

## DSECT-Developed section 1.7

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DSECT is an application for AUTOCAD 2002-2018 or BricsCAD V15-V17, 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](http://www.rcad.eu) (not from other websites)!

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:

  - Tools

    - Load Application

    - Add

    - C:/DSECT/DSECT.DES

    - Load

The shareware version (of trial) of the DSECT program has the limitations:

- run only 4 times,
- the first 3 segments of POLYLINE will be interpreted.

Available commands:

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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

## LAUNCHING -----

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!

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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!

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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!