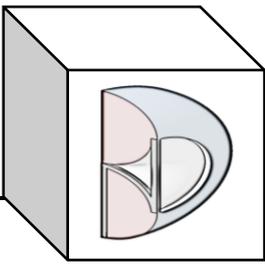


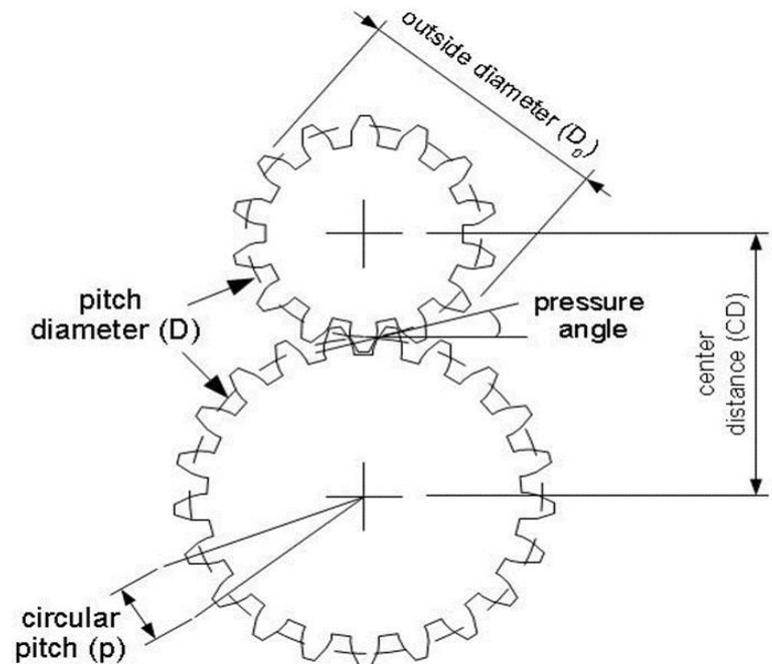
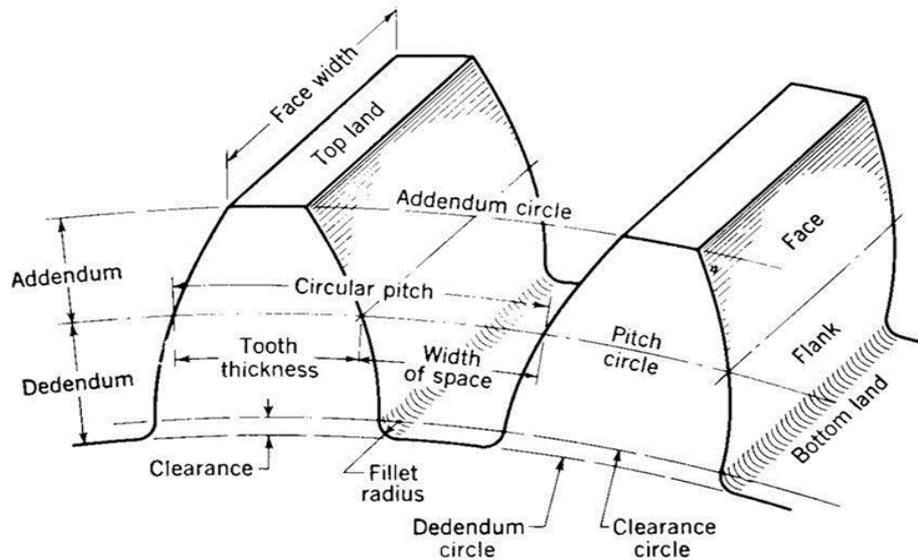
## **Designing Approximate Spur Gear in CATIA V5 (Method 4 of 4: creating/using Macro)**

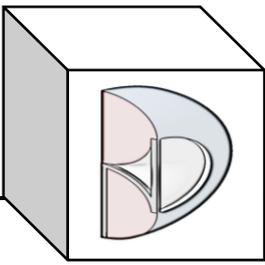




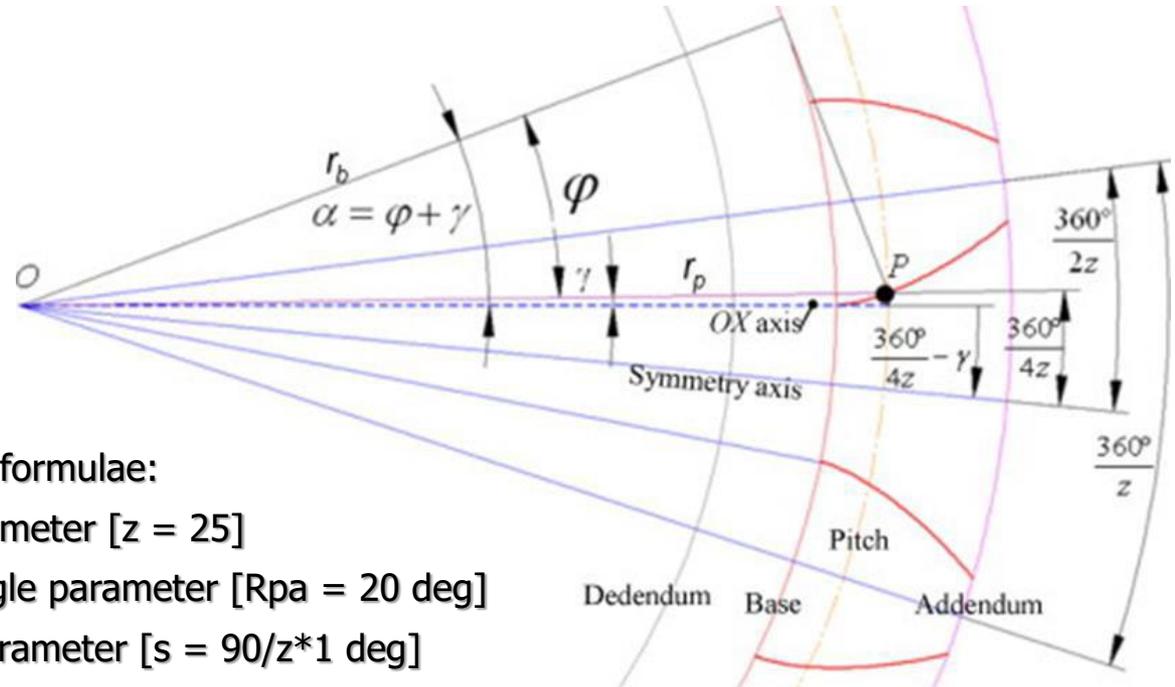
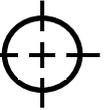
- This is step by step guide of how to create an *approximate* involute spur gear using CATIA V5.
- This document assumes that you know basic spur gear geometry.

## GEAR NOMENCLATURE





# BND TechSource



Some basic nomenclature and formulae:

$z$  -number of teeth // real parameter [ $z = 25$ ]

$R_{pa}$  -ref. pressure angle // angle parameter [ $R_{pa} = 20 \text{ deg}$ ]

$s$  -symmetry angle // angle parameter [ $s = 90/z * 1 \text{ deg}$ ]

$m$  -module // length parameter [ $m = 3\text{mm}$ ]

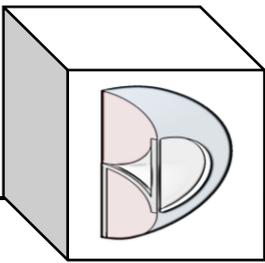
$P_d$  -Pitch diameter // length parameter [ $P_d = z * m$ ]

$B_d$  -Base diameter // length parameter [ $B_d = P_d * \cos(R_{pa}[\text{radians}])$ ]

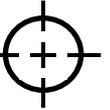
$A_d$  -Addendum diameter // length parameter [ $A_d = P_d + (2 * m)$ ]

$D_d$  -Dedendum diameter // length parameter [ $D_d = P_d - (2.5 * m)$ ]

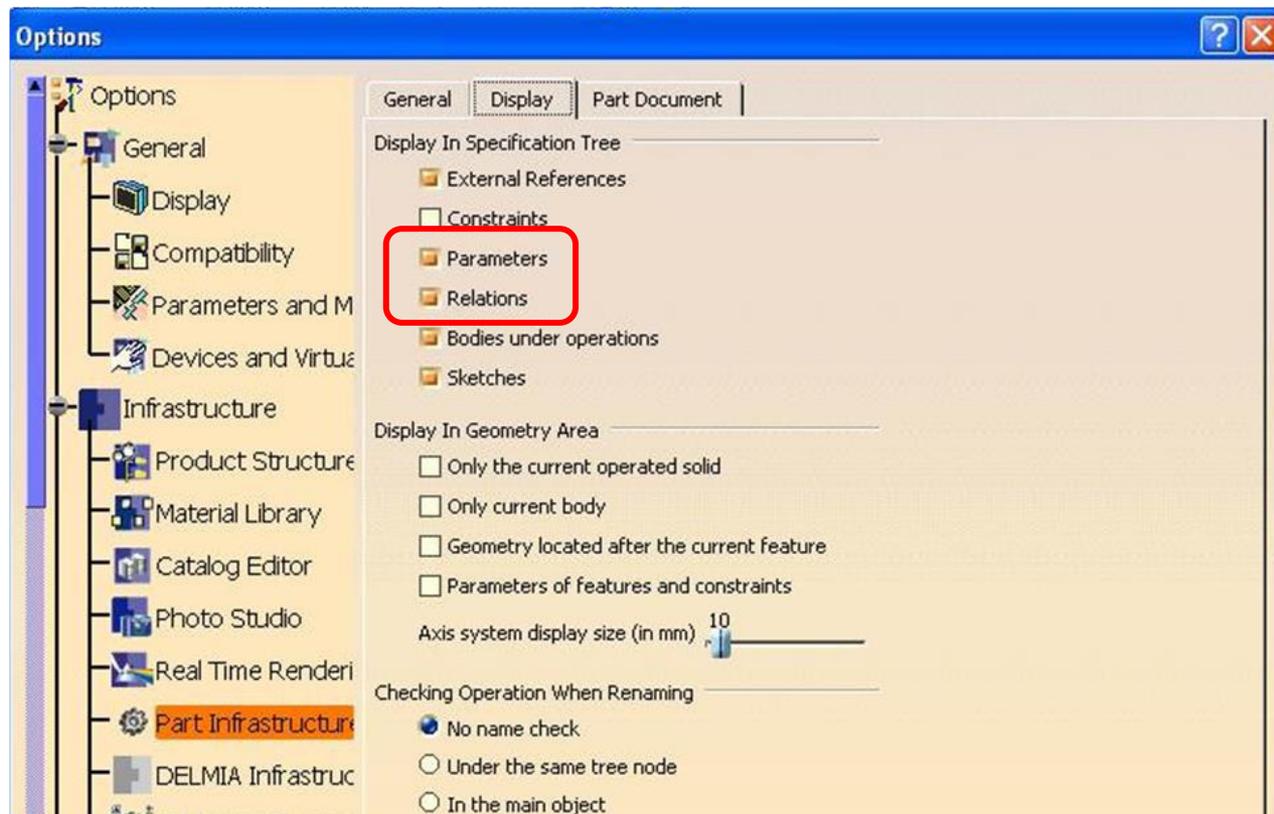
$tr$  -tooth radius at dedendum circle // length parameter [ $0.38 * m$ ]

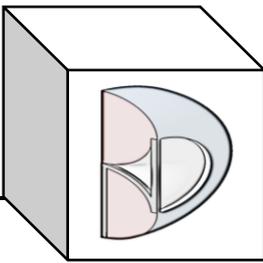


# BND TechSource

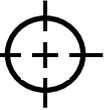


- When you start CATIA, go to TOOLS->OPTIONS->Infrastructure->
- Part Infrastructure and in Display select “Parameters” and “Relations”.

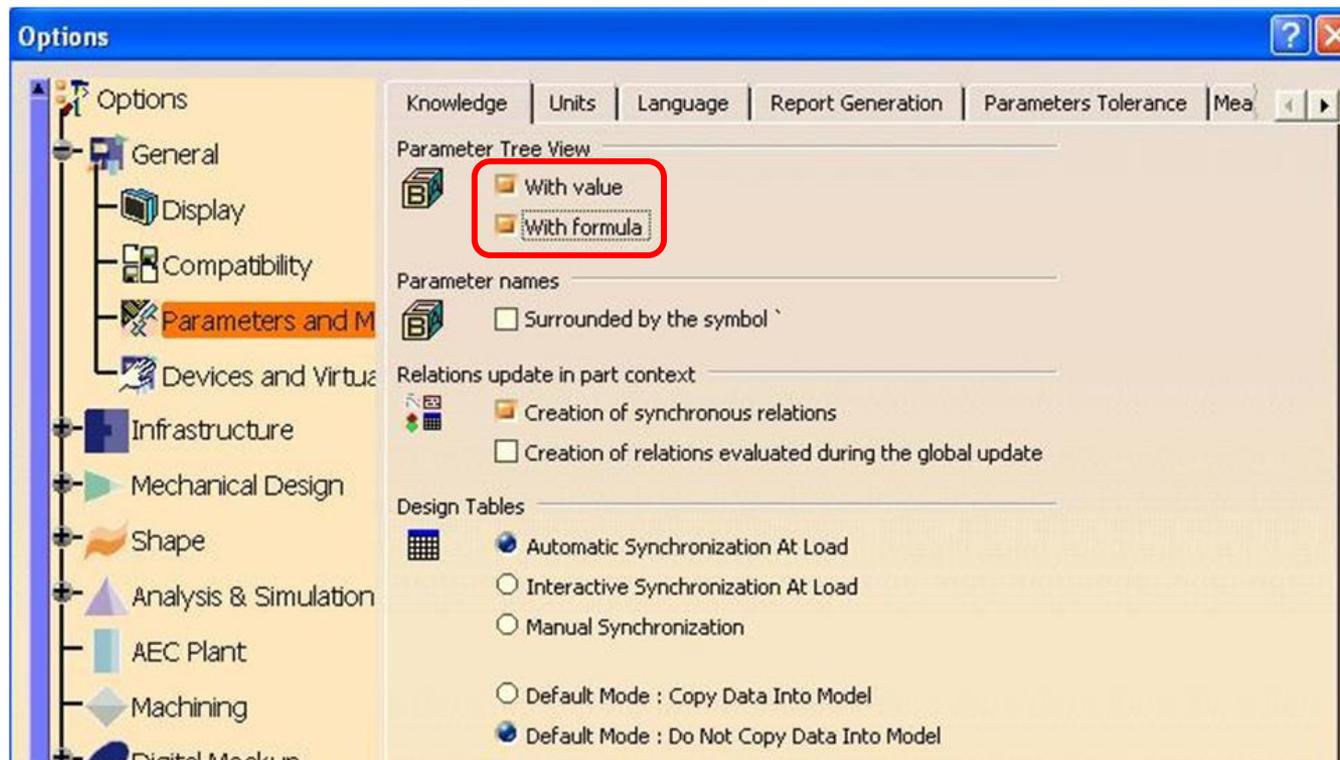


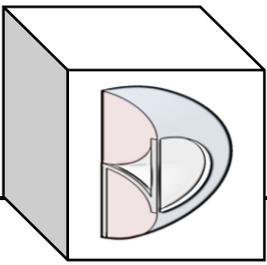


# BND TechSource

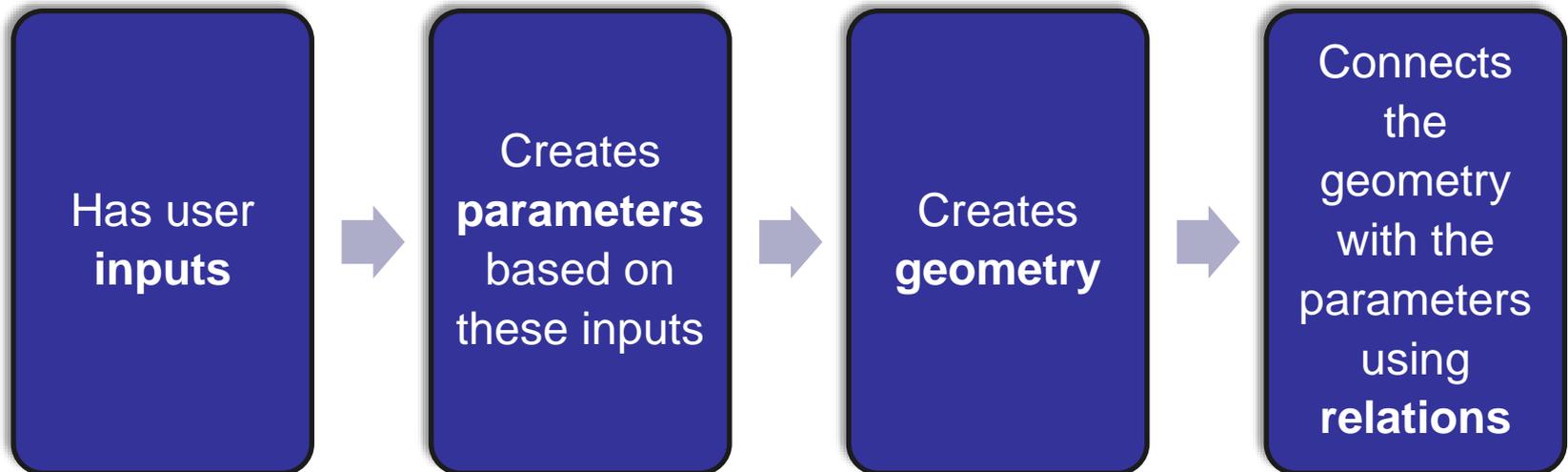


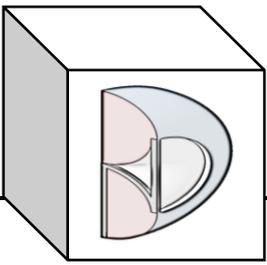
- Then in Options->General in Parameters and Measures select “With Value” and “With Formula” in Parameters Tree View.





- This macro:

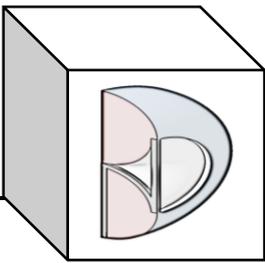




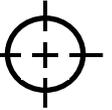
# BND TechSource



**Here is the finished  
macro...**



# BND TechSource



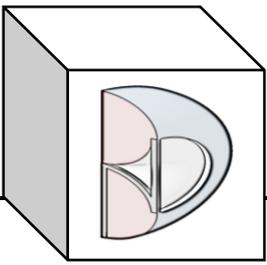
- Run the macro.

a) Go to Tools + Macro + Macro, or use Alt+F8

b) Select the Spur Gear Macro

c) Pick "Run"

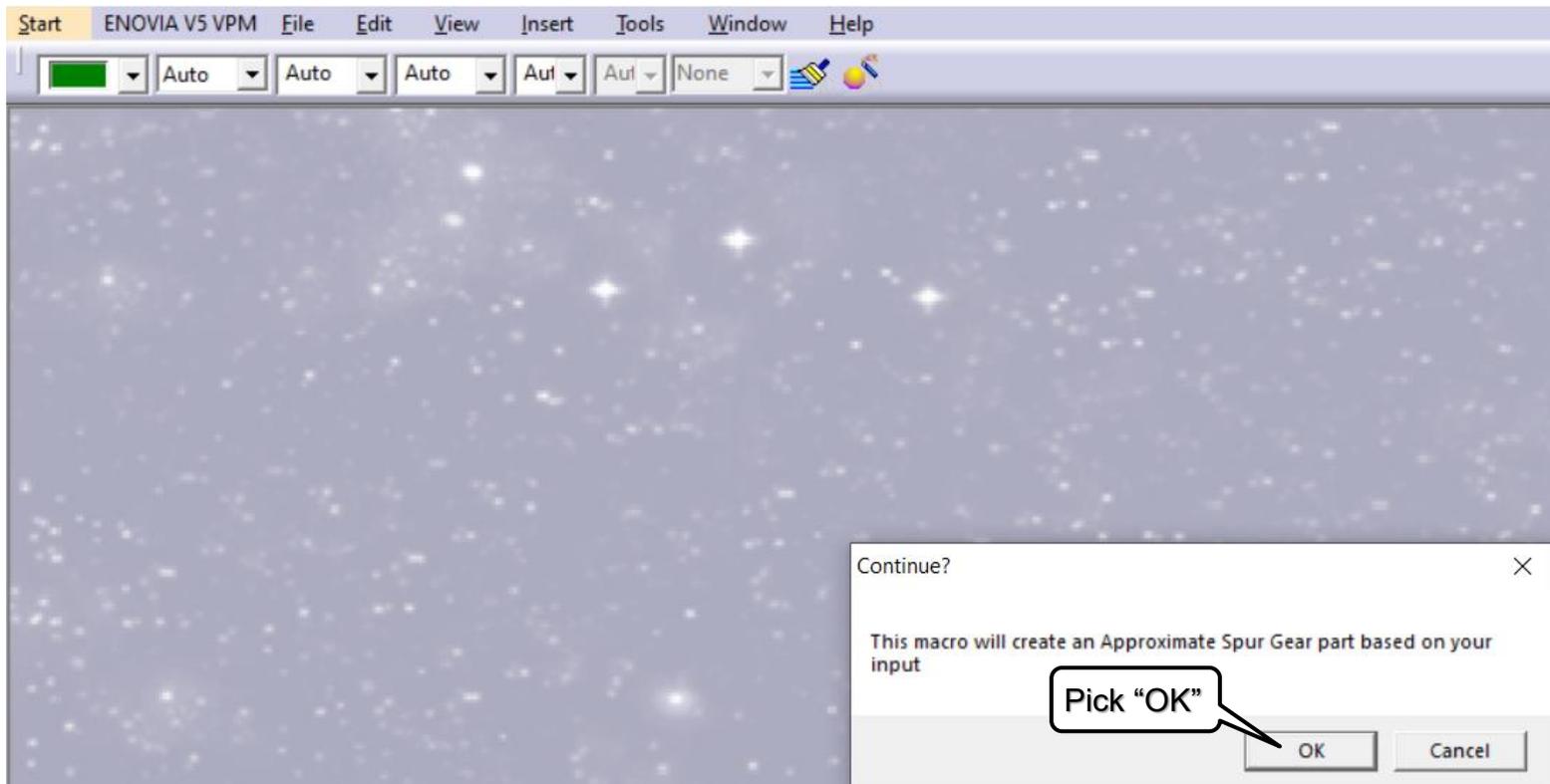
Name	Language
Approximate Spur Gear.catvbs	MS VBScript
Geometry.catvbs	MS VBScript
Input.catvbs	MS VBScript
Params.catvbs	MS VBScript
Relations.catvbs	MS VBScript

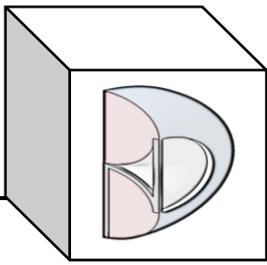


# BND TechSource



- Run the macro.

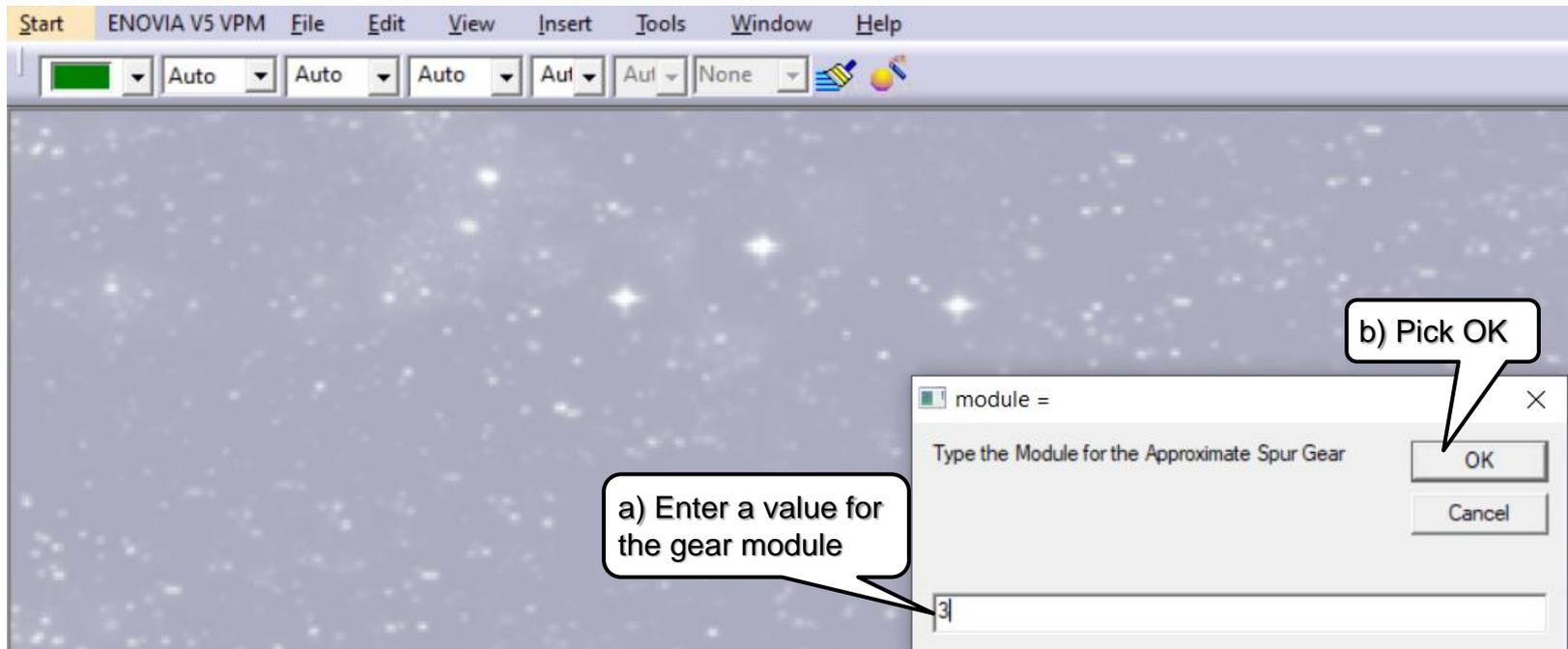


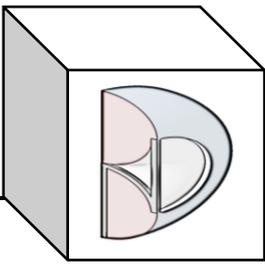


# BND TechSource

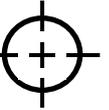


- Enter a value for the gear module.

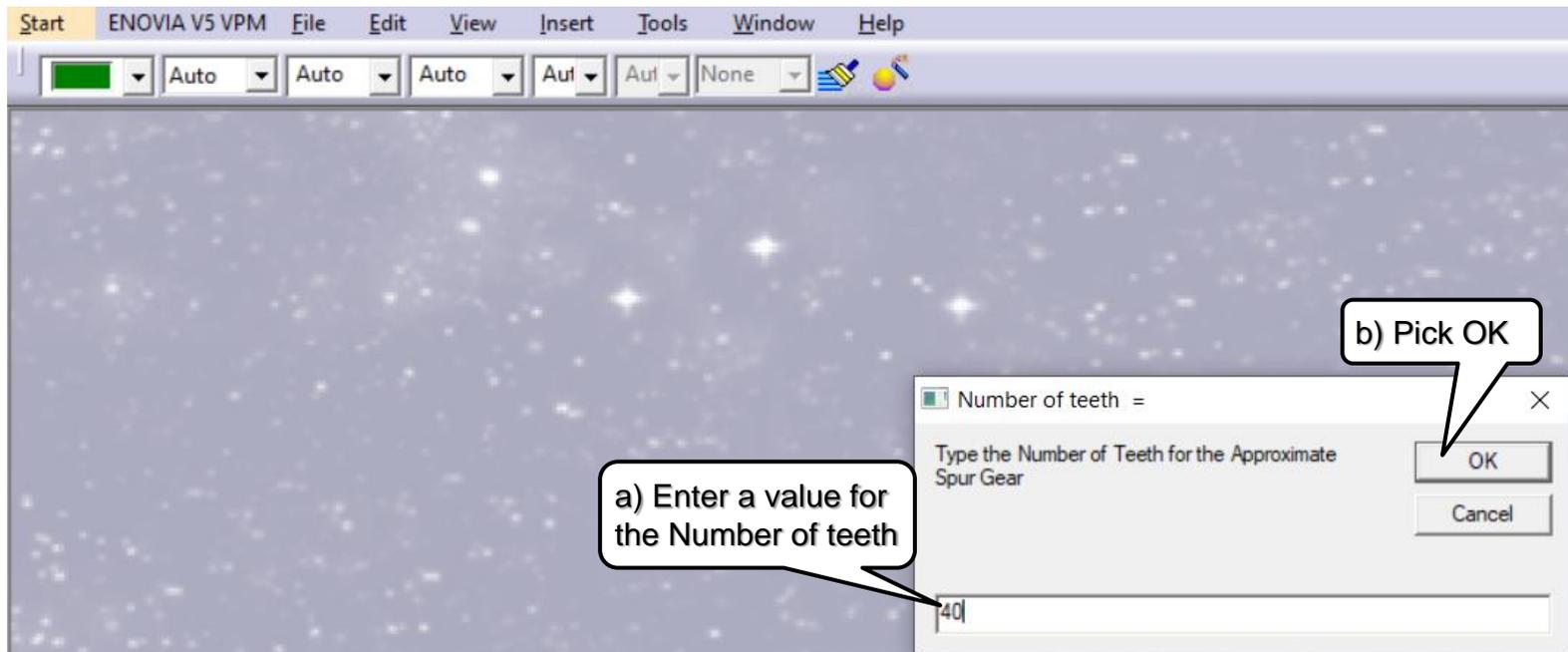


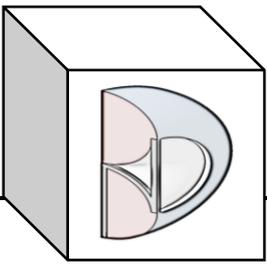


# BND TechSource



- Enter a value for the Number of teeth.

