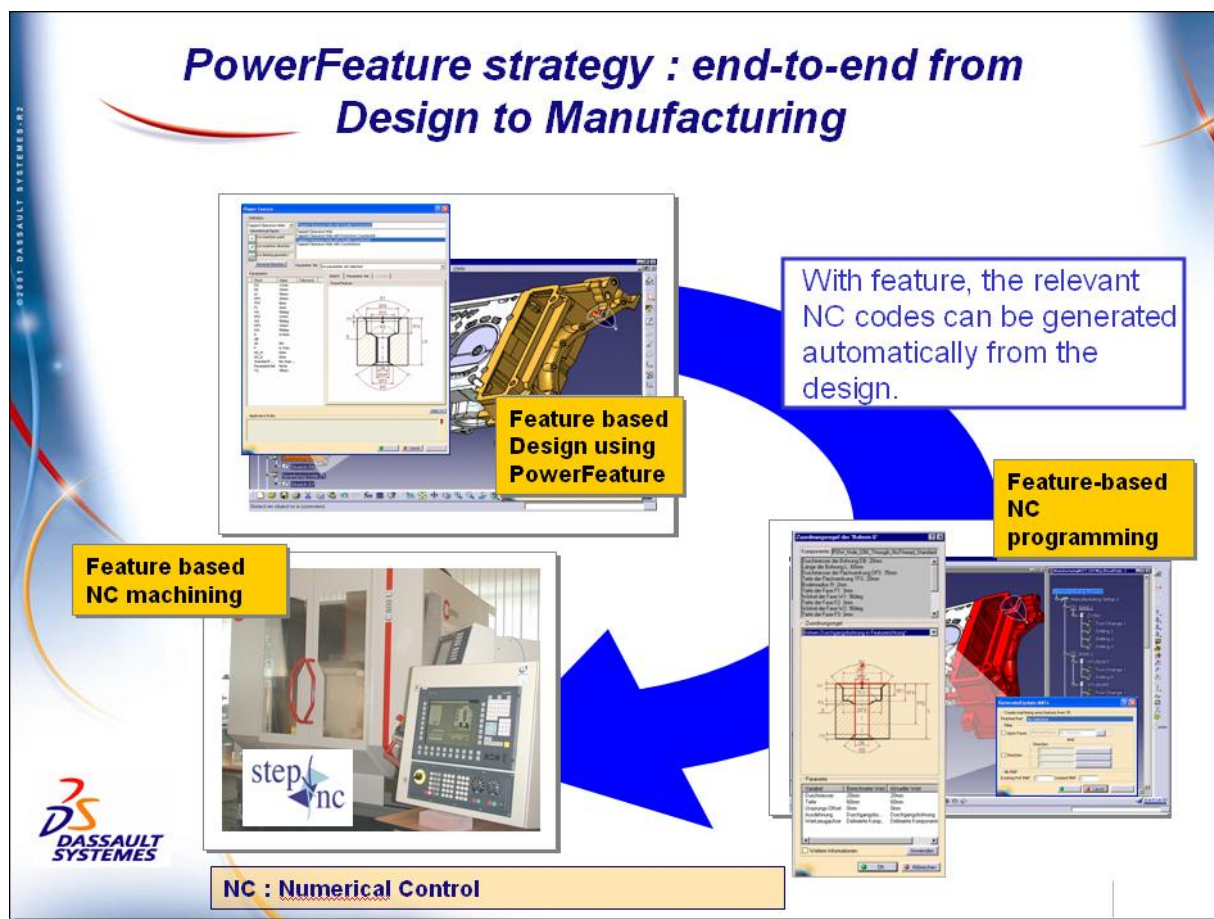


Power Feature (PF9)

Implementation guide



BPA Delivery 7 for V5R19 (V5.7) Version 0.0

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Attention:

This document is a short presentation of the BPA Power Feature. So, for a good utilization of the BPA, you have to read the user's guide given in the pack. This guide describes the functionalities through an example.

1 Introduction

Power Feature 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. Some time Power Feature is symbolized with Pwf.

2 Power Feature Implementation Guide

The Power Feature tool allows you to implement holes from a catalog to your 3D document. First we will describe the CAD part and all the functionalities, then in the NC machining workbench.

The Power Feature 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.

3 Expert User CAD

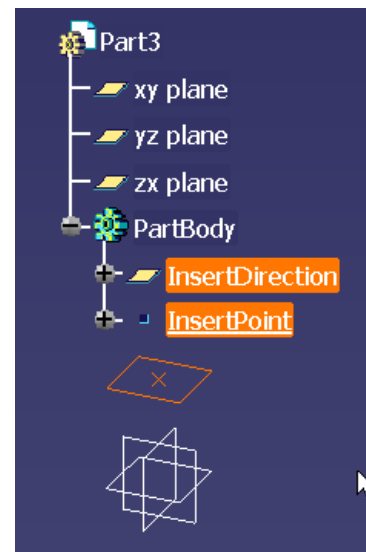
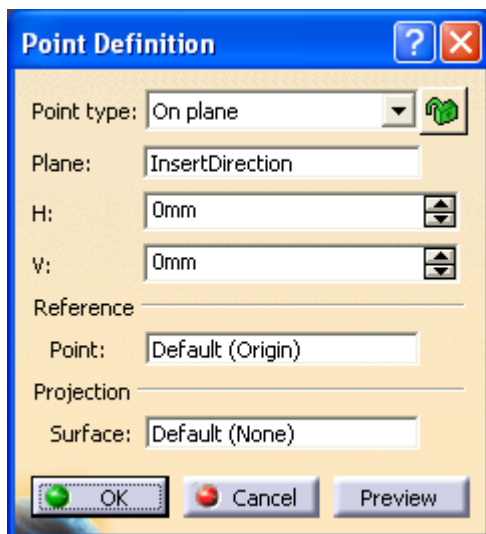
In this part, we will explain how a CAD can provide a new Power Feature

3.1 *Create an UDF*

A power feature is based on User Feature concept. So the first step is to define a User Feature.

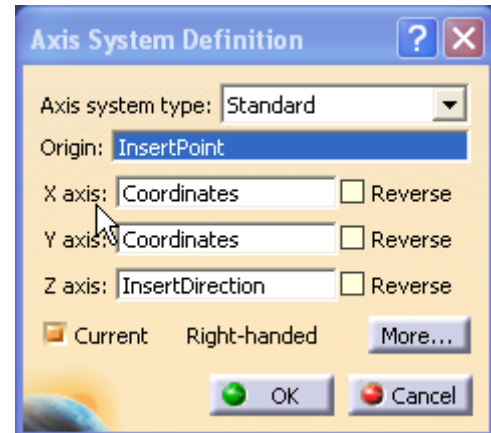
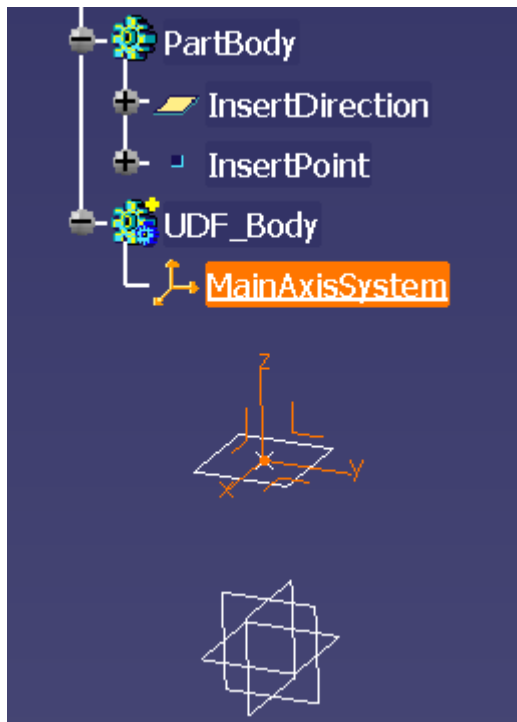
3.1.1 Create the inputs

- 🔸 Create a plane : InsertionDirection under the Main Part body
- 🔸 Create a point : InsertionPoint under the Main Part body
- 🔸 The insertion point must be on the insertionDirection plane.



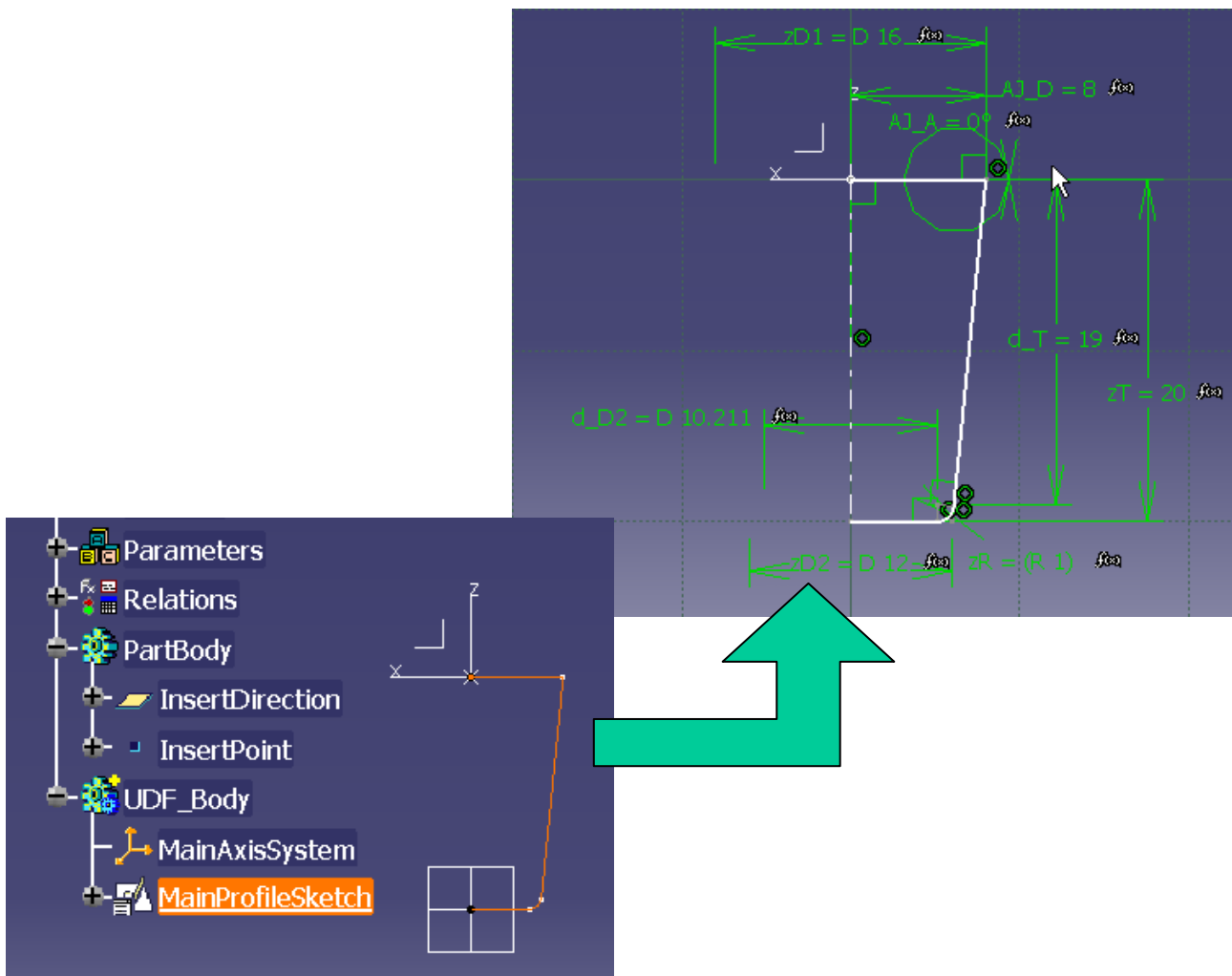
You have now to define the main body and the axis system.

- 🔸 Create a Body : UDF_Body
- 🔸 Create an axis system under UDF_Body : MainAxisSystem with:
 - 🔸 Origin = InsertionPoint
 - 🔸 Z axis = InsertionDirection



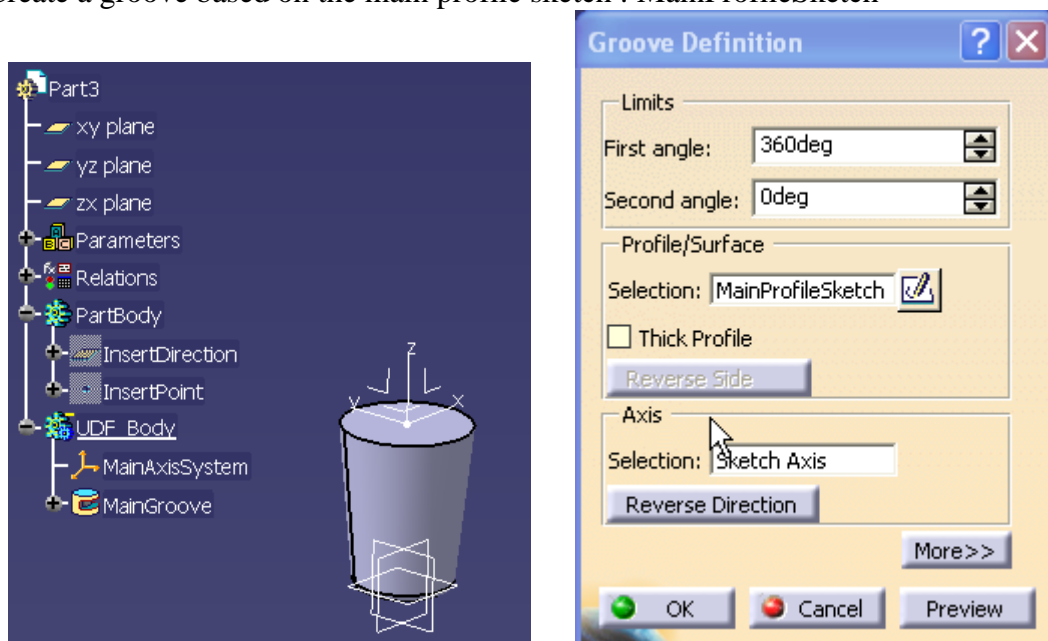
3.1.2 The sketch

- Create your own sketch MainProfileSketch positioned under UDF_Body
- Manage the needed parameters and create the geometrical and applicative relations and functions.



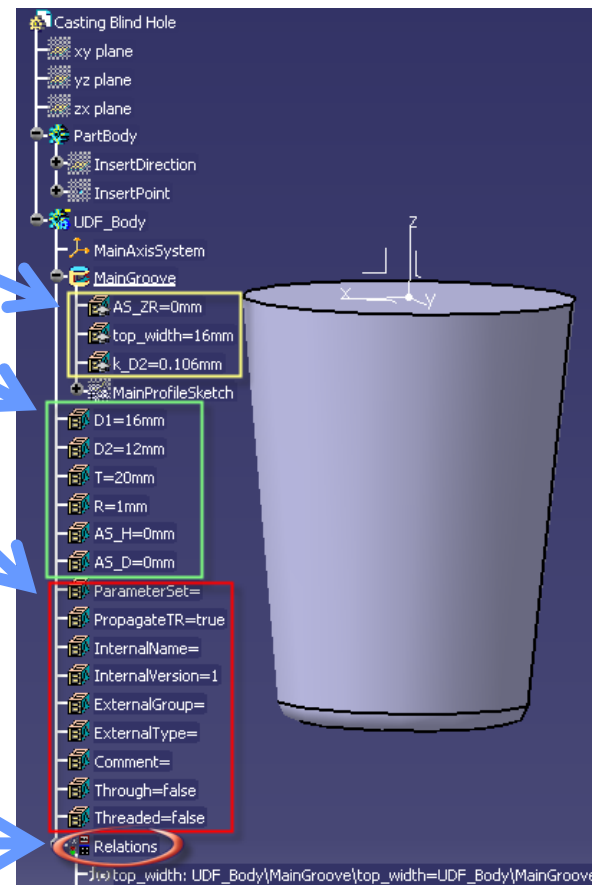
3.1.3 The groove operation

- Create a groove based on the main profile sketch : MainProfileSketch



3.1.4 Relations & parameters

- Attach your parameters:
 - Parameters Lying to the sketch
 - Input/output parameters published by the UDF and managed by the Power Feature Application.
 - Mandatory parameter used by the Power Feature application
- Attach the rules and functions under the UDF_Body
- All the parameters under UDF_Body will be used to generate Technological Results. The parameters which are not directly under the body will not be taken into account (i.e. : the application will not generate a technological Result for AS_ZR or top_width or K_D2 in this example)

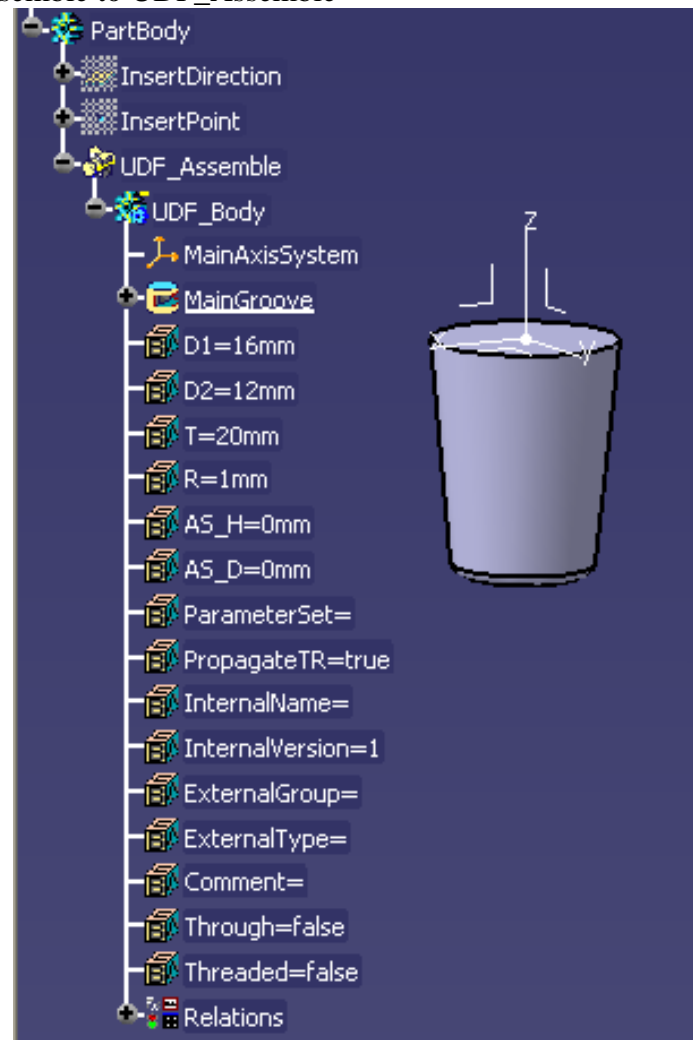


- Some parameters must be set directly under the UDF_Body to be able to generate the Technological results. Otherwise the Machining process will not take those parameters into account.
- Input/output parameters published by the UDF and managed by the Power Feature Application will be shown in the left side of the Power Feature panel.
- The parameter name must not be modified:
 - ParameterSet : string parameter
 - PropagateTR : boolean parameter (must be set by the expert default = true)
 - InternalName : string parameter
 - InternalVersion : integer parameter
 - ExternalGroup : string parameter
 - ExternalType : string parameter
 - Comment : string parameter
 - Through : Boolean parameter (must be set by the expert default = false)
 - Threaded : Boolean parameter (must be set by the expert default = false)
- Important Rules:
 - To avoid problem, parameter publication name must NOT include space (" ") character.

- On threaded Power Feature, Standard Type parameter must be published with “Standard Type” (with space character). This is the only known exception of above rule.

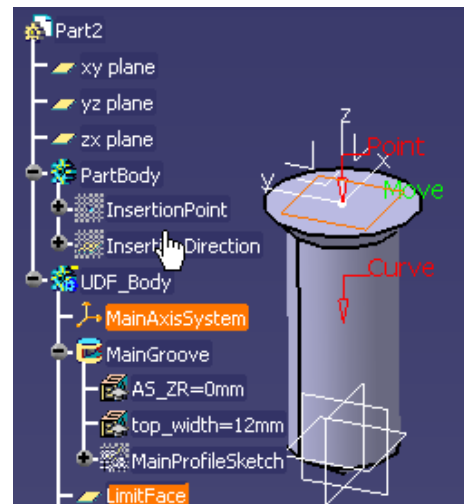
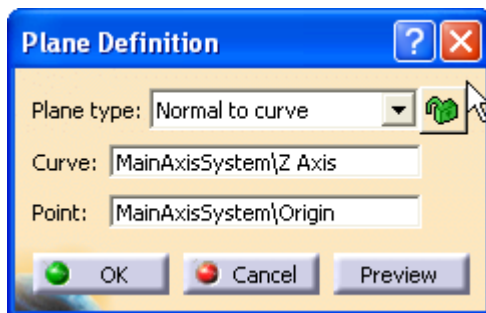
3.1.5 Assemble

- Set PartBody as the in work object
- Select UDF_Body
- Use the Boolean operation tool bar and launch the assemble command
- Rename the assemble to UDF_Assemble

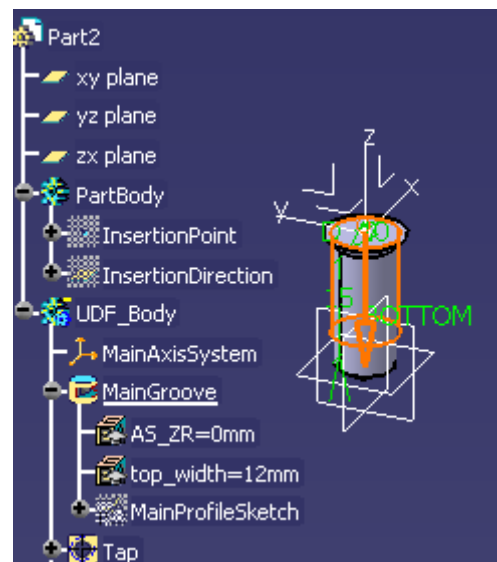
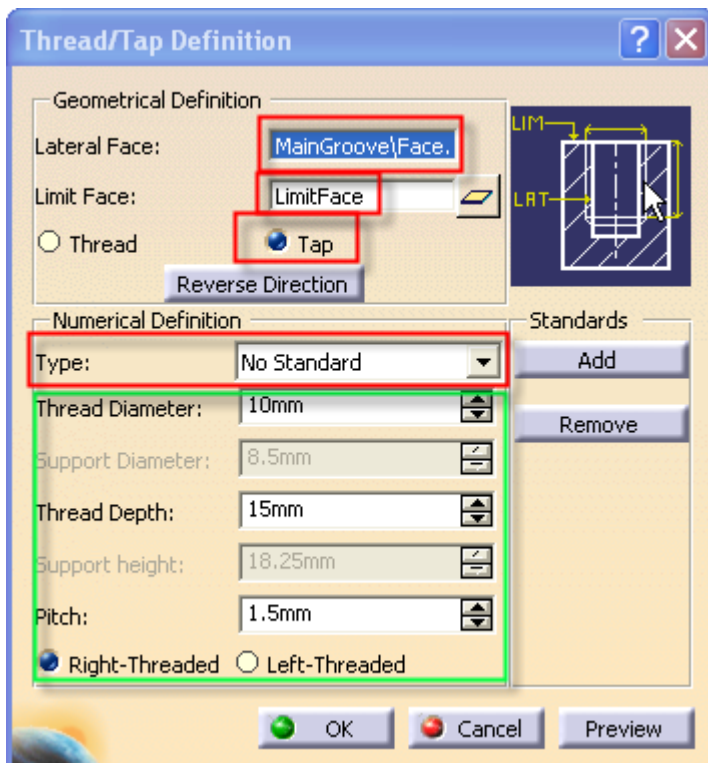


