3D Modeling with NX 12

Introduction, Form Features and Feature Operations

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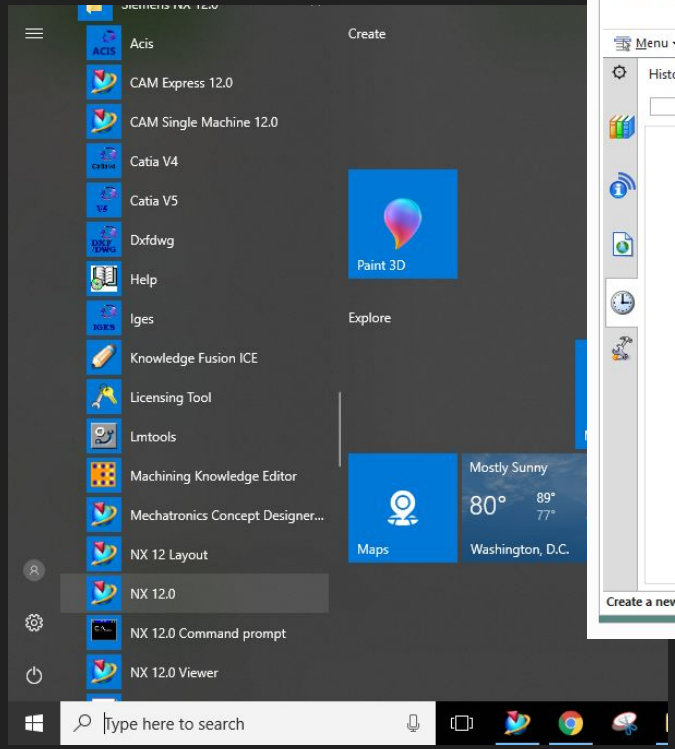
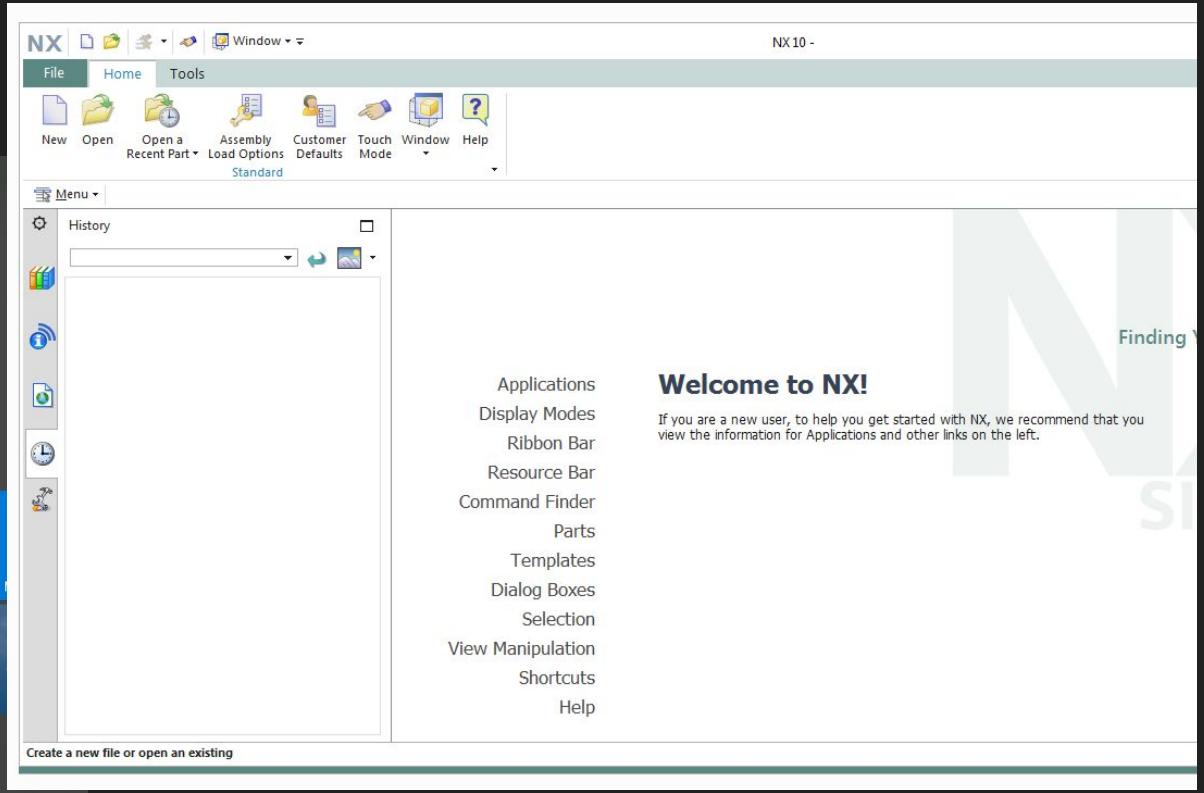
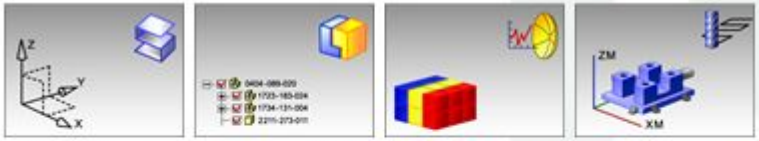
Starting NX





There’s no magic here. Find NX in your device and start it.

Template



When you create a new NX file, select a template in the **File New** dialog box. Most Drawing, Manufacturing, and Simulation templates automatically select the 3D geometry part you are working on as the part to reference. This automates the *master model* approach.

Your company can create custom templates in the **File New** dialog box. For example, your company can define standard drawing frames and views once in a template, and you can reuse these for every new drawing.

Parts

You can open as many parts as you like at the same time.

Press Ctrl+Tab or choose **Switch Window** on the Quick Access toolbar to scroll through loaded parts.

When you open a file that contains an assembly, NX also opens the file for each component, as needed.

When you create a drawing, create it in a new file that references the part or assembly as the master model. Different users can then work on the master model and the drawing at the same time.

Part models, assemblies, drawings, and most other objects that you create in NX are saved as datasets in Teamcenter, and are identified by part numbers. If you run NX without Teamcenter, files are saved with the .prt extension.

