

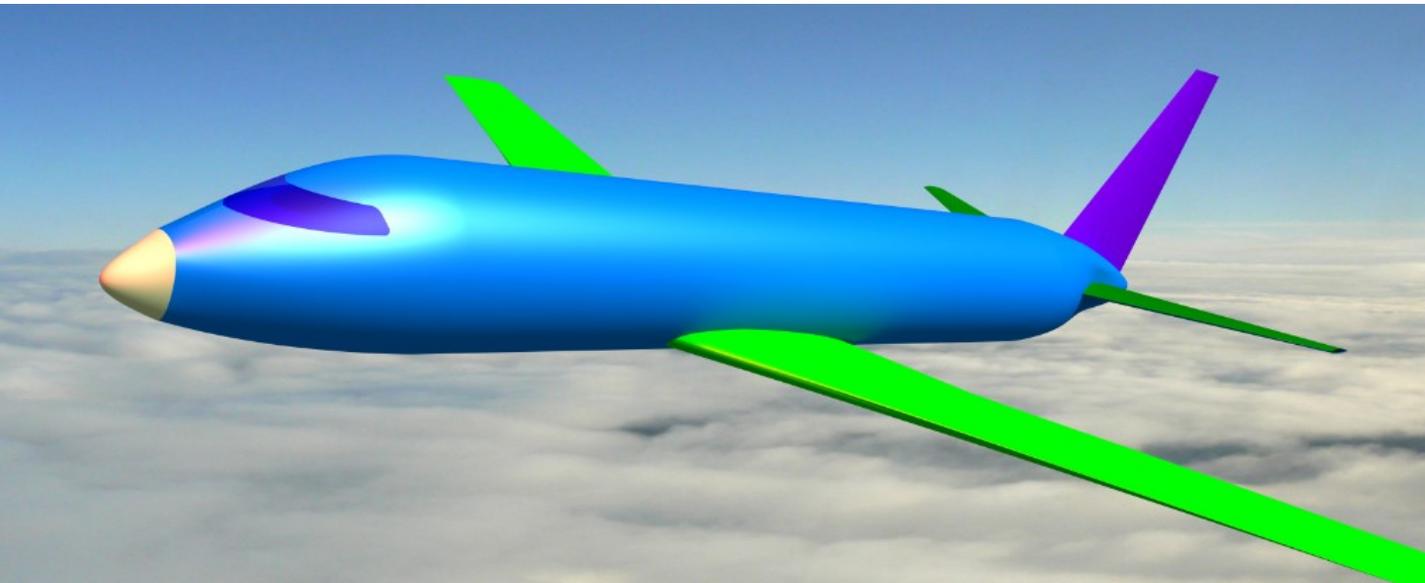


# MiniCurso



## Parametrização em Catia usando *Knowledge*

**Equipe Dragonfly e prof. Mateus Miranda**  
mateusmiranda@unb.br





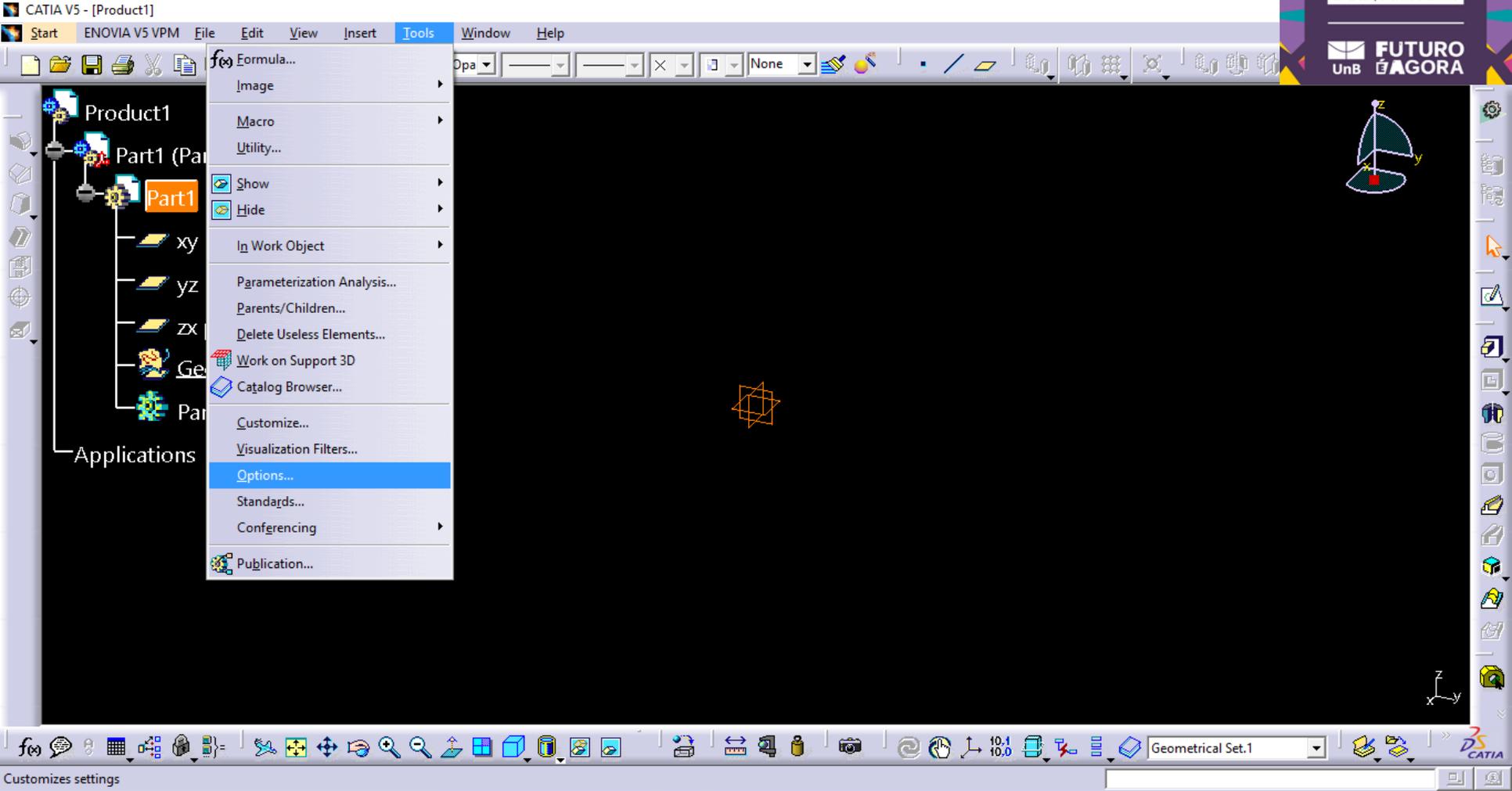
## Escopo do curso:

- Setagem do Catia para trabalhar com o knowledge.
- Treinamento das ferramentas principais de knowledge com um modelo simples;
- Aplicação dos conhecimentos em um modelo de aeronave.





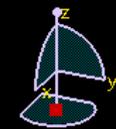
# Setagem do Catia





# Setagem do Catia

The screenshot displays the CATIA V5 software interface. On the left, the Product Tree shows a hierarchy starting with 'Product1', followed by 'Part1 (Part1.1)', and then 'Part1'. Below 'Part1', there are three coordinate planes: 'xy plane', 'yz plane', and 'zx plane'. A 'Geometrical Set.1' is also visible. The main workspace shows a 3D model of a sailboat. The 'Options' dialog box is open, with the 'Parameters and Measure' tab selected. The dialog box has several sections: 'Parameter Tree View' with checkboxes for 'With value' and 'With formula'; 'Parameter names' with a checkbox for 'Surrounded by the symbol'; 'Relations update in part context' with checkboxes for 'Creation of synchronous relations' and 'Creation of relations evaluated during the global update'; and 'Design Tables' with radio buttons for 'Automatic Synchronization At Load', 'Interactive Synchronization At Load', and 'Manual Synchronization', and a 'Default Mode' section with radio buttons for 'Copy Data Into Model' and 'Do Not Copy Data Into Model'. The 'OK' and 'Cancel' buttons are at the bottom right of the dialog box. The Windows taskbar at the bottom shows the system tray with the time 16:56 and date 07/10/2016.

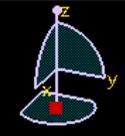




# Setagem do Catia



The screenshot displays the CATIA V5 software interface. On the left, the 'Product1' tree structure is visible, including 'Part1 (Part1.1)', 'xy plane', 'yz plane', 'zx plane', 'Geometrical Set.1', and 'PartBody'. The main workspace shows a 3D coordinate system. The 'Options' dialog box is open, with the 'Knowledge Environment' tab selected. The 'Language' section is expanded, showing 'Load extended language libraries' and 'All packages' checked. Below this, a list of packages is shown: AnalysisBasisPackage, AssyJoints, AutomotiveBiWFastening, BasicConstraintPackage, BasicProduct, and BasicWireframePackage. The 'Packages to load' list is currently empty. The 'Reference Directory For Types' field is also empty. The 'Knowledge Environment' section at the bottom shows 'Architect Resources Creation Path'. The Windows taskbar at the bottom shows the time as 16:57 on 07/10/2016.





# Setagem do Catia

Ativar Parameters e Relations

The screenshot shows the CATIA V5 interface with the 'Options' dialog box open. The 'Display' tab is selected, and the 'Parameters' and 'Relations' options are checked under the 'Display In Specification Tree' section. A red box highlights these two options, and a red arrow points from the text 'Ativar Parameters e Relations' to the 'Parameters' option.

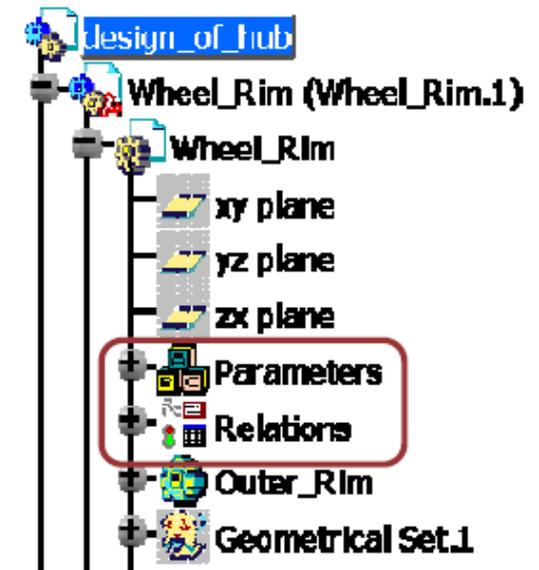
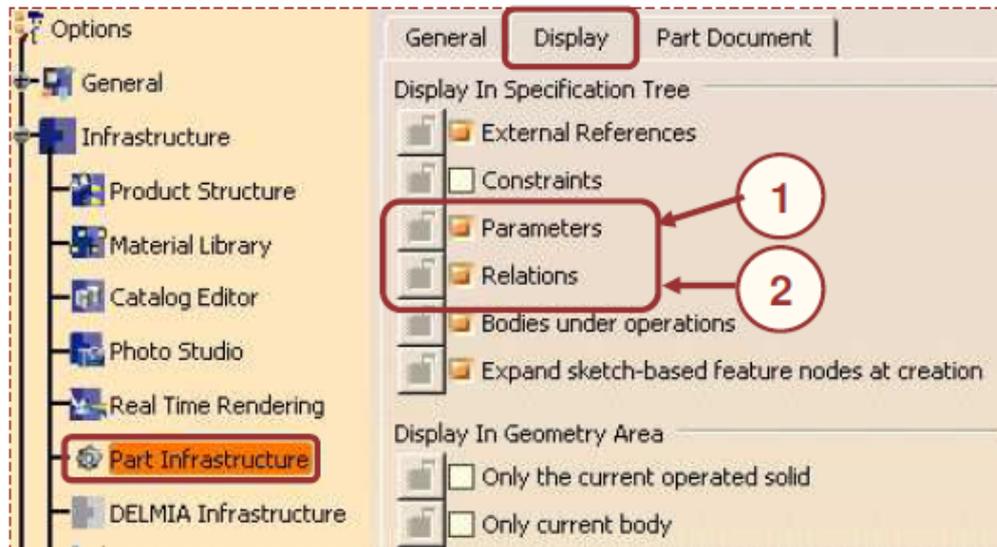
**Options Dialog Box - Display Tab**