3.1.6 Threaded holes

- Create a plane LimitPlane based n the main axis system : MainAxisSystem
- Plane type : Normal to curve
- Curve : Z Axis of MainAxisSystem
- Point : Origin of MainAxisSystem











- Create a Tap with:
- Lateral Face is the lateral face of the groove.
- Limit Face : the plane created below LimitPlane
- Thread type must be set to No Standard



 Mandatory

 Optional

- Add the following parameters under UDF_Body:
 - ST: Standard Type as String
 - GB : Thread Description as String
 - DG: diameter as length
 - DK: length core diameter as Length

-  P: pitch as length
-  TG: thread length as length
-  SR: Rotation side as Multiple values (LH , RH)
-  Add the following Formulas inside the relations of UDF_Body:
 -  DG : the DG parameter is equal to Tap Diameter (UDF_Body\Tap\Diameter)
 -  TG : the TG parameter is equal to Tap Depth (UDF_Body\Tap\Depth)
 -  P : the P parameter is equal to Tap Pitch(UDF_Body\Tap\Pitch)
 -  GB : the GB parameter is equal to Tap Description (UDF_Body\Tap\ThreadDescription)

3.1.7 Mapping of the thread attributes

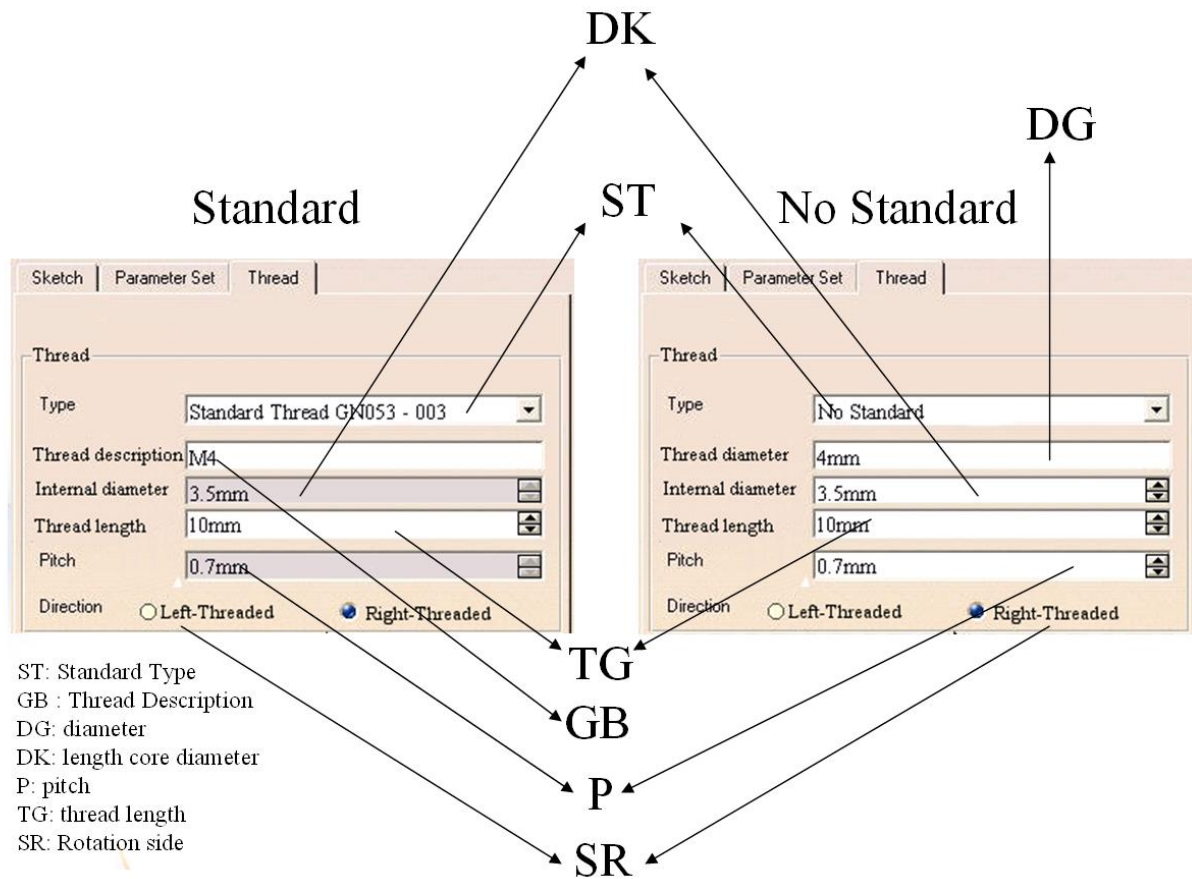
To make a relation between Thread parameters name and panel edition, you need to provide in the GeneralData\resources\msgcatalog\ISPPwfThreadMappingRules.CATNls file the mapping.

Exemple :

```
//TappedHole
TappedBlindHole_1.Thread.Type="Standard Type";
TappedBlindHole_1.Thread.Description ="GB";
TappedBlindHole_1.Thread.Internal_Diameter="DK";
TappedBlindHole_1.Thread.Diameter ="DG";
TappedBlindHole_1.Thread.Length="TG";
TappedBlindHole_1.Thread.Pitch="P";
TappedBlindHole_1.Thread.Direction="SR";
```

If some parameters names are the same for all type (blind or through), you can directly use:

```
//Generic Case
Blind.Thread.Diameter = "DG";
```



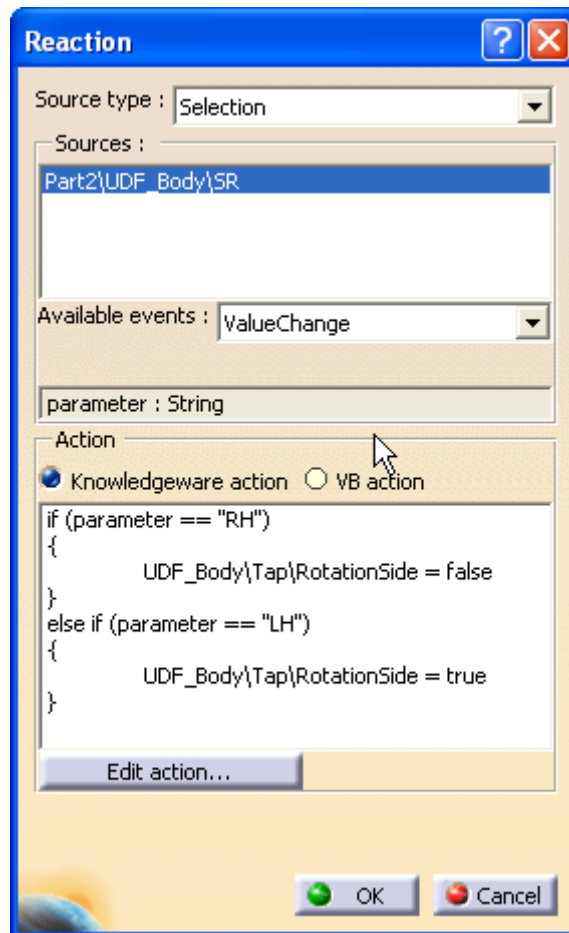
3.1.8 How to manage the thread attributes

- Create a Reaction to manage the Thread Rotation Side.
- Source type : Selection
- Sources : SR parameter
- Event : ValueChange
- Action: Copy and paste the text bellow.

```

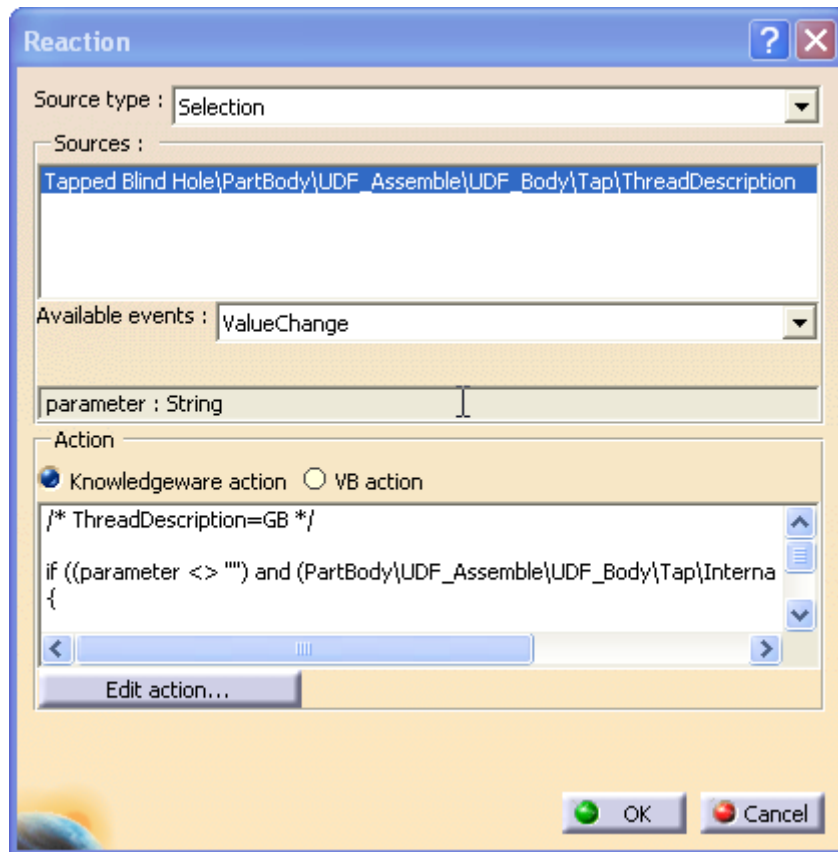
if (parameter == "RH")
{
    UDF_Body\Tap\RotationSide = false
}
else if (parameter == "LH")
{
    UDF_Body\Tap\RotationSide = true
}

```



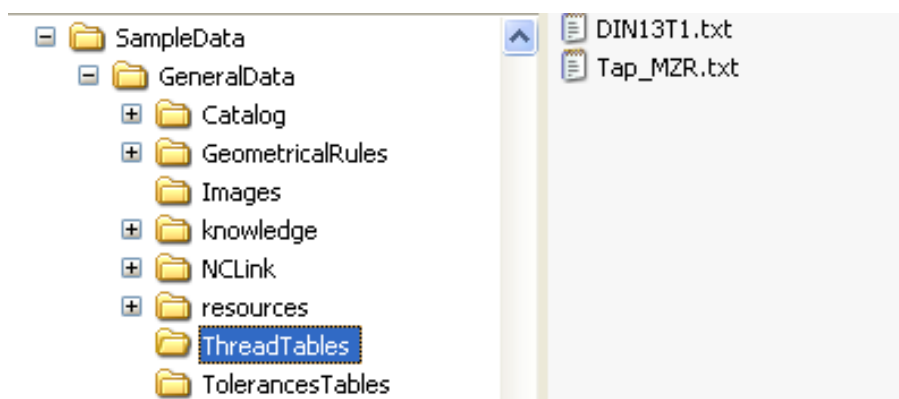
- Create a Reaction to manage the Thread description.
- Source type : Selection
- Sources : ThreadDescription tap parameter
- Event : ValueChange
- Action: Copy and paste the text bellow.


```
if ((parameter <> "") and (PartBody\UDF_Assemble\UDF_Body\Tap\InternalDiameter > 0mm))
{
    PartBody\UDF_Assemble\UDF_Body\DK =
PartBody\UDF_Assemble\UDF_Body\Tap\InternalDiameter
}
```



3.1.9 How to provide new thread standard

- Company can provide its own thread standard.
- It must be located on ThreadTables on General data directory



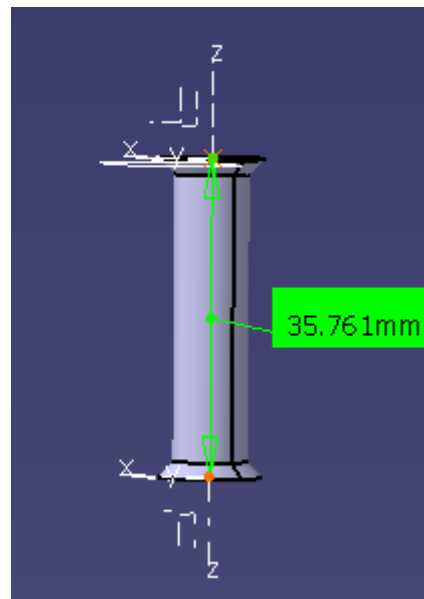
- The format must be standard one's. For more information see CATIA document :
 \$v5doc/doc17/online/cfyugprt_C2/cfyugprthole3.htm#hj-reusing

3.1.10 Clearance holes

- To manage the clearance hole you need to create another input under the main part body : LimitingSketchAxis
- This plane can be used:
 - 👤 To compute the position of the limiting plane of the clearance hole
 - 👤 To be used as it
- Create a new parameter for the Length of the clearance hole:
 - 👤 Name of this parameter could be any valid Name
- Create a new formula which automatically compute the value of this Parameter:
- Specify the name of the Parameter in ISPPwfParametersDescription.CATNls file
- 👤 Example.

```
//TappedClearanceHole
TappedClearanceHole_1.Depth ="LK";

//ClearanceHole
ClearanceHole_1.Depth ="L";
```



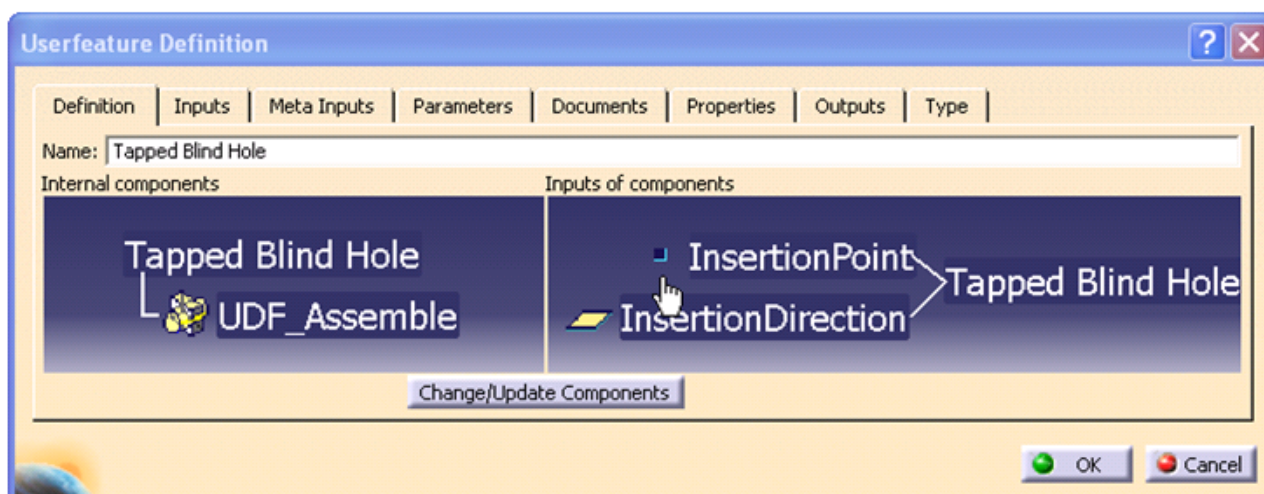
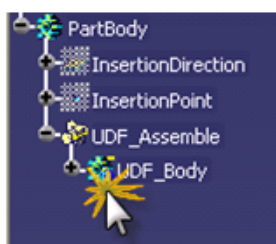
3.1.11 Parameters used by the application

- A specific Knowledgeware string parameter is created and aggregated into the UDF_Body for each fitted Diameter. Its name is 'Fit_Tolerance_XXX', where XXX is the name of fitted Diameter. Ex: FitTolerance_DB=H7.
- Some parameters need to be managed by the application but we never need to see them inside the Power Feature panel. Those parameters will be called 'UIHidden_xxx' where xxx is the name of the hidden parameter.
- additional_length is a specific parameter to manage a custom additional length.

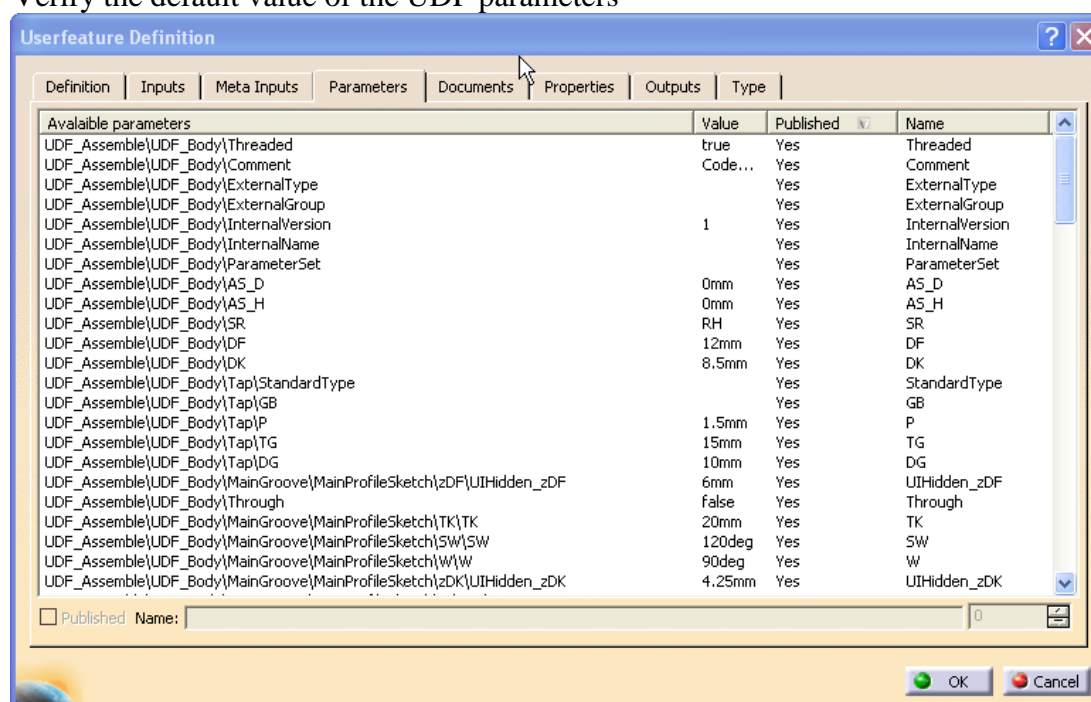
3.1.12 UDF Definition

- Create an UDF based on UDF_Assemble

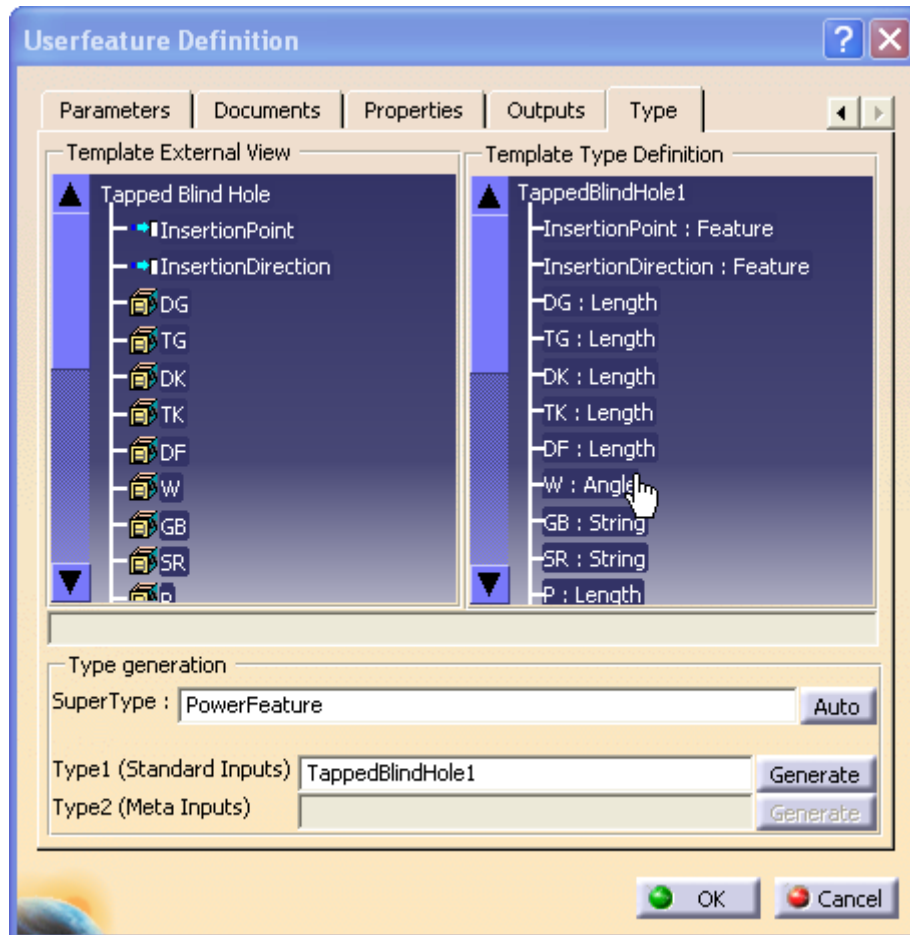
- Verify the Inputs components contain
 - InsertionPoint and InsertionDirection for simple or threaded holes.
 - InsertionPoint, InsertionDirection and LimitingSketch for Clearance holes.



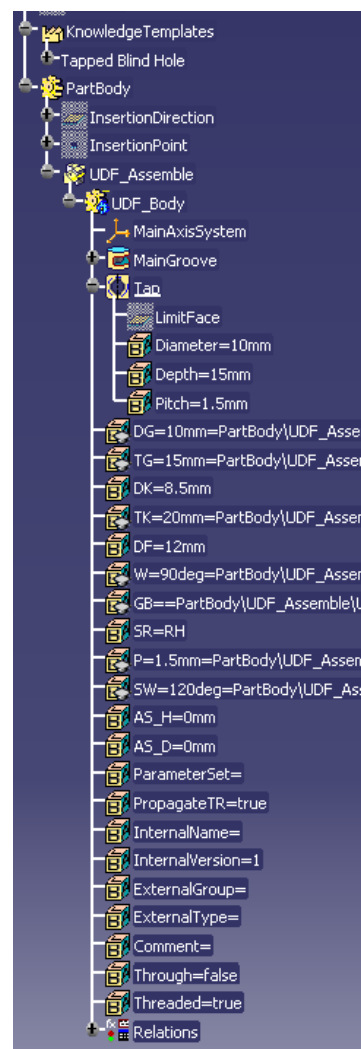
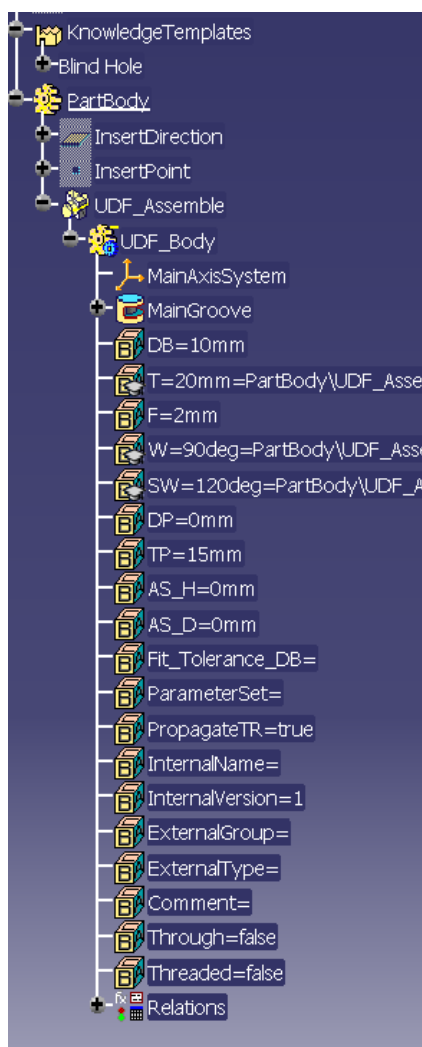
- Publish all the needed parameters
- Rename the parameters by removing the local path parameter
- Verify the default value of the UDF parameters



- Define the SuperType : Power Feature
- Push the Generate push button
- Define Type1 which is normally automatically filled
- The Template Type Definition will contain the tree of the new Type. If it is not the case then verify your environment variable CATKnowledgePath and contact your administrator.



