

RCAD Help

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1. RCAD

RCAD is a graphic core distributed together with different applications (like [RTOPO](#) or [VDAFS TO DXF and NC Converter and Viewer](#)).

See:

- [Installation](#)

- Commands: [File](#), [Display](#), [Format](#), [Draw](#), [Layer toolbar](#), [Modify](#)

- [Special keys](#).

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1.1 Installation

Requirements:

- graphic card with **OpenGL** hardware acceleration and **32-bit color depth**,
- Windows 98, XP, Vista, 7, 8 or 10 (32 or 64 bits),
- **computer administrator account**.

Create "C:\rcad" directory.

Unzip "rcad.zip" in this directory.

On Windows 98 and XP:

"Right mouse on desktop", "Properties", "Appearance" ("Advanced" on XP), "Item"->"Active Window Border", "Size"->1, "Apply", "Item"->"Inactive Window Border", "Size"->1, "Apply", "OK".

On Windows Vista:

Change "Border Padding", "Active Window Border" and "Inactive Window Border": "right mouse on desktop", "Personalize", "Window Color and Appearance", "Advanced", "Item"->"Border Padding", "Size"->0, "Item"->"Active Window Border", "Size"->1, "Item"->"Inactive Window Border", "Size"->1, "OK".

On Windows 7:

Change "Border Padding", "Active Window Border" and "Inactive Window Border": "right mouse on desktop", "Personalize", "Window Color", "Item"->"Border Padding", "Size"->0, "Apply", "Item"->"Active Window Border", "Size"->1, "Apply", "Item"->"Inactive Window Border", "Size"->1, "Apply", "OK".

On Windows 8/10:

1) Run the program as an administrator: right mouse on "c:\rcad\rcad.exe", "Compatibility", "Run this program as an administrator",

2) set "BorderWidth" and "PaddedBorderWidth" to 0: "Run" command (move your mouse to the upper right corner until the "Charms Bar" pops out, select "Apps" from the list and type "run" in the search box; click on "Run" from the search results), Regedit, HKEY_CURRENT_USER\Control Panel\Desktop\WindowMetrics, set BorderWidth 0 and PaddedBorderWidth 0, restart the computer.

Run RCAD.EXE.

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RCAD can use up to 4 GB of memory on Windows 64-bit and up to 2GB on Windows 32-bit. If this memory is exceeded, RCAD crashes without any message or the messages "Error-insufficient memory ..." or "OPENGL_ERROR ..." are displayed and then RCAD ends its execution. In this case you should take measures to work with fewer entities (for example, set ERase_prev to 1) or you can work without "Flat Shaded" or "Lighting".

-----Settings problems-----

If the graphics card has not OpenGL or the OpenGL drivers are not installed, the image may be upside down or can remain traces on the screen, when you move the mouse.

Similar effects can occur if the color depth is not 32-bit.

If you don't set the Window Border and Border Padding to 1 and 0, the toolbars appear incomplete or distorted.

If you don't run the program from a Windows administrator account or as an administrator (Windows 8), the program exits with an error message (privilege violation or the trial version is expired).

----- SCRIPTS -----

You can call "rcad.exe" from other programs and can also indicate a file of commands to be executed.

The file must be named "c:\rcad\rcad.scr". If this file exists, RCAD will execute first its commands.

The accepted commands are:

DXFIN file_name.dxf

STLIN file_name.stl

DXFOUT file_name.dxf

RTRIANGULATION

VOLUME volume_increment_factor

QUIT

For the RTRIANGULATION and VOLUME commands all the entities will be automatically selected.

If the VOLUME command is not followed by "volume_increment_factor" the volume increment will have the recommended value (see [Format](#)). If it exists, "volume_increment" will be the product of the recommended value and "volume_increment_factor". For example, the value of 0.5 means that "volume_increment" will be half of the recommended value.

The volume will be written in the file "c:\rcad\rcad.vol". If there are not 3DFACE entities or they exist but do not define a body the volume will be 0. At the beginning of the volume calculation "c:\rcad\rcad.vol" will be deleted.

QUIT command ends the execution of the RCAD program.

1.2 Commands

1.2.1 File

New - Delete all existing entities and initialize session.

DXFln - Load a DXF file; LINE, POINT, ARC (in XOY plane), CIRCLE (in XOY plane), 3DPOLY or POLYLINE (in XOY plane), INSERT (without rotation and INSERT in INSERT), TEXT and 3DFACE are interpreted.

3DMESH entities are also loaded and converted in 3DFACE entities in a layer having the name "3DMESHnumber_name". "Number" starts with 1 and is incremented by 1 for each new 3DMESH. "Name" is the original layer name of the 3DMESH entity. All the 3DFACE entities of a 3DMESH entity can be selected using the "LAYER" option (see [Modify](#)).

If the gauge of the drawing is not within the range (-100000, 100000), the displaying may be wrong because OpenGL (software interface for graphics hardware used by the RCAD core) works in simple precision (six significant digits). In this case, move all entities (with the Move command) between the limits (-100000, 100000).

DXFOut - Save the current drawing in a DXF file.

STLin - Load a STL ascii files generates by AutoCAD for solids.

Print - Print the current view of the drawing (representing window) in the rectangle defined by "Paper size" (viewport). Through a dialogue you can set the scale, origin, rotation, sizing or hidden line removal (hide). There are not taken into account the [Flat Shaded and LIGHTing](#) settings.

Landxml - converts a LandXML file type to a XYZ file of points, which can be loaded with [Load points \(POIntin\)](#) 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) and AutoCAD Civil 3D).

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 - saves the points (POINT entities) and triangles (TIN-"Triangulated irregular network" represented by 3DFACE entities that are displayed) 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) and AutoCAD Civil 3D), 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 loaded using [Load points \(POIntin\)](#). If you want to save the POINT entities from a DXF file, load first the DXF file (using DXFIn command), create a XYZ point file (of TXT extension) with [Extract points \(RTExtract\)](#) command and then, load the TXT file.

The second file will have the name of the last loaded TXT file, plus "_surfaces". It will contain the 3DFACE entities that are displayed (resulting from a triangulation or from the loading of a DXF file).

Quit - Exits the program.

1.2.2 Display

Zoom - Enlarges or reduces the display of the active drawing; options: Window/Extents/Previous/<Scale>. "Zoom Extents" from menu recomputes the drawing limits.

Pan - Move the drawing in any direction by specifying two points.

Hide - Creates a hidden-line view.

VPoint - Selects the viewpoint for a three-dimensional look at your drawing.

LIGHTing - Controls lighting; 0=off, 1=on. Lighting is really active if "Flat Shaded" (from VIEW TOOLBAR) is on and exists 3DFACE entities.

HIGHLIGHT - Controls object highlighting for SElect command; 0=off, 1=on. Turn off HIGHLIGHT when you have many entities and you use commands to modify the objects.