- Display In Specification Tree
  - External References
  - Constraints
  - Parameters
  - Relations
  - Bodies under operations
  - Expand sketch-based feature nodes at creation
- Display In Geometry Area
  - Only the current operated solid
  - Only current body
  - Geometry located after the current feature
  - Parameters of features and constraints
- Axis system display size (in mm): 10
- Checking Operation When Renaming
  - No name check
  - Under the same tree node
  - In the main object

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UnB



# Setagem do Catia





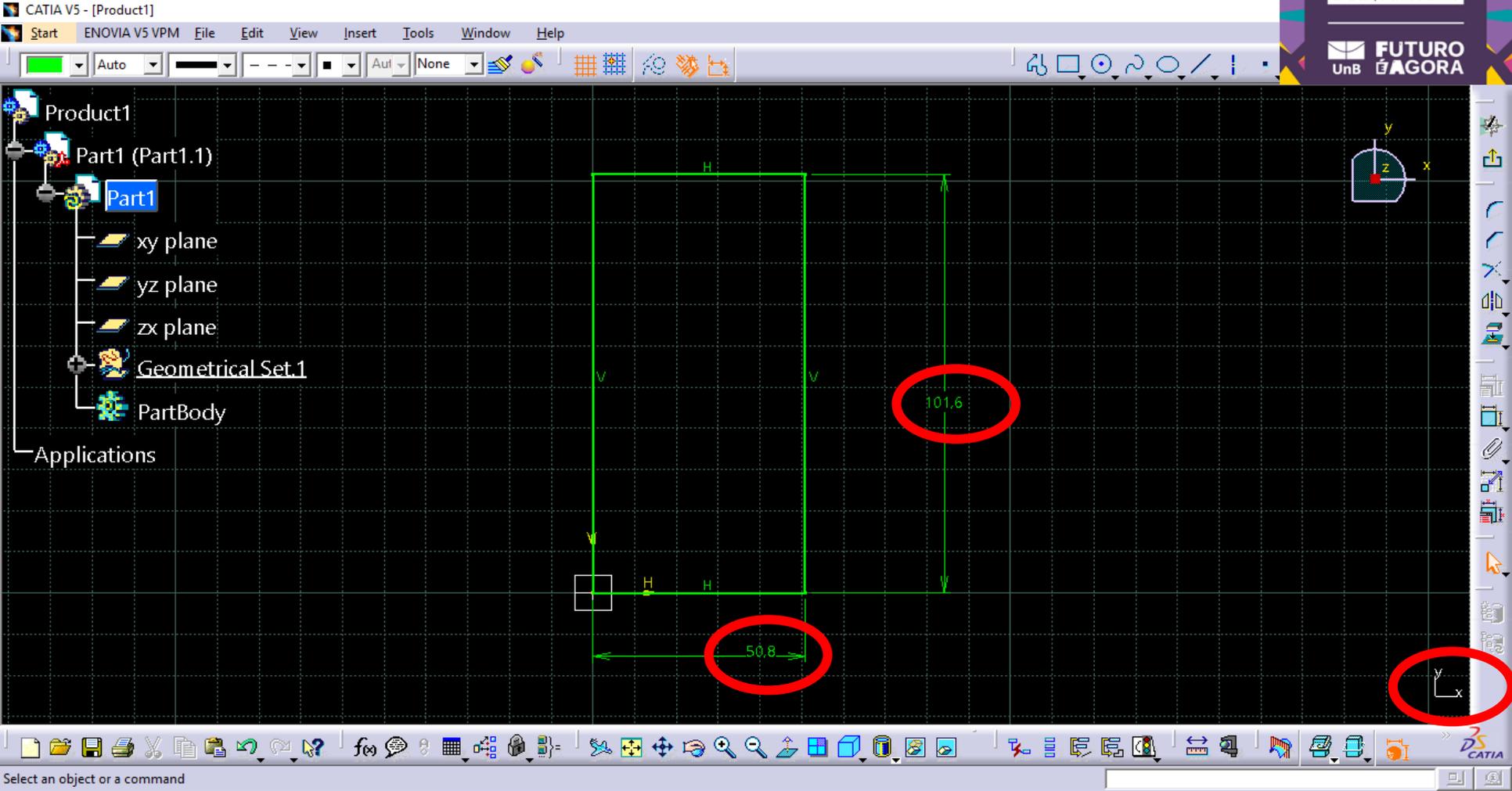
## Geração dos dados para os parâmetros em MS Excel

Part Num	espessura	comprimento	altura	dia furo
teste 1	76,2	101,6	127	25,4
teste 2	82,55	127	139,7	31,75
teste 3	88,9	152,4	152,4	38,1
teste 4	95,25	177,8	165,1	44,45

X

Part Num	espessura (mm)	comprimento (in)	altura	dia furo (mm)
teste 1	76,2	4	127mm	25,4
teste 2	82,55	5	5,5in	31,75
teste 3	88,9	6	152,4mm	38,1
teste 4	95,25	7	6,5in	44,45

# Geração do box teste Plano XY – 101,6 x 50,8 mm

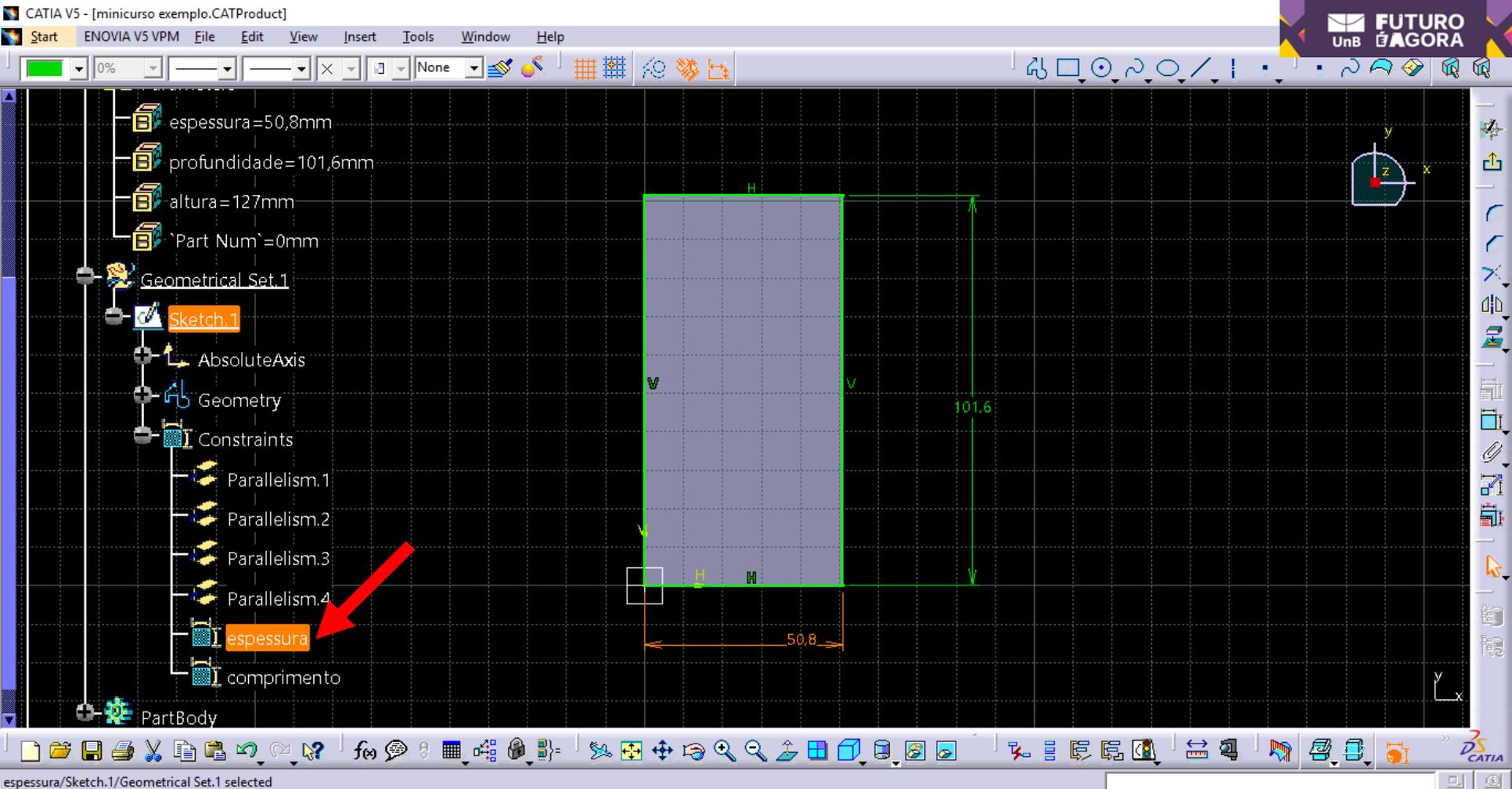


Select an object or a command



# Geração do box teste

## Renomear cotas: espessura e comprimento





# Geração do box teste

## Renomear cotas: espessura e comprimento



CATIA V5 - [minicurso exemplo.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

0%

None

espessura=50,8mm

profundidade=101,6mm

altura=127mm

'Part Num'=0mm

Geometrical Set.1

Sketch.1

AbsoluteAxis

Geometry

Constraints

- Parallelism.1
- Parallelism.2
- Parallelism.3
- Parallelism.4
- espessura
- comprimento

PartBody

2 elements selected



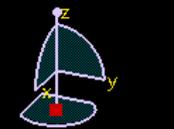
# Geração do box teste Gerar o PAD com 127 mm



The screenshot displays the CATIA V5 software interface. The main window shows a 3D model of a rectangular box with dimensions 127mm (height), 101.6mm (width), and 50.8mm (depth). The top surface is labeled 'LIM1' and the bottom surface is labeled 'LIM2'. A 'Pad Definition' dialog box is open, showing the following settings:

- First Limit: Type: Dimension, Length: 127mm, Limit: No selection
- Profile/Surface: Selection: Sketch.1
- Thick:
- Mirrored extent:
- Buttons: Reverse Side, Reverse Direction, More>>, OK, Cancel, Preview

The software interface includes a menu bar (Start, ENOVIA V5 VPM, File, Edit, View, Insert, Tools, Window, Help), a toolbar, a tree view on the left (Product1, Part1 (Part1.1), Part1, xy plane, yz plane, zx plane, Geometrical Set.1, PartBody, Pad.1), and a status bar at the bottom (PartBody, Dim1 | 127mm).





# Geração dos Parâmetros (Fórmulas) – F(x)



Formulas: Pad.1

Filter On Pad.1  
Filter Name: \*  
Filter Type: User parameters

Parameter	Value	Formula	Active
-----------	-------	---------	--------

New Parameter of type: Length With: Single Value

Creates a Length

Buttons: Add Formula, Delete Formula, OK, Apply, Cancel

Creates a new Length user parameter



# Geração dos Parâmetros (Fórmulas) – F(x)

## Tipo Length

Formulas: Parameters

Filter On  
Filter Name: \*  
Filter Type: User parameters

Double click on a parameter to edit it

Parameter	Value	Formula	Active
esessura	50,8mm		

Edit name or value of the current parameter

esessura 50,8mm

New Parameter of type Length With Single Value

Creates a Length parameter

Buttons: Add Formula, Delete Formula, OK, Apply, Cancel

1 2 3

Creates a new Length user parameter

Pesquisar na Web e no Windows

17:08 07/10/2016



# Geração dos Parâmetros (Fórmulas) – F(x) Tipo String

Formulas: Parameters

Filter On

Filter Name: \*

Filter Type: User parameter

Double click on a parameter

Parameter
espessura
profundidade
altura
Length.4

Edit name or value of the

Length.4

New Parameter of type

Delete Parameter

- Real
- Integer
- String
- Boolean
- Length
- Angle
- Time
- Mass
- Volume
- Density
- Area
- Inertia Product
- Energy
- Force
- Inertia
- Mass flow
- Moment
- Pressure
- Temperature
- Linear mass
- Stiffness Constant
- Volumetric flow
- Frequency
- Electric power
- Voltage
- Electric resistance
- Electric intensity
- Acceleration
- Angular Acceleration
- Strain Energy
- Volumic force
- Surfacic mass
- Velocity
- Linear feed rate
- Angular feed rate
- Linear spindle speed
- Angular spindle speed
- Surface spindle speed
- List

Import...

