

GLOSSARY

Introductory and Intermediatory
Skills to 3D-Modeling

With Specialization in Blender and Fusion 360 for
industry based design and manufacturing.

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Introductory and Intermediary Skills to 3D-Modeling

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industry based design and manufacturing.

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(Glossary Terms Introduction)

Whilst learning fusion 360 the following terms should aid your learning as it will help you:

- (1) Communicate with your peers and teacher in a more concise and accurate manner thus allowing you to be more efficient.
- (2) Search for solution to problems you may encounter on the internet more accurately to aid your learning (ex. Instead of searching for “how to make a box from a drawing” and getting limited results, you may search for “how to *extrude* out a *mesh* from a *sketch*”. Notice how the key terms are in italics.)

Some important points to note are:

- (1) Terms in the glossary are sorted alphabetically as shown on the top and bottom of each page.
- (2) While this is by no means a comprehensive list of terms, however it should provide a good overview of terms to know.

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(Terms starting with “2”)

- (1) 2-Point Circle
Creates a circle defined by two points. Specify two points on the diameter of the circle.
- (2) 2-Point Rectangle
(Keyboard shortcut “R”) Creates a rectangle using two points for the diagonal corners. Select the first point as the start of the rectangle. Select the second point or specify the width and height values.
- (3) 2-Tangent Circle
Creates a circle tangent to two sketch lines. Select two lines then specify the radius of the circle.

(Terms starting with “3”)

- (1) 3-Point Arc
Creates an arc using three points. Select the start point, end point, and then a point on the arc.
- (2) 3-Point Circle
Creates a circle defined by three points. Specify three points on the circumference of the circle. The points define the size and the position of the circle.
- (3) 3-Point Rectangle
Creates a rectangle using three points to define width, direction, and height. Select the first point as the start of the rectangle. Select the second point or specify a distance value and pick a point. Select the third point or specify a distance value.
- (4) 3-Tangent Circle
Creates a circle tangent to three sketch lines. Select three lines that the circle will be tangent to.
- (5) 3D Print
Converts the selected body to a mesh body and outputs to STL or a 3D print utility (software). Select the body to output. Set the mesh controls and specify the print utility to output to.

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(Terms starting with “A”)

(1) Align

Moves an object (component, body, sketch, work geometry) by aligning geometry selected from the object with geometry selected elsewhere. Geometry can be a point, line, plane, circle, or coordinate system. Snaps are treated as full coordinate systems for full position and orientation alignment. Select the point on the object to move then select the point on the face to align to. Click Flip and Angle to rotate the object.

(2) Animation Workspace

Creates animations of how the design should be operated or assembled.

(3) Appearance

(keyboard shortcut “A”) The appearance affects the color of the bodies, components, and faces. Appearances override the color assigned from the physical material. Appearances do not affect engineering properties. Drag the appearance from the dialog to the body, component, or face.

(4) Application bar

Access the Data Panel (that allows quick access to your design files), file operations, save, undo and redo.

(5) As-Built Joint

Positions components relative to one another and defines the relative motion. Components maintain their current position. Select components to join. Specify the joint type and the position of the joint origin.

(6) Attached Canvas

Places an image on a planar face or sketch plane. Select a face then select an image to import.

(7) Axis Perpendicular at Point

Creates a construction line normal to the selected face at the area selected. Select a face at the desired area.

(8) Axis Perpendicular to Face at Point

Creates a construction line normal to the selected face and passing through the selected point. Select a face or sketch profile then select a point.

(9) Axis Through Cylinder/Cone/Torus

Creates a construction axis coincident with the center axis of a cylindrical or conical face. Select a cylinder or cone.

(10) Axis Through Edge

Creates a construction line from the selected linear edge or sketch curve.

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Select an edge or sketch curve.

(11) Axis Through Two Planes

Creates a construction line coincident with the intersection of two planes or planar faces. Select two planes or planar faces.

(12) Axis Through Two Points

Creates a construction line through two selected work points, sketch points or vertices. Select two work points, sketch points or vertices.

(Terms starting with “B”)

(1) Boundary Fill

Creates, joins, or removes volumes using bounding volumes formed by tool selections. Select solids, surfaces or workplanes as tools to form volumes (or cells). These cells can be used to cut existing solids or to join to existing solids or to create a new solid.

