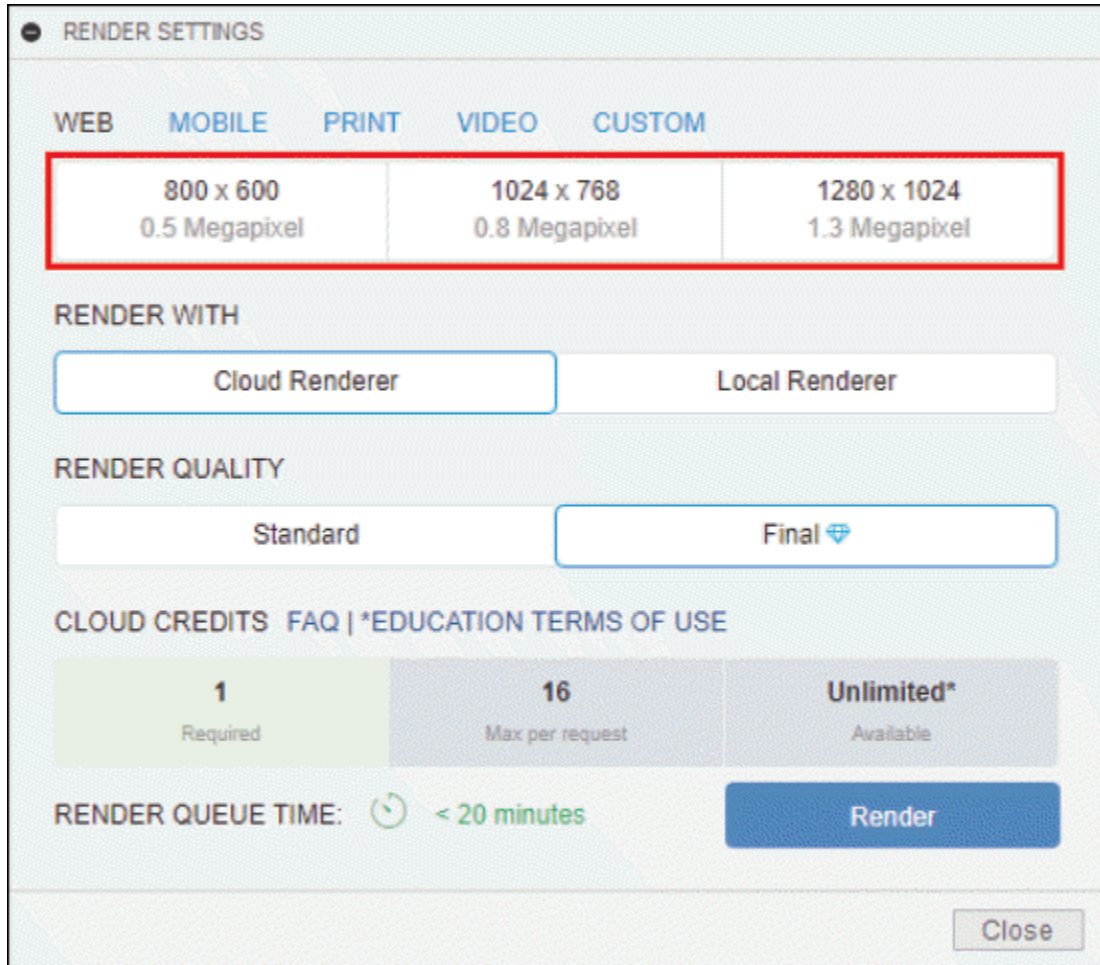


Figure-23. Updated WEB tab

- With **Local Renderer** option selected, click on the **Advanced settings** toggle button from **WEB** tab to manually set the render quality value. The **RENDER QUALITY** slider will be displayed; refer to [Figure-24](#).

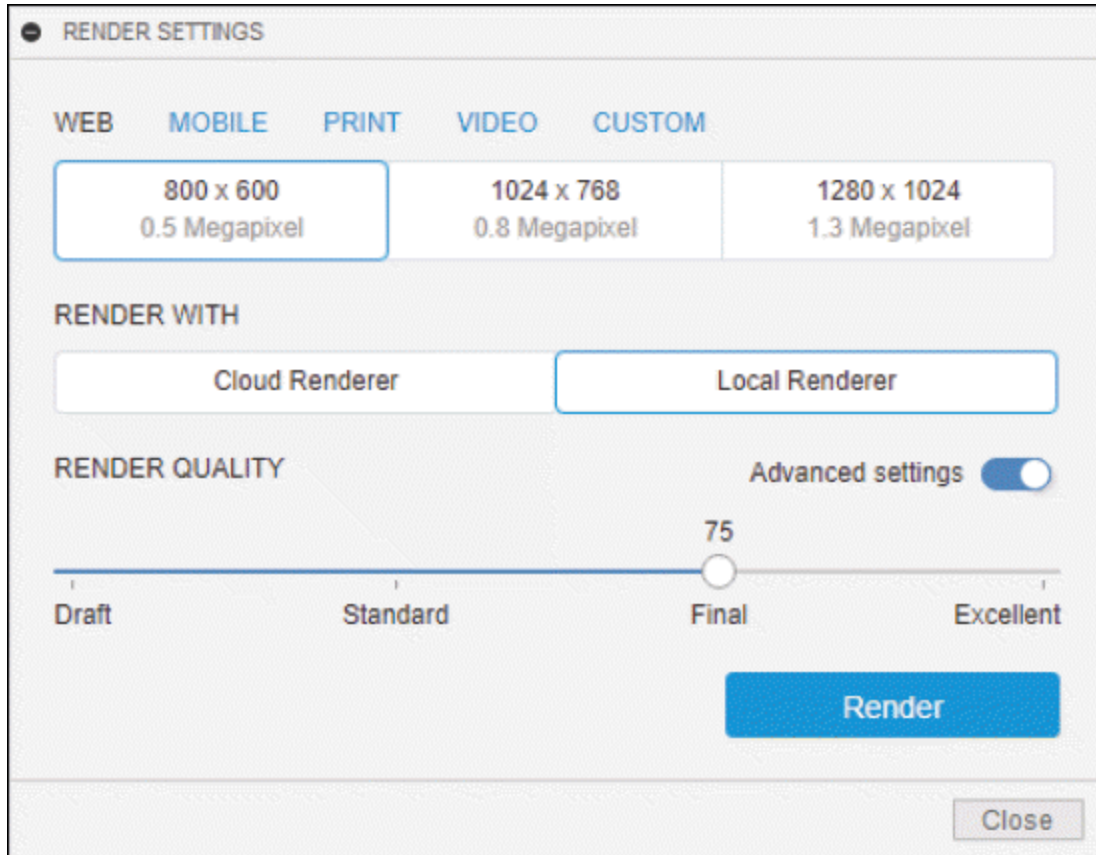


Figure-24. RENDER QUALITY slider in WEB tab

- Set the value of quality from slider as required. If you want to automatically set the quality of render then again click on the **Advanced settings** toggle button. The **Standard(50)** and **Final(75)** option will be displayed.
- Click on required option from **RENDER QUALITY** section and click on the **Render** button from **WEB** tab. The rendering process will start.
- The completed rendered file will be saved in **RENDERING GALLERY**; refer to [Figure-25](#).

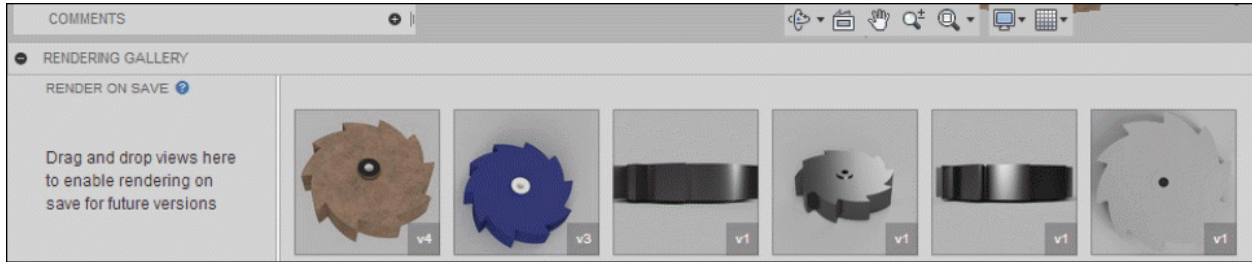


Figure-25. RENDERING GALLERY

Render Gallery

- To view the enlarged size of rendered image, click on that particular image. The image will be displayed in **RENDERING GALLERY** dialog box; refer to [Figure-26](#).

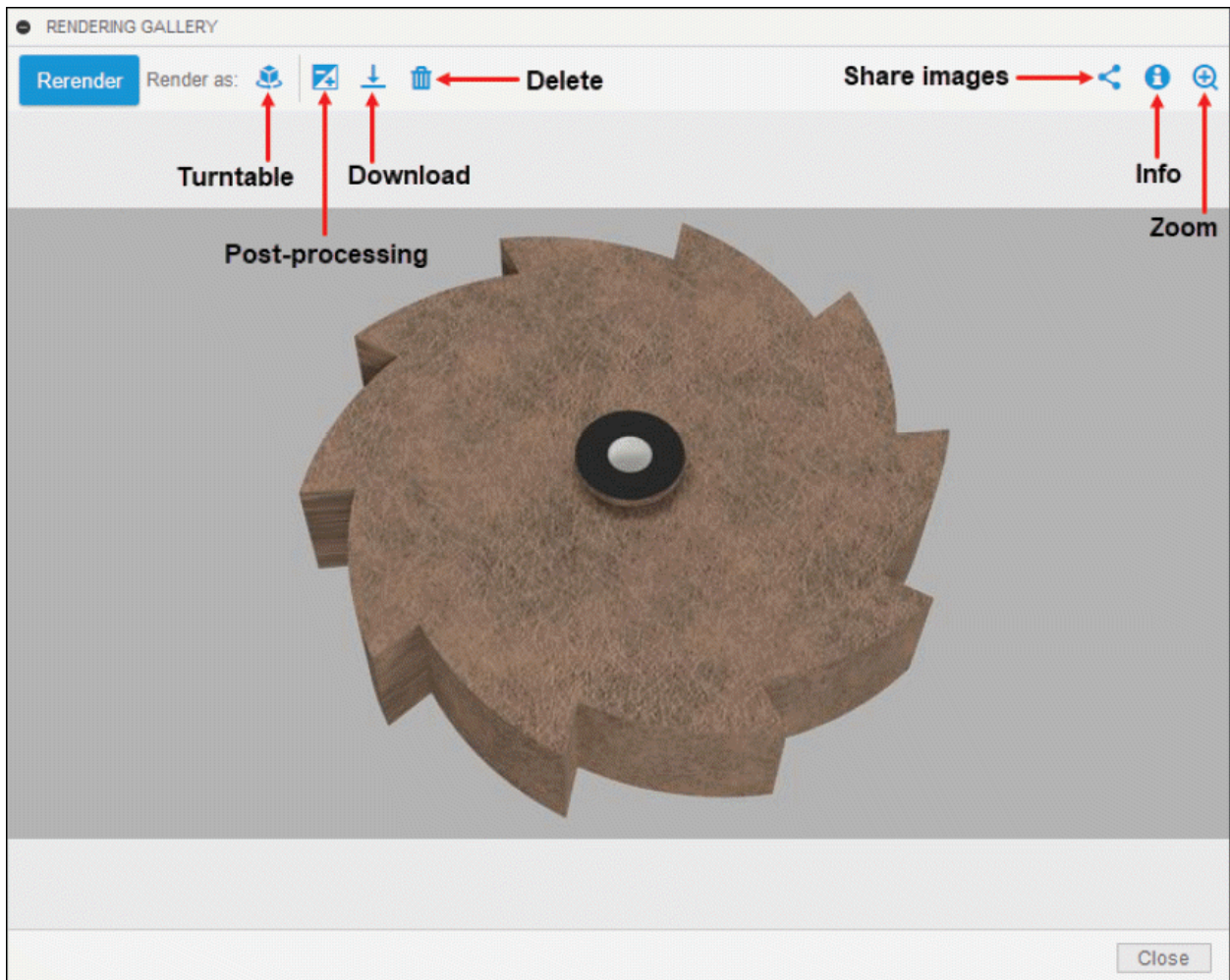


Figure-26. RENDERING GALLERY dialog box

- If you want to create a video of the rotating model then click on the **Turntable** button from **RENDERING GALLERY** dialog box. The **Render Settings - Turntable** dialog box will be displayed; refer to [Figure-27](#).

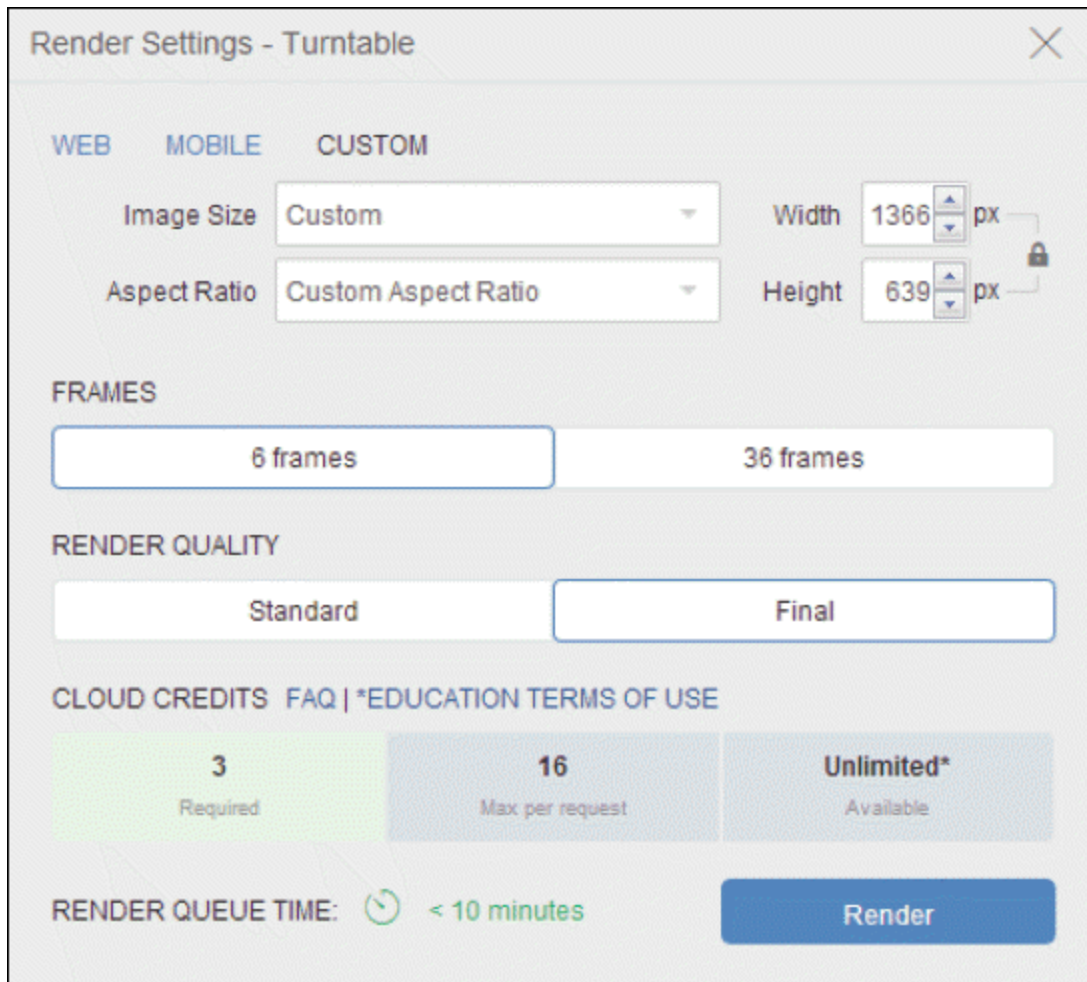


Figure-27. Render Settings-Turtable dialog box

- Specify desired parameters and click on the **Render** button. The file will be re-created and displayed in **RENDERING GALLERY**.

Post Processing

- If you want to adjust the exposure of model then click on

the **Post-processing** button. The **Post-processing** dialog box will be displayed; refer to [Figure-28](#).

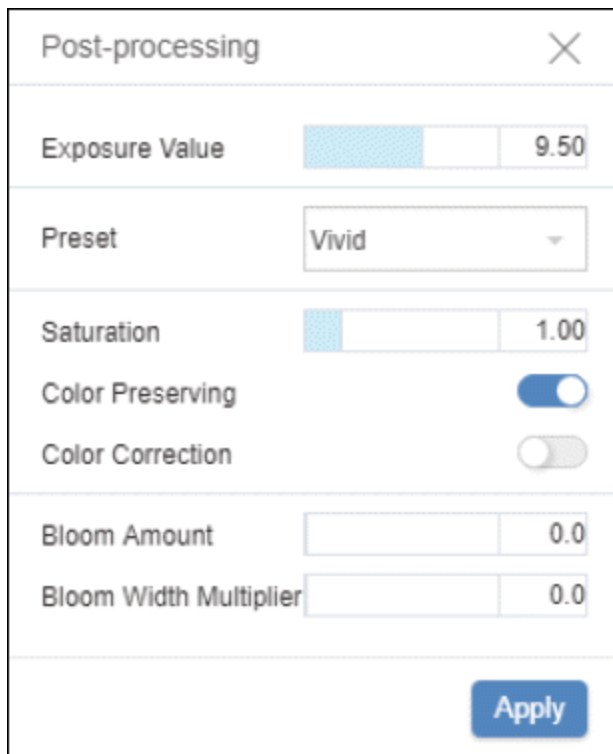


Figure-28. Post processing dialog box

- The value of exposure can be adjusted by using the slider for **Exposure Value**.
- Select desired option from the **Preset** drop-down. The options in the dialog box will be automatically adjusted presentation settings for rendering.
- Similarly, you can set the other parameters in the dialog box. After setting the parameters, click on the **Apply** button.

Downloading Render Image File

- If you want to download the current file then click on the

Download button from **RENDERING GALLERY** dialog box. The drop-down will be displayed as shown in [Figure-29](#).

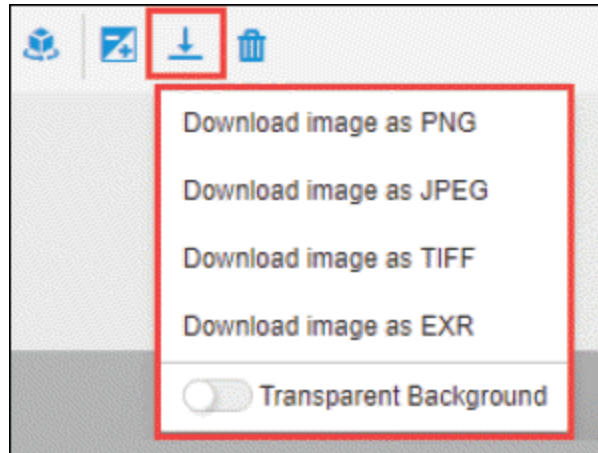


Figure-29. Download drop-down

- Select the **Transparent Background** toggle button if you want to make background of model transparent. On selecting this toggle button, you will be able to save the image file in PNG or TIFF format only.
- Select desired **Download image as...** button. The **Save File** dialog box will be displayed; refer to [Figure-30](#).

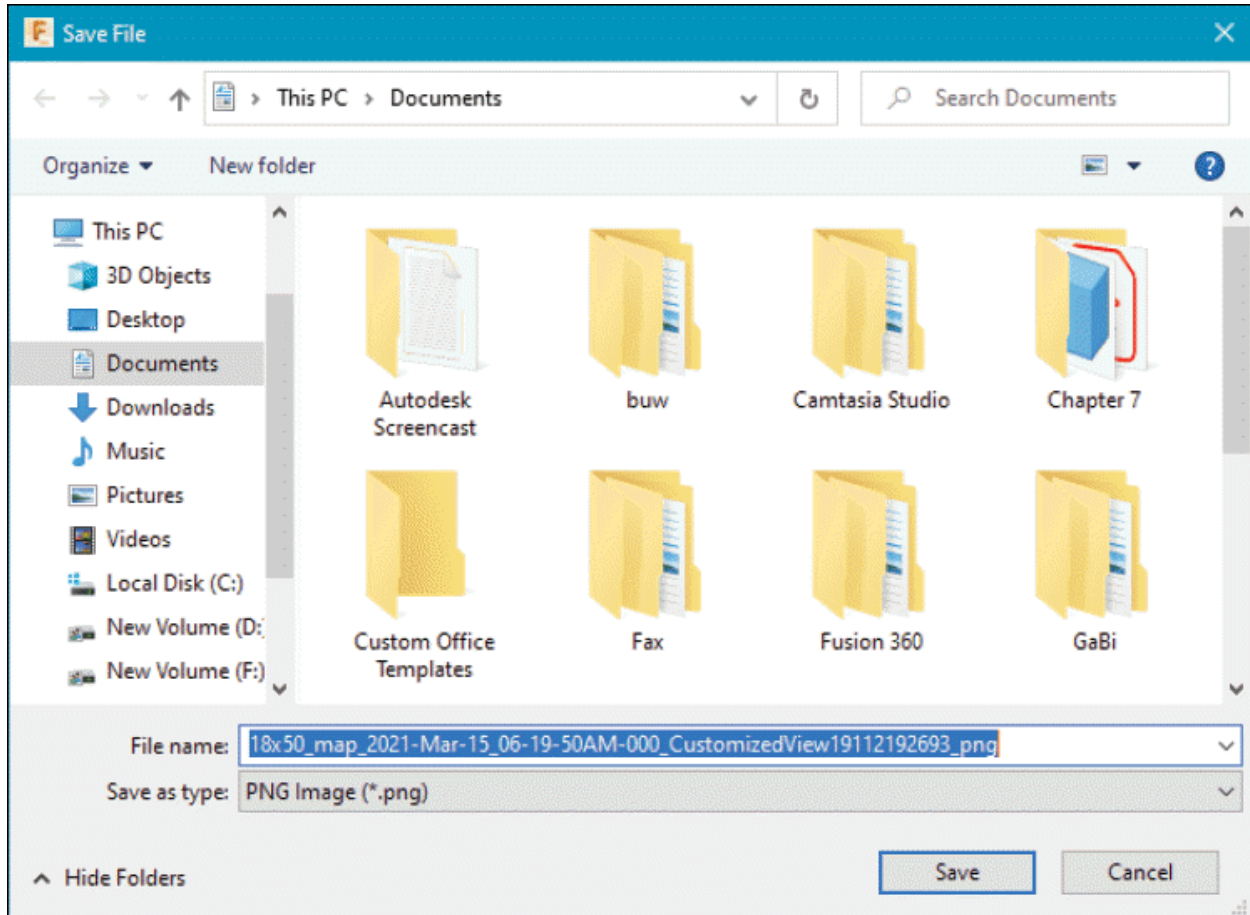


Figure-30. Save File dialog box

- Select the location for saving file and click on the **Save** button from the dialog box to save the file.
- If you want to delete the current file from **RENDERING GALLERY** then click on the **Delete** button from **RENDERING GALLERY** dialog box. The **Delete Image** dialog box will be displayed; refer to [Figure-31](#).

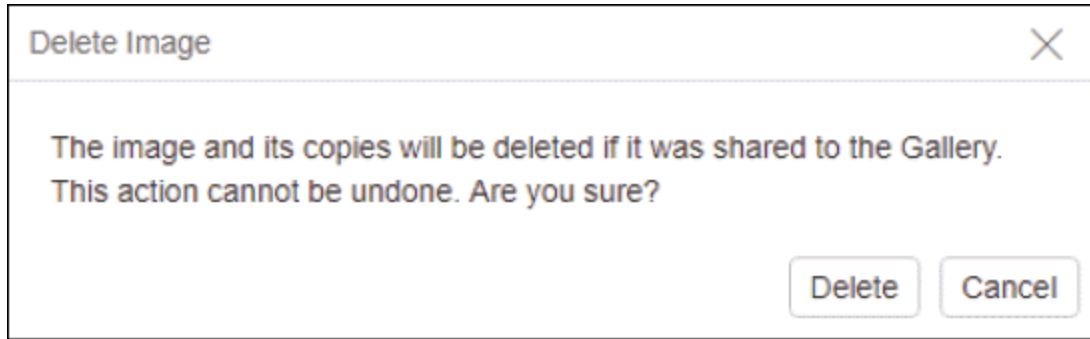


Figure-31. Delete Image dialog box

- If you are sure about deleting the particular file then click on the **Delete** button from **Delete Image** dialog box otherwise click on **Cancel** button.
- If you want to share the image to Fusion 360 Gallery or A360 then click on the **Share images** button at the top right in **RENDERING GALLERY** dialog box and select the respective option.
- If you want to see the information of the current file then click on the **Info** button from **RENDERING GALLERY** dialog box.
- If you want to zoom-in/zoom-out the rendered image then click on the **Zoom** button from the dialog box.
- After rendering the file, click on the **Close** button from **RENDERING GALLERY** dialog box to exit the rendering process.

ANIMATION

The **Animation** workspace is used to animate the joints of assembly. This workspace is also used to create exploded

view of assembly. You can create the exploded view automatically as well as manually in this workspace. To activate the workspace, click on the **ANIMATION** option from the **Workspace** drop-down. The tools to perform animation will be displayed.

New Storyboard

A storyboard is automatically generated for each model when you switch to **ANIMATION** workspace. In the process of creating animation, additional storyboard can be added at any time as well. Each new storyboard has a default name “Storyboardx” where x is a number. You can edit this name by double-clicking on it at the bottom in application window. Storyboard acts as base for animation. To create different animations of same assembly, you can create different storyboards. The procedure to create new storyboard is discussed next.

- Click on the **New Storyboard** tool from **STORYBOARD** drop-down of **Toolbar** in **ANIMATION** workspace; refer to [Figure-32](#). The **NEW STORYBOARD** dialog box will be displayed; refer to [Figure-33](#).

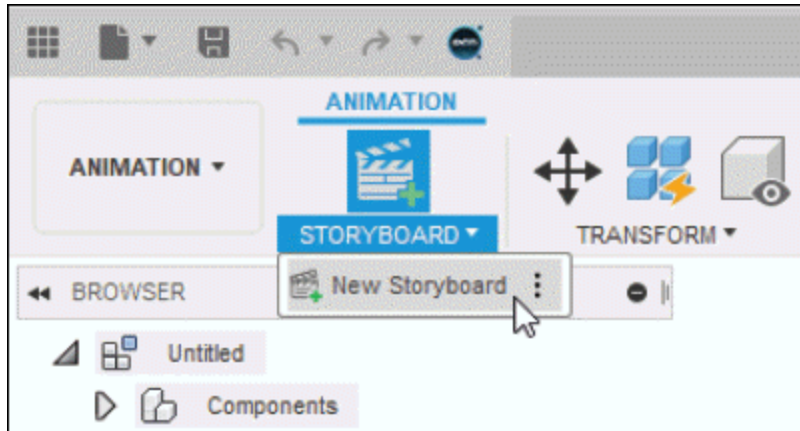


Figure-32. New Storyboard tool

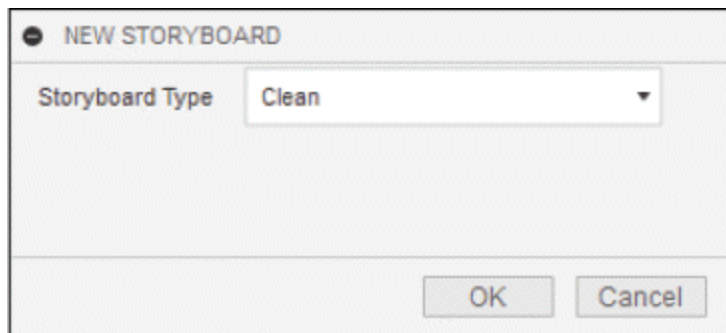


Figure-33. NEW STORYBOARD dialog box

- Select the **Clean** option from **Storyboard Type** drop-down if you want to delete all previous actions of animation and want to create a new storyboard.
- Select the **Start from end of previous** option from **Storyboard Type** drop-down if you want to use position of components at the end of previous animation as the starting position for the new animation.
- After setting desired options, click on the **OK** button from the dialog box. The created storyboard will be added at the bottom of **ANIMATION TIMELINE**; refer to [Figure-34](#).

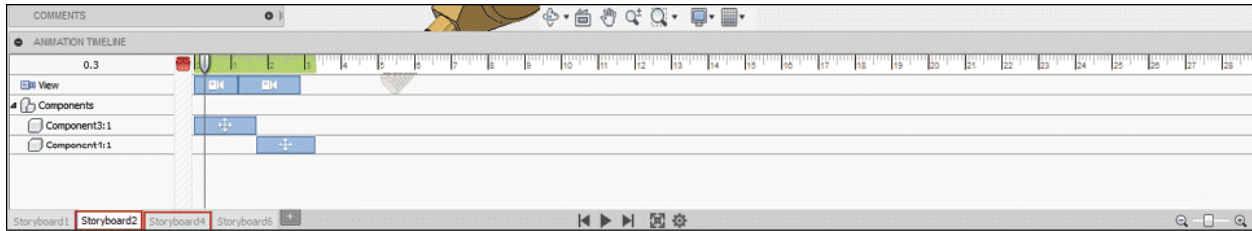


Figure-34. ANIMATION TIMELINE

Transform Components

The **Transform Components** tool is used to move and rotate the component. Note that the movement and rotation performed in this workspace will be recorded in animation. The procedure to use this tool is discussed next.

- Click on the **Transform Components** tool of **TRANSFORM** drop-down; refer to [Figure-35](#). The **TRANSFORM COMPONENTS** dialog box will be displayed; refer to [Figure-36](#).

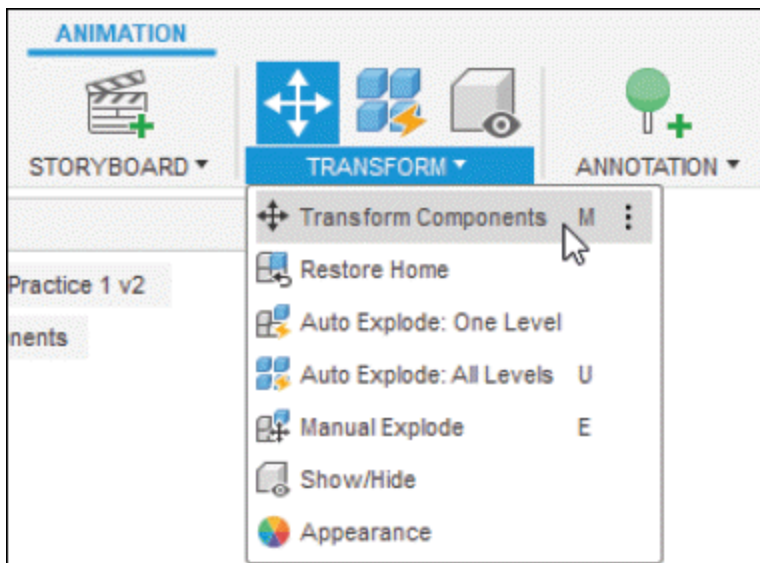


Figure-35. Transform Components tool

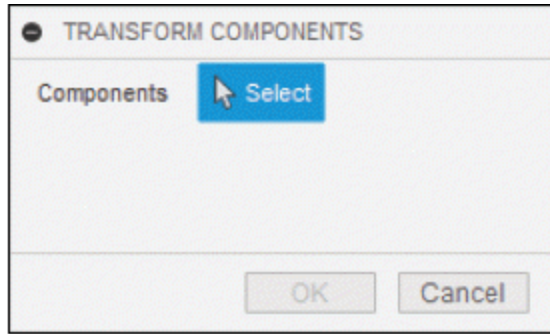


Figure-36. TRANSFORM COMPONENTS dialog box

- The **Select** button of **Components** section is active by default. Click on the component that you want to move or rotate. On selection of component, a manipulator will be displayed on the selected component along with updated **TRANSFORM COMPONENTS** dialog box; refer to [Figure-37](#).

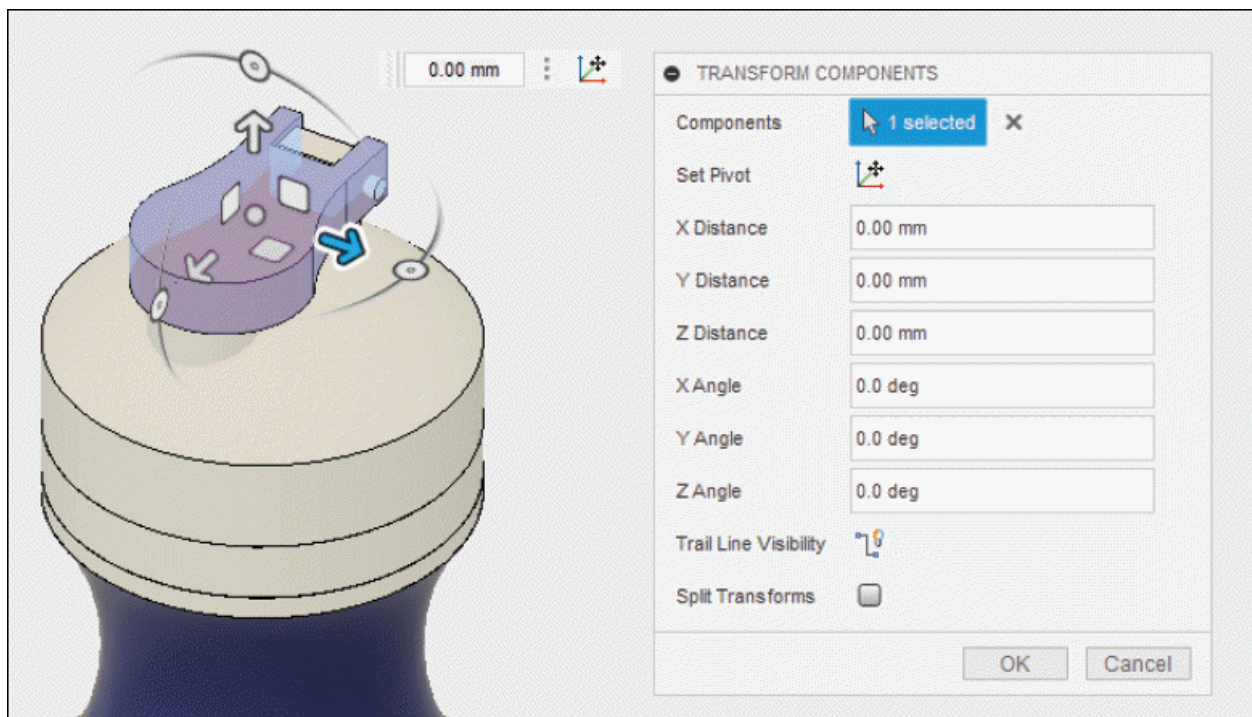


Figure-37. Manipulator on selected component

- The actions like moving or rotating the components are

registered as animation in storyboard. To register the movement, drag the playhead in **ANIMATION TIMELINE** area to time value upto which the selected transformation operation will be completed. For example, if you want to move the component shown in figure by **60** mm in **2** seconds then first move the playhead of timeline to 2 seconds and then move the component by **60** mm; refer to [Figure-38](#).

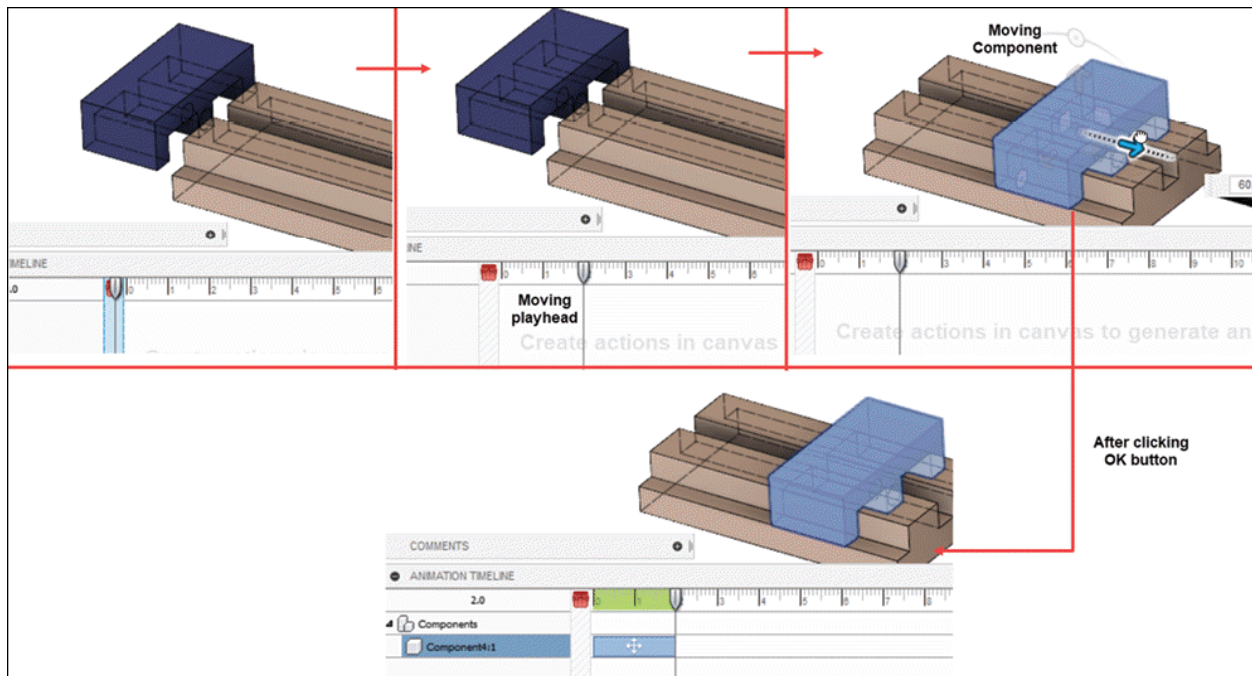


Figure-38. Registering movement of component in timeline

- You can specify desired values for movement in the **TRANSFORM COMPONENTS** dialog box or you can set the parameters by moving the manipulator.
- After specifying the parameters, click on the **OK** button from **TRANSFORM COMPONENTS** dialog box to add the action in animation timeline.

- To play the animation, you need to click on the **Play** button from **ANIMATION TIMELINE**.

Restore Home

The **Restore Home** tool is used to restore the model position as it is at the end of animation.

- Click on the **Restore Home** tool from **TRANSFORM** drop-down; refer to [Figure-39](#). The position of model will be restored.

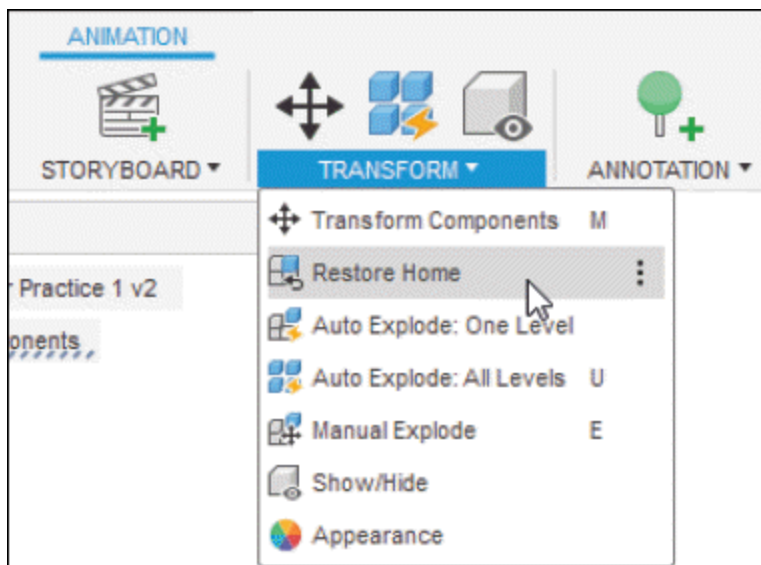


Figure-39. Restore Home tool

Auto Explode: One Level

The **Auto Explode: One Level** tool is used to automatically separate all selected components down to one level in hierarchy. The procedure to use this tool is discussed next.

- Select the parts of assembly to be exploded (separated from each other) while holding the **CTRL** key. You can also select all components using window selection; refer to

[Figure-40.](#)

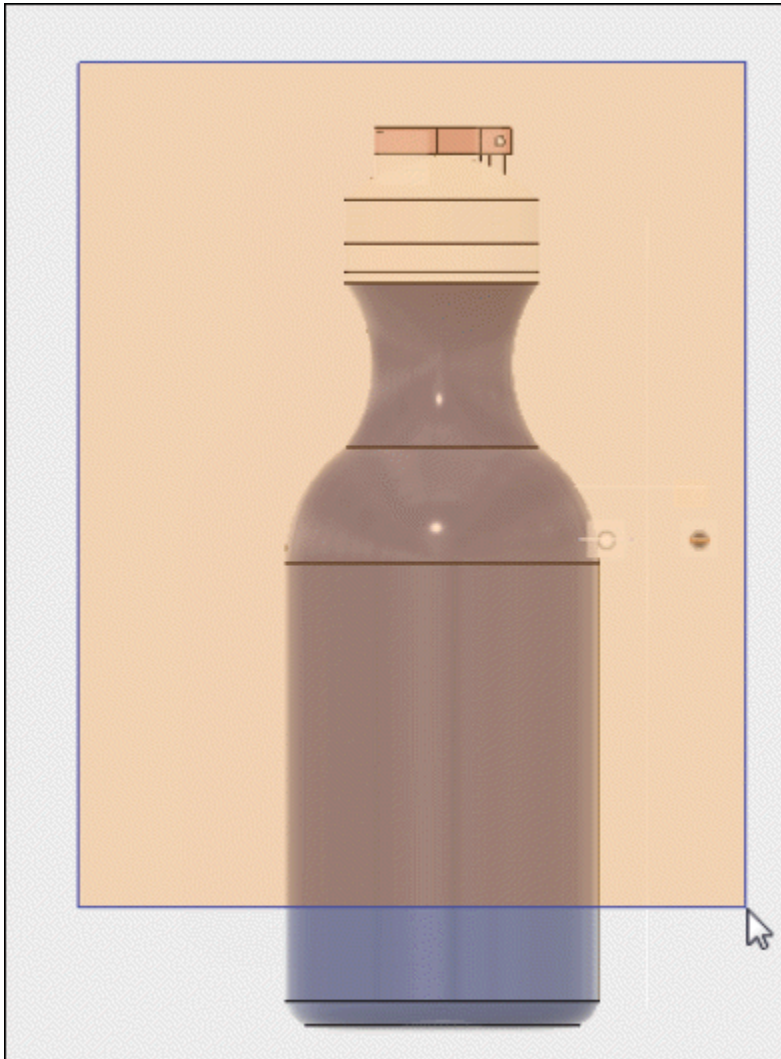


Figure-40. Window selection of component

- Click on the **Auto Explode: One Level** tool from **TRANSFORM** drop-down; refer to [Figure-41](#). The components will be separated on the canvas screen; refer to [Figure-42](#).

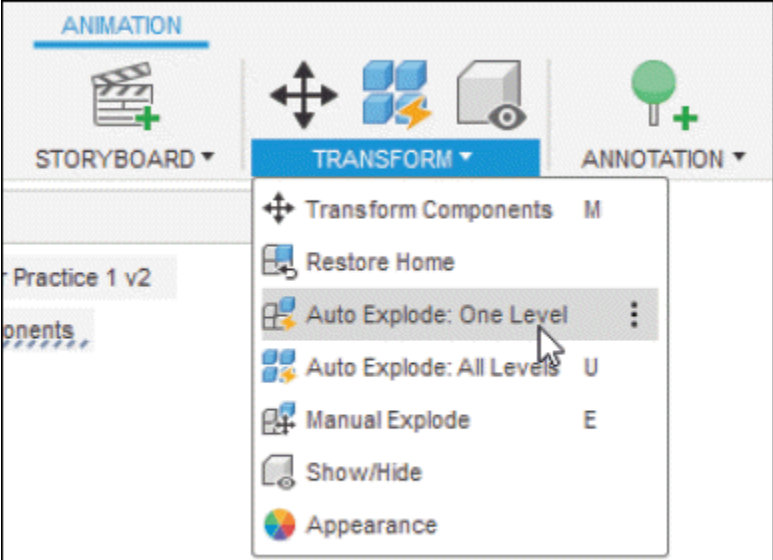


Figure-41. Auto Explode One Level tool

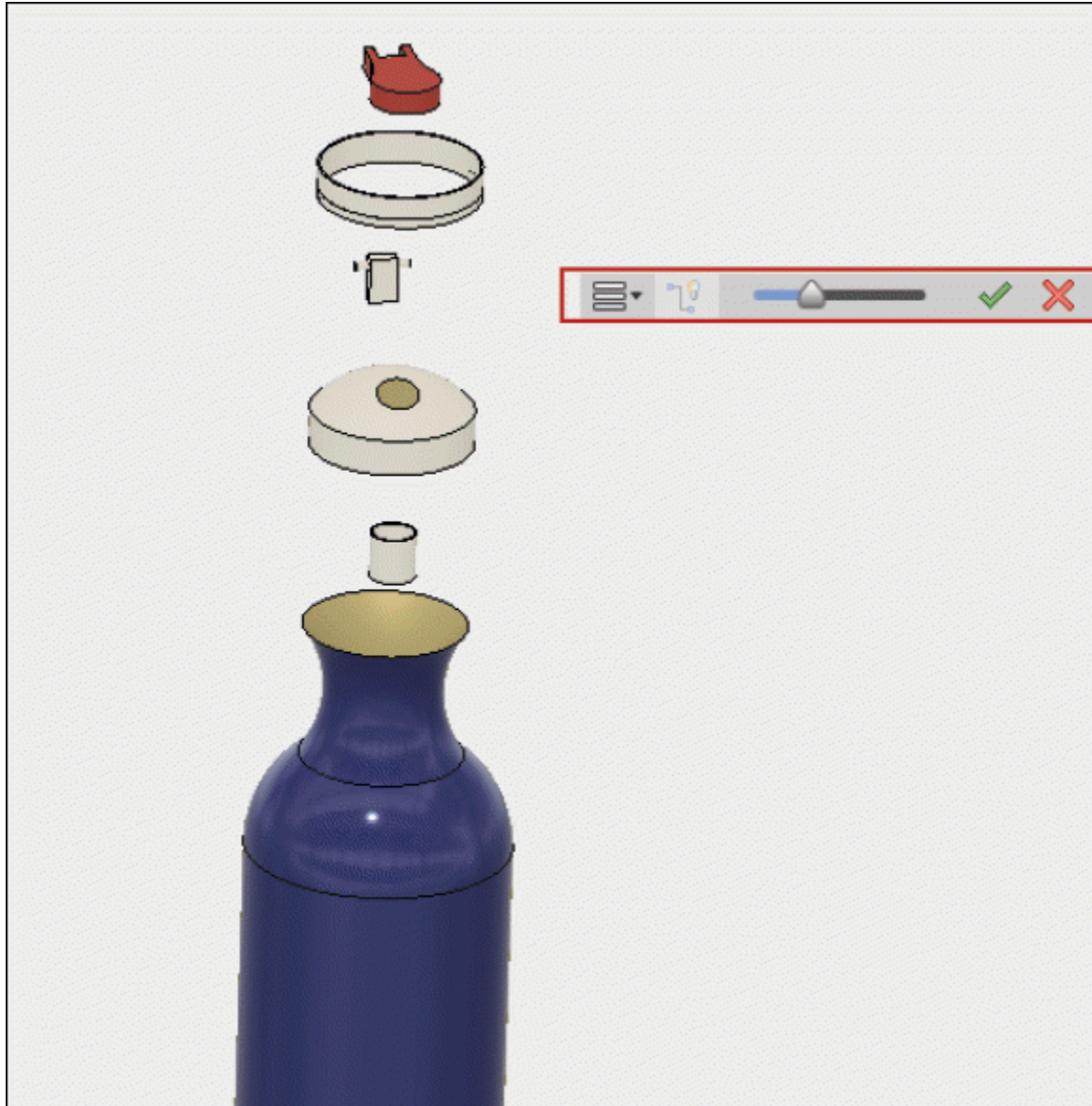


Figure-42. Preview of Auto Explode One Level

- By moving the highlighted slider (**Explosion Scale**), you can set the distance between component.
- After adjusting the space between components, click on the **OK** button from slider.

Auto Explode: All Levels

The **Auto Explode: All Levels** tool is used to automatically

separate all selected components down to all levels in component hierarchy (including sub-assemblies). The procedure to use this tool is discussed next.

- Select the parts of components to separate by holding the **CTRL** key. You can also select the components using window selection.
- Click on the **Auto Explode: All Levels** tool from **TRANSFORM** drop-down; refer to [Figure-43](#). The components of selected model will be separated.

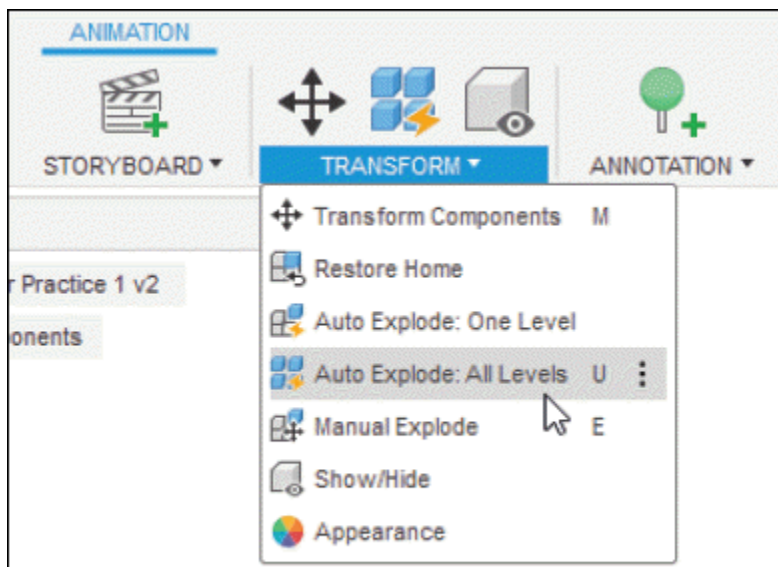


Figure-43. Auto Explode All Levels tool

- Click on the screen to view the separated component properly; refer to [Figure-44](#).

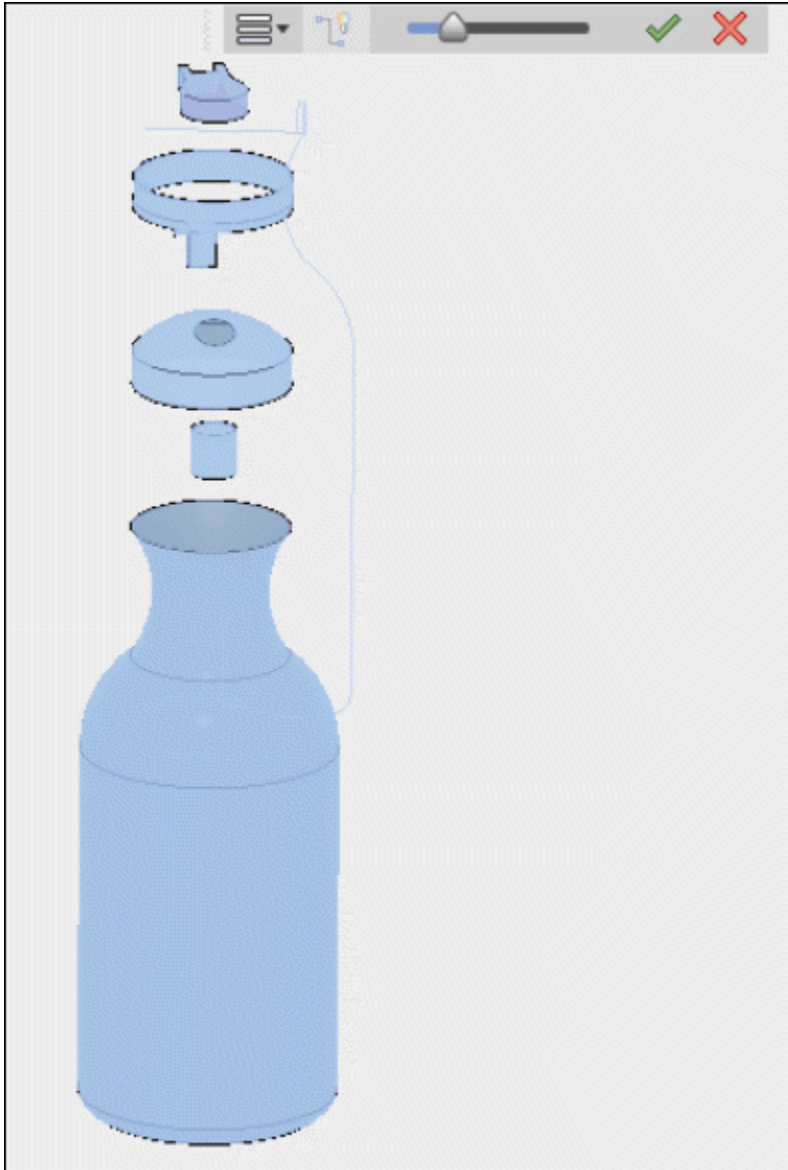


Figure-44. Preview of Explode

- Move the slider to adjust the distance between separated components of model.
- After adjusting the components, click on the **OK** button. The exploded view of assembly will be displayed; refer to [Figure-45](#).

