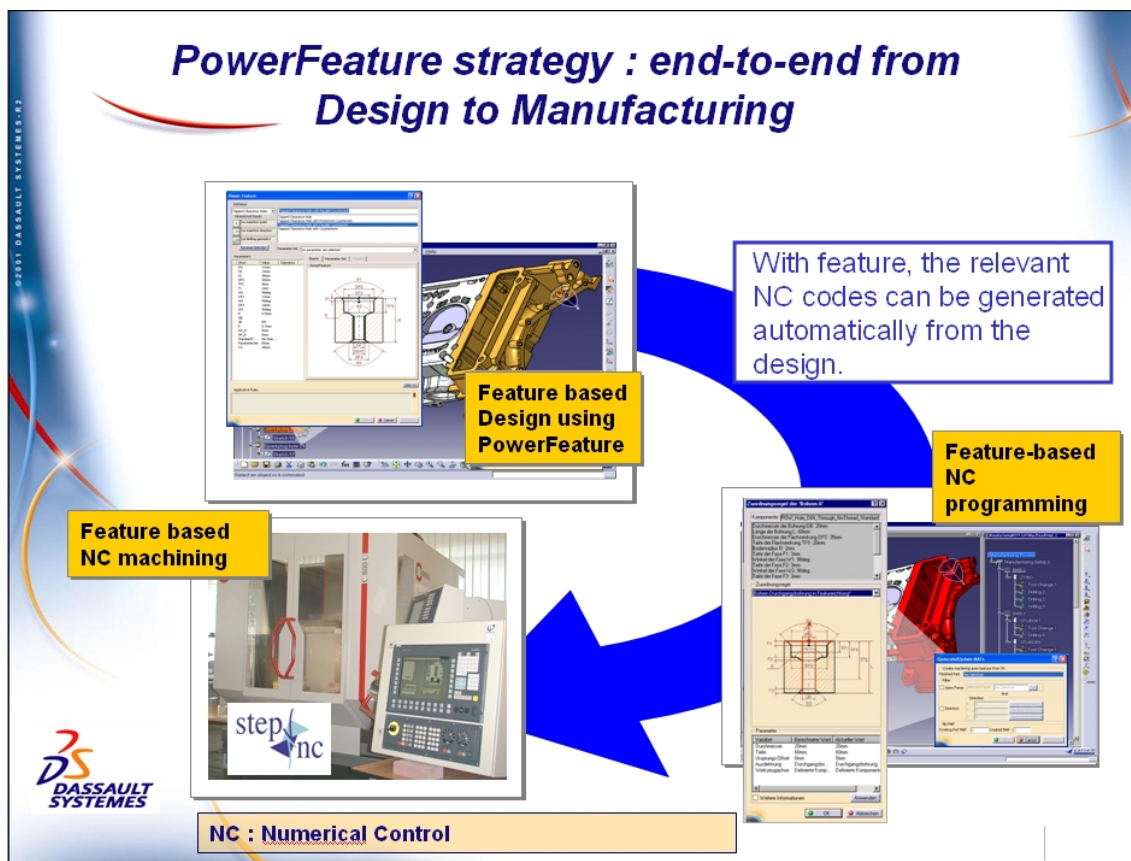


PowerFeature (PF9)

PowerFeature user's guide



BPA Delivery 6 for V5R19 (V5.6) Version 1.4

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1 Introduction

PowerFeature is a CATIA V5 tool which allows designers to implement holes features in an efficient way that makes it possible to define a construction CATIA V5 and to preserve the logic of the process design manufacture while passing by the quality control by respecting as well the international standards as those interns with the company.

2 PowerFeature User's Guide

The PowerFeature tool allows you to implement holes from a catalog into your 3D document. First we will describe the function's tool, then the concept of PowerFeature and then we will go through an example.

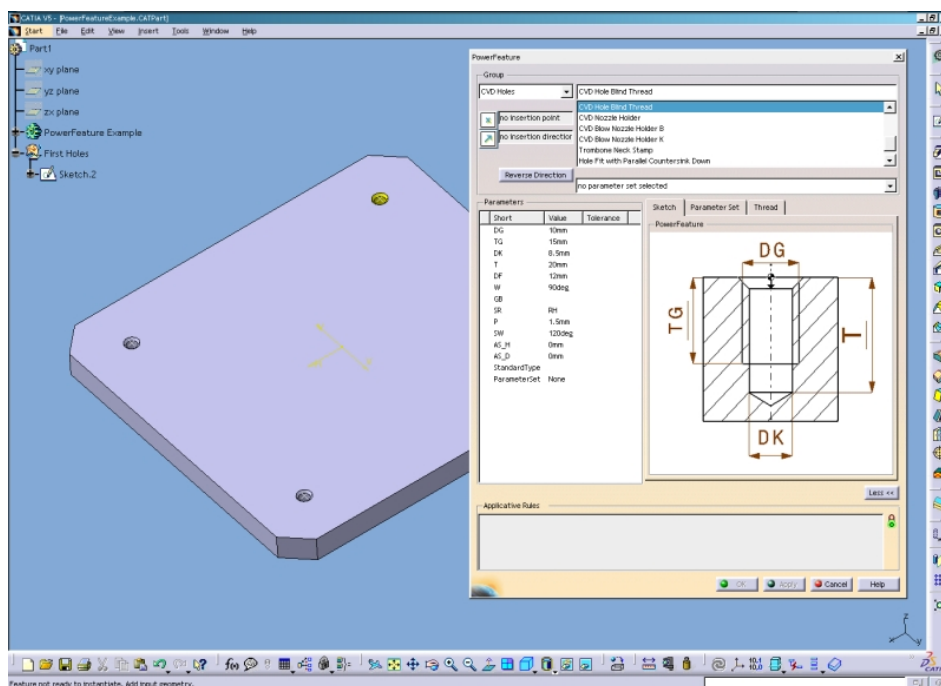
2.1 PowerFeature Tool

The PowerFeature tool is available in Part Design Workbench. It is completely integrated in CATIA V5 environment.

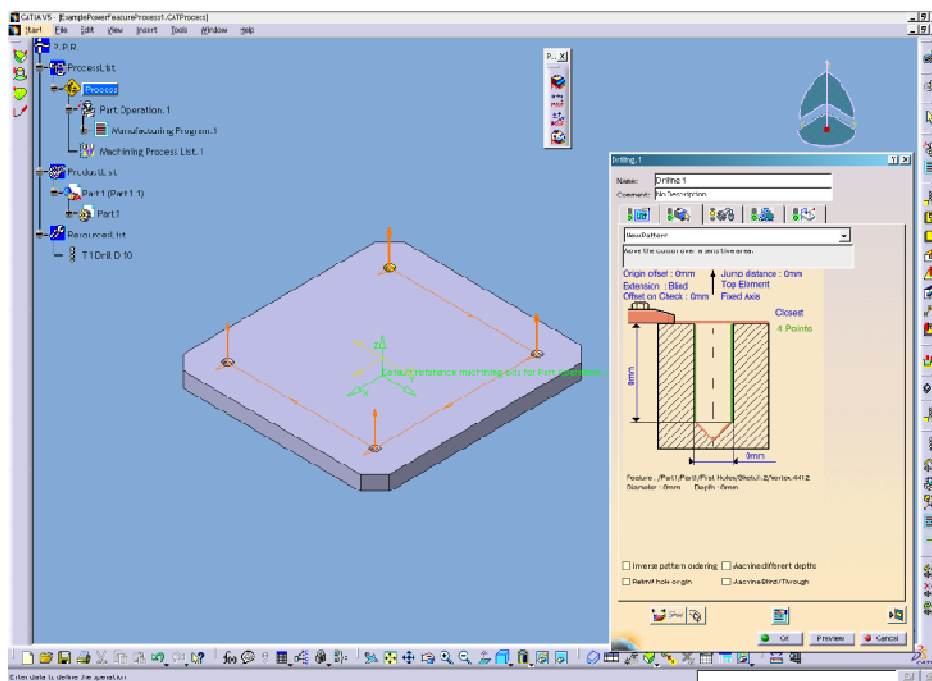
The application is available for the following operating systems: windows XP, AIX, HP-UX, SUN. For more information read installation guide.

Example of process:

Definition of the holes in the conception context



Extraction of information in the machining context

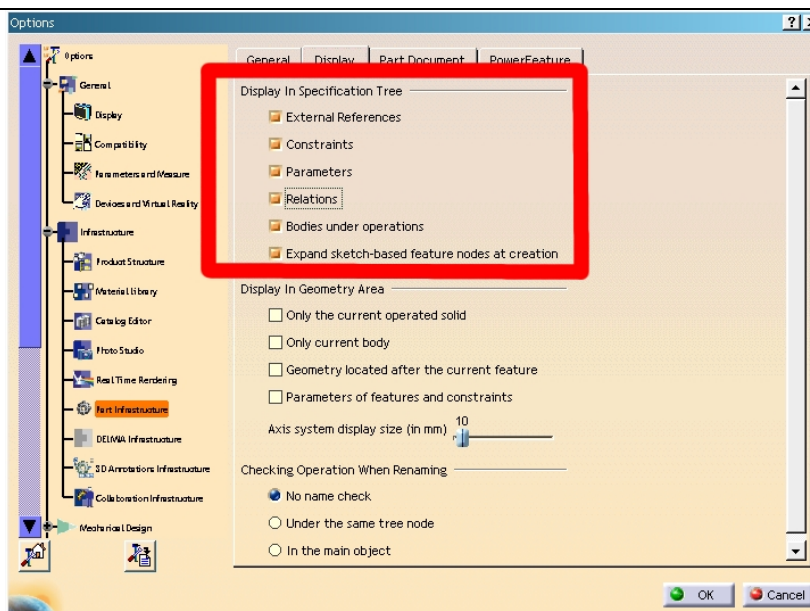


PowerFeature – Business Process Accelerator
User Guide R19D6 v1.4

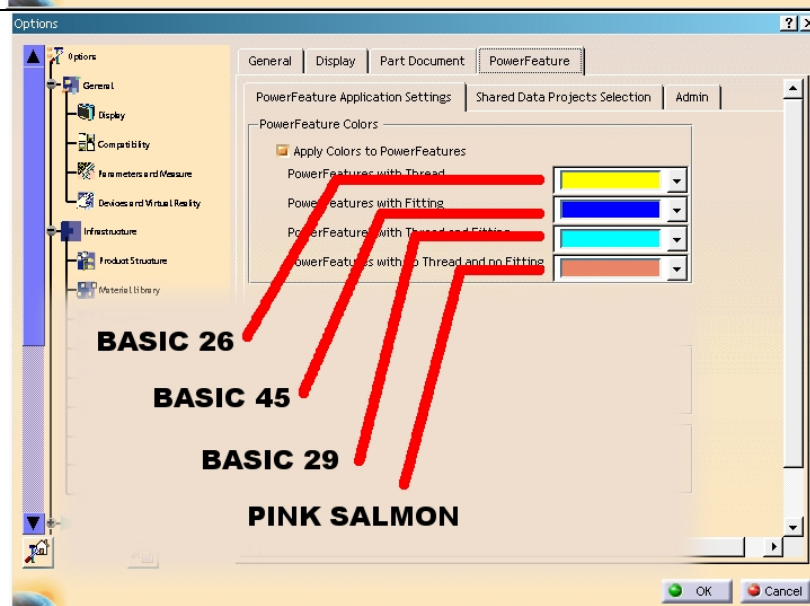
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2.2 Options in Part Design Workbench

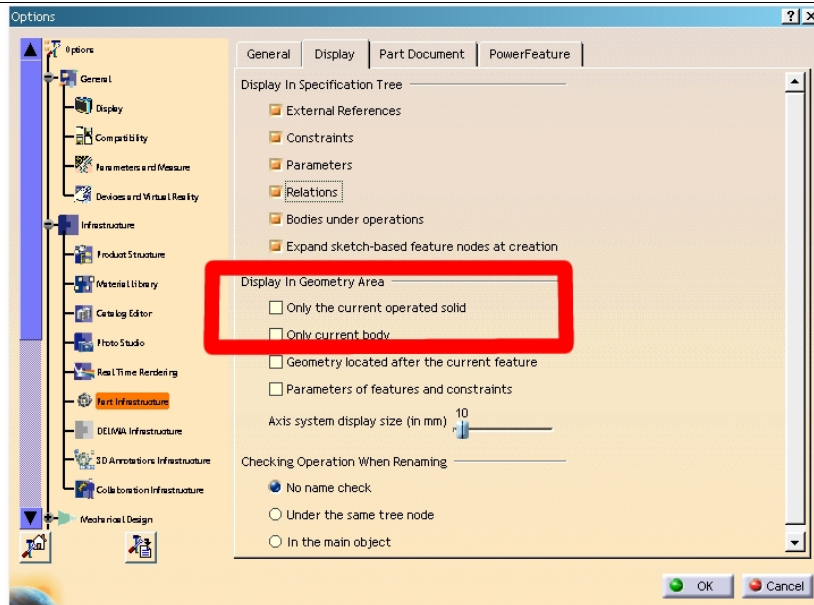
All options below
„display in
Specification Tree
“are to be checked



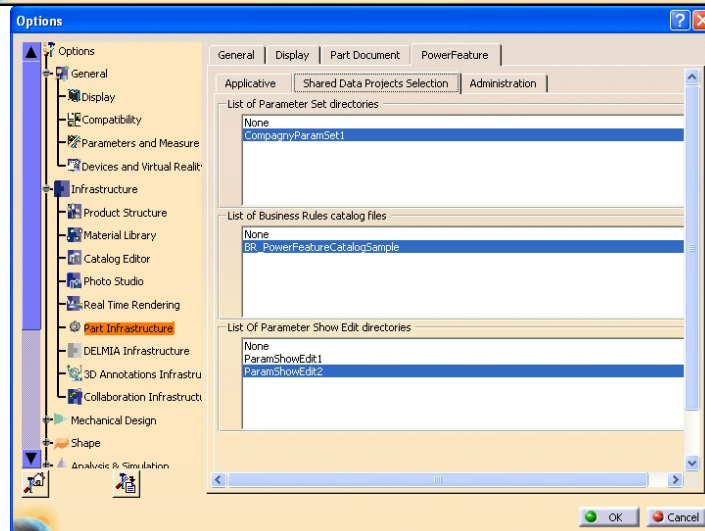
Realized due to a
requirement from
CAM process.
Preset colors are
internal and in the
VDA committee
coordinate (are
suitable also for
colorblind).
This option must be
checked.
Definitions of the
colors are not to be
changed by the user
(standardization)



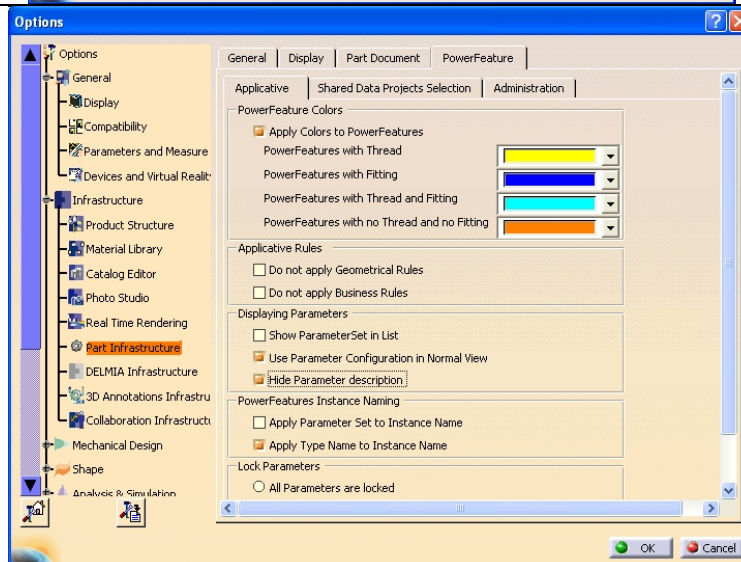
You can check the following options



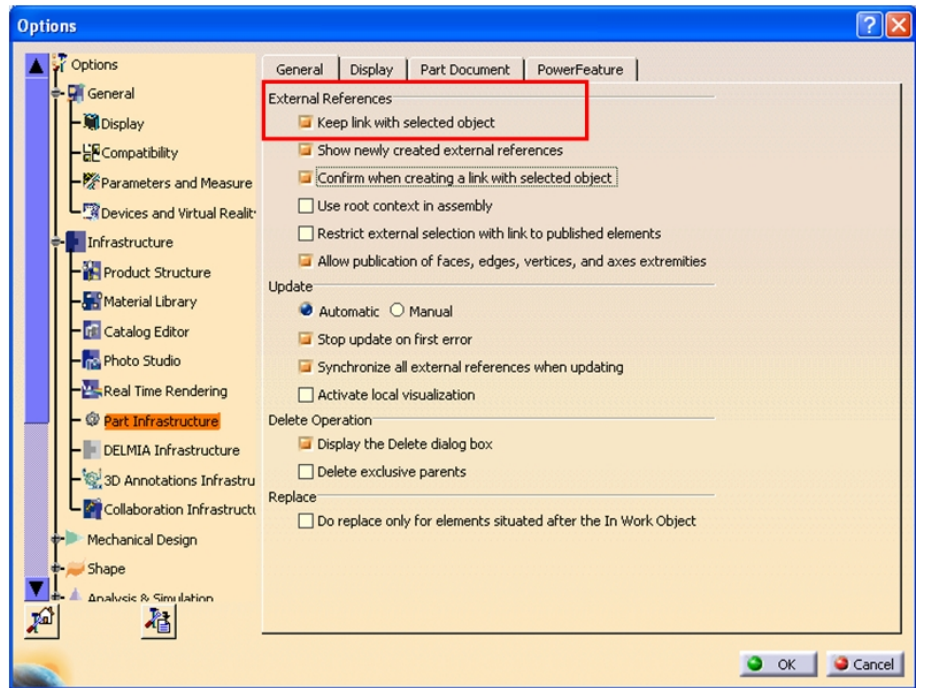
You can see that the power feature has a set of parameters (defined by a name, a value and a tolerance). For each type of power feature, the set of parameters may be different. This applies also to “Applicative Rules” and “parameter show Edit” (if these are available)



Options concerning the way is displayed the parameters for the PowerFeature and in the specification tree



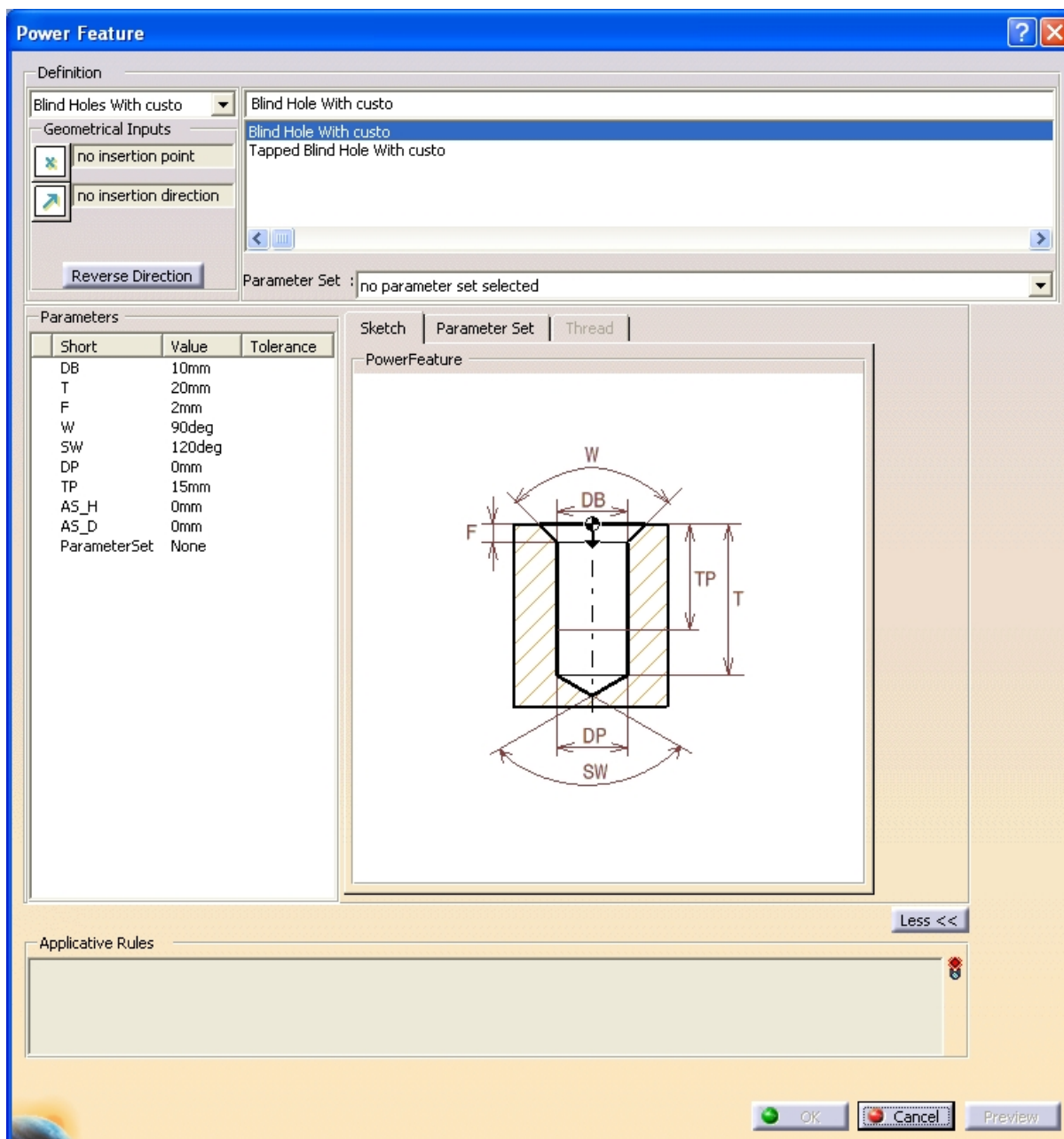
Check “Keep link with selected object” when necessary



2.3 The PowerFeature Tool



When the PowerFeature tool is selected, the main window below is displayed and will be explained:



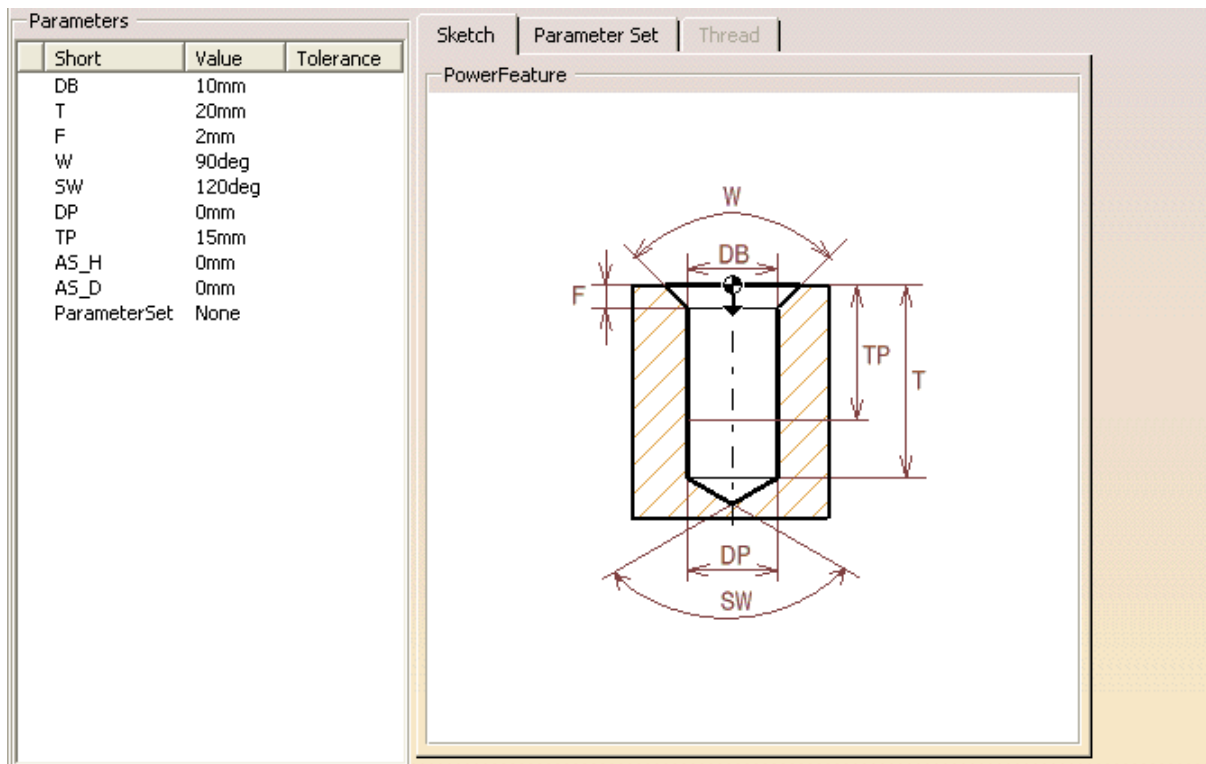
2.3.1 The Group

Expert designer provide to end user a set of group.

