Autodesk Fusion 360 Book (2nd Edition)

COLORED

By

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(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** (2nd Edition) is the second edition of our series on Autodesk Fusion 360. The book is updated on Autodesk Fusion 360 Ultimate, Student V 2.0.4116. 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, Rendering & Animation, Sculpting, Mesh Design, CAM, Simulation, Sheetmetal, 3D printing, 3D PDFs, 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 CAM, Simulation, and Mesh 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 easy 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 **1700** small and large illustrations that make the learning process effective.

Tutorial point of view

At the end of concept's explanation, the tutorial make 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

Free 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.

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*** 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 in Design and Development department.

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***
- ***File Menu***
- ***User Account drop-down***
- ***Data Panel***
- ***Simulation Browser***

OVERVIEW OF AUTODESK FUSION 360

Autodesk Fusion 360 is a new 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 3D CAD, CAM, and CAE tool of its kind. It connects your entire product development process in a single cloud-based platform that works on both Mac and PC; refer to Figure-1.

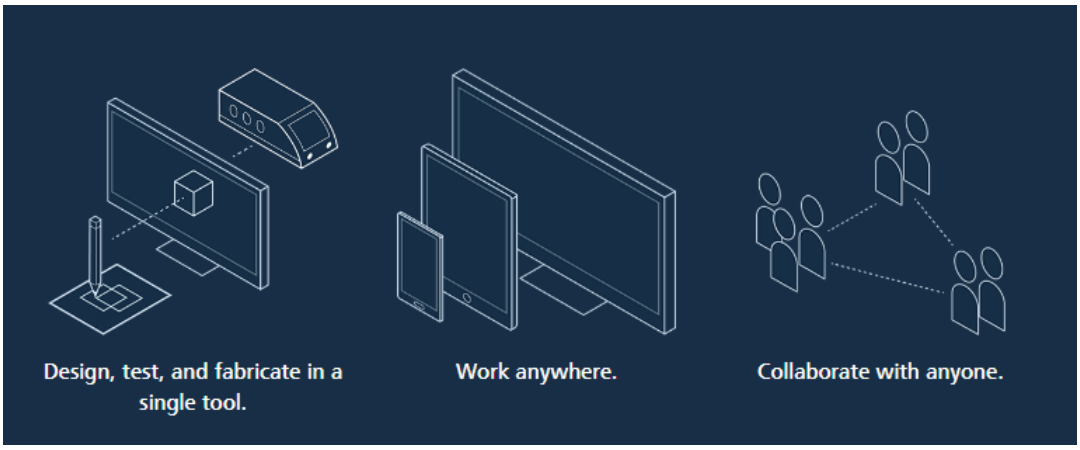

Figure-1. Overview

It combines industrial and 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 is a good software for those individual and companies who want to develop themselves. This software is much affordable than any other software offered by Autodesk. To use this software, one need to pay the 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 his saved file and also able to share files with anyone from anywhere as long as he has the software and good internet connection. Also, the pricing of this software is cost effective so anyone can use it for manufacturing of tools and parts. Autodesk Fusion 360 has been built to work in multi body manner: both parts and assemblies built 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 login to **http://www.autodesk.com/products/fusion-360/students-teachers-educators** as shown in Figure-2.

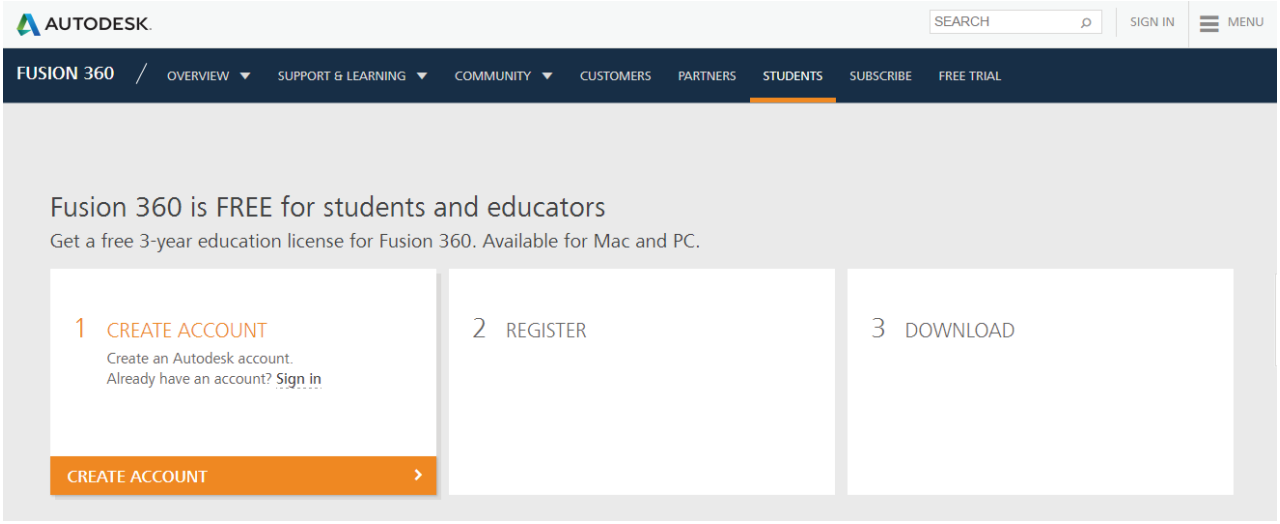

Figure-2. Autodesk website

• Click on the **Create Account** button and on next page, select your Country and Education role. In **Country** drop-down, select your Country and in **Education role** drop-down, you need to select **Student** (There is free subscription for students with license term of 3 years) as shown in Figure-3.

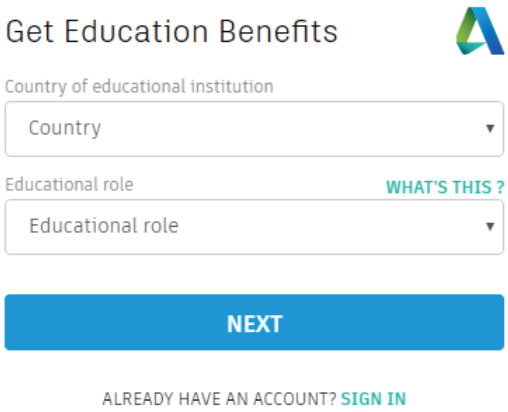

Figure-3. Select country and education role

• After filling the details, click on **NEXT** button. The next page of this website opens. In this page, you need to fill your personal details and then click on **Create Account** as shown in Figure-4.

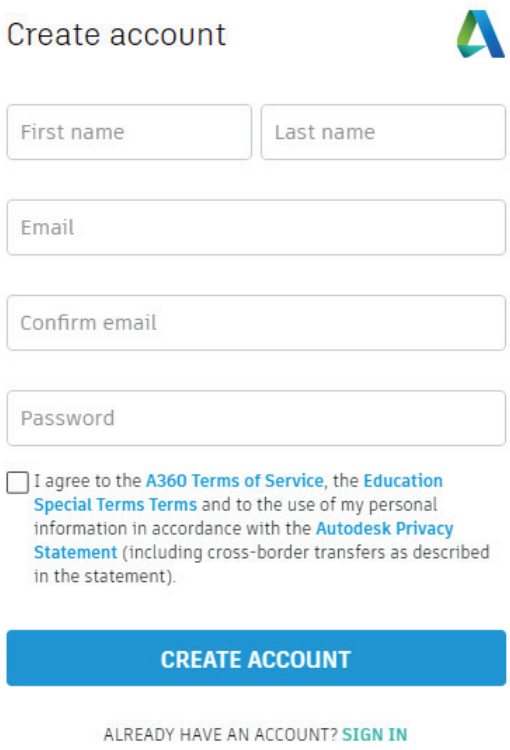

Figure-4. Creating account

- After creating account, you need to verify your E-mail address by login into your E-mail account. After verifying E-mail address, you need to give your **Education details**.
- After completion of these processes, you need to sign-in your Autodesk Account and search the **Autodesk Fusion 360 software**.
- Download the **Autodesk Fusion 360 software**.
- Open the downloaded setup file and follow the instructions as per the setup instruction.
- The software will be installed in 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, click on **All Programs** folder, and then on **Autodesk folder**. Select the Autodesk Fusion 360 icon; refer to Figure-5.

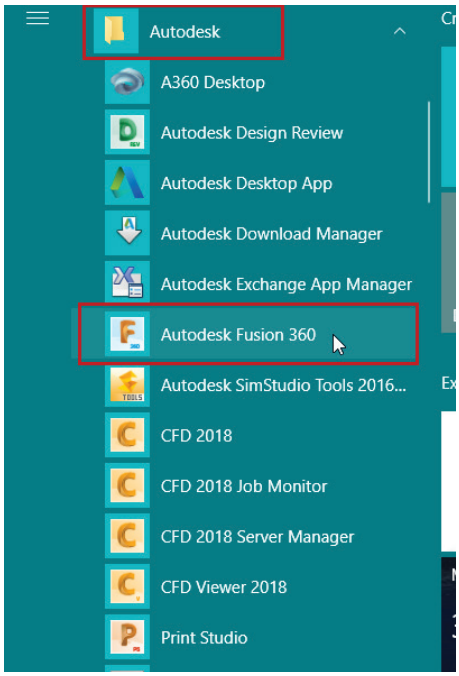

Figure-5. Start Menu

- 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, then right-click on the **Autodesk Fusion 360** icon in the **Start** menu and select the **Send To-> Desktop(Create icon)** option from the shortcut menu displayed. If you are using Windows 8 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-6.

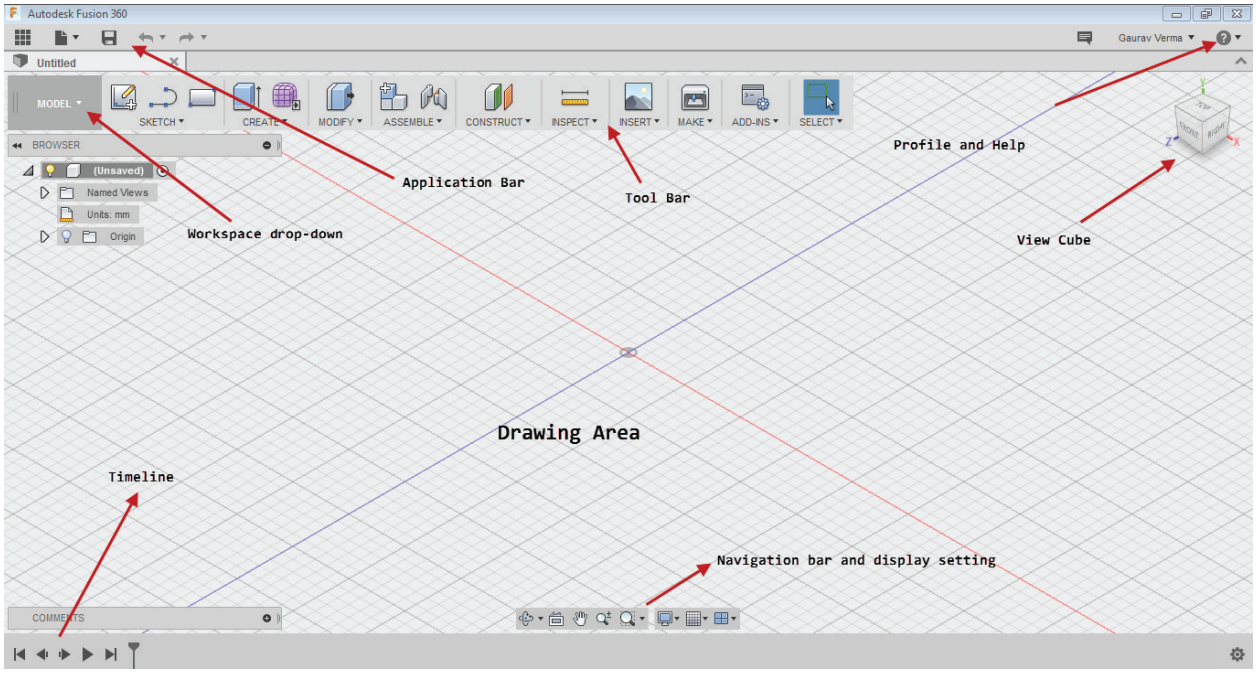

Figure-6. Autodesk Fusion 360 application Window

STARTING A NEW DOCUMENT

- Click on the **File** drop-down and select the **New design** tool as shown in Figure-7. A new document will open.

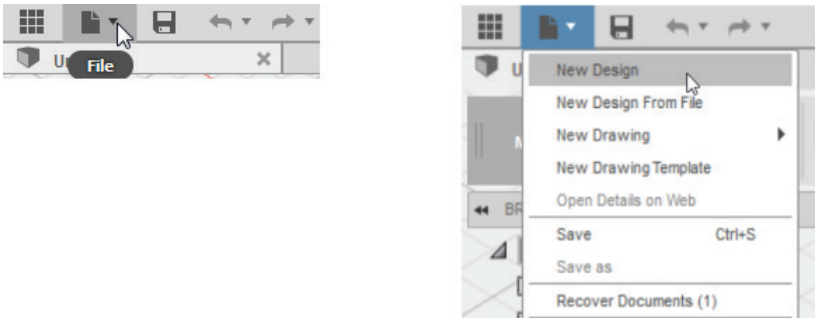

Figure-7. File Menu

- Select the desired workspace from the **Workspace** drop-down; refer to Figure-8.

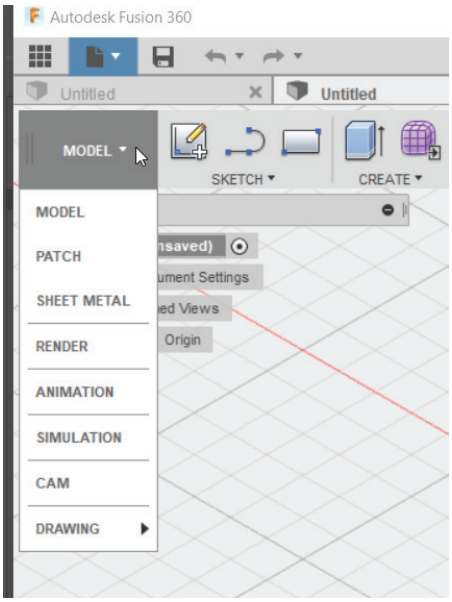

Figure-8. Workspace drop-down

- There are seven buttons available in **Workspace** drop-down; **MODEL, PATCH, RENDER, ANIMATION, SIMULATION, CAM,** and **DRAWING.**

- The **MODEL** button is used to activate workspace in which you can create solid bodies.

- The **PATCH** button is used to activate workspace for designing surface.

- The **SHEET METAL** button is used to activate workspace for designing sheet metal parts.

- The **RENDER** button is used to activate workspace for preparing 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 for Engineering Analysis.
- The **CAM** button is used for producing CNC codes.
- The **DRAWING** button is used for generating drawing and sketch.

You will learn more about these work-spaces 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.

New Design From File

This is used to import the file into Fusion 360 software. The steps to do so are given next.

- Click on the **New Design From File** tool from the **File** menu; refer to Figure-9. The **Open** dialog box will be displayed.

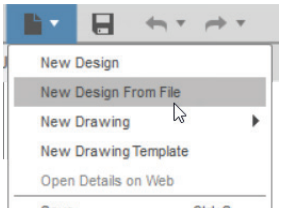

Figure-9. New Design From File tool

- Select the desired format from the **File Type** drop-down; refer to Figure-10 and double-click on the file to be imported.

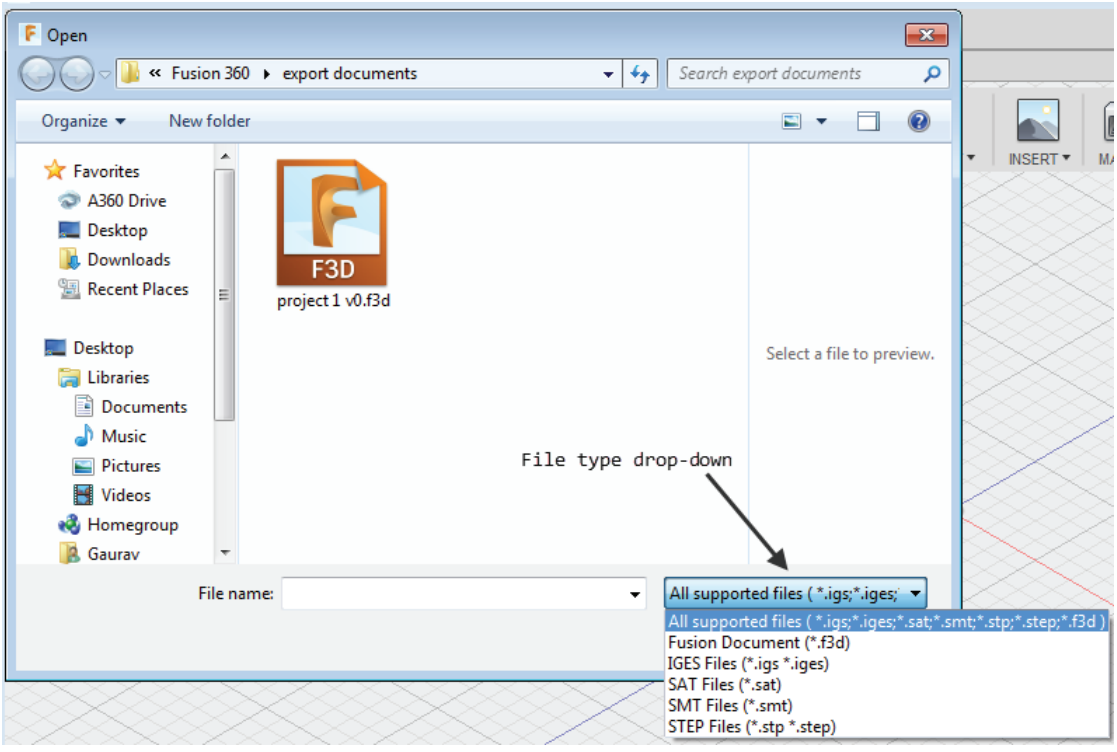

Figure-10. Open dialog box

This command supports file formats:

- Autodesk Fusion 360 Archive Files (*.f3d)
- IGES (*ige, *iges, *igs)
- SAT/SMT Files (*.sat, *.smt)
- STEP Files (*.step, *.stp)

New Drawing

The **New Drawing** tool is used for initiating a new drawing from animation or design; refer to Figure-11. The method to use this tool will be discussed later in the book.

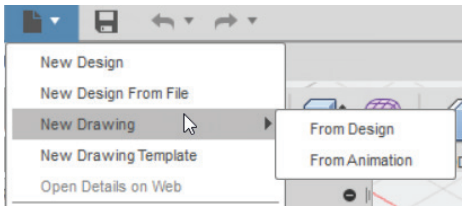

Figure-11. New Drawing

New Drawing Template

The New Drawing Template tool is used to create a template for generating drawings of the model; refer to Figure-12. You will learn about this option later in this book.

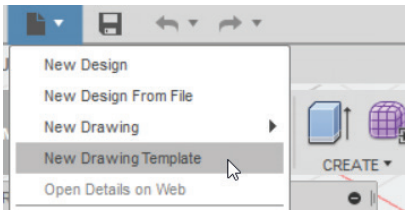

Figure-12. Related Data

Open details On Web

The **Open Details On Web** tool is used to display the details of current file in web browser. You can use this option only after you have saved your file on Autodesk cloud.

Save

The **Save** tool is used to save the current file. You can press **CTRL+S** key from keyboard to save file.

Figure-13. Save tool

- Click on the **Save** tool from **File** menu or select the **Save** button from application bar as shown in Figure-13. The **Save** dialog box will be displayed asking you to specify the **Name** and **Location** of your file; refer to Figure-14.

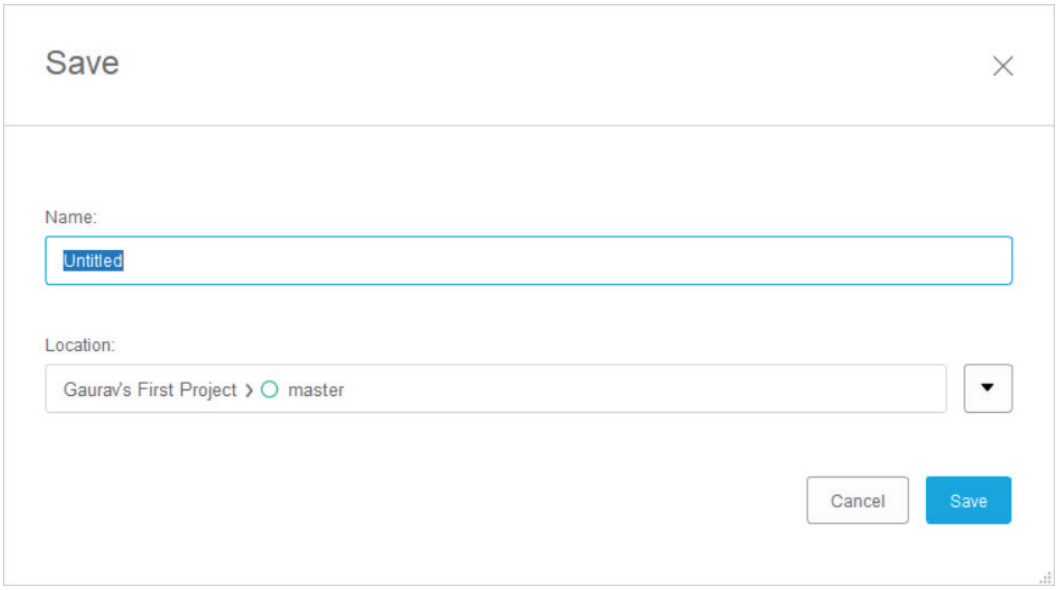

Figure-14. Save dialog box

- Specify the desired name in the **Name** edit box.
- Click on the down arrow next to **Location** edit box in the dialog box. A list of locations available in Autodesk Cloud will be displayed; refer to Figure-15.