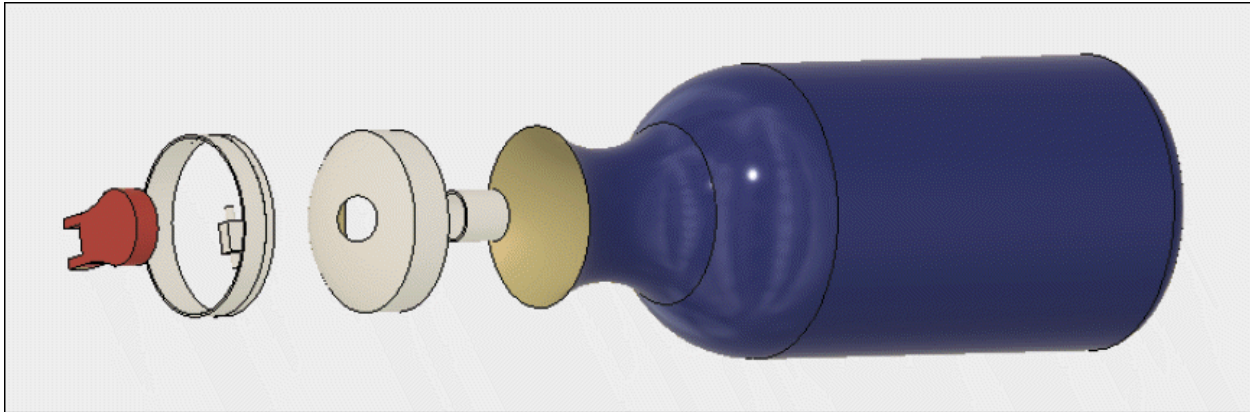


Figure-45. Exploded view of water bottle assembly

Manual Explode

The **Manual Explode** tool is used to manually separate the selected component. The procedure to use this tool is discussed next.

- Click on the **Manual Explode** tool of **TRANSFORM** drop-down from **Toolbar**; refer to [Figure-46](#).

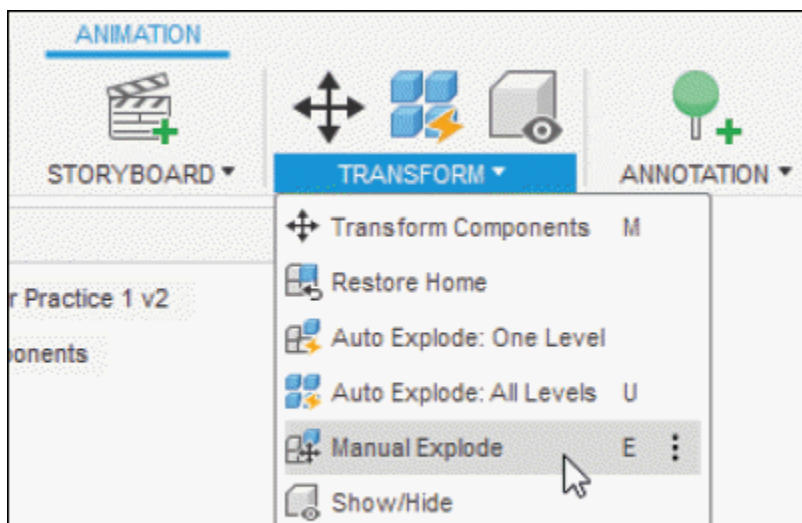


Figure-46. Manual Explode tool

- You are asked to select the components to be moved during exploding. Click on the component to separate;

refer to [Figure-47](#).

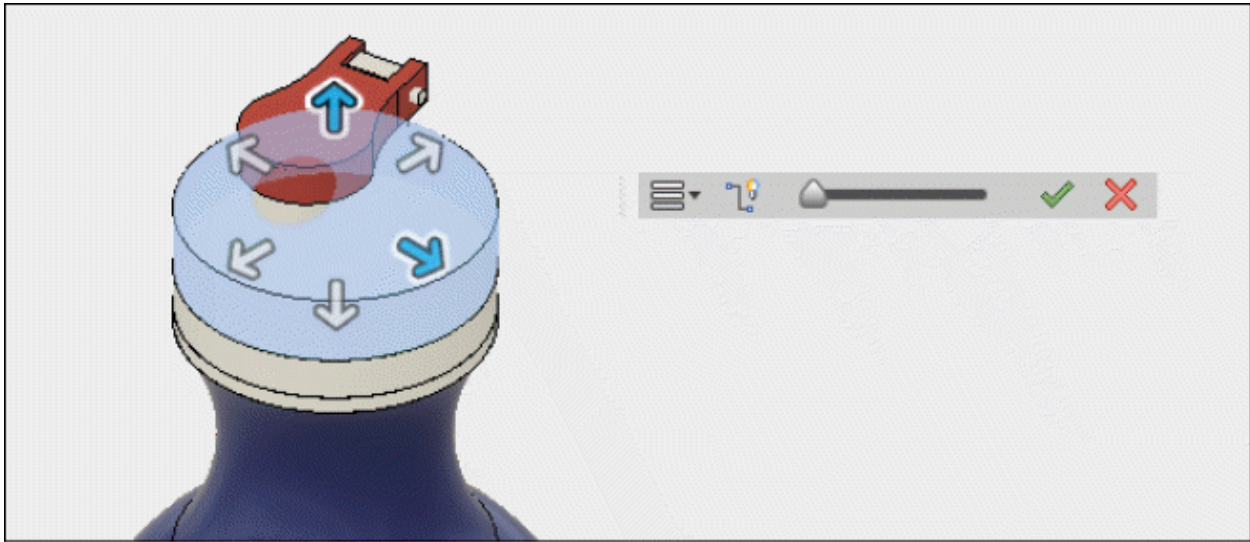


Figure-47. Separating component manually

- On selecting a component, number of arrows will be displayed on the selected component. You need to click on the arrow of direction in which you want to move the selected component.
- Move the **Explosion Scale** slider to separate the selected component from remaining model; refer to [Figure-48](#).

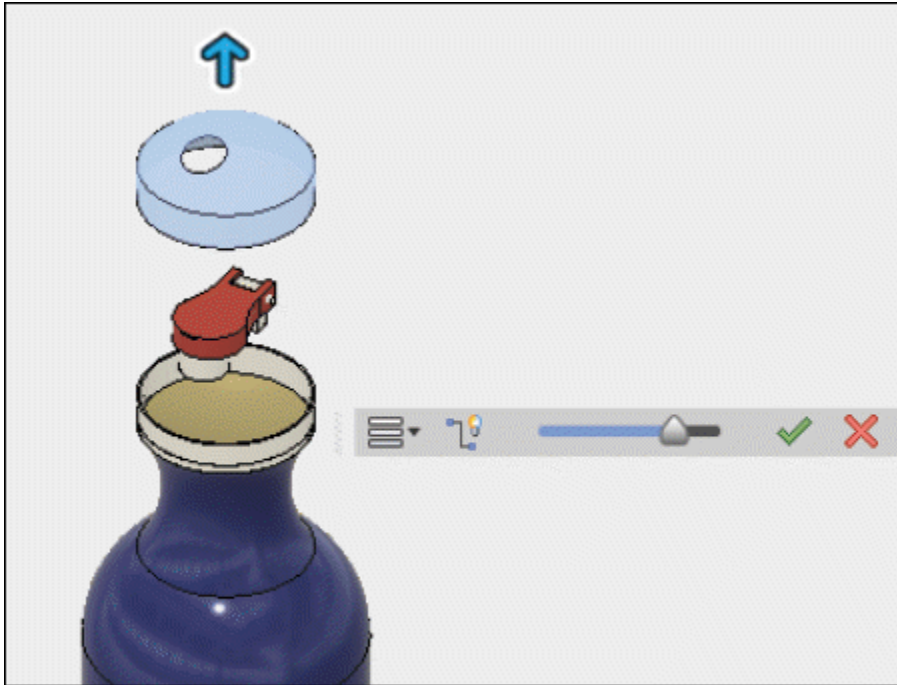


Figure-48. Separating the selected component

Show/Hide

The **Show/Hide** tool is used to show or hide the selected component. The procedure to use this tool is discussed next.

- Click on desired component from model to show or hide.
- Now, click on the **Show/Hide** tool from **TRANSFORM** drop-down; refer to [Figure-49](#). The selected component will be shown or hidden based on its condition.

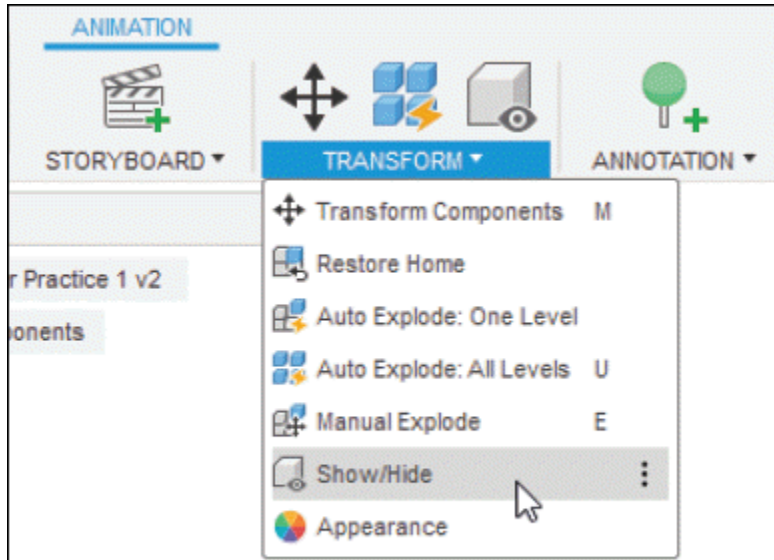


Figure-49. Show/Hide tool

- To display the hidden component, click on the crossed eye button from **Browser**; refer to [Figure-50](#).

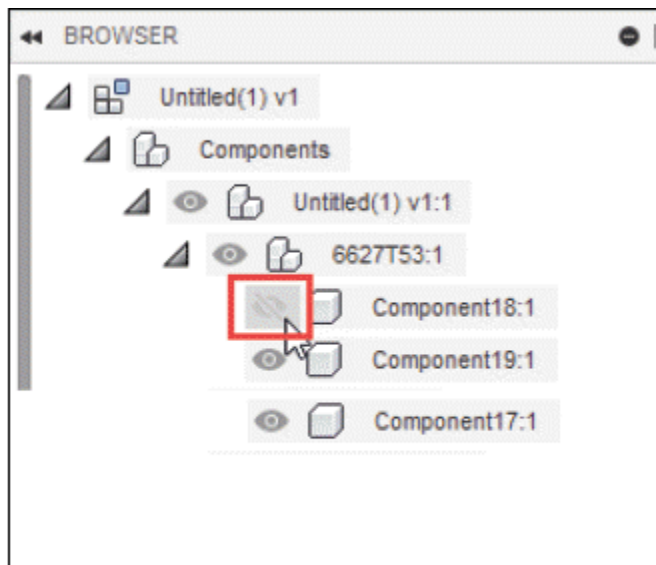


Figure-50. Viewing the hidden component

Create Callout

The **Create Callout** tool is used to add annotation to the animation. The procedure to use this tool is discussed next.

- Click on the **Create Callout** tool from **ANNOTATION** panel; refer to [Figure-51](#). The callout symbol will be attached to the cursor.

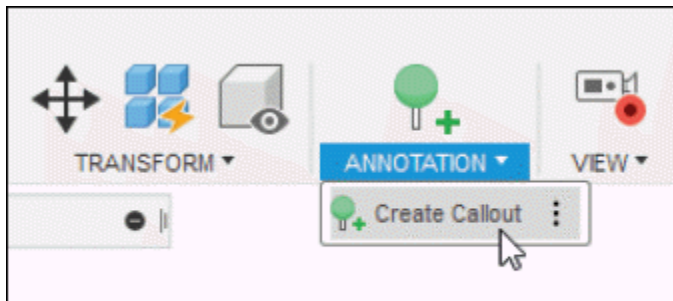


Figure-51. Create Callout tool

- Click on the component to add the callout. A text box will be displayed; refer to [Figure-52](#).

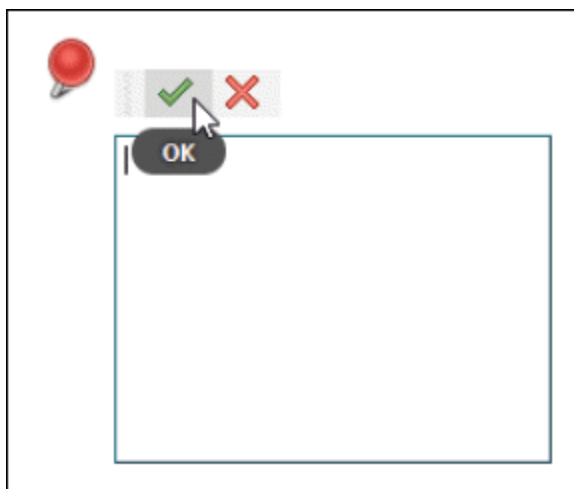


Figure-52. Callout text box

- Type desired text in the text box and click on **OK** button. The annotation callout will be added to the selected component.

View

The **View** tool is used to start and stop the recording of the

actions or commands occurring on the model or component. When **View** tool is activated then it captures the actions like move for animation and when the **View** tool is deactivated it does not record the process; refer to [Figure-53](#).



Figure-53. View tool

Publish Video

The **Publish Video** tool is used to publish the video. The procedure to use this tool is discussed next.

- Click on the **Publish Video** tool from the **PUBLISH** panel in the **Toolbar**; refer to [Figure-54](#). The **Video Options** dialog box will be displayed; refer to [Figure-55](#).



Figure-54. Publish Video tool

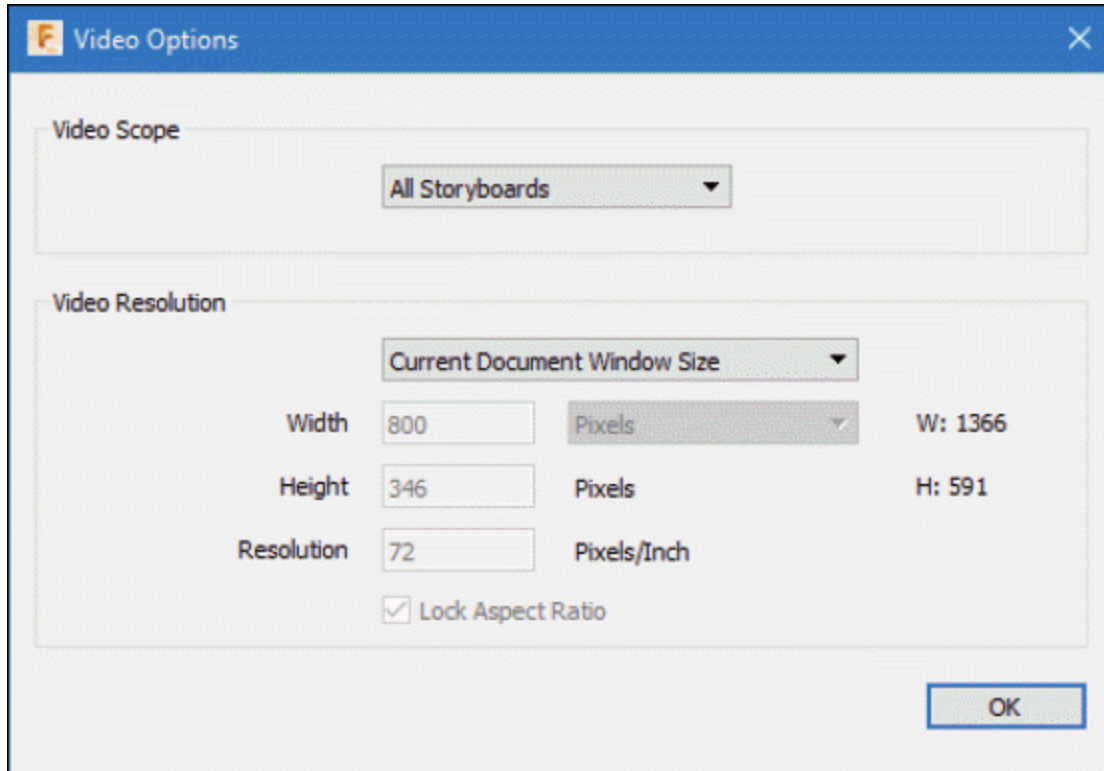


Figure-55. Video Options dialog box

- Select **Current Storyboard** option from **Video Scope** section if you want to create a video of current storyboard animation only.
- Select **All Storyboards** option from **Video Scope** section if you want to create a video of all storyboard animations combined.
- Click on the **Current Document Window Size** drop-down from **Video Resolution** section or select required video resolution.
- After specifying the parameters, click on the **OK** button from **Video Options** dialog box. The **Save As** dialog box will be displayed; refer to [Figure-56](#).

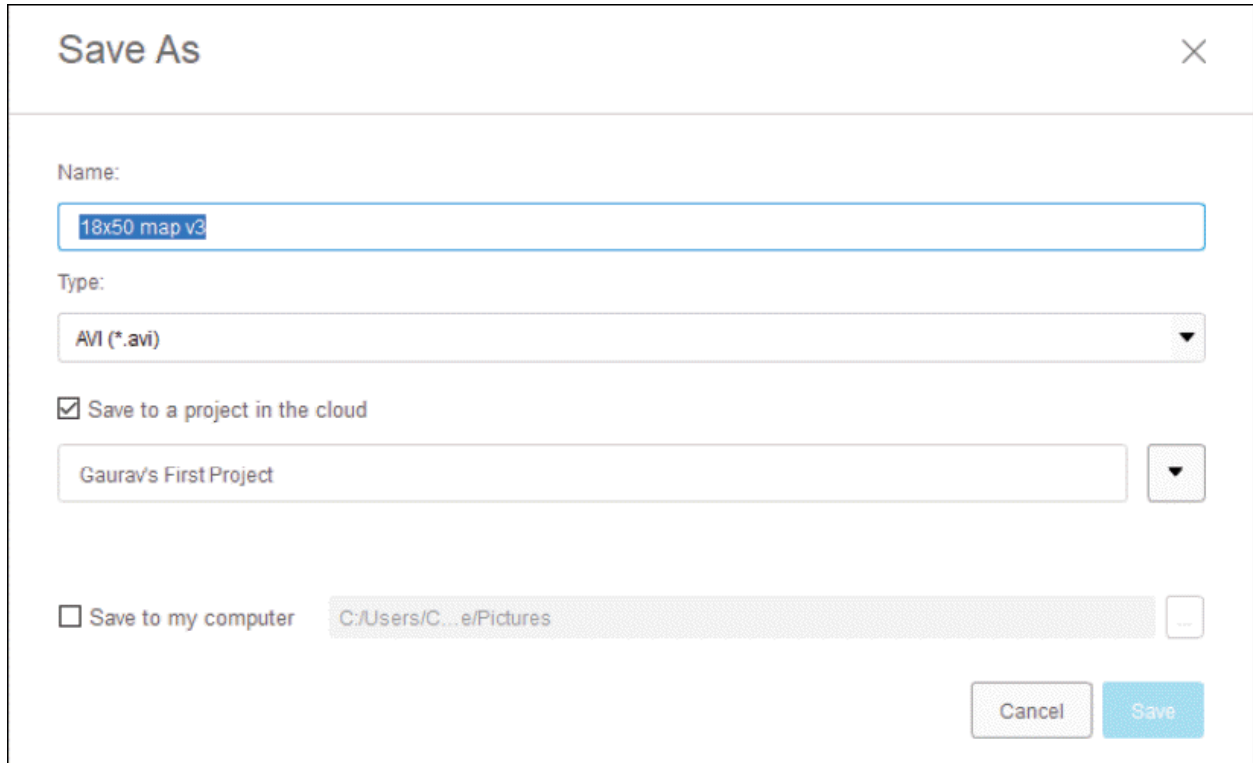


Figure-56. Save As dialog box for Publish Video tool

- Click in the **Name** edit box and specify the name of file.
- Select the **Save to a project in the cloud** check box if you want to save the file online or in your Autodesk account.
- Select the **Save to my computer** check box if you want to save the file in your computer.
- After specifying the parameters, click on the **Save** button from **Save As** dialog box. The **Publish Video** progress dialog box will be displayed which tells about the progress of publishing video; refer to [Figure-57](#).

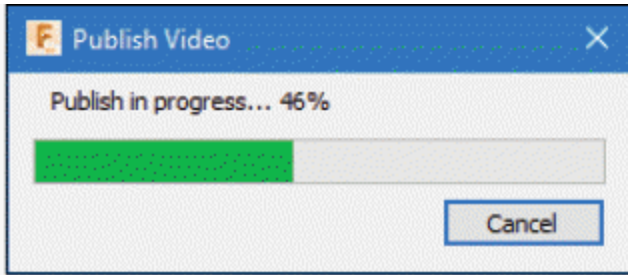


Figure-57. Publish Video progress dialog box

The video will be published online and saved to the computer at preferred location.

SELF ASSESSMENT

Q1. What is Rendering?

Q2. The is used to create the photo-realistic image from a scene of model.

Q3. The tool is used to control the lighting, background color, ground effects, and camera rendering.

Q4. From where can you get the environments for background as well as for model?

Q5. How can you display the shadow and reflections of model on ground canvas?

Q6. A textured ground plane where the environment image is mapped as a texture can be enabled by selecting **Ground Plane** check box. (T/F)

Q7. The reflection of model on the ground can be displayed by selecting the **Reflections** check box. (T/F)

Q8. What is Depth of Field and how it can be enabled?

Q9. If you want to change the environment of the model then

drag a specific environment from section and drop it to the current model or in the **Current Environment** section.

Q10. The **Appearance** tool is used to set the orientation of the texture applied to the face or model. (T/F)

Q11. What is the use of **Reset Transform** button in **Box** dialog box?

Q12. What happens when **In-canvas Render** button is selected?

Q13. What is the use of Animation Workspace?

Q14. Which tool is used to add annotation to the animation?

Chapter 11

Drawing

Topics Covered

The major topics covered in this chapter are:

- *Introduction to Drawing Workspace*
- *Specifying Drawing Preferences*
- *Drawing View*
- *Projected view*
- *Section View*
- *Detail View*
- *Break View*
- *Moving and Rotating Views*
- *Center line*
- *Center Mark and Center Mark Pattern*
- *Edge Extension*
- *Creating Sketch*
- *Dimensioning*
- *Annotation Settings*
- *Editing Dimensions*
- *Creating Text and Note*
- *Applying Surface Texture*
- *Feature Control Frame*
- *Datum Identifier*
- *Generating Tables and Balloon*
- *Bend Identifier and Table*
- *Renumbering and Aligning Balloons*
- *Output DWG, CSV, DXF, and PDF*

INTRODUCTION

Drawing is the engineering representation of a model on the paper. For manufacturing a model in the real world, we need some means by which we can tell the manufacturer what to manufacture. For this purpose, we create drawings from the models. These drawings have information like dimensions, material, tolerances, objective, precautions, and so on.

STARTING DRAWING WORKSPACE

In Autodesk Fusion 360, we create drawings in the

DRAWING workspace. The procedure to open a file in **DRAWING** workspace is discussed next.

- Open desired model from **Data Panel** or saved directory.
- Click on **From Design** option from **DRAWING** cascading menu in **Change Workspace** drop-down. The **CREATE DRAWING** dialog box will be displayed along with the model; refer to [Figure-1](#).

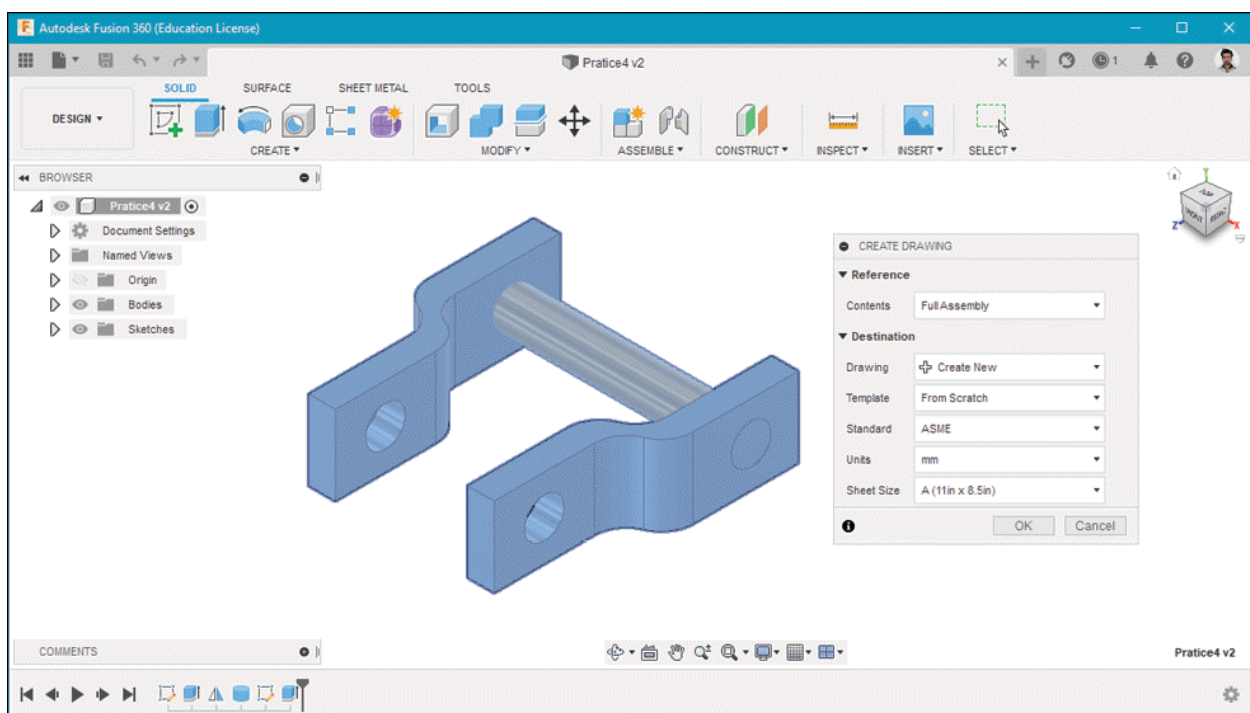


Figure-1. CREATE DRAWING dialog box

- Select the **Full Assembly** option from **Contents** drop-down of **Reference** area in the dialog box if you want to create drawing using whole assembly. Select the **Visible Only** option from **Contents** drop-down to automatically select only visible components of assembly. Select the **Select** option from **Contents** drop-down to manually

select components in the canvas or browser.

- Click on the **Standard** drop-down from **CREATE DRAWING** dialog box and select the standard for the drawing. **ASME** option can use the third angle projection as well as first angle projection but **ISO** option uses first angle projection for placing views. There are also many other differences between two standards which you can find better in relevant books. Note that these projections are default settings but you can override them using **Preferences** dialog box.
- Click on the **Units** drop-down from **CREATE DRAWING** dialog box and select the unit for drawing.
- Click on the **Sheet Size** drop-down from **CREATE DRAWING** dialog box and select the size of sheet for drawing. Note that there is difference between A (8.5x11) and A (11x8.5) sheet size. The size A (8.5x11) will generate Portrait sheet whereas size A(11x8.5) will generate Landscape sheet.
- After specifying the parameters, click on the **OK** button from **CREATE DRAWING** dialog box. The **DRAWING** workspace will be activated along with **DRAWING VIEW** dialog box in a new drawing file; refer to [Figure-2](#).

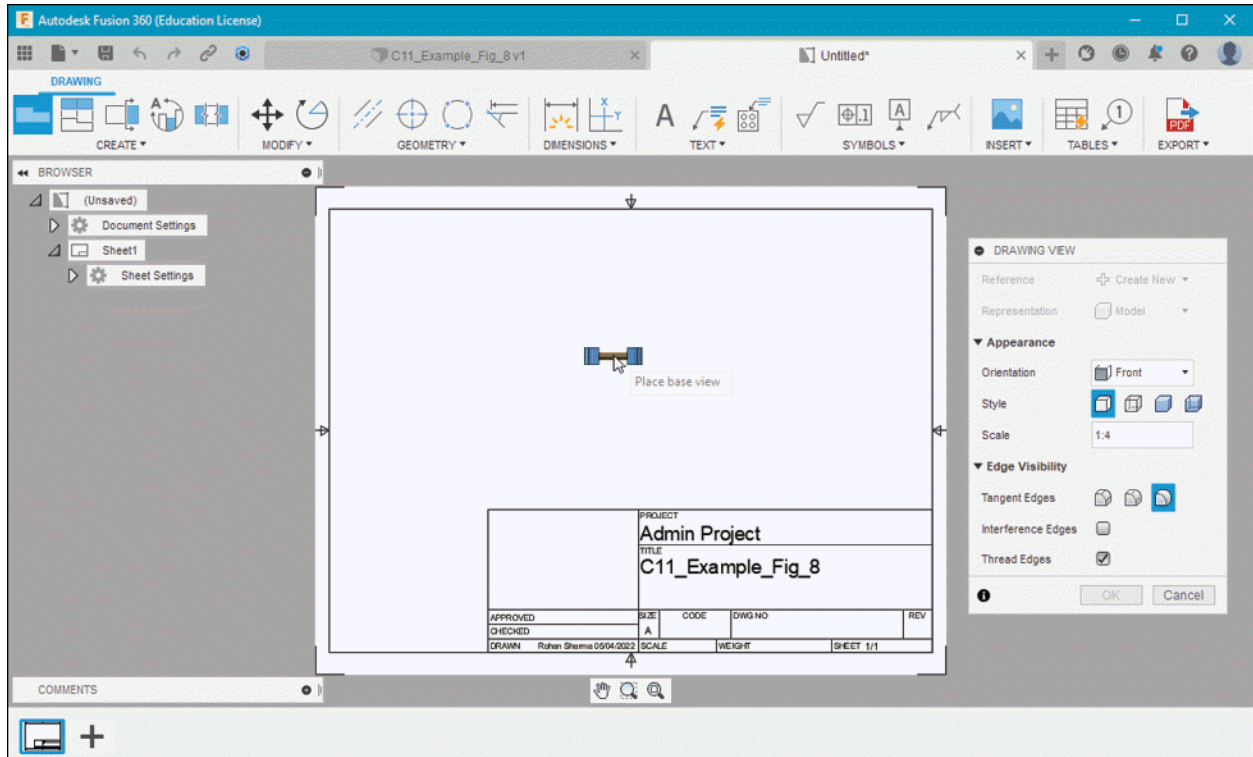


Figure-2. DRAWING workspace

- The model will be attached to the cursor. Click on the drawing sheet to place the base view.

Parameters in DRAWING VIEW dialog box

- Click on the **Orientation** drop-down and select the base view of model. Here, **Front** view is default option. When you change the view from **Front** to **Top**, the view of model will automatically changed.
- Select the style of model from the **Style** section.
- Click in the **Scale** drop-down and select required scaling factor for model.
- Select required option from **Tangent Edges** section to apply the edges with the selected view.

- Select the **Interference Edges** check box if you want to keep interfering edges within the selected view. Interfering edges are overlapping edges from two features of the model.
- Select the **Thread Edges** check box if you want to display the threads of model in current view.
- After specifying the parameters in **DRAWING VIEW** dialog box, click on **OK** button. The selected view of model will be placed; refer to [Figure-3](#).

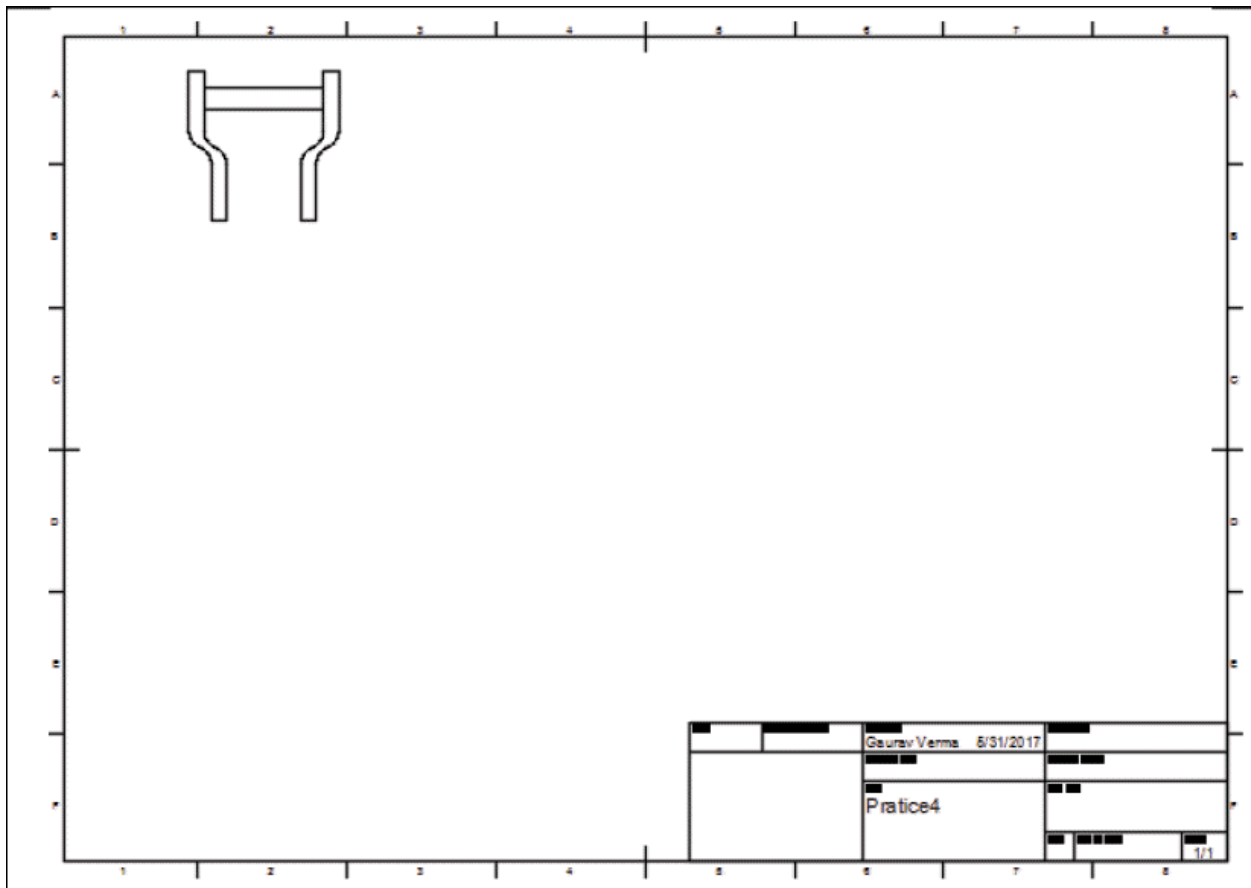


Figure-3. Top view of model

SPECIFYING DRAWING PREFERENCES

- Click on the **Preferences** option from the **Autodesk Account** drop-down. The **Preferences** dialog box will be displayed.
- Select the **Drawing** option from the **General** node at the left in the dialog box. The options will be displayed as shown in [Figure-4](#).

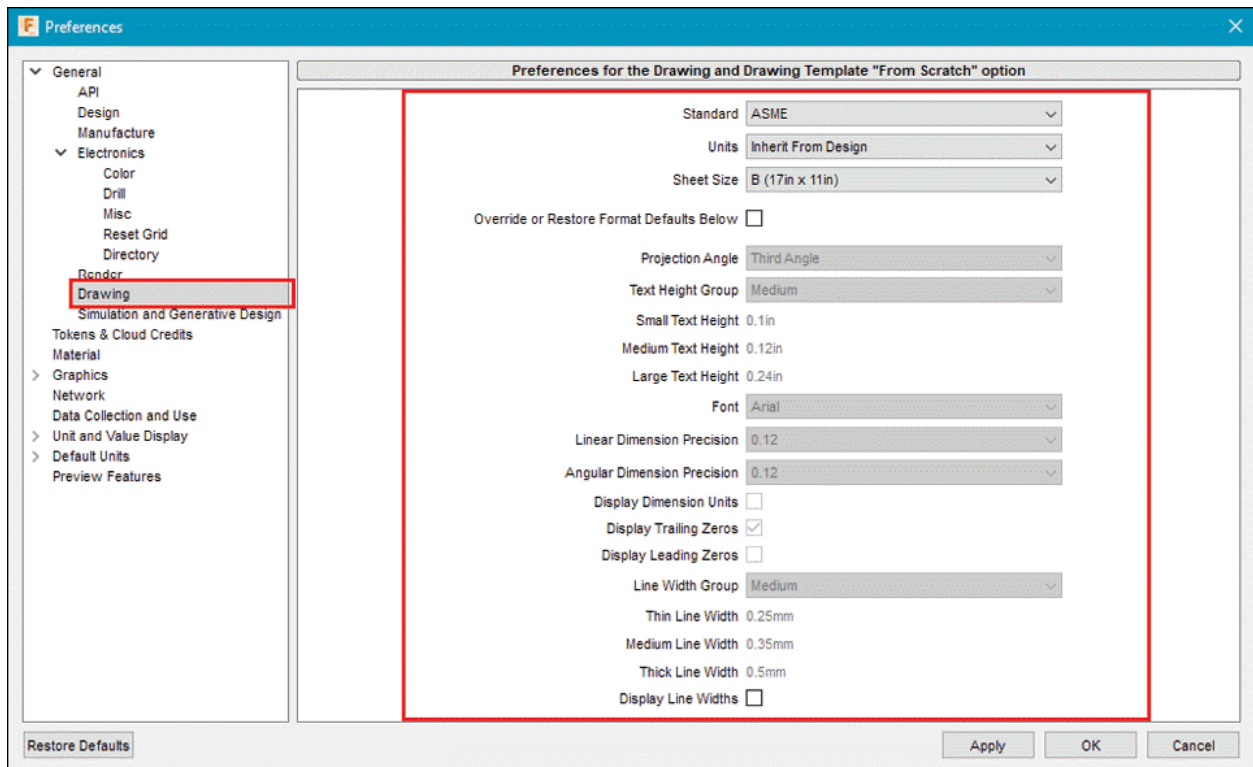


Figure-4. Drawing node in Preferences dialog box

- Select desired option from the **Standard** drop-down to define which standard to use for drafting. You can select **ASME** or **ISO** option from the drop-down to set it default for drafting or you can select the **Inherit From Design** to define the standard by 3D Model.
- Select desired unit of length from the **Units** drop-down.

You can select inch or mm as unit.

- Select desired default size of sheet from the **Sheet Size** drop-down.
- Select the **Override or Restore Format Defaults Below** check box if you want to use parameters different from selected standard. The options below this check box will be activated; refer to [Figure-5](#).

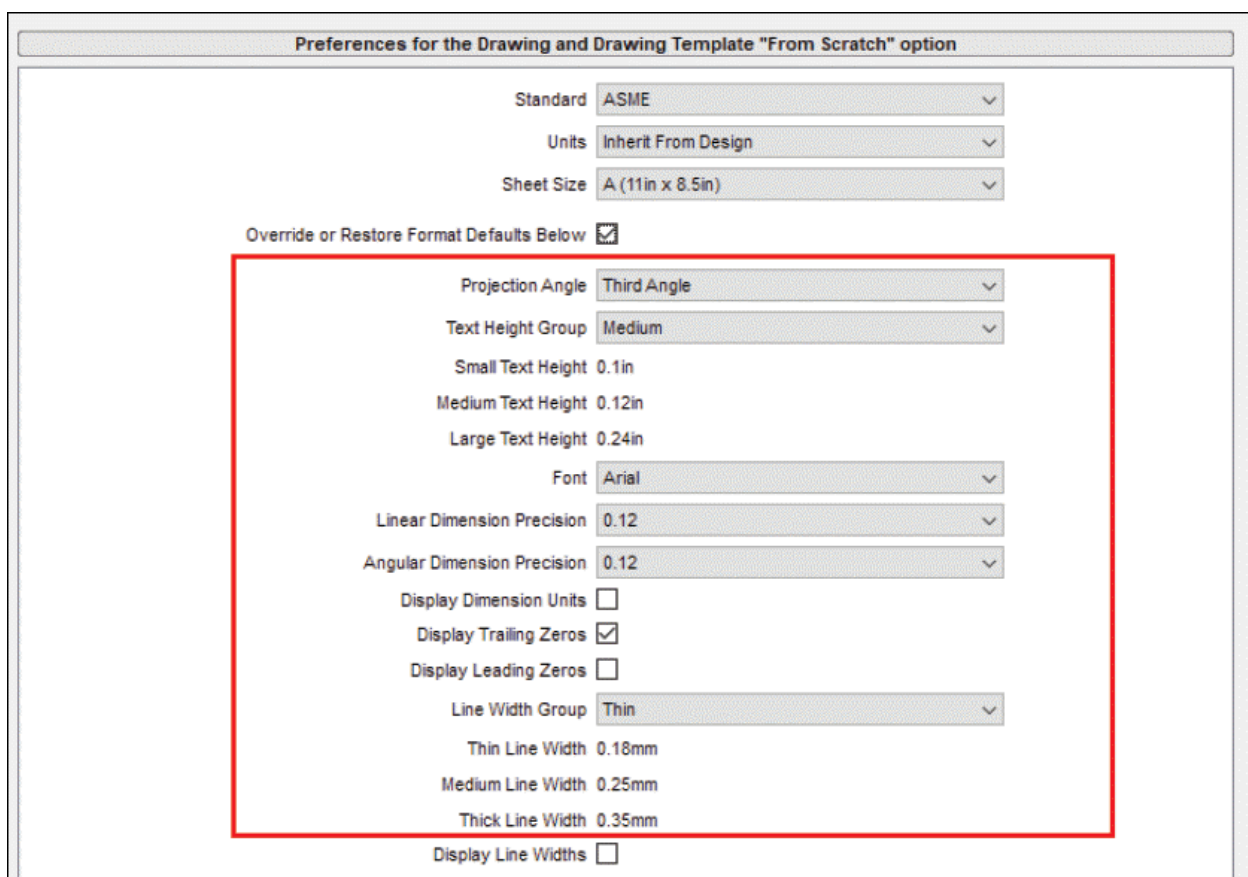


Figure-5. Override options

- Select desired option from the **Projection Angle** drop-down to define the view projection method for drawing.
- Select desired value for height of annotations in **Text Height Group** drop-down.

- Select desired font from **Font** drop-down to define font of all the annotations in drawing area.
- Select desired precision level for linear and angular dimensions in the **Linear Dimension Precision** and **Angular Dimension Precision** drop-downs, respectively.
- Select the **Display Dimension Units** check box to display unit of annotation in drawing.
- Select the **Display Trailing Zeros** check box to display zeros after decimal places if the dimension is not up to specified precision level.
- Select the **Display Leading Zeros** check box if you want to display zeros before the dimension values in the annotations.
- Select desired value for line width for drawings in **Line Width Group** drop-down.
- Select the **Display Line Widths** check box if you want to display lines in drawing by their specified widths rather than using same width for all lines.
- Click on the **OK** button from the **Preferences** dialog box.

CREATING BASE VIEW

The **Base View** tool is used to create base view derived directly from a 3D model. You can create various views of your model using this view. Note that you should have only one base view and other views should be projection of that view. The procedure to use this tool is discussed next.

- Click on the **Base View** tool from **CREATE** panel; refer to [Figure-6](#). The **DRAWING VIEW** dialog box will be displayed.

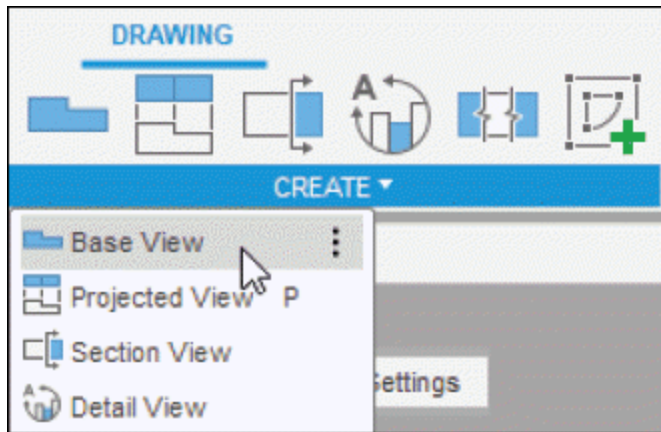


Figure-6. Base View tool

- Click at desired location to place the view and specify desired parameters in the dialog box. The options of **DRAWING VIEW** dialog box have been discussed earlier.
- Click on the **OK** button from the dialog box to generate the view.

CREATING PROJECTED VIEW

The **Projected View** tool is used to create drawing views generated from an existing base or drawing view. The procedure to use this tool is discussed next.

- Click on the **Projected View** tool from **CREATE** panel in the **Toolbar**; refer to [Figure-7](#). The tool will be activated.

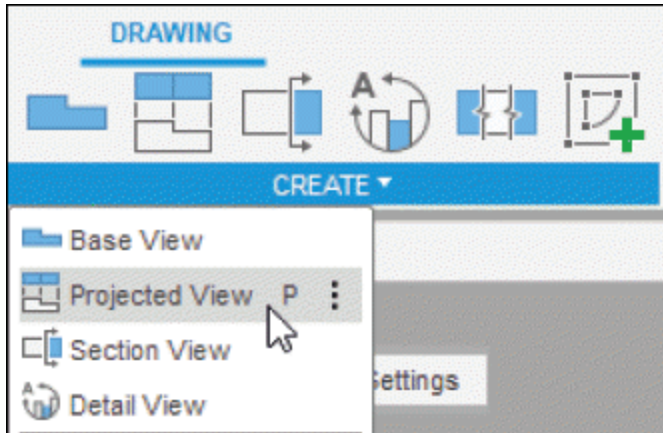


Figure-7. Projected View tool

- Click on the pre-existing base view from drawing sheet to select the view as parent view; refer to [Figure-8](#). The projected view will be attached to the cursor; refer to [Figure-9](#).

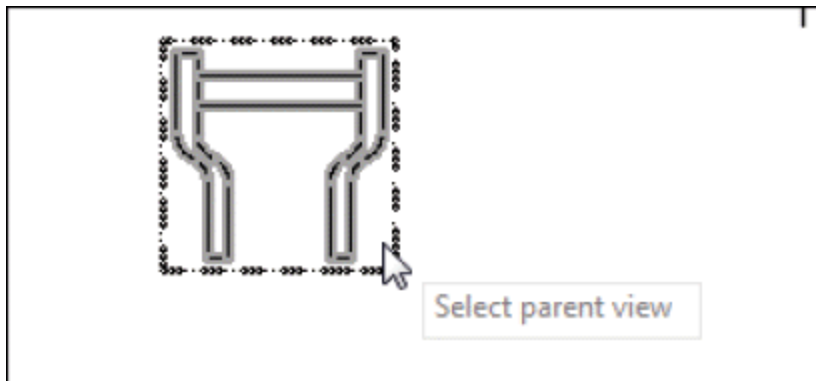


Figure-8. Selecting parent view

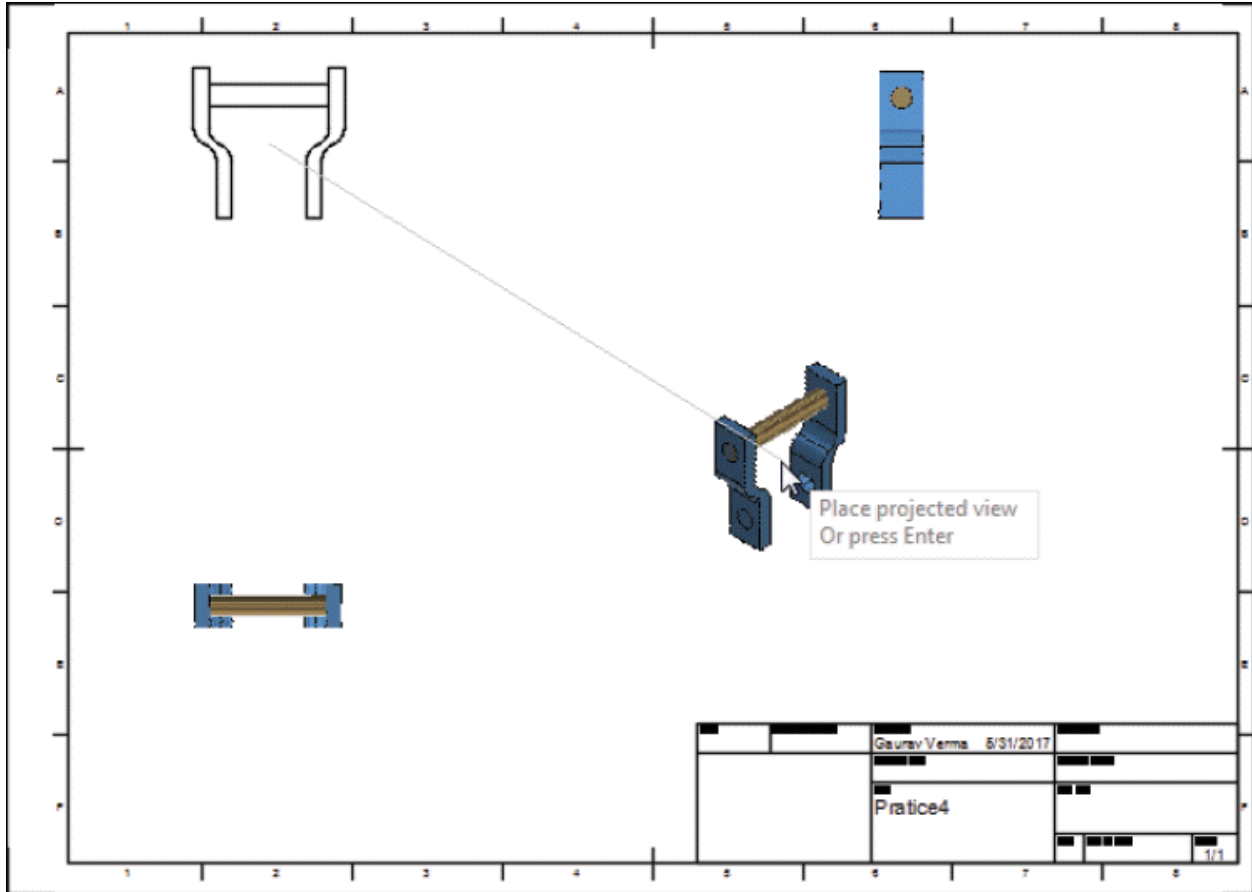


Figure-9. Placing projected view

- Place the projected views of model at desired locations and press **ENTER**. The projected view(s) are created on drawing sheet; refer to [Figure-10](#).

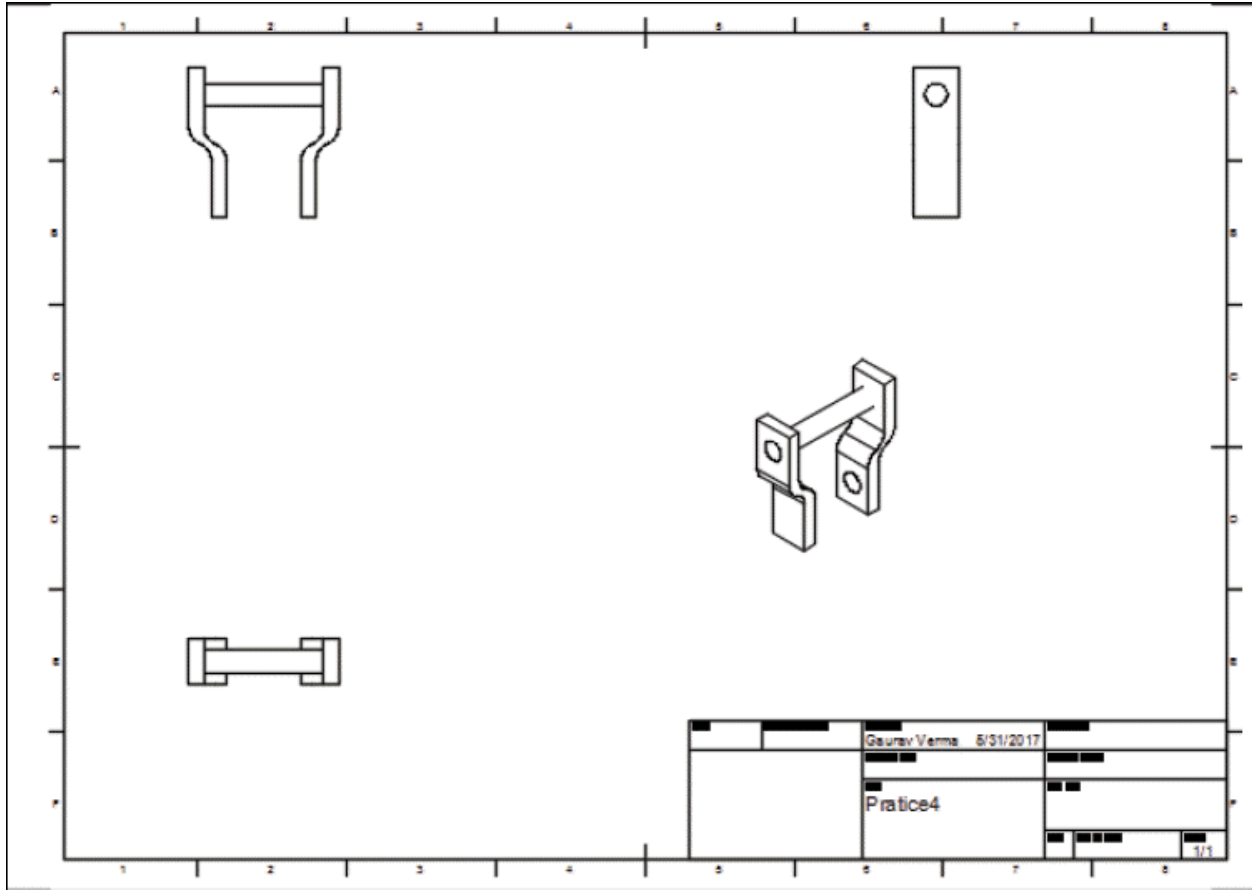


Figure-10. Projected views created

CREATING AUXILIARY VIEW

The **Auxiliary View** tool is used to create a projected view using view point normal to selected edge. The procedure to use this tool is given next.