GRips - Controls (similar to PDMode command) the appearance of the grips for the CHange command, applied to Line (PLine) entities ; range: 0 to 4, 32, 33, 34, 35, 36, 64, 65, 66, 67, 68, 96, 97, 98, 99, 100. (1=without drawing). The color of the grips is the current color.

GRIPsize - Controls the grip display size (similar to PDsize command). The 0 value specifies a grip size 5% of the drawing height. A value greater than 0 specifies the absolute size, and a value less than 0 specifies a percentage size from viewport.

ANimation - It generates the animation on display by the movement over Line (PLine) entities.

You will first select a Line (PLine) entity. If you have selected 2D PLine, it is advisable to round the vertex using Fillet command. If you have selected a 3DPOLY entity (Line by more than 2 vertex), the results are better if it is interpolated (see [PEdit,Spline](#)).

You will then enter "Step" of movement. A higher value would imply a higher speed of movement. The speed that result will be displayed at the end of animation. The movement speed also depends on the speed of the computer.

During execution you can influence the movement by the keys:

- **ESC**: will stop the execution
- **Left arrow**: increases the horizontal angle (Hor_angle); the sight is rotated to the left; when the angle is 0 you look in front (after the tangent of PLine entity)
- **Right arrow**: decreases the horizontal angle (Hor_angle); the sight is rotated to the right; the values will be negative when you are in the right of 0 value
- **Up arrow**: increases the vertical angle (Ver_angle); the sight is rotated up; when the angle is 0 you look horizontal, parallel to XOY
- **Down arrow**: decreases the vertical angle (Ver_angle); the sight is rotated down; the values will be negative when you are below the value 0
- **Home**: increases the length of the window (L_window); you will see more of the drawing; the bottom side will be centered on the current point of PLine
- **End**: decreases the length of the window (L_window); you will see less of the drawing
- **PageUp**: increases the height of sight (Sight_height); actually you rise above the current PLine point; for 0 value you are exactly in the current PLine point
- **PageDown**: decreases the height of sight (Sight_height); the values will be negative when you are below the current PLine point.

"Hor_angle", "Ver_angle", "L_window" and "Sight_height" will be displayed in the title bar of the program. The angles are in sexagesimale degrees and the distances are in drawing units.

The animation will be done under current settings for the variables LIGHTing (see above) and SPLFrame (see [Format](#)) and under the settings "Wireframe" and "Flat shaded" from VIEW TOOLBAR (see [Special keys](#)).

To save the animation in a file you need to use specialized programs to capture the image from a window of a program.

LEGend - It controls the drawing mode of the 4D points in [POIntin](#) RTOPO command. If it is 1, the 4D points will be colored according to a legend of 20 colors. If it is 0, the 4D points will be colored according to their color.

STOred_points - It sets the number of the 4D points that will be permanently stored, meaning the points that can be modified, saved and printed. The rest of the points of the total points loaded with [POIntin](#), will be drawn temporarily (you can make zoom, pan or orbit, with different values for P Dsize), will not be printed and will not be saved in the DXFOUT command. If your computer has 4 GB of memory (the maximum that can be accessed by the program), you can set the STOred_points to about 9200000 (the value depends also of the graphics card performance).

1.2.3 Format

COlor - Establishes the color for all entities that you draw after you choose the command.

LAyer - Creates named drawing layers and assigns color and linetype properties to those layers. Also sets the current layer, locks/unlocks (unused in this version), rename and turns layers on/off. In "Rename", you can choose a filename having RLN extension, which can contain multiple items with two layer name. First layer name of the article will be renamed with the second name. "Rename" will run automatically for all items in the file.

Using the "Dialog" option you can open a window dialog for setting the layers.

Linetype - Defines linetypes (sequences of alternating line segments, dots, and spaces).

APerture - Sets the selection area for snapping to entities.

LOGFILEON - Writes all commands in RCAD.LOG file.

LOGFILEOFF - Stops writing the commands started with LOGFILEON.

LTscale - Sets the scale factor you want to apply to all linetypes within the current drawing.

PDMode - Controls (like in AutoCAD) the appearance of the POINT entities; range: 0 to 5, 32, 33, 34, 35, 36, 64, 65, 66, 67, 68, 96, 97, 98, 99, 100. 0 value means a pixel irrespective of P Dsize, 1=none, 2 a upright cross, 3 a rotated cross, 4 a upward line, 5 means a point and must be used for the case where there are many points. For the value of 5, P Dsize means pixels. If at the values 1-4 it will add 32, a circle will be drawn in addition. If at the values 1-4 it will add 64, a square will be drawn in addition. If at the values 1-4 it will add 96, a circle and square will be drawn in addition. The most favorable value (in terms of speed and memory) is 5. Then, 0, 2, 3, and 4. The cases with 96 and 32 are the worst.

P Dsize - Controls the point display size. The 0 value specifies a size 5% of the drawing height. A value greater than 0 specifies the absolute size, and a value less than 0 specifies a percentage size from the drawing height. For PDMode = 5, P Dsize means pixels.

P lckbox - Specifies the target height for entity selection.

SPLFrame - Controls the display of invisible edges of 3D faces; 0= Does not display, 1= display.

SPLInesegs - Sets the number of line segments to be generated for each segment of interpolated 3DPOLY.

CONcat_prec - Sets the concatenation precision of the commands [Join](#), [SEParate](#), [3DInt](#) and [SMooth](#). Two end points of the segments are considered identical if the distance between them is less than concatenation precision!

ERAsE_prev - Controls the permanent erasure of entities immediately after the use in some commands; 0 = "off"-without erasing, 1 = "on"-with erasing. It is useful to free the memory to process as many entities. The UNdo command will not restore the deleted entities! An example of command affected by ERAsE_prev is [RTRlangulation](#) (are deleted the point entities before the generation of 3DFace entities).

3DInt_type - Controls the mode of work of the [3DInt](#) command.
If the value is 1, 3DInt will generate only 3DPOLY of intersection.

If the value is 2, 3DInt will generate only the division and concatenation (the joint) of 3DFace entities, along the intersection.

If the value is 3, 3DInt will generate both (3DPOLY and division and concatenation).

CUrvature - represents percents with possible values between 5-100, used to establish the curvature (the inverse of the radius of the curvature) for the commands [PEdit](#)->Spline, [SMooth_3dface](#) and [ISolines](#). The smaller "Curvature" is the larger flattening.

OSnap - Specifies running object snap modes, which remain active until you set them NONE; options: ENDpoint, MIDpoint, CENter, QUADrant, NODE, NEARest.

List - List information about selected entities. For Line/Pline, Arc or Circle entities the length is also listed.

Dist - Reports 3D distance between 2 points.

ID - Lists the X, Y, and Z values of the specified point.