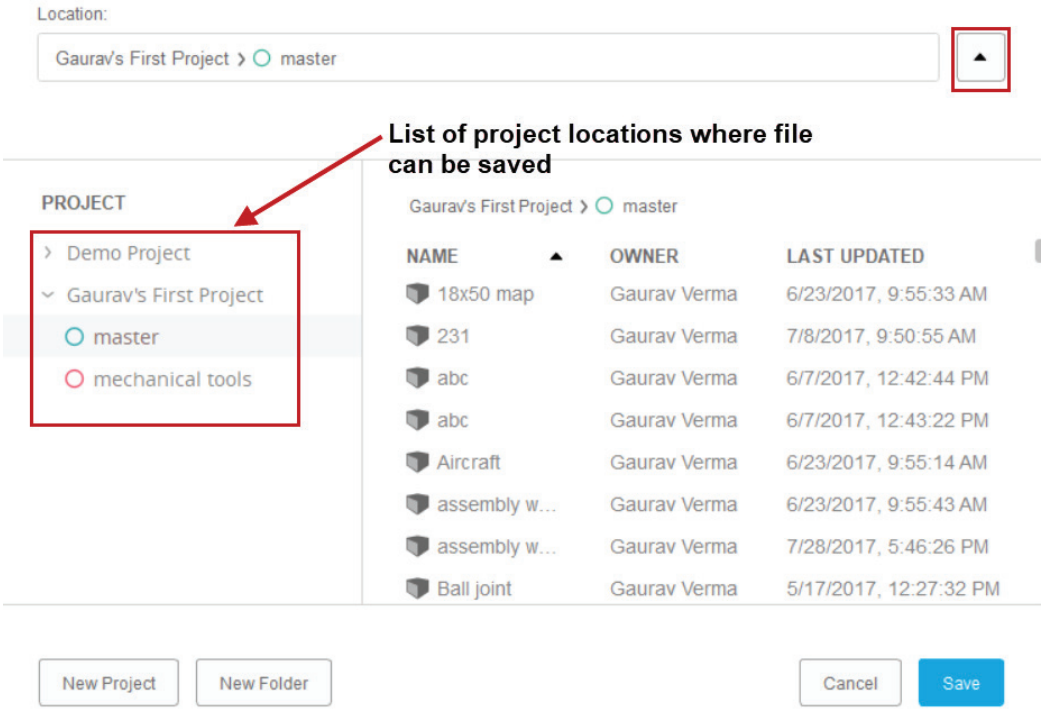

Figure-15. Locations to save file

- 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-16.

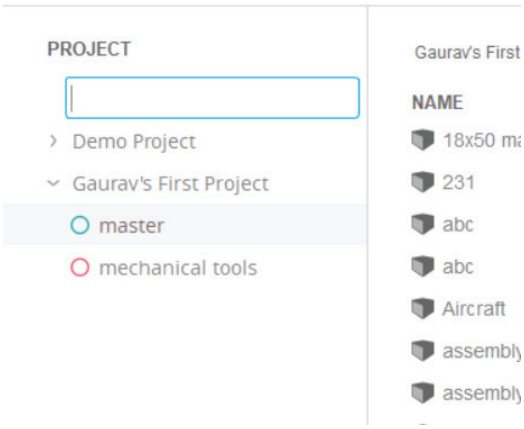

Figure-16. Specifying Project name

- Specify desired name for the project in displayed edit box and click anywhere in blank area of the dialog box.
- Click on the arrow displayed before your new project. The master folder will be created automatically under the new project and the location will be selected as save location for your file; refer to Figure-17.

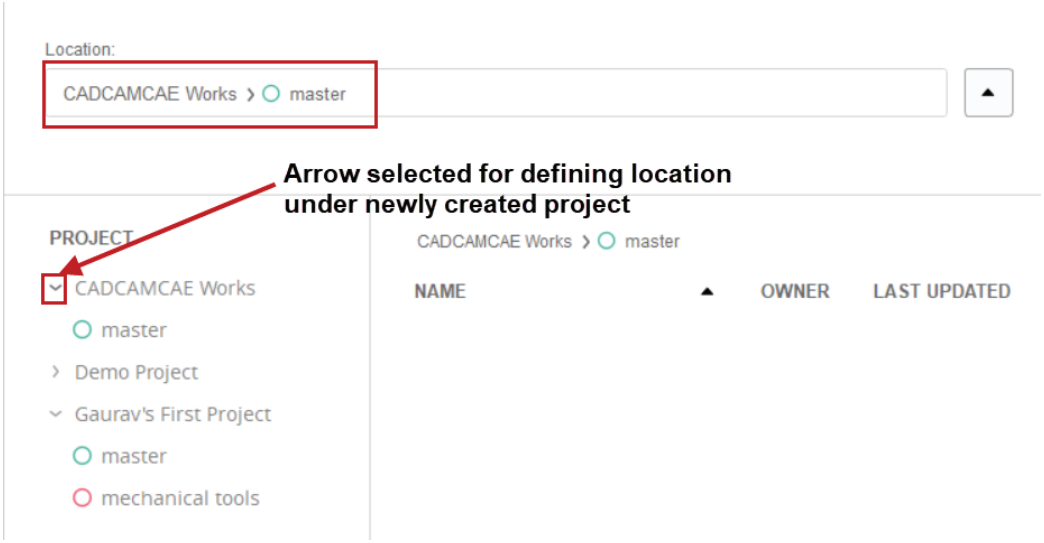

Figure-17. Location selected for saving file

- 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.
- After setting the desired parameters, click on the **Save** button. The file will be saved at specified location.

Save As

Using the **Save As** tool, you can save the file in desired format. The procedure to use this tool is discussed next.

- Click on **Save As** tool from **File** menu; refer to Figure-18. The **Save As** dialog box will be displayed as shown in Figure-19.

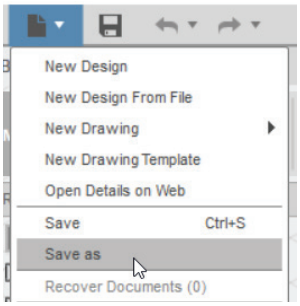

Figure-18. Save As Tool

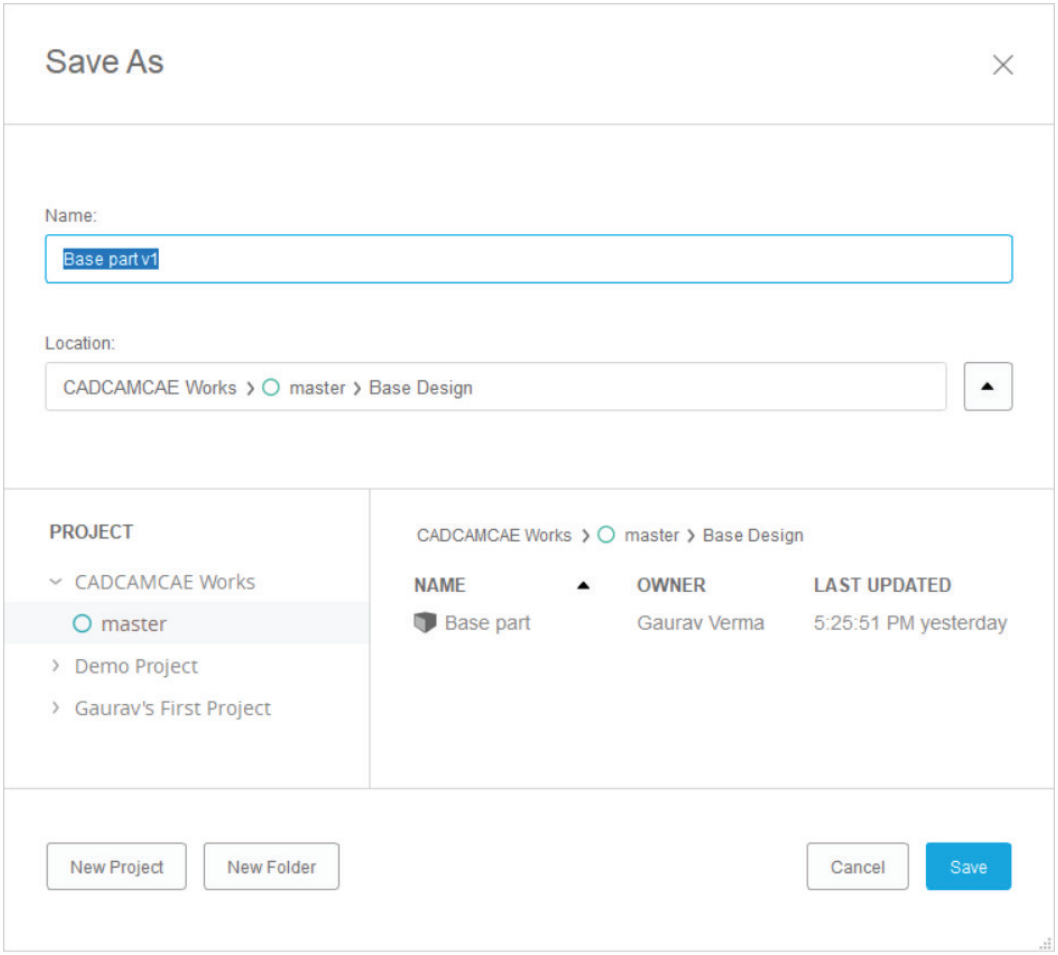

Figure-19. Save As dialog box

- Specify desired name and location for the file, and click on the **Save** button.

Recover Documents

The **Recover Documents** tool is used to recover your last unsaved file; refer to Figure-20. Note that this tool will be active only when you have some unsaved work and software closes unexpectedly. The procedure to use this tool is given next.

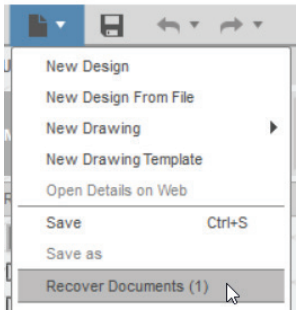

Figure-20. Recover Documents tool

- Click on the **Recover Documents** tool from the **File** menu. The **File Recovery** dialog box will be displayed where you need to select your unsaved file; refer to Figure-21.

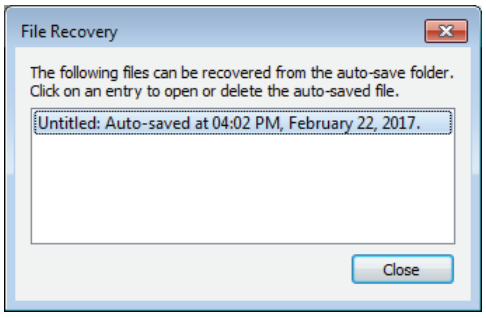

Figure-21. File Recover

- Double-click on the file that you want to be recovered and click on the Open button from the displayed menu. The file will open in **Autodesk Fusion 360**.

Export

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

- Click on the **Export** tool from the **File** menu; refer to Figure-22. The **Export** dialog box will be displayed; refer to Figure-23.

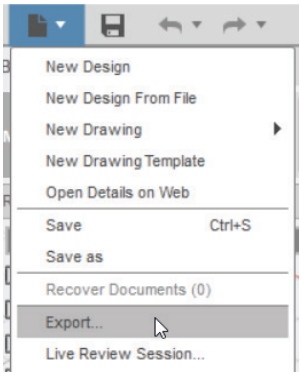

Figure-22. Export tool

- In the **Type** drop-down, there are five options to specify export format; **F3D, IGES, SAT, STM,** and **STEP**. Select the desired format from the drop-down.

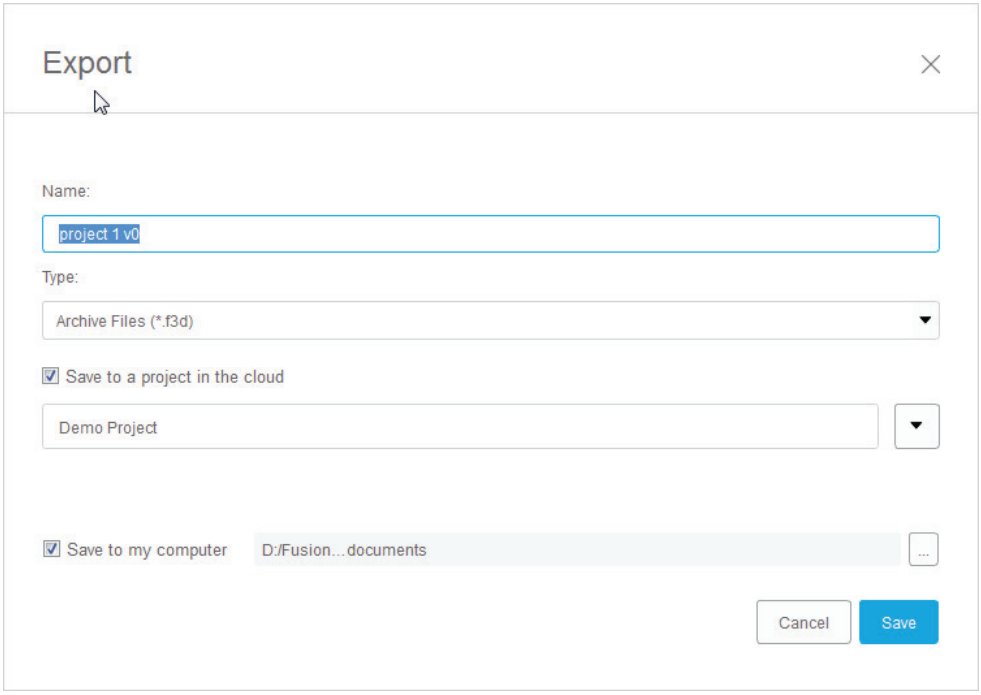

Figure-23. Export dialog box

- Select the **Save to a project in the cloud** check box if you want to save the file on cloud otherwise make it clear. Note that this check box is available only for **Archive Files (*.f3d)** format. If you are saving this file for the first time on cloud then the **Save to a project in the cloud** dialog box will be displayed on clicking the **Save** button; refer to Figure-24. Select the desired project from the **Project** drop-down in the dialog box and click on the **Select Project**. The file will be saved in selected project on cloud.

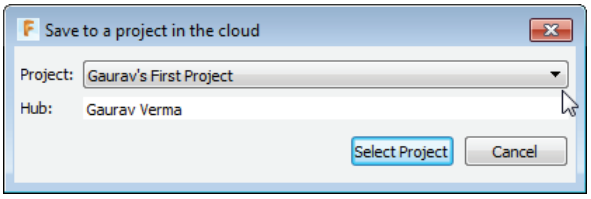

Figure-24. Save to a project in the cloud

- You can also save the file to your computer by selecting the **Save to my computer** check box. On selecting this option, click on the browse button next to **Save to my computer** field to specify the location of file where you want to save the file.
- Click on the **Save** button to save the file.

Live Review Session

Till the time of writing this book, the **Live Review Session** tool is under preview mode and not an active part of software. To use this tool, you must be in online mode with file saved on cloud. The **Live Review Session** tool is used to collaborate with your colleagues on current model. The procedure to use this tool is given next.

- Click on the **Live Review Session** tool from the **File** menu; refer to Figure-28. The **Fusion 360** dialog box will be displayed asking you to enable preview of the feature if you have not opted to display preview tools earlier while installing. Click on the **Enable** button from the dialog box. The **LIVE REVIEW SESSION** dialog box will be displayed in the left of application window; refer to Figure-25.

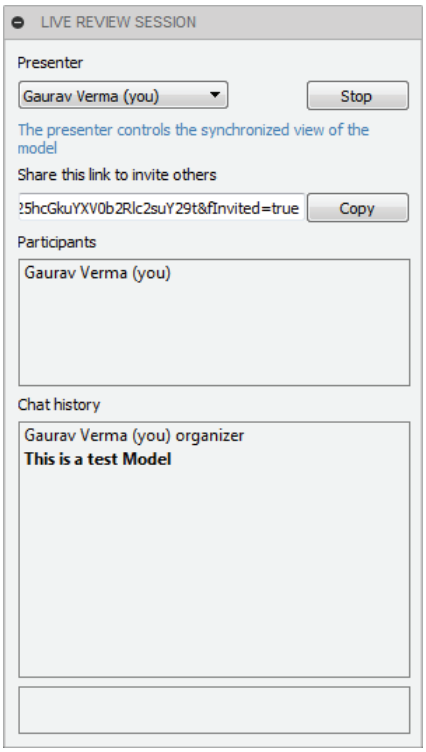

Figure-25. Live Review Session dialog box

- Copy the link in **Share this link to invite others** edit box and distribute it to participants via E-mail or other methods. (Your participants need to open the link in their web browser to participate. Note that their web browser should support OpenGL to display model.)
- Type desired comments in the input box at the bottom and press **ENTER** key. The message will be displayed to all the participants in **Chat history** area of the box.
- On opening the link in web browser, your participants will see the A360 window as shown in Figure-26. Participant need to specify his/her name and click on the **Join** button. On doing so, the **Live Review** window of part will be displayed; refer to Figure-27.

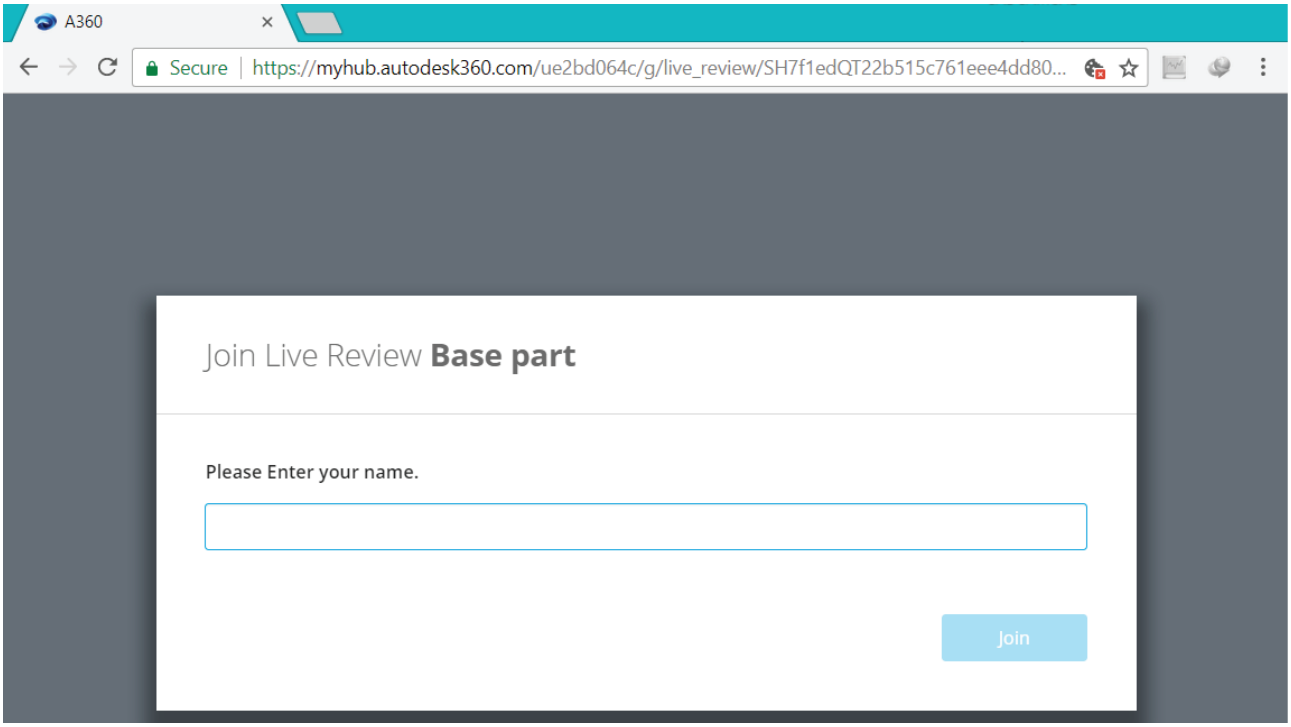

Figure-26. A360 window displayed

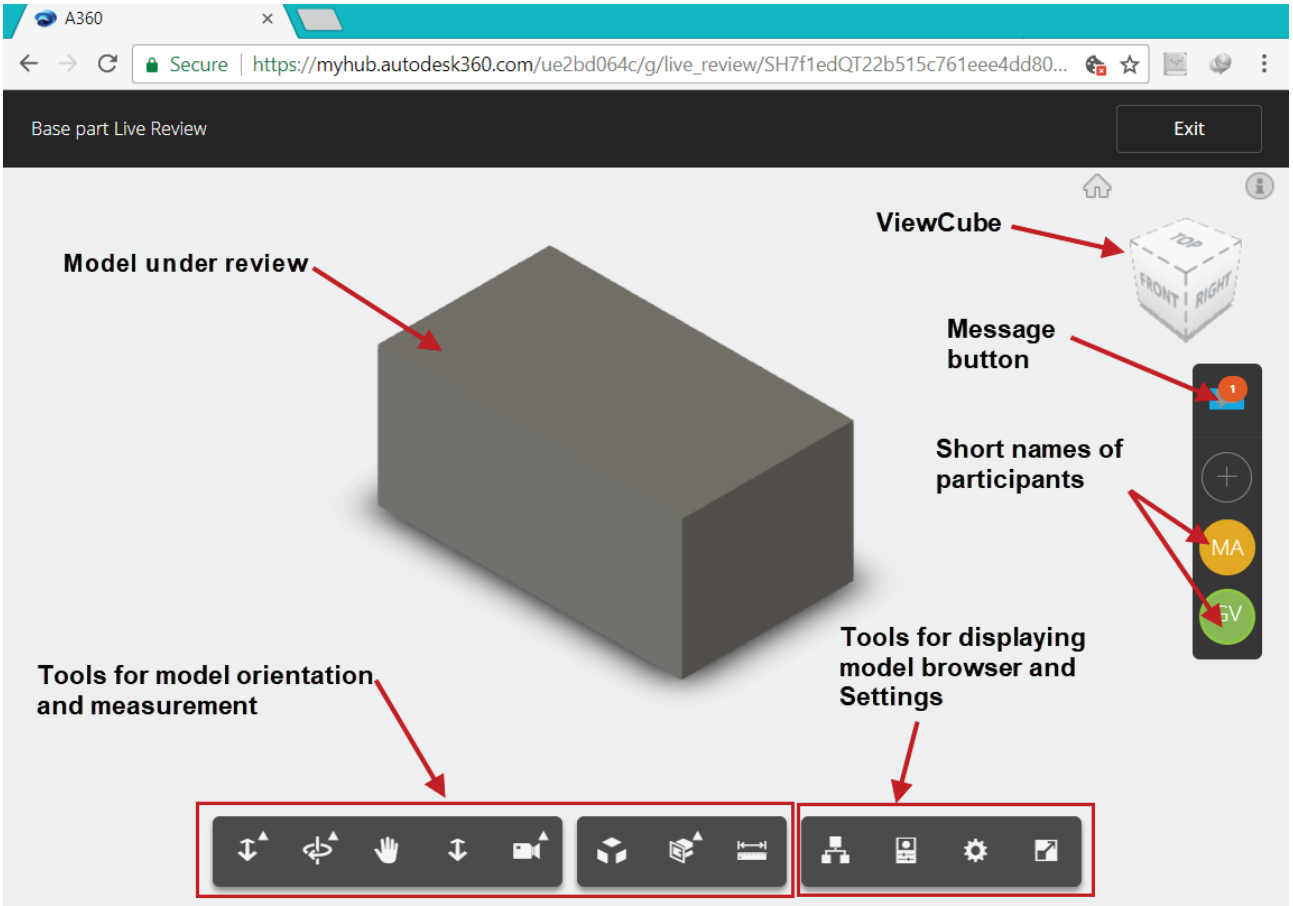

Figure-27. Live Review window

- After checking the model and giving desired instructions, your participants can click on the **Exit** button at the top right in their web browser. Note that your

participants cannot modify the model but any modifications you perform during the review session will be reflected immediately in their model.