(2) Box

Creates a solid box. Select a plane, draw a rectangle then specify the height of the box.

(3) Break

Break curve entities into two or more sections. Pause the cursor over the segment to break from the overall curve. Select the curve to break.

(4) Browser

The browser lists objects in your design (everything from planes and sketches to parts and assemblies). Use the browser to make changes to objects and control visibility of objects.

(Terms starting with “C”)

(1) CAM Workspace

Generates tool-path strategies for the design to be fabricated.

(2) Canvas and Marking Menu

Left click to select objects in the canvas. Right-click to access the marking menu. The marking menu contains frequently used commands in the wheel and all

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commands in the overflow menu.

(3) Capture Image

Captures the current canvas as an image. Set the image size and resolution then set the file type (*.png, *jpg, *.tiff) and file name.

(4) Center Diameter Circle - (keyboard shortcut "C") Creates a circle using a center point and diameter. Select to define the center point then specify the diameter.

(5) Center of Mass

Displays a glyph at the location of the Center of Mass for the selected objects. You can measure from the COM to other objects in your design. Select components and solid bodies to include in the Center of Mass calculation.

(6) Center Point Arc

Creates an arc using three points. Select the center point, then select the start point and then select the endpoint or specify an angle value.

(7) Center Point Slot

Creates a linear slot defined by a center point, the location of slot arc centers, and by slot width. Specify the slot center, and place the center of a slot arc. Click to specify slot width, or enter the diameter of the slot arcs.

(8) Center Rectangle

Creates a rectangle using two points to define the center and one corner. Select the first point as the center of the rectangle. Select a corner point or specify width and height values.

(9) Center to Center Slot

Creates a linear slot defined by placement and distance of slot arc centers, and by slot width. Specify centers of both slot arcs. Click to specify slot width, or enter the diameter of the slot arcs.

(10) Center Two Point Arc Slot

Creates an arc slot defined by a center point, two-point center arc and slot width. Specify the center point of the slot. Specify the start and end point of the slot center arc. Click to specify slot width or enter diameter or radius of the slot arc.

(11) Chamfer

Applies a bevel to one or more edges. Select the edges then specify a distance.

(12) Change Parameters

Displays the Parameters dialog box. Edit the name, expression, and comments for parameters. Create User Parameters to use in other expressions.

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(13) Circular Pattern

Creates duplicate faces, features, bodies, or components and arranges them in an arc or circular pattern. Select the objects to pattern, the axis to revolve around, and the quantity.

(14) Circular Pattern (Sketch)

Duplicates selected sketch curves in an arc or circular pattern. Select the curves to pattern, the point to revolve around, and the quantity.

(15) Circumscribed Polygon

Creates a polygon using the center point and midpoint of one edge. Select the center point of the polygon. Specify a value for the number of sides. Select the midpoint of an edge or specify a distance and select a point.

(16) Coil

Creates a solid coil. Select a plane then draw a circle to specify the major diameter of the coil. Finish defining the coil using the manipulators or dialog box.

(17) Combine

Performs boolean operations between solid bodies. Select the target body then select one or more tool bodies to perform a join, cut, or intersect operation on the target.

(18) Component Color Cycling Toggle

(keyboard shortcut "Shift + N") Applies different colors to each component to help differentiate between components. Click the command to toggle this setting on or off.

(19) Compute All

(keyboard shortcut "CMD + B") Computes all parameters in the design.

(20) Conic Curve

Creates a curve driven by endpoints and Rho value. Depending on the Rho value, the curve can be elliptical, parabolic, or hyperbolic. Select the start point, end point, and then the top of the vertex. Use the guidelines to create a tangency constraint. Specify the Rho value for the desired shape of the conic curve.

(21) Constrained Orbit

Revolves the object 360 degrees around selected point.

(22) Create Base Feature

Enters direct editing mode and inserts a history-free feature in the timeline. Actions performed while in the Base Feature are not recorded in the timeline. Select Finish Base Feature to exit out of the mode.

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(23) Create Form

Enters Form mode and inserts a form feature into the timeline. Use form tools to create and edit history-free bodies by pushing and pulling on vertices, edges, and faces. This is commonly referred to as T-Spline modeling. Select Finish Form to exit out of the mode.