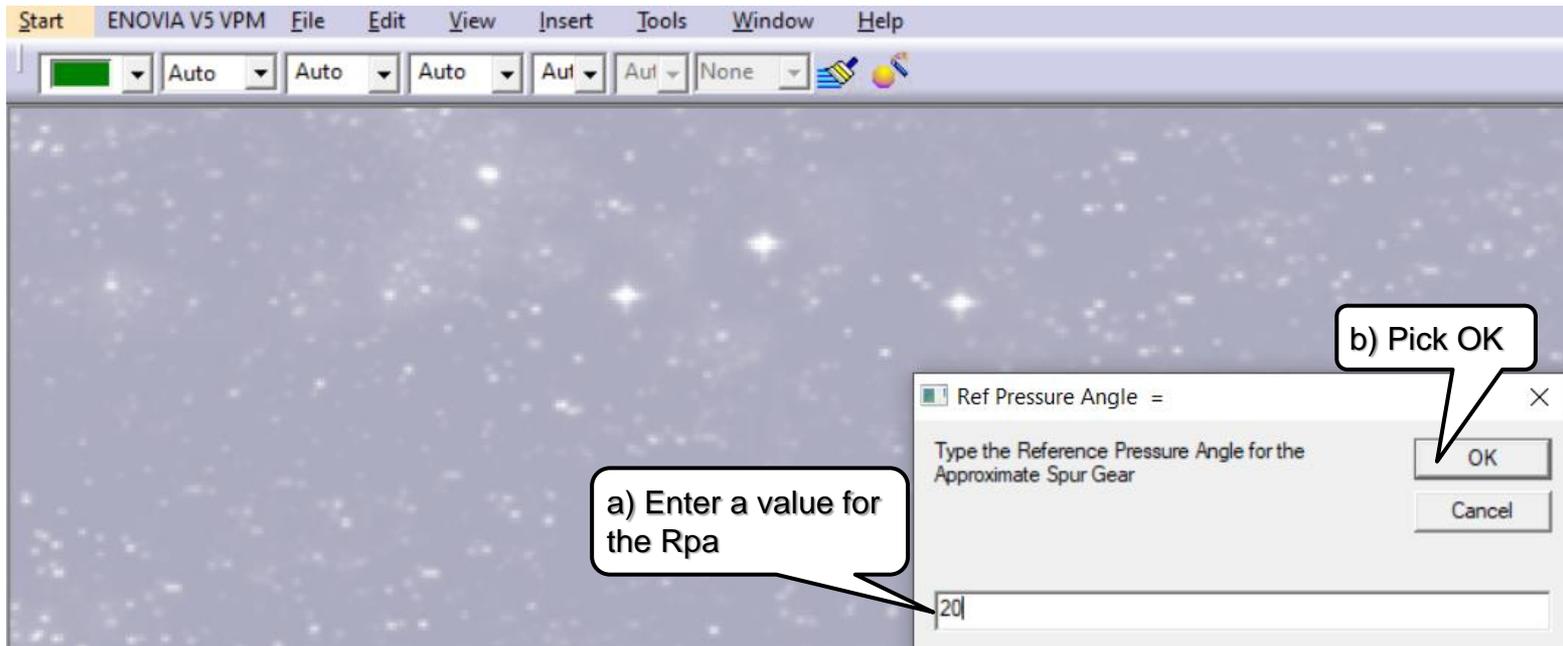


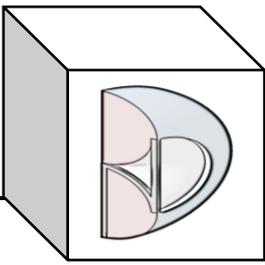


# BND TechSource



- Enter a value for the Reference Pressure Angle.

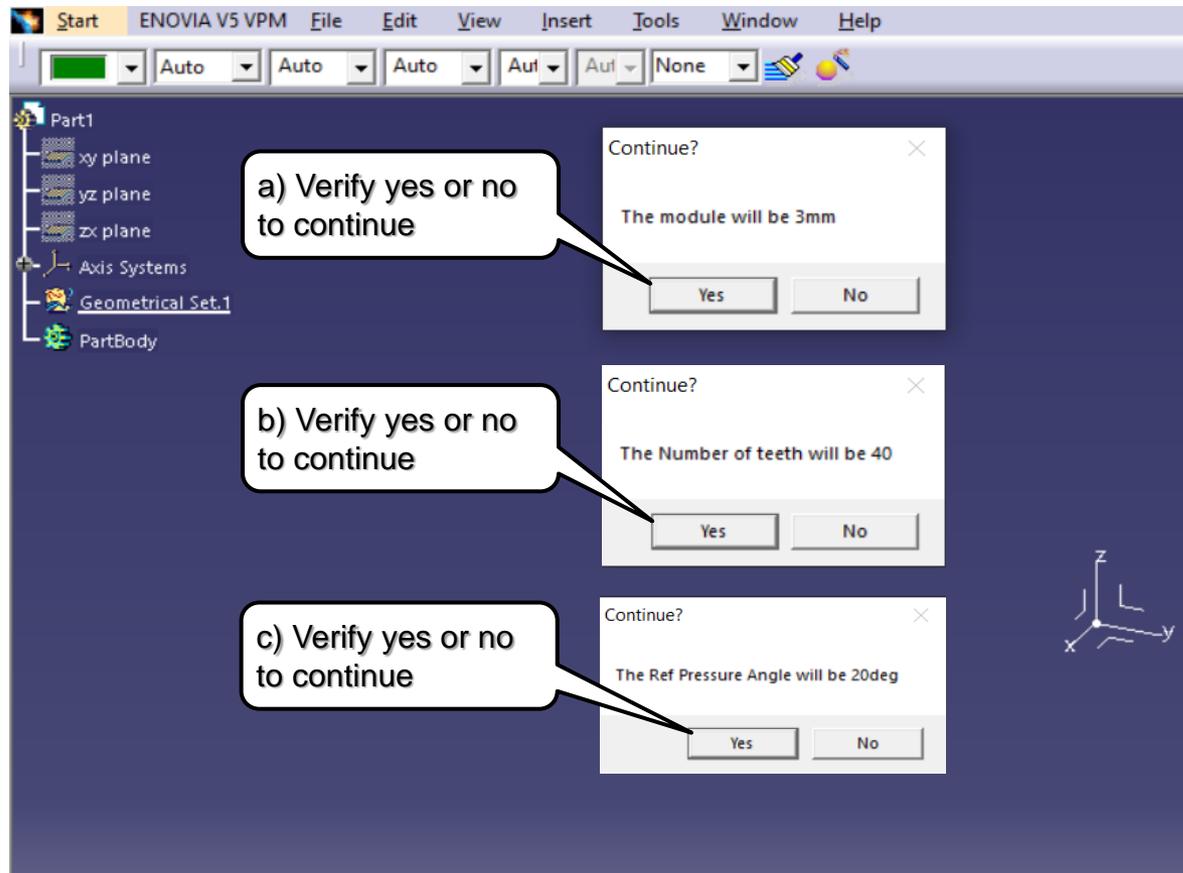


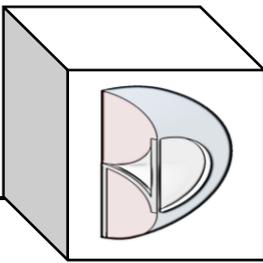


# BND TechSource

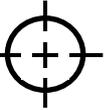


- Data verification.





# BND TechSource



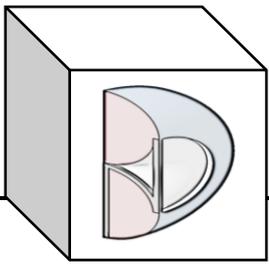
- Approximate Spur Gear part is complete.

The screenshot displays the ENOVIA V5 VPM software interface. The main window shows a 3D model of an approximate spur gear. The left-hand side contains a tree view with the following structure:

- Approximate Spur Gear [m=3, z=40, Rpa= 20]
  - xy plane
  - yz plane
  - xz plane
  - Axis Systems
  - Parameters
    - Number of teeth: z = 40
    - Ref Pressure Angle: Rpa = 20deg
    - Symmetry angle: s = 2.25deg = 90 / Number of teeth: z \* 1 deg
    - Module: m = 3mm
    - Pitch diameter: Pd = 120mm = Number of teeth: z \* Module: m
    - Base diameter: Bd = 112.763mm = Pitch diameter: Pd \* cos(Ref Pressure Angle: Rpa)
    - Addendum diameter: Ad = 126mm = Pitch diameter: Pd + (2 \* Module: m)
    - Dedendum diameter: Dd = 112.5mm = Pitch diameter: Pd - (2.5 \* Module: m)
    - tooth radius at dedendum circle: tr = 1.14mm = 0.38 \* Module: m
  - Relations
    - Formula.1: Symmetry angle: s = 90 / Number of teeth: z \* 1 deg
    - Formula.2: Pitch diameter: Pd = Number of teeth: z \* Module: m
    - Formula.3: Base diameter: Bd = Pitch diameter: Pd \* cos(Ref Pressure Angle: Rpa)
    - Formula.4: Addendum diameter: Ad = Pitch diameter: Pd + (2 \* Module: m)
    - Formula.5: Dedendum diameter: Dd = Pitch diameter: Pd - (2.5 \* Module: m)
    - Formula.6: tooth radius at dedendum circle: tr = 0.38 \* Module: m
    - Formula.7: Body.2\CircPattern.1\AngularNumber = Number of teeth: z
    - Formula.8: Body.2\CircPattern.1\AngularSpacing = 360 deg / Number of teeth: z
    - Formula.9: Geometrical Set.1\Sketch.1\Radius.1\Radius = Pitch diameter: Pd / 2
    - Formula.10: Geometrical Set.1\Sketch.1\Radius.2\Radius = Base diameter: Bd / 2
    - Formula.11: Geometrical Set.1\Sketch.1\Radius.14\Radius = Dedendum diameter: Dd / 2
    - Formula.12: Geometrical Set.1\Sketch.1\Radius.15\Radius = Addendum diameter: Ad / 2
    - Formula.13: Geometrical Set.1\Sketch.1\Radius.18\Radius = tooth radius at dedendum circle: tr
    - Formula.14: Geometrical Set.1\Sketch.1\Radius.21\Radius = tooth radius at dedendum circle: tr

Three callout boxes provide additional information:

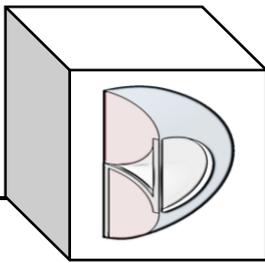
- a) Notice the part has been renamed
- b) Notice all the parameters are complete
- c) Notice all of the relations are complete



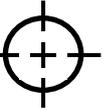
# BND TechSource



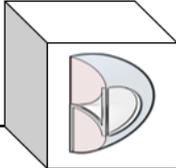
## Create the “Input” Block



# BND TechSource



- We know from the manual input (method 1) there are three input parameters from which all others are built.

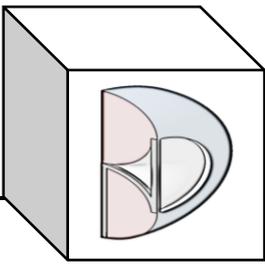


## BND TechSource



- Create the following Nine formulae:
- Known:
  - $z$  // real parameter [**Number of teeth:  $z = 25$** ]
  - $Rpa$  // angle parameter [**Ref Pressure Angle:  $Rpa = 20$  deg**]
  - $m$  // length parameter [**Module:  $m = 3$ mm**]
- Resultant:
  - $s$  // angle parameter [**Symmetry angle:  $s = 90/z*1$  deg**]
  - $Pd$  // length parameter [**Pitch diameter:  $Pd = z*m$** ]
  - $Bd$  // length parameter [**Base diameter:  $Bd = Pd * \cos(Rpa[\text{radians}])$** ]
  - $Ad$  // length parameter [**Addendum diameter:  $Ad = Pd+(2*m)$** ]
  - $Dd$  // length parameter [**Dedendum diameter:  $Dd = Pd-(2.5*m)$** ]
  - $tr$  // length parameter [**tooth radius at dedendum circle:  $tr = 0.38*m$** ]

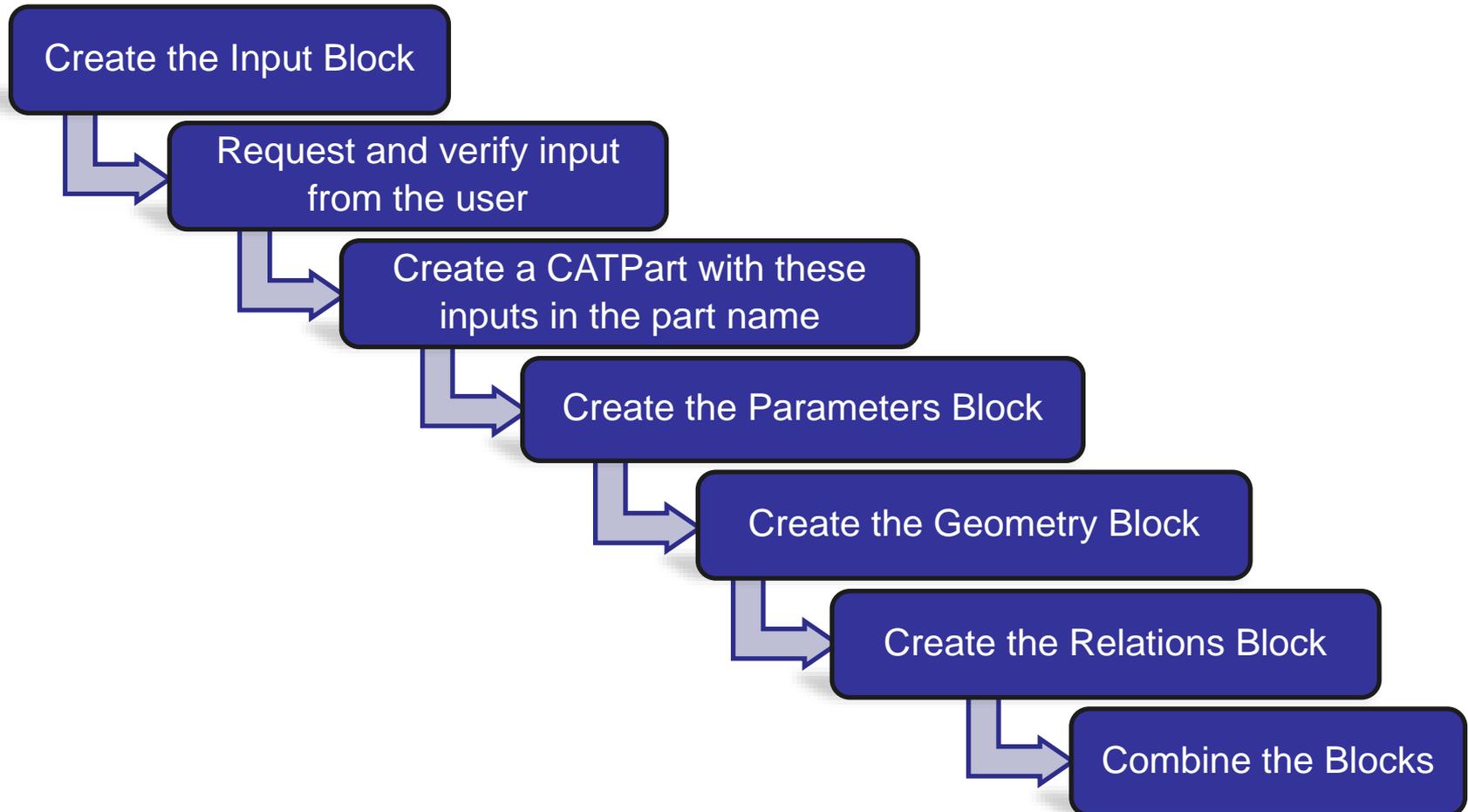
CATIA sees a diametral dimension as a radius. Even though the dimension displays as a diameter.

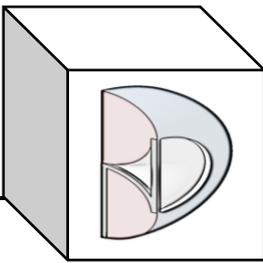


# BND TechSource



- Macro process...





# BND TechSource



- Create the Input Block portion of the macro.

**d)** This macro was written in VBScript. There are also VBAScript and CATScript available.

The coding for the Input portion of our macro must be created (not recorded).  
For those without a lot of coding experience, we suggest looking online to forums such as “stack overflow” for VBScript insight.

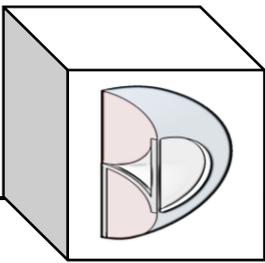
**a)** Alt+F8 to open the macro directory

**b)** Select the “Input” block.

**c)** Pick “Edit” to work on this portion of the macro.

```
Macros Editor - [D:\-William\CATIA Tutorials\Approximate Spur Gear\Input.catvbs]
File Edit View Help
language= "VBSCRIPT"
Sub CATMain()
a = MsgBox ("This macro will create an Approximate Spur Gear")
If a = vbCancel Then
Exit sub
ElseIf a = vbOK Then
module = ""
module = InputBox ("Type the Module for the Approximate Spur Gear", "module = ", module)
teeth = ""
teeth = InputBox ("Type the Number of Teeth for the Approximate Spur Gear", "Number of teeth = ", teeth)
Rpa = ""
Rpa = InputBox ("Type the Reference Pressure Angle for the Approximate Spur Gear", "Ref Pressure Angle = ", Rpa)
Set documents1 = CATIA.Documents
Set partDocument1 = documents1.Add("Part")
Set product1 = partDocument1.GetItem("Part 1")
result = MsgBox ("The module will be " & module, vbYesNo, "Continue?")
If vbNo Then
Do until result = vbYes
module = ""
module = InputBox ("Type the module for the Approximate Spur Gear", "module = ", module)
result = MsgBox ("The module will be " & module, vbYesNo, "Continue?")
Loop
Else
End if
result = MsgBox ("The Number of teeth will be " & teeth, vbYesNo, "Continue?")
```

Name	Language
Approximate Spur Gear.catvbs	MS VBScript
Geometry.catvbs	MS VBScript
<b>Input.catvbs</b>	<b>MS VBScript</b>
Params.catvbs	MS VBScript
Relations.catvbs	MS VBScript



# BND TechSource



- Test the Macro for the Input Block.

Macros

Current macro library or document:  
D:\~William\CATIA Tutorials\Approximate Spur Gear

Available macros:

Name	Language
Approximate Spur Gear.catvbs	MS VBScript
Geometry.catvbs	MS VBScript
input.catvbs	MS VBScript
Params.catvbs	MS VBScript
Relations.catvbs	MS VBScript

Pick "Run"

Continue?

This macro will create an Approximate Spur Gear part based on your input

module =

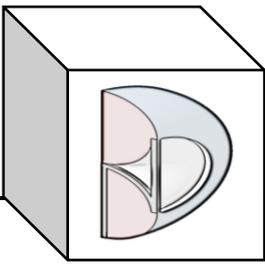
Type the Module for the Approximate Spur Gear

Ref Pressure Angle =

Type the Reference Pressure Angle for the Approximate Spur Gear

Number of teeth =

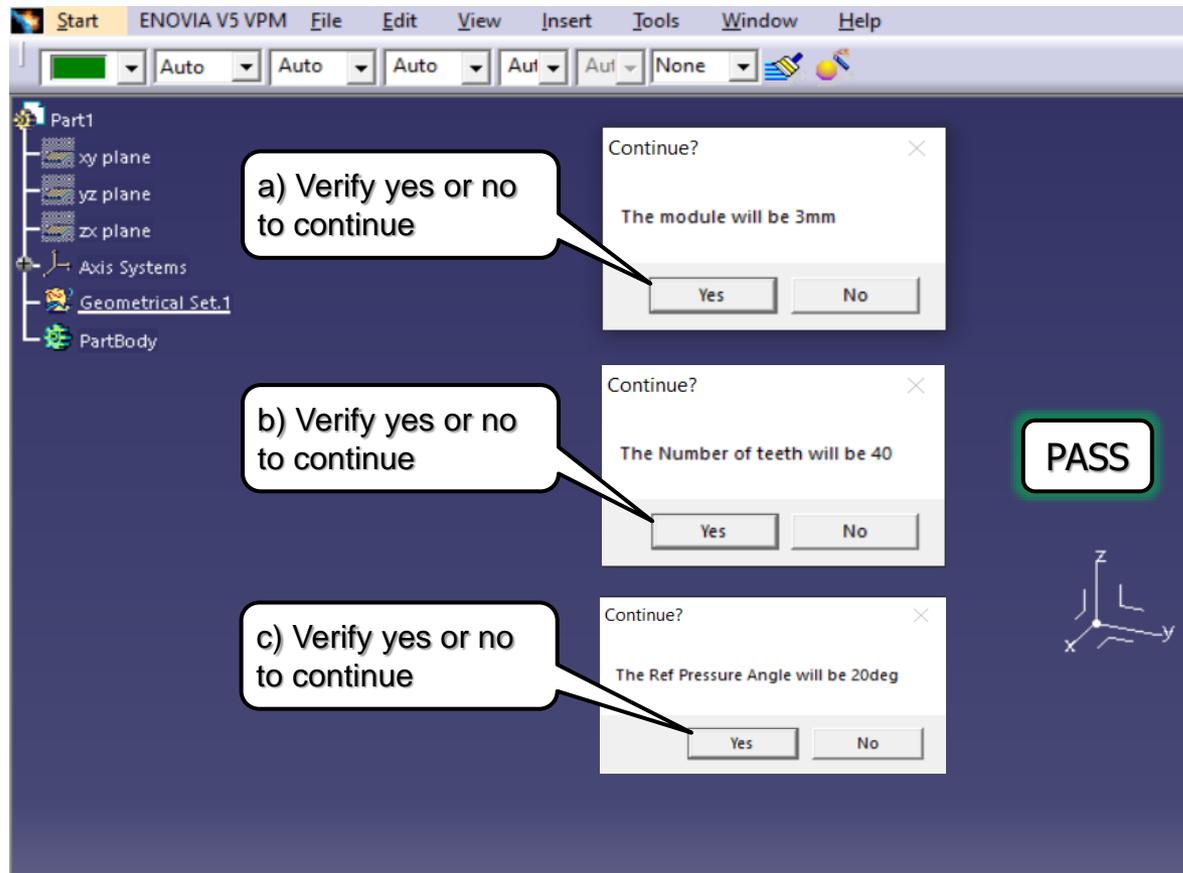
Type the Number of Teeth for the Approximate Spur Gear

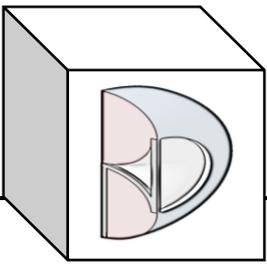


# BND TechSource



- Test the Macro for the Input Block.

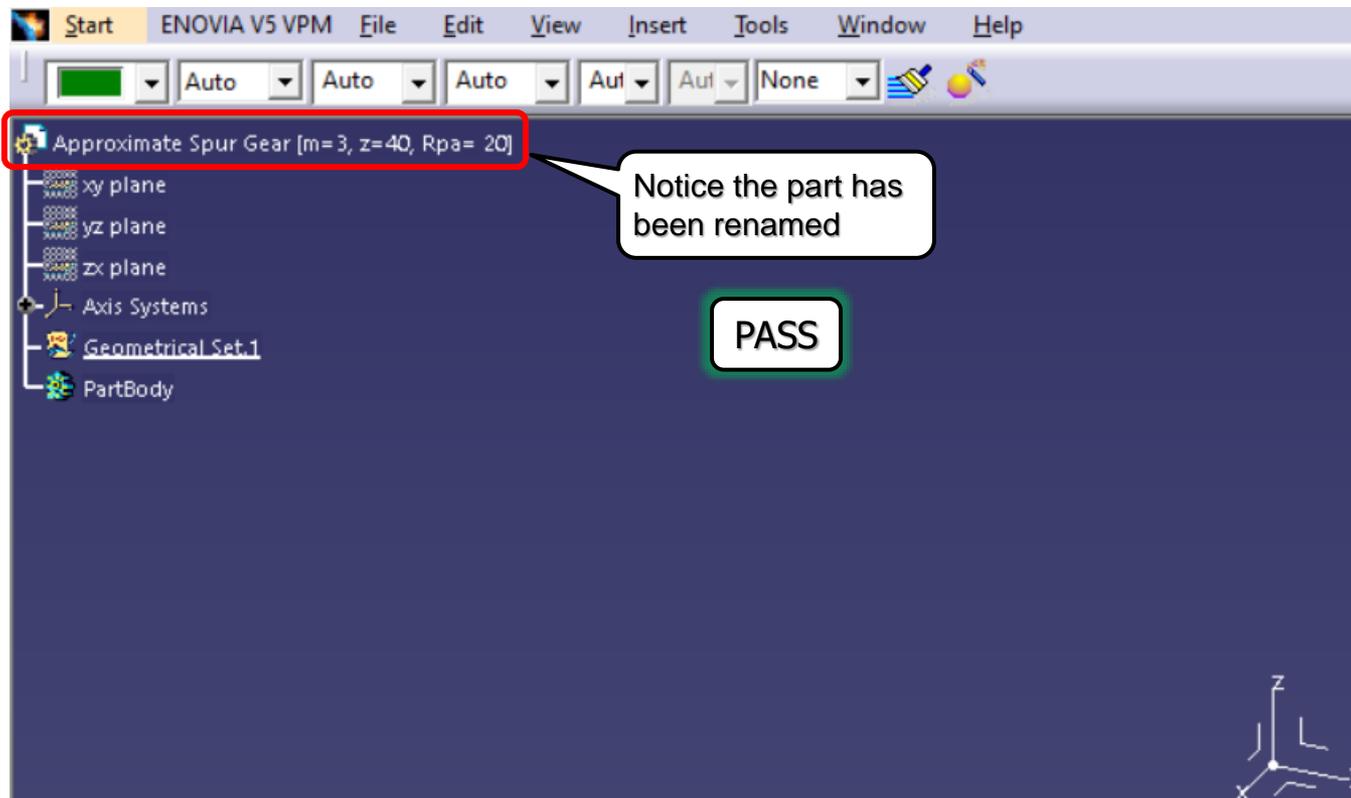


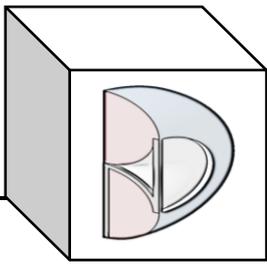


# BND TechSource



- Test the Macro for the Input Block.

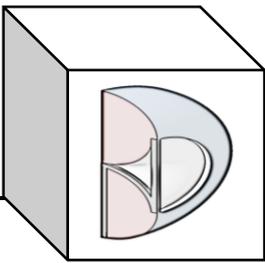




# BND TechSource



## Create the “Parameters” Block



# BND TechSource



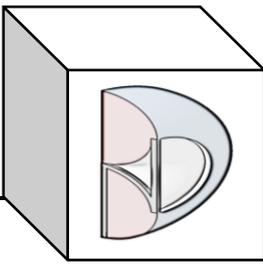
- **Record the Macro while creating the parameters.**

a) Go to Tools + Macro + Start Recording

b) Rename the Macro "Parameters"

c) Pick "Start"

d) Notice the Record "Stop" icon appears



# BND TechSource



- Open the Formulas window.