This set depends of each company. For example TRANSCAT provide DIN group.

2.3.2 The parameter Set

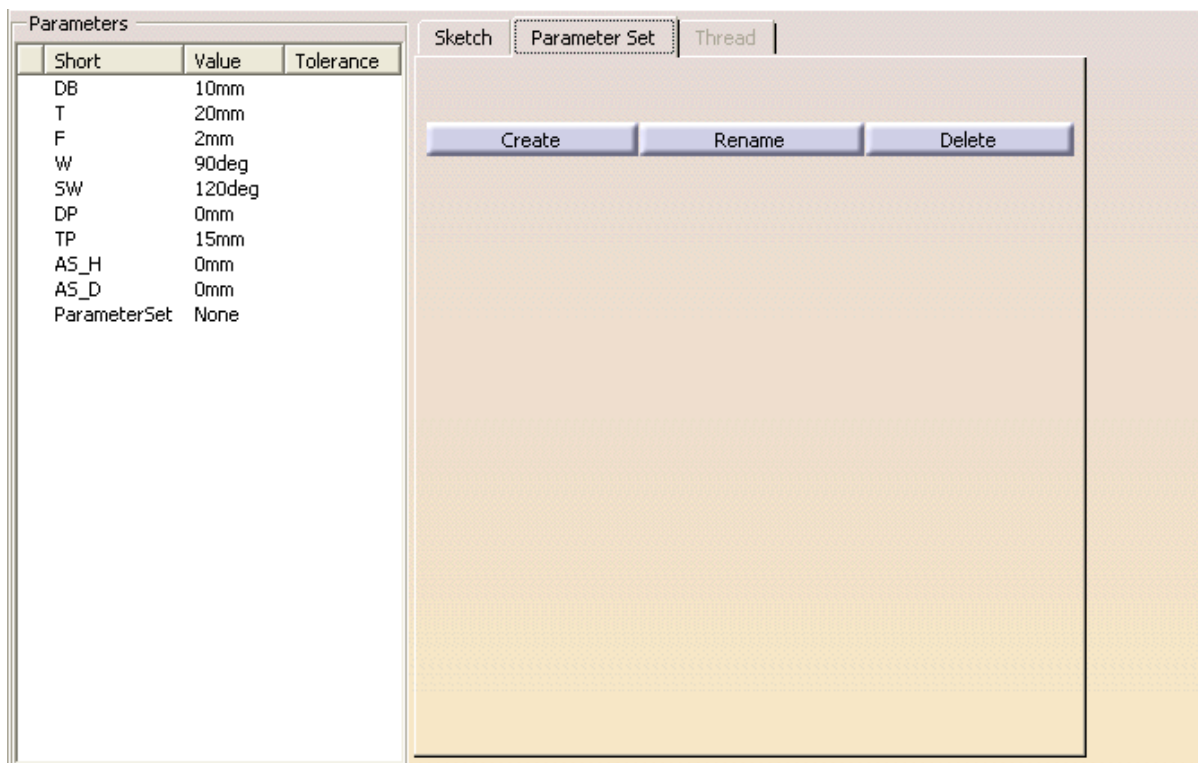
2.3.2.1 Sketch tab



There is a specific set of parameters for each types of PowerFeature. Each parameter is shown in the graphic.

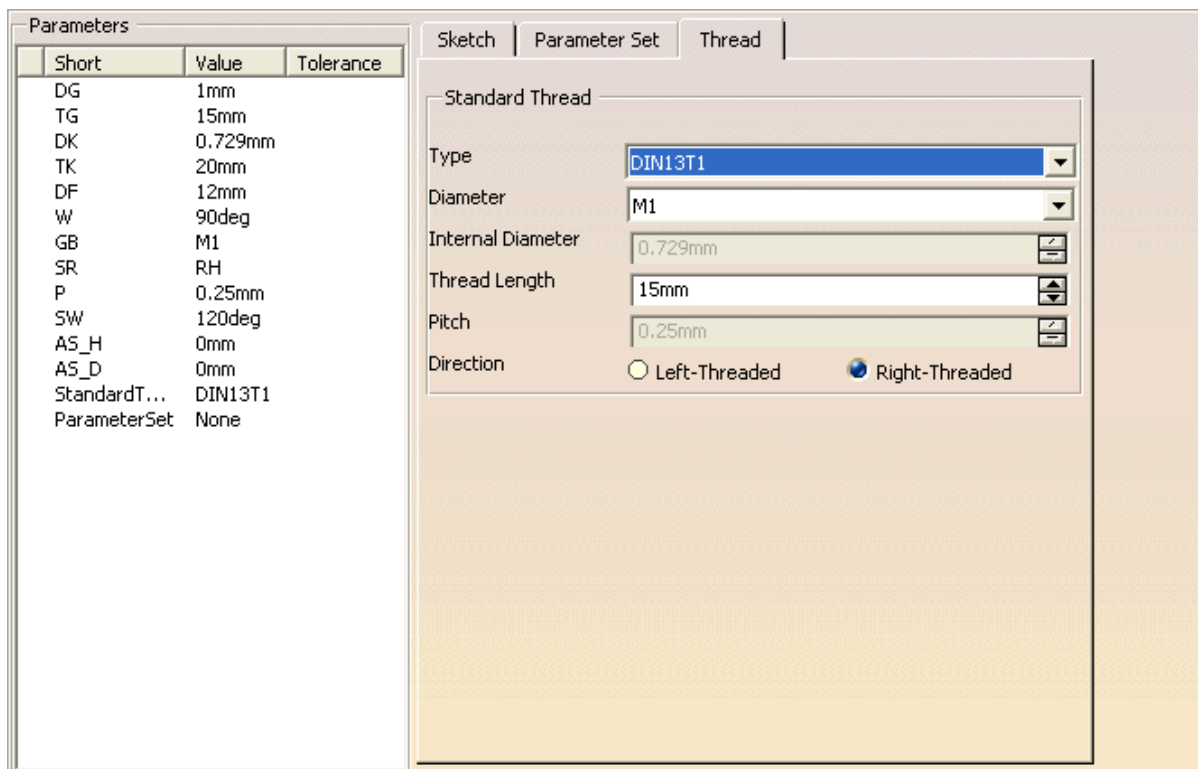
Each value of the parameters is editable so that you can suit the UDF to your needs.

2.3.2.2 Parameter Set Tab



You can create, rename, and delete your own parameter sets.

2.3.2.3 Thread Tab



The thread properties for some UDF can be managed and some standard are available.

2.3.3 Input geometry

The following table show you which geometry is allowed to define a PowerFeature:

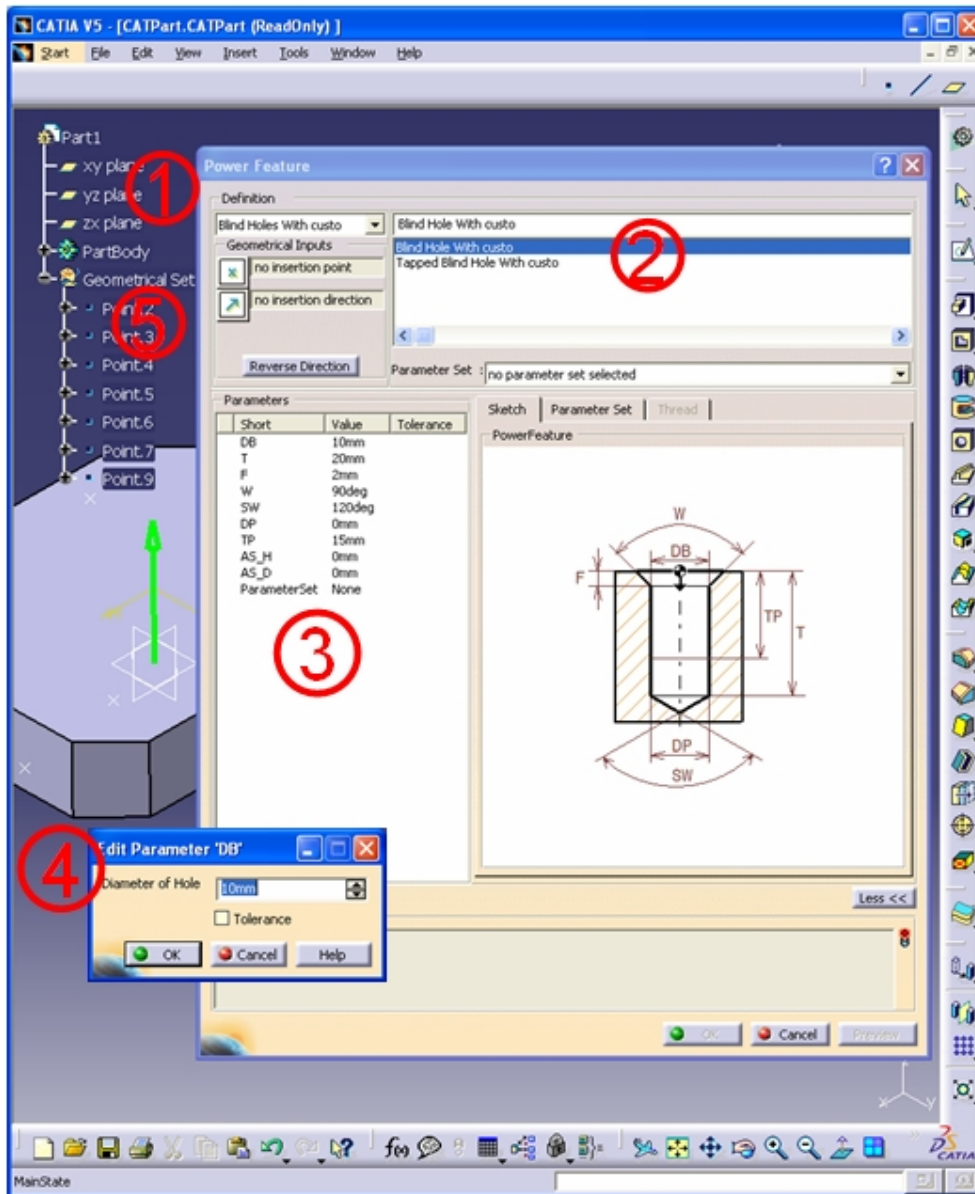
Input Point	Input Direction
Sketch (S)	Plane / Planar Face (F)
	Edge (E)
	2-D Line (L)
	3-D Line or output line of a sketch (L1)
2-D Point (P) in multi-point Sketch (S)	
OR	
One end 2-D Point (P) of a Line	Plane / Planar Face (F)
OR	
Intersection/Projection point (P)	Edge (E)
	2-D Line (L)
	3-D Line or output line of a sketch (L1)
3-D Point or output point of a sketch (P)	Plane / Planar Face (F)
	Edge (E)
	2-D Line (L)
	3-D Line or output line of a sketch (L1)
Vertex (V)	Plane / Planar Face (F)
	Edge (E)
	2-D Line (L)
	3-D Line or output line of a sketch (L1)

2.4 PowerFeature Process

The following example will show you the process to implement a PowerFeature.

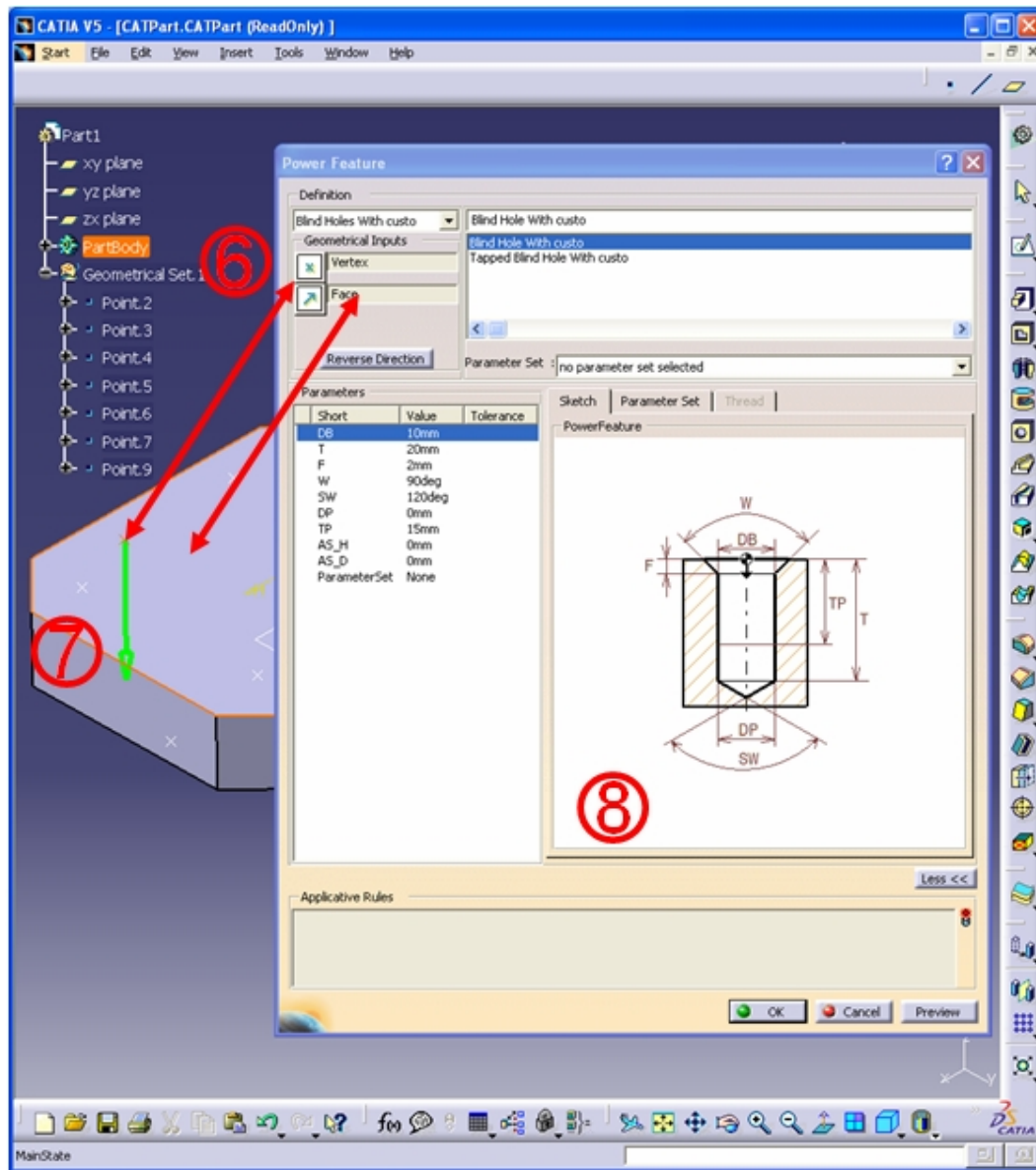
A simple pad and a point in a sketch have been designed and we have activated the PowerFeature tool.

2.4.1 Implementation



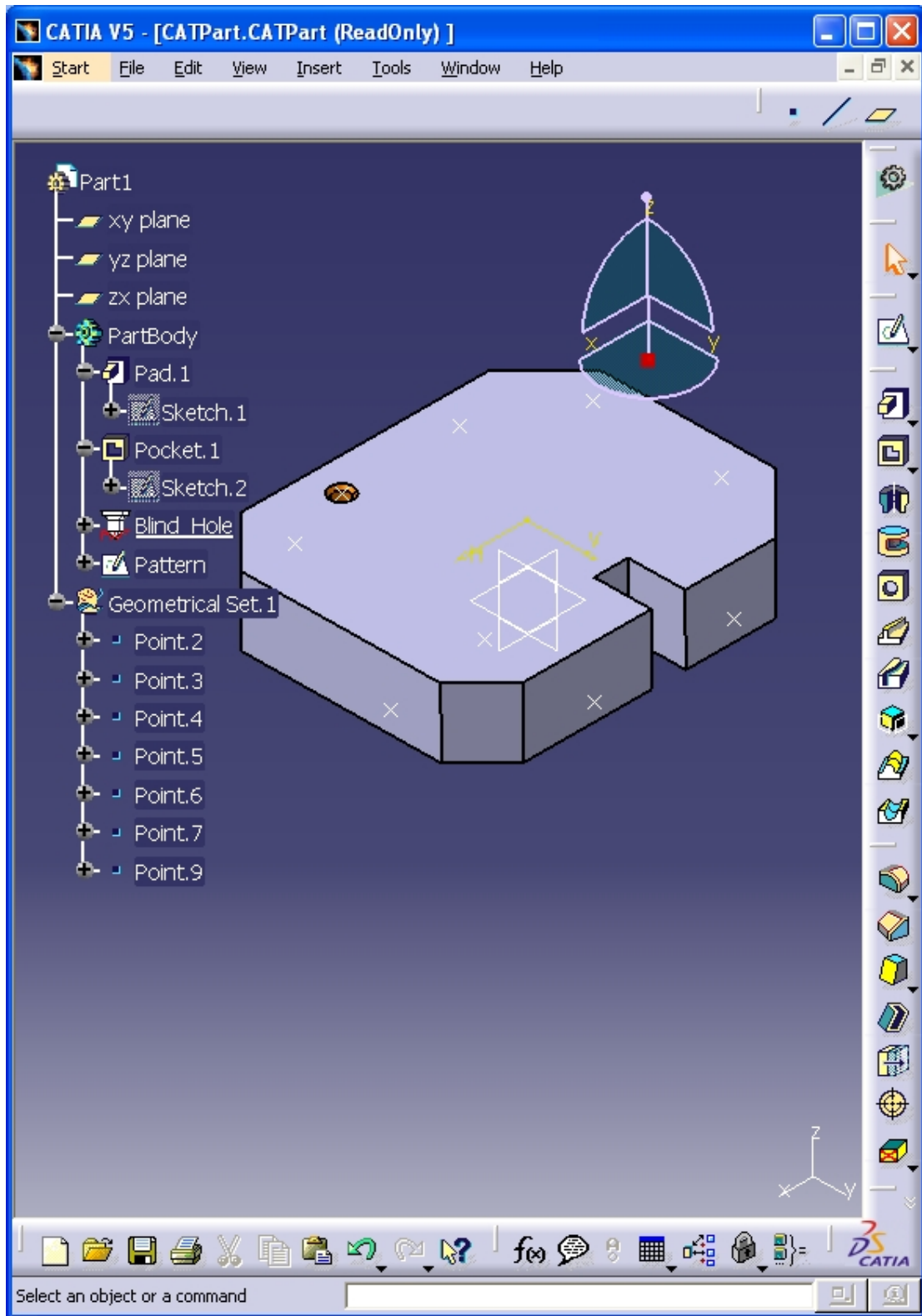
1. Select the PowerFeature group to implement
2. Select the type of the PowerFeature
3. Select one or more variables to edit their values if needed.
4. Modify the value parameters

5. Indication of selected geometry



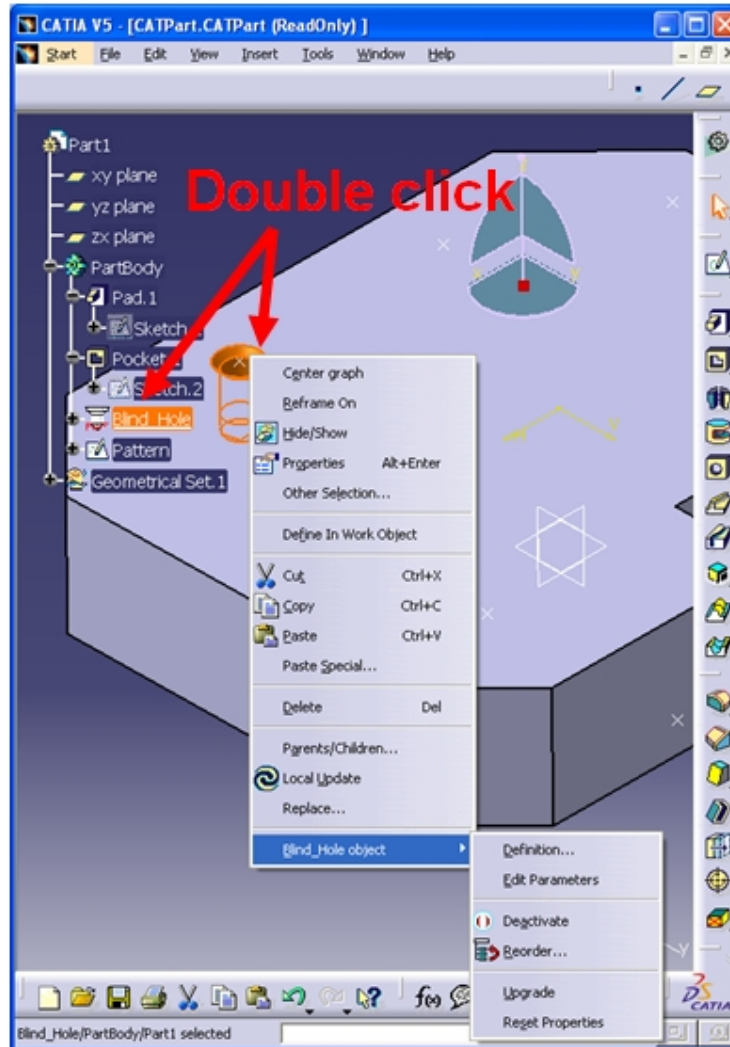
6. A vertex and a face were selected. The vertex is a 2D point created in a sketch, and the face is belonging to the pad
7. Verify that the arrow has the good direction (This direction has to go towards the matter).
8. If you modify the value of a parameter you can verify graphically the good sense of it thanks to picture beside the parameters

9. By clicking on OK, the PowerFeature is implemented in the geometry and in the specification tree. The name in the tree corresponds to the type of the PowerFeature.

