TRIANGULATION for BricsCAD - 2.3

TRIANGULATION for BricsCAD is an application for BricsCAD V14-V20 , which makes the triangulation of a set of POINT entities, the intersection curves (isolines) between a set of 3DFACE entities and a set of equidistance plans, horizontally or vertically and the volume and center of gravity of a set of bodies or between surfaces composed of 3DFACE entities. You can load and draw a file of points having the format: Number X Y Z Code. You can also make the projection over a triangulation of a 2D POLYLINE and generate cross sections and a longitudinal profile corresponding at projection. The POLYLINE may contain arcs.
In addition, you can draw a DWG file in Google Earth by generating a KML file type. The DWG file can be in any projected coordinate system.
It is also possible to determine the flatness of the surfaces and you can import and export LandXML file type.
No matter the language of BricsCAD. *TRIANGULATION for BricsCAD* works, but his commands remains in English. But there are user guides for different languages, in PDF format.

You can also try our RTOPO program (see www.rcad.eu\rtopo) running under Windows 98/XP/VISTA/7/8/10 ! This program has many other functions (for example: several formats for loading of points, the intersection of two triangulations, **the interactive modification of the points** (modify, delete or add); you can modify the coordinates, the code and the position (in 3DPOLY or SPLINE), the interpolation of a set of 3DFACE entities, using quartic triangular patches with G1 continuity, its own CAD kernel, motion path animation).
Often it is necessary to edit interactive the points and the curves (3DPOLY or SPLINE). This editing can not be done in TRIANGULATION for BricsCAD, but can be done in RTOPO! Therefore we recommend the purchasing of both programs.

Installation

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

You must have:
- computer administrator account,
- UAC disable (Start, Control Panel, User Accounts, Change User Account Control Settings, Never notify, then restart the computer).
Run BricsCAD as an administrator (Right mouse button on BricsCAD icon, Properties, Compatibility, Run this program as an administrator).

Follow these steps:
1) Unpack A_TRIANG.ZIP in the directory "C:\A_TRIANG".
2) Enter in BricsCAD as an administrator (Right mouse button on bricscad.exe, Properties, Compatibility, Run this program as an administrator)
  - launch the menu function:
   Tools
   Load Application
   Add
     c:\a_triang\a_triang.des
   Load
3) Enter **TRIA** command and choose from the displayed options.

The shareware version:
- runs only 18 times
- only 75000 triangles are drawn at triangulation,
- the calculated volume is not displayed in clear,
- only 9 profiles are drawn at PROFILE command,
- only 25% of what is displayed on the screen will be generated in the file of KML type,
- only 1500 points are loaded from a LandXML file,
- only 1500 points and 1500 triangles (3DFace) will be saved in a LandXML file.

Loadp

LOADP option - loads and draws a XYZ file of points having the format:
Number X Y Z Code
for example:

```
1238  420385.445  658974.376  455.38  AX01
```

The separating character of the fields may be: space, tab or ", " .
The accepted formats are:

```
point_number X Y Z Code
X Y Z Code   (point_number will be 990)
X Y Z        (point_number will be 991 and Code will be "L")
X Y          (point_number will be 992, Z will be 0 and Code will be "L")
```

“Code” contains 2 parts: the first part (code_name) contains only letters. The second part is an integer number.
POINT, TEXT, 3DPOLY and BLOCK entities will be generated, according to the Loadp-draw variable.
The texts height and the points size are established by iso-Htext variable.
LOADP_DRAW variable must be setted in advance and may have the values:

none=0, number=1, code=2, Z=4, treating=8 and any sum of the previous values.

For example, to draw number and code LOADP_DRAW will be 3 (1+2).
The number, code and Z will be drawn as TEXT entities. In the points X,Y,Z are drawn POINT entities having "pdsize"=ISO_HTEXT and according to "pdmode" (BricsCAD variable).