- TR definition: All parameters that are under the UDF body definition will be propagated by the TR.
- a boolean Parameter named "PropagateTR" is mandatory and must be set to TRUE to benefit TR behavior

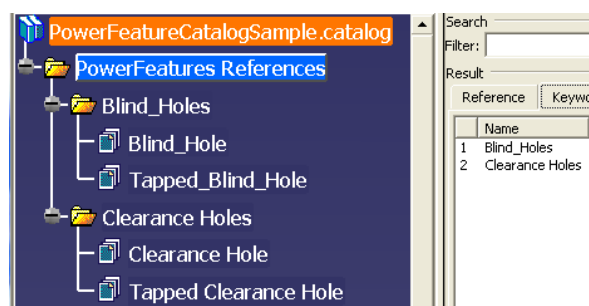


3.2 Power Feature Catalog

If a Power Feature catalog doesn't exist, you need to create it. If it exists, you could add your new power feature in it.

3.2.1 Architecture

Power Feature panel gives a specific view of the power feature catalog. Following catalog:



It is seen as bellow:



3.2.2 Architecture rules

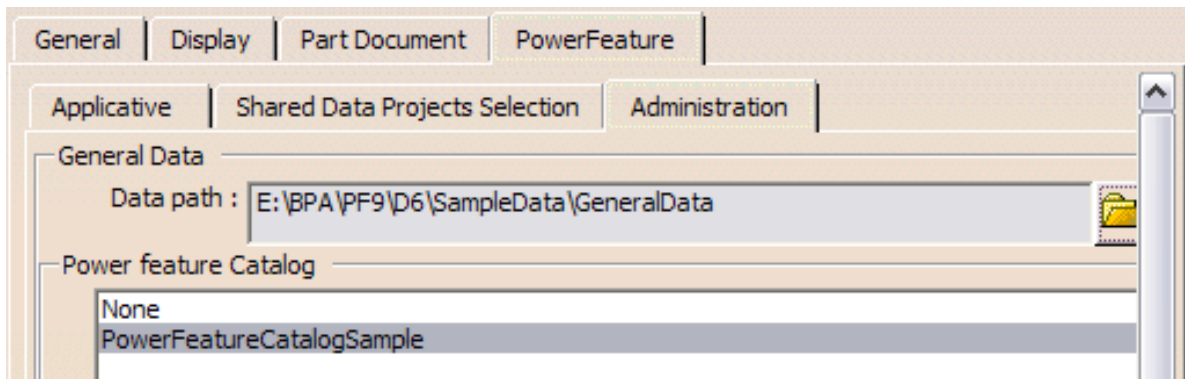
- No rule about the name of the file, neither the name of the root of the file
- Root must contain only chapter that correspond to the group
- A group can not contain other groups
- A group contains only family that represents a type
- No keyword on the group is taken in account (except "Name" of course)
- A family / type must have one keyword named « Version » that corresponds to the version of the UDF
- The family pointed a document that contains one and only one UDF
- The name keyword must not contain any blank. The name is called « Internal Name » in the code and the other descriptions. It is used to find file. It must be unique in the catalog.
- It's always the last version of the UDF that we can instantiate (the highest Version number even if it's not the last in list of UDF)

To be able to customize Power Feature panel command, avoid using blank in the group and type name.

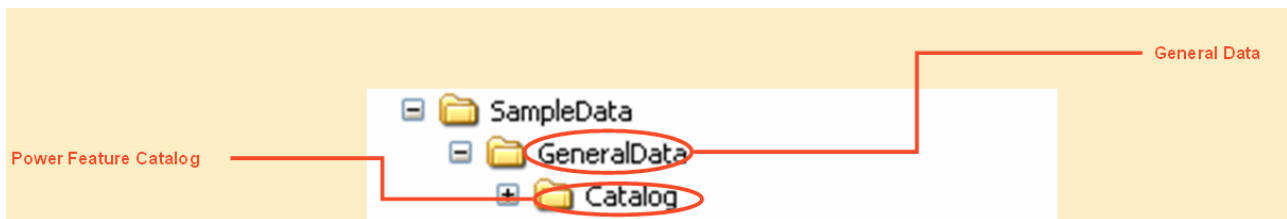
You can add as many group and type as you want.

3.2.3 Catalog and Power Feature settings

Tools -> Settings

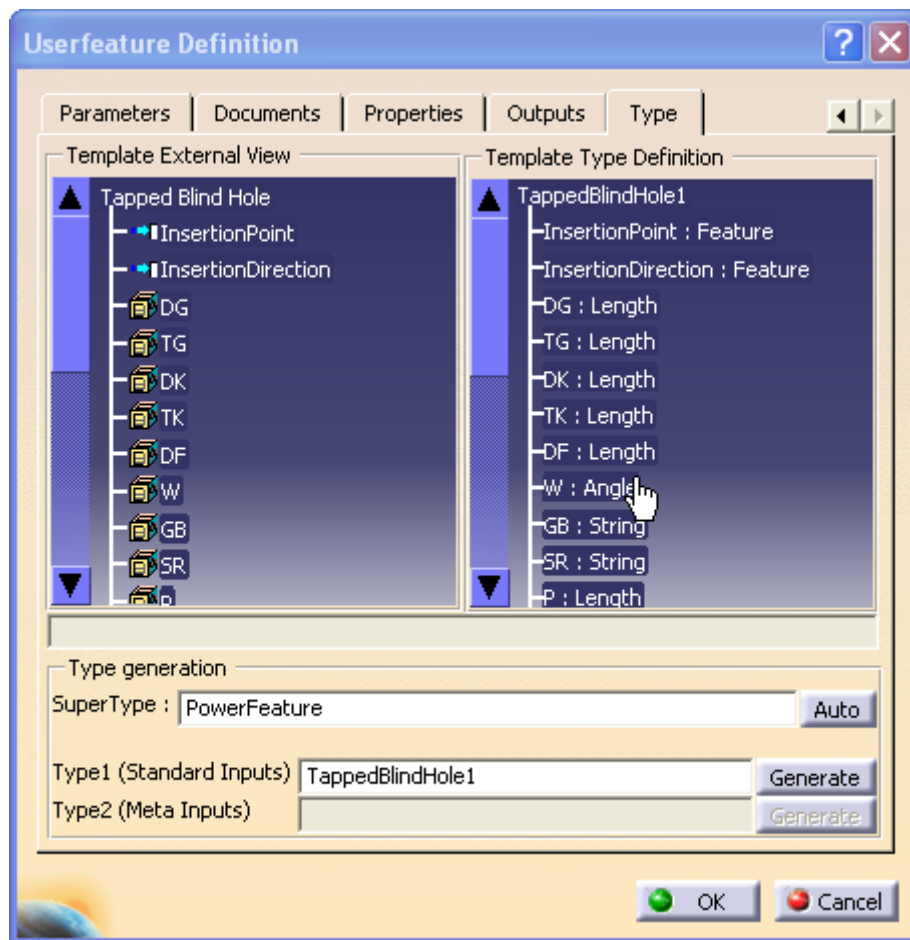


- Catalog is found through the setting.
- Activation of the active catalog is mandatory
- Only one udf catalog could be active
- Catalog must be located in General Data /Catalog.
- « Catalog » directory name is defined as is and can not be change. Nevertheless the catalog file name is free.
- It may contain as many catalogs you want
- CATPart document could be located anywhere. In this case, on Data directory we store UDF definition CATPart document is located under catalog directory.

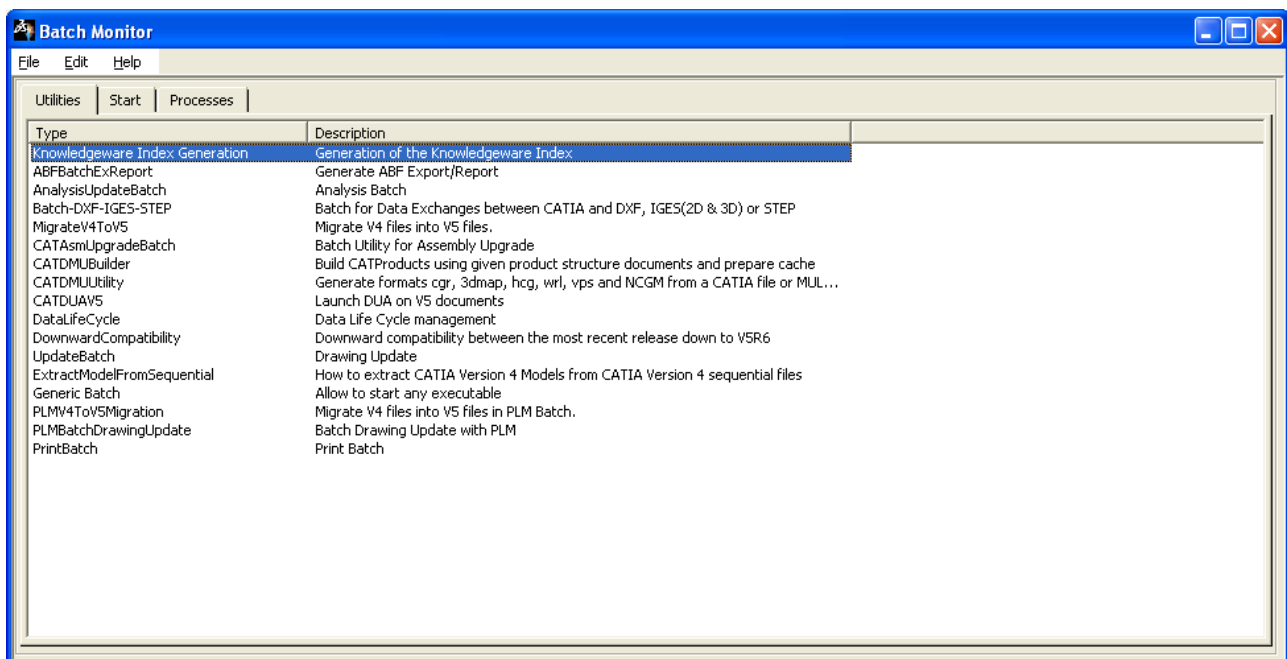


3.3 Knowledge index

To be able to use a Power Feature in Rules, A knowledge index file must be generated. Generate the type through “generate command” in User Feature Definition



Create a file in GeneralData\knowledge\knowledgeTypesCustom (if CATKnowledgePath is well set). Generate index create a binary index file that define the Power feature as a new feature type. (Run Tools->Utilities ->



For more information have a look to [\\$v5doc/online/cfyugkwr_C2/cfyugkwrarm0001.htm](#)

3.4 Applicative rules

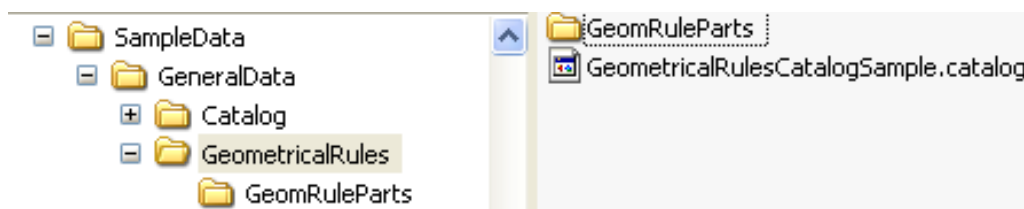
Now, we are about to see how to create the applicative rules:

- The first idea of the applicative rules is to define geometrical rules to avoid geometrical update problem.
- Business are define later one to warn the end user that parameters he/she defined could create problem (to the manufacturing for example)

3.4.1 Catalog architecture and location

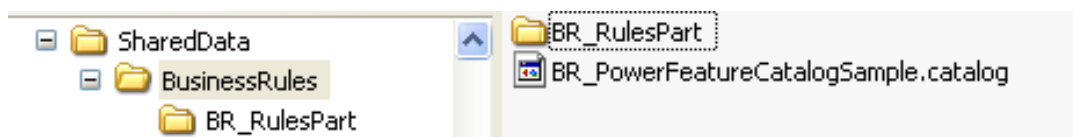
Geometrical and Business catalogs must have the same architecture as power Feature catalog. Group and type must have the same name. The geometrical catalog must have the as many group, type and corresponding CATPart. The Business catalog could miss some group/Type.

- For Geometrical rules, a directory named “GeometricalRules” must exist in General Data directory
 - It may contain as many catalogs as you want



- Corresponding CATPart must be located anywhere

- For Business rules, a directory named “BusinessRules” must exist in Shared Data directory
 - It may contain as many catalogs as you want

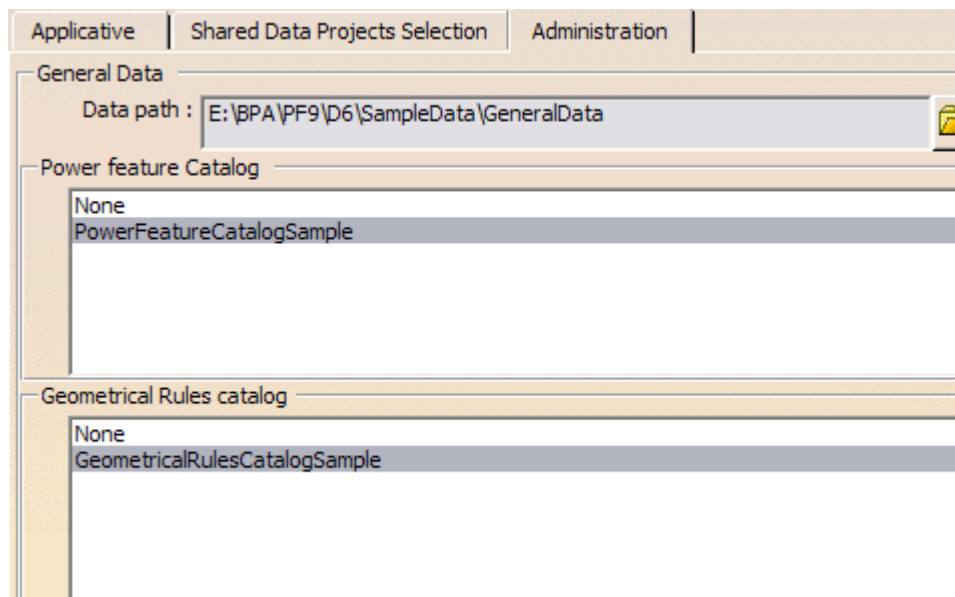


3.4.2 Catalog Settings:

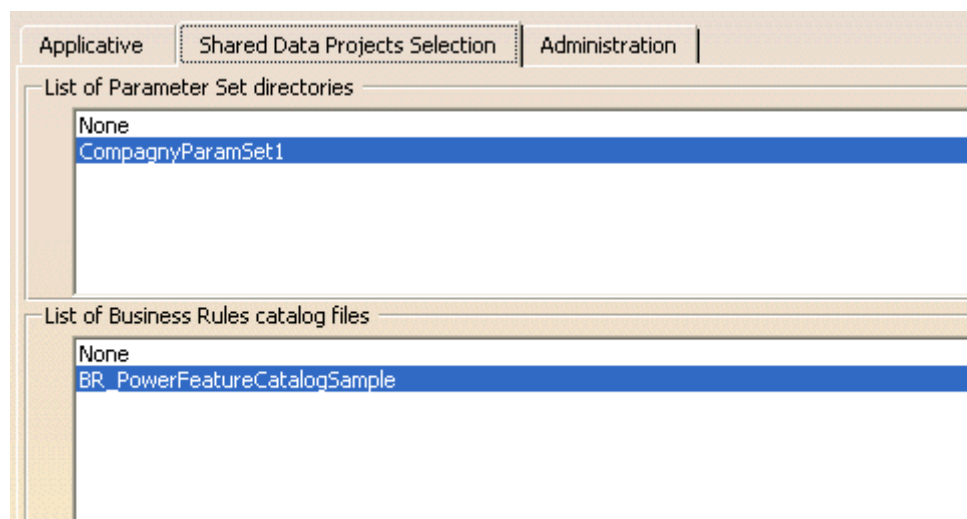
- Geometrical and Business rules could be apply or not depending of the settings
- In tools->Options-> Infrastructure -> Part Infrastructure -> Power Feature ->Applicative



- In tools->Options-> Infrastructure -> Part Infrastructure -> Power Feature ->Admin, a list gives the catalog the application
- Administrator should set a catalog as active one for the Geometrical one's. A catalog for Geometrical Rules is mandatory.

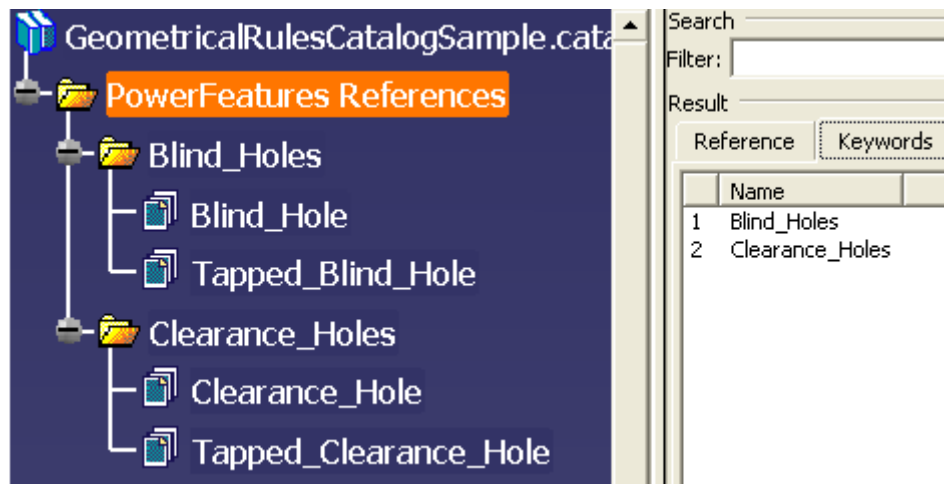


- In tools->Options-> Infrastructure -> Part Infrastructure -> Power Feature ->Shared Data, a list gives the catalog the application
- Administrator should set a catalog as active one for the Business one's. The Business Rules catalog can be used optionally.

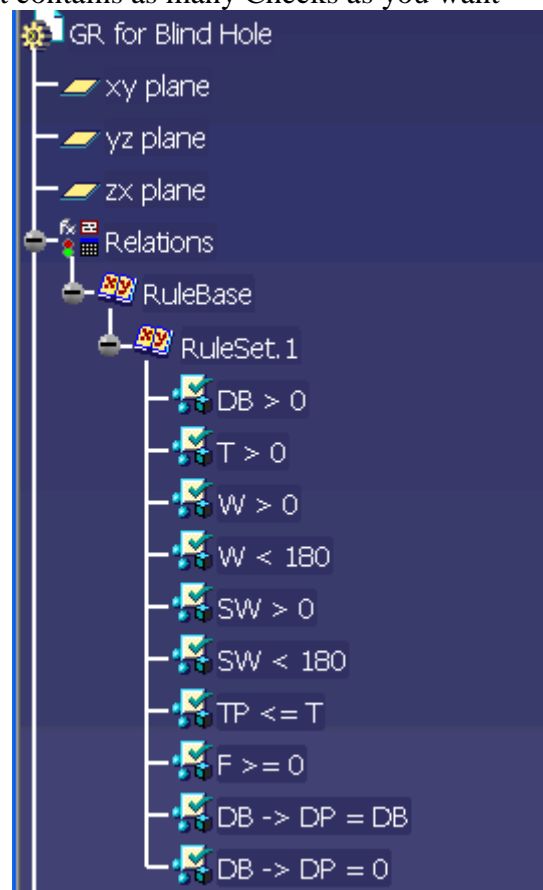


3.4.3 Catalog rules content

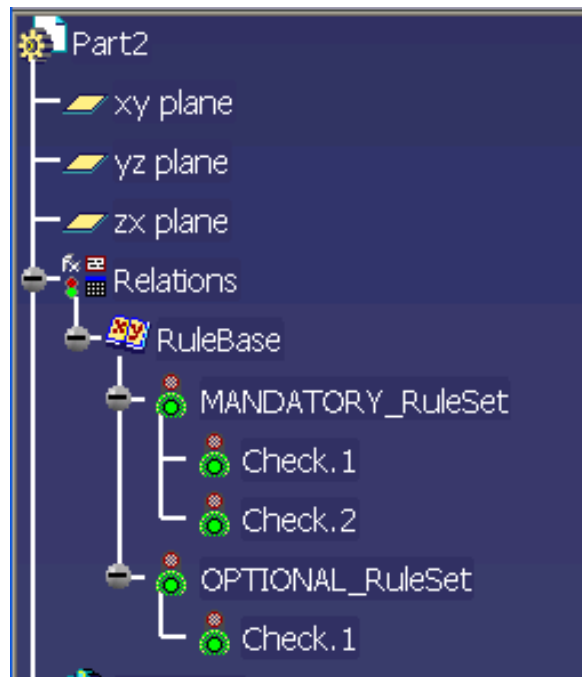
The Applicative Rule catalogs must exactly correspond to the UDF catalog structure. This means catalogs contain the superimposed structure of all Power Feature UDF catalogs.



- There is no specific architecture concerning the CATPart including the Geometrical Rule: Usually, one RuleSet contains as many Checks as you want

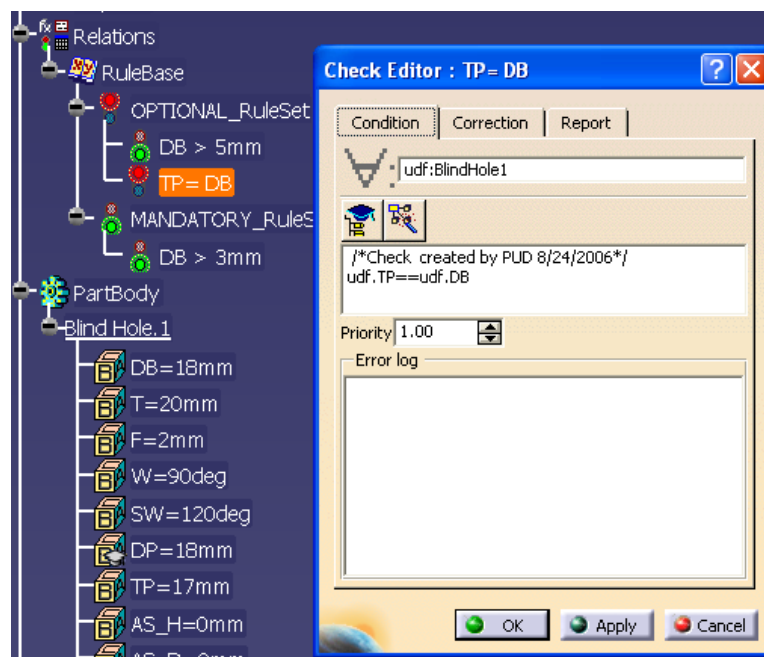


- The architecture of the CATPart including the Business Rule is constraint. The mandatory Rules Set must have the prefix string "MANDATORY_" and the optional set must have the prefix string "OPTIONAL_" in its name.



