AutoCAD 3D – Chapter 8
3D Model Objects
8.1 Wireframes

A wireframe model is a skeletal description of a 3D object. There are no surfaces in a wireframe model; it consists only of points, lines, and curves that describe the edges of the object. With AutoCAD you can create wireframe models by positioning 2D objects anywhere in 3D space. AutoCAD also provides some 3D wireframe objects, such as 3D polylines (that can only have a CONTINUOUS linetype) and splines. Because each object that makes up a wireframe model must be independently drawn and positioned, this type of modeling can be the most time-consuming.

Wireframe lines in 3D
8.2 Surfaces

Surface modeling is more sophisticated than wireframe modeling in that it defines not only the edges of a 3D object, but also its surfaces. The AutoCAD surface modeler defines faceted surfaces using a polygonal mesh. Because the faces of the mesh are planar, the mesh can only approximate curved surfaces.
Solid modeling is the easiest type of 3D modeling to use. With the AutoCAD solid modeler, you can make 3D objects by creating basic 3D shapes: boxes, cones, cylinders, spheres, wedges, and tori (donuts). You can then combine these shapes to create more complex solids by joining or subtracting them or finding their intersecting (overlapping) volume. You can also create solids by sweeping a 2D object along a path or revolving it about an axis.
AutoCAD 3D – Chapter 9
2D Solids and 3D Faces
9.1 2D Solid

Creates solid-filled triangles and quadrilaterals.

1. Type SOLID at the command prompt.
   Command: solid
   First point: P1
   Second point: P2
   Third point: P3
   Fourth point: P4
   Third point: enter
9.2 3D Faces

3DFACE creates a three or four sided surface anywhere in 3D space. You can specify different Z coordinates for each corner point of a 3D face. 3DFACE differs from SOLID, which creates a three- or four-sided surface that is parallel to the current UCS and can be extruded.

1. Type 3DFACE at the command prompt.

   Command: 3dface

   First point: pick

   Second point: pick

   Third point: pick

   Fourth point: pick

   Third point: enter

3D Wireframe Lines  3D Face
9.3  3D Face Invisible Edge

With 3DFACE, you control which edges of a 3D face are visible, allowing accurate modeling of objects with holes. Entering i or invisible before the first point of an edge makes the edge invisible. Type 3DFACE at the command prompt.

1. Type 3DFACE at the command prompt.

Command: 3Dface

First point: P1
Second point: P2
Third point: i P3
Fourth point: P4
Third point: i P5
Fourth point: P6
Third point: P7
Fourth point: P8
Third point: enter

NOTE: You must enter an “i” for invisible before the face is chosen.
9.4 Edge Command

1. Type EDGE at the command prompt.

   Command: edge

   Specify edge of 3dface to toggle visibility or [Display]: pick edge

   ** Regenerating 3DFACE objects...done.

   Specify edge of 3dface to toggle visibility or [Display]: d

   Enter selection method for display of hidden edges [Select/All]
   <All>: a

   ** Regenerating 3DFACE objects...done.

   Specify edge of 3dface to toggle visibility or [Display]: press enter
9.5 PFace

Creates a three-dimensional polyface mesh vertex by vertex

1. Type PFACE at the command prompt.
   Command: pface
   Specify location for vertex 1: pick point 1
   Specify location for vertex 2 or <define faces>: pick point 2
   Specify location for vertex 3 or <define faces>: pick point 3
   Specify location for vertex 4 or <define faces>: pick point 4
   Specify location for vertex 5 or <define faces>: pick point 5
   Specify location for vertex 6 or <define faces>: pick point 6
   Specify location for vertex 7 or <define faces>: pick point 7
   Specify location for vertex 8 or <define faces>: enter
   Face 1, vertex 1:
   Enter a vertex number or [Color/Layer]: type 1
   Face 1, vertex 2:
   Enter a vertex number or [Color/Layer] <next face>: type 2
   Face 1, vertex 3:
   Enter a vertex number or [Color/Layer] <next face>: type 6
   Face 1, vertex 4:
   Enter a vertex number or [Color/Layer] <next face>: type 7
   Face 1, vertex 5: enter
   Enter a vertex number or [Color/Layer] <next face>:
   Face 2, vertex 1:
   Enter a vertex number or [Color/Layer]: type 2
   Face 2, vertex 2:
   Enter a vertex number or [Color/Layer] <next face>: type 3
   Face 2, vertex 3:
   Enter a vertex number or [Color/Layer] <next face>: type 4
   Face 2, vertex 4:
Enter a vertex number or [Color/Layer] <next face>: type 6
Face 2, vertex 5:

Enter a vertex number or [Color/Layer] <next face>:

Face 3, vertex 1:

Enter a vertex number or [Color/Layer]: type 4
Face 3, vertex 2:

Enter a vertex number or [Color/Layer] <next face>: type 5
Face 3, vertex 3:

Enter a vertex number or [Color/Layer] <next face>: type 6
Face 3, vertex 4:

Enter a vertex number or [Color/Layer] <next face>:
Face 4, vertex 1: enter

Enter a vertex number or [Color/Layer]:
10.1 Box

1. Type AI_BOX at the command prompt.
   Command: ai_box
   Initializing... 3D Objects loaded. Corner of box: pick point
   Specify length of box: 4
   Specify width of box or [Cube]: 2
   Specify height of box: 2
   Specify rotation angle of box about the Z axis or [Reference]: 0
10.2 Pyramid

1. Type AI_PYRAMID at the command prompt.
   Command: \texttt{ai\_pyramid}
   Specify first corner point for base of pyramid: \texttt{pick point}
   Specify second corner point for base of pyramid: \texttt{4}
   Specify third corner point for base of pyramid: \texttt{4}
   Specify fourth corner point for base of pyramid or [Tetrahedron]: \texttt{4}
   Specify apex point of tetrahedron or [Top]: \texttt{.xy}
   of \texttt{pick}
   (need Z): \texttt{4}
10.3 Wedge

1. Type AI_WEDGE at the command prompt.

   Command: ai_wedge
   Specify corner point of wedge: pick point
   Specify length of wedge: 4
   Specify width of wedge: 2
   Specify height of wedge: 2
   Specify rotation angle of wedge about the Z axis: 0
1. Type AI_DOME at the command prompt.
   Command: ai_dome
   Specify center point of dome: pick point
   Specify radius of dome or [Diameter]: 3
   Enter number of longitudinal segments for surface of dome <16>: 20
   Enter number of latitudinal segments for surface of dome <8>: 10
10.5 Sphere

1. Type `AI_SPHERE` at the command prompt.

   Command: `ai_sphere`
   Specify center point of sphere: `pick point`
   Specify radius of sphere or [Diameter]: `3`
   Enter number of longitudinal segments for surface of sphere <16>: `25`
   Enter number of latitudinal segments for surface of sphere <16>: `25`
10.6 Cone

1. Type AI_CONE at the command prompt.

Command: **ai_cone**
Specify center point for base of cone: **pick point**
Specify radius for base of cone or [Diameter]: 2
Specify radius for top of cone or [Diameter] <0>: 5
Specify height of cone: 3
Enter number of segments for surface of cone <16>: **enter**
10.7 Torus

1. Type AI_TORUS at the command prompt.

   Command: ai_torus
   Specify center point of torus: pick point
   Specify radius of torus or [Diameter]: 6
   Specify radius of tube or [Diameter]: 1
   Enter number of segments around tube circumference <16>: enter
   Enter number of segments around torus circumference <16>: enter
10.8 Dish

1. Type AI_DISH at the command prompt.

Command: ai_dish
Specify center point of dish: pick point
Specify radius of dish or [Diameter]: 3
Enter number of longitudinal segments for surface of dish <16>: 20
Enter number of latitudinal segments for surface of dish <8>: 15
10.9 Mesh

1. Type `ai_mesh` at the command prompt.

   Command: **ai_mesh**
   Initializing... 3D Objects loaded. Specify first corner point of mesh: 1,1,1
   Specify second corner point of mesh: 4,1,1
   Specify third corner point of mesh: 4,4,2
   Specify fourth corner point of mesh: 1,4,1
   Enter mesh size in the M direction: 20
   Enter mesh size in the N direction: 10
AutoCAD 3D – Chapter 11
Complex Surfaces
11.1 Revolved Surfaces