Pick "Real" for the first formula

Pick the Function icon

Pick "OK"

Open the drop-down list

## 1) z // real parameter [Number of teeth: z = 40]

Formulas: Approximate Spur Gear [m=3, z=40, Rpa= 20]

Filter On Approximate Spur Gear [m=3, z=40, Rpa= 20]

Filter Name:

Filter Type: All

Double click on a parameter to edit it

Parameter	Value	Formula	Active
'Absolute Axis System\Activity'	true		
'Approximate Spur Gear [m=3, z=40, Rpa= 20]\Part Number'	Approximat		
'Approximate Spur Gear [m=3, z=40, Rpa= 20]\Nomenclature'			
'Approximate Spur Gear [m=3, z=40, Rpa= 20]\Revision'			
'Approximate Spur Gear [m=3, z=40, Rpa= 20]\Product Descri...			
'Approximate Spur Gear [m=3, z=40, Rpa= 20]\Definition'			
<b>Number of teeth: z</b>	<b>40</b>		

Edit name or value of the current parameter

Number of teeth: z

New Parameter of type Real With Single Value

Delete Parameter

Add Formula

Delete Formula

OK Apply Cancel

1

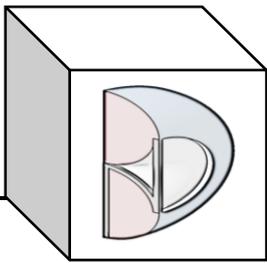
a) Pick New Parameter

b) Type "Number of teeth: z" for the name

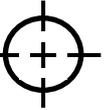
c) Set 40 for the number of teeth

d) Pick OK





# BND TechSource



## 2) Rpa // angle parameter [Ref Pressure Angle: Rpa = 20 deg]

Formulas: Approximate Spur Gear

Filter On Approximate Spur Gear  
Filter Name:   
Filter Type: All

Double click on a parameter to edit it

Parameter	Value	Formula	Active
'Number of teeth: z'	25		
'Approximate Spur Gear\Part Number'	Approximat...		
'Approximate Spur Gear\Nomenclature'			
'Approximate Spur Gear\Revision'			
'Approximate Spur Gear\Product Description'			
'Approximate Spur Gear\Definition'			
<b>'Ref Pressure Angle: Rpa'</b>	<b>20deg</b>		

Edit name or value of the current parameter

Ref Pressure Angle: Rpa      20deg

New Parameters of type Angle With Single Value

Delete Parameter      Add Formula      Delete Formula

OK      Apply      Cancel

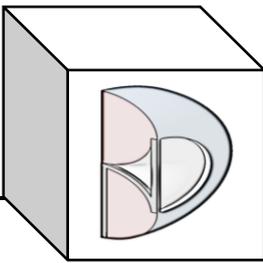
a) Pick New Parameter

b) Type "Ref Pressure Angle: Rpa" for the name

c) Set 20 for the Ref Pressure Angle

d) Pick OK

2



# BND TechSource



3)  $s$  // angle parameter [**Symmetry angle:  $s = 90/z * 1$  deg**]

**d) Type 90/Number of teeth: z \* 1 deg**

**e) Pick OK**

**b) Type "Symmetry angle: s" for the name**

**c) Pick Add Formula**

**f) Pick Apply**

**a) Pick New Parameter**

**3**

Formula Editor: 'Geometrical Set.1\Sketch.4\Activity'

Geometrical Set.1\Sketch.4\Activity =

Formulas: Approximate Spur Gear

90/Number of teeth: z \* 1 deg

Dictionary	Members of Parameters	Members of Real
Parameters	Boolean	'Absolute Axis System\XAxis\Z'
Design Table	Length	'Absolute Axis System\YAxis\X'
Operators	CstAttr_Mode	'Absolute Axis System\YAxis\Y'
Pointer on value function	Angle	'Absolute Axis System\YAxis\Z'
Point Constructors	Integer	'Absolute Axis System\ZAxis\X'
Law	Real	'Absolute Axis System\ZAxis\Y'
Operations Constructors	String	'Absolute Axis System\ZAxis\Z'
<	Feature	'Number of teeth: z'

Number of teeth: z 40

OK Cancel

Parameter

Relations\Formula.17\Activity

'Approximate Spur Gear (manual input)\Nomenclature'

'Approximate Spur Gear (manual input)\Revision'

'Approximate Spur Gear (manual input)\Product Description' Approximat...

'Approximate Spur Gear (manual input)\Definition'

'Symmetry angle: s' 0deg

Edit name or value of the current parameter

Symmetry angle: s 0deg

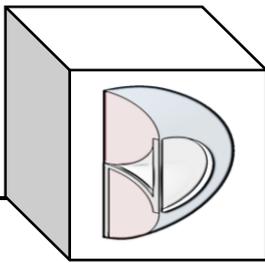
New Parameter of type Angle With Single Value

Delete Parameter

Add Formula

Delete Formula

OK Apply Cancel



# BND TechSource

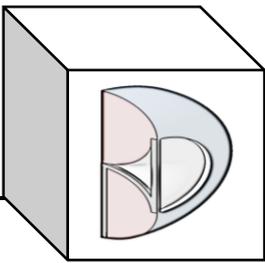


- The next Six Parameters will be Length type.

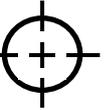
Pick "Length" for the next five formulae

Pick the Function icon

Pick "OK"



# BND TechSource



4) m // length parameter [**Module: m = 3mm**]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Approximate Spur Gear

xy plane  
yz plane  
zx plane  
Axis Systems  
Geometrical Set.1  
Sketch.1  
PartBody

Formulas: Approximate Spur Gear

Filter On Approximate Spur Gear  
Filter Name:   
Filter Type: All

Double click on a parameter to edit it

Parameter	Value	Formula	Active
'Ref Pressure Angle: Rpa'	20deg		
'Approximate Spur Gear\Part Number'	Approximat...		
'Approximate Spur Gear\Nomenclature'			
'Approximate Spur Gear\Revision'			
'Approximate Spur Gear\Product Description'			
'Approximate Spur Gear\Definition'			
'Module: m'	3mm		

Edit name or value of the current parameter

Module: m 3mm

New Parameter of type Length With Single Value

Delete Parameter

Add Formula

Delete Formula

OK Apply Cancel

4

a) Pick New Parameter

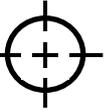
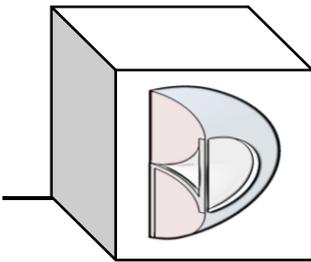
b) Type "Module: m" for the name

c) Set 3mm for the module

d) Pick OK

# BND TechSource

## 5) Pd // length parameter [**Pitch diameter: Pd = z\*m**]



The screenshot shows the ENOVIA software interface. On the left is a tree view with 'Parameters' selected. The main area is divided into two windows:

- Formula Editor: 'Pitch diameter: Pd'**: Shows the formula  $\text{Pitch diameter: Pd} = \text{Number of teeth: } z * \text{Module: } m$ . It includes a dictionary of parameters and a list of members.
- Parameters dialog**: Shows a list of parameters. 'Pitch diameter: Pd' is selected with a value of 120mm. The formula  $\text{Pitch diameter: Pd} = \text{Number of teeth: } z * \text{Module: } m$  is visible in the background.

Numbered callouts (a-f) point to specific actions in the interface:

- a) Pick New Parameter (points to the 'New Parameter of type' button)
- b) Type "Pitch diameter: Pd" for the name (points to the 'Pitch diameter: Pd' text field)
- c) Pick Add Formula (points to the 'Add Formula' button)
- d) Type Number of teeth: z \*Module: m (points to the formula input field)
- e) Pick OK (points to the 'OK' button in the Formula Editor)
- f) Pick Apply (points to the 'Apply' button in the Parameters dialog)

A circled number '5' is located in the bottom left corner of the screenshot.

## 6) Bd // length parameter [**Base diameter: $Bd = Pd \cdot \cos(Rpa[\text{radians}])$** ]

The screenshot shows the SolidWorks interface with the Formula Editor and Parameter dialog boxes open. The Formula Editor displays the formula:  $\cos(\text{Ref Pressure Angle: Rpa}) * \text{Pitch diameter: Pd}$ . The Parameter dialog box shows the parameter name 'Base diameter: Bd' and its value '112.763mm'. Annotations with callouts provide step-by-step instructions:

- a) Pick New Parameter
- b) Type "Base diameter: Bd" for the name
- c) Pick Add Formula
- d) Type Pitch diameter: Pd \*cos(Ref Pressure Angle: Rpa)
- e) Pick OK
- f) Pick Apply

Note: Ref Pressure Angle: Rpa is read as radians.

Parameter	Value	Formula
'Absolute Axis System\Origin\X'		
'Absolute Axis System\Origin\Y'		
'Absolute Axis System\Origin\Z'		
'Module: m'		
'Pitch diameter: Pd'	120mm	= 'Number of teeth: z'*Modu...
'Base diameter: Bd'	112.763mm	= cos('Ref Pressure Angle: Rpa...)

6

## 7) Ad // length parameter [Addendum diameter: $Ad = Pd + (2 * m)$ ]

The screenshot shows the ENOVIA V5 Formula Editor interface. The main window displays the formula for 'Addendum diameter: Ad' as  $Pd + (2 * Module: m)$ . The interface includes a Dictionary, Members of Parameters, and Members of All sections. The parameter list shows 'Addendum diameter: Ad' with a value of 126mm. The formula editor shows the formula being entered. Callouts a-f provide step-by-step instructions for creating the parameter.

**a) Pick New Parameter**

**b) Type "Addendum diameter: Ad" for the name**

**c) Pick Add Formula**

**d) Type Pitch diameter: Pd + (2 \* Module: m)**

**e) Pick OK**

**f) Pick Apply**

## 8) Dd // length parameter [Dedendum diameter: $Dd = Pd - (2.5 * m)$ ]

The screenshot shows the ENOVIA V5 software interface with several callouts:

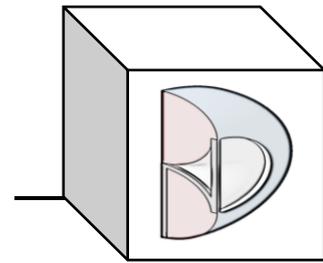
- a) Pick New Parameter**: Points to the 'New Parameter of type' button in the parameter creation dialog.
- b) Type "Dedendum diameter: Dd" for the name**: Points to the text input field for the parameter name.
- c) Pick Add Formula**: Points to the 'Add Formula' button in the parameter creation dialog.
- d) Type Pitch diameter: Pd - (2.5 \* Module: m)**: Points to the formula input field in the Formula Editor.
- e) Pick OK**: Points to the 'OK' button in the Formula Editor.
- f) Pick Apply**: Points to the 'Apply' button in the parameter creation dialog.

The Formula Editor window shows the following table:

Dictionary	Members of Parameters	Members of All
Parameters	All	'Absolute Axis System\Origin\X'
Design Table	Renamed parameters	'Absolute Axis System\Origin\Y'
Operators	Length	'Absolute Axis System\Origin\Z'
Pointer on value function	Real	'Absolute Axis System\XAxis\X'
Point Constructors	Boolean	'Absolute Axis System\XAxis\Y'
Law	Angle	'Absolute Axis System\XAxis\Z'
Operations Constructors	String	'Absolute Axis System\YAxis\X'
	Feature	

The parameter list shows the following table:

Parameter	Value	Formula
'Absolute Axis System\Origin\Y'		
'Absolute Axis System\Origin\Z'		
'Module: m'		
'Pitch diameter: Pd'		
'Base diameter: Bd'	112.763mm	= cos( Ref Pressure Angle: Kp...
'Addendum diameter: Ad'	126mm	= 'Pitch diameter: Pd' + (2 * M...
<b>'Dedendum diameter: Dd'</b>	<b>112.5mm</b>	<b>= 'Pitch diameter: Pd'</b>



## 9) tr // length parameter [tooth radius at dedendum circle: $tr = 0.38 * m$ ]

d) Type 0.38 \* Module: m

b) Type "tooth radius at dedendum circle: tr" for the name

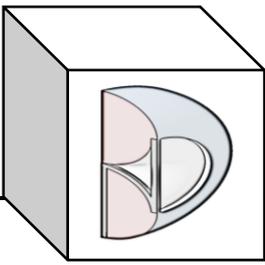
a) Pick New Parameter

c) Pick Add Formula

e) Pick OK

f) Pick Apply

9

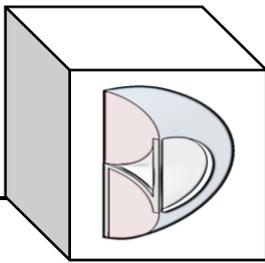


# BND TechSource



- **Stop recording the Parameters block of this Macro.**





# BND TechSource



- Check the Parameters content of this Macro.

The image shows a screenshot of the CATIA Macros Editor. The left pane displays a VBS script for creating a macro. The right pane shows the macro directory with a table of macros. Three callouts provide instructions on how to interact with the macro directory.

```
Language="VBSSCRIPT"

Sub CATMain()

Set partDocument1 = CATIA.ActiveDocument

Set part1 = partDocument1.Part

Set parameters1 = part1.Parameters

Set realParam1 = parameters1.CreateReal("", 0.000000)

realParam1.Rename "Number of teeth: z"

realParam1.Value = 40.000000

Set parameters2 = part1.Parameters

Set angle1 = parameters2.CreateDimension("", "ANGLE", 0.000000)

angle1.Rename "Ref Pressure Angle: Rpa"

angle1.Value = 20.000000

Set parameters3 = part1.Parameters

Set angle2 = parameters3.CreateDimension("", "ANGLE", 0.000000)

angle2.Rename "Symmetry angle: s"

Set relations1 = part1.Relations

Set formula1 = relations1.CreateFormula("Formula.1", "", angle2, "90/Number of teeth: z `*1 deg")

formula1.Rename "Formula.1"

Set parameters4 = part1.Parameters

Set length1 = parameters4.CreateDimension("", "LENGTH", 0.000000)

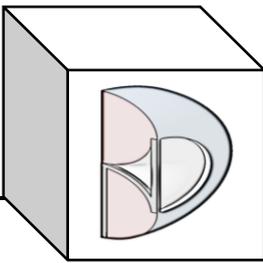
length1.Rename "Module: m"
```

Name	Language
Approximate Spur Gear.catvbs	MS VBScript
Geometry.catvbs	MS VBScript
Input.catvbs	MS VBScript
Parameters_1.catvbs	MS VBScript
Relations.catvbs	MS VBScript

a) Alt+F8 to open the macro directory

b) Select the "Parameters" block.

c) Pick "Edit" to work on this portion of the macro.



# BND TechSource



- Test the Parameters block of this Macro.

d) Notice all the parameters have been added to the part.

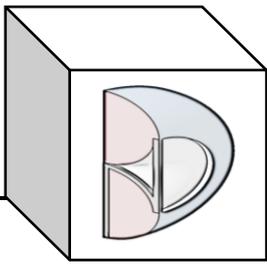
**PASS**

a) Alt+F8 to open the macro directory

b) Select the "Parameters" block.

c) Pick "Run" to test this portion of the macro.

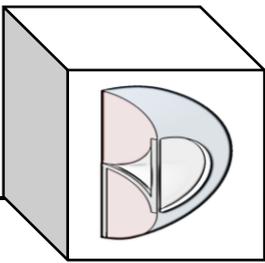
Name	Language	Buttons
Approximate Spur Gear.catvbs	MS VBScript	Run, Edit...
Geometry.catvbs	MS VBScript	Create...
Input.catvbs	MS VBScript	Rename...
Parameters_1.catvbs	MS VBScript	Delete, Select...
Relations.catvbs	MS VBScript	Obfuscate...



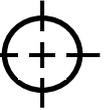
# BND TechSource



## Create the “Geometry” Block



# BND TechSource



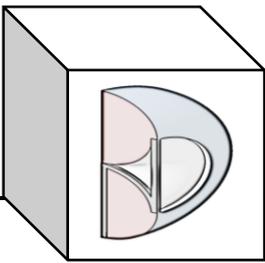
- **Start Recording** the Macro while creating the geometry.

a) Go to Tools + Macro + Start Recording

b) Rename the Macro "Geometry"

c) Pick "Start"

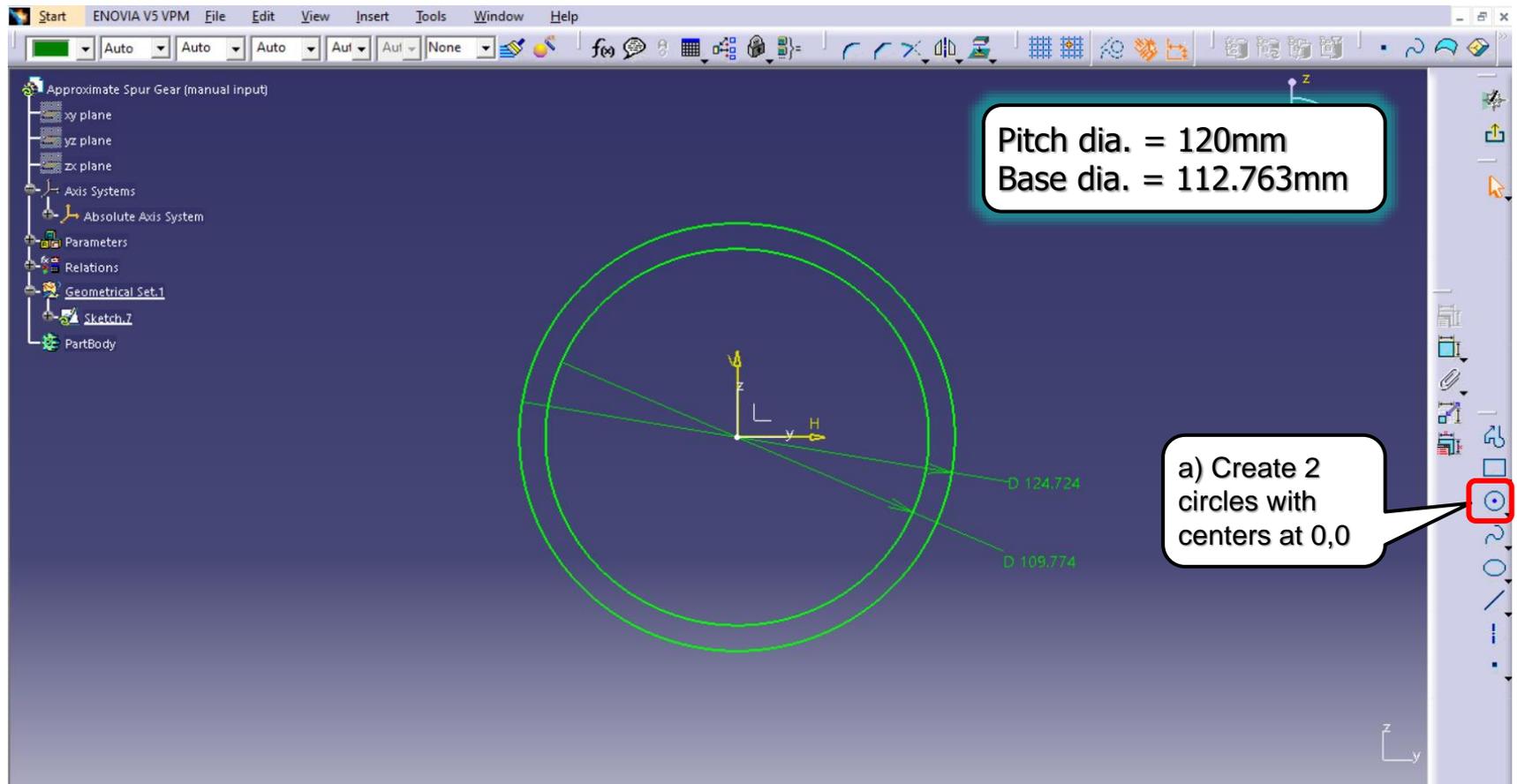
d) Notice the Record "Stop" icon appears

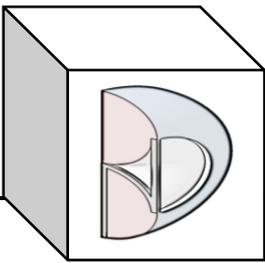


# BND TechSource



- Create the pitch and base circles in a sketch.





# BND TechSource



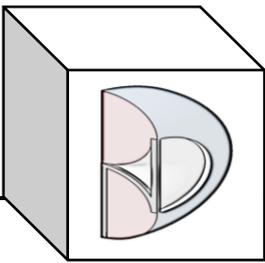
- Create tooth geometry reference lines.

a) Change the circles to reference geometry

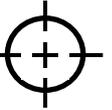
b) Create two reference lines from 0,0

c) Create an angular dimension

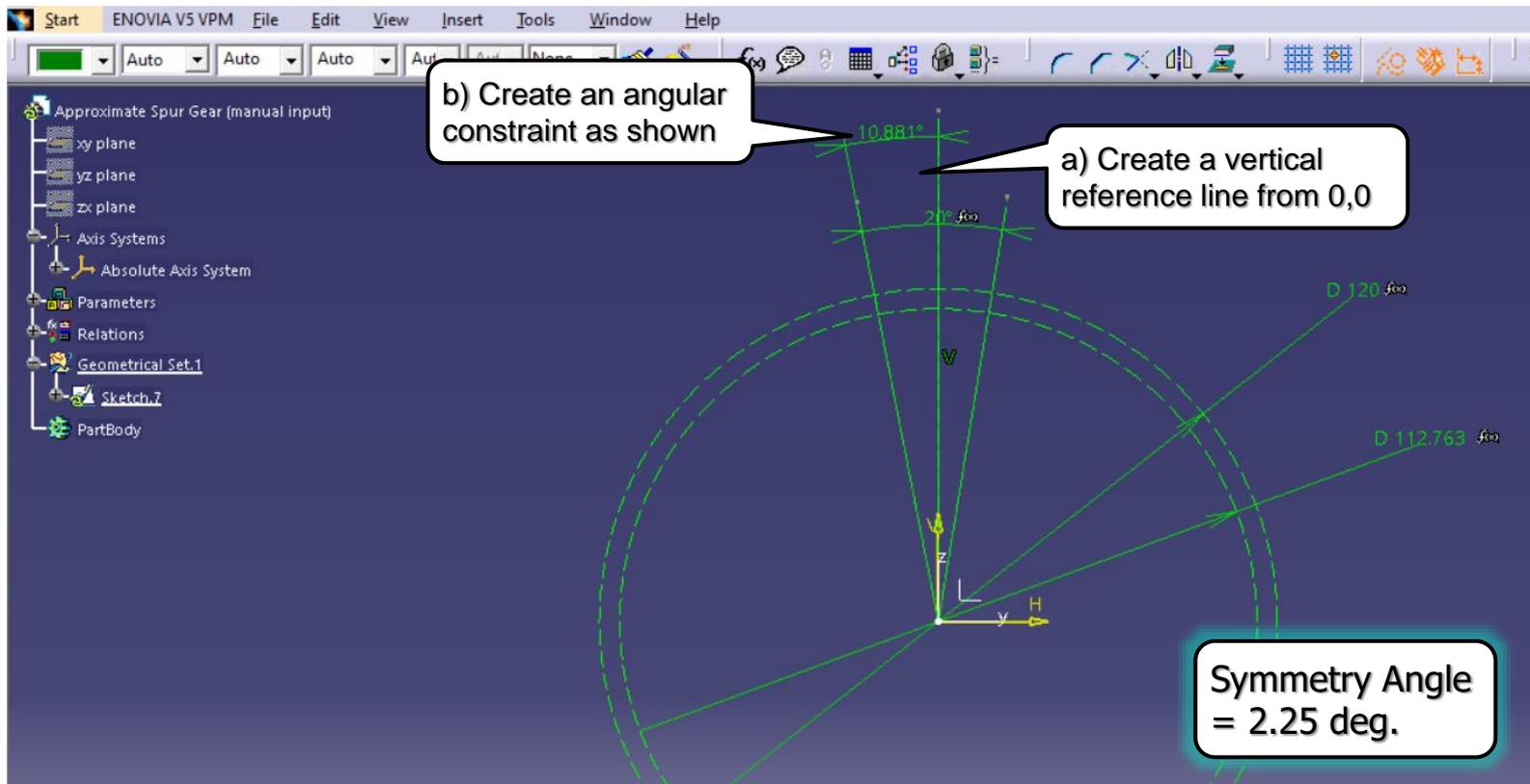
Ref. Pressure Angle = 20 deg.

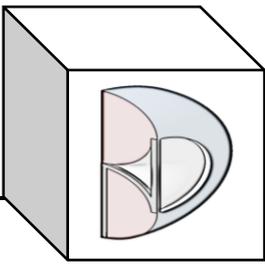


# BND TechSource



- Create tooth geometry reference lines.

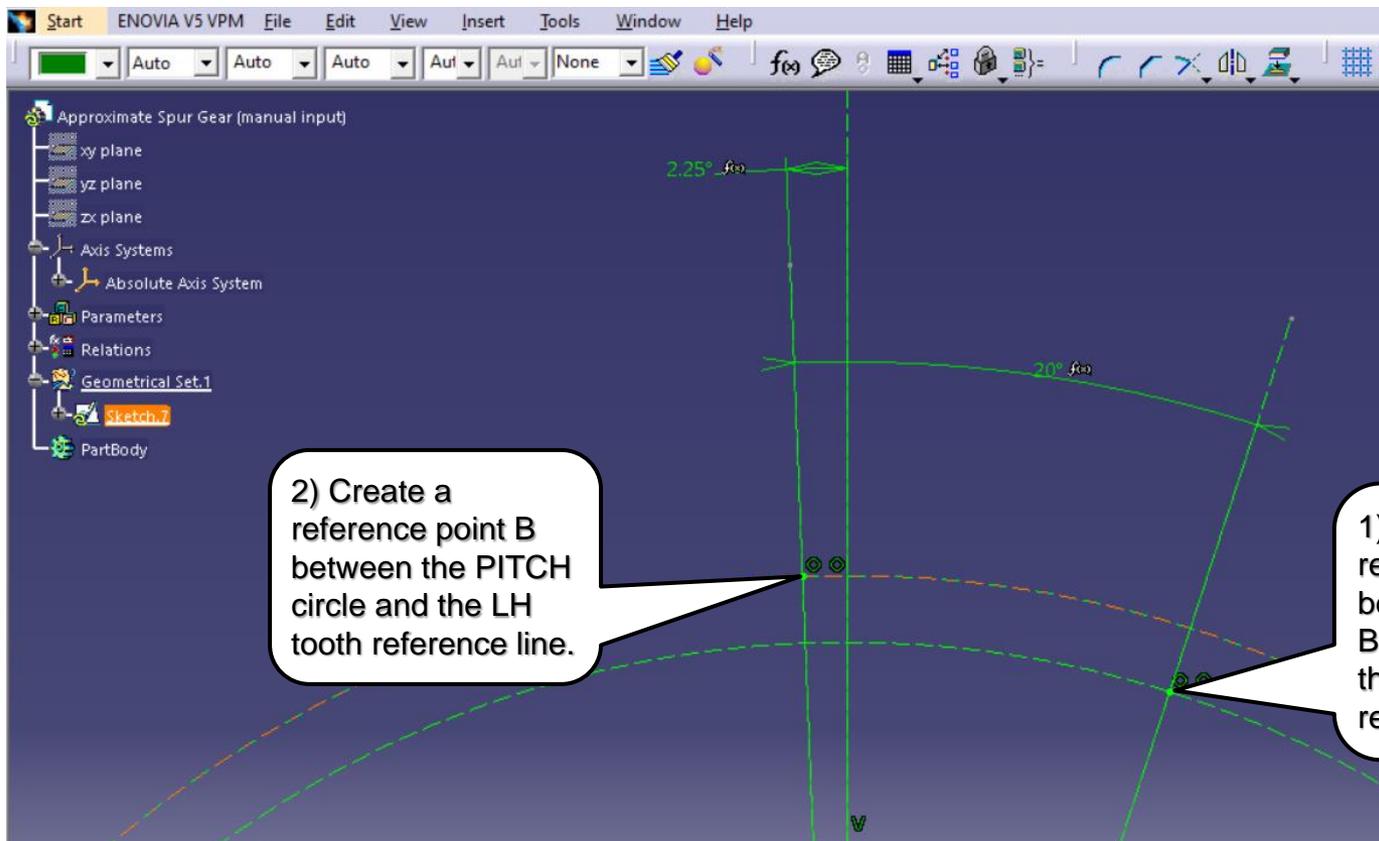


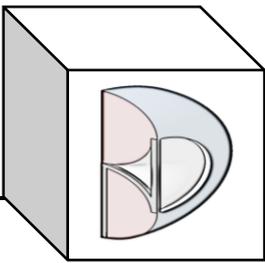


# BND TechSource

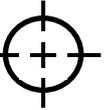


- Create reference points A & B for the construction of the approximated tooth profile.

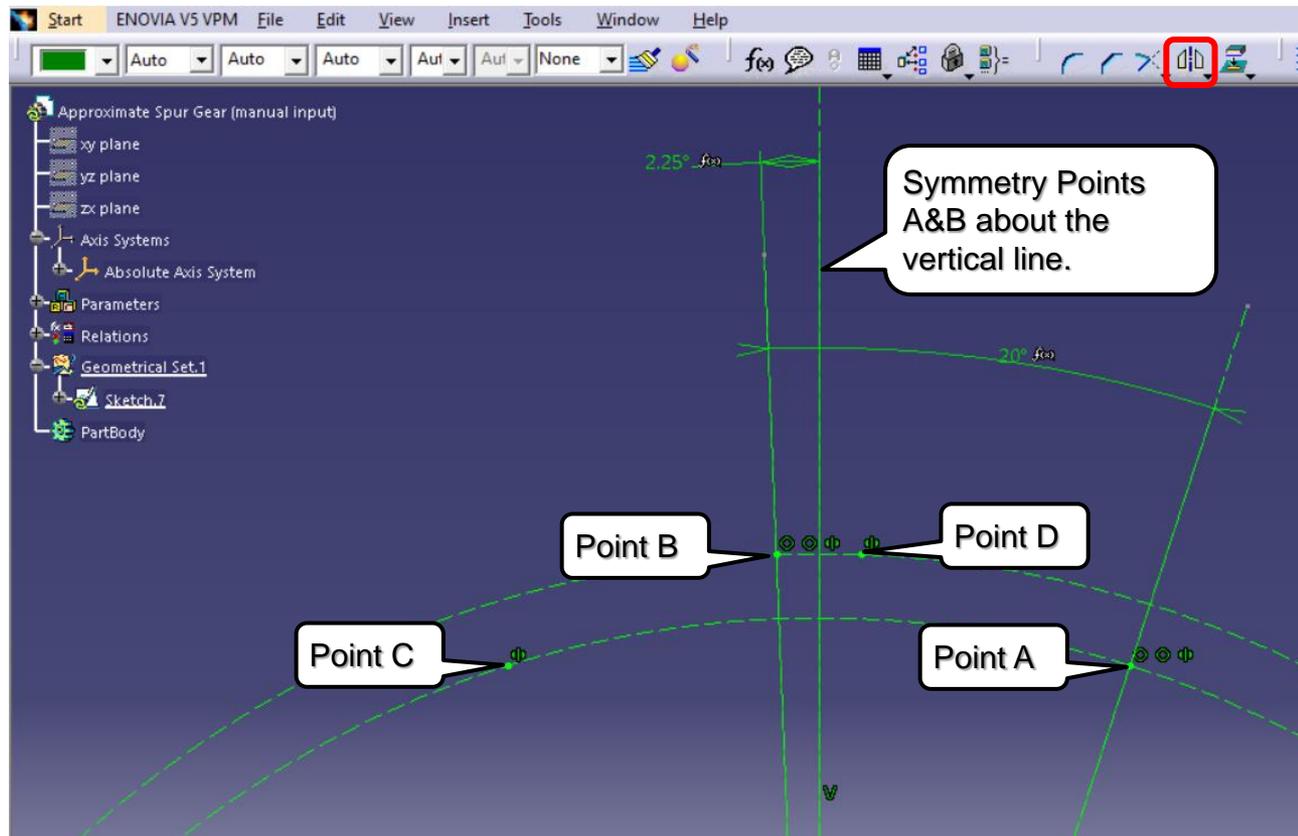


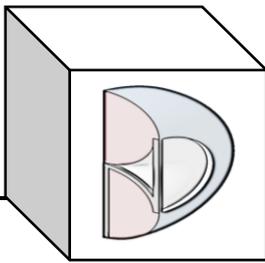


# BND TechSource



- Create symmetrical reference points C & D for the construction of the approximated tooth profile.