- Click on the **Auxiliary View** tool from the **CREATE** drop-down in the **Toolbar**. You will be asked to select the edge for generating auxiliary view.
- Select desired edge from the model. The auxiliary view will get attached to cursor; refer to [Figure-11](#).

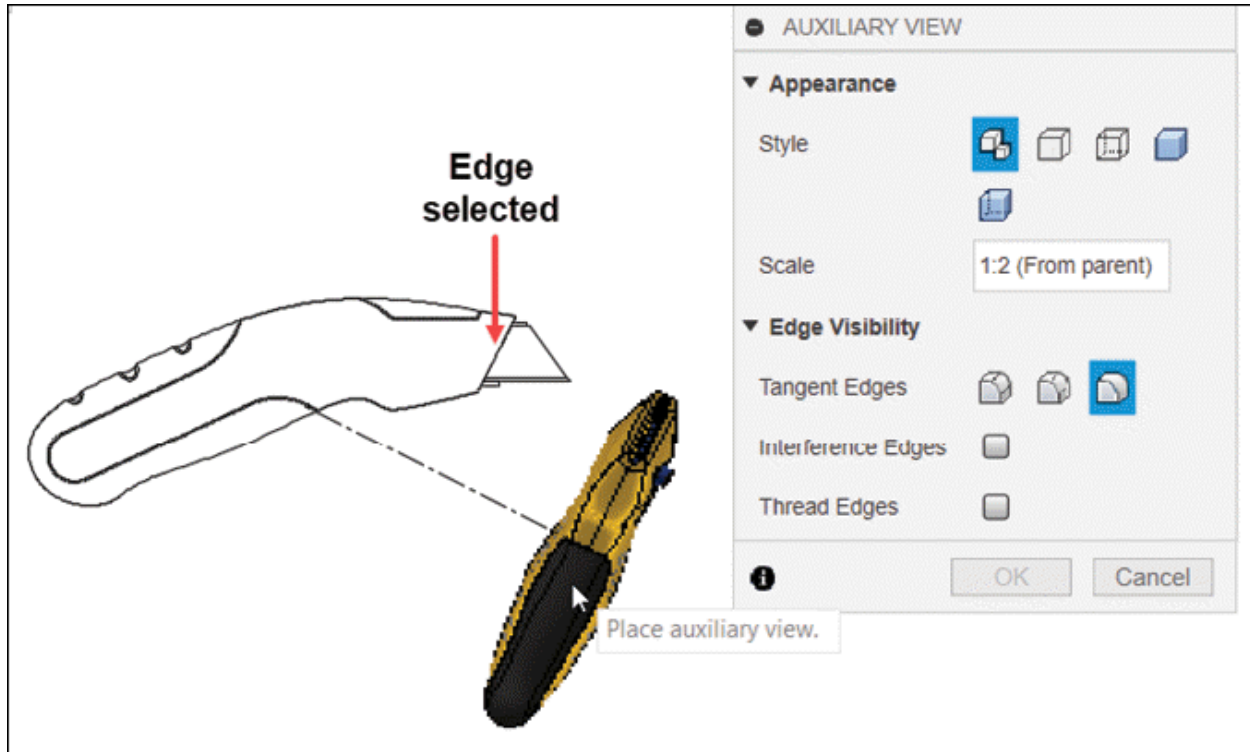


Figure-11. Auxiliary view attached to cursor

- Click at desired location in the drawing area to place the view and set the parameters in **AUXILIARY VIEW** dialog box as discussed earlier.
- Click on the **OK** button from dialog box to create the view.

CREATING SECTION VIEW

The **Section View** tool is used to create full, half, offset, or aligned section views. The procedure to use this tool is discussed next.

- Click on the **Section View** tool from **CREATE** panel; refer to [Figure-12](#). The tool will be activated.

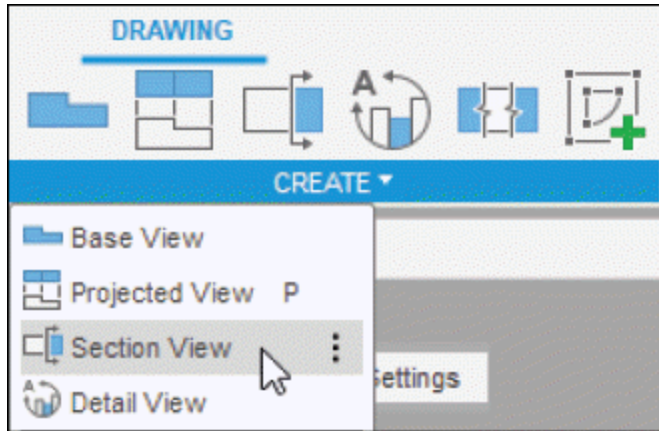


Figure-12. Section View tool

- Click on the projected or base view from drawing sheet to be used for section view. The **DRAWING VIEW** dialog box will be displayed.
- You need to divide the selected parent view into two parts to create half section; refer to [Figure-13](#).
- Click on the parent view to specify the start point and end point. You can create different shape of section line to create desired type of section. Note that you can use snap point in right-click shortcut menu to select points for section line; refer to [Figure-14](#). After specifying the start point, intermediate points, and end point, click on **ENTER** key. The section view will be created and attached to the cursor; refer to [Figure-15](#).

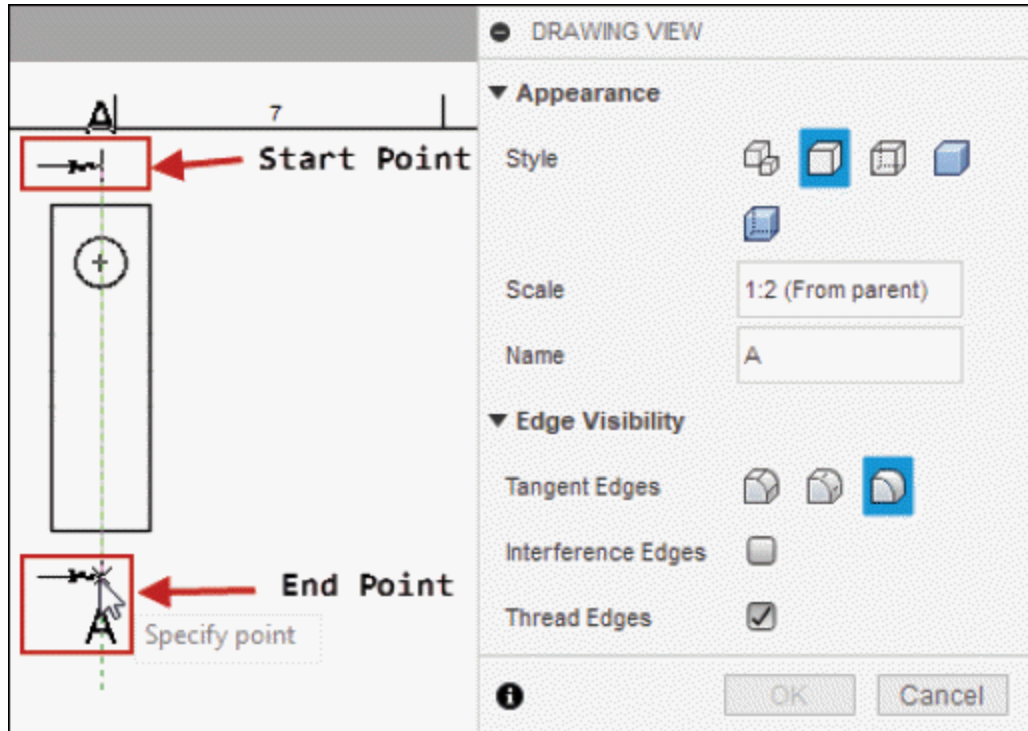


Figure-13. Creating section view

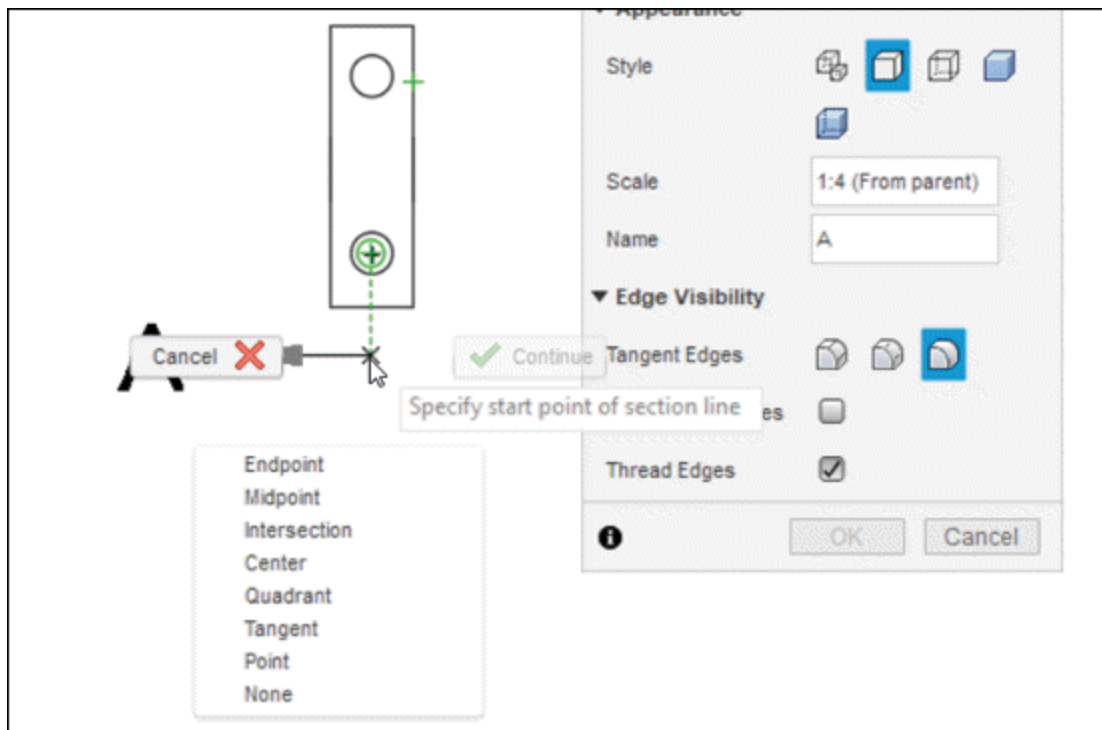


Figure-14. Shortcut menu for snap points

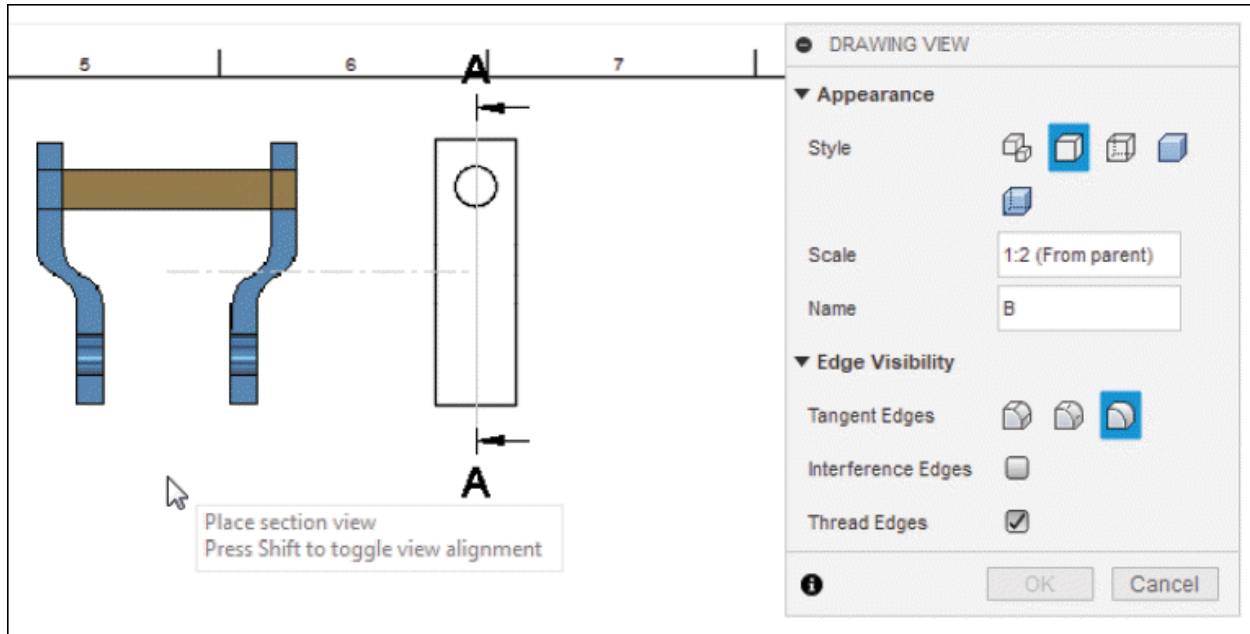


Figure-15. Section view attached to the cursor

- Place the section view at desired location/side of base model and click on **OK** button from **DRAWING VIEW** dialog box. The section view will be created on drawing sheet; refer to [Figure-16](#). Note that arrows of section view will be modified automatically based on placement location of section view.

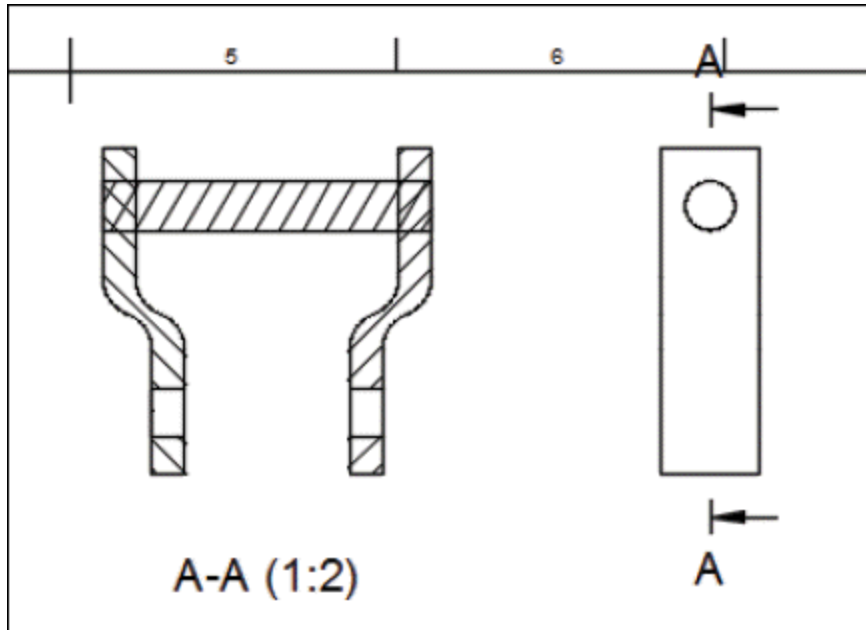


Figure-16. Section View created

- Note that the hatching is created automatically in the section view. To edit hatching, double-click on it. The **HATCH** dialog box will be displayed. Specify desired values and click on the **Close** button.
- If you want to edit the section annotation then double-click on it. The **TEXT** dialog box will be displayed; refer to [Figure-17](#).

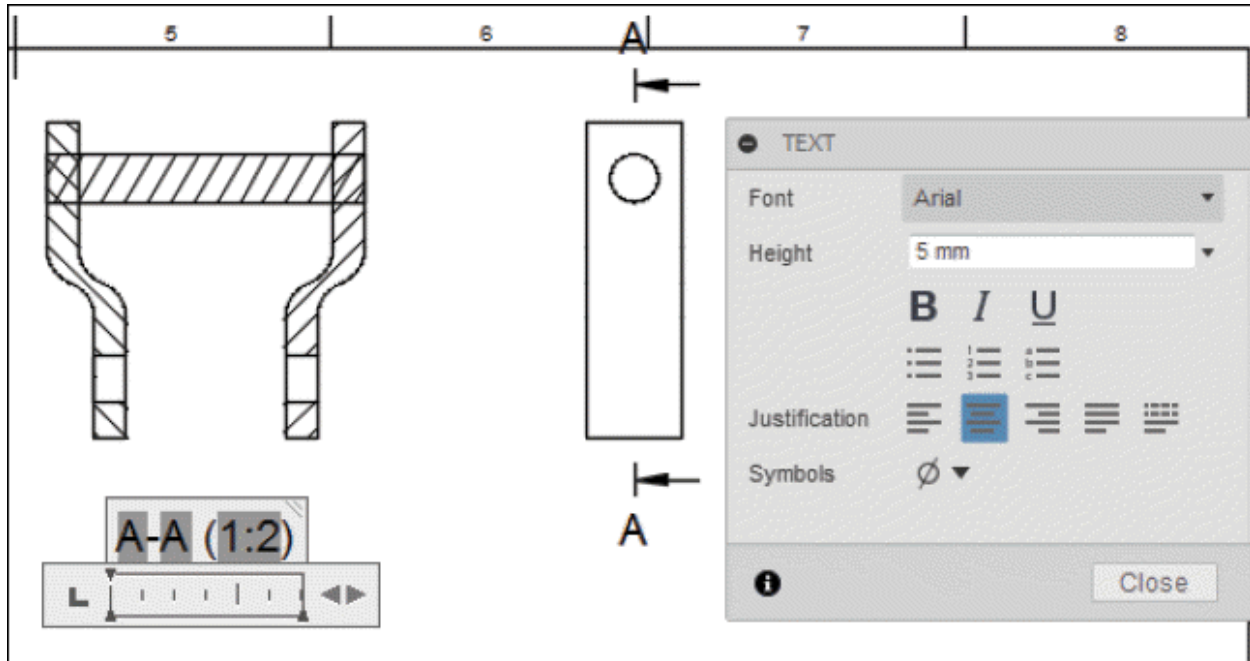


Figure-17. TEXT dialog box for annotation

- Click on the **Font** drop-down and select required font from the list.
- Click in the **Height** edit box and specify the height of text and select the writing style of text from **TEXT** dialog box.
- Click on desired text alignment from **Justification** section.
- If you want to insert any special symbol then click on it from **Symbols** section of the dialog box.
- After specifying the parameters for the text, click in the text box from drawing sheet and enter required text.
- Click on the **Close** button from **TEXT** dialog box to complete the process. The text will be modified according to parameters specified.

CREATING DETAIL VIEW

The **Detail View** tool is used to create detail view from any projected or base view. The procedure to use this tool is discussed next.

- Click on the **Detail View** tool of **CREATE** panel; refer to [Figure-18](#). The tool will be activated and you will be asked to select a view whose selected section is to be included in detail view.

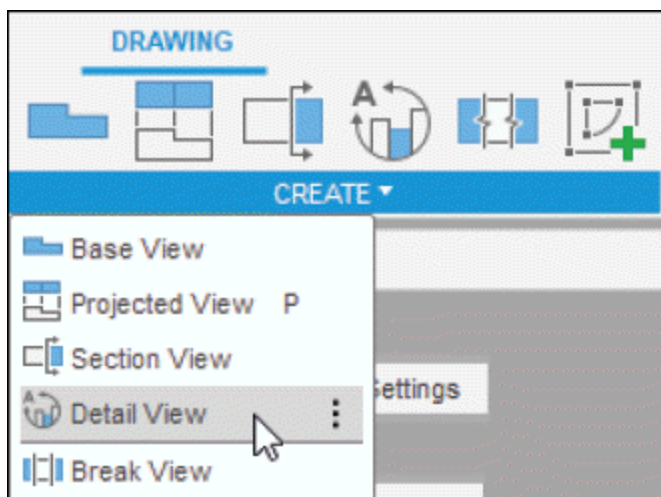


Figure-18. Detail View tool

- Select desired view from drawing sheet; refer to [Figure-19](#). The **DRAWING VIEW** dialog box will be displayed.

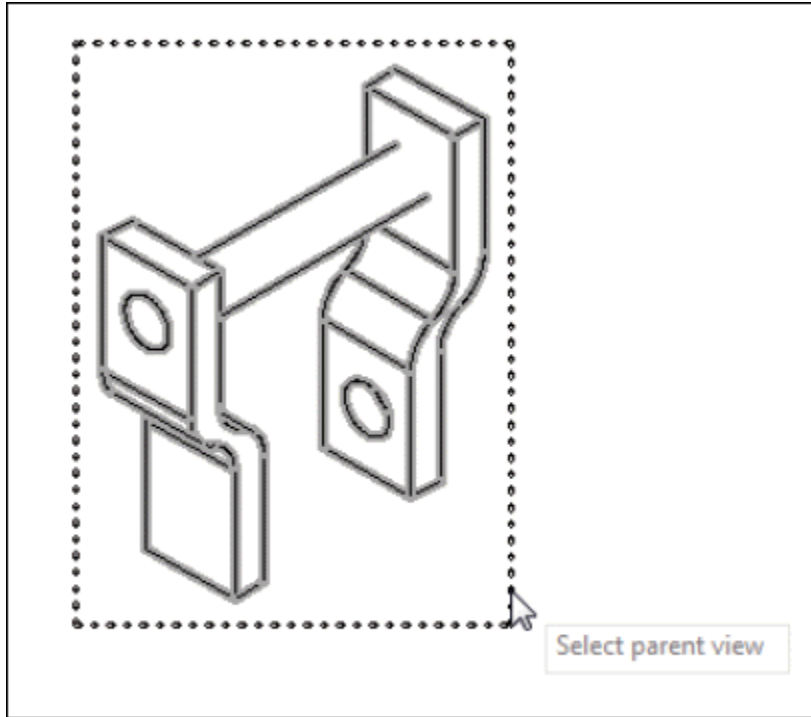


Figure-19. Selecting parent view for detail view

- Click at desired location to specify the center point for boundaries of detail view; refer to [Figure-20](#).

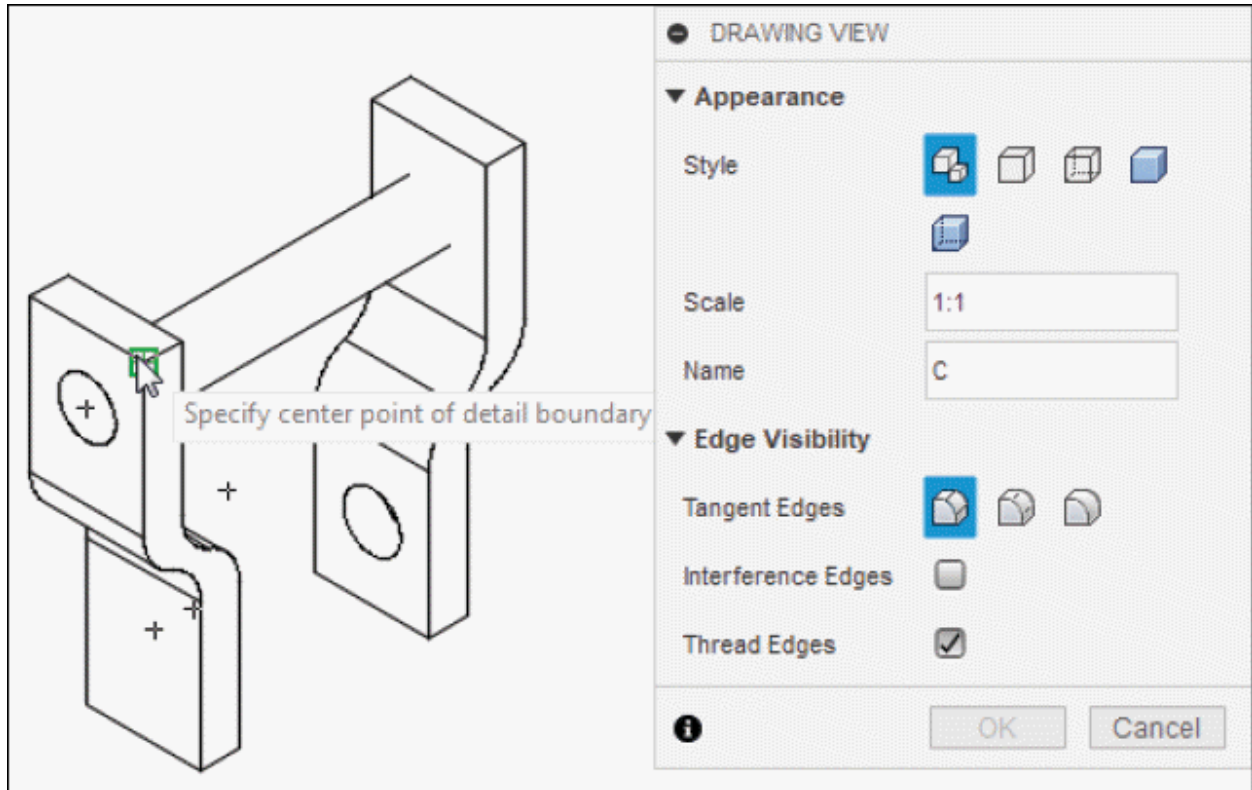


Figure-20. Specifying center point for boundary

- Create the boundary on parent view as required; refer to [Figure-21](#).

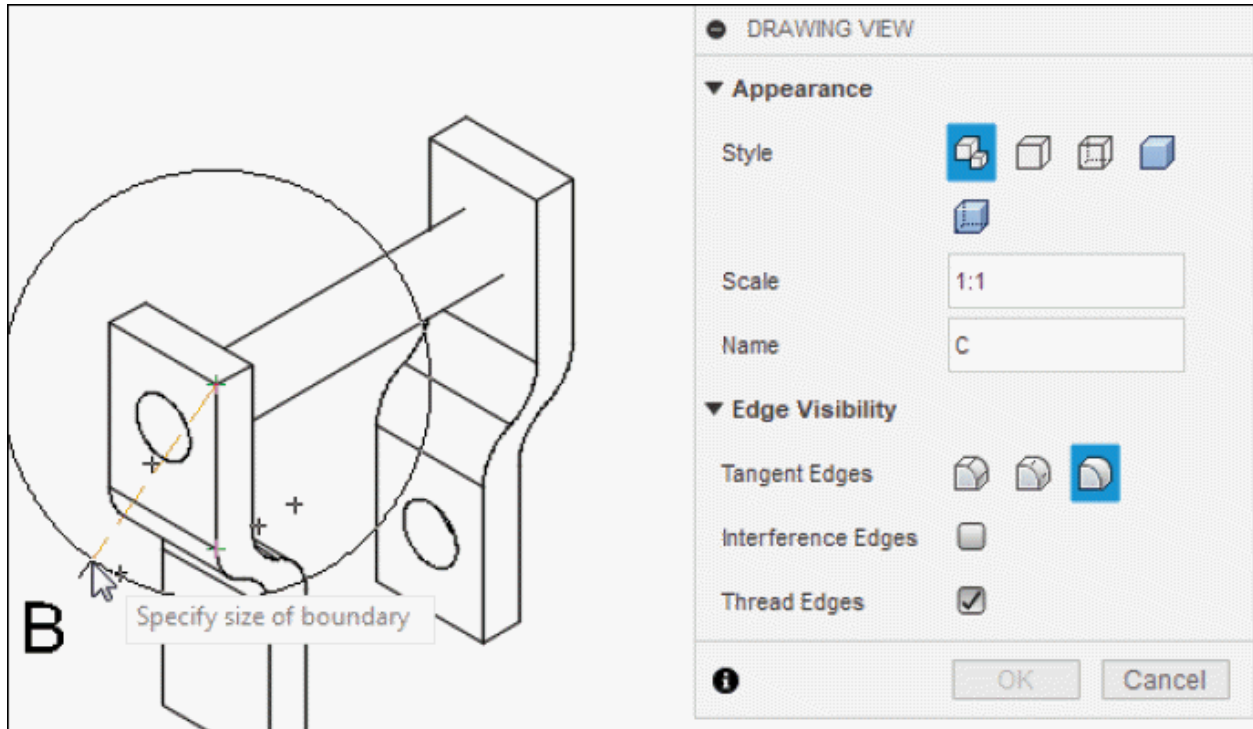


Figure-21. Creating boundary

- On creating the circular boundary, the detail view of model will be attached to the cursor; refer to [Figure-22](#). Place it as desired.

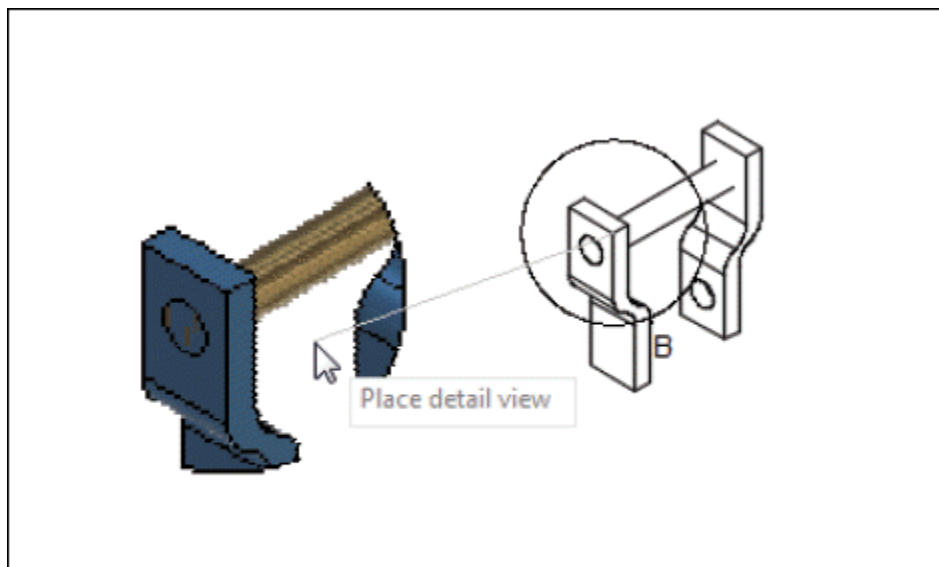


Figure-22. Placing detail view

- After placing the view, set the parameters of detail view in **DRAWING VIEW** dialog box and click on **OK** button. The detailed view will be created; refer to [Figure-23](#).

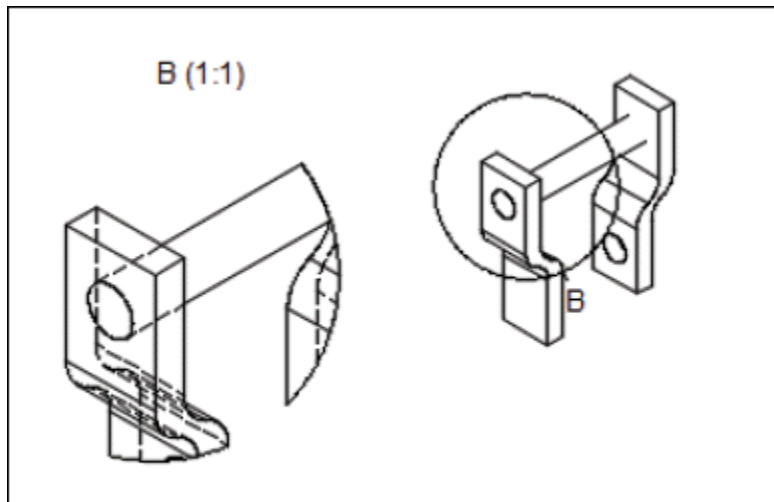


Figure-23. Detailed view created

CREATING BREAK VIEW

The **Break View** tool is used to shorten an existing drawing view by removing a portion of the design and indicating the missing section in the drawing. Broken view is generally created to place long views. The procedure to use this tool is discussed next.

- Click on the **Break View** tool from **CREATE** panel; refer to [Figure-24](#). The **BREAK VIEW** dialog box will be displayed and you will be asked to select the drawing view to be broken; refer to [Figure-25](#).

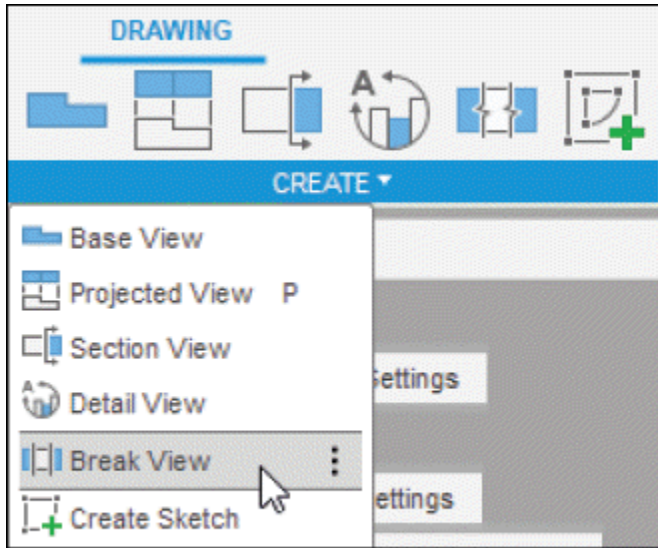


Figure-24. Break View tool

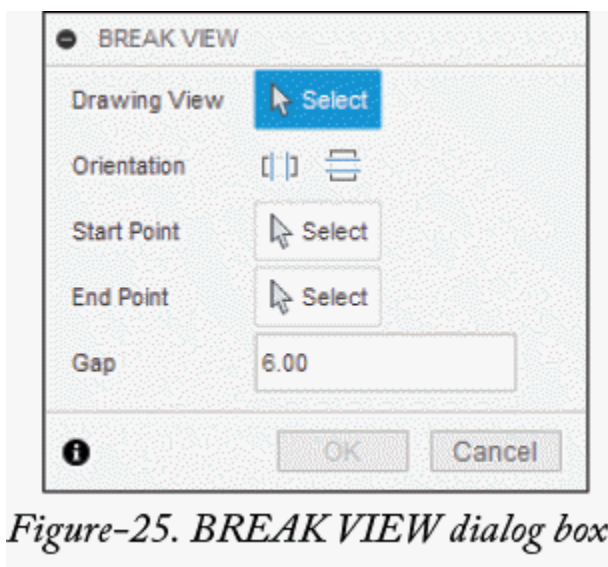


Figure-25. BREAK VIEW dialog box

- The **Select** button of **Drawing View** section is active by default. Select desired view from drawing sheet which you want to break; refer to [Figure-26](#). You will be asked to select break position.

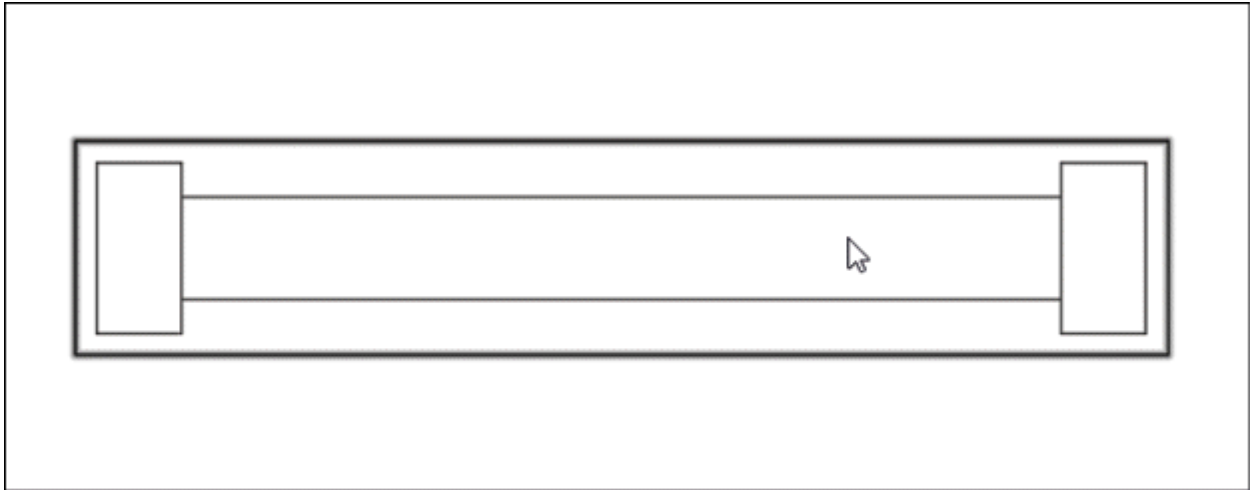


Figure-26. Selecting drawing view for broken view

- Select the start point of break position using the **Start Point** selection button; refer to [Figure-27](#). You will be asked to select end point of break position.

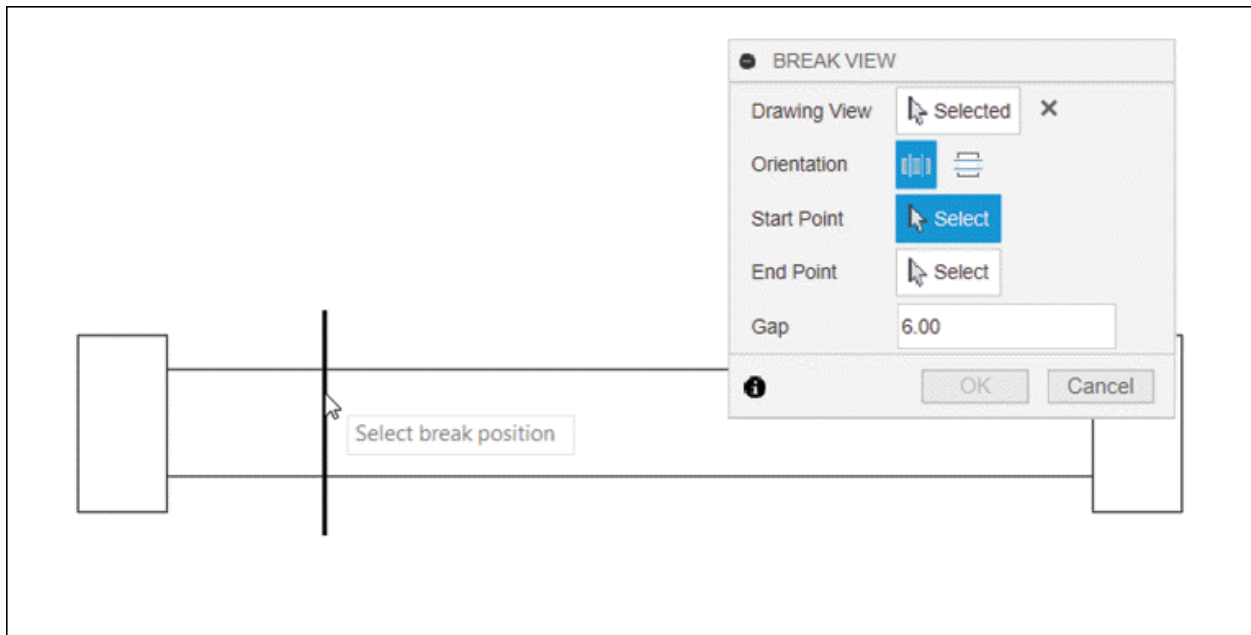


Figure-27. Selecting start point of break position

- Move the cursor away and click at desired location to specify end position of break position in the **End Point**

selection button; refer to [Figure-28](#).

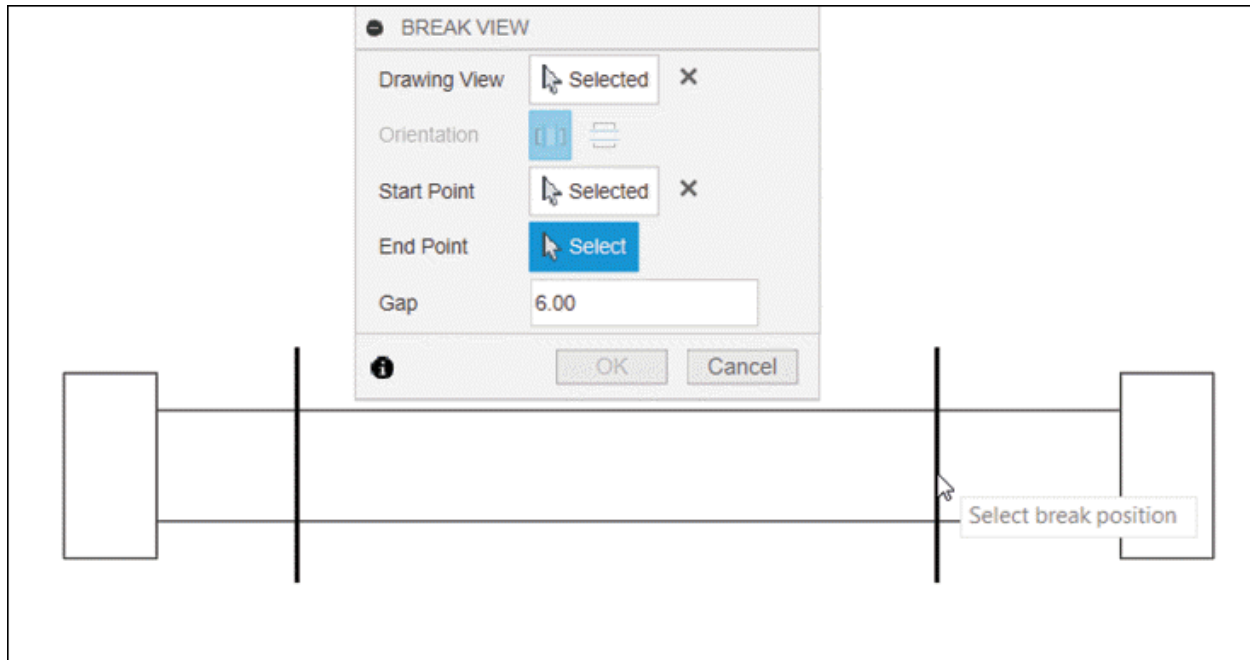


Figure-28. Selecting end point of break position

- Specify desired value in **Gap** edit box of the dialog box to specify distance for the gap between the two sections.
- After specifying parameters for break view, click on the **OK** button from the dialog box. The broken view will be created; refer to [Figure-29](#).

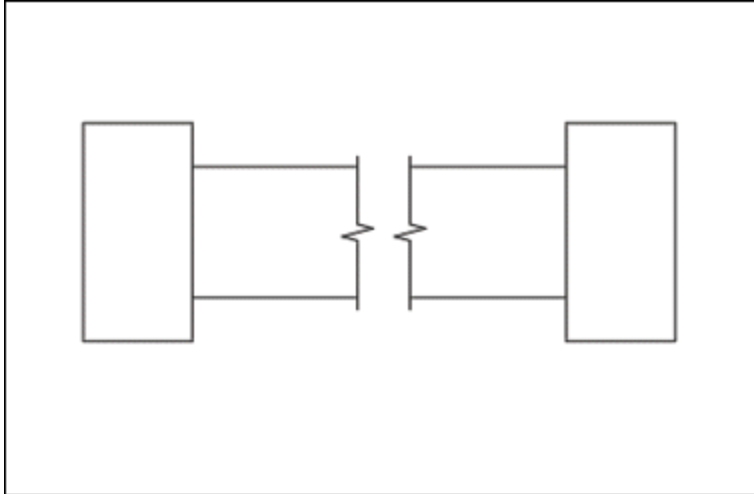


Figure-29. Broken view created

CREATING SKETCH

The **Create Sketch** tool is used to the sketch mode and create a new sketch on the current sheet. The procedure to use this tool is discussed next.

- Click on the **Create Sketch** tool from **CREATE** panel in the **Toolbar**; refer to [Figure-30](#). The **SKETCH** contextual tab will be displayed along with **SKETCH** dialog box; refer to [Figure-31](#).

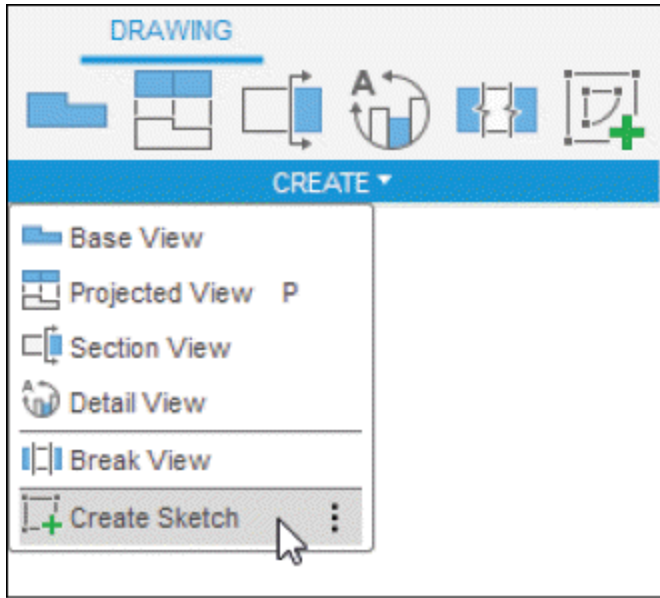


Figure-30. Create Sketch tool

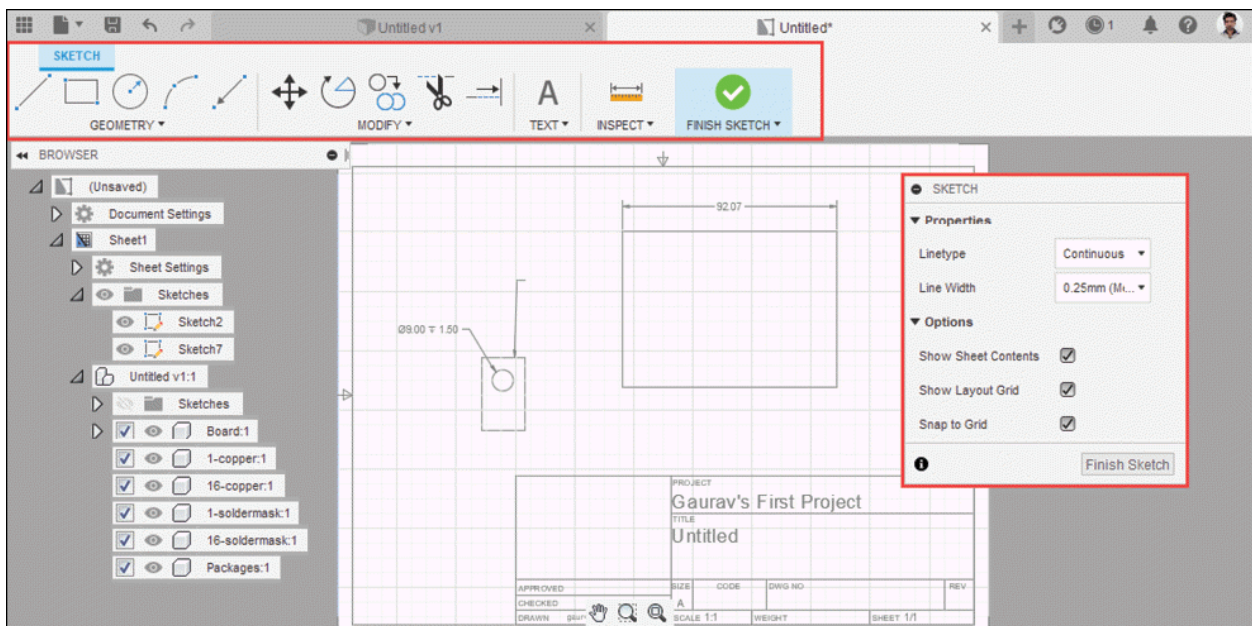


Figure-31. SKETCH contextual tab with SKETCH dialog box

- Select desired option from **Linetype** drop-down of **Properties** area in the dialog box to change the linetype of new or selected sketch geometry.
- Select desired option from **Line Width** drop-down of

Properties area in the dialog box to change the line width of new or selected sketch geometry.

- Select **Show Sheet Contents** check box from **Options** area to show the contents of the sheet as you edit the sketch.
- Select **Show Layout Grid** check box from **Options** area to show the layout grid in the canvas as you edit the sketch.
- Select **Snap to Grid** check box from **Options** area to snap to the layout grid in the canvas as you edit the sketch.
- After specifying parameters in the **SKETCH** dialog box, create desired sketch from **SKETCH** tab in the **Toolbar**.
- After creating the sketch, click on **Finish Sketch** button from **Toolbar** or from the dialog box to exit the tab.

MOVING VIEWS

The **Move** tool is used to move selected view from one place to another. The procedure to use this tool is discussed next.

- Click on the **Move** tool from **MODIFY** drop-down; refer to [Figure-32](#). The **MOVE** dialog box will be displayed; refer to [Figure-33](#).

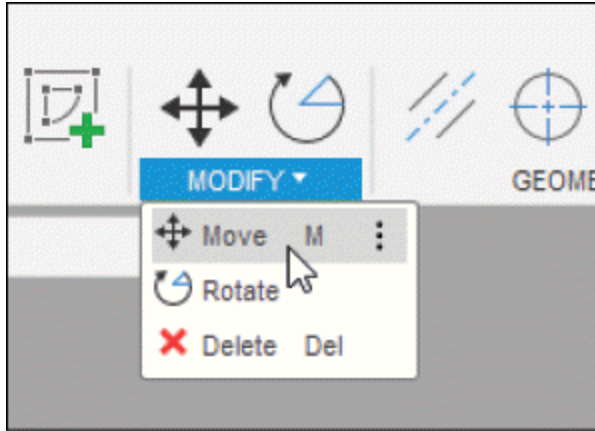


Figure-32. Move tool

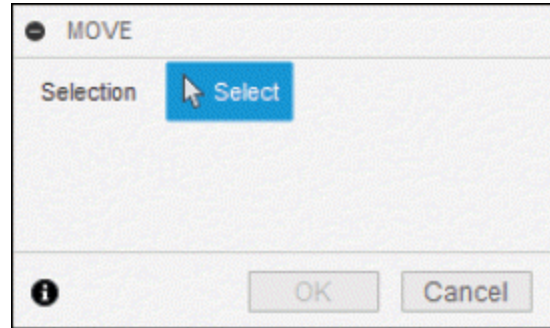


Figure-33. MOVE dialog box

- The **Select** button of **Selection** section in **MOVE** dialog box is active by default. Click on the view to select the view to be moved. The updated **MOVE** dialog box will be displayed; refer to [Figure-34](#).

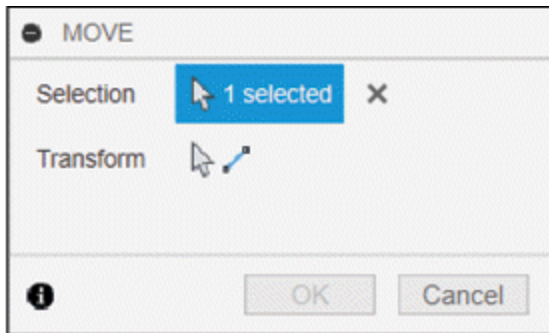


Figure-34. Updated MOVE dialog box

- Click on **Transform** button from **MOVE** dialog box and select a base point from the selected view. The selected view will get attached to the cursor to move the view from one place to another; refer to [Figure-35](#).

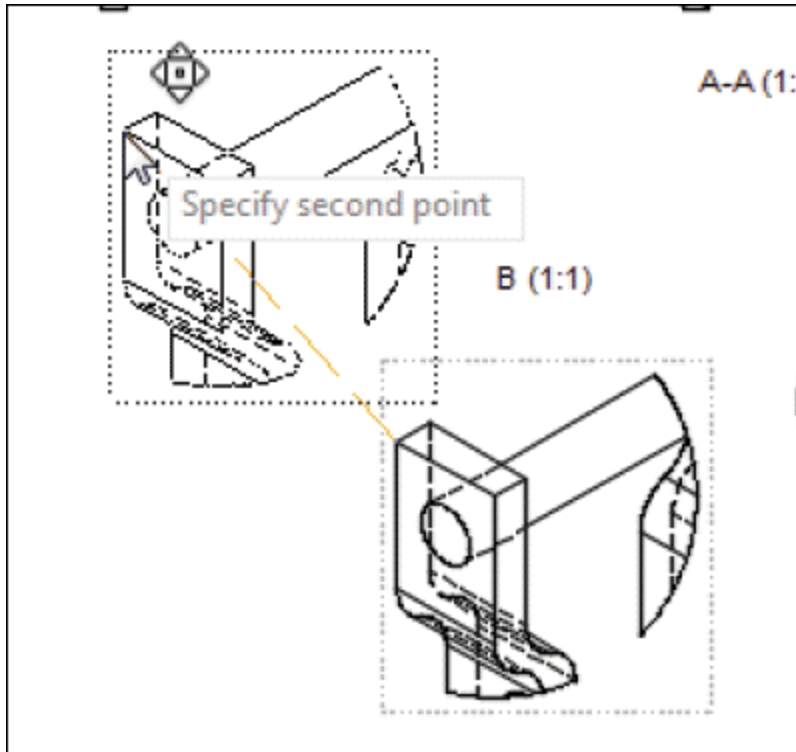


Figure-35. Specifying second point

- Click on the sheet to place the view. The view will be moved from previous position to new position.
- Click on the **OK** button from **MOVE** dialog box to complete the process.