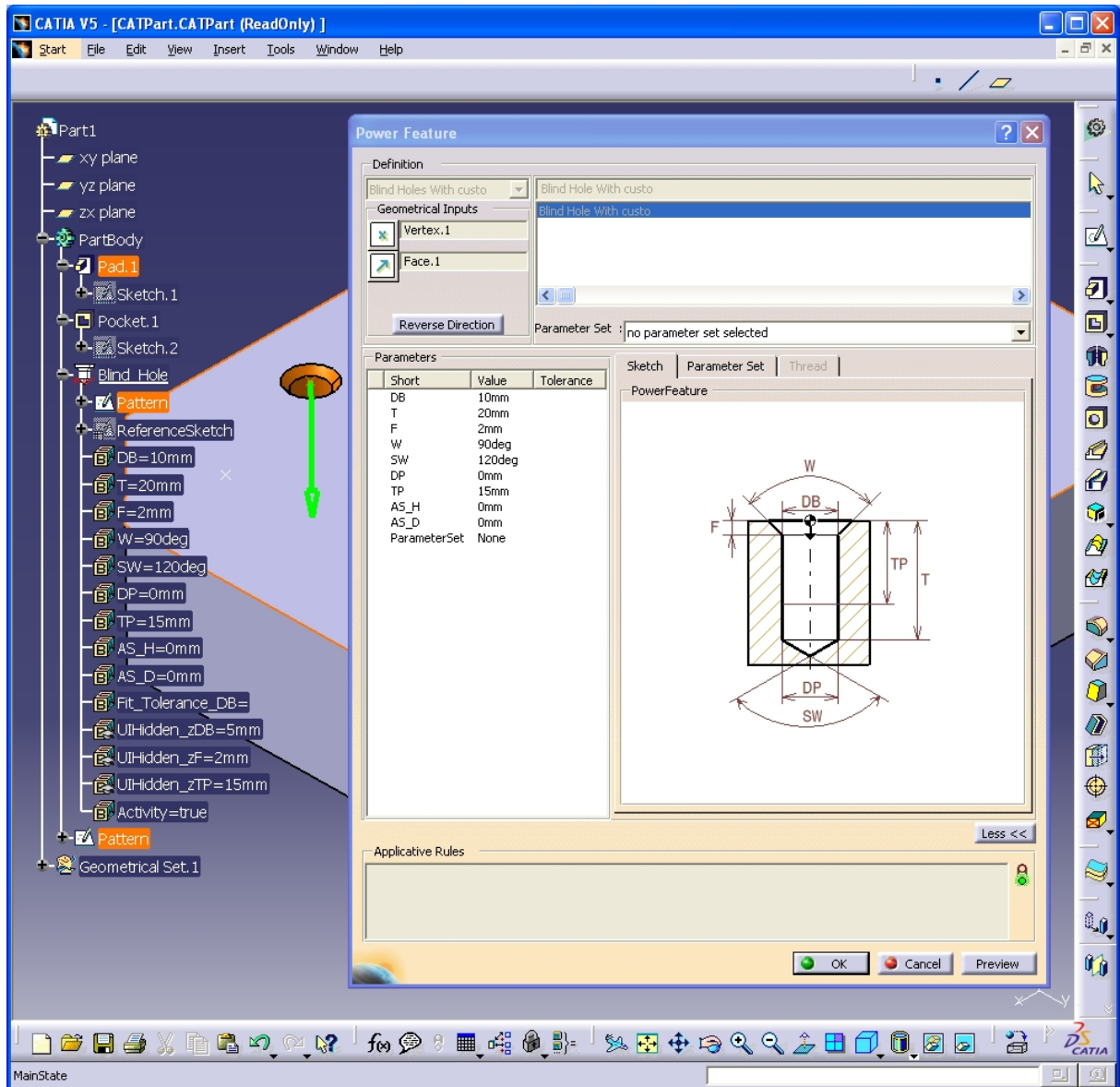


2.4.2 Modification of a PowerFeature

1. The PowerFeature can be edited by double-click on it in the 3D screen or in the specification tree, or by contextual menu on it.



- Value parameters can be modified. Three parameters have been modified. Then click on “Apply” to update the values and to check the Applicative rules. Click “OK” to confirm the choice.



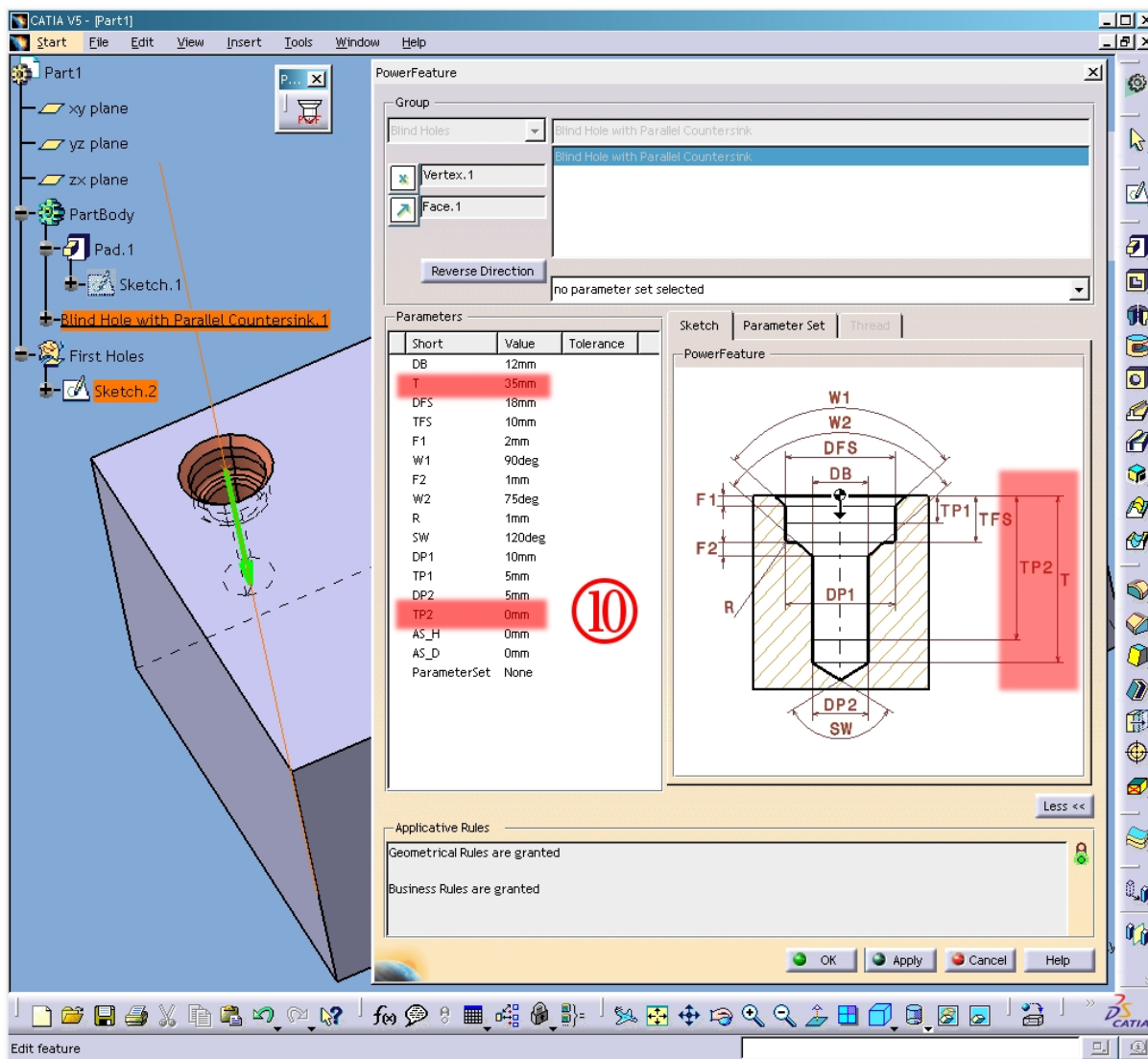
Group and type of the PowerFeature can not be changed.

2.5 Advanced PowerFeature

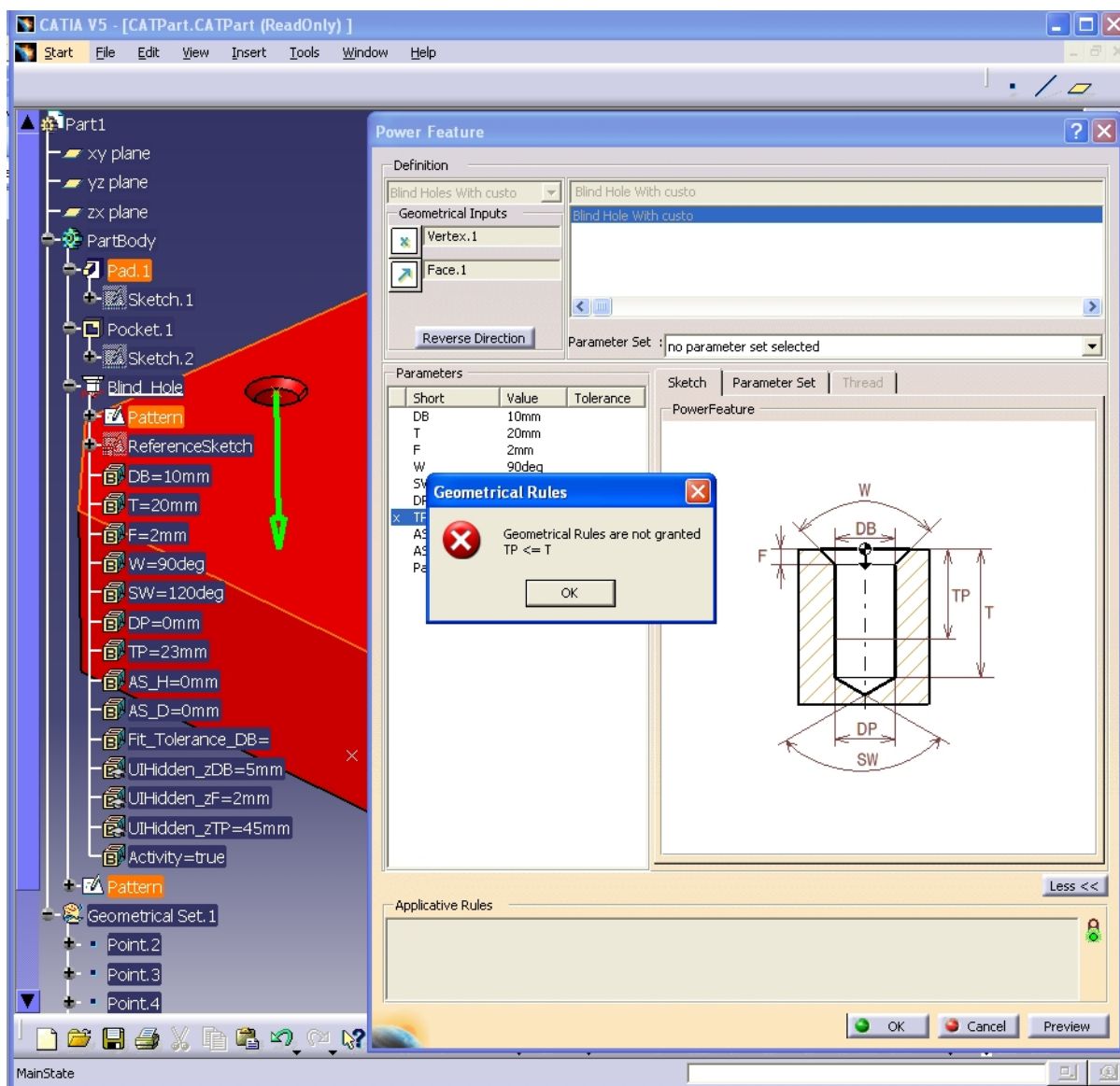
The values parameters are checked by the Applicative rules to guarantee the coherence of the PowerFeature. Tolerance can be added to the values parameters. The Parameter set can be modified so that they correspond to the needs.

2.5.1 Applicative Rules

You can edit the PowerFeature and modify some parameters by double-clicking on it in the specification tree or in the 3D geometry. We will modify a value of a parameter that will not be granted by the rules. In the picture it is logical to have $T > TP$.



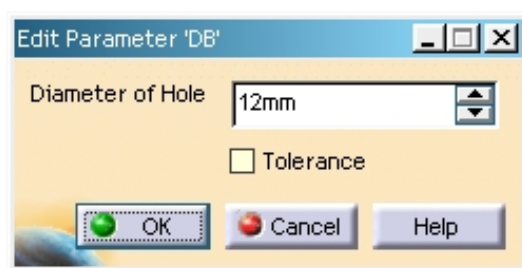
Modifying the TP2's value to 23 mm makes the following windows to be displayed:



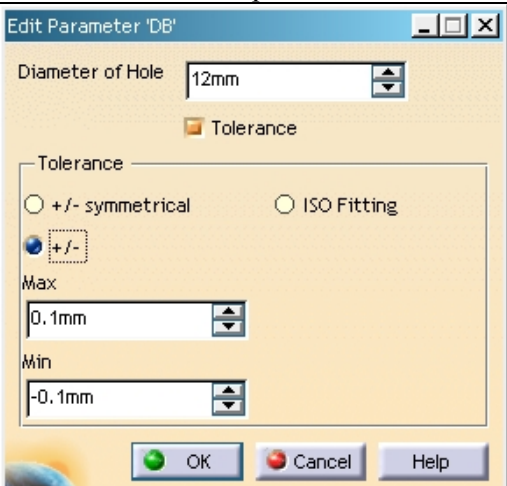
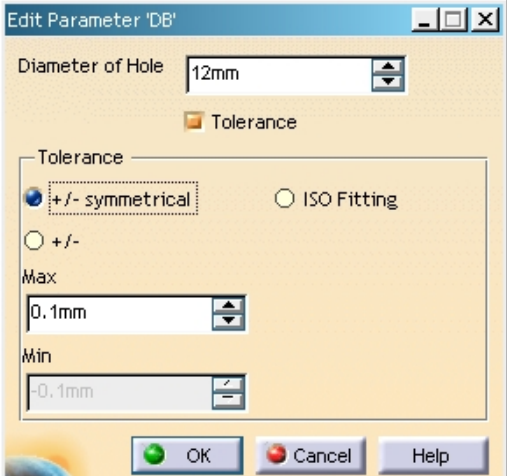
It informs you that the value is not correct according to the other values parameter. Usually an explicative message is provided by the rule, giving the reason of the fail (not provided on the previous picture).

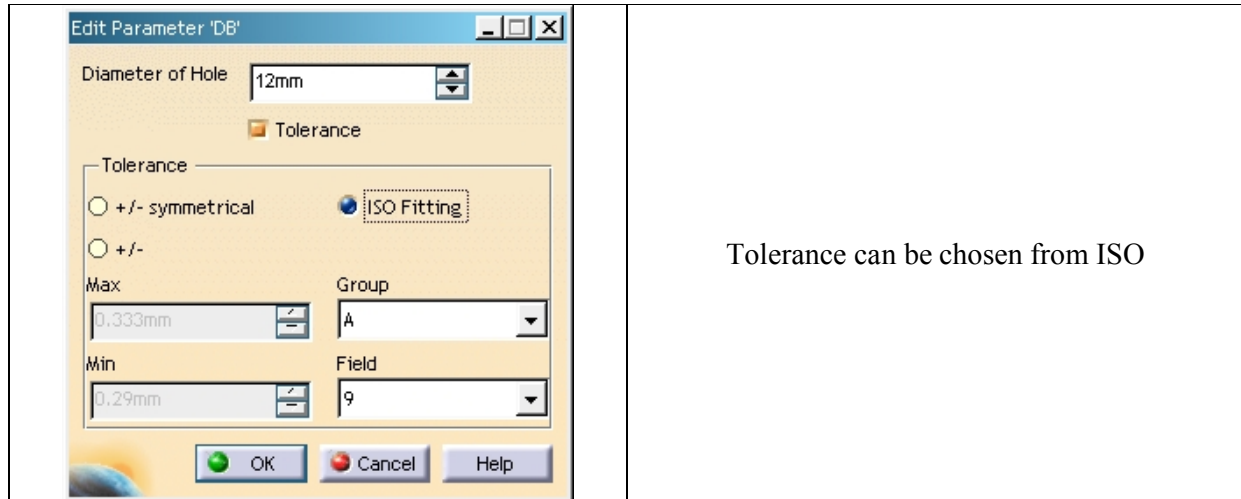
2.5.2 Tolerance value

Tolerance can be added to the value by editing the parameter. The "Edit parameter" window is displayed and by default "Tolerance" is not checked



When the “Tolerance” option is checked then more information can be added:

	<p>The tolerance is a special one and not an ISO one</p>
	<p>For symmetrical tolerance</p>

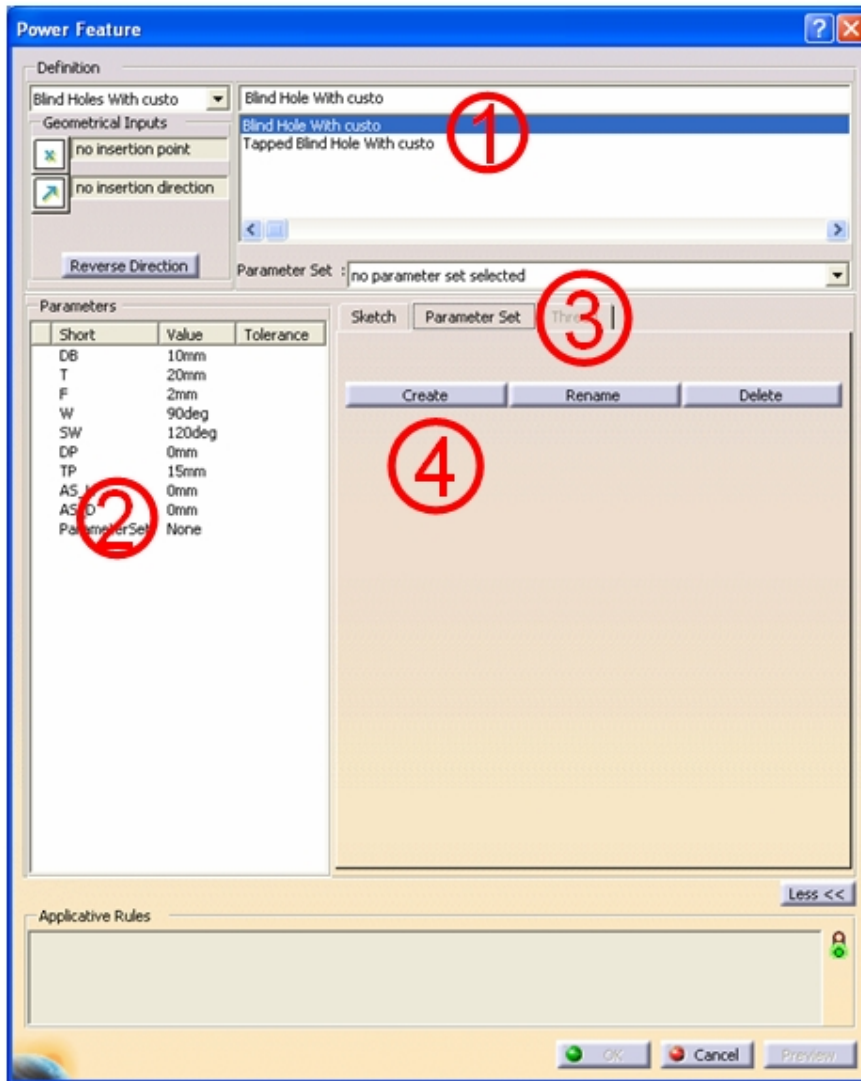


2.5.3 The parameter set

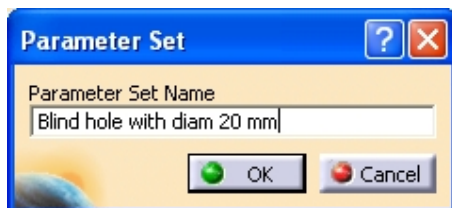
The parameter set allow having a unique list of features shared between all engineers. But parameter set with useful values can be stored by the engineer to be reused later easily.