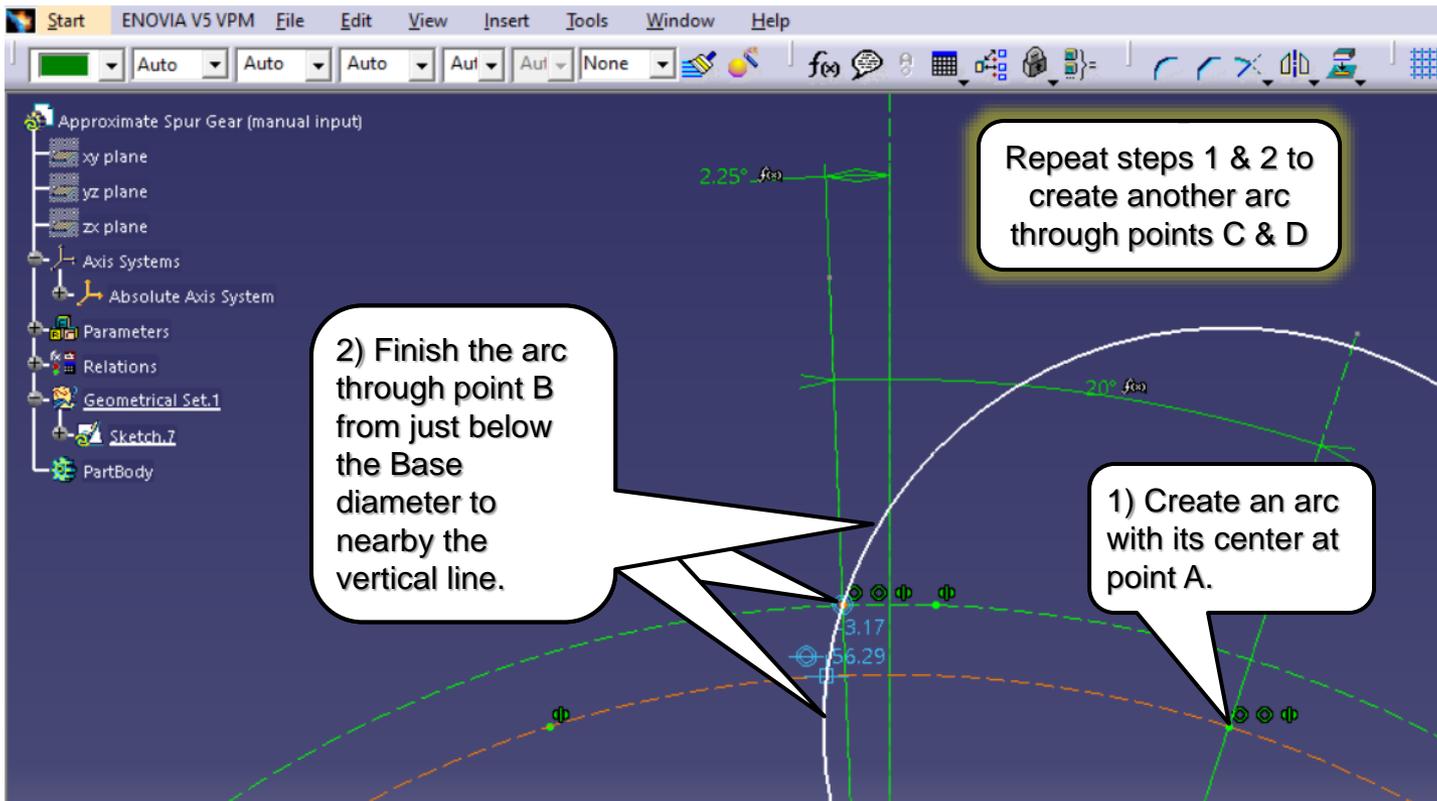


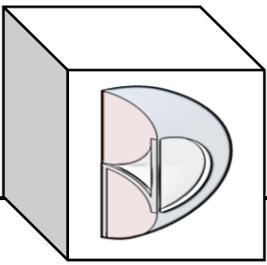


# BND TechSource



- Create the approximated (circular) tooth profile. *Actual* tooth profile would be an *involute* curve.

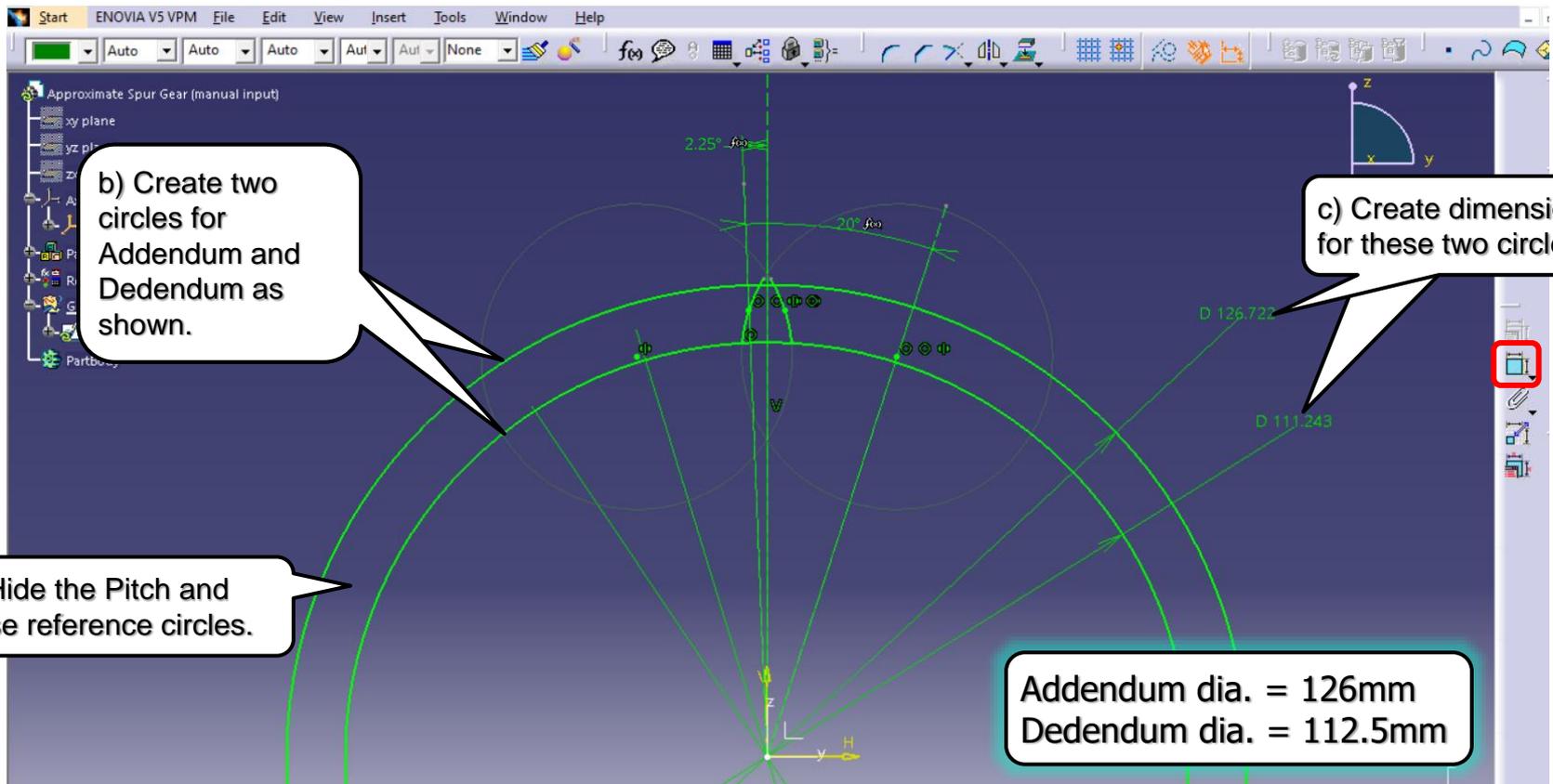


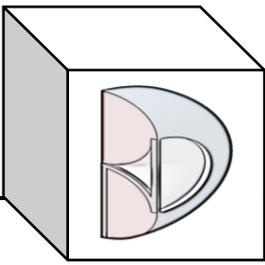


# BND TechSource



- Create the Addendum and Dedendum construction circles.

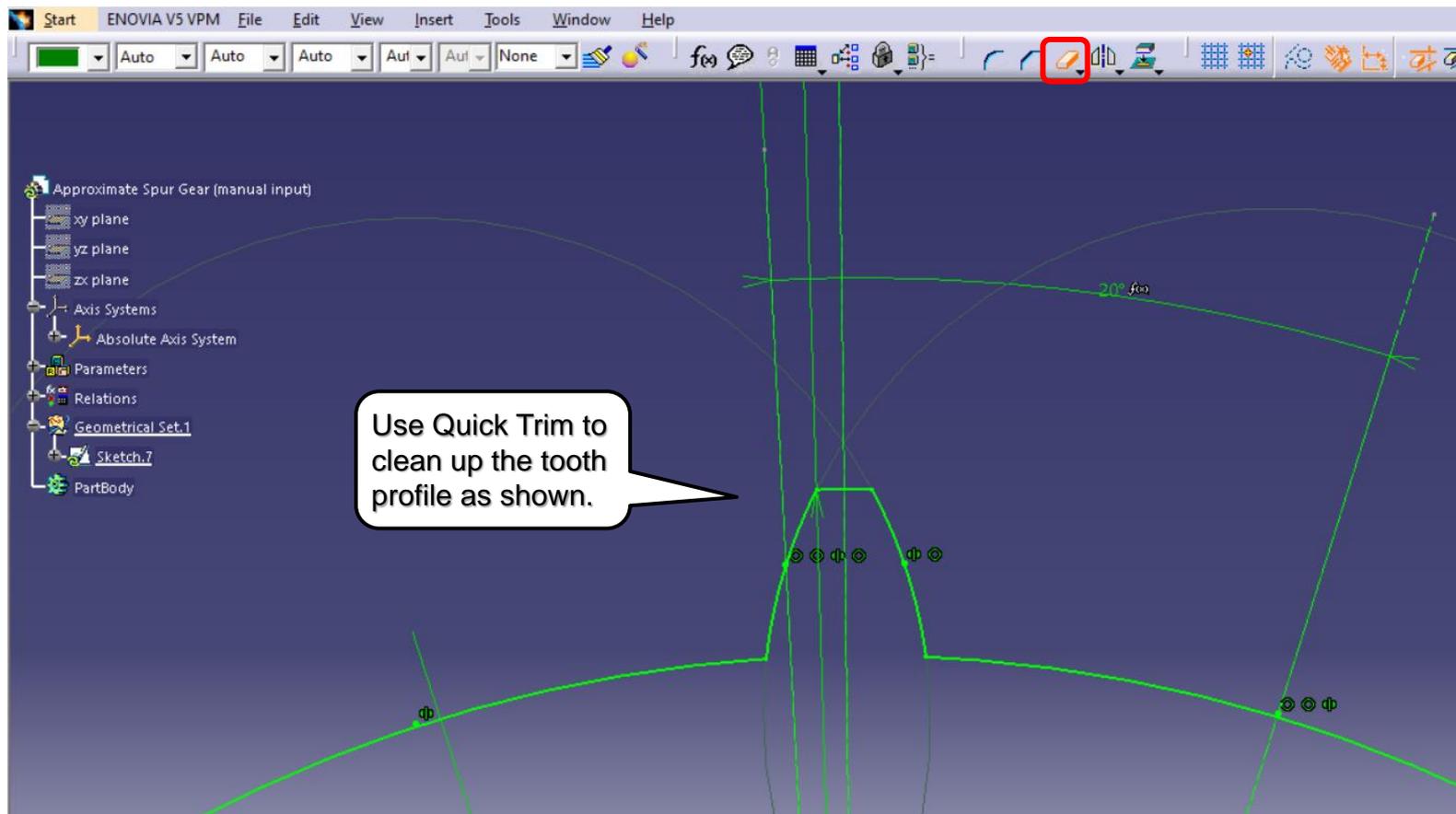


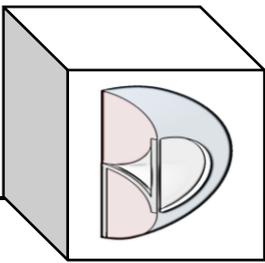


# BND TechSource



- Clean up the tooth profile.

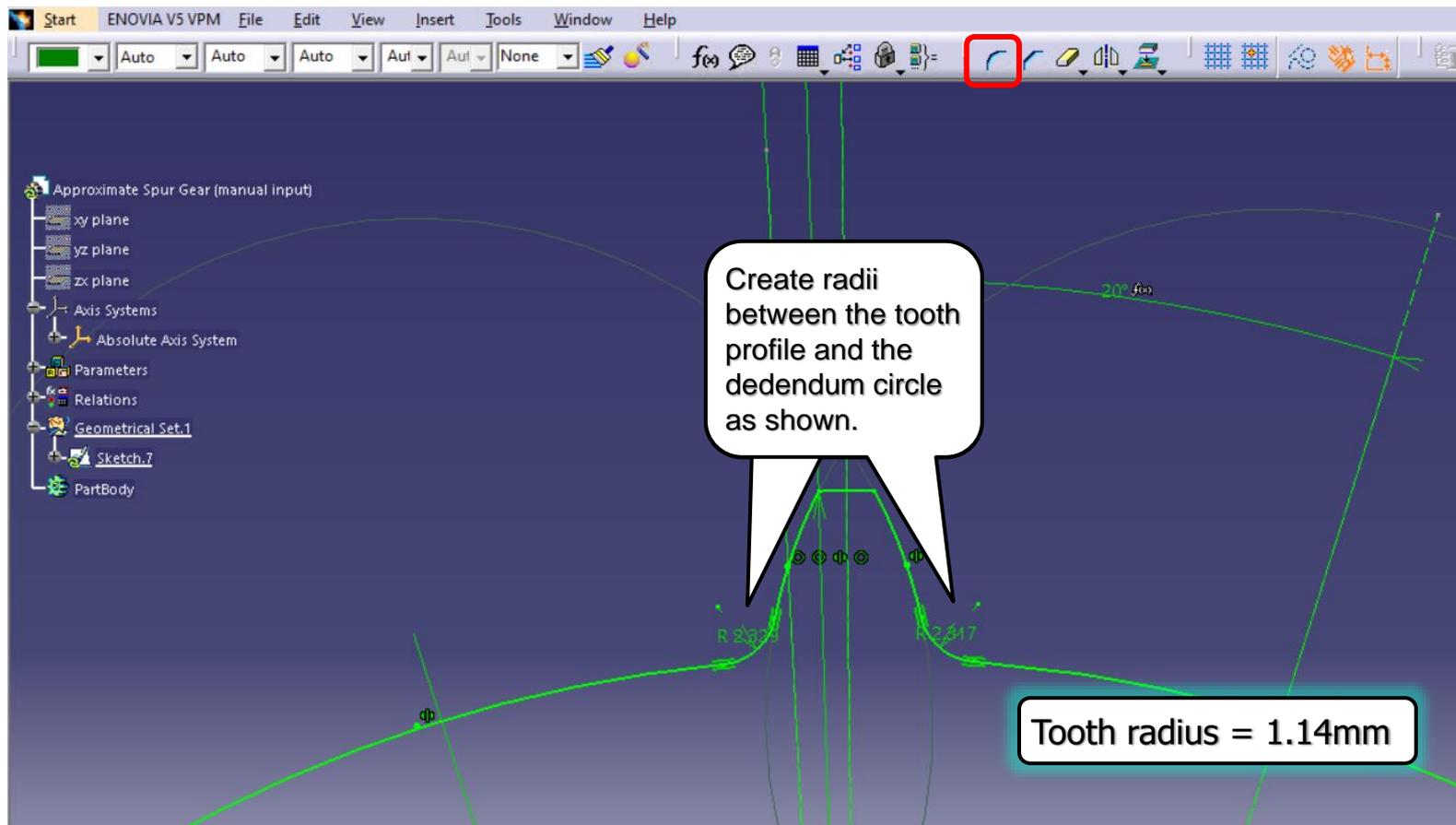


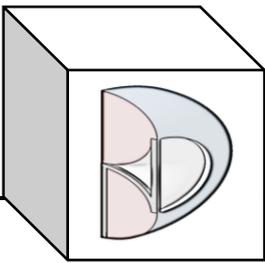


# BND TechSource



- Create the radius between the tooth profile and the dedendum circle.





# BND TechSource



- Create a solid pad using the sketch.

The screenshot shows the ENOVIA V5 VPM software interface. The main workspace displays a 3D model of a gear tooth with a sketch of a rectangular profile. A callout box labeled 'a) Insert a body.' points to the 'Body.3' entry in the left-hand tree view. Another callout box labeled 'b) Create a Pad using the sketch as the profile selection; set length to 10mm; use mirrored extent.' points to the 'Pad Definition' dialog box. The dialog box shows the 'Length' set to '10mm' and the 'Mirrored extent' checkbox checked. A third callout box labeled 'c) Pick OK' points to the 'OK' button in the dialog box.

ENOVIA V5 VPM File Edit View Insert Tools Window Help

Approximate Spur Gear (manual input)

- xy plane
- yz plane
- zx plane
- Axis Systems
  - Absolute Axis System
- Parameters
- Relations
- Geometrical Set.1
  - Sketch.7
  - Body.3
- PartBody

2.25° fco26 fco

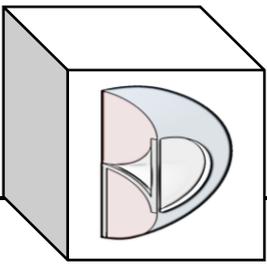
10

10mm

Sketch.7

Mirrored extent

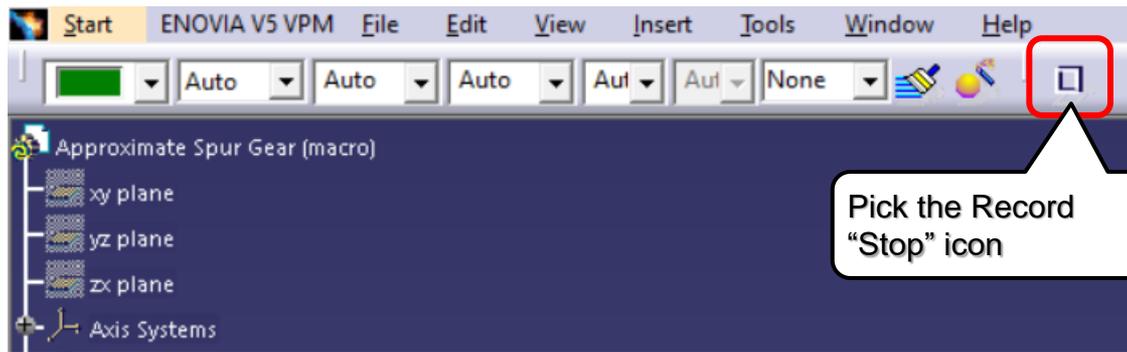
OK Cancel Preview

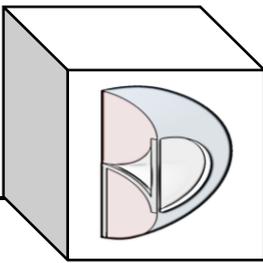


# BND TechSource



- **Stop recording the Geometry block of this Macro.**





# BND TechSource



- Check the Geometry content of this Macro.

The image shows a screenshot of the CATIA Macros Editor and the Macro Libraries dialog box. The Macros Editor window on the left displays the following VBS script:

```
language="VBSCRIPT"
Sub CATMain()
Set partDocument1 = CATIA.ActiveDocument
Set part1 = partDocument1.Part
Set hybridBodies1 = part1.HybridBodies
Set hybridBody1 = hybridBodies1.Item("Geometrical Set.1")
Set sketches1 = hybridBody1.HybridSketches
Set reference1 = part1.CreateReferenceFromName("Selection_RSUR:(Face:(Brp:(AxisSystem.1;2);None();Cf11:0));AxisSyst
Set sketch1 = sketches1.Add(reference1)

Dim arrayOfVariantOfDouble1(8)
arrayOfVariantOfDouble1(0) = 0.000000
arrayOfVariantOfDouble1(1) = 0.000000
arrayOfVariantOfDouble1(2) = 0.000000
arrayOfVariantOfDouble1(3) = 0.000000
arrayOfVariantOfDouble1(4) = 1.000000
arrayOfVariantOfDouble1(5) = 0.000000
arrayOfVariantOfDouble1(6) = 0.000000
arrayOfVariantOfDouble1(7) = 0.000000
arrayOfVariantOfDouble1(8) = 1.000000
sketch1.SetAbsoluteAxisData arrayOfVariantOfDouble1

part1.InWorkObject = sketch1

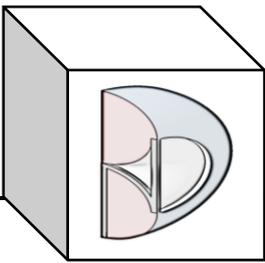
Set factory2D1 = sketch1.OpenEdition()

Set geometricElements1 = sketch1.GeometricElements
Set axis2D1 = geometricElements1.Item("AbsoluteAxis")
Set line2D1 = axis2D1.GetItem("HDirection")
line2D1.ReportName = 11
```

The Macro Libraries dialog box on the right shows a list of available macros. The 'Geometry.catvbs' macro is highlighted. Callouts provide instructions:

- a) Alt+F8 to open the macro directory
- b) Select the "Geometry" block.
- c) Pick "Edit" to work on this portion of the macro.

Name	Language
Approximate Spur Gear.catvbs	MS VBScript
<b>Geometry.catvbs</b>	MS VBScript
Input.catvbs	MS VBScript
Parameters.catvbs	MS VBScript
Relations.catvbs	MS VBScript



# BND TechSource



- Test the Geometry block of this Macro.

d) Notice the geometry has been added to the part.

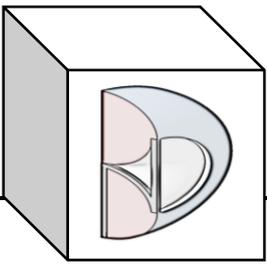
**PASS**

a) Alt+F8 to open the macro directory

b) Select the "Geometry" block.

c) Pick "Run" to test this portion of the macro.

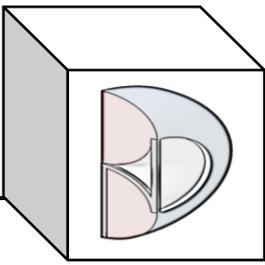
Name	Language
Approximate Spur Gear.catvbs	MS VBScript
Geometry.catvbs	MS VBScript
Input.catvbs	MS VBScript
Parameters.catvbs	MS VBScript
Relations.catvbs	MS VBScript



# BND TechSource



## **Create the 1<sup>st</sup> “Relations” Block**

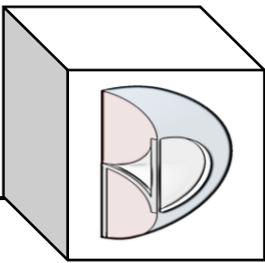


# BND TechSource

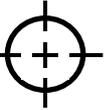


- Must **start** and **stop** recording the Macro while “inside” the Sketch to properly record the Relations.

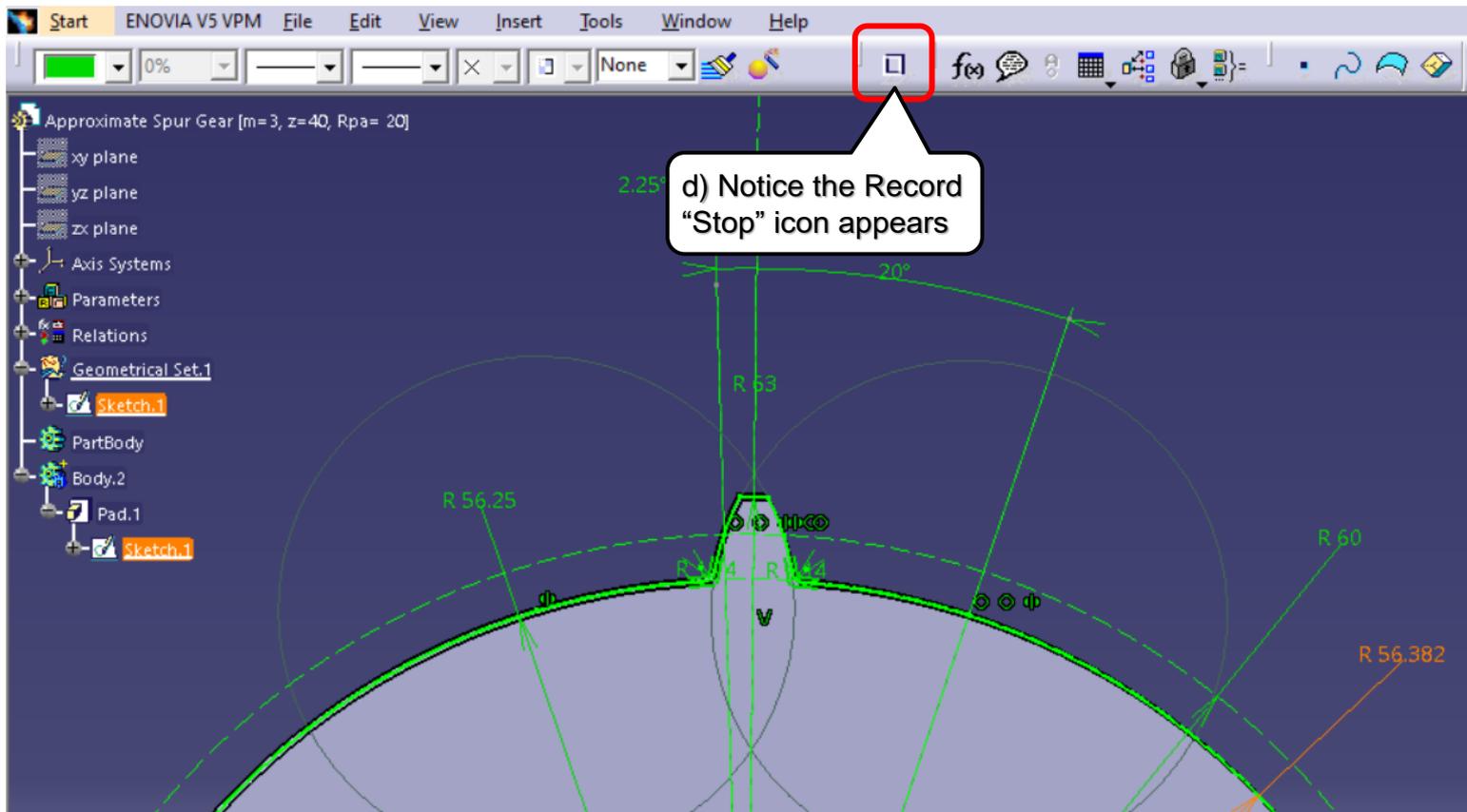
The screenshot shows the CATIA software interface. The 'Tools' menu is open, and the 'Macro' option is selected, which has opened a sub-menu with 'Start Recording...' highlighted. A callout box labeled 'a) Go to Tools + Macro + Start Recording' points to this menu path. The 'Record macro' dialog box is open, showing the 'Current macro library or document' as 'D:\~William\CATIA Tutorials\Approximate Spur Gear', the 'Language used' as 'MS VBScript', and the 'Macro name' as 'Relations\_1.catvbs'. A callout box labeled 'b) Rename the Macro "Relations\_1"' points to the macro name field. At the bottom right of the dialog box, the 'Start' button is highlighted, with a callout box labeled 'c) Pick "Start"' pointing to it.

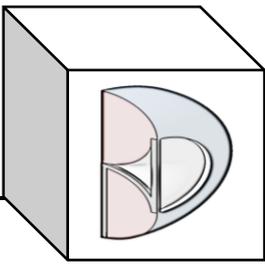


# BND TechSource



- Must **start** and **stop** recording the Macro while “inside” the Sketch to properly record the Relations.





# BND TechSource



- Apply the formula (Pitch diameter: Pd) to the pitch circle.

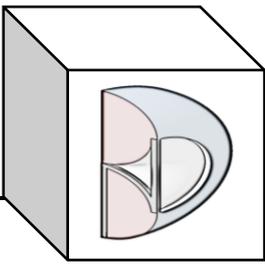
The screenshot shows the ENOVIA V5 VPM software interface. The main workspace displays a 3D model of a gear with a pitch circle highlighted in green. A dimension line is drawn across the pitch circle, labeled 'D 121.734'. A context menu is open over the dimension line, and a sub-menu is open over the 'Radius.54 object'.