- After completing session, click on the **Stop** button and then click on the **End Session** button to close live review session. The Session has been ended message will be displayed in your participants web browsers.

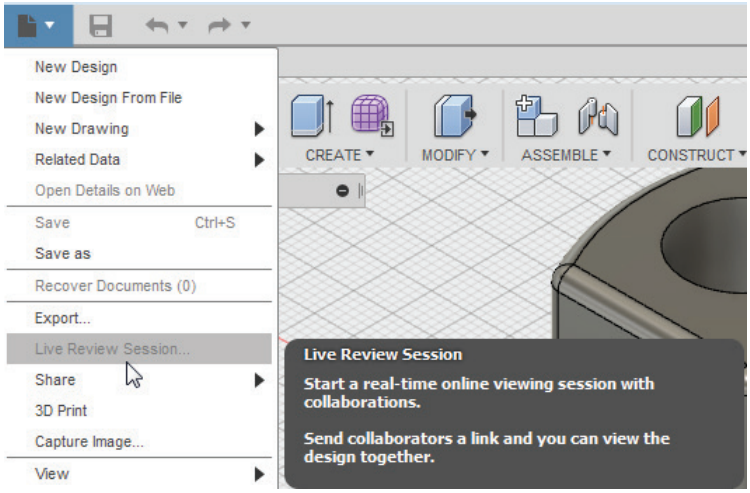

Figure-28. Live Review Session

Share

The **Share** tools are used to share the file or project on various portals. Click on the **Share** tool from **File** menu. Three options will be displayed in **Share** cascading menu; refer to Figure-29.

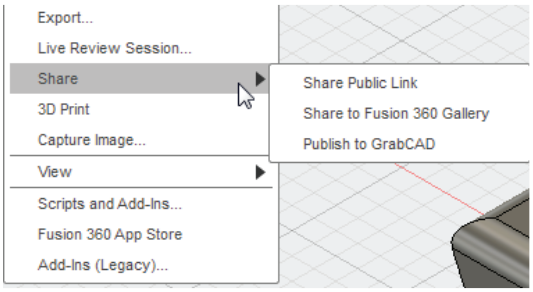

Figure-29. Share cascading menu

Share Public Link

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

- Click on **Share Public 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-30.
- 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 active to allow downloading of file.

Figure-30. Share Public Link dialog box

Share To Fusion 360 Gallery

The **Share To Fusion 360 Gallery** tool is used to share the file online to the gallery of Fusion 360 where the file is accessible to everyone registered with the gallery. The procedure to use this tool is given next.

- Click on **Share To Fusion 360 Gallery** tool from **Share** cascading menu. The **SHARE TO FUSION 360 GALLERY** dialog box will be displayed; refer to Figure-31.
- Click on the **NEW PROJECT** button from the dialog box. The options will be displayed as shown in Figure-31.

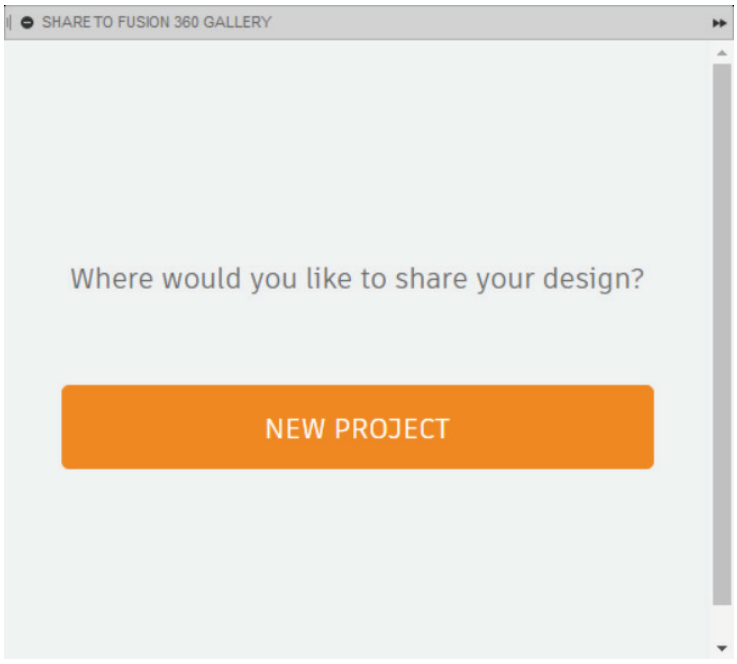

Figure-31. SHARE TO FUSION 360 GALLERY dialog box

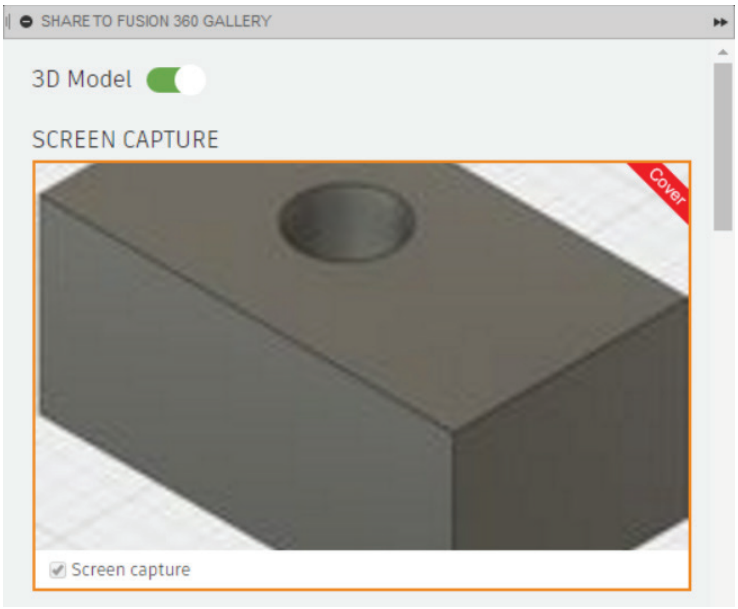

Figure-32. Share To Fusion 360 Gallery dialog box

- Scroll down in the dialog box and specify the information as required.
- Once all the desired information is specified, select the **I have read and agree to the Autodesk Terms of Use** check box and 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 **Publish to GrabCad** tool from **Share** cascading menu. The **PUBLISH TO GRABCAD** dialog box will be displayed; refer to Figure-33.
- 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 and following the instructions.
- Once you have login to your account, the dialog box will be displayed as shown in Figure-34.

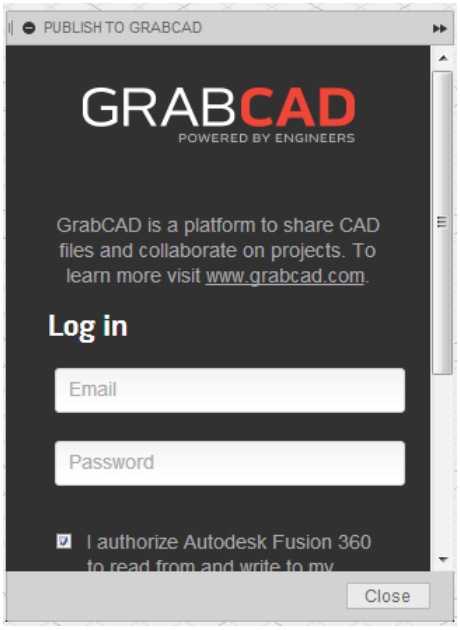

Figure-33. PUBLISH TO GRABCAD dialog box

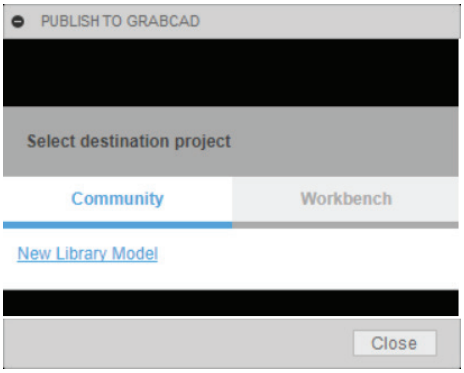

Figure-34. PUBLISH TO GRABCAD dialog box

- Click on the **New Library Model** link button if you want to save model in community. If you want to create a workbench model 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-35.

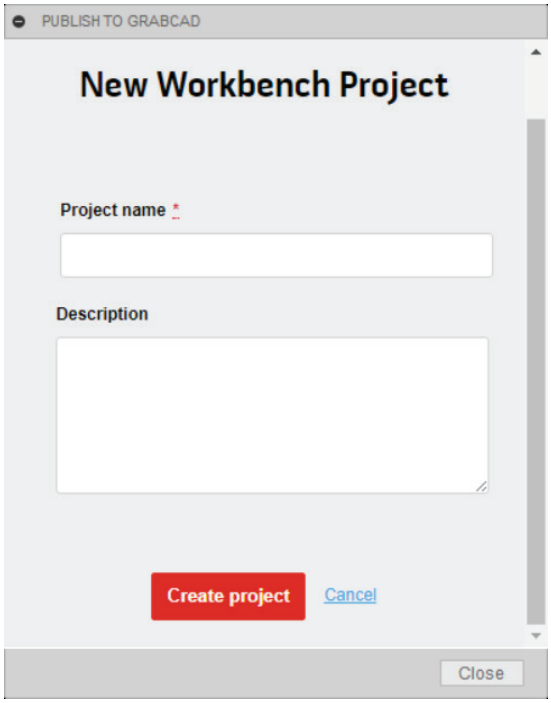

Figure-35. New Workbench Project details

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

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** and **STL**. 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 **Toolbar**; refer to Figure-36. The **3D-PRINT** dialog box will be displayed; refer to Figure-37.

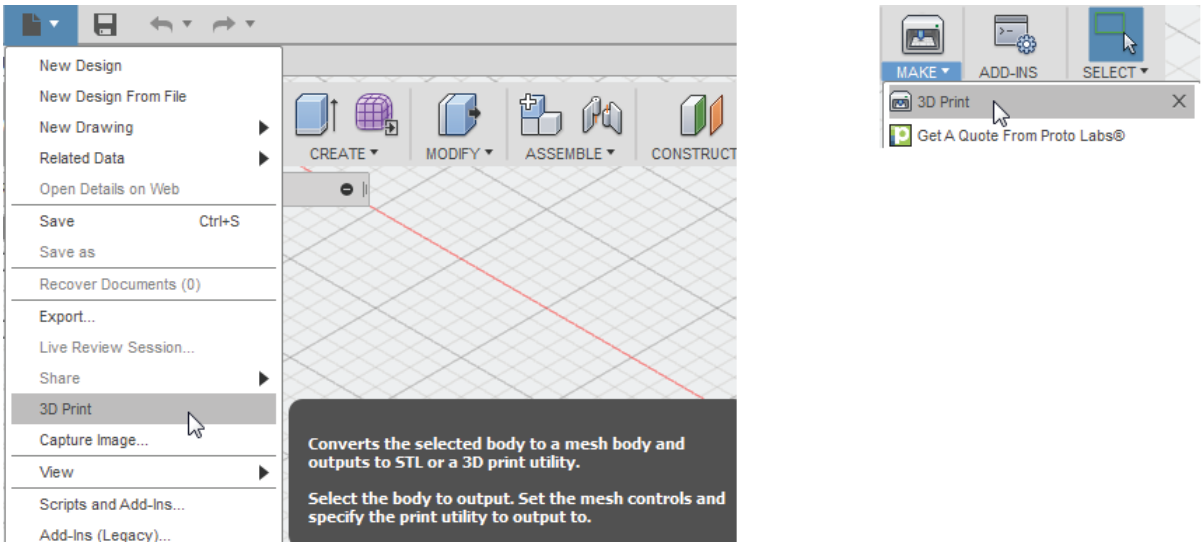

Figure-36. 3D Print

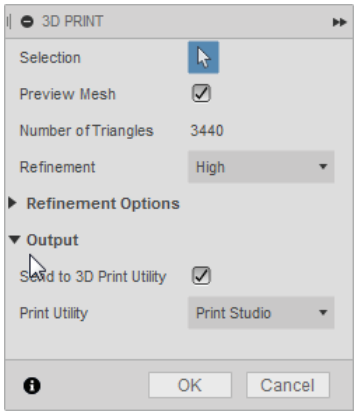

Figure-37. 3D-Print Dialog box

- 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.
- Now, select the **Preview Mesh** check box to see the number of triangles forming in the selected body.
- Select the 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-38.

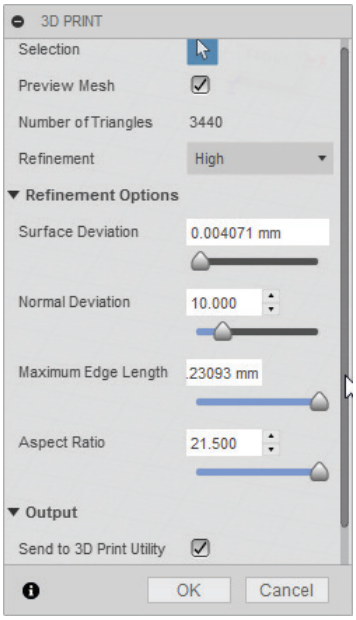

Figure-38. Refinment

- 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 in **Print Utility** drop-down; refer to Figure-39. Note that 3D printing utilities are not installed automatically during Fusion 360 installation. On selecting the utility in **Print Utility** drop-down, an option to download and install the utility is displayed in the dialog box; refer to Figure-40. Download and install the desired utility.

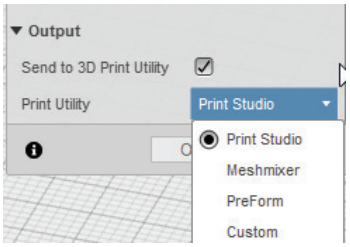

Figure-39. Output 3D-Print

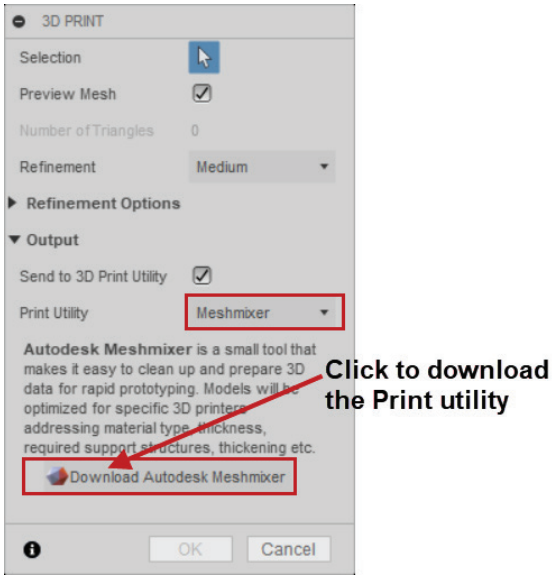

Figure-40. Option to download Print utility

- Clear the **Send To 3D Print Utility** check box if you want to save the project file in **STL** format so that you can open this file later in any 3D Printing software.
- In our case, we have selected **Print Studio** option from the **Print Utility** drop-down (Needless to say, we have installed it also). Click on the **OK** button from the dialog box. The **Print Studio** application window will be displayed; refer to Figure-41.

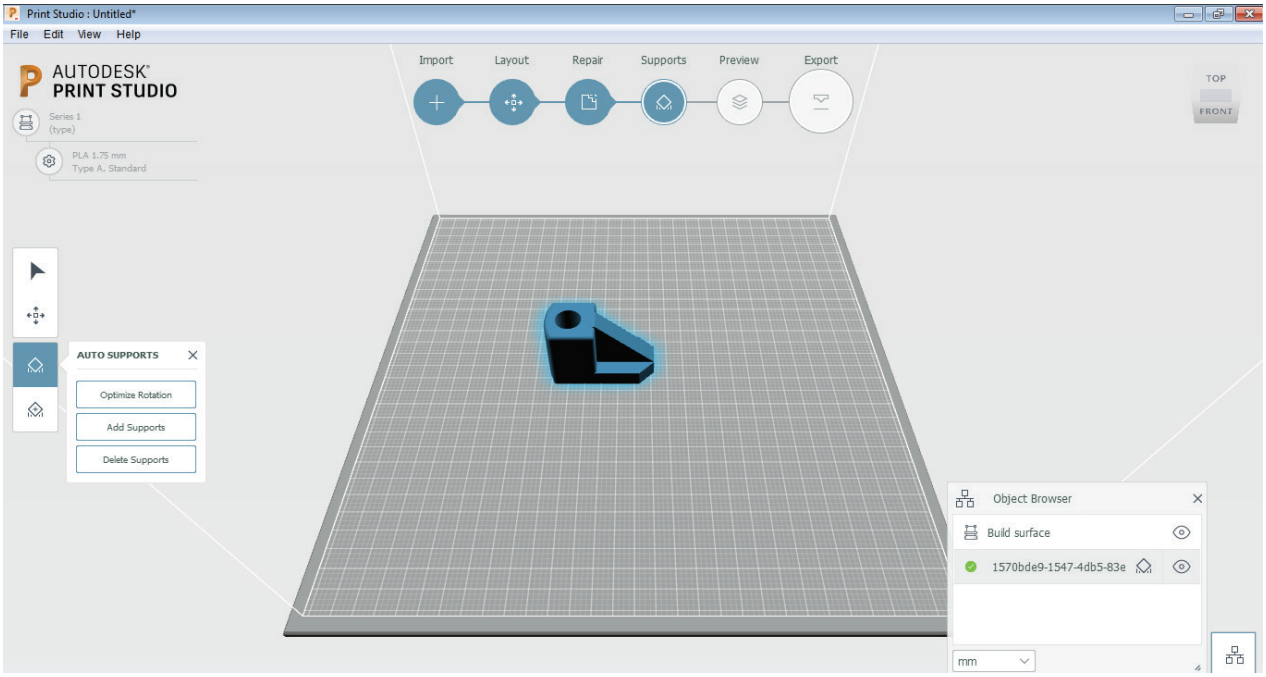

Figure-41. Print Studio Window

The tools of this application are discussed next.

Print studio

On selecting the **OK** button from **Print Studio** dialog box, the file is exported from Autodesk Fusion 360 and imported to Autodesk Print Studio.

Specifying 3D Printer

- To set desired 3D Printer, click on the **Change Printer** button at upper left corner; refer to Figure-42. The **Printers** dialog box will be displayed. In this dialog box, you can select the printer connected via USB or connected at network.

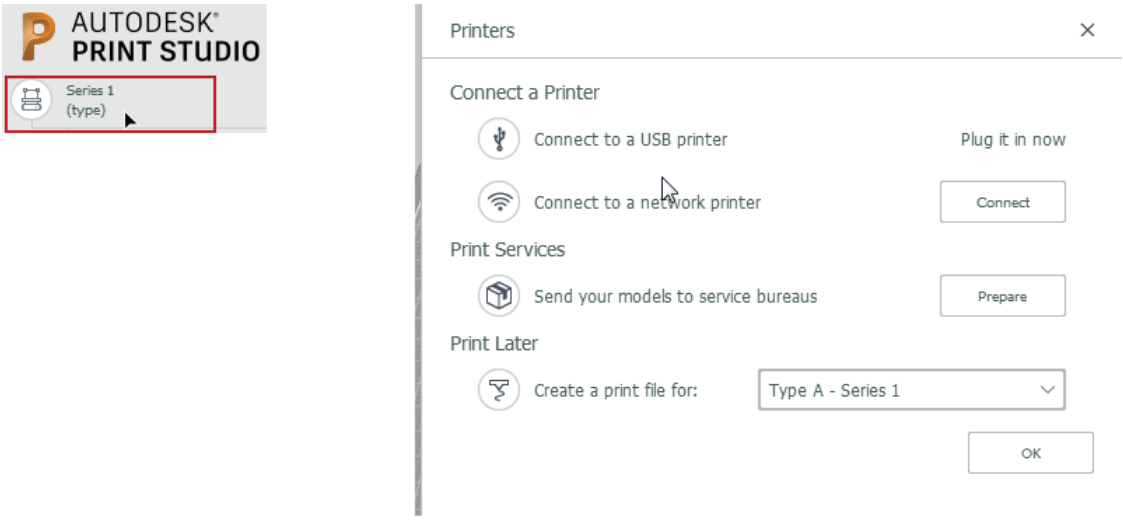

Figure-42. Selecting a 3D printer

- If you want to use the **Print Services** tools to modify parts then click on the **Prepare** button in the **Print Services** Section.

- If you want to print later and want to create printable file for your printer then select the desired printer model from **Create a print file for** drop-down and click **OK** button of **Printers** dialog box.

Change Material and Printer Setting

This tool is used to change the material and other settings to use 3D printer optimally.

- To change material, click on the **Change Material and Printer Setting** button below the **Change the Printer** button in the left toolbar; refer to Figure-43. The **Settings** dialog box will be displayed; refer to Figure-43. Select the **Profile** according to your needs in **Profiles** drop-down. In **Basic Setting** section, specify the desired parameters like height of each layer, pattern of infill in printing, infill density etc. Select the Enable raft check box if you are using raft at the base for better adhesion and stability of printed model.