Application

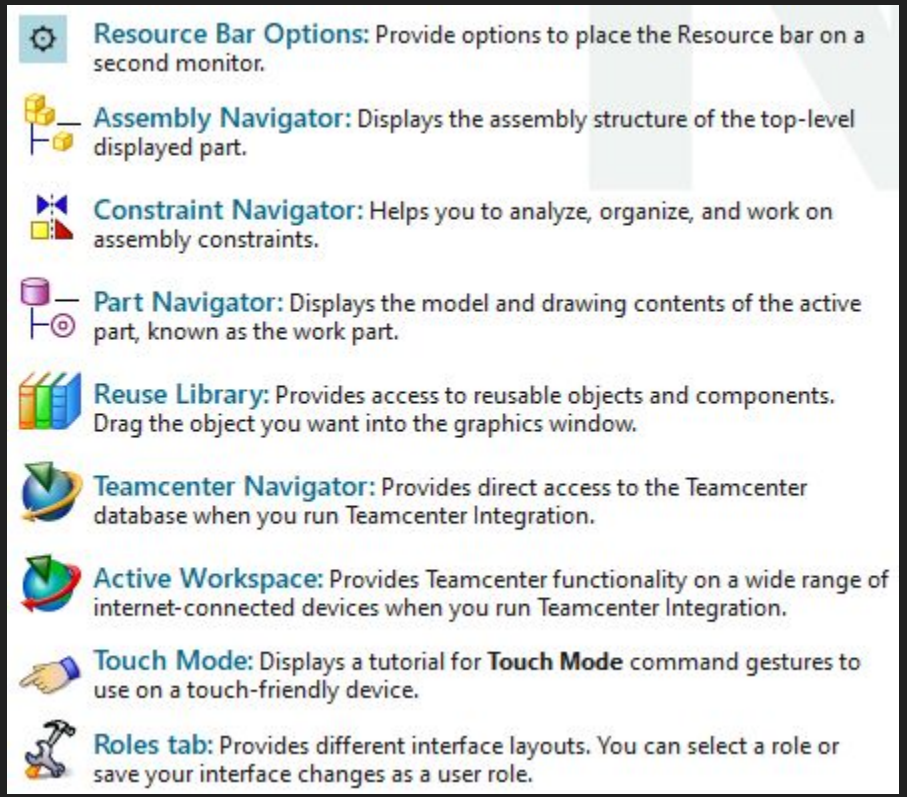
NX contains a series of applications that support different workflows, such as creating parts, building an assembly, or producing a drawing.

When you open a part, NX opens it in the application in which it was last saved.

When you create a part, NX starts the appropriate application for the template that you select.

You can switch between applications from the **Application** tab on the Ribbon bar.

Resource Bar

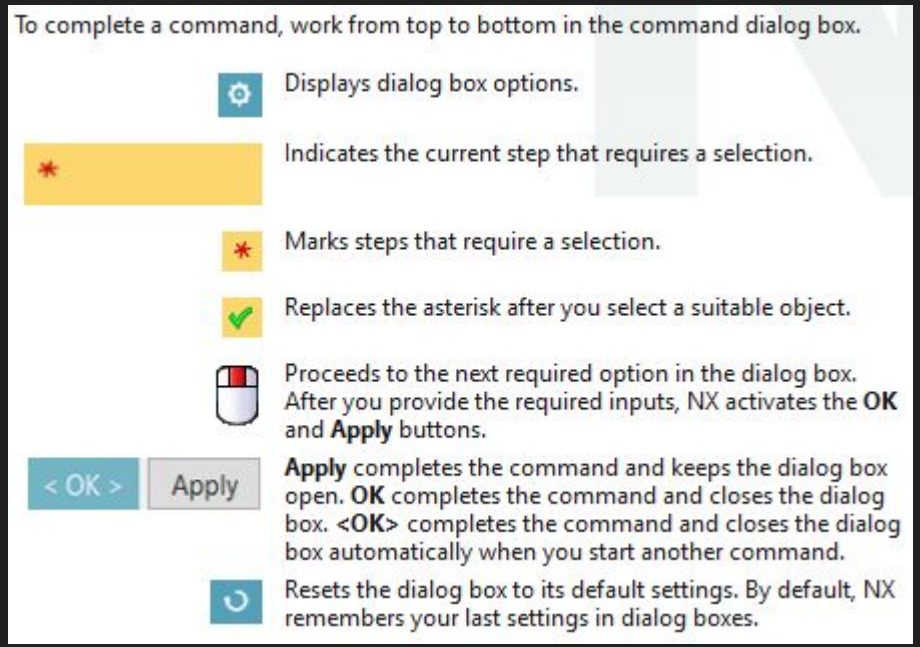


Command Finder

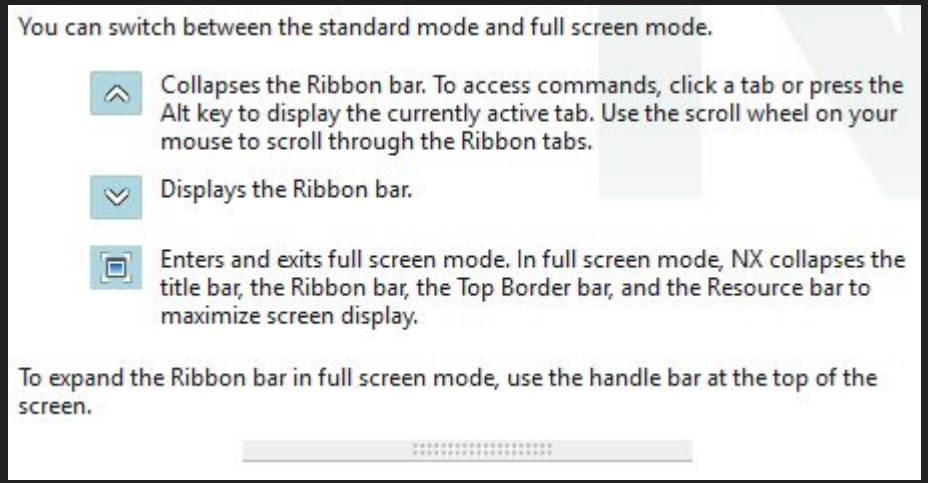
Displays the command you search for and other related commands in the **Command Finder** dialog box.