**a) Right-click the dimension**

**b) Mouse-over the object and pick edit formula**

The context menu options are:

- Center graph
- Reframe On
- Hide/Show space
- Properties Alt+Enter
- Other Selection...
- Parents/Children...
- Cut Ctrl+X
- Copy Ctrl+C
- Paste Ctrl+V
- Paste Special...
- Delete Del
- Radius.54 object
  - Definition...
  - Edit Formula**
  - Lock Parameter
  - Edit Equivalent Dimensions
  - Rename Parameter
  - Deactivate
  - Display with Leader
  - Name Display



# BND TechSource



- Apply the formula (Pitch diameter: Pd) to the pitch circle.

b) Divide by 2 because CATIA sees Diameters as Radius objects

a) Double pick the formula Pitch diameter: Pd

c) Pick OK

Geometrical Set.1\Sketch.7\Radius,54\Radius

'Pitch diameter: Pd' / 2

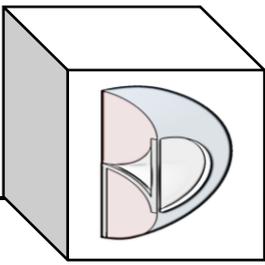
Dictionary	Members of Parameters	Members of Length
Parameters	All	'Absolute Axis System\Origin\Y'
Design Table	Renamed parameters	'Absolute Axis System\Origin\Z'
Operators	Boolean	'Module: m'
Pointer on value function	Length	'Pitch diameter: Pd'
Point Constructors	CstAttr_Mode	'Base diameter: Bd'
Law	Real	'Addendum diameter: Ad'
Operations Constructors	Angle	'Dedendum diameter: Dd'
	String	

Pitch diameter: Pd 120mm

OK Cancel

D 124.724

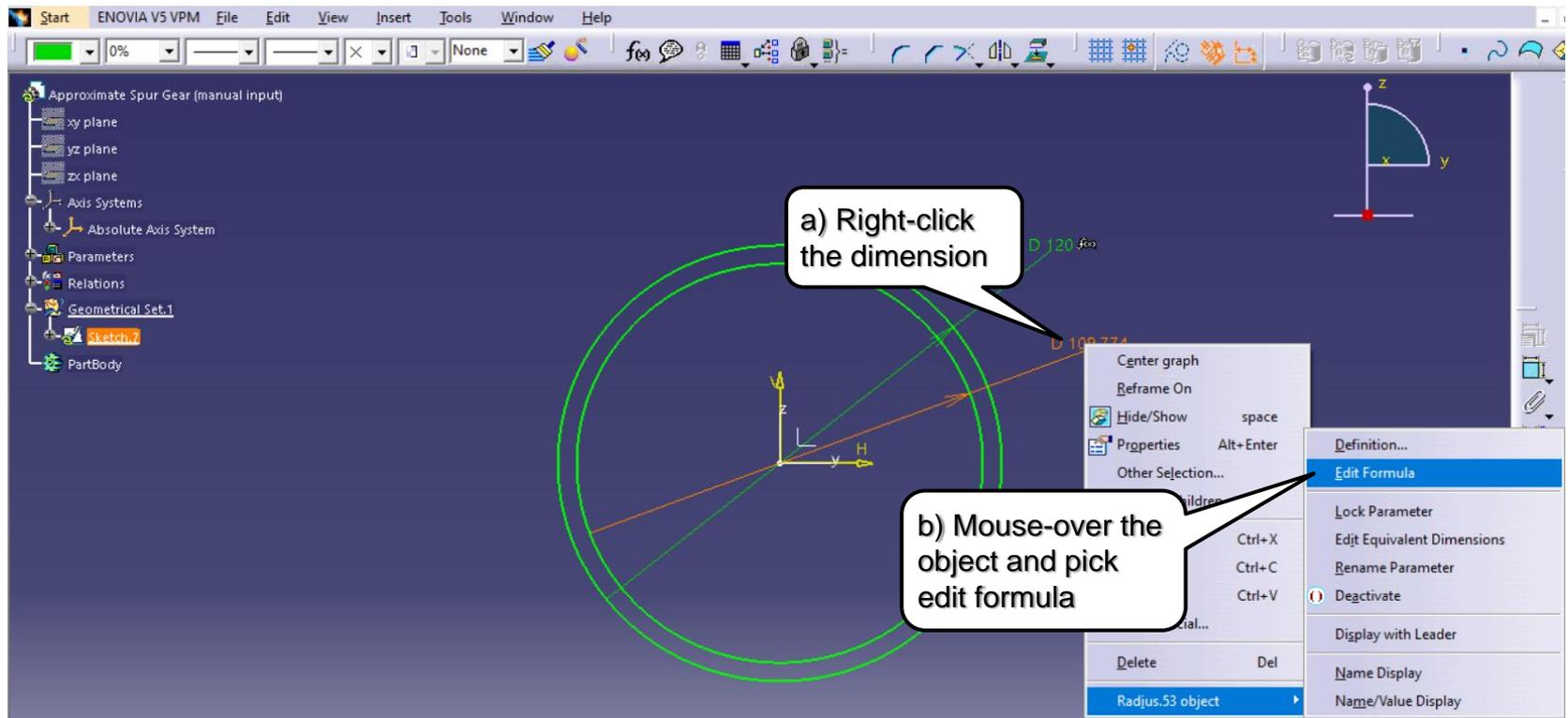
D 109.774

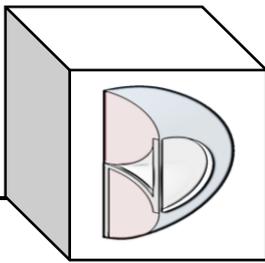


# BND TechSource



- Apply the formula (Base diameter:  $B_d$ ) to the base circle.





# BND TechSource



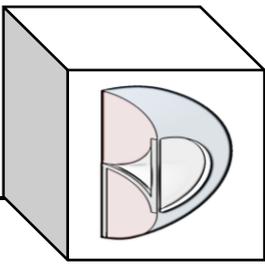
- Apply the formula (Base diameter: Bd) to the base circle.

b) Divide by 2 because CATIA sees Diameters as Radius objects

a) Double pick the formula Base diameter: Bd

c) Pick OK

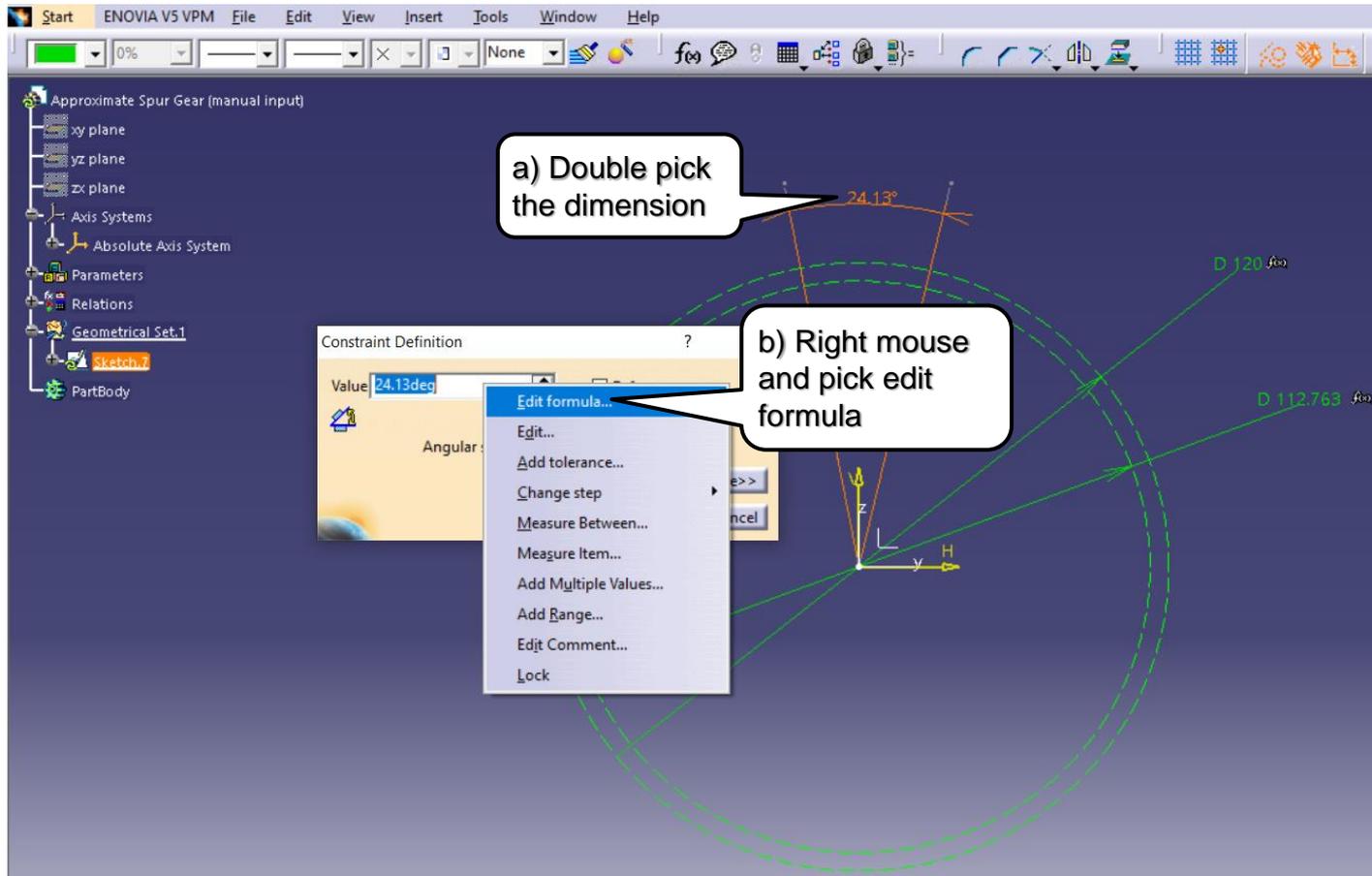
Dictionary	Members of Parameters	Members of Length
Parameters	All	'Absolute Axis System' Origin\Y'
Design Table	Renamed parameters	'Absolute Axis System' Origin\Z'
Operators	Boolean	'Module: m'
Pointer on value function	Length	'Pitch diameter: Pd'
Point Constructors	CstAttr_Mode	'Base diameter: Bd'
Law	Real	'Addendum diameter: Ad'
Operations Constructors	Angle	'Dedendum diameter: Dd'
	String	

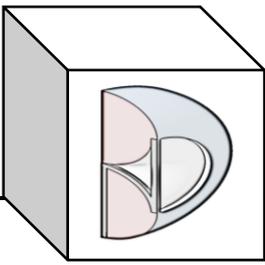


# BND TechSource



- Apply Ref Pressure Angle: Rpa.





# BND TechSource

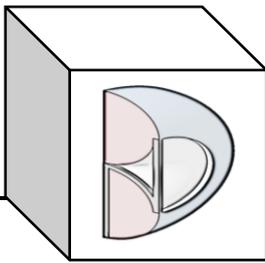


- Apply Ref Pressure Angle: Rpa.

The screenshot shows the CAD software interface with the Formula Editor open. The main window displays a sketch of a gear with a green angle dimension of 24.13°. The Formula Editor window is titled "Formula Editor: 'Geometrical Set.1\Ske..." and shows the formula for the "Ref Pressure Angle: Rpa" parameter. The formula is set to "20deg". The "Members of Parameters" list includes "Angle", "Real", and "String". The "Members of Angle" list includes "Geometrical Set.1\Ske... Angle.56\Angle", "Ref Pressure Angle: Rpa", and "Symmetry angle: s". A callout box labeled "a) Double pick the formula Ref Pressure Angle: Rpa" points to the "Ref Pressure Angle: Rpa" entry in the "Members of Angle" list. Another callout box labeled "b) Pick OK" points to the "OK" button at the bottom right of the Formula Editor window.

a) Double pick the formula  
Ref Pressure Angle: Rpa

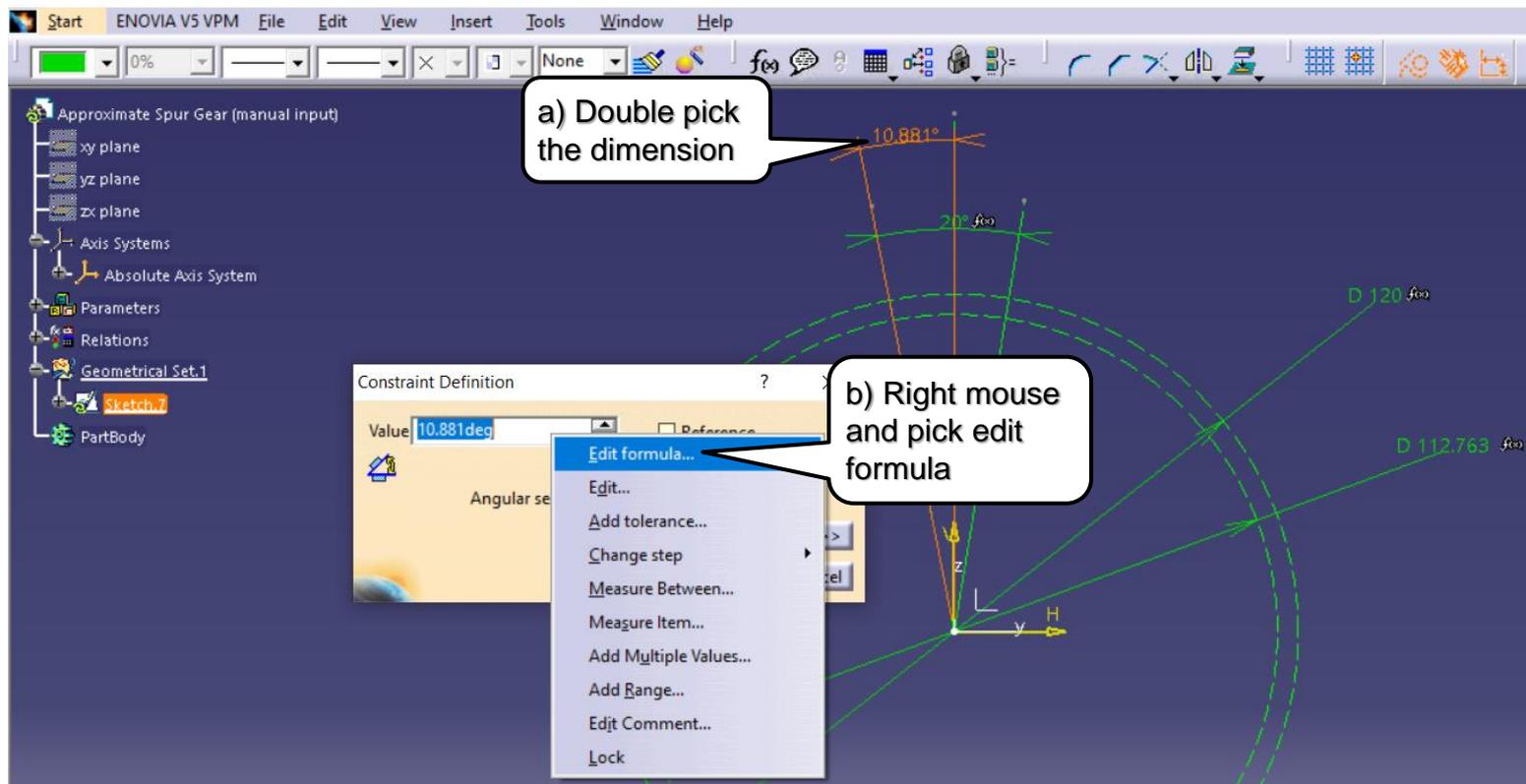
b) Pick OK

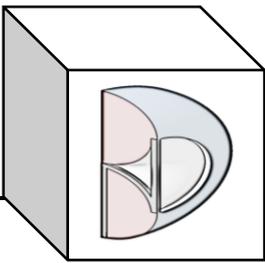


# BND TechSource



- Apply Symmetry angle: s





# BND TechSource



- Apply Symmetry angle: s

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Approximate Spur Gear (manual input)

- xy plane
- yz plane
- zx plane
- Axis Systems
- Absolute Axis System
- Parameters
- Relations
- Geometrical Set.1
- Sketch.7
- PartBody

Formula Editor: "Geometrical Set.1\Sketch.7\Angle.59\Angle"

Geometrical Set.1\Sketch.7\Angle.59\Angle =

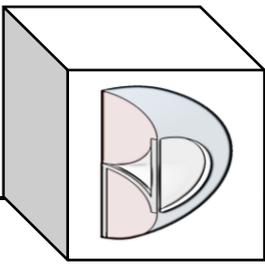
"Symmetry angle: s"

Dictionary	Members of Parameters	Member Angle
Parameters	All	"Geometrical Set.1\Sketch.7\Angle.56\Angle"
Design Table	Renamed parameters	"Geometrical Set.1\Sketch.7\Angle.59\Angle"
Operators	Boolean	"Ref Pressure Angle: Rpa"
Pointer on value function	Length	"Symmetry angle: s"
Point Constructors	CstAttr_Mode	
Law	Angle	
Operations Constructors	Real	
	String	

Symmetry angle: s 2.25deg

a) Double pick the formula  
Symmetry angle: s

b) Pick OK

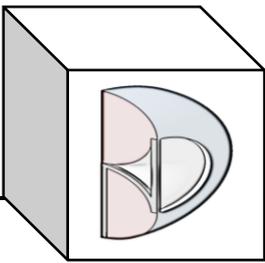


# BND TechSource

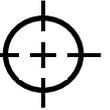


- Apply the formula (Addendum diameter:  $A_d$ ) to the addendum circle.

The screenshot displays the ENOVIA V5 VPM software interface. The main workspace shows a gear sketch with several dimensions and geometric constraints. A dimension of  $2.25^\circ$  is shown, and another dimension is labeled  $20^\circ$ . A dimension value of  $D 126.722$  is visible. A context menu is open over the dimension  $D 126.722$ , showing options such as "Center graph", "Reframe On", "Hide/Show", "Properties", "Other Selection...", "Parents/Children...", "Cut", "Copy", "Paste", "Paste Special...", "Delete", "Definition...", "Edit Formula", "Lock Parameter", "Edit Equivalent Dimensions", "Rename Parameter", and "Deactivate". A callout box points to the "Edit Formula" option with the text "b) Mouse-over the object and pick edit formula". Another callout box points to the dimension  $D 126.722$  with the text "a) Right-click the dimension". The software interface includes a menu bar (Start, ENOVIA V5 VPM, File, Edit, View, Insert, Tools, Window, Help), a toolbar, and a tree view on the left showing the model structure (Approximate Spur Gear, xy plane, yz plane, zx plane, Axis Systems, Absolute Axis System, Parameters, Relations, Geometrical Set.1, Sketch.7, PartBody).



# BND TechSource



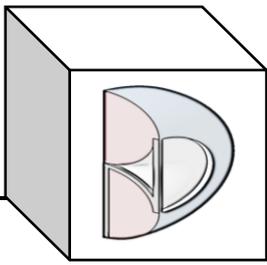
- Apply the formula (Addendum diameter: Ad) to the addendum circle.

b) Divide by 2 because CATIA sees Diameters as Radius objects

a) Double pick the formula Addendum diameter: Ad

c) Pick OK

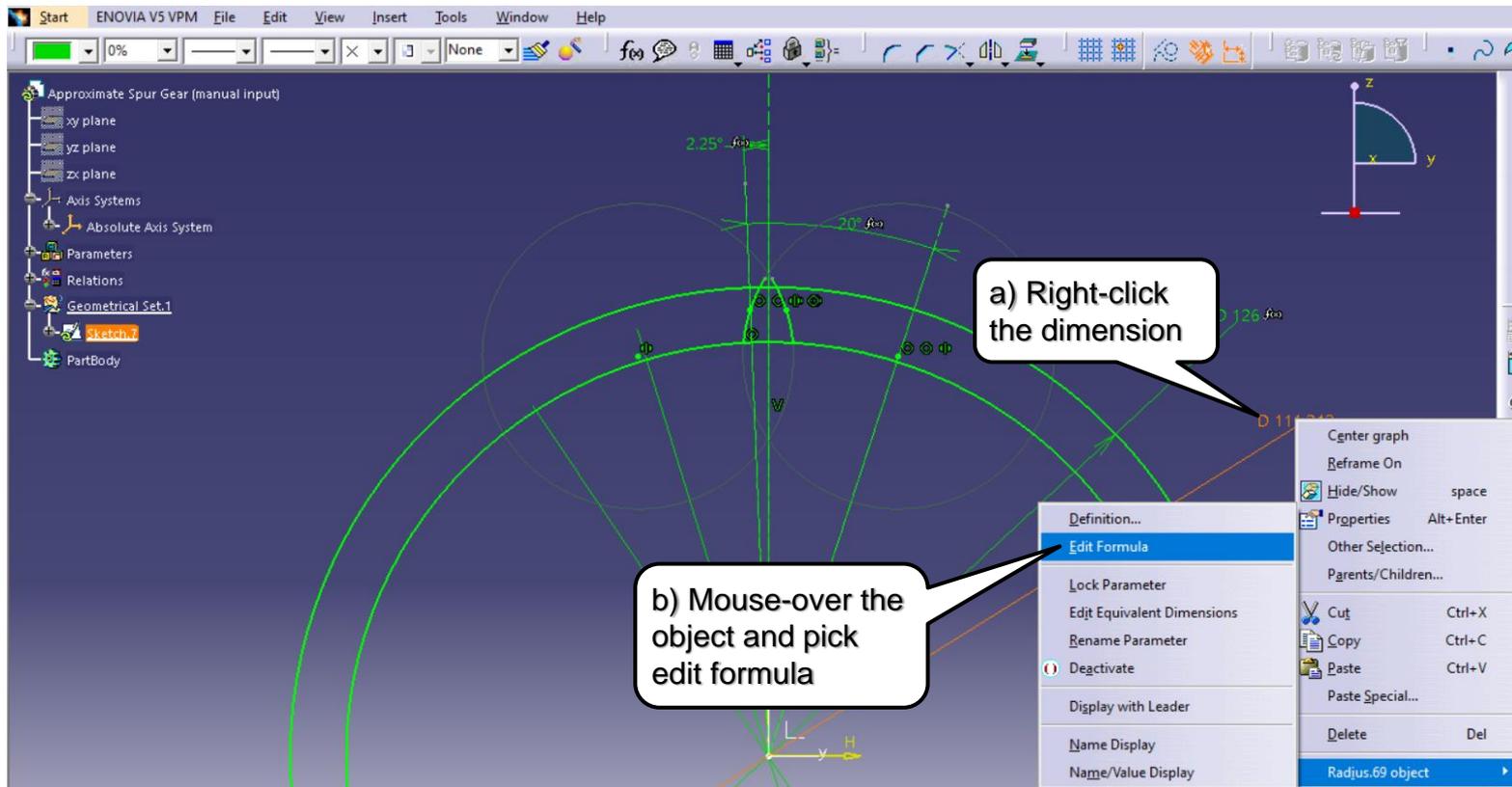
Dictionary	Members of Parameters	Members of Let
Parameters	All	'Module: m'
Design Table	Renamed parameters	'Pitch diameter: P'
Operators	Boolean	'Base diameter: Bd'
Pointer on value function	Length	'Addendum diameter: Ad'
Point Constructors	CstAttr_Mode	'Dedendum diameter: Dd'
Law	Angle	'tooth radius at dedendum circle: tr'
Operations Constructors	Real	
	String	

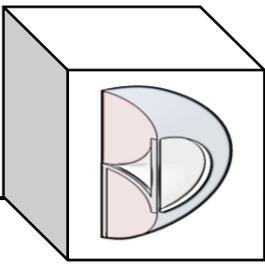


# BND TechSource



- Apply the formula (Dedendum diameter:  $D_d$ ) to the dedendum circle.





# BND TechSource



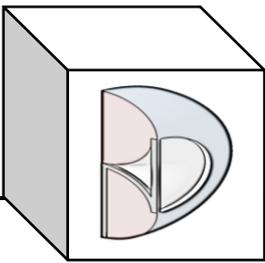
- Apply the formula (Dedendum diameter: Dd) to the dedendum circle.

b) Divide by 2 because CATIA sees Diameters as Radius objects

a) Double pick the formula Dedendum circle radius: Rd

c) Pick OK

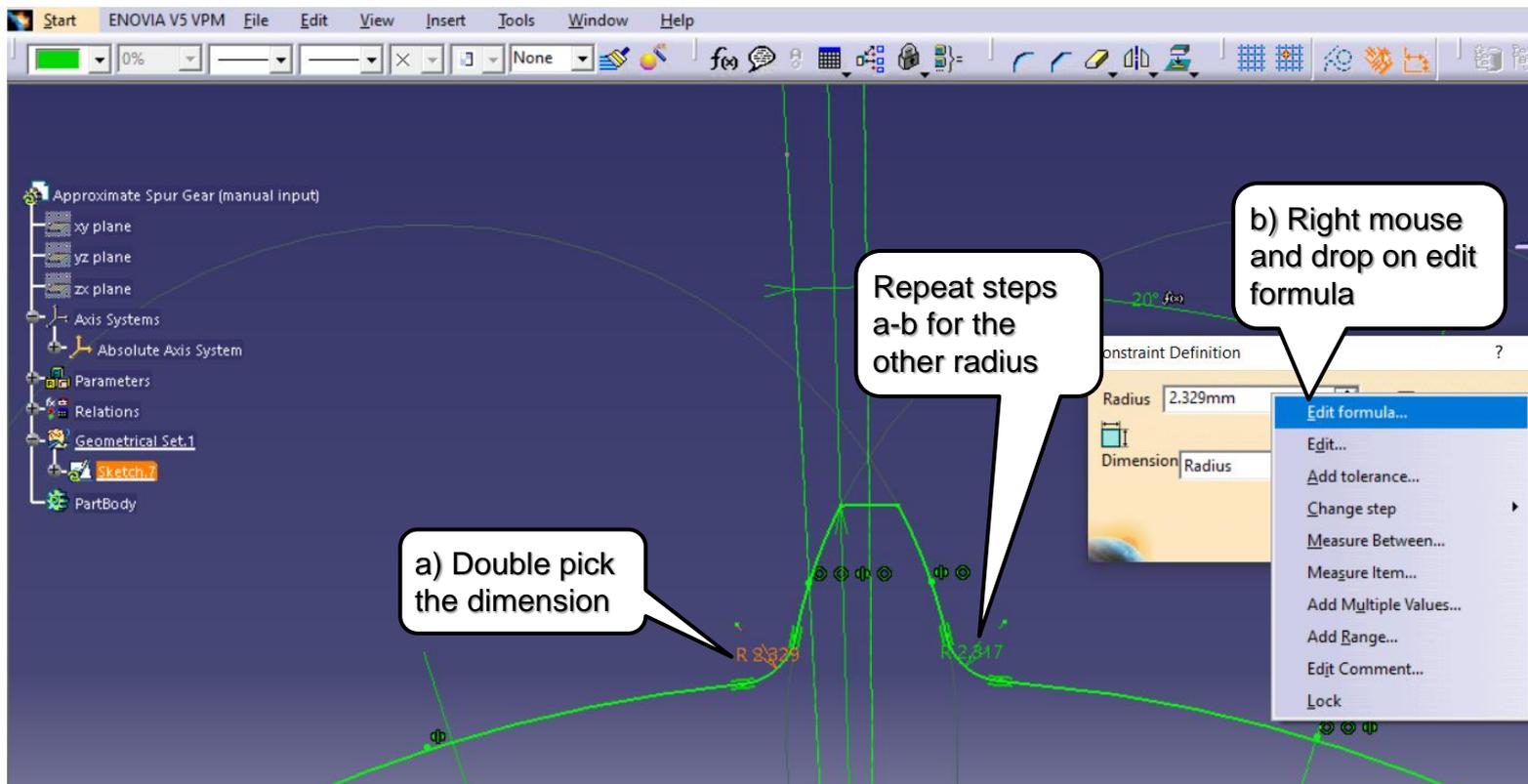
Dictionary	Members of Parameters	Members of Length
Parameters	All	'Module: m'
Design Table	Renamed parameters	'Pitch diameter: Pd'
Operators	Boolean	'Base diameter: Bd'
Pointer on value function	Length	'Addendum diameter: Ad'
Point Constructors	CstAttr_Mode	'Dedendum diameter: Dd'
Law	Angle	'tooth radius at dedendum circle: tr'
Operations Constructors	Real	
	String	

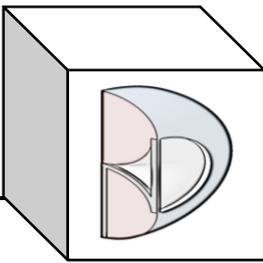


# BND TechSource



- Apply tooth radius at dedendum circle: tr to both radii.





# BND TechSource



- Apply tooth radius at dedendum circle: tr to both radii.

Formula Editor: 'Geometrical Set.1\Sketch.7\Radius.80\Radius'

Geometrical Set.1\Sketch.7\Radius.80\Radius

'tooth radius at dedendum circle: tr'

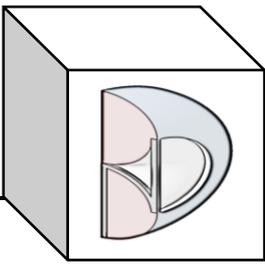
Dictionary	Members of Parameters	Members of Length
Parameters	All	'Module: m'
Design Table	Renamed parameters	'Pitch diameter: Pd'
Operators	Boolean	'Base diameter: Bd'
Pointer on value function	Length	'Addendum diameter: Ad'
Point Constructors	CstAttr_Mode	'Dedendum diameter: Dd'
Law	Angle	'tooth radius at dedendum circle: tr'
Operations Constructors	Real	
	String	

tooth radius at dedendum circle: tr 1.14mm

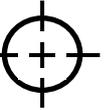
OK Cancel

a) Double pick the formula tooth radius at dedendum circle: tr

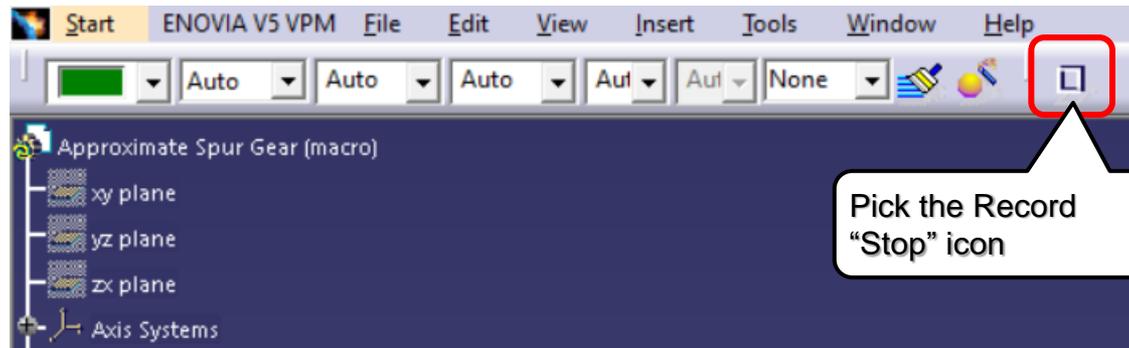
b) Pick OK

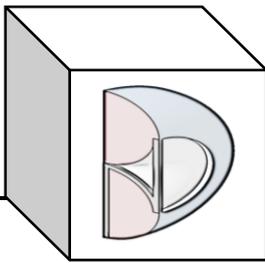


# BND TechSource



- **Stop recording** the 1<sup>st</sup> Relations block of this Macro (while still “inside” the sketch).





