Modeling Basics for 3D Printing Using AutoCAD®
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The AutoCAD file format has always fully supported three-dimensional geometry but it was not until AutoCAD 2007, with the introduction of commands such as PRESSPULL and the introduction of visual styles, that AutoCAD began to fully leverage the power of 3D. In this class we will look at the set of AutoCAD commands best suited to modeling three-dimensional geometry that will translate well into the STL file format most commonly used by 3D Printers regardless of the specific 3D printing technology employed. Along the way we will discuss best practices that will help ensure that you produce a successful 3D print.

Learning Objectives
At the end of this class, you will be able to:

- Explain the fundamentals of 3D printing
- Create 3D geometry that is suitable for 3D printing
- Clean and validate a 3D model
- Output an STL format file

About the Speaker
William Work is an architect and CAD manager with a firm of more than three hundred professional staff working on projects spanning a wide range of project types. On a daily basis, he works closely with the design staff providing impromptu one-on-one Q&A style AutoCAD® support and tutorials. On a regular basis he develops and delivers structured classes. He has also worked directly with resellers providing a variety of consulting services throughout the United States. These services have included AutoCAD deployment and customization, on-site training, and CAD standards development and documentation.
Fundamentals of 3D Printing

The term 3D printing refers to any of various processes whereby a three-dimensional object is produced from a 3D computer model. It is generally an additive process that lays down successive layers of material under computer control. Most commonly the materials used will be colored plastic but other materials including metal are available. There are also subtractive processes that remove material from a laminated stack of stock.

In 1986, Charles (Chuck) Hull was issued a patent for stereolithography, a term he coined from the Greek words stereo (solid body), litho (stone) and graphien (to write). The associated STL file format describes only the surface geometry of a three-dimensional object without any representation of color, texture or other common CAD model attributes. It is unitless and represents the 3D model as unstructured triangulated surfaces. AutoCAD has supported stereolithography (STL) files since AutoCAD 2010.

Models produced for the purpose of 3D printing must be cleanly created and watertight. Watertight means that there are no gaps in the geometry. In most cases, the appropriate object type for creating these models in AutoCAD will be solids although meshes can be used as well. Solids are more easily modified in ways familiar to AutoCAD users so in this class we will focus on modeling with 3D solids. The ideal 3D model should be thought of as a shell. When modeling small elements for 3D printing it will often be necessary to exaggerate elements to fall within the resolution of the intended 3D printer.

Printing time and material can often be saved by giving thoughtful consideration to creating your model in parts that can be assembled. For example, if you were designing a building with two masses stacked one upon the other; you would want to consider separating them so that they sit side-by-side on the printer bed. In this way the number of passes made by the print head would be minimized thus saving time. Depending on the 3D printing technology and resolution selected the time to create even a small item can run into several hours.

Just as in creating a 2D drawing, you will create your model at full size and scale it during the output process. The minimum element size that can be printed will depend on the device you are printing with. My firm has 3D printers from two manufacturers; they are the Objet 30 Pro 3D and the MakerBot Replicator2. The Objet has a build plate that can accommodate pieces up to 11.57" (292 mm) x 7.55" (192 mm) and has a limiting height of 5.85" (148.6 mm). The MakerBot has a build plate that can accommodate pieces up to 11.2" (284.48 mm) x 6.1" (154.94 mm) and has a limiting height of 6" (152.4 mm). Service bureaus will likely have larger printers.

Creating 3D Geometry

Before you begin working in 3D you will want to prepare your work environment and acquaint yourself with visual styles as described below. After having set up our 3D modeling environment we will then look at the 3D modeling tools available to you in AutoCAD.
Preparing Your Work Environment for 3D Modeling

AutoCAD has a number of settings that will generally assist you in your work or are specifically related to working in 3D. The following is a list of those system variables that are especially useful. The first step, before adjusting any of the settings listed, is to make the 3D Modeling workspace current. As you work more in 3D you may decide to make further adjustments to your work environment. To preserve many of those adjustments, you can create a user profile specifically for use when you are doing 3D modeling.

- **DELOBJ=0** [default: 3, range: -3 to 3, saved in: registry]
  This system variable determines whether objects used when extruding, revolving, sweeping or using the presspull command, are retained or deleted. When set to 0, all source objects are retained. Note that if the surfaceassociativity system variable is set to 1 (the default) defining geometries are not deleted when an associativesurface is created.

