Special Strategy

«Magic Words»

- CUTHOLDERS
- LOCK
- NOSEL
- NOROT
- UNDERCUT
- DMIN
- DMAX
- BREAK
- SPIRAL
- SLOPE
- DVAR
- OSO
- MILLCONE
- PORTION
- MATRES

- LTH
- TS
- MOA
- DMIN0
- IOF
- MVSY
- BFH
- DINC
- MINC
- MAO
- T:
- EZS
- ZI
- 4A-5A
- MT
- …..

<CUTHOLDERS>

This function is used for the trimming connector. It works with pocketing machining. It allows to completely trim the connector while following the shape of the crown.

There are three options:

<CUTHOLDERS>
<CUTHOLDERS1> is like <CUTHOLDERS> and it has the same behavior.

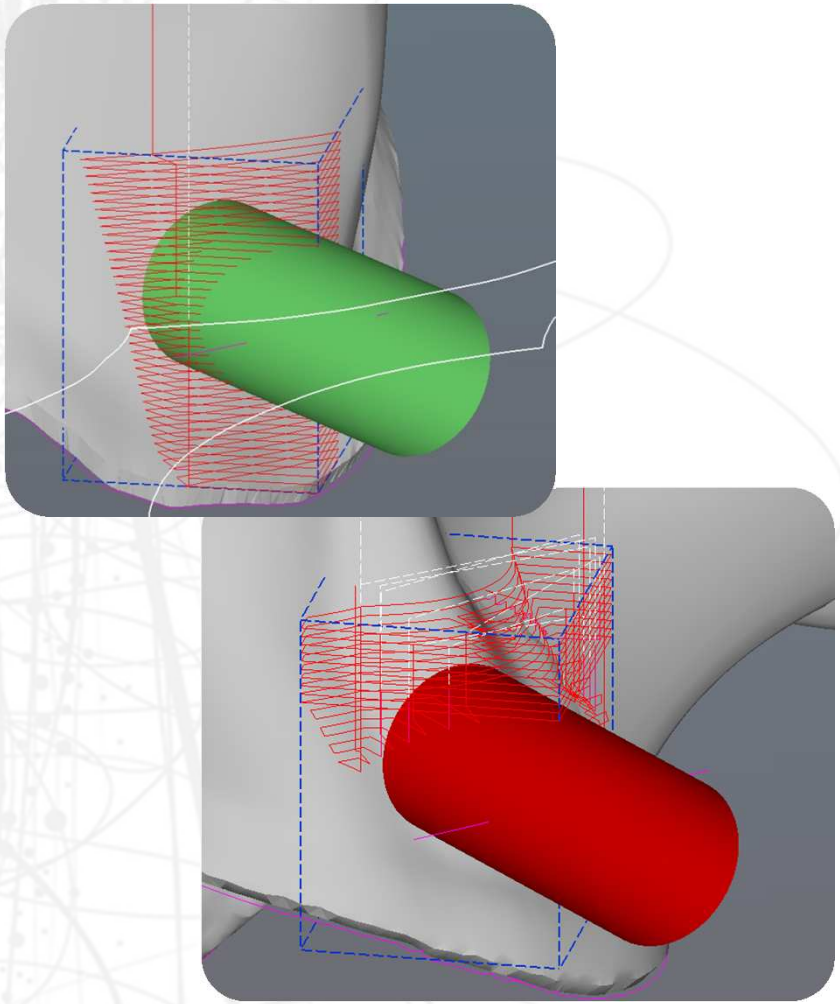
<CUTHOLDERS2> is used with the "red" links and can be used to make a partial trim in contact with the tooth.

<CUTHOLDERS>

There can be both of these options in the same strategy:
<CUTHOLDERS...>:
<CUTHOLDERS1> with a Z limit (for example between -2 and 2) ;
<CUTHOLDERS2> with a z limit 0 and 2.

In this case the red link will be reduced near to the tooth surface, without being cut.

Sample:
TRIM CONNECTOR<CUTHOLDERS>



<LOCK>

This function indicates that the parameter of machining and the selection of the curves / surfaces can not be automatically modified in the machining process.