# BND TechSource



- Check the content of the Relations\_1 Macro.

Macros Editor - [D:\~William\CATIA Tutorials\Approximate Spur Gear\Relations\_1.catvbs]

```
Set constraint1 = constraints1.Item("Radius.1")
Set length1 = constraint1.Dimension
Set formula1 = relations1.CreateFormula("Formula.8", "", length1, "Pitch diameter: Pd / 2")
formula1.Rename "Formula.8"
Set relations2 = part1.Relations
Set constraint2 = constraints1.Item("Radius.2")
Set length2 = constraint2.Dimension
Set formula2 = relations2.CreateFormula("Formula.9", "", length2, "Base diameter: Bd / 2")
formula2.Rename "Formula.9"
Set relations3 = part1.Relations
Set constraint3 = constraints1.Item("Angle.3")
Set angle1 = constraint3.Dimension
Set formula3 = relations3.CreateFormula("Formula.10", "", angle1, "Ref Pressure Angle: Rpa ")
formula3.Rename "Formula.10"
Set relations4 = part1.Relations
Set constraint4 = constraints1.Item("Angle.5")
Set angle2 = constraint4.Dimension
Set formula4 = relations4.CreateFormula("Formula.11", "", angle2, "Symmetry angle: s ")
formula4.Rename "Formula.11"
Set relations5 = part1.Relations
Set constraint5 = constraints1.Item("Radius.15")
```

a) Alt+F8 to open the macro directory

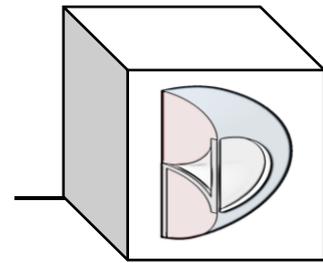
b) Select the "Relations\_1" block.

c) Pick "Edit" to work on this portion of the macro.

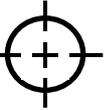
File name	Language
Approximate Spur Gear.catvbs	MS VBScript
Geometric Relations.catvbs	MS VBScript
Input Parameters.catvbs	MS VBScript
Parameters.catvbs	MS VBScript
Relations.catvbs	MS VBScript
Relations_1.catvbs	MS VBScript

Macro libraries...

Run Edit... Create... Rename... Delete Select... Obfuscate... Close



# BND TechSource



- Test the block of the Relations\_1 Macro.

## Before running the macro

Approximate Spur Gear [m=3, z=40, Rpa= 20]

- xy plane
- yz plane
- zx plane
- Axis Systems
- Parameters
- Relations
  - Formula.1: Symmetry angle:  $s = 90 / \text{Number of teeth} \cdot z \cdot 1 \text{ deg}$
  - Formula.2: Pitch diameter:  $Pd = \text{Number of teeth} \cdot z \cdot \text{Module} \cdot m$
  - Formula.3: Base diameter:  $Bd = \text{Pitch diameter} \cdot Pd$
  - Formula.4: Addendum diameter:  $Ad = \text{Pitch diameter} \cdot Pd$
  - Formula.5: Dedendum diameter:  $Dd = \text{Pitch diameter} \cdot Pd$
  - Formula.6: tooth radius at dedendum circle:  $tr = 0.38 \cdot \text{Module} \cdot m$

b) Select the "Relations\_1" block.

Language	Run
Approximate Spur Gear.catvbs	MS VBScript
Geometry.catvbs	MS VBScript
Input.catvbs	MS VBScript
Parameters.catvbs	MS VBScript
Relations.catvbs	MS VBScript
Relations_1.catvbs	MS VBScript

c) Pick "Run" to test this portion of the macro.

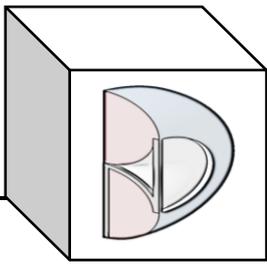
## After running the macro

Approximate Spur Gear [m=3, z=40, Rpa= 20]

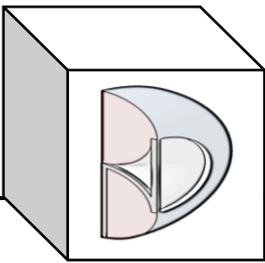
- xy plane
- yz plane
- zx plane
- Axis Systems
- Parameters
- Relations
  - Formula.1: Symmetry angle:  $s = 90 / \text{Number of teeth} \cdot z \cdot 1 \text{ deg}$
  - Formula.2: Pitch diameter:  $Pd = \text{Number of teeth} \cdot z \cdot \text{Module} \cdot m$
  - Formula.3: Base diameter:  $Bd = \text{Pitch diameter} \cdot Pd \cdot \cos(\text{Ref Pressure Angle})$
  - Formula.4: Addendum diameter:  $Ad = \text{Pitch diameter} \cdot Pd + (2 \cdot \text{Module} \cdot m)$
  - Formula.5: Dedendum diameter:  $Dd = \text{Pitch diameter} \cdot Pd - (2.5 \cdot \text{Module} \cdot m)$
  - Formula.6: tooth radius at dedendum circle:  $tr = 0.38 \cdot \text{Module} \cdot m$
  - Formula.7: tooth radius at addendum circle:  $tr = 0.38 \cdot \text{Module} \cdot m$
  - Formula.8: Geometrical Set.1\Sketch.1\Radius.1\Radius = Pitch diameter:  $Pd \cdot /2$
  - Formula.9: Geometrical Set.1\Sketch.1\Radius.2\Radius = Base diameter:  $Bd \cdot /2$
  - Formula.10: Geometrical Set.1\Sketch.1\Angle.3\Angle = Ref Pressure Angle:  $Rpa$
  - Formula.11: Geometrical Set.1\Sketch.1\Angle.5\Angle = Symmetry angle:  $s$
  - Formula.12: Geometrical Set.1\Sketch.1\Radius.15\Radius = Addendum diameter:  $Ad \cdot /2$
  - Formula.13: Geometrical Set.1\Sketch.1\Radius.14\Radius = Dedendum diameter:  $Dd \cdot /2$
  - Formula.14: Geometrical Set.1\Sketch.1\Radius.18\Radius = tooth radius at dedendum circle:  $tr$
  - Formula.15: Geometrical Set.1\Sketch.1\Radius.21\Radius = tooth radius at dedendum circle:  $tr$

PASS

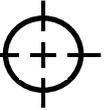
d) Notice the new relations have been added to the part.



## **Create the 2<sup>nd</sup> “Relations” Block**



# BND TechSource



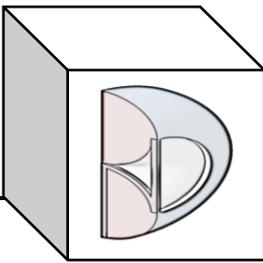
- Start recording the 2nd “Relations” Block.

The screenshot shows the CATIA software interface. The 'Tools' menu is open, and the 'Macro' option is selected. The 'Record macro' dialog box is displayed, showing the current macro library or document path, the language used (MS VBScript), and the macro name 'Relations\_2'. The 'Start' button is highlighted.

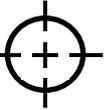
a) Go to Tools + Macro + Start Recording

b) Rename the Macro “Relations\_2”

c) Pick “Start”



# BND TechSource

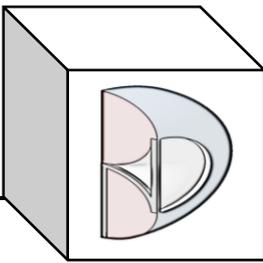


- Create the pattern for the number of teeth.

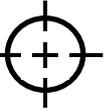
The screenshot displays the ENOVIA V5 VPM software interface. The main window shows a 3D model of a spur gear. The left-hand side contains a tree view with the following items:

- Approximate Spur Gear [m=3, z=40, Rpa= 20]
- xy plane
- yz plane
- zx plane
- Axis Systems
- Parameters
- Relations
  - fixed Formula.1: Symmetry angle:  $s = 90 / \text{Number of teeth} \cdot z \cdot 1 \text{ deg}$
  - fixed Formula.2: Pitch diameter:  $Pd = \text{Number of teeth} \cdot z \cdot \text{Module} \cdot m$
  - fixed Formula.3: Base diameter:  $Bd = \text{Pitch diameter} \cdot Pd \cdot \cos(\text{Ref Pressure Angle} \cdot Rpa)$
  - fixed Formula.4: Addendum diameter:  $Ad = \text{Pitch diameter} \cdot Pd + (2 \cdot \text{Module} \cdot m)$
  - fixed Formula.5: Dedendum diameter:  $Dd = \text{Pitch diameter} \cdot Pd - (2.5 \cdot \text{Module} \cdot m)$
  - fixed Formula.6: tooth radius at dedendum circle:  $tr = 0.38 \cdot \text{Module} \cdot m$
  - fixed Formula.8: Geometrical Set.1\Sketch.1\Radius.1\Radius:  $= \text{Pitch diameter} \cdot Pd / 2$
  - fixed Formula.9: Geometrical Set.1\Sketch.1\Radius.2\Radius:  $= \text{Base diameter} \cdot Bd / 2$
  - fixed Formula.10: Geometrical Set.1\Sketch.1\Angle.3\Angle:  $= \text{Ref Pressure Angle} \cdot Rpa$
  - fixed Formula.11: Geometrical Set.1\Sketch.1\Angle.5\Angle:  $= \text{Symmetry angle} \cdot s$
  - fixed Formula.12: Geometrical Set.1\Sketch.1\Radius.15\Radius:  $= \text{Addendum diameter} \cdot Ad / 2$
  - fixed Formula.13: Geometrical Set.1\Sketch.1\Radius.14\Radius:  $= \text{Dedendum diameter} \cdot Dd / 2$
  - fixed Formula.14: Geometrical Set.1\Sketch.1\Radius.18\Radius:  $= \text{tooth radius at dedendum circle} \cdot tr$
  - fixed Formula.15: Geometrical Set.1\Sketch.1\Radius.21\Radius:  $= \text{tooth radius at dedendum circle} \cdot tr$
- Geometrical Set.1
- PartBody
- Body.2

The 'Circular Pattern Definition' dialog box is open, showing the 'Crown Definition' tab. The 'Instance(s) & angular spacing' section is active, with 'Instance(s)' set to 4 and 'Angular spacing' set to 45deg. A context menu is open over the 'Instance(s)' field, with the 'Edit formula...' option selected. A speech bubble points to this option with the text: 'a) Right mouse "Instance" and drop on edit formula'. The 'Object to Pattern' section is set to 'Current Solid'. The 'Keep specifications' checkbox is unchecked. The 'OK', 'Cancel', and 'Preview' buttons are visible at the bottom of the dialog.



# BND TechSource



- Create the pattern for the number of teeth.

Formula Editor : Body.2\CircPattern.1

Body.2\CircPattern.1\AngularNumber

Number of teeth: z

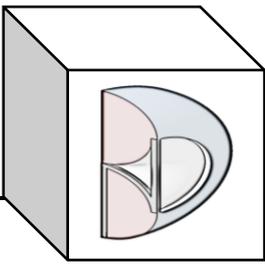
Dictionary	Members of Parameters	Members of R
Parameters	All	'Absolute Axis System\ZAxis\Z'
Design Table	Renamed parameters	'Absolute Axis System\ZAxis\X'
Operators	Boolean	'Absolute Axis System\ZAxis\Y'
Pointer on value function	Length	'Absolute Axis System\YAxis\Z'
Point Constructors	CstAttr_Mode	'Absolute Axis System\ZAxis\X'
Law	Real	'Absolute Axis System\ZAxis\Y'
Operations Constructors	Integer	'Absolute Axis System\ZAxis\Z'

Number of teeth: z 40

OK Cancel

b) Double pick the formula  
Number of teeth: z

c) Pick OK



# BND TechSource



- Create the pattern for the number of teeth.

b) See next slide

a) Right mouse "Angular Spacing" and drop on edit formula

d) Reference element; pick the X axis.

Definition

Instance | Crown Definition

Instance(s) & angular spacing

Angular spacing : 14.40

Total angle : 345.6

Reference Direction

Reference element: No

Reverse

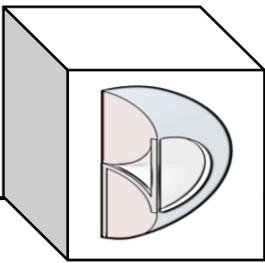
Object to Pattern

Object: Current Solid

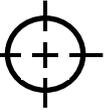
Keep specifications

More>>

OK Cancel Preview



# BND TechSource



- Create the pattern for the number of teeth.

Approximate Spur Gear (manual input)

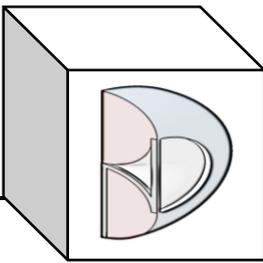
- xy plane
- yz plane
- zx plane
- Axis Systems
- Absolute Axis System
- Parameters
- Relations
- Geometrical Set.1
- Sketch.7
- Body.3
- PartBody

Formulas: Body.2\CircPattern.1\AngularSpacing

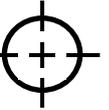
Body.2\CircPattern.1\AngularSpacing = 360 deg/Number of teeth: z

Dictionary	Members of Parameters	Members of All
Parameters	All	'Geometrical Set.1\Sketch.1\Activity'
Design Table	Renamed parameters	'Geometrical Set.1\Sketch.1\AbsoluteAxis\
Operators	Boolean	'Geometrical Set.1\Sketch.1\Radius.1\Radi
Pointer on value function	Length	'Geometrical Set.1\Sketch.1\Radius.1\Mod
Point Constructors	CstAttr_Mode	'Geometrical Set.1\Sketch.1\Radius.1\Acti
Law	Angle	'Geometrical Set.1\Sketch.1\Radius.2\Radi
Operations Constructors	Real	'Geometrical Set.1\Sketch.1\Radius.2\Mod
	Integer	

b) Pick OK



# BND TechSource

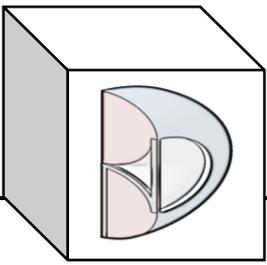


- Update the part.

The screenshot displays the ENOVIA V5 VPM software interface. The main window shows a 3D model of a gear with a circular face and teeth. The left-hand side contains a tree view with the following sections:

- z: plane
- Axis Systems
- Parameters
  - Number of teeth: z = 40
  - Ref Pressure Angle: Rpa = 20deg
  - Symmetry angle: s = 2.25deg = 90 / Number of teeth: z \* 1 deg
  - Module: m = 3mm
  - Pitch diameter: Pd = 120mm = Number of teeth: z \* Module: m
  - Base diameter: Bd = 112.763mm = Pitch diameter: Pd \* cos(Ref Pressure Angle: Rpa)
  - Addendum diameter: Ad = 126mm = Pitch diameter: Pd + (2 \* Module: m)
  - Dedendum diameter: Dd = 112.5mm = Pitch diameter: Pd - (2.5 \* Module: m)
  - tooth radius at dedendum circle: tr = 1.14mm = 0.38 \* Module: m
- Relations
  - Formula.1: Symmetry angle: s = 90 / Number of teeth: z \* 1 deg
  - Formula.2: Pitch diameter: Pd = Number of teeth: z \* Module: m
  - Formula.3: Base diameter: Bd = Pitch diameter: Pd \* cos(Ref Pressure Angle: Rpa)
  - Formula.4: Addendum diameter: Ad = Pitch diameter: Pd + (2 \* Module: m)
  - Formula.5: Dedendum diameter: Dd = Pitch diameter: Pd - (2.5 \* Module: m)
  - Formula.6: tooth radius at dedendum circle: tr = 0.38 \* Module: m
  - Formula.7: Body.2\CircPattern.1\AngularNumber = Number of teeth: z
  - Formula.8: Body.2\CircPattern.1\AngularSpacing = 360 deg / Number of teeth: z
  - Formula.9: Geometrical Set.1\Sketch.1\Radius.1\Radius = Pitch diameter: Pd / 2
  - Formula.10: Geometrical Set.1\Sketch.1\Radius.2\Radius = Base diameter: Bd / 2
  - Formula.11: Geometrical Set.1\Sketch.1\Radius.14\Radius = Dedendum diameter: Dd / 2
  - Formula.12: Geometrical Set.1\Sketch.1\Radius.15\Radius = Addendum diameter: Ad / 2
  - Formula.13: Geometrical Set.1\Sketch.1\Radius.18\Radius = tooth radius at dedendum circle: tr
  - Formula.14: Geometrical Set.1\Sketch.1\Radius.21\Radius = tooth radius at dedendum circle: tr
  - Formula.15: Geometrical Set.1\Sketch.1\Angle.3\Angle = Ref Pressure Angle: Rpa
  - Formula.16: Geometrical Set.1\Sketch.1\Angle.5\Angle = Symmetry angle: s
- Geometrical Set.1
- Sketch.1
- PartBody

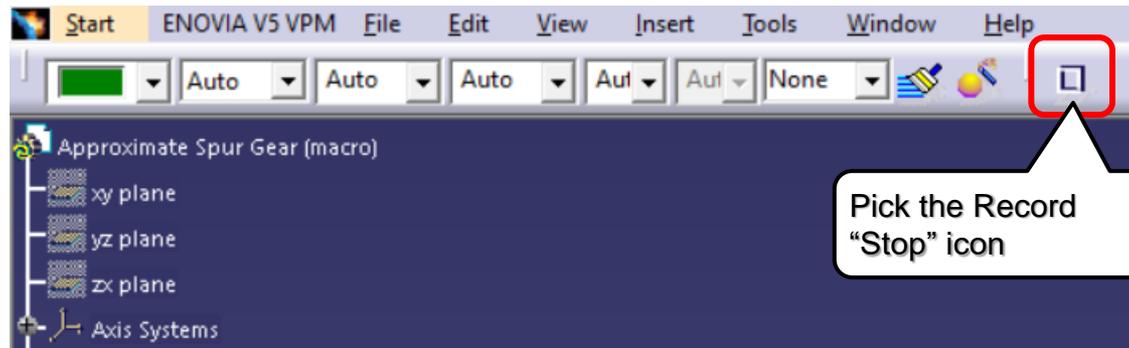
The bottom of the interface shows a toolbar with various icons. A red circle highlights the 'Update' icon (a circular arrow), with a callout box containing the text 'Pick Update'.

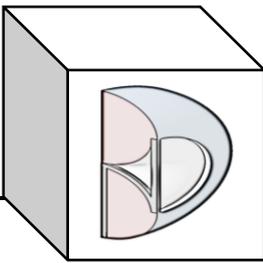


# BND TechSource



- **Stop recording** the 2<sup>nd</sup> Relations block of this Macro.





# BND TechSource



- Check the content of the Relations\_2 Macro.

Macros Editor - [D:\~William\CATIA Tutorials\Approximate Spur Gear\Relations\_2.catvbs]

```
Set relations1 = part1.Relations
Set angularRepartition1 = circPattern1.AngularRepartition
Set intParam1 = angularRepartition1.InstancesCount
Set formula1 = relations1.CreateFormula("Formula.17", "", intParam1, "Number of teeth: z ` ")
formula1.Rename "Formula.17"
Set relations2 = part1.Relations
Set angularRepartition2 = circPattern1.AngularRepartition
Set angle1 = angularRepartition2.AngularSpacing
Set formula2 = relations2.CreateFormula("Formula.18", "", angle1, "360 deg/Number of teeth: z ` ")
formula2.Rename "Formula.18"
Set axisSystems1 = part1.AxisSystems
Set axisSystem1 = axisSystems1.Item("Absolute Axis System")
Set reference3 = part1.CreateReferenceFromBRRepName("FEEdge:(Edge:(Face:(Brp:(AxisSystem.1;1);None();Cf11:0));Face...)")
circPattern1.SetRotationAxis reference3
part1.UpdateObject circPattern1
Set hybridBodies1 = part1.HybridBodies
Set hybridBody1 = hybridBodies1.Item("Geometrical Set.1")
Set sketches1 = hybridBody1.HybridSketches
Set sketch1 = sketches1.Item("Sketch.1")
Set constraints1 = sketch1.Constraints
Set constraint1 = constraints1.Item("Angle.3")
```

a) Alt+F8 to open the macro directory

b) Select the "Relations\_2" block.

c) Pick "Edit" to work on this portion of the macro.

Document:	Language	Run
Geometry.ca	MS VBScript	Edit...
Input.catvbs	MS VBScript	Create...
Parameters.ca	MS VBScript	Rename...
Relations.catv	MS VBScript	Delete
Relations_1.catv	MS VBScript	Select...
Relations_2.catv	MS VBScript	Obfuscate...

# BND TechSource

- Test the block of the Relations\_2 Macro.

The screenshot shows the ENOVIA V5 VPM interface with a gear model. The Relations list on the left contains the following formulas:

- Formula.1: Symmetry angle:  $s = 90 / \text{Number of teeth} : z * 1 \text{ deg}$
- Formula.2: Pitch diameter:  $Pd = \text{Number of teeth} : z * \text{Module} : m$
- Formula.3: Base diameter:  $Bd = \text{Pitch diameter} : Pd * \cos(\text{Ref Pressure Angle} : Rpa)$
- Formula.4: Addendum diameter:  $Ad = \text{Pitch diameter} : Pd + (2 * \text{Module} : m)$
- Formula.5: Dedendum diameter:  $Dd = \text{Pitch diameter} : Pd - (2.5 * \text{Module} : m)$
- Formula.6: tooth radius at dedendum circle: tr
- Formula.7: Geometrical Set.1\Sketch.1\Radius.15\Radius.15
- Formula.8: Geometrical Set.1\Sketch.1\Radius.14\Radius.14
- Formula.9: Geometrical Set.1\Sketch.1\Radius.13\Radius.13
- Formula.10: Geometrical Set.1\Sketch.1\Angle.10\Angle.10
- Formula.11: Geometrical Set.1\Sketch.1\Angle.11\Angle.11
- Formula.12: Geometrical Set.1\Sketch.1\Radius.15\Radius.15
- Formula.13: Geometrical Set.1\Sketch.1\Radius.14\Radius.14
- Formula.14: Geometrical Set.1\Sketch.1\Radius.13\Radius.13
- Formula.15: Geometrical Set.1\Sketch.1\Radius.12\Radius.12
- Formula.16: Geometrical Set.1\Sketch.1\Radius.11\Radius.11
- Formula.17: Body.2\CircPattern.1\AngularNumber=Number of teeth: z
- Formula.18: Body.2\CircPattern.1\AngularSpacing=360 deg/Number of teeth: z

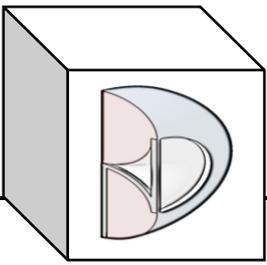
The Macro Directory shows the following table:

Macro	Language
Input.catvbs	MS VBScript
Parameter.catvbs	MS VBScript
Relations.catvbs	MS VBScript
Relations_1.catvbs	MS VBScript
Relations_2.catvbs	MS VBScript

Callout instructions:

- a) Alt+F8 to open the macro directory
- b) Select the "Relations\_2" block.
- c) Pick "Run" to test this portion of the macro.
- d) Notice the new relations have been added to the part.

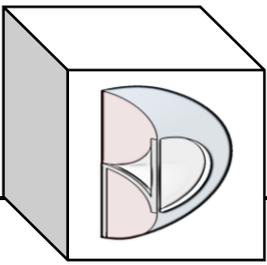
PASS



# BND TechSource



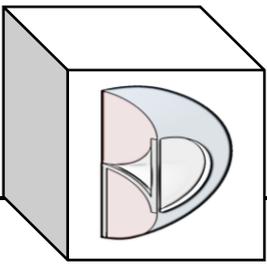
## Test the “Relations” blocks



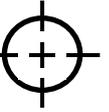
# BND TechSource



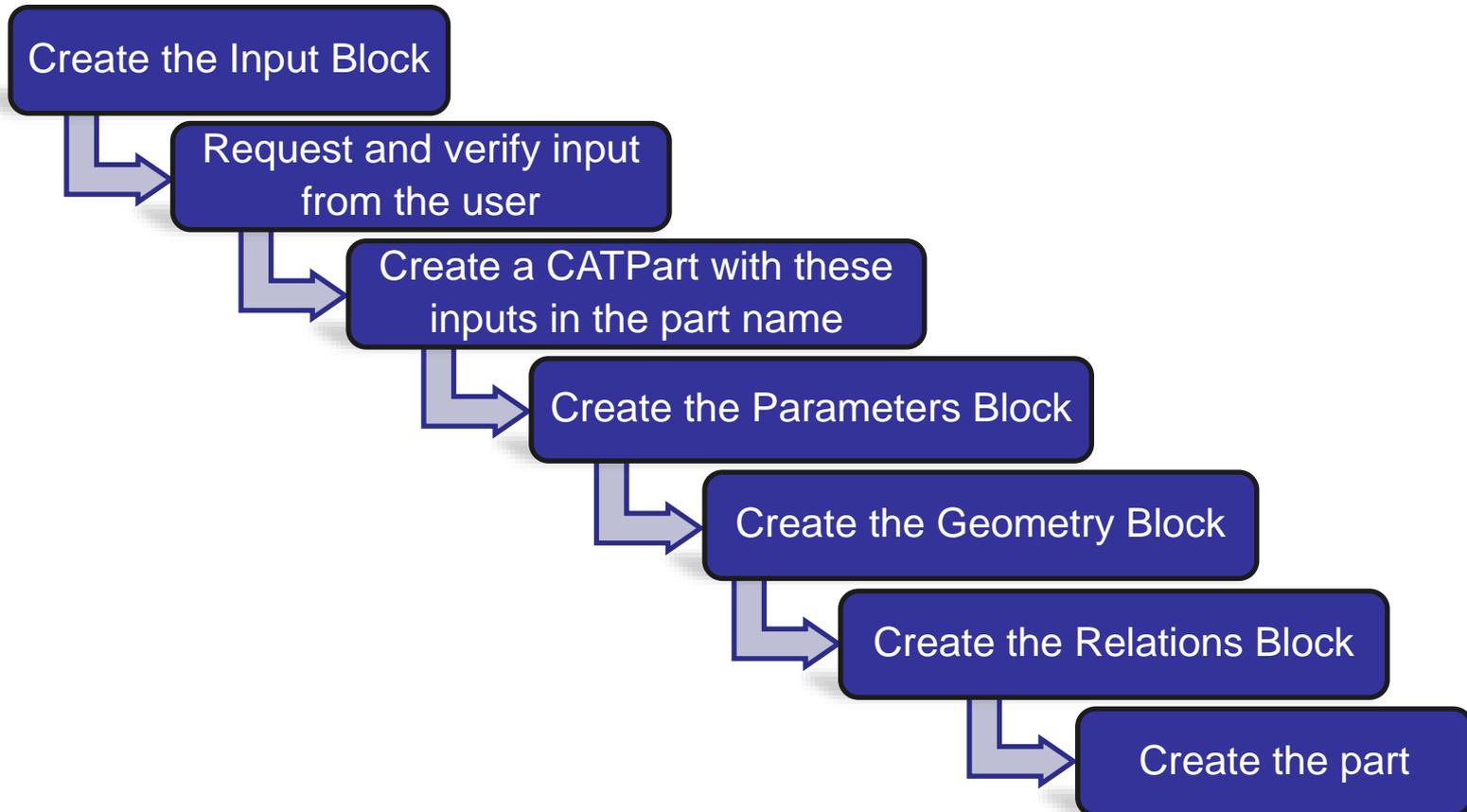
## Combine the Macro Blocks

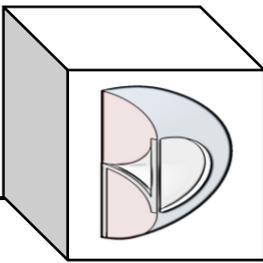


# BND TechSource



- Once each block runs independently, it's time to combine them into a single macro that will run through the complete process.





# BND TechSource



- Combine the blocks into one complete macro.