- To start a command, click it.
- To show a currently hidden command, right-click the command name and choose **Show**.
- To add a command to your preferred location, right-click the command and choose one of the Add options.
- To access context-specific help for a command, right-click the command and choose **Help**.
- To find the command in another application, right-click the command to start the application.

Dialog Boxes



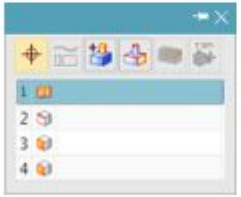
Display Mode



Selection

Shortcut toolbar: Provides access, at the cursor location, to the commands you are most likely to use on the selected object.

Selection shortcut toolbar: Provides access, at the cursor location, to selection filter settings. To display this toolbar, right-click in the background of the graphics window.



QuickPick: Lets you select a specific object when multiple objects are close together. To display this dialog box, hover over the object, and when three dots appear next to the cursor, click.

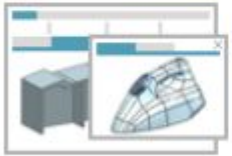
Selecting objects: To select an object, click it. To select multiple objects, continue to click objects, or use the rectangle or lasso gesture on the Top Border bar. To deselect an object, hold Shift and click the object, or press Esc to deselect all objects.

Multiple Windows

Open parts in tabbed windows: By default, NX opens a part in a tabbed window next to the active tab.

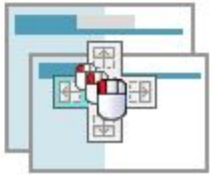
View multiple parts at once: To divide the main NX window into groups, choose a layout on the **View** tab. You can also drag windows between the groups, and simultaneously view multiple parts or different views of the same part.

Grouping windows to display multiple parts is especially useful when you work with assemblies and drawings. When you display related designs and edit a part, you can view the update to the related assemblies and drawings in real time.



Dock and float windows: To display a part in its own window, drag its window tab. To redock the window, right-click the window tab and choose **Dock to Main**, or drag the window tab over another window's **Dock Control**.

Work seamlessly across windows: To switch between parts, click any window tab or in any window. NX automatically starts the application and restores most navigator settings, so you can continue working where you left off.



View Manipulation

Viewing parts: The easiest way to orient a view is with a 3D input device. You can also orient the view using the mouse.

rotate *pan* *zoom*

To find other view options, click the **View** tab or right-click in the background of the graphics window.

Rendering styles: To change the rendering style, click the **Render** tab or right-click in the background of the graphics window and choose a style from the **Rendering Style** menu.

True shading: To quickly set up a photo-realistic real-time display, click the **Render** tab.

Customization

Right-click a command or group to quickly add it to a location such as a Border bar or the Quick Access toolbar.

Use the **Customize** command to further customize the user interface to suit your workflows.

You can hide the Ribbon bar or customize it as follows:

- Create, show, or hide tabs or groups of commands.
- Drag commands between groups.
- Change icon sizes.

You can also show or hide tooltips, and assign your own shortcut keys.

To find a command quickly, use the **Search** box on the **Commands** tab.

Shortcuts

Radial toolbars: Hold Ctrl+Shift and press a mouse key to see a radial toolbar. The toolbar you see depends on the application that you are in and the mouse key that you press.

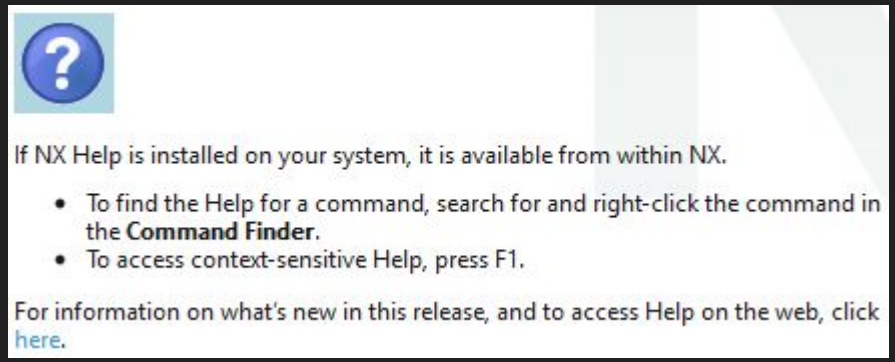
Radial shortcut toolbars: Hold the right mouse button on an object to see a radial shortcut toolbar.

View shortcut toolbar: Click in the background of the graphics window or hold the Ctrl key and click an object to see this toolbar.

View radial toolbar: Hold the right mouse button to see this radial toolbar.

Shortcut keys: Type *shortcuts* in the **Command Finder** to find a complete list of keyboard shortcuts.

Help



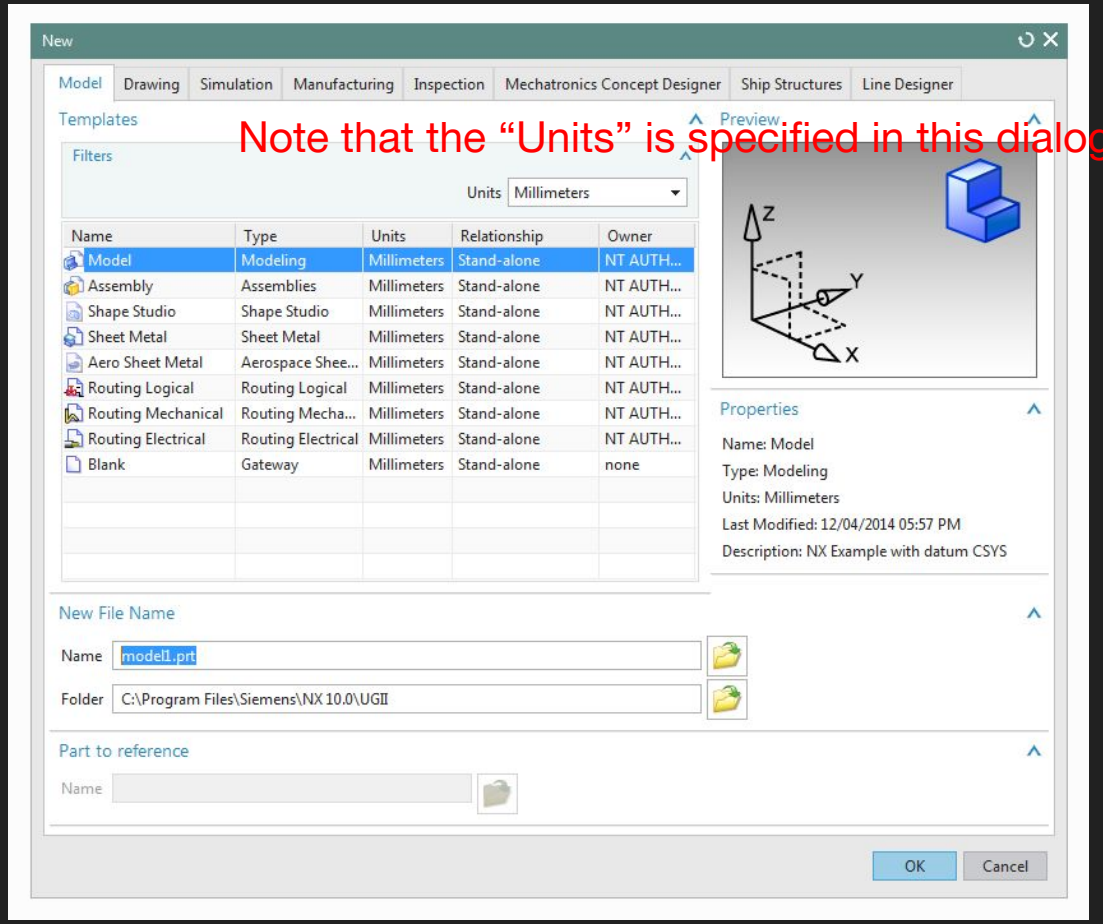
If NX Help is installed on your system, it is available from within NX.