3.4.4 Rule description

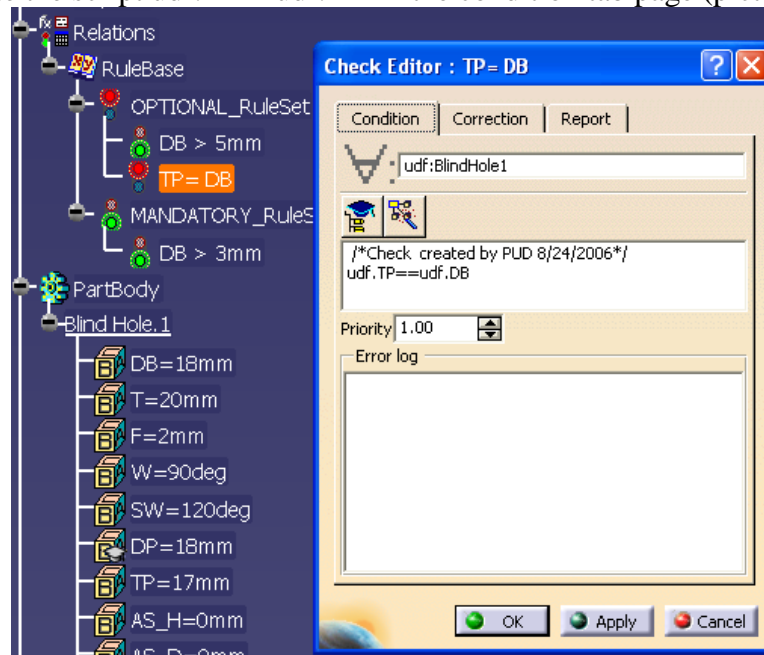
- In Condition tab page, you must define the condition that set the Rule OK or KO
- If the rule is KO, the BPA displays the text set in « Report » table page. Fill this tab page is mandatory to inform the end-user about the Problem
- If the rule is KO, a corrective action could be launch



3.4.5 Corrective action

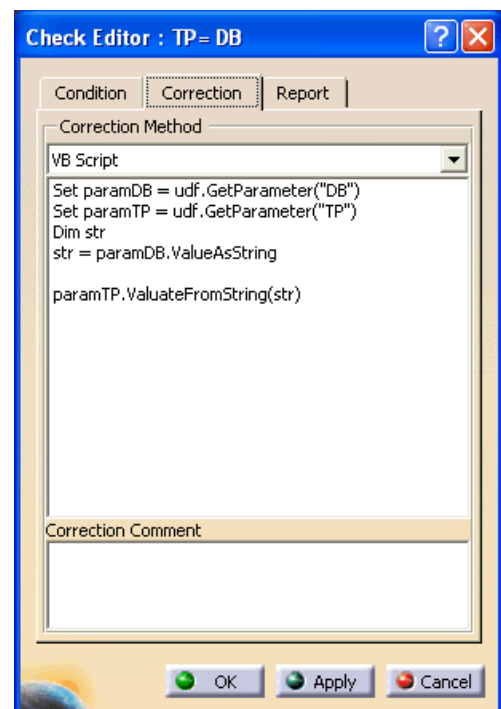
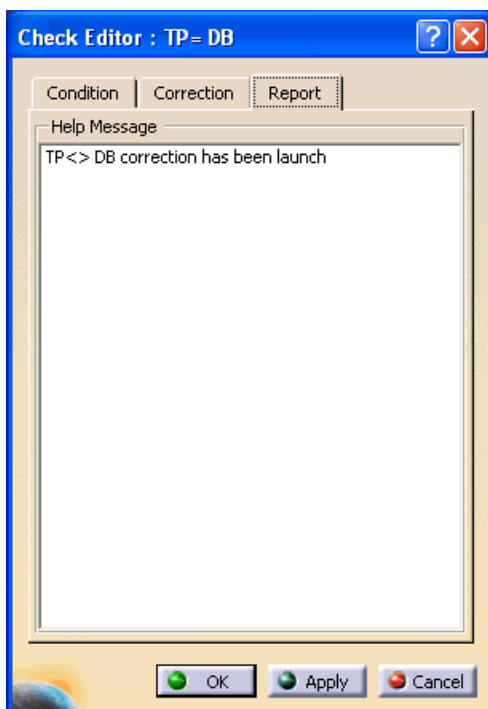
- If a rule is not validated, the BPA launch the correction function if it exists
- Process :

- 👤 Open the CATPart containing the Business rule on the UDF
- 👤 Create a rule named for example TP=DB
- 👤 Write the script `udf.TP==udf.DB` in the condition tab page (picture 1)



Process :

- 👤 In the report tab page write the error message
- 👤 And in the correction tab page, write the script to correct the condition. The script will be automatically run in the BPA.



3.4.6 Applicative rule process

- Retrieve CATPart document where Applicative rules are stored for this Pwf
- Retrieve Rule base object
- Instantiate
- Set Pwf instance as object to test
- Apply rule and retrieve result
- If one rules is red, read all the report to be display in the panel and run corrective function if exist
- First Geometrical Rules are test, and if there are OK, test Business rules

3.4.7 Rules

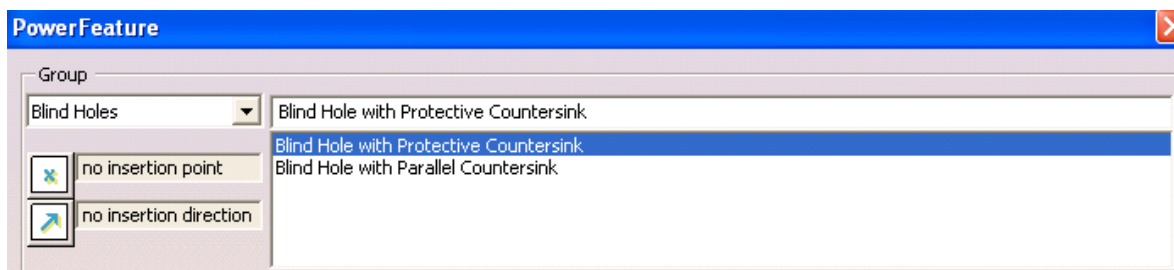
- 1 - Rule catalog is not set in tools-Options:
 - 👤 Geometrical rules: the application must not continue. It is a show stopper. No update/modification of UDF is allowed.
 - 👤 Business rules: the application continues. It's not a show stopper. A warning inform the end-user
- 2 - The catalog is set, but we can not be opened : (e.g. due wrong rights)
 - 👤 Geometrical rules: the application must not continue. It is a show stopper. Show warning access problem
 - 👤 Business rules: the application continues. It's not a show stopper. A warning inform the end-user
- 3 - The catalog is set, the document is opened, but we can not found the Group and type in the structure :
 - 👤 Geometrical rules: the application must not continue. It is a show stopper.
 - 👤 Business rules: the application continues. It's not a show stopper A warning inform the end-user
- 4 - The catalog is set, the document is opened, we found the Group and type in the structure, but we can not open the target file :
 - 👤 Geometrical rules: the application must not continue. It is a show stopper.
 - 👤 Business rules: the application continues. It's not a show stopper... A warning informs the end-user "rule file not found".
- 5 - The catalog is set, the document is opened, we found the Group and type in the structure, we open the target file, but the CATIA Rules can not be found :
 - 👤 Geometrical rules: the application must not continue. It is a show stopper.
 - 👤 Business rules: the application continues. It's not a show stopper A warning inform the end-user
- 6 - The catalog is set, the document is opened, we found the Group and type in the structure, we open the target file, we found the rule(s) :
 - 👤 Geometrical rules: if the rules are not granted, the application must not continue. It shows and panel. The application doesn't check the business rules? The application stops here until the rules get granted (are Ok/true).
 - 👤 Business rules: if the mandatory rule is not granted, the application must not continue. It shows and panel .But; does the application check the optional rules? >> No, there is no need in going on.

- 7 - If Geometrical Rules are ok, then check Mandatory-Business Rules, if Mandatory-Business Rules are Ok, then check Optional-Business Rules. If the optional rules are not granted, the application continues but display a panel.
- 8 - If none is set as Business rule catalog, no warning panel appears when click Ok on Pwf instantiation panel command. When click on Apply, a message warn the end-user that no BR has been applied.


3.5 Group/Type customization

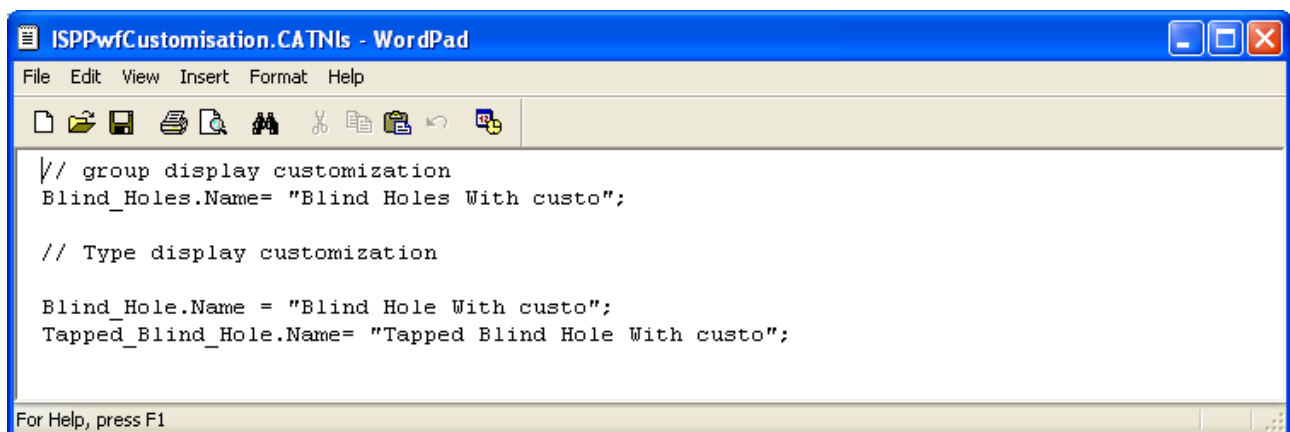
3.5.1 Use

- Power Feature CAD command gives a list of Groups and types.
- By default names are retrieved through Power feature catalog. You can customize the display of these names through ISPPwfCustomisation.CATNls file.



This file must be located in resources/msgcatalog and added to the CATMsgCatalogPath CATIA variable

- The syntax inside the document is
 « Group or type catalog name».Name="Customized name" ;



- Restriction: In the power feature catalog document, the group/Type name must not contain blank character.

3.6 Parameter Set

For each feature, the application can provide a set of parameter to avoid to set each parameter each time. In one shot, end-user set all parameters.

3.6.1 Architecture

- These parameter sets are stored in a xml file
- Two level of parameter set file exists :
 - On a the level of the shared data directory
 - One at the level of the User Profile directory

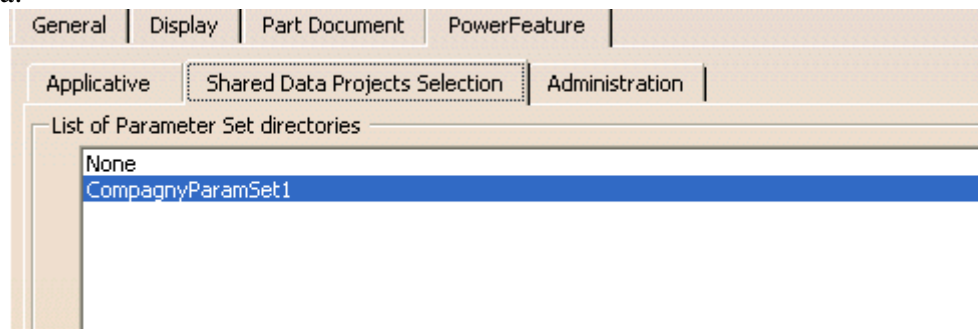


- For Share data, parameter set are stored in a directories
- For User data, parameter set are stored in the Parameter set root directory

3.6.2 Settings

User parameter sets are always accessible, independent

Shared Data:



End user can set or not a directory containing parameters sets.

User data: If in Administration tag page, a path is set to the data Path, the user list of parameters set are always available (no possibility to customize it).

3.6.3 File architecture

The parameter set files are the same for Shared and User data. It exist one parameters set file per power feature per data.

Files are xml file with the following architecture

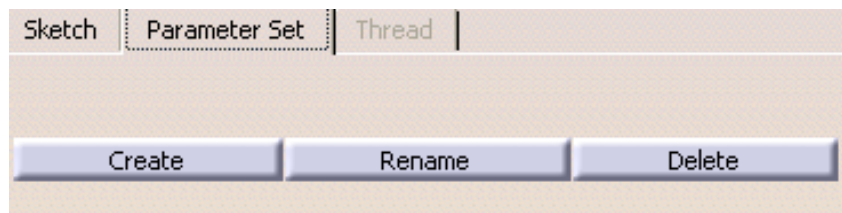
Feature Type name is the concatenation of InternalName+"_" +Version .

```
- <Parameters>
- <FeatureType Name="BlindHole_1">
- <ParameterSet Name="compagnyParamStandard">
  <Parameter Max="0.0" Min="0.0" Name="DB" Unit="mm">10</Parameter>
  <Parameter Max="0.0" Min="0.0" Name="T" Unit="mm">20</Parameter>
  <Parameter Max="0.0" Min="0.0" Name="F" Unit="mm">2</Parameter>
  <Parameter Max="0.0" Min="0.0" Name="W" Unit="deg">90</Parameter>
  <Parameter Max="0.0" Min="0.0" Name="SW" Unit="deg">120</Parameter>
  <Parameter Max="0.0" Min="0.0" Name="DP" Unit="mm">0</Parameter>
  <Parameter Max="0.0" Min="0.0" Name="TP" Unit="mm">15</Parameter>
  <Parameter Max="0.0" Min="0.0" Name="AS_H" Unit="mm">0</Parameter>
  <Parameter Max="0.0" Min="0.0" Name="AS_D" Unit="mm">0</Parameter>
  <Parameter Max="0.0" Min="0.0" Name="ParameterSet" Unit="">None</Parameter>
</ParameterSet>
</FeatureType>
</Parameters>
```

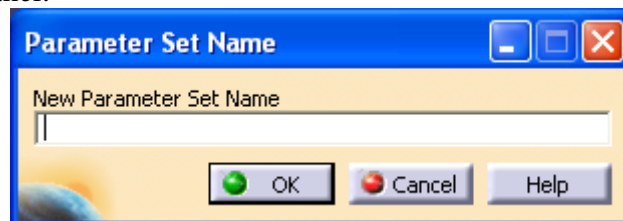
Parameter set contains value and tolerance

3.6.4 User interface

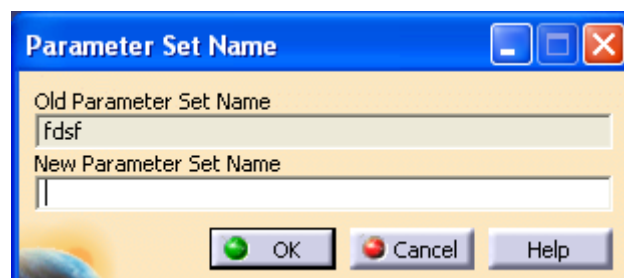
For each Power feature, you can create, Rename or delete a parameter set. Two parameter set with the same name are no possible. In the second table page or the main panel you have :



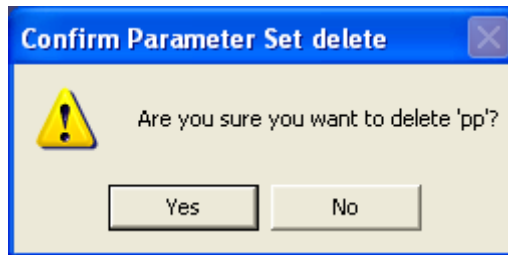
Parameter set creation panel:



Parameter set rename panel:



Parameter set delete warning:



To provide parameter set to an end user, expert user creates their parameter set locally and copies it to the shared data of the end user.

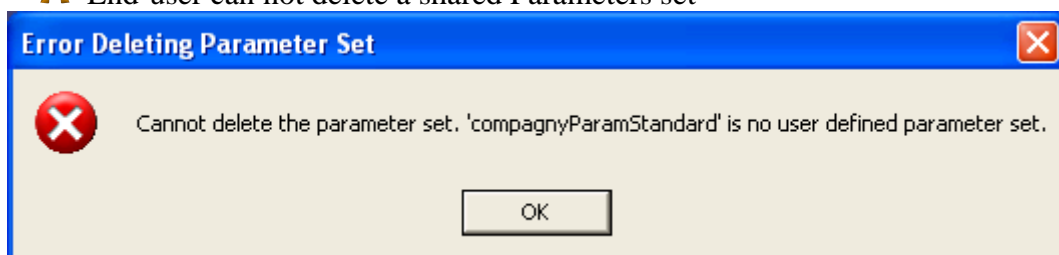
3.6.5 Process

Creation :

- 👤 When end-user create a parameter set corresponding to a Power feature, a file named « InternalName_Version.xml » is created under User profile directory.
- 👤 Administrator need to put it under SharedData\ParameterSet directory or created a dedicated directory at the same localization and put it on.

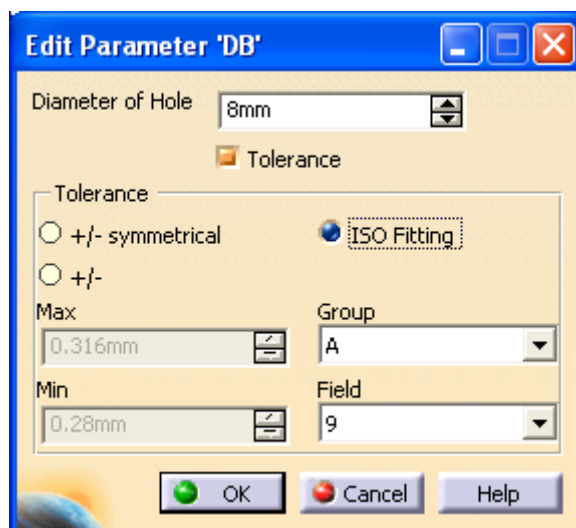
Deletion :

- 👤 End-user can not delete a shared Parameters set



3.7 Fitting & tolerance

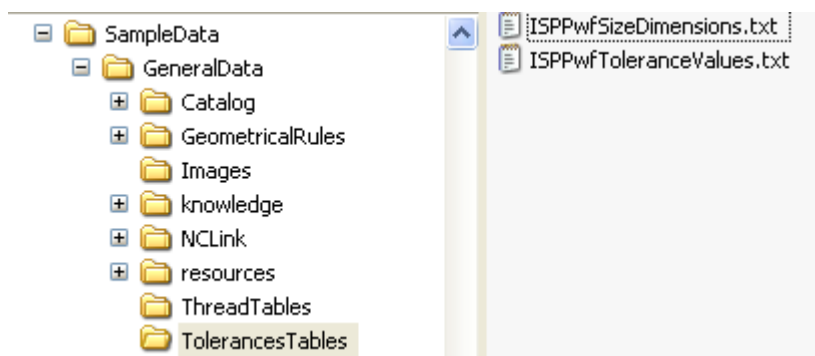
For Power feature parameter, allows to provide tolerance value and/or fitting depending of the value



3.7.1 Location file

- A file name ISPPwfSizeDimensions.txt must exist into TolerancesTables directory
 - ✎ Before R18, this file was named POWSizeDimensions.txt. It must be renamed.
- A file name ISPPwfToleranceValues.txt must exist into TolerancesTables directory
 - ✎ Before R18, this file was named POWToleranceValues.txt. It must be renamed.

These files must be located to..\\SampleData\\GeneralData\\TolerancesTables



3.7.2 File description

ISPPwfSizeDimension.txt must have this architecture

DimensionId	LowerVal	UpperVal
1	1	3
2	3	6
3	6	10
4	10	14
5	14	18
6	18	24

- ✎ First column : IdNumber
- ✎ Second column : parameter's low value range
- ✎ Third column: parameter's upper value range

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- From this file, we identify the range corresponding to an Id Number. For example IdNumber 2 corresponds to length in range $3\text{mm} < \text{IdNumber}2 \leq 6\text{ mm}$
- ISPPwfToleranceValues.txt must have this architecture

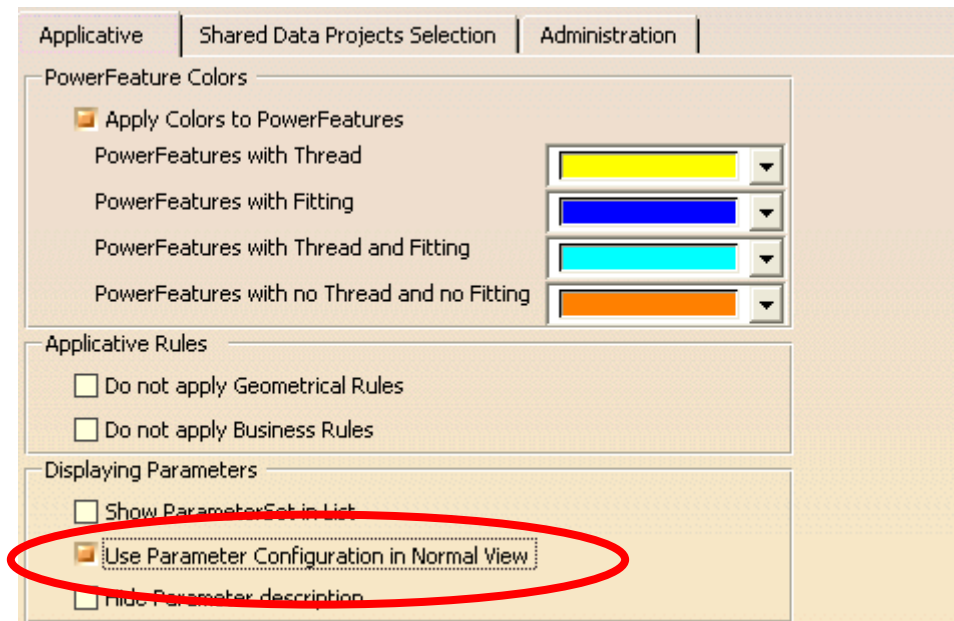
letter	number	sizedim	minival	maxval
A	9	1	0.27	0.295
A	11	1	0.27	0.33
A	9	2	0.27	0.3
A	10	2	0.27	0.318
A	9	3	0.28	0.316
A	12	3	0.28	0.43
A	13	3	0.28	0.5
A	9	4	0.29	0.333
A	10	4	0.29	0.36
A	11	4	0.29	0.4
A	12	4	0.29	0.47
A	13	4	0.29	0.56
A	9	5	0.29	0.333
A	10	5	0.29	0.36
A	13	6	0.3	0.63

- First column : Fit Letter
- Second column : fit number
- Third column: corresponding IdNumber
- First column : corresponding Minimum tolerance Value
- First column : corresponding Maximum tolerance Value
- Knowing the Id Number of the length, the application filter to get all corresponding Fit Letter/Number. For example, for Id Number 2, authorized Fit are A9 and A10.
- When setting a Fit tolerance to the parameter A:
 - If a string parameter named « Fit_Tolerance_A » is published by the Power Feature , it is valuated with the fit value.
 - In all case, a comment is added in the Power Feature parameter named «Comment ».The added comment is « Tol_A=FitValue ». This is mandatory to retrieve this information when edit the power feature.