ROTATING VIEWS

The **Rotate** tool is used to rotate selected view as desired. The procedure to use this tool is discussed next.

- Click on the **Rotate** tool from **MODIFY** panel; refer to [Figure-36](#). The **ROTATE** dialog box will be displayed; refer to [Figure-37](#).

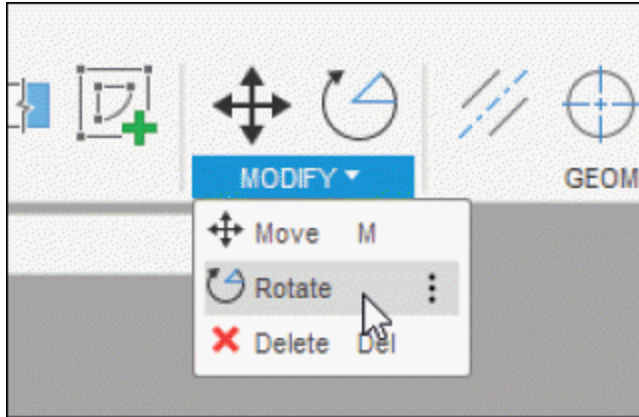


Figure-36. Rotate tool

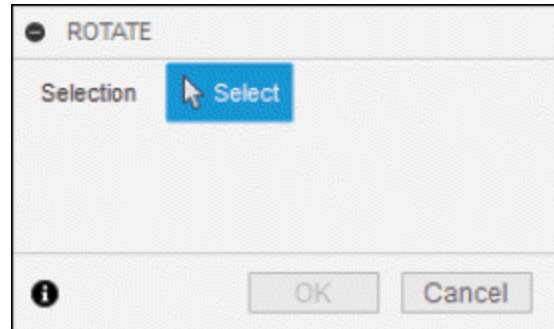


Figure-37. ROTATE dialog box

- The **Select** button of **Selection** section in **ROTATE** dialog box is active by default. Click on the view to be rotated.
- Click on **Transform** button from **ROTATE** dialog box and select the base point from selected view. The view will be attached to the cursor; refer to [Figure-38](#).

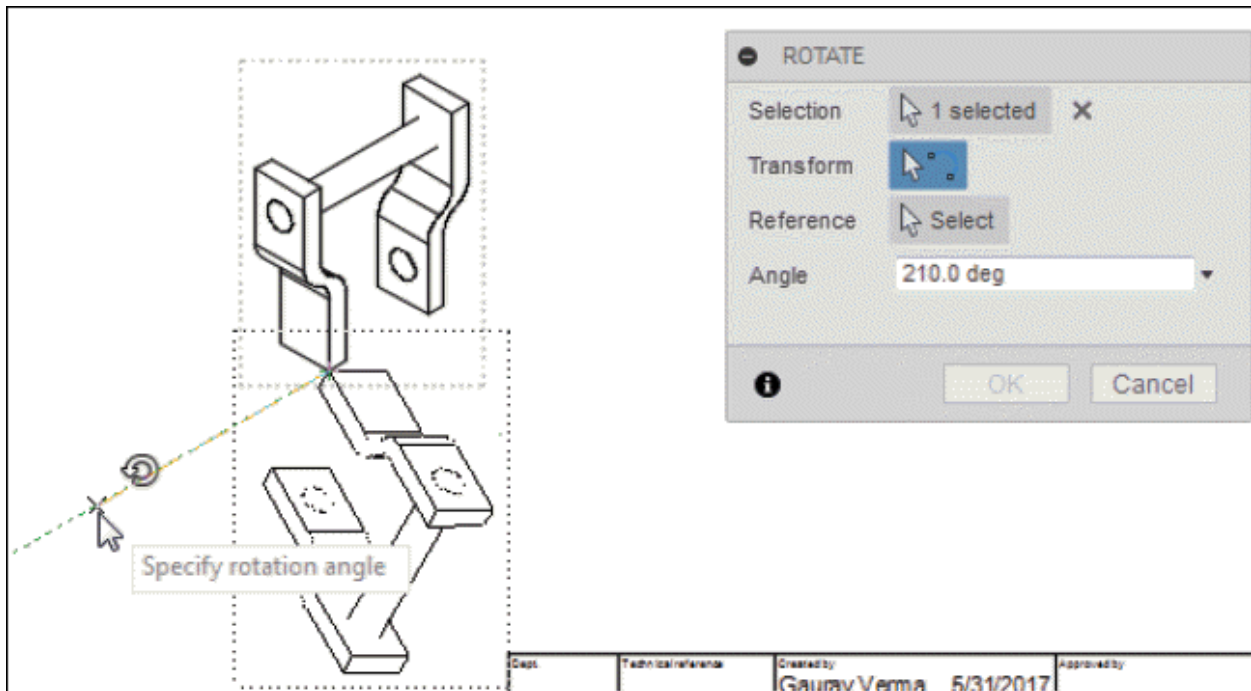


Figure-38. Rotating view

- You can adjust the angle of rotation of view by moving the cursor around parent view and place as required. If you want to specify the value of rotation then click in the **Angle** edit box of **ROTATE** dialog box and specify the value.
- You can also set the reference for rotation by clicking on **Select** button of **Reference** section and selecting the reference point from parent view.
- After specifying the parameters, click on **OK** button from **ROTATE** dialog box to complete the process.

CREATING CENTERLINE

The **Centerline** tool is used to create centerline. The procedure to use this tool is discussed next.

- Click on the **Centerline** tool from **GEOMETRY** panel; refer to [Figure-39](#). The tool will be activated.

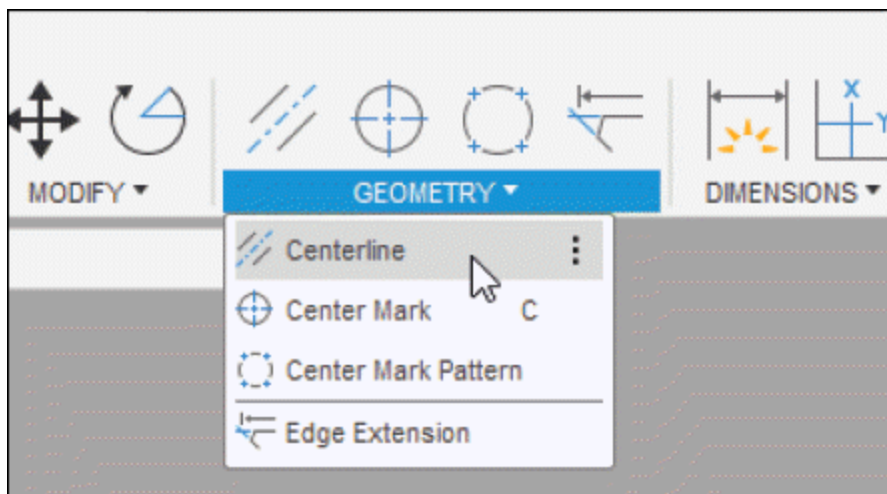


Figure-39. Centerline tool

- You need to select parallel edges from a view to create a

center line. Click on the desired edge to select; refer to [Figure-40](#). You will be asked to select the second edge.

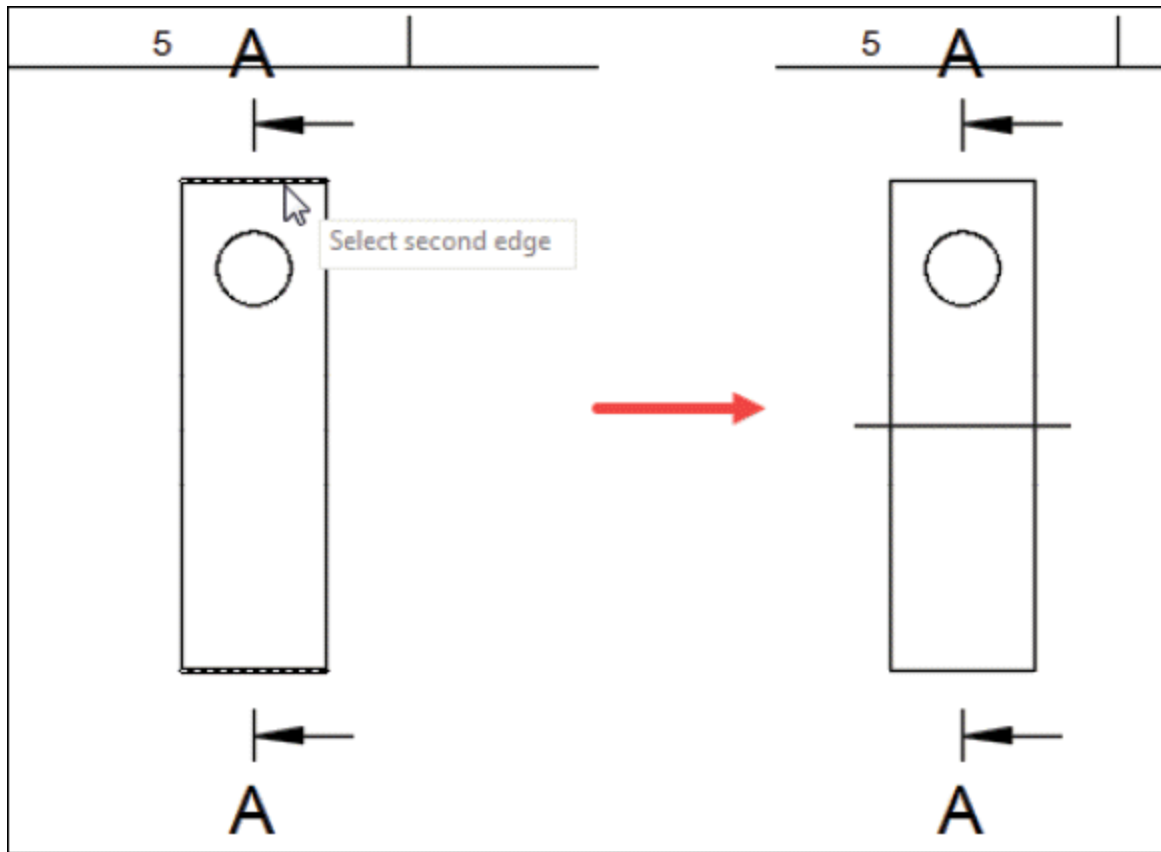


Figure-40. Creating centerline

- On selecting second edge, the center line will be created.

CREATING CENTER MARK

The **Center Mark** tool is used to create center mark for rounds and circles. The procedure to use this tool is discussed next.

- Click on the **Center Mark** tool from **GEOMETRY** panel; refer to [Figure-41](#). The tool will be activated. You need to select a round edge from any view.
- On selecting the edge, the center mark will be created for

selected round edge; refer to [Figure-42](#).

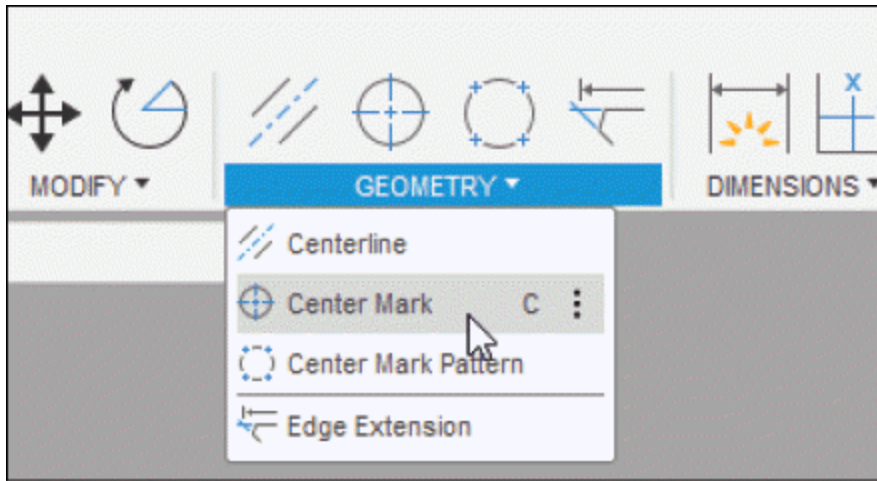


Figure-41. Center Mark tool

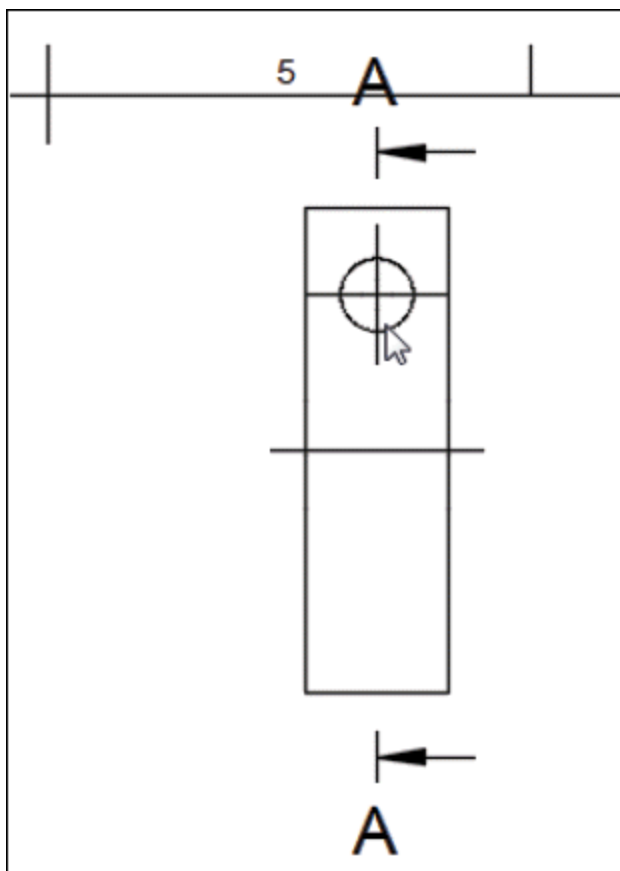


Figure-42. Creating center mark

CREATING CENTER MARK PATTERN

The **Center Mark Pattern** tool is used to generate pattern of center marks in the drawing view. The procedure to use this tool is given next.

- Click on the **Center Mark Pattern** tool from the **GEOMETRY** panel of **Toolbar**; refer to [Figure-43](#). The **CENTER MARK PATTERN** dialog box will be displayed; refer to [Figure-44](#). You will be asked to select a hole or round edge whose center mark is to be created.

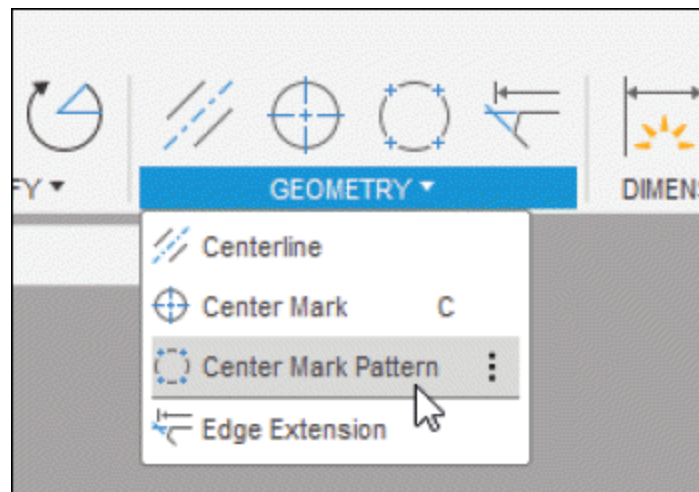


Figure-43. Center Mark Pattern tool

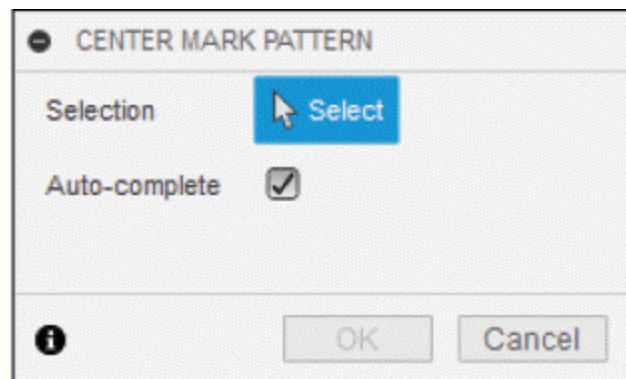


Figure-44. CENTER MARK PATTERN dialog box

- Click on desired holes one by one. If the holes are in

circular pattern then the dialog box will be displayed as shown in [Figure-45](#). Select the **Center Mark** check box to generate center mark for pitch circle. Click on the **OK** button to generate center marks.

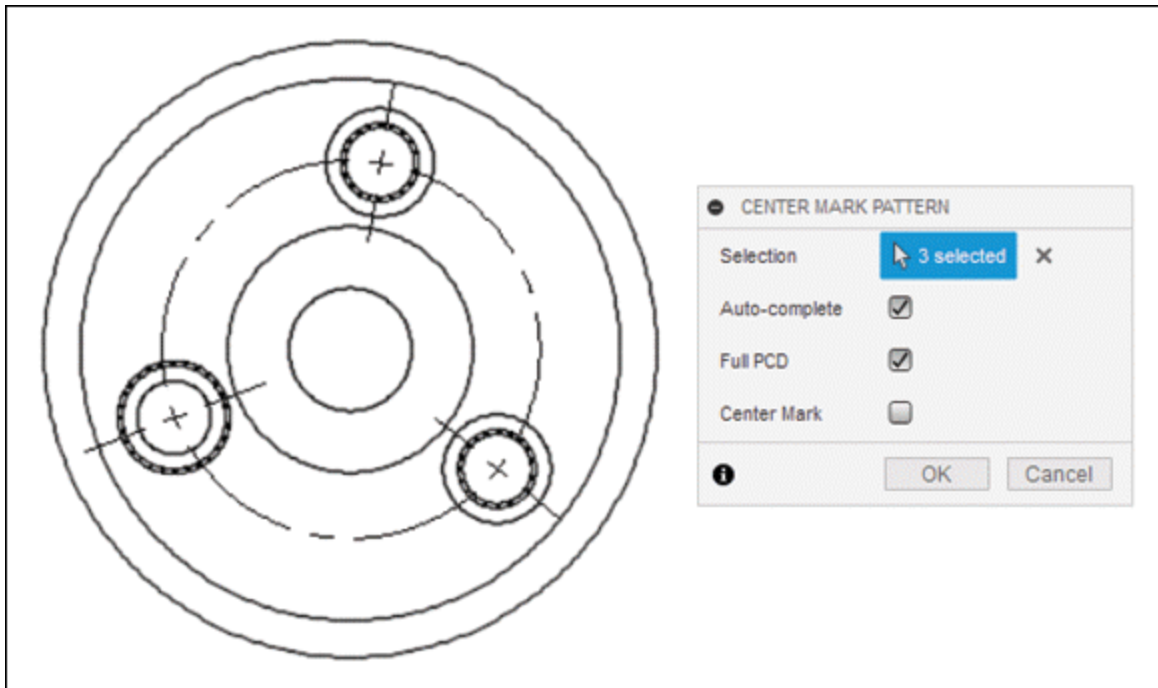


Figure-45. Holes selected in circular pattern

- In cases where you want to specify the PCD reference manually, click on the **Reference Edge** selection button after selecting holes and then select the reference edge; refer to [Figure-46](#).

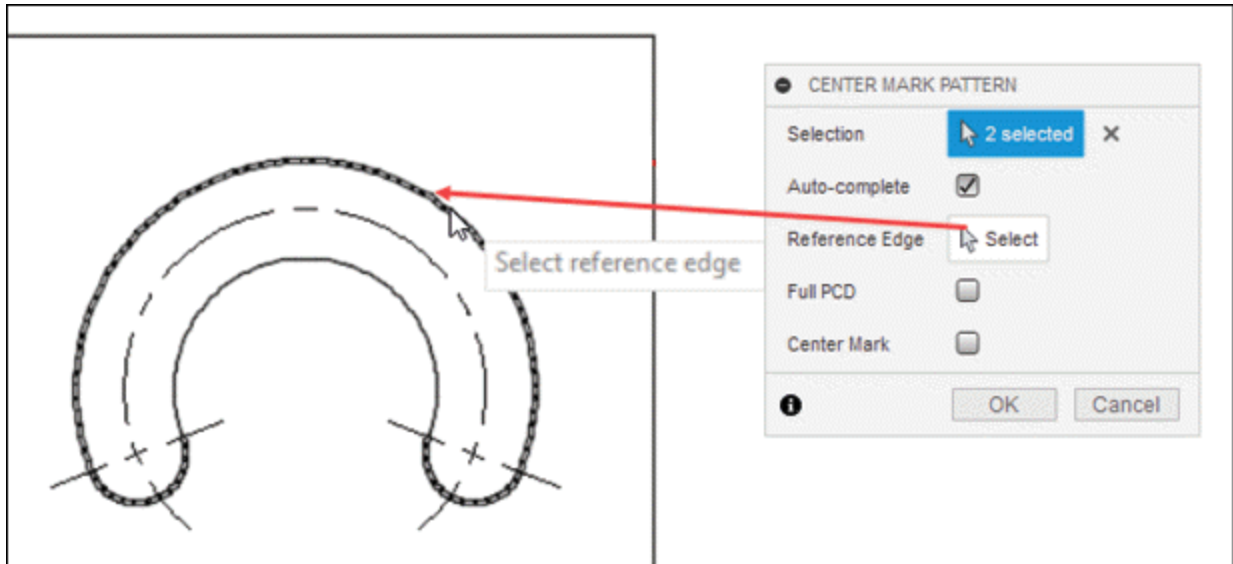


Figure-46. Selecting reference edge

- In case of selecting holes in linear direction, you can select maximum 3 holes for center mark.

EDGE EXTENSION

The **Edge Extension** tool is used to create an extension of associative edge for two intersecting (unequal) edges within an existing drawing view. The procedure to use this tool is discussed next.

- Click on the **Edge Extension** tool from **GEOMETRY** panel; refer to [Figure-47](#). The tool will be activated.

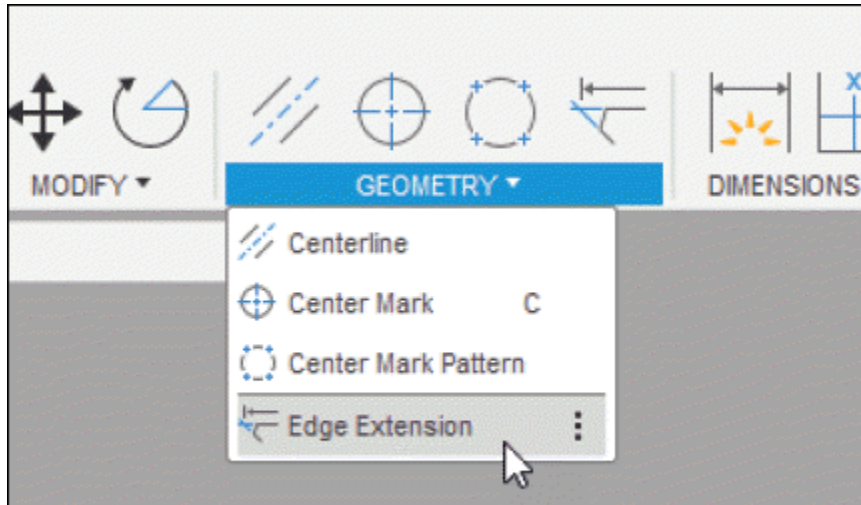


Figure-47. Edge Extension tool

- You need to select the two non-parallel (going to intersect) edges to create the extension. Click on the edge to select; refer to [Figure-48](#).

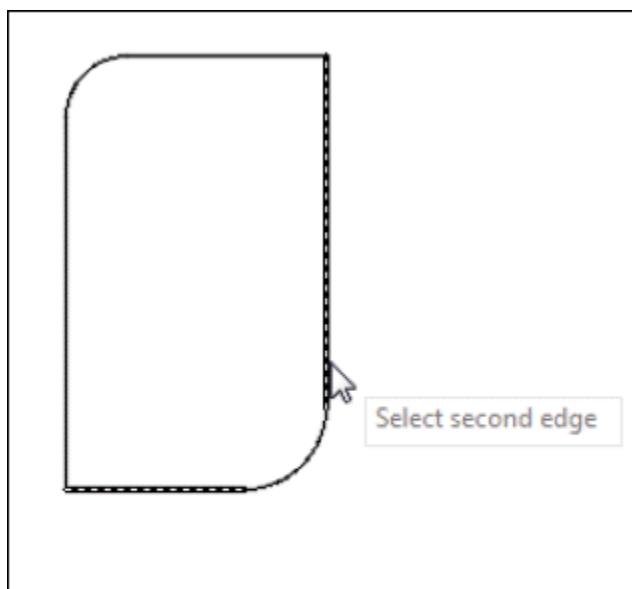


Figure-48. Selecting second edge

- On selecting the second edge, the extended edge will be displayed; refer to [Figure-49](#).

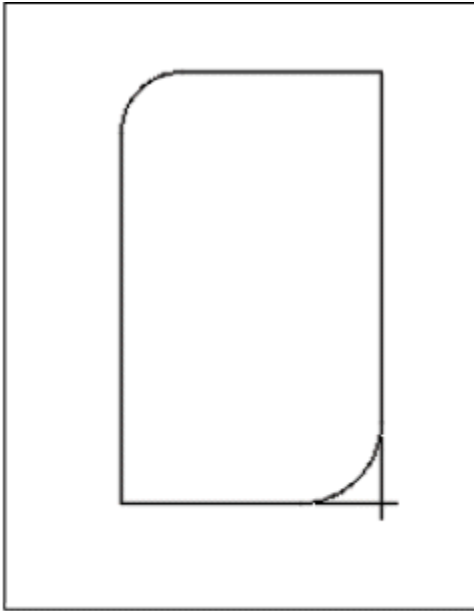


Figure-49. Edge extension created

DIMENSIONING

The **Dimension** tool is used to apply dimension to model. The procedure to use this tool is discussed next.

- Click on the **Dimension** tool from **DIMENSIONS** panel; refer to [Figure-50](#). The tool will be activated.

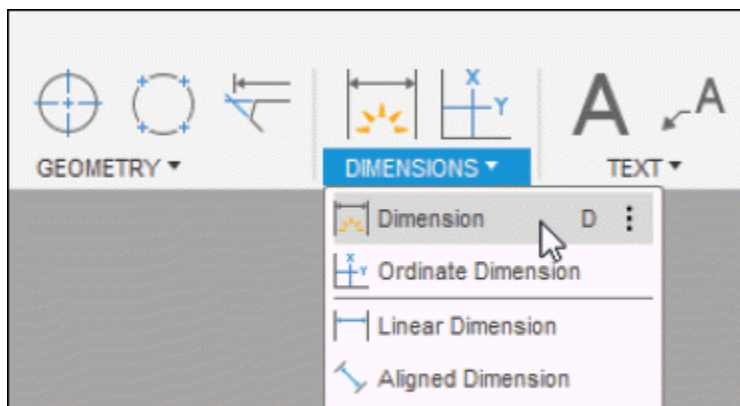


Figure-50. Dimension tool

- When you hover the cursor over on any geometry, the measurement of that dimension will be displayed; refer to

[Figure-51.](#)

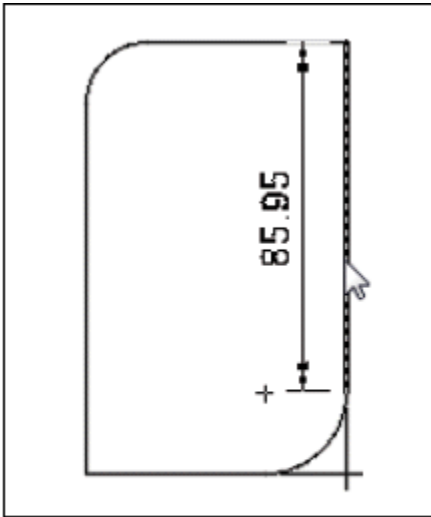


Figure-51. Checking dimension

- If you want to mention that dimension then click on the line. The dimension will be attached to the cursor.
- Place the dimension at desired distance by clicking.

Ordinate Dimensioning

The **Ordinate Dimension** tool is used to apply ordinate dimensioning. The procedure to use this tool is discussed next.

- Click on the **Ordinate Dimension** tool from **DIMENSIONS** panel; refer to [Figure-52](#). The tool will be activated and you will be asked to specify origin for ordinate dimensioning.

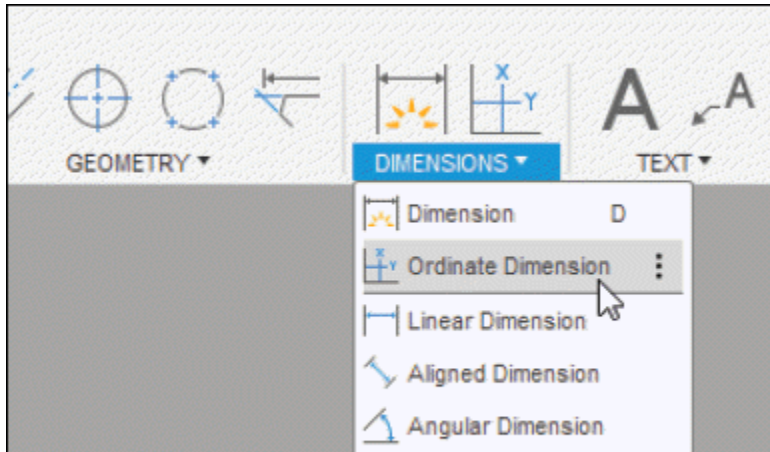


Figure-52. Ordinate Dimension tool

- Click at desired location to place origin of ordinate dimensioning. The ordinate dimension will be attached to the cursor; refer to [Figure-53](#).
- Click at desired location to place dimension text. Similarly, you can dimension horizontally/vertically.

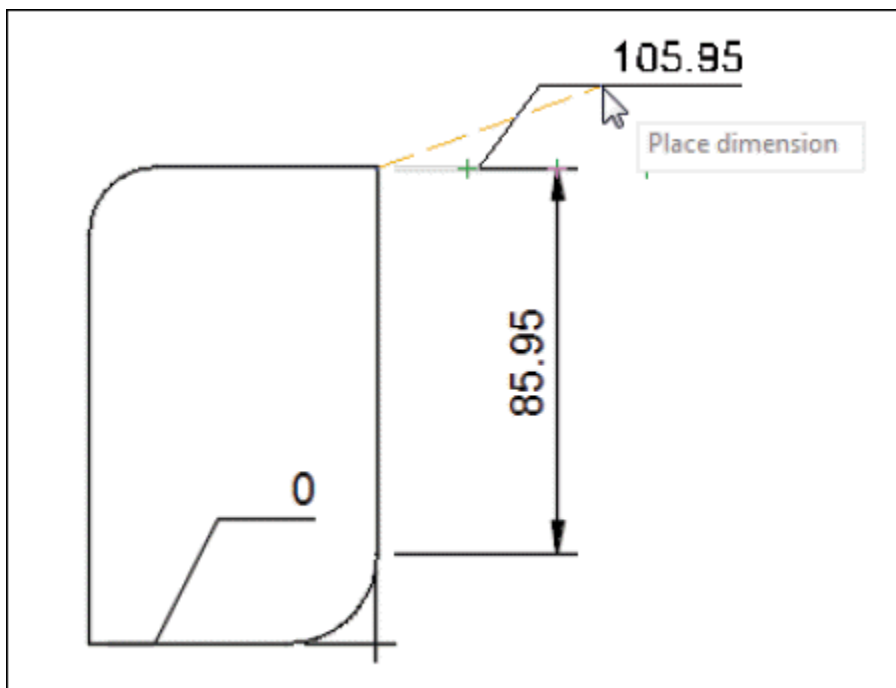


Figure-53. Calculating ordinate dimension

Creating Linear Dimension

The **Linear Dimension** tool is used to create linear dimensions in horizontal/vertical directions. The procedure to use this tool is given next.

- Click on the **Linear Dimension** tool from the **DIMENSIONS** panel in the **Toolbar**; refer to [Figure-54](#). You will be asked to select points.

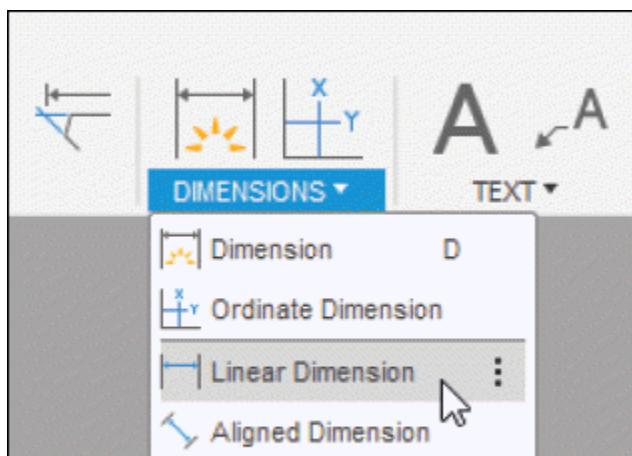


Figure-54. Linear Dimension tool

- Select the two points one by one to create horizontal/vertical distance dimension. The dimension will get attached to cursor.
- Move the cursor in desired direction to create vertical or horizontal dimension if the two points are not in aligned vertically/horizontally; refer to [Figure-55](#).

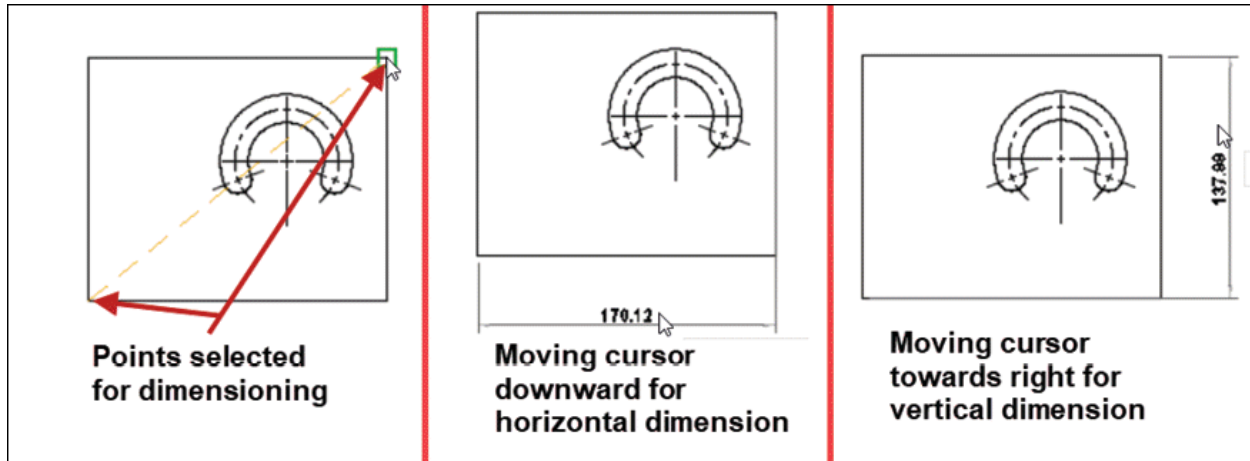


Figure-55. Creating linear dimension

- Click at desired location to place the dimension.

Creating Aligned Dimension

The **Aligned Dimension** tool is used to create aligned dimension between two selected points. The procedure is given next.

- Click on the **Aligned Dimension** tool from **DIMENSIONS** panel in the **Toolbar**; refer to [Figure-56](#). You will be asked to select points.

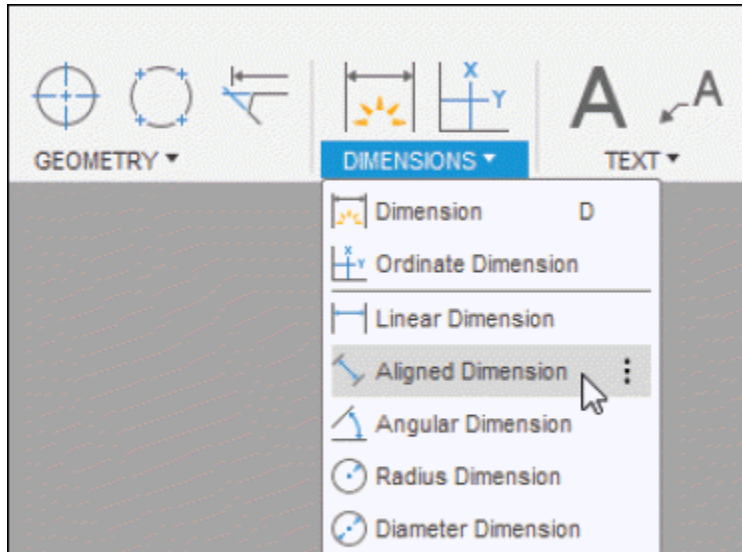


Figure-56. Aligned Dimension tool

- Click at desired two points. The distance dimension between two selected points will get attached to cursor; refer to [Figure-57](#).

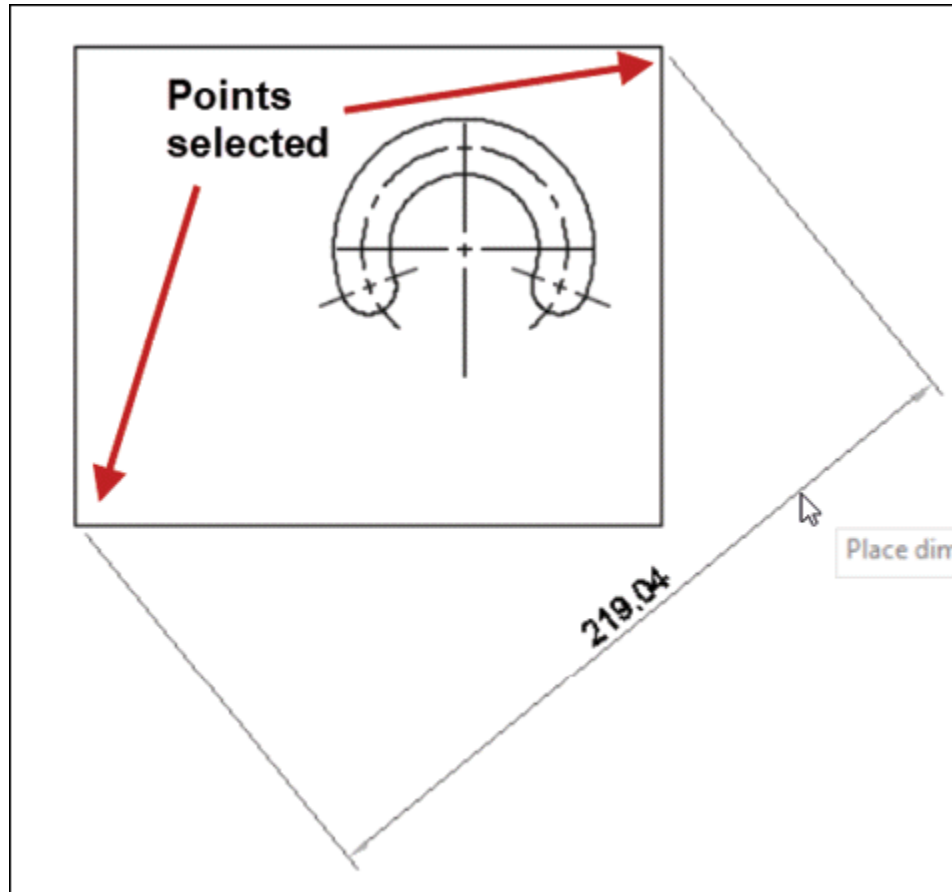


Figure-57. Placing Aligned dimension

- Click at desired location to place the dimension.

Creating Angular Dimension

The **Angular Dimension** tool is used to create angle dimension between two selected lines/edges. If you select an arc then its angular span is created as dimension. The procedure to use this tool is given next.

- Click on the **Angular Dimension** tool from the **DIMENSIONS** panel in the **Toolbar**; refer to [Figure-58](#). You will be asked to select entities.

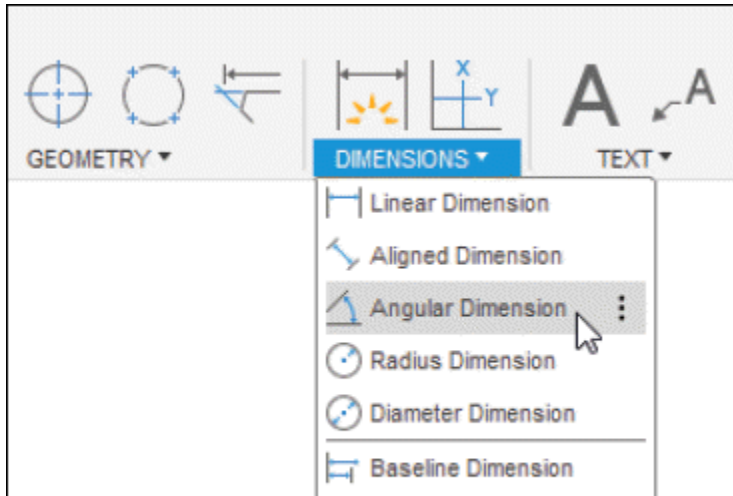


Figure-58. Angular Dimension tool

- Select two lines between which you want to create angular dimension or select an arc whose angular dimension is to be created. The dimension will get attached to cursor; refer to [Figure-59](#).

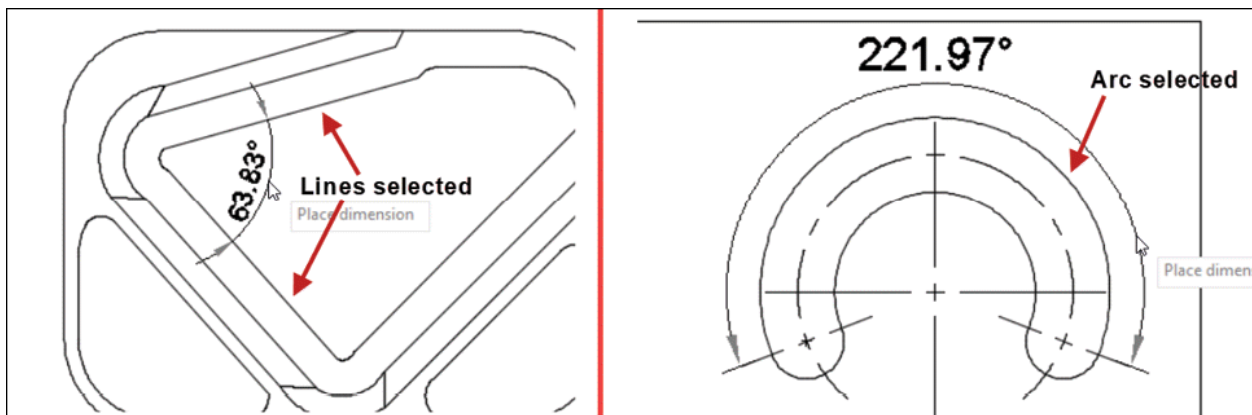


Figure-59. Creating angular dimension

- Click at desired location to place the dimension.

Creating Radius Dimension

The **Radius Dimension** tool is used to apply radius dimension to selected arc, circle, and circular edges. The procedure to use this tool is given next.

- Click on the **Radius Dimension** tool in **DIMENSIONS** panel of **Toolbar**; refer to [Figure-60](#). You will be asked to select circular edge or circle in the drawing.

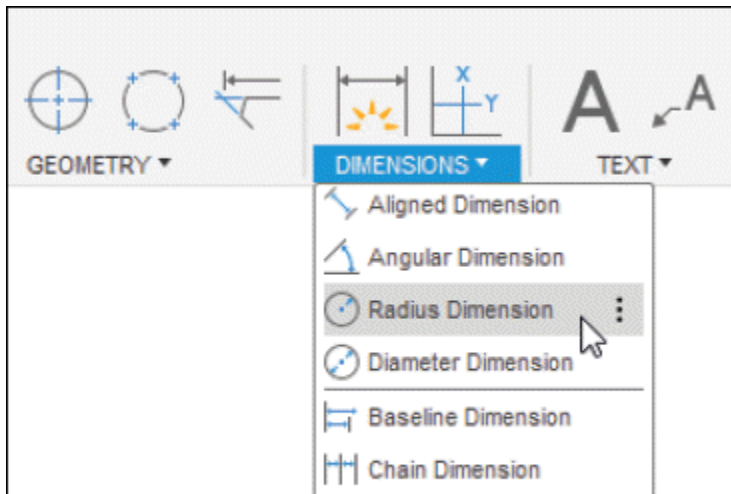


Figure-60. Radius Dimension tool

- Click at desired entity. The radius dimension will get attached to cursor; refer to [Figure-61](#).

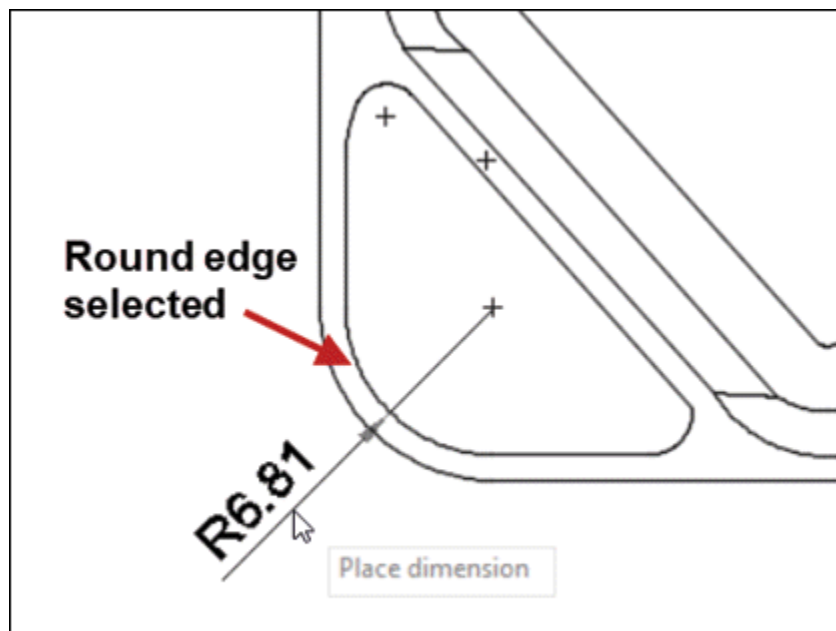


Figure-61. Creating radius dimension

- Click at desired location to place the dimension.

Creating Diameter Dimension

The **Diameter Dimension** tool is used to create diameter dimension for selected round edges, circles, and fillets in the drawing; refer to [Figure-62](#). The procedure to use this tool is similar to **Radius Dimension** tool.

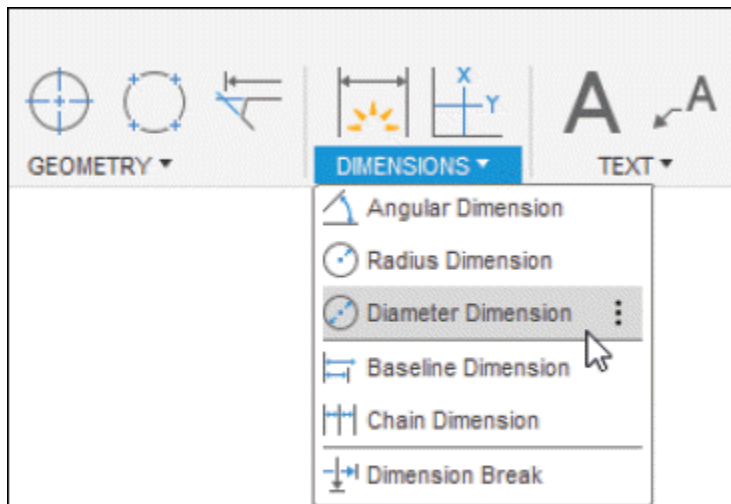


Figure-62. Diameter Dimension tool

Baseline Dimensioning

The baseline dimensioning is performed when you want to create linear dimensions with respect to one common base reference. The procedure to create baseline dimensions is given next.

- Click on the **Linear Dimension** tool from the **DIMENSIONS** panel of **Toolbar**; refer to [Figure-63](#), and create a linear dimension which will be used as base line; refer to [Figure-64](#).

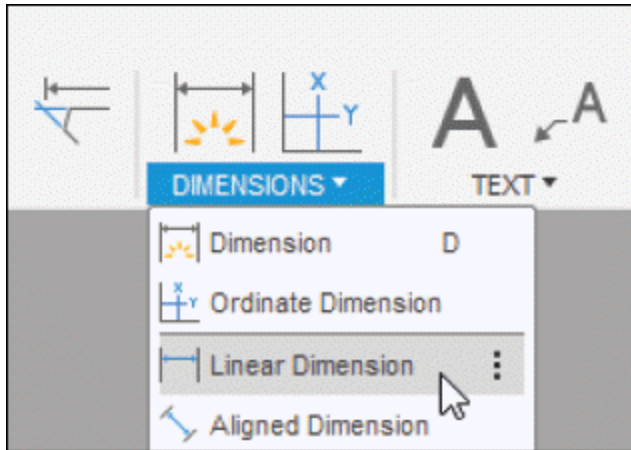


Figure-63. Linear Dimension tool

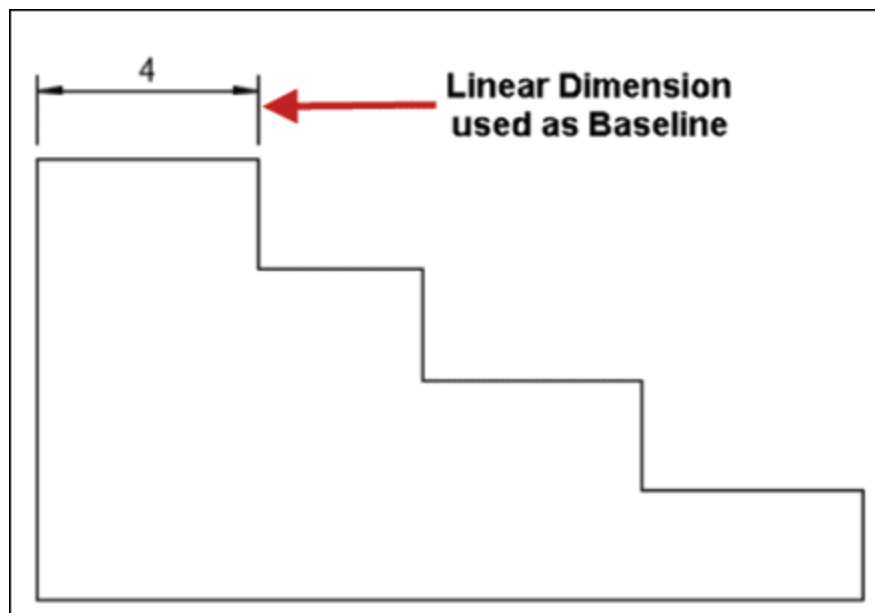


Figure-64. Linear dimension used as baseline

- Click on the **Baseline Dimension** tool from the **DIMENSIONS** panel of the **Toolbar**; refer to [Figure-65](#), and select one of the leg of base dimension.

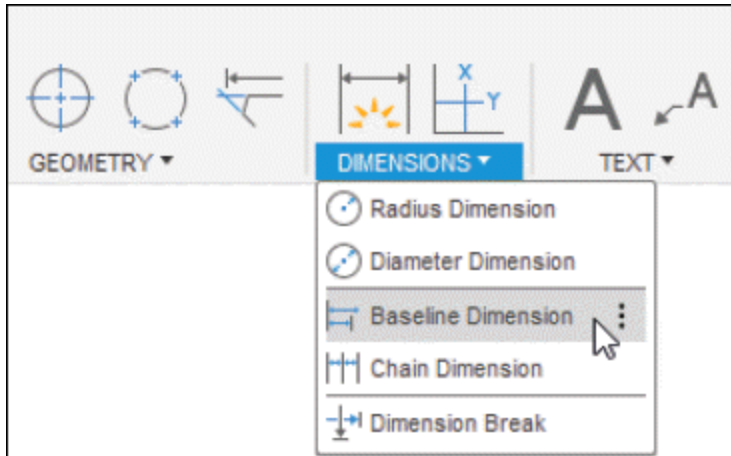


Figure-65. Baseline Dimension tool

- A new dimension will be generated with one end point fixed to selected leg and other end point of dimension attached to cursor; refer to [Figure-66](#).

