

Extrude macro v1.1

(for DataCAD¹)

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Whilst it is free (or because of this), I would like and expect that if you can think of any improvements or spot any bugs (or even spelling or formatting errors in the documentation) that you would let me know. Your feedback will help with future development of the macro.

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1 General Description

This macro will allow you to create entities by defining a profile and then sweeping that profile along a defined path.

The profile can be defined by one of the following methods:

- selecting an entity or group of entities, and then selecting a reference point (which is the point that will follow the defined sweep path). A number of different entity types are supported for this (see [section 2.1.1.1](#)).
- selecting a symbol that contains suitable entity types. The symbol's insertion point will be used as the reference point in this case.
- Using parameters to define a regular polygon (with from 3 to 20 sides) or circle. The reference point will be the centre of the polygon or circle.

The sweep path can be defined by either of the following 2 methods:

- Tracing a path and selecting points with the cursor. You can specify the Z of the extrusion to be either the current Z-Height or Z-Base, or to be a custom Z value.
- Selecting 1 or more existing entities. Many entity types are suitable for this selection (if you select an entity type that is not available it will be ignored).

For 2D entity types several options are available to define the Z of the extrusion. For 3D entities some of the same options are available but you may need to experiment a little as the results may not always be what you expect (see details of Z options in [Section 2.3.2.2](#) of this document). The entities created by the macro are a series of polygons, and the macro allows options such as enlarging the profile as it moves along the path.

This macro has predominantly been tested using DataCAD 22, but some testing has also been done using DataCAD versions 19 and 21. Although I have not tested it with earlier versions, I believe it should be compatible with DataCAD versions 14 and onwards.

2 Using the Macro

The Section Profile menu will be displayed when you invoke the macro. Select any required options (Enlarge, Thickness etc.), and then select the profile shape by either selecting an entity (or entities using the Group or Area options), selecting a Symbol File, or specifying parameters to define a circle or regular polygon. Once you have selected the required profile, the Extrude Path menu will display. You can return to the Section Profile menu by selecting the **S7 Chg Profile** option from the Extrude Path menu.

The sweep path can be defined by tracing a path on the screen or by selecting entity(s) to define the path. Once the path is defined the extrusion can be created immediately, or you can select to have a confirmation dialog displayed first.

2.1 Section Profile

2.1.1 Defining the Profile



The profile can be defined using one of the methods detailed below:

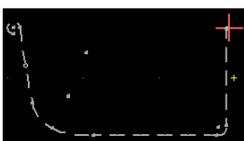
2.1.1.1 Selecting Entity(s) from the Drawing

Select one of the options, **F1 Entity**, **F2 Group**, or **F3 Area** (together with the appropriate **F6 Layer Search** setting) from the Section Profile menu, then select the required entities as you would for other standard DataCAD functions.

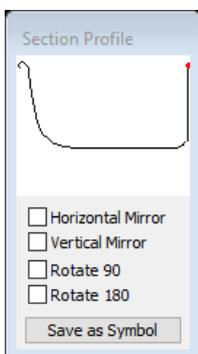
Note that only entities of the following types are acceptable:

- 2D Lines
- 2D Arcs, Circles and Ellipses
- Bezier curves
- BSpline Curves
- Polygons ²
- Polylines ²

If multiple entities are selected, they do not all need to be of the same type, but any that are not one of the above types will be ignored.

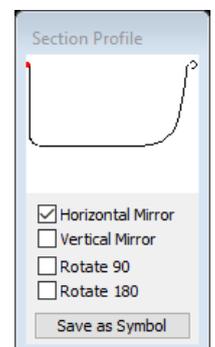


Selected entities will be highlighted and you will be prompted to select a reference point for the profile. In the example at left, I selected the point shown by the red cursor as the reference point.



After the reference point is selected, a Section Profile window will display³ (or will be updated if it is already displayed) as shown at left. The red dot on the diagram represents the reference point. If you expect to use this profile again you can save it as a symbol file.

You can select any (or all) of the checkboxes on the Section Profile window to mirror and/or rotate the profile as shown at right.



² If you select polygons or polylines that are not in the horizontal plane, the profile shape used will be shape that you see when you look at the entity in plan view.

³ Default behaviour is described here. Advanced Options allow you to suppress the display of the Section Profile window.

