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MicroStation XM Training Manual 3D Level 3

The first eight pages of Module 5 are shown below. The first two pages are typical for all Modules and provide the Module title and set out the learning objectives. The suggested time for completion of the Module is given at the end of Page 5-2.

Pages 5-3 to 5-8 are instructional pages and, in this case, discuss the Extrude tool. The information and step-by-step instruction is typical throughout the 3D training manual.

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MicroStation XM 3D LEVEL 3

Module 5

3D CONSTRUCT TOOLS

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Module Information

Prerequisites:

Module 4 MicroStation 3D

Introduction:

AccuDraw provides a selection of tools to derive 3D drawings from existing geometry. These are a very powerful set of tools that greatly increase the range of 3D shapes that can be drawn.

Objective(s):

5.1 Construct elements using the Extrude tool.
5.2 Construct revolved elements using the Construct Revolution tool.
5.3 Construct elements using the Extrude Along A Path tool.
5.4 Construct elements using the Shell Solid tool.
5.5 Construct elements using the Thicken To Solid tool.
5.6 Construct elements using the Loft Block to Circle tool.

Time:

This Module should be completed within 4 hours.

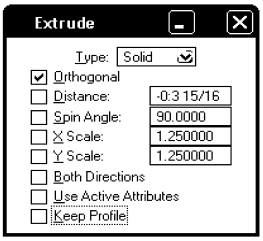
DISCUSSION:

The 3D Construct tool box contains five tools that derive 3D shapes from existing geometry. This is extremely useful because it is often easier to edit existing geometry than to create new geometry. The first two tools are a perfect example. Both the Extrude and Construct Revolution allow you to use a 2D shape to generate a 3D element.

You will find the 3D Construct tool box on the 3D Main tool frame, next to the Primitives tool box.

Open a new design file called "Mod 5 3D" before starting.

5.1 EXTRUDE TOOL



 You looked briefly at the *Extrude* tool in Module 3, but now you need to look closely at this invaluable tool. The Extrude tool creates a 3D complex element by projecting a *boundary* element or *profile*.

Start the tool and look at the available options:

Type
The two options are the standard *Solid* or *Surface*.

Orthogonal
Constrains the center line of the extrusion to be orthogonal to the profile.

Distance
Used to preset the distance the profile is extruded.

Spin Angle
Sets the angle through which the extrusion is twisted (similar to the Rotate Element tool). There are limitations to extent of the spin angle as you will see.

X and Y Scales
Sets the scale factors of the extrusion (similar to the Scale Element tool).

Both Directions
The profile can be extruded in both positive and negative directions at the same time.

Use Active Attributes
If ON, uses the active attributes. If OFF, uses the attributes of the profile.

Keep Profile
The original profile (or boundary) element can be retained in the design.

The profile being projected can be a line, line string, arc, ellipse, multi-line, complex chain, complex shape, or B-spline curve. You can even project text. This wide range of element types leads to very sophisticated drawing procedures.

Before your start experimenting with projected elements there are some points you should think about:

1. You use a *profile* or a *boundary* as the element to be projected. Technically a profile is an *open* element, whereas a boundary is a *closed* element. MicroStation uses the term *profile* for open or closed elements and in this manual I will do the same.

2. While you can project a *set* of *separate* but *connected* elements, it is best to use *joined* or *closed* elements (as in line strings, complex chain, shapes, complex shapes, composite curves, etc.). If you try to extrude a set of separate elements you will likely see a warning message advising you that your action may result in *overlapping geometry*. This is not permitted and the extrusion will not be completed.

3. After extruding a profile, you may not want to see or even keep the *original* profile element. There are several ways of achieving this: draw the profile on a level different from the extrusion, use a *construction class* element as the profile, set the *Keep Profile* option to OFF in the Tool Settings window, or simply delete the profile element after extrusion.

4. Point 3 above becomes significant because the profile element is *not* incorporated into the extrusion and remains *entirely separate* from the extrusion. This may not be an advantage, especially when developing hollow surface extrusions.

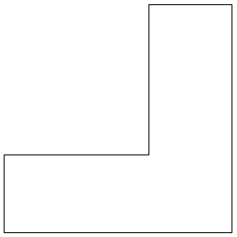
5. The *attributes* of the *extruded element* are *derived* from the *profile element*. However, the *level* of the projected element can be set differently from the profile element.