Regenerates a three-dimensional model with hidden lines

1. Type Revsurf at the command prompt.

Command: revsurf
Current wire frame density: SURFTAB1=6 SURFTAB2=6
Select object to revolve: pick path curve
Select object that defines the axis of revolution: pick line
Specify start angle <0>: enter
Specify included angle (+=ccw, -=cw) <360>: enter
11.2 Surftab Variables

Sets the number of tabulations for both directions to be generated for RULESURF and TABSURF. Also sets the mesh density in ROTATE3D the M direction for REVSURF and EDGESURF commands.

1. Type Surftab1 at the command prompt.
   Command: `surftab1`
   Enter new value for SURFTAB1 <6>: 30

2. Type Surftab2 at the command prompt.
   Command: `surftab2`
   Enter new value for SURFTAB2 <6>: 30

3. Type Revsurf at the command prompt.
   Command: `revsurf`
   Current wire frame density: SURFTAB1=30 SURFTAB2=30
   Select object to revolve: pick path curve
   Select object that defines the axis of revolution: pick line
   Specify start angle <0>: enter
   Specify included angle (+=ccw, -=cw) <360>: enter
11.3 Tabulated Surfaces

1. Type TABSURF at the command prompt.

Command: tabsurf
Select object for path curve: Select object for direction vector:
11.4 Ruled Surfaces

1. Type RULESURF at the command prompt.

Command: rulesurf
Current wire frame density: SURFTAB1=6
Select first defining curve: P1
Select second defining curve: P2
11.5 Edge Surfaces

Regenerates a three-dimensional model with hidden lines

1. Type EDGESURF at the command prompt.

Command: `edgesurf`

Current wire frame density: SURFTAB1=20 SURFTAB2=10

Select object 1 for surface edge: P1
Select object 2 for surface edge: P2
Select object 3 for surface edge: P3
Select object 4 for surface edge: P4
AutoCAD 3D – Chapter 12
Creating Solids
12.1 Solid Primitives

Solid primitives can easily be drawn from both the Draw pulldown menu and from Autodesk’s dashboard.

1. Choose Draw, Modeling, and one of the following solid primitives.

2. Choose the solid primitive from AutoCAD’s Dashboard.
12.2 Polysolid Command

With the POLYSOLID command, you can convert an existing line, 2D polyline, arc, or circle to a solid with a rectangular profile. A polysolid can have curved segments, but the profile is always rectangular by default.

1. Open a drawing with a closed 2D polyline and display in a 3D view.
2. Choose Draw, Modeling, Polysolid.
   or
3. Type POLYSOLID at the command prompt.
   Command: polysolid
   Specify start point or [Object/Height/Width/Justify] <Object>: h
   Specify height <0'-4">: 10'
   Specify start point or [Object/Height/Width/Justify] <Object>: o
   Select object: Pick polygon
12.3 Helix

Creates a 2D or 3D spiral.

1. Begin a new drawing.
2. Choose Draw, Helix.
   or
3. Type HELIX at the command prompt.
   Command: helix
   Number of turns = 3.0000    Twist=CCW
   Specify center point of base: pick point
   Specify base radius or [Diameter] <1.0000>: enter or drag and pick
   Specify top radius or [Diameter] <11.0776>: enter or drag and pick
   Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <1.0000>: enter or drag and pick
12.4 Extrude

Creates unique solid primitives by extruding existing two-dimensional objects. You can extrude multiple objects with EXTRUDE.

1. Type EXTRUDE at the command prompt.

   Command: extrude
   Current wire frame density: ISOLINES=4
   Select objects: pick objects
   Select objects: enter
   Specify height of extrusion or [Direction/Path/Taper angle]: 2
12.5 Extrude with Taper

   or
2. Type EXTRUDE at the command prompt.