2.1.1.2 Selecting a Symbol File

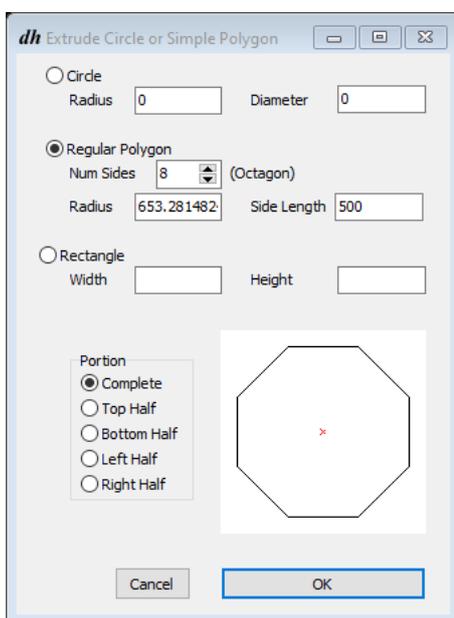
Selecting the **F8 Symbol File** option from the Section Profile menu will display a File Open Dialog to allow you to specify the symbol file to be used. Note that the file dialog displayed by the macro does not contain a preview window, so it is necessary to know which symbol file you wish to use before you start selecting this option.

Although the selected symbol may contain a variety of different entity types, only the appropriate types (as specified in section 2.1.1.1) will be used. Any other entities included in the symbol will be ignored.

The symbol insertion point will be used as the reference point for the profile (you will not be prompted to select a different point).

Once a symbol file is selected, an illustration of the profile will be displayed in the Symbol Profile window.

2.1.1.3 Using a Circle or Regular Polygon defined by parameters



Selecting **F9 Circle/Plygn** from the Section Profile menu will display a dialog window as shown at left.

For circles, you can enter either the Radius or Diameter (and the other corresponding value will be calculated and displayed).

For Regular Polygons, you can enter either the radius of the circle within which the polygon is inscribed, or else you can enter the length of a single side of the required polygon. In either case, the other corresponding value will be calculated and displayed (e.g. in the illustration at left, I have entered a side length of 500mm and the corresponding radius has been calculated).

For a Rectangle, both the width and height must be entered. A square can be defined by entering the same value for both, or alternately define a square using a 4 sided regular polygon.

Only one of the Complete, Top Half, Bottom Half, Left Half or Right Half radio buttons can be selected at a time.

2.1.1.4 Using the Previous Profile

If you have selected a profile in this same drawing previously, then you can select the same profile again by selecting the **F0 Prev Profile** option. This option is only available when you first enter the macro.

This option will not display if the previous profile was based on selection of entity(s) and the entities no longer exist, or if a symbol file was used and that file no longer exists. If previously selected entities have been modified, then the modified entities will be used to define the profile.

2.1.2 Saving a Profile

You can save a profile that has been created from entity(s) or with the Circle/Polygon option. The profile will be saved as a symbol.

Selecting the **Save as Symbol** button on the Section Profile window will open a File Dialog where you can specify the name and path for the saved symbol.

2.1.3 Normals

The polygons created by the macro will have their positive normal⁴ to the right hand side of the direction of the lines in your profile. Note that an entire chain of lines will take on the direction of the first line processed in the event that they do not all travel in the same direction (you can control this using colour – see [Section 2.1.4](#) below).

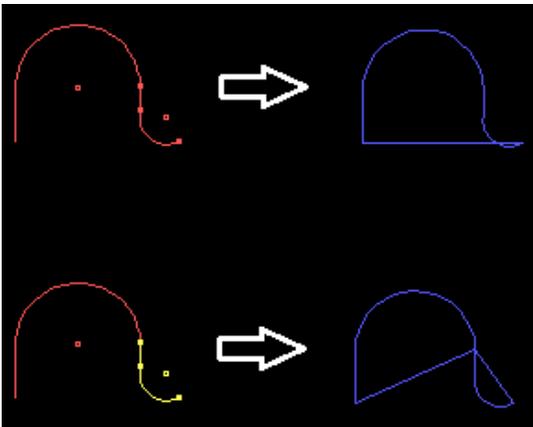
If a thickness is applied to the profile, the positive normal will always be the outside of the profile shape.

You can change the positive normal side of polygons if required after the extrusion is created using the **S6 FlipNrml** option on DataCAD's Polygon Edit menu.

2.1.4 Use of Colour in Profile Definition

The extrusion will always be created using the current colour. However, the colour of entities used to define the profile can affect the result if you are using either the cap or thickness options.

When you use entities to define your profile, entities that connect to each other will be 'chained together' if they are of the same colour. Entities of different colours will not be chained.



An example of the effect of colour on the cap is illustrated at left.