If LOADP_DRAW also contains 8 (treating), the points having the same code (letters+number) will be joined by 3DPOLY or drawn by a block, according to the “points_treating.ini” file.
“Points treating.ini” file has items of the form:

code_name	treating	description	color	lineweight	linetype

for example:

AR	3dpoly	arable	3	0.00	DASHED

"Treating" can be: "3dpoly" , "spline" (3dpoly interpolated), "symbol" (block) or "nothing".
The “Symbol” is drawn with a block defined in the “rtopo_simbol.dxf” file. You can modify a block from this file using BricsCAD commands (OPEN file, BEDIT and SAVE)! The modified block must have the gauge 1 (one unit)! The blocks will be drawn with the ISO_HTEXT scale .

If you have many points, set LOADP_DRAW to 0 and PDMODE to 0 or 2! In this case, LOADP will run faster.

BricsCAD doesn't draw "linetype" for "3dpoly" and "spline" entities!
“Points treating.ini” file can be edited with any text editor.

A grid of axes is also drawn. The axes steps may be established by the variables X-axis-step or Y-axis-step. If a value is 0, the axes will not be drawn!
The points that have the code defined in Numbered-codes are numbered, starting with 1.

Intermediary

Generates intermediary points on the selected “3dpoly” and “spline” entities, according to “intermediary points distance”. This aids the triangulation!

“Intermediary points distance” must be the minimum distance between 2 “3dpoly” or “spline” entities. “3dpoly” and “spline” entities are actually break lines for triangulation.

Triangulation

It will triangulate a set of selected POINT entities.
It will launch first a fast triangulation. It can work with a a relative uniform distribution of the points, but not with a constant step. If it doesn't work for the given set of the points, it will resume automatically using a normal

triangulation. If the normal triangulation will fail, the triangulation will resume with total checks (which will take longer).

The triangulation is performed on a convex hull of a set of points.
3DFACE entities are generated in the TRIANG layer.
You can triangulate million points!
You can aid the triangulation, using Intermediary option.

You can also simulate the break lines in the following way:
- define SPLINE or 3DPOLY entities, passing through existing points (using OSNAP NODE)
- generate additional points on SPLINE and 3DPOLY using MEASURE command with a step smaller than the smallest distance between the existing points
- enter now Triangulation option.

ISolines

ISolines generates the intersection of a selected 3DFACE set with horizontal plans. POLYLINE entities, optional interpolated, will be generated.
Possible options:
1) Yes - intersection with several parallel and equidistant plans, introduced by the minimum and maximum position of the plans and the distance between them; in order to introduce an exact quota (Z), write the coordinates of a point, Y beeing actually Z; if you wish, for example, level curves between the Z quotas 690 and 730, introduce the points 0,690 and 0,730 (value of the fist coordinate (X) is not liable to interpretation); the points (0,zmin) and (0,zmax) are implicitly displayed
2) No - intersection with a single plane, one point crossed by the plan will be mentioned
3) iso-mDist - the isolines with Z a multiple of "iso-mDist" will be drawn thickened according to the value of "iso-Width"
4) iso-Width - the thickness of isolines having Z a multiple of "iso-mDist"; is available if you choose to interpolate
5) iso-Htext - the height of the Z-texts attached to isolines
6) iso-Intseg - the interpolation segment length if you choose to join and interpolate the segments of intersection; if the time of interpolation is too large enter a higher value
7) CUrvature - represents percents with possible values between 5-100, used to establish the curvature of the interpolated lines; a smaller "CUrvature" represents a larger flattening
8) CLipint - if is 1, an intersection clipping will be operated on the rectangle used for the selection of the 3DFACE entities; if is 0 no clipping will be operated
Finally, the user will have to select entities of 3DFACE type, by a rectangle ("First corner") or by "All" option.
You can still opt for the joining of the segments. If you answered yes, you can opt for the interpolation of the polylines. The interpolated polylines pass through the points of intersection between plans and the 3DFACE entities.
Each curve is located in a LAYER holding in denomination the Z expressed in 0.001 units. Their colors are according to a drawn legend depending on Z. These colors may be altered, by the following BricsCAD command:

(setq cul_leg (list 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15))

introduced in the "acad.lsp" file. Following "list", 15 integer numbers must be introduced, representing colors from Zmax to Zmin.