Command: extrude
Current wire frame density: ISOLINES=4
Select objects to extrude: pick object to extrude
Select objects to extrude: enter
Specify height of extrusion or [Direction/Path/Taper angle]
<8.9509>: T
Specify angle of taper for extrusion <0>: 15
Specify height of extrusion or [Direction/Path/Taper angle]
<8.9509>: 4
12.6 Extrude with Path

1. Choose **Draw, Modeling, Extrude.**
   
   or

2. Type EXTRUDE at the command prompt.

   Command: **extrude**

   Current wire frame density: ISOLINES=4

   Select objects to extrude: 1 found

   Select objects to extrude: **pick circle (P1)**

   Specify height of extrusion or [Direction/Path/Taper angle] <4.0000>: p

   Select extrusion path or [Taper angle]: **pick P2**
12.7 Revolve Command

1. Open a drawing with 3D objects and display in a 3D view.
2. Choose Draw, Modeling, Revolve.
   Or
3. Type REVOLVE at the command prompt.

Command: revolve

Current wire frame density: ISOLINES=4

Select objects: pick profile

Select objects: enter

Specify start point for axis of revolution or define axis by [Object/X (axis)/Y (axis)]: o

Select an object: pick axis

Specify angle of revolution <360>: enter
12.8 Sweep Command

1. Open a drawing with 2D objects to sweep and display in a 3D view.
2. Choose **Draw, Modeling, Sweep.**
   or
3. Type SWEEP at the command prompt.
   
   Command: **sweep**
   
   Current wire frame density: ISOLINES=4
   
   Select objects to sweep: **pick arc**
   
   Select objects to sweep:
   
   Select sweep path or [Alignment/Base point/Scale/Twist]: **pick path**

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![Sweep Command Diagram](image-url)
12.9 Loft Command

1. Open a drawing with 2D objects to sweep and display in a 3D view.
2. Choose **Draw, Modeling, Loft**.
   or
3. Type LOFT at the command prompt.

Command: **loft**

Select cross-sections in lofting order: **pick circles**
Specify opposite corner: 5 found

Select cross-sections in lofting order:
Enter an option [Guides/Path/Cross-sections only] <Cross-sections only>: **enter**

![Loft Settings dialog box](image)

Click OK.
AutoCAD 3D – Chapter 13
3D Edits
13.1 Convert to Solid

Converts polylines and circles with thickness to 3D solids. With the CONVTOSOLID command, you can convert the following objects into extruded 3D solids:

- Uniform-width wide polylines with thickness
- Closed, zero-width polylines with thickness
- Circles with thickness

**Note** You cannot use CONVTOSOLID with polylines that contain vertices with 0 width or that contain segments of variable width

1. Open a drawing with 2D polylines or circles with thicknesses and display in a 3D view.
2. Choose **Modify, 3D Operation, Convert to Solid.**
   
   or

3. Type **CONVTOSOLID** at the command prompt.

   **Command:** `convtosolid`
   Select objects: pick circle or polyline 1 found
   Select objects: press enter
13.2 **Convert to Surface**

Converts polylines and circles with thickness to surfaces.

1. Open a drawing with 2D polylines or circles with thicknesses and display in a 3D view.
2. Choose *Modify, 3D Operation, Convert to Surface*.
   
   or

3. Type CONVTSURFACE at the command prompt.

Command: `convtosurface`

Select objects: 1 found

Select objects:
13.3 3D Move

Displays the move grip tool in a 3D view and moves objects a specified distance.

1. Open a drawing with 3D objects and display in a 3D view.
2. Choose Modify, 3D Operations, 3DMove.
   or
3. Type 3DMOVE at the command prompt.

Command: 3Dmove
Select objects: pick object to move
1 found
Select objects: enter
Specify base point or [Displacement] <Displacement>: D
Specify displacement <0.0000, 0.0000, 0.0000>: 0,0,2
13.4 3D Rotate

1. Open a drawing with 3D objects and display in a 3D view.
2. Choose **Modify, 3D Operations, 3D Rotate**.

   or

3. Type 3DROTATE at the command prompt.
   
   Command: **3DROTATE**

   Current positive angle in UCS: ANGDIR=counterclockwise  
   ANGBASE=0

   Select objects: **pick object and press enter**

   Specify base point: **pick point**

   Pick a rotation axis: **select X axis**

   Specify angle start point: -90
13.5 3DAlign

1. Open a drawing with 3D objects and display in a 3D view.
2. Choose **Modify, 3D Operations, 3DAlign**.
   or
3. Type 3DALIGN at the command prompt.