- To find the Help for a command, search for and right-click the command in the **Command Finder**.
- To access context-sensitive Help, press F1.

For information on what's new in this release, and to access Help on the web, click here.

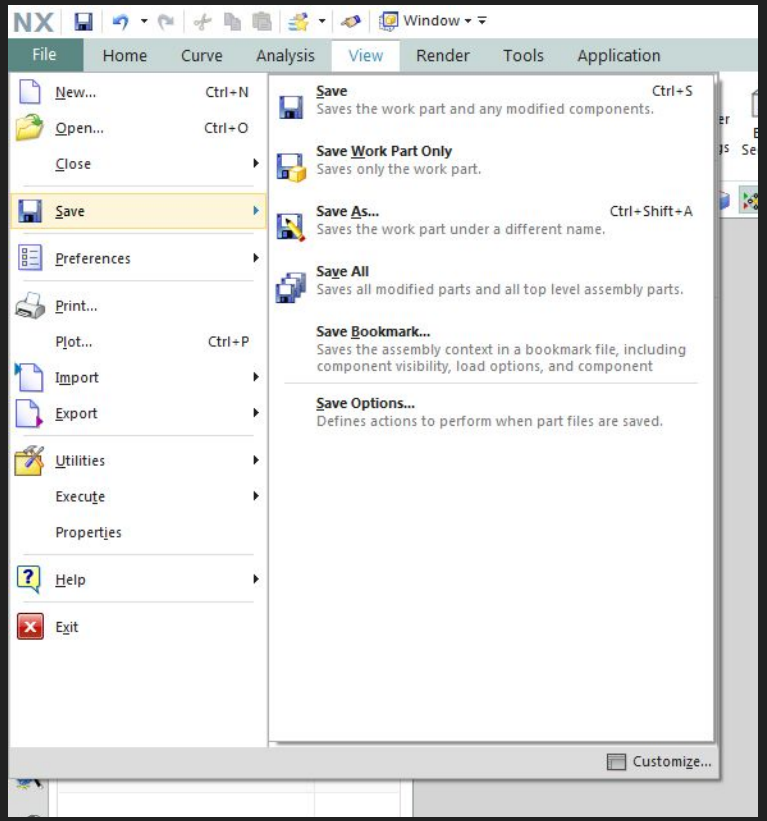
File Options

- New
  To create a new file from a template
- Open



File Save

- SAVE will save the part with the same name
- SAVE WORK PART ONLY will save the active part on the screen
- SAVE AS will save the part using a different name
- SAVE ALL will save all the opened part files with their existing names
- SAVE BOOKMARK will save screenshot of the current model on the screen as JPEG files and bookmarks