2.5.3.1 Use of parameter set

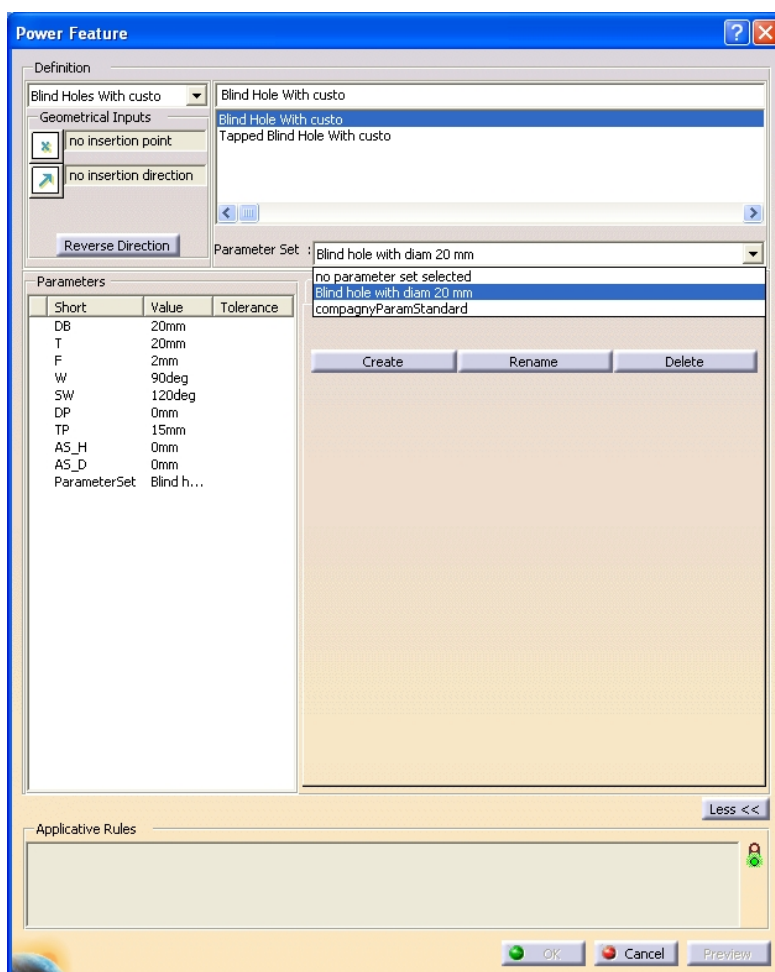
2.5.3.1.1 *Creation of parameter set*



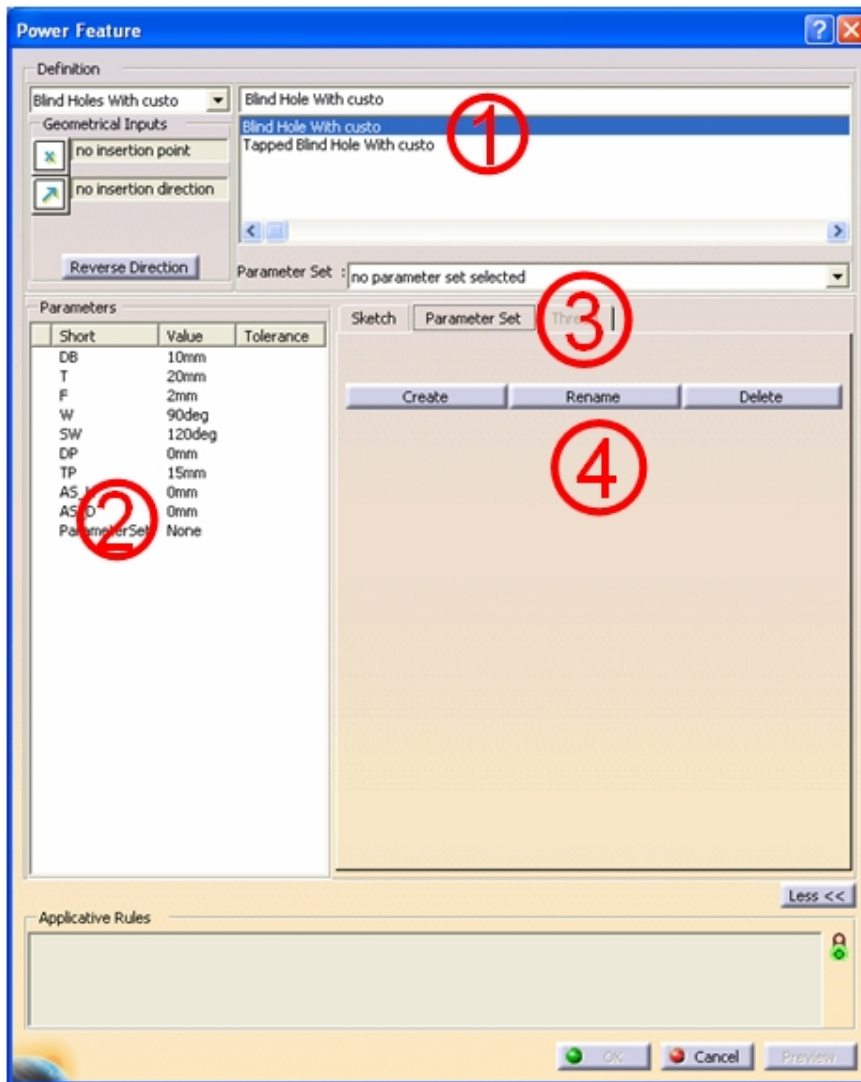
1. Select the type of the PowerFeature
2. select and modify values of parameters
3. Switch to “Parameter Set” tab
4. Click on “Create” button to add this new parameter set in the list.
5. A window is displayed to name the parameter set.



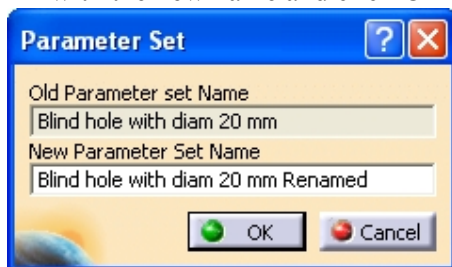
6. The new parameter set is added to the list



2.5.3.1.2 Rename a parameter set



1. Select the group of PowerFeature
2. Select the type of PowerFeature
3. Select the parameter set to rename
4. Click on rename button
5. fill with the new name and click OK to confirm



2.5.3.1.3 *Delete a parameter set*

The same as in 2.5.3.1.2 but with the delete button

2.5.3.2 The thread tab

Some features with thread can be implemented in the construction. In that case the thread tab is activated.

?

×

Power Feature

Definition

Blind Holes With custo

Geometrical Inputs

no insertion point

no insertion direction

Reverse Direction

Tapped Blind Hole With custo

Blind Hole With custo

Tapped Blind Hole With custo

Parameter Set : no parameter set selected

Parameters

Short	Value	Tolerance
DG	10mm	
TG	15mm	
DK	8.5mm	
TK	20mm	
DF	12mm	
W	90deg	
GB		
SR	RH	
P	1.5mm	
SW	120deg	
AS_H	0mm	
AS_D	0mm	
StandardT...		
ParameterSet	None	

Sketch

Parameter Set

Thread

Standard Thread

Type

No Standard

Diameter

10mm

Internal Diameter

8.5mm

Thread Length

15mm

Pitch

1.5mm

Direction

☐ Left-Threaded
 ☒ Right-Threaded

Less <<

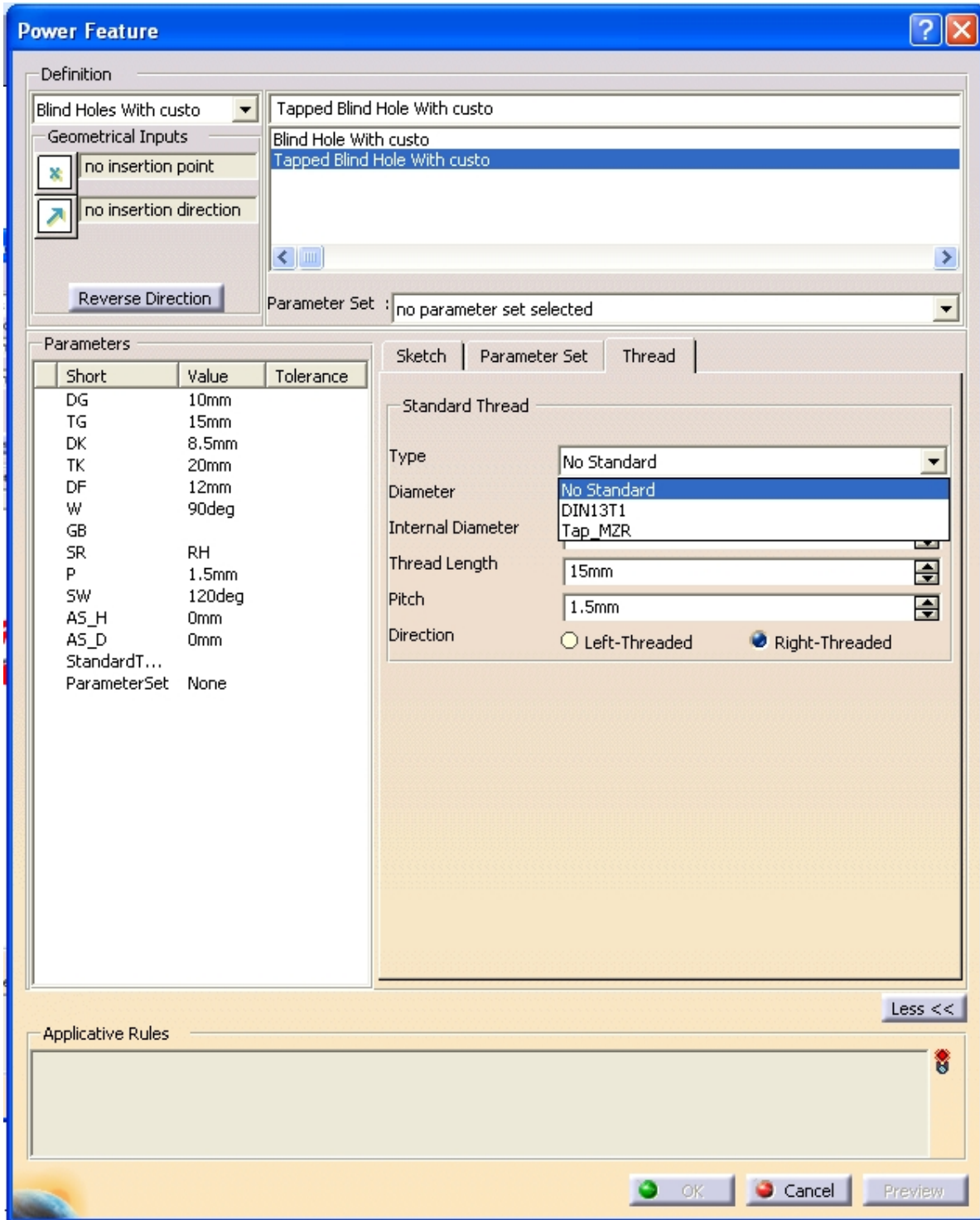
Applicative Rules

OK

Cancel

Preview

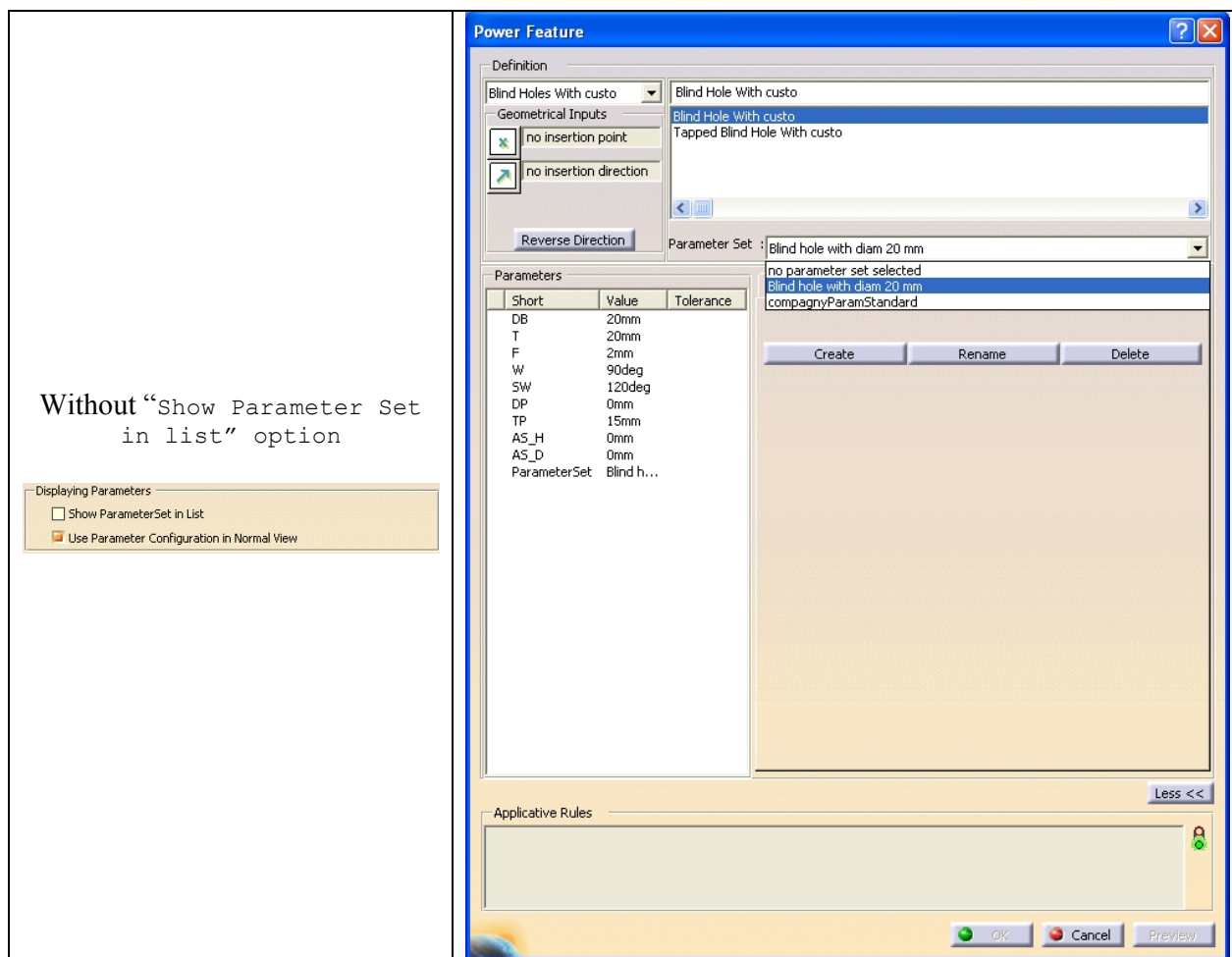
The thread tab

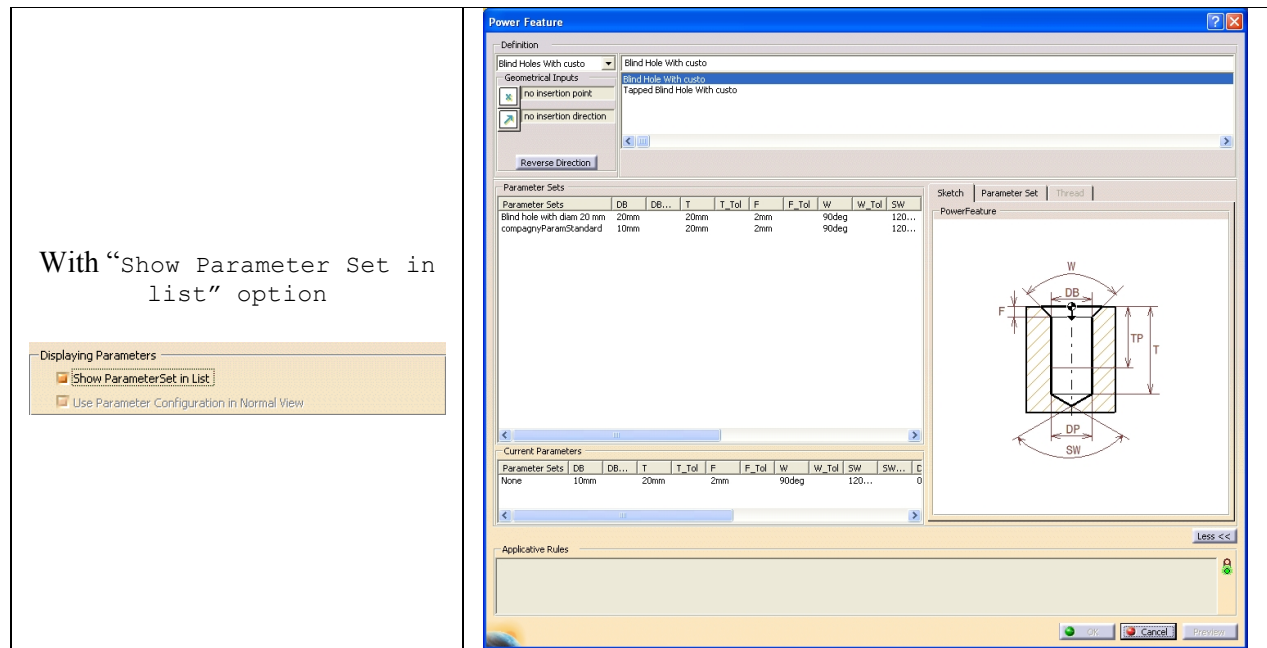


1. Different standard can be applied
2. Thread parameters values
3. type of thread

2.5.4 Important notes

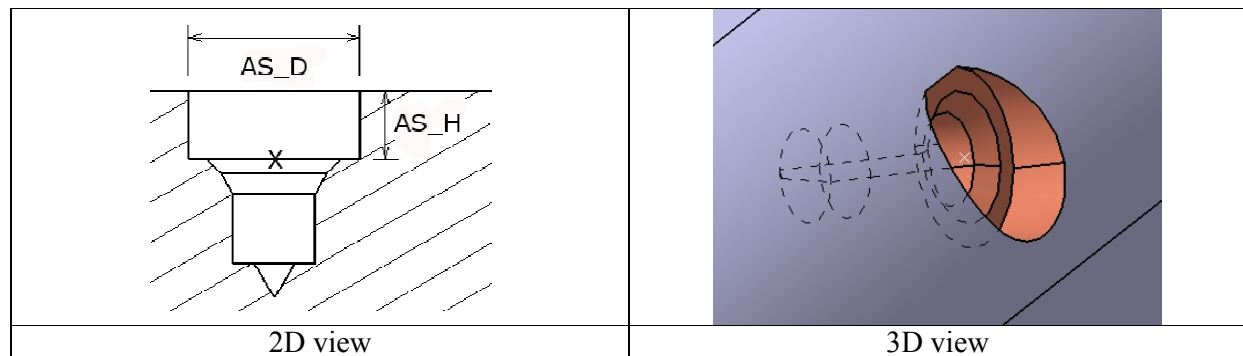
1. Parameter Set view
depending on the option you have checked the parameters can be viewed in two different ways:



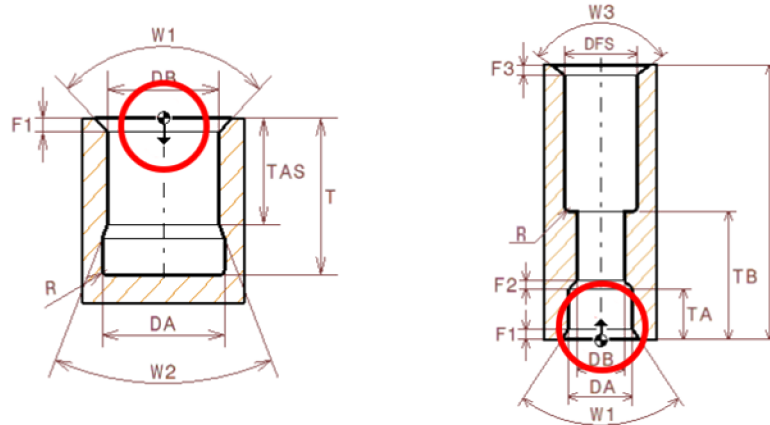


2. AS_H and AS_D parameters.

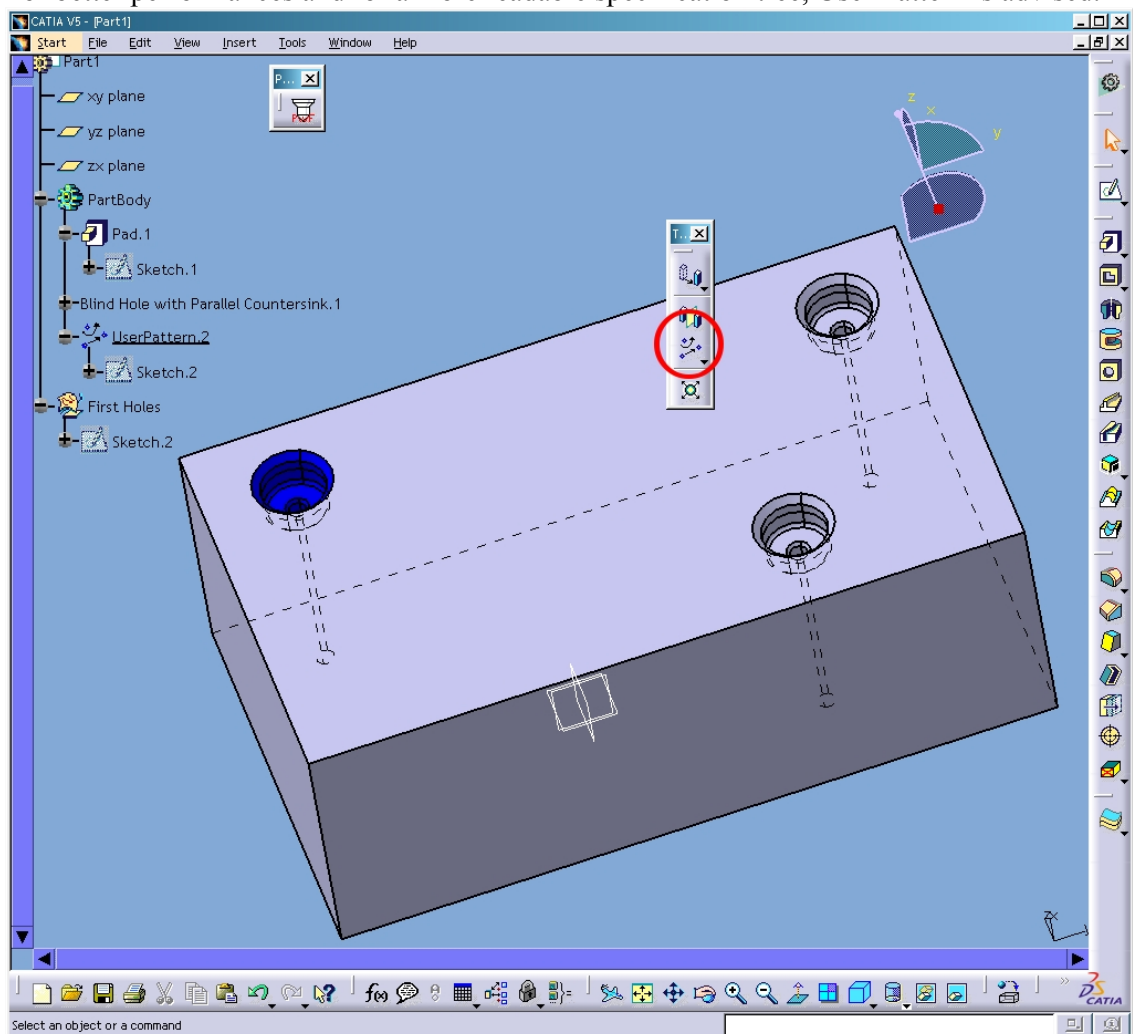
Those two parameters allow you to add a cylinder to the PowerFeature geometry. This cylinder serves for the production of the break-through of the PowerFeature geometry. AS_H correspond to the height's cylinder and AS_D is the value of the diameter's cylinder.