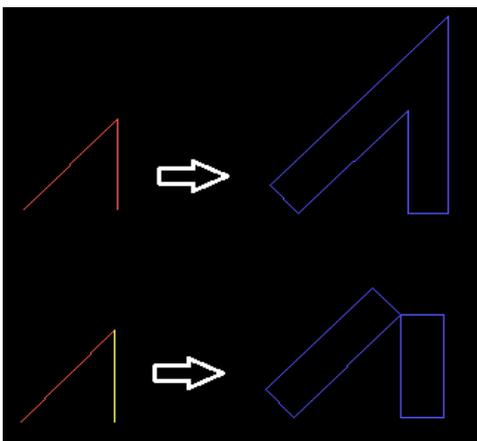
In both cases, the profile was defined using a line, an arc and a polyline. In the bottom case, the polyline is a different colour to the line and arc.

When the polyline is the same colour as the arc and line, a single cap is created (and in this case results in a cap polygon which crosses over itself).

When the polyline is a different colour, two separate caps are created (one for the line/arc, and a separate one for the polyline).

You could also use this feature to split the end cap into multiple parts when the number of points becomes an issue (max number of points for a single cap entity is 256 – see [Section 2.2.3.3](#))

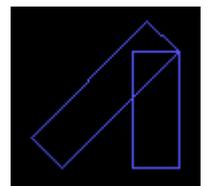
Different colours used in the profile definition can also affect the way that thickness is processed as shown below:



When the entities defining the profile are the same colour, they are treated as a continuous path that is offset to produce the thickness shown in the top example.

When the entities are a different colour, they are treated as discreet paths and offset individually to produce the thickness shown in the bottom example.

Note that the direction of the lines is also important: If the red and yellow lines in the bottom example at left went in opposite directions then the result may be as shown at right.

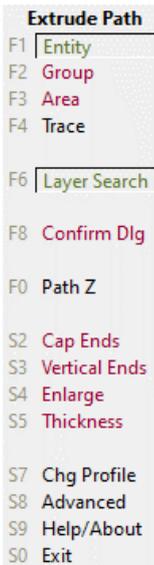


⁴ The positive normal is the face that will be rendered by default if your rendering program (such as the O2C Object Viewer) does not render both faces. In O2C this rendering behaviour is controlled by the material setting and the default when you create a new material is to render both sides, but materials can be defined without this option enabled in which case only the positive normal side will be rendered

2.1.5 Profile Divisions

Use the **S5 Advanced** option to specify the number of full-circle divisions that will be used by the macro if there are curves in a profile. See [Section 3.1](#) for further details.

2.2 Extrude Path



The Extrude Path menu will display after you select the profile.

You can select **S7 Chg Profile** to return to the Section Profile Menu.

2.2.1 Selecting the Path

Use **F1 Entity** / **F2 Group** / **F3 Area** options (in conjunction with **F6 Layer Search**) to select path entities using the normal DataCAD selection. The path will follow selected entities that are of suitable types.

Selecting **F4 Trace** will allow you to select points to define a path that is not necessarily related to any existing entities. Paths defined using the Trace option will always be horizontal with the Trace menu having options allowing you to select the Z of the extrusion.

2.2.2 Acceptable Path Entities

You may commonly use 2D lines/arcs and Polylines to define the path, but some other entity types are also suitable and will give varying results as described below. Curved paths are segmented into straight line segments when extruded by the macro. You can specify the number of full-circle divisions used (except for text) using the [Advanced Options](#) in the macro.

2.2.2.1 2D Lines/Arcs/Circles/Ellipses/Beziers/B-Splines

2D entities will be 'chained together' where they have the same Z-Base/Height and connect to each other at their endpoints. Chained lines will be processed as if they all travel in the same direction (even if, for example, two lines are joined at the starting point of each line).

Z-Base and Z-Height options will create extrusions at the Z-base and Z-Height of the selected entities.

If the Verticals option is used by itself, then vertical extrusions (between Z-Base and Z-Height) will be produced at each line end, or at intervals around curves according to the division setting.

If Verticals is used in conjunction with Z-Base and/or Z-Height, then vertical extrusions will only be produced at the endpoints of the lines or at the endpoints of the chain if multiple lines are chained together (or not at all if the chained entities form a closed shape).

2.2.2.2 PolyLines/Slabs/Cylinders/Blocks

These entities will be treated much like a closed chain of 2D lines or 2D arcs/circles, except that one side of the entity will be treated as Z-Base and the other side as Z-Height. As these entities can be rotated in 3D space, the face treated as Z-base may not always be the face with the lowest Z values.

2.2.2.3 3D Lines, Arcs and Polygons

Z-Base, Z-Height and Sloped Z settings will all produce the same result for 3D lines or Polygon paths. Verticals will not produce any extrusion. 3D lines are *not* chained together in the same way as 2D lines.

2.2.2.4 Cones

Z-Base setting will produce an extrusion around the circular base of the cone. Z-Height and Slope settings will not produce any extrusion.