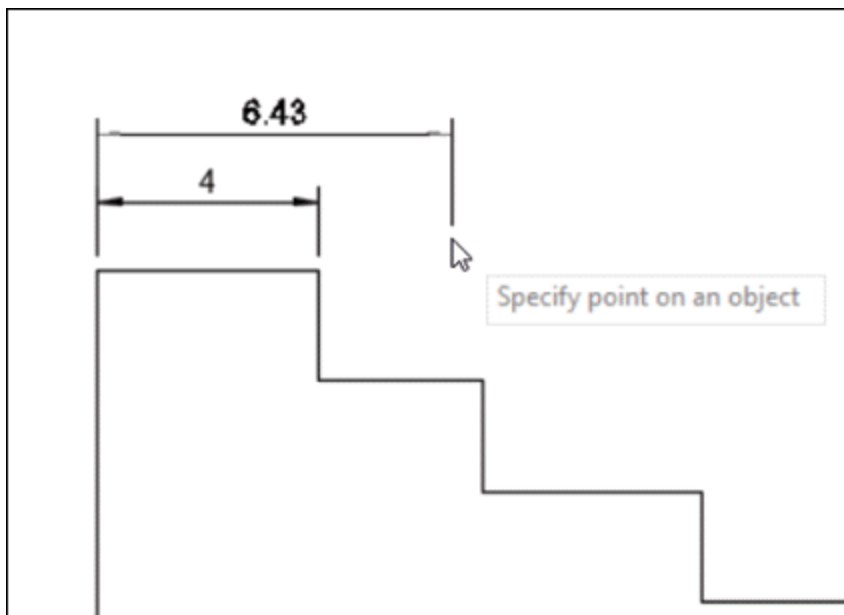


Figure-66. Placing baseline dimension

- Select desired point of sketch to create the dimension. You will be asked to specify end point for another dimension. Select desired points to create required dimensions. Once

you have created required dimensions, press **ENTER** from keyboard to complete the process. The dimensions will be generated; refer to [Figure-67](#).

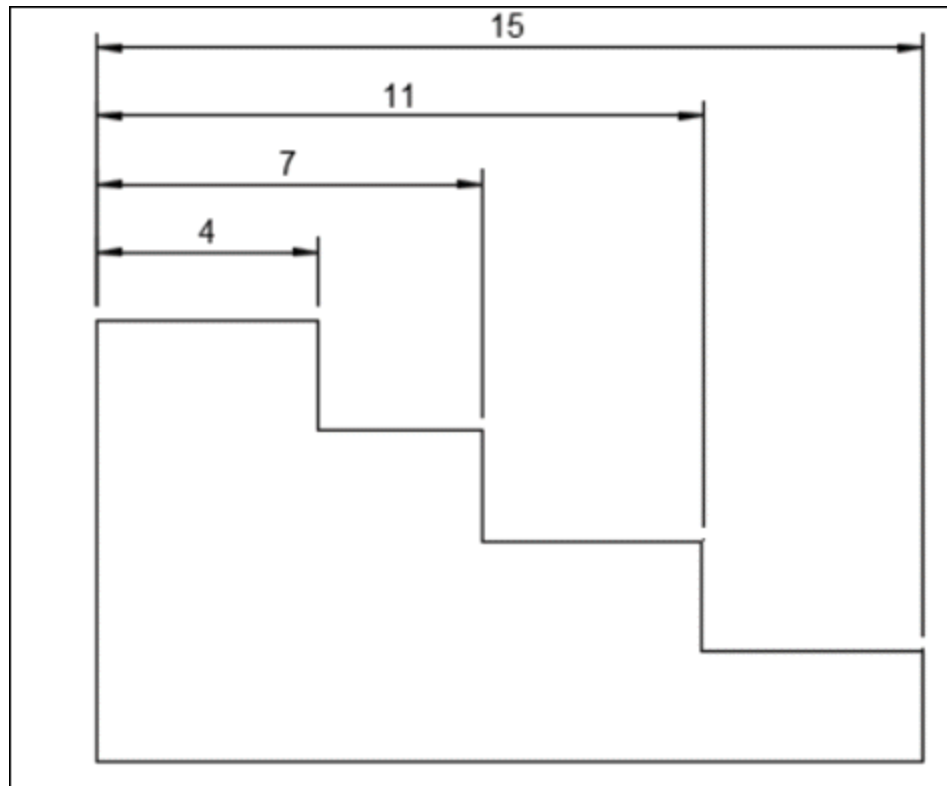


Figure-67. Baseline dimension created

Chain Dimensioning

In chain dimensioning, all the linear dimensions are placed side by side. The procedure to use this tool is given next.

- Create a linear dimension to used as reference for chain dimensioning as discussed in previous section for baseline dimensioning.
- Click on the **Chain Dimension** tool from the **DIMENSIONS** panel of the **Toolbar**; refer to [Figure-68](#), and select the base dimension. You will be asked to specify

end point of new dimension; refer to [Figure-69](#).

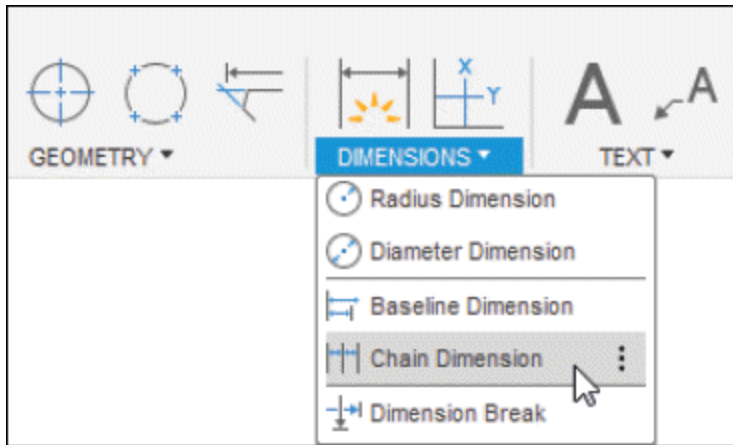


Figure-68. Chain Dimension tool

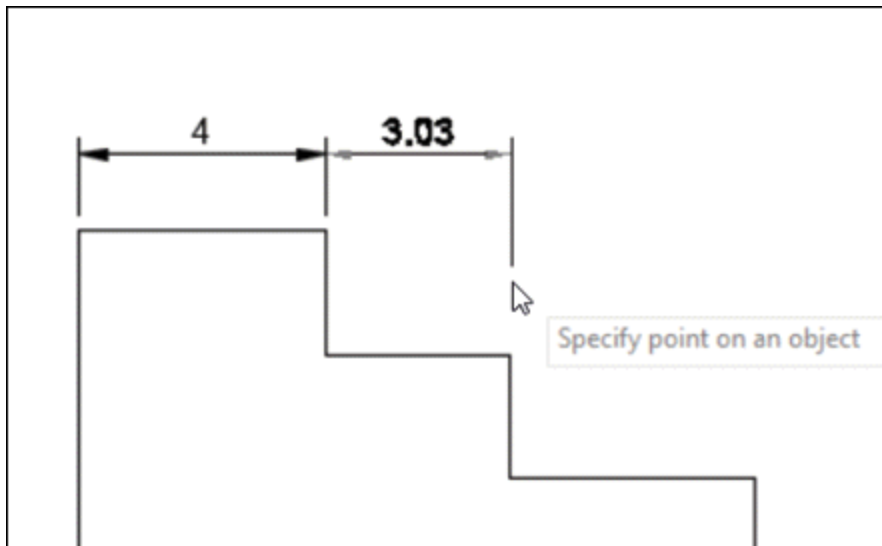


Figure-69. Placing chain dimensions

- Click at desired location to specify end point. You will be asked to specify end point of another dimension.
- Click at desired locations to create required dimensions and then press **ENTER** to exit. The dimensions will be created; refer to [Figure-70](#).

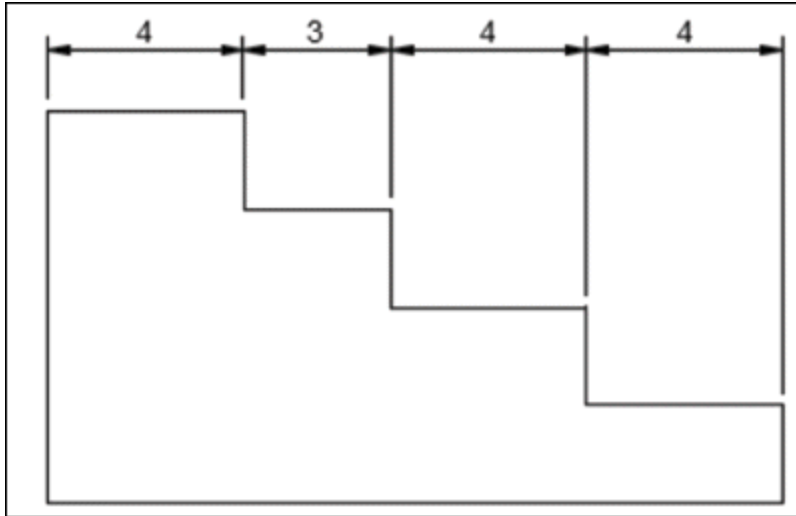


Figure-70. Chain dimensions created

Arrange Dimensions

The **Arrange Dimension** tool is used to stack or align linear or angular dimensions in the **DRAWING** workspace. The procedure to use this tool is discussed next.

- Click on the **Arrange Dimensions** tool from **DIMENSIONS** panel of the **Toolbar**; refer to [Figure-71](#). The **ARRANGE DIMENSIONS** dialog box will be displayed; refer to [Figure-72](#).

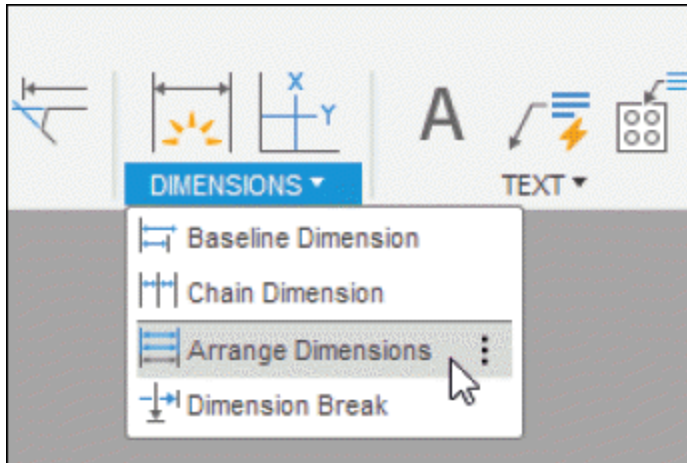


Figure-71. Arrange Dimensions tool

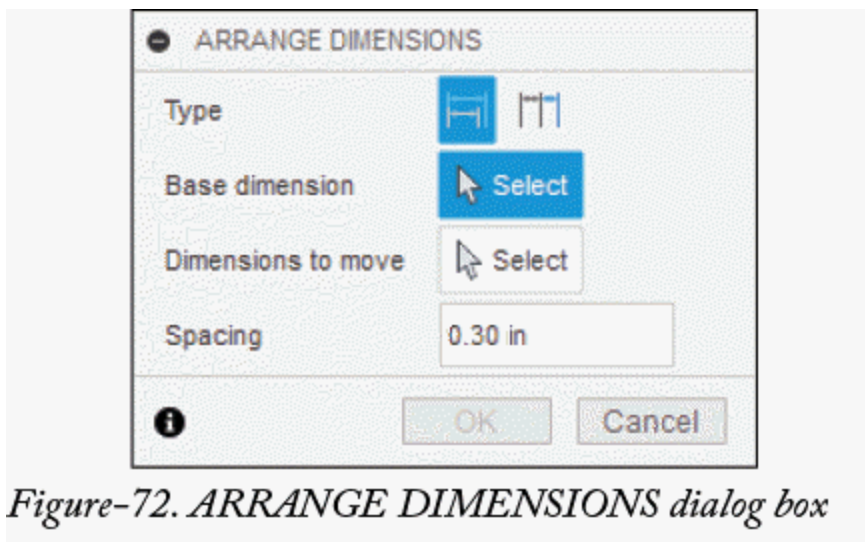


Figure-72. ARRANGE DIMENSIONS dialog box

- Select **Stack** or **Align** option from **Type** section of the dialog box to create the dimension.
- The **Base dimension** selection option is active by default. Select the base dimension that all other dimensions will be stacked above/below; refer to [Figure-73](#).

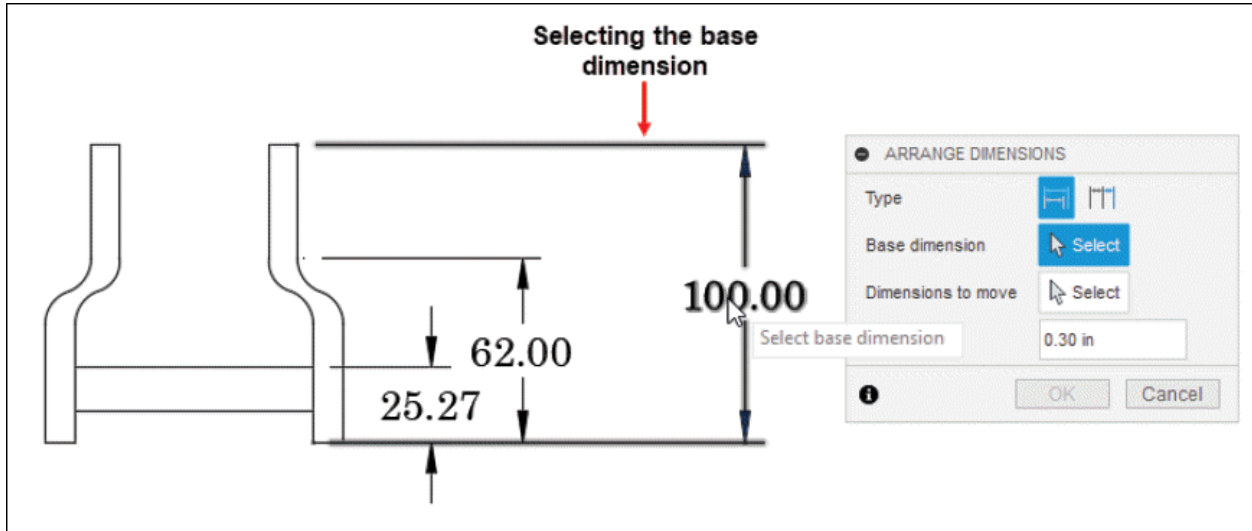


Figure-73. Selecting the base dimension

- After selecting the base dimension, select the other dimensions that you want to stack; refer to [Figure-74](#).

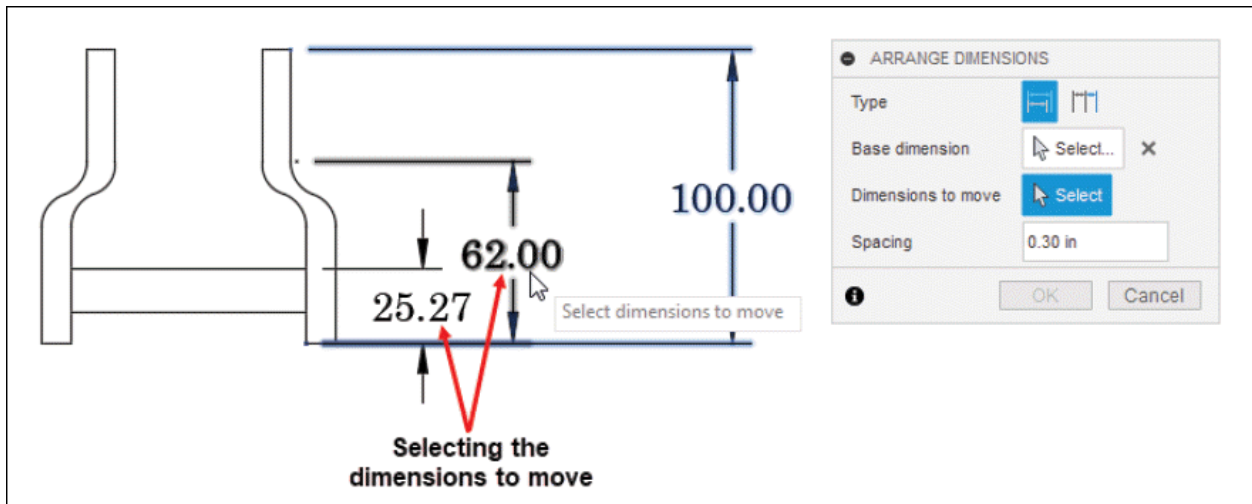


Figure-74. Selecting the dimensions to move

- Specify desired spacing between stacked dimensions in the **Spacing** edit box of the dialog box.
- After specifying desired parameters, click on the **OK** button from the dialog box. The dimensions will be arranged; refer to [Figure-75](#).

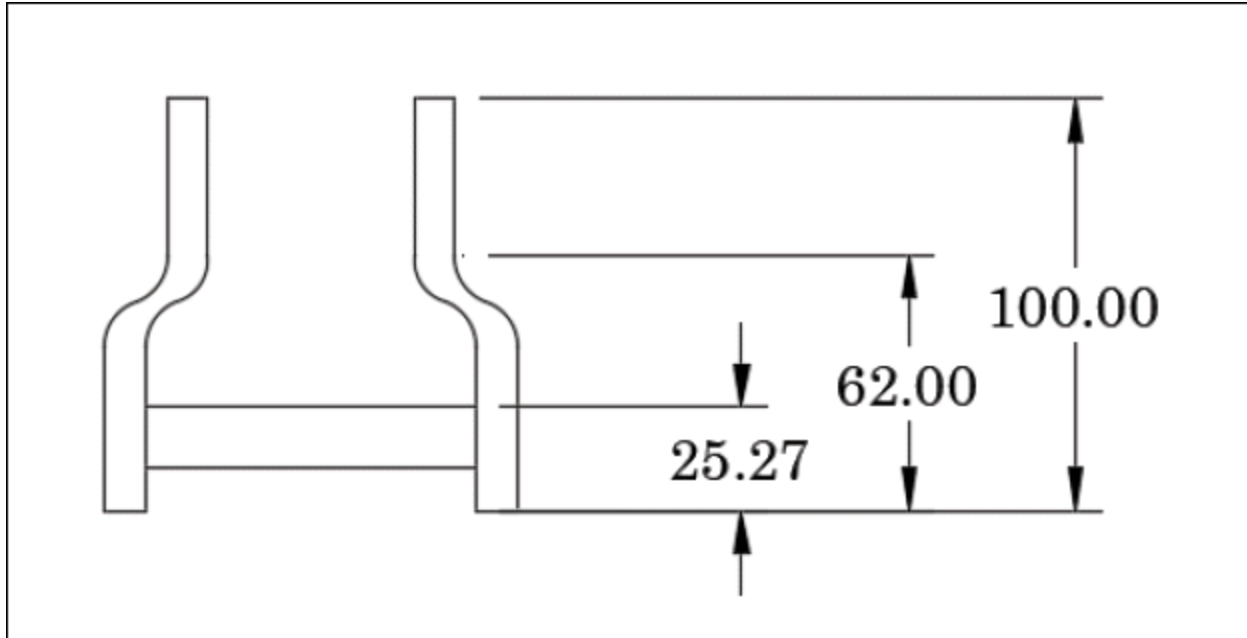


Figure-75. Dimensions arranged

Dimension Break

The **Dimension Break** tool is used to resolve intersecting dimensions in the drawing. This tool adds/removes break in selected dimension. The procedure to use this tool is given next.

- Click on the **Dimension Break** tool from the **DIMENSIONS** panel of the **Toolbar**; refer to [Figure-76](#). The **DIMENSION BREAK** dialog box will be displayed; refer to [Figure-77](#), and you will be asked to select the dimension to add or remove break.

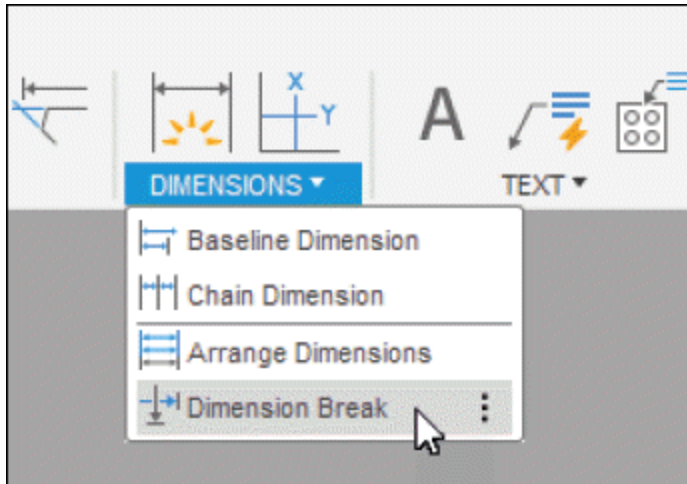


Figure-76. Dimension Break tool

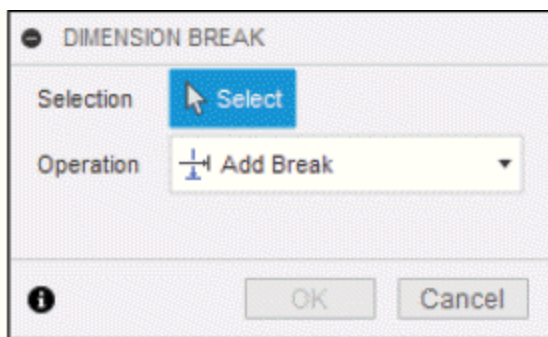


Figure-77. DIMENSION BREAK dialog box

- Select desired dimension and then select the **Add Break** or **Remove Break** option from the **Operation** drop-down to perform respective action. In our case, we are selecting **Add Break** option from the drop-down to add break in intersecting dimension.
- Click on the **OK** button from the dialog box. The selected operation will be performed on the dimension; refer to [Figure-78](#).

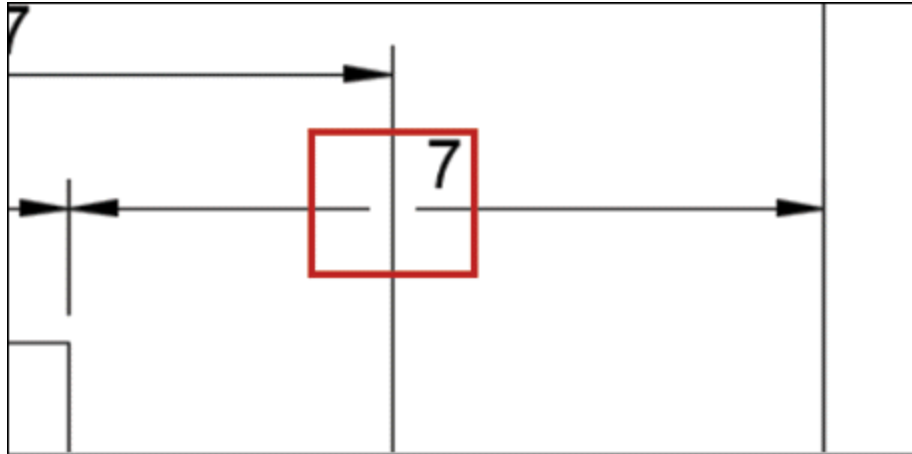


Figure-78. Dimension after applying Add Break

EDITING DIMENSIONS

Once you have generated dimensions, there are some cases where you need to provide special symbols with dimensions or you want to modify the value of dimension. The procedure to edit dimension is given next.

- Double-click on the dimension in drawing that you want to edit. The **DIMENSION** dialog box will be displayed; refer to [Figure-79](#).

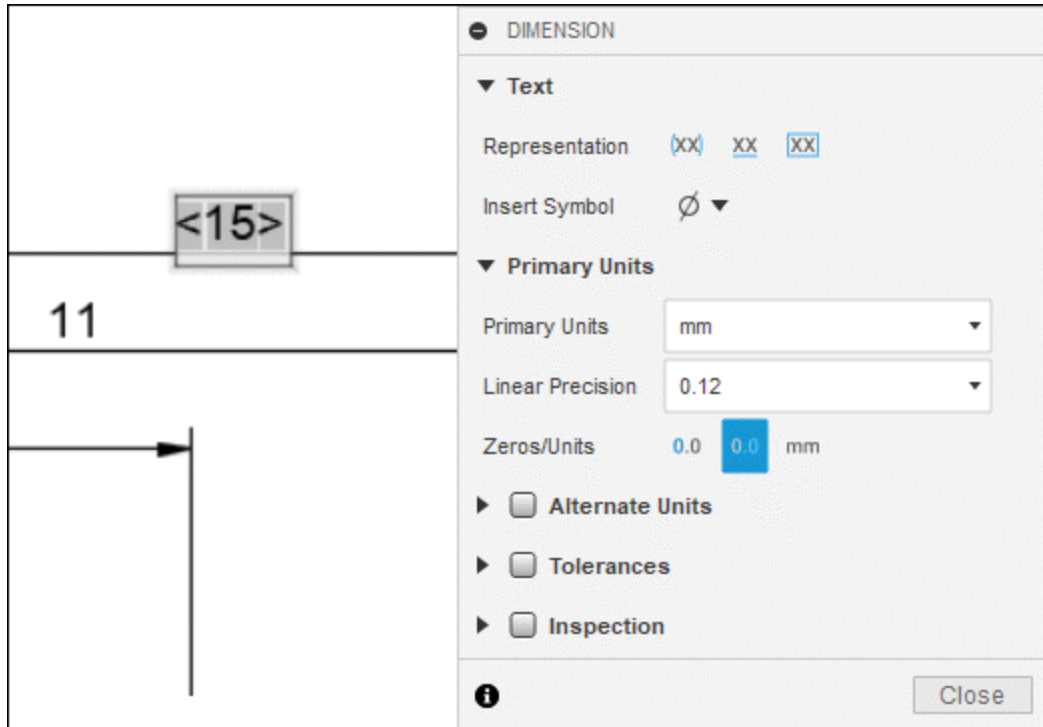


Figure-79. DIMENSION dialog box

- Select desired button of Representation section from Text node to make the dimension as reference dimension, not to scale, and theoretically exact; refer to [Figure-80](#).

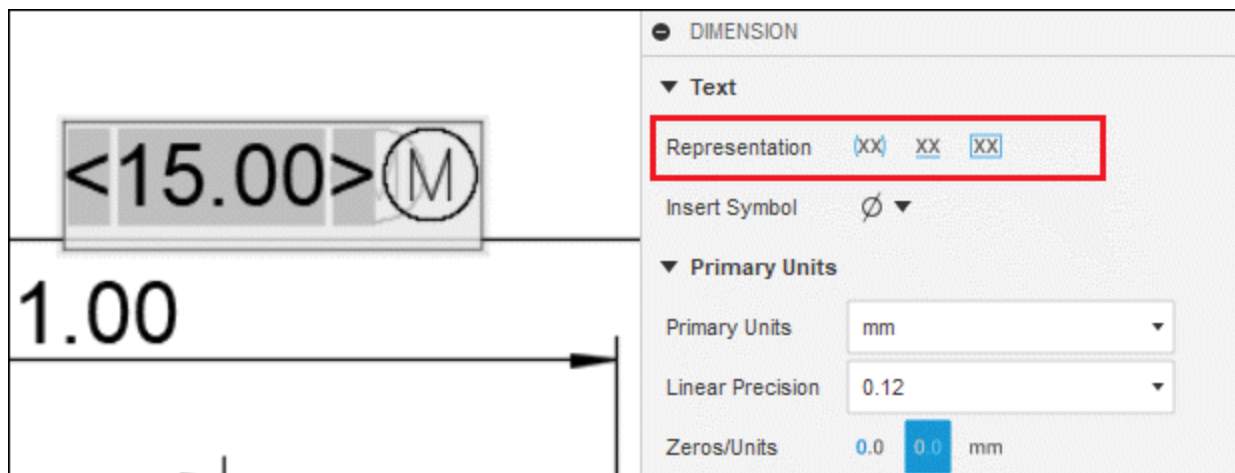


Figure-80. Representation options

- Click at desired place in dimension box and select desired

symbol to be inserted. The symbol will be added to dimension; refer to [Figure-81](#).



Figure-81. Symbol added to dimension

- Select desired unit from **Primary Units** drop-down in the **Primary Units** node.
- Select desired precision level from **Linear Precision** drop-down.
- Select desired option from **Zeros/Units** section to add/remove zeros and units to/from the dimension.
- Select the **Alternate Units** check box to also display dimensions in inches if they are in mm and vice-versa; refer to [Figure-82](#).

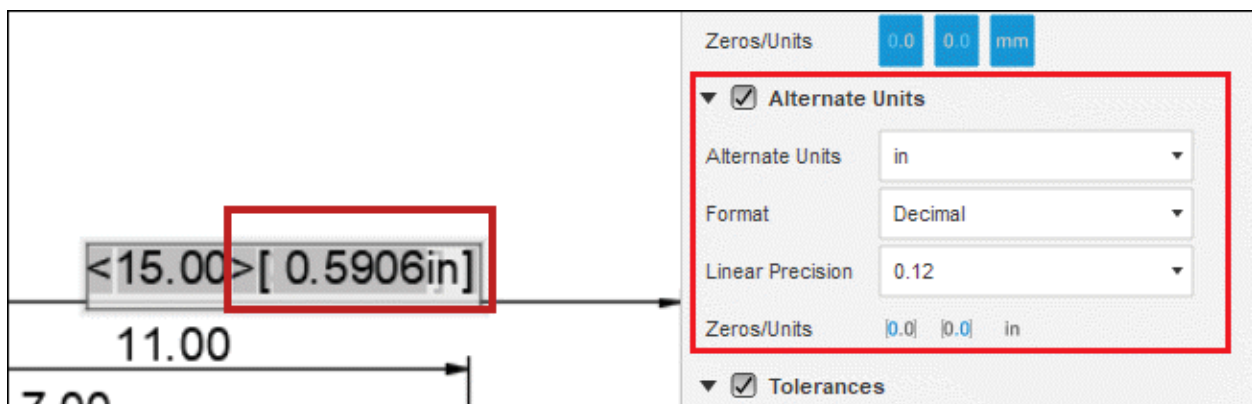


Figure-82. Alternate units

- Select the **Tolerances** check box and select desired type of

tolerance from the **Type** drop-down to apply tolerance to dimension. Select the **Symmetrical** option from the **Type** drop-down if tolerance is equal in both negative and positive directions. Select the **Deviation** option from the **Type** drop-down if tolerance is different in positive and negative directions. Select the **Limit** option from the **Type** drop-down if you want to display upper and lower limit of dimension; refer to [Figure-83](#).

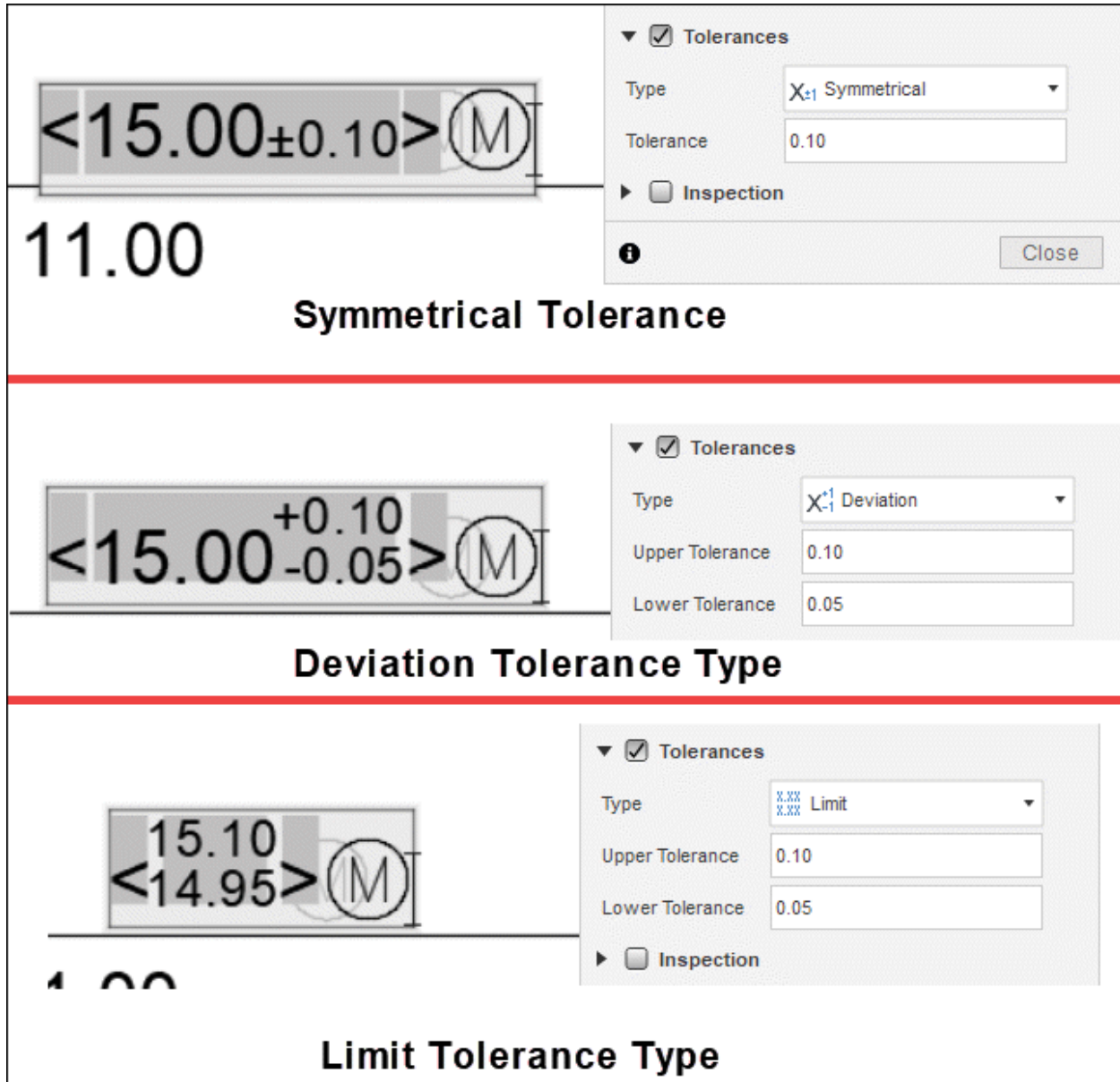


Figure-83. Tolerance types

- After selecting desired option from drop-down, specify desired value of tolerance in edit boxes below it in **Tolerances** node.
- Similarly, select the **Inspection** check box and set desired parameters if you want to define inspection rate of current dimension.

- After specifying parameters, click on the **Close** button or click anywhere in blank space of drawing to exit the tool.

CREATING TEXT

The **Text** tool is used to insert text into the active drawing. The procedure to use this tool is discussed next.

- Click on the **Text** tool from **TEXT** panel; refer to [Figure-84](#). The **Text** tool is activated and you will be asked to draw a text box.

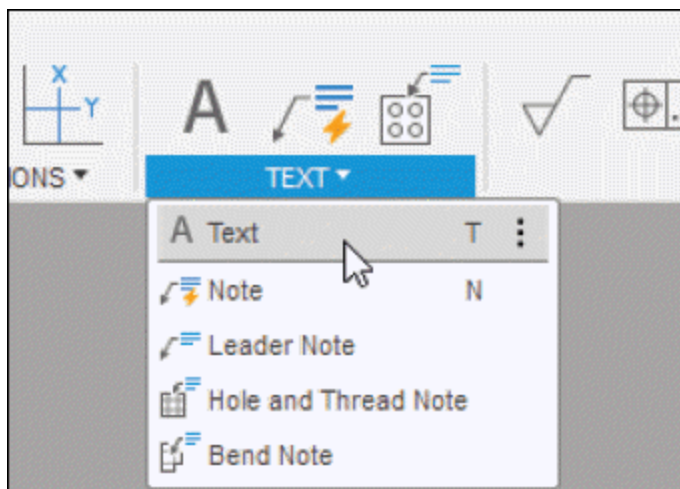


Figure-84. Text tool

- Click to specify the first corner.
- After specifying the first corner, click on the screen to specify the second corner; refer to [Figure-85](#). The **TEXT** dialog box will be displayed; refer to [Figure-86](#).

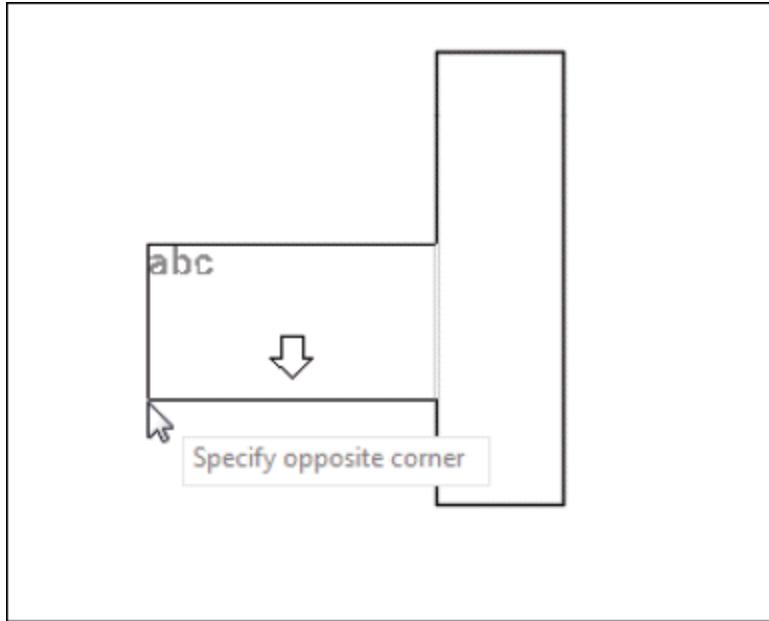


Figure-85. Specifying second corner

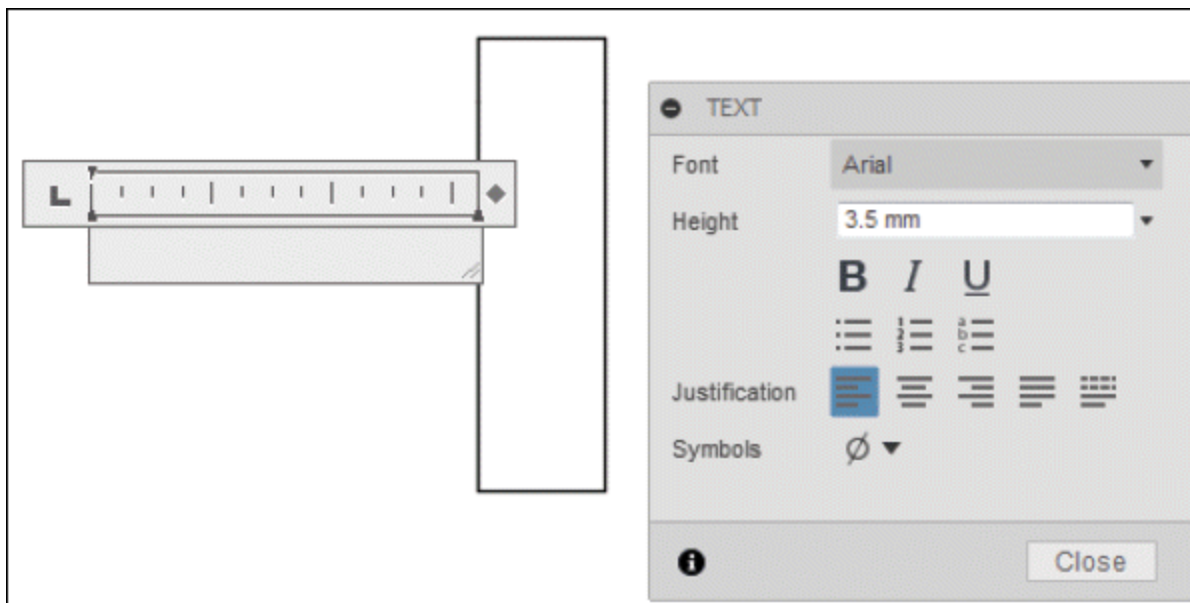


Figure-86. TEXT dialog box displayed

- Click on the **Font** drop-down and select required font for text.
- Click in the **Height** edit box and specify the height of text.

- Set the other parameters as required and type desired text in the **Text** box.
- Click on **Close** button. The text will be added in drawing sheet.

CREATING NOTE

The **Note** tool automatically creates different type of note based on the type of object you select on the current sheet. The procedure to use this tool is discussed next.

- Click on the **Note** tool from **TEXT** panel in the **Toolbar**; refer to [Figure-87](#). The **NOTE** dialog box will be displayed; refer to [Figure-88](#).

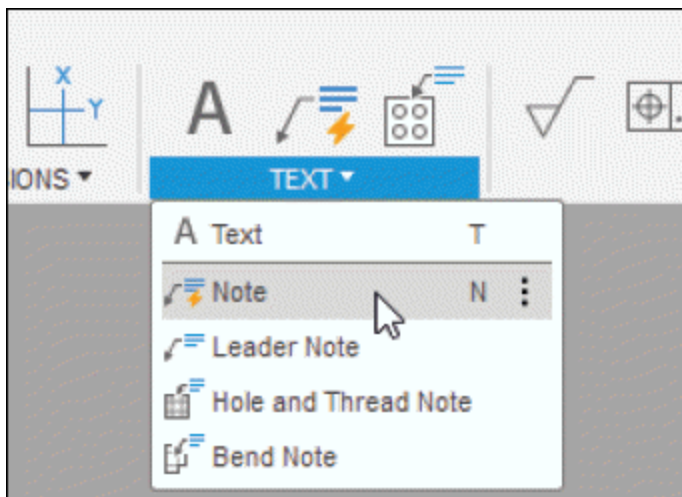


Figure-87. Note tool

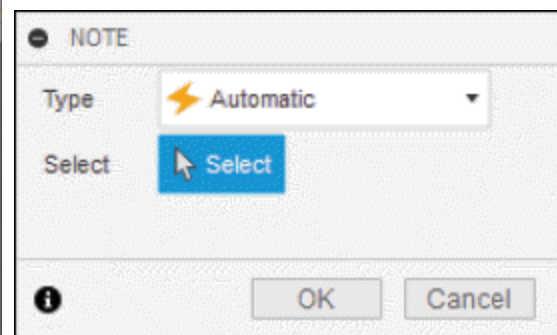


Figure-88. NOTE dialog box

- The **Automatic** option in **Type** drop-down of **NOTE** dialog box is selected by default.
- Select **Leader Note** option from **Type** drop-down to create a multiline text object with a leader that associates the note to a component or feature.

- Select **Hole and Thread Note** option from **Type** drop-down to create a hole or thread note associated with a hole or thread feature.
- Select **Bend Note** option from **Type** drop-down to create a bend note associated with a sheet metal flat pattern bend.
- Here, we selected **Leader Note** option in **Type** drop-down of dialog box or click on **Leader Note** tool from **TEXT** panel of **Toolbar**; refer to [Figure-87](#). The tool will be activated.
- The **Select** button of **Select** section is active by default. Click on the drawing to select the start point. An arrow will be attached to the cursor.
- Click on the screen to place text box. The text box will be displayed along with **TEXT** dialog box; refer to [Figure-89](#).

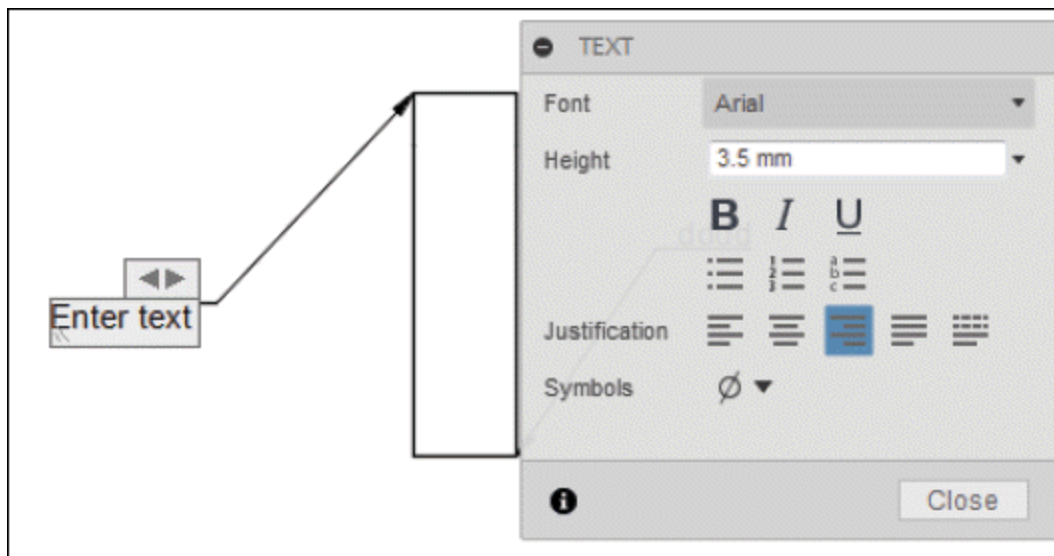


Figure-89. Leader text box

- Set desired parameter in **TEXT** dialog box and type the

text in text box.

- After specifying the parameters, click on **Close** button from **TEXT** dialog box.

Similarly, you can use the **Hole and Thread Note** tool and **Bend Note** tool.

APPLYING SURFACE TEXTURE

The **Surface Texture** tool is used to apply surface finish symbol to objects in drawing. The procedure to use this tool is discussed next.

- Click on the **Surface Texture** tool from **SYMBOLS** panel of **Toolbar**; refer to [Figure-90](#). The tool will be activated.

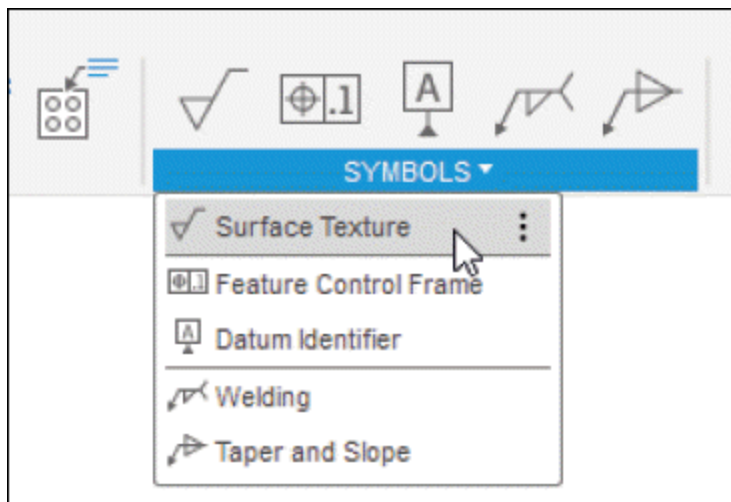


Figure-90. Surface Texture tool

- Click on the model in a drawing view to select object. An arrow will be displayed on the selected part.
- Click to select the second point for leader or press **ENTER** key to place the symbol directly on selected face. The **SURFACE TEXTURE** dialog box will be displayed; refer to

[Figure-91.](#)

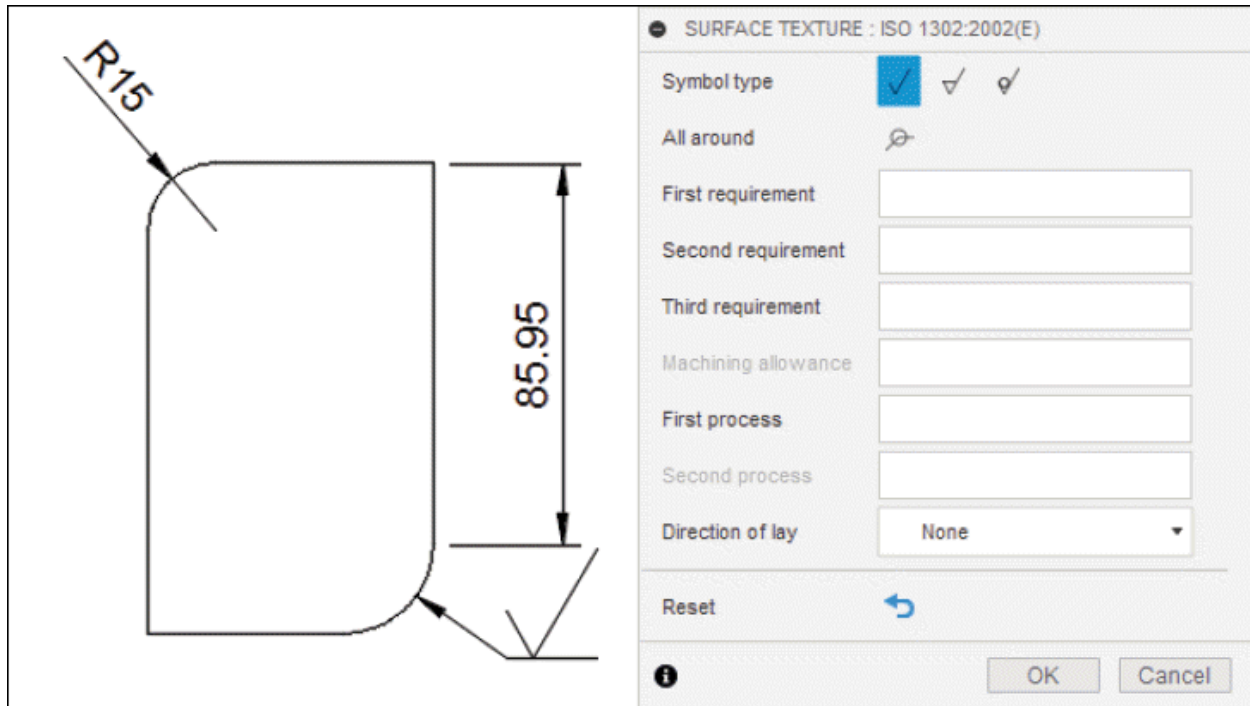


Figure-91. SURFACE TEXTURE dialog box

- Select desired surface finish symbol button from **Symbol type** section.
- Select the **All around** button from **SURFACE TEXTURE** dialog box if you want to apply surface finish to all the surfaces in current view.
- Click in the **First requirement** edit box of **SURFACE TEXTURE** dialog box and enter the first parameter of surface finish symbol. Similarly, specify the other parameters in the edit boxes of the dialog box.
- Select the **Material removal required** symbol type to specify desired machining allowance in the **Machining allowance** edit box of the dialog box.

- Select the lay direction option as desired from **Direction of lay** drop-down.
- Click on the **Reset** button to reset all the properties to default values.
- After specifying the parameters, click on the **OK** button from **SURFACE TEXTURE** dialog box; refer to [Figure-92](#).

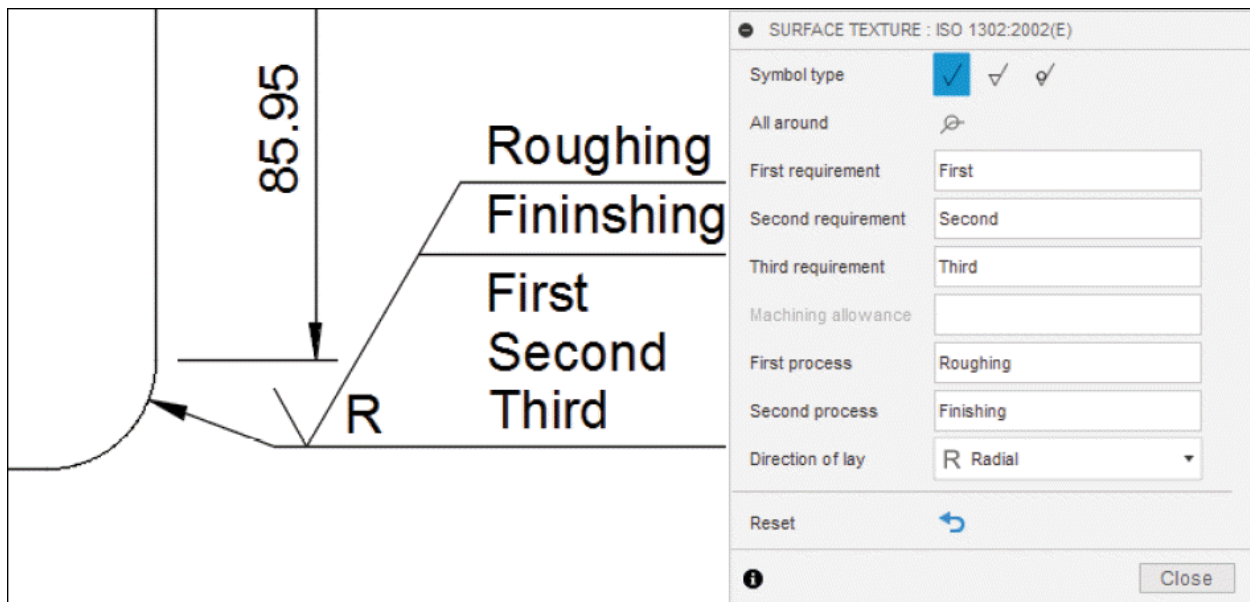
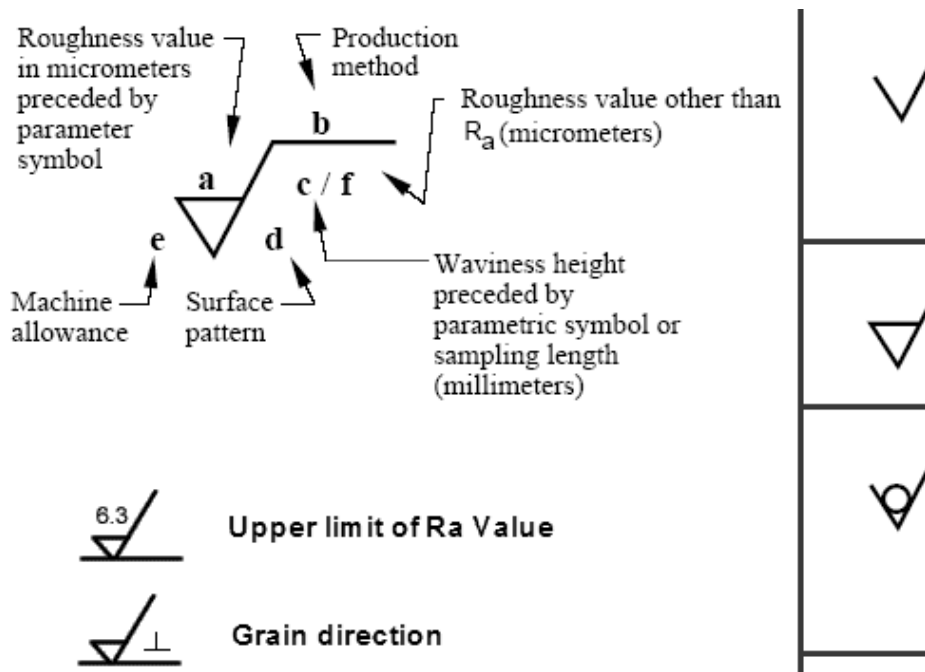


Figure-92. Applying Surface Texture

Surface Texture Symbols or Surface Roughness Symbols



CREATING FEATURE CONTROL FRAME

The **Feature Control Frame** contains data for controlled specifications. It shows the characteristics symbol, datum reference, and tolerance value. It is a rectangular frame which is divided into two or more sections. The feature control frame is also known as GD&T box in laymen's language. The method to insert Feature Control Frame in drawing is same as discussed for **Surface Texture** symbol. In GD&T, a feature control frame is required to describe the conditions and tolerances of a geometric control on a part's feature. The feature control frame consists of four pieces of information:

1

. GD&T symbol or control symbol

2

. Tolerance zone type and dimensions

3

. Tolerance zone modifiers: features of size, projections...