<NOSEL>

This function indicates that the selection of surface can not be automatically modified in the machining process

<NOROT>

This function indicates that the machining is done in 3-axis without using optimization angle during the importation

<UNDERCUT>

This function works with Zig-Zag, Z-level Contouring or 3D Profile Pocketing machining.

With 3D Profile Pocketing it works in 5-axis in continuous and there are 4 parameters (always optional).

Example: <UNDERCUT20/20/0,5> max undercut 20°, start angle on border 20°, final angle (max depth) 0 degree, angular step on the XY 5 degree.

Besides these parameters it is possible to perform 5 axis continuous machining with different increments adding the <SPIRAL> parameter. For instance, it is possible to work with the codes: <UNDERCUT15/10/10><SPIRAL0/5/0.05><SPIRAL5/0/0.15> to machine the first 5 mm with step 0.05 and the rest of the cavity with step 0.15.

<UNDERCUT>

With Zig-Zag machining for both sides, it is possible to indicate where the material was milled from the opposite machining. To activate this option, the MZ (Mirror Z) parameter is required. Example: <UNDERCUT0.2,MZ>. The value is optional and it allows to manage different Z value increment. These values reduce the stress of the tool.

In case of 5 axis simultaneous axis cavity machining, the behavior can be optimized when the starting angle has not been defined.

The actual behavior suggests its use only by setting the maximum inclination angle, without the indication of the starting angle (eg.: <UNDERCUT10> instead of <UNDERCUT10,10>). The execution time will be reduced, maintaining the same machining quality.