Selecting Verticals on a cone path will create extrusions along lines from the tip of the cone to the end of each arc segment (as set by the divisions setting), or to the ends of the base arc only for a partial cone when used in conjunction with a Z-Base extrusion. No vertical extrusions will be produced if both Verticals and Z-Base are selected on a complete cone (as the base 'arc' is a full circle with no ends).

2.2.2.5 Text

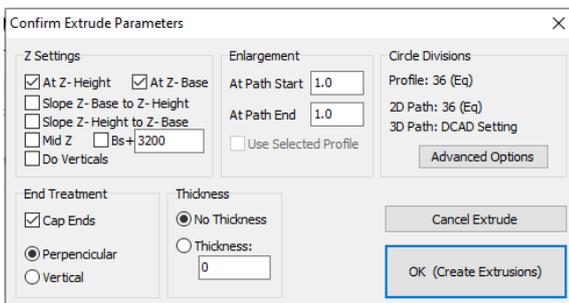
You can use text entities as a path. The macro uses default DataCAD settings when segmenting text curves and does not have control over the number of divisions used. Because of this the Divisions settings in the Advanced Options are not applicable to text entities (in my testing, shx fonts typically produced about 20 divisions for a capital 'O'; TTF fonts generally used many more divisions).

Note that some fonts may produce undesired results (particularly some of the 'bold' shx fonts that use multiple lines to create 'fill').



2.2.3 Extrude Path Options

2.2.3.1 Confirmation Dialog

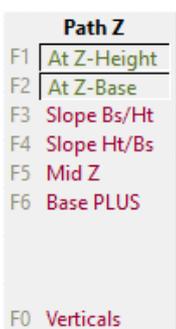


Enabling **F8 Confirm Dlg** will cause a Confirmation Dialog to display immediately before the extrusion is created.

You can use this dialog to adjust options if required.

For traced paths the Z Settings allow you to specify a specific Z only. For other paths the Z Settings are as illustrated at left.

2.2.3.2 Path Z



The **F0 Path Z** option displays the Path Z menu. The options referring to Base, Height or Mid refer to the Z values of the selected path entities (not the current Z settings) and are fairly self-explanatory for 2D lines/curves used to define the path. The 'Mid Z' option will create an extrusion midway between the Z-Base and Z-Height of the path entity, whilst 'Base PLUS' will create the extrusion at a specified distance above the Z-Base of the path entity. The 'Verticals' option is perhaps a little less obvious and is explained in a little more detail in the sections below.

See notes on the various path entities ([Section 2.2.2](#) above) for details about how the various Z options work with different type of entities.

Note that some of these options can be selected together whilst the Slope options can only be selected individually. For example, if you selected both Z-Height and Z-Base options for a path defined by 2D lines then two separate extrusions would be created (one at Z-Height of the lines, one at Z-Base). If you selected

Base, Height, *and* Verticals for a single 2D line path, then a single extrusion would be created that wrapped around the four sides of the line.

2.2.3.2.1 Verticals

Select **F0 Verticals** by itself to create extrusions along the ‘vertical’ edges of selected objects. The exact application will depend on the type of entity selected to define the path. As an example, if you select a horizontal slab entity, then using this option would create a vertical extrusion at each of the corners of the slab (note that if this same slab had been rotated in 3D, then these same corner edges would be used to define the extrusion even though they may not in fact be vertical).

For circles, arcs, and ellipses, vertical extrusions will be created at the end of each division (based on the path divisions setting in the Advanced Options (see [Section 3.1](#)). The cyan extrusions illustrated at right were created using a cylinder path with 12 equal divisions set.

For polylines, vertical extrusions will be created at the vertices (and not along curves regardless of the number of divisions specified). The vertical extrusions will be oriented according to the directions of the adjacent sides.

2.2.3.2.2 Verticals In Conjunction with Z-Base and/or Z-Height

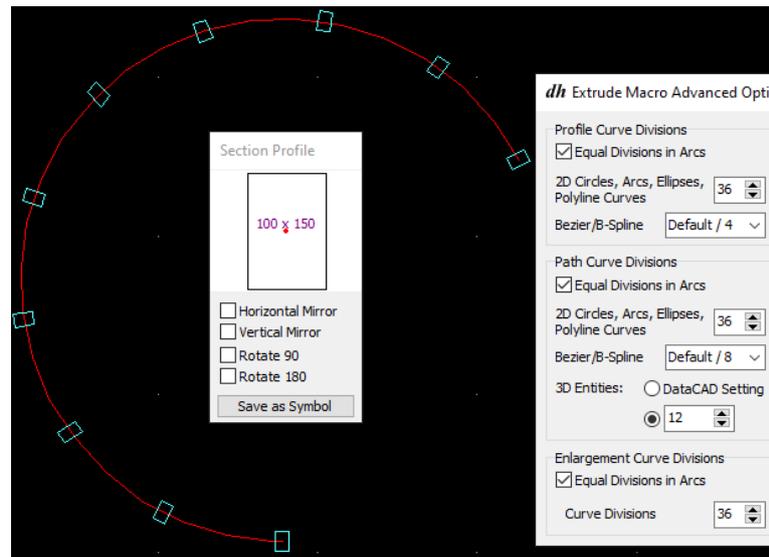
If the Verticals setting is used in conjunction with Z-Base and/or Z-Height, then vertical extrusions will only be created at the ends of the path. If the path is closed (and therefore does not have any ends), no vertical extrusions will be created (even though Verticals is selected).