Active

Add Formula

Delete Formula

OK Apply Cancel



# Geração dos Parâmetros (Fórmulas) – F(x) Tipo String → Part\_Num

Formulas: teste 1

Filter On teste 1  
Filter Name: \*  
Filter Type: String

Double click on a parameter to edit it

Parameter	Value	Formula	Active
'teste 1\Part Number'	teste 1	DesignTable.1	yes
'teste 1\Nomenclature'			
'teste 1\Revision'			
'teste 1\Product Description'			
'teste 1\Definition'			
String.1			

Edit name or value of current parameter  
Part\_Num

New Parameter of type String With Single Value

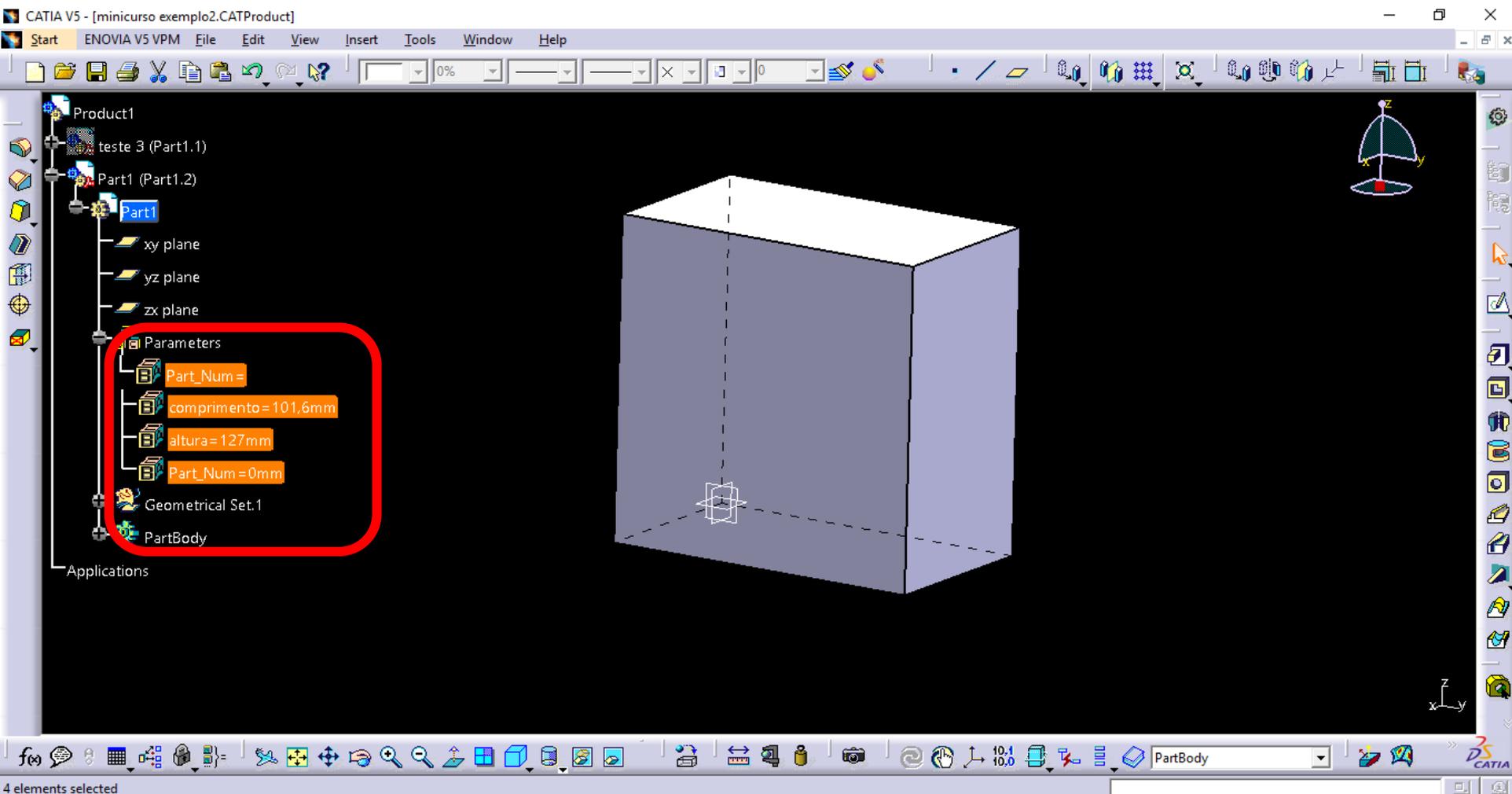
Buttons: Add Formula, Delete Formula, OK, Apply, Cancel



# Geração dos Parâmetros (Fórmulas) – F(x)

Resultado das fórmulas na árvore. Se Part\_Num = 0mm => **ERRO (Lenght)**

Correto: Part\_Num =





# Inserção da tabela MS Excel no Catia via Design Table

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Auto None

Product1

- teste 3 (Part1.1)
- Part1 (Part1.2)
  - Part1
    - xy plane
    - yz plane
    - zx plane
    - Parameters
      - Espessura=50,8mm
      - comprimento=101,6mm
      - altura=127mm
      - Part\_Num =
    - Relations
    - Geometrical Set.1
    - PartBody

Applications

Creation of a Design Table

Name: DesignTable.1  
 Comment: DesignTable created by Mateus 13/10/2016

Create a design table from a pre-existing file  
 Create a design table with current parameter values

Orientation:  Vertical  Horizontal

For Excel or Lotus 1-2-3 sheets, sheet index: 1

You should create a design table:  
 either from a text file, an Excel sheet or a Lotus 1-2-3 sheet (on NT).  
 Here is an example of a design table:

PadHeight (mm)	PadWidth (mm)	Material
15	12	Steel
17	1.3 cm	Aluminium

In a text file, columns should be separated by tabulations.

Destination:  
 Part1\Relations

OK Cancel

1

Change the name and the comment of the design table and choose the option of creation



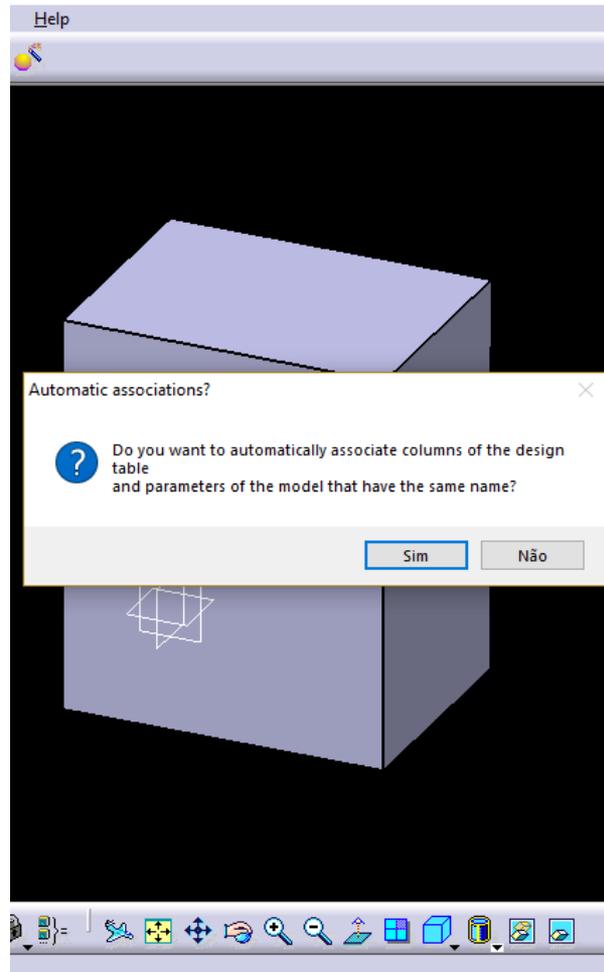
# Inserção da tabela excel no Catia via Design Table

Selecione o arquivo e dê sim,

The screenshot shows the CATIA V5 software interface. The main window displays a CAD model tree on the left with 'Part1' selected. A 'File Selection' dialog box is open in the center, showing a file named 'minicurso catia knowledge.xlsx' selected in a list. A tooltip for this file shows details: 'Tipo: Planilha do Microsoft Excel', 'Autores: Mateus Miranda', 'Tamanho: 8,81 KB', and 'Data de modificação: 13/10/2016 14:29'. To the right, a 'Design Table' dialog box is partially visible, showing a table with columns for 'Date' and 'Material'. The 'Date' column contains '13/10/2016' and the 'Material' column contains 'Steel' and 'Aluminium'. The 'Design Table' dialog has 'OK' and 'Cancel' buttons at the bottom.

Date	Material
13/10/2016	Steel
	Aluminium

Inserção da tabela excel no Catia via Design Table  
Selecione o arquivo e dê sim, associar automaticamente.





# Inserção da tabela excel no Catia via Design Table

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Product1

- teste 3 (Part1.1)
- Part1 (Part1.2)
  - Part1
    - xy plane
    - yz plane
    - zx plane
    - Parameters
      - Espessura=50,8mm
      - comprimento= 101,6mm
      - altura= 127mm
      - Part\_Num =
    - Relations
    - Geometrical Set.1
    - PartBody

Applications

DesignTable.1 active, configuration row: 1

Design Table Properties

Name: DesignTable.1

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter: Edit...

Line	comprimento	altura
1	101,6mm	127mm
2	127mm	139,7mm
3	152,4mm	152,4mm
4	177,8mm	165,1mm

This list contains the possible configurations of the design table. The selected row is the current one.

Edit table...  Duplicate data in CATIA model

OK Apply Cancel

PartBody

This list contains the available configurations of the design table. The selected row is the current one.

Pesquisar na Web e no Windows

15:42 13/10/2016



# Inserção da tabela excell no Catia via Design Table

## Associação da tabela excell com os parâmetros.

DesignTable.1 active

Design Table Properties

Name: DesignTable.1

Comment: Design Table created by Mateus 10/10/2016

Configurations: Associations

Filter On Product: \*

Filter Name: \*

Filter Type: All

Parameters	Columns
Part1\Geometrical Set.1\Sketch.1\Act	Part Num
Part1\Geometrical Set.1\Sketch.1\Abs	espessura
Part1\Geometrical Set.1\Sketch.1\Para	profundidade
Part1\Geometrical Set.1\Sketch.1\Para	altura
Part1\Geometrical Set.1\Sketch.1\Para	dia furo
Part1\Geometrical Set.1\Sketch.1\Para	

Buttons: Associate, Dissociate, Create parameters..., Rename associated parameters, Edit table..., Duplicate data in CATIA model, OK, Apply, Cancel



# Inserção da tabela excell no Catia via Design Table Associação da tabela excell com os parâmetros.

The screenshot shows the CATIA V5 interface with the Design Table dialog box open. The dialog box is titled "DesignTable.1 active" and contains the following sections:

- Design Table Properties:** Name: DesignTable.1, Comment: DesignTable created by Mateus 13/10/2016.
- Configurations:** Filter On Part1: Filter Name: \*, Filter Type: Length.
- Associations:** A table showing the mapping between parameters and columns.

Parameters	Columns
'Geometrical Set.1\Sketch.1\espessura	Part Num
'Geometrical Set.1\Sketch.1\comprimento	dia furo
PartBody\Sketch.2\espessura\Length	comprimento
PartBody\Sketch.2\comprimento\Length	altura
PartBody\Pad.1\FirstLimit\Length	espessura
PartBody\Pad.1\SecondLimit\Length	
PartBody\Pad.1\ThickThin1	
PartBody\Pad.1\ThickThin2	
Espessura	

Red arrows labeled 1, 2, and 3 point to the "Associate" button, the "Parameters" list, and the "Columns" list respectively. A tooltip at the bottom of the dialog reads: "Adds an association between the selected parameter and the selected column".