4

. Datum references (if required by the GD&T symbol)

This information provides everything you need to know about geometry of part like what geometrical tolerance needs to be on the part and how to measure or determine if the part is in specification; refer to [Figure-93](#). The common elements of feature control frame are discussed next.

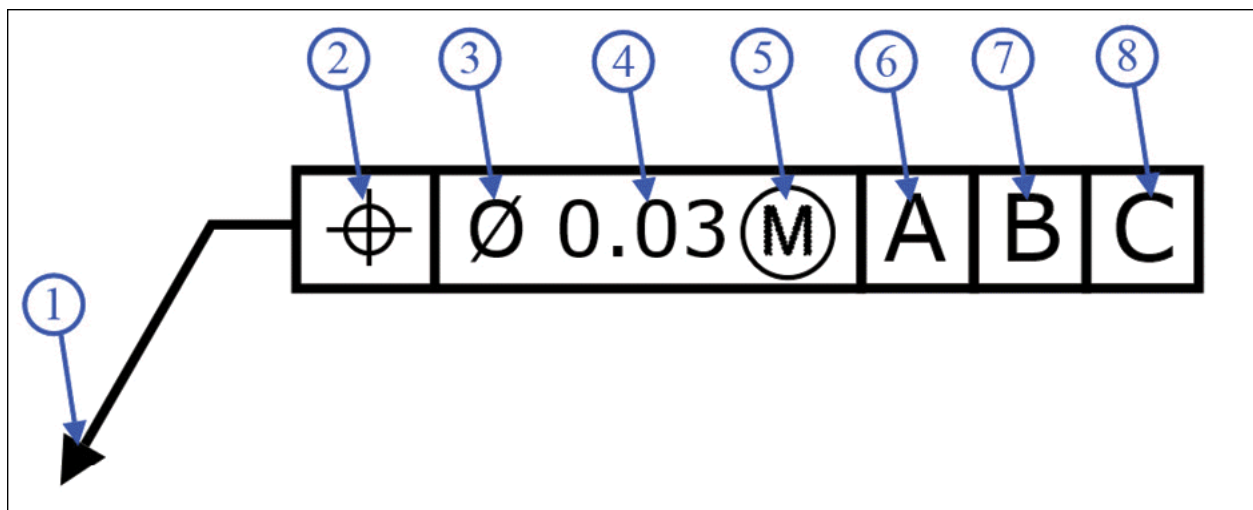


Figure-93. Feature control frame

1

. **Leader Arrow** - This arrow points to the feature then the geometric control is placed on. If the arrow points to a surface then the surface is controlled by the GD&T. If it points to a diametric dimension then the axis is controlled by GD&T. The arrow is optional but helps clarify the

feature being controlled.

2

- . **Geometric Symbol** – This is where your geometric control is specified.

3

- . **Diameter Symbol (if required)** – If the geometric control is a diametrical tolerance then the diameter symbol (\emptyset) will be in front of the tolerance value.

4

- . **Tolerance Value** – If the tolerance is a diameter you will see the \emptyset symbol next to the dimension signifying a diametric tolerance zone. The tolerance of the GD&T is in same unit of measure that the drawing is written in.

5

- . **Feature of Size or Tolerance Modifiers (if required)** – This is where you call out max material condition or a projected tolerance in the feature control frame.

6

- . **Primary Datum (if required)** – If a datum is required, this is the main datum used for the GD&T control. The letter corresponds to a feature somewhere on the part which will be marked with the same letter. This is the datum that must be constrained first when measuring the part. Note: The order of the datum is important for measurement of the part. The primary datum is usually held in three places to fix 3 degrees of freedom.

7

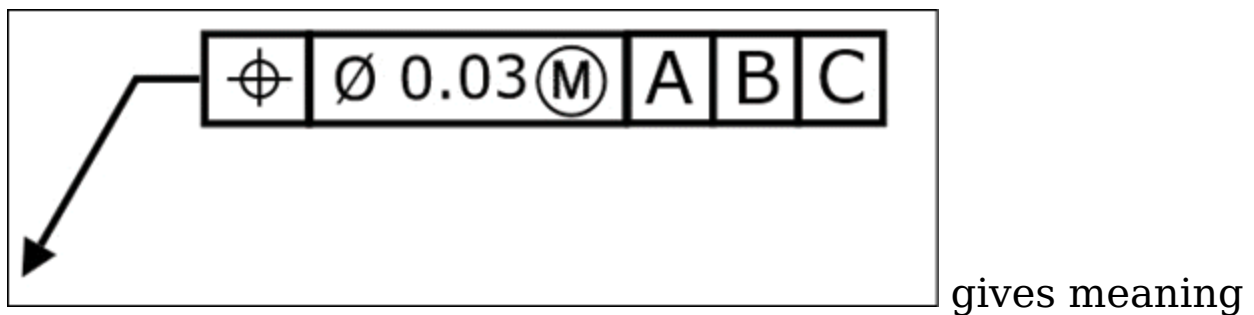
- . **Secondary Datum (if required)** - If a secondary datum is required, it will be to the right of the primary datum. This letter corresponds to a feature somewhere on the part which will be marked with the same letter. During measurement, this is the datum fixated after the primary datum.

8

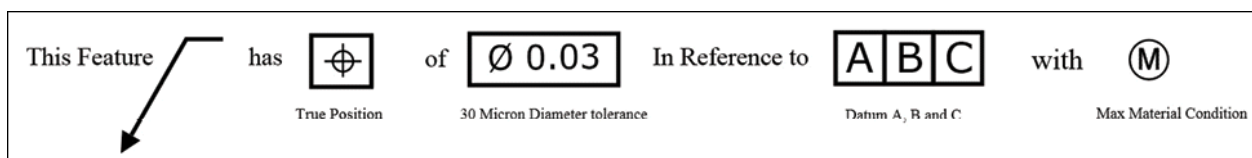
- . **Tertiary Datum (if required)** - If a third datum is required, it will be to the right of the secondary datum. This letter corresponds to a feature somewhere on the part which will be marked with the same letter. During measurement, this is the datum fixated last.

Reading Feature Control Frame

The feature control frame forms a kind of sentence when you read it. Below is how you would read the frame in order to describe the feature.



of



Meaning of various geometric symbols are given in [Figure-94](#).














SYMBOL	CHARACTERISTICS	CATEGORY
—	Straightness	Form
	Flatness	
	Circularity	
	Cylindricity	
	Profile of a Line	Profile
	Profile of Surface	
	Angularity	Orientation
	Perpendicularity	
	Parallelism	
	Position	Location
	Concentricity	
	Symmetry	
	Circular Runout	Runout
	Total Runout	

Figure-94. Geometric Symbols

[Figure-95](#) and [Figure-96](#) shows the use of geometric tolerances in real-world.

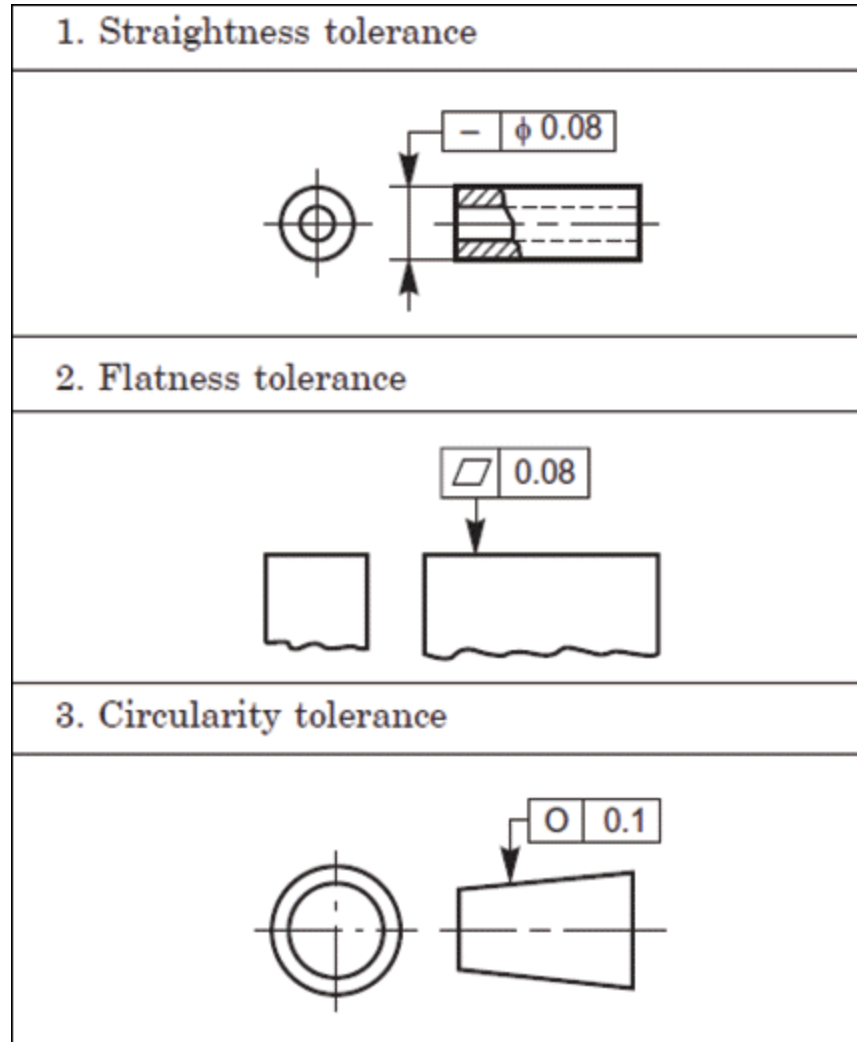


Figure-95. Use of geometric tolerance 1

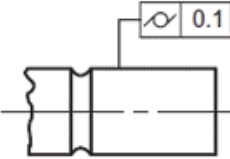
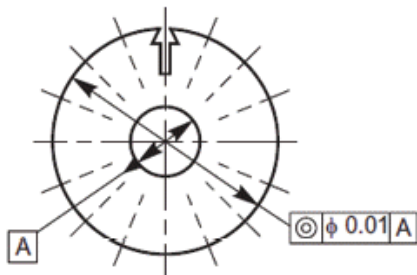
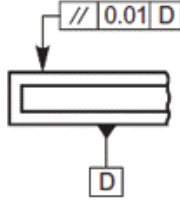
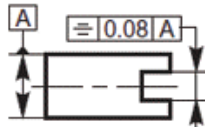
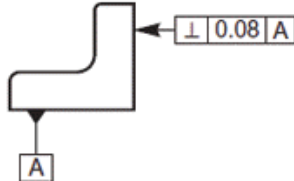
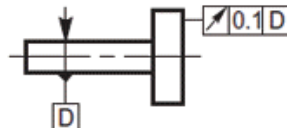
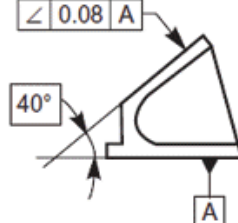
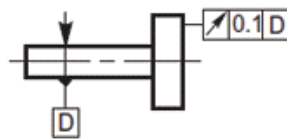
<p>4. Cylindricity tolerance</p> 	<p>8. Concentricity and coaxiality tolerance</p> 
<p>5. Parallelism tolerance</p> 	<p>9. Symmetry tolerance</p> 
<p>6. Perpendicularity tolerance</p> 	<p>10. Radial run-out</p> 
<p>7. Angularity tolerance</p> 	<p>11. Axial run-out</p> 

Figure-96. Use of geometric tolerance 2

Note that in applying most of the geometrical tolerances, you need to define a datum plane like in Perpendicularity, Parallelism, and so on. There are a few dimensioning symbols also used in geometric dimensioning and tolerances, which are given in [Figure-97](#).













Symbol	Meaning	Symbol	Meaning
Ⓛ	LMC – Least Material Condition		Dimension Origin
Ⓜ	MMC – Maximum Material Condition		Counterbore
Ⓣ	Tangent Plane		Countersink
Ⓟ	Projected Tolerance Zone		Depth
ⓕ	Free State		All Around
∅	Diameter		Between
R	Radius		Target Point
SR	Spherical Radius		Conical Taper
S∅	Spherical Diameter		Slope
CR	Controlled Radius		Square
	Statistical Tolerance		
	Basic Dimension		
(77)	Reference Dimension		
5X	Places		

Figure-97. Dimensioning symbols

The procedure to use this tool is discussed next.

- Click on the **Feature Control Frame** tool from **SYMBOLS** panel of **Toolbar**; refer to [Figure-98](#). The tool will be activated.

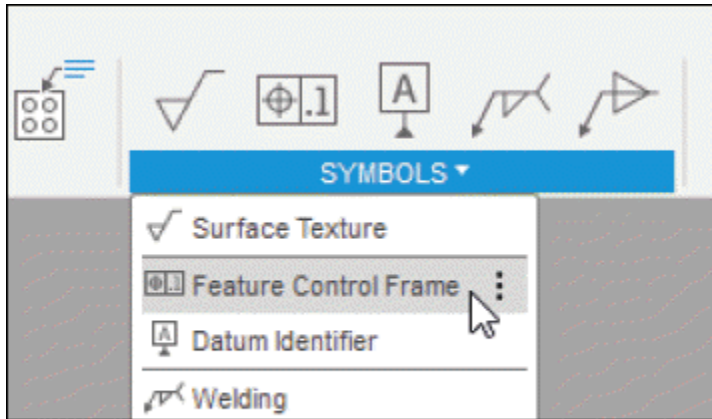


Figure-98. Feature Control Frame tool

- Click on the drawing to select the object. An arrow will be attached to the cursor; refer to [Figure-99](#).

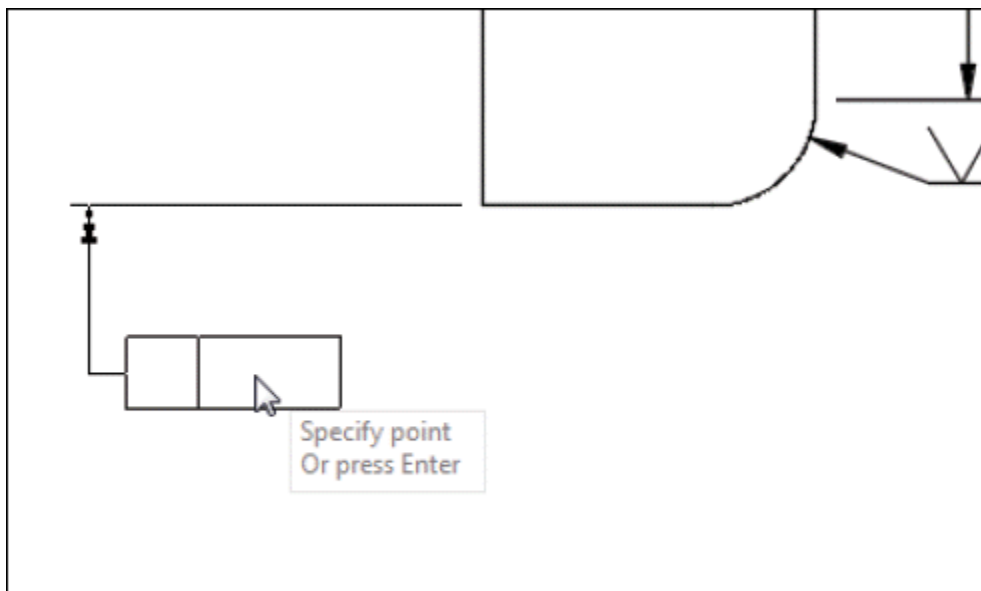


Figure-99. Placing the rectangular box

- Click on the screen to place the rectangular box and press **ENTER** key. The **FEATURE CONTROL FRAME** dialog box will be displayed; refer to [Figure-100](#).

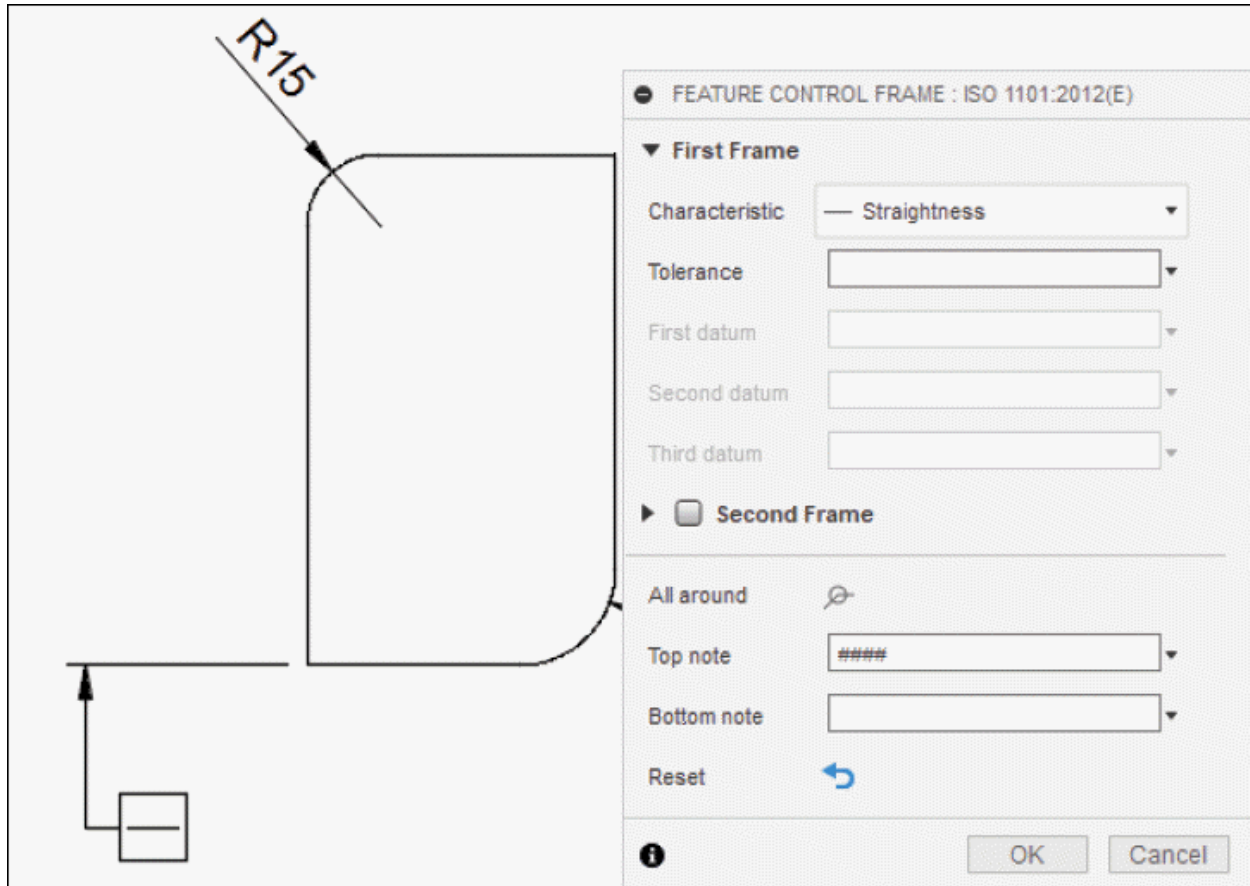


Figure-100. FEATURE CONTROL FRAME dialog box

- Expand the **First Frame** node and select required symbol from **Characteristic** drop-down. The selected symbol will be displayed in the feature frame rectangular box.
- Click in the **Tolerance** edit box and enter the value of tolerance. You can also add the required symbol; refer to [Figure-101](#).

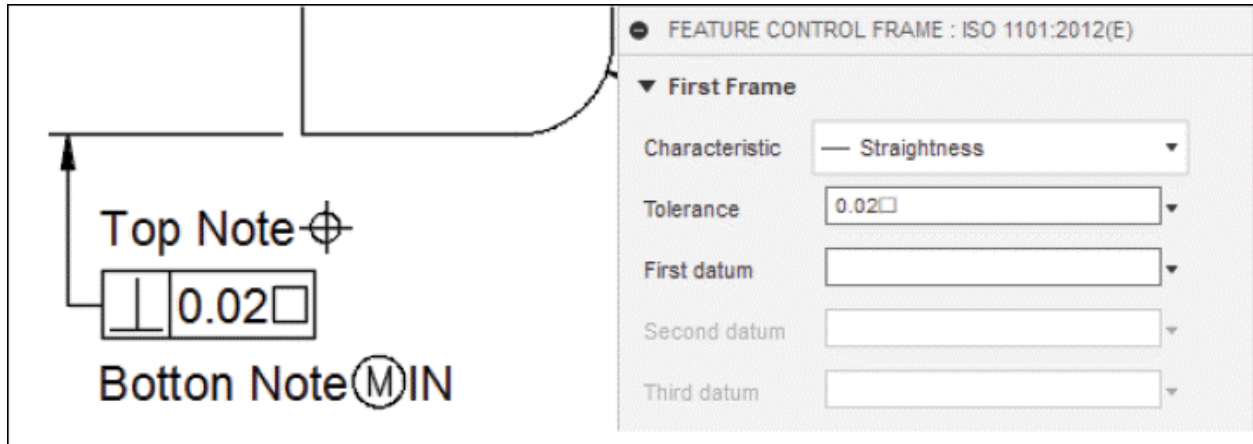


Figure-101. Adding tolerance

- Click on the **First datum** edit box and enter the value of datum as required; refer to [Figure-102](#).

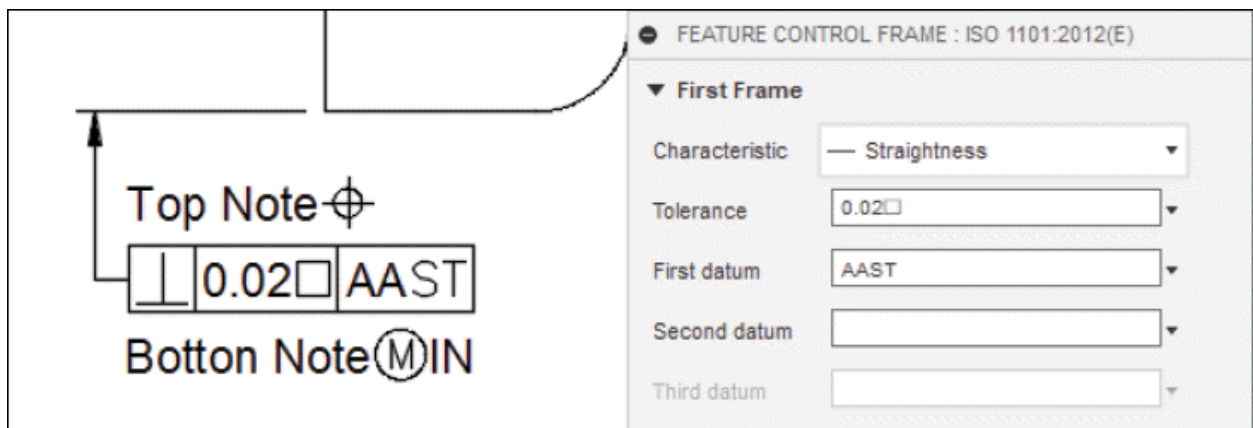


Figure-102. Specifying datum value

- Similarly enter the remaining parameters in respective edit boxes.
- Select the **Second Frame** check box of **FEATURE CONTROL FRAME** dialog box if you want to add second frame for the drawing. Feature Control Frames with two frames is also called Composite Feature Control Frame.
- Click in the **Top Note** edit box and enter the note. If you

want to add any symbol then click on the drop-down button. A list of symbols will be displayed; refer to [Figure-103](#).

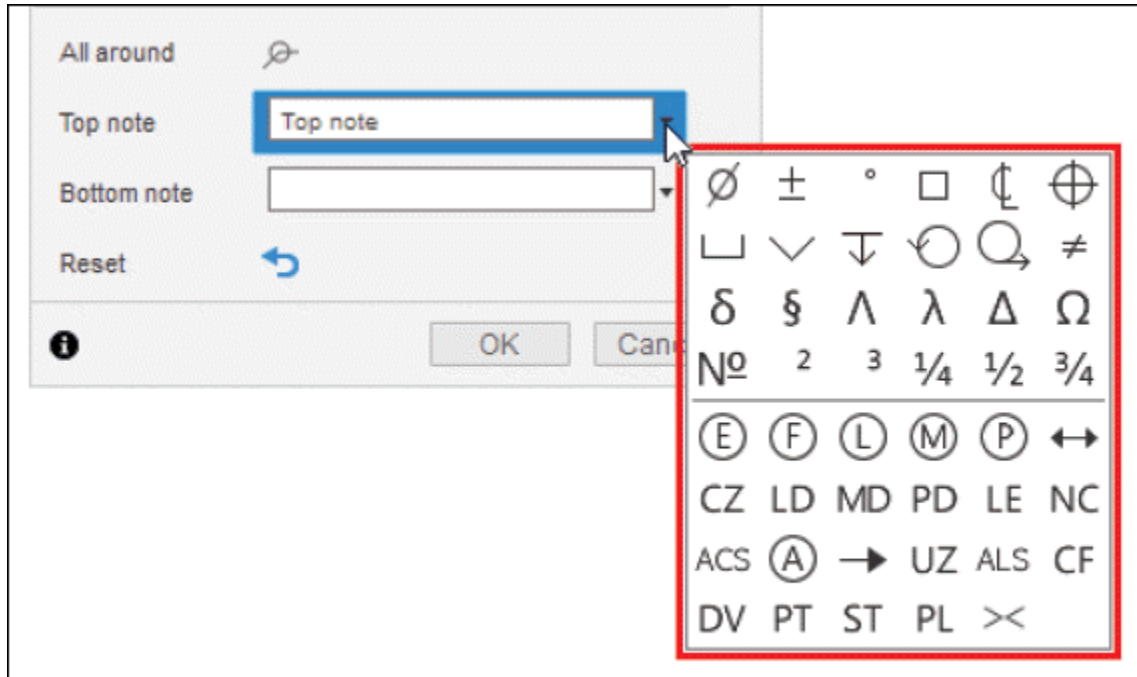


Figure-103. Adding symbols

- Click on required symbol. The selected symbol will be added over the feature control frame box. Similarly, you can set the symbol for **Bottom Note** edit box.
- After specifying the parameters, click on the **OK** button from **FEATURE CONTROL FRAME** dialog box to complete the process.

CREATING DATUM IDENTIFIER

The **Datum Identifier** tool identifies a datum feature for a feature control frame symbol. Datum identifier are circular frames divided in two parts by a horizontal line. The lower

half represents the datum feature and the upper half is for additional information, such as dimensions of the datum target area; refer to [Figure-104](#).

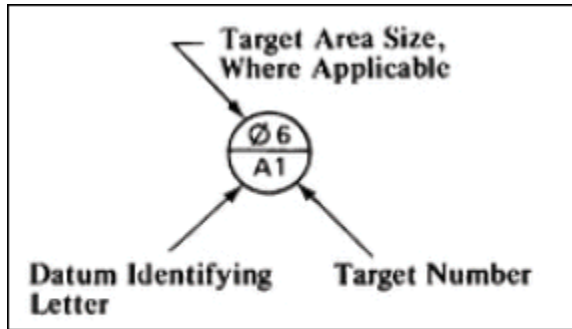


Figure-104. Datum target area

The procedure to use this tool is discussed next.

- Click on the **Datum Identifier** tool from **SYMBOLS** panel of **Toolbar**; refer to [Figure-105](#). The tool will be activated.

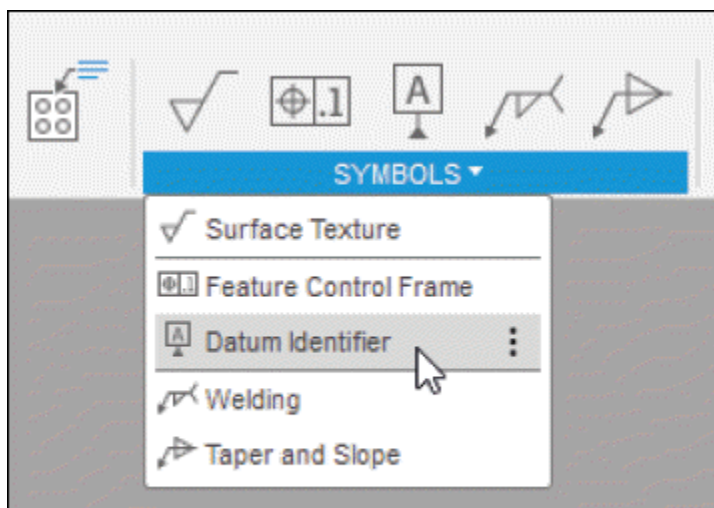


Figure-105. Datum Identifier tool

- Click on the drawing to select object. An arrow will be attached to the cursor.
- Specify the point and press **ENTER** key. The datum ID

symbol will be displayed along with **DATUM IDENTIFIER** dialog box; refer to [Figure-106](#).

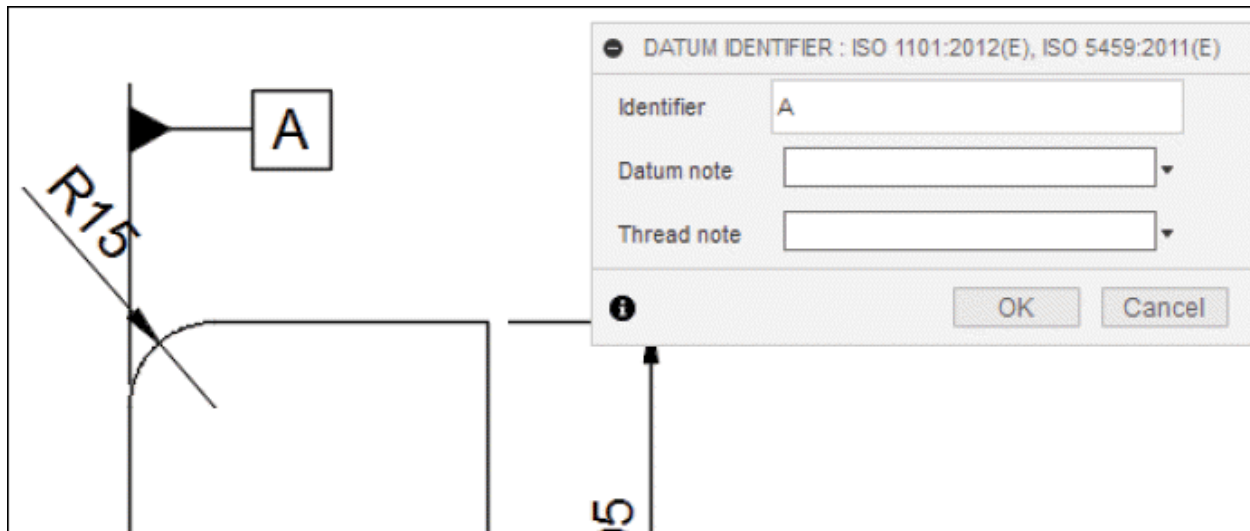


Figure-106. *DATUM IDENTIFIER* dialog box

- Click in the **Datum note** edit box and enter desired note.
- Click in the **Thread note** edit box and enter desired text. You can also add the required symbol from **Symbol** drop-down.
- After specifying the parameters, click on the **OK** button from **DATUM IDENTIFIER** dialog box; refer to [Figure-107](#).

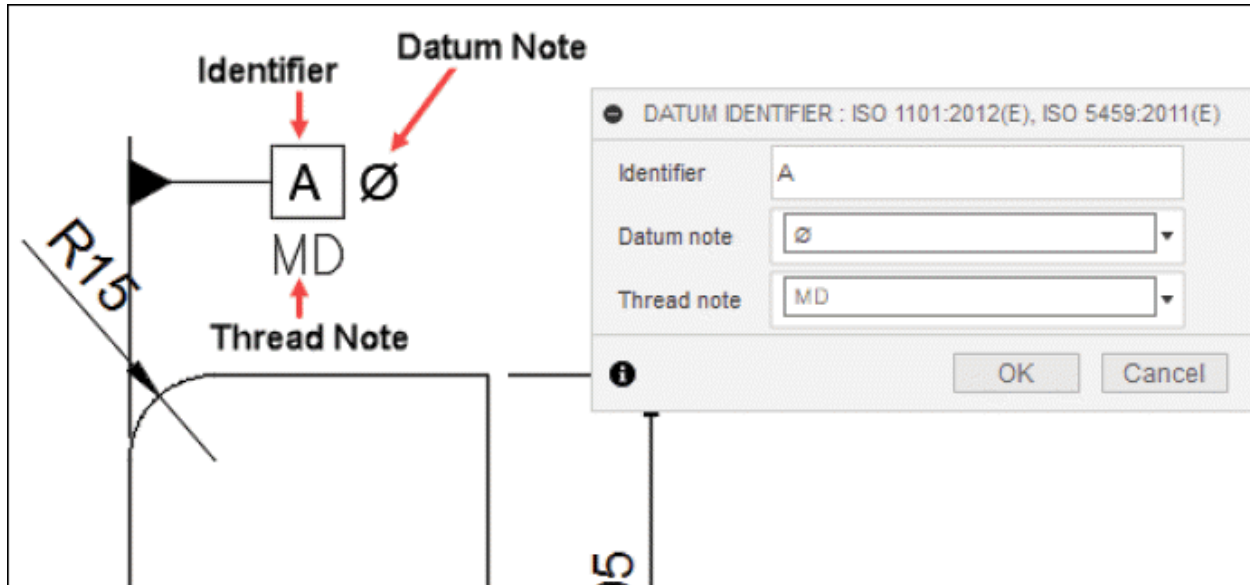


Figure-107. Updated DATUM IDENTIFIER dialog box

CREATING WELDING SYMBOL AND OTHER SYMBOLS

The **Welding** tool in **SYMBOLS** drop-down of **Ribbon** is used to insert welding symbols applied to selected edge/face. Using this tool, you can apply annotation to the edges where weld is to be performed. The procedure to place a weld symbol is similar to placing Feature Control Frame discussed earlier. Similarly, you can use the **Taper and Slope** tool from the **SYMBOLS** drop-down in the **DRAWING** tab of **Ribbon** to apply symbol defining taper/slope on selected edge.

GENERATING TABLES

The **Table** tool is used to generate empty table, parts list, and bend table of the assembly model linked with drawing. The procedure to use this tool is discussed next.

- Click on the **Table** tool from **TABLES** drop-down in the

Toolbar; refer to [Figure-108](#). The **TABLE** dialog box will be displayed along with attached empty table; refer to [Figure-109](#).

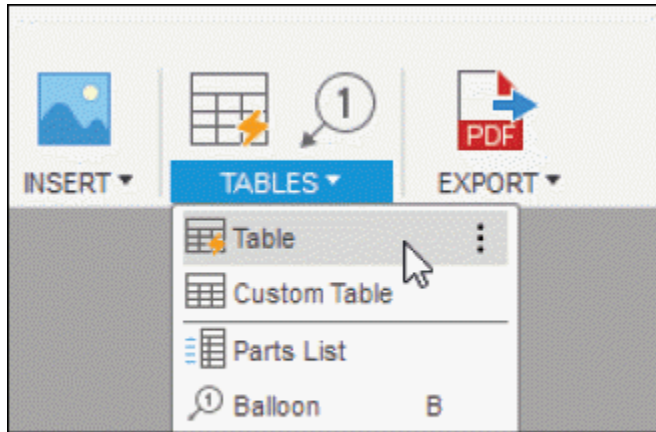


Figure-108. Table tool

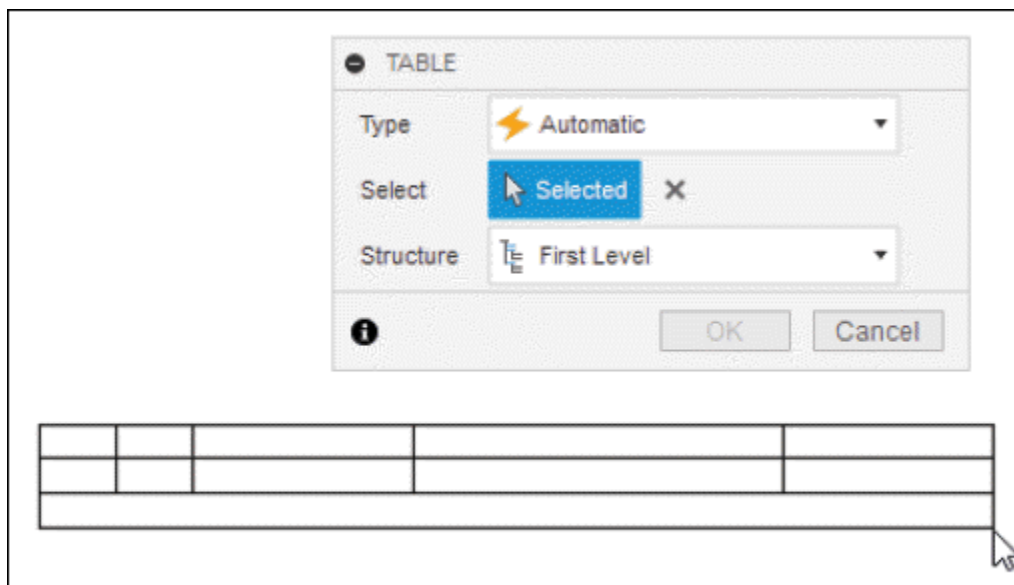


Figure-109. TABLE dialog box along with empty table

- The **Automatic** option in **Type** drop-down of **TABLE** dialog box is selected by default. The **Automatic** option automatically creates a parts list or bend table, based on the contents of the current sheet.

- Select **Custom Table** option from **Type** drop-down to create an empty table.
- Select **Parts List** option from **Type** drop-down to create a parts list based on an assembly, storyboard, component, or sheet metal component reference. Note that Parts list is also called Bill of Materials or BOM.
- Select **Bend Table** option from **Type** drop-down to create a bend table based on a sheet metal flat pattern reference.
- Select **Hole Table** option from **Type** drop-down to create a hole table based on the origin you place in a drawing view.
- Select **Revision History** option from **Type** drop-down to create a revision history for the active sheet.
- Here, we are selecting the **Parts List** option from **Type** drop-down of dialog box or click **Parts List** tool from **TABLES** drop-down in the **Toolbar**; refer to [Figure-108](#).
- The **Select** button of **Select** section is active by default. Select desired view on the current sheet.
- Select **First Level** option from **Structure** drop-down to include first level components only.
- Select **All Levels** option from **Structure** drop-down to include all components in the assembly.
- Click at desired location in the drawing to place the parts list. Parts list will be generated with balloons; refer to [Figure-110](#).

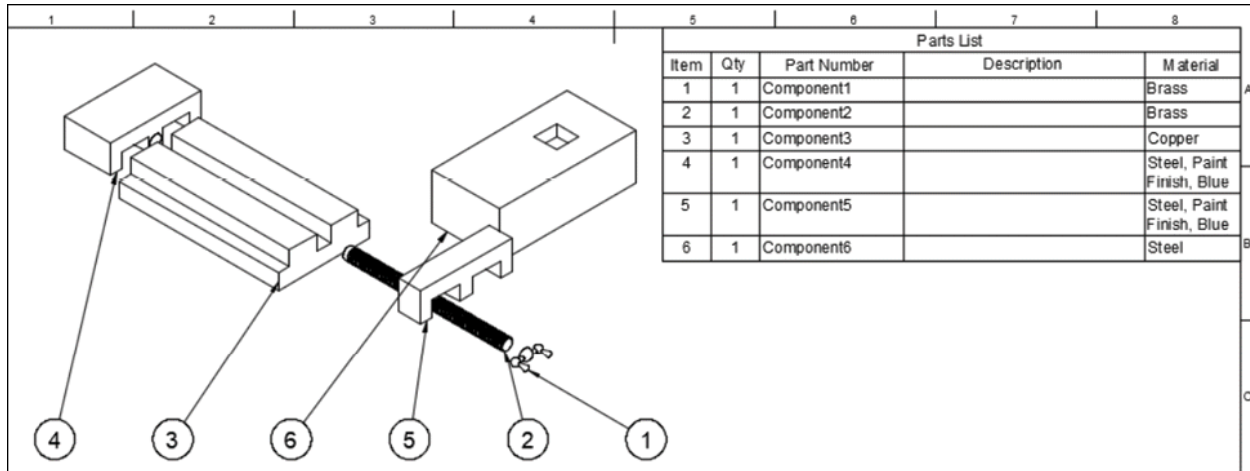


Figure-110. Parts List of assembly

Similarly, you can use **Custom Table (Empty Table)** tool, **Bend Table** tool, **Hole Table** tool, and **Revision History** tool. Note that you can edit the fields of empty table by double-clicking in them.

GENERATING BALLOON

The **Balloon** tool is used to generate balloons for components in drawing. The procedure to use this tool is discussed next.

- Click on the **Balloon** tool from **TABLES** drop-down; refer to [Figure-111](#). The **BALLOON** dialog box will be displayed; refer to [Figure-112](#) and you will be asked to select the component.

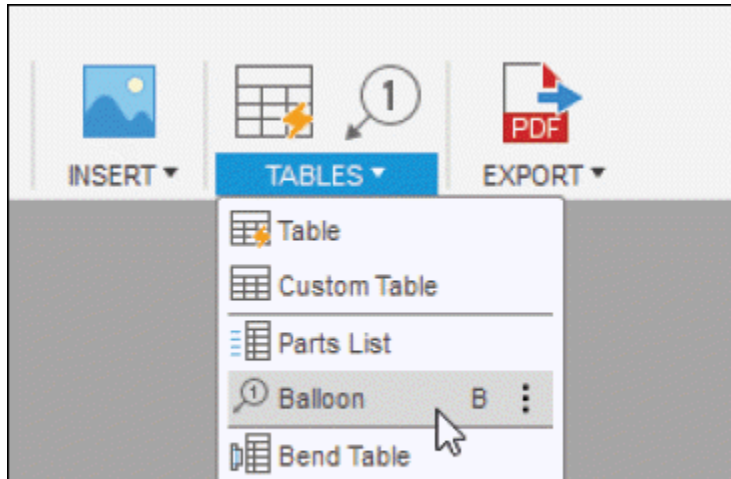


Figure-111. Balloon tool

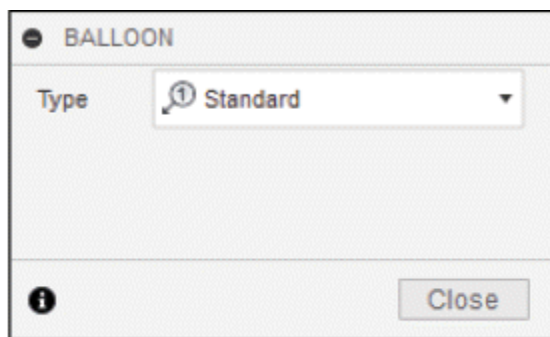


Figure-112. BALLOON dialog box

- By default, the **Standard** option is selected in the dialog box, so straight lines are created with balloon. If you want to create spline attached to balloon in place of line then select the **Patent** option from the **Type** drop-down.
- Click on desired edge of component. On selection of edge, an arrow will be attached to the cursor.
- Click at desired location to place the balloon; refer to [Figure-113](#) and [Figure-114](#).

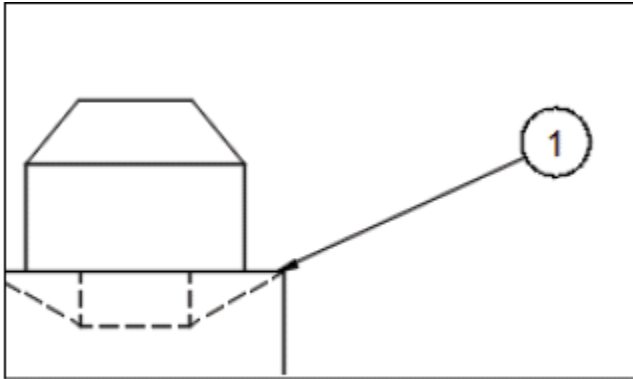


Figure-113. Placing balloon with Standard option

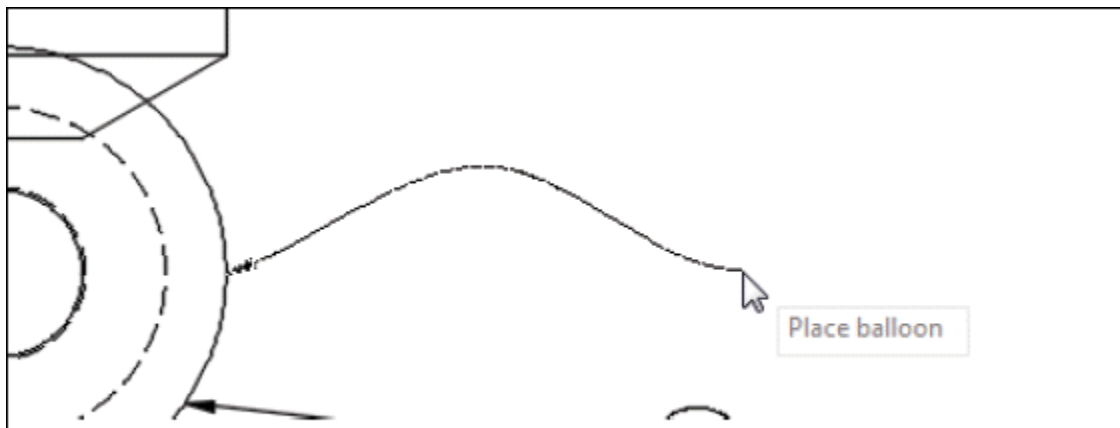


Figure-114. Placing balloon with Patent option

BEND IDENTIFIER AND TABLE

The **Table** tool discussed earlier works in a different way for sheet metal component. If you are generating drawing for a sheetmetal part then **Table** tool will generate bend table for the part and like balloons in assembly, bend identifiers are used to identify bends as per the table. The process is given next.

- After starting drawing environment, using flat pattern of a sheet metal part, click on the **Bend Identifier** tool from the **TABLES** drop-down in the **Toolbar**; refer to [Figure-](#)

[115](#). You will be asked to select the bends in flat pattern.

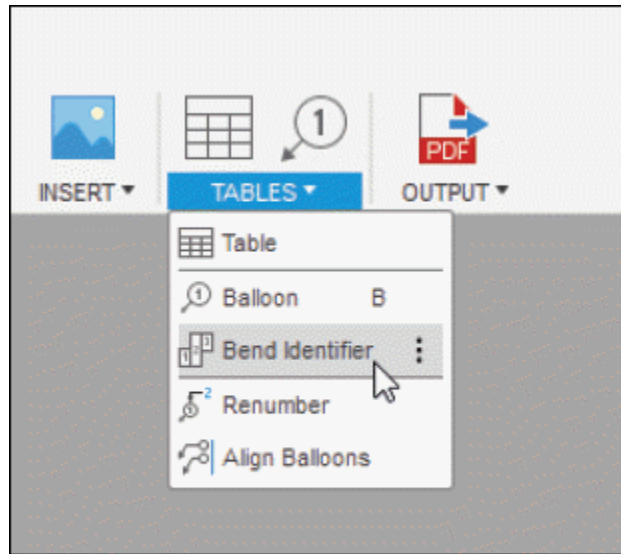


Figure-115. Bend Identifier tool

- Click on the sheet metal bends in drawing one by one. Bend identifiers will be assigned to the bends; refer to [Figure-116](#).

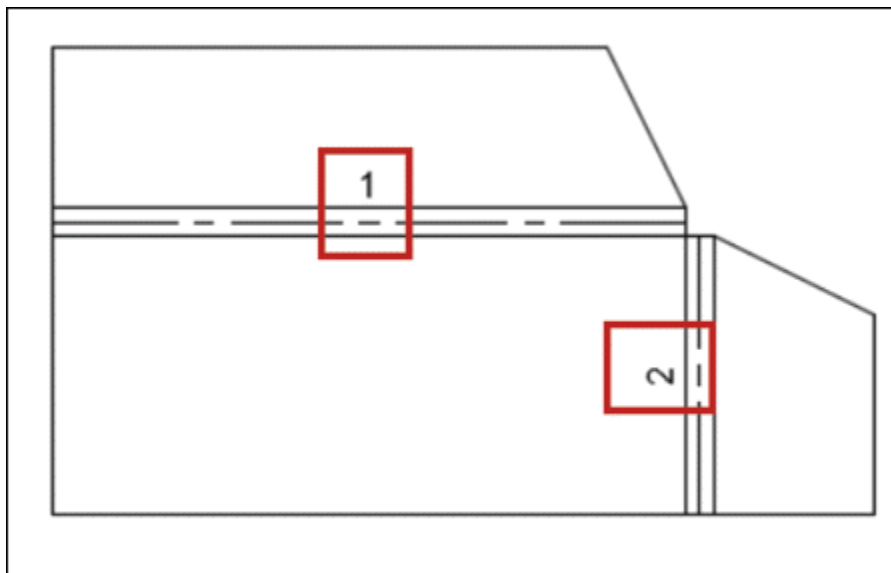


Figure-116. Bend identifier assigned

- Press **ESC** to exit the tool.

- Click on the **Table** tool from **TABLES** drop-down in the **Toolbar** and place the bend table at desired location; refer to [Figure-117](#).

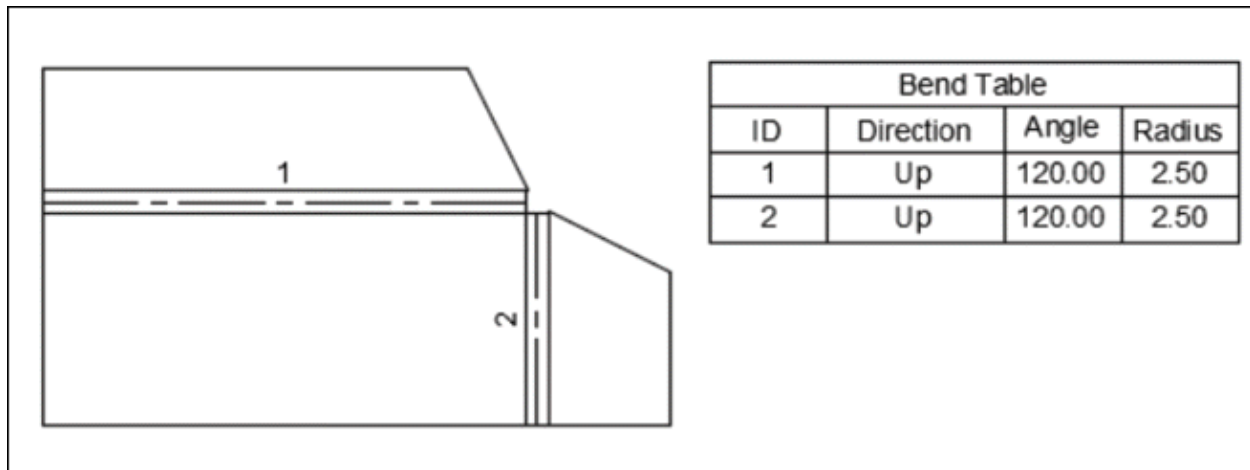


Figure-117. Flat pattern with bend table

REVISION MARKER

The **Revision Marker** tool is used to create non-associative and associative revision markers with and without a leader in a **DRAWING** workspace. The procedure to use this tool is discussed next.