(24) Create Mesh

Enters Mesh mode and inserts a mesh feature into the timeline. Use Mesh tools to modify and repair mesh geometry. Select Finish Mesh to exit out of the mode.

(25) Create Mesh Section

Creates a new sketch containing a mesh cross section as reference geometry. Select a mesh body then move the section plane. To fit sketch curves to the mesh section, follow with the Fit Mesh Section Curves command.

(26) Create Sketch

Enters Sketch mode and creates a new sketch on a selected plane or face. Select Finish Sketch to exit out of the mode.

(27) Curvature Comb Analysis

Shows the amount of curvature at sample points along the selected edges. Select the edges to evaluate then set the comb density and scale.

(28) Curvature Map Analysis

Displays a color gradient on the faces of selected bodies to help evaluate an area of high and low surface curvature. Select the bodies to evaluate. Use the options to refine the color gradient display.

(29) Cylinder

Creates a solid cylinder. Select a plane, draw a circle then specify the height of the cylinder.

(Terms starting with “D”)

(1) Data Panel

Opening the Data Panel gives you access to your Project Files and Folders. Click the Show Data Panel button in the upper left of the Fusion window. The Panel will expand on the left side and display the project folders. You can also add members to projects and Import data from other sources.

(2) Decal

Places an image on a selected face. Select a face then select an image to import.

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- (3) Delete
Deletes the selected entity from the design. Select components, bodies, sketches, and construction entities to remove.
- (4) Display Settings
Change display settings including the visual style, mesh display, environment, effects, object visibility, camera perspective, and ground planes.
- (5) Draft
Applies a draft angle to specified planar faces. Select a plane or planar face for the neutral plane then select the faces to be drafted. Specify the draft angle for the faces.
- (6) Draft Analysis
Displays a color gradient on the faces of selected bodies to help evaluate the manufacturability of your design. Select the bodies to evaluate and the axis to define the direction. Use the options to refine the gradient display.
- (7) Drive Joints - Specify the rotation angle or distance value for joint degrees of freedom.

(Terms starting with “E”)

- (1) Edge Polygon
Creates a polygon by defining a single edge and the position of the polygon. Select the start and end points of one edge or select the start point then specify a distance and angle. Specify the number of sides. Select a point to define the orientation of the polygon.
- (2) Ellipse
Creates an ellipse defined by a center point, major axis and a point on the ellipse. Select the center point of the ellipse. Select the second point to define the first axis. Select the third point to define a point on the ellipse.
- (3) Enable All Contact
Activates contact analysis for all components.
- (4) Enable Contact Sets
Activates contact analysis between components in contact sets. Contact sets are managed in the browser.
- (5) Export
Export files to your computer. Opens the Export dialog box and allows you to

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choose a name, file type, and location to save the file to.

(6) Extend

Extends a curve to the nearest intersecting curve or boundary geometry. Pause the cursor over the curve to preview the portion to be extended. Select the curve to extend it.

(7) Extrude

Adds depth to a closed sketch profile or planar face. Select the profile or planar face then specify the distance to extrude.

(Terms starting with “F”)

(1) Fillet

Places an arc of a specified radius at the intersection of two lines or arcs. Select the vertex or the two lines or arcs. Specify a radius for the fillet.

(2) Fillet

Adds fillets or rounds to one or more edges. Select the target edges then specify the radius value.

(3) Fit

(keyboard shortcut “f6”) Makes the current file fill the screen.

(4) Fit Mesh Section Curves

Fits sketch curves to a mesh cross-section. Choose the sketch curve type, then select the start and end points on the mesh section. Create multiple curves within the same command with various join conditions.

(5) Free Orbit

Revolves the object 360 degrees around the center point of the object.

(6) Freeform Selection

(keyboard shortcut “2”) Define a freeform border by mouse down drag. Selection from left to right will select any target within the boundary. Selection from right to left will select any target crossed by the boundary.

(7) Fusion 360 App Store

Your portal to both community and professional built Add-Ins for Fusion 360. Install apps to further enhance the functionality of Autodesk Fusion 360.