Sample: INTERNAL FINISHING<UNDERCUT10/3/3>

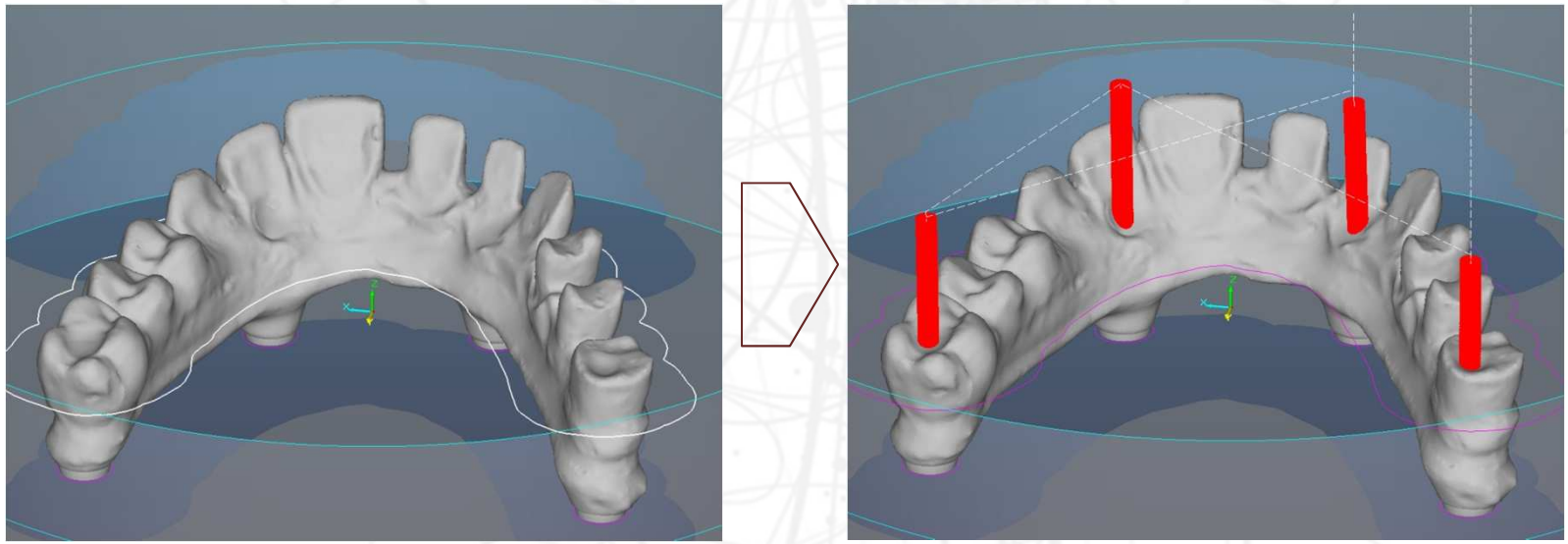
<DMIN> .. <DMAX>

This function indicates the minimum/maximum diameter for machining holes

<BREAK>

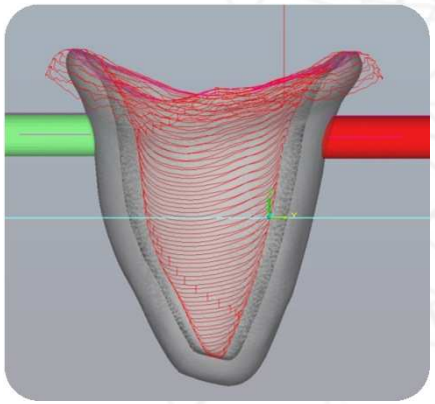
In SUM3Dental we can mill an Implant Bridge without cylindrical holes inside the model.

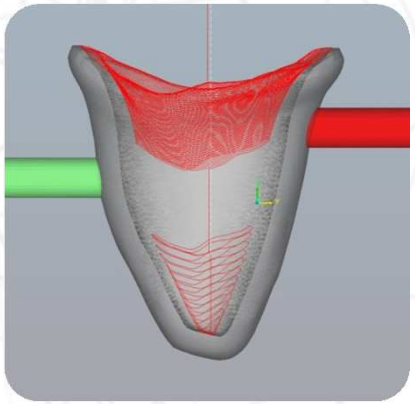
In machining name add the "BREAK" word and the software will calculate the drilling processes also without cylinders.



<SPIRAL>

Strategy for "3d curves pocketing", for a 3-axis continuous spiral machining. Either a "standard“ or a "lollipop“ tool with a thinner stick, can be used to eliminate the "undercut“ . This strategy also includes the possibility to set the distance from the border where to start and end the machining. For instance, it is possible to input: <SPIRAL0/5> or <SPIRAL2/10> or <SPIRAL5> or <SPIRAL-5>. Moreover, it is possible to create a machining with different zones and different increments. For instance, it is possible to input, inside the machining's name, the words: "<SPIRAL0/5/0.05><SPIRAL5/0/0.15>" to set which kind of machining to execute: with 0.05 mm increment for the first 5 millimeters of the cavity, starting from the "prep-line", and with 0.15mm increment for the rest of the cavity.





<SPIRAL>

The parameters of the "<SPIRAL...>" strategy are 3: start, end and increment. The start and the end can be negative, in this case the machining will begin from the back and not from the border. The third parameter (increment) is not mandatory, in case it is not specified the applied increment will be the value shown in the table of machining parameters. The increment value shown in the table of machining parameters must be always specified.

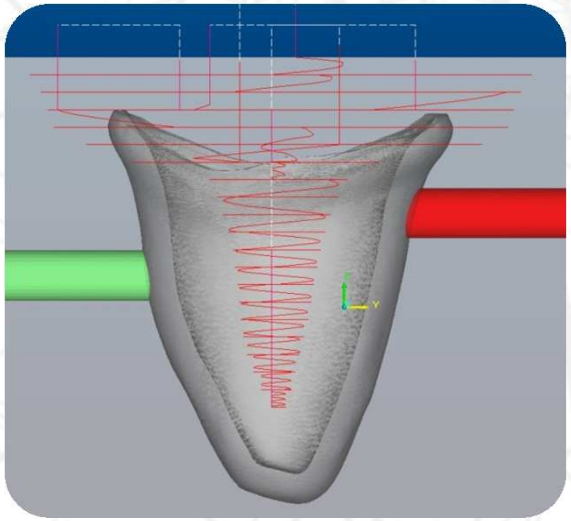
Sample: INTERNAL FINISHING<SPIRAL0/2/0.05><SPIRAL2/0/0.1>

<SLOPE>

This function is used in Z level Contouring and it allows to indicate two angles: one for the stock and one for the model. The machining will "draft" and the lateral side of tools will not be in contact with stock/model.