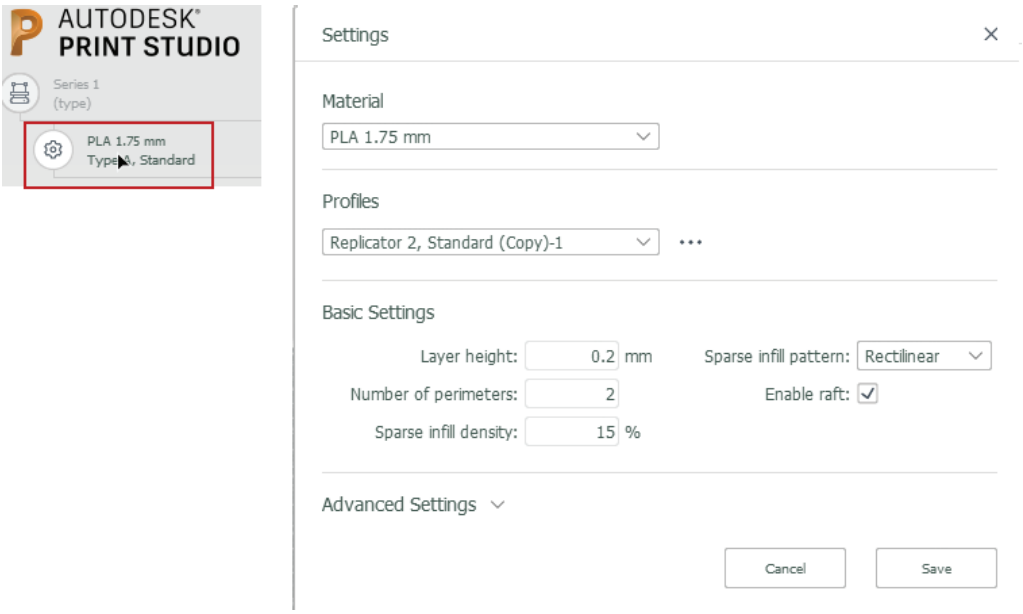

Figure-43. Changing Material and Printer Setting

- If you want to specify advanced parameters like, support dimensions, Infill pattern, extruder temperature, and so on then, click on **Advanced Setting** button and specify the desired parameters.
- After setting the desired parameters, click on **Save** button from **Settings** dialog box.

Toolbar

The tools in the **Toolbar** of **Print Studio** application are used to finalize the project for 3D printer step by step; refer to Figure-44.

Figure-44. Print Studio Toolbar

Import Tools

Import tool is used to import the file to Autodesk Print Studio. We have already imported the file so the button is displayed selected in the toolbar in the Figure-44.

Layout Tools

Layout tool is used to position the model on printing tray. On selecting this tool, the toolbar for layout will be displayed on the left of the application window; refer to Figure-45.

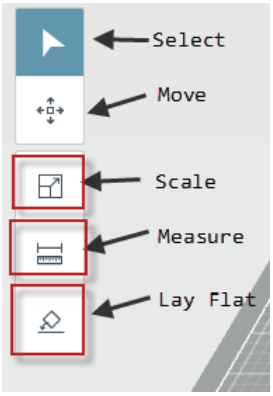

Figure-45. Layout

- Click on the **Move** tool and move the model at desired location by dragging.
- Click on the **Scale** button from left toolbar if you want to scale up or scale down the model. The **SCALE** box will be displayed next to the Scale button. Specify the desired percent scale value in **Scale Factor** edit box and press **ENTER**.
- Click on the **Measure** button if you want to measure your model before printing. You will be asked to specify start and end points for measurement.
- If you want to change the face of model to be resting on the printer bed then click on the **Lay Flat** button from the left toolbar and select the desired face.

Repair Tools

After setting the parameters of **Layout** tool, next step is to repair the model. On selecting the **Repair** tool from **Toolbar** section, a repair tool box will be displayed at the left of screen. This tool detects the model issues like holes and other geometry defects cannot be printed by 3D printer. You can resolve these issues manually or automatically. When issues are resolved, this tool display a notification i.e. **No Problems**; refer to Figure-46.

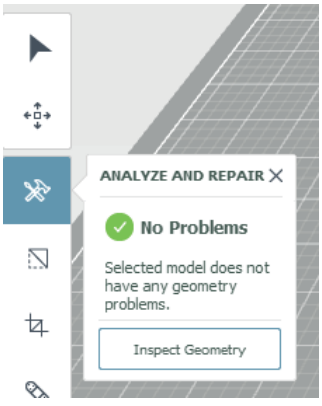

Figure-46. Repair in Print Studio

- Click on the **Find and Repair** button to automatically find and repair faults in the model. The **ANALYZE AND REPAIR** toolbox will be displayed; refer to Figure-46. Click on the **Inspect Geometry** button from the toolbox to find problematic areas. The problematic areas will be highlighted in the viewport; refer to Figure-47. Click on the **Auto Repair** button to automatically repair the model.

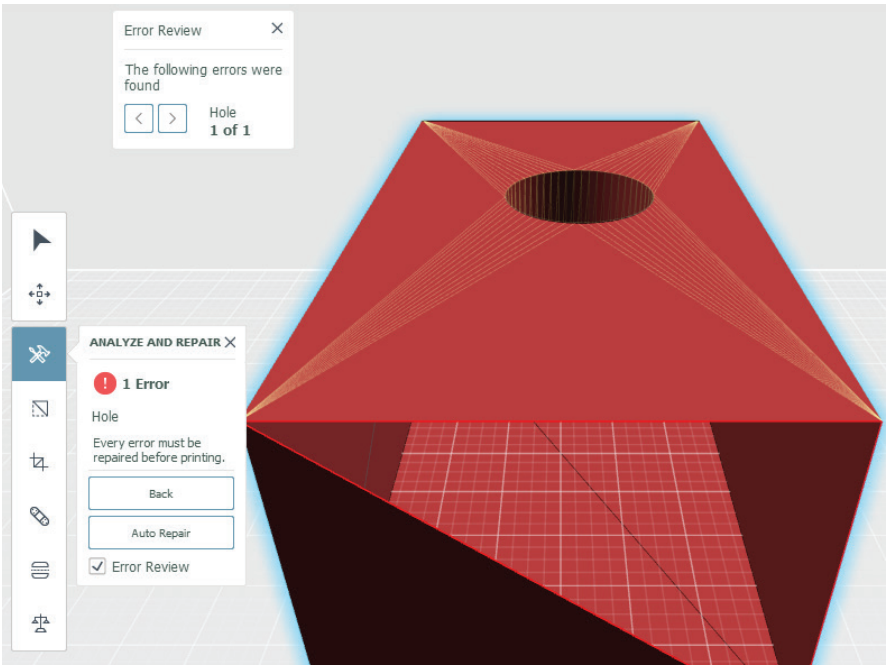

Figure-47. Finding and repairing errors

- Click on the **Select Regions** tool from the left toolbar to select different regions of model and apply basic operations. The **Select Regions** toolbox will be displayed; refer to Figure-48.

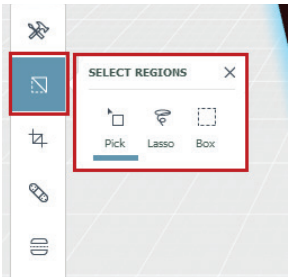

Figure-48. SELECT REGIONS toolbox

- Click on the desired button from the toolbox. Click on the **Pick** button to select individual feature of the model. Click on the **Lasso** button to select features of model using freehand selection region. After selecting this tool, click on the model and drag cursor to form selection region; refer to Figure-49. Click on the **Box** button and draw a rectangle to select features that fall inside the created rectangle.

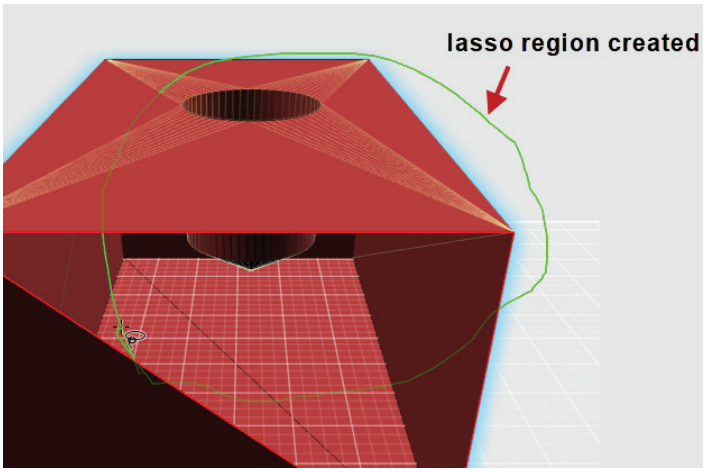

Figure-49. Lasso region created

- On selecting features, a round button (Action icon) is displayed in the viewport. Click on this button to display context toolbar (Action wheel); refer to Figure-50. Click on the desired button to perform respective operation on selected faces.
- Click on the **Crop** button to isolate selected region and delete rest of the model for printing; refer to Figure-51. You can modify the selection by using rotation and translation handles displayed on the box. Click on the round **OK** button at the bottom of selection box to applying cropping. Press **ESC** to exit cropping mode.

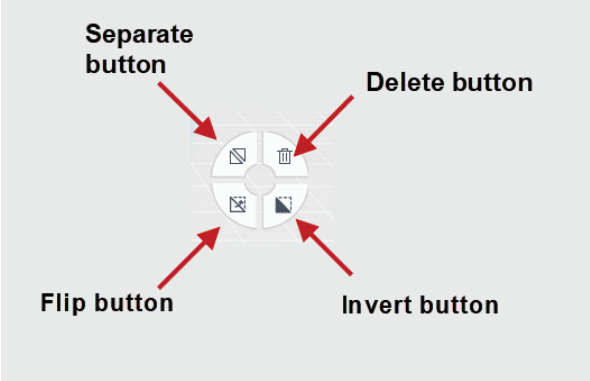

Figure-50. Context toolbar

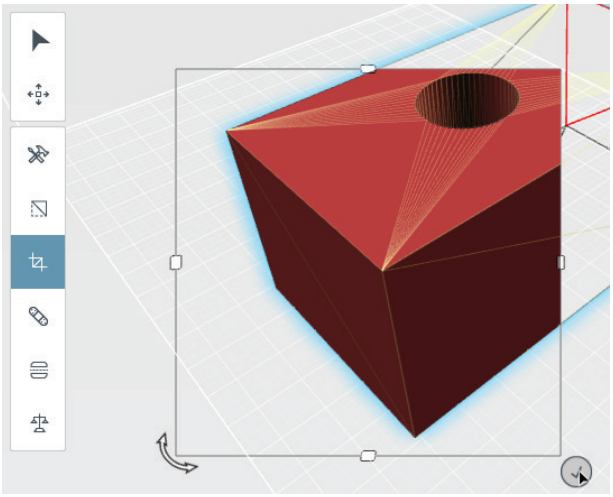

Figure-51. Region created for croppping

- Click on the **Patch Holes** tool to close open areas of the model. The **PATCH HOLES** toolbox will be displayed; refer to Figure-52. Click on the **Bridge** button to create boundaries for filling. Click on the **Fill** button after selecting the open region to fill it; refer to Figure-53.

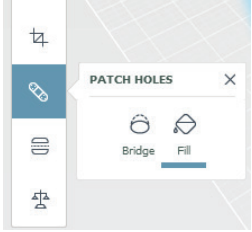

Figure-52. PATCH HOLES toolbox

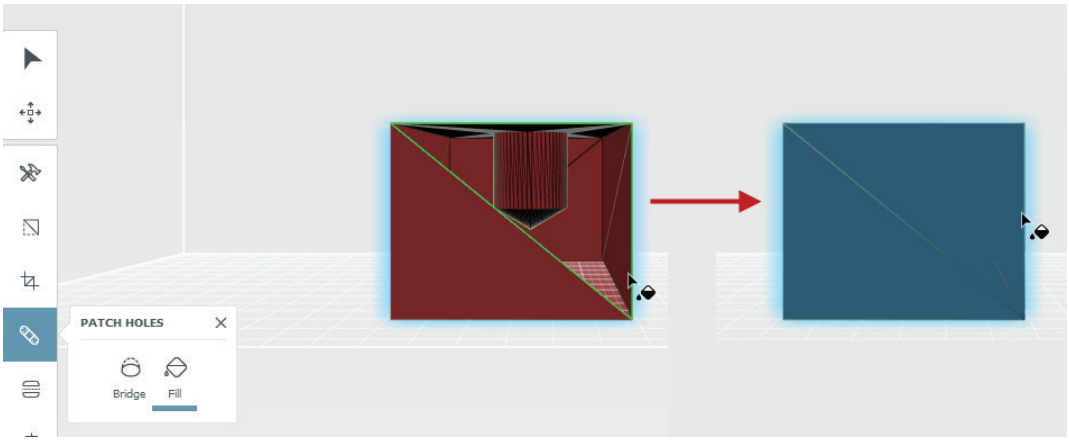

Figure-53. Filling open region

- Click on the **Plane Cut** tool from the left toolbar to check insight of model. The **PLANE CUT** toolbox will be displayed with **X**, **Y**, and **Z** buttons. Click on the desired button create section. Press **ESC** to exit the tool.

- Click on the **Balance** tool from the left toolbar to check whether the part will be stable during 3D printing or not. The **BALANCE** toolbox will be displayed. Click on the **Suggest Base** button from the toolbox to display the faces that can be used as base; refer to Figure-54. Select the desired face to be used as base.

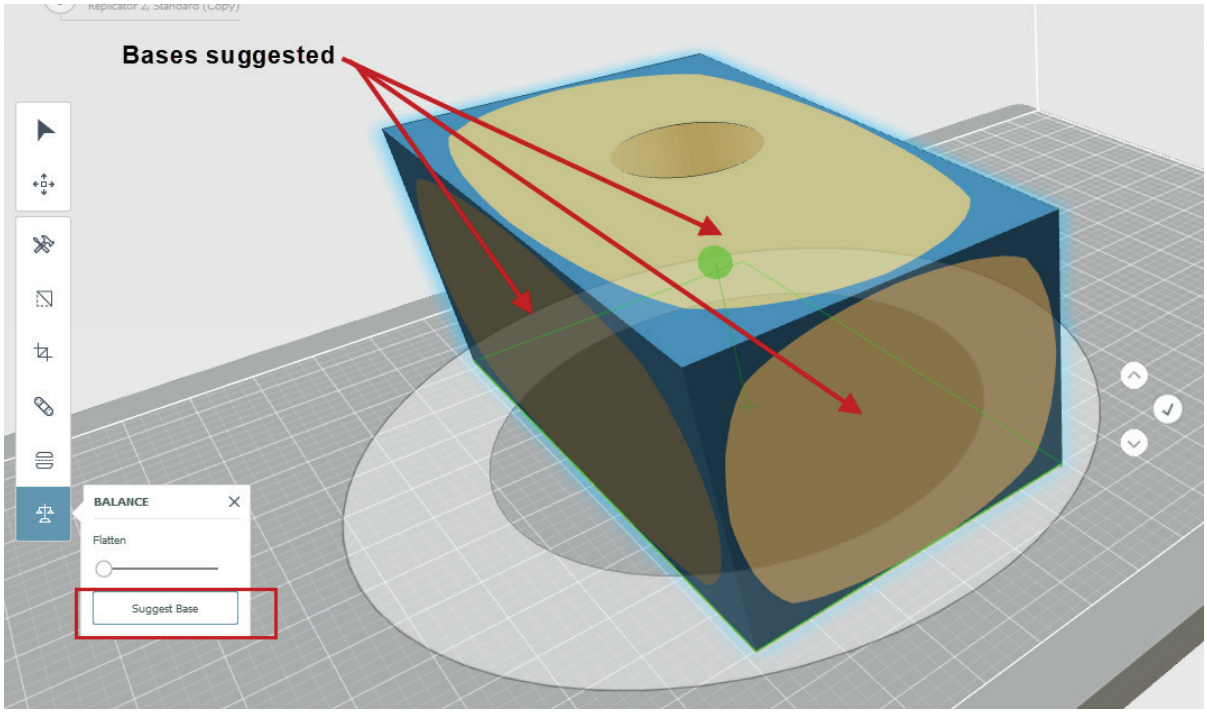

Figure-54. Bases suggested by Balance tool

Supports Tools

The **Supports** tools are used for giving support to model which do not have flat base. To apply supports, click on the **Supports** tool from the top toolbar. The **AUTO SUPPORTS** toolbox will be displayed in the left of screen; refer to Figure-55.

- Click on the **Optimize Rotation** tool to orient the model at optimum position for 3D printing. This tool considers the dimensions of part and available space on bed.

- Click on the **Add Supports** tool from the **AUTO SUPPORTS** toolbox to automatically apply supports to the model which is not stable during 3D printing; refer to Figure-55.

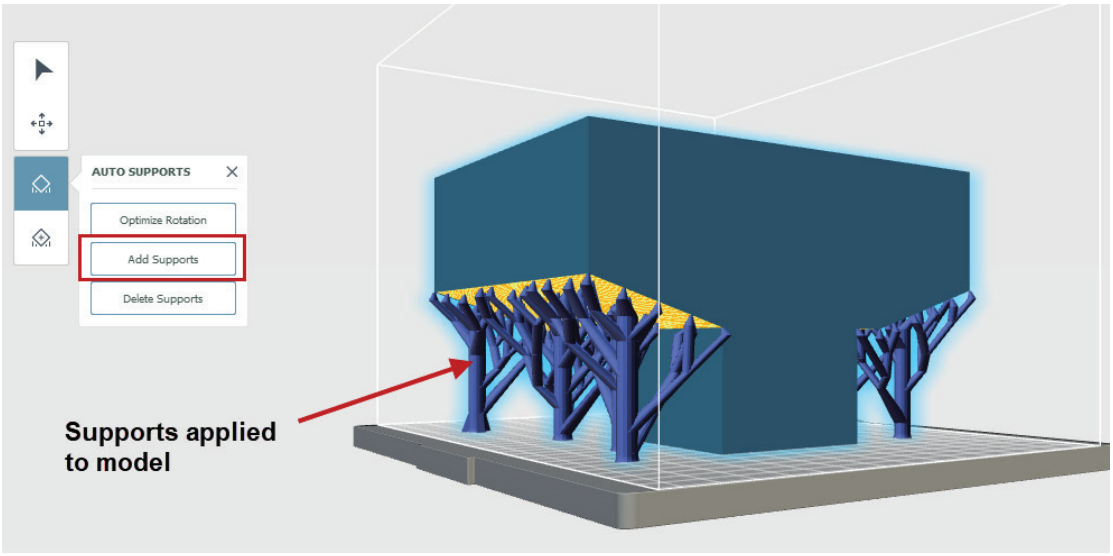

Figure-55. Applying supports to model

- Click on the **Delete Supports** tool to delete automatically applied supports.
- Click on the **Manual Supports** tool from the left toolbar to manually define supports for the model. You will be asked to specify start and end points of the support. Click at the desired locations on model and printer bed to create support; refer to Figure-56. Create as many supports as required.

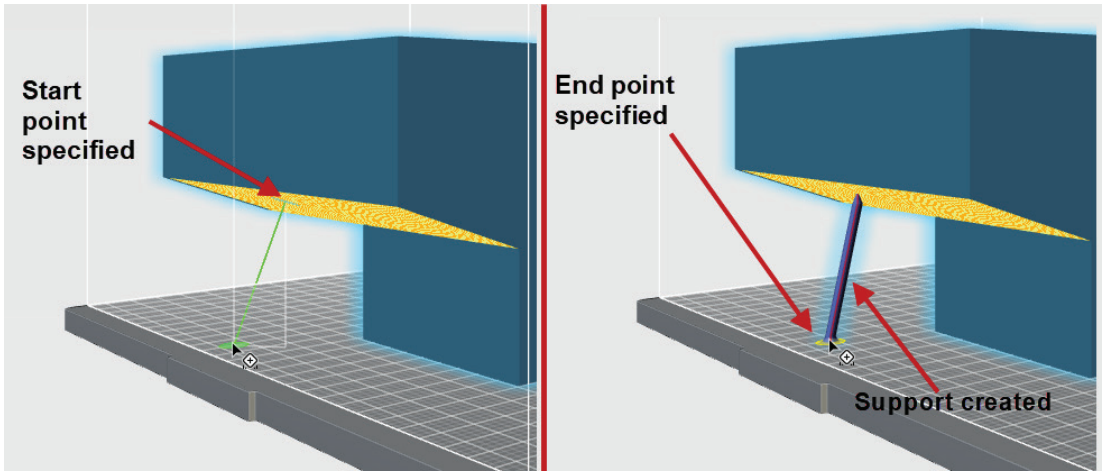

Figure-56. Creating support manually

Preview

- Click on the **Preview** tool from top toolbar to check preview of 3D printed model. System will start generating toolpath and the **PREVIEW SLICES** toolbox will be displayed once toolpath generation is finished; refer to Figure-57.
- In **Preview Slices** Box, you can check the time taken by printer and volume of model in **Estimated time** and **Estimated volume** section. By moving the slider in

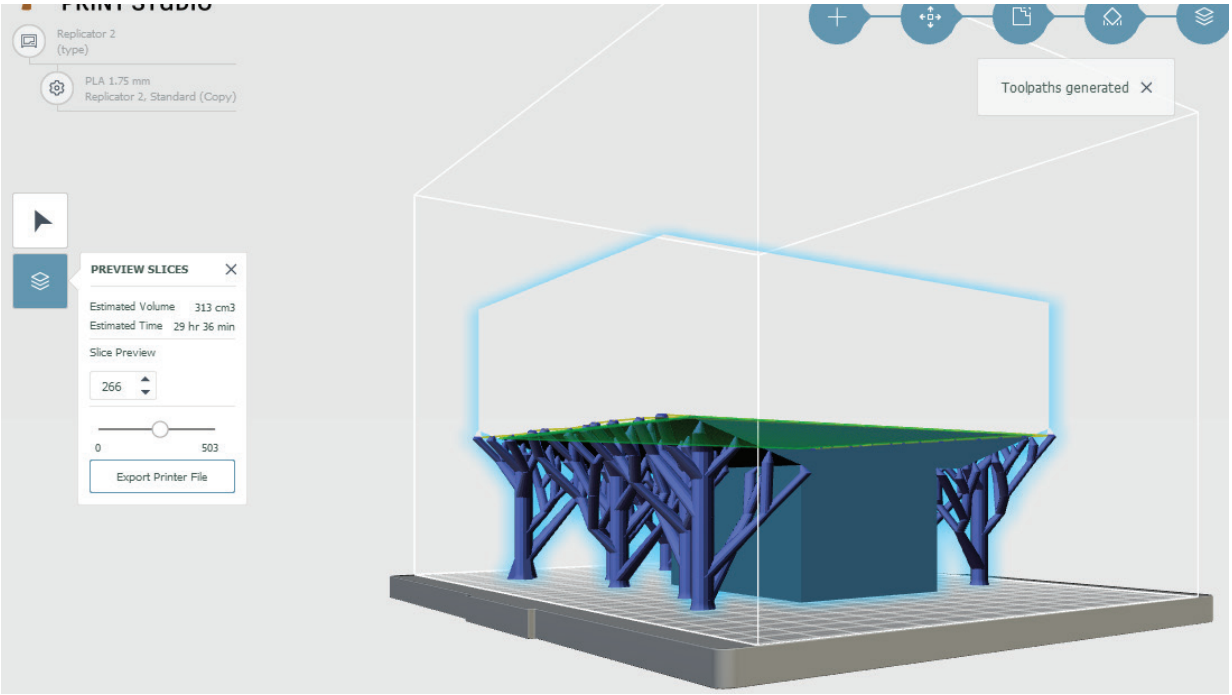

Figure-57. Preview tool

Click on the **Export Printer File** from **PREVIEW SLICES** toolbox or click on the **Export** tool from top toolbar to save the printer ready file. The **Save As** dialog box will be displayed; refer to Figure-58. Specify desired name and location for the file, and click on the **Save** button.

