

LAORT CADD LECTURE NOTES TUESDAY 11/12

CREATING OBJECTS TO DEFINE REVOLVE/EXTRUDE SOLIDS

READING

PEDIT	pgs. 638-642
JOIN	
SPLINE FIT	pgs. 648-650
SPLINE	pgs. 651-652
OFFSETTING SPLINES	see help files

OVERVIEW

Because a vast majority of all solid models will be made using the **REVOLVE** and **EXTRUDE** commands, the ability to produce valid objects for these functions in an accurate and rapid manner is essential.

First, we must review what makes an object (or curve) valid for these two functions. The three fundamental requirements are:

- *The Object Must Be Planar*
- *The Objects Must Be Closed*
- *The Objects Must Not Be Self Intersecting*

Just for reference let's look at the definition of planar.

planar

SYLLABICATION: pla·nar
 PRONUNCIATION: plnr, -när
 ADJECTIVE: 1. Of, relating to, or situated in a plane.
 2. Flat: *a planar surface*.
 3. Having a two-dimensional characteristic.
 ETYMOLOGY: Late Latin *plnris*, flat, from Latin *plnus*. See plain.
 OTHER FORMS: **pla·nari·ty** (pl-nr-t) —NOUN

These two requirements mean that only the following Autocad objects are valid for making solids using the **EXTRUDE** or **REVOLVE** commands.

- **CIRCLE**
- **ELLIPSE**
- **Closed planar SPLINE**
- **3DFACE**
- **SOLID**
- **Closed Planar POLYLINE (this includes POLYGONS)**
- **REGION**

➤ *DONUTS*

In order to create complex shapes we usually find ourselves using the most complex of these objects, the PLINE and REGION.

More complex shapes typically drawn using the SPLINE command may be used to produce solids if the user is aware of the restrictions to these curves and how they can be manipulated to create a valid object for solid modeling.

This leads to the primary focus of this document, creating POLYLINES or REGIONS that contain SPLINE curves.

CREATING OBJECTS WITH SPLINE CURVES FOR SOLIDS

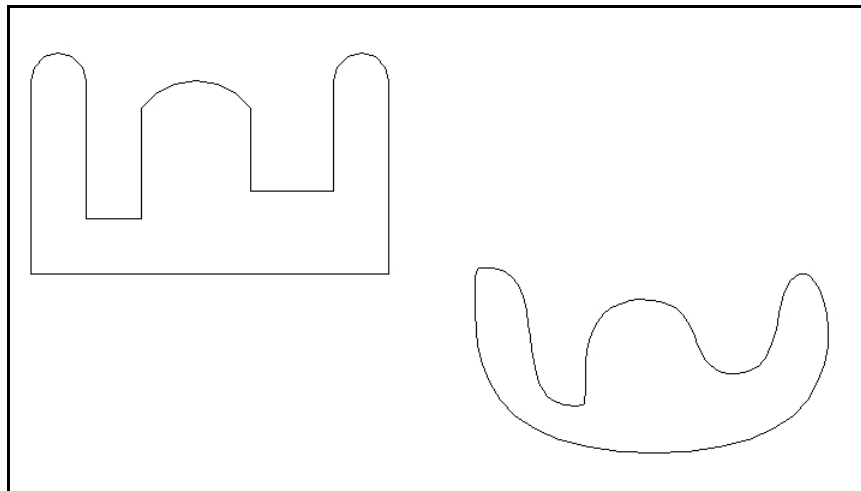
The most common method of creating a composite object for use in solid modeling is to draw the desired shape using LINES and ARCS and create a POLYLINE from them using the PEDIT command. This is typically the fastest and most reliable way to create a useful object.

Because we cannot use ellipses or splines drawn using the SPLINE command to define a POLYLINE we must look at alternative methods.

USING POLYLINES

In many cases we do not need to define a precise spline curve to serve our needs. This allows the use of SPLINE FIT POLYLINE curves as a valid type of object for certain modeling situations.