Example: <SLOPE5> indicates a descent angle of 5° to stock. <SLOPE2/20> or <SLOPE25/3> indicate two different angles.

Sample : INTERNAL ROUGH<SLOPE2>



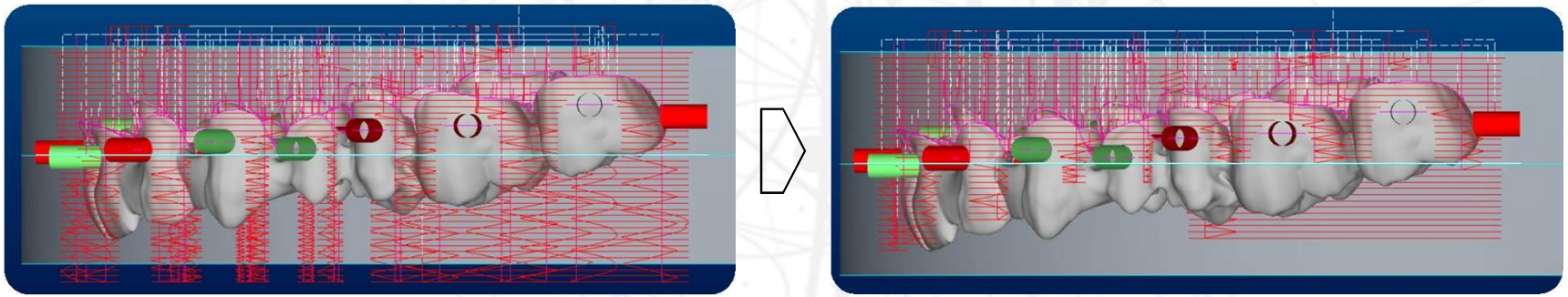
<DVAR>

This function is used in drilling operations and allows to add (subtract if it is negative) the value at hole diameters

<OSO>

Strategy that is used to mill the element until the end of the disk in only one side. This machining is used to generate a quick trial implant to test on the patient. To use this strategy add in the name the option <ONESIDEONLY> or <OSO>. Currently the supported machining types are: pocketing and Zlevel contouring. With this option the machining will be done until the end of the disk instead of the max external point.

Sample: RESTMATEXTFINISM&S5A<OSO><LTH><TS5,6><T:/D1B/REFMS5-ST-?>



<MILLCONE>

Strategy that allows to mill, with flat tip tools, the conical parts of the screws by means of a drilling machining.

When drilling with a spiral milling, the conical parts need specific tools with the tip shaped to fit the support part of the screw's head.

Alternatively, these parts can be milled by means of "XY contouring" machining, using flat tip tools . So it is possible to execute a drilling with a spiral mill using flat tip tools, obtaining a radial re-machining of the conical parts. To enable this operation, please write <MILLCONE> in the name of the machining or <MILLCONE,20> if, for example, you want to set 20 as the number of "radial" cuts to be performed.