3. The position of the inserted point depends on the type of the PowerFeature.



4. For better performances and for a more readable specification tree, User Pattern is advised.



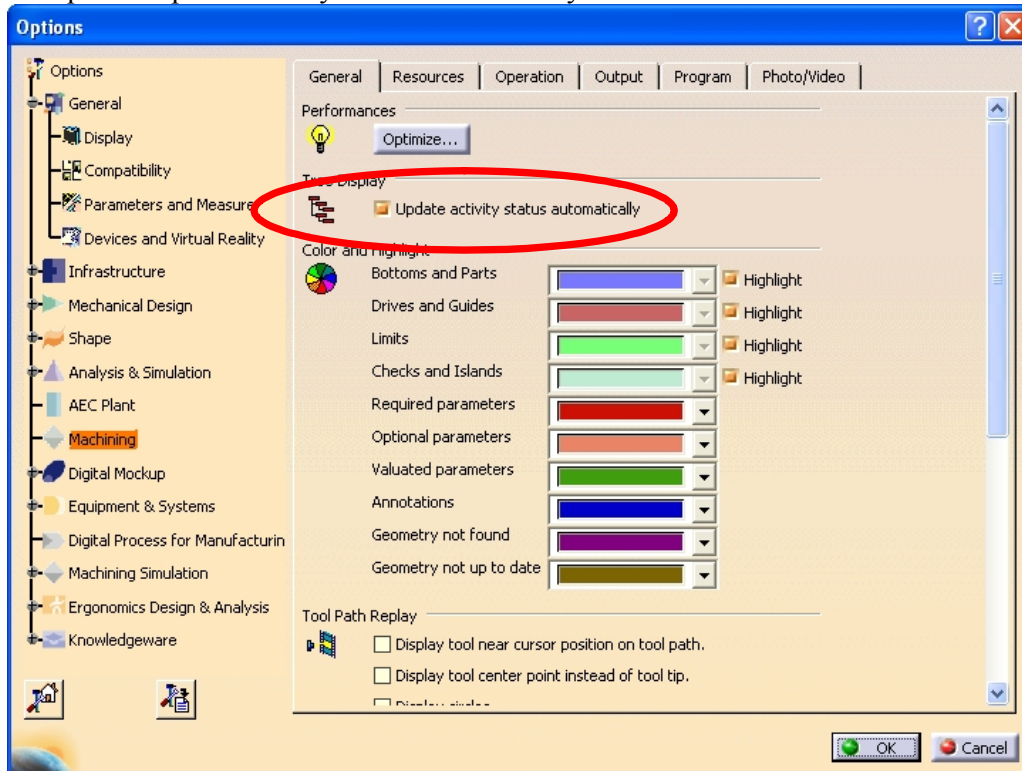
5. Modify the values parameter only with PowerFeature tool

2.6 PowerFeature prismatic machining process

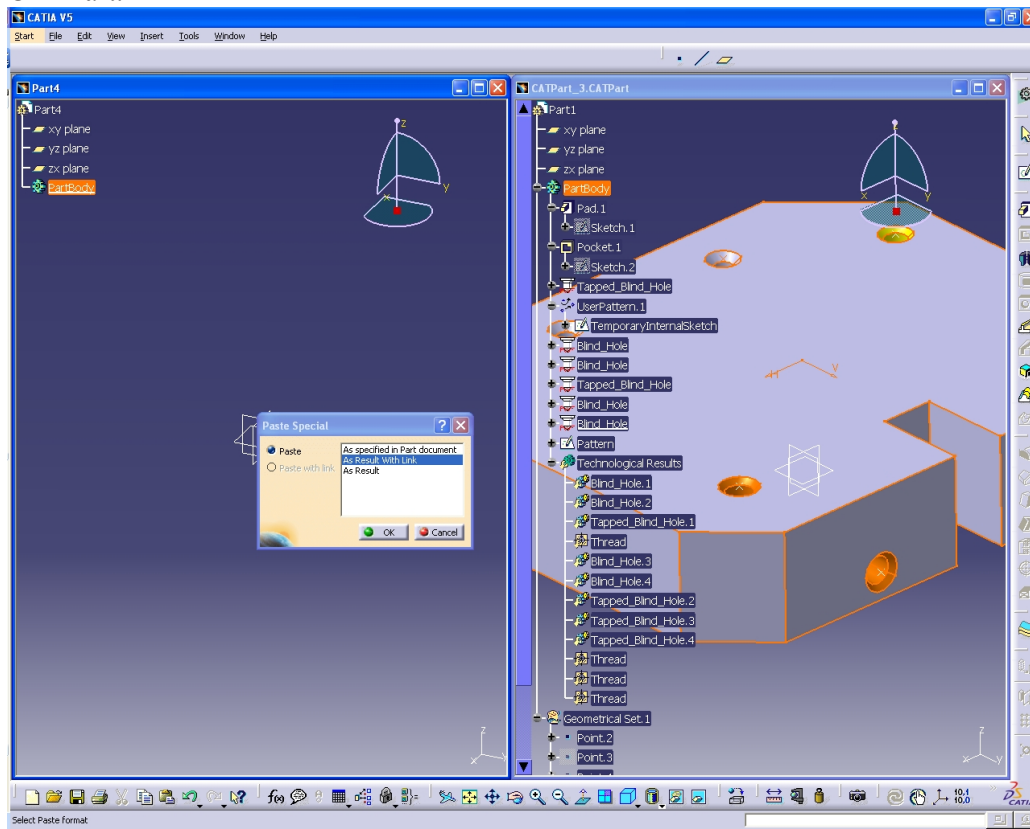
Once PowerFeature geometries are instanced, it is possible to create Technological Result (TR) to be used in the Prismatic machining Workbench. The idea is to work the machining process with a copy of the solid. In that a “Copy/Paste as Result with Link” is used.

2.6.1 Introduction

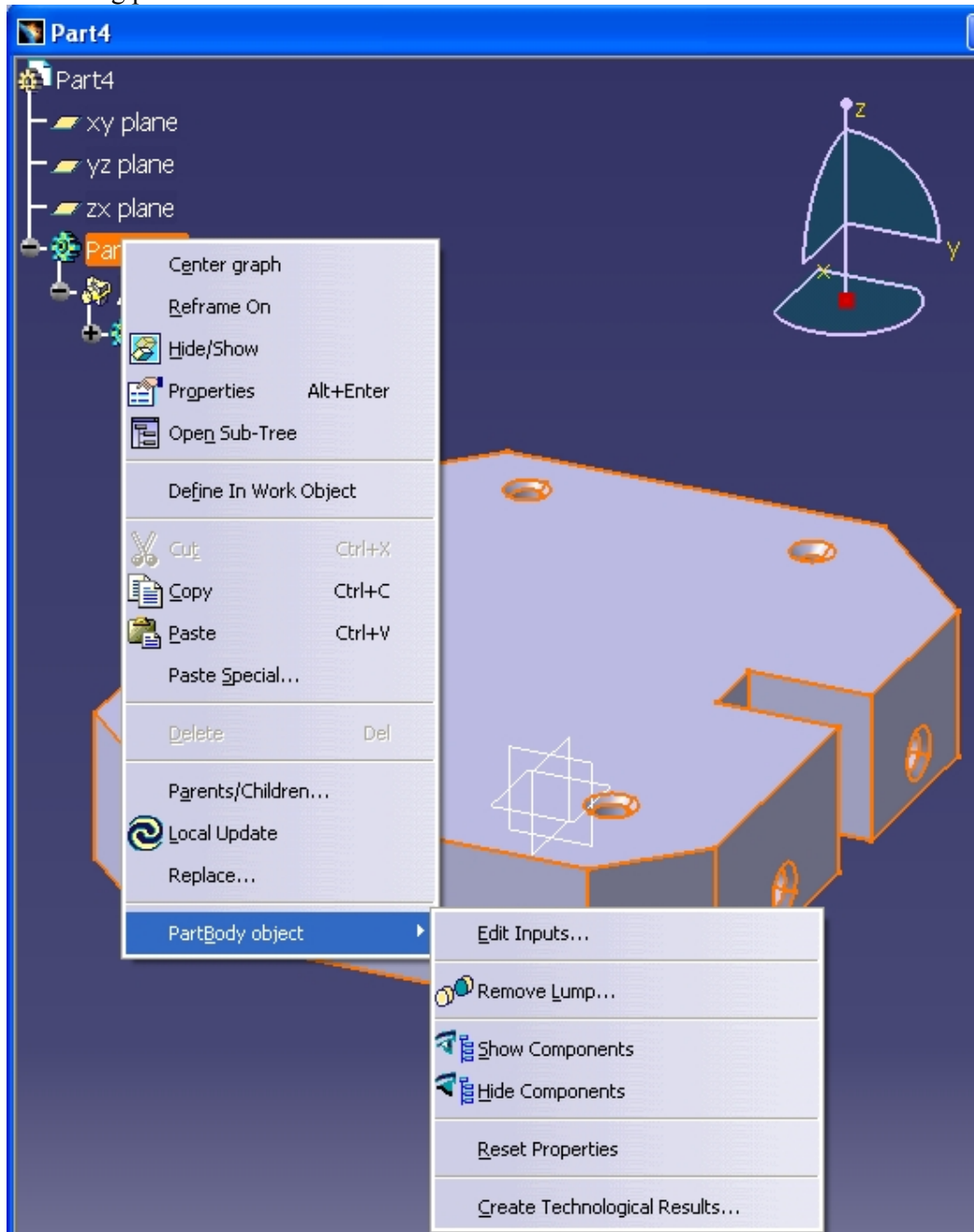
In this part, we are now about to pass in the prismatic machining part. You have to check the option “Update activity status automatically”.



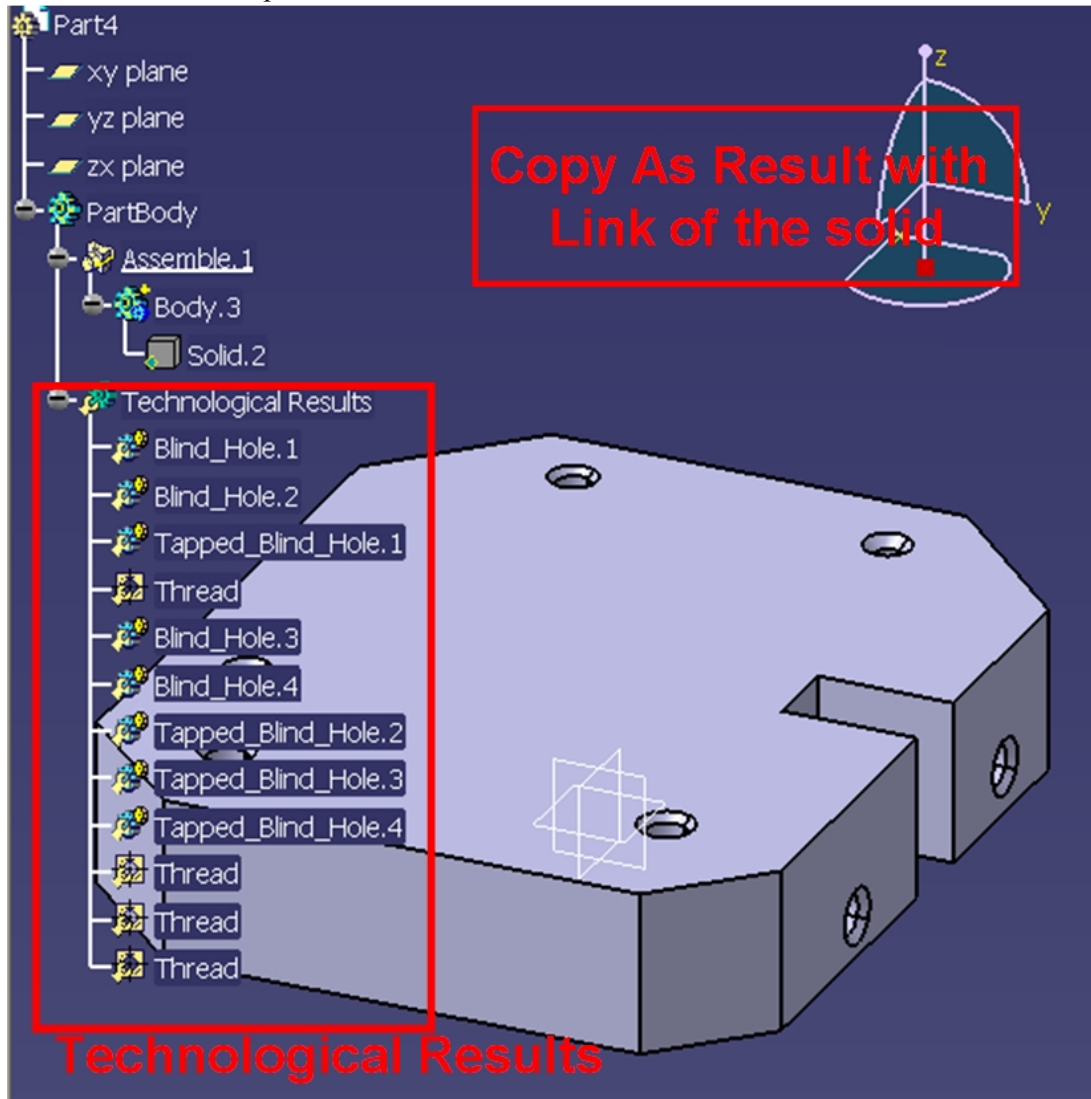
1. But first create TR from the original solid, and then copy/paste the solid to another CATPart.



2. Recreate TR (Technological Result) to have a link between PowerFeature geometries and machining process.

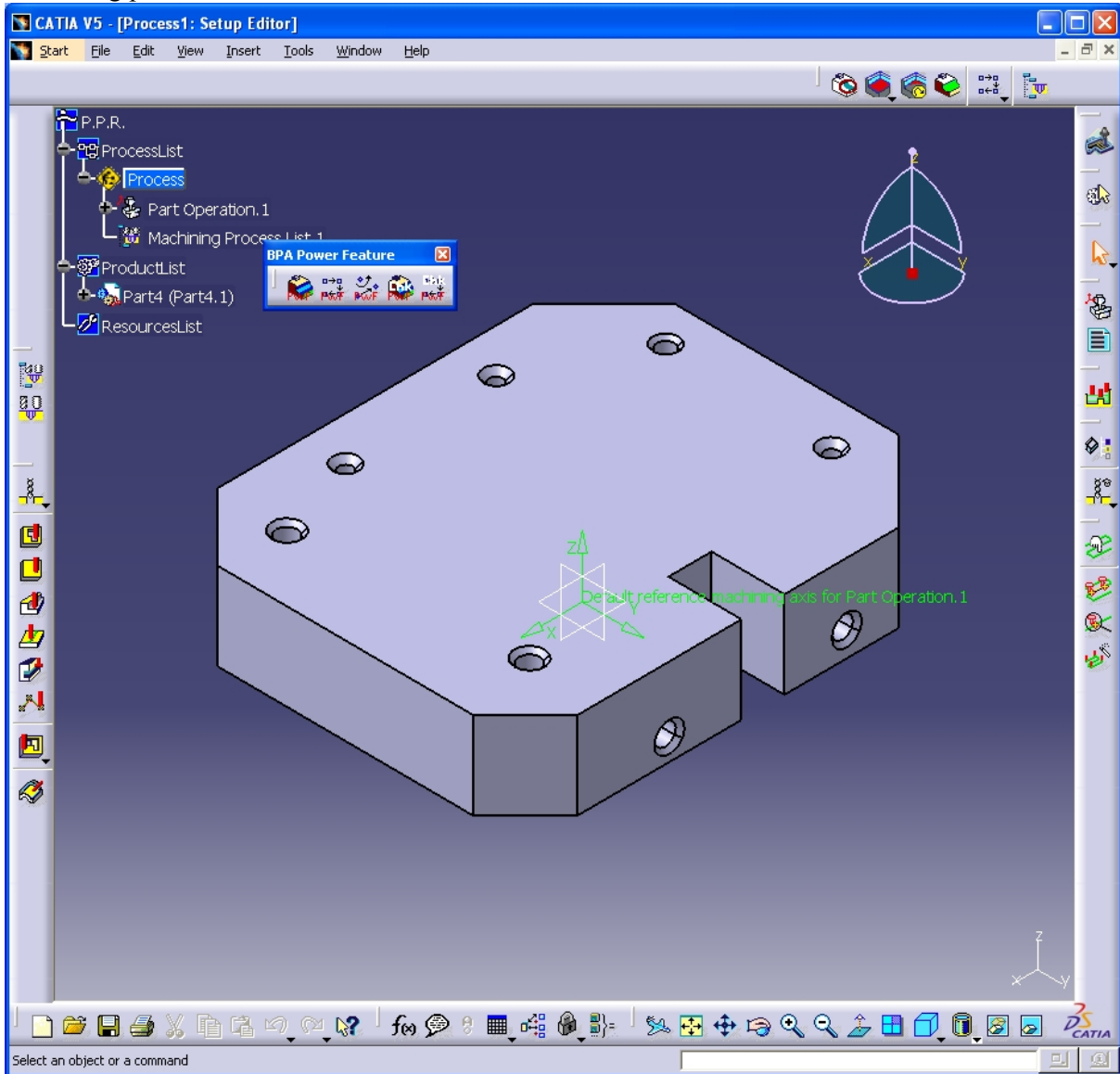


3. TR are added to the specification tree. Check that the TR are identical.



Switch to Prismatic machining workbench. Prismatic PowerFeature Toolbar is displayed (in the red box in the picture) and the part is inserted in the PPR product. It is possible now to begin the

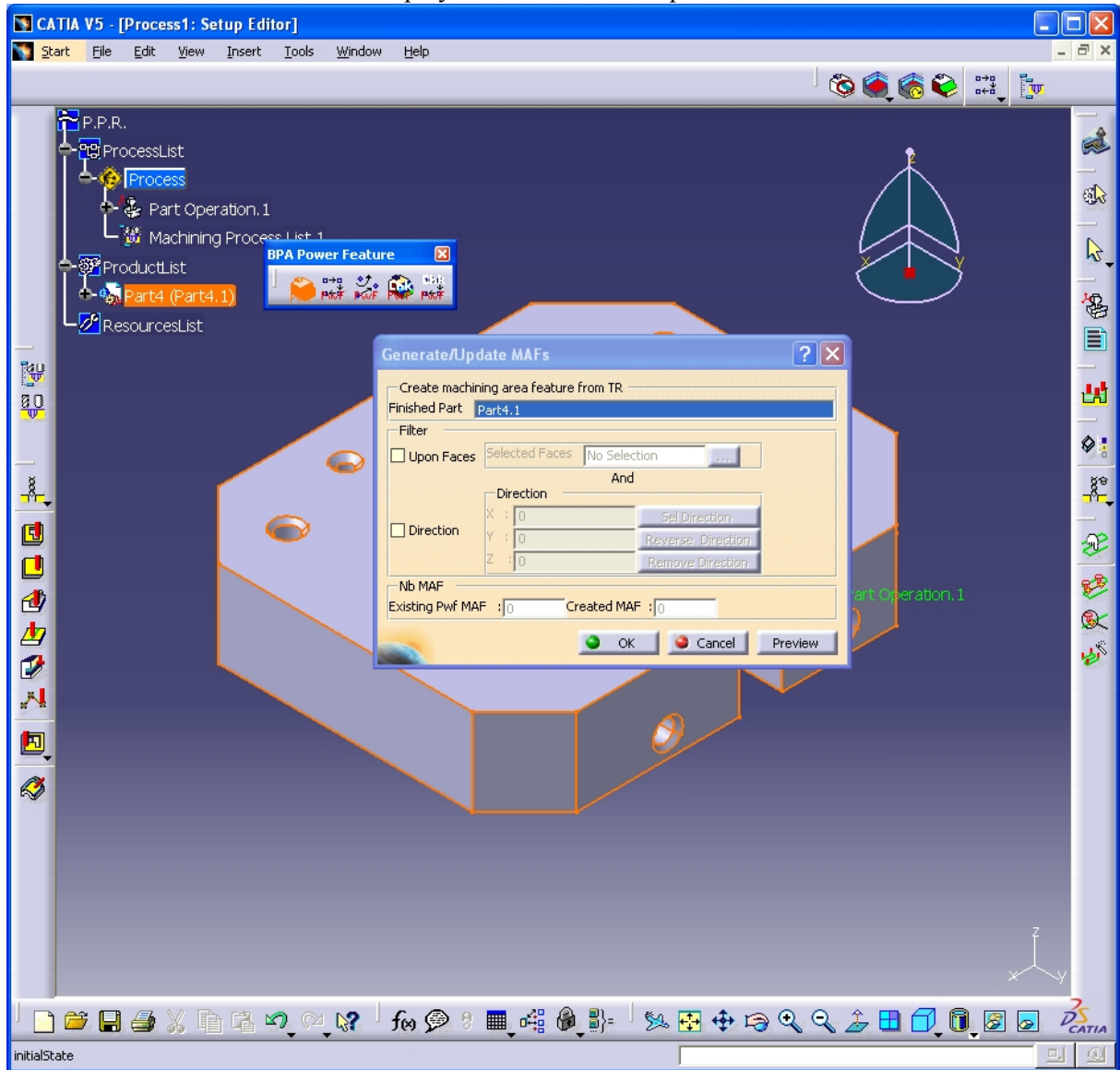
machining process.



As you can see, there is a new icon in the Prismatic PowerFeature toolbar. The function is described in this guide.

2.6.2 Create and update Mafs

1. Creation of Machining Area Feature (MAF) by selecting the first icon of the toolbar:
The “Generate MAF from TR” is displayed and the finished part have to be selected.