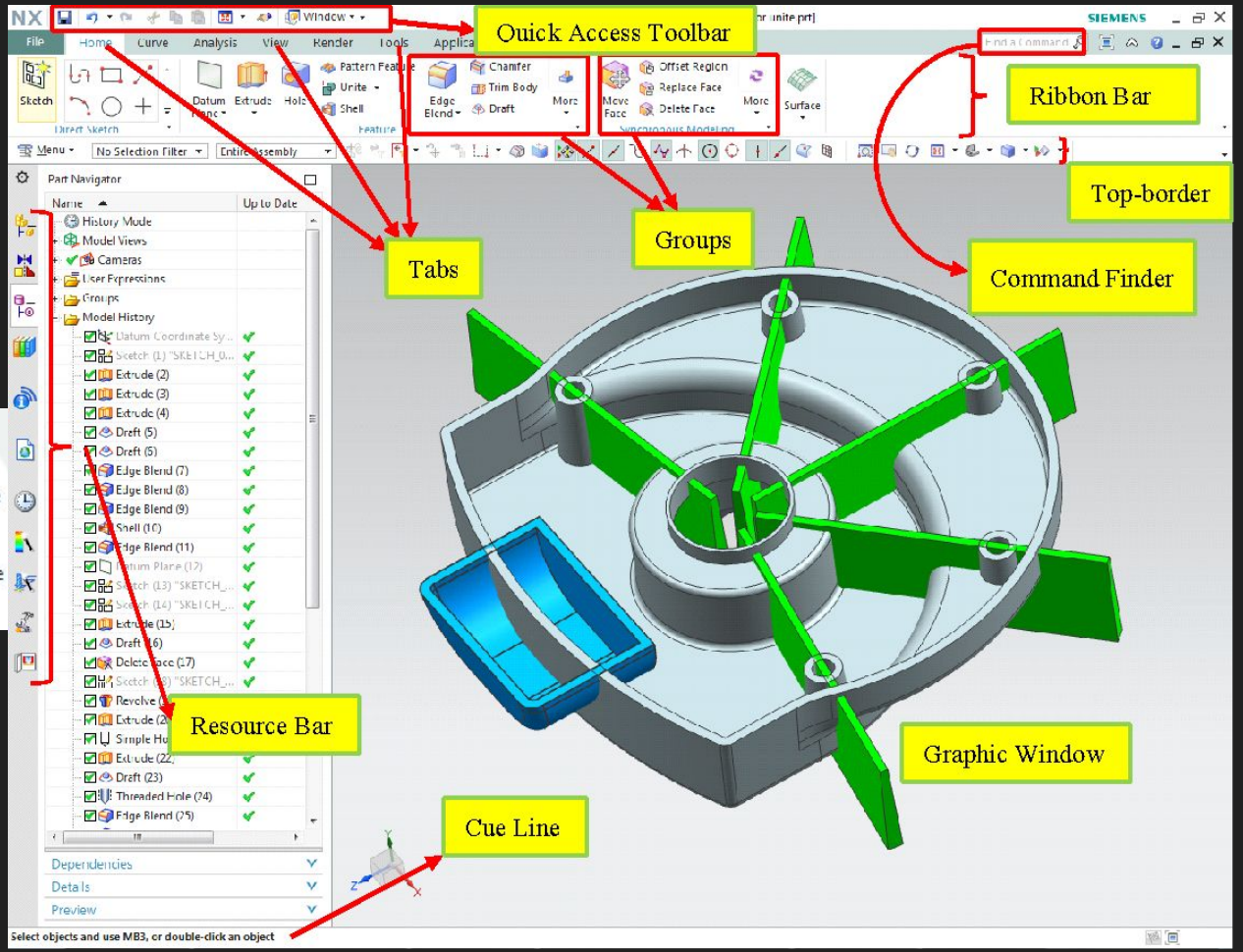


Gateway

To rotate, use the middle mouse button.

To pan, hold Shift and use the middle mouse button, or use the middle and right mouse buttons.

To zoom, use the scroll wheel, or use the left and middle mouse buttons.

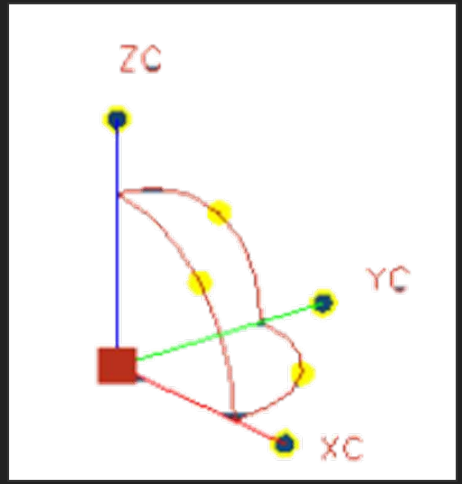


Coordinate Systems

There are two types of coordinate system:

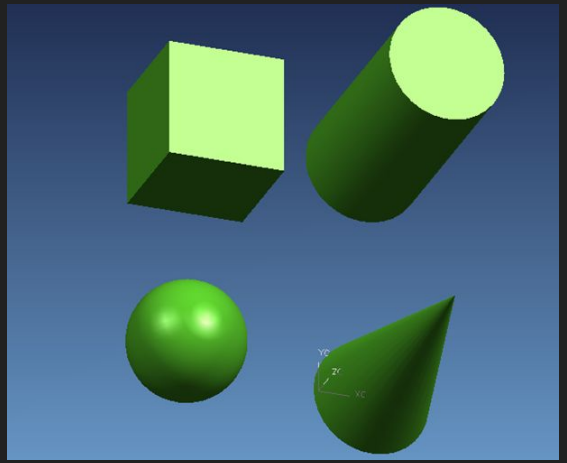
- Absolute Coordinate System

- Work Coordinate System
  Move the WCS
  Translate the WCS
  Rotate the WCS



Form Features

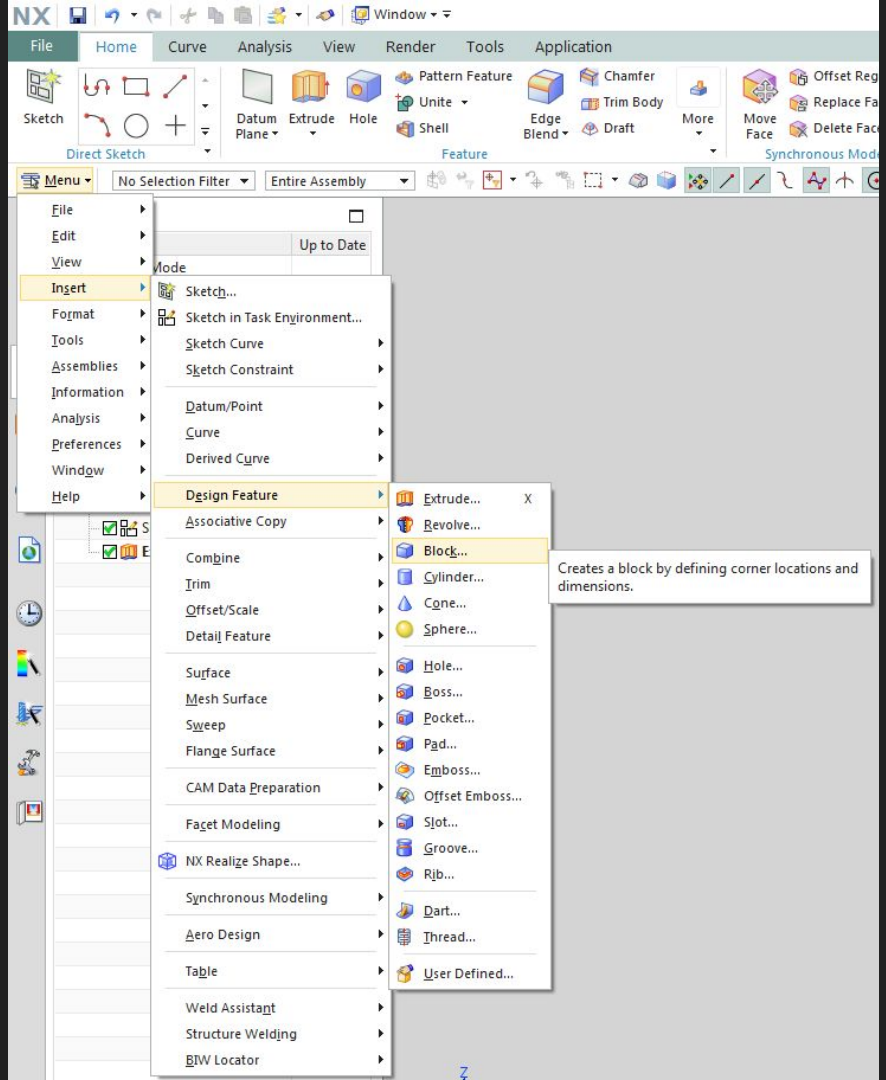
Features are objects that are associatively defined by one or more parents and that retain within the model the order of its creation and modification, thus capturing its history.
For example, some common features include blocks, cylinders, cones, spheres, extruded bodies, and revolved bodies.