Sample: PRE-HOLE SIDE 0<DMIN2.11><ZI1><MILLCONE,40><TS3,4,12>

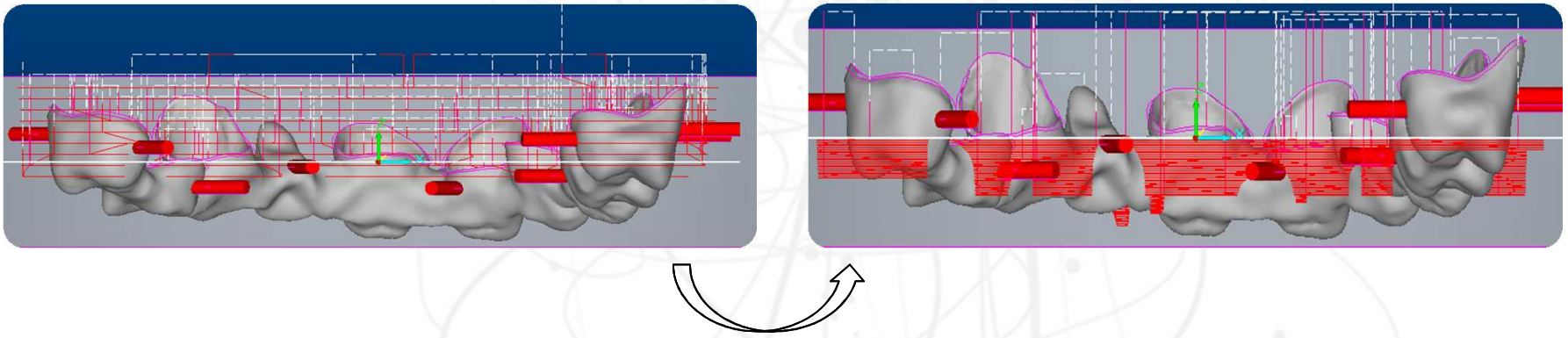
<PORTIONn/m>

This strategy in the pocketing phase allows to split the roughing into phases in Z.

The aim of this modification is to preserve the tool from breaking and overheating.

During the roughing of the first side, the tool tends to turn and go down rapidly in cavities, occasionally very small, as sometimes happens to the space between two connectors. In these cases the material (e.g.: Zirconium) which is not removed could "block" the tool, impeding the cooling and stressing it too much with the risk to break it, especially on high speed machines.

To avoid this problem it is now possible to split the roughing of the same side in "phases" and to perform the roughing on the opposite side between these "phases". The aim is to make a hole in the disk and consent to get back on the previous side to go on with the machining avoiding a dangerous overstock of the material removed.



<PORTIONn/m>

To set the "phase“, it is necessary to input the code <PORTIONn/m> in the name of the machining, where 'n' sets the number of the phase and 'm' sets the number of total phases. For instance: <PORTION1/2> or <PORTION2/2>

It is also possible to perform a further split of the portion, in order to machine the first millimeters with parameters different from the rest of the portion. To reach this aim, the code DEEP (or DEPTH) has been added that allows to set the depth of the machining to perform (positive value) or the depth of the machining to skip (negative value). For instance, <PORTION1/2,DEPTH1.5> will machine only the first 1.5 millimeters of the portion 1, while <PORTION1/2,DEPTH-1.5> > will machine the portion 1 avoiding the first 1.5 millimeters.

The code DEEP (or DEPTH) can also be performed alone (without <PORTION...>). E.g.: <DEEP2.3>

<PORTIONn/m>

Example of machining sequence:

- roughing of the cavities side with <PORTION1/2>
- roughing of the opposite side, without codes "<PORTION...>"
- roughing of the part still to be machined of the cavities' side, with <PORTION2/2>
- other machinings

Otherwise:

- roughing of the first portion of the cavities side with <PORTION1/2>
- eventual elimination of the stock interference with the axes of the cavity, to free the access to the cavities with strong inclination
- roughing of the cavities
- finishing of the borders
- finishing of the cavities
- roughing of the opposite side, without codes "<PORTION...>"
- roughing of the part still to be machined of the cavities' side, with <PORTION2/2>
- finishing of both sides

<MATRES>

For the "pocketing" machining there has been added the chance to set the tolerance to create the dynamic raw. The abbreviated form <MATRES...> added in the name of the machining can cause two effects:

1) to reset the dynamic raw.

2) to assign the value set as tolerance for the further eventual update of the machined model (the "dynamic raw").

For instance <MATRES0.04> means that before performing the machining, the "dynamic raw" will be reset and the tolerance of 0.04mm will be assigned to the next operation of update of the machined model.

<MATRES>

This new option brings two main advantages:

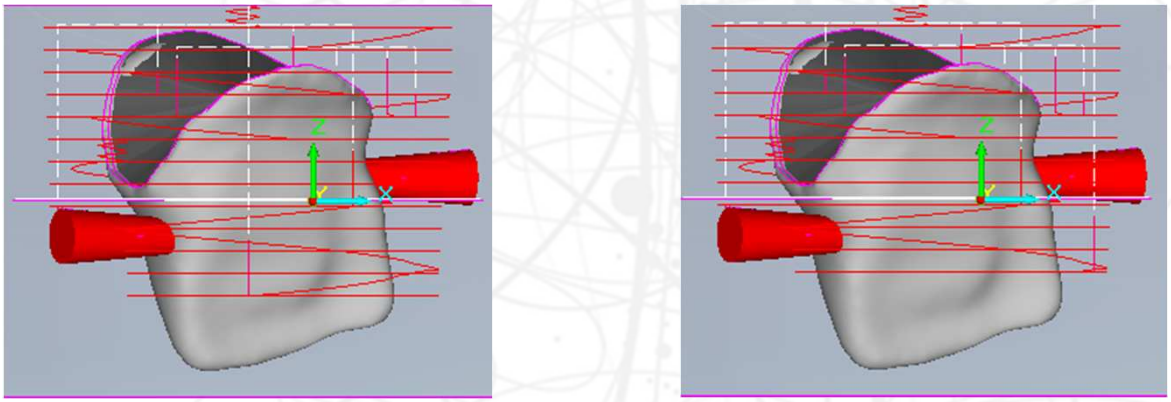
a) to manage different tolerances in different situations, with the purpose to apply always the best possible choice between processing time and precision of results. This allows to set a higher tolerance value to manage the "dynamic raw" only during roughing, or to set a lower tolerance value to manage the "dynamic raw" in finishing strategies (identification of the rest material in finishing).

b) to reset the "dynamic raw" in the different phases of the machining. For instance: to perform a general roughing using tools with different diameter, which identify the rest material to be removed on the basis of the condition of the model. Then to "reset" the machined model to perform the roughing with different tools inside the cavity, eventually reducing the tolerance of the "dynamic raw" and working with a model that is lighter than the one obtained at the end of the general roughings.

<LTH>

When working with pocketing or XY contouring, this strategy allows the deep machining until the center of the connectors and not any further. In case the object has areas to be pocket-machined on both sides, this feature allows to stop at the connectors preventing the machining to reach the end. In this way you will avoid the tool-paths without removal when roughing the second part.

Sample: RESTMATEXTFINISM&S5A<OSO><LTH><TS5,6><T:/D1B/REFMS5-ST-?>



<TYPESEL / TS>

During the import of the strategies, any operations that require <TS...> (or <TYPESEL...>) that are not included in the working file, have been deleted.

The values which can be associated to the <TYPESEL...> command are:

1 for single crowns

2 for bridges

3 for abutments

4 for implant bridges

5 for models

6 for stumps

7 for single ceramic-glass teeth

8 for the inlays-onlays-Veneers

9 for the anatomic crowns

10 for the anatomic bridges

11 for Bite Splints

12 for anatomic bridge for implant

13 for Framework

14 for telescopic (only from Exocad)

15 for bar

Moreover, the possibility to use the code <TYPESEL-...> (or <TS-...>) has been added to exclude object types applied to the machining, instead of selecting them.

For instance:

<TS1,2> sets a machining to be performed only to the items kind 1 and 2.

<TS-1,-2> sets a machining to be performed to all items, except the kind 1 and 2.

<DMIN0>

To avoid to go deep in the hole, while working on the interface's opposite side during the implants' drilling function, there has been added the chance to set <DMIN0> to prevent the machining of the part of the hole with the lower diameter, corresponding to the one through the hole.

<IOF>

For some machining like pocketing, contouring, 3D curves' pocketing and zig-zag, the <IOF...> code is active in order to apply to the "internal" delimitation curves (e.g.: the margin lines) an offset different from the one set in the offset parameter for the selected delimitation curves (in this case it will be applied only to the "external" curve). This parameter will be useful in case the offset for the usual "external" curves (e.g.: "-DM-0-") is different from the one to be applied to the internal curves (e.g.: margin lines).

For example, in the bridge's finishing, using a tool whose diameter is 1, the offset could be -0.8, considering the eventual "<SLOPE...>" used in the roughing, but you could decide to apply a lower offset (e.g.: <IOF-0.2>) to the "internal" margin curves to be sure to also machine the bridge's parts close to the margin lines.