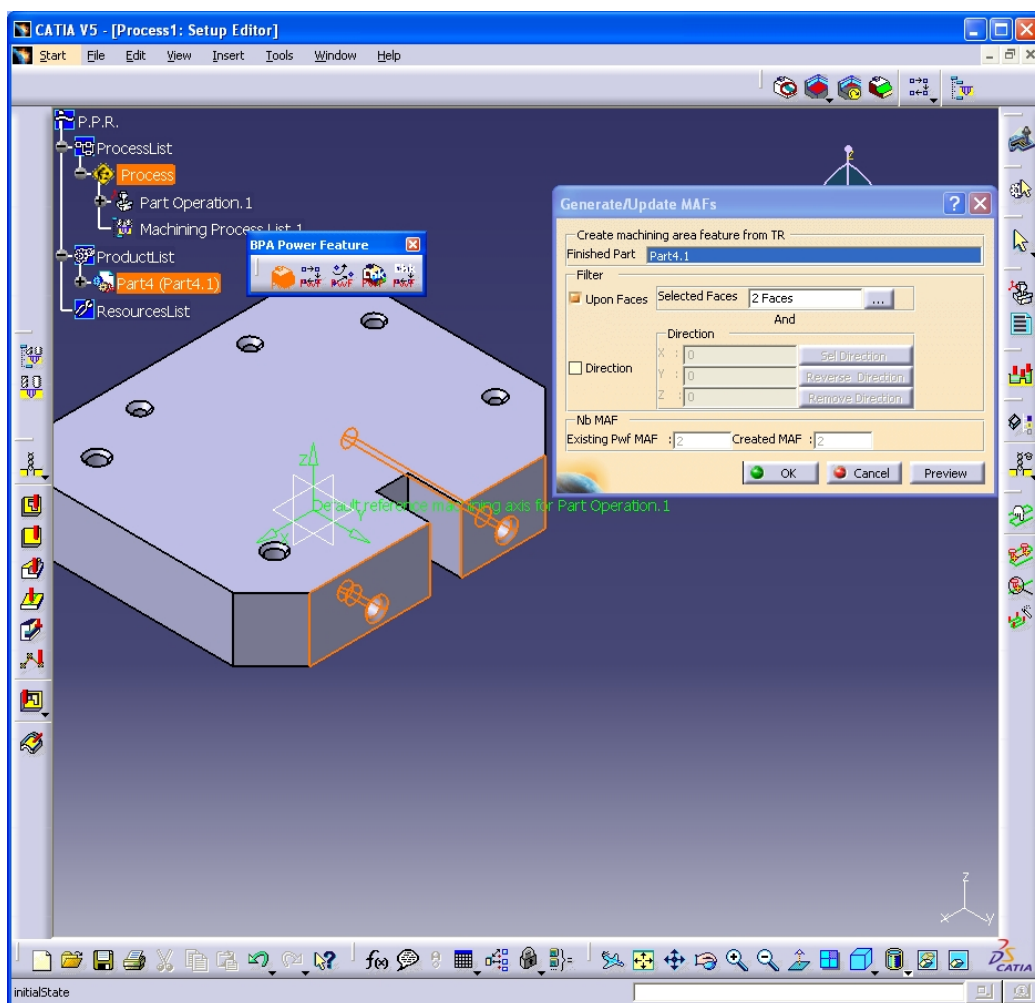
The filter function is used, to generate a Maf, when:

- You want to find Power Features which are upon one of the selected faces
- You want to find Power Features which have a common direction

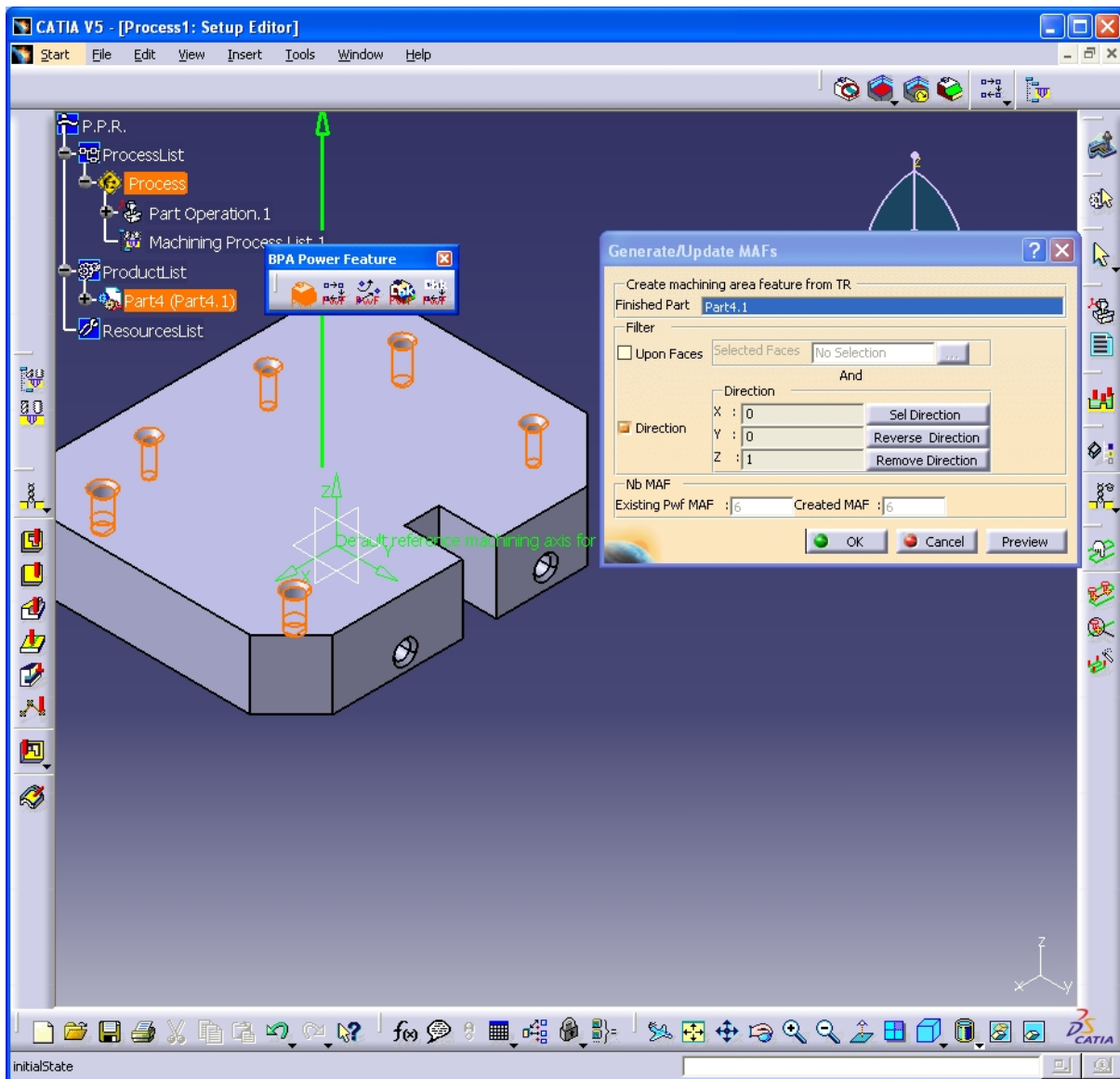
Filter options are managed like the ‘and’ function

Take examples:

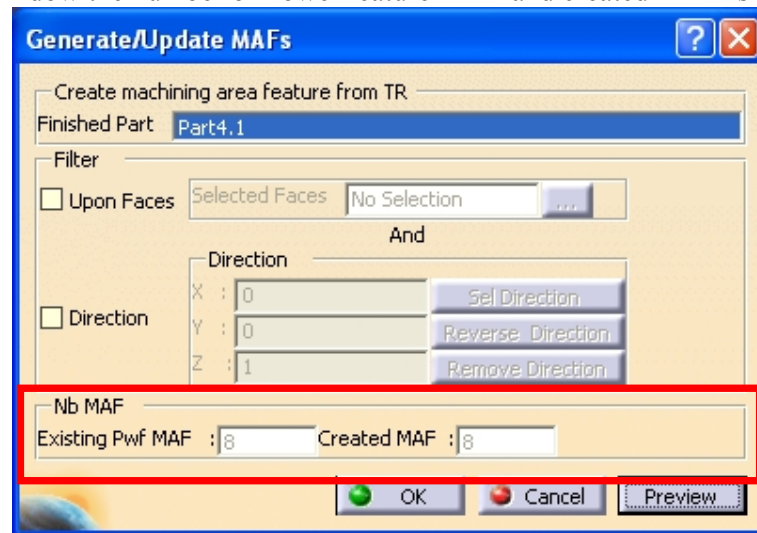
Selected the two side face allows to Generate/Update only 2 Pwf MAFs (instead of 8)



Selected the face surface as direction allows to Generate/Update only 6 Pwf MAFs (instead of 8)

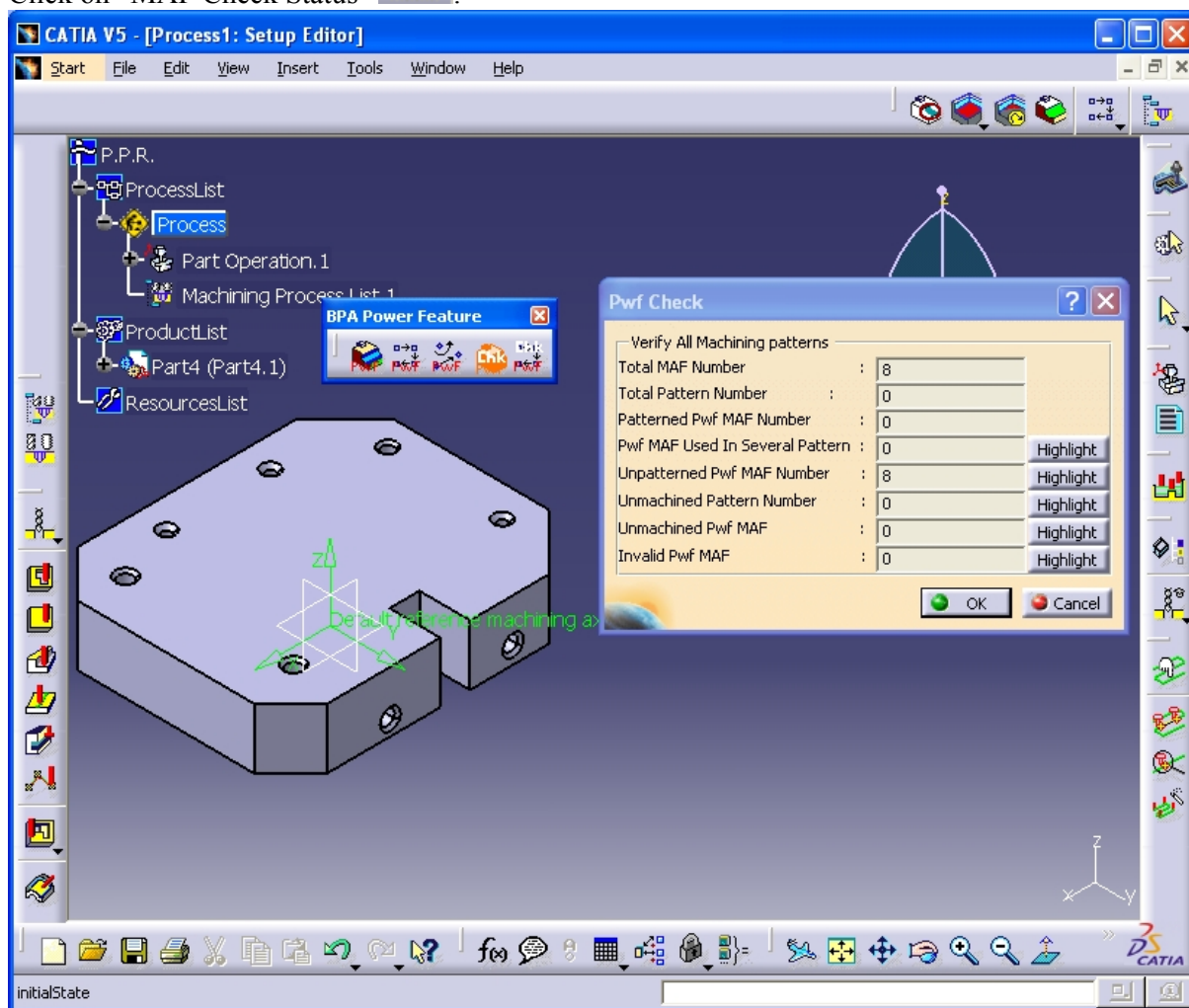


2. The preview button allows you to see in the 3D the result for the selected finished part. In the window the number of PowerFeature MAF and created MAF is shown.



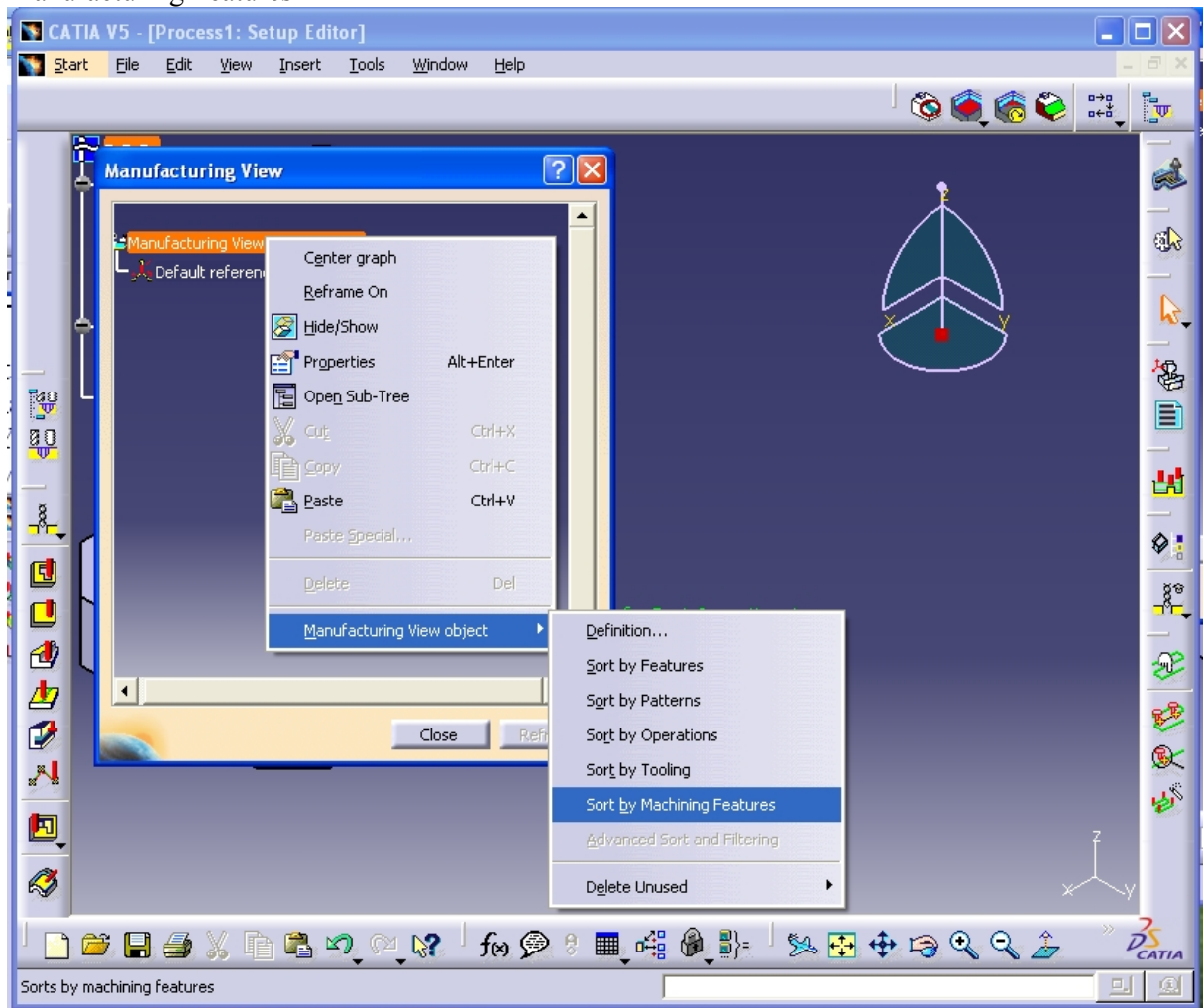
3. Click on OK button to confirm the creation of MAF.

4. Click on “MAF Check Status”  :

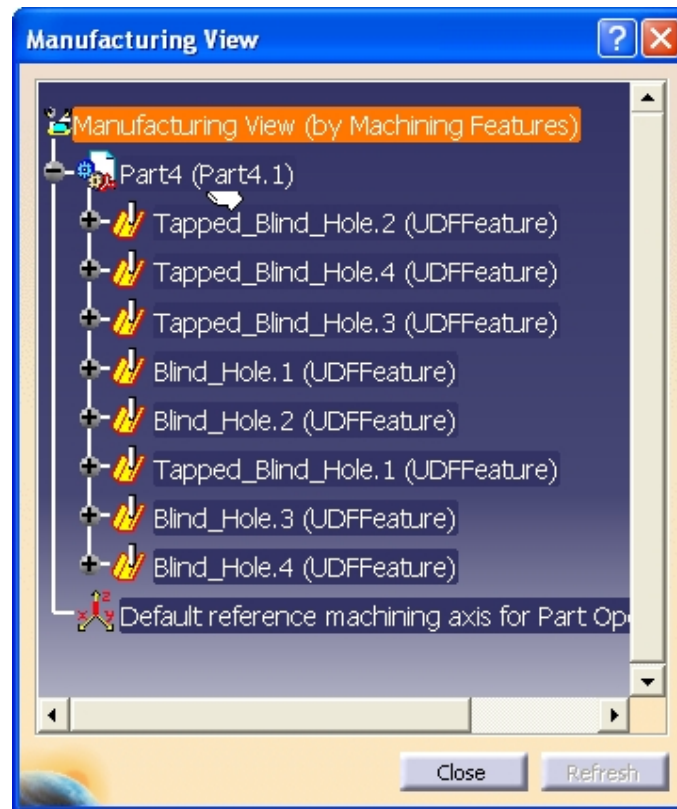


It is possible to highlight the elements in different status to display them clearly and then to have a good status of the MAF.

5. You can see the MAF as features by activating “Manufacturing View” and sort by Manufacturing Features




6. The MAF are generated, it is time to create the pattern of MAF to group same machining “process”.

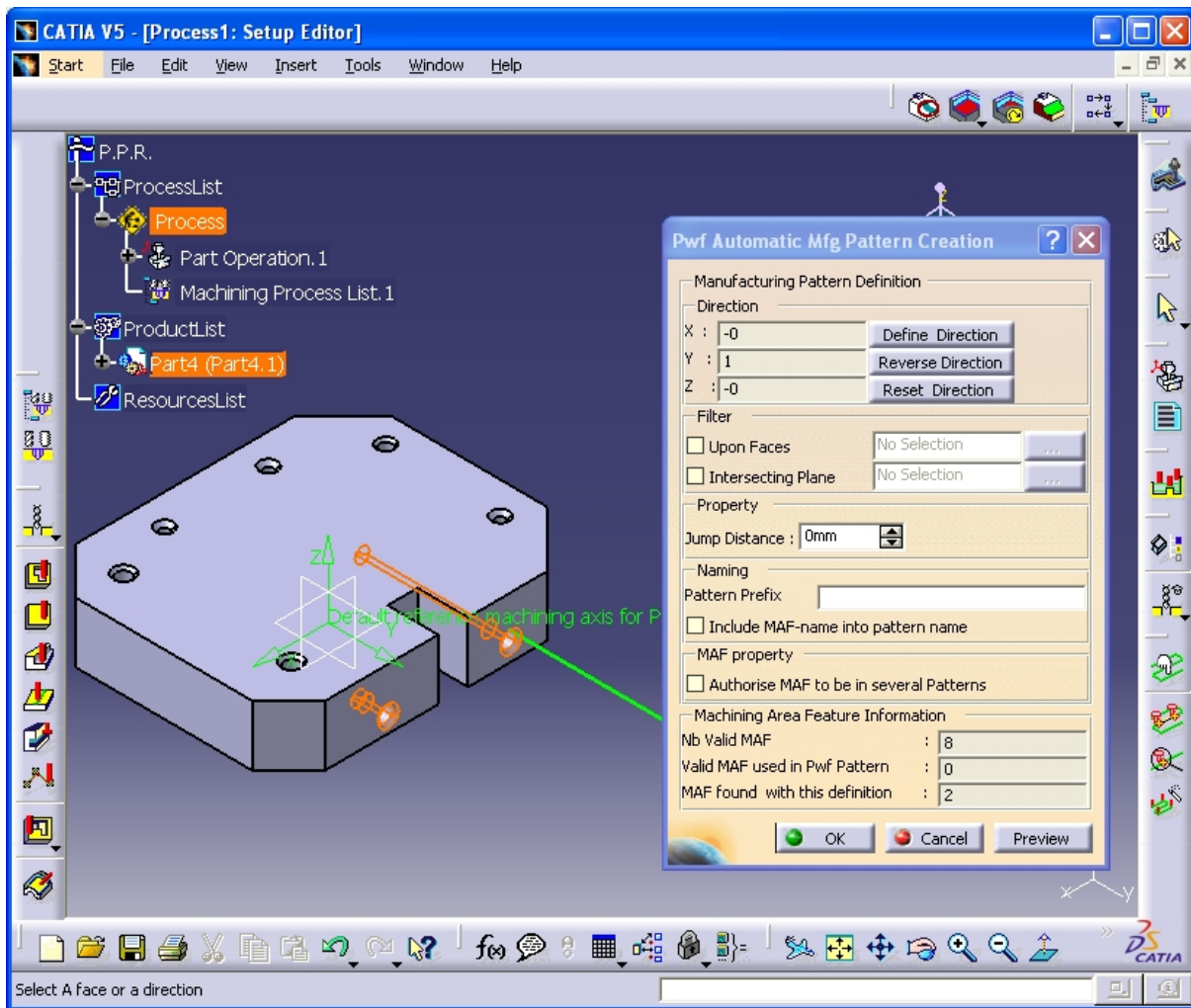


2.6.3 Patterns

2.6.3.1 Automatic pattern

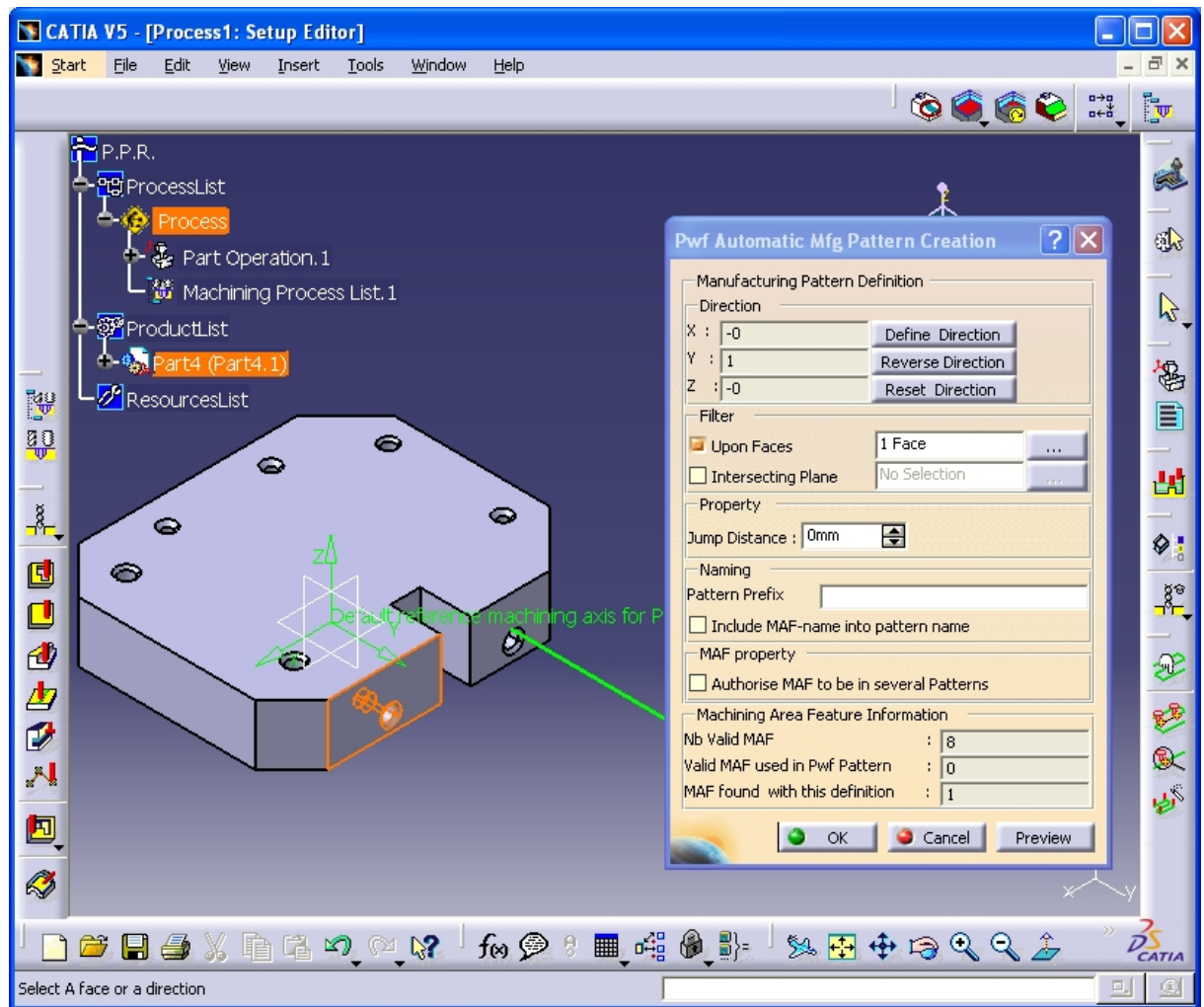


- 1 Activate “Create pattern from MAF”: . Here the pattern is not the pattern used in “Part Design” Workbench.
 - a. Filter by direction



I select the edge 1 to define the direction 2. So, the system finds the holes which are defined in this direction.