AutoCAD recognizes both fit polylines and spline-fit polylines. A spline-fit polyline uses a curve fit, similar to a B-spline. There are two kinds of spline-fit polylines, quadratic and cubic. Both of these polylines are controlled by the SPLINETYPE system variable.



NORMAL FIT POLYLINE ON THE LEFT AND THE SPLINE-FIT POLYLINE ON THE RIGHT THAT IS CREATED FROM IT USING THE PEDIT/SPLINE

A fit polyline uses standard curves for curve fit and utilizes any tangent directions set on any given vertex.

You can change the splined polyline back to a polyline using the Decurve option. The illustration shows a polyline edited with the Spline option.

However, some editing actions, such as trimming, breaking, and any grip editing, remove the spline definition of a fit or splined polyline. For this reason, it is better to use a true spline, which maintains its spline definition.

If you plan to use a SPLINED POLYLINE in a solid modeling application it is essential that you become familiar with the following system variables to achieve the desired results.

- *SPLFRAME* – how it is displayed
- *SPLINESEGS* – number of line segments
- *SPLINETYPE* – spline type (cubic or quadric)

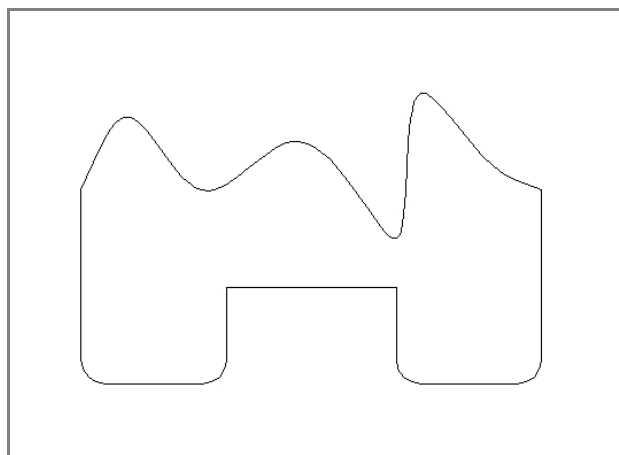
These variables define how Autocad produces and displays a spline from your input geometry.

USING REGIONS

A much better option than the use of SPLINE FIT POLYLINES is to use planar SPLINE curves to create a REGION. The only essential requirement of a SPLINE that is to be used for this purpose is that it is PLANAR and that it is in the same plane as any other curves that make up the edges of the REGION.

Regions made in this manner have much more flexibility in shape and are also more mathematically precise. A shape such as the one above would be nearly impossible using POLYLINE but is simple using lines, arcs, and splines to make up the edges of the resulting REGION.

Care must be taken when drawing the original spline curve to make sure that it is planar. It is a good idea to create a UCS in the plane that the region is to be define in and create the SPLINE with that UCS current.



REGION MADE WITH LINES ARCS AND A SPLINE

To verify that a SPLINE is planar we use the either the PROPERTIES command or use the LIST command. LIST will say the word PLANAR right after the properties

label on about the sixth line of the output. The properties form has PLANAR as an item in the MISC category.

Once you have verified that you have valid splines you must convert your edge curves to a REGION. This should be done either by using the REGION command or by using the BOUNDARY command with the Object Type set to REGION. Attempting to create a POLYLINE first before creating the REGION will fail because the PLINE command does not accept splines as valid input.

OFFSETTING SPLINES

In order to use splines effectively for applications as discussed above it is useful to be able to perform editing commands on them in the same manner as you would with lines and arcs. Offsetting is one of the most essential of these functions.