Before you start working with the Extrude tool it will help if you set the *Isometric view* to render the extrusions automatically. Make the following settings in the *Isometric* view:

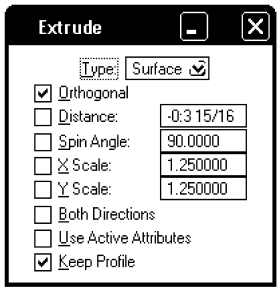
Step 1 Select *Display Mode Smooth.*

Start by looking at an angle shape:

Step 1 *Create* a new *Model* called *Extrude*.
Step 2 In the *Isometric* view, *draw* the L-shape using *SmartLine* and the *Front* compass orientation. No specific dimensions, but be sure the surface is closed.

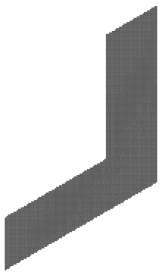


Front view.



Step 3 Start the *Extrude* tool and make the settings shown at the left.

You are going to keep the profile with the extrusion for the moment. The profile should be displayed in the Isometric view as a rendered shape.

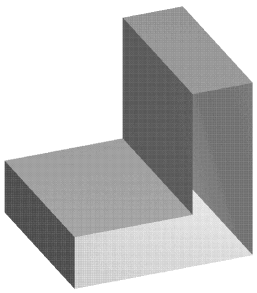


Rendered shape.

Extrude the profile as a *Surface* element first:

Step 4 *Select* the profile in the *Top* view and *drag* the extrusion to the lower right.

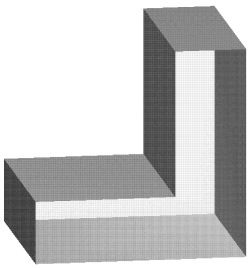
Step 5 *Data-point* to set the extrusion distance.



Extruded profile.

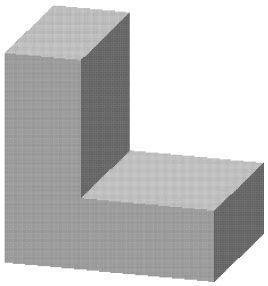
Since you extruded this as a surface type it should be possible to see through the interior of the angle:

Step 6 *Dynamically rotate* the angle to see the interior.



Rotated extrusion.

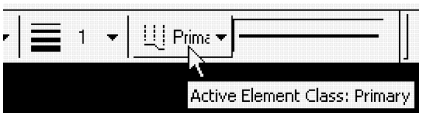
Notice that the extrusion is indeed a surface type and that you can see the original profile at the back of the angle (shown on the left). To be sure about this, rotate the angle again to see the "back" of the angle (shown on the right). Clearly, the original profile is still in place.



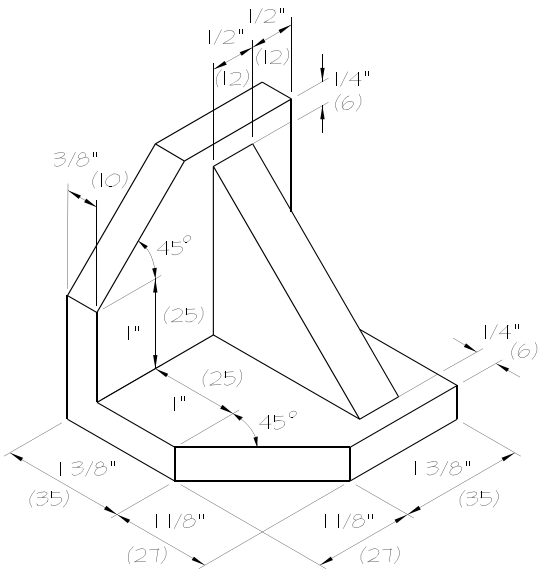
Rotated extrusion.

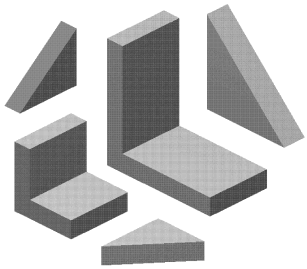
In this example you have kept the profile, but you can always delete it if necessary. Alternatively, if you had known that you did *not* want to keep the profile you could have set the *Keep Profile* option OFF before extrusion. To *keep* the profile but *not have it display*, you can either put it on a separate level or make it a *construction class* element.