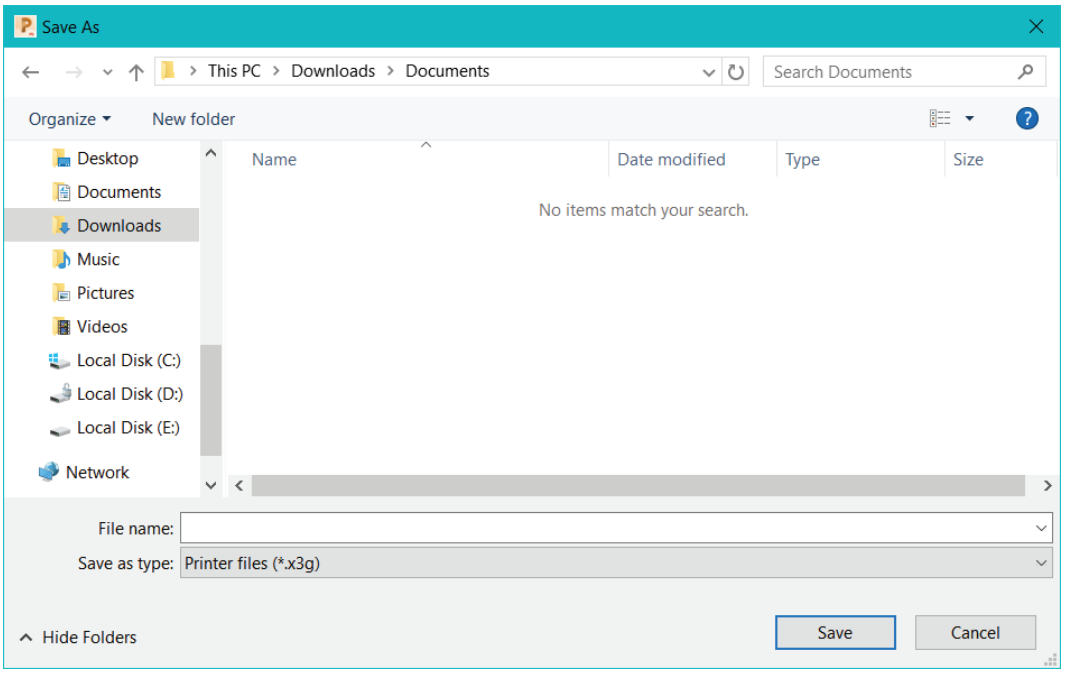

Figure-58. Save As dialog box for 3D printing file

You can save the Print Studio file of current model by using **CTRL+S** in 3PS format if you want to store and share the file with your colleagues. Now, you can close the **Print Studio** window by pressing **CTRL+X** or clicking on the **Close** button at top right corner.

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-59. The **Image Options** dialog box will be displayed.

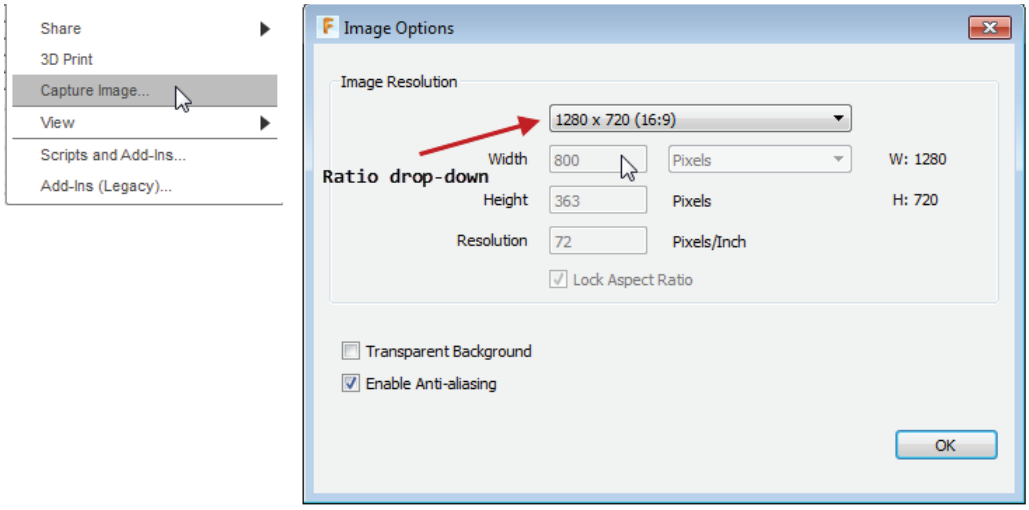

Figure-59. Capture Image tool

- Select the 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 **Inch** 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 of **Image Options** dialog box. The **Save As** dialog box will be displayed; refer to Figure-60.

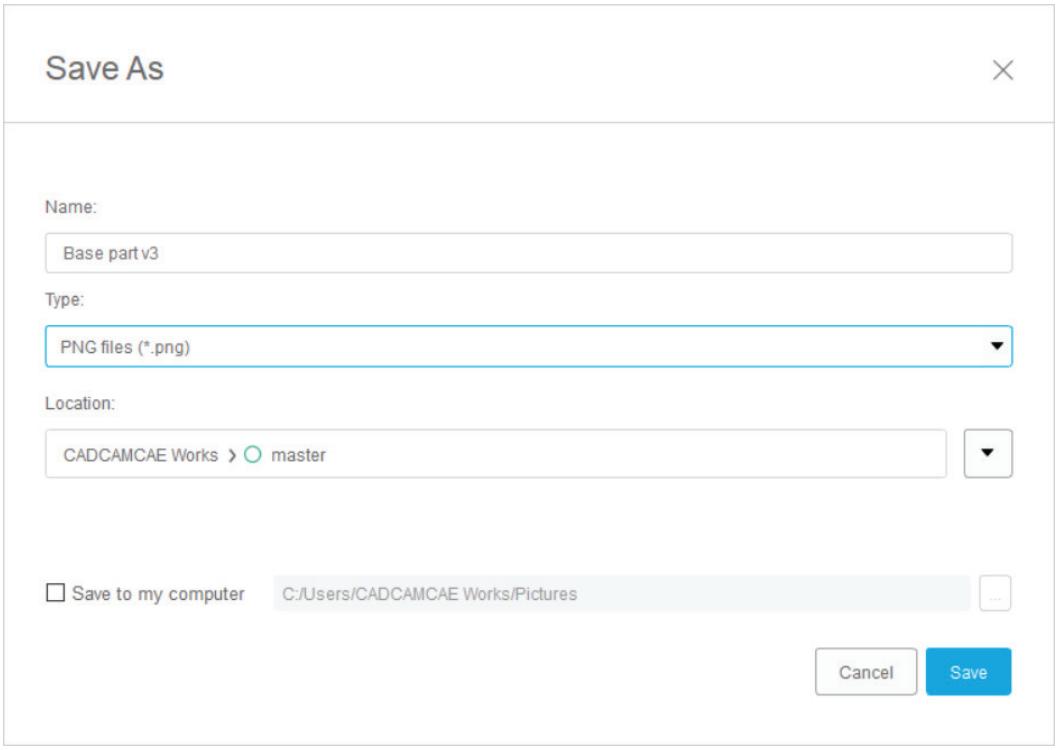

Figure-60. Save As dialog box for image capturing

- Select the 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.

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-61.

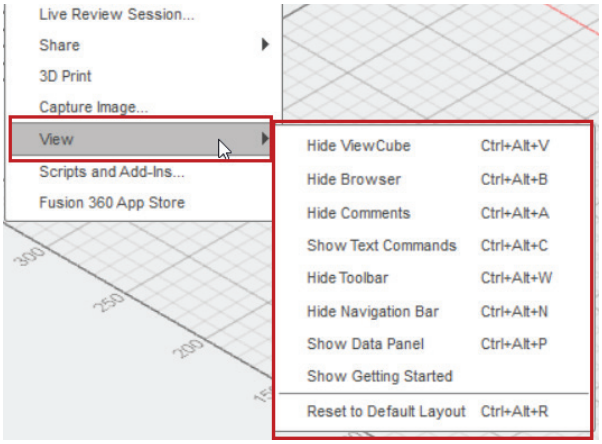

Figure-61. View cascading menu

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 **Hide 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 Comment

The **Hide Comment** tool is used to show and hide the **Comments** from Fusion 360 software. If the **Hide 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 **Show Text Commands** 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 Toolbar

The **Hide Toolbar** tool is used to hide the **Toolbar**. If the **Hide Toolbar** is already hidden then **Show Toolbar** tool will be displayed in place of it. This tool can also be used by pressing **CTRL+ALT+W** 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.

Show Getting Started

The **Show Getting Started** tool is used to display **GETTING STARTED** dialog box. Select the desired unit system and navigation control scheme from the dialog box. Click on the **START HERE** button to check tutorials on Autodesk Fusion 360. Click on the **Close** button from the dialog box to close it.

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.

Script and Add-Ins

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

- Click on **Script and Add-Ins** tool from **File** menu. The **Script and Add-Ins** dialog box will displayed; refer to Figure-62.

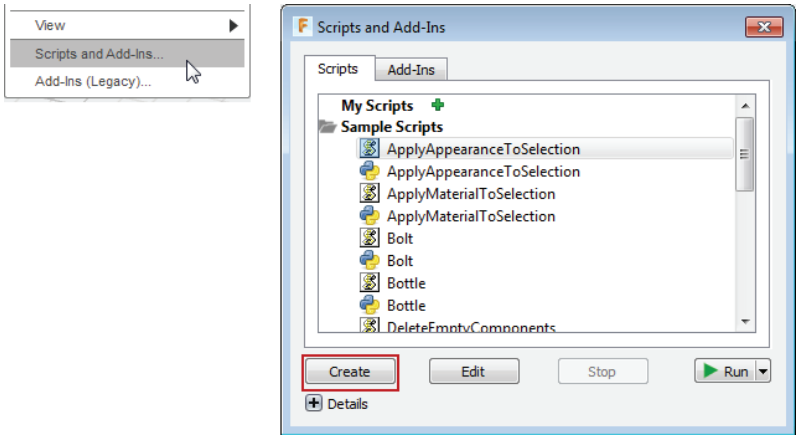

Figure-62. Script and Add-ins

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-Ins** dialog box will be displayed; refer to Figure-63.

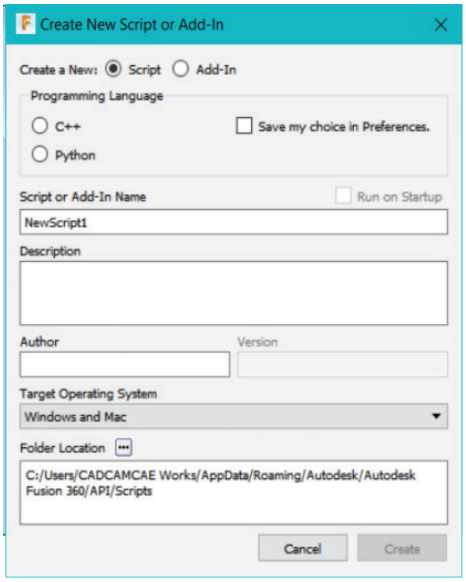

Figure-63. Creating New Script

- Select **Script** radio button to create a script or select the **Add-In** radio button to create an Add-In.
- Select the 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 Spyder will start installing and once installation is complete, codes will be displayed; refer to Figure-64. 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-63. Programming interface will be displayed based on language of selected script/Add-In.

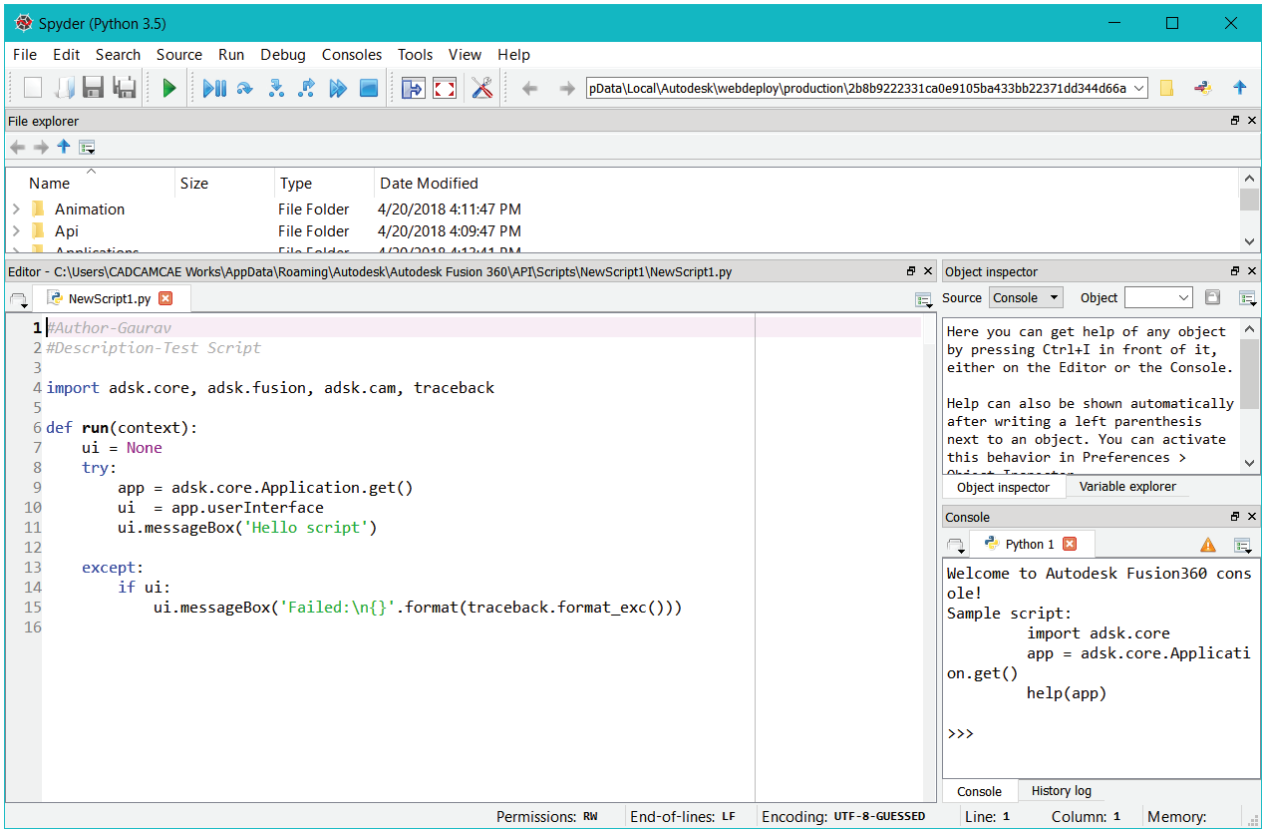

Figure-64. Spyder application window

Fusion 360 App Store

The **Fusion 360 App Store** tool in **File** menu is used to download and install third party Apps in Autodesk Fusion. On clicking this tool. a webpage is displayed in your default web browser; refer to .

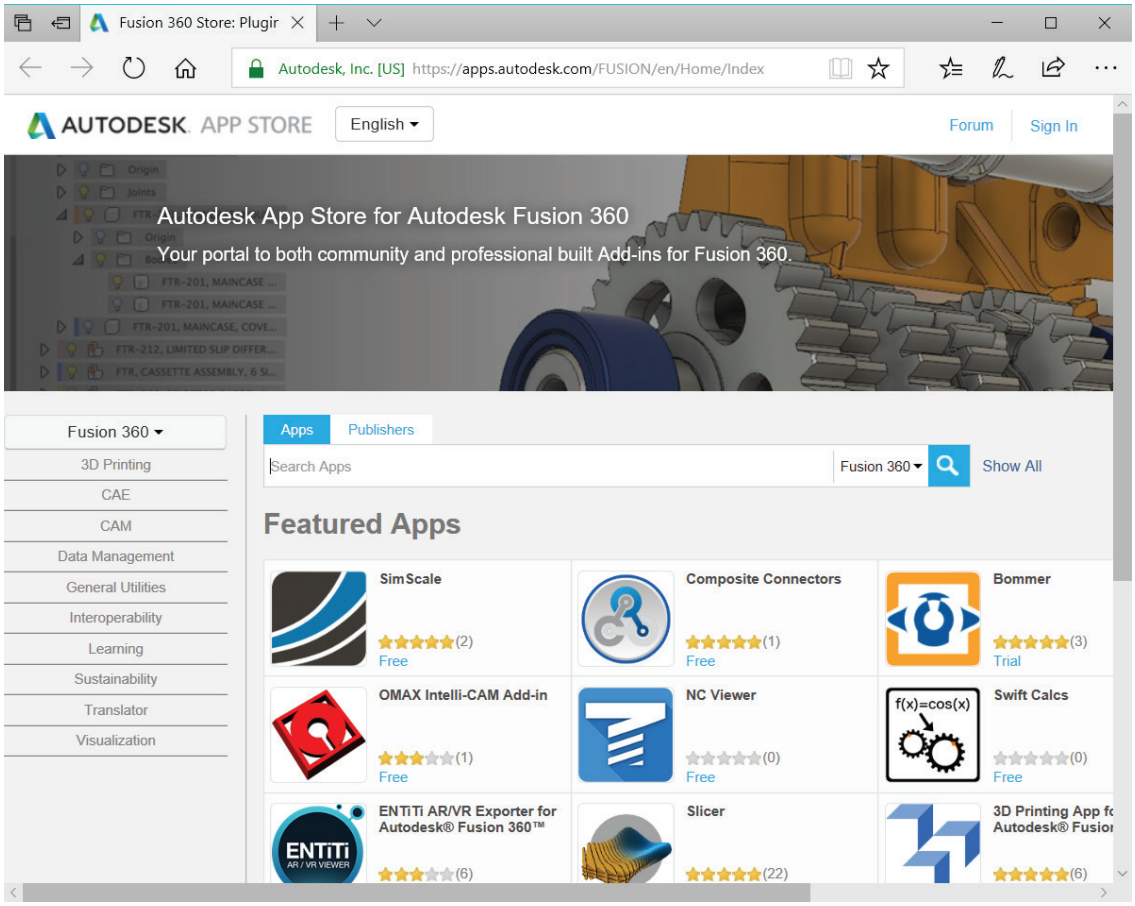

Figure-65. Fusion 360 Store webpage

- Login to your Autodesk account after clicking **Sign In** button at the top right in the web page.
- Select the desired app, set the desired Operating system and click on the **Download** button from web page; refer to Figure-66. The software will start downloading.

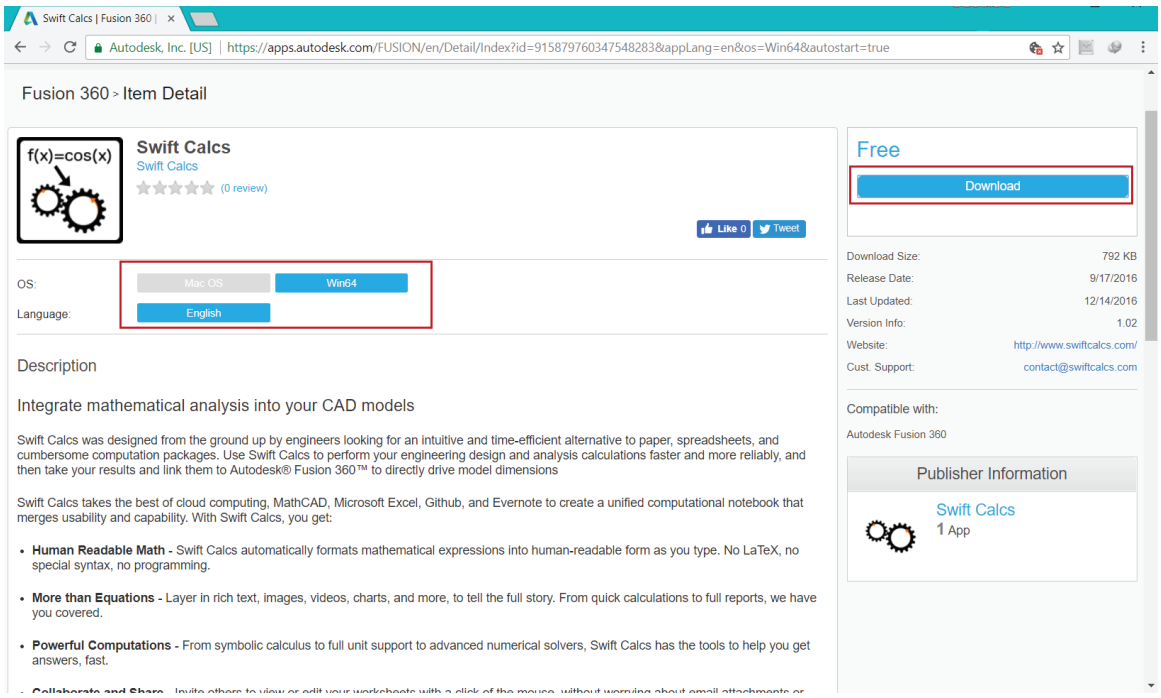

Figure-66. Downloading App

- Once download, 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-67.