3.8 Parameter show edit (PSE)

This behavior allows customizing Power feature parameter display.

- For each feature :
 - Display or not the parameter in the list of the panel
 - Display/authorize or not tolerance on a parameter
- This behavior is activated/deactivated through setting



3.8.1 User interface

When using PSE, parenthesis specifies that parameter could not be edit.

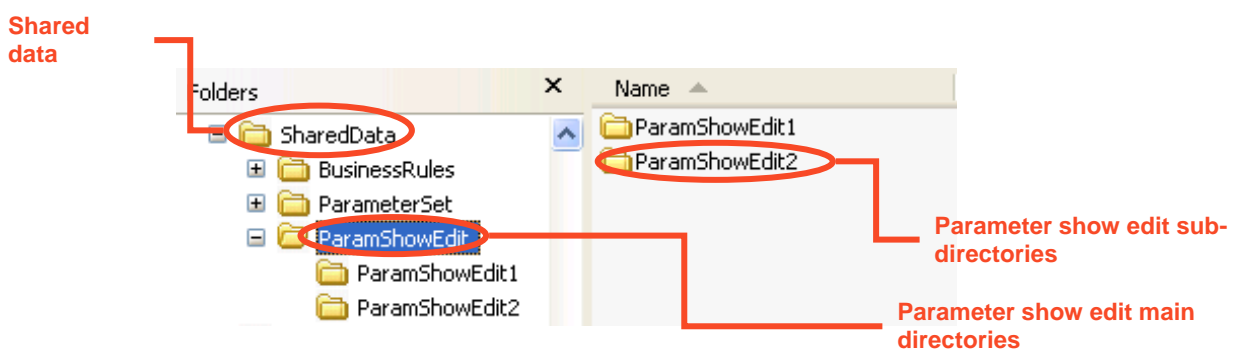
Parameters		
Short	Value	Tolerance
DB	10mm	
T	20mm	
F	2mm	
W	90deg	
SW	(120deg)	
DP	(0mm)	
TP	(15mm)	

Parameter Sets									
Parameter Sets	DB	DB...	T	T_Tol	F	W	SW	DP	TP
MyFavoriteStandard	10mm		20mm		2mm	90deg	(12...	(0mm)	(15...
compagnyParamStandard	10mm		20mm		2mm	90deg	(12...	(0mm)	(15...

Current Parameters									
Parameter Sets	DB	DB...	T	T_Tol	F	W	SW	DP	TP
None	10mm		20mm		2mm	90deg	(12...	(0mm)	(15...

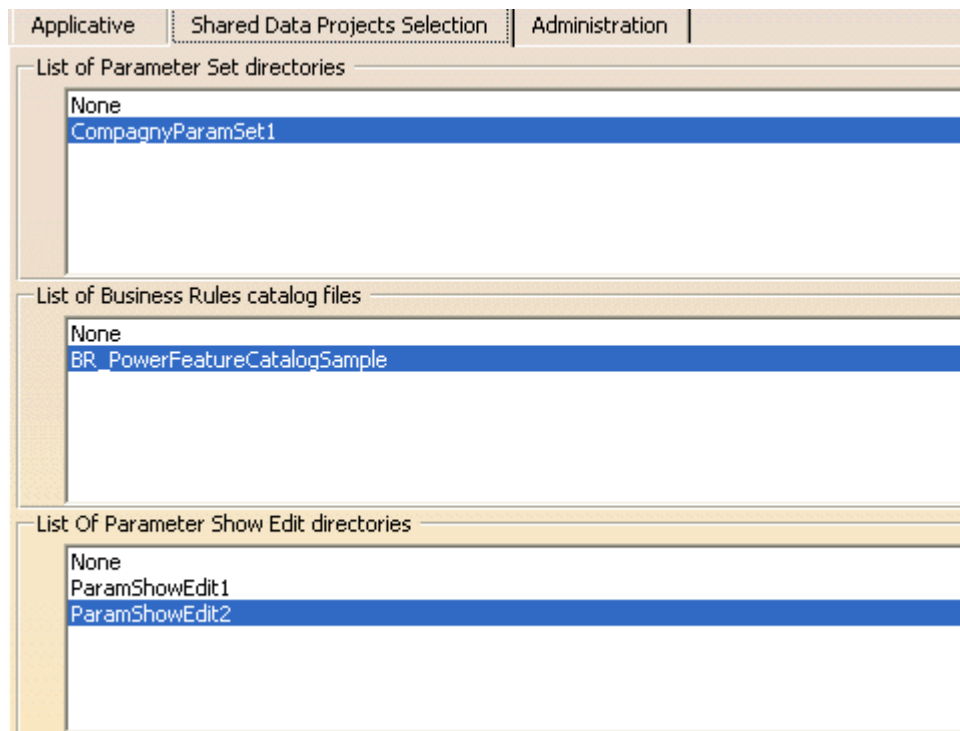
3.8.2 Settings and shared directory

You can provide several configurations. These configurations correspond to the name of the directory under SharedData\ParamShowEdit.

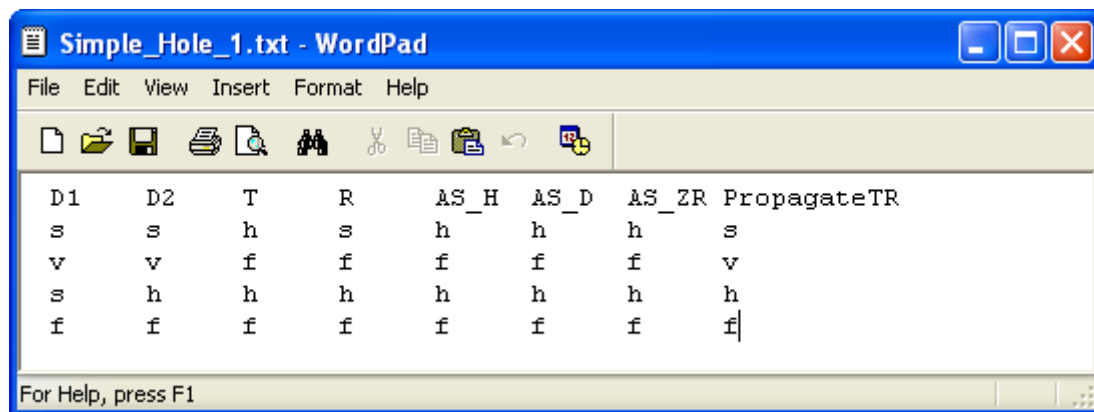


In each configuration :

- For each feature we want to apply PSE:
 - A .txt file must exist on the directory under ParamShowEdit
 - If not, all parameters are editable and could have tolerance*
 - The name of the file must be InternalName_Version.txt
- End user can use or not PSE through the setting



3.8.3 Description



- One column by parameter
- Following line are :
 - 1st => Show/Hide Parameter (s or h character)
 - 2de => Edit/noEdit Parameter (v or f character)
 - 3rd => Show/Hide Tolerance (s or h character)
 - 4th => Edit/NoEdit Tolerance (v or f character)
- Show/Hide => s/h
- Edit/noEdit => v/f
- In the fact : only Show and Edit characters are fixed but Hide and no Edit characters could be any characters

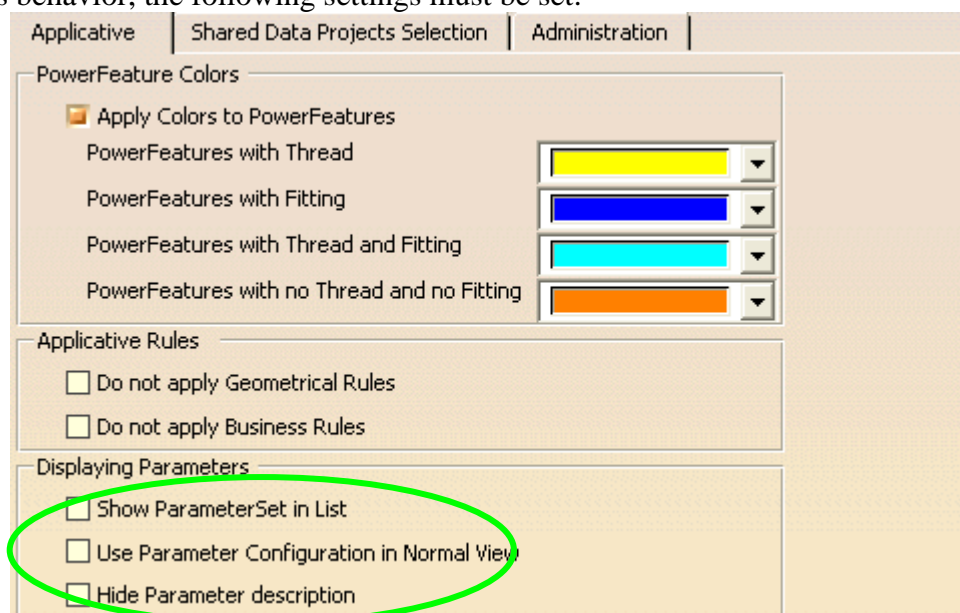
3.9 Parameter's description

When end-user uses Normal display, we could display a text explaining the aim of the power feature parameter.

Parameters				
Short	Value	Tolerance	Description	
DB	10mm		Diameter of Hole	
L	36.183mm		Length of Hole	
F1	2mm		Depth of Chamfer 1	
W1	90deg		Angle of Chamfer 1	
F2	3mm		Depth of Chamfer 2	
W2	60deg		Angle of Chamfer 2	
DP	10mm		Diameter of Fit	
TP	25mm		Depth of Fit	
AS_H	0mm		Height of Adjunct Cylinder	
AS_D	0mm		Diameter of Adjunct Cylinder	
ParameterSet	None			

3.9.1 Settings

To have this behavior, the following settings must be set:



Applicative | Shared Data Projects Selection | Administration

PowerFeature Colors

☒ Apply Colors to PowerFeatures

PowerFeatures with Thread: [Yellow]

PowerFeatures with Fitting: [Blue]

PowerFeatures with Thread and Fitting: [Cyan]

PowerFeatures with no Thread and no Fitting: [Orange]

Applicative Rules

☐ Do not apply Geometrical Rules

☐ Do not apply Business Rules

Displaying Parameters

☐ Show ParameterSet in List

☐ Use Parameter Configuration in Normal View

☐ Hide Parameter description

3.9.2 Data

For each Power feature, the description of the parameter must be written in ISPPwfParametersDescription.CATNls.

If the description of a parameter name is the same for all power feature types, you can specify only by the name of the power feature Parameter:

Example:

```
// general Description
// --- Parameters for basic geometry (blue) -----
DB= "Diameter of Hole";
```

```
T= "Depth of Hole";
F= "Depth of Chamfer";
W= "Angle of Chamfer";
DP= "Diameter of Fit";
TP= "Depth of Fit";
AS_H= "Height of Adjunct Cylinder";
AS_D= "Diameter of Adjunct Cylinder";
```

If the description is specific to the power feature, we use ‘internal name’ + version + parameter name to describe it.

Example:

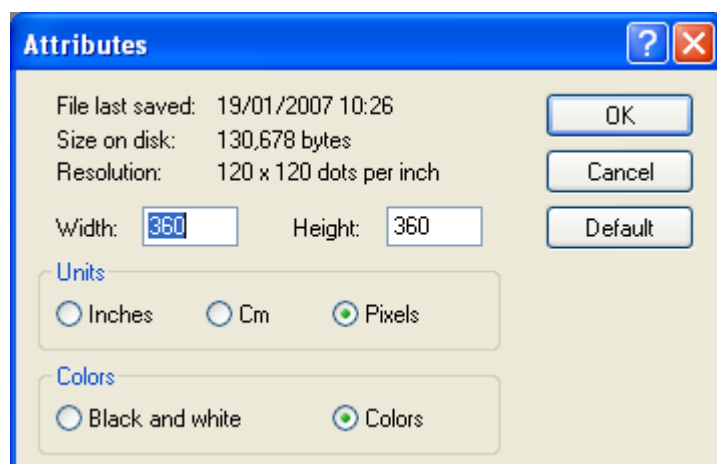
```
//BlindHole
BlindHole_1.DB="Diameter of Hole for Blind Hole";
BlindHole_1.T="Depth of Hole for Blind Hole ";
BlindHole_1.F=" Depth of Chamfer for Blind Hole";
```

3.10 Power Feature Images

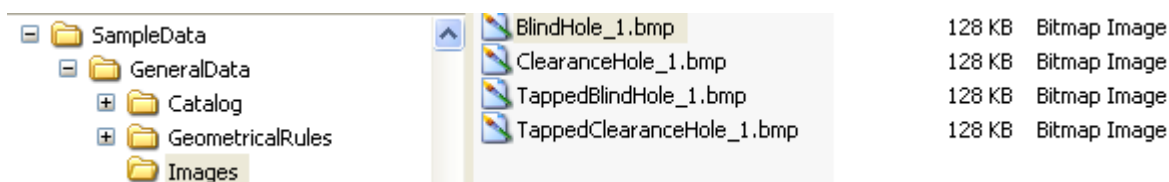
Power Feature main command displays a panel that contains a tag page named ‘Sketch’. We can provide a picture to help the end-user to choice is Power Feature. Usually the picture represents a sketch of the Power feature with parameter signification.

The picture must follow the rules:

- 1- it is a bmp file
- 2- its dimensions must be :



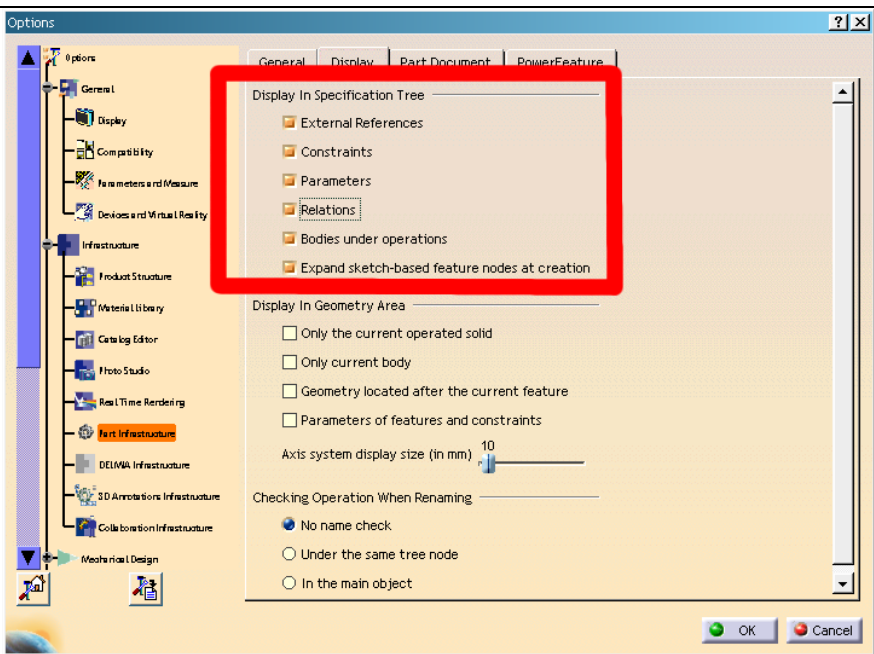
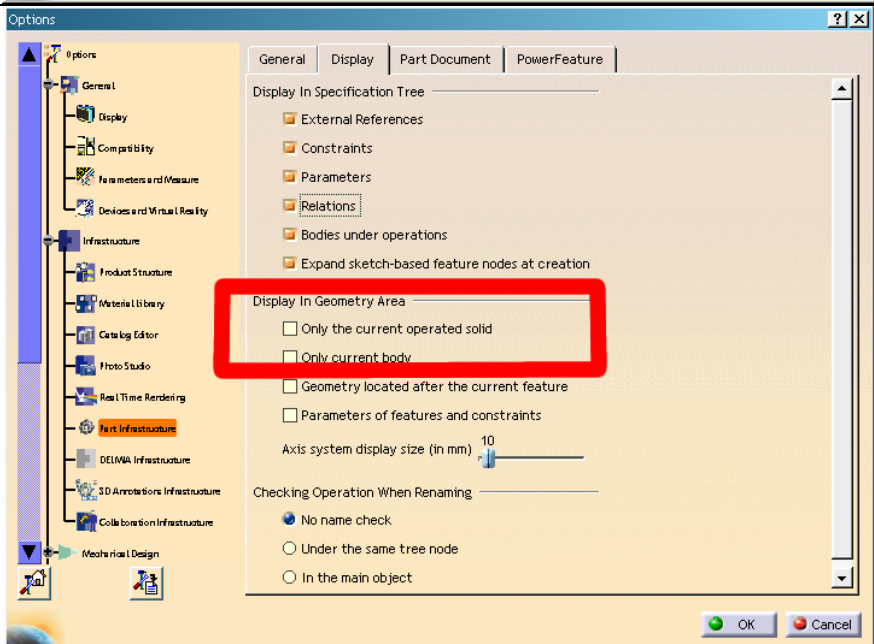
- 3- file must be located under GeneralData\Images



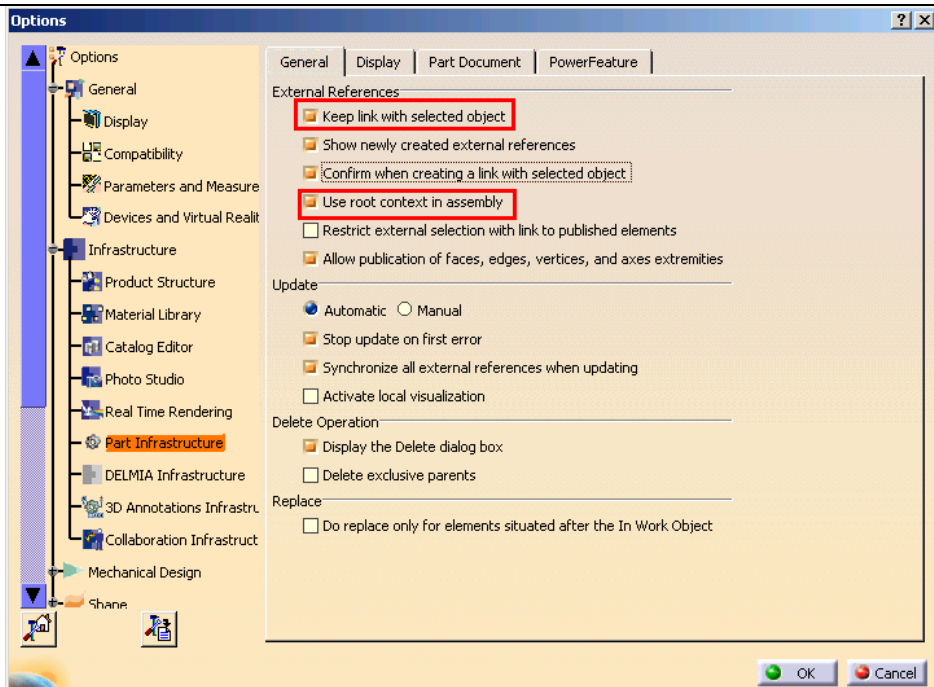
4 End User CAD

4.1 Settings

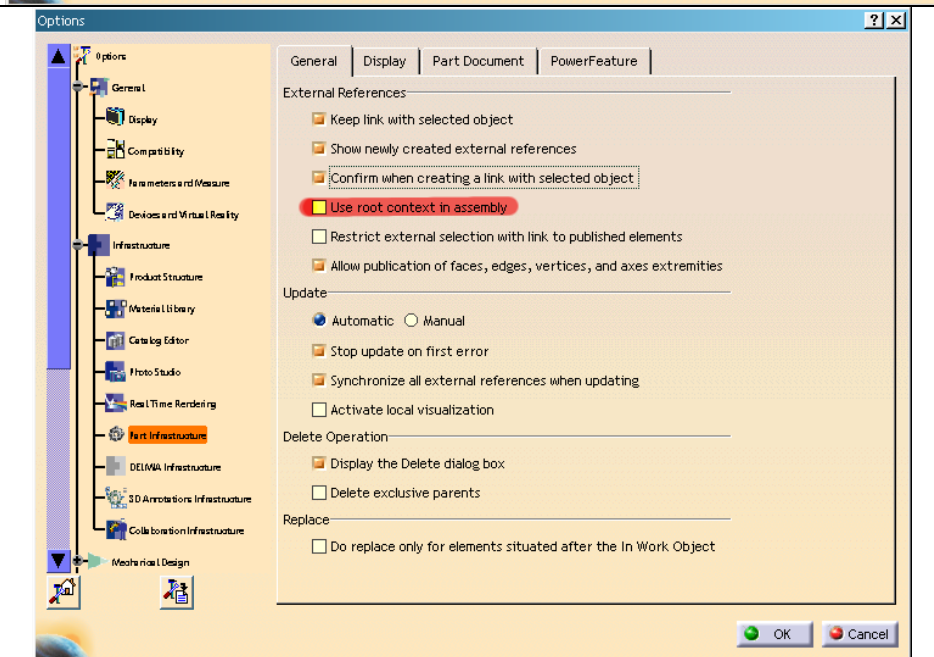
For an efficient use, you have to check some items:

<p>All options below „display in Specification Tree “are to be checked</p>	
<p>You can check the following options</p>	

Check “Use root context in assembly” and “Keep link with selected object” when necessary