VOLume - calculates the volume and center of gravity of a set of bodies or between surfaces composed of 3DFACE entities. 3DFACE entities from BLOCKs are not considered.

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 decrease the increment, the accuracy and computing time increase!

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 the center of gravity will be displayed. In the center of gravity will be generated a POINT entity.

AREa - Calculates 3D and 2D area for a set of 3DFACE entities. 2D area corresponds to the projection in XOY plane of the 3DFACE entities.

1.2.4 Draw

Line - Draws one or multiple lines (3DPOLY in this case). You can use coordinate filters (.x, .y, .z, .xy, .xz, .yz) to specify points.

PLine - Executes actually LINE command.

Point - Draws a point. The PDMODE and PDSIZE system variables control the appearance of the points.

3Dface - Draws four-sided surface anywhere in 3D space; option: invisible edges

Circle - Draws a circle parallel to XOY plane; options: center point and ray/diameter, 3P, 2P, Ttr

RTRlangulation - Generates a triangulation with required links for a selected set of closed Line (PLine) and POINT entities.

Line (PLine) entities must define an external contour and possibly more interior contours. The contours must be closed (use [PEdit](#) command for that)! The contours edges are the required links for the triangulation! If there are reversion points (at 180 degrees) on the contours or the contours are intersected, the triangulation may be wrong!

If there are POint entities, the command will consider only those points located between the exterior and the interior contours!

It will generate 3DFace entities that will fill the surface between the exterior and the interior contours! If no contours, 3DFACE entities will be generated on the convex hull of the points.

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If `ERASE_prev` variable is 1, are deleted the point entities before the generation of 3DFace entities.

If triangles with intersected sides will result, the triangulation will resume automatically with maximum checks, but with a lower speed. This situation can occur if there are points with small deviations from a line, near contours or convex hull of the set of points.

RTRIangulation is useful for topography, for the generation of triangular finite elements or to define the separation plan of a mold (in combination with [SEParate](#) command).

You can simulate the break lines in the following way:

- define Line/PLine entities, passing through existing points (using OSNAP NODE)
- generate additional points on Line/PLine using MEasure command with a step smaller than the smallest distance between the existing points
- enter now RTRIangulation command

FTRIangulation - Generates a triangulation, 3 times faster. It is recommended for a large number of points (maximum 2 million), with a relative uniform distribution, but not with a constant step. If it doesn't work for the given set of the points, it will resume automatically using the TRIangulation command. You can interrupt FTRIangulation with ESC.

SMooth_3dface - Interpolates a set of 3DFACE entities. For each 3DFACE entity is determined a quartic (of degree 4) triangular surface patch of Bezier type. It provide a continuity of type G1 (the continuity of the tangent plane along the common edge of two 3DFACE entities). The surface passes through the initially vertices of the 3DFACE.

Each initially 3DFACE will be replaced with more 3DFACE located on the surface, according to the user setting for "number of subdivisions" (the number of edges of new 3DFACE on every edge of initially 3DFACE).

If [ERASE_prev](#) variable is 1, the initially 3Dface entities are deleted.

You can control the curvature of the surface, setting in advance the [CURvature](#) variable with a values between 5 and 100. A lower value indicates a greater flattening (vicinity to the plane).

It can result one or more zones that consist of 3DFACE entities that are in contact (with [CONcat_prec](#) accuracy). Each zone will have another color. Coloring is from 1 to 1 starting with 1 color.

The 3DFACE entities that will result will be in the "smooth_3dface" layer.

MEasure - Generates POINT entities, at a given distance, on one or more LINE (PLINE) selected entities.

It will be indicated:

- "Line_step" - the step used for the generation of points over a straight line
- "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 the arcs will be maximum of "Minimum_Arc_step" and the step resulting from "Chord_arc_Deviation".
- "Polyline" - with values 1 (yes) or 0 (no) ; for 0 will be generated points; for 1 will be generated a PLINE entity passing through calculated points.
- "Continuous" - with values 1 (yes) or 0 (no); for 1, the points will be continuously generated with constant step "Line_step" measured on PLINE; for 0, the points will be generated in the end of the line or arcs and between ends, with the steps for lines or arcs.
- "Single_layer" - with values 1 (yes) or 0 (no); for 1 will be generated points in a single layer, having the name "measure"; for 0, the points will be generated in a layer that is formed from the PLINE layer name, plus a number that begins with 1 and is incremented by 1 for each new PLINE having the same layer.

DIVIde_3dface - Divides each of the selected 3DFace entities in triangular 3DFace entities in accordance with the indicated step "Length or number of segments".

If the value entered is > 0, it represents a distance that is the step of division of the longest edge of 3DFace. It could result in this case adjacent triangles with unequal edges in contact (this case is unfavorable for the concatenation from the 3DInt command).

If the value entered is < 0 (favorable case for 3DInt), it represents the number of segments on each edge.

The command must be executed before the 3DInt command, to increase the accuracy of division.

3DInt - Intersects two sets of 3DFace entities.

If the [3DINT_type](#) variable is 1, 3DInt will generate in the "3DINT" layer only 3DPOLY (LINE) entities representing the intersection. They will be colored from 1 to 1, beginning with the color 1. The joints in 3DPOLY of the intersection segments will be made according to [CONcat_prec](#) variable.

If the 3DINT_type variable is 2, will be made only the division and concatenation (the joint) of 3DFace entities, along the intersection. A quadrilateral 3DFace will be decomposed into two triangular 3DFace. A triangular 3DFace can be divided only in two pieces, along the intersection of two of its sides! If a triangular 3DFace is intersected several times, the results will be inaccurate! After division, zones will be generated, by concatenation of 3DFace which will receive the same color. A zone is formed by 3DFace entities that are placed on the same side of an 3DPOLY of intersection and in contact with common sides (respecting [CONcat_prec](#) accuracy).

The zones will be colored from 1 to 1, beginning with the color 1. If it results bad zones, divide first the large 3DFace entity with [DIVIde_3dface](#) command (see above) and/or increase the value of [CONcat_prec](#). The 3DFace entities located in zones

will be in their original layers. You can select a zone using the "COLOR" option (see [Modify](#)).

If the 3DINt_type variable is 3, will be generated 3DPOLY of intersection and the division and concatenation of 3DFace entities.

TABSurf - Generates a set of 3DFace entities by moving of one PLine entity along a vector. The set of 3DFace entities actually defines a 3DMESH entity.

You must select a PLine entity and then, you need to give a direction vector by two points. The arcs in PLine are considered line segments. You can approximate the arcs using first the MEasure command with "Polyline" option (see above).

The 3DFACE entities will receive the current color and will be located in a layer having the name "3DMESHnumber_name. "Number" starts with 1 and is incremented by 1 for each new 3DMESH. "Name" is the original layer name of the PLine entity. All the 3DFACE entities of a 3DMESH entity can be selected using the "LAYER" option (see [Modify](#)).