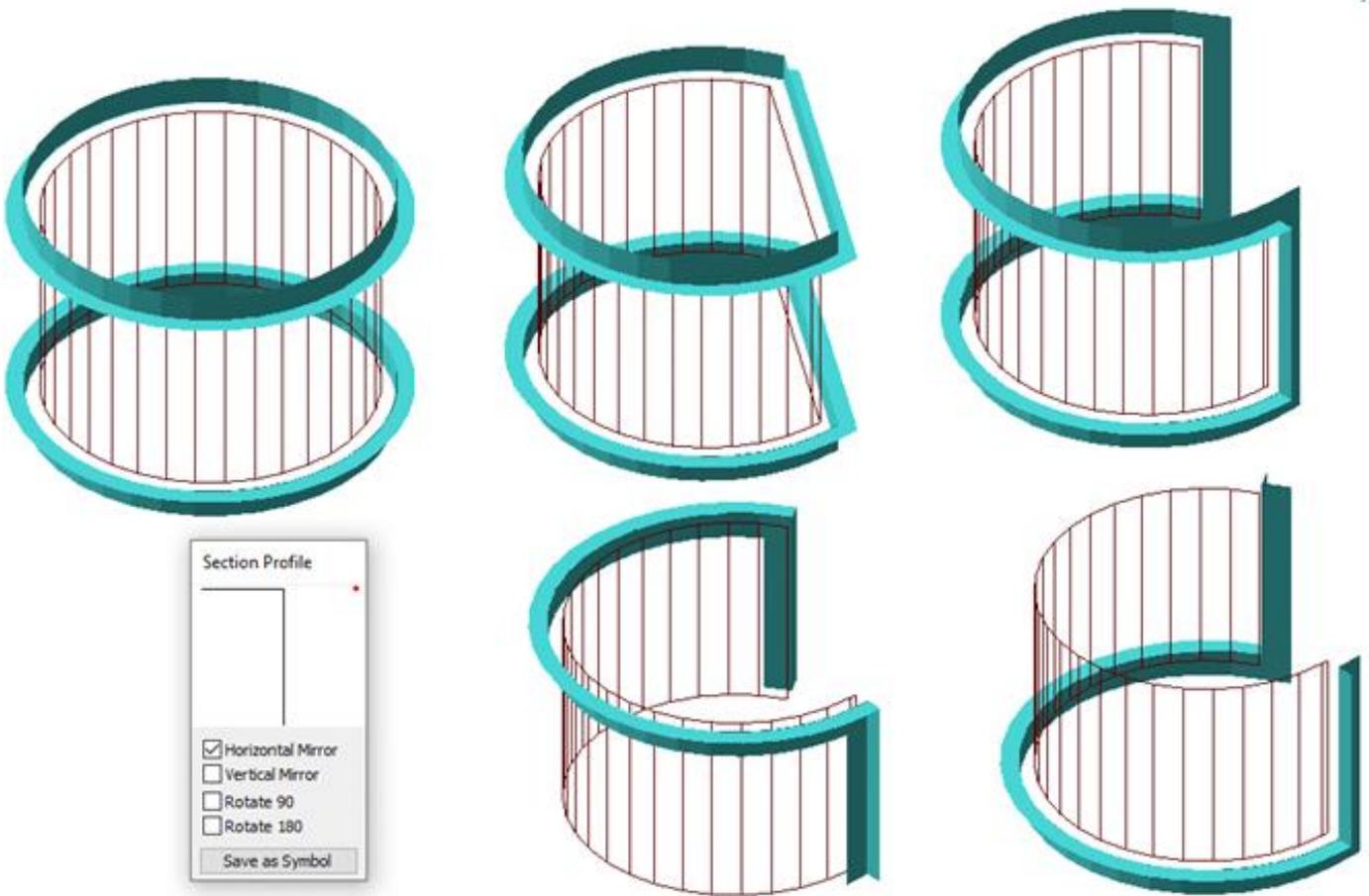
If both Z-Base and Z-Height are selected together with Verticals, then the extrusion will be wrapped around from the Z-Base to the Vertical and then to the Z-Height. This results in the extrusion at Z-height being upside down. (Note that even the path is closed then the Z-Height extrusion will be upside down even though no Vertical extrusion is created).