While splines can be offset there are some restrictions. Understanding these will help to avoid problems or figure out how to recognize and solve them. The most important factor is:

- *The SPLINE must be PLANAR*

TIPS

COMMON CAUSES OF PROBLEMS and ERRORS

- *OPEN POLYLINE*
- *POLYLINE WITH CURVES LAYING ON TOP OF EACH OTHER*
- *INVALID POLYLINE*

WAYS TO AVOID THESE PROBLEMS

- *WHEN USING PEDIT DO NOT USE ALL OR WINDOW SELECT WHEN SELECTING THE CURVES TO JOIN*
- *DO NOT SELECT THE ORIGINAL CURVE IN THE PEDIT COMMAND WHEN YOU PICK THE CURVES TO JOIN*
- *AVOID ADDING SMALL RADIUS FILLETS UNTIL YOU HAVE MADE THE POLYLINE OR THE SOLID*
- *BEFORE EXITING THE PEDIT COMMAND MAKES SURE THAT THE FIRST OPTION IN THE COMMAND LINE IS "OPEN". THIS TELLS YOU THAT IT IS CLOSED.*

SOLID EDITING - BOOLEAN OPERATIONS

READING

BOOLEAN OPERATIONS

pgs. 920-924

UNION

SUBTRACT

INTERSECT

Other reading CADDEX lesson T304.pdf page 10

OVERVIEW

The foundation of all solid modeling programs is the ability to combine multiple bodies in order to produce the desired final result.

Some of the first solid modelers approached the process by using a *Material Removal* method. This could best be described as starting out with a chunk of material larger than your ultimate part and chipping away at it until you get what you want.

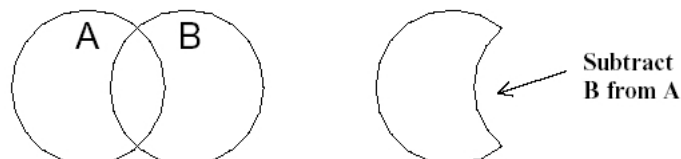
The more common approach is to use both *Material Addition* and *Material Removal* to develop you design. In order to accomplish this we must have three fundamental modeling functions. These functions are known as the BOOLEAN operators.

Just to put this one question to bed, here is the definition of BOOLEAN from the dictionary.

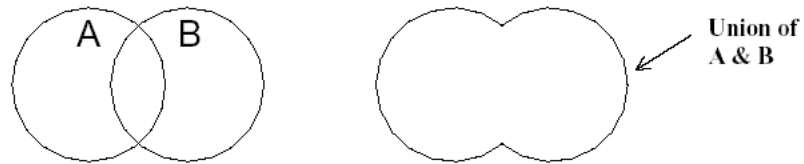
Boolean SYLLABICATION:Bool·e·n ADJECTIVE:1. Of or relating to a logical elements, through the operators AND, OR, NOT, and XOR: *a browser that supports Boolean searches*. 2. Of or relating to a data type or variable in a programming language that can have one of two values, true or false. ETYMOLOGY:After George Boole

I hope that clears that up! The three operations SUBTRACT, UNION, and INTERSECT are the saw, drill, hammer and nails in your modeling toolbox.

SUBTRACT, which is used to remove material is the most commonly used of the three operators. This could be thought of as a cutting tool, or chisel, that is used to shape an object by using other objects as cutting tools.



UNION is used to combine bodies together, eliminating redundant volumes. You could think of this as the way you would glue things together.



INTERSECT is the least commonly used of the three operators. This is because it is the most difficult to visualize and doesn't have a good analogy.



TIPS

- *Be aware that the result of any the three Boolean commands is a single solid. This means that all layer/color properties from the first object selected will be used for the resulting object. Don't spend a lot of time creating fancy layer/color schemes for your objects prior to combining them together.*
- *The sequence of executing your Boolean operations is critical when making complex parts. It is a good idea to plan your strategy out before you start making your part.*
- *When using a solid to cut a hole from another object, make sure the cutter extends clearly through the object it is being used to cut. This will help avoid ambiguity in some of the faces.*
- *Make sure that you have your precision set to a small enough value. If you are working in architectural units with a precision of 1/8", don't try to draw things using numbers like 1.05".*