VIntersection

VIntersection generates the intersection of a selected 3DFACE entities with vertical plans. POLYLINE entities, optional interpolated, will be generated.

Previously, the "cutting line" through 2 points will be specified. Intersection plans will be parallel to a vertical plan crossing the "cutting line". Next dialogue is similar to the ISolines dialogue. Furthermore, under the name VERT, is saved the UCS with a XOY in the vertical plan crossing the "cutting line". In order to visualize the curves along of a direction perpendicular at the intersection plans, the following BricsCAD commands may be operated: UCS, RESTORE, VERT, PLAN and then press Enter.

Color-filled: color-filled contour map

Color-filled intersects and divides a set of 3DFACE entities with 14 equidistant and horizontal plans.
The intersected 3DFACE entities will be divided along the line of the intersection.

The 3DFACE entities that are located between two neighboring planes will receive the same color.
It will result 15 colored areas according a color legend (as in ISolines command) depending on Z.
Will be introduced first the points (0, Zmin) and (0, zmax). Between these will be made the division in 15 areas. The step of the plans will be (Zmax-Zmin)/15.
Finally, the user will have to select entities of 3DFACE type.
After the execution of the option you can launch SPLFRAME (0 or 1) followed by REGEN and then SHADE or 3DORBIT commands!

Profile

It generates cross sections and a longitudinal profile for:
- the projection over a triangulation of a 2D POLYLINE or
- of a "3dpoly" or "spline" entity having a given "Code" (see Loadp).
The 2D POLYLINE may contain arcs.

Possible options:
- "**Line_step**" - the step used for the generation of points over a line segment of the 2D POLYLINE
- "**chord_arc_Deviation**" - admitted chord-arc deviation, whence arise the number of the points on the arcs
- "**minimum_Arc_step**" - the minimum distance between 2 consecutive points generated on the arcs

The distance between 2 consecutive points generated on an arc of the 2D POLYLINE will be the maximum of "minimum_Arc_step" and the step resulting from "chord_arc_Deviation".

- "**longitudinal_y_Scale**" - y-axis (actually Z) scale of the longitudinal profile (e.g. 10 means 10:1 scale); x-axis scale is always 1:1
- "**sec-Htext**" - the height of the texts at projection, cross sections and longitudinal profile

"Line_step", "chord_arc_Deviation" and "minimum_Arc_step" will not be used if you choose "Code" option. In this case will be generated cross sections just in the points having the given "Code". It must have been previously loaded some points with the Loadp option.

Finally you will have to enter the left and right perpendicular distance in every points generated on POLYLINE. Up to these distance it will be generated the cross sections (at 1:1 scale).

In the longitudinal profile, the maximum slope is colored red.

Volume

Volume calculates the volume and center of gravity of a set of bodies or between surfaces composed of 3dface entities.

First, the user will have to select 3DFACE entities. The end of selection is indicated by pressing ENTER.

You must enter then the volume increment. The recommended value for 100 MB of used memory will be displayed.

The calculation of the volume is based on the intersection with vertical lines of the rectangular gauge (in XOY plane) of the set of 3DFACE entities. The step on X and Y of these line is the increment of volume. The volume will be considered for a line if there is an even number (2, 4, 6 ...) of intersections. The total volume will be approximated with the sum of the parallelepipedic volumes (with a square base having "volume increment" as a side) between the intersections 1-2, 3-4 and so on, upwards of Z of each line. So, the bodies may have holes! If on a line exists only one intersection (or none) is not considered a volume on that line. The vertical 3DFACE entities are not taken into account (they may not exist) when calculating the volume!

If you have a surface consisting of 3DFACE entities for which you want to calculate the volume to a plane of given Z, describe manually a 3DFACE entity (higher than the XOY gauge of the set of 3DFACE entities), its points having desired Z.

The volume and the coordinates of center of gravity will be displayed. In the center of gravity will be generated a POINT entity.

Redraw

It redraws the points already loaded with "Loadp", according to the setting of the LOADP_DRAW variable. Preliminarily, all entities will be erased.