REVSurf - Generates a set of 3DFace entities by rotating of one PLine entity around a vector. The set of 3DFace entities actually defines a 3DMESH entity.

You must select a PLine entity and then, you need to give a rotational axis vector by two points, the included angle and the steps number. The arcs in PLine are considered line segments. You can approximate the arcs using first the MEasure command with "Polyline" option (see above).

The 3DFACE entities will receive the current color and will be located in a layer having the name "3DMESHnumber_name. "Number" starts with 1 and is incremented by 1 for each new 3DMESH. "Name" is the original layer name of the PLine entity. All the 3DFACE entities of a 3DMESH entity can be selected using the "LAYER" option (see [Modify](#)).

Offset - Generates an equidistant PLine from a selected PLine. The equidistance will be on the right if the distance is positive and on the left if the distance is negative. The sense of the PLine can be seen using [PEdit](#) command.

The selected PLine may be 3D. Z of an equidistance point will be the Z of the corresponding point of the selected PLine.

The arcs of the selected Pline will be considered as straight lines. If you still want offset for arcs, generate first a polyline that approximates the arcs by line segments, using MEasure command (see above), with 1 option for "Polyline", 0 for "Continuous" and the appropriate setting for "chord_arc_Deviation" and "minimum_Arc_step".

Due to an excessive number of points with small distance between them and small deviations from straight lines, can result an offset which intersects itself. Use in this case the MEasure command , with 1 option for "Polyline", 1 for "Continuous" and a distance greater than the distance between points. Thereby you obtain a PLine with a smaller number of points. Make offset to this last PLine!

1.2.5 Layer toolbar

LAYER - Open a window dialog for setting the layers.

MOLC - Make object's layer current.

LAYISO - Isolate object's layer.

LAYOFF - Turn object's layer off.

LAYALL - Turn all layers on.

1.2.6 Modify

Erase - Erase selected entities.

COPY - Copy selected entities. The two points you specify define a displacement. You can use coordinate filters (.x, .y, .z, .xy, .xz, .yz) to specify points.

MOVE - Move selected entities. The two points you specify define a displacement. You can use coordinate filters (.x, .y, .z, .xy, .xz, .yz) to specify points.

ROTATE - Rotate selected entities in XOY plane or about a three-dimensional axis.

SCALE - Scale selected entities uniform or proportionally in the X, Y, and Z directions. The base point you specify identifies the point that remains in the same location. You can specify the scale using one or three different SX, SY, and SZ values. Specifying negative value you can make mirror.

Fillet - Round the vertices of a 2D PLINE entity. You can indicate a radius or you can choose between the rounding of one point (Vertex) or rounding all vertices (Polyline). You can round only the vertices that are between 2 straight lines!

PEdit - Edit a Line (PLINE) entity.

The options are: Next, Previous, Insert, Erase, Break, CURvature, Spline, Decurve, Close, Open, eXit or the modification of the current position of the point.

"Erase" delete the current point.

"Break" break PLine in 2 parts and the editing will continue with the first part.

"Spline" will interpolate according to the SPLInesegs and CURvature variables.

"Decurve" eliminate the intermediate points under the current setting of SPLInesegs.

"CUrvature" represents percents with possible values between 5-100, used to establish the curvature of the interpolated lines. The smaller "Curvature" is the larger flattening (vicinity to the straight line).

Grips will be displayed according to GRips and GRipsize variables (see [Display](#)).

If the segments containing the point of the LINE (PLINE) are arcs, they will be transformed into a straight line.

You can use coordinate filters (.x, .y, .z, .xy, .xz, .yz) to specify the destination point.

Join - Joins the Line (2D or 3D) or PLine entities selected to form a single PLine entity. The joint will be made according to the CONcat_prec variable (see [Format](#)). Any arcs will turn into straight lines. It starts from the first Line/PLine entity of the selected set. At its ends will be repeatedly added other Line/Pline entities having common ends. It will result a single new PLine having the color and layer of the first segment.

MJoin - runs repeatedly Join command, joining all Line/PLine of the selected set. It results more Pline entities. Undo is not available for this command.

CHange - Is launched automatically when a command is not active and you select with the mouse a Line (PLine) or a 3DFACE entity. You can change the position of the nearest point of the entity. If the entity is Line (PLine) will be displayed the options of PEdit command. In the STANDARD TOOLBAR will be displayed the entity attributes (layer, color and linetype).

SEParate - Intersect a set of 3DFace entities with a horizontal plan passing through an indicated point.

Each intersected 3DFace entity will be separate in two parts according to the segment of the intersection.

The 3DFace entities located above the intersection will be transferred to the layer "separate_up" (having cyan color).

The 3DFace entities located below the intersection will be transferred to the layer "separate_down" (having magenta color).

The segments resulting from the intersection will be automatically joined in Line (PLine) entities located in the layer "separate_polyline" and colored from 1 to 1, beginning with the color 2. The joints will be made according to the CONcat_prec variable (see [Format](#)). These contours (eventually corrected with PEdit command) and contours defined explicitly with the command Line (PLine) can define the separation plane of a mold (by using [RTRlanquation](#) command)!

If you press the left mouse button and in the cursor position is not a LINE (PLINE) or a 3DFACE entity, it automatically enters in the Crossing selection mode (of the SElect command). When you are in the SElect command and are selected entities you can change the layer, color or linetype through a proper selection in STANDARD TOOLBAR.

The modification commands are automatically preceded by SElect command.

The selection options are: Window, Crossing, CPline, ALL, Previous, Add, Remove, Single, Last, LAyer and COlor.

The "CPline" option will select all entities having a point inside a selected PLine.

Using "LAyer" option and selecting an entity, all entities having the same layer name with the selected entity, will be automatically selected.

Using "COlor" option and selecting an entity, all entities having the same color with the selected entity, will be automatically selected.

The options "LAyer" or "COlor" are valid only for the next selection, and "Remove" remains valid until it selects the "Add" (and vice versa).

Turn off HIGHLIGHT variable (see [Display](#)) when you have many entities.

UNdo - Reverses the effect of a number of commands of modification.

UNdo1 - Reverses the effect of one command of modification.

1.3 Special keys

You can move the toolbars by dragging them from any side (preferably left). You can also turn a toolbar from vertical to horizontal and vice versa (dragging the appropriate side). You can not put off a toolbar. If you are bothered by a toolbar, drag it so that the toolbar area to be outside of the window.

F1 - Display Help

F2 - The zone of the commands is displayed on the full screen or at its previous size. You can reduce or increase the number of displayed rows by positioning the cursor above the topmost displayed row. Press the left mouse button and then move the mouse up or down.

F8 - Turns on/off Ortho mode (which work in the projection plane, identical to that of the screen)

ESC - Cancel a command in progress

Press mouse left button - Orbit (change VPOINT in real time) when you move the mouse.