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(Terms starting with “G”)

- (1) Get a Quote from Proto Labs
Get an interactive quote with free design analysis within hours, from Proto Labs.
- (2) Get Parts Made with MakeTime
When your parts are ready to be machined, MakeTime is your direct access to thousands of qualified suppliers across the US. MakeTime manages your order from production to delivery so that you can focus on designing more great parts.
- (3) Get Quotes from 100kGarages.com
The 100kGarages service enables you to request a quote for manufacturing your part, compare quotes against other fabricators, and get your product made with a seamless process.
- (4) Grid and Snaps
Toggle on/off the layout grid, layout grid lock, snap to grid, and incremental move. Edit grid settings and set increments.

(Terms starting with “H”)

- (1) Hole
Creates a hole based on user-specified values and selections. Select a face to place the hole then select edges to position the hole on the face, or select sketch points to place multiple holes. Specify the hole type, tap type, and size values.

(Terms starting with “I”)

- (1) Include 3D Geometry
Include edges, work geometry, and sketch curves into the current sketch. Select the objects to include.
- (2) Inscribed Polygon - Creates a polygon using the center point and a vertex.
Select the center point of the polygon. Specify a value for the number of edges. Select a vertex point or specify a distance and select a point.
- (3) Insert a Manufacturer Part
Parts4Cad lets you browse and configure millions of parts from more than 400

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supplier and manufacturer catalogs and insert them into your designs.

(4) Insert DXF

Imports a DXF file into the active sketch. Select a face or plane then select the DXF file to insert. Set the position of the geometry on the selected plane.

(5) Insert McMaster-Carr Component - Insert a McMaster-Carr Component into the active document. Browse to the desired component, click the Product Detail CAD icon, select the file type, then click save.

(6) Insert Mesh

Inserts the selected OBJ or STL mesh file into the active design. Choose an OBJ or STL file to insert, then use the options and manipulators to orient and position the mesh.

(7) Insert SVG

Imports an SVG file into the active sketch. Use SVG files for detailed sketches such as logos or text.

(8) Interference

Reports the interference between selected solid bodies or components. Select the components then click compute. Choose to create components from the interferences.

(9) Intersect

Projects the points, model edges, work geometries, and sketch curves that intersect the active sketch plane. Use the selection filter to project a specific type of geometry or the entire model.

(10) Intersection Curve

Creates a 3D sketch curve from intersecting geometry. The first selection is the sketch curve. The second selection is the sketch curve or faces to intersect with. You can select multiple curves or faces to intersect with.

(11) Invert Selection

Invert the active selection by obeying the options of Selection Tools and Selection Filters.

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(Terms starting with “J”)

(1) Joint

(keyboard shortcut “J”) Positions components relative to one another and defines the relative motion. Select geometry or joint origins to define the joint. Specify the type to define the relative motion.

(2) Joint Origin

Positions a joint on a component. Joint origins define the geometry used to relate to a joint’s components. Select geometry to define the joint origin.

(Terms starting with “L”)

(1) Line

(keyboard shortcut “L”) Creates lines and arcs. Select a start and endpoint to define a line segment. Click and drag the endpoint of a segment to define an arc.

(2) Live Review Session

Start a real-time online viewing session with collaborators. Send collaborators a link and you can view the design together.

(3) Loft

Creates a transitional shape between two or more sketch profiles or planar faces. Select a series of profiles or planar faces to define a shape. Optionally select rails or a centerline to guide the shape.

(4) Look At

Re-arranges screen to look at selected object.

(Terms starting with “M”)

(1) Manage Materials

Opens the material browser. Manage materials the Fusion 360 material library. Upload or alter existing materials.

(2) Measure

(keyboard shortcut “I”) Reports distance, angle, area, or position data of the

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selected object. Select a vertex, edge, face, body, or component.

(3) Midplane

Creates a construction plane at the midpoint between two faces or work planes. Select two faces or planes.

(4) Mirror

Mirrors the selected sketch curves about a selected sketch line. Select the curves to mirror then select the line to mirror about.

(5) Mirror

Makes a mirrored copy of selected faces, features, bodies, or components at equal distances across a plane. Select the objects to mirror then the plane to mirror around.

(6) Model Workspace

Creates mechanical designs that contain mostly prismatic geometry. Access commands to create solid bodies.

(7) Motion Link

Defines the rotational and translational relationship between joint degrees of freedom. Select the joints then specify the values.