Form Features

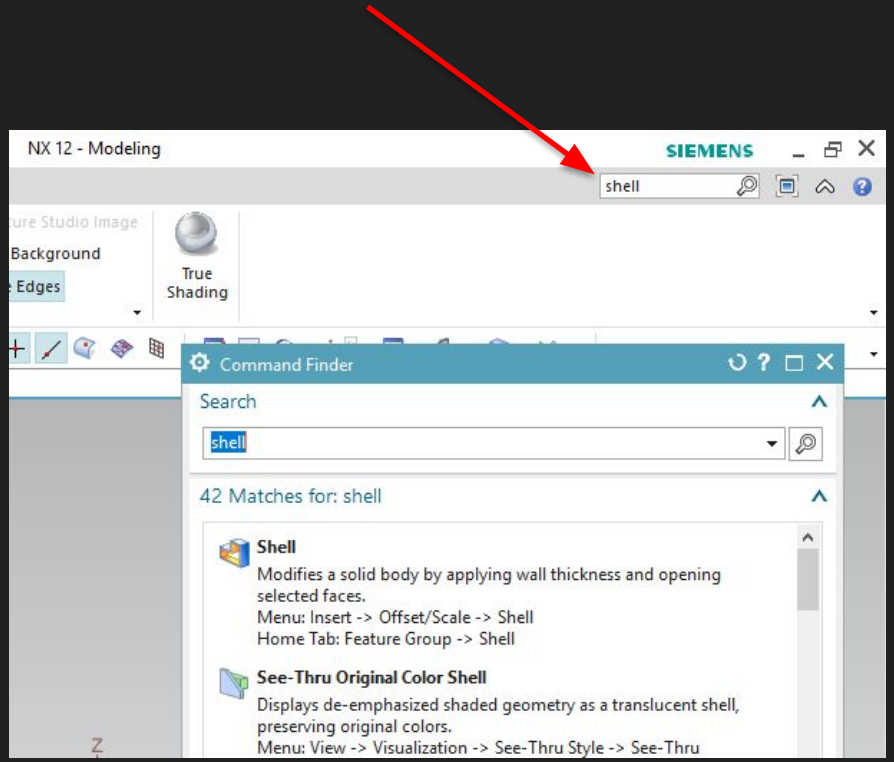
There’re lots of functions. All of them can be accessed through the ribbon bar or “Menu”

- Extrude
- Revolve
- Block
- Cylinder
- Sphere
- Hole
- Boss
- Pocket
- …



Command Finder

If you couldn’t find a specific function, use the search box to locate it.



Types of Form Features

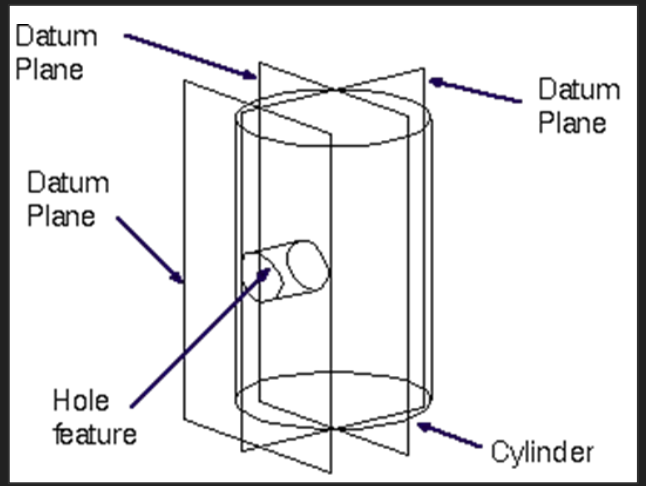
These form features can be categorized into different groups:

- Reference Features: datum planes, datum axis, ...
- Swept Features: extrusion, revolve, sweep along guide, ...
- Remove Features: hole, boss, pocket, ...
- Extract Features: extract, sheet from curves, ...
- Primitives: block, cylinder, cone, ...

Datum Plane/Reference Plane

Datum Planes are reference features that can be used as construction tools in the building of a model. Datum planes are used for:

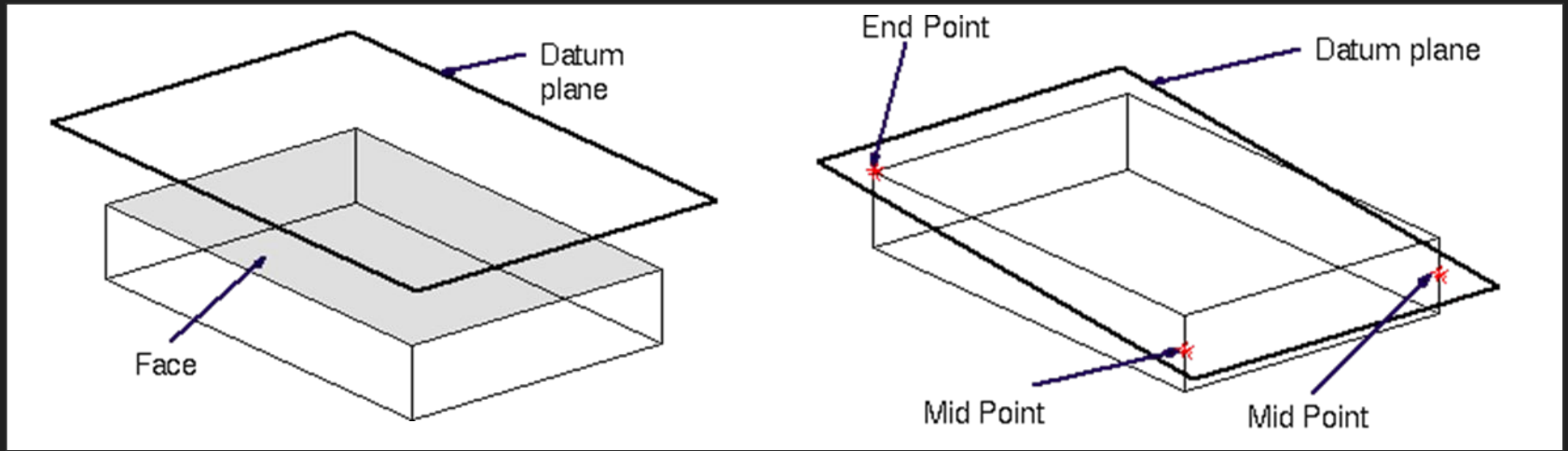
- to create features on non-planar faces
- to provide a planar location for a sketch
- to provide a planar trimming object