Press wheel button - PAN in real time when you move the mouse

Rotate mouse wheel - ZOOM SCALE in real time

VIEW TOOLBAR

Splframe - controls the display of invisible edges of 3D faces; 0= Does not display, 1= display.

Wireframe - displays 3DFACE using lines

Flat Shaded - shades the objects

LIGHTing - controls the lighting; 0=off, 1=on. Lighting is really active if "Flat Shaded" is on and exists 3DFACE entities.

STANDARD TOOLBAR

Picking +/- in the layer box you can switch layer on/off

When a command is not active you can pick and select the current layer, color or linetype.

When you are in the SElect command and are selected entities you can pick and change the layer, color or linetype of the selected entities.

2. RTOPO

Is useful in topography for the:

- "Digital Terrain Modeling" (DTM) by "Triangulated Irregular Network" (TIN),
- representation (by 3DPOLY, SPLINE or BLOCK entities) and editing (modify, delete or add) of 3D points ("x y z code" format),
- triangulation,
- isolines (contour lines),
- cross sections,
- longitudinal profile,
- 2D entities (Line or PLine) projection over a DTM,
- conversion of TIN to a regular grid,
- conversion of existing isolines resulting from maps in points and then in TIN.
- calculation of the area of a triangulation, in 3D or 2D (in XOY plane), using the [AREa](#) command,
- calculation of the volume between two triangulations (or one and a plane) using the [VOLUME](#) command,
- intersection of two triangulations, using [3DInt](#) command
- transformation of a drawing containing entities as PPoint, PLine (3dpoly), Line, Block and Text, in points having codes and then in a triangulation (see [Extract points](#)),
- representation of the 4D point cloud: "x y z r g b" or "x y z s" formats (see [Load points](#)).

(AREa, VOlume and 3DInt are commands of the RCAD graphic core),

- import and export of LandXML file type.

RTOPO is delivered together with the [RCAD](#) graphic core.

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Select from menu or toolbar :

RTOPO

Load points

"c:\rcad\Load points example.txt"

Intermediary points

Triangulation

Isolines

Sections

Edit points

Settings

Redraw points

draw a PLine entity

make Fillet on PLine entity

Projection

select PLine

2.1 Points treating

Sets the drawing mode of the points (see [Load points \(POIntin\)](#)). "Lineweight" is not yet implemented.

2.2 Settings

Sets:

- text height - used to number, Z, code, longitudinal profile and cross sections, the scale of the inserted blocks; for longitudinal profile and sections (see [Sections](#)) the recommended value is 1.8
- axes step - the grid axis step (if it is 0, the axes are not drawn)
- intermediary points distance (see [Intermediary points](#))
- longit.Yscale - drawing scale of the longitudinal profile
- autotranslation (see below)
- numbered codes - will be numbered starting with 1, the points having the codes introduced here (separated by comma ",")
- draw number - will be drawn the number of each point
- draw z - will be drawn the Z of each point
- draw code - will be drawn the code of each point.

"AutoTranslation" is useful when X or Y coordinates are ≥ 100000 , because OPENGL works in single float precision (6 significant digits) and the displaying may be corrupted. "X translation" and "Y translation" will be added to coordinates when you load points and will be subtracted when you save or extract points.

2.3 Redraw points

Redraws the points according to [Settings](#). Preliminarily, all entities will be erased.

2.4 Intermediary points

Generates intermediary points on the "3dpoly" and "spline" RTOPO entities according to "intermediary points distance" (see [Settings](#)). This aids the triangulation! "Intermediary points distance" must be the minimum distance between 2 entities "3dpoly" or "spline". "3dpoly" and "spline" RTOPO entities are actually break lines for triangulation.

2.5 Load points (POIntin)

It loads and draws a file of points (of TXT type) having a format with 2, 3, 4, 5, 6 or 7 fields. The item type is given by the number of fields. The cases 4, 6 or 7 fields can be drawn in 4D (see below).

The separating character of the fields may be: space, tab or ", ". A file must have items with the same number of fields.

X and Y coordinates are affected by [AutoTranslation](#) setting.

Types of items that can be processed with RTOPO:

a) **5 fields: point_number X Y Z Code**

for example: 1238 420385.445 658974.376 455.38 AX01

b) **4 fields: X Y Z Code** (point_number will be 990; Code must begin with a letter)

c) **3 fields: X Y Z** (point_number will be 991 and Code will be "L")

d) **2 fields: X Y** (point_number will be 992, Z will be 0 and Code will be "L")

The coordinates are considered in [m].

POINT, 3DPOLY and BLOCK entities will be generated.

"Code" contains 2 parts: the first part contains only letters (according to [Points treating](#)). The second part is an integer number. The points heaving the same code (letters+number) will be joined according to "Points treating", with "3dpoly" or "spline". "Spline" is actually a 3DPOLY entity, interpolated according to the [SPLInsegs](#) variable. The order of the connection is the one of the TXT file. If the order is wrong, you can correct this using [Edit points](#) command and the options: "code_After_point" or "code_Before_point".

The "Symbol" is drawn by a block defined in "rtopo_simbol.dxf" file. You can modify this file in AutoCAD or IntelliCAD!

Modifying the Block in AutoCAD: BEDIT (Block Editor), select the block, OK, modifications, Close Block Editor: Save the changes, in the menu: "Save as" C:\RCAD\rtopo_simbol.dxf.

New Block in AutoCAD: around the point 0,0 and 1-2 units gauge, draw AutoCAD entities, Block command, Objects: Select objects and Delete option, Name, Base point 0,0 , in the menu: "Save as" C:\rcad\rtopo_simbol.dxf. Set then in RTOPO: "Points treating", New button, fill Name, Treating (symbol) and Description. The Blocks will be inserted at the [text_height](#) scale.

The POINT entities will be drawn according to the settings of [PDsize](#) and [PDMode](#) variables (that can be entered from the keyboard or selected from RCAD-Format menu). We recommend PDMode = 5 (for drawing speed and minimum memory used). The next recommended value would be PDMode = 2.

Before the loading of a new TXT file, are deleted all the existing entities!

Types of articles that will be represented 4D (considering the [LEGend and STORed_points](#) settings):

- a) **4 fields: X Y Z s** (s is a number: integer (color) or decimal (scalar)); if LEGend = 1, "s" is considered to be a scalar and the points will be drawn according to a legend of 20 colors, between the maximum and minimum values of "s"; if legend = 0, it is considered that "s" is a color (it will actually use the integer rest of "s" divided to 256; 0 color will turn in 1)
- b) **6 fields: X Y Z r g b** (r, g, b, represents the color in RGB format (Red, Green, Blue)); r, g, b will be converted first to a standard color, between 1 and 255; then, LEGend is set 0 and it will draw according to the standard color of each point
- c) **7 fields: X Y Z r g b s** or **X Y Z s r g b** (r, g, b, represents the color in RGB format (Red, Green, Blue); s is a decimal number (scalar)); if LEGend = 1, it will draw according to the legend of 20 colors, between the maximum and minimum values of "s"; if LEGend = 0, r g b will be converted first to a standard color, between 1 and 255; then it will draw according to the standard color of each point.