(8) Motion Study

Performs kinematic motion analysis based on joints. Select the joints to participate then specify points and values for the motion.

(9) Move/Copy

Moves the selected face, body, sketch, or construction geometry a specified distance angle. Select the objects to modify then specify the distance or angle. Use Set Pivot to reposition the manipulator.

(Terms starting with “N”)

(1) Navigation Bar and Display Settings

The navigation bar contains commands used to zoom, pan, and orbit your design. The display settings control the appearance of the interface and how designs are displayed on the canvas.

(2) New Component

Creates a new empty component or converts existing bodies to components. When creating an empty component, enter a name and select the parent. When converting bodies, select the bodies to convert.

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(3) New Component

Creates a new empty component or converts existing bodies to components. When creating an empty component, enter a name and select the parent. When converting bodies, select the bodies to convert.

(4) New Design

Creates a new tab with a blank canvas.

(5) New Design from File

Opens computer dialog box and allows you to select a pre-existing CAD file to start with.

(6) New Drawing from Animation

Creates a new drawing for the animation file that is currently open.

(7) New Drawing from Design

Creates a new drawing for the file that is currently open.

(8) New Drawing Template

Creates a new drawing that can be saved as a template.

(Terms starting with “O”)

(1) Offset

Copies the selected sketch curves a specified distance from the original curves. Select the curves to offset then specify the offset distance.

(2) Offset Plane

Creates a construction plane that is offset from the selected face or plane. Select a face, plane, or sketch profile, then specify the offset distance.

(3) Open Details on Web

Opens current file in the Fusion 360 hub.

(4) Overall Slot

Creates a linear slot defined by orientation, length, and width. Specify the start and end on the slot center line. Click to specify slot width, or enter the diameter of the slot arcs.

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(Terms starting with “P”)

- (1) Paint Selection
(keyboard shortcut “3”) Selection by mouse down drag over targets.
- (2) Pan
Shifts the view without changing the viewing direction or magnification.
- (3) Path Workspace
Creates construction surfaces and repairs surface geometry. Access commands to create surface bodies.
- (4) Pattern on Path
Creates duplicate faces, features, bodies, or components and arranges them along a path. Select the objects to pattern and the path to pattern along. Specify the distance and quantity.
- (5) Physical Material
Physical materials affect the color and engineering properties of bodies and components. Drag the physical material from the dialog to the body or component.
- (6) Pipe
Creates a solid pipe that follows a selected path. Select the path then specify the section and size.
- (7) Plane Along Path
Create a construction plane normal to an edge or sketch profile. Select the path then specify the position of the plane along the path.
- (8) Plane at Angle
Creates a construction plane through an edge, axis, or line at a specified angle. Select a linear edge, axis, or sketch line to position the plane, then specify the rotation angle.
- (9) Plane Tangent to Face at Point
Creates a construction plane tangent to a face and aligned to a point. Select a face then point or vertex.
- (10) Plane Through Three Points
Creates a construction plane through three selected construction points, sketch points or vertices. Select three points.
- (11) Plane Through Two Edges
Creates a construction plane passing through two linear edges or axes. Select

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two edges or axes. The edges or axes must be coplanar.

(12) Point

Creates a sketch point.

(13) Point at Center of Circle/Sphere/Torus

Creates a construction point at the center of a sphere or circular edge. Select a circular edge or spherical face.

(14) Point at Edge and Plane

Creates a construction point at the intersection of a construction plane, planar face, or sketch profile and an axis or sketch line. Select a plane, planar face, or sketch profile and an axis or sketch line.

(15) Point at Vertex

Creates a construction point at a selected point or vertex. Select a point or vertex.

(16) Point through Three Planes

Creates a construction point at the intersection of three planes or planar faces. Select three planes or planar faces.

(17) Point through Two Edges

Creates a construction point at the intersection of two linear edges or sketch lines. Select two linear edges or sketch lines.

(18) Press Pull

(keyboard shortcut "Q") Modifies the selected geometry using offset, extrude, or fillet commands. The operation depends on the geometry selected.

(19) Profile and Help

In the profile, you can control your profile and account settings, or use the help menu to continue your learning or get help in troubleshooting.

(20) Project

Projects the body silhouette, edges, work geometries and sketch curves into the active sketch plane. Use the selection filter to project a specific type of geometry or the body silhouette.