# Inserção da tabela excell no Catia via Design Table

## Fazer para espessura, comprimento, altura e part num.

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

yz plane  
zx plane  
Parameters  
Espessura=50,8mm  
comprimento= 101,6mm  
altura= 127mm  
Part\_Num =  
Relations  
Geometrical Set.1  
Sketch.1  
AbsoluteAxis  
Geometry  
Constraints  
PartBody

DesignTable.1 active, configuration row: 1

Design Table Properties  
Name: DesignTable.1  
Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter On Part1  
Filter Name: \*  
Filter Type: Length

Parameters	Columns
PartBody\Sketch.2\espessura\Length	Part Num
PartBody\Sketch.2\comprimento\Length	dia furo
PartBody\Pad.1\FirstLimit\Length	altura
PartBody\Pad.1\SecondLimit\Length	
PartBody\Pad.1\ThickThin1	
PartBody\Pad.1\ThickThin2	
Espessura	
comprimento	
altura	

Associations between parameters and columns

Parameters	Columns
'Geometrical Set.1\Sketch.1\...	espessura
'Geometrical Set.1\Sketch.1\...	comprimento

Associate Dissociate

Create parameters... Rename associated parameters

Edit table... Duplicate data in CATIA model

OK Apply Cancel

PartBody

InitialState

Pesquisar na Web e no Windows

15:46  
13/10/2016



# Inserção da tabela excell no Catia via Design Table Fazer para espessura, comprimento, **altura** e part num.

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

yz plane  
zx plane  
Parameters  
Espessura=50,8mm  
comprimento= 101,6mm  
altura= 127mm  
Part\_Num =  
Relations  
Geometrical Set.1  
Sketch.1  
AbsoluteAxis  
Geometry  
Constraints  
PartBody

DesignTable.1 active, configuration row : 1

Design Table Properties  
Name : DesignTable.1  
Comment : DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter On Part1  
Filter Name : \*  
Filter Type : Length

Parameters	Columns
PartBody\Sketch.2\espessura\Length	Part Num
PartBody\Sketch.2\comprimento\Len	dia furo
PartBody\Pad.1\FirstLimit\Length	altura
PartBody\Pad.1\SecondLimit\Length	
PartBody\Pad.1\ThickThin1	
PartBody\Pad.1\ThickThin2	
Espessura	
comprimento	
altura	

Associate Dissociate

Create parameters... Rename associated parameters

Edit table... Duplicate data in CATIA model

OK Apply Cancel

PartBody

InitialState

Pesquisar na Web e no Windows

15:46  
13/10/2016

# Inserção da tabela excel no Catia via Design Table Fazer para espessura, comprimento, altura e **part num**.

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Auto None

DesignTable.1 active, configuration row : 1

Design Table Properties

Name: DesignTable.1

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter On Part1

Filter Name: \*

Filter Type: Length

Parameters	Columns
PartBody\Sketch.2\espessura\Length	Part Num
PartBody\Sketch.2\comprimento\Length	dia furo
PartBody\Pad.1\SecondLimit\Length	
PartBody\Pad.1\ThickThin1	
PartBody\Pad.1\ThickThin2	
Espessura	
comprimento	
altura	
Part_Num	

Associations between parameters and columns

Parameters	Columns
'Geometrical Set.1\Sketch.1\...	espessura
'Geometrical Set.1\Sketch.1\...	comprimento
PartBody\Pad.1\FirstLimit\L...	altura

Associate Dissociate

Create parameters... Adds an association between the selected parameter and the selected column's

Edit table...

Duplicate data in CATIA model

OK Apply Cancel

yz plane

zx plane

Part\_Num =

Espessura=50,8mm

comprimento= 101,6mm

altura= 127mm

Part\_Num = 0mm

Relations

Geometrical Set.1

Sketch.1

AbsoluteAxis

Geometry

Constraints

PartBody

Applications

PartBody

# Inserção da tabela excell no Catia via Design Table Fazer para espessura, comprimento, altura e **Part Num.**

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

yz plane  
zx plane  
Parameters  
Espessura=50,8mm  
comprimento=101,6mm  
altura=127mm  
Part\_Num =  
Relations  
Geometrical Set.1  
Sketch.1  
AbsoluteAxis  
Geometry  
Constraints  
PartBody

Applications

DesignTable.1 active, configuration row : 1

Design Table Properties  
Name: DesignTable.1  
Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter On Part1  
Filter Name: \*  
Filter Type: String

Parameters	Columns
Part1\Nomenclature	dia furo
Part1\Revision	
Part1\Product Description	
Part1\Definition	

Associations between parameters and columns

Parameters	Columns
Geometrical Set.1\Sketch.1\...	espessura
Geometrical Set.1\Sketch.1\...	comprimento
PartBody\Pad.1\FirstLimit\L...	altura
teste 1\Part Number	Part Num

List of parameters that can be associated with a column in the design table

Associate Dissociate

Create parameters... Rename associated parameters

Edit table... Duplicate data in CATIA model

OK Apply Cancel

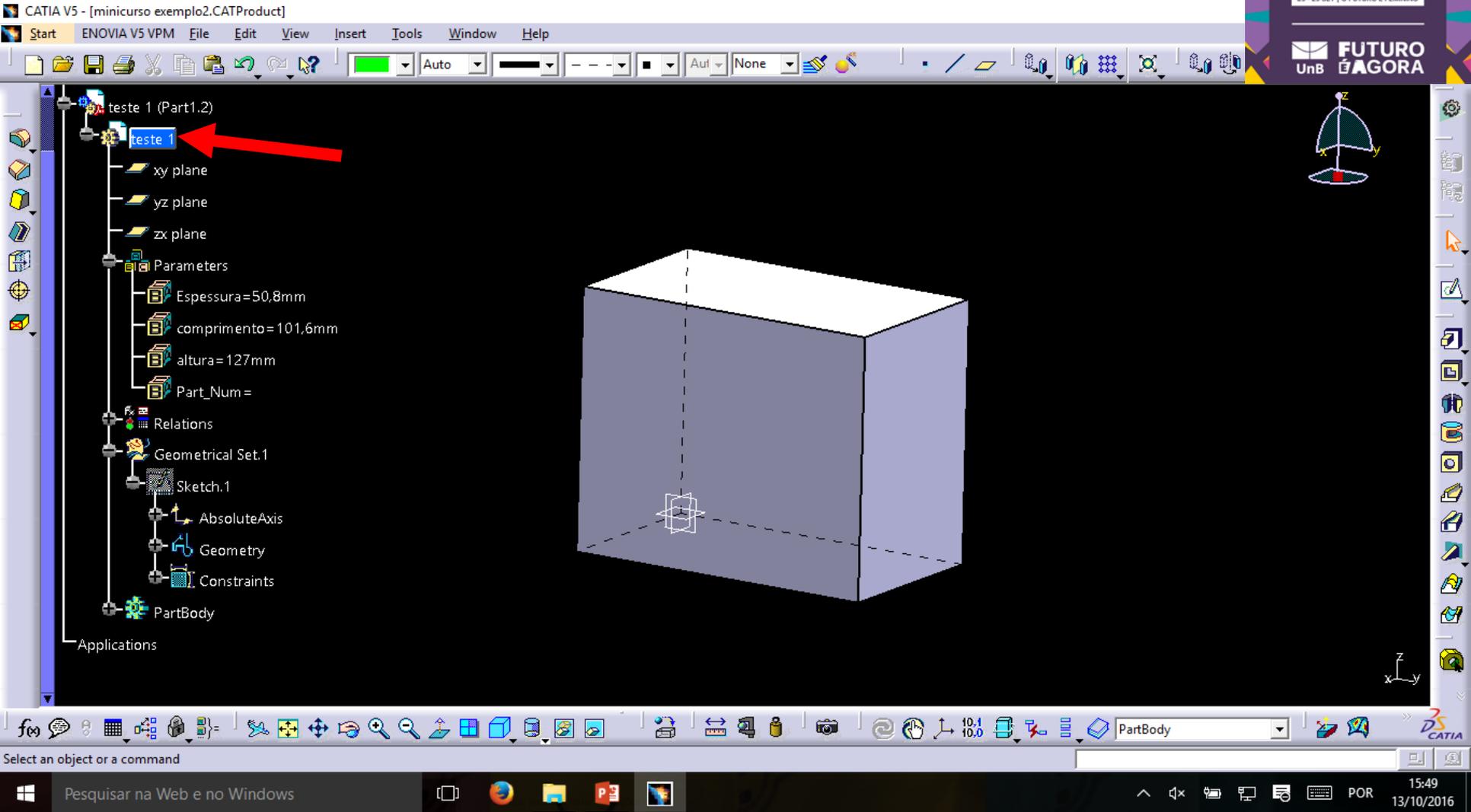
1  
2  
3

PartBody

13/10/2016 15:48

List of parameters that can be associated with a column in the design table

# Inserção da tabela excel no Catia via Design Table Resultado na árvore.





# Mudando os parâmetros usando Relations – Design Table

teste 1 (Part1.2)

- teste 1
  - xy plane
  - yz plane
  - zx plane
  - Parameters
    - espessura=50,8mm
    - comprimento=101,6mm
    - altura=127mm
    - Part\_Num =
  - Relations
    - DesignTable.1
      - Configuration=1
    - Sheet.1
  - Geometrical Set.1
  - Sketch.1
  - PartBody
  - Pad.1

Applications

Edit Parameter

Relations\DesignTable.1\Configu 1

OK Cancel

Edit parameter

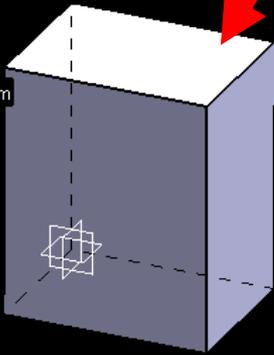
# Mudando os parâmetros usando Relations – Design Table

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

teste 1 (Part1.2)

- teste 1
  - xy plane
  - yz plane
  - zx plane
  - Parameters
    - espessura=50,8mm
    - comprimento=101,6mm
    - altura=127mm
    - Part\_Num =
  - Relations
    - DesignTable.1
    - Geometrical Set.1
    - Sketch.1
    - PartBody
    - Pad.1



DesignTable.1 active, configuration row : 1

Design Table Properties

Name: DesignTable.1

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter:

Line	'Geometrical Set.1\Sketch.1\espessura\Length'	'Geometrical Set.1\Sketch.1\comprimento\Length'	PartBody\Pad.1\FirstLir
1	76,2mm	101,6mm	127mm
<2>	82,55mm	127mm	139,7mm
3	88,9mm	152,4mm	152,4mm
4	95,25mm	177,8mm	165,1mm