- **DISPSILH=1** [default: 0, range: 0 or 1, saved in: drawing]
  This variable controls the display of silhouette edges of 3D solid objects in a 2D Wireframe or 3D Wireframe visual style.

- **FACETRES=1** [default: 0.5, range: 0.01 to 10.0, saved in: drawing]
  This variable controls the smoothness of shaded and rendered objects and objects with hidden lines removed. When the output for the 3D printer is created, this value is automatically set to 10.

- **ISOLINES=0** [default: 4, range: 0 to 2047, saved in drawing]
  This variable specifies the number of contour lines displayed per surface on objects. It has no effect on the smoothness of exported solids. Reducing the number of isolines from its default of 4 simplifies the display of curved objects making the drawing less cluttered and easier to edit. For best results, setting ISOLINES to 0 should be combined with setting DISPSHIL to 1. Regen the drawing to see the effect of a change to this value.

- **NAVBARDISPLAY=1** [default: 1, range: 0 or 1, saved in: registry]
  The navigation bar provides easy, on screen, access to several navigation tools from a single onscreen interface.

- **NAVVCUBEDISPLAY=3** [default: 3, range: 0 to 3, saved in: drawing]
  Use the navigation cube to change easily your view of the model.

- **OSNAPZ=0** [default: 0, range: 0 or 1, not saved]
  This variable controls whether object snaps are automatically projected onto a plane parallel to the XY plane of the current UCS at the current elevation.

- **SOLIDCHECK=1** [default: 1, range: 0 or 1, not saved]
  This variable turns 3D solid validation on and off for the current session. An example error message that you may see would be "Modeling Operation Error: Gap cannot be filled."
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- **UCSDETECT=1**  
  [default: 1, range 0 or 1, saved in: registry]
  When UCS detection is on, AutoCAD will temporarily align with the UCS of a face to facilitate drawing and editing.

- **VPCONTROL=1**  
  [default: 1, range: 0 or 1, saved in: registry]
  This variable controls whether the menus for viewport tools, views, and visual styles that are located in the upper-left corner of every viewport are displayed.

**Visual Styles**

In AutoCAD, a visual style is a collection of settings that allow you to control how objects appear in a viewport. Visual styles are stored in drawings and can be customized to suit your needs. Using the DesignCenter you can add visual styles from one drawing to another.

While working on a 3D model you use visual styles to control the image displayed on the screen. This can help make the model easier to understand. For example, you might change from the Wireframe style to the Shades of Gray style. To change the current visual style, use the text buttons found in the upper left corner of the graphics screen.

Setting VPCONTROL system variable to 1 will display the viewport controls in the upper left corner of the drawing area.

**AutoCAD’s 3D Modeling Tools**

In order to access the 3D modeling tools in AutoCAD, make the 3D Modeling workspace current; you should then see tabs on the ribbon labeled Solid, Surface, and Mesh. As stated above, we will concentrate on the Solids. Surface models cannot be 3D printed and while meshes are perfectly valid object types for creating 3D printed objects; I have found solids to be the more versatile object type for creating the required output for 3D printing.

**Solid Primitives**

Think of the solid primitives as your basic building blocks. A 3D model will often start off as one or more primitives that you manipulate in various ways to achieve the desired result. There are seven primitives. They are the box, the cylinder, the cone, the sphere, the pyramid, the wedge, and the torus.

![The Seven Solid Primitives](image)

The solid primitives have object grips that allow you to reshape, move, and manipulate them in a variety of ways.

**Polysoild**

The polysoild is a unique 3D solid object that creates a wall-like object. Polysolids can be generated from 2D geometry including lines, circles, arcs, ellipses, and
polylines. By default, the polysolid will have a rectangular cross section but object grips provide many ways to manipulate the object.

**Solids Generation**

Solids can be generated using closed objects that define profiles and paths. There are four commands that create solids in this way. They are Extrude, Loft, Revolve and Sweep.

Solids generated in this way have object grips that allow you to reshape, move, and manipulate them in a variety of ways. It is often possible to create directly the exact shape required. For example, a profile can be extruded to create a moulding. A handrail can be generated by extruding its cross section along a path. Use Loft to model an object with varying cross sections. A column capital might be created directly by revolving a profile. A sweep can be used to bend a profile around a curve in three dimensions. It includes options to twist and scale the profile as it is generated.

