

DSECT-Developed section 1.8

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DSECT is an application for AUTOCAD or BricsCAD, which determines the developed (unwrapped) section of one set of 3DSOLID, 3DFACE or 3DMESH entities and vertical planes passing through a 2D POLYLINE which may contain arcs, in XOY plane. Starting with AutoCAD 2010, the 3DMESH entities must be created using 0 value for the MESHTYPE variable.

Developed section is generated in the vertical plane passing through first segment of POLYLINE, in the "dsect" layer. The intersection is generated in the form of 3DPOLY entities, colored from 1 to 1, beginning with the color 1, corresponding to every vertical plane.

Two viewports for printing are created, with the dimensions specified by the user. The first viewport contains the initial view and the second contains only the section, in the unwrapped plan.

No matter the language of AutoCAD. DSECT works, but his commands remain in English.

INSTALLATION

Always download the latest shareware version from www.rcad.eu (not from other websites)!

The shareware version runs only 8 times and only the first 3 segments of POLYLINE will be interpreted at DSECT command.

Set OFF "Create viewport for new layouts":

- in BricsCAD:
 - CREATEVIEWPORTS, Off or
 - Settings, "Viewports,layouts and tabs", "Automatic viewport creation", Off,
- in AutoCAD : Tools, Options, Display, Create Viewport in new layout, Off.

Follow these steps:

1) Unpack DSECT.ZIP in the directory "C:\DSECT".

The folder "c:\DSECT" is mandatory!

2) in AutoCAD

- starting with Autocad 2014, set SECURELOAD variable to 0 or TRUSTEDPATHS to c:\DSECT

- launch the menu function:

```
Tools
  Load Application
    Startup Suite
      Add
        C:/DSECT/DSECT.VLX
```

in BricsCAD

- launch the menu function (ori APPLOAD command):

```
Tools
  Load Application
    Add
      C:/DSECT/DSECT.DES
    Load
```

Available commands:

DSECT - determines the developed section of one set of 3DSOLID, 3DFACE or 3DMESH entities,
DSECT_CP - sets the "concatenation precision" (default 0.000001) of the segments of intersection,
DTP - generates drawings in PDF format, for several rectangular areas from DWG.

LAUNCHING DSECT command

Within an AutoCAD or BricsCAD session the **DSECT** command will be introduced first. Then, it will appear the question:

"Select POLYLINE in XOY plane containing only lines",

the user will have to select a 2D POLYLINE in XOY plane.

The following question will be:

"Developed

section(Yes,No,Select,Length,chord_arc_Deviation,minimum_Arc_step,Format)<Yes>"

The user will have to enter:

- "Yes" for developed section or "No" for nondeveloped section
 - "Select" if you want to explicitly select 3DSOLID, 3DFACE or 3DMESH entities; implicitly are selected the entities passing through POLYLINE (in the sense of "Fence" selection from AutoCAD)
 - "Length" (implicitly 0.05) is the segment length for the approximation of ELLIPSE, ARC, CIRCLE or SPLINE entities resulting from the section of 3DSOLID entities; if the execution time is too long, augment "Length"!
 - "chord_arc_Deviation" - admitted chord-arc deviation, whence arises the number of the segments on the arcs
 - "minimum_Arc_step" - minimum length of the segments which approximates the arcs; the length of the segments generated on the arcs will be the maximum of "minimum_Arc_step" and the length resulting from "chord_arc_Deviation"
 - "Format" - the dimensions on X and on Y of the two viewports for printing that will be created; the first viewport contains the initial view and the second contains only the section, in the unwrapped plan; When calculating the scale, the unit in model space is considered [m] and in paper space is considered [mm].
- The settings are valid only for the current session!

There is also the DSECT_CP command that you can set "concatenation precision" (default 0.000001) of the segments of intersection. Two end points of segments are considered identical if the distance between them is less than "concatenation precision"! It is a good idea to enter higher values than the default value when you intersect 3DSOLID entities or 3DFACE entities that are not joined perfectly on their edges! The setting is valid only for the current session!

The AutoCAD message "Object isn't that long" is warning!

"Modeling Operation Error.Inconsistent containment of intersection curve.SECTION failed for selected solid."

or "Cannot explode REGION" are AutoCAD errors! In this case the developed section can be incomplete!

DTP Command

The command generates drawings in PDF format, for several rectangular areas from DWG.

Without this new command you would have wasted time, manually entering the PLOT command and its options, in repeatedly.

The DWG must contain several areas to be printed in Paper Space, as in the example "c:\dsect\DWG to PDF example.dwg". In this example, there are several Viewport entities, through which different areas of the Model Space can be seen. You can also use the DTP command if you drew directly in Paper Space. The drawing unit must be millimeter.

First you will indicate the lower-left point of the first area to be listed, in response to "The lower-left point of the first format".

Then you can opt for:

- F (or Format), by which you indicate the dimensions of a rectangular area, in the X and then Y direction,
- P (or Printer), by which you indicate the name of the printer driver in PDF format, which you can find in the PLOT command of AutoCAD (or BricsCAD) and then in Printer/plotter; for AutoCAD 2022 it is for example "AutoCAD PDF (General Documentation)", and for BricsCAD V22 it is "Print as PDF.pc3" (the name is entered without the characters "),
- A (or pAper) by which you indicate the name of the paper size, which you can find under Paper Size in the PLOT command; for AutoCAD 2022 it is for example "ISO full bleed A3 (420.00 x 297.00 MM)", and for BricsCAD V22 it is also "ISO full bleed A3 (420.00 x 297.00 MM)" (the name is entered without the " characters); the Paper size will be correlated with the dimensions from Format (printing will be done on a 1:1 scale);
- D (or format-Distance) which indicates the distance between the rectangular areas, on X and Y; the distance is 5 in the example "c:\dsect\DWG to PDF example.dwg",
- O (or plot-Offset) indicating the offset on X and Y from the PLOT command; 2 values will be entered separated by a comma, as if you were entering a point.

The values set above will be saved in a file with the name and in the folder of the DWG file and the extension "var". When you want to print the DWG again, the values from this file will be loaded automatically.

You can then choose to print several rectangular areas in the X direction, if you indicate a number of rectangular areas.

If you opt for W (or Window), you can indicate a number of rectangular areas on the X direction (Number of drawings on X) and then a number of rectangular areas on the Y direction (Number of drawings on Y). In this case, (Number of drawings on X)*(Number of drawings on Y) rectangular areas will be printed.

After printing, a line will be temporarily drawn on the screen, along the diagonal of each printed rectangular area.

For each drawing, a PDF file will be generated, having the name and folder of the DWG file, to which a number has been added, starting with 1. It is possible that these PDF files are automatically loaded in Acrobat Reader. In this case, do not forget to close Acrobat Reader when you launch the DTP command again.

When printing, "plot styles" are not taken into account, but "object lineweights" will be taken into account.