(21) Project To Surface

Project sketch geometry, edge, vertex, or workpoint to selected faces.

(22) Publish to GrabCad

Publishes model to grabcad.com, an online platform hosting millions of free CAD files.

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(Terms starting with “R”)

- (1) Recover Documents
Reopen files that were autosaved. Select the file name and click open.
- (2) Rectangle Pattern (Sketch)
Duplicates sketch curves in rows and columns. Select the curves to pattern then specify the direction, quantity, and distance.
- (3) Rectangular Pattern
Creates duplicate faces, features, bodies, or components and arranges them in rows and columns. Select the objects to pattern then specify the direction, quantity, and distance.
- (4) Redo
Re-applies the most recent action that was undone.
- (5) Render Workspace
Generates realistic renderings of the design. Apply materials, scenery, and lighting.
- (6) Replace Face
Replaces one or more part faces with a different face. The new face must completely intersect the part. Select the face to remove then select the new face.
- (7) Reset to Default Layout
Resets browser tree and dialog boxes to their default locations in Fusion 360.
- (8) Revolve
Revolves a sketch profile or planar face around a selected axis. Select a profile or planar face then select the axis to revolve around.
- (9) Rib
Uses an open sketch curve to create a thin feature. The rib is created planar (parallel) to the sketch plane. Select a curve then specify the thickness.
- (10) Rigid Group
Locks the relative position of the selected components. The components are treated as a single object when moved or when joints are applied. Select the components to group together.
- (11) Rule Fillet
Adds fillets or rounds based on specified rules instead of specified edges. Select a face or feature then specify a radius. The fillet will be applied to all edges of the face or feature.

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(Terms starting with “S”)

- (1) Save
(keyboard shortcut CMD/CTRL+S) Saves the file that is currently open.
- (2) Save As
Allows you to rename a file and save a duplicate to an alternative location.
- (3) Scale
Scales sketch objects, bodies, or components. Select the objects to scale then specify the scale factor.
- (4) Scripts and Add-Ins
Displays the Scripts and Add-Ins dialog box. Create, edit, run, stop, debug and manage Scripts and Add-Ins.
- (5) Section Analysis
Generates a cutaway view of the model at a single section plane. Select a face or plane to use as the cutting plane. Specify offset and angle values to position the section plane.
- (6) Select
The selection mode controls how objects are select when you drag in the canvas.
- (7) Select Body Priority
Sets selection priority to bodies. Click to select only bodies in the model. Click again to remove priority and allow selection of all objects.
- (8) Select By Boundary
Selects objects defined by boundaries. Set the centroid, shape, and shape direction of the boundary. Specify if objects are selected within the boundary and if they can intersect the boundary.
- (9) Select By Size
Selects objects based on size. Use the slider bars or directly input minimum and maximum size values to select objects.
- (10) Select Component Priority
Sets selection priority to components. Click to select only components in the model. Click again to remove priority and allow selection of all objects.
- (11) Select Edge Priority
Sets selection priority to edges. Click to select only edges in the model. Click again to remove priority and allow selection of all objects.

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(12) Select Face Priority

Sets selection priority to faces. Click to select only faces in the model. Click again to remove priority and allow selection of all objects.

(13) Selection Tools

Select object by name. Enter the object name and set the type. Find the object and select if appropriate.

(14) Share Public Link

Creates a unique Autodesk link that can be shared with others. Links can be set up to allow users to download the file. Links can also be password protected.

(15) Share to Fusion 360 Gallery

Opens up the Share to Fusion 360 Gallery dialog box. Share files with the Fusion 360 community by filling out relevant info and automatically publishing a screenshot and title of your file.

(16) Sheet Metal Workspace

Creates sheet metal designs. Use these commands to set rules, create sheet metal bodies, and produce manufacturing data.

(17) Shell

Removes material from a part interior, creating a hollow cavity with walls of a specified thickness. Select faces then specify a thickness. The selected faces will be removed and the body will become hollow.

(18) Show/Hide Browser

Toggles the browser from visible to hidden.

(19) Show/Hide Comments

Toggles comments from visible to hidden.

(20) Show/Hide Data Panel

Toggles the data panel from visible to hidden.

(21) Show/Hide Navigation Bar

Toggles the navigation bar from visible to hidden.