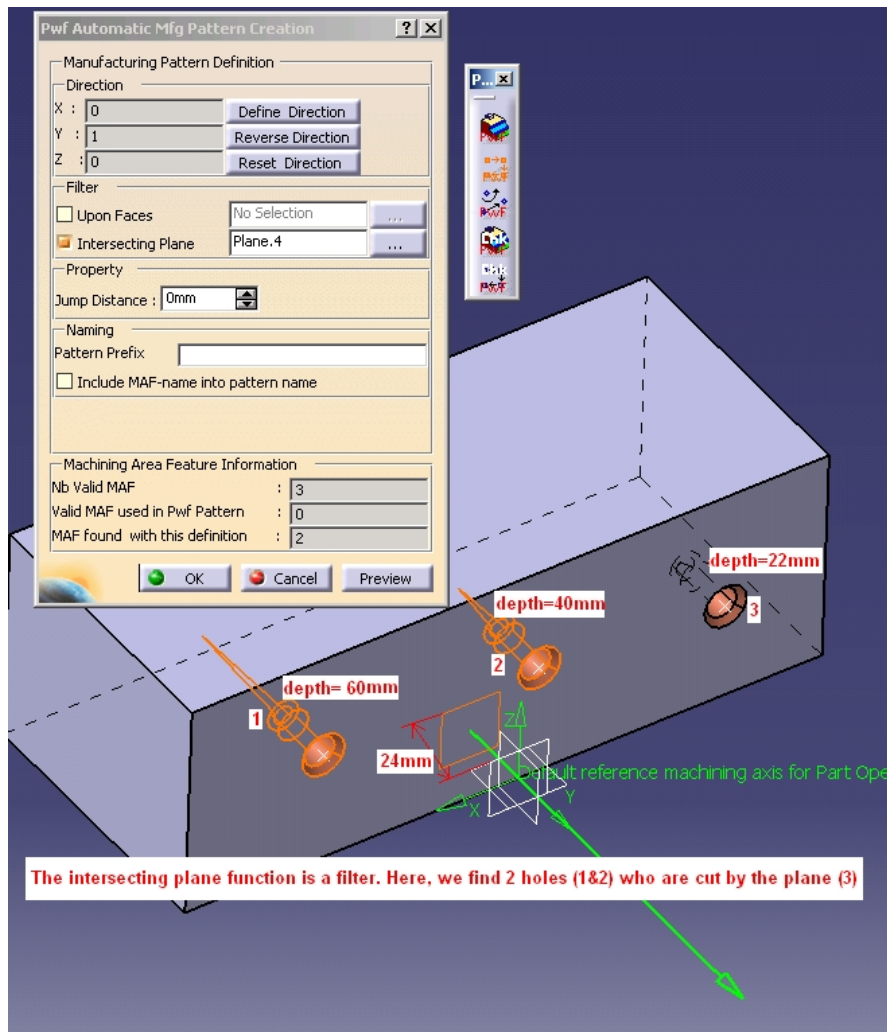
b. Filter by faces



Select the face of the solid to filter the MAF. The direction filter “b” is automatically filled.

Click on OK to create the operation

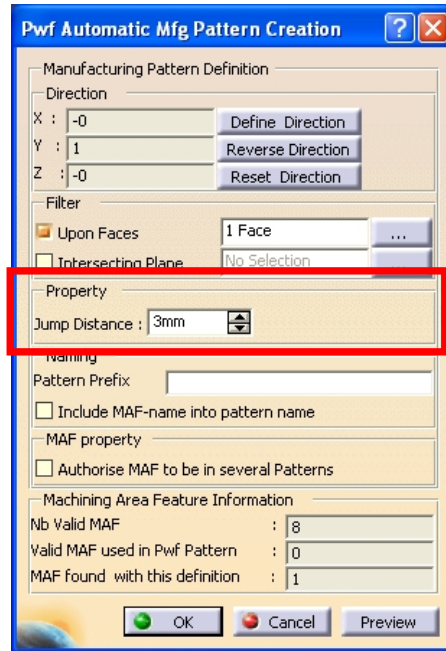
c. Filter by intersecting plane



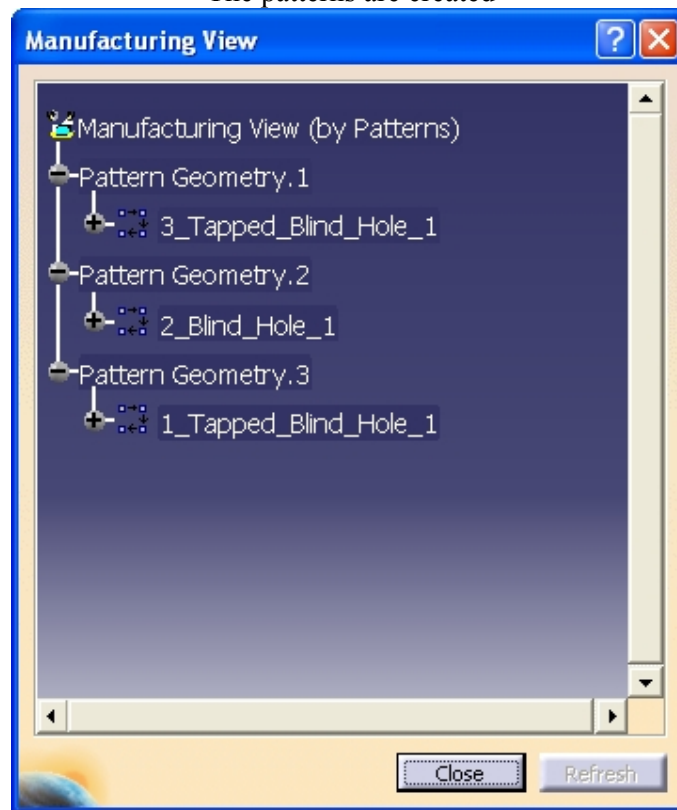
d. The jump distance property

It is the distance of release of the tool.

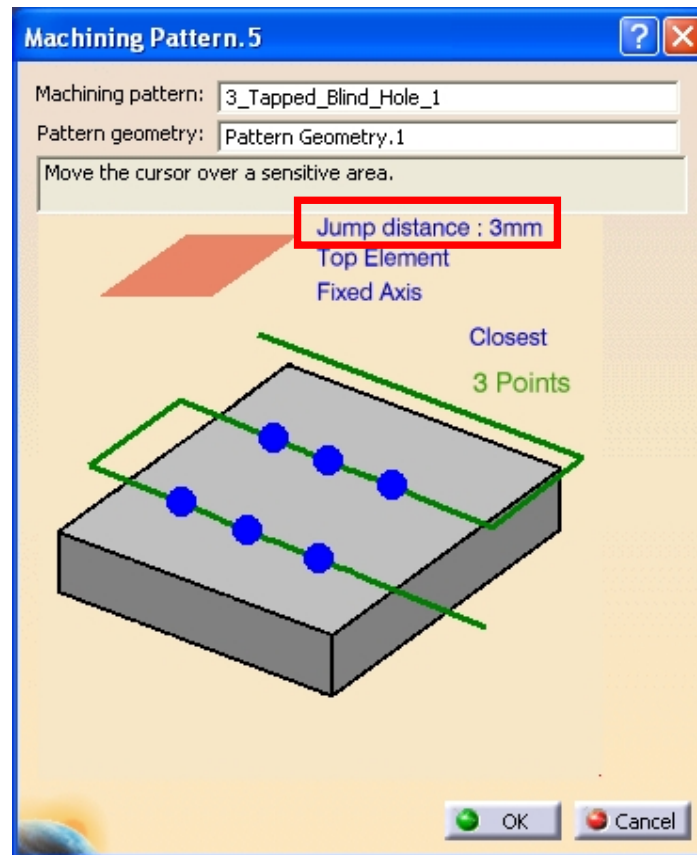
We have created a pattern with a jump distance = 10 mm. We check the option and enter the value



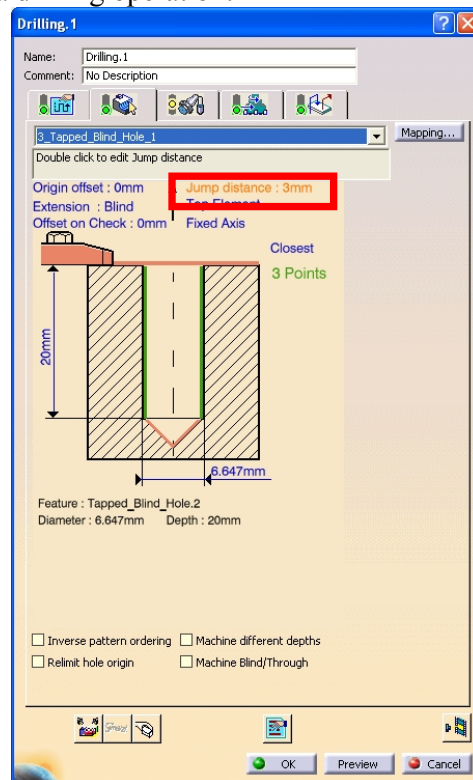
The patterns are created



We find the jump value




What we obtain when we do a drilling operation:

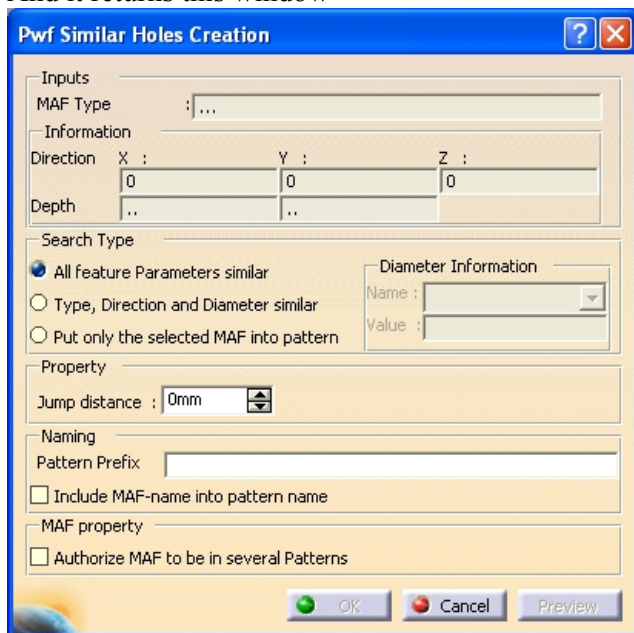


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2.6.3.2 Similar hole pattern

It's the second method to create a pattern. This function is activated with the button . And it returns this window



Pwf Similar Holes Creation

Inputs
MAF Type : ...

Information
Direction X : Y : Z :
0 0 0
Depth : : :
.. ..

Search Type
☒ All feature Parameters similar
☐ Type, Direction and Diameter similar
☐ Put only the selected MAF into pattern

Diameter Information
Name :
Value :

Property
Jump distance : 0mm

Naming
Pattern Prefix :
☐ Include MAF-name into pattern name

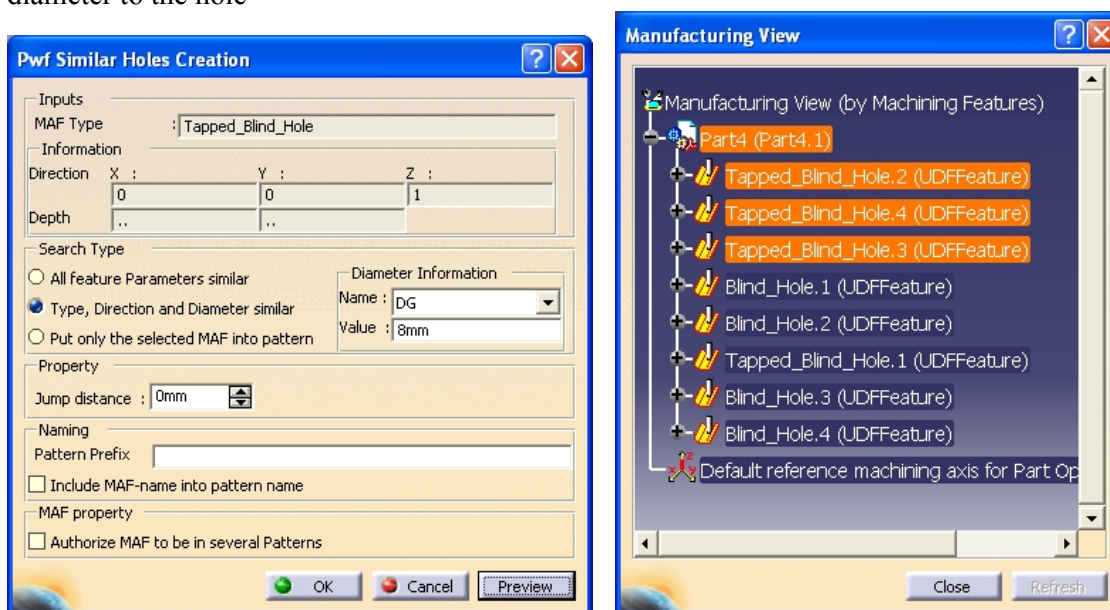
MAF property
☐ Authorize MAF to be in several Patterns

OK Cancel Preview

First case: I want create a pattern with the holes which are identical to selected MAF

Preview action returns 4 Mafs

Second case: I want to create a pattern with the holes wich have same type, direction an DG diameter to the hole



Pwf Similar Holes Creation

Inputs
MAF Type : Tapped_Blind_Hole

Information
Direction X : Y : Z :
0 0 1
Depth : : :
.. ..

Search Type
☐ All feature Parameters similar
☒ Type, Direction and Diameter similar
☐ Put only the selected MAF into pattern

Diameter Information
Name : DG
Value : 8mm

Property
Jump distance : 0mm

Naming
Pattern Prefix :
☐ Include MAF-name into pattern name

MAF property
☐ Authorize MAF to be in several Patterns

OK Cancel Preview

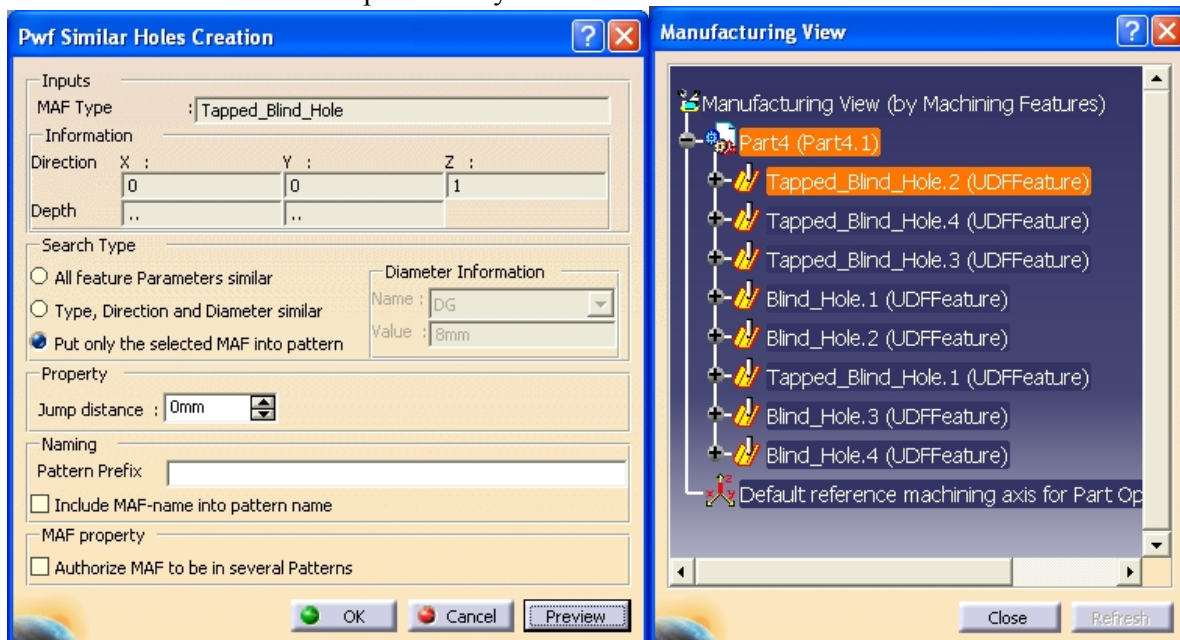
Manufacturing View

Manufacturing View (by Machining Features)

- Part4 (Part4.1)
- Tapped_Blind_Hole.2 (UDFFeature)
- Tapped_Blind_Hole.4 (UDFFeature)
- Tapped_Blind_Hole.3 (UDFFeature)
- Blind_Hole.1 (UDFFeature)
- Blind_Hole.2 (UDFFeature)
- Tapped_Blind_Hole.1 (UDFFeature)
- Blind_Hole.3 (UDFFeature)
- Blind_Hole.4 (UDFFeature)
- Default reference machining axis for Part Op

Close Refresh

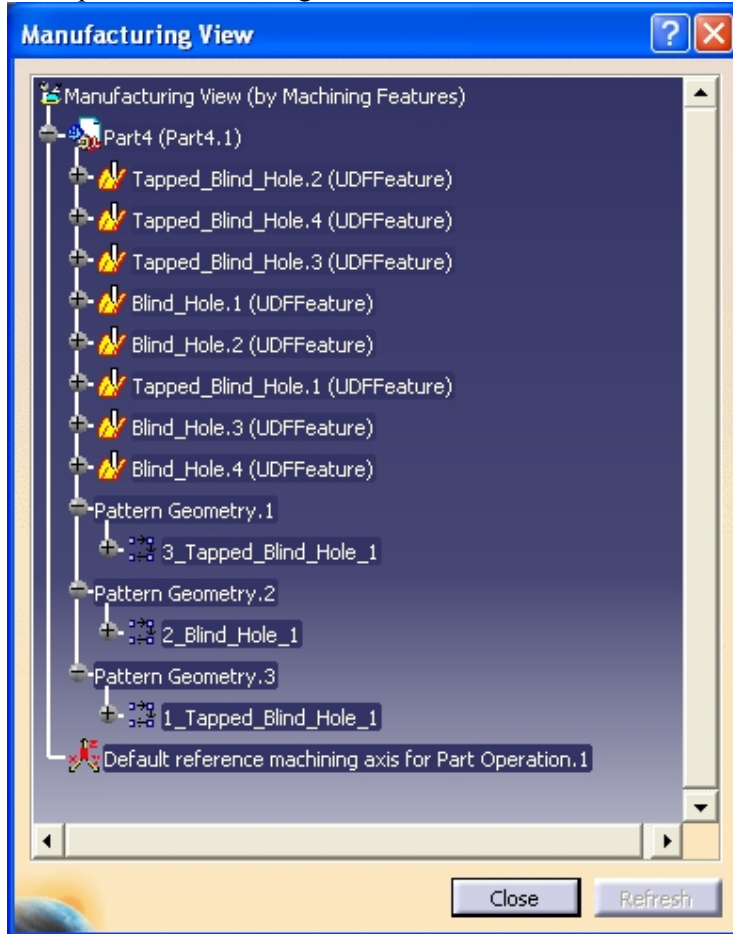
Third case: I want to create a pattern only with 1



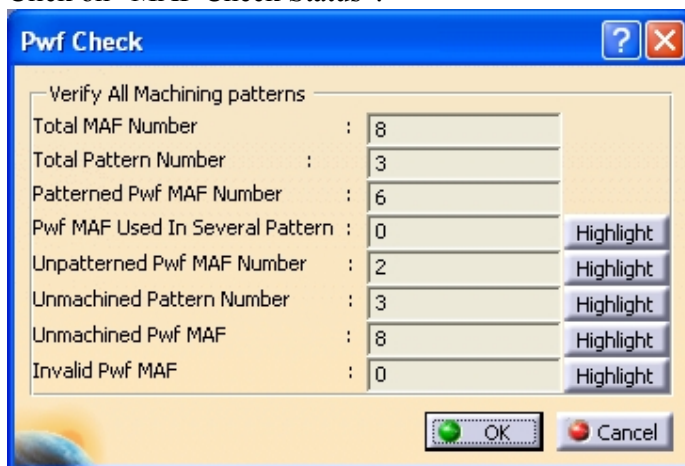
There is only one MAF in this pattern.