In the case of a 4D Draw, PDMode is automatically set to the value 5. As well, the points can not be edited ([Edit points](#)) or saved ([Save points](#)). The points can be saved using [Extract points](#) command and with the Other_points set to 2.

You can redefine the legend of 20 colors at the [ISolines](#) command!

2.6 Save points (POINTOut)

It saves all the points (which are loaded with [Load points](#) and possibly edited with [Edit points](#)) in a TXT file. Will not be saved the points coming from a DXF file or from the PPoint command! To save these points use [Extract points](#) with Other_points=1 or 2.

2.7 Extract points (RTExtract)

It saves in a TXT file, only the selected points, in the "point_number x y z code" format. There are 4 options: Rtopo_points, Other_points, Text_insertion and Block_insertion. You can use any combination of values for these options.

1) Rtopo_points - if it is 1 will be saved points loaded with [Load points](#) and possibly edited by [Edit points](#). If "Rtopo_points" is 0, will not be saved this kind of points.

This option is useful when there are many points and is more convenient to correct them in different parts or when in different areas of the land have been used the same codes (letters+number) for points.

To save only the points that are at a certain distance from an existing PLine:

- make the left and right Offset of the PLine (see [Offset](#))
- draw a Line at the ends of the offsets,
- join the line and offsets in a single PLine (using Join command),
- run RTExtract and use CPline option to select entities.

2) Other_points - if it is 1 or 2, will be saved points not loaded with [Load points](#) and possibly edited by [Edit points](#). For "Other_points" =1, the point code will be the layer of the point. For "Other_points" =2, the point code will be the color of the point and the format is "x y z s" (is useful when 4D points were loaded).

If "Other_points" is 0, will not be saved this kind of points.

"Other_points" =1 option is useful to transform several 3D Line/PLine entities in points with codes.

Previously, you must run the commands:

- [MJoin](#), to join all segments in PLine entities,
- [LAyer](#), Rename, File, to rename the layers as codes of the library (see [Points treating](#)); RLN file type must be created in advance.
- [MEasure](#) with the options: "Polyline" 0, "Continuous" 0 and "Single_layer" 0.

3) Text_insertion - if it is 1 will be saved the insertion points of the TEXT entities; Z-coordinate will be the one of the insertion point. If it is 2, Z will be the one mentioned in TEXT. If "Text_insertion" is 0, will not be saved this kind of points. The point code will be the layer of the TEXT.

4) Block insertion - if it is 1 will be saved the insertion points of the INSERT (of block) entities; the point code will be the layer of the INSERT. If it is 2, the point code will be the name of the block. If "Block_insertion" is 0, will not be saved this kind of points.

2.8 Edit points (RTedit)

It edits interactively the points.

You can change:

- the position (X,Y), indicating the new values (explicitly or interactive), possibly with OSNAP options,
- Z: by value, of point (by copying the Z of another point indicated in an interactive mode), by increment or by interpolation between 2 or 3 points,
- the code,
- the code and the position relative to another point in "3dpoly" or "spline" (code_After_point, code_Before_point); the point will receive the code of another selected point and will be placed (in terms of the succession in "3dpoly" or "spline") after or before the selected point,
- you can split "3dpoly" or "spline"; the first part ends at the selected point and the second part starts of the next point and receives the same letters of code and a number incremented by 1.

You can also make global changes to all points ("All of code") having the same code (letters + number): delete, reverse order, move or change the code.

You can also insert or delete one point.

After each change, all the entities will be automatically deleted, except those who are in the layer "kept". It follows that, if you want to make changes relative to some entities that you have uploaded from a file (with DXFIn) or by RCAD commands, these entities should be moved (see [Modify-SElect](#)) to the layer "kept", to not be deleted after each editing in RTedit .

2.9 TRlangulation

It generates a triangulation of a set of POINT entities.

3DFACE triangular entities will be generated.

If you set "Offset to AX01 code", temporary points will be generated on offset and after triangulation, the triangles having sides on offset and the temporary points will be erased. Use this option to avoid triangles which bind far points.

It can exist a drilling file "points_file_name.dri" having sequences (preceded by "****") of the form:

point_number_1 delta_Z_1

point_number_2 delta_Z_2

...

An article contains a point number and a Delta_Z value added to his Z. Delta_Z (<0) is the distance between ground and the geological layer! Each sequence will be triangulated independently. The cross sections and the longitudinal profile will contain the intersections with these triangulations.

If ERase_prev variable is 1 (see RCAD-Format Help), are deleted the point entities before the generation of 3DFace entities.

"3dpoly" and "spline" RTOPO entities (see [Load points \(POIntin\)](#)) are actually break lines for triangulation.

You can use also [RTRlangulation](#) command which generates a triangulation with required links for a selected set of closed Line (PLine) and POINT entities.

You can intersect two triangulations (TIN), if they have different colors, doing something like this:

- load the first triangulation using the [DXFIn](#) command or by the triangulation of a set of points

- change the color of the 3DFace entities by their selection and then choose a new color in the [STANDARD TOOLBAR](#)
- save the triangulation using [DXFOut](#) command
- load the second triangulation using the DXFIn command or by the triangulation of a set of points
- load the first triangulation using the DXFIn command
- launch the [3DInt](#) command and select the first triangulation (as the first set) and then the second (as the second set) using the option [COlor](#) (are automatically selected all the entities that have the selected entity color).

Will result some intersection 3DPOLY entities and the 3DFace entities are divided into zones (colored differently) limited by intersections. You can then delete some zones or calculate the volume (using the [VOLUME](#) command).

2.10 ISolines

It generates the intersection of 3DFACE entities with equidistant horizontal planes.

3DPOLY entities or colored-filled zones will be generated. You must run first [TRlangulation](#) or [RTRlangulation](#).

3DPOLY may be optional interpolated and over them, Z will be written.

3DPOLY or colored-filled zones will be colored according to a legend.

You will enter the minimum and maximum Z for the isolines. In advance you can redefine the legend of 20 colors ("Legend_color") or you can return to the original colors ("Reset").

You will then enter the distance between the planes (resulting isolines (contour lines) in the form of 3DPOLY) or you can choose "Colored_filled" for divide in 20 intervals the distance zmax-zmin (resulting colored-filled zones).

In the case of 3DPOLY isolines you can choose between: interpolation and joined segments ("Yes"), without interpolation and joined segments ("No") or without interpolation and not joined segments ("noT_join"). The last option is useful when you initially opted for the interpolation and the insufficient memory message ("Error-malloc ...") is displayed. You will then enter the "Curvature" which represents percents with possible values between 5-100, used to establish the curvature of the interpolated lines. The smaller "Curvature" is the larger flattening (vicinity to the straight line).