If only one of Z-Base and Z-Height is used in conjunction with Verticals, the horizontal extrusion will be the correct way up but will wrap around to the vertical extrusions.

The illustration below shows cylinder entities (illustrated as wire frame) that have been used as the path to create the cyan extrusions using Z settings as follows:

- Z-Base, Z-Height and Verticals setting were all turned on for the top 3 extrusions. No verticals were created for the first 2 as they were closed paths (the first being a complete cylinder and the next being a partial closed cylinder).
- Verticals were created for the last cylinder on the top row because it was not a closed path (partial cylinder, not closed).
- Z-Height and Verticals was used for the partial (not closed) cylinder on the left of the bottom row.
- Z-Base and Verticals was used for the partial (not closed) cylinder on the right of the bottom row.





Note that any *enlargement settings will be ignored* if Verticals are specified in conjunction with either Z-Base or Z-Height (enlargement settings *are* used if Verticals setting is used alone). If Z-Height and Z-Base are selected together (without Verticals) then two extrusions will be produced (both will be oriented the same way up).

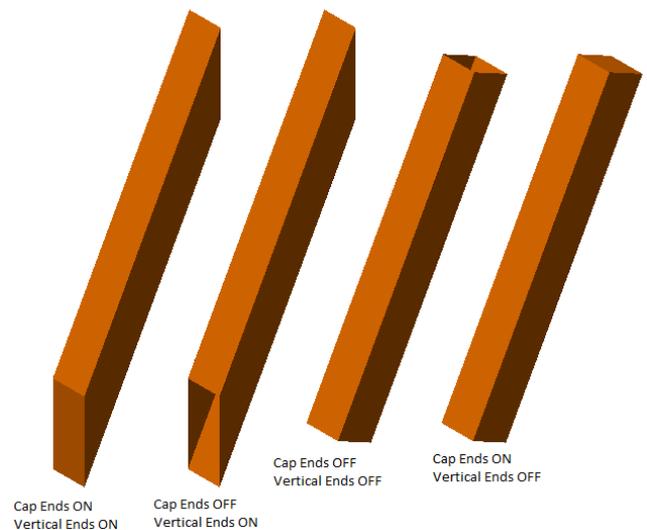
2.2.3.3 Cap Ends

If **S2 Cap Ends** option is enabled additional polygons will be created to cover each end of the extrusion. Results of using this setting are illustrated at right.

Note that end caps will not be created if the profile has more than 256 points⁵ (including points resulting from curves being segmented).

2.2.3.4 Vertical Ends

This option is applicable only when Tracing or using 2D entities to define the path. It will cause the ends of sloped extrusions to be modelled vertically. If this option is disabled, then ends will be modelled square to the direction of the extrusion.



⁵ If a thickness is specified, then end caps may not be created if the profile has more than 128 points.

2.2.3.5 Enlarge

Enlargement	
F1	Enlrgmt Strt
F2	Enlrgmt End
F3	Set Both
F5	Enlrgmt Pfl
F8	Reset

The Enlargement menu allows you to specify how (and if) the extrusion profile will be enlarged (or reduced) along the path.

You can specify different enlargements at the start and end of the path using the **F1 Enlrgmt Strt** and **F2 Enlrgmt End** options, or use **F3 Set Both** to set both starting and ending enlargements to the same value.

Note that any enlargement settings will be ignored if you have Z settings that include Verticals combined with either Z-Base or Z-Height (see section [2.2.3.2](#)).

2.2.3.5.1 Enlargement Profile

Use **F5 Enlrgmt Pfl** to set a non-linear enlargement along the path..

Selecting this option will prompt you to select a polyline to define the enlargement. You should select a polyline that travels in one direction along the x-axis and which has only positive y values.

After selecting a suitable polyline you will be prompted to select a point representing an enlargement factor of 1. Only the y value of the selected point is important. The y value of all the points along the selected polyline will be divided by the y value of the point to determine the enlargement factor.

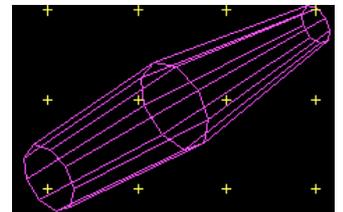


Once you have selected a polyline and point, an Enlargement Profile window will be displayed similar to the one shown at left.