Settings

With it you can set:
- **X-axis-step**: step on X of the vertical coordinate axes
- **Y-axis-step**: step on Y of the horizontal coordinate axes; If X-axis-step or Y-axis-step is 0, the vertical and horizontal axes will not be drawn!
- **Loadp-draw**: determines what is drawn at "Loadp" or "Redraw"; it have the values:
  none=0, number=1, code=2, Z=4, treating=8 and any sum of the previous values;
for example, to draw number and code LOADP_DRAW will be 3 (1 +2)
- **Numbered-codes:** will be numbered starting with 1, the points having the codes introduced here (separated by space)
- **iso-Htext:** sets the texts height, points size and block size at "Loadp" and texts height at "ISolines".
- **iso-Intseg** - the interpolation segment length for the options: "Loadp" (for "spline" treating), "ISolines" or "VIntersection".

- **Kml** - is used to generate a file of KML extension, the interface with Google Earth; it can have the values 1, 2 or 3:
  1) for **kml = 1,** it is considered that: X and Y coordinates are in [m] and they can be in any projection system, X increases to the right and Y to upward, X is oriented to east and Y to north;
    - you will be asked to define a base point in your system (possibly using Osnap interactive options) and then, its equivalent in Google Earth; this definition should be made only once, for your local coordinate system;
    - in Google Earth, you must first set the displaying of lat / lon in decimal degrees (in menu: Tools, Options, 3D View, Show Lat/Long, Decimal Degrees);
    - then, select (in Google Earth) Add, Placemark and put a Placemark on the map, at the corresponding base point from DWG; in the displayed window, you must copy the Longitude and Latitude (one at a time) and then, insert them in BricsCAD (with Paste), separated by ","; the south latitude and west longitude must be negative;
    - the base point will be used to align the DWG in Google Earth; the base point may be the corner of a building, the center of a crossroads or any point in DWG, for which you can find the corresponding point in Google Earth;
    - at the coordinates from DWG, DX-kml, DY-kml and DZ-kml will be added and then, the coordinates will be rotated with DA-kml (see below);
    - the XY plan of the DWG is considered tangent to the WGS84 ellipsoid (by which is approximated the terrestrial globe), in the base point; if DZ-KML is different from 0, the plan will be moved DZ-kml [m], along the normal to the ellipsoid, in the base point; if DZ-kml is negative the plan will be moved closer to the ellipsoid center,
    - the projection of the points from the XY plane of DWG on the WGS84 ellipsoid is along the ellipsoid normals.

  2) kml = 2 and the user has defined (in AutoLisp) a conversion function from X,Y to Google Earth coordinates (WGS84 - geographic coordinates (longitude and latitude) in decimal degrees) - this conversion function will be used; AutoLisp function must have the name "xy_to_wgs" and two input variables, "x" and "y"; the output must be a list of two variables, "East" (longitude) and "North" (latitude) in WGS84 decimal degrees; below is an example:

```
(defun xy_to_wgs (x y / east north)
  ;Conversion formulas from "x" and "y" to "east" and "north"
  (list east north)
)
```

  3) for kml = 3 , X,Y coordinates are considered in [m], in Stereo70 projection system (from Romania); at the X,Y coordinates will be added also DX-kml and DY-kml (see below); DZ-kml and DA-kml are not used.

- KML3D - can have the values "Yes" or "No";

for "**Yes**" a 3D KML file will be generated from the visible entities 3DSOLID, 3DMESH, 3DFACE and POLYLINE; all these entities will be automatically decomposed into 3D POLYLINE and will be generated in the KML file as LineString entities having Z coordinate; the KML file will be drawn in Google Earth as a 3D wireframe,

for "**No**" all the entities from the part displayed on the display will be drawn 2D in Google Earth; the entities will be projected on the ground, without taking into account the Z coordinate.