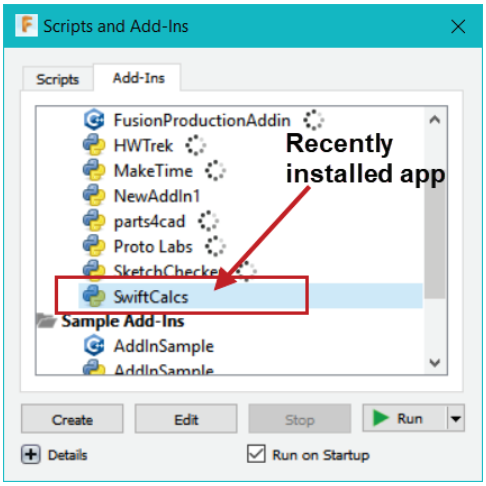

Figure-67. Installed app

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-68.
- You can't undo some actions, like clicking commands on the **File** tab or saving a file. If you can't undo an action, the Undo command changes to Can't Undo.

Figure-68. Undo and Redo

- 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 only appears 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-69. The tools in this drop-down are discussed next.

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-69. The **Preferences** dialog box will be displayed where you can specify various parameters for application; refer to Figure-70.

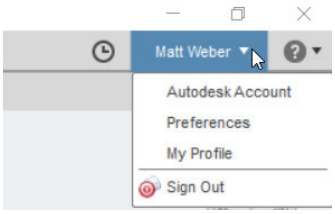

Figure-69. User Account drop-down

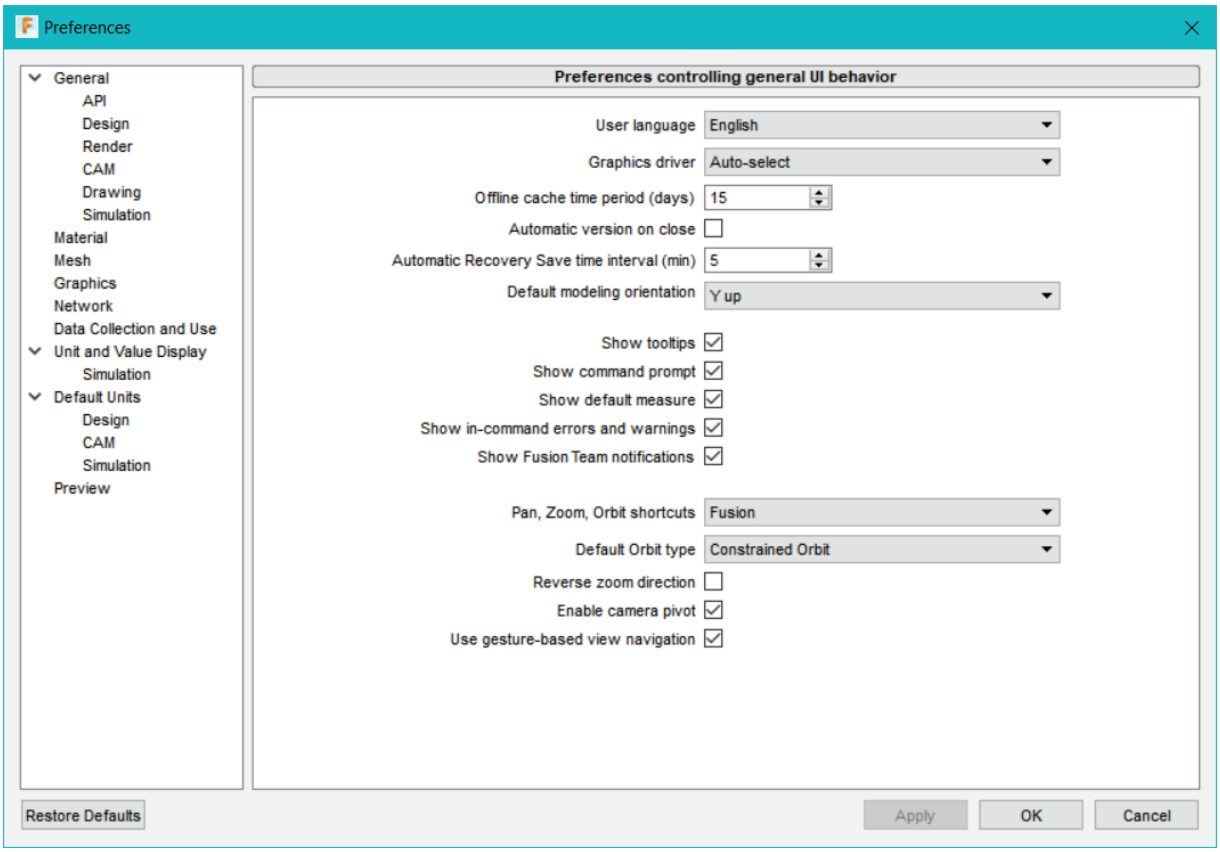

Figure-70. Preferences dialog box

General

In **General** node, you can specify the preferences for language, graphics, mouse functioning, default workspace and so on. Various important options are discussed next.

- Click on the **General** node at the left in the dialog box to display general options.
- Click on the **User language** drop-down and select the desired language for Autodesk Fusion interface.
- Click on the **Graphics driver** drop-down to select the graphics driver to be used by Autodesk Fusion.
- 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.
- Select the **Automatic version on close** check box to automatically save the newer version of file when you close Autodesk Fusion.

- Specify the desired time in minute after which a recovery copy of your model will be created in **Automatic Recovery Save time interval** edit box.
- Click on the **Default modeling orientation** drop-down and select the 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.
- Set the other options as required in General page.
- Click on the **API** option under **General** node in the left of the dialog box and set the desired parameters to define default location and language of scripts/Add-Ins.
- 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-71.

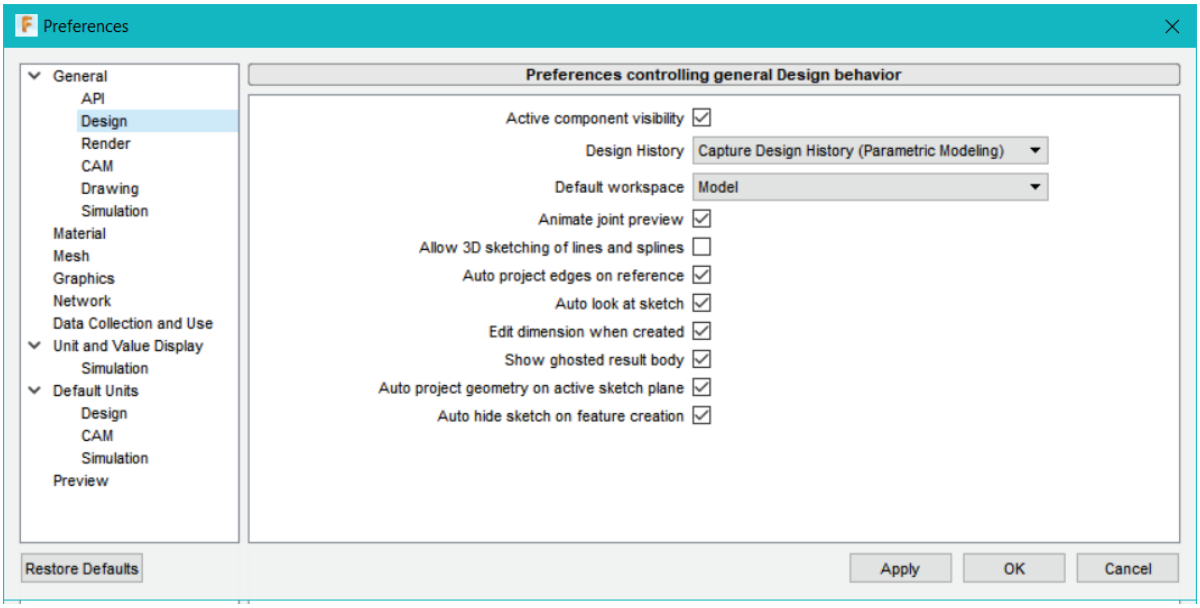

Figure-71. Design page

- 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. If you want to save space and 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 most of the options will be active after you restart Autodesk Fusion.
- Set the default workspace in **Default** workspace drop-down.
- 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.
- 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.

The **Autodesk Account** tool in **User Account** drop-down is used to manage your Autodesk account 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-72. Click on this button to toggle between offline and online mode of Autodesk Fusion.

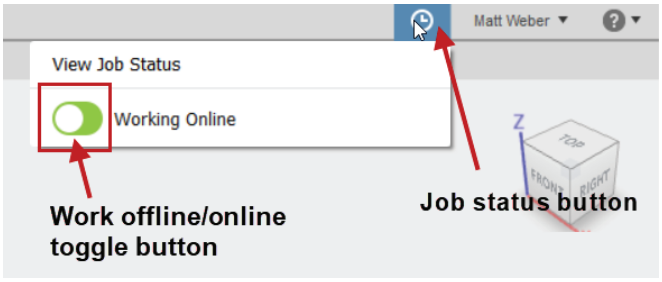

Figure-72. Work Offline

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-73. Tools in the drop-down are given next.

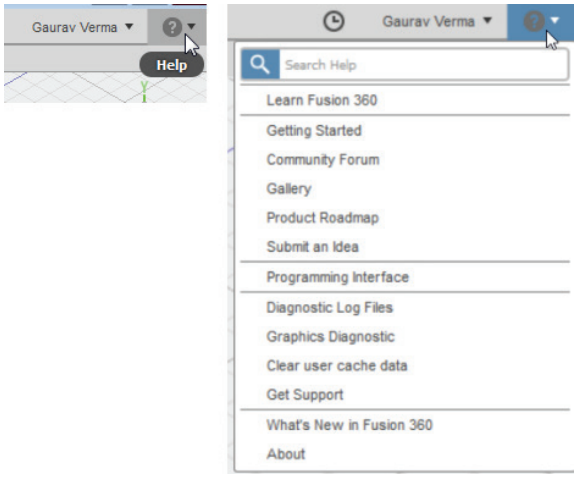

Figure-73. Help

Search Help Box

When you are facing any problem regarding tools and terms in the software,then you need to type your problem in **Search Box** and press **ENTER** to find a suitable solution to your problems; refer to Figure-74.

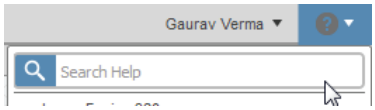

Figure-74. Search box

Learn Fusion 360

The **Learn Fusion 360** tool is used to learn about tools and topics related to software which are available on Autodesk website. On clicking this button, website of Autodesk Fusion 360 will be opened, where you can find the information related to software; refer to Figure-75.

Figure-75. Learn fusion 360

Getting Started

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

- Click on **Getting Started** tool from **Help** drop-down. The **GETTING STARTED** dialog box will be displayed.
- The navigation functions of mouse can be set as Autodesk Inventor, SolidWorks or Alias by using this tool and you can also set the units for your project; refer to Figure-76.

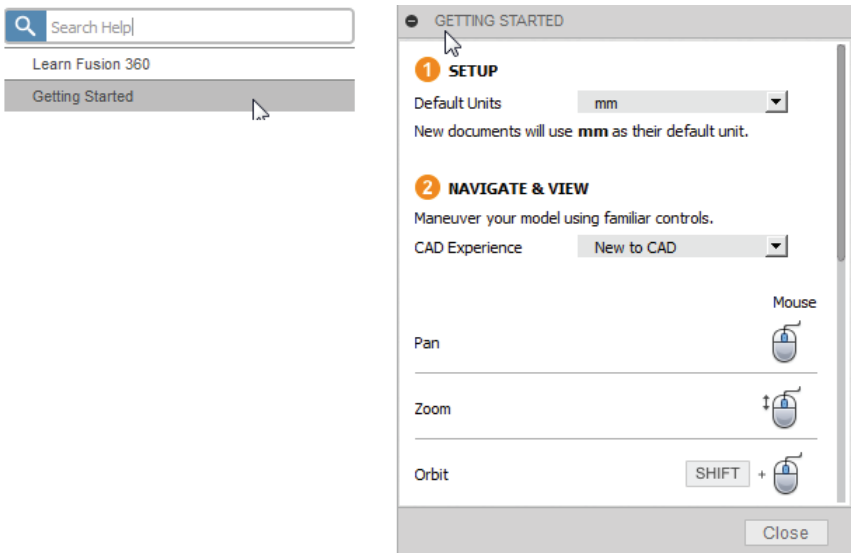

Figure-76. Getting Started

- If you are new to CAD, then you can select the **New to Cad** in **Cad Experience** drop-down menu. The mouse will function as per Autodesk Fusion 360 programming.

Similarly, you can use **Community Forum, Gallery, Product Roadmap, Programming Interface** and **Submit an idea** tool to get help from Autodesk Fusion 360 or provide feedback.

Diagnostic log Files

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 in **Help** drop-down. The **Diagnostic Log Files** dialog box will be displayed; refer to Figure-77.

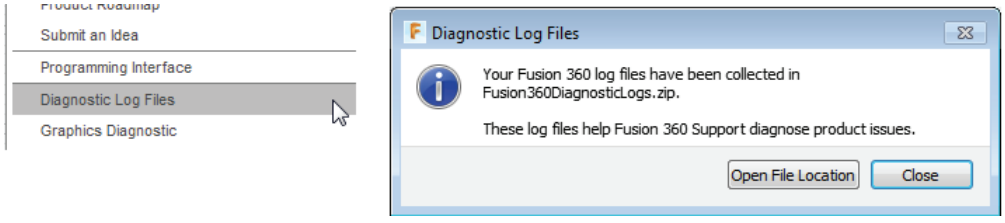

Figure-77. Daignostic Log Files

- 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

Data Panel is used to manage the Autodesk Fusion 360 projects. The projects you save in cloud will be shown here. 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 the **Show Data Panel** or **Hide Data Panel** button at upper left corner of the Fusion 360 window; refer to Figure-78.

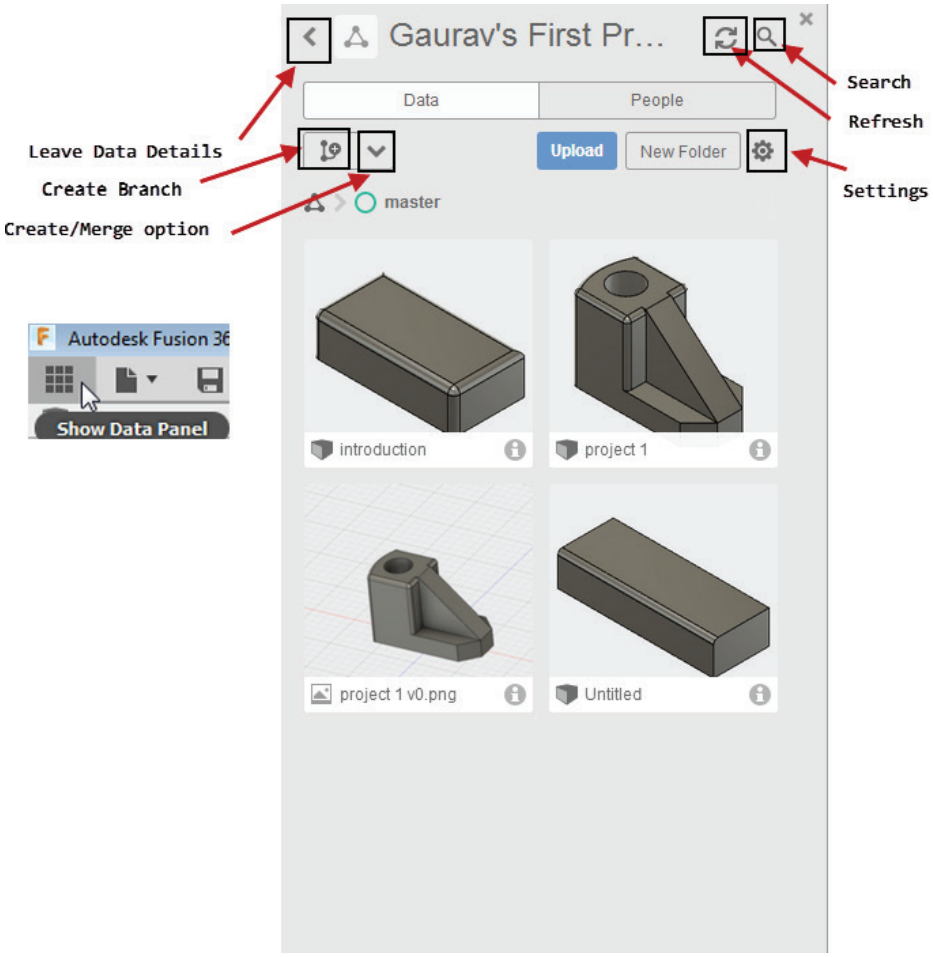

Figure-78. Data Panel

- A box will be displayed with files of current project. In this box, there are two sections; **Data** and **People**.
- In **Data section**, the files which are saved earlier by you in current project are displayed. You will be able to open any file by double-clicking on it.
- In **People** section, the information of users will be displayed who can access the files; refer to Figure-79.

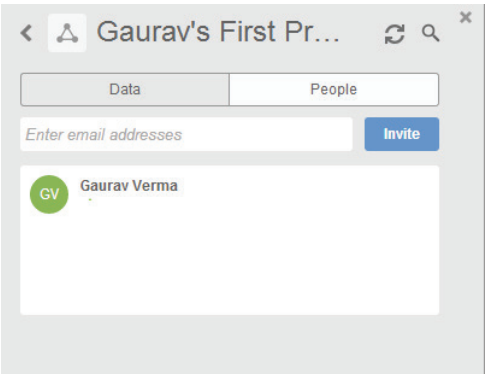

Figure-79. People section in Data panel

Branch / Merge Options

The tools in this drop-down are used to create various categories of the current project like, we have master as our main branch and we are creating mechanical tools as another branch. At any point in entire project, you are able to create the branches. This helps you to manage your projects in a different category; refer to Figure-78. The steps to use this tool are given next. (Note that Autodesk Fusion was in the process of closing preview of this tool in version 2.0.4099 so you might not be able to utilize this tool any further.)

- Click on **Create Branch** button from **Branch / Merge Options** drop-down; refer to Figure-80.

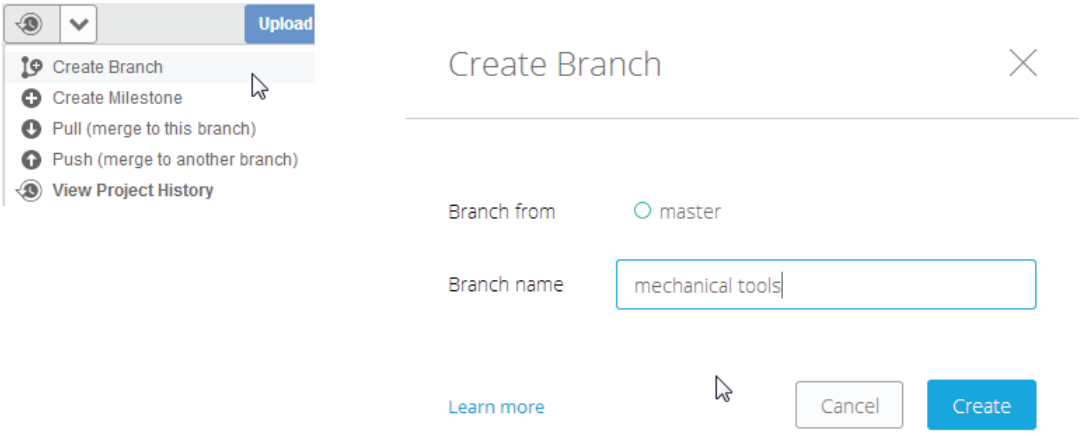

Figure-80. Create Branch

- On selecting the **Create Branch** button, the **Create Branch** dialog box will be displayed. You need to specify the location for branch in **Branch from** section and name in **Branch name** edit box. Note that once you have created a branch then you will not be able to rename or delete any created branch. File in the branch cannot be moved outside the branch or project. You can only move files from branch to master or master to master.
- After specifying the parameters, click on the **Create** button of **Create Branch** dialog box to create the Branch.

Merge Push or **Pull** commands are only available, when there are more than one branch in your project. At least one file should be available in the folder to merge branch. Empty folders will not merge.

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 **Data Panel**. The **Upload** dialog box will be displayed; refer to Figure-81. Click on the Select Files button or drag & drop the files to be uploaded.
- If you want to change the location where files will be uploaded then click on the **Change Location** button and select the desired location from the **Change Location** dialog box; refer to Figure-82. After selecting desired location, click on the **Select** button. You will return to **Upload** dialog box with updated location.
- After specifying the desired parameters, click on **Upload** button from **Upload** dialog box. File will be uploaded.