Command: `_3dalign`

Select objects: **pick and press enter**

Specify source plane and orientation ...

Specify base point or [Copy]:

Specify second point or [Continue] <C>:

Specify third point or [Continue] <C>:

Specify destination plane and orientation ...

Specify first destination point:

Specify second destination point or [eXit] <X>:

Specify third destination point or [eXit] <X>:
13.6 3D Mirror

1. Open a drawing with 3D objects and display in a 3D view.
2. Choose Modify, 3D Operations, 3DMirror.
   or
3. Type MIRROR3D at the command prompt.
   Command: mirror3D
   Select objects: pick the circle
   Select objects: enter
   Specify first point of mirror plane (3 points) or
   [Object/Last/Zaxis/View/XY/YZ/ZX/3points] <3points>: P1
   Specify second point on mirror plane: P2
   Specify third point on mirror plane: P3
   Delete source objects? [Yes/No] <N>: enter

Circle Mirrored around 3 Points
13.7 3D Rectangular Array

1. Open a drawing with 3D objects and display in a 3D view.
2. Choose **Modify, 3D Operations, 3DArray**.
   
   or
3. Type 3DARRAY at the command prompt.

   Command: **3darray**

   Select objects: pick object and press enter

   Enter the type of array [Rectangular/Polar] <R>: R

   Enter the number of rows (---) <1>: 3

   Enter the number of columns (|||) <1>: 4

   Enter the number of levels (...) <1>: 2

   Specify the distance between rows (---): 5

   Specify the distance between columns (|||): 4

   Specify the distance between levels (...): 8
13.8 3D Polar Array

1. Open a drawing with 3D objects and display in a 3D view.
2. Choose **Modify, 3D Operations, 3DArray.**
   
or
3. Type 3DARRAY at the command prompt.
   
   Command: **3darray**
   Select objects: **pick cube**
   Select objects: **enter**
   Enter the type of array [Rectangular/Polar] <R>: **P**
   Enter the number of items in the array: **5**
   Specify the angle to fill (+=ccw, -=cw) <360>: **enter**
   Rotate arrayed objects? [Yes/No] <Y>: **enter**
   Specify center point of array: **mid of axis line**
   Specify second point on axis of rotation: **pick**
13.9 Extract Edges

1. Open a drawing with 3D objects and display in a 3D view.
2. Choose **Modify, 3D Operations, Extract Edges**.
   
   or

3. Type XEDGES at the command prompt.

   Command: `_xedges`

   Select objects: **pick object**

   Select objects: **enter**
13.10 Facetres

Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed. Valid values are from 0.01 to 10.0.

1. Open a drawing with 3D objects and display in a 3D view.
2. Type FACETRES at the command prompt.

Command: FACETRES

Enter new value for FACETRES <0.5000>: 10
13.11 Isolines

1. Open a drawing with 3D objects and display in a 3D view.
2. Type ISOLINES at the command prompt.
   Command: isolines
   Enter new value for ISOLINES <4>: 20
   Command: regen
   REGEN Regenerating model.
AutoCAD 3D – Chapter 14
Solid Composites
14.1 Union

1. Open a drawing with 3D objects and display in a 3D view.
2. Choose **Modify, Solids Editing, Union.**
   or
3. Type UNION at the command prompt.

   **Command:** union
   Select objects: **pick objects to union**
   Select objects: **enter**

---

**Solid Objects Unioned Together**
14.2 Subtract

1. Open a drawing with 3D objects and display in a 3D view.
2. Choose **Modify, Solids Editing, Subtract**.
   
or
3. Type SUBTRACT at the command prompt.

Command: `subtract`

SUBTRACT Select solids and regions to subtract from...
Select objects: **pick the box**
Select objects: **(press enter)**
Select solids and regions to subtract... Select objects: pick the cylinder Select objects: enter

---

Objects Subtracted from Box

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14.3 Intersect

1. Choose **Modify, Solids Editing, Intersect**

   or

2. Type INTERSECT at the command prompt.

   Command: `intersect`

   Select objects: **pick objects**

   Select objects: **enter**

*Intersection of Cylinder and Box*
14.4 Thicken

 Creates a 3D solid by thickening a surface.