**Solid Editing**

Beyond the straightforward creation of 3D objects using primitives and the solid generation tools, a number of powerful tools are available to further manipulate the 3D objects you have created.

**Composite Solids**

Composite 3D objects are created by combining, subtracting or finding the intersection of two or more 3D solids. AutoCAD provides three commands for these purposes; they are Union, Subtract, and Intersect.

When you perform one of these Boolean operations on your solids they will lose many of the grips that are available with the simpler underlying objects.

In order to create a 3D print it will be necessary to union the individual parts you have created in your 3D model.
Editing Subobjects (Faces, Edges and Vertices)

Using the subobject filters found on the Ribbon Solid tab Subobject panel you can select one or more subobjects and modify them independently from the associated solid by using the grips that become available. Alternatively, you can press and hold the Ctrl key and use the cursor to select the set of subobjects to manipulate.

The easiest command to use when editing faces is PRESSPULL found on the Ribbon Solid tab Modeling panel. Using this command you click inside a bounded area and drag to generate a positive or negative extrusion. For example, you might create the pattern in a Roman lattice by using PRESSPULL to pull a shape right through the panel. Of course the same result could be achieved in a number of other ways.

The Ribbon Solid tab Solid Editing panel offers many options for editing your solids. A very useful command option when modeling objects for 3D printing is the shell command. Use this command to convert your solid block into a shell. Be sure to provide a thickness for the shell that, when scaled down for 3D printing, will have sufficient thickness to withstand the cleaning process and subsequent handling.

Fillets and Chamfers

The ordinary fillet and chamfer commands are just as effective in working with 3D solids as they are with 2D objects. They are the perfect commands for easing edges.

Creating 3D Solids From Other Objects

In this class we have focused our attention on creating 3D solids directly but AutoCAD is by no means limited to modeling with solids. Some complex shapes can more easily be modeled using surface or mesh tools.

There are a variety of commands and methods provided in AutoCAD that you can use to convert surfaces and meshes in your drawing to 3D solids. The DELOBJ system variable controls whether the objects you select are deleted when the 3D solid is created.

Use the CONVERTOSOLID command to convert closed polylines and circles with thickness to extruded 3D solids. The command can also be used convert 3D meshes to 3D solids. The smoothness and number of faces in the resulting 3D solid are controlled by the SMOOTHMESHCONVERT system variable.

Apply the SURFSCULPT command to a group of surfaces that enclose a watertight volume to produce 3D solid.

You can apply the THICKEN command to a complex curved surface to create a 3D solid that would be difficult to create otherwise.
3D Object Snaps
When working with 3D objects there are six object snap modes that you can use to facilitate your object creation. Use the DSETTINGS command or, more directly, the 3DOSNAP command to set running 3D object snaps. The F4 key toggles the snaps. Autodesk advises that, “Because 3D object snaps can slow performance, select only the ones you need.”

Cleaning and Validating the 3D Model

Before outputting your model to the STL format required by 3D printers, there are two steps that you can take to help insure that you have a usable 3D solid.

Run the CLEAN command found on the Ribbon Solid tab Solid Editing panel SOLIDEDIT flyout. This command will help produce a cleaner solid by removing redundant lines from your 3D solid object.

Run the CHECK command found on the Ribbon Solid tab Solid Editing panel Solid Editing flyout. This command is used to validate the 3D Solid object you have modeled. If the 3D solid is valid, the command will return the message "This object is a valid ShapeManager solid." If the 3D solid is not valid, you will get no feedback and the command will simply repeat its request that you "Select a 3D solid:"

Creating the 3D Output

Having created and validated your 3D model you are now ready output it to the required STL format. At the command line, type 3DPRINT and select the model that you are printing. Remember that only 3D solids and watertight meshes will be output even if there are other selected objects. It is common to model more than one 3D object in a file but for the purposes of 3D printing, each model should be output separately. Once you have selected your model, the Send to 3D Print Service dialog box will appear with a preview of the output. The Output preview In the Output dimensions area of the dialog box, enter the output scale. For example, if your 3D print is to be at 1/4"=1'-0" scale, type 1/48. AutoCAD will convert the fraction to a decimal for you. Glance at the Length, Width and Height values provided to confirm that the result is in line with what you are expecting and that the model will fit within the printer's 3D print envelope. Click OK to create the STL file.
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3D Print Dialog Box