(22) Show/Hide Text Commands

Toggles the text command box from visible to hidden. Type text commands to control Fusion 360 with text instead of the user interface.

(23) Show/Hide Toolbar

Toggles the toolbar from from visible to hidden.

2 3 A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

2 3 A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

(24) Show/Hide ViewCube

Toggles the viewcube from visible to hidden.

(25) Silhouette Split

Splits a body using surfaces generated from its silhouette curves. Set the view direction by selecting a plane or axis then select the body and choose a split operation.

(26) Simulation Workspace

Performs simulation to determine how loads lead to deformation and failure, helping you understand if and how the part will fail.

(27) Sketch Dimension

Create sketch dimensions for sketch geometries. Use dimensions to control the size or position of sketch objects. Select the sketch curves to dimension then select an area to position the dimension.

(28) Sketch Scale

Scales sketch geometry. Select sketch geometry to scale, then specify the scale factor.

(29) Sphere

Creates a solid body sphere. Select a plane then specify the center point of the sphere and the diameter.

(30) Spline

Creates a spline curve through the selected points. Select the first point to start the spline. Select additional points as fit points.

(31) Split Body

Creates new bodies by dividing selected bodies using a profile, face, or plane. Select the bodies to modify then select the profile, face, or plane that divides the bodies.

(32) Split Face

Divides faces using a selected profile, face, or plane as the cutting tool. Select the faces to modify then select the profile, face, or plane that divides the source face.

(33) Stop Sketch

Exits the sketch that is currently open.

(34) Sweep

Sweeps a sketch profile or planar face along a selected path. Select a series of profiles or planar faces to define a shape. Optionally select rails or a centerline guide the shape.

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(Terms starting with “T”)

- (1) Tangent Arc
Creates an arc with tangency. Select two points.
- (2) Tangent Plane
Creates a construction plane that is tangent to a cylindrical or conical face. Select the cylinder or cone that the plane will be tangent to. Specify the position by entering an angle or selecting a reference plane and then adding an angle.
- (3) Text
Inserts text into the active sketch. Use text as a profile to create 3D geometry. Select the insertion point and input the text. Modify the format of the text in the Text dialog box.
- (4) Thicken
Adds thickness to surface faces to make a solid. Select the faces to thicken then specify a thickness value.
- (5) Thread
Add internal or external threads to cylindrical geometry. Threads can be cosmetic or modeled on the geometry. Select a cylindrical face then set the thread specifications.
- (6) Three Point Arc Slot
Creates an arc slot defined by a three-point center arc and slot width. Specify the start and end point of the slot center arc. Specify the third point of the center arc, or enter center arc radius. Click to specify slot width, or enter diameter or radius of the slot arcs.
- (7) Timeline
The timeline lists operations performed on your design. Right-click operations in the timeline to make changes. Drag operations to change the order they are calculated.
- (8) Toolbar
Use the Toolbar to select the workspace you want to work in, and the tool you want to use in the workspace selected.
- (9) Torus
Creates a solid torus. Specify the center point and diameter of the axis of revolution of the torus.
- (10) Trim
Trims a sketch curve to the nearest intersecting curve or boundary geometry.

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2 3 A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

Pause the cursor over the curve to preview the portion to be trimmed. Select the curve to trim.

(Terms starting with “U”)

(1) Undo

Reverses the effects or results of the most recent action.

(Terms starting with “V”)

(1) ViewCube

Use the ViewCube to orbit your design or view the design from standard view positions.

(2) Viewports

Allows you to view file from multiple views at once.

(Terms starting with “W”)

(1) Web

Uses an open sketch curve to create a thin feature. The web is created normal (perpendicular) to the sketch plane. Select the curve then specify the thickness.

(2) Window Selection

(keyboard shortcut “1”) Define a window by mouse down drag. Selection from left to right will select any target within the boundary. Selection from right to left will select any target crossed by the boundary.

(Terms starting with “Z”)

(1) Zebra Analysis

Displays a stripe pattern on the selected bodies. Use Zebra Analysis to determine continuity. Select the surfaces then set the analysis options.

(2) Zoom

Zooms in and out as the mouse is moved from left to right.

2 3 A B C D E F G H I J K L M N O P Q R S T U V W X Y Z