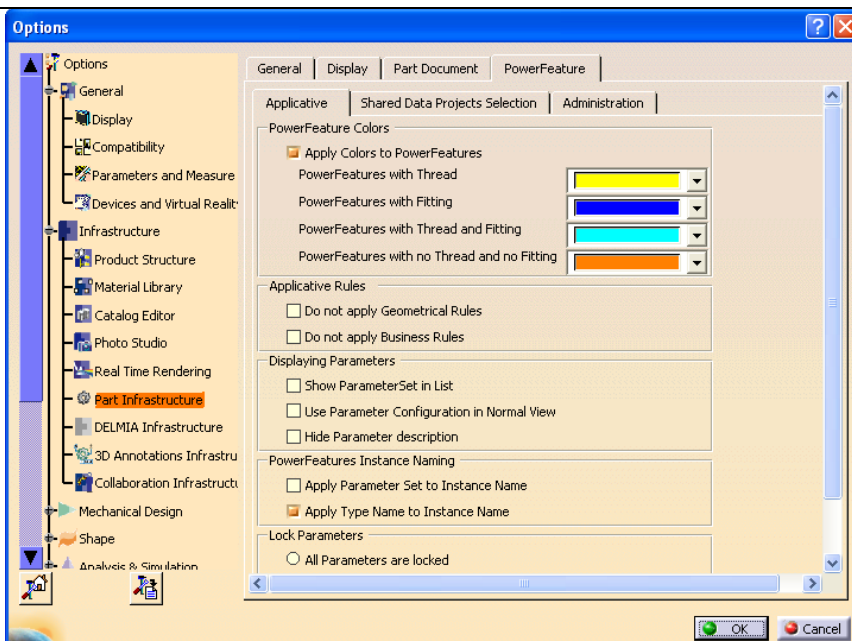


Check “Use in root context” when necessary

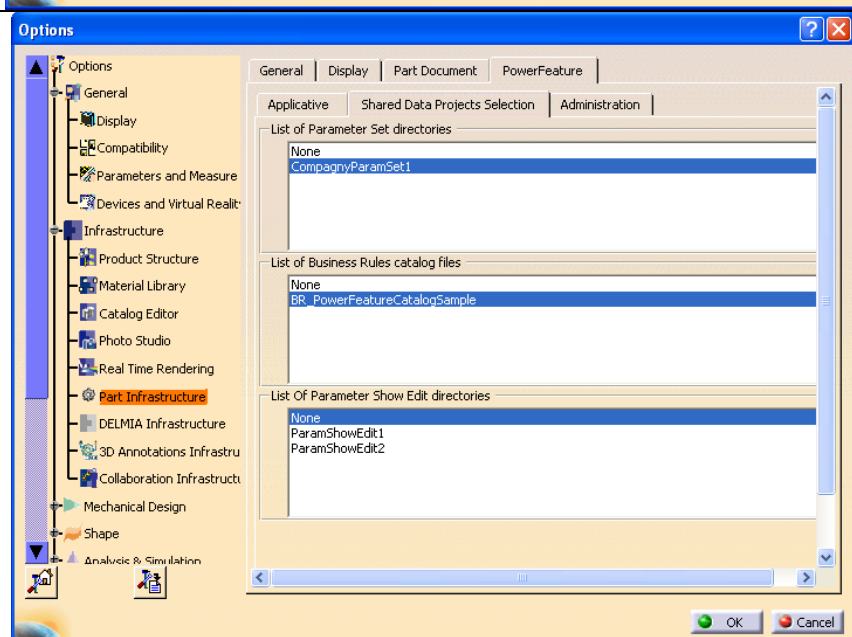


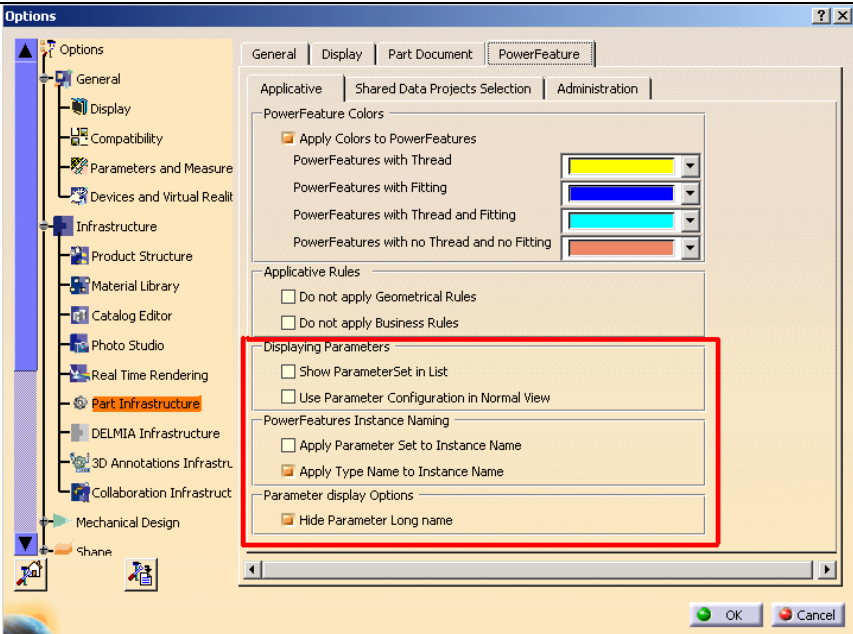
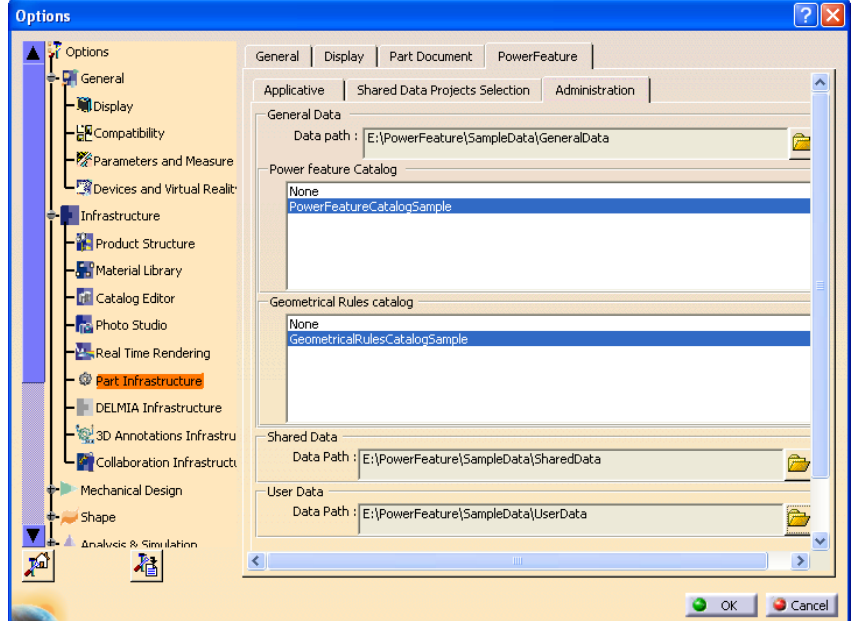
ABOUT Pwf :

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 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 available)



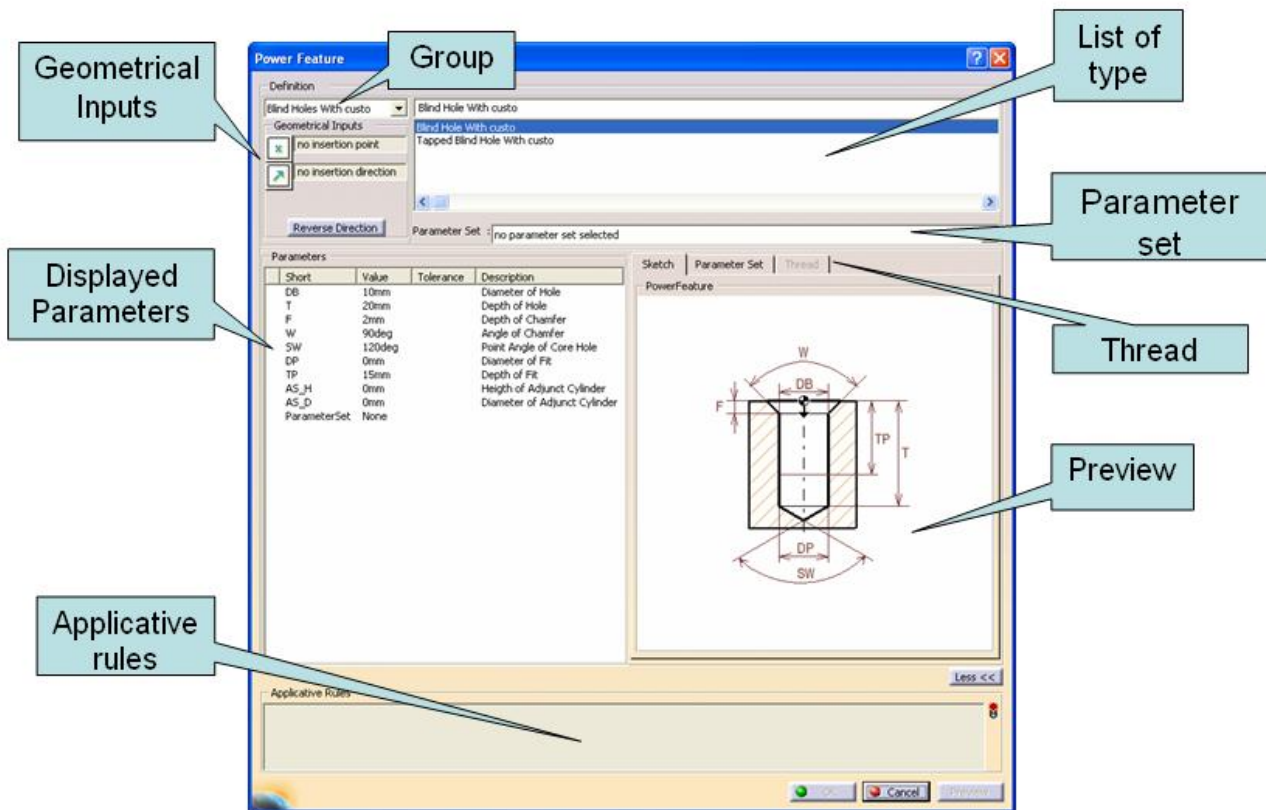
<p>Options concerning the way is displayed the parameters for the Power Feature and in the specification tree</p>	
<p>Administration : Enter the paths where we find</p> <p>Power Feature catalog</p> <p>Geometrical rules catalog</p> <p>Shared data</p> <p>User parameter sets</p>	

4.2 Power Feature tool

In the part design, there is only one icon about the BPA Power Feature :

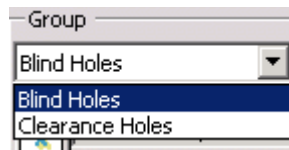


When you click on this button, the window appears:



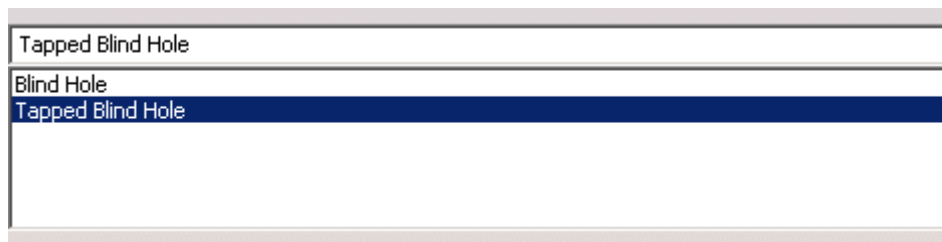
4.3 Group

It's the list where there are all the principal group of holes



4.4 The selected hole type in the List

When you have selected a category (group), the list of specific holes appears:



4.5 Inputs

Necessary Information for instantiation of Power Feature



The point where the Power Feature will be put



The direction of the Power Feature .

When we want to instantiate a clearance hole, we have an added field



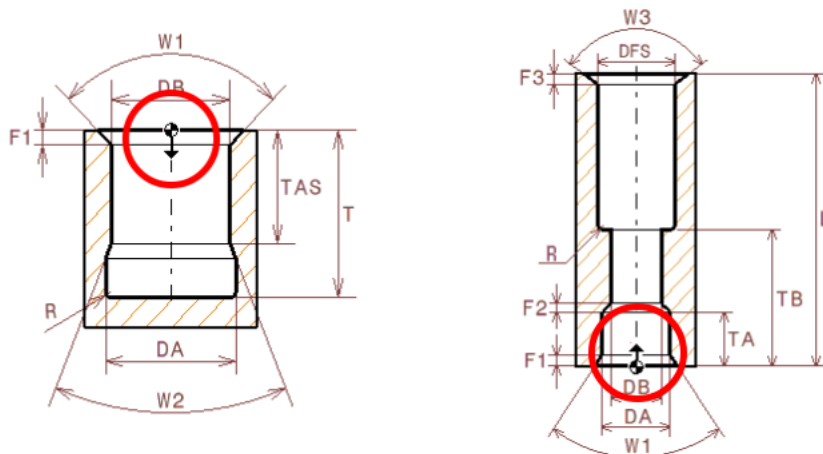
To define the limiting geometry for a clearance hole.

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

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)

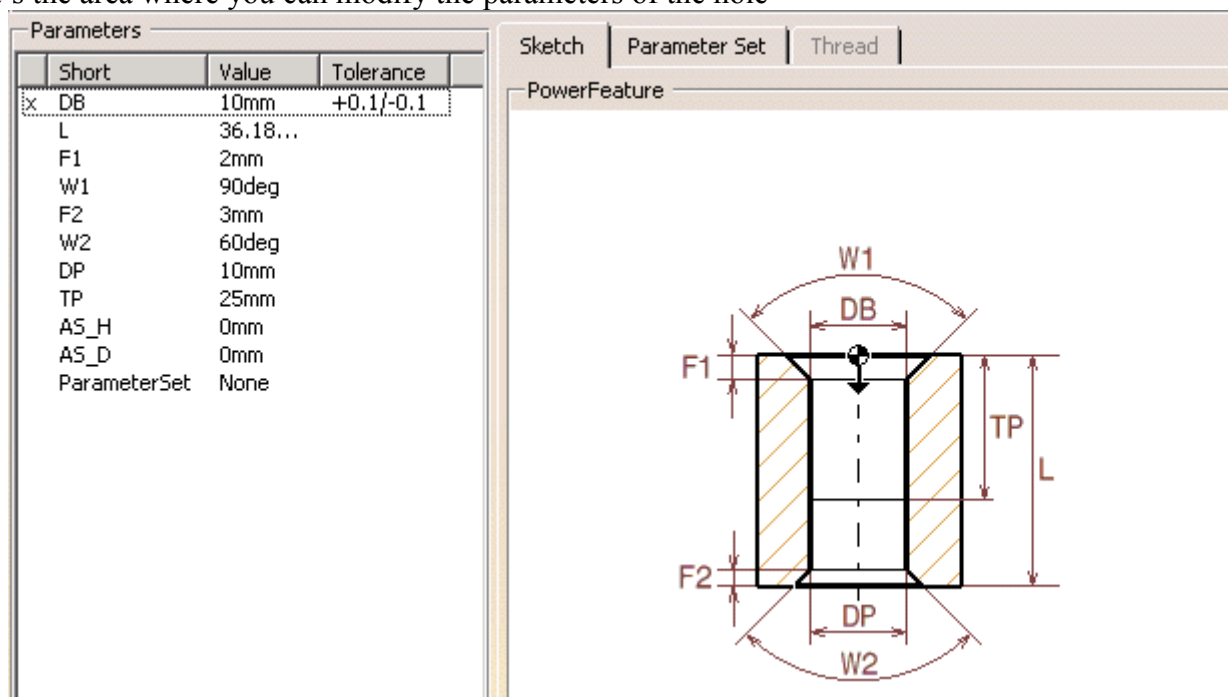
Note:

The position of the inserted point depends on the type of the Power Feature .



4.6 Parameters

It's the area where you can modify the parameters of the hole

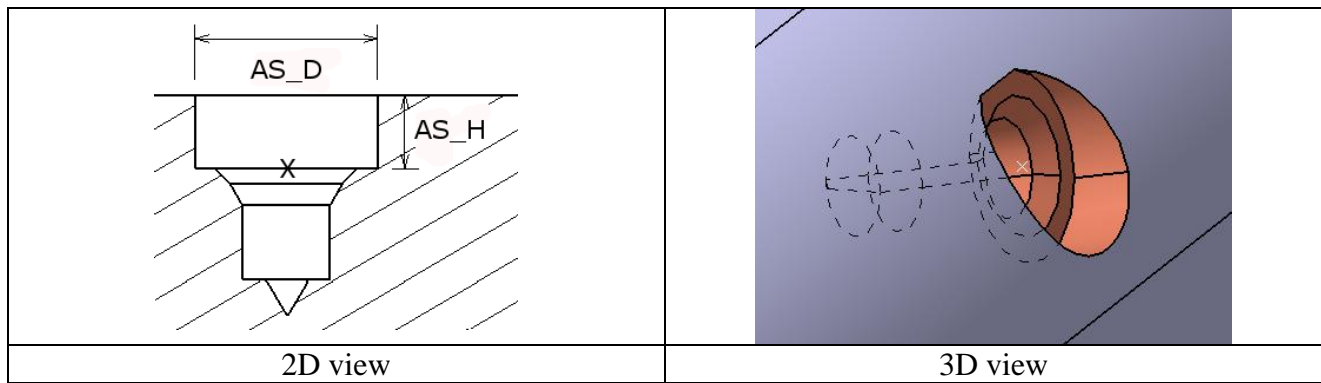


You can also add a tolerance. For example, we can see that:

- The length TP is equal to 25mm
- $W1=90^\circ$
- ...

Note:

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



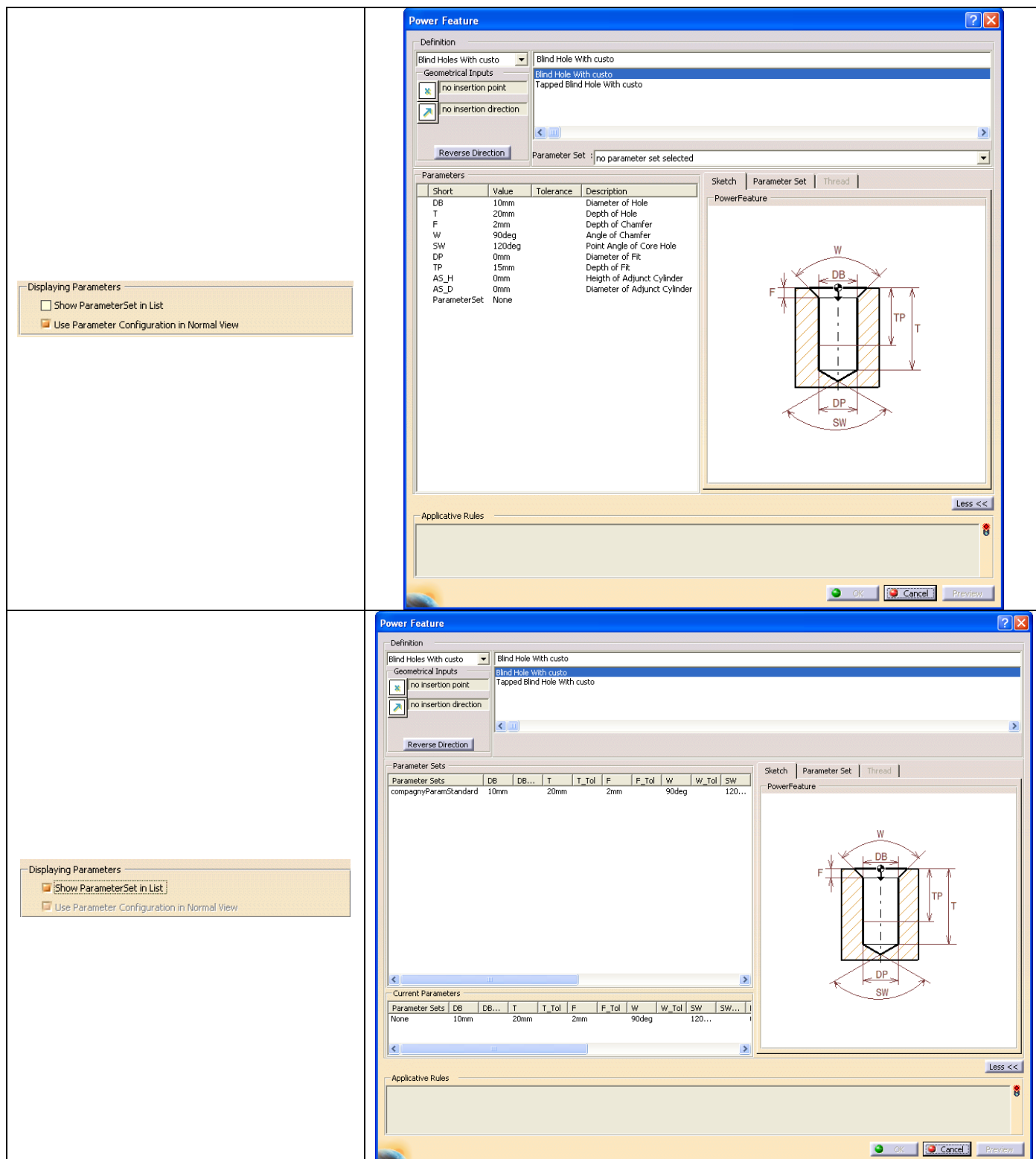
4.7 Parameter set

It's an interesting part: If we have two sort of tapped blind holes "a" & "b" for example, we can retrieve "a" & "b" easily.

In this example: for "a", L=22mm and W1=85° and for "b", L=21mm and W1=90°. So with the parameter set, it's not necessary to re-enter the parameter of "a" when you need "a" and re-enter the parameter of "b" when you need "b". You only have to select "a" or "b" in the parameter set list. The parameters are memorized. You can create, rename, and delete your own parameter sets.

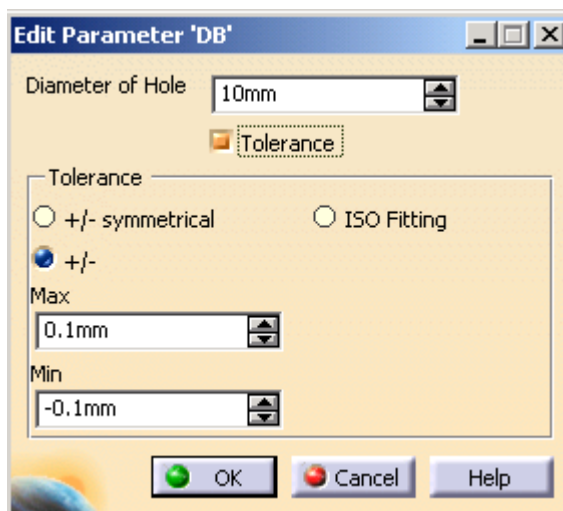


Parameter Set view depending on the option you have checked the parameters can be viewed in two different ways:



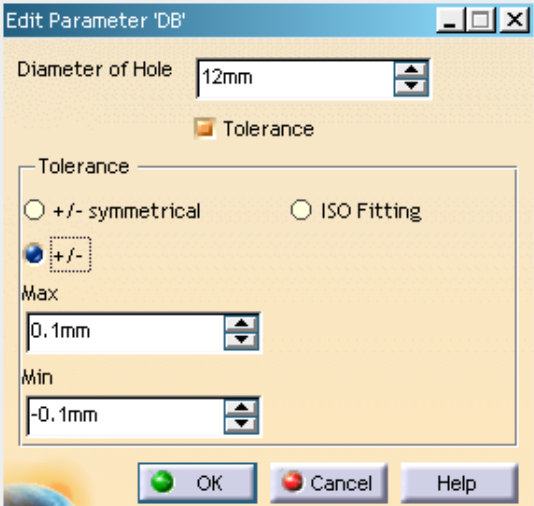
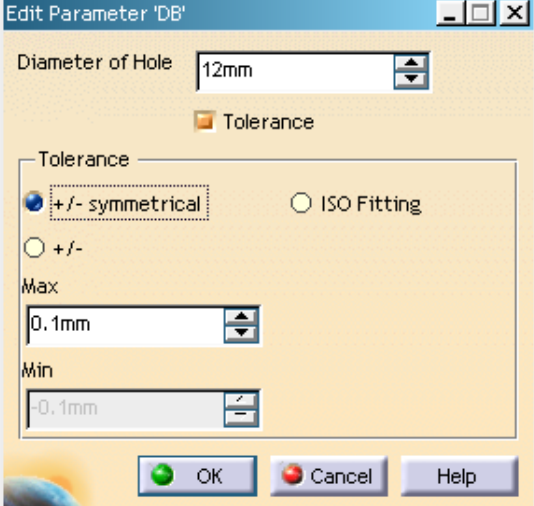
4.8 Tolerance