You can set the number of line segments to be generated for each segment of interpolated 3DPOLY by the [SPLinesegs](#) variable.

If [ERase_prev](#) variable is 1, are deleted the 3DFace entities before the generation of the isolines or of the colored-filled zones.

You can get a better approximation of the surface represented by the 3DFACE set, running first the [SMooth_3dface](#) command and then ISolines, without interpolation. In this way it is made an intersection with horizontal planes of a continuous 3D surfaces that passes through the vertices of the 3DFACE set.

2.11 Sections

It generates for a code: cross sections (1:1 scale) and longitudinal profile (1:1 scale on X and [Longit.Yscale](#):1 on Y). The cross sections will be generated to the normal of the curve having the given code, up to the indicated distances, "left distance" and "right distance".

In the longitudinal profile, the largest slope (%) is colored red. "S 2D|3D" are the cumulative distances from the first point: 2D (of the projection on XOY plane) and 3D.

If there are geological layers (see [TRlangulation](#)) will be drawn the intersections with them with the color 21.

If there are OG01 codes (left waterside) and OG02 (right waterside) will be also drawn (with the color 160) the water level, as follows:

- in the longitudinal profile, Z of the middle of the segment that connects the points of intersection between the normal to the curve (of the given code) and OG01 and OG02 curves; in table will be written Z of the middle of the segment and its deviation

- in the cross sections, the segment that connects the points of intersection between the normal to the curve (of the given code) and OG01 and OG02 curves.

In addition to OG01 and OG02, will be also drawn (if they exist) other waterside pairs of the form: OG04-OG05, OG07-OG08, OG10-OG11, ... (repeatedly gather 3 to each code number).

To draw OG curves, you must give values large enough for "left distance" and "right distance" (you can find the maximum distance between the curve of the given code and an OG curve, using "Distance" command)!

You must run first [TRlangulation](#) .

3 input files, for the MicroPiste Setra program, are also generated :

- 1) "*points_file_name-code.PO*" having format: POI Ppoint_number X Y
- 2) "*points_file_name-code-L.PO*" having format: POI Ppoint_number S Z

S is the accumulated 3D distance from the first point.

- 3) "*points_file_name-code.PRF*" having format:

```
point_number X Y

XY_distance_from_axis Z [C] XY_distance_from_axis Z [C]

XY_distance_from_axis Z [C] XY_distance_from_axis Z [C]

...
```

XY_distance_from_axis represents the XY distance from the left (-) to right (+) between a cross section point and axis (the point having selected code). Character 'C' appear between the couples "XY_distance_from_axis Z" corresponding to the curves having the codes "CS01" (left road bed) and "CD01" (right road bed). The three files are generated in the same directory as TXT points file.

2.12 Projection

It generates points with a certain code and step on one or more entities "Line" (PLine) and it generate the projection of these points over a triangulation.

The projection will be generated over the visible (layer is on) "3DFace" entities.

The projected entities may be "Line" or "PLine" which may contain arcs. A number of points with a certain step indicated by the user will be generated for each straight line or arc segment. The entities may be introduced interactively or may be imported by the [DXFIn](#) command. In case of DXFIn and if there are "AutoTranslation" (see [Settings](#)) and points loaded with POIntin, the imported entities will be translated automatically with the values "X translation" and "Y translation."

You can round the vertex of the "Line" ("PLine") entities using the RCAD "Fillet" command.

You can modify the "Line" ("PLine") vertex selecting them by the mouse (RCAD "CHange" command) and indicating the new position.

After the entities selection you must enter:

- "Only_generation" - possible values 0, 1, 2;

0 - will generate points and will make their projection; only the projection of the points is preserved

1 - only the generation of the points will be made; this option is useful when the "Line" (PLine) entities are actually isolines (having real Z) resulting from scanned maps and you wish to obtain the 3D digital model of the terrain

2 - will generate points and will make their projection; keep both the projection of points as well as the generated points that have not projection; this option is useful when you want to transform the digital terrain model from an irregular triangulation of points in a regular network of NxM points

- "Line_step" - the step used for the generation of points over a straight line

- "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 the arcs will be the maximum of the "Minimum_Arc_step" and the step resulting from "Chord_arc_Deviation". If you want a continuous and constant step on PLINE, use first the [MEasure](#) command with the options Continuous=1 and Polyline=1.

You'll have to indicate in the final the points code which will result. If you project more entities, to the code will be automatically added the number of entity (from 1 to the maximum number of projected entities).

Because "Redraw points" is not made automatically after the projection, you can still launch [RTSection](#) command for the code used in projection. You can get in this way the cross sections and the longitudinal profile for an existing triangulation and a desired route.

If you use for projection a family of parallel straight lines (horizontal or vertical) having equal lengths and you set "Only_generation"=2, you can transform the digital terrain model from an irregular triangulation of points in a regular network of NxM points (having constant steps on the X and Y axis)! You can use for this, the horizontal or vertical straight lines of the axis grid!

3. VDAFS TO DXF and NC Converter and Viewer

Converts VDAFS file to DXF file (in 3DFACE entities) and generates a NC program for milling a piece composed from 3DFACE entities.

VDAFS TO DXF and NC Converter and Viewer is an application for [RCAD](#) graphic core (Copyright RCAD SOFTWARE).

In VDAFS file will be interpreted the entities of type PSET, FACE (created from the entities of type SURF and CONS) and possibly those of type SURF (see "VDafsin" command).

3DFACE entities may result also from 3DFACE or 3DMESH entities from DXF files, from STL ASCII files generated by AutoCAD for solids (see RCAD "STLin" command) and from RCAD "3DFace" command.

You can calculate 3D [AREa](#), [VOLUME](#) and center of gravity of some bodies limited by 3DFACE entities.

3DFace entities which describe the surface can be intersected or not perfectly joined! The milling cutter with hemispherical head will be always above and tangent to the assembly of 3DFace entities!

3D bodies described by 3DFACE entities can be divided into 2 parts by a horizontal plane ([SEParate](#) command). You can define then the separation plane for each part, using RCAD commands:

- [PLine](#) to define the exterior contour
- [PEdit](#) and [Join](#) on the contours resulting from the SEParate command
- [RTRlangulation](#) on the contours above.

3D bodies can be also divided into two or more zones by surfaces composed of several 3DFace, using the [3DInt](#) command (of RCAD core).

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3.1 VDafsin

Loads a file of VDAFS format. 3DFACE triangular entities will be generated.

"Surface tolerance" will be introduced first. "Surface tolerance" is maximum admitted distance (in [mm]) between any triangular facet and a parallel plan to the facet and tangential to the corresponding scrap of the theoretical surface. The triangle points are on the surface.