- **DX-kml, DY-kml** and **DZ-kml**, considered in [m], will be added at the coordinates of DWG, before the projection on WGS84 ellipsoid; DX-kml and DY-kml have the effect of a displacement of the drawing in XOY plan; DZ-kml has the effect of nearby in the XY plan ("-" sign) or of removal ("+" sign) of the points, compared to the base point;
- **DA-kml** is in decimal degrees (the right angle is 90 degrees), with minus sign for clockwise; the coordinates of the DWG will be rotated with DA-kml;

For the case of KML = 1, the order of the transformation is: DX-kml, DY-kml, then DA-kml and DZ-kml;

If it is needed a displacement with DX-kml and DY-kml, proceed as follows:
- set first the value 0 for DX-kml, DY-kml, DZ-kml and DA-kml and generate a KML file,
- display in Google Earth the KML file (double click on file),
- measure the necessary displacement in Google Earth, in the following way:
- Zoom in (by turning the mouse wheel up),
- press "n" to force North to be upwards,
- press "u" to have a look perpendicular to the ground,
- select in menu: Tools, Ruler and set the unit of measurement in [m],
- draw a horizontal line between a point displayed in KML file and the necessary point on the map and notice the length displayed; enter this value in BricsCAD, at the setting of DX-kml,
- draw a vertical line between a point displayed in KML file and the necessary point on the map and notice the length displayed; enter this value in BricsCAD, at the setting of DY-kml,
- generate a new KML file in BricsCAD,
- draw in Google Earth the KML file (by double click on file) and see if you need of a finer correction of the displacement.

If it is needed of a rotation with DA-kml:
- first, you will make the displacement with DX-kml and DY-kml,
- set DA-kml to 0 and generate the KML file,
- draw in Google Earth the KML file,
- notice if needed and what value must be the rotation, in decimal degrees (with the sign "-" for clockwise); usually, 0.5-1.0 decimal degrees are required,
- enter this value in BricsCAD, at the setting of DA-kml,
- generate a new KML file in BricsCAD,
- draw in Google Earth the KML file (by double click on file) and see if you need of a finer correction; 1-2 iterations are usually sufficient.

If the points have to move more away (sign "+" for DZ-kml) or closer (sign "-") towards the base point, set a value for DZ-kml. Usually, the values are of the order of thousands of meters.

- **LKml** - defines in [m] the side of a square; the DWG area, shown on display, will be automatically split in squares, having a side of LKML meters; each square will be displayed on the entire display and then, will be converted into the KML; for a display resolution of 1280x1024 , LKML = 2500m is enough; if the DWG has small details (circles, text, polyline, etc.) that can not be seen well in Google Earth (they appear like a scrawl), you must decrease LKML; a smaller LKML increases the generation time of the KML file.
- **Wkml** - is the width, in pixels, of all the lines that will be drawn in Google Earth. It can be an integer with values greater than 0.

Kmlout

With Kmlout you can draw in Google Earth:
- in ***3D mode***; in this case the 3D entities (3DSOLID, 3DMESH, 3DFACE and POLYLINE) from DWG will be selected automatically (KML3D is "Yes"; see above) ; in this case it is good to set the BricsCAD variables: "FACETERGRIDRATIO" 0, "FACETERMAXEDGELENGTH" 0 and "FACETERSMOOTHLEV" -1,
or
- in ***2D mode***; in this case all entities from the part displayed on the display will be selected automatically, (KML3D is "No").

The X,Y coordinates of the DWG can be in any projection system and from any part of the globe (KML is 1; see above). The alignment of the DWG in Google Earth is made by means of a base point, having known coordinates in DWG and in Google Earth. It can also make alignment corrections by an eventual displacement and rotation (see Settings).

You can define your own function in AUTOLISP, for alignment (KML is 2).

An automatic alignment can be done if the coordinates X, Y are in the projection system Stereo70 from Romania (KML is 3).

When KML3D is "No" all entitties will be drawn, except IMAGE, SOLID, hatch with "Solid" and the texts having "True Type" style.
The "True Type" styles need to be replaced with vector styles (ie Romans), using BricsCAD "Style" command.
To increase the speed, you should set (in BricsCAD) "Fillmode" and "Textfill" 0 and put "Freeze" the layers that are not necessary.