- Click on the **Revision Marker** tool from **TABLES** panel in the **Toolbar**; refer to [Figure-118](#). The **REVISION MARKER** dialog box will be displayed; refer to [Figure-119](#).

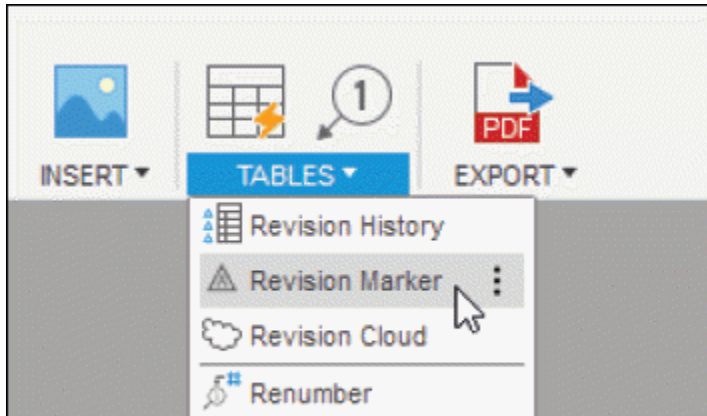


Figure-118. Revision Marker tool

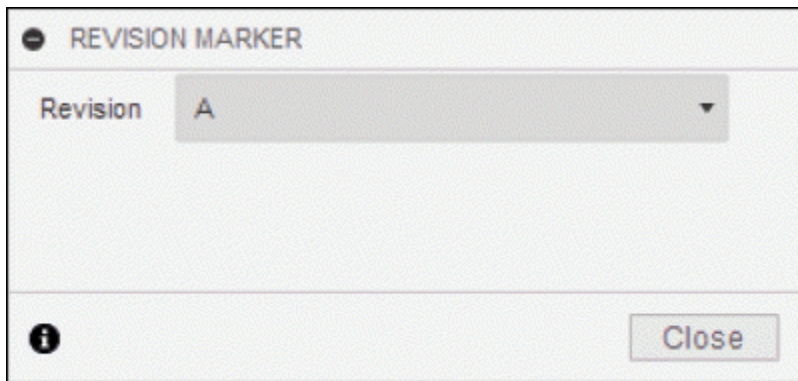


Figure-119. REVISION MARKER dialog box

- Select desired revision from **Revision** drop-down, if more revisions are available.
- Select an object or specify a point to place the revision marker; refer to [Figure-120](#).

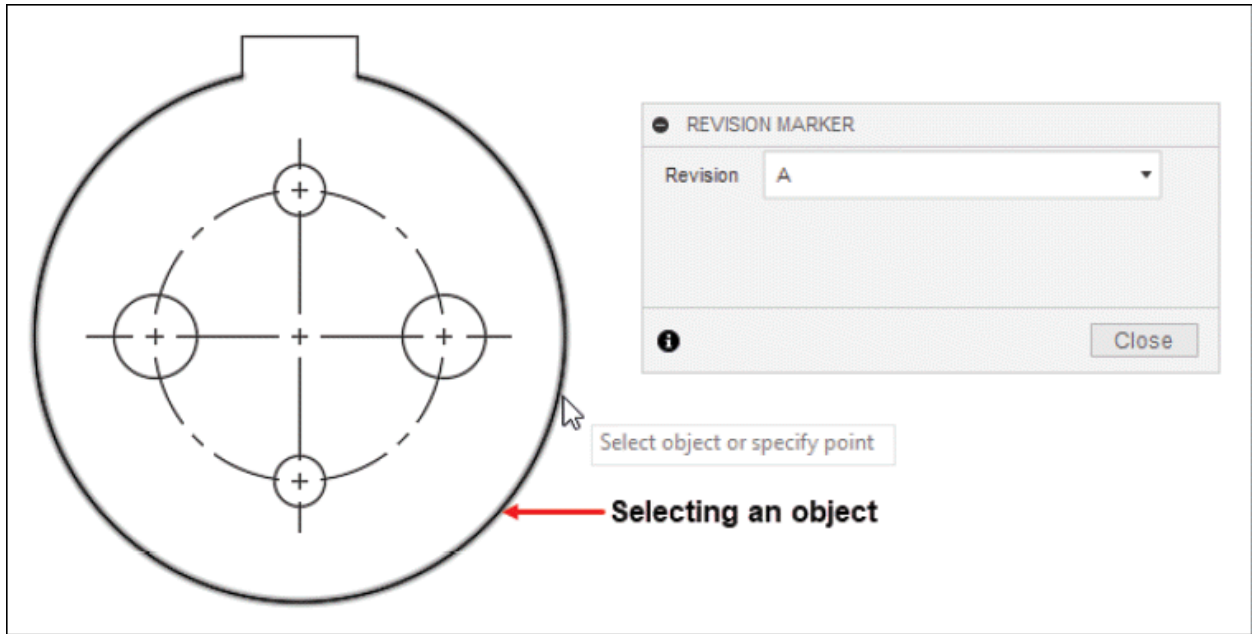


Figure-120. Selecting an object

- After selecting the object, an arrow will be attached to the cursor.
- Click at desired location to place the revision marker; refer to [Figure-121](#).

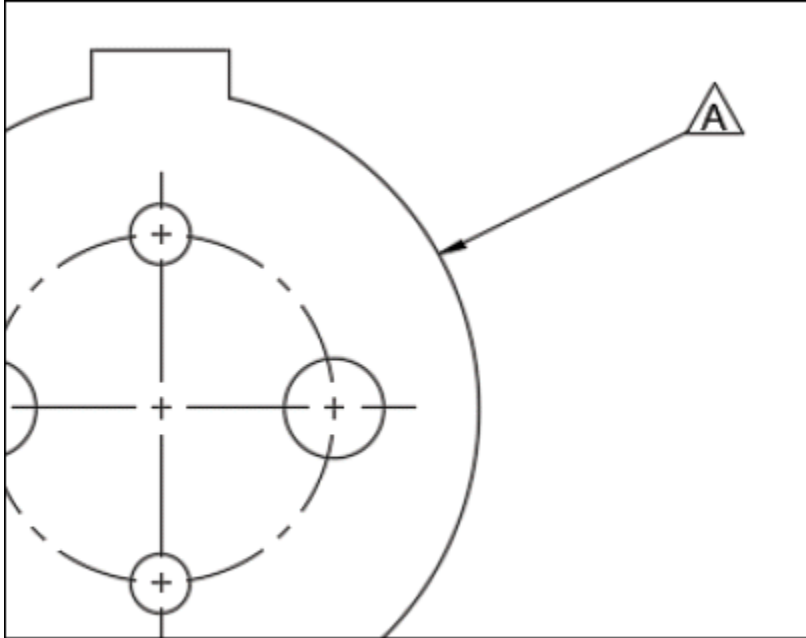


Figure-121. Revision marker placed

- Click on the **Close** button from the dialog box to exit the tool.

REVISION CLOUD

The **Revision Cloud** tool is used to create cloud-shaped objects to call attention to specific changes or portions of each drawing in the **DRAWING** workspace. The procedure to use this tool is discussed next.

- Click on the **Revision Cloud** tool from the **TABLES** panel in the **Toolbar**; refer to [Figure-122](#). You will be asked to specify the first point.

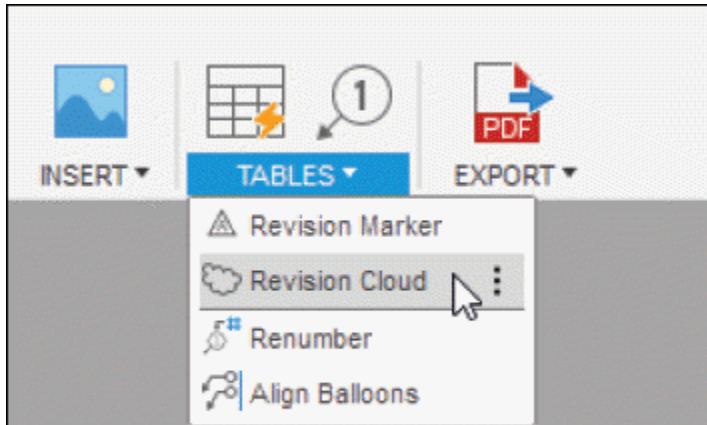


Figure-122. Revision Cloud tool

- Click in the canvas at desired location to specify the first point of the revision cloud.
- Move the cursor away and click at desired location to specify the next point of the revision cloud; refer to [Figure-123](#). Optionally, you can specify the more points of the revision cloud.

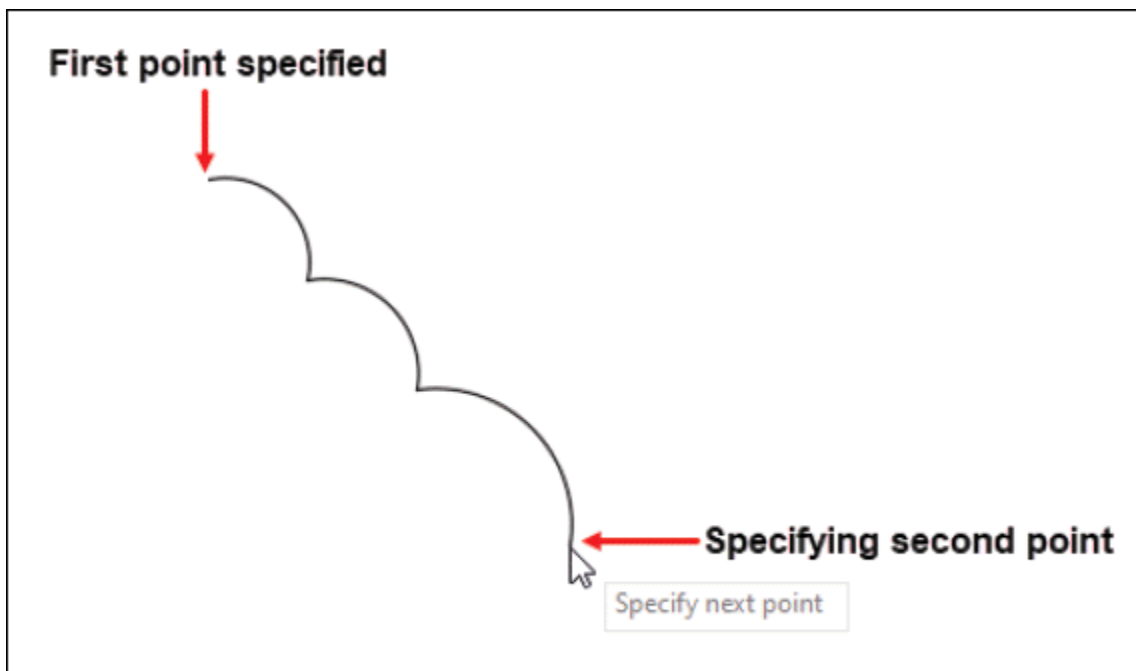


Figure-123. Specifying points of revision cloud

- After specifying desired points, press **Enter** from the keyboard. The revision cloud will be created; refer to [Figure-124](#).

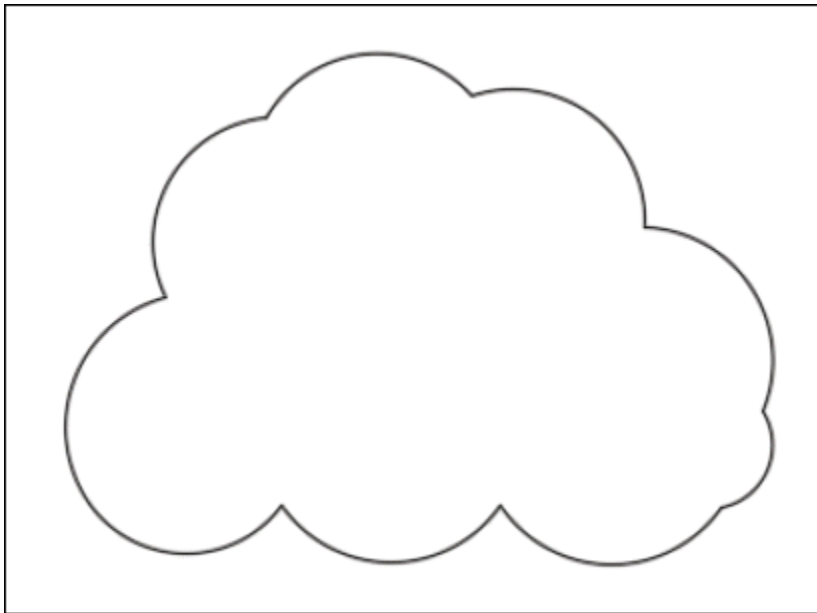


Figure-124. Revision cloud created

REVISION HISTORY

The **Revision History** tool is used to generate a table of list of revisions in the drawing. After performing revisions then generating revision markers, click on the **Revision History** tool from the **TABLES** drop-down in **Toolbar**. The revision history table will be created; refer to [Figure-125](#).

Revision History			
Rev	Description	Date	Approved
A		23-01-2023	




Figure-125. Revision History table

RENUMBERING

The **Renumber** tool is used to renumber the existing balloons. The procedure to use this tool is discussed next.

- Click on the **Renumber** tool from the **TABLES** drop-down in the **Toolbar**; refer to [Figure-126](#). The **RENUMBER** dialog box will be displayed; refer to [Figure-127](#).

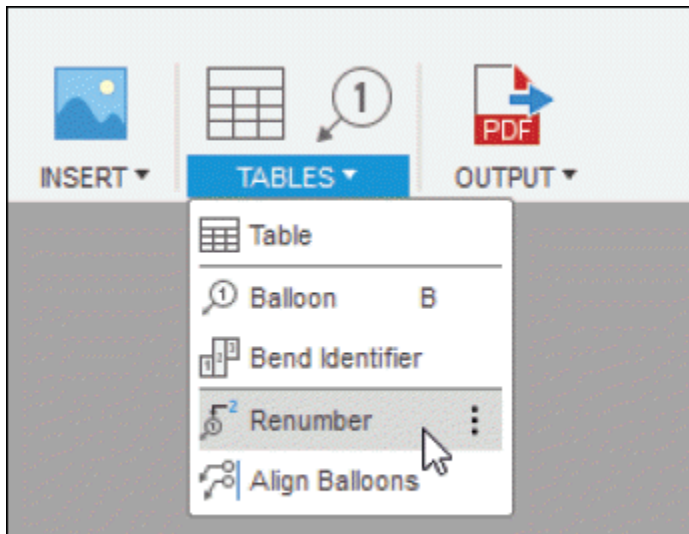


Figure-126. Renumber tool

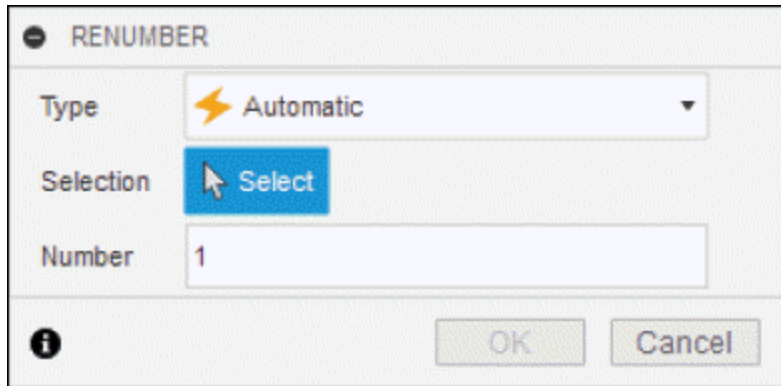


Figure-127. RENUMBER dialog box

- Select **Automatic** option from **Type** drop-down to renumber the balloons or bend identifiers, based on the selection. Select **Balloon** option from **Type** drop-down to renumber the balloons for components in a parts list. Select **Bend Identifier** option from **Type** drop-down to renumber the bend identifiers for sheet metal bends in a bend table.
- Click in the **Number** edit box and enter desired number.
- Click on **Select** button from **Selection** section and click on the balloons as per their sequences to renumber them. The balloon numbers will be changed; refer to [Figure-128](#).

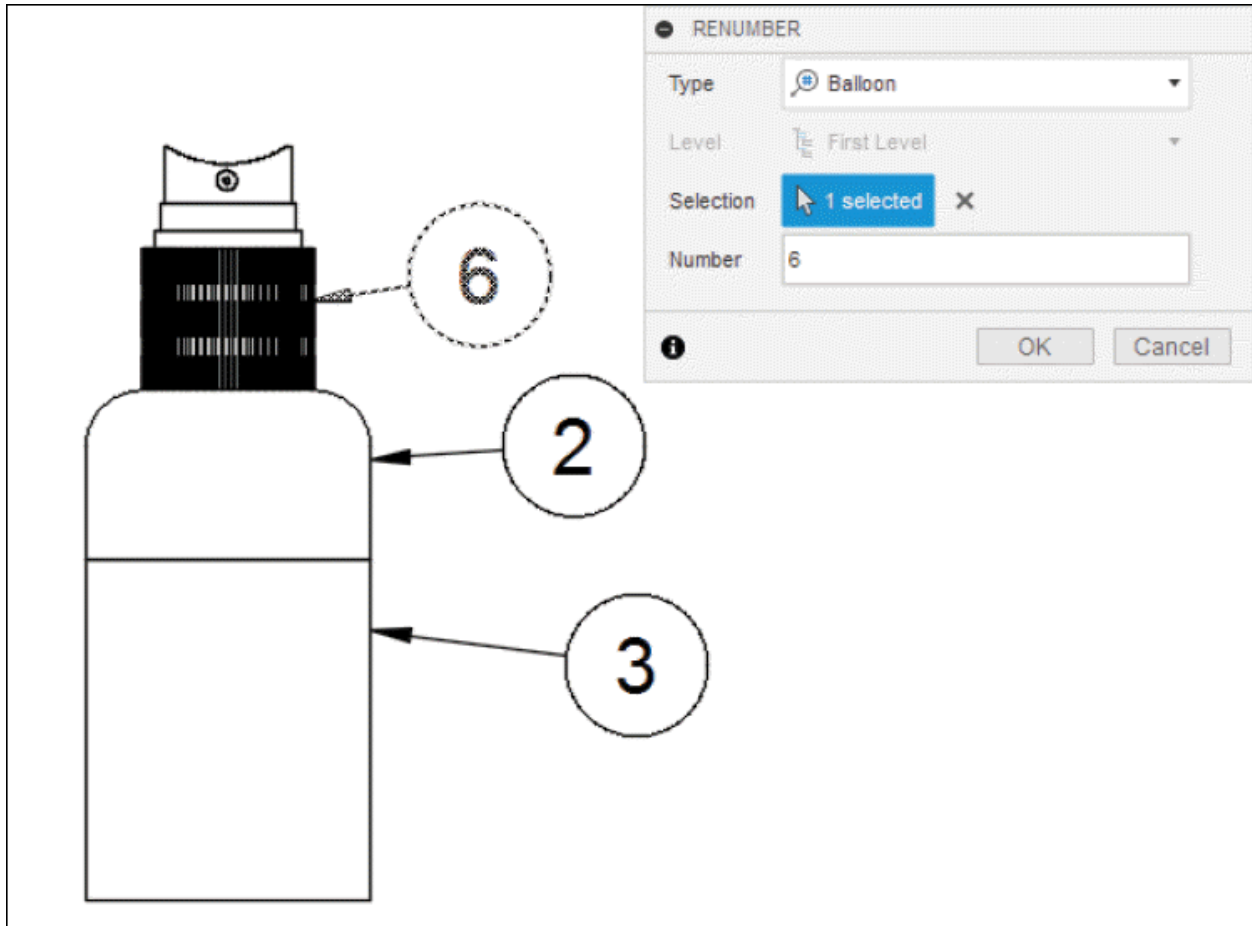


Figure-128. Changed balloon number

- After specifying the parameters, click on **OK** button from **RENUMBER** dialog box to complete the process.

ALIGNING BALLOONS

The **Align Balloons** tool is used to align all the pre-existing balloons. The procedure to use this tool is discussed next.

- Click on the **Align Balloons** tool from **TABLES** drop-down in **Toolbar**; refer to [Figure-129](#). The tool will be activated.

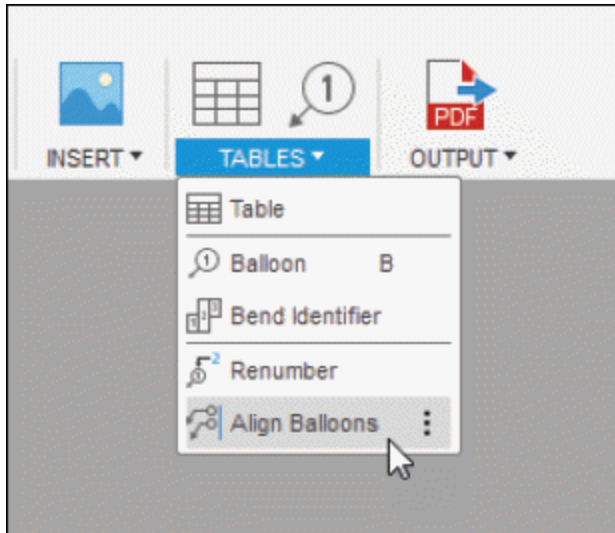


Figure-129. Align Balloons tool

- Click to select all the balloons to aligned; refer to [Figure-130](#) and press **ENTER**. You will be asked to specify starting point of line to be used for alignment of balloons.

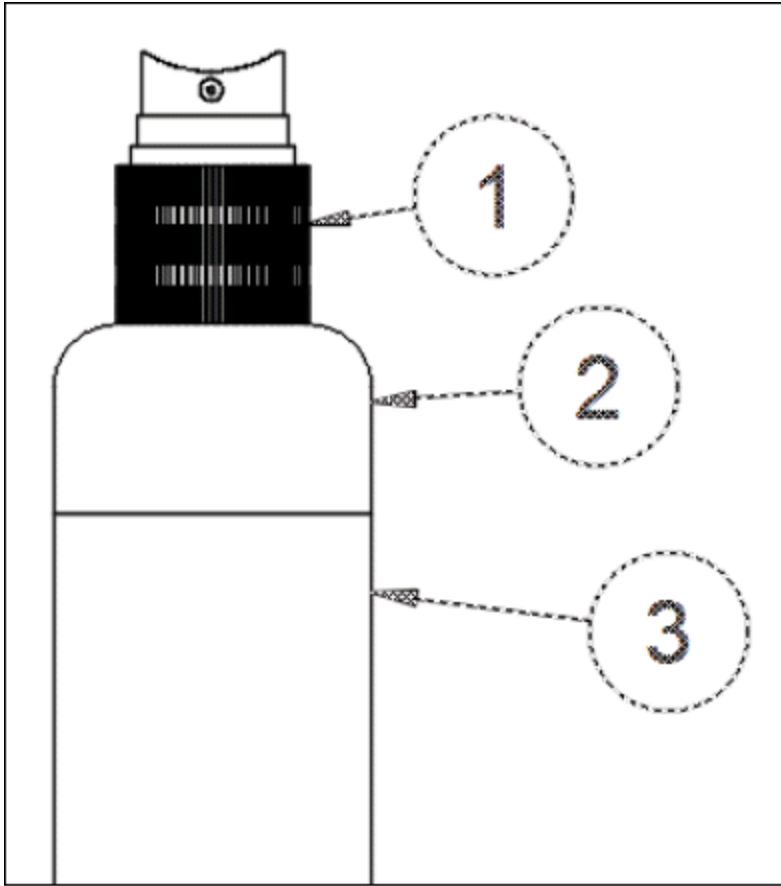


Figure-130. Selection of balloons for alignment

- Click on the screen to specify the start point. End point of alignment line will be attached to cursor.
- Click on the screen at desired location to define alignment of balloons; refer to [Figure-131](#).

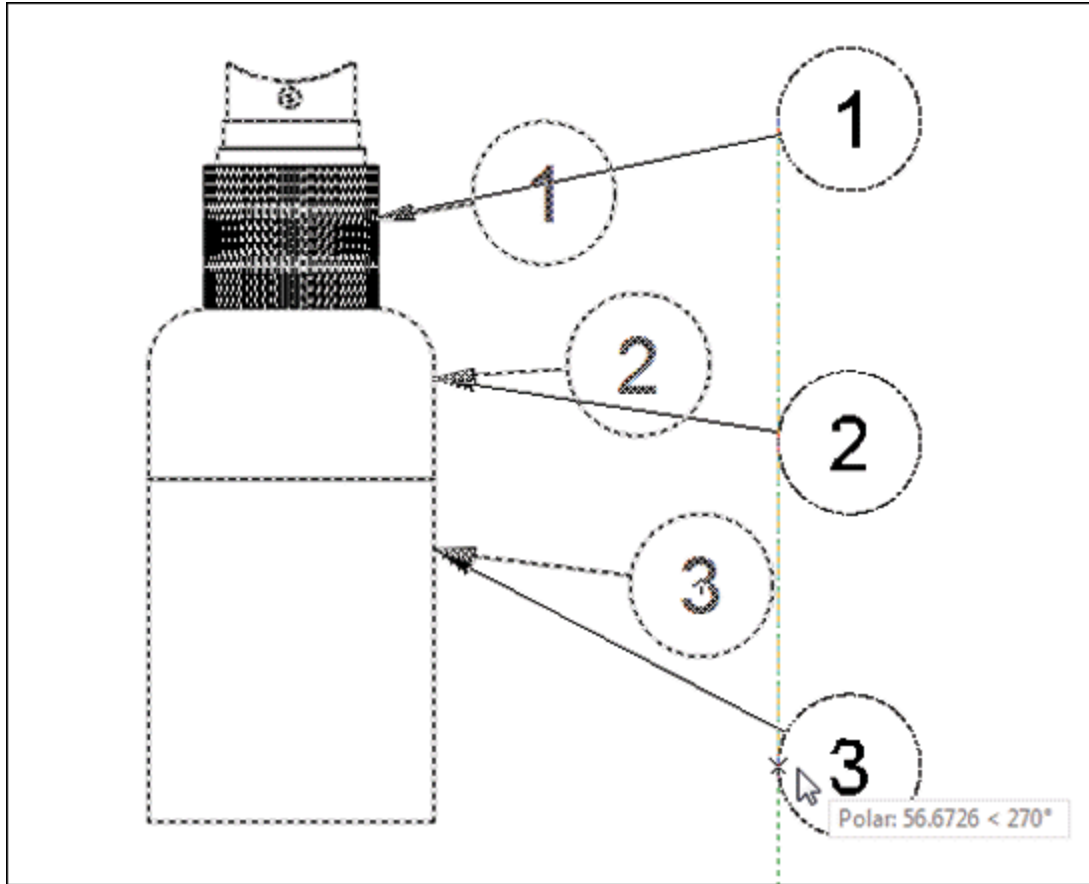


Figure-131. Performing alignment process

EXPORT PDF

The **Export PDF** tool is used to output the drawing in PDF format. The procedure to use this tool is discussed next.

- Click on the **Export PDF** tool of **EXPORT** drop-down in **Toolbar**; refer to [Figure-132](#). The **EXPORT PDF** dialog box will be displayed; refer to [Figure-133](#).

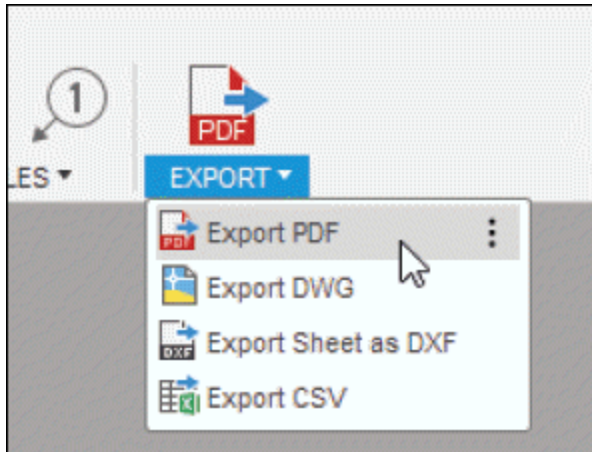


Figure-132. Export PDF tool

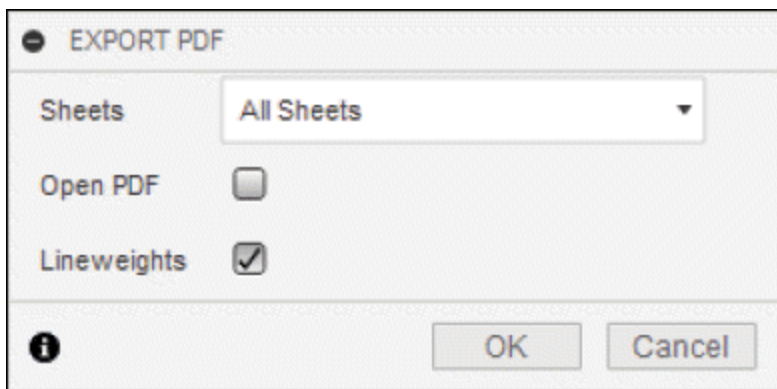


Figure-133. EXPORT PDF dialog box

- Click on the **Sheets** drop-down from the **EXPORT PDF** dialog box and select required sheet for output. You can also use **SHIFT** key or **CTRL** key to select multiple sheets.
- Select the **Open PDF** check box if you want to open the output PDF file.
- Select the **Lineweights** check box if you want to make bold lines of drawing otherwise, clear it.
- After specifying the parameters, click on the **OK** button from **EXPORT PDF** dialog box. The **Export PDF** dialog

box will be displayed to save the file; refer to [Figure-134](#).

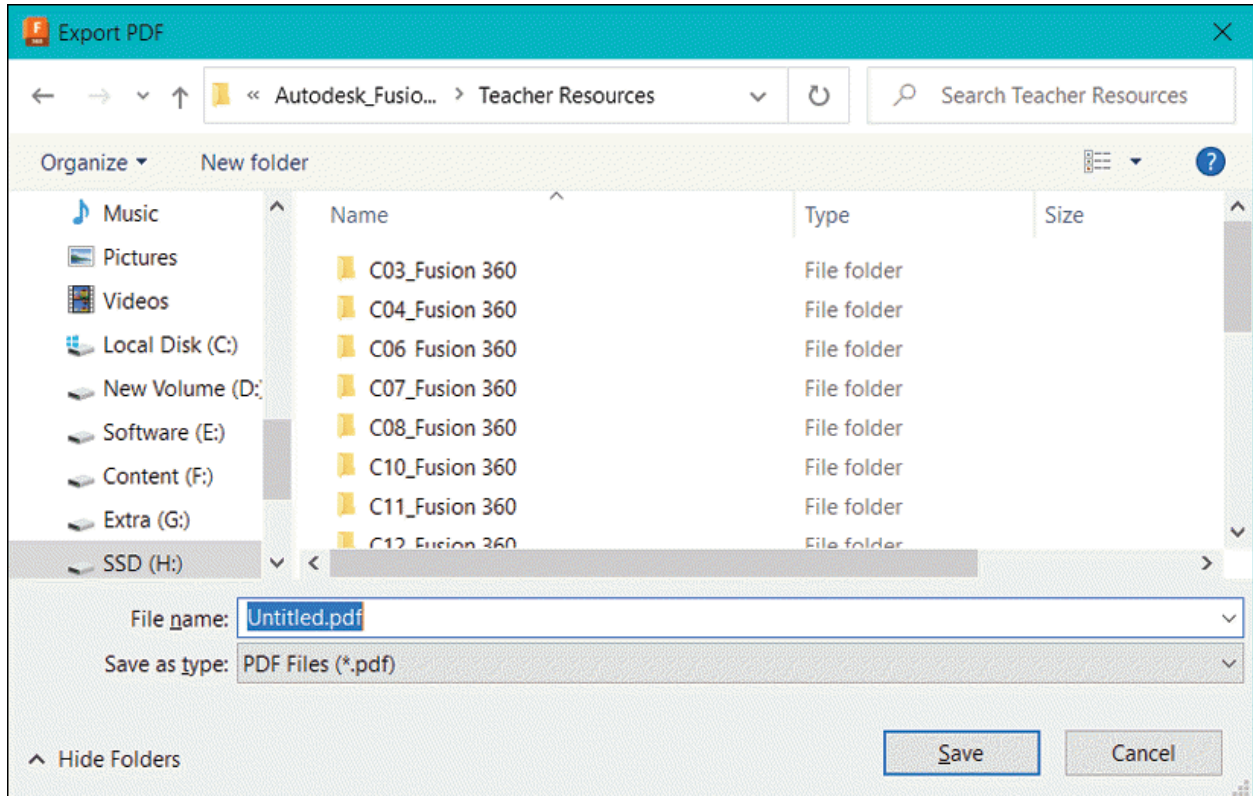


Figure-134. Export PDF dialog box for saving file

- Click in the **File name** edit box and specify desired name for pdf file.
- Browse to desired location where you want to save the file and click on the **Save** button. The PDF file will be created and opened if you had selected the **Open PDF** check box from **EXPORT PDF** dialog box.

EXPORTING DWG

The **Export DWG** tool is used to create an output file of drawing in **DWG** format. The procedure to use this tool is discussed next.

- Click on the **Export DWG** tool from **EXPORT** drop-down

in the **Toolbar**; refer to [Figure-135](#). The **EXPORT DWG** dialog box will be displayed to save the file; refer to [Figure-136](#).

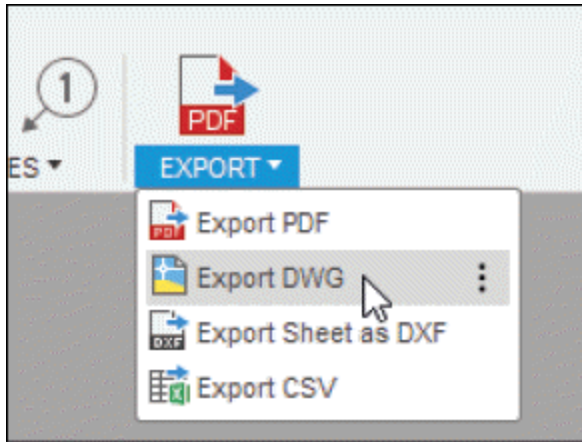


Figure-135. Export DWG tool

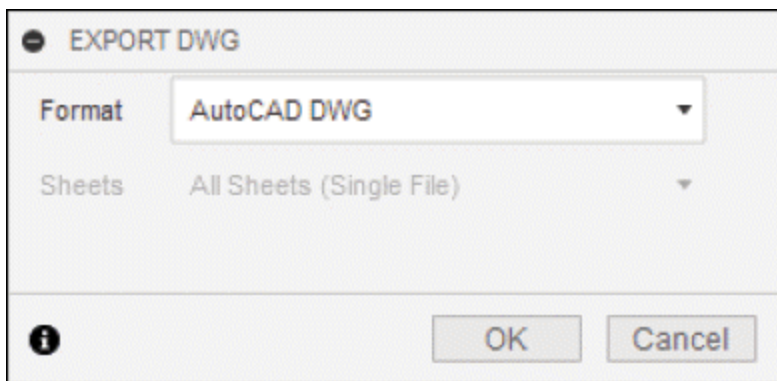


Figure-136. EXPORT DWG dialog box

- Select **Simplified DWG** option from **Format** drop-down of dialog box to export the contents of the current sheet as a DWG file, including views, title block, border, and all annotations as 2D geometry in Model Space.
- Select **AutoCAD DWG** option from **Format** drop-down to export the contents of all sheets to a DWG file. Each sheet is exported as a separate Layout tab.

- Click on the **Sheets** drop-down from **EXPORT DWG** dialog box and select required sheet for output. You can also use **SHIFT** key or **CTRL** key to select multiple sheets.
- After specifying the parameters, click on the **OK** button from **EXPORT DWG** dialog box. The **Output DWG** dialog box will be displayed to save the file; refer to [Figure-137](#).

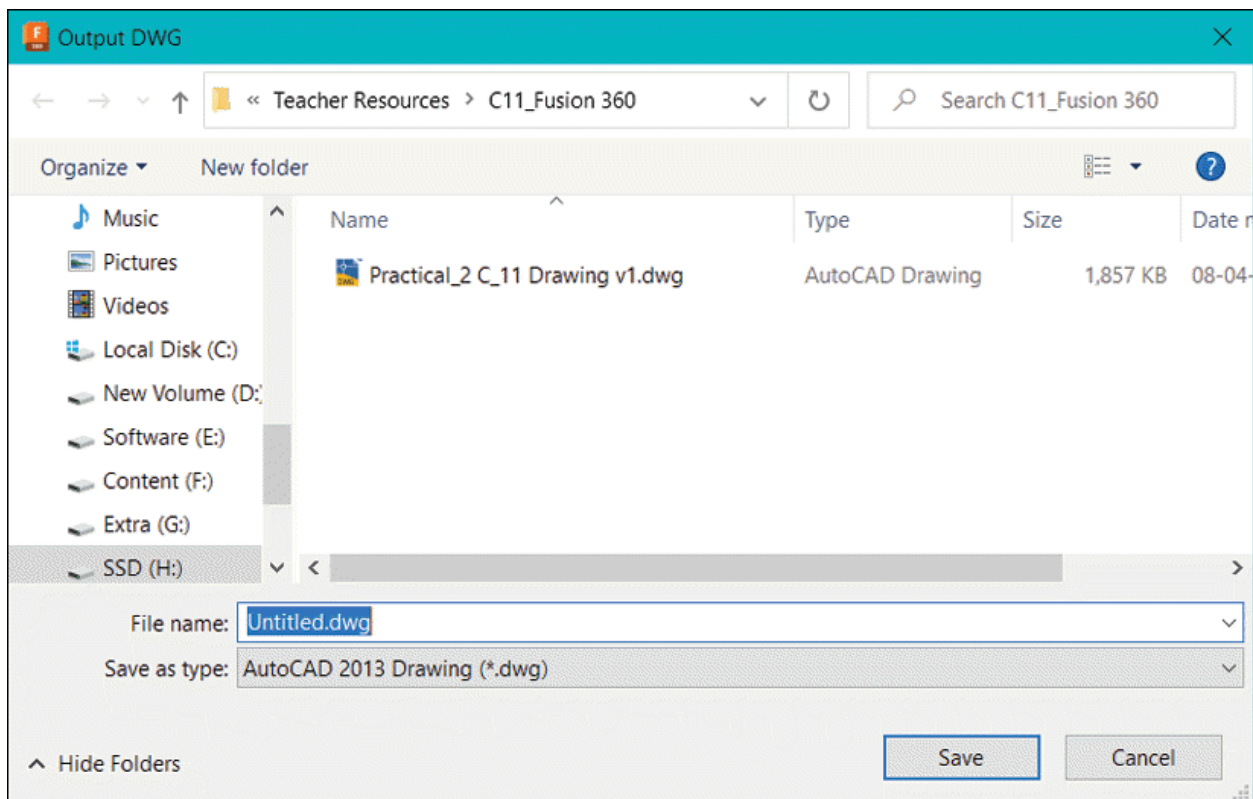


Figure-137. Output DWG dialog box for saving the file

- Select desired location to save the file.
- After specifying the parameter, click on the **Save** button from **Output DWG** dialog box to save the file.

EXPORTING DXF FILE

The **Export Sheet as DXF** tool from **EXPORT** drop-down in

the **Toolbar** exports the sheets in a drawing as a DXF file. The procedure to use this tool is similar to the **Simplified DWG** option in the **Format** drop-down of **EXPORT DWG** dialog box.

EXPORTING CSV FILE

The **Export CSV** tool is used to create the output of the current file in CSV format. CSV is a format used by database management software and point cloud software. The procedure to use this tool is discussed next.

- Click on the **Export CSV** tool from **EXPORT** drop-down in the **Toolbar**; refer to [Figure-138](#).

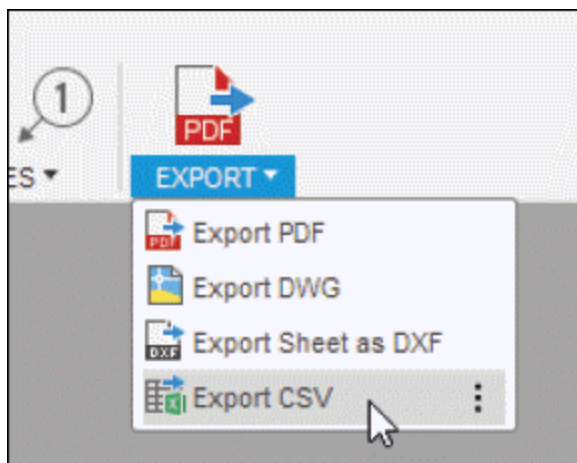


Figure-138. Export CSV tool

- Select the table, parts list, or bend table in the Model Space. The **Output Table** dialog box will be displayed; refer to [Figure-139](#).

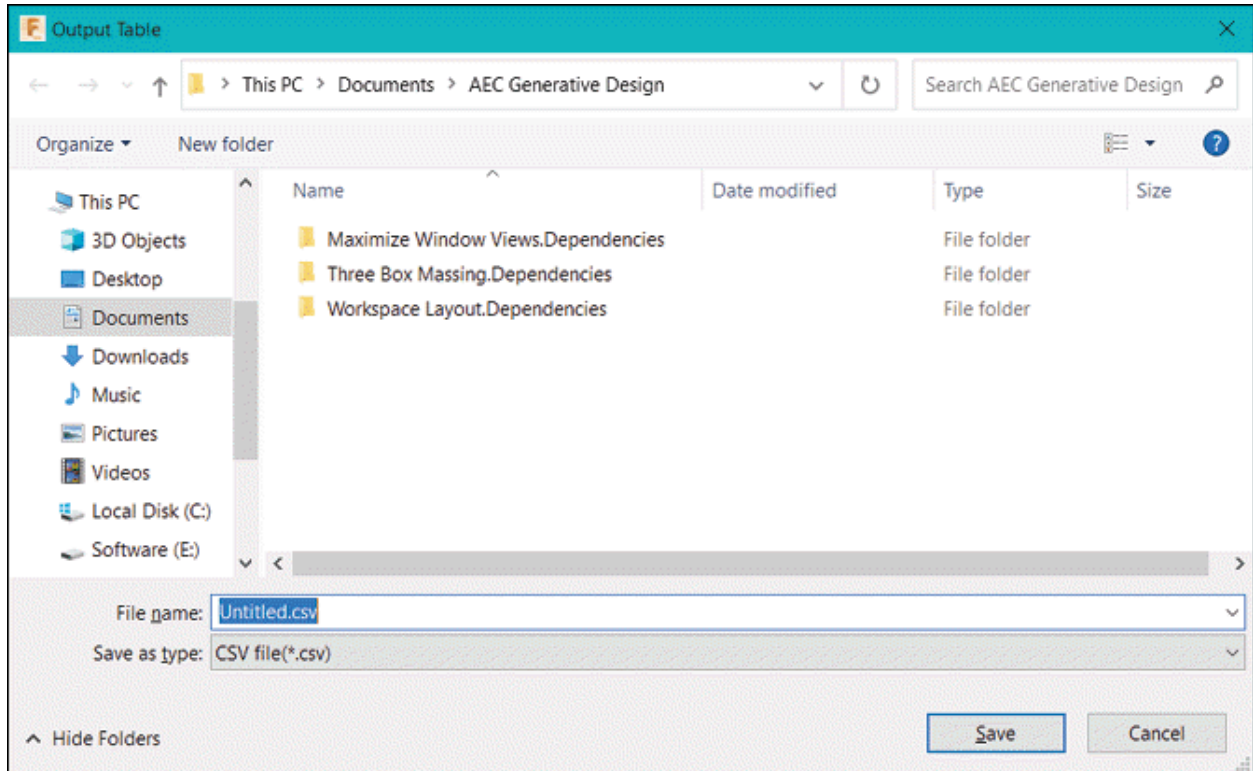


Figure-139. Output Table dialog box

- Select desired name and location of file.
- After specifying the parameters, click on the **Save** button to save the file.

PRACTICAL 1

Generating part drawing for the model as shown in [Figure-140](#).

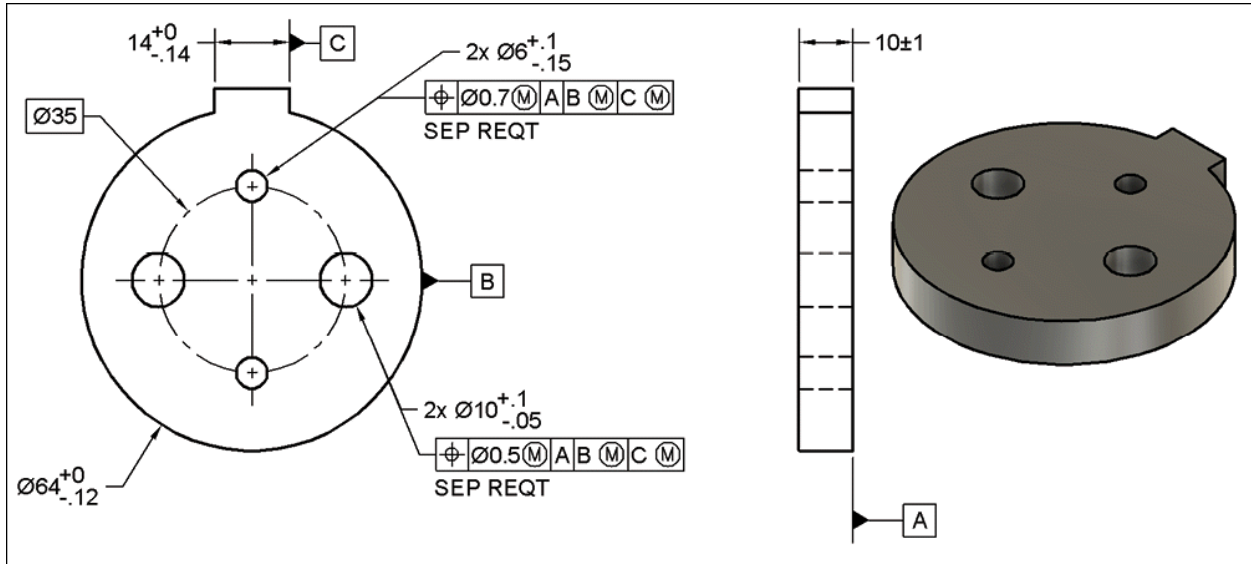


Figure-140. Practical 1 Drawing

Starting a New Drawing using Part

- Set the parameters as shown in [Figure-141](#) for preference.

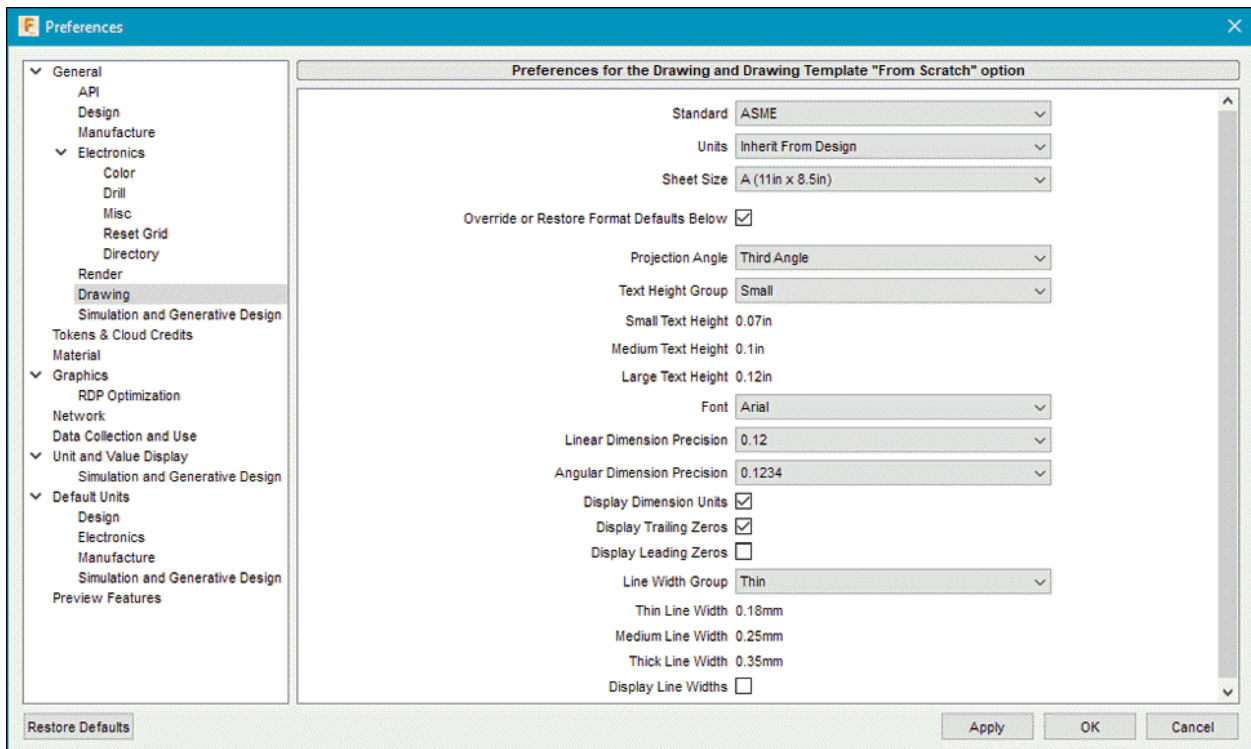


Figure-141. Specifying drawing preferences

- Open the model file for Practical 1 of Chapter 11 in the

resource kit. The model will be displayed in the canvas.

- Click on the **From Design** option from **New Drawing** cascading menu of the **File** menu; refer to [Figure-142](#). The **CREATE DRAWING** dialog box will be displayed.

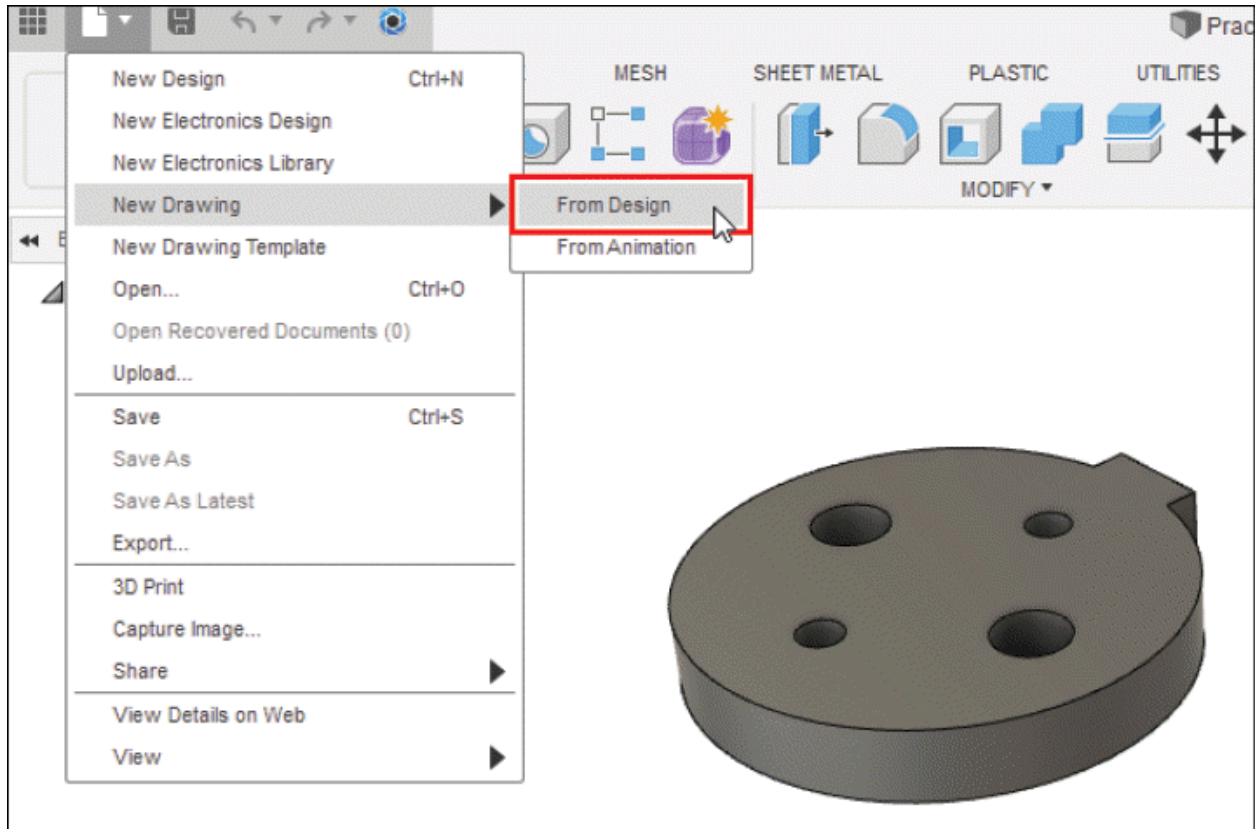


Figure-142. Starting new drawing

- Make sure **ASME** is selected in the **Standard** drop-down of **CREATE DRAWING** dialog box and click on the **OK** button. The **DRAWING VIEW** dialog box will be displayed with preview of drawing view; refer to [Figure-143](#).