Macros Editor - [D:\~William\CATIA Tutorials\Approximate Spur Gear\Approximate Spur Gear.catvbs]

File Edit View Help

Language="VBSSCRIPT"

```
Sub CATMain()  
  
*****input*****  
a = MsgBox ("This macro will create an Approximate Spur Gear part based on your input", vbOKCancel, "Continue?")  
If a = vbCancel Then  
Exit sub  
Elseif a = vbOK Then  
Set partDocument1 = CATIA.ActiveDocument  
Set part1 = partDocument1.Part  
*****parameters*****  
module = ""  
module = InputBox("Module", "Module", module)  
teeth = ""  
teeth = InputBox("Number of teeth", "Number of teeth", teeth)  
Rpa = ""  
Rpa = InputBox("Pressure angle", "Pressure angle", Rpa)  
Set document = partDocument1  
Set parameters2 = part1.Parameters  
Set partDocum = partDocument1  
Set angle1 = parameters2.Parameter("Angle")  
Set product1 = angle1.Rename("Gear")  
result = MsgBox("Continue?", vbYesNo, "Continue?")  
If vbNo Then  
Do until result = vbYes  
module = InputBox("Module", "Module", module)  
module = InputBox("Number of teeth", "Number of teeth", teeth)  
result = MsgBox("Continue?", vbYesNo, "Continue?")  
Loop  
Else  
End if  
result = MsgBox("Continue?", vbYesNo, "Continue?")  
Set parameters3 = part1.Parameters  
Set angle2 = parameters3.Parameter("Angle")  
result = MsgBox("Continue?", vbYesNo, "Continue?")  
Loop  
Set relations1 = part1.Relations  
Set formula1 = relations1.CreateFormula("Formula.7", "", intParam1, "Number of teeth: z `")  
formula1.Rename "Formula.7"  
Set angularRepartition1 = circPattern1.AngularRepartition  
Set angularRepartition2 = circPattern1.AngularRepartition  
Set angle1 = angularRepartition2.AngularSpacing  
Set formula1 = relations1.CreateFormula("Formula.8", "", angle1.Value, "360/teeth")  
part1.InWorkObject = sketch1  
Set factor2D1 = sketch1.O
```

Macros

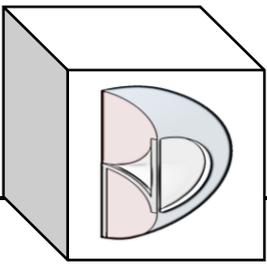
Current macro library or document:  
D:\~William\CATIA Tutorials\Approximate Spur Gear

Available macros:

Name	Language	Run
Approximate Spur Gear.catvbs	MS VBScript	Run
Geometry.catvbs	MS VBScript	Edit...
Input.catvbs	MS VBScript	Create...
Parameters.catvbs	MS VBScript	Rename...
Relations.catvbs	MS VBScript	Delete
		Select...
		Obfuscate...

Close

It is advisable to bring two blocks together and test the macro before combining the next block.



# BND TechSource



- Create Input+Params macro.

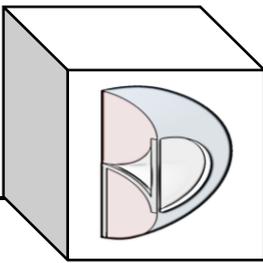
The screenshot shows the 'Macros' dialog box in CATIA. The 'Current macro library or document' is set to 'D:\~William\CATIA Tutorials\Approximate Spur Gear'. The 'Available macros' list is empty. The 'Create a new macro' sub-dialog box is open, showing the 'Macro language' set to 'MS VBScript' and the 'Macro name' set to 'Input+Params.catvbs'. The 'OK' button is highlighted.

a) Alt+F8 to open the macro directory

b) Pick "Create"

c) Type "Input+Params"

d) Pick "OK"



# BND TechSource



- Copy from Input macro and Paste into Input+Params macro.

Macros Editor - [D:\~\William\CATIA Tutorials\Approximate Spur Gear\Input.catvbs]

File Edit View Help

result = MsgBox ("The module will be " & module, vbYesNo, "Continue?")

If vbNo Then  
Do until result = vbYes  
module = ""  
module = InputBox ("Type the module for the Approximate Spur Gear", "module = ", module)  
result = MsgBox ("The module will be " & module, vbYesNo, "Continue?")  
Loop

Else  
End if

result = MsgBox ("The Number of teeth will be " & teeth, vbYesNo, "Continue?")

If vbNo Then  
Do until result = vbYes  
teeth = ""  
teeth = InputBox ("Type the Number of teeth for the Approximate Spur Gear", "teeth = ", teeth)  
result = MsgBox ("The Number of teeth will be " & teeth, vbYesNo, "Continue?")  
Loop

Else  
End if

result = MsgBox ("The Ref Pressure Angle will be " & Rpa, vbYesNo, "Continue?")

If vbNo Then  
Do until result = vbYes  
Rpa = ""  
Rpa = InputBox ("Type the Reference Pressure Angle for the Approximate Spur Gear", "Rpa = ", Rpa)  
result = MsgBox ("The Ref Pressure Angle will be " & Rpa, vbYesNo, "Continue?")  
Loop

Else  
End if

product1.PartNumber = "Approximate Spur Gear [m=" & module & ", z=" & teeth & ", Rpa=" & Rpa & "]"

End if

End Sub

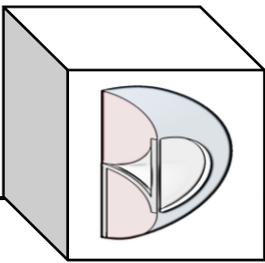
d) Close the editor window

c) Select all the text and Copy

a) Select "Input" macro

b) Pick "Edit"

Name	Language
Approximate Spur Gear.catvbs	MS VBScript
Geometry.catvbs	MS VBScript
<b>Input.catvbs</b>	<b>MS VBScript</b>
Input+Params.catvbs	MS VBScript
Input+Params_2.catvbs	MS VBScript
Input+Params+Geom+Relations.catvbs	MS VBScript
Parameters.catvbs	MS VBScript
Relations_1.catvbs	MS VBScript
Relations_2.catvbs	MS VBScript



# BND TechSource



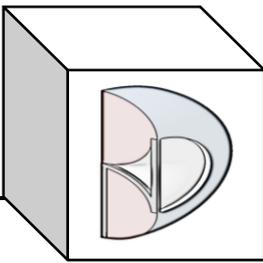
- Copy from Input macro and Paste into Input+Params macro.

The image consists of two screenshots from the CATIA software interface, illustrating the steps to copy and paste macros.

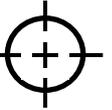
**Left Screenshot:** Shows the 'Macros Editor' window. The 'Edit' menu is open, and the 'Paste' option is highlighted. A callout box points to the 'Paste' option with the text: "c) Select all the text and Paste".

**Right Screenshot:** Shows the 'Available macros' dialog box. The 'Input+Params.catvbs' macro is selected in the list. A callout box points to the 'Edit...' button with the text: "b) Pick 'Edit'". Another callout box points to the 'Input+Params.catvbs' entry in the list with the text: "a) Select 'Input+Params' macro".

Name	Language
Approximate Spur Gear	MS VBScript
Geometry.catvbs	MS VBScript
Input.catvbs	MS VBScript
Input+Params.catvbs	MS VBScript
Input+Params_2.catvbs	MS VBScript
Input+Params+Gear Relations.catvbs	MS VBScript
Parameters.catvbs	MS VBScript
Relations_1.catvbs	MS VBScript
Relations_2.catvbs	MS VBScript



# BND TechSource



- Copy from Input macro and Paste into Input+Params macro.

The screenshot shows the CATIA Macros Editor window with a VBA script for an 'Approximate Spur Gear' macro. The script includes prompts for the number of teeth and reference pressure angle. A callout box labeled 'a) File + Save' points to the 'Save' option in the File menu. Another callout box labeled 'b) Close the editor window' points to the window's close button. In the bottom right, a 'Macro' dialog box is open, showing a list of available macros. The 'Input+Params.catvbs' macro is selected and highlighted in blue.

```
File Edit View Help
New
Open...
Save vbYes
Save As...
Exit

Else
End if

result = MsgBox ("The Number of teeth will be " & teeth, vbYesNo, "Continue?")

If vbNo Then
Do until result = vbYes
teeth = ""
teeth = InputBox ("Type the Number of teeth for the Approximate Spur Gear", "Number of teeth = ", teeth)
result = MsgBox ("The Number of teeth will be " & teeth, vbYesNo, "Continue?")
Loop

Else
End if

result = MsgBox ("The Ref Pressure Angle will be " & Rpa, vbYesNo, "Continue?")

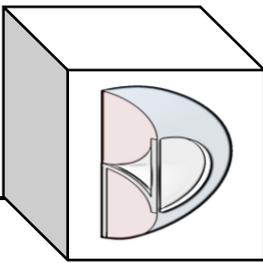
If vbNo Then
Do until result = vbYes
Rpa = ""
Rpa = InputBox ("Type the Reference Pressure Angle for the Approximate Spur Gear", "Ref Pressure Angle = ", Rpa)
result = MsgBox ("The Ref Pressure Angle will be " & Rpa, vbYesNo, "Continue?")
Loop

Else
End if

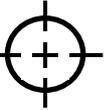
product1.PartNumber = "Approximate Spur Gear [m=" & module & ", z=" & teeth & ", Rpa=" & Rpa & "]"

End if
End Sub
```

Name	Language
Approximate Spur Gear.catvbs	MS VBScript
Geometry.catvbs	MS VBScript
Input.catvbs	MS VBScript
<b>Input+Params.catvbs</b>	<b>MS VBScript</b>
Input+Params_2.catvbs	MS VBScript
Input+Params+Geom+Relations.catvbs	MS VBScript
Parameters.catvbs	MS VBScript
Relations_1.catvbs	MS VBScript
Relations_2.catvbs	MS VBScript



# BND TechSource



- Copy from Parameters macro and Paste into Input+Params macro.

Macros Editor - [D:\~William\CATIA Tutorials\Approximate Spur Gear\Parameters.catvbs]

File Edit View Help

Language= "VBSCRIPT"

```
Sub CATMain()  
Set partDocument1 = CA  
Set part1 = partDocumen  
Set parameters1 = part1.Parameters  
Set realParam1 = parameters1.CreateReal("...", 0.000000)  
realParam1.Rename "Number of teeth: z"  
realParam1.Value = 40.000000  
Set parameters2 = part1.Parameters  
Set angle1 = parameters2.CreateDimension  
angle1.Rename "Ref Pressure Angle: Rpa"  
angle1.Value = 20.000000  
Set parameters3 = part1.Parameters  
Set angle2 = parameters3.CreateDimension  
angle2.Rename "Symmetry angle: s"  
Set relations1 = part1.Relations  
Set formula1 = relations1.CreateFormula("Formula.1", "", angle2, "90/Number of teeth: z *1 deg")  
formula1.Rename "Formula.1"  
Set parameters4 = part1.Parameters  
Set length1 = parameters4.CreateDimension("", "LENGTH", 0.000000)  
length1.Rename "Module: m"
```

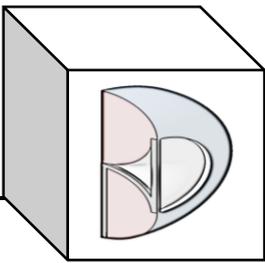
c) Select all the text between "Sub CATMain()" and "End Sub" and Copy

d) Close the editor window

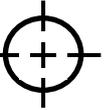
a) Select "Parameters" macro

b) Pick "Edit"

Name	Language
Approximate Spur Gear.catvbs	MS VBScript
Geometry.catvbs	MS VBScript
Input.catvbs	MS VBScript
Input+Params.catvbs	MS VBScript
Input+Params_2.catvbs	MS VBScript
Input+Params+Gear Relations.catvbs	MS VBScript
Parameters.catvbs	MS VBScript
Relations_1.catvbs	MS VBScript
Relations_2.catvbs	MS VBScript



# BND TechSource



- Copy from Parameters macro and Paste into Input+Params macro.

The image shows two overlapping windows from the CATIA software. The left window is the 'Macros Editor' for the file 'Input+Params.catvbs'. It displays the following VBA code:

```
result = MsgBox ("The module will be " & module, vbYesNo, "Continue?")
Loop

Else
End if

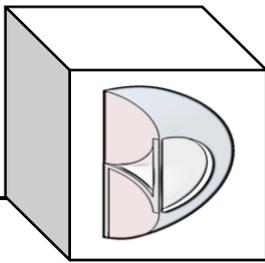
result = MsgBox ("The Number of teeth will be " & teeth, vbYesNo, "Continue?")

If vbNo Then
Do until result = vbYes
teeth = ""
teeth = InputBox ("Type the Number of teeth for the Approximate Spur Gear", "Number of teeth = ", teeth)
result = MsgBox ("The Number of teeth will be " & teeth, vbYesNo, "Continue?")
Loop

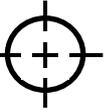
Else
    Undo
    Cut
    Copy
    Paste
    Delete
    Select All
    Right to left Reading order
    Show Unicode control characters
    Insert Unicode control character
    Open IME
    Reconversion
End
End Sub
```

The right window is the 'Macros' dialog box, which lists available macros. The 'Input+Params.catvbs' macro is selected. The 'Edit...' button is highlighted. Callouts provide instructions:

- a) Select "Input+Params" macro
- b) Pick "Edit"
- c) Paste just before "End Sub"



# BND TechSource



- Copy from Parameters macro and Paste into Input+Params macro.

The screenshot shows the CATIA Macros Editor window with the File menu open and the 'Save' option highlighted. A callout box points to the 'Save' option with the text 'a) File + Save'. Another callout box points to the 'Close' button in the 'Macros' dialog box with the text 'b) Close the editor window'. The 'Macros' dialog box is open, showing the current macro library or document as 'D:\~\William\CATIA Tutorials\Approximate Spur Gear' and a list of available macros. The 'Input+Params.catvbs' macro is selected in the list.

```
Macros Editor - [D:\~\William\CATIA Tutorials\Approximate Spur Gear\Input+Params.catvbs... - □ ×
```

File Edit View Help

New  
Open...  
Save  
Save As...  
Exit

a) File + Save

b) Close the editor window

Parameters

```
ers7.CreateDimension("", "LENGTH", 0.000000)  
endum diameter: Ad'  
Set relations4 = part1.Relations  
Set formula4 = relations4.CreateFormula("Formula.4", "", length4, "Pitch diameter: Pd + (2*Module: m`")  
formula4.Rename "Formula.4"  
Set parameters8 = part1.Parameters  
Set length5 = parameters8.CreateDimension("", "LENGTH", 0.000000)  
length5.Rename "Dedendum diameter: Dd"  
Set relations5 = part1.Relations  
Set formula5 = relations5.CreateFormula("Formula.5", "", length5, "Pitch diameter: Pd -(2.5*Module: m`")  
formula5.Rename "Formula.5"  
Set parameters9 = part1.Parameters  
Set length6 = parameters9.CreateDimension("", "LENGTH", 0.000000)  
length6.Rename "tooth radius at dedendum circle: tr"  
Set relations6 = part1.Relations  
Set formula6 = relations6.CreateFormula("Formula.6", "", length6, "0.38*Module: m`")  
formula6.Rename "Formula.6"  
End Sub
```

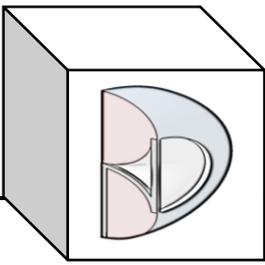
Macros

Current macro library or document:  
D:\~\William\CATIA Tutorials\Approximate Spur Gear Macro libraries...

Available macros:

Name	Language	
Approximate Spur Gear.catvbs	MS VBScript	Run
Geometry.catvbs	MS VBScript	Edit...
Input.catvbs	MS VBScript	Create...
Input+Params.catvbs	MS VBScript	Rename...
Input+Params_2.catvbs	MS VBScript	Delete
Input+Params+Geom+Relations.catvbs	MS VBScript	Select...
Parameters.catvbs	MS VBScript	Obfuscate...
Relations_1.catvbs	MS VBScript	
Relations_2.catvbs	MS VBScript	

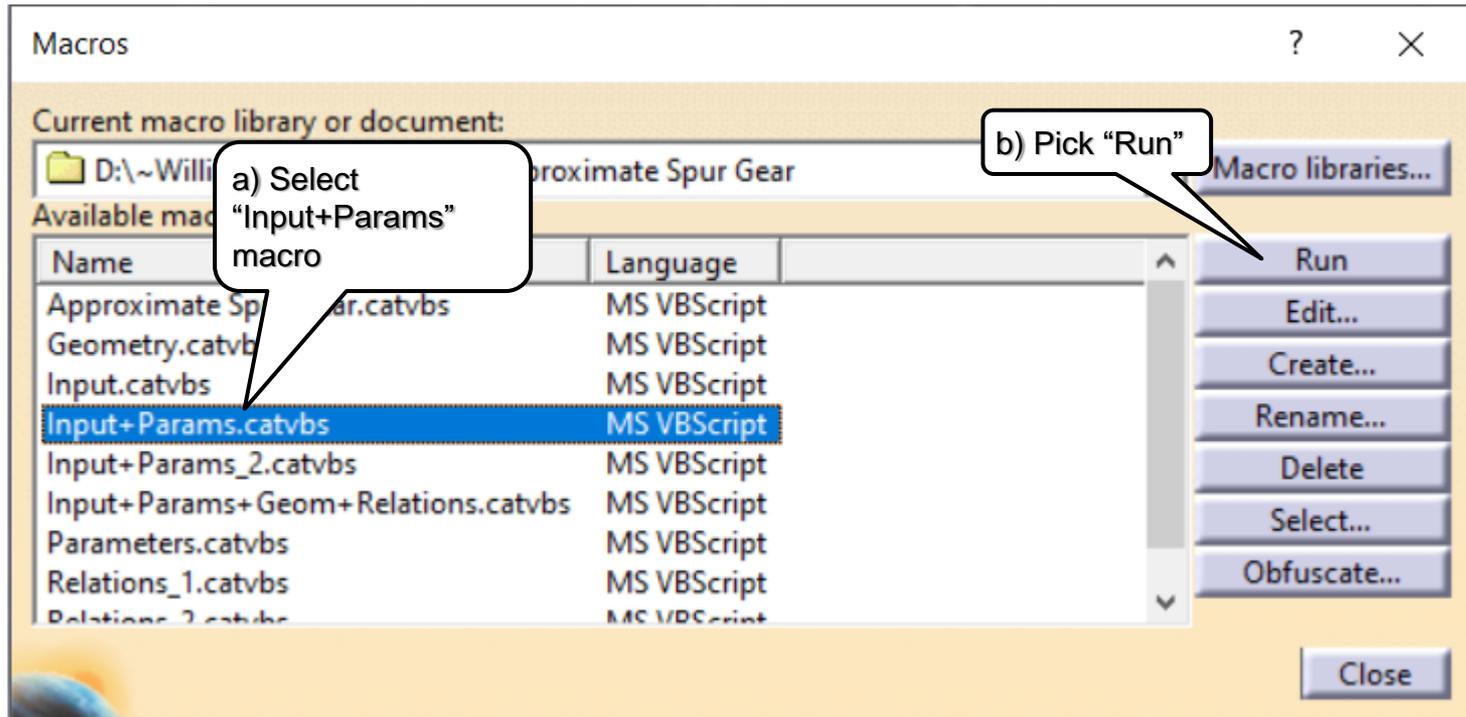
Close

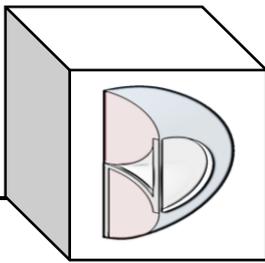


# BND TechSource



- Test the Input+Params macro.





# BND TechSource



- Test the Input+Params macro.

Test 1 Input:

$$m = 3$$

$$z = 40$$

$$Rpa = 20$$

Test 2 Input:

$$m = 3$$

$$z = 25$$

$$Rpa = 20$$

Approximate Spur Gear [m=3, z=40, Rpa= 20]

- xy plane
- yz plane
- zx plane
- Axis Systems
- Parameters
  - Number of teeth:  $z = 40$
  - Ref Pressure Angle:  $Rpa = 20deg$
  - Symmetry angle:  $s = 2.25deg = 90 / \text{Number of teeth: } z * 1 deg$
  - Module:  $m = 3mm$
  - Pitch diameter:  $Pd = 120mm = \text{Number of teeth: } z * \text{Module: } m$
  - Base diameter:  $Bd = 112.763mm = \text{Pitch diameter: } Pd * \cos(\text{Ref Pressure Angle: } Rpa)$
  - Addendum diameter:  $Ad = 126mm = \text{Pitch diameter: } Pd + (2 * \text{Module: } m)$
  - Dedendum diameter:  $Dd = 112.5mm = \text{Pitch diameter: } Pd - (2.5 * \text{Module: } m)$
  - tooth radius at dedendum circle:  $tr = 1.14mm = 0.38 * \text{Module: } m$

a) This data all looks good.

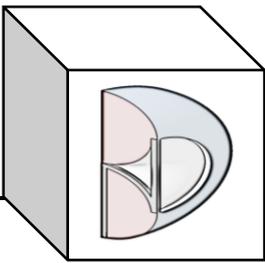
**PASS**

Approximate Spur Gear [m=3, z=25, Rpa= 20]

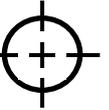
- xy plane
- yz plane
- zx plane
- Axis Systems
- Parameters
  - Number of teeth:  $z = 40$
  - Ref Pressure Angle:  $Rpa = 20deg$
  - Symmetry angle:  $s = 2.25deg = 90 / \text{Number of teeth: } z * 1 deg$
  - Module:  $m = 3mm$
  - Pitch diameter:  $Pd = 120mm = \text{Number of teeth: } z * \text{Module: } m$
  - Base diameter:  $Bd = 112.763mm = \text{Pitch diameter: } Pd * \cos(\text{Ref Pressure Angle: } Rpa)$
  - Addendum diameter:  $Ad = 126mm = \text{Pitch diameter: } Pd + (2 * \text{Module: } m)$
  - Dedendum diameter:  $Dd = 112.5mm = \text{Pitch diameter: } Pd - (2.5 * \text{Module: } m)$
  - tooth radius at dedendum circle:  $tr = 1.14mm = 0.38 * \text{Module: } m$

b) We have a bug to fix.

**FAIL**



# BND TechSource



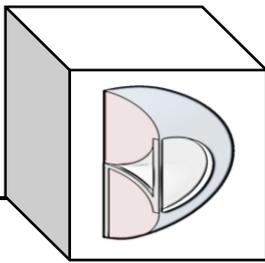
- Fix the bug in the Input+Params macro.

```
Macros Editor - [D:\~William\CATIA Tutorials\Approximate Spur Gear\Input+Params_2.catvbs]
File Edit View Help
[Icons]
module = ""
module = InputBox ("Type the Module for the Approximate Spur Gear", "module = module)
teeth = ""
teeth = InputBox ("Type the Number of Teeth for the Approximate Spur Gear", "Number of teeth = teeth)
Rpa = ""
Rpa = InputBox ("Type the Reference Pressure Angle for the Approximate Spur Gear", "Ref Pressure Angle = Rpa)
```

a) Input has these declarations.

```
realParam1.Value = teeth was 40.000
angle1.Value = Rpa was 20.000
length1.Value = module was 3.000
```

b) To fix the bug we must change the values to declarations in the Parameters section



# BND TechSource



- Test the Input+Params macro again.

Test 1 Input:

$$m = 3$$

$$z = 40$$

$$Rpa = 20$$

Test 2 Input:

$$m = 4$$

$$z = 25$$

$$Rpa = 14.5$$

Approximate Spur Gear [m=3, z=40, Rpa= 20]

- xy plane
- yz plane
- zx plane
- Axis Systems
- Parameters
  - Number of teeth:  $z = 40$
  - Ref Pressure Angle:  $Rpa = 20\text{deg}$
  - Symmetry angle:  $s = 2.25\text{deg} = 90 / \text{Number of teeth: } z * 1 \text{ deg}$
  - Module:  $m = 3\text{mm}$
  - Pitch diameter:  $Pd = 120\text{mm} = \text{Number of teeth: } z * \text{Module: } m$
  - Base diameter:  $Bd = 112.763\text{mm} = \text{Pitch diameter: } Pd * \cos(\text{Ref Pressure Angle: } Rpa)$
  - Addendum diameter:  $Ad = 126\text{mm} = \text{Pitch diameter: } Pd + (2 * \text{Module: } m)$
  - Dedendum diameter:  $Dd = 112.5\text{mm} = \text{Pitch diameter: } Pd - (2.5 * \text{Module: } m)$
  - tooth radius at dedendum circle:  $tr = 1.14\text{mm} = 0.38 * \text{Module: } m$

a) This data all looks good.

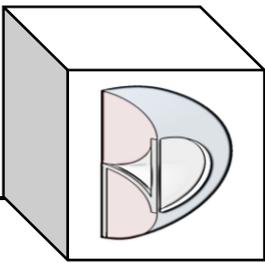
**PASS**

Approximate Spur Gear [m=4, z=25, Rpa= 14.5]

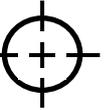
- xy plane
- yz plane
- zx plane
- Axis Systems
- Parameters
  - Number of teeth:  $z = 25$
  - Ref Pressure Angle:  $Rpa = 14.5\text{deg}$
  - Symmetry angle:  $s = 3.6\text{deg} = 90 / \text{Number of teeth: } z * 1 \text{ deg}$
  - Module:  $m = 4\text{mm}$
  - Pitch diameter:  $Pd = 100\text{mm} = \text{Number of teeth: } z * \text{Module: } m$
  - Base diameter:  $Bd = 96.815\text{mm} = \text{Pitch diameter: } Pd * \cos(\text{Ref Pressure Angle: } Rpa)$
  - Addendum diameter:  $Ad = 108\text{mm} = \text{Pitch diameter: } Pd + (2 * \text{Module: } m)$
  - Dedendum diameter:  $Dd = 90\text{mm} = \text{Pitch diameter: } Pd - (2.5 * \text{Module: } m)$
  - tooth radius at dedendum circle:  $tr = 1.52\text{mm} = 0.38 * \text{Module: } m$

a) This data all looks good.

**PASS**



# BND TechSource

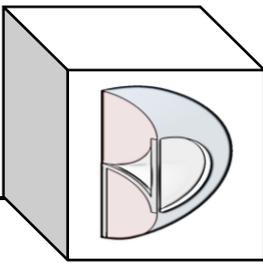


- Rename Input+Params to Input+Params+Geom.

The screenshot shows the 'Macros' dialog box in a CAD application. The 'Current document' is 'IA Tutorials\Approximate Spur Gear'. A table lists macros, with 'Input+Params.catvbs' selected. A 'Rename...' button is visible. A secondary dialog box is open, showing the new name 'Input+Params+Geom.catvbs' entered in the text field. Five callout boxes provide step-by-step instructions:

- a) Alt+F8 to open the macro directory
- b) Select the Input+Params Macro
- c) Pick "Rename"
- d) Rename to Input+Params+Geom
- e) Pick "OK"

Name	Language
Approximate Spur Gear.catvbs	MS VBScript
Geometry.catvbs	
Input.catvbs	
<b>Input+Params.catvbs</b>	
Parameters.catvbs	
Relations_1.catvbs	
Relations_2.catvbs	



# BND TechSource



- Copy from Geometry macro and Paste into Input+Params+Geom macro.

The image shows two overlapping windows from the CATIA software. The top window is the 'Macros Editor' for a file named 'Geometry.catvbs'. The code is in VBScript and includes a 'Sub CATMain()' block. A context menu is open over the code, with 'Copy' selected. A callout bubble points to this menu with the text: 'c) Select all the text between "Sub CATMain()" and "End Sub" and Copy'. The bottom window is the 'Macro Libraries' dialog, showing a list of macros. 'Geometry.catvbs' is selected in the list. A callout bubble points to the 'Edit...' button for this macro with the text: 'b) Pick "Edit"'. Another callout bubble points to the 'Close' button of the editor window with the text: 'd) Close the editor window'. A third callout bubble points to the 'Geometry' macro in the list with the text: 'a) Select "Geometry" macro'.

Macros Editor - [D:\-William\CATIA Tutorials\Approximate Spur Gear\Geometry.catvbs]

File Edit View Help

Language="VBSCRIPT"

Sub CATMain()

Set partDocument1 = CATIA.ActiveDocument

Set part1 = partDocument1.Part

Set hybridBody1 = part1.HybridBodies(1)

Set hybridBody2 = part1.HybridBodies(2)

Set sketches1 = part1.Sketches(1)

Set reference1 = part1.References(1)

Set sketch1 = part1.Sketches(1)

Dim arrayOfVariantOfDouble1(8)

arrayOfVariantOfDouble1(0) = 0.000000

arrayOfVariantOfDouble1(1) = 0.0

arrayOfVariantOfDouble1(2) = 0.0

arrayOfVariantOfDouble1(3) = 0.0

arrayOfVariantOfDouble1(4) = 1.0

arrayOfVariantOfDouble1(5) = 0.0

arrayOfVariantOfDouble1(6) = 0.0

arrayOfVariantOfDouble1(7) = 0.0

arrayOfVariantOfDouble1(8) = 1.0

sketch1.SetAbsoluteAxisData arrayOfVariantOfDouble1

part1.InWorkObject = sketch1

Set factory2D1 = sketch1.OpenEditor

Set geometricElements1 = sketch1.GeometricElements

Set axis2D1 = geometricElements1.Axis2D1

Set line2D1 = axis2D1.GetItem("H0direction")

line2D1.ReportName = 11

Undo

Cut

Copy

Paste

Delete

Select All

Right to left Reading order

Show Unicode control characters

Insert Unicode control character >

Open IME

Reconversion

Macros

Current macro library or document:  
D:\-William\CATIA Tutorials\Approximate Spur Gear

Name	Language	Run
Approximate Spur Gear.catvbs	MS VBScript	Edit...
Geometry.catvbs	MS VBScript	Run
Input.catvbs	MS VBScript	Edit...
Input+Params+Geom.catvbs	MS VBScript	Create...
Parameters.catvbs	MS VBScript	Rename...
Relations_1.catvbs	MS VBScript	Delete
Relations_2.catvbs	MS VBScript	Select...

Macro libraries...