It will generate a KML file type according to the settings from "Settings". Double clicking on this file, it will be loaded in Google Earth (which must be installed). The KML file is organized on colors that can be selected (by clicking) or inhibited in "Sidebar" in Google Earth. In "Sidebar", "Base Point 1" can be also selected (if kml=1).
"Kmlout" settings will be loaded and saved in the folder and with the name of the current DWG.

ILandxml

ILandxml converts a LandXML file type to a XYZ file of points, which can be loaded with *Loadp* command.
LandXML is a file format that serves to exchange data between programs for surveying and civil engineering (eg ARD-Advanced Road Design (Civil Site Design)).
From LandXML file type will be interpreted Pnts entities of the <Surfaces> elements and CgPoint entities of the <CgPoints> elements.
It will create a file with the same name as the XML file, but with a TXT extension.
LandXML file type must have newline after each item. If not, load it in Google Chrome (which will automatically add newline), make Copy at everything that is displayed, make Paste in a new Word document and save the document as a new XML file.

OLandxml

OLandxml saves the points (POINT entities) and triangles (TIN-Triangulated irregular network represented by 3DFACE entities) of the current DWG in a LandXML file type.
So you have an interface to other programs for surveying or civil engineering (eg Advanced Road Design (Civil Site Design)), software that can import LandXML files.
It creates two XML (LandXML) files.
The first file will have the name of the last loaded TXT file, plus "_cgpoints". It will contain the POINT entities from DWG.
The second file will have the name of the last loaded TXT file, plus "_surfaces". It will contain the 3DFACE entities from DWG.
If before OLandxml, there was no Loadp command, it will generate a XYZ file of points, of TXT extension and with the name of DWG. You can get in this way, an XYZ file of points, from a DWG containing POINT entities!

ROughness

ROughness determines the flatness of the surface defined by the 3DFACE entities (triangulation) in the current DWG.
You must either open a DWG with 3DFACE entities or upload a point file with the "Loadp" option and then to run the "Triangulation" option.
The user will first enter 3 values: "Ruler dimension", "Step" and "Tolerance".
The XOY gauge of the 3DFACE area will be scanned by a 2D square ruler, having "Ruler dimension" as length of the side. "Step" is the scan increment on X and Y.
In each position of the ruler, a plane is determined. The normal of the plane is a average of the triangles normals in the square of the ruler. The position of the plan will be set so that the sum of the distance of the points from the plan is minimal. Relatives to this plan, will be determined the positive (above) or negative (underneath) deviation of each point (vertex) of the 3DFACE from the square of the ruler.
If "Step" is smaller than "Ruler dimension", the deviation for a point will be calculated several times, taking into account 3DFACEs that are in the square of the ruler in the current position. In this case will be stored the maximum deviation from that point.
The recommended value for "Step" is 75% of the "Ruler dimension". Ideally, "Step" would be the size of the smallest triangle (3DFACE) from the entire DWG, but running could take a long time in this case.
Besides the "Ruler dimension" and "Step", the user will also enter the "Tolerance" value. After executing the command, triangles with vertices with deviations higher than "Tolerance" will be marked with POINT entities. POINT

entities will have the color "red" if the deviation is positive (the points are above the plane corresponding to the respective position of the ruler) and the "blue" color if the deviation is negative (the points are below the plane).
So in the DWG only the points that have a deviation from flatness greater than the "Tolerance" value will be marked!
It will also be generated a file of points 4D (with articles of form "X Y Z deviation") with the centers of all the triangles (3DFACE) and the deviation value. The file name is the same as the DWG, but the extension is TXT.
4D files can be represented graphically with different programs, such as our RTOPO http://rcad.eu/rtopo/ program. In RTOPO use the "Load points" button, indicate the name of the TXT file and uncheck "Flat shaded" in VIEW TOOLBAR. You will be able to see the deviations of all the points, based on a color legend.

In "c: \a_triang \ Rough_example.dwg" you have a test example. In it there are 2 zones with deviations of +0.006 and -0.006 and 2 zones with deviations of -0.5 and 0.5. You can test the ROughness command with different values for "Tolerance" (for example, 0.006 or 0.5).

Help

This help.

Undo

Undo reverses the effect of a TRIA option.