TOOL TIP !
A quick way to apply *Construction Class* to elements is to display the *Active Element Class* tool in the *Attributes* tool box. *Right-click* in the *Attributes* tool box to see the list of options and *tick* the *Active Element Class* tool.

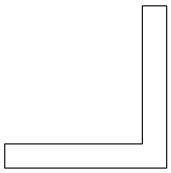


Since you are working with an angle example it would be interesting to redraw the original angle and wedge assembly from Module 3. It was a lengthy process using just faces, but extrusions are easier.





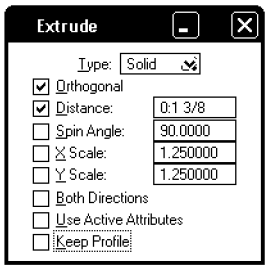
You will need a total of five extrusions as shown in the illustration on the right. In a later Module you will encounter the Cut Solid tool which will reduce the number of extrusions to two.



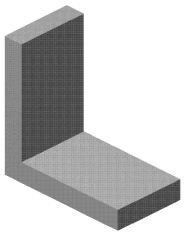
Step 1.

Step 1 In the *Right* view, draw the *large* L-shaped profile as a closed shape to the dimensions given above. Check the Isometric view to ensure the correct orientation of the profile.

Step 2 Start the *Extrude* tool and set *Type* to *Solid*, *Orthogonal ON*, and *Distance* to 1 3/8" (35). All other settings OFF.



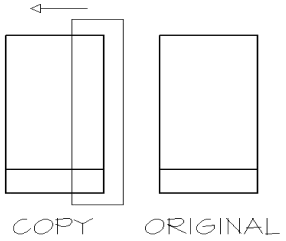
Step 2.



Step 3.

Step 3 *Data-point* on the profile in the *Isometric* view and move the cursor to the *rear* of the profile.

Next you are going to copy the extrusion and, so that the 45^0 diagonal line can be maintained, use the *Stretch* tool to reduce the length of the legs and the length of the copied extrusion:

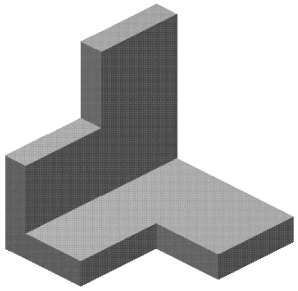


Applying a fence in the Front view.

Step 4 In the *Front* view, *copy* the extrusion to the *left*. Use the View compass orientation to maintain a horizontal alignment between the two extrusions.

Step 5 In the Front view, on the *copy, reduce* the *length* of *each leg* by 1 1/8" (35) and the length of the angle by 1/4" (10) using *fences* and the *Stretch* tool.

It is safer to apply a fence to the legs in the orthographic views since doing so in the Isometric view can lead to unexpected results. More on fences later.

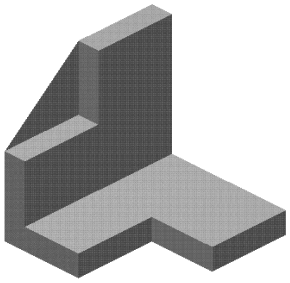


Step 6.

Step 6 In the Isometric view, *move* the edited *copy* to snap to the original angle (feel free to switch in and out of Wireframe).

The next step is to add the two diagonal extrusions to the copied and stretched angle.

Step 7 In the *Isometric* view, use *SmartLine* to draw a *triangle* between the *three points* of the *diagonal* in the vertical leg.

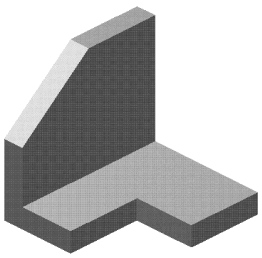


Step 7.

When the triangle is placed it immediately displays as a rendered planar triangle. Now *extrude* it by snapping to the leg thickness:

Step 8 *Start* the *Extrude* tool and turn OFF the *Distance* setting.

You must identify the triangle for extrusion and you need two snap points for the distance. You can *both* identify the triangle *and* the first snap point at the same time if you manually snap to a corner of the triangle. Otherwise you will need to use the "O" shortcut for the first snap point.