Sample: BRIDGE FINISHING<IOF-0.3>

<MSVY / MVSN>

MILLVERTICALSIDEYES> or <MVSY> and <MILLVERTICALSIDENO> or <MVSN>

These codes have been introduced to be used in "roughing" and " XY contouring" for the optimization of the models' machining. They are used in case you want to calculate, with <MVSY>, the tool-path of vertical areas (in Z) or in case you want to exclude these zones from the machining with <MVSN>. Usually the machining is calculated both from upside and from downside, to reach the "middle Z level" of the item. Some geometrical items, like bars or implants, could present large vertical areas.

SUM3D Dental includes these areas in the machining "side 180" (the one which is generally performed first). In some strategies it could be necessary to invert this behavior so, to this end, we added the possibility to set these codes.

<BFH>

This code is applicable in "roughing" and " XY contouring“ in order to start the machining from the deepest connector. This code could be useful, for example, to perform the roughing on one side with the code <LTH> and then a "XY contouring" (which is faster) starting from the end of the roughing, setting <BFH>.

<DINC>

It was noticed that during the roughing, when the tool dips in blind-end narrow cavities, it overheats and/or breaks because it cannot output the rest material. For this reason, an option was introduced to apply a "peck drilling cycle" by setting <DINC...> (Depth Increment for Narrow Cavities). "<DINC…>" // Depth Increment for Narrow Cavities allows to set a "peck drilling cycle" when the tool descends in narrow and deep cavities. The set value represents the "Z" step increment to be applied for the "peck drilling cycle". When the tool descends in very narrow cavities, every "xxx" millimeters (value set by <DINC...>) of milled depth, the tool rises outside the cavity and then descends again to the previous milled Z level, to reach the lower preset raising point. By means of this raising the tool can cool down and output devices (water, air or suction) can clear the chip. Also in case the "<DINC...>" parameter is not set, a "default" value of one/third of the tool's diameter will be applied.

Sample: CROWN ROUGH<DINC0.2>

<MINC>

To complete the options given from the <DINC...> code during the "pocketing" operations, it is now possible to set the code <MINC...> to specify a minimum "diameter" of the cavities to mill. For instance, writing "<MINC5>" in the "pocketing" name, the cavities with a diameter lower than 5mm will not be milled. We introduced this option to avoid to descend into cavities which are just a little bit larger than the tool. It will be possible to mill those cavities later on with a lower diameter's tool, which uses the "dynamic rough " to detect the areas where the previous tool has left the material

<T:/>

A new “keyword”, recognizable by the machine name has been added to be able to recall a tool from the tools archive: <T:.....>.

After the sign ':' and before '>' there is the "name" (description) of the tool that you want to "call" from the tools archive.

A tool with the "name" requested from those assigned to the selected machine tool, or from those that do not have it.

Since every tool can have different cutting parameter for several material, from the archive, the tools associated to the related material have been extracted.

This code has been added in order to simplify the strategy changes, intervening on the tools archive instead of the strategy files, to change the tool length or the cutting parameters of the recalled tools.

If the called tools are not in the archive, a table with the with the list of missing tools is shown and the machining is not started.

<T:/>

Using codes TOOLSTABLE and REPOTOOLS, now is possible to use the same tool number with name partially different.

Comparing the tool names (description) the text after "/" is ignored (including the "/" char).

It means that two tools with description "D1/A" and "D1/B", with the use of TOOLSTABLE and REPOTOOLS, are considered as the same tool.

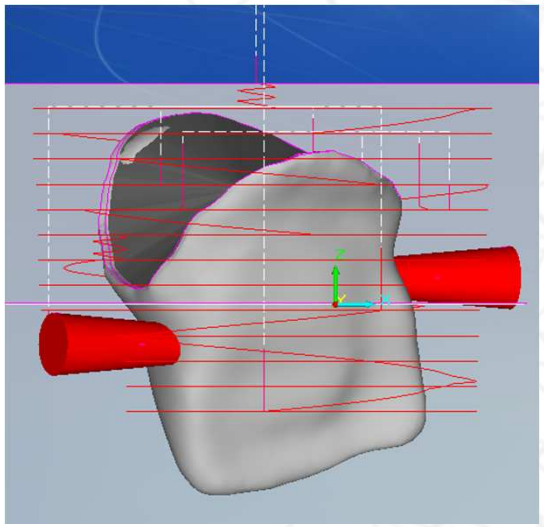
This possibility has been added in order to recall the same tool with different parameters using the "<T:....>" code. The use of several copies of the same tool of the tool archive, is needed when you want to use it for different machining, applying different machining parameters.