1. Open a drawing with 3D surface and display in a 3D view.
2. Choose Modify, 3D Operation, Thicken. 
   or
3. Type THICKEN at the command prompt.
   Command: `thicken`
   Select surfaces to thicken: 1 found
   Select surfaces to thicken:
   Specify thickness <0.0000>: 1
14.5 Interference Checking

Highlights 3D solids that overlap.

1. Open a drawing with 3D objects that overlap and display in a 3D view.
2. Choose Modify, 3D Operations, Interference Checking.
   or
3. Type INTERFERE at the command prompt.
   Command: interfere
   Select first set of objects or [Nested selection/Settings]: pick objects and press enter.

Select second set of objects or [Nested selection/check first set] <check>: pick rectangle and press enter
The following dialog box allows you to change the zoom and display of the drawing to better see the interfering objects.

You can also choose to retain the interference objects by turning the check box off for “Delete interference objects created on Close”
AutoCAD 3D – Chapter 15
Modifying Solid Faces
15.1 Extrude Face

1. Choose **Modify, Solid Editing, Extrude face.**
   
   NOTE: Must be a solid to extrude the face.

2. Choose a face to extrude. If you choose more than one face, hold the SHIFT key to deselect unwanted faces.

3. **Press ENTER.**

4. Specify height of extrusion or [Path]: .25

5. Specify angle of taper for extrusion <0>: 45

---

**Select Face**

---

**Extruded Face**
15.2 Taper Face

1. Choose **Modify, Solid Editing, Taper face.**
2. Choose a face(s) to taper. If you choose more than one face, hold the SHIFT key to deselect unwanted faces.
3. **Press ENTER.**
4. Specify the base point: **pick the back left corner**
5. Specify another point along the axis of tapering: **pick point**
6. Specify the taper angle: **45**
   Solid validation started. Solid validation completed.
AutoCAD 3D Tutorial

15.3 Delete Face

1. Choose **Modify, Solid Editing, Delete face**.
2. Choose a face to delete. If you choose more than one face, hold the SHIFT key to deselect unwanted faces.
3. **Press ENTER**.
4. Choose the face to delete.

![Diagram of a 3D model showing a face to be deleted. The original face is shown in solid lines, and the face to be deleted is shown in dashed lines. After deleting the face, the model is shown without the dashed lines.](image)
15.4 Copy Face

1. Choose Modify, Solid Editing, Copy face.
2. Choose a face to copy. If you choose more than one face, hold the SHIFT key to deselect unwanted faces.
3. Press ENTER.
4. Pick the solid face to copy.
5. Pick a new location.
15.5 Color Face

1. Choose Modify, Solid Editing, Color face.
2. Choose a face to change the color of. If you choose more than one face, hold the SHIFT key to deselect unwanted faces.
3. Press ENTER.
4. Choose a color to change the face to.
15.6 Imprint

Imprints an object on the selected solid. The object to be imprinted must intersect one or more faces on the selected solid in order for imprinting to be successful. Imprinting is limited to the following objects: arcs, circles, lines, 2D and 3D polylines, ellipses, splines, regions, bodies, and 3D solids

1. Extrude objects (i.e. walls) to create a solid object.
2. Change the UCS and draw an object on a face of one of the walls.
3. Choose Modify, Solid Editing, Imprint.
4. Select a 3D solid: pick solid
5. Select an object to imprint: pick circle
6. Delete the source object <N>: Y
15.7 Extrude Imprint

1. Choose **Modify, Solid Editing, Extrude Faces**.
2. Select the circle that was imprinted in 15.6.
3. Deselect any unwanted faces.
4. **Type -8** as the height of extrusion (or desired height).
15.9 Shell

You can create a shell or a hollow thin wall with a specified thickness from your 3D solid object. AutoCAD creates new faces by offsetting existing ones inside or outside their original positions. AutoCAD treats continuously tangent faces as single faces when offsetting.

1. Choose Modify, Solid Editing, Shell.
2. Select a 3D solid: pick solid
   Remove faces or [Undo/Add/ALL]: enter
   Enter the shell offset distance: .5