1

2

3

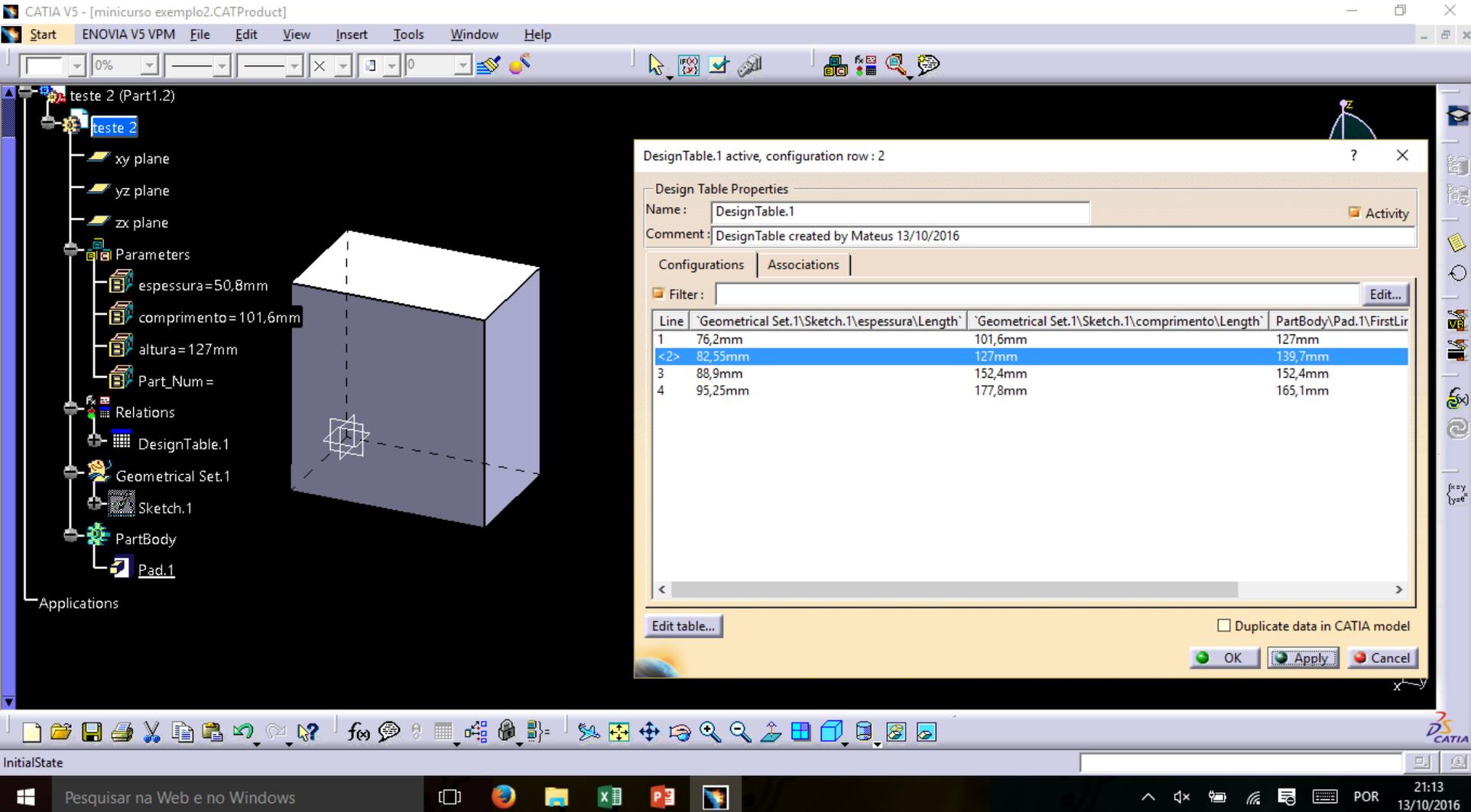
Apply

InitialState

Pesquisar na Web e no Windows

21:13 13/10/2016

# Mudando os parâmetros usando Relations – Design Table



CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

teste 2 (Part1.2)

- teste 2
  - xy plane
  - yz plane
  - zx plane
  - Parameters
    - espessura=50,8mm
    - comprimento=101,6mm
    - altura=127mm
    - Part\_Num =
  - Relations
    - DesignTable.1
    - Geometrical Set.1
    - Sketch.1
    - PartBody
    - Pad.1

DesignTable.1 active, configuration row : 2

Design Table Properties

Name: DesignTable.1

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter:

Line	'Geometrical Set.1\Sketch.1\espessura\Length'	'Geometrical Set.1\Sketch.1\comprimento\Length'	PartBody\Pad.1\FirstLir
1	76,2mm	101,6mm	127mm
<-2>	82,55mm	127mm	139,7mm
3	88,9mm	152,4mm	152,4mm
4	95,25mm	177,8mm	165,1mm

Edit table...

Duplicate data in CATIA model

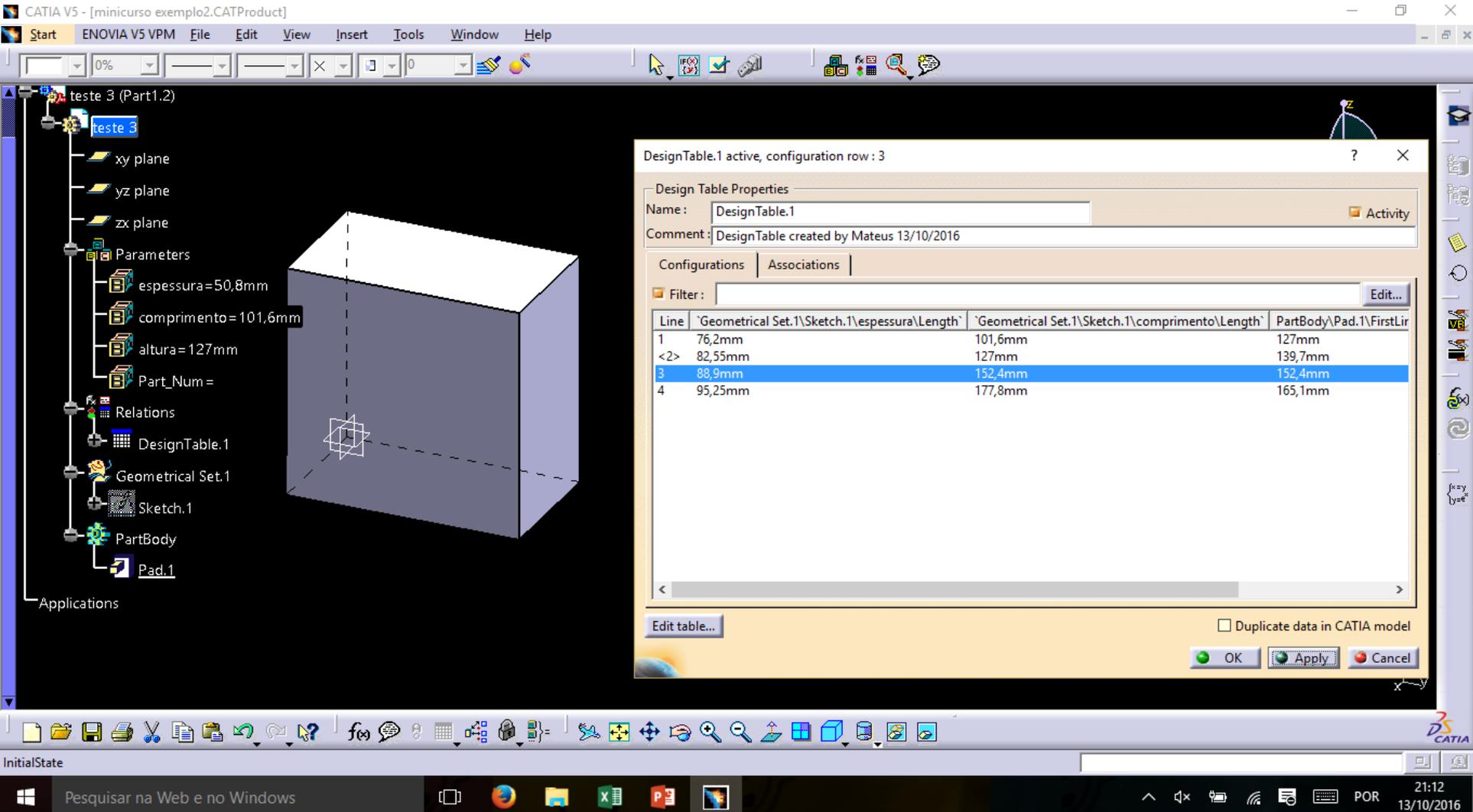
OK Apply Cancel

InitialState

Pesquisar na Web e no Windows

21:13 13/10/2016

# Mudando os parâmetros usando Relations – Design Table



CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

teste 3 (Part1.2)

- teste 3
  - xy plane
  - yz plane
  - zx plane
  - Parameters
    - espessura=50,8mm
    - comprimento=101,6mm
    - altura=127mm
    - Part\_Num =
  - Relations
    - DesignTable.1
    - Geometrical Set.1
    - Sketch.1
    - PartBody
      - Pad.1

Applications

DesignTable.1 active, configuration row : 3

Design Table Properties

Name: DesignTable.1 Activity

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter: Edit...

Line	'Geometrical Set.1\Sketch.1\espessura\Length'	'Geometrical Set.1\Sketch.1\comprimento\Length'	PartBody\Pad.1\FirstLir
1	76,2mm	101,6mm	127mm
<2>	82,55mm	127mm	139,7mm
3	88,9mm	152,4mm	152,4mm
4	95,25mm	177,8mm	165,1mm

Edit table...  Duplicate data in CATIA model

OK Apply Cancel

InitialState

Pesquisar na Web e no Windows

21:12 13/10/2016



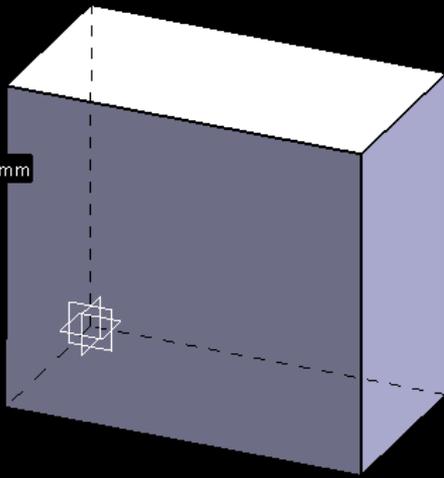
CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

0% [Navigation icons]

teste 4 (Part1.2)

- teste 4
  - xy plane
  - yz plane
  - zx plane
  - Parameters
    - espessura=50,8mm
    - comprimento=101,6mm
    - altura=127mm
    - Part\_Num=0mm
  - Relations
  - DesignTable.1
  - Geometrical Set.1
  - Sketch.1
  - PartBody
  - Pad.1



DesignTable.1 active, configuration row : 4

Design Table Properties

Name: DesignTable.1 Activity

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter:  Edit...

Line	'Geometrical Set.1\Sketch.1\espessura\Length'	'Geometrical Set.1\Sketch.1\comprimento\Length'	PartBody\Pad.1\FirstLir
1	76,2mm	101,6mm	127mm
<2>	82,55mm	127mm	139,7mm
3	88,9mm	152,4mm	152,4mm
4	95,25mm	177,8mm	165,1mm

Edit table...  Duplicate data in CATIA model

OK Apply Cancel

[Toolbars]

InitialState

# Mudando os parâmetros usando Relations – Design Table

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Parameters

- espessura=50,8mm
- comprimento=101,6mm
- altura=127mm
- Part\_Num =

Relations

- DesignTable.1
  - Configuration=2
  - Sheet.1
- Geometrical Set.1
  - Sketch.1
    - AbsoluteAxis
    - Geometry
    - Constraints
      - espessura
      - comprimento
      - Parallelism.1
      - Parallelism.2
      - Parallelism.3
      - Parallelism.4

DesignTable.1 active, configuration row : 2

Design Table Properties

Name: DesignTable.1

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter:

Line	'Geometrical Set.1\Sketch.1\espessura\Length'	'Geometrical Set.1\Sketch.1\comprimento\Length'	PartBody\Pad.1\FirstLimit\Length	teste 2\Part Number
1	76,2mm	101,6mm	127mm	teste 1
<2>	82,55mm	127mm	139,7mm	teste 2
3	88,9mm	152,4mm	152,4mm	teste 3
4	95,25mm	177,8mm	165,1mm	teste 4

Edit table...

Duplicate data in CATIA model

OK Apply Cancel

InitialState

Pesquisar na Web e no Windows

20:55  
13/10/2016

# Linkando os parâmetros criados com as cotas e constrains do modelo

The screenshot displays the CATIA V5 software interface. The main window shows a 3D model of a rectangular block with a context menu open over a dimension parameter named 'espessura object'. The menu options include 'Definition...', 'Open in New Window', 'Edit formula...', 'Hide', 'Reorder...', 'Lock', and 'Create equivalent dimensions...'. The left-hand side of the interface shows the 'Parameters' tree with 'espessura = 10' selected, and the 'Geometrical Set' tree with 'Sketch.1' selected. The bottom status bar indicates 'Creates/Edits a formula between parameters'. The Windows taskbar at the bottom shows the time as 20:56 on 13/10/2016.