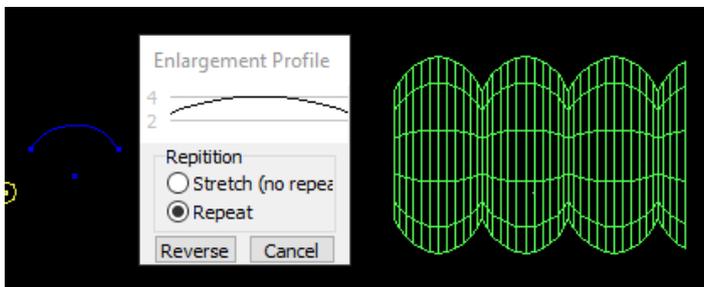
In this example I selected the green polyline shown and snapped to the left end of the line to define the enlargement factor of 1.

Using the above enlargement profile with an octagonal section profile produced the result shown in isometric view at right.

This example uses a very simple polyline with the Stretch (no repeat) option when applying the enlargement profile. You can of course use more complex polylines including curves. The Repeat option detailed below is only available when the starting and ending enlargement factors are the same.



If the repeat option is used the length of each repetition will be the same as the x-axis length of the selected enlargement polyline. A partial repetition will result if the length of the path is not an exact multiple of the enlargement profile length.



In the example at left, the blue polyline was selected as the enlargement profile and the centre of the curve was selected as the unity enlargement point. The green extrusion was produced using a dodecagon section profile⁶.

The aspect ratio of the Enlargement Profile diagram will change as the window is resized.

⁶ Any section profile shape can be used in conjunction with an enlargement profile, and curved paths can be used. Note that for a circular section profile along a straight path it would likely be a better solution to use of a Surface of Revolution entity.

2.2.3.6 Thickness

This option allows you to define a thickness that will be applied to section profiles.

For Circle/Polygon profiles, the thickness is defined towards the inside of the shape. For profiles defined by selecting entities, the thickness is defined towards the left hand side of the line. If you wish the thickness to be in the other direction, specify a negative distance.

Note that lines which connect to each other at their endpoints will be 'chained together' by the macro if they are the same colour. Where these lines go in different directions (e.g. the first point of each line is the same and they head away from each other towards the next point), the macro will use the direction of the first line it finds. If you selected by group, then the first line will be the one you selected to specify the group. However, if you selected by area, you cannot guarantee which will be the first line, but you can always specify a negative value if the thickness goes the wrong way.

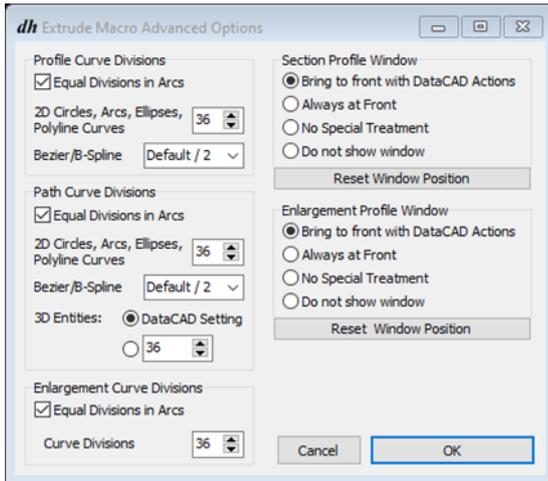
The **S4 Thickness** button will display red text if a thickness of zero has been specified or green text if a non-zero value has been entered.

Note that you may get unexpected (and undesired) results for thickness where the profile contains indents that are less than twice the thickness wide. The thickness is displayed in the Profile Window so that you can see what the result can be before actually creating the extrusion.

2.2.4 Path Divisions

The macro will segment any curves included in the extrusion path. The number of divisions can be specified in the Advanced Options dialog (see [Section 3.1](#) below).

3 Advanced Options



Selecting **S8 Advanced** from the Section Profile or Extrude Path menu will open the Advanced Options form shown at left.

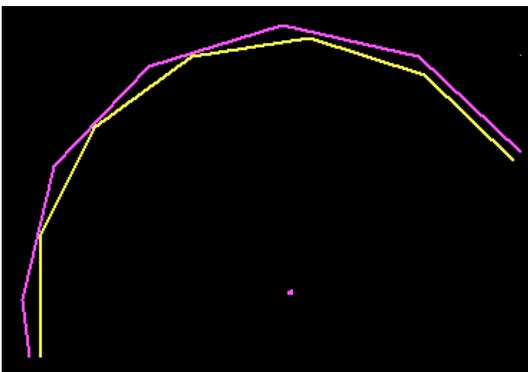
This allows you to specify the number of full circle divisions that will be used by the macro as it processes various type of curves. It also includes options for how the Section and Enlargement Profile windows are displayed.