Datum Planes

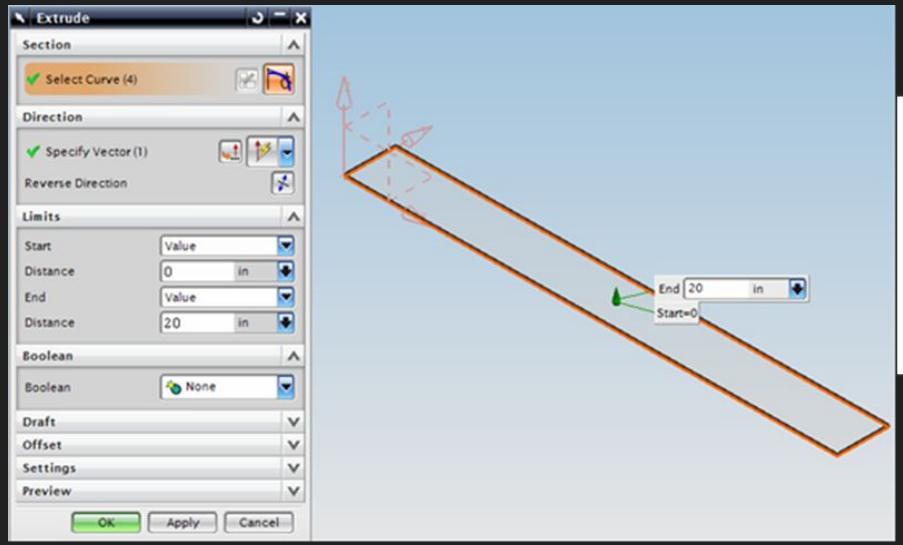
There’re different ways to creating a datum plane:

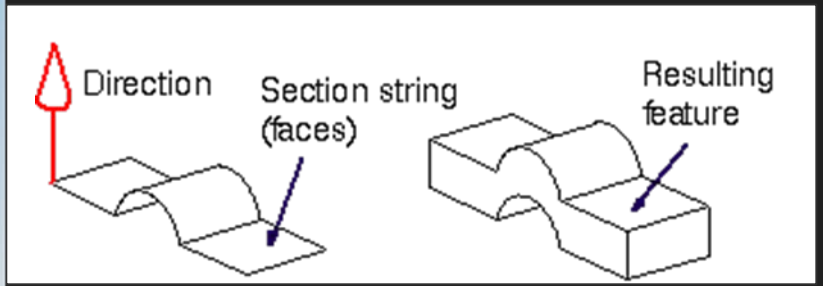
- Creating a Datum Plane Offset from a Face
- Creating a Datum Plane Through Three Points
- Creating a Datum Through an Edge and at an Angle to a Face



Extrude

The Extruded Body option lets you create a solid or sheet body by sweeping generator geometry (curves, solid faces, solid edges, sheet body) in a linear direction for a specified distance.



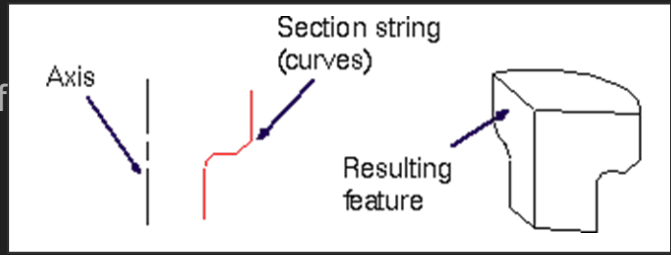


Revolve

A revolved body is created with a section string that is rotated around a specified axis.

Rules for Revolving Section String Objects:

- A solid or sheet body is created based on the closure condition of the curves and current body type setting under Preferences →Modeling



Exceptions apply as follows:

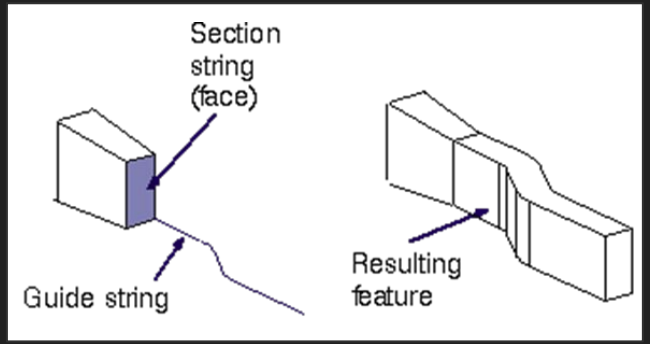
- Revolving open section strings will cause the system to automatically cap the end faces to produces a solid body; however, the rotation must be a full 360o and the Modeling Preferences Body Type switch must be set to Solid.
- The Right Hand rule determines direction of revolve.

Sweep

The Sweep along Guide body is one that has been swept along a guide string.

Rules for Revolving Section String Objects:

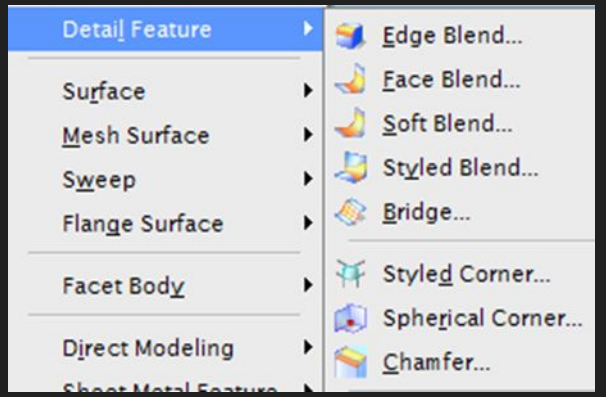
- An open section string swept along a guide path that forms an enclosed loop will automatically cap the end faces, provided the Modeling Preferences Body Type is set to Solid.
- Open section strings will always be swept into a solid body when using the sweep with offset option.