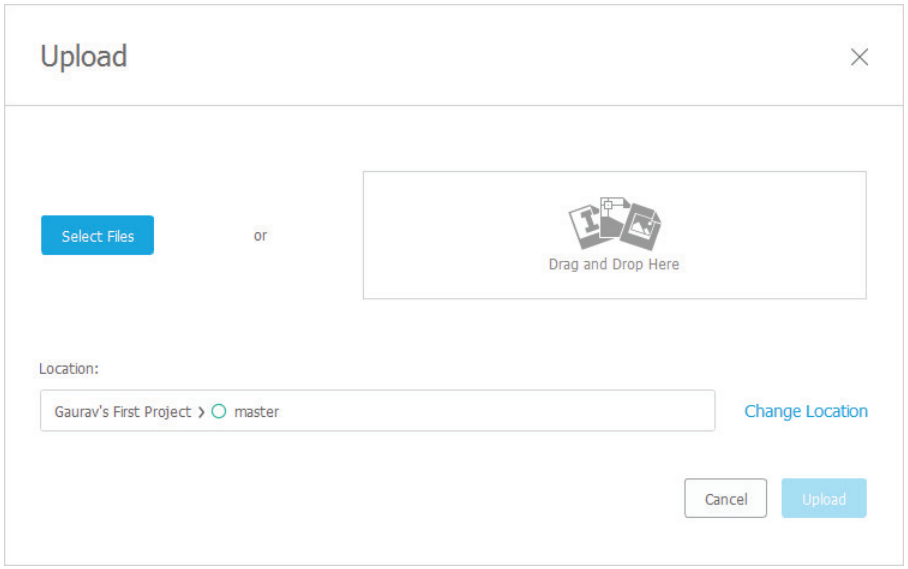

Figure-81. Upload file Data panel

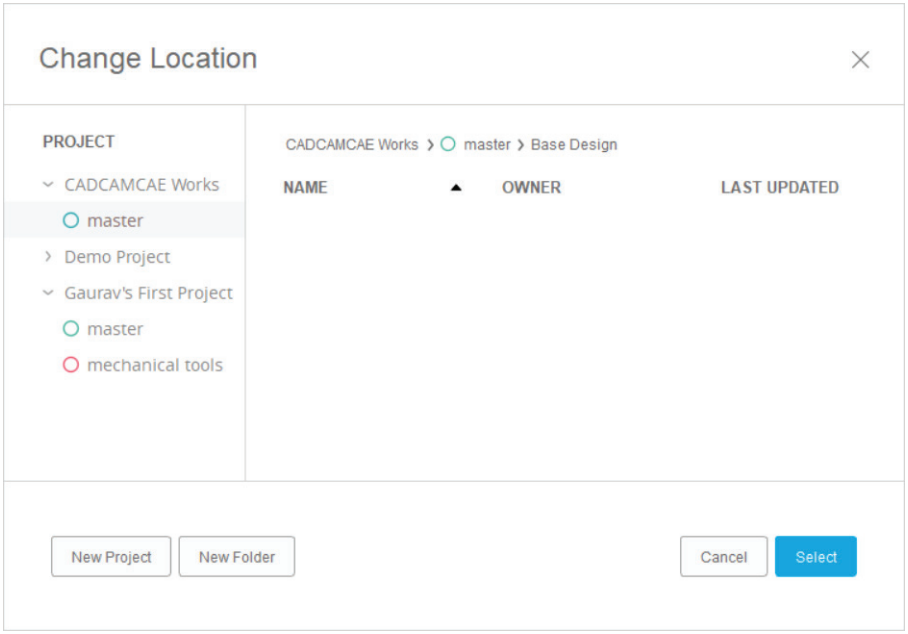

Figure-82. Change Location dialog box

The **New Folder** button in **Data Panel** is used to create a new folder. With the help of this tool, you can categorise your parts in different folders.

Cloud Account of User

To open the cloud account of user, you need to select the arrow button (**Leave Data Details** button) which is displayed at the upper left corner of the **Data Panel** box. On selecting this button, details of user's cloud storage will be displayed; refer to Figure-50.

- If you want to return to the previous screen, then click on the **Back** button which is highlighted in the figure.
- In **Projects** section, your projects are displayed. If you want to select the **My Recent Data, Demo Project** and any other project then double-click on the respective link.
- In **Samples** section, there are various samples and tutorial of design and project provided by Autodesk. To open any one of these samples double-click on the link.

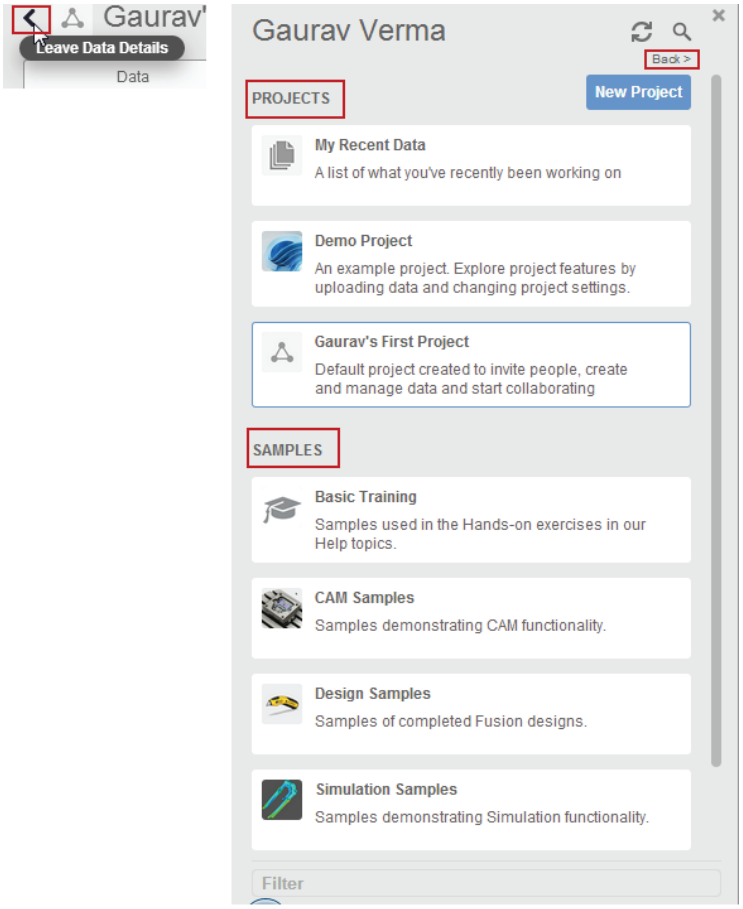

Figure-83. Leave Data Details

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-51. When you select a feature or component in the **BROWSER**, it is also highlighted in the graphics window.

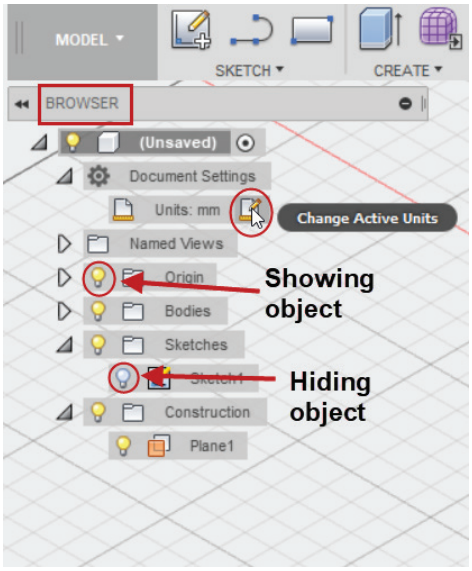

Figure-84. Browser

- Click on the **Light bulb** 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 the 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 provide access of navigation commands for design. To start a navigation command, click on any tool from the **Navigation Bar**; refer to Figure-85.

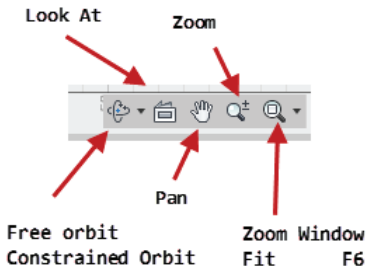

Figure-85. Navigation Bar

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.

Look At- This tool is used to view the selected of model parallel to screen. To use this tool, select the 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 and drag mouse upward/downward to zoom out/in respectively. You can do the same function by rolling the mouse wheel up/down.

Zoom Window and Fit- **Zoom window** tools is 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-86.

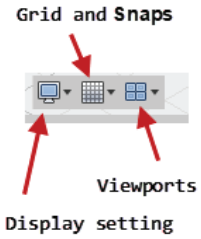

Figure-86. Display Setting

Display Setting- The **Display Setting** tool is used to enable or disable various commands like **Desired Visual Style, Visibility of objects,** and **Camera settings**, and so on. If you want to run Autodesk Fusion 360 in full screen then press **CTRL+SHIFT+F** from keyboard.

Grid And Snaps- The **Grid and Snaps** flyout is used to activate or deactivate various interface objects 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 **Viewports** tool is used to display the model in four viewports at the same time on the Fusion 360 screen. You can set these views asynchronous by clearing the **Synchronize Views** check box from **Viewports** flyout after activating multiple viewports mode. 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-54. Right-click on the **ViewCube** to access options related to perspective mode and other view modes.

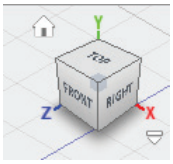

Figure-87. ViewCube

Mouse Functions

You can use the mouse shortcuts to zoom in/out, pan the view, and orbit the view, refer Figure-88.

- Scroll the Middle Mouse Button downward/upward to zoom in or zoom out, respectively.
- Click and hold the Middle Mouse Button to pan the view.
- Use the **Shift** + Middle Mouse Button to orbit the view.
- Click on the left mouse button select any object or tool.
- Click on the right mouse button (right-click) to access shortcut menus in the software.

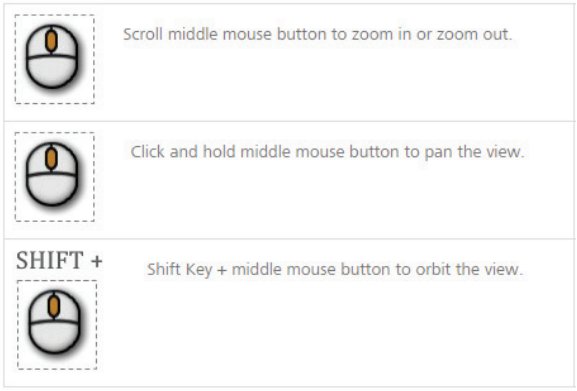

Figure-88. Mouse shourtcut keys

Timeline

The **Timeline** bar is very useful for saving time while editing. It records the design feature in chronological order; refer to Figure-56. 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-89. 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 the desired changes and apply the parameters. The final design will be generated automatically with the applied changes; refer to Figure-90.

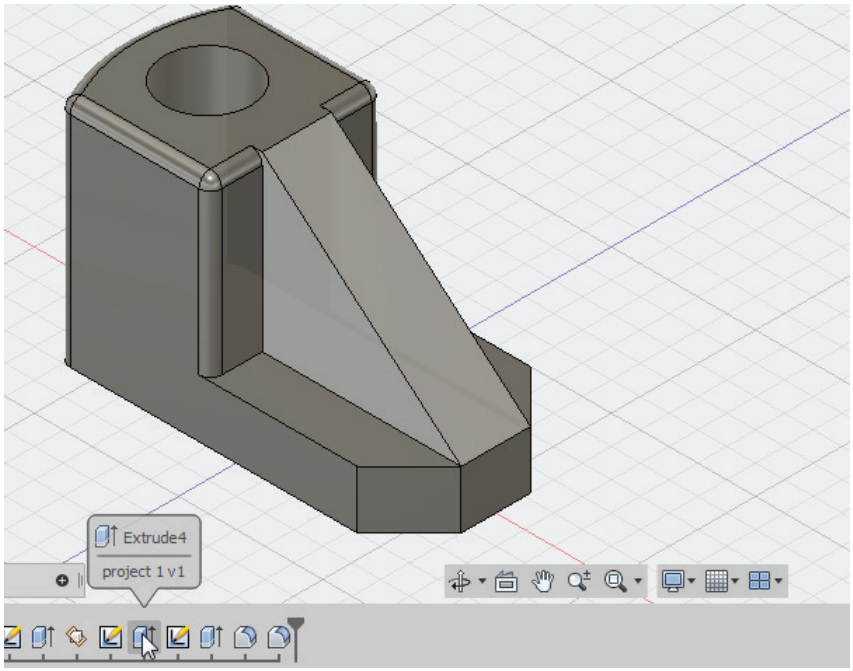

Figure-90. Editing model from timeline

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.

For Student Notes

Chapter 2

Sketching

Topics Covered

The major topics covered in this chapter are:

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

INTRODUCTION

In Engineering, sketches are based on real 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 form base feature for various 3D operations. Note that the sketching environment is the base of 3D Models so you should be proficient in sketching.

In this chapter, we will be working in **Model Workspace**. 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 **SKETCH** drop-down in **Toolbar**; refer to Figure-1. The three primary planes will be displayed on canvas screen; refer to Figure-2.

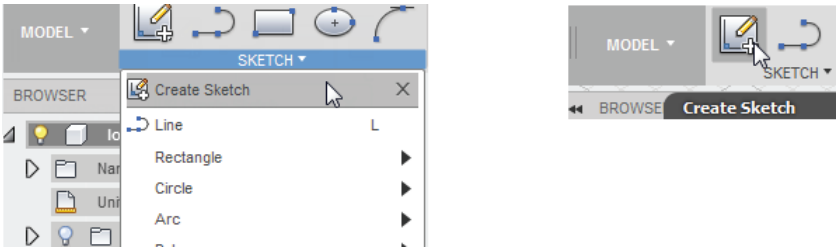

Figure-1. Create Sketch

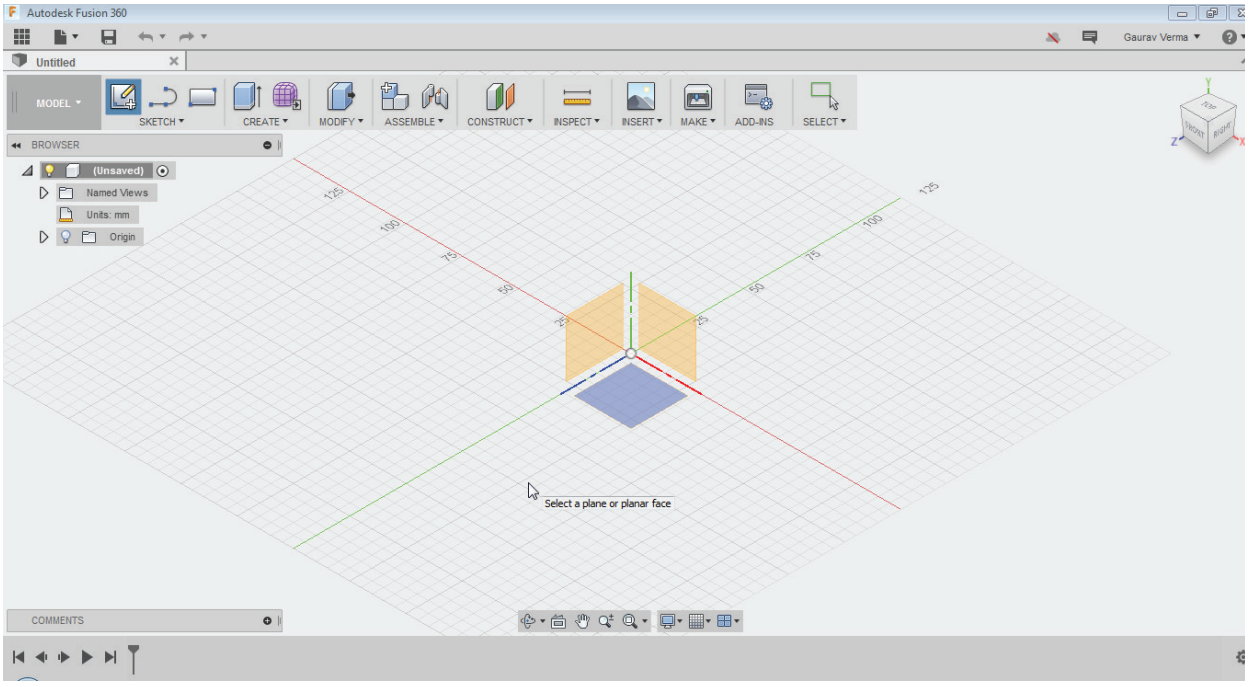

Figure-2. Select Plane for Sketch

- Click on the desired plane from the canvas screen. The selected plane will become parallel to the screen and act as current sketching plane. Now, we are ready to draw sketch on the selected plane.

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

SKETCH CREATION TOOLS

In **SKETCH** drop-down, there are various tools for creating sketch entities. These tools are discussed next.

Line

The **Line** tool is used to create a line. The procedure to use this tool is discussed next.

- Click on **Line** tool of **SKETCH** drop-down from **Toolbar**; refer to Figure-3. You can also press **L** key to select the **Line** tool. You will be asked to specify start point of the line.

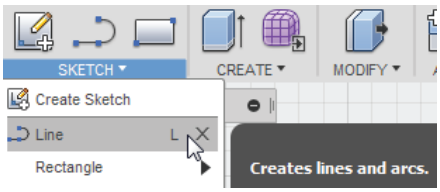

Figure-3. line

- 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** edit boxes will be displayed with the preview of line.
- Enter the desired value of length in the **Length** edit box. Press **TAB** key to toggle to **Angle** edit box; refer to Figure-4.

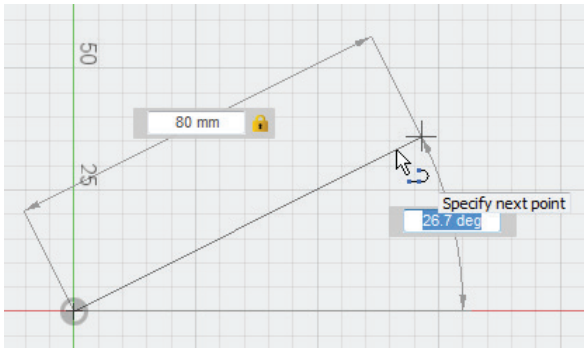

Figure-4. making of line

- If you are making 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-5. (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-6.

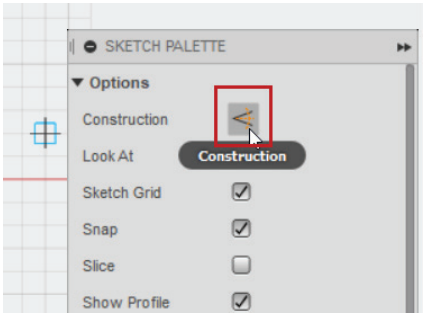

Figure-5. Construction button in SKETCH PALETTE

Marking menu

It is the radical display of most frequently used tool and tools related to selected entities. This menu also provide 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.

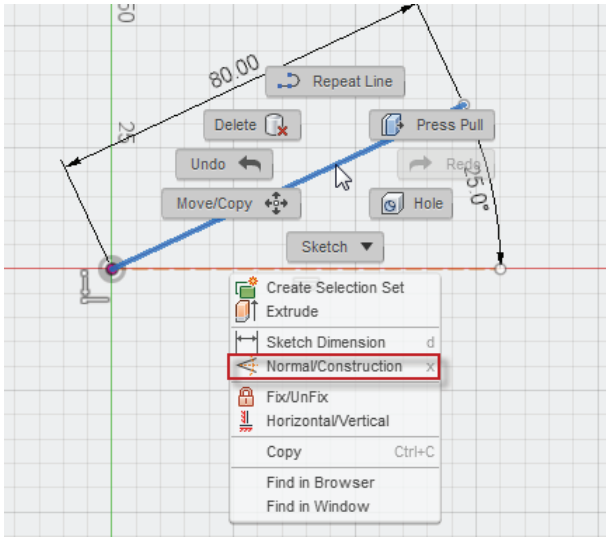

Figure-6. Marking menu

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-7.

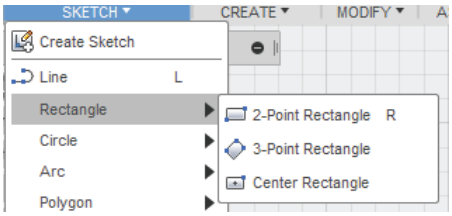

Figure-7. Rectangle drop-down

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

2-Point Rectangle

- Click on the **2-Point Rectangle** tool of the **Rectangle** cascading menu from **SKETCH** drop-down. 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-8.

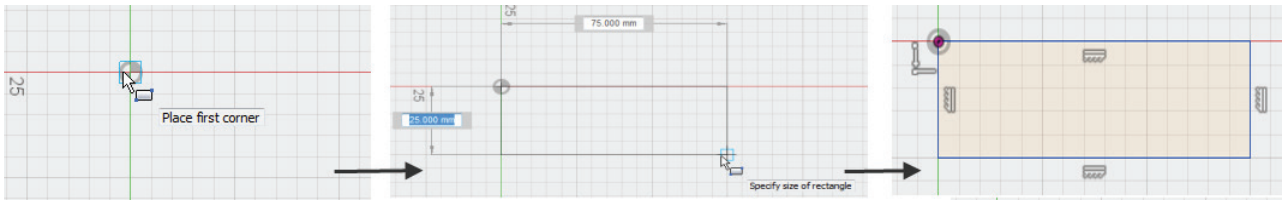

Figure-8. 2-Point Rectangle tool

- 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 last 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 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 **SKETCH** drop-down. You will be asked to specify location of first corner point of rectangle.
- Click at the desired location to specify first corner point of the rectangle on the sketch canvas; refer to Figure-9.

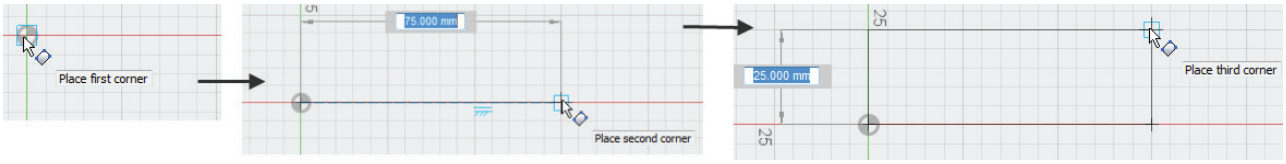

Figure-9. Creation of 3-point rectangle

- Click to specify the second corner point (end point of base like of rectangle) on the canvas or specify the dimension in dimension box.
- Click to specify the third corner point of the rectangle to create the complete rectangle creation.

Center rectangle

- Click on the **Center Rectangle** tool of the **Rectangle** cascading menu from **SKETCH** 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-10.
- Enter the dimension for rectangle in the edit boxes and press **ENTER** to create the rectangle.