3.1 Curve Divisions

Settings made to the curve divisions settings will be saved in the current drawing, but future drawings will revert to the default values. If you wish to save different defaults for new drawings you can invoke the macro in your default drawing and save the required settings there.

Note that the curve divisions specified do not apply to text entities used as the path (see [Section 2.2.2.5](#)).

3.1.1 Equal Divisions In Arcs



DataCAD uses a curve division size that is the same for full circles as for arcs. This results in a ‘partial division’ at the end of any arc that is not an exact multiple of the division size as illustrated by the purple arc shown at left (which uses 12 full-circle divisions, but the angle spanned by the arc is not a multiple of 30 degrees).

If the Equal Division checkbox *is not* checked, then the macro will calculate curves using the same method as DataCAD.

If the Equal Divisions checkbox *is* checked the size of each division will be adjusted so that all are the same length (as illustrated by the yellow arc).

3.1.2 Circle, Arc and Ellipse Divisions

Specify a value between 6 and 128 (inclusive) that will be used for 2D Circles, Arcs, Ellipses and Polyline Curves. Separate values are specified for curves in Section Profiles, Paths, and Enlargement Profiles.

3.1.3 Bezier/B-Spline

The macro uses DataCAD to calculate Bezier and B-Spline curves and the number of divisions is not based on a specific full-circle number. For less segments, you can instruct the macro not to use all the points returned by DataCAD (e.g. if you select ‘Default/3’ then the macro will use every third point). Separate values are specified for curves in Section Profiles and Paths.

3.1.4 3D Entities

3D entities (cylinders, cones, truncated cones) can be used to specify an extrusion path, and you can select to either specify a number of divisions for these or to use the divisions being used by DataCAD. Note that if you select to use the DataCAD Setting then equal division will not be used (regardless of the setting of the Equal Divisions checkbox).

You can select a specific number of divisions between 6 and 128 (inclusive) in which case the Equal Division setting is used when calculating paths.

3.2 Section and Enlargement Window Options

The Reset Window Position buttons will take effect immediately. Changes to other options may not take place immediately. It is recommended that you exit the macro and restart it after making changes to these options.

Changed options will have effect in subsequent drawings (unless a different option has previously been selected in other drawings in which case the setting in the drawing will be used).

3.2.1 Bring to front with DataCAD Actions

This option is the default when the macro is installed. The window can be hidden behind other windows (for example if you leave DataCAD to use another application whilst the macro is running), but will automatically be brought back to the front when you perform some action in the macro (i.e. select a menu option or click in the drawing area). Note that simply changing DataCAD to the active application may not bring the windows to the front.

3.2.2 Always at Front

The windows will remain in front even if you use another application.

3.2.3 No Special Treatment

The window may be hidden behind other applications. If the window position is within the DataCAD window it will be hidden behind DataCAD when using the macro, but you can bring it to the front using normal Windows methods such as Alt-Tab.

3.2.4 Do not show Window

The window will not be displayed at all. The macro will still function normally, but you will need to remember the profile that you have chosen and will not be able to change options (such as mirror profile or repeat enlargement) that are included on the windows.

3.2.5 Reset Window Position

You can manually drag the Profile windows to any position on your monitor display and it should not normally be necessary to use the Reset Window Position buttons.

The most likely scenario when you will need to use these buttons is if you have changed your monitor setup (e.g. you may have removed a second monitor). In this case it is possible that the Profile windows may be positioned outside the display area. Clicking on the Reset Window Position button will reposition the appropriate Profile Window to the top left corner of your primary monitor (from where you can drag it to your desired position).

4 Known Issues and Bugs

4.1 Possible Crash

There is an issue in DataCAD that potentially allows you to select another macro whilst the Extrude macro is still running, and doing so can cause DataCAD to crash. This has been fixed in versions of DataCAD from 21.01.04.01 onwards.

The possibility of the crash can be avoided by selecting '[Do Not Show Window](#)' in the Advanced Options for both the Section and Enlargement Profile Windows⁷.

Whilst I expect that it would not be common for you to invoke another macro whilst using Extrude, you should consciously avoid doing so if you are using a version of DataCAD prior to 21.01.04.01.

5 Future Enhancements

I do intend to continue to develop this macro, and have a few new features in mind for future versions. If you have particular comments about the macro or suggestions for future enhancements, I would be pleased to receive an email from you at dhsoftware1@gmail.com.

To ensure that you are advised about future updates to the macro please [register your use of it at my web site](#).

⁷ Testing has shown that the potential crash is avoided if you select this option, but there may still be other (less serious but none-the-less undesirable) symptoms such as memory leakage if you select another macro whilst Extrude is still running.