2.6.4 Pattern & MAFs checks

1. Once pattern have been generated, the result can be seen in the “Manufacturing View”:

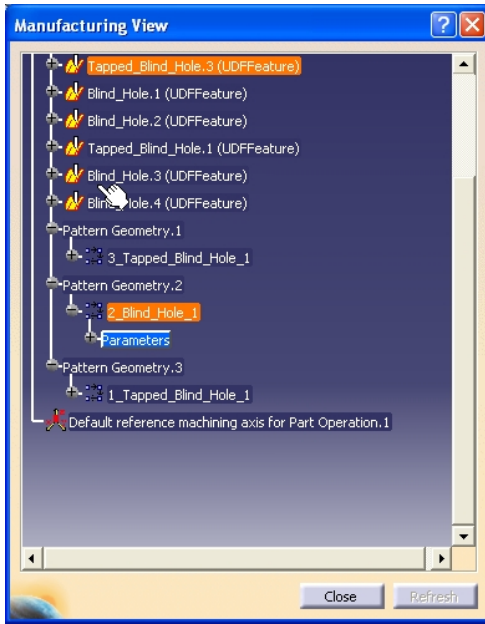


2. Click on “MAF Check Status”:



3 patterns have been added

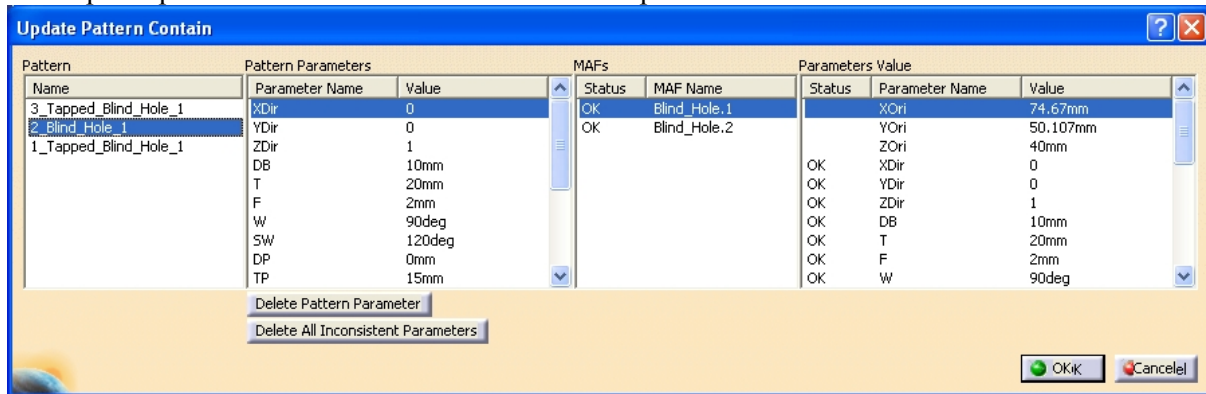
3. “update pattern contains” command



“2_Blind_Hole_1” is a pattern with power feature MAFs corresponding to 2 instance of CAD Power Feature. These two instances have the same value of DB.

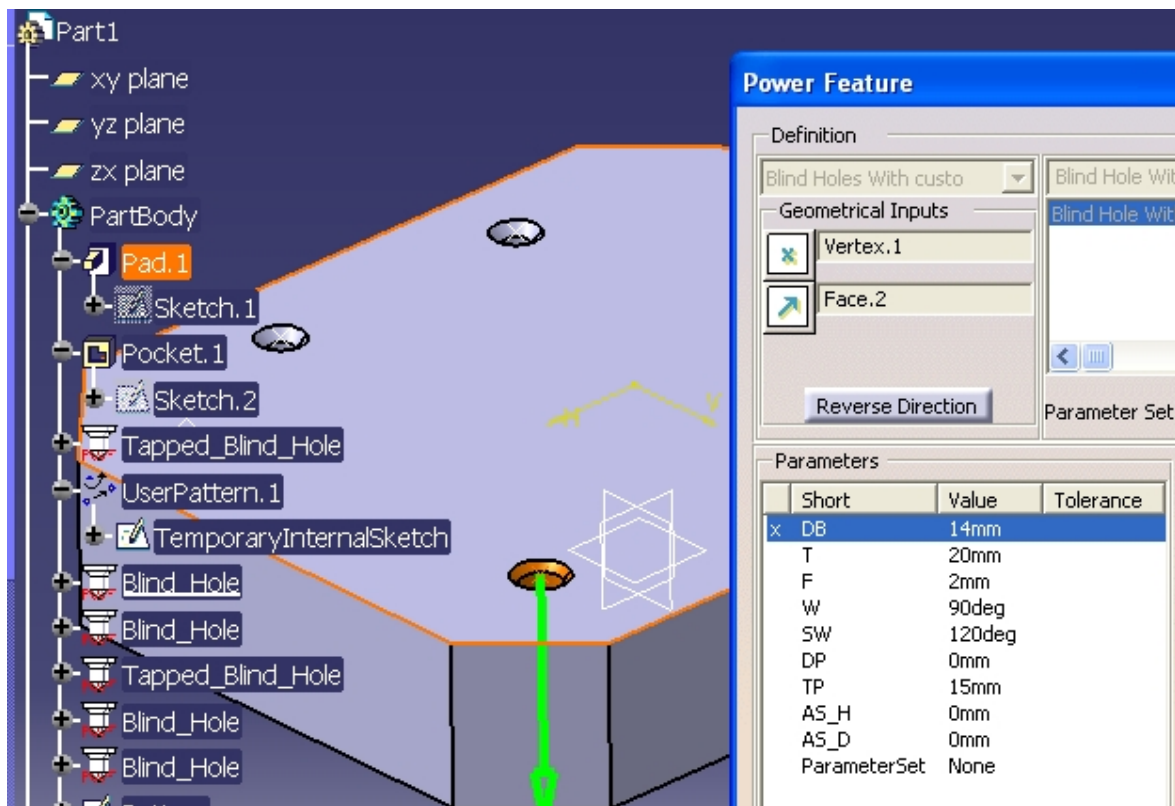


Run Update pattern command. And select the pattern:

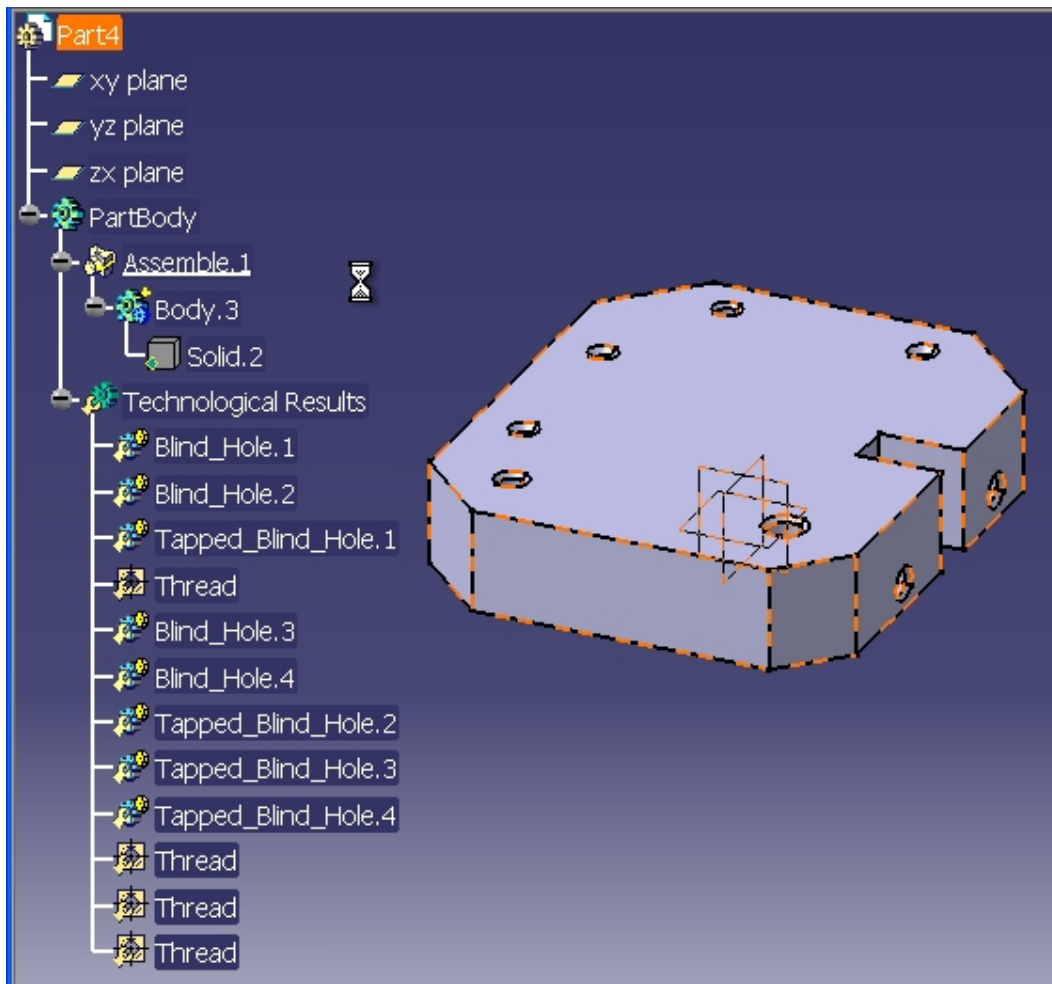


All aggregated MAF are OK.

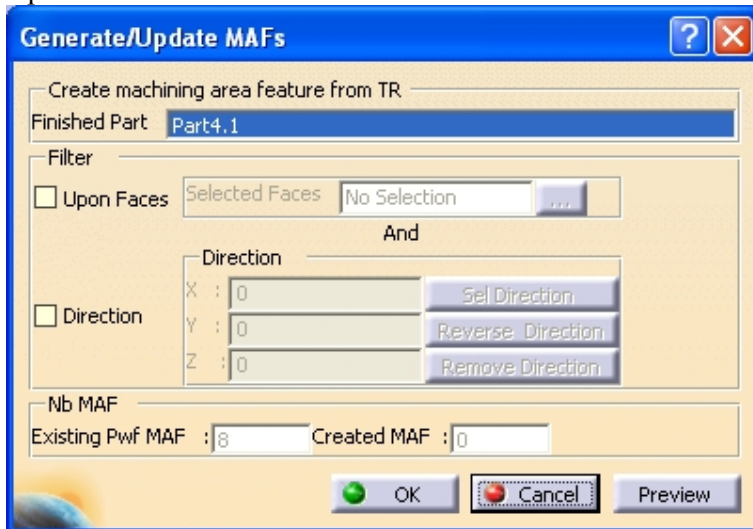
Now go back to CATPart where CAD Power Features are instantiated. Edit on of these Power feature and modify DB value.



Update CATPart where CPWL has been done:



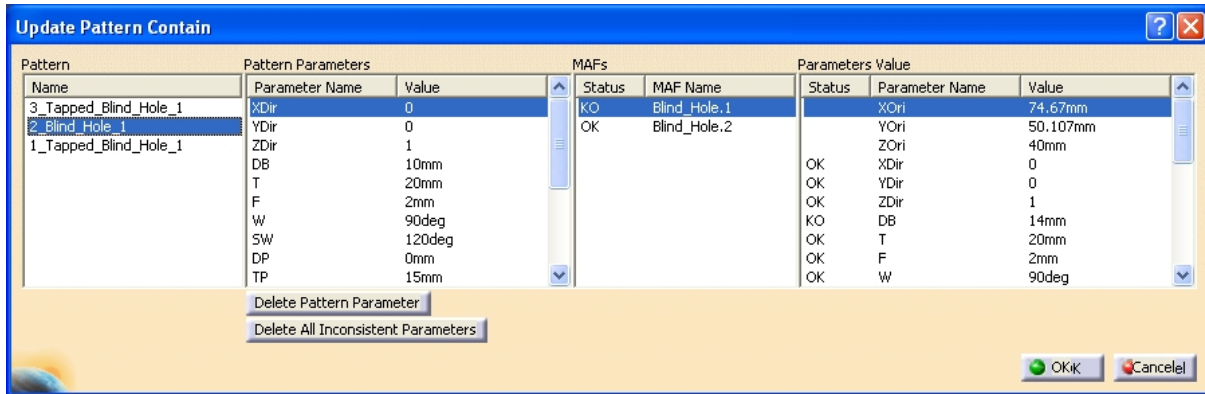
Update MAFs



Run Update Pattern command and select 2-Blind_Hole_1 pattern.

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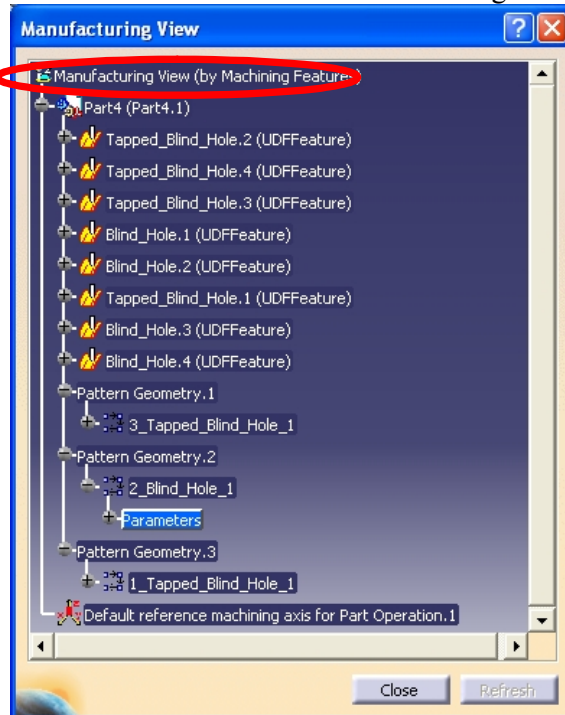


Now you can see that you have a MAF that doesn't correspond anymore to the criteria used to pattern creation. You need to edit the corresponding pattern and remove the MAF.

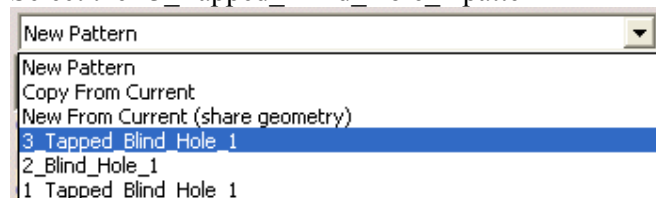
2.6.5 NC link (To create an operation)

1. Creation of a drilling operation

- select the "Drill" icon
- select the root node in the Manufacturing View



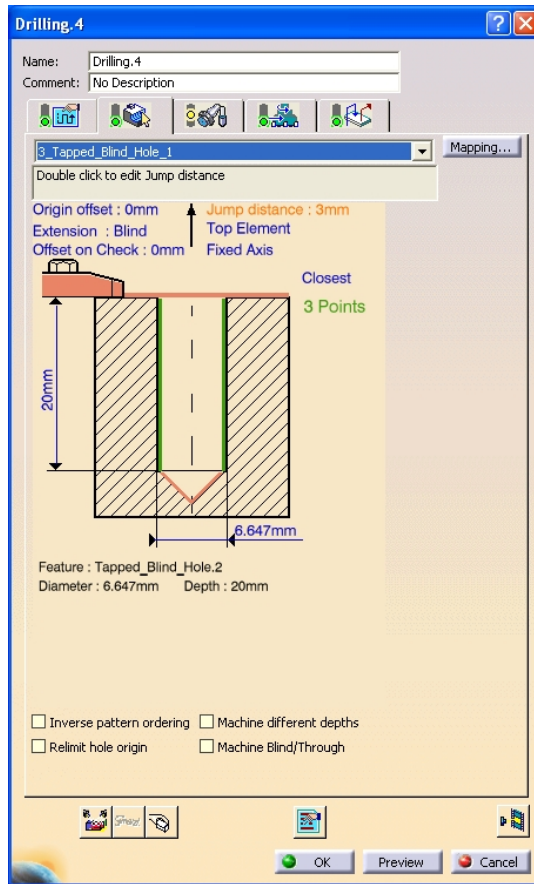
- Select the "3_Tapped_Blind_Hole_1 pattern



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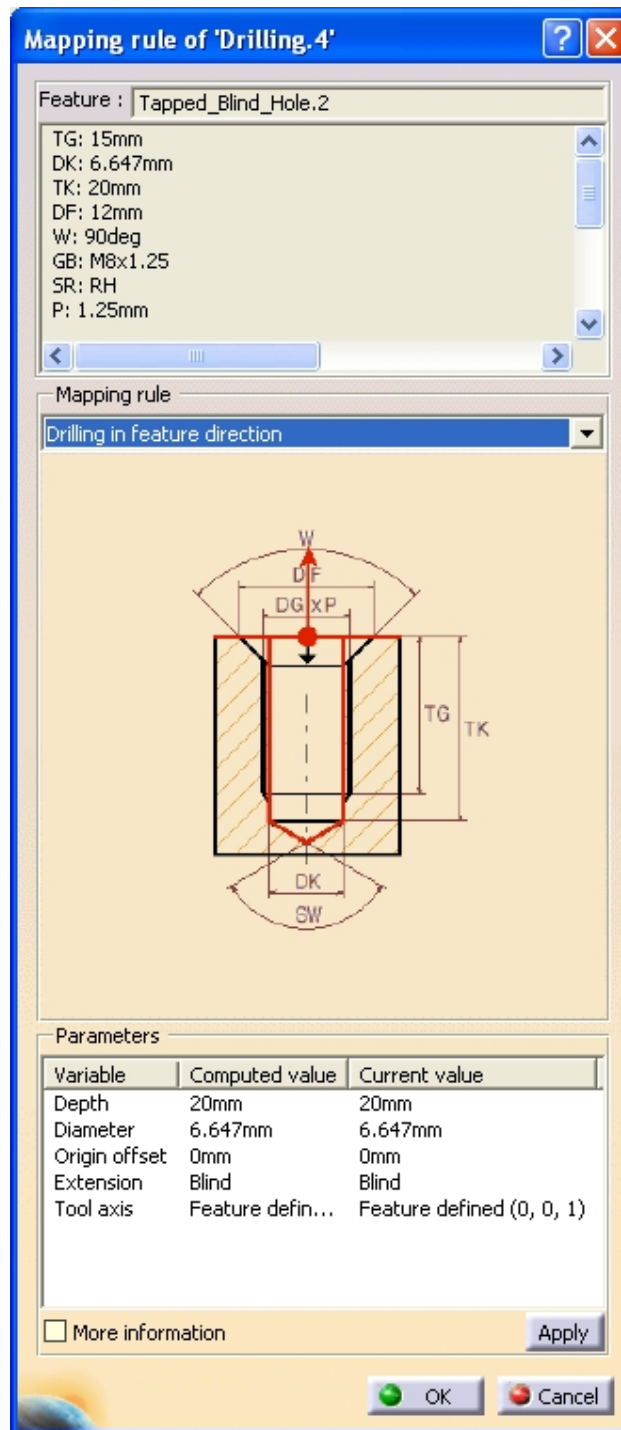
- d. Positions are retrieve and all pertinent MAFs information (diameter, Depth, ..)
- e.



- f. now click on the “Mapping...” button and ...

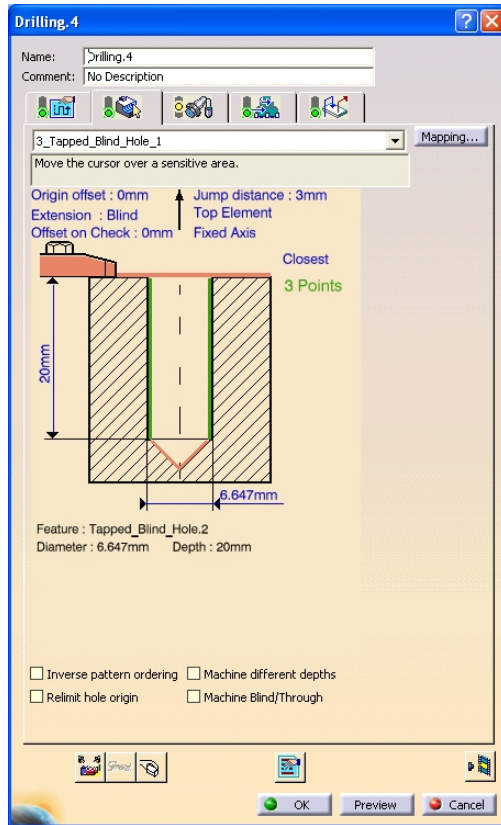


select “Drilling in feature direction”

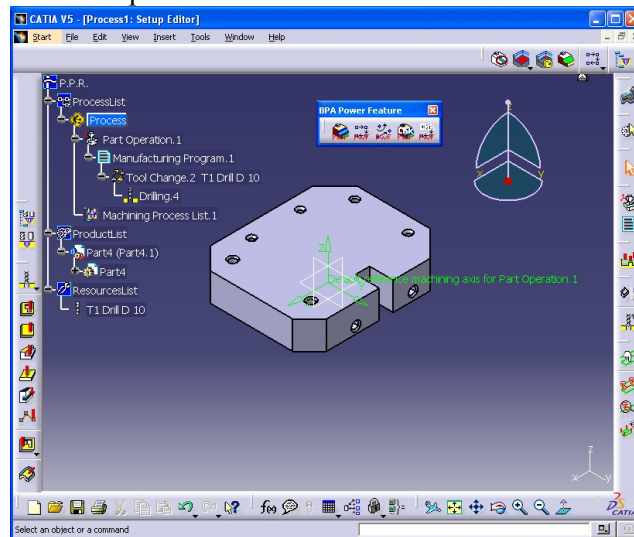


In Points 1 and 2 the PowerFeature parameters have been retrieve. Validate by clicking on “apply”.

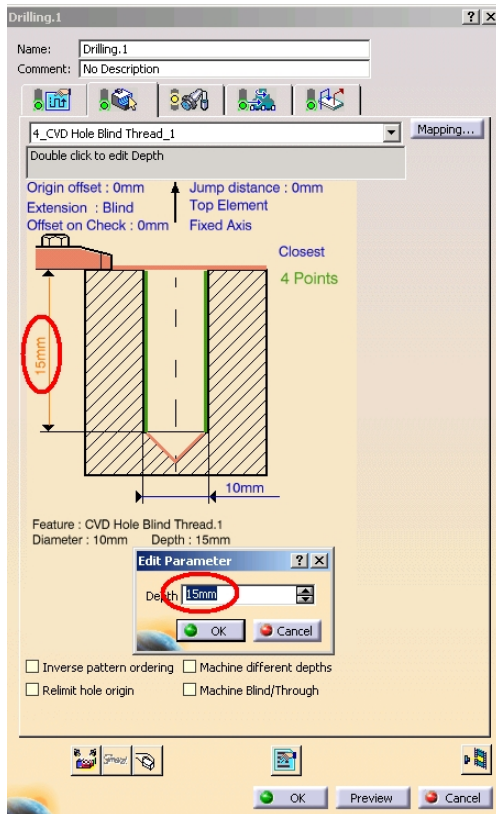
- g. Diameter as be modified corresponding to the rules



h. And the operation have been created



i. If no Mapping rules are applied, you can modify Drilling parameters by double clicking on:



If there is a mapping rule, you have to use $f(x)$ function to change the parameter.