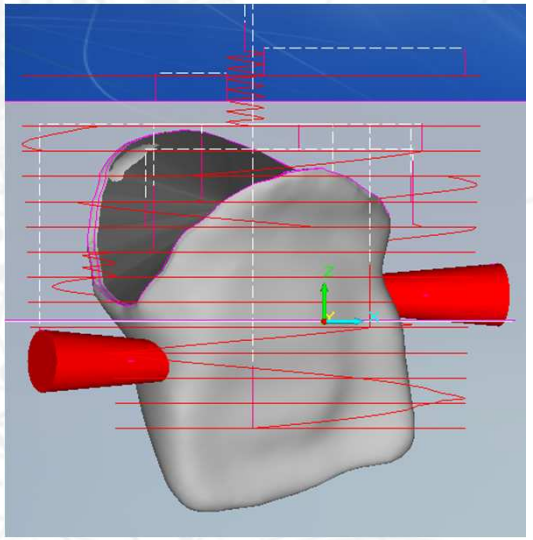
Sample: FINISHING EDGE<FF><TS3,4,12,15><T:/D06F/FE-SP-?>

<EZS>

This code (Extend Z Start) allows to start the roughing machining a bit higher. The value set is used to extend the Z of roughing machining start. It could be useful if you suspect the use of irregular raw part.

Sample: INTERNAL ROUGH<SLOPE2><EZS0.3>





<ZI>

In case of deep holes milling, the tool overheating could happen. To help its cooling, the code <ZI…> has been added in the holes milling. If this parameter is set, an up movement to the hole starting level is applied at every achievement depth corresponding to the set value. For example, <ZI3> indicates that every 3 millimeters of depth, an up movement was applied before continuing with the machining.

Sample: PRE-HOLE SIDE 180<TS3,4,12><ZI1><FF><T:/D1F/PH180-SP-?>

<4A/5A>

In order to use the same “strategy” both on 5 and 4 axis machines, we have added the possibility to set in the name of the machining the abbreviations:

“<4A>” for the machining that must be used only on 4 axis machines.

“<5A>” for the machining that must be used only on 5 axis machines.

For example, if a specific operation (e.g.: finishing of internal cavity of a crown) has to be executed in a different way on 4 axis and 5 axis machines, it is possible to set both operations in the same “strategy”, setting the abbreviation “<4A>” in the name of the machining that has to be used on 4 axis machines, and “<A5>” in the name of the machining that has to be used on 5 axis machines.

Sample: INT FIN 5 AXIS CONT<FF><UNDERCUT10><5A><TS1,2,9,10,14><T:/D1B/IF5-?>

<MT>

In the name of machining, the <MT:...> code has been added, in order to point out that the machining is applicable (or not) to the listed materials. The name of material must be written using the following acronym:

Zirconium: zr
Zirconium translucent: zrt
Pmma: pmma
Composit Pmma: pmmac
Wax: wax
Polyurethane: pu
Gypsum: gyps
Titanium: ti
Titanium grade 2: ti2
Titanium grade 5: ti5
Chrome cobalt: crco
Aluminium: al
Glass ceramic: gc
Ultimate: ult

<MT>

It is possible to set several materials, separated by a comma. To specify that the machining is not applicable to a material, you have to set the letter '-' before the name of material.

Examples:

<MT:zr,zrt>
<MT:pmma>
<MT:-pmma>
<MT:-ti,-ti2,-crco,-al>

Sample: INTERNAL ROUGH<#STANDARD//1><RR><SLOPE2><MT:-PMMA><T:/D2B/IR-ST-?>

Contact us

Luigi Fanin
fanin@cimsystem.com
CIMSYSTEM s.r.l.
Via Monfalcone, 3
20092 Cinisello Balsamo (MI) Italy

Phone +39 02 87213185
Fax +39 02 6129306
Web Site:
www.sum3ddental.com
www.cimsystem.com
www.rhinoplugins.com
Technical Support : support@cimsystem.com