There will be interpreted the entities of type PSET, FACE (created from the entities of type SURF and CONS) and possibly those of type SURF (see on). The outlines which define FACE must not (auto) intersect. If they intersect because of a too high tolerance or because of a mistake of the person who projected them the error message "Decrease tolerance" appears. The previous tolerance is divided automatically by 4 and triangulation is tested again. This process is repeated two times. If the error still appears, is tried for the third time with the initial tolerance and with total verifications of intersections.

If entities of type PSET exist, it will be generated a surface (with a sole Z for an X, Y given) through the triangulation of the points from PSETs.

If it exists a file of DEF extension with the same name and directory as the file of VDA extension, then only the entities of SURF and FACE indicated in the file of extension DEF will be interpreted.

Here it is its structure:

- Key word "surf", the number of entities which follows of SURF type, and then the name of these entities
- Key word "face", the number of entities which follows of FACE type, and then the name of these entities
- Instead of the number and then the name of entities, it may appear the key word "all"; in this case all entities of SURF or FACE type will be interpreted
- Key words, numbers and names of entities must be separated, at least by a character "space", "tab", CR or LF

Example 1:

surf 20

surf0010 surf0016 surf0018 surf0019 surf0021 surf0022 surf0028 surf0029

surf0030 surf0034 surf0035 surf0036 surf0037 surf0038 surf0039 surf0040

surf0041 surf0042 surf0043 surf0044

face 25

face0001 face0002 face0003 face0004 face0005 face0006 face0007 face0008

face0009 face0010 face0011 face0012 face0013 face0014 face0015 face0016

face0017 face0018 face0019 face0020 face0021 face0022 face0023 face0024

face0025

Example 2:

surf all

face 1

face0001

3.2 NCSettings

It allows the setting of the technologic parameters for generating the NC program.

The gauge limits of the surface, the number of triangles and the geometric rugosity, due to the secondary step will be displayed.

You can make some settings for postprocessor by N-start, N-max, G-modal and XYZ-modal.

SURFACE TOLERANCE is a repetition of that of the [VDafsin](#) command. Introducing the value for SURFACE TOLERANCE it leads to redefinition:

- the main step becomes equal with SURFACE TOLERANCE,
- the secondary step is calculated in such a way that rugosity to be equal with SURFACE TOLERANCE,
- $\text{separation_border} = \max(\text{Xstep}, \text{Ystep}) + \text{ray} + \text{principal_step} + 0.1,$
- $\text{z_reference} = \text{zmax} + \text{ray} + \text{principal_step}.$

RAY defines, in millimeters, the ray (radius) of hemispherical-headed tool. The redefinition presented at SURFACE TOLERANCE will be also made.

X, Y RAPID - Selecting the axis according to which the tool moves more rapidly can be made by commuting between X RAPID and Y RAPID (see X, Y STEP). The corresponding step to the axis of rapid movement is called main/principal step. The other step is called secondary step.

X, Y STEP - Through X STEP and Y STEP are defined, in millimeters, the movement steps on the axes X and Y of the center of the tool. The processing is made in zigzag by trajectories equidistant at surface, having the projections in XOY, parallel right lines to the axes. It results that the tool machine must not have 3D geometrical interpolator, but only 2D in XY, XZ and YZ. Initially, axis X is the one after which the tool will move faster (the main step is parallel to axis X). By commuting between X RAPID and Y RAPID (see X, Y RAPID), it can be selected the axis after which the tool can move faster.

If the vertical of a given X, Y cuts more surfaces the surface of Z maxim will be considered.

It is forbidden to introduce a principal step > SURFACE TOLERANCE !

It is forbidden to introduce a secondary step > 2* RAY !

After introducing the steps, it is redefined Z REFERENCE = zmax+ray+principal step, and

$$\text{SEPARATION BORDER} = \max(\text{Xstep}, \text{Ystep}) + \text{ray} + \text{principal_step} + 0.1.$$

Because the verification of the interference with the piece is only made after making a step, it is recommended the step by which the tool moves faster (the principal step) to be sufficiently small. However, when generating the NC program, the segments belonging to the same straight line will be converted into a single segment. Deviation from perpendicularity or the maximum supplement left without being removed on a facet almost vertical is equal to the principal step. If the secondary step is not executed in the separation plan, it will have to be smaller or equal to the principal step.

It will be displayed the geometric RUGOSITY due to the secondary step. The rugosity will be calculated with the formula $a=r\sqrt{r^2-p^2/4}$, where 'r' is the radius of the tool and 'p' is the secondary step.

In the case of a vertical plan, parallel to the principal direction of processing, the maximum supplement that can be left without being removed on the wall is equal to the secondary step. In order to decrease this supplement left without being removed, either the secondary step will be decreased, or two crossed (first with principal step by X and then by Y) will be made.

Z REFERENCE allows defining the Z reference, at which the tool withdraws for getting through positioning into another areas. When a new file is loaded, Z REFERENCE becomes $z_{max}+ray+ principal_step$. At the actual version, Z REFERENCE has significance only for the beginning and the end of the NC program.

SAFETY DISTANCE defines a distance in millimeters. The tool will descend with G0 to this surface distance, and then it will go through this distance with G1. When loading a new file, SAFETY DISTANCE becomes $z_{max}-z_{min}$. At the actual version, SAFETY DISTANCE has significance only for the beginning of the NC program.

Z SEPARATION allows defining the Z of a rectangular separation facet of the piece. The rectangle has X, Y identical centers to those of the base of the gauge parallelepiped. The dimensions of the rectangle can be obtained from those of the rectangle base, to which are added algebraically the SEPARATION BORDER, on each side of the rectangle. During execution of the command [NCGenerate](#), it will be processed the separation facet and the part of piece (composed by 3dface entities) above it. When a new file is loaded, Z SEPARATION becomes z_{min} . By varying Z SEPARATION you can divide headpiece processing.

SEPARATION BORDER allows introducing a distance that is added on the sides of the gauge XOY rectangle, to form the separation plan (see Z SEPARATION). When loading a new file or when modifying the value for RAY or X, Y STEP, SEPARATION BORDER becomes $\max(Xstep, Ystep)+ray+ principal_step+0.1$.

The SEPARATION BORDER facet is not processed if it is inside the contour (in XOY) of the piece! You can do this giving negative values for SEPARATION BORDER!

3.3 NCGenerate

It generates a NC program according to [NCSettings](#). During execution, it will be processed the separation facet and the part of piece (composed by 3DFace entities) above it.

The NC equipment must have 3D interpolator.

It will be indicated a file name of implicit NC extension.

To eliminate the interference milling machine - piece, due to the principal step, it will be generate a trajectory of the corresponding tool to a $ray=ray+principal_step$.

3.4 NCLoad

Loads and draws a NC file. It will be interpreted G0, G1, X, Y and Z address.