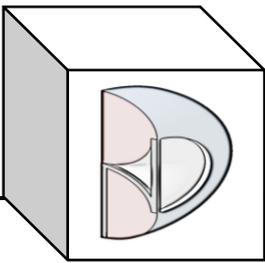
Close

d) Close the editor window

c) Select all the text between "Sub CATMain()" and "End Sub" and Copy

a) Select "Geometry" macro

b) Pick "Edit"



# BND TechSource

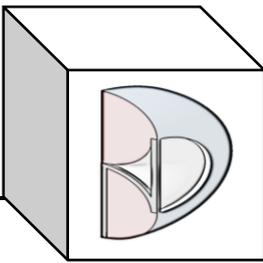


- Copy from Geometry macro and Paste into Input+Params+Geom macro.

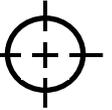
The image shows two windows from the CATIA software. The left window is the 'Macros Editor' for a file named 'Input+Params+Geom.catz'. It contains several lines of VBA code for setting parameters, creating dimensions, and defining formulas. A context menu is open over the text 'Paste', with 'Paste' highlighted. A callout bubble points to the 'Paste' option with the text: 'c) Paste just before "End Sub"'. The right window is the 'Available macros' dialog, showing a list of macros. The macro 'Input+Params+Geom.catz' is selected, and the 'Edit...' button is highlighted. A callout bubble points to the 'Edit...' button with the text: 'b) Pick "Edit"'. Another callout bubble points to the selected macro name with the text: 'a) Select "Input+Params+Geom" macro'.

```
Set parameters7 = part1.Parameters
Set length4 = parameters7.CreateDimension("", "LENGTH", 0.000000)
length4.Rename "Addendum diameter: Ad"
Set relations4 = part1.Relations
Set formula4 = relations4.CreateFormula("Formula.4", "", length4, "Pitch diameter: Pd' + (2*Module: m `)")
formula4.Rename "Formula.4"
Set parameters8 = part1.Parameters
Set length5 = parameters8.CreateDimension("", "LENGTH", 0.000000)
length5.Rename "Dedendum diameter: Dd"
Set relation:
Set formula:
formula5.Re
Set paramet
Set length6
length6.Re
Set relation:
Set formula:
formula6.Re
End Sub
```

Name	Language	Run
Approximate Spur Ge	MS VBScript	Edit...
Geometry.catz	MS VBScript	Create...
Input.catz	MS VBScript	Rename...
<b>Input+Params+Geom.catz</b>	<b>MS VBScript</b>	<b>Edit...</b>
Parameters.catz	MS VBScript	Delete
Relations_1.catz	MS VBScript	Select...
Relations_2.catz	MS VBScript	Obfuscate...



# BND TechSource



- Copy from Geometry macro and Paste into Input+Params+Geom macro.

The screenshot shows the CATIA Macros Editor window with the following macro script:

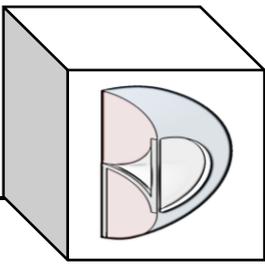
```
part1.UpdateObject sketch1
Set bodies1 = part1.Bodies
Set body1 = bodies1.Add()
part1.InWorkObject = body1
Set shapeFactory1 = part1.ShapeFactory
Set reference40 = part1.CreateReferenceFromName("")
Set pad1 = shapeFactory1.AddNewPadFromRef(reference40, 20.000000)
Set limit1 = pad1.FirstLimit
Set length1 = limit1.Dimension
length1.Value = 10.000000
Set reference41 = part1.CreateReferenceFromObject(sketch1)
pad1.SetProfileElement reference41
pad1.IsSymmetric = True
part1.UpdateObject pad1
part1.Update
End Sub
```

Two callout boxes provide instructions:

- a) File + Save (pointing to the Save option in the File menu)
- b) Close the editor window (pointing to the close button in the window title bar)

The Macros dialog box is open, showing the current macro library or document as "D:\-William\CATIA Tutorials\Approximate Spur Gear" and a list of available macros:

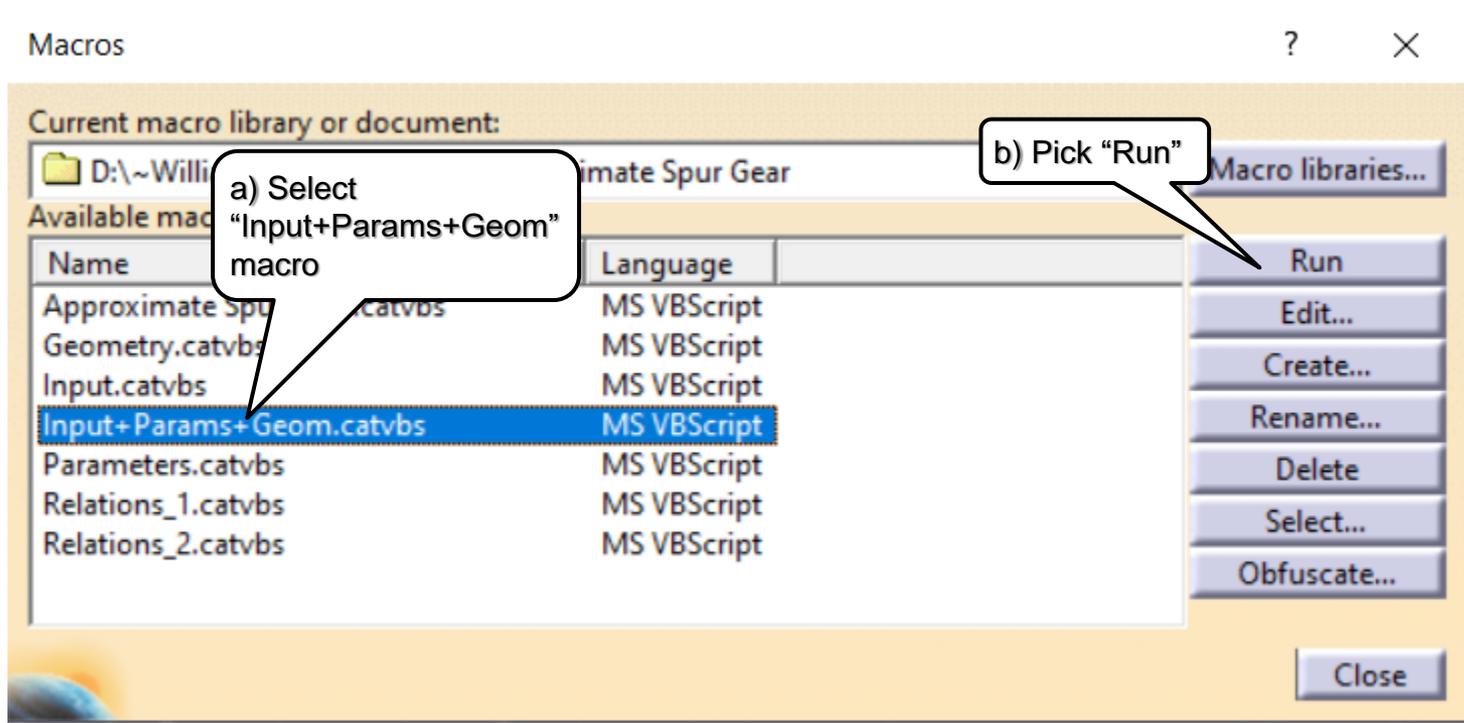
Name	Language	Run
Approximate Spur Gear.catvbs	MS VBScript	Run
Geometry.catvbs	MS VBScript	Edit...
Input.catvbs	MS VBScript	Create...
Input+Params+Geom.catvbs	MS VBScript	Rename...
Parameters.catvbs	MS VBScript	Delete
Relations_1.catvbs	MS VBScript	Select...
Relations_2.catvbs	MS VBScript	Obfuscate...

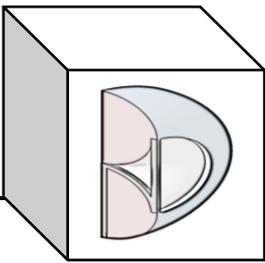


# BND TechSource



- Test the Input+Params+Geom macro.





# BND TechSource



- Test the Input+Params+Geom macro.

a) Notice that Pd, Bd, Ad, and Dd do not match the geometry.

b) This is because at this point there are no relations between the parameters and the geometry.

Test Input:  
 $m = 3$   
 $z = 25$   
 $Rpa = 20$   
**PASS**

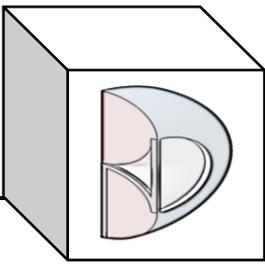
Parameters:

- Number of teeth:  $z = 25$
- Ref Pressure Angle:  $Rpa = 20\text{deg}$
- Symmetry angle:  $s = 3.6\text{deg} = 90 / \text{Number of teeth: } z * 1 \text{ deg}$
- Module:  $m = 3\text{mm}$
- Pitch diameter:  $Pd = 75\text{mm} = \text{Number of teeth: } z * \text{Module: } m$
- Base diameter:  $Bd = 70.477\text{mm} = \text{Pitch diameter: } Pd * \cos(\text{Ref Pressure Angle: } Rpa)$
- Addendum diameter:  $Ad = 81\text{mm} = \text{Pitch diameter: } Pd + (2 * \text{Module: } m)$
- Dedendum diameter:  $Dd = 67.5\text{mm} = \text{Pitch diameter: } Pd - (2.5 * \text{Module: } m)$
- tooth radius at dedendum circle:  $tr = 1.14\text{mm} = 0.38 * \text{Module: } m$

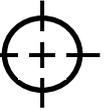
Relations:

- Geometrical Set.1
  - Sketch.1
- PartBody
- Body.2

Geometric features:  $D 126$ ,  $D 120$ ,  $R 56.382$ ,  $R 56.25$



# BND TechSource



- Rename Input+Params+Geom to Input+Params+Geom+Rel.

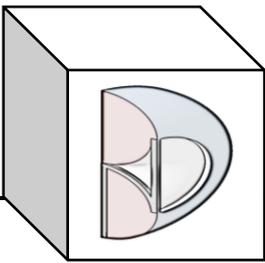
a) Alt+F8 to open the macro directory

b) Select the Input+Params+Geom Macro

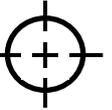
c) Pick "Rename"

d) Rename to Input+Params+Geom+Rel

e) Pick "OK"



# BND TechSource



- Copy from Relations\_1 macro and Paste into Input+Params+Geom+Rel macro.

The screenshot shows the CATIA Macros Editor window and the Macro Libraries dialog. The editor window displays a VBScript macro with the following code:

```
Sub CATMain()  
Set partDoc = PartDoc  
Set part1 = partDoc.Part1  
Set relations1 = part1.Relations  
Set hybridBody1 = hybridBody1  
Set sketches1 = hybridBody1.Sketches  
Set sketch1 = sketches1.Item("Angle.3")  
Set constraints1 = sketch1.Constraints  
Set constraint1 = constraints1.Item("Base diameter: Bd / 2")  
Set length1 = constraint1.Dimension  
Set formula1 = relations1.CreateFormula("Formula.9", "", length1, "Base diameter: Bd / 2")  
formula1.Rename "Formula.9"  
Set relations2 = part1.Relations  
Set constraint2 = constraints1.Item("Angle.3")  
Set length2 = constraint2.Dimension  
Set formula2 = relations2.CreateFormula("Formula.9", "", length2, "Base diameter: Bd / 2")  
formula2.Rename "Formula.9"  
Set relations3 = part1.Relations  
Set constraint3 = constraints1.Item("Angle.3")  
End Sub
```

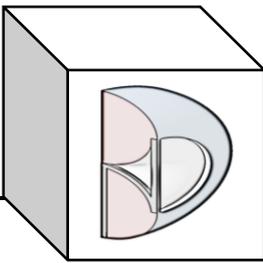
The Macro Libraries dialog shows the following table:

Name	Language	Run
Approximate Spur Gear.catvbs	MS VBScript	Edit...
Geom...catvbs	MS VBScript	Create...
Input...vbs	MS VBScript	Rename...
Input+Params+Geom+Rel.catvbs	MS VBScript	Delete
Params...catvbs	MS VBScript	Select...
Relations_1.catvbs	MS VBScript	Obfuscate...
Relations_2.catvbs	MS VBScript	

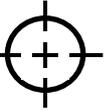
Callouts in the image provide the following instructions:

- a) Select "Relations\_1" macro
- b) Pick "Edit"
- c) Select all the text between "Sub CATMain()" and "End Sub" and Copy
- d) Close the editor window





# BND TechSource



- Copy from Relations\_1 macro and Paste into Input+Params+Geom+Rel macro.

The screenshot shows the CATIA Macros Editor window with a macro script and a 'Macros' dialog box. The macro script contains the following code:

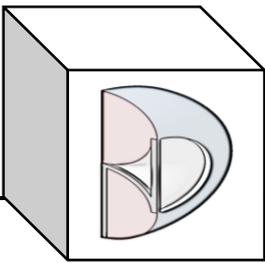
```
formula5.Rename "Formula.12"  
Set relations6 = part1.Relations  
Set constraint6 = constraints1.Item("Radius.14")  
Set length4 = constraint6.Dimension  
Set formula6 = relations6.CreateFormula("Formula.13", "", length4, "Dedendum diameter: Dd / 2")  
formula6.Rename "Formula.13"  
Set relations7 = part1.Relations  
Set constraint7 = constraints1.Item("Radius.18")  
Set length5 = constraint7.Dimension  
Set formula7 = relations7.CreateFormula("Formula.14", "", length5, "tooth radius at dedendum circ")  
formula7.Rename "Formula.14"  
Set relations8 = part1.Relations  
Set constraint8 = constraints1.Item("Radius.21")  
Set length6 = constraint8.Dimension  
Set formula8 = relations8.CreateFormula("Formula.15", "", length6, "tooth radius at dedendum circ")  
formula8.Rename "Formula.15"  
End Sub
```

Two callouts are present: 'a) File + Save' pointing to the 'Save' option in the 'File' menu, and 'b) Close the editor window' pointing to the 'X' button in the window's title bar.

The 'Macros' dialog box shows the current macro library or document as 'D:\~\William\CATIA Tutorials\Approximate Spur Gear' and lists available macros:

Name	Language	Run
Approximate Spur Gear.catvbs	MS VBScript	Run
Geometry.catvbs	MS VBScript	Edit...
Input.catvbs	MS VBScript	Create...
Input+Params+Geom+Rel.catvbs	MS VBScript	Rename...
Parameters.catvbs	MS VBScript	Delete
Relations_1.catvbs	MS VBScript	Select...
Relations_2.catvbs	MS VBScript	Obfuscate...

The 'Input+Params+Geom+Rel.catvbs' macro is highlighted in blue. The 'Close' button is visible at the bottom right of the dialog box.



# BND TechSource



- Copy from Relations\_2 macro and Paste into Input+Params+Geom+Rel macro.

The screenshot shows the CATIA Macros Editor window. The title bar reads 'Macros Editor - [D:\~\William\CATIA Tutorials\Approximate Spur Gear\Relat...'. The menu bar includes 'File', 'Edit', 'View', and 'Help'. The main area contains a list of macros, with 'Relations\_2' selected. A context menu is open over the selected macro, showing options like 'Undo', 'Cut', 'Copy', 'Paste', 'Delete', 'Select All', etc. The 'Copy' option is highlighted. A callout box 'c) Select all the text between "Sub CATMain()" and "End Sub" and Copy' points to the macro text. Another callout box 'a) Select "Relations\_2" macro' points to the macro name in the list. A third callout box 'b) Pick "Edit"' points to the 'Edit...' button in the context menu. A fourth callout box 'd) Close the editor window' points to the 'Close' button in the bottom right corner of the window.

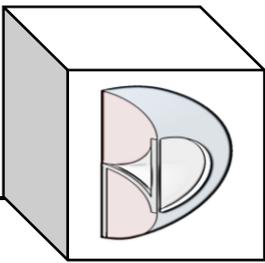
d) Close the editor window

c) Select all the text between "Sub CATMain()" and "End Sub" and Copy

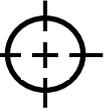
a) Select "Relations\_2" macro

b) Pick "Edit"

Name	Language
Approximate Spur Gear.catvbs	MS VBScript
Geometry.catvbs	MS VBScript
Input+Params+Geom+Rel.catvbs	MS VBScript
Parameters.catvbs	MS VBScript
Relations_1.catvbs	MS VBScript
Relations_2.catvbs	MS VBScript



# BND TechSource



- Copy from Relations\_2 macro and Paste into Input+Params+Geom+Rel macro.

Macros Editor - [D:\~William\CATIA Tutorials\Approximate Spur Gear\Input...]

File Edit View Help

```
Set length3 = constraint5.Dimension
Set formula5 = relations5.CreateFormula("Formula.12", "", length3, "Addendum diameter: Ad' /2")
formula5.Rename "Formula.12"
Set relations6 = part1.Relations
Set constraint6 = constraints1.Item("Radius.14")
Set length4 = constraint6.Dimension
Set formula6 = relations6.CreateFormula("Formula.13", "", length4, "Dedendum diameter: Dd' /2")
formula6.Rename "Formula.13"
Set relations7 = part1.Relations
Set c
Set l
Set f
form
Set r
Set c
Set l
Set f
form
End Sub
```

Undo  
Cut  
Copy  
Paste  
Delete  
Select All  
Right to left Reading order  
Show Unicode control characters  
Insert Unicode control character  
Open IME  
Reconversion

Available macros:

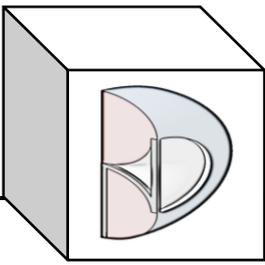
Name	Language
Approximate Spur Gear...catvbs	MS VBScript
Geometry.catvbs	MS VBScript
Input.catvbs	MS VBScript
<b>Input+Params+Geom+Rel.catvbs</b>	<b>MS VBScript</b>
Parameters.catvbs	MS VBScript
Relations_1.catvbs	MS VBScript
Relations_2.catvbs	MS VBScript

Run  
Edit...  
Create...  
Rename...  
Delete  
Select...  
Obfuscate...  
Close

a) Select "Input+Params+Geom+Rel" macro

b) Pick "Edit"

c) Paste just before "End Sub"



# BND TechSource



- Copy from Relations\_2 macro and Paste into Input+Params+Geom+Rel macro.

The screenshot shows the CATIA Macros Editor window with the following macro script:

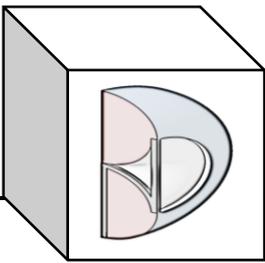
```
Set axisSystems1 = part1.AxisSystems
Set axisSystem1 = axisSystems1.Item("Absolute Axis System")
Set reference3 = part1.CreateReferenceFromBRepName("FEEdge:(Edge:(Face:(Brp:(AxisSystem.1);1);N
circPattern1.SetRotationAxis reference3
part1.UpdateObject circPattern1
Set hybridBodies1 = part1.HybridBodies
Set hybridBody1 = hybridBodies1.Item("Geometrical Set.1")
Set sketches1 = hybridBody1.HybridSketches
Set sketch1 = sketches1.Item("Sketch.1")
Set constraints1 = sketch1.Constraints
Set constraint1 = constraints1.Item("Angle.3")
constraint1.AngleSector = catCstAngleSector0
Set constraint2 = constraints1.Item("Angle.5")
constraint2.AngleSector = catCstAngleSector0
part1.Update
End Sub
```

Two callout boxes provide instructions:

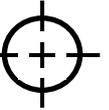
- a) File + Save (pointing to the Save option in the File menu)
- b) Close the editor window (pointing to the Close button in the Macro Libraries dialog)

The Macro Libraries dialog box shows the current macro library or document as D:\~\William\CATIA Tutorials\Approximate Spur Gear. The available macros are listed below:

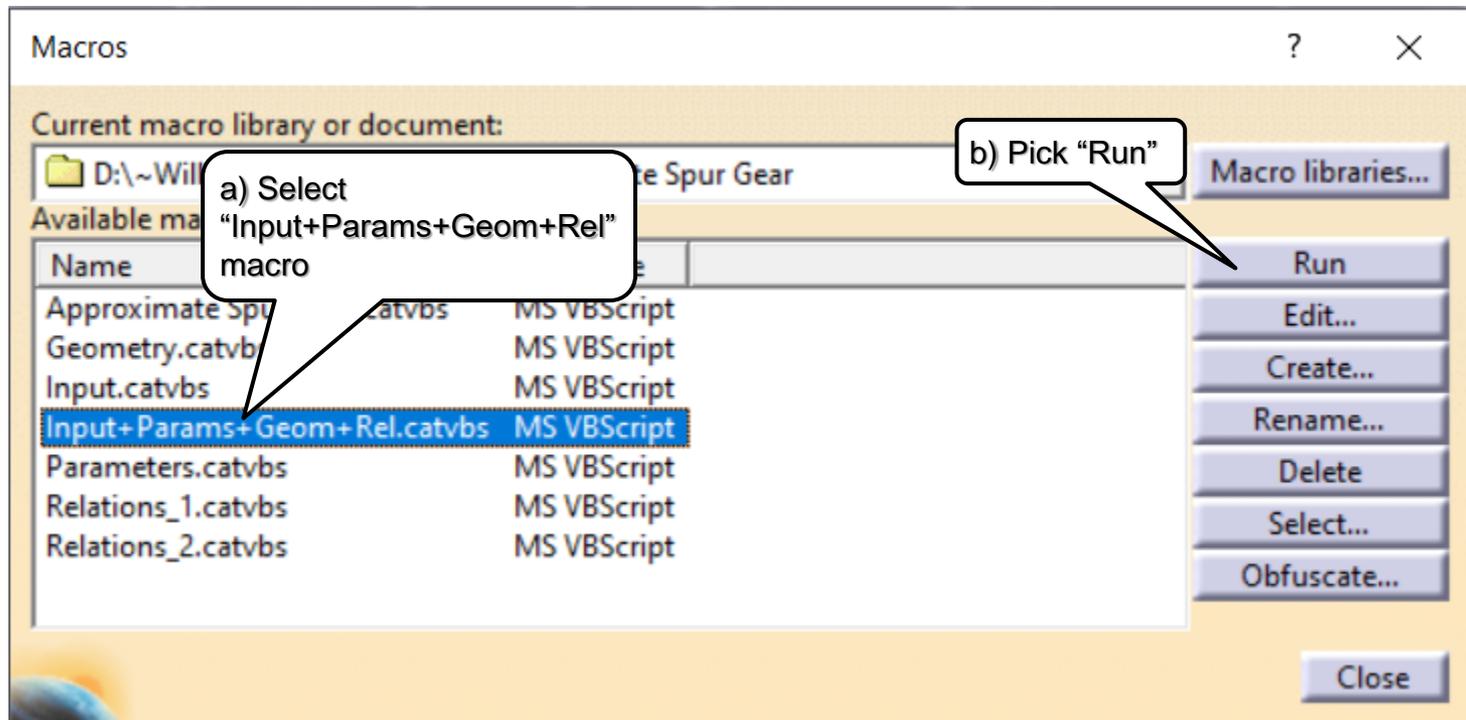
Name	Language	Run
Approximate Spur Gear.catvbs	MS VBScript	Edit...
Geometry.catvbs	MS VBScript	Create...
Input.catvbs	MS VBScript	Rename...
Input+Params+Geom+Rel.catvbs	MS VBScript	Delete
Parameters.catvbs	MS VBScript	Select...
Relations_1.catvbs	MS VBScript	Obfuscate...
Relations_2.catvbs	MS VBScript	

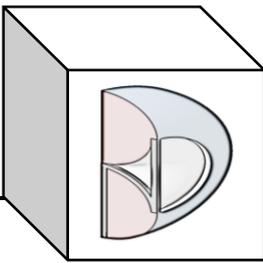


# BND TechSource



- Test the Input+Params+Geom+Rel macro.





# BND TechSource

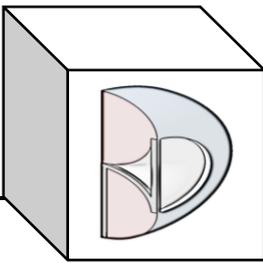


- Test the Input+Params+Geom+Rel macro.

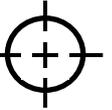
The screenshot displays the ENOVIA V5 VPM software interface. The main window shows a 3D model of a spur gear. The left-hand side contains a tree view with the following items: Approximate Spur Gear [m=3, z=25, Rpa= 20], xy plane, yz plane, zx plane, Axis Systems, Parameters, Relations, Geometrical Set.1, PartBody, and Body.2. The Parameters list is highlighted with a red box and contains the following entries:

- Number of teeth:  $z = 25$
- Ref Pressure Angle:  $Rpa = 20deg$
- Symmetry angle:  $s = 3.6deg = 90 / \text{Number of teeth: } z * 1 deg$
- Module:  $m = 3mm$
- Pitch diameter:  $Pd = 75mm = \text{Number of teeth: } z * \text{Module: } m$
- Base diameter:  $Bd = 70.477mm = \text{Pitch diameter: } Pd * \cos(\text{Ref Pressure Angle: } Rpa)$
- Addendum diameter:  $Ad = 81mm = \text{Pitch diameter: } Pd + (2 * \text{Module: } m)$
- Dedendum diameter:  $Dd = 67.5mm = \text{Pitch diameter: } Pd - (2.5 * \text{Module: } m)$
- tooth radius at dedendum circle:  $tr = 1.14mm = 0.38 * \text{Module: } m$

Overlaid on the screenshot is a white box with the text "Test Input:" followed by the values  $m = 3$ ,  $z = 25$ , and  $Rpa = 20$ . A green box with the word "PASS" is positioned to the right of these values. A speech bubble points to the gear model with the text: "a) Notice that all the parameters now match the geometry."



# BND TechSource



- And there you have it. A macro that produces an Approximate Spur Gear based upon user input!

The screenshot displays the ENOVIA V5 VPM software interface. The main window shows a 3D model of an approximate spur gear. The left-hand side contains a tree view with the following sections:

- Approximate Spur Gear [m=3, z=25, Rpa= 20]
- xy plane
- yz plane
- zx plane
- Axis Systems
- Parameters
- Relations

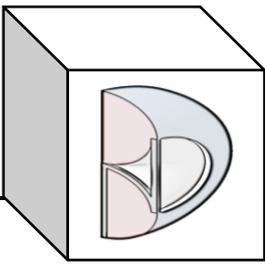
The Parameters section lists the following values:

- Number of teeth: z = 25
- Ref Pressure Angle: Rpa = 20deg
- Symmetry angle: s = 3.6deg = 90 / Number of teeth: z \* 1 deg
- Module: m = 3mm
- Pitch diameter: Pd = 75mm = Number of teeth: z \* Module: m
- Base diameter: Bd = 70.477mm = Pitch diameter: Pd \* cos(Ref Pressure Angle: Rpa)
- Addendum diameter: Ad = 81mm = Pitch diameter: Pd + (2 \* Module: m)
- Dedendum diameter: Dd = 67.5mm = Pitch diameter: Pd - (2.5 \* Module: m)
- tooth radius at dedendum circle: tr = 1.14mm = 0.38 \* Module: m

The Relations section lists the following formulas:

- Formula.1: Symmetry angle: s = 90 / Number of teeth: z \* 1 deg
- Formula.2: Pitch diameter: Pd = Number of teeth: z \* Module: m
- Formula.3: Base diameter: Bd = Pitch diameter: Pd \* cos(Ref Pressure Angle: Rpa)
- Formula.4: Addendum diameter: Ad = Pitch diameter: Pd + (2 \* Module: m)
- Formula.5: Dedendum diameter: Dd = Pitch diameter: Pd - (2.5 \* Module: m)
- Formula.6: tooth radius at dedendum circle: tr = 0.38 \* Module: m
- Formula.7: Body.2\CircPattern.1\AngularNumber = Number of teeth: z
- Formula.8: Body.2\CircPattern.1\AngularSpacing = 360 deg / Number of teeth: z
- Formula.9: Geometrical Set.1\Sketch.1\Radius.1\Radius = Pitch diameter: Pd / 2
- Formula.10: Geometrical Set.1\Sketch.1\Radius.2\Radius = Base diameter: Bd / 2
- Formula.11: Geometrical Set.1\Sketch.1\Radius.14\Radius = Dedendum diameter: Dd / 2
- Formula.12: Geometrical Set.1\Sketch.1\Radius.15\Radius = Addendum diameter: Ad / 2
- Formula.13: Geometrical Set.1\Sketch.1\Radius.18\Radius = tooth radius at dedendum circle: tr
- Formula.14: Geometrical Set.1\Sketch.1\Radius.21\Radius = tooth radius at dedendum circle: tr
- Formula.15: Geometrical Set.1\Sketch.1\Angle.3\Angle = Ref Pressure Angle: Rpa

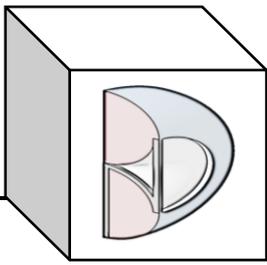
The 3D model of the gear is shown in the center of the interface, with a coordinate system (x, y, z) overlaid on it.



# BND TechSource



- Our challenge to you:
  - Edit this macro to include both the input and creation for the gear thickness.
  - Good Luck!
- We hope this series on creating Approximate Spur Gear parts has been useful.
- We chose the Spur Gear as an example; however, we are confident our users will be able to explore many new ways to use this information for their many other parts.



- Conclusion:

This is an example of Designing Approximate Spur Gear in CATIA V5 (Method 4: creating/using Macro).

We hope this will help those who need this type of simulation.

As always, we are open to any discussions this may bring.

Please ***subscribe*** to our YouTube channel!