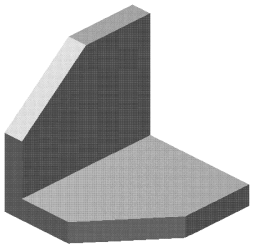


Steps 9 and 10.

Step 9 *Identify* the triangle by *manually snapping* to a *corner* of the *triangle*.

Step 10 *Extrude* the triangle by *snapping* to the corresponding thickness *corner*.

You could, of course, have entered the distance in the Tool Settings window but it is usually quicker to use snaps.



Step 11.

Step 11 *Repeat* Steps 7 to 10 to place the *diagonal extrusion* in the *horizontal* leg.

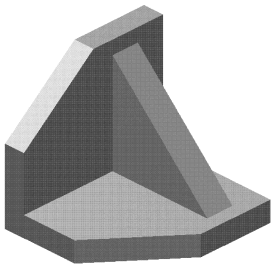
You may be thinking that you could have placed the second diagonal extrusion using the *Rotate Element* tool. You can, indeed, use this tool, but the procedure is not immediately obvious. I will show you how this is done after you have finished the full angle.

Now place the interior web extrusion by placing a triangle located 3/4" (18) from the back of the object. You could place the triangle at one of the web face locations, but I want you to use the *Both Directions* option of the Extrude tool:

Step 12 With *SmartLine*, *place* the *triangle*. Use AccuDraw to locate top and bottom points of the triangle while keeping a close eye on the compass orientations.

Step 13 *Start* the *Extrude* tool and turn *Both Directions* ON.

Step 14 *Identify* the *web triangle* and *extrude* it 1/4" (6) in any direction.



Steps 12 to 14.

With *Both Directions* ON, the dimension is repeated on both sides of the triangle.

Now that the assembly is complete you can look at the action of the *Rotate Element* tool to place the two diagonal extrusions.

Step 1 *Make* a *copy* of the angle assembly and *delete* the *horizontal* diagonal extrusion.

You are now ready to apply the Rotate Element tool to the vertical diagonal extrusion, but you must think carefully about how the tool operates and how best to use it in the drawing. Remember that the tool requires a *point* about which to *rotate* the extrusion and place a copy.

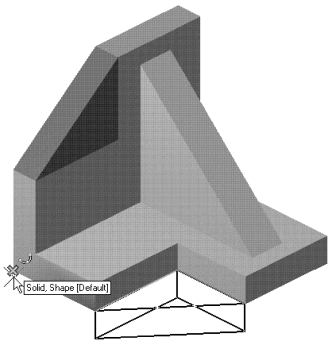
Step 2 *Select* the *vertical* diagonal extrusion with the *Element Selection* tool.

Step 3 *Start* the *Rotate Element* tool, turn *Copy Element* ON, and set the *Active Angle* to 90^0.

In which view should you locate the point to rotate about? The obvious candidates are the Isometric and the Right views since these are the only ones in which you will be able to see the 90^0 rotation you need. Try the Isometric view first:

Step 4 *Switch* to the *Side* compass orientation (the rotation is about the Z-axis).

Step 5 *Find tentative points* at the *inner* and *outer corners* of the angle to see if either snap point is acceptable.



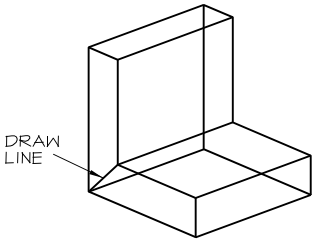
Tentative point at outer corner.

You will find that neither tentative point is acceptable. One places the extrusion too high and one too low. The reason is that the tool is rotating the extrusion by its *centroid* (center of the object's *volume*). Because of this you will need to find the center of the distance between the inner and outer corners of the angle which will be in alignment with the extrusion's centroid.

You can do this with AccuDraw, but it's easier just to draw a construction line from corner to corner and snap to its midpoint.

Step 6 *Place* a *line* between the *inner* and *outer* corners of the angle.

Step 7 *Start* the *Rotate Element* tool and snap to the *midpoint* of the line.



Step 6.

The extrusion should now be copied to the horizontal leg at its correct location.

Why did you need a *positive* 90^0 angle? Because you are working with the Side compass orientation which is orientated with the Right view and the positive 90^0 angle is a correct counterclockwise rotation in that view. For practice, undo the copy and the line and repeat the process in the Right view to see how it works there.