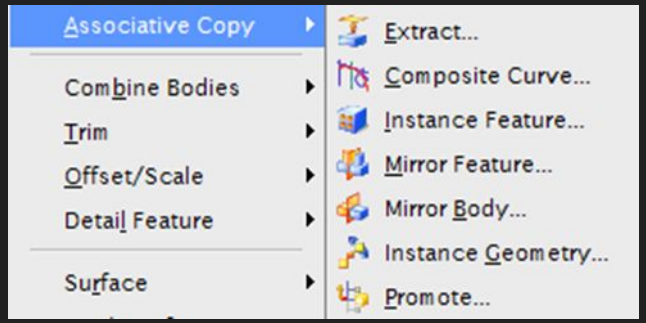


Feature Operations

Some commonly used feature operations are:

- Edge Blend
- Chamfer
- Mirror feature
- Pattern feature
- …

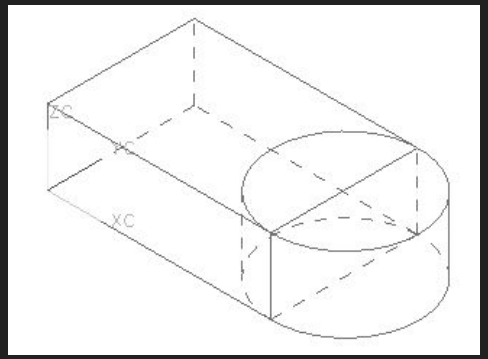




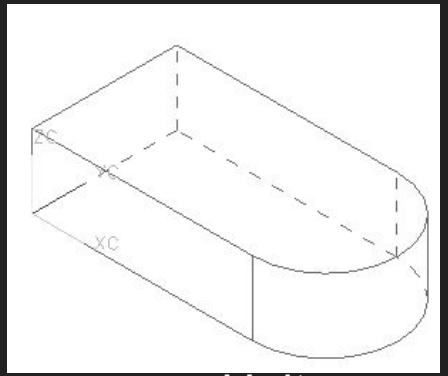
Boolean Operations

There are three Boolean operations:

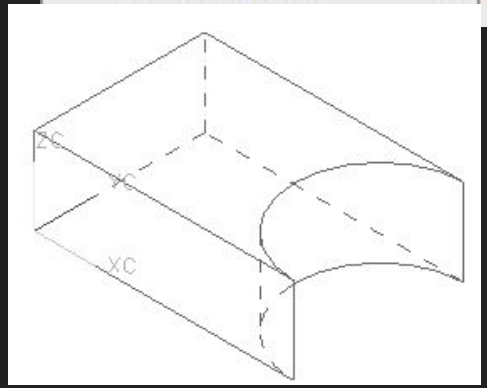
- Unite
- Subtract
- Intersect



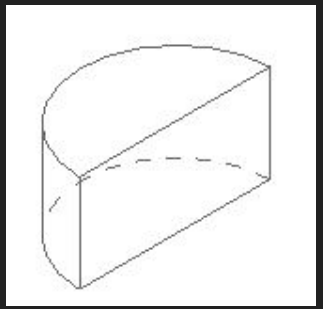
Combined body



Unite



Subtract



Intersect

Talk is cheap.

Show me the ~~code~~. CAD

Linus Torvalds

Demo

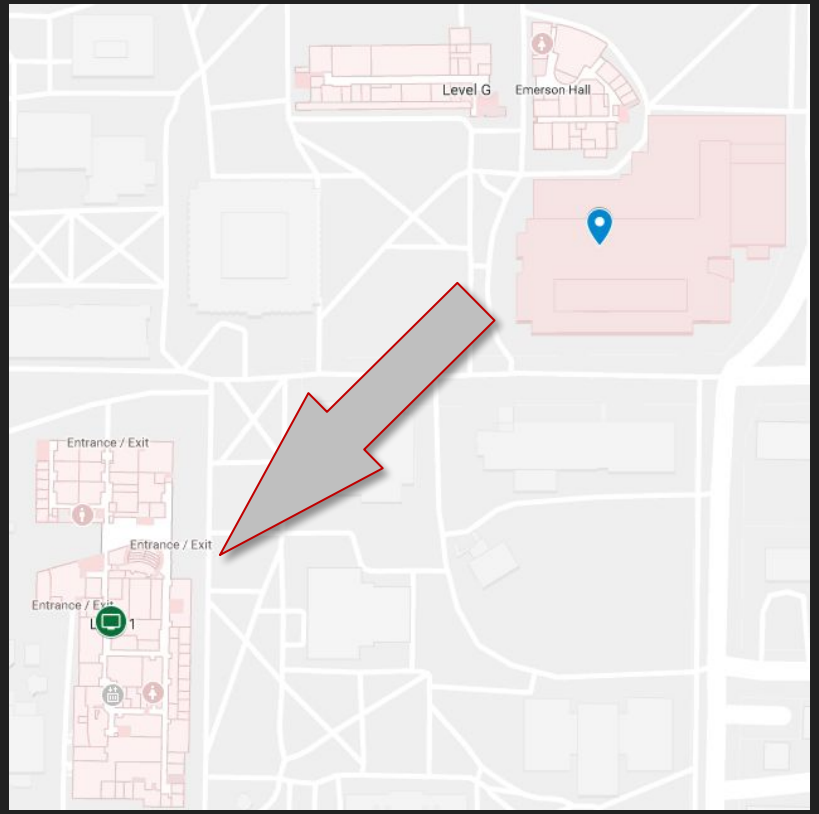
- Extrusion
- Revolve
- Datum plane
- Sweep
- Split
- Trimming
- Boolean operation

Self Practice

Get Started.

Project 1: https://me5763.github.io/lab/pages/project-1.html

For on-campus students: Toomey Hall 200. See you there!



Toomey Hall 200

CLC
COMPUTER LEARNING CENTER

For Distance Students

There are CLC machines available for you to access software remotely.
The setup instructions are provided on Canvas.