# Linkando os parâmetros criados com as cotas e constrains do modelo

Formula Editor : espessura

espessura =

'Geometrical Set.1\Sketch.1\espessura\Length'

Dictionary	Members of Parameters	Members of Length
Parameters	All	'Geometrical Set.1\Sketch.1\espessura\Length'
Design Table	Renamed parameter	'Geometrical Set.1\Sketch.1\comprimento\Length'
Operators	Boolean	PartBody\Pad.1\FirstLimit\Length
Pointer on value function	CstAttr_Mode	PartBody\Pad.1\SecondLimit\Length
NC Manufacturing	Length	PartBody\Pad.1\ThickThin1
Point Constructors	Integer	PartBody\Pad.1\ThickThin2
Law	String	espessura
Operations Constructors	Feature	comprimento
Line Constructors	Plane	altura
Circle Constructors	Solid	Part_Num
String	Curve	
Direction Constructors	Constraint	
Electrical Functions	Set of parameters	
List	Set Of Relations	
Measures	Design Table	
Surface Constructors	Sheet	
Wireframe Constructors		
NC Manufacturing resour		
Generative functions		
Part Measures		
Plane Constructors		

Geometrical Set.1\Sketch.1\espessura\Length 76,2mm

OK Cancel

**3 - Duplo clique!**

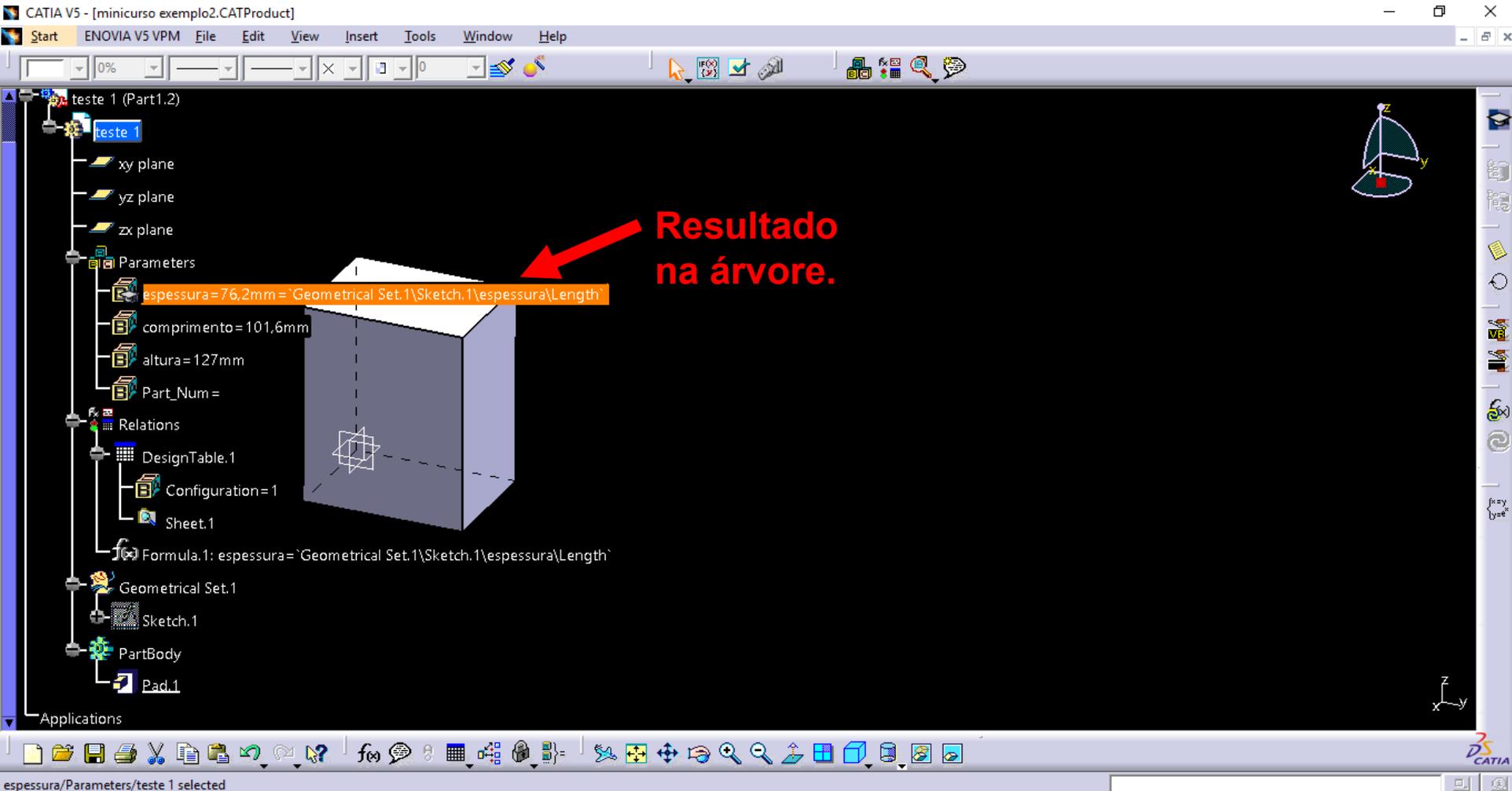
# Linkando os parâmetros criados com as cotas e constrains do modelo

The screenshot displays the CATIA V5 interface. On the left, the 'teste 1 (Part.1.2)' tree shows parameters: `espessura=50,8mm`, `comprimento=101,6mm`, `altura=127mm`, and `Part_Num =`. A 3D model of a rectangular part is shown with a dimension of 76.2 mm. The 'Formula Editor : espessura' window is open, showing the formula: `espessura = 'Geometrical Set.1\Sketch.1\espessura\Length'`. A red arrow points from the text 'Resultado' to the result value '76,2mm' in the formula editor. The dictionary lists various categories, and the 'Members of Length' list includes the selected formula path.

Dictionary	Members of Parameters	Members of Length
Parameters	All	'Geometrical Set.1\Sketch.1\espessura\Length'
Design Table	Renamed parameters	'Geometrical Set.1\Sketch.1\comprimento\Length'
Operators	Boolean	PartBody\Pad.1\FirstLimit\Length
Pointer on value function	CstAttr_Mode	PartBody\Pad.1\SecondLimit\Length
NC Manufacturing	Length	PartBody\Pad.1\ThickThin1
Point Constructors	Integer	PartBody\Pad.1\ThickThin2
Law	String	espessura
Operations Constructors	Feature	comprimento
Line Constructors	Plane	altura
Circle Constructors	Solid	Part_Num
String	Curve	
Direction Constructors	Constraint	
Electrical Functions	Set of parameters	
List	Set Of Relations	
Measures	Design Table	
Surface Constructors	Sheet	
Wireframe Constructors		
NC Manufacturing resour		
Generative functions		
Part Measures		
Plane Constructors		



# Linkando os parâmetros criados com as cotas e constrains do modelo



**Resultado na árvore.**

espessura/Parameters/teste 1 selected



Linkando os parâmetros criados com as cotas e constrains do modelo – fazer para todos.

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

0%

teste 1 (Part1.2)

- teste 1
  - xy plane
  - yz plane
  - zx plane
  - Parameters
    - espessura=76,2mm = 'Geometrical Set.1\Sketch.1\espessura\Length'
    - comprimento=101,6mm = 'Geometrical Set.1\Sketch.1\comprimento\Length'
    - altura=127mm
    - Part\_Num =
  - Relations
    - DesignTable.1
      - Configuration=1
      - Sheet.1
    - Formula.1: espessura='Geometrical Set.1\Sketch.1\espessura\Length'
    - Formula.2: comprimento='Geometrical Set.1\Sketch.1\comprimento\Length'
  - Geometrical Set.1
    - Sketch.1
    - PartBody
      - Pad.1

Formula Editor : altura

altura =

PartBody\Pad.1\FirstLimit\Length

Dictionary	Members of Parameters	Members of Length
Parameters	All	'Geometrical Set.1\Sketch.1\espessura\Length'
Design Table	Renamed parameters	'Geometrical Set.1\Sketch.1\comprimento\Length'
Operators	Boolean	PartBody\Pad.1\FirstLimit\Length
Pointer on value function	CstAttr_Mode	PartBody\Pad.1\SecondLimit\Length
NC Manufacturing	Length	PartBody\Pad.1\ThickThin1
Point Constructors	Integer	PartBody\Pad.1\ThickThin2
Law	String	espessura
Operations Constructors	Feature	comprimento
Line Constructors	Plane	altura
Circle Constructors	Solid	Part_Num
String	Curve	
Direction Constructors	Constraint	
Electrical Functions	Set of parameters	
List	Set Of Relations	
Measures	Design Table	
Surface Constructors	Formula	
Wireframe Constructors	Sheet	
NC Manufacturing resour		
Generative functions		
Part Measures		
Plane Constructors		

PartBody\Pad.1\FirstLimit\Length 127mm

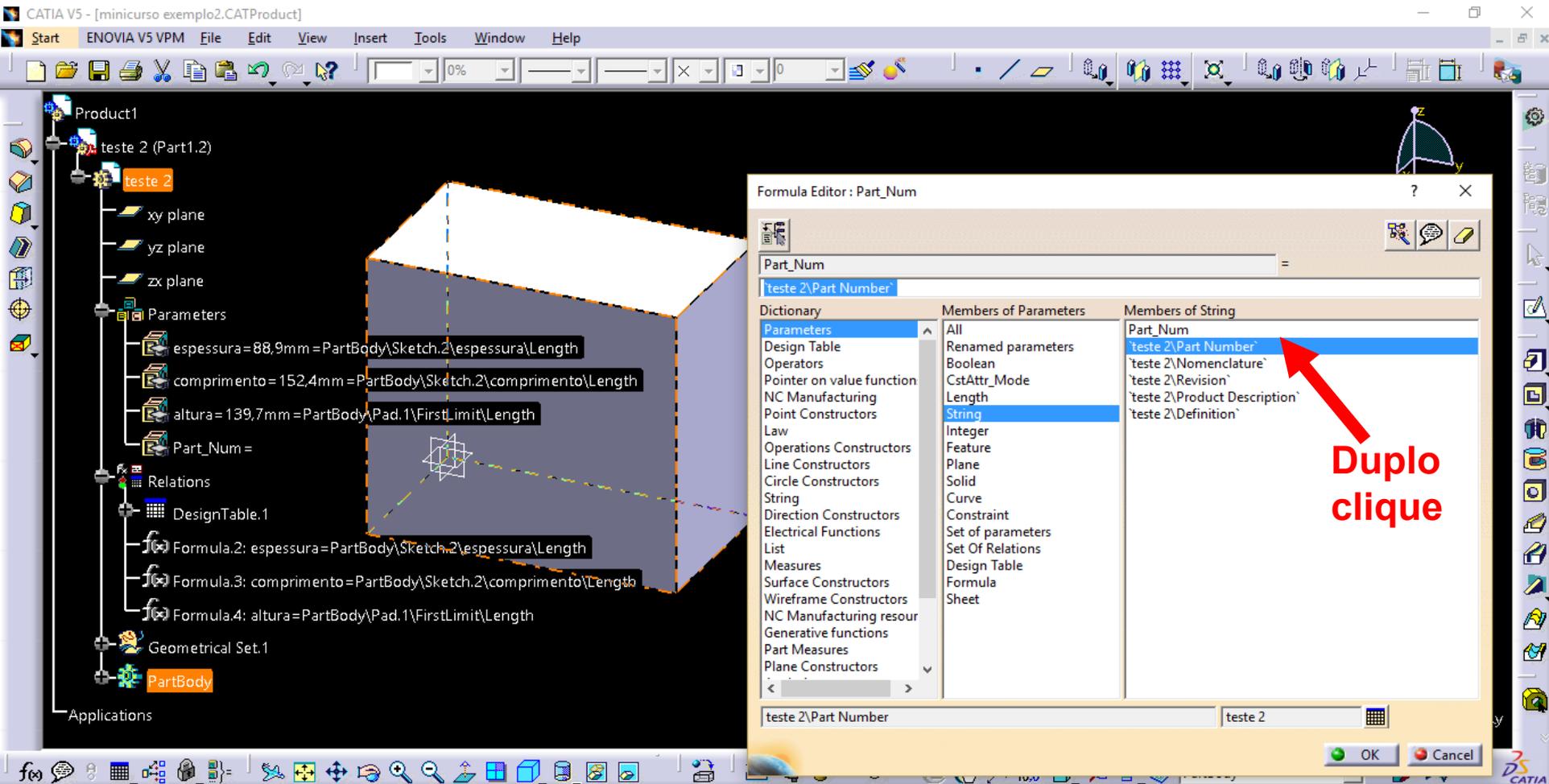
OK Cancel

Linkando os parâmetros criados com as cotas e constrains do modelo – fazer para todos – Tipo string.