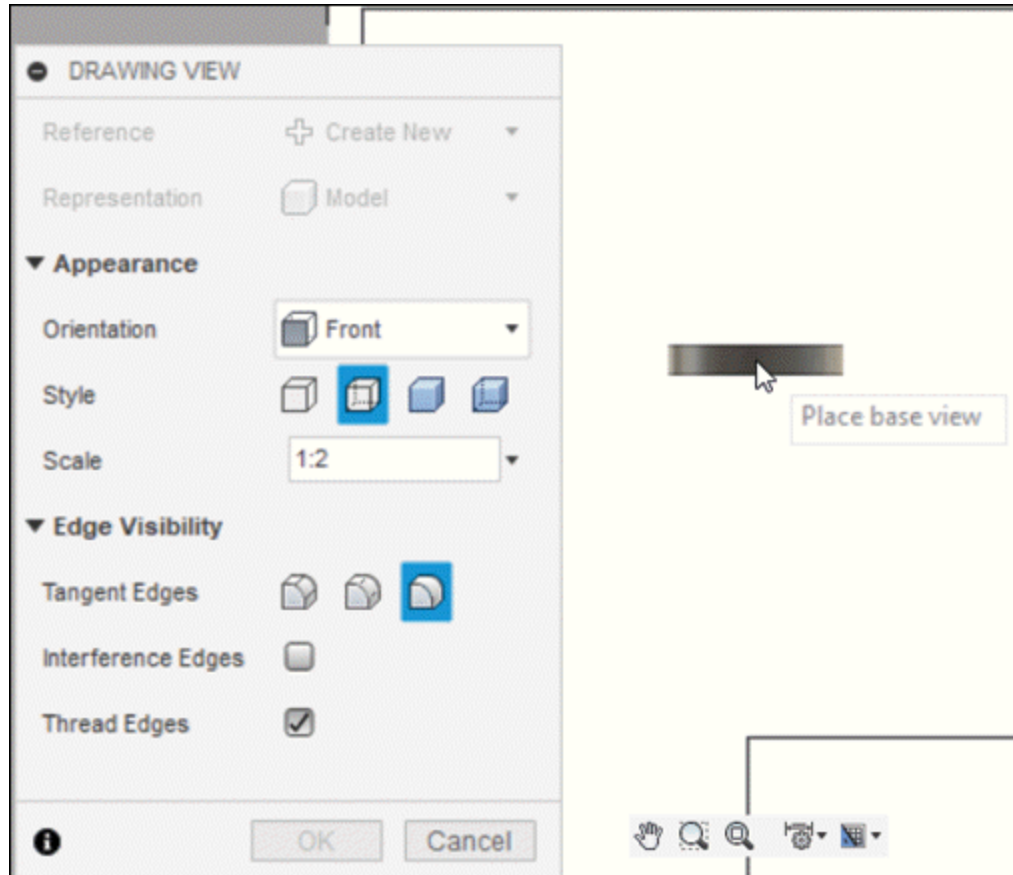


Figure-143. Preview of drawing view

Placing Views

- Select the **Top** option from the **Orientation** drop-down, **Visible and Hidden Edges** button from **Style** section, and **1:1** in **Scale** drop-down from the dialog box.
- Click at desired location to place the view; refer to [Figure-144](#).
- Click on the **OK** button from the dialog box to create the view.
- Click on the **Projected View** tool from the **CREATE** panel in the **Toolbar**. You will be asked to select the base view.

- Select the view earlier placed and move the cursor towards right. The view will get attached to the cursor; refer to [Figure-145](#).

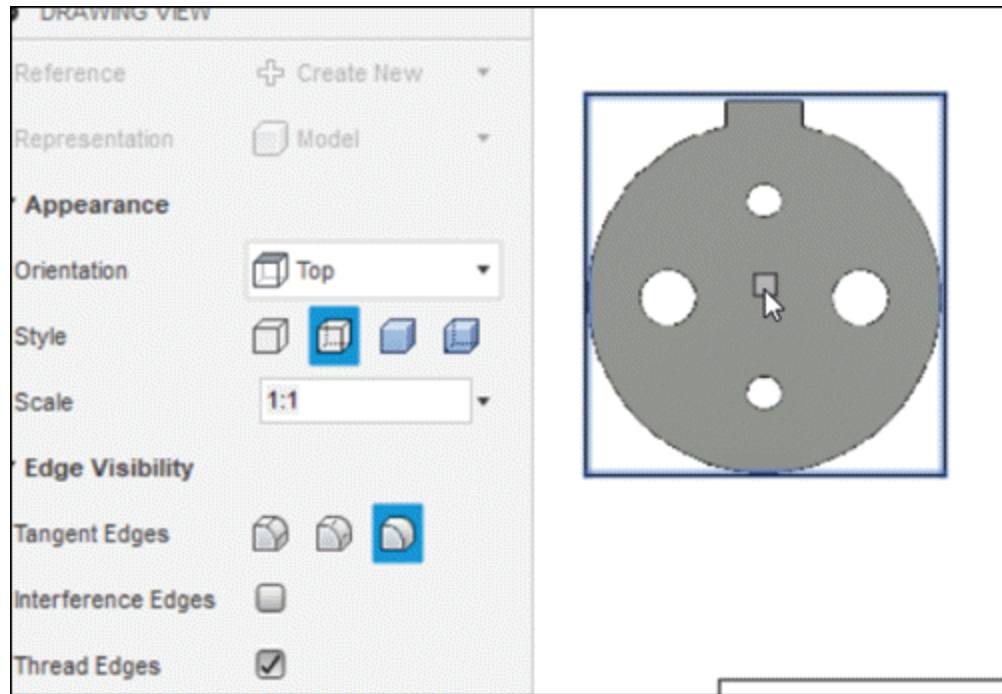


Figure-144. Placing base view

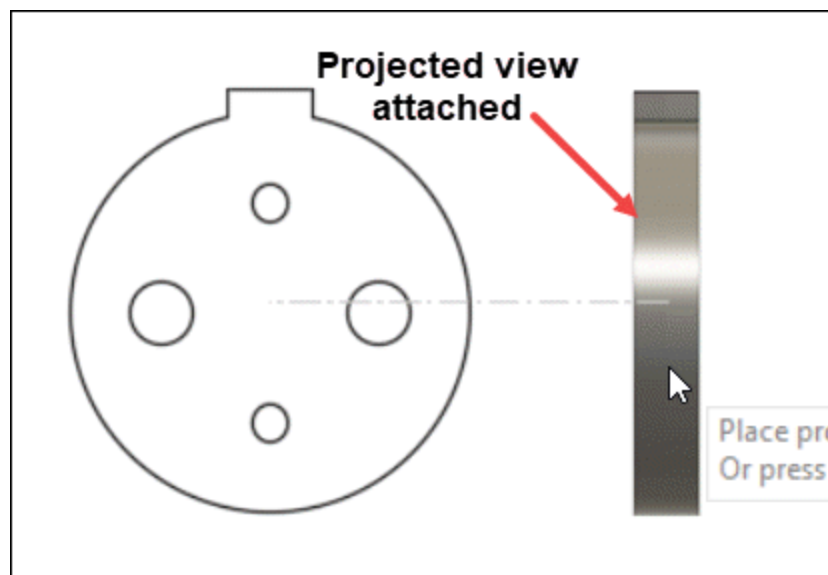


Figure-145. Projected view attached to cursor

- Click at desired location to place the view and press **ESC** to exit.

Applying Annotations

- Click on the **Center Mark Pattern** tool from the **GEOMETRY** panel in the **Toolbar**. The **CENTER MARK PATTERN** dialog box will be displayed.
- Click on one of the holes in the base view and select the **Center Mark** check box. The preview of center mark pattern will be displayed; refer to [Figure-146](#).

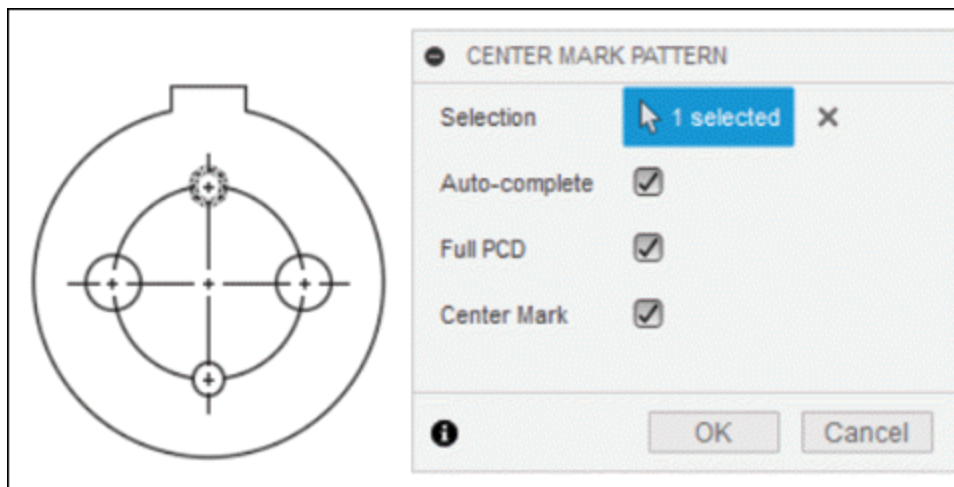


Figure-146. Preview of center mark pattern

- Click on the **OK** button from the dialog box.
- Click on the **Dimension** tool from the **DIMENSIONS** panel in the **Toolbar** and apply dimensions as shown in [Figure-147](#).
- Click on the **Datum Identifier** tool from the **SYMBOLS** panel in the **Toolbar** and place the datum identifiers as shown in [Figure-148](#).

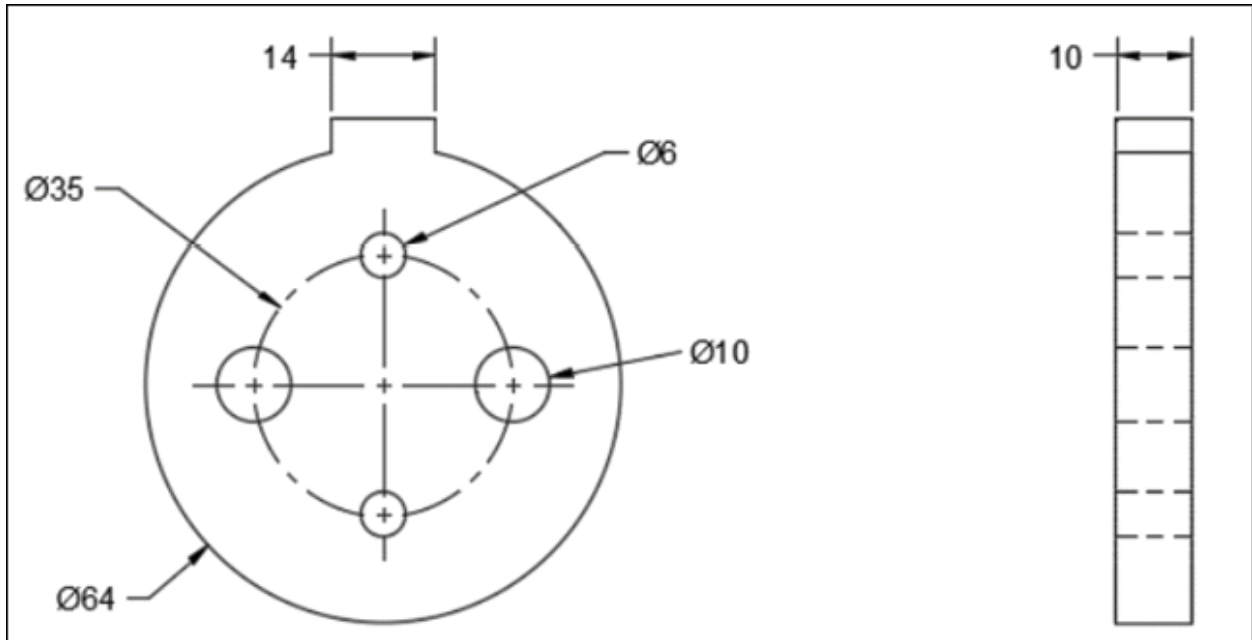


Figure-147. Applying dimensions

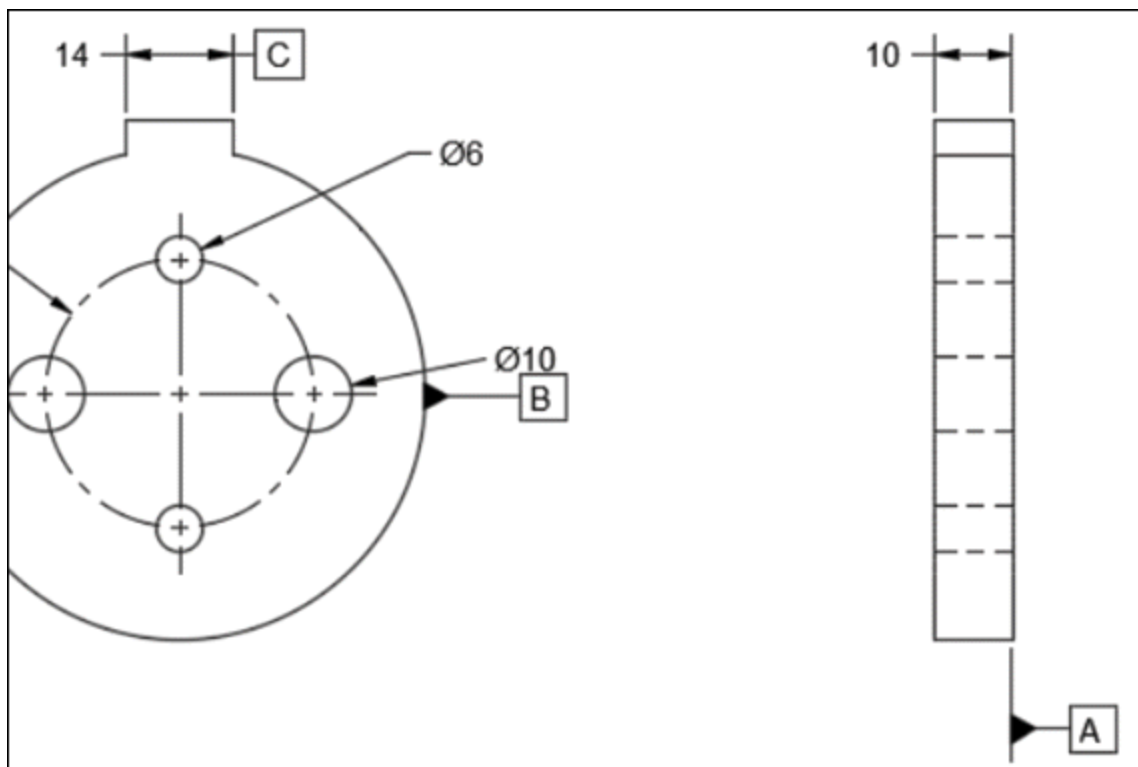


Figure-148. Applying datum identifier

Adding Tolerances to Dimension

- Double-click on the dimension with **10** value. The **DIMENSION** dialog box will be displayed; refer to [Figure-149](#).

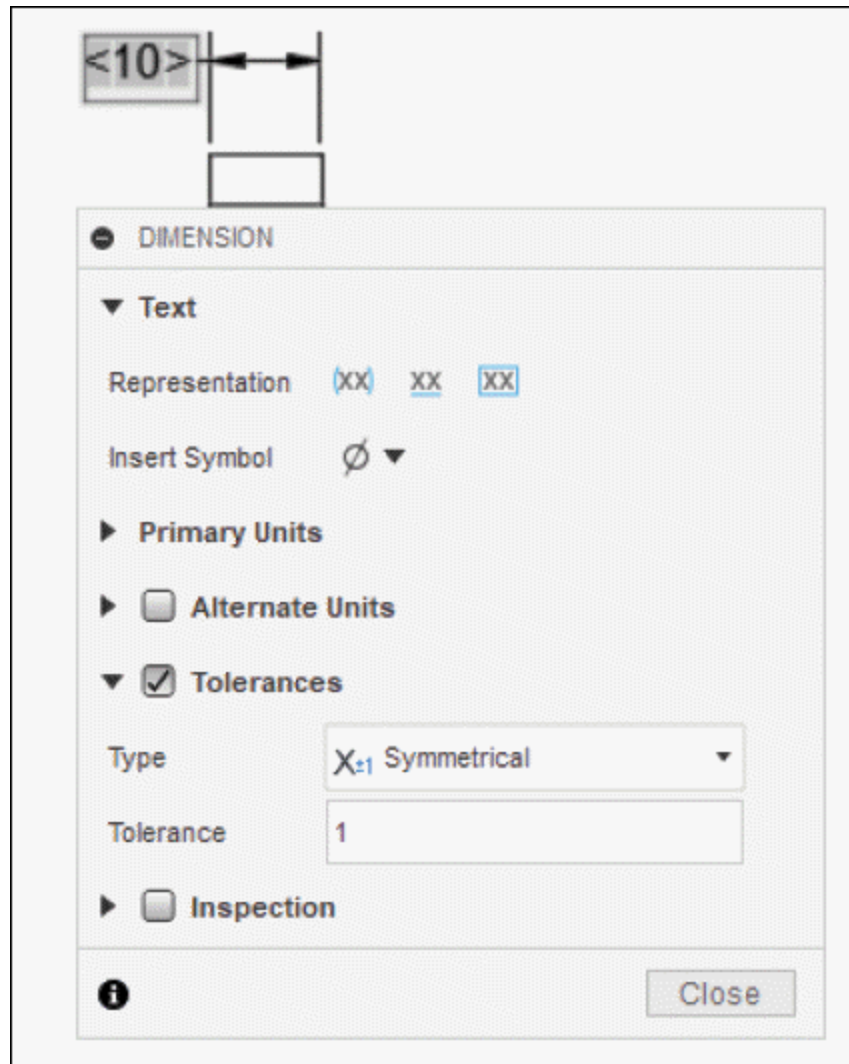


Figure-149. DIMENSION dialog box

- Select the **Symmetrical** option from the **Type** drop-down in **Tolerances** node of dialog box and specify the value as **1** in **Tolerance** edit box. Click on the **Close** button to exit the dialog box.
- Similarly, set the tolerances for other dimensions; refer to

[Figure-150.](#)

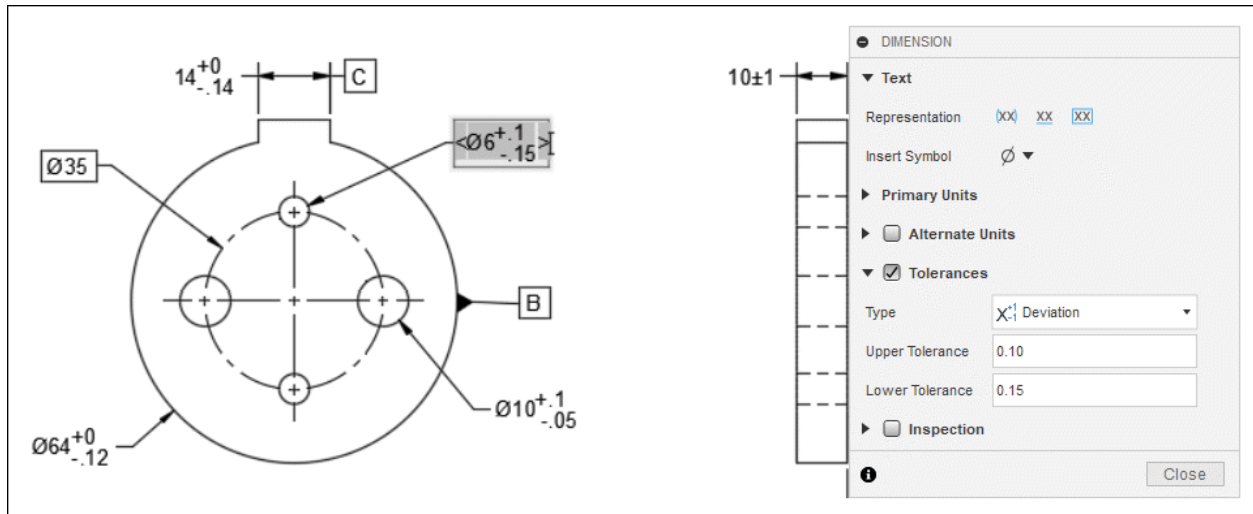
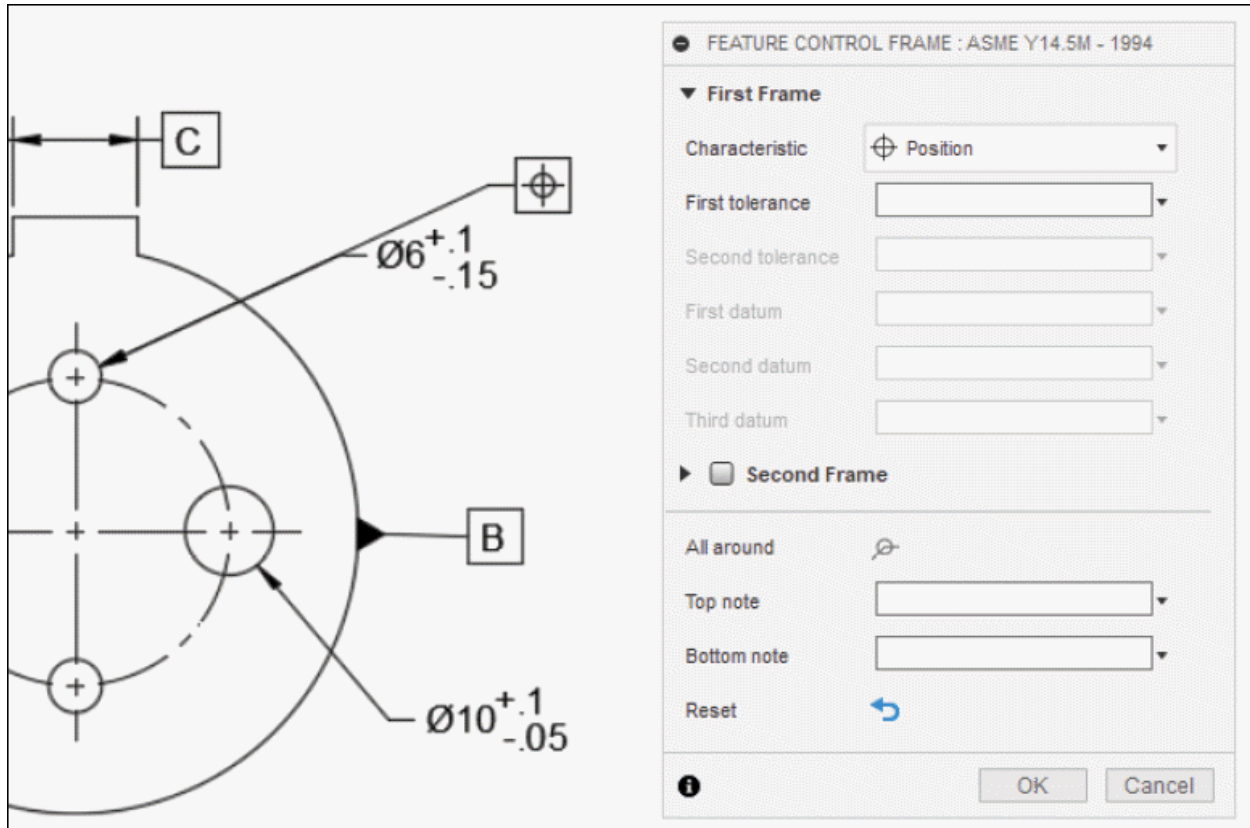


Figure-150. Applying tolerances

- Click on the **Feature Control Frame** tool from the **SYMBOLS** panel in the **Toolbar** and select the hole of diameter **6**. The control frame will get attached to cursor.
- Click at desired location to place the control frame and press **ENTER**. The **FEATURE CONTROL FRAME** dialog box will be displayed; refer to [Figure-151](#).



- Specify the parameters as shown in [Figure-152](#) and click on the **Close** button.

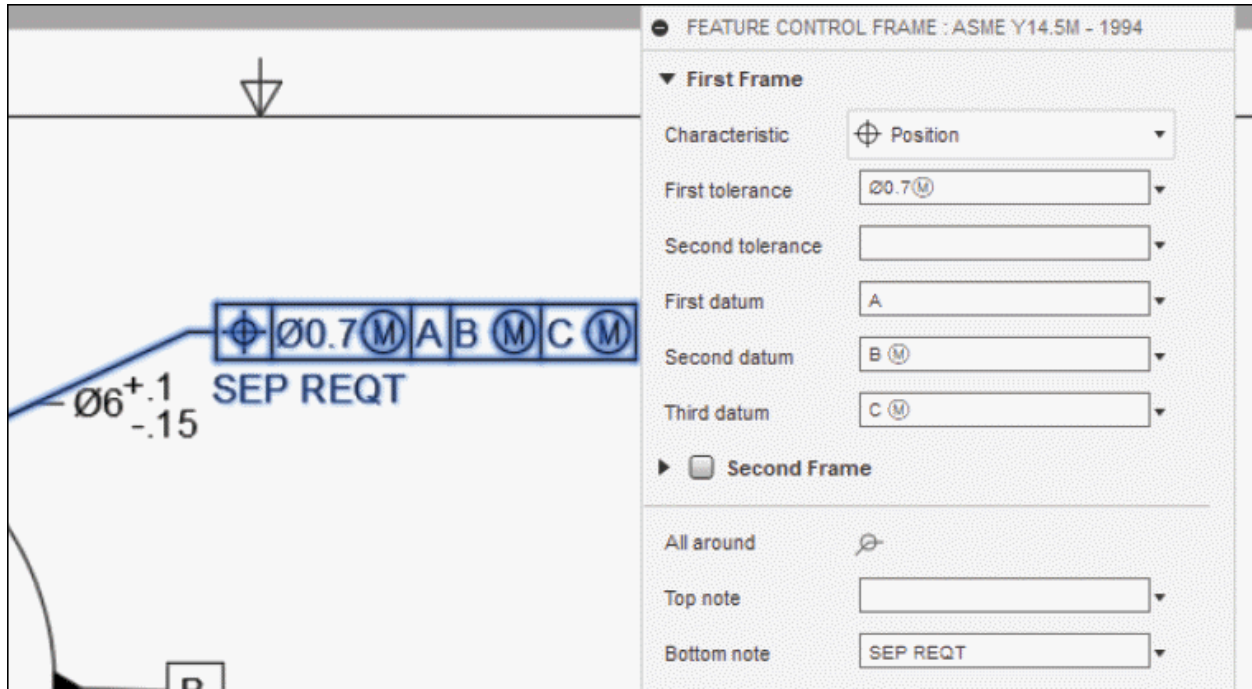


Figure-152. Creating feature control frame

Similarly, define position feature control frame for other hole and place another base view as 3D NE Isometric view; refer to [Figure-153](#).

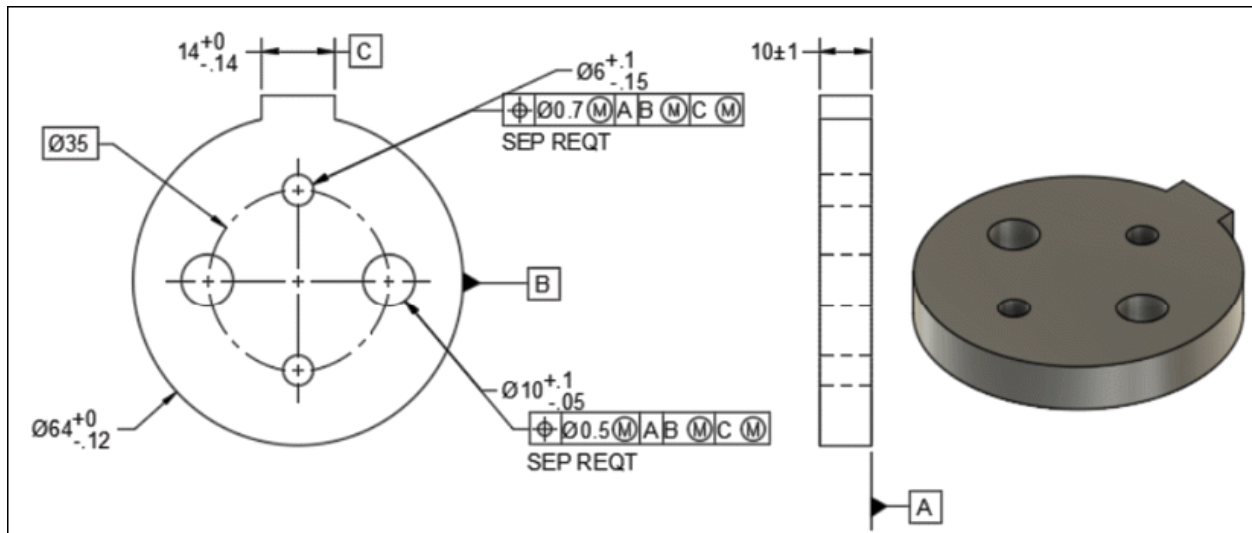


Figure-153. Completed drawing

PRACTICAL 2

Generate part drawing for the model as shown in [Figure-154](#).

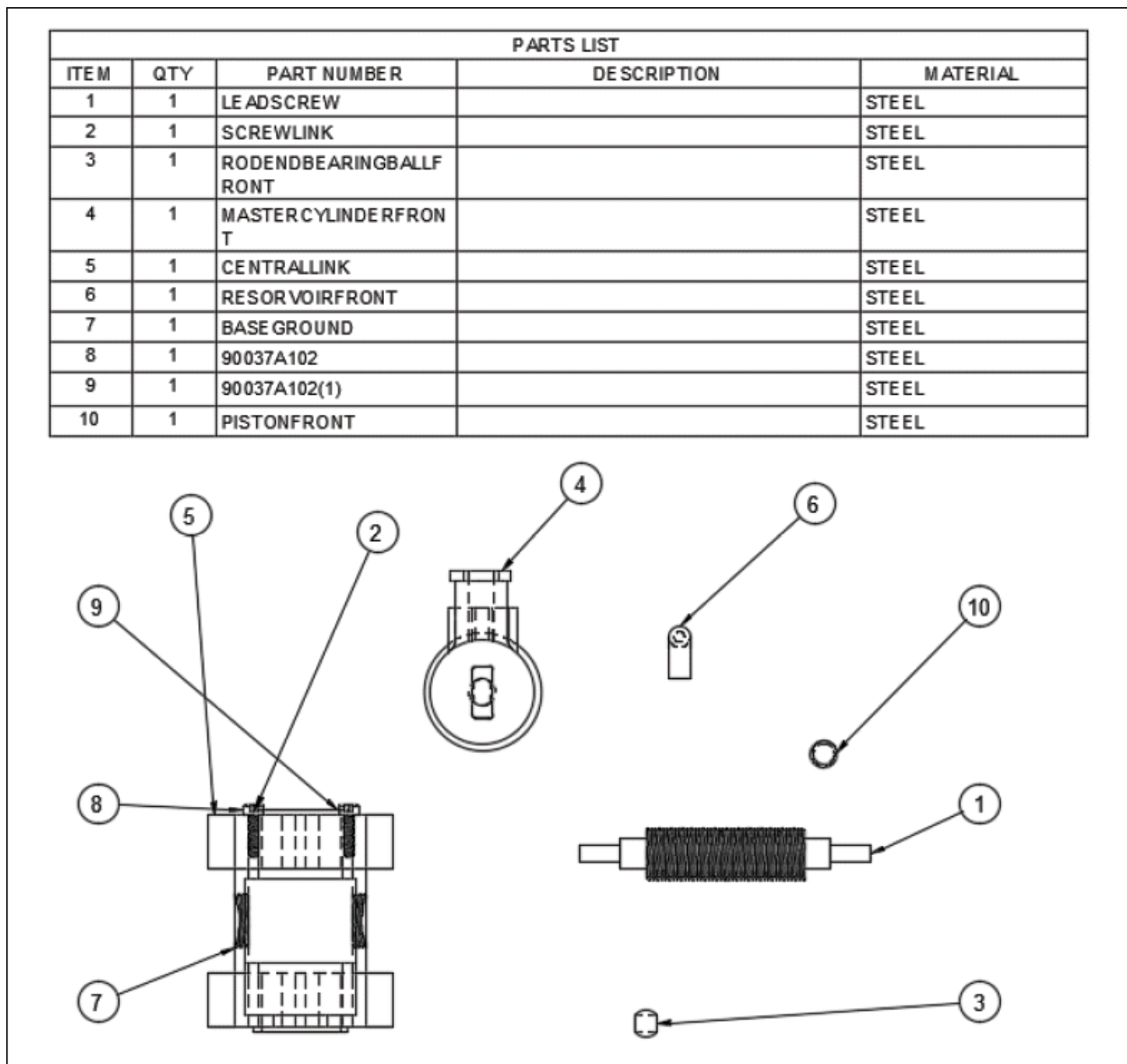


Figure-154. Practical 2 drawing

- Open the model file for Practical 2 of Chapter 11 in the resource kit. The model will be displayed in the canvas.
- Click on the **From Design** option from **New Drawing** cascading menu of the **File** menu; refer to [Figure-155](#). The

CREATE DRAWING dialog box will be displayed.

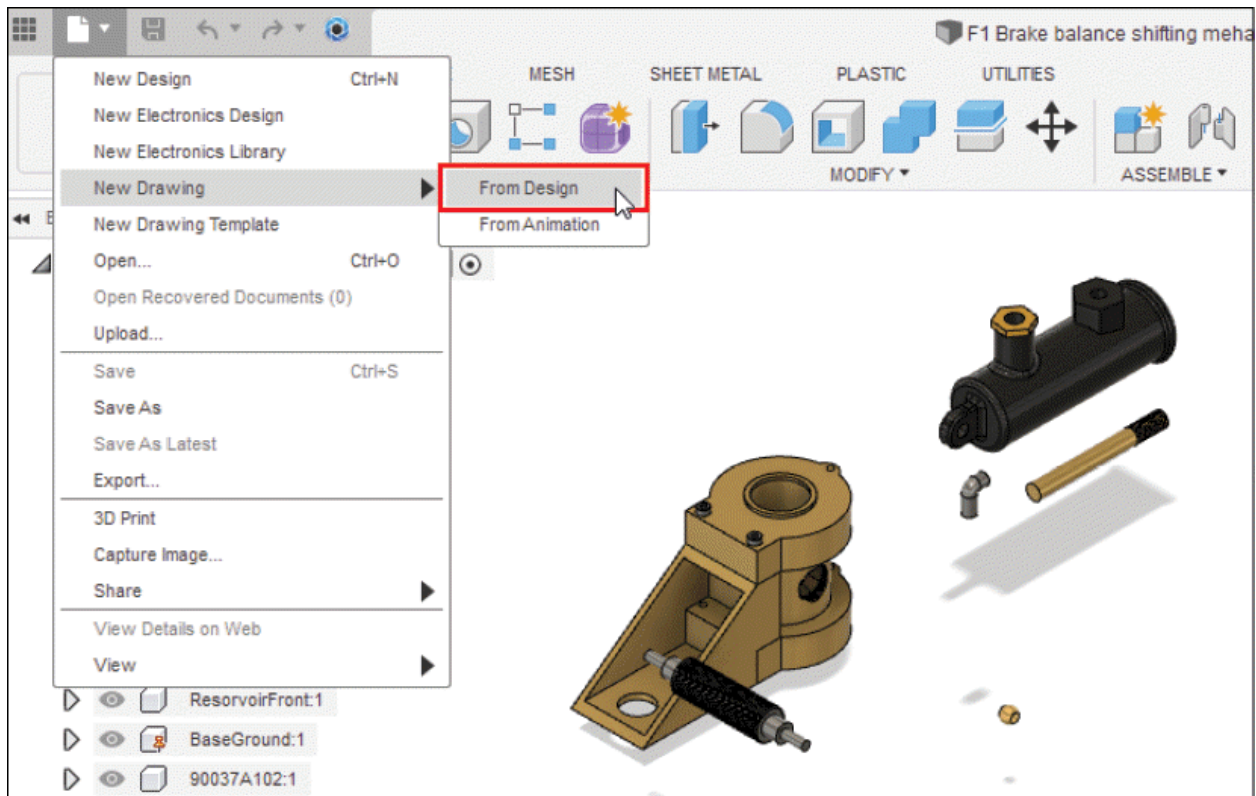


Figure-155. Starting new drawing

- Make sure **ASME** is selected in the **Standard** drop-down of **CREATE DRAWING** dialog box and click on the **OK** button. The **DRAWING VIEW** dialog box will be displayed with preview of drawing view; refer to [Figure-156](#).

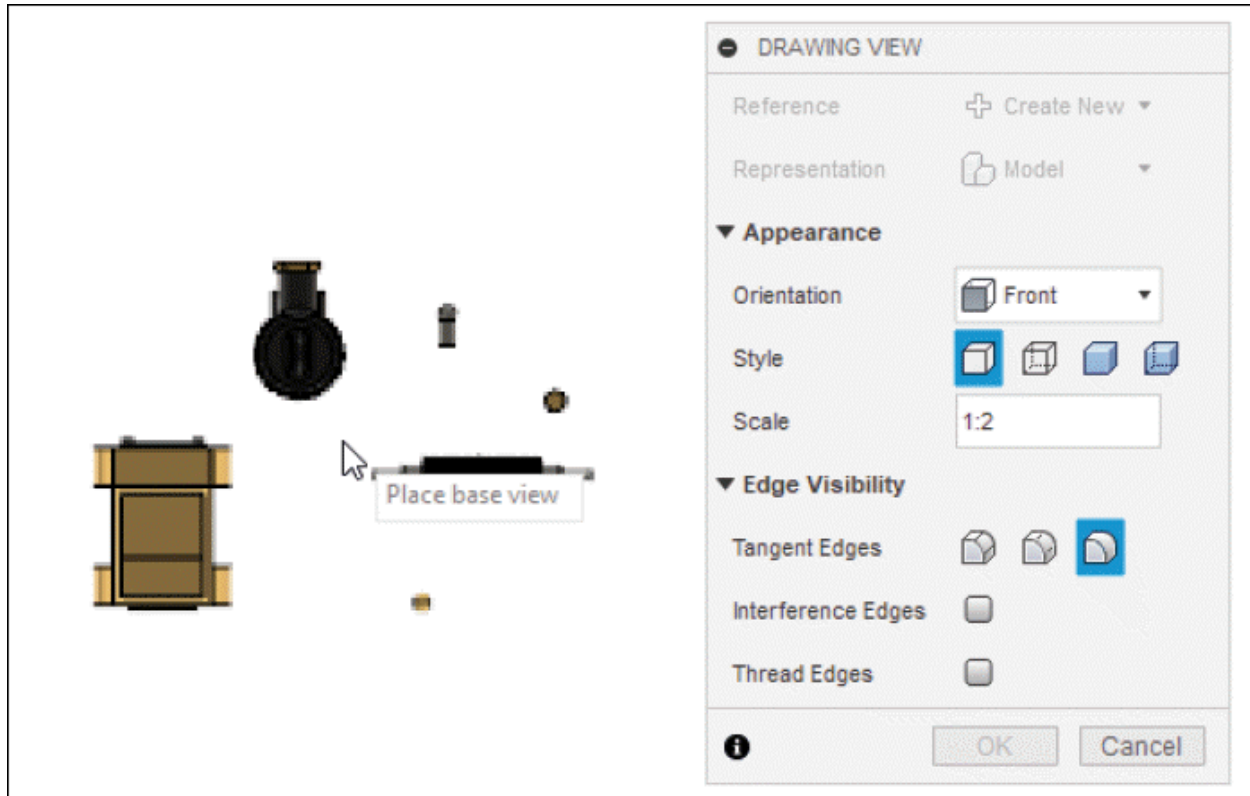


Figure-156. Preview of drawing view

Placing Views

- Select the **Front** option from the **Orientation** drop-down, **Visible and Hidden Edges** button from **Style** section, and **1:1** in **Scale** drop-down from the dialog box.
- Click at desired location to place the view; refer to [Figure-157](#).

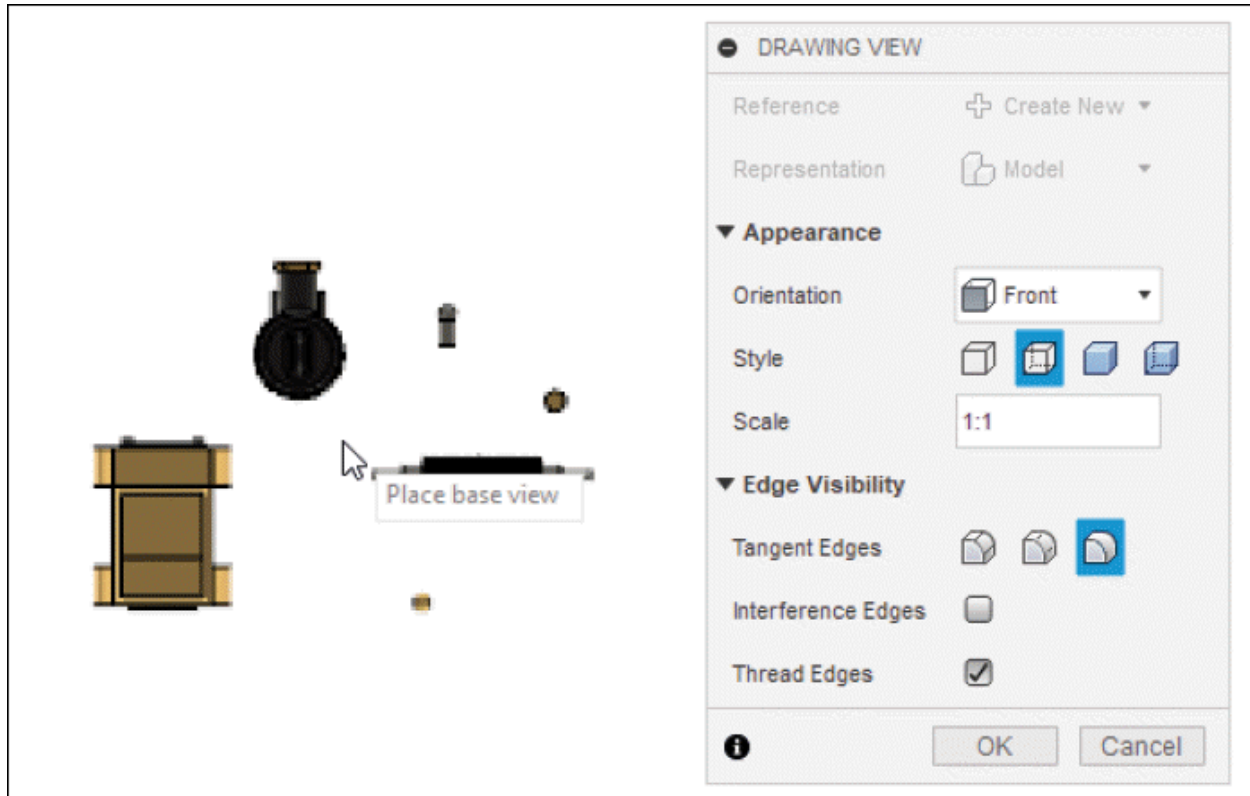


Figure-157. Placing base view

- Click on the **OK** button from the dialog box to create the view; refer to [Figure-158](#).

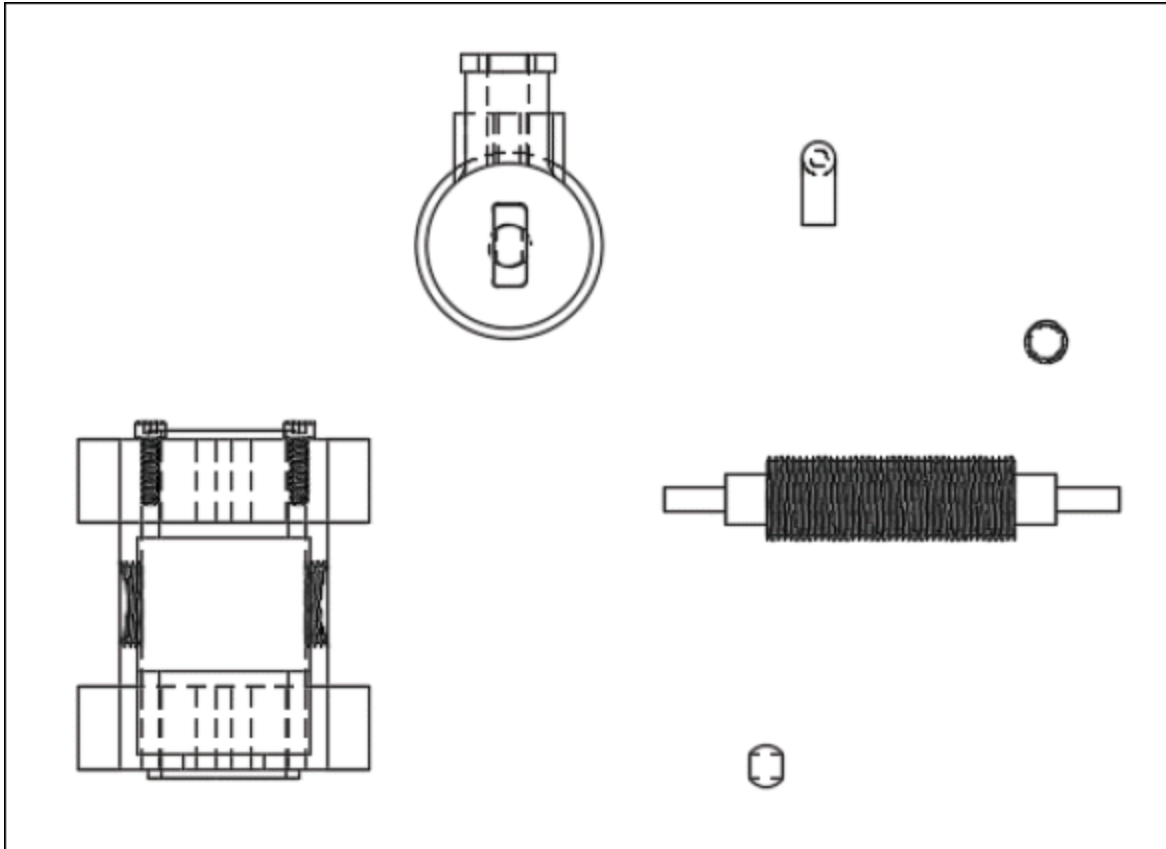


Figure-158. Base view created

Creating Table

- Click on the **Table** tool from **TABLES** drop-down in the **Toolbar**. The **TABLE** dialog box will be displayed along with attached empty table; refer to [Figure-159](#).

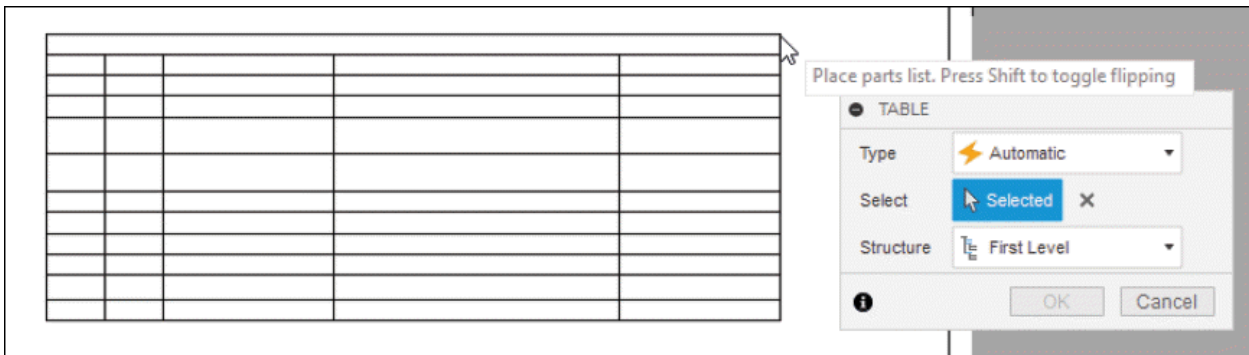


Figure-159. TABLE dialog box along with empty table

- Select **Parts List** option from **Type** drop-down in the dialog box and the current drawing view will be selected automatically.
- Click at desired location in the drawing sheet to place the parts list. Parts list will be generated with balloons; refer to [Figure-160](#).

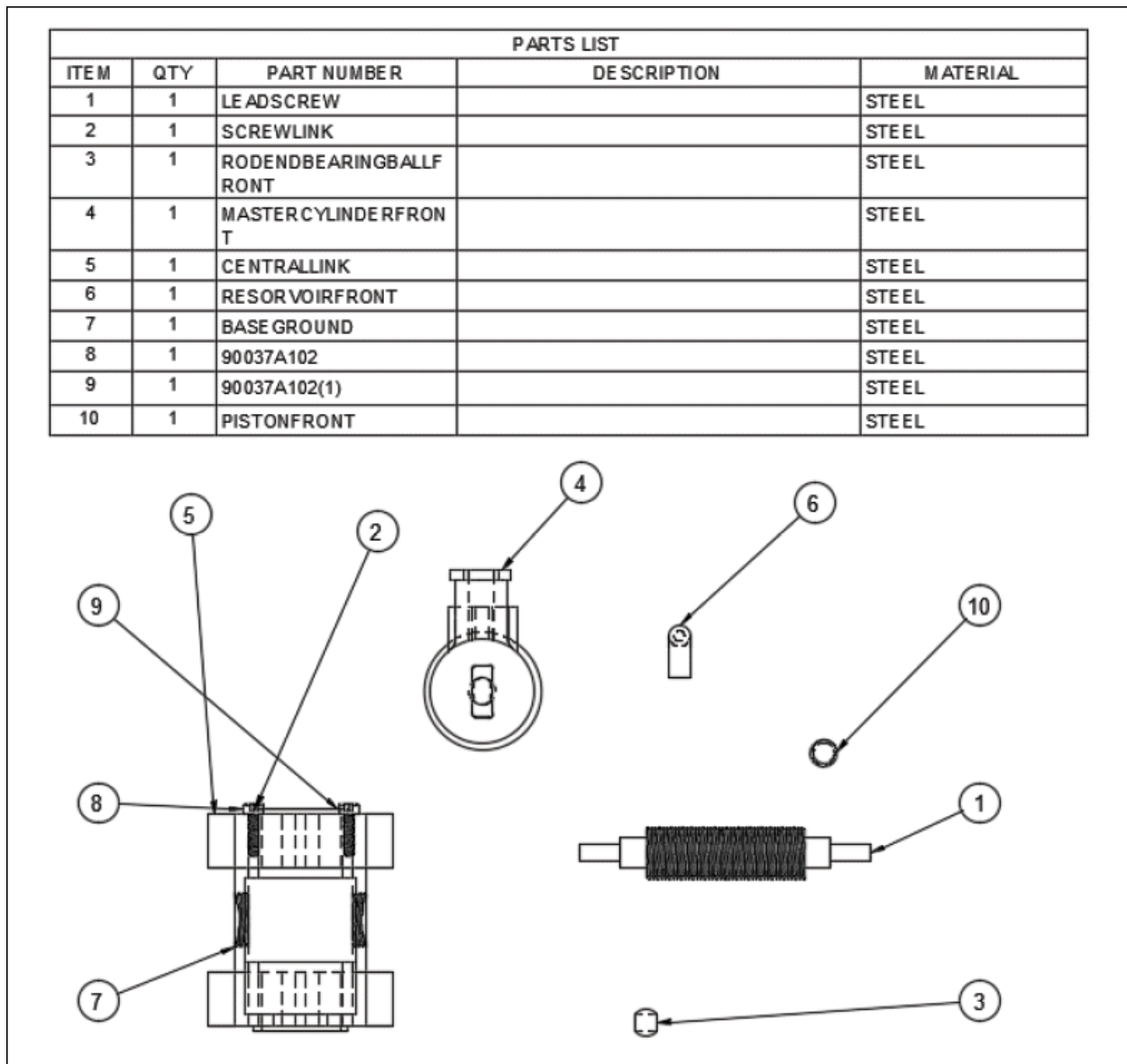


Figure-160. Parts list of assembly generated

PRACTICE

Create drawings for the models discussed in practical and practice sections of Chapter 5 of this book.

SELF ASSESSMENT

Q1. The tool is used to create drawing views generated from an existing base or drawing view.

Q2. What is the difference between **ASME** and **ISO** option in **CREATE DRAWING** dialog box?

Q3. In which view of drawing, hatching is automatically created?

Q4. Text can be aligned by **Justification** section. (T/F)

Q5. The drawing view can be move from one place to another with the help of **Move** button. (T/F)

Q6. What is the use of **Edge extension** tool?

Q7. Two parallel edges should be selected to create the extension. (T/F)

Q8. When is the baseline dimensioning required?

Q9. What is Feature Control Frame?

Q10. What is the use of **Datum Identifier** tool?

Q11. The tool is used to create an output file of drawing in DWG format.

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