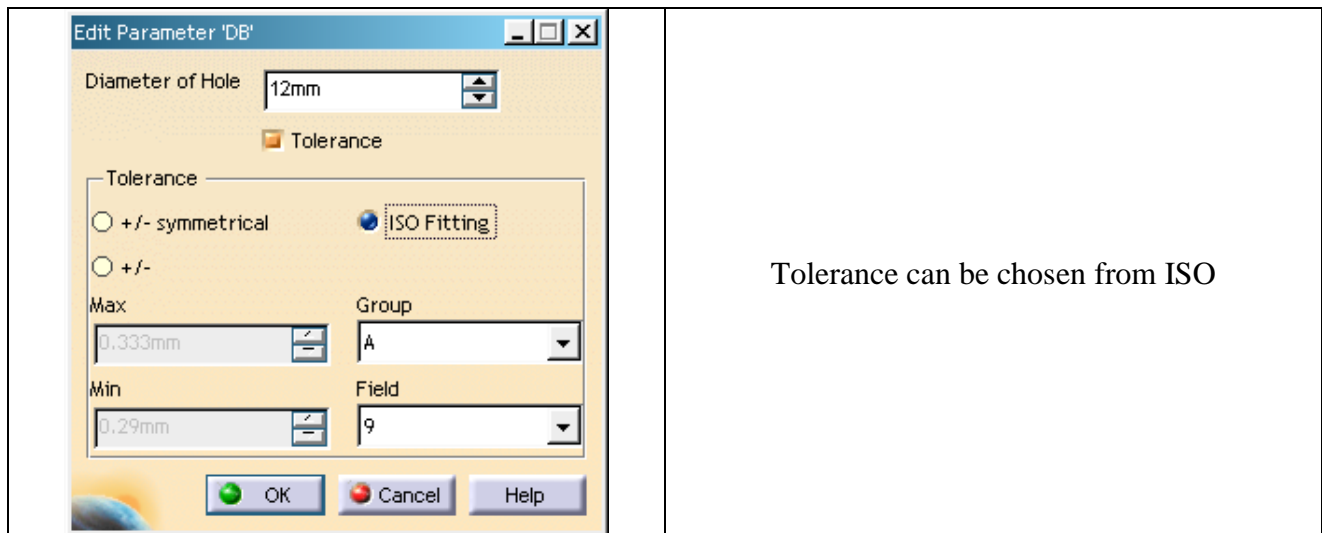
When we edit a parameter, we can apply a tolerance on a length. The display panel is



It's a dialog box where you find the nominal value of the selected length and where you can add a tolerance:

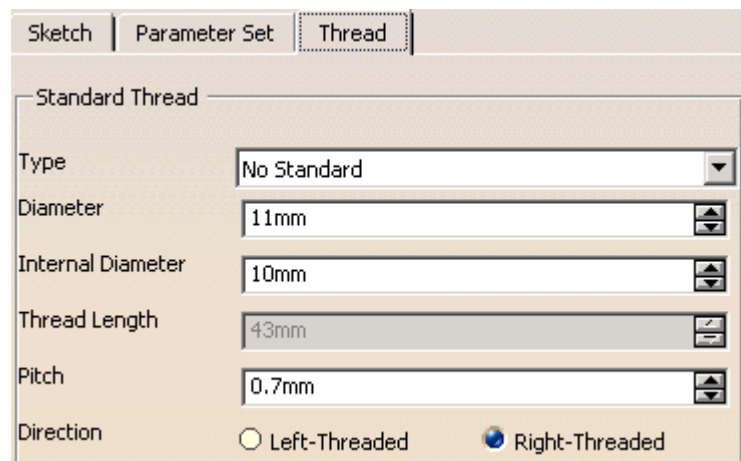
- +/-
- +/- symmetrical
- ISO Fitting

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

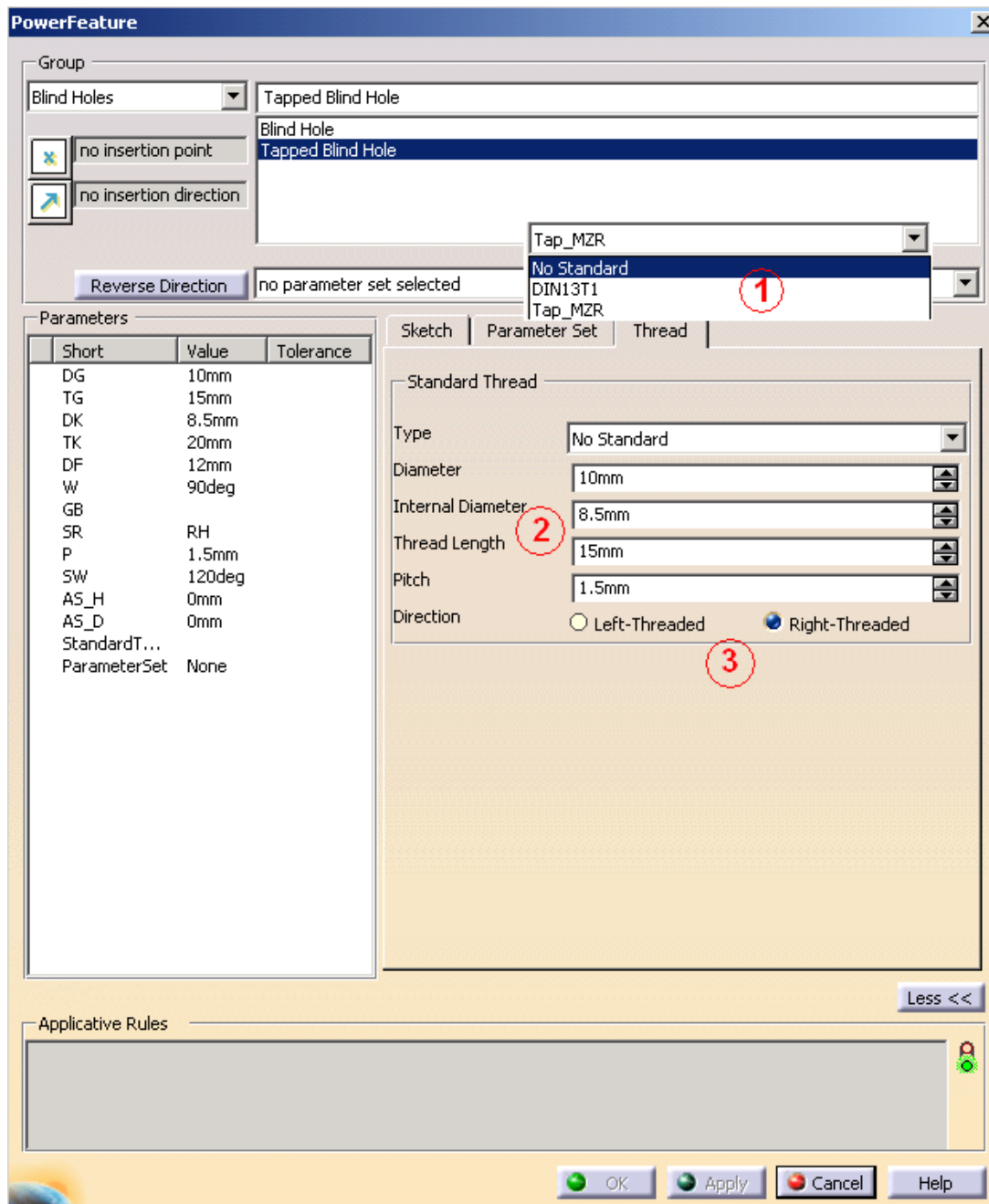


4.9 Thread hole

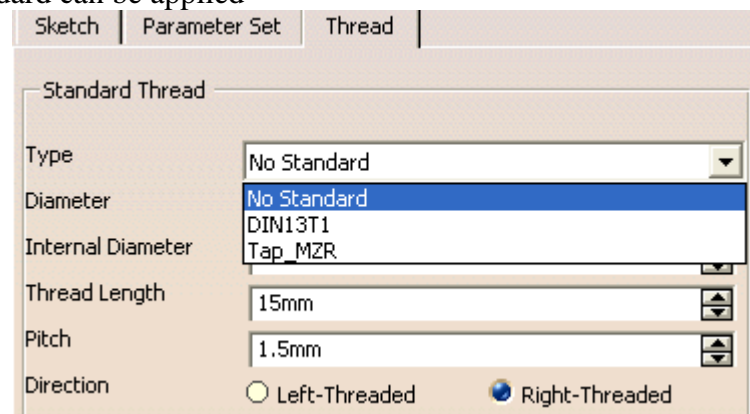
When the hole is threaded, you have a special dialog box:



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



1. Different standard can be applied



2. Thread parameters values

Sketch | Parameter Set | Thread

Standard Thread

Type: No Standard

Diameter: 1mm

Internal Diameter: 0.729mm

Thread Length: 15mm

Pitch: 0.25mm

Direction: ☐ Left-Threaded ☒ Right-Threaded

Sketch | Parameter Set | Thread

Standard Thread

Type: DIN13T1

Diameter: M1

Internal Diameter: 0.729mm

Thread Length: 15mm

Pitch: 0.25mm

Direction: ☐ Left-Threaded ☒ Right-Threaded

3. Direction of thread

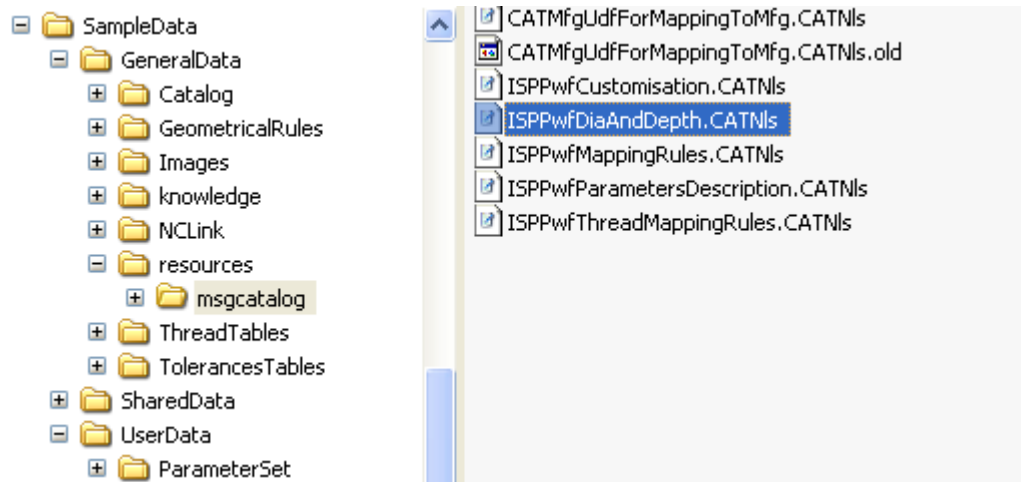
Direction: ☐ Left-Threaded ☒ Right-Threaded

When a power feature is threaded, thread parameters become editable only through the thread tab page, neither in the Parameters list.

5 Expert machining User

5.1 *MAFs diameter and depth customization*

To map machining operation parameter to MAF data, it is also necessary to provide a Ressources file named: ISPPwfDiaAndDepth.CATNls. This file must be located in resources/msgcatalog and added to the CATMsgCatalogPath CATIA variable



ISPPwfDiaAndDepth.CATNls contains :

If all Power feature as the same mapping with NC parameter, you just have to map Blind/Thread behavior with NC parameter as follow :

```
Blind_Thread.Diameter = "BTD";
Blind_NoThread.Diameter = "BND";
Through_Thread.Diameter = "TTD";
Through_NoThread.Diameter = "TND";
```

If not, for each feature, give the corresponding parameter name :

```
//BlindHole
BlindHole_1.Diameter="DB";
BlindHole_1.Depth ="T";
BlindHole_1.CBDiameter="DP";
BlindHole_1.CBDepth="F";
BlindHole_1.CSAngle="W";
```

In the Process to retrieve this information, the code try to retrieve the information from the feature first, and it does not find, it try from the general parameter

NC parameters are :

- Diameter : The hole diameter
- Depth : The hole depth
- CBDiameter : The hole head diameter

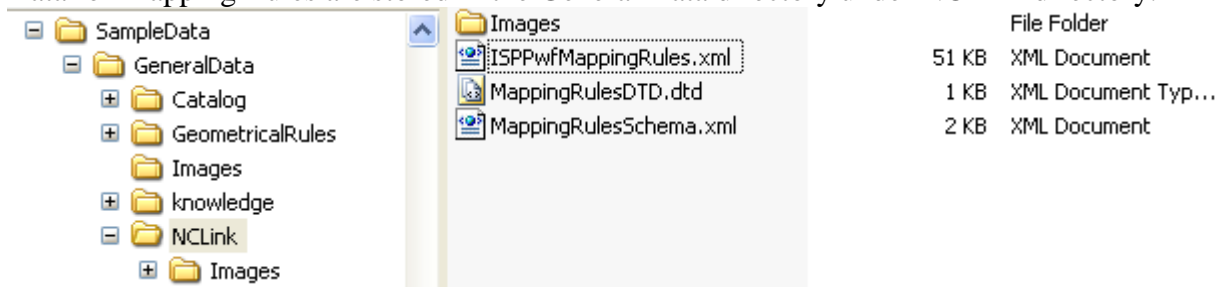
- 👤 CBDepth : The hole head depth
- 👤 CSAngle : The hole head angle
- 👤 THDiameter : The hole thread diameter
- 👤 THDepth : The hole thread depth
- 👤 THPitch : The hole thread pitch
- 👤 THSense : The hole thread side. (0: right threaded 1: left threaded)
- 👤 Extension : The hole extension. (1: blind 2: through) done by Through parameter of R16 UDF

5.2 Mapping Rules

Mapping rules allows to provide for each Power Feature a list of customized machining Operation (activity). This customization corresponds to create formula between Machining operation parameters (Ex: Drilling) and MAF parameters. Mapping Rules is called NC Link sometime.

5.2.1 Data

Data for Mapping Rules are stored in the General Data directory under NCLink directory:



- 👉 It must contain:
 - 👤 mapping rule Image in Images directory
 - 👤 Mapping rule xml file containing formula between MO parameters and MAF parameters
- 👉 The XML file must Follow the Rules schema example
- 👉 Example in xml file for mapping rule named « DoNothing » for Pwf internal name « BlindHole » Version 1:

```

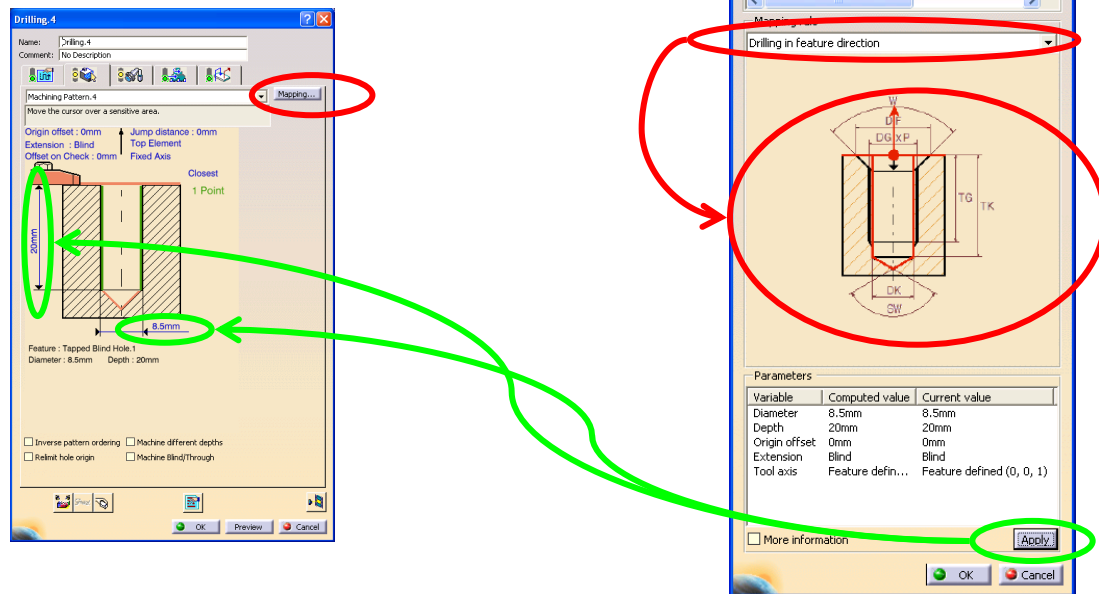
<!--TappedBlindHole Version 1 Gewindegrundloch R16 -->
<FEATURE NAME="TappedBlindHole" VERSION="1">
  <ACTIVITY NAME="Drilling">
    <RULE NAME="DoNothing" DEFAULTRULE="yes">
      <IMAGE/>
      <FORMULA>
        <VARIABLE NAME="Depth"/>
      </FORMULA>
    </RULE>
    <RULE NAME="DrillingUpwards">
      <IMAGE>1/DrillingUpwards.bmp</IMAGE>
      <FORMULA>
        <VARIABLE NAME="Depth">#MfgFeatPackage.MfgMachinableAreaFeature.TK#</VARIABLE>
        <VARIABLE NAME="Diameter">#MfgFeatPackage.MfgMachinableAreaFeature.DK#</VARIABLE>
        <VARIABLE NAME="MfgHoleExtension">MfgBlind</VARIABLE>
        <VARIABLE NAME="ToolAxis">+1</VARIABLE>
        <VARIABLE NAME="OriginOffset">Omm</VARIABLE>
      </FORMULA>
    </RULE>
  </ACTIVITY>
  <ACTIVITY NAME="DrillingDwellDelay">
    <RULE NAME="DoNothing" DEFAULTRULE="yes">
      <IMAGE/>
      <FORMULA>
        <VARIABLE NAME="Depth"/>
      </FORMULA>
    </RULE>
    <RULE NAME="DrillingDwellDelayUpwards">
      <IMAGE>1/DrillingDwellDelayUpwards.bmp</IMAGE>
      <FORMULA>
        <VARIABLE NAME="Depth">#MfgFeatPackage.MfgMachinableAreaFeature.TK#</VARIABLE>
        <VARIABLE NAME="Diameter">#MfgFeatPackage.MfgMachinableAreaFeature.DK#</VARIABLE>
        <VARIABLE NAME="MfgHoleExtension">MfgBlind</VARIABLE>
        <VARIABLE NAME="ToolAxis">+1</VARIABLE>
        <VARIABLE NAME="OriginOffset">Omm</VARIABLE>
      </FORMULA>
    </RULE>
  </ACTIVITY>

```

Unfortunately, Power Feature doesn't provide any tools to add values in this xml file. You must do it using a standard xml editor or text editor.

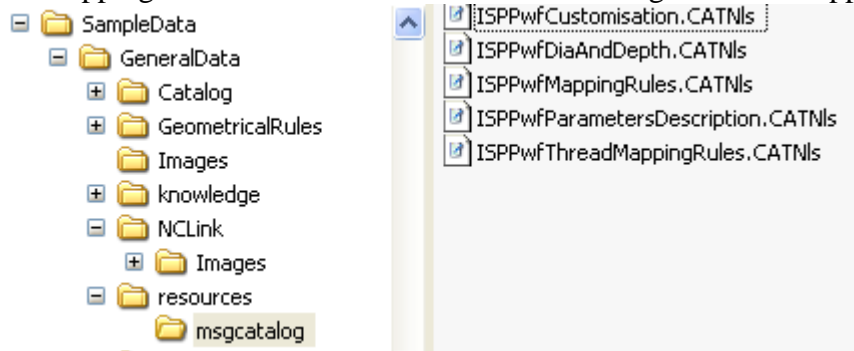
5.2.2 Directories

- Rules are provided through xml file :



Mapping rules images are located
\$GeneralData\NCLink\Images\InternalName\Version\MappingRulesNames.bmp directory

Mapping rules name could be customized through ISPPwfMappingRules.CATNls file

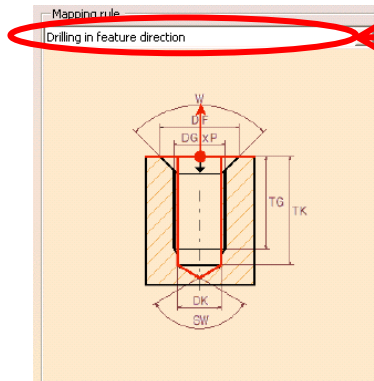


ISPPwfMappingRules.xml

```
</RULE>
<RULE NAME="DrillingUpwards">
  <IMAGE>1/DrillingUpwards.bmp</IMAGE>
  <FORMULA>
    <VARIABLE NAME="Depth">#HfgFeatPackage.HfgMachinableAreaFeature.DP</VARIABLE>
    <VARIABLE NAME="Diameter">#HfgFeatPackage.HfgMachinableAreaFeature.DS</VARIABLE>
    <VARIABLE NAME="HfgHoleExtension">HfgBlind</VARIABLE>
    <VARIABLE NAME="ToolAxis">+1</VARIABLE>
    <VARIABLE NAME="OriginOffset">0mm</VARIABLE>
  </FORMULA>
</RULE>
```

ISPPwfMappingRules.CATNIs

```
// Drilling
Drilling= "Drilling";
DrillingUpwards= "Drilling in feature direction";
DrillingUpwards1= "Drilling in feature direction 1";
DrillingUpwards2= "Drilling in feature direction 2";
```



5.3 Machining Process

The Expert Machining can provide CATProcess document containing the machining corresponding to a Power Feature.

This behavior is a standard of Prismatic Machining workbench.

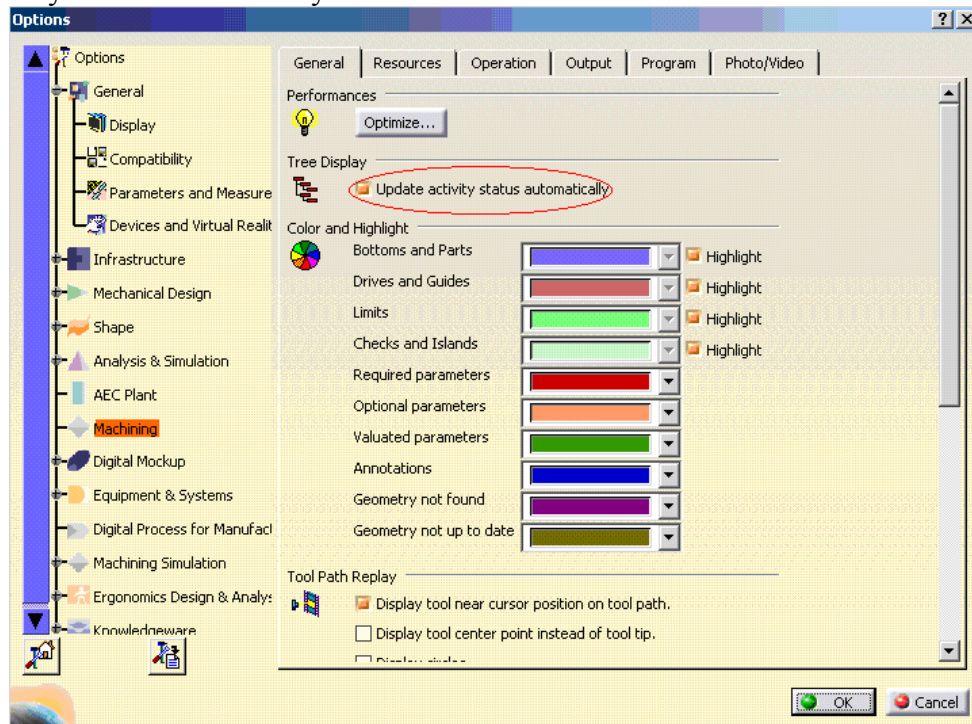
For more information, have a look to [\\$v5doc/online/pmgug_C2/pmgugbt0502.htm](#).

In this machining process, expert machining user can use the mapping rules defined in this document.