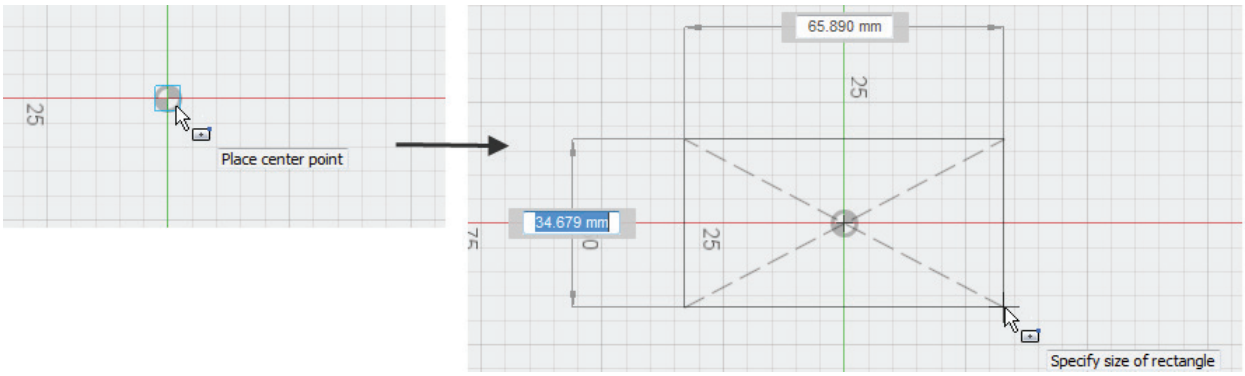

Figure-10. Creation of center rectangle

- If you want to create free style rectangle then click on the screen to select the center point and corner point of rectangle.

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-11.

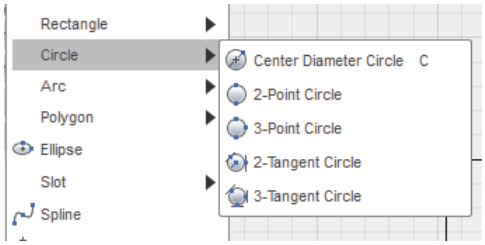

Figure-11. Circle tool

The shortcut key to use the **Center Diameter Circle** tool is 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.

- Click on the **Center Diameter Circle** tool from **Circle** cascading menu.
- Click to specify the center point for the circle; refer to Figure-12.

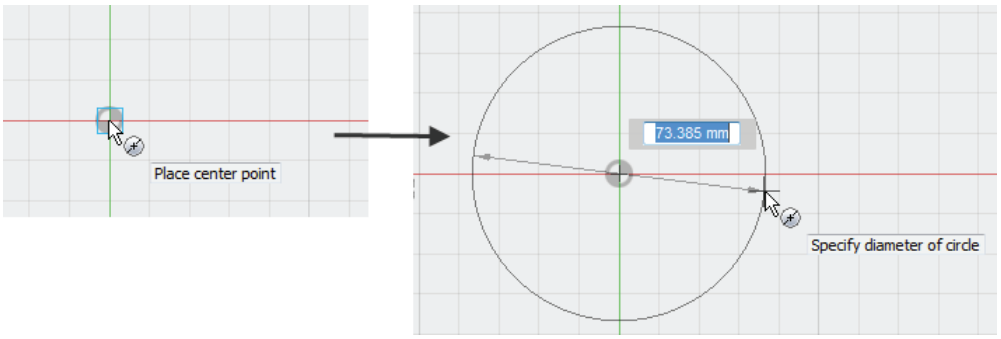

Figure-12. Creation of center diameter circle

- 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 **SKETCH** drop-down and specify the first point for circle; refer to Figure-13.

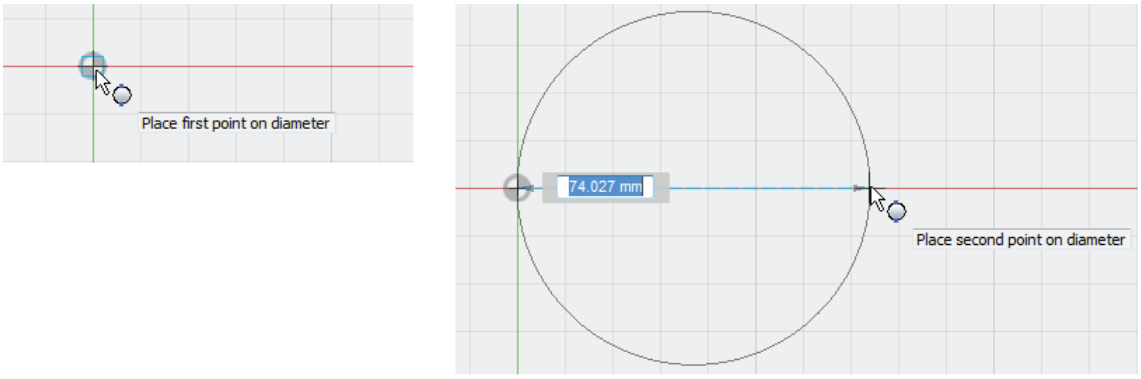

Figure-13. Creation of 2-point circle

- Specify the second point on the canvas screen or enter 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 from **Circle** cascading menu of **SKETCH** drop-down.
- Click on the screen to specify the first point, second point and third point of circle; refer to Figure-14.

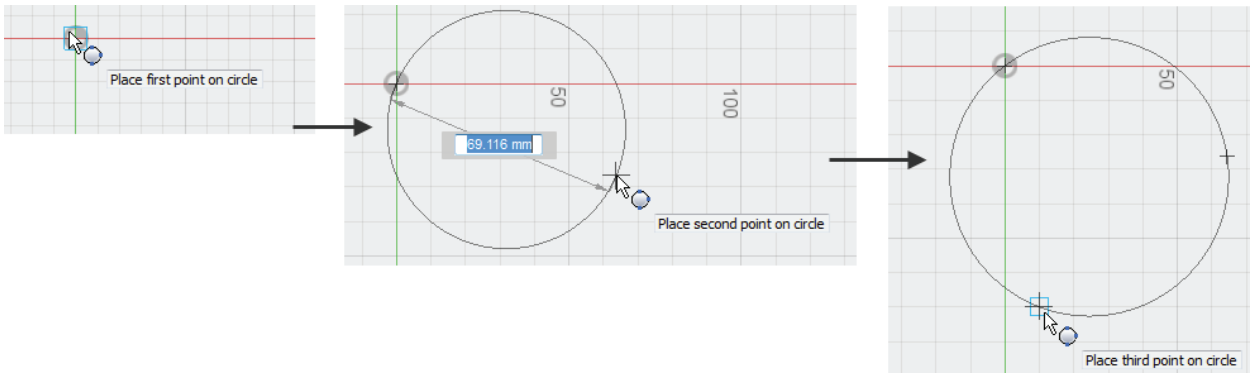

Figure-14. Creation of 3-point circle

- 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 **SKETCH** drop-down.
- Click at desired locations on lines that should be tangent to the circle; refer to Figure-15. Preview of circle will be displayed

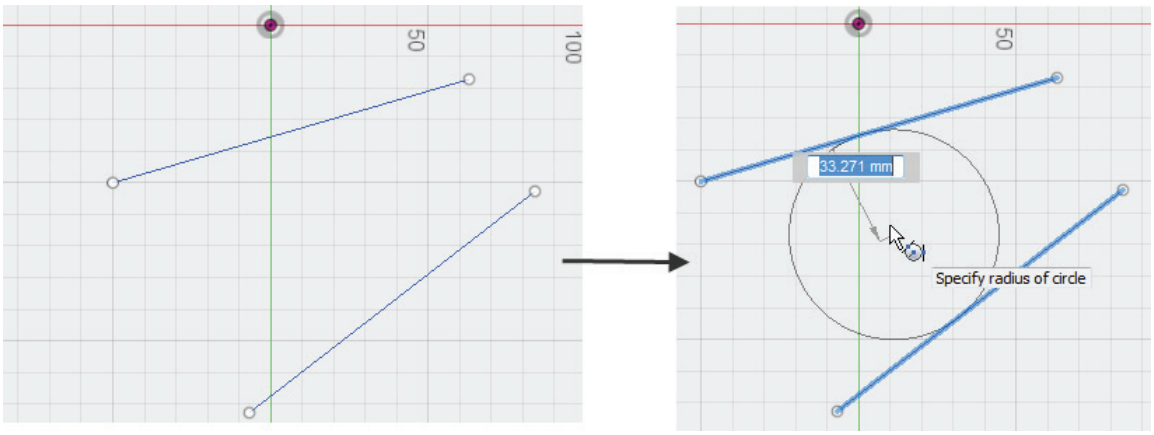

Figure-15. Creation of 2-tangent circle

- Enter the desired radius of the circle in the floating window and press **ENTER** to create a circle or click at the desired location to create 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 **SKETCH** drop-down.
- Select three lines to be tangent to the circle. You can use perform window selection to select the lines; refer to Figure-16.
- A preview of circle will be displayed. 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 tool.

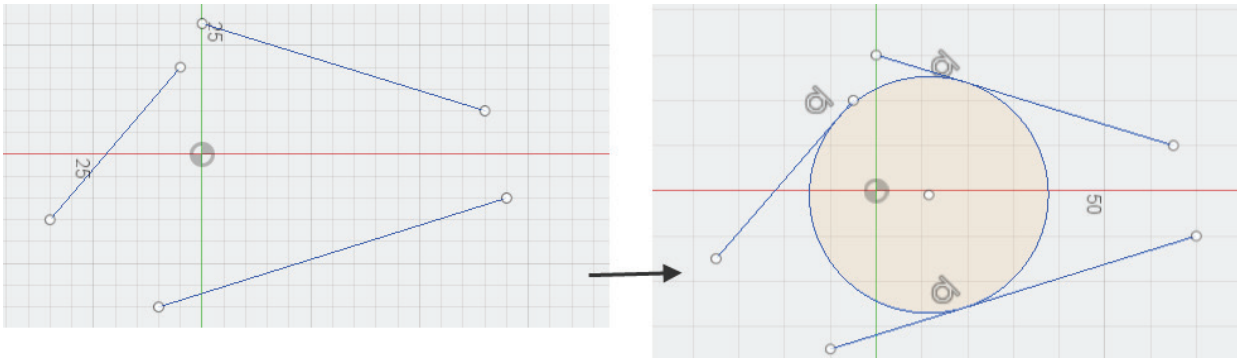

Figure-16. Creation of 3-tangent circle

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-17.

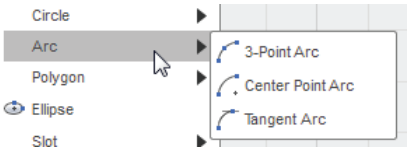

Figure-17. Arc

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 **SKETCH** drop-down. Select the desired plane or face from canvas screen if not selected yet.
- Click on the screen to specify the start point of the arc; refer to Figure-18.

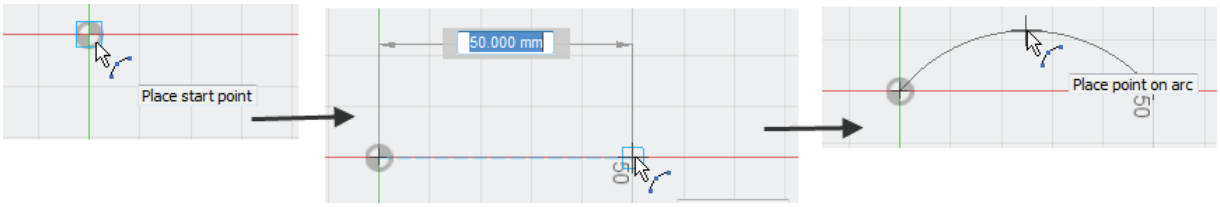

Figure-18. Creation of 3-point arc

- 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 **SKETCH** 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 enter desired value of radius in dynamic input box.
- Enter the 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-19.

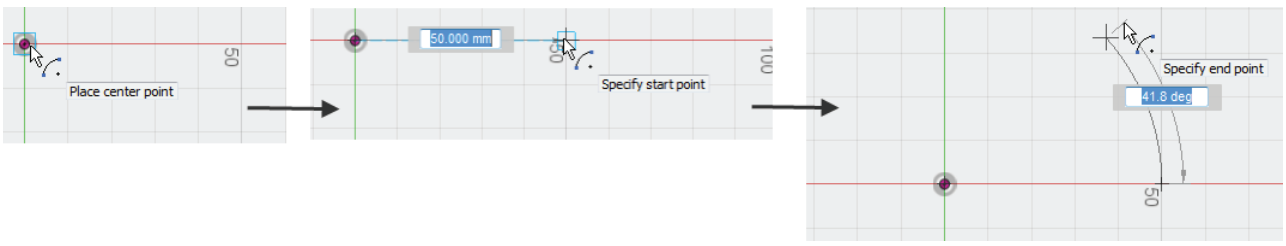

Figure-19. Creation of center point arc

- 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 from **Arc** cascading menu in **SKETCH** 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-20. A rubber-band arc will be attached to cursor starting from selected point.
- Click at the desired location to specify the end point of the arc.

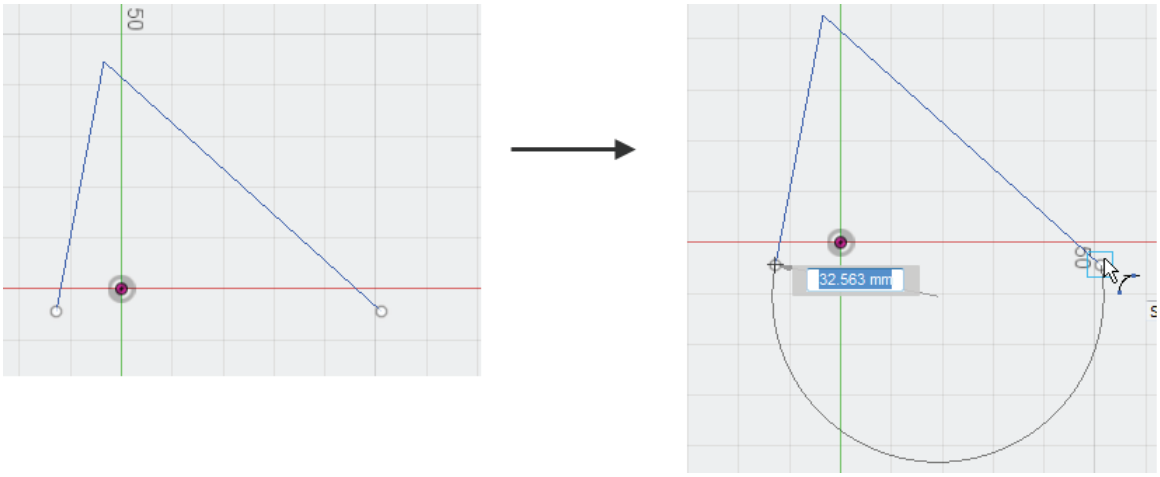

Figure-20. Creation of tangent arc

- 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-21. 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-22.

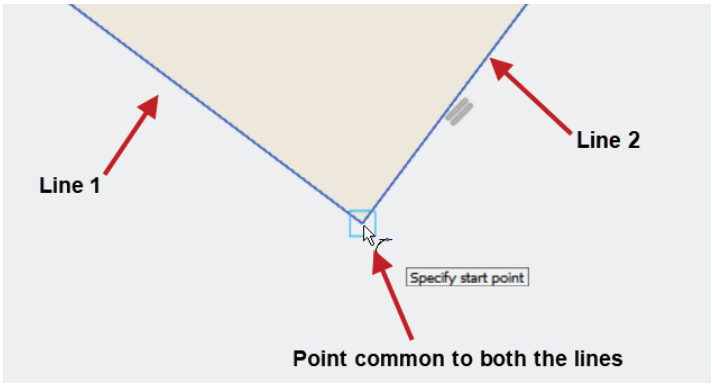

Figure-21. Tangent arc problem at corner point

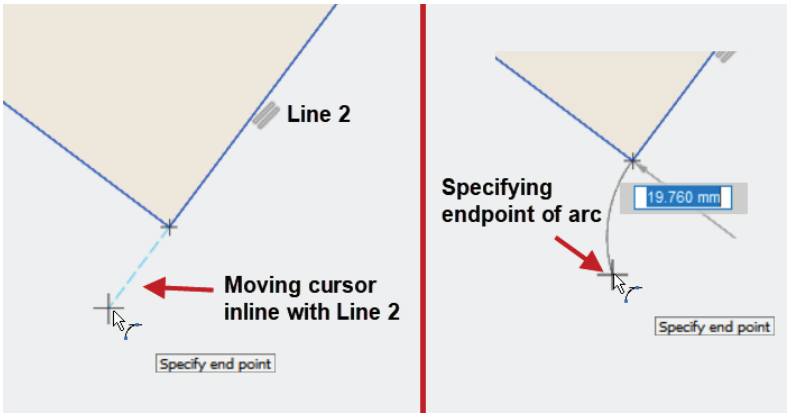

Figure-22. Creating arc tangent to line 2

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-23. The tools to create the polygons are discussed next.

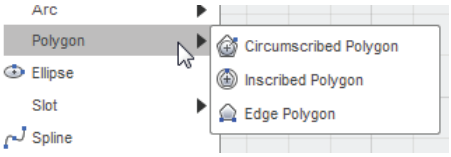

Figure-23. Polygon

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 from **Polygon** cascading menu in **SKETCH** drop-down; refer to Figure-23. 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-18. You will be asked to specify radius of reference circle.

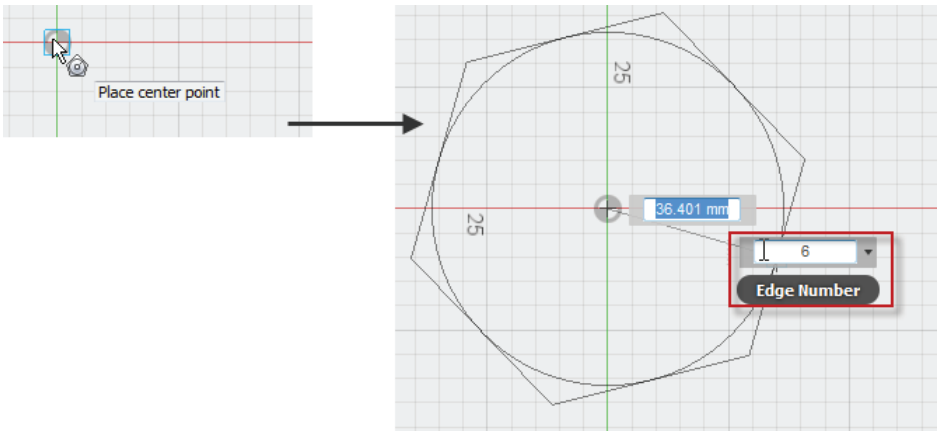

Figure-24. Creation of circumscribed polygon

- Enter 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 the **Polygon** cascading menu from **SKETCH** 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-25.
- Enter the distance between center point of polygon and vertex of polygon edge in dynamic input box.

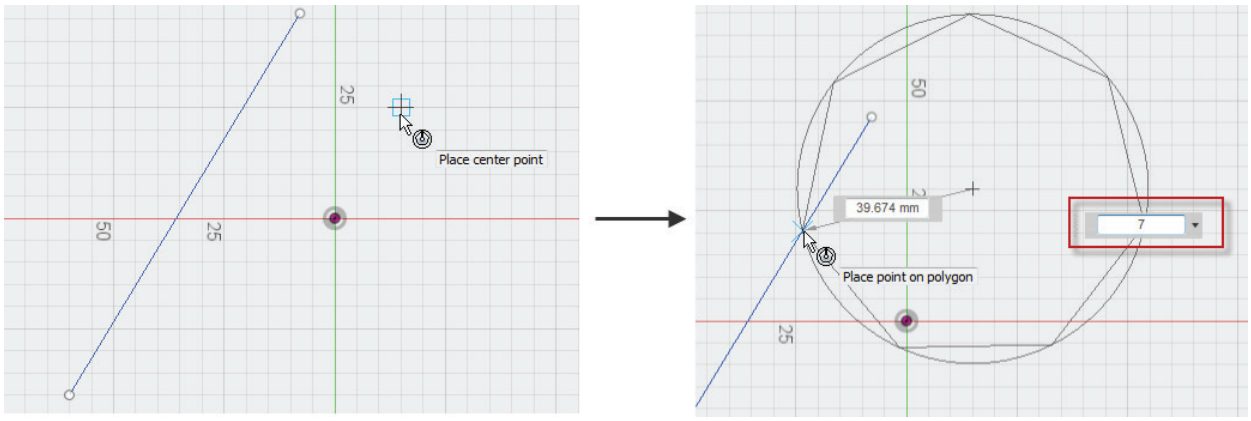

Figure-25. Creation of inscribed polygon

- Press **TAB** and specify the value of number of edges in **Edge Number** dynamic input box.
- Press **ENTER** to create 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 the **Edge Polygon** tool of the **Polygon** cascading menu from **SKETCH** drop-down.
- Click on the screen to select the start point and end point of edge. You can also specify the value of angle and distance of the edge in dynamic input boxes; refer to Figure-26.
- Specify the value of number of edges of polygon in **Edge Number** edit box.

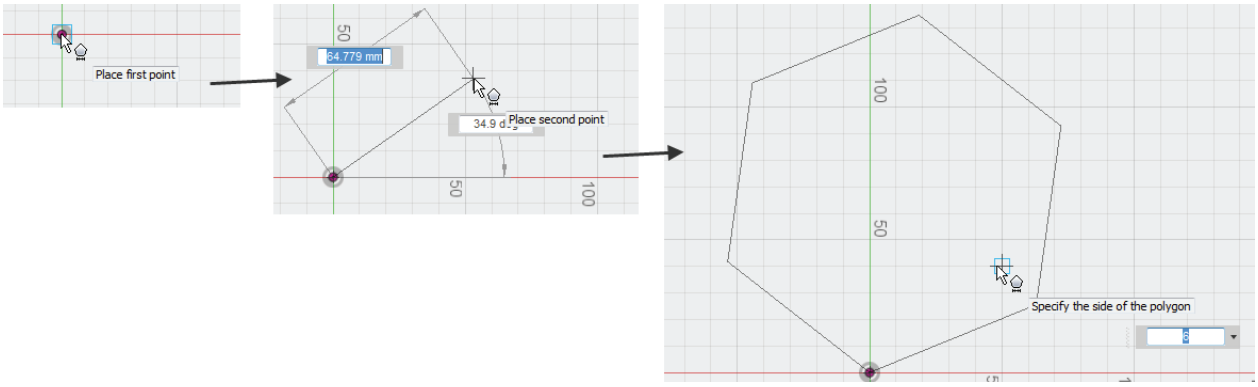

Figure-26. Creation of edge polygon