6 End User machining

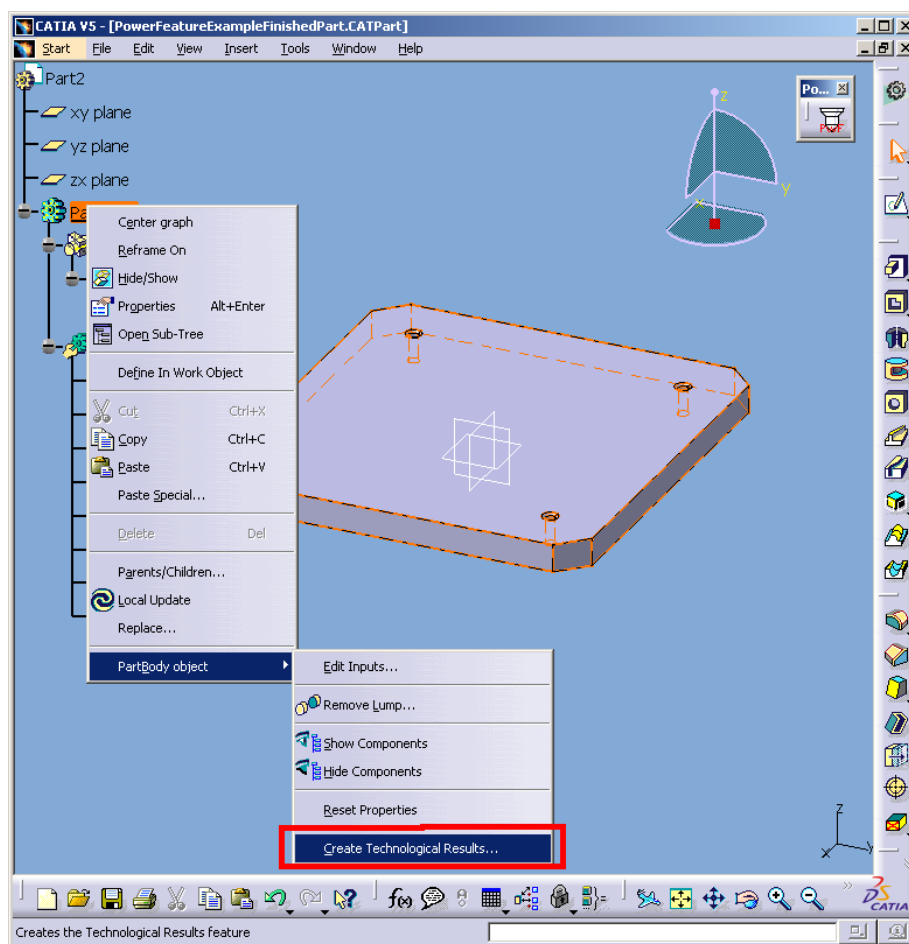
6.1 Settings

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



6.2 Technological Result

You have to create technological result to link CATPart & CATProcess. Then you can switch to prismatic machining workbench.



6.3 Presentation

In this workbench, there is a new icon and we are about to see the different functions.



6.4 Create/update MAFs

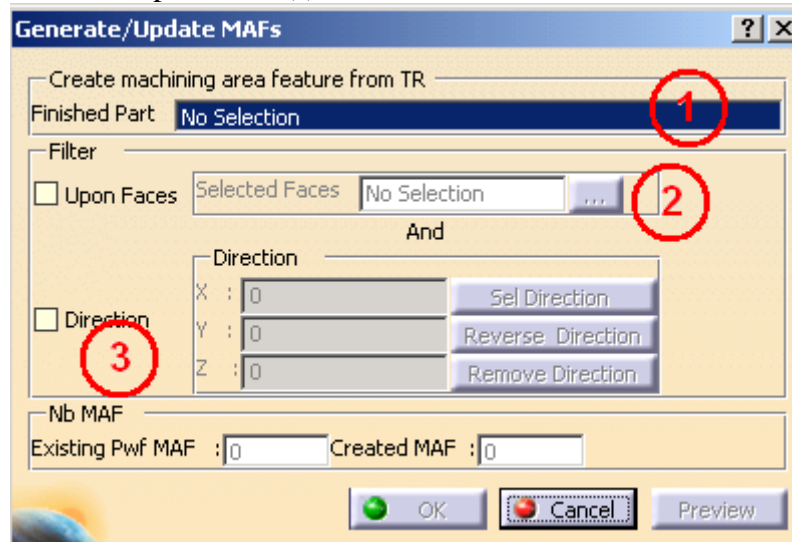
A MAF is a Machining Area Feature that means an area to manufacturing. MAF exists in standard CATIA. Power Feature creates its own MAFs, name Power Feature MAF. A Power Feature MAF is a Standard MAF that corresponds to a CAD Power Feature.

To create & update MAFs, use:



This command allows creating or updating after geometry modification the MAFs.

The dialog box to create and update Maf(s) is:



Only one finished part is taking as input for MAF generation (1).

If no filter are specify, we create/update all MAFs. The available filters are:

- 1 – One or several geometry face s
- 2 – A direction



☞ The filter function is used, to generate a MAF, when:

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

Filter options are managed like the 'and' function

6.5 Create pattern from MAFs

A pattern is a group of MAFs which respond to parameters criteria of acceptance. To create CATIA


pattern, we use  command. To create Power Feature pattern, we use  or  command

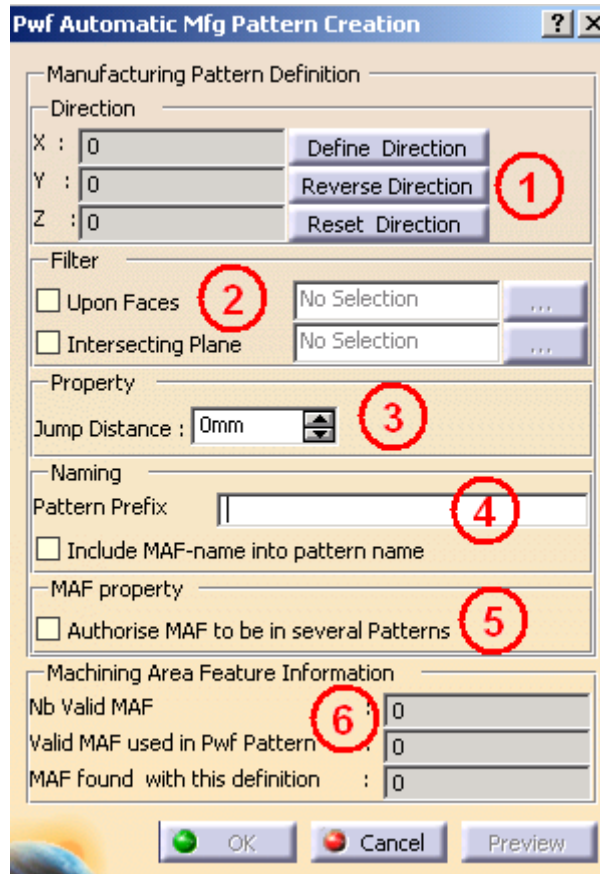
Power Feature pattern are CATIA customized patterned. The added values are:

- 1 – a Power Feature pattern contains only same kind of Power Feature MAFs, that means MAF comes from same kind of CAD Power Feature
- 2 – Parameters used to filter the MAF are stored into the pattern. The geometrical filters are not take in account.

Note: manufacturing pattern is not the pattern used in “Part Design” Workbench.

6.5.1 Automatic Pattern

This command () allows to generate several pattern in one shoot.




The dialog box is titled "Pwf Automatic Mfg Pattern Creation". It contains several sections with numbered callouts (1-6) pointing to specific fields or buttons:

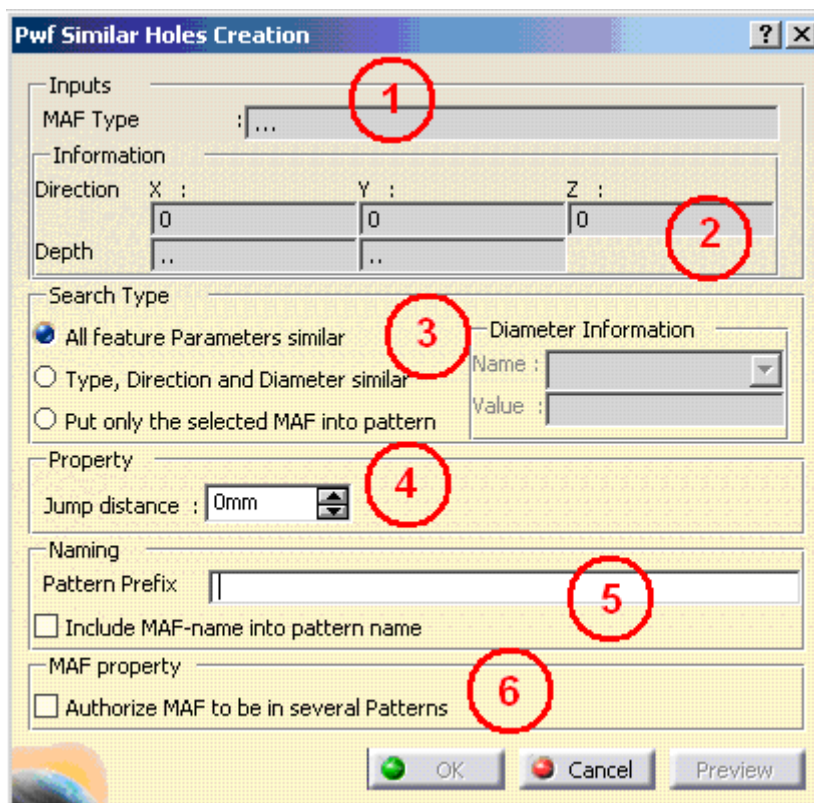
- Direction:** Fields for X, Y, and Z coordinates, each set to 0. To the right are buttons for "Define Direction", "Reverse Direction", and "Reset Direction". Callout 1 points to the "Reverse Direction" button.
- Filter:** Two checkboxes: "Upon Faces" (callout 2) and "Intersecting Plane". To the right of each is a "No Selection" button with a dropdown arrow.
- Property:** A "Jump Distance" field set to "0mm" with a spinner control. Callout 3 points to the spinner.
- Naming:** A "Pattern Prefix" text field (callout 4) and an unchecked checkbox "Include MAF-name into pattern name".
- MAF property:** An unchecked checkbox "Authorise MAF to be in several Patterns" (callout 5).
- Machining Area Feature Information:** Three rows of fields: "Nb Valid MAF" (callout 6), "Valid MAF used in Pwf Pattern", and "MAF found with this definition", all set to 0.

At the bottom are "OK", "Cancel", and "Preview" buttons.

- 1: A direction is mandatory to create a Power Feature pattern.
- 2: Two kinds of filter could be applied:
 - a – all MAFs which are upon one ore several geometrical faces (geometrical face must be in the finished part)
 - b – all MAFs which intersect a GSM Plane. Only one plane is selectable.
- 3: A jump distance (the distance of release of the tool) could be specified.
- 4: A prefix and the name of the Power Feature could be specified in pattern's name to easily know pattern content
- 5: This option allows a MAF to be in several patterns.
- 6: Information frame

6.5.2 Create pattern from similar MAFs

This command () allows to generate only pattern that contain the same kind of Power feature MAFs.



Pwf Similar Holes Creation

Inputs
MAF Type : ... (1)

Information
Direction X : 0 Y : 0 Z : 0 (2)
Depth : ..

Search Type
☒ All feature Parameters similar (3)
☐ Type, Direction and Diameter similar
☐ Put only the selected MAF into pattern

Property
Jump distance : 0mm (4)

Naming
Pattern Prefix : (5)
☐ Include MAF-name into pattern name

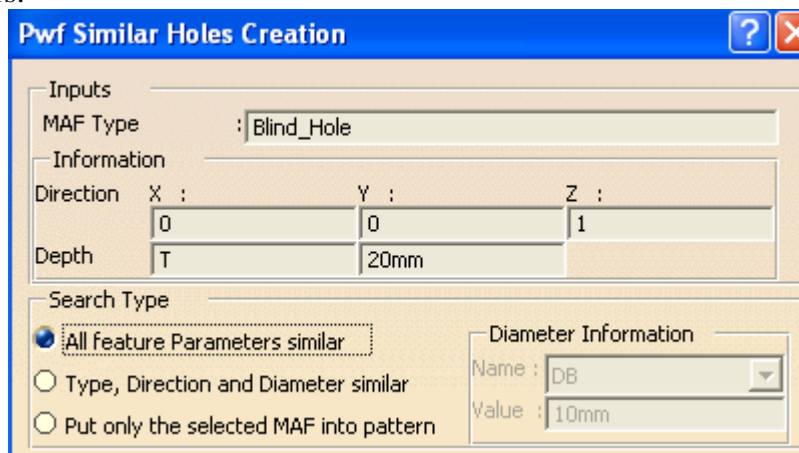
MAF property
☐ Authorize MAF to be in several Patterns (6)

OK Cancel Preview

With this command instead of specify a direction you specify a Power Feature MAFs. The created MAF will contain only this kind of selected MAFs. Direction and Depth are automatically filled if data are provided.

We can use 3 kind of search:

- 1- All parameters similar. We create patterns where MAFs have the same value for all parameters.



Pwf Similar Holes Creation

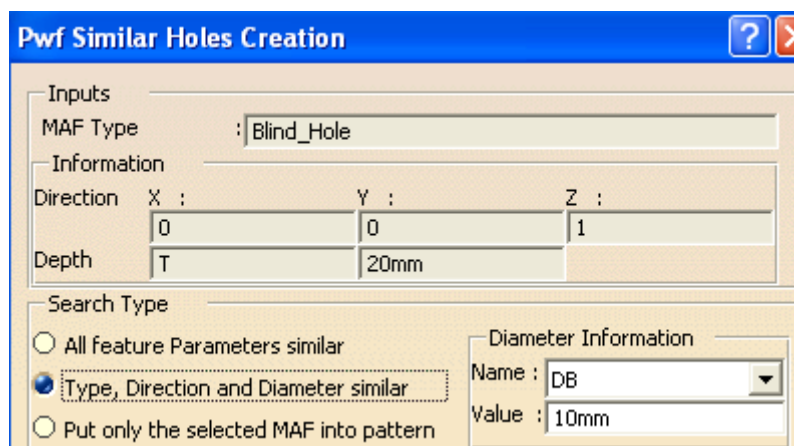
Inputs
MAF Type : Blind_Hole

Information
Direction X : 0 Y : 0 Z : 1
Depth : T 20mm

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

Diameter Information
Name : DB
Value : 10mm

- 2- Type, Direction and Diameter similar. We create patterns where MAFs have same direction and the specific diameter value.



Pwf Similar Holes Creation

Inputs
MAF Type : Blind_Hole

Information
Direction X : 0 Y : 0 Z : 1
Depth T 20mm

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

Diameter Information
Name : DB
Value : 10mm

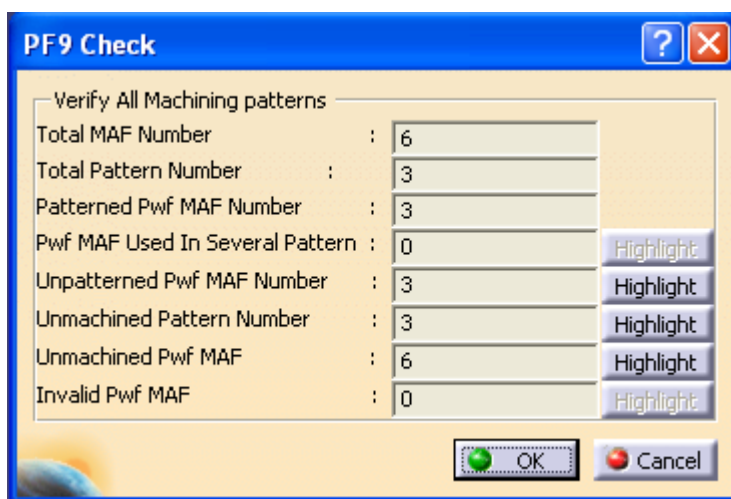
3- “Put only ...”. Only the selected MAF will be put in the pattern

6.6 Check MAFs & pattern status

The standard Power Feature Process take as input that all power Feature MAF must be in a Power Feature pattern. This tool allows to check this pre-requisite;



This function summarizes the different 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.



PF9 Check

Verify All Machining patterns

Total MAF Number	: 6	
Total Pattern Number	: 3	
Patterned Pwf MAF Number	: 3	
Pwf MAF Used In Several Pattern	: 0	Highlight
Unpatterned Pwf MAF Number	: 3	Highlight
Unmachined Pattern Number	: 3	Highlight
Unmachined Pwf MAF	: 6	Highlight
Invalid Pwf MAF	: 0	Highlight

OK Cancel

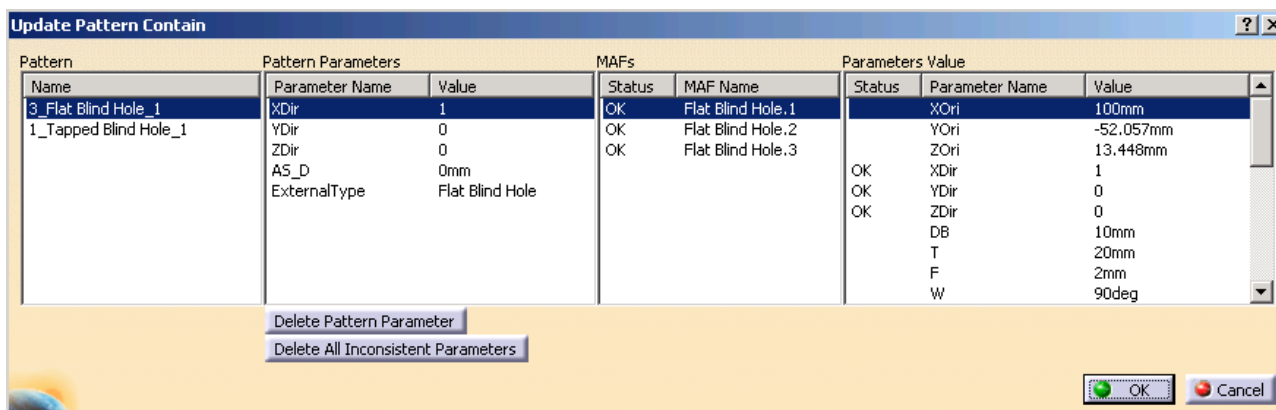
Note: You can see the MAF as features by activating “Manufacturing View” and sort by manufacturing features

6.7 Update pattern contains

During Process, geometry could change; So a Power Feature MAF that was included in a Power Feature pattern could not correspond anymore to the pattern criteria.



This command (Pwf) checks the consistency between the parameters pattern criteria and the power Feature MAFs parameters. We don’t take in account the geometrical filter use to create the pattern.













If parameters Power Feature are changed, we know if it doesn't respond to the condition. If you change the diameter DB of Flat blind Hole, he becomes KO.

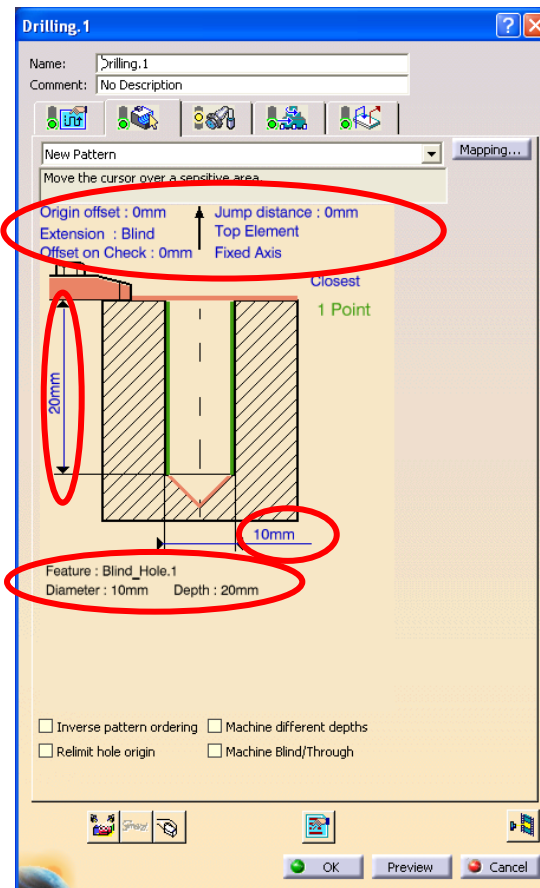
Note: When the diameter is changed, the TR is updated (if we are in auto update) but you have to update the MAFs. A click on "delete all inconsistent parameters" "solves" the problem. At this moment, DP is not a necessary condition for the pattern and the flat blind hole becomes OK

6.8 Machining Operation and MAFs diameter and depth mapping

Thanks to the expert machining process delivery, when an end-user creates a machining operation on a MAF position, the following parameters are automatically set. Parameters are:

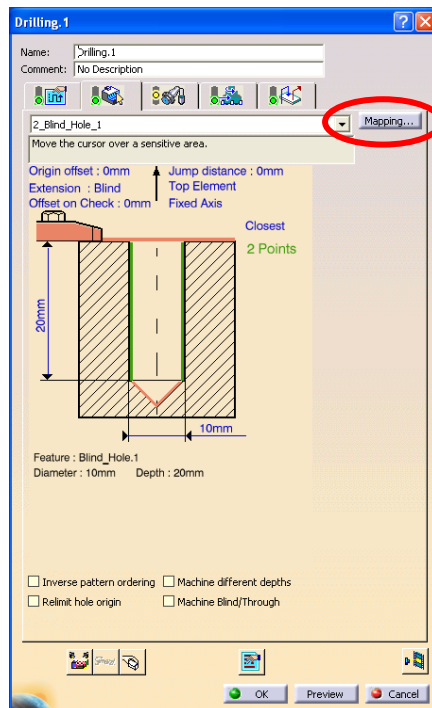
-  Diameter : The hole diameter
-  Depth : The hole depth
-  CBDiameter : The hole head diameter
-  CBDepth : The hole head depth
-  CSAngle : The hole head angle
-  THDiameter : The hole thread diameter
-  THDepth : The hole thread depth
-  THPitch : The hole thread pitch
-  THSens : The hole thread side. (0: right threaded 1: left threaded)
-  Extension : The hole extension. (1: blind 2: through) done by Through parameter of Power Feature (upper R16)

Example:

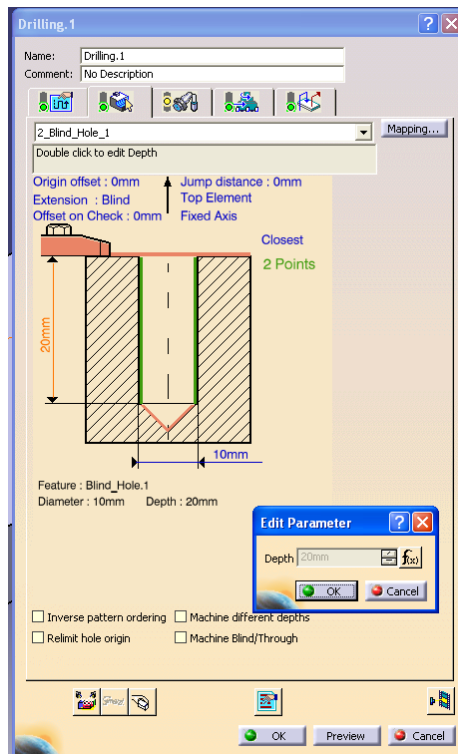


6.9 Mapping Rules and Machining Operation = NC link

When you define a machining operation, we have some advantages to use BPA Power Feature. For each Power Feature MAFs, lists a customized machining operation are provided. After selected a MAFs or patterns, a mapping button is display in the second machining operation tab page. This push button gives access to the list of customized machining operation



When a mapping rule is applied, it's different to change the value of the parameter of an operation. The $f(x)$ function is applied.



6.10 Power Feature MAFs/ Pattern and Machining process

Power Feature MAFs and Pattern can be used with Machining process as any standard MAFs and patterns. For more information, have a look to [\\$v5doc/online/mfgug_C2/mfgugwd0208.htm](#)

7 Knowing Bugs

7.1 Catalogs

If catalogs (Power feature, Geometrical rules and business rules) don't follow the architecture rule, error panel appear and CAD Power feature command need to be restarted.

7.2 Applicative rules CATPart

If Applicative rules CATPart (Geometrical and business) don't follow the architecture rule, error panel appear and CAD Power feature command need to be restarted.