The screenshot displays the CATIA V5 interface. On the left, the CAD tree shows a product structure with parameters defined as strings, such as `espessura=3750mm='teste 4\Geometrical Set.1\Sketch.1\espessura\Length'`. The main 3D view shows a part with a dashed line indicating a constraint or dimension. The 'Formula Editor' dialog is open, showing the configuration for the 'Part Num' parameter. The 'Dictionary' list includes 'Parameters', and the 'Members of String' list contains several string-based parameters from the model.

Dictionary	Members of Parameters	Members of String
Parameters	All	'teste 4\Part Number'
Design Table	Renamed parameters	'teste 4\Nomenclature'
Operators	Boolean	'teste 4\Revision'
Pointer on value function	CstAttr_Mode	'teste 4\Product Description'
NC Manufacturing	Length	'teste 4\Definition'
Point Constructors	Integer	
Law	String	
Operations Constructors	Feature	
Line Constructors	Plane	
Circle Constructors	Solid	
String	Curve	
Direction Constructors	Constraint	
Electrical Functions	Set of parameters	
List	Set Of Relations	
Measures	Design Table	
Surface Constructors	Formula	
Wireframe Constructors	Sheet	
NC Manufacturing resour		
Generative functions		
Part Measures		
Plane Constructors		

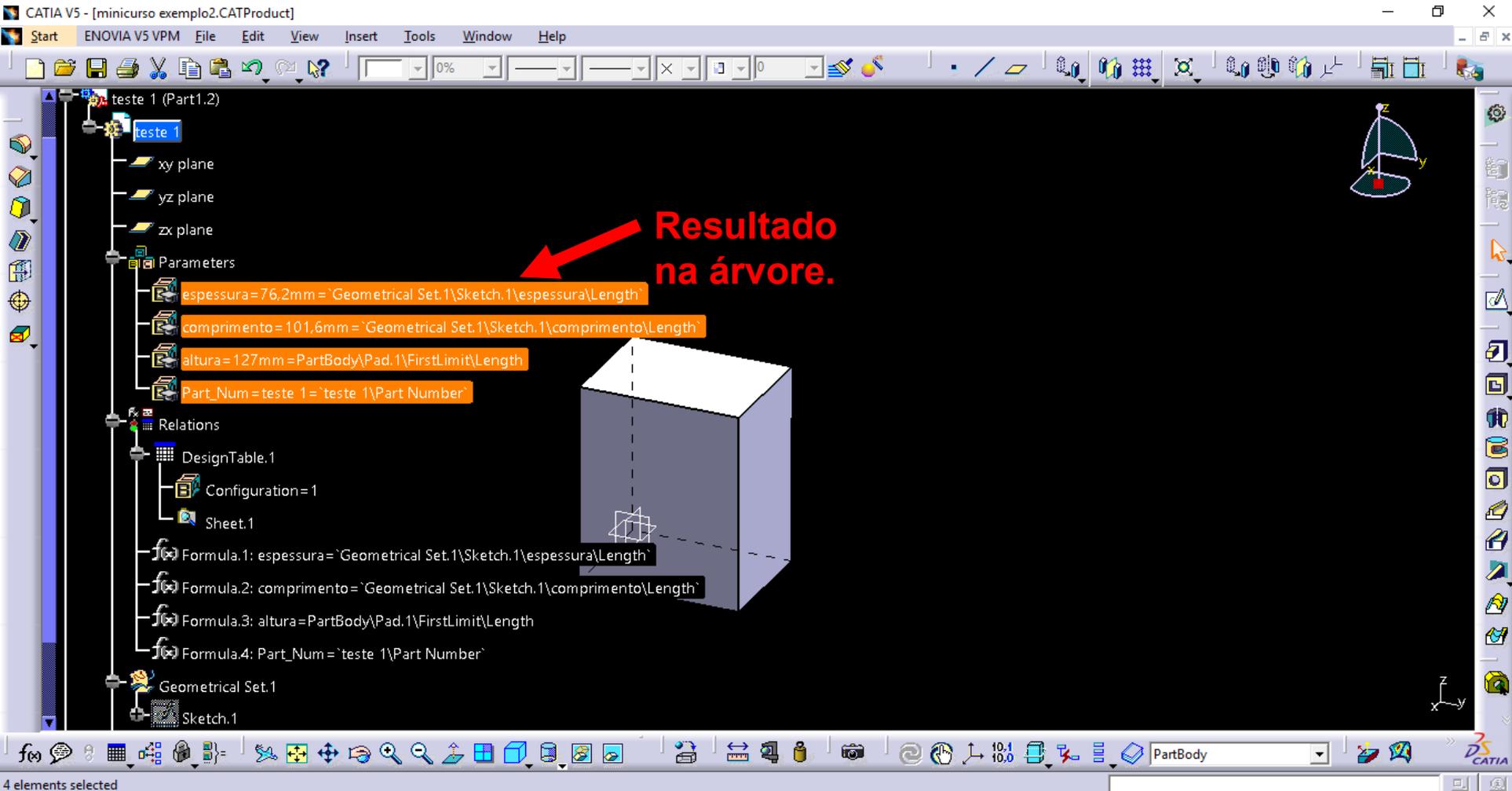
Linkando os parâmetros criados com as cotas e constrains do modelo – fazer para todos – Tipo string.



The screenshot shows the CATIA V5 interface with a 3D model of a rectangular part. The left-hand tree structure includes 'Product1', 'teste 2 (Part1.2)', 'teste 2', 'xy plane', 'yz plane', 'zx plane', 'Parameters', 'Relations', 'DesignTable.1', and 'PartBody'. The 'Parameters' list includes 'espessura=88,9mm=PartBody\Sketch.2\espessura\Length', 'comprimento=152,4mm=PartBody\Sketch.2\comprimento\Length', and 'altura=139,7mm=PartBody\Pad.1\FirstLimit\Length'. The 'Relations' list includes 'Formula.2: espessura=PartBody\Sketch.2\espessura\Length', 'Formula.3: comprimento=PartBody\Sketch.2\comprimento\Length', and 'Formula.4: altura=PartBody\Pad.1\FirstLimit\Length'. The 'PartBody' is highlighted in orange.

The 'Formula Editor: Part\_Num' dialog box is open, showing the formula 'Part\_Num = 'teste 2\Part Number''. The 'Dictionary' list includes 'Parameters', 'Design Table', 'Operators', 'Pointer on value function', 'NC Manufacturing', 'Point Constructors', 'Law', 'Operations Constructors', 'Line Constructors', 'Circle Constructors', 'String', 'Direction Constructors', 'Electrical Functions', 'List', 'Measures', 'Surface Constructors', 'Wireframe Constructors', 'NC Manufacturing resour', 'Generative functions', 'Part Measures', and 'Plane Constructors'. The 'Members of Parameters' list includes 'All', 'Renamed parameters', 'Boolean', 'CstAttr\_Mode', 'Length', 'String', 'Integer', 'Feature', 'Plane', 'Solid', 'Curve', 'Constraint', 'Set of parameters', 'Set Of Relations', 'Design Table', 'Formula', and 'Sheet'. The 'Members of String' list includes 'Part\_Num', 'teste 2\Part Number', 'teste 2\Nomenclature', 'teste 2\Revision', 'teste 2\Product Description', and 'teste 2\Definition'. A red arrow points to the 'teste 2\Part Number' entry in the 'Members of String' list with the text 'Duplo clique'.

Linkando os parâmetros criados com as cotas e constrains do modelo – fazer para todos – Tipo string.



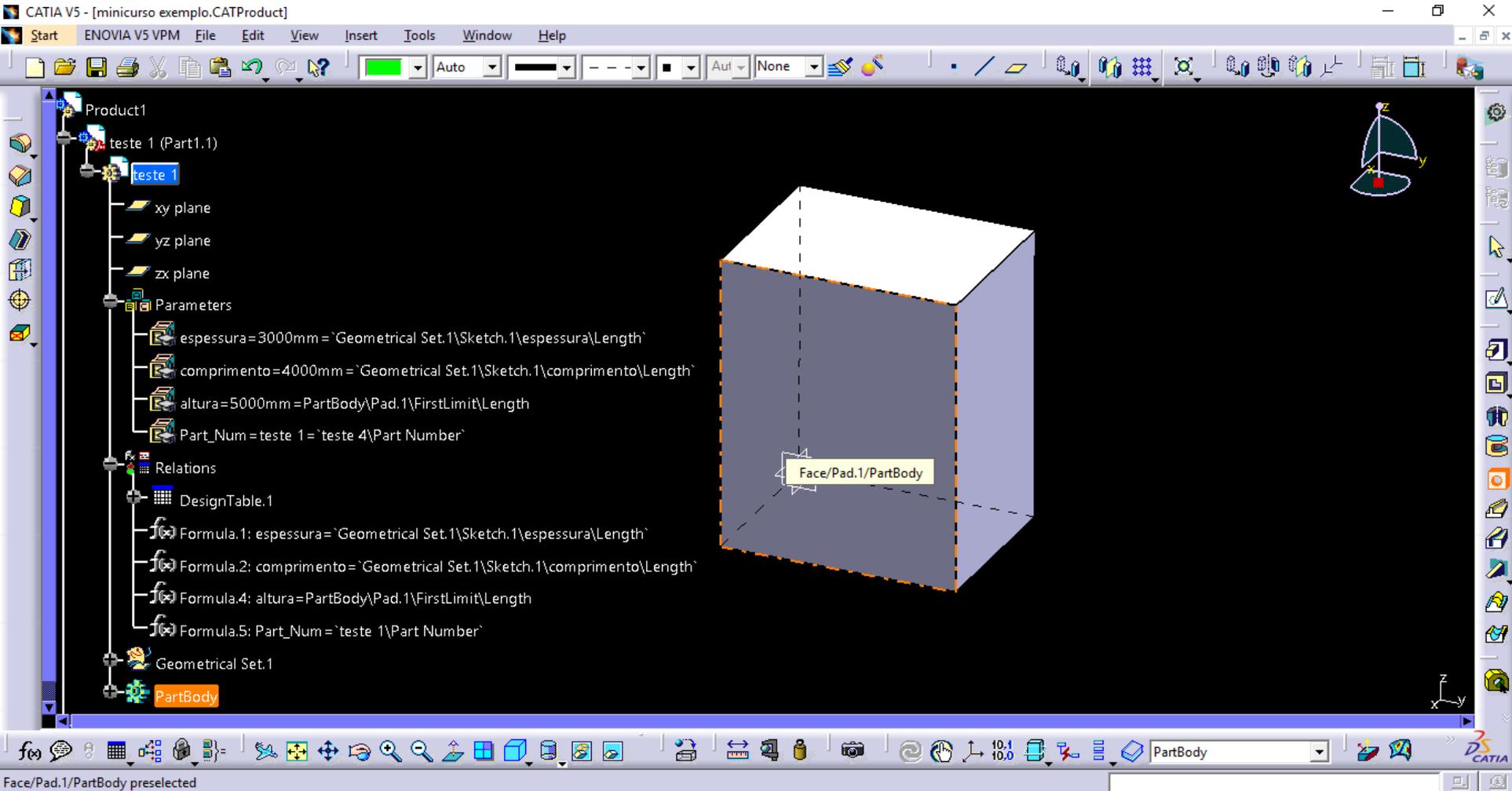


# Inserindo um furo

Parâmetro “diâmetro do furo” e setando a sua posição



# Gerando um furo usando HOLE na face frontal do modelo



Face/Pad.1/PartBody preselected

# Gerando um furo usando HOLE na face frontal do modelo

The image shows the CATIA V5 interface for creating a hole. The 'Hole Definition' dialog box is open, with the following settings:

- Extension: Blind
- Type: Blind
- Thread Definition: (empty)
- Diameter: 25mm
- Depth: 25mm
- Limit: No selection
- Offset: 0mm
- Direction: Normal to surface
- Bottom: Flat, Angle: 120deg

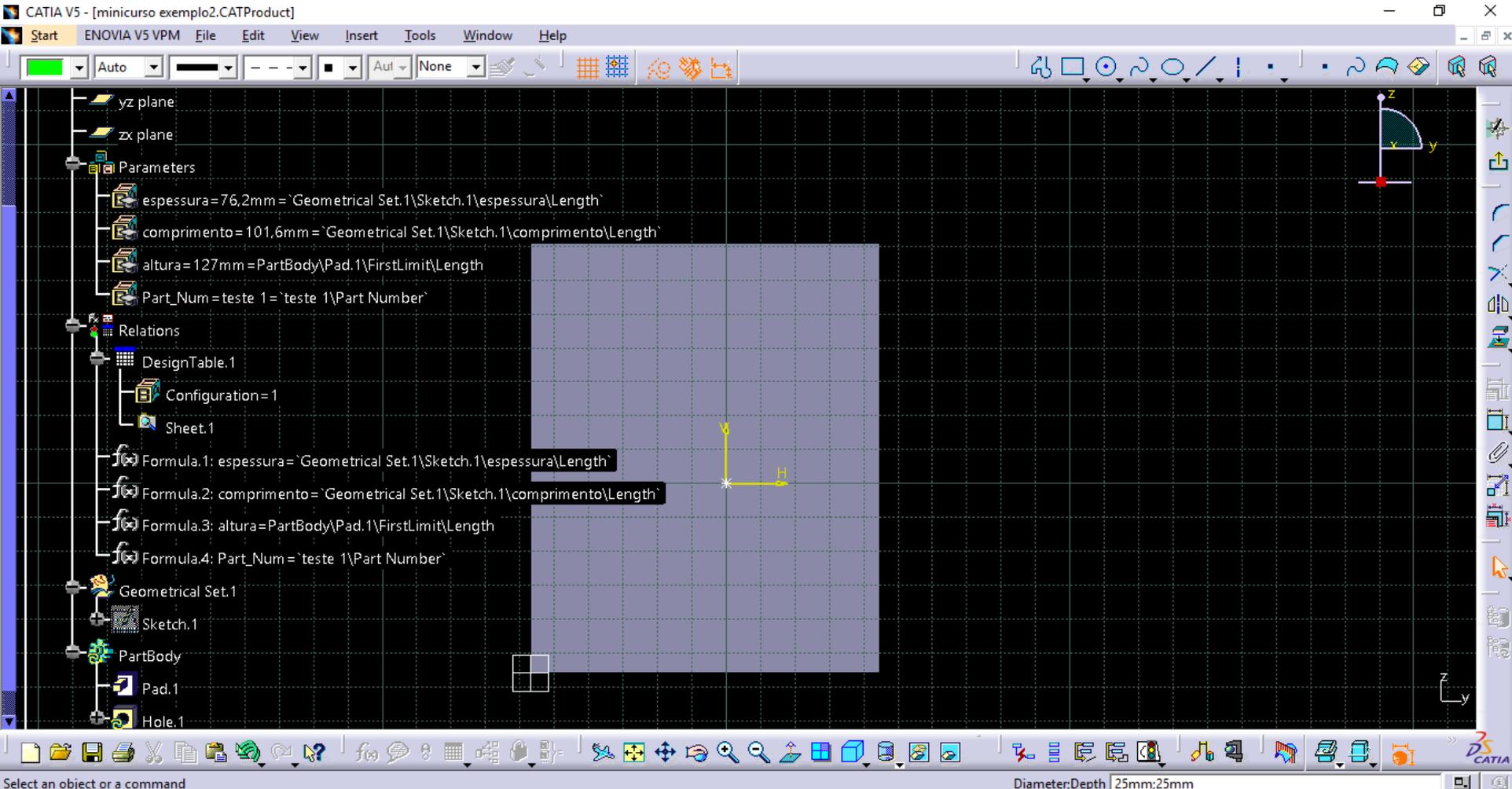
Red arrows in the image point to the 'Blind' type dropdown (labeled 1) and the 'Positioning Sketch' icon (labeled 2). The 3D model shows a rectangular part with a coordinate system and labels 'BOTTOM' and 'HOLE'.

Enter new data to define the hole

Diameter;Depth | 25mm;25mm

# Gerando um furo usando HOLE

## Setagem das constrains de posição do furo.



Select an object or a command

Diameter;Depth | 25mm;25mm



# Gerando um furo usando HOLE

## Setagem das constrains de posição do furo.

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Auto None

yz plane

zx plane

Parameters

- espessura=76,2mm = 'Geometrical Set.1\Sketch.1\espessura\Length'
- comprimento=101,6mm = 'Geometrical Set.1\Sketch.1\comprimento\Length'
- altura=127mm = PartBody\Pad.1\FirstLimit\Length
- Part\_Num = teste 1 = 'teste 1\Part Number'

Relations

- DesignTable.1
  - Configuration=1
  - Sheet.1
- Formula.1: espessura='Geometrical Set.1\Sketch.1\espessura\Length'
- Formula.2: comprimento='Geometrical Set.1\Sketch.1\comprimento\Length'
- Formula.3: altura=PartBody\Pad.1\FirstLimit\Length
- Formula.4: Part\_Num='teste 1\Part Number'

Geometrical Set.1

- Sketch.1
- PartBody
- Pad.1
- Hole.1

71,077

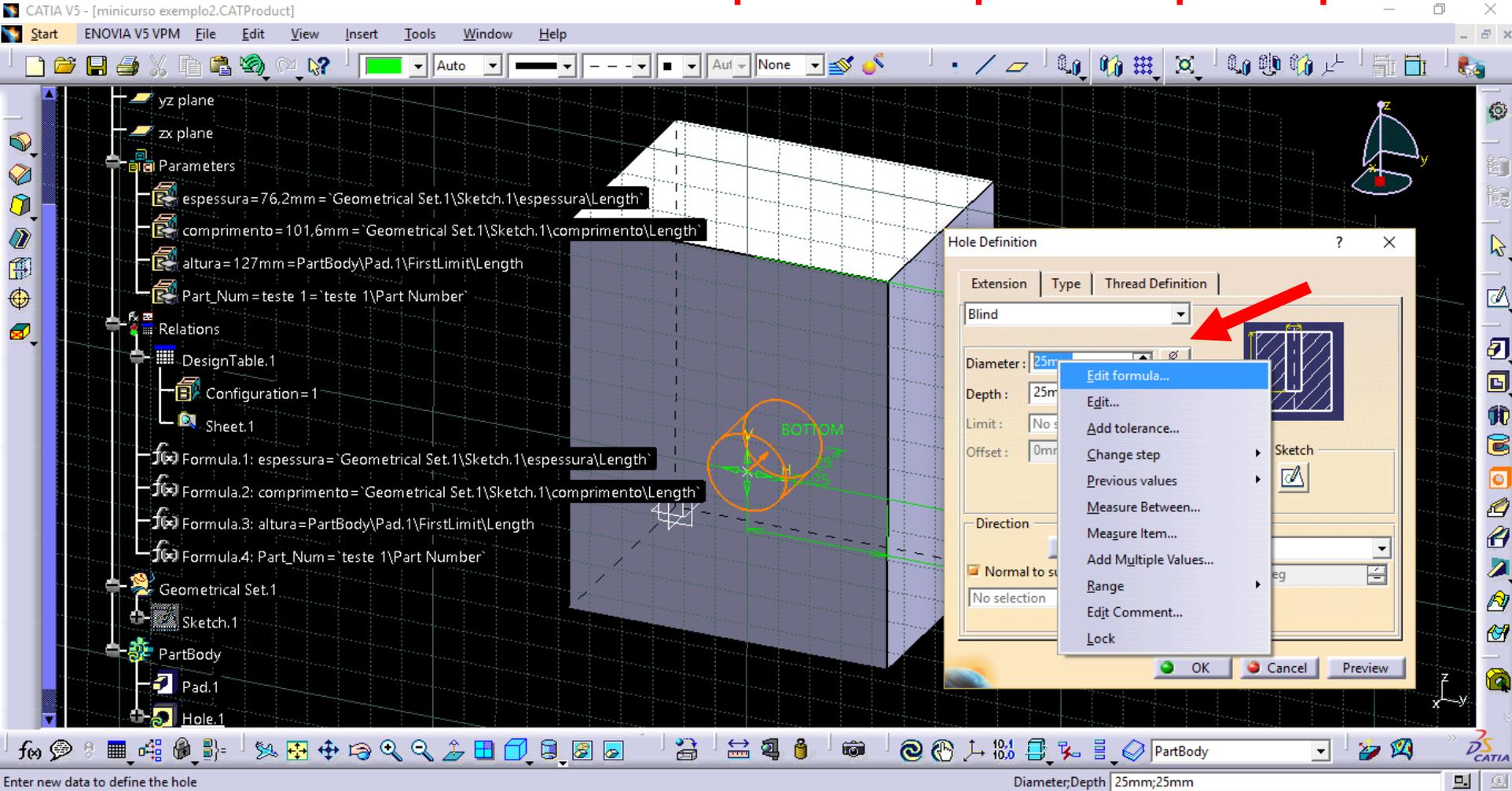
44,738

Diameter;Depth | 25mm;25mm

Select an object or a command

Inserção do parâmetro dia furo. Botão dir sobre Diameter...

Sem parâmetros para usar por enquanto.



Enter new data to define the hole

Diameter;Depth | 25mm;25mm

# Inserção do parâmetro dia furo.

## Associando primeiro o parâmetro “dia furo”.

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Parameters

- espessura=76,2mm = 'Geometrical Set.1\Sketch.1\espessura\Length'
- comprimento=101,6mm = 'Geometrical Set.1\Sketch.1\comprimento\Length'
- altura=127mm = PartBody\Pad.1\FirstLimit\Length
- Part\_Num = teste 1 = 'teste 1\Part Number'

Relations

- \*DesignTable.1
- Configuration=1
- Sheet.1
- Formula.1: espessura = 'Geometrical Set.1\Sketch.1\espessura\Length'
- Formula.2: comprimento = 'Geometrical Set.1\Sketch.1\comprimento\Length'
- Formula.3: altura = PartBody\Pad.1\FirstLimit\Length
- Formula.4: Part\_Num = 'teste 1\Part Number'

Geometrical Set.1

- Sketch.1
- PartBody
- Pad.1
- Hole.1

Applications

\*DesignTable.1 active, configuration row : 1

Design Table Properties

Name: \*DesignTable.1

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter: Edit...

Line	'Geometrical Set.1\Sketch.1\espessura\Length'	'Geometrical Set.1\Sketch.1\comprimento\Length'	PartBody\Pad.1\FirstLimit\Length
< 1 >	76,2mm	101,6mm	127mm
2	82,55mm	127mm	139,7mm
3	88,9mm	152,4mm	152,4mm
4	95,25mm	177,8mm	165,1mm

Edit table...

Push this button to edit the design table values

OK Apply Cancel

Push this button to edit the design table values

Pesquisar na Web e no Windows

22:18 13/10/2016



CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help



minicurso catia knowledge.xlsx - Excel Mateus Miranda

Arquivo Página Inicial Inserir Desenhar Layout da Página Fórmulas Dados Revisão Exibir Diga-me o que você deseja fazer Compartilhar

Fonte: Calibri 11

Alinhamento: Geral

Número: % 000

Estilo: Formatação Condicional, Formatar como Tabela, Estilos de Célula

Células: Inserir, Excluir, Formatar

Edição: Classificar e Filtrar, Localizar e Selecionar

Área de Trabalho: E1 dia furo (mm)

Part Num	espessura (mm)	comprimento (in)	altura	dia furo (mm)
teste 1	76,2	4	127mm	25,4
teste 2	82,55	5	5,5in	31,75
teste 3	88,9	6	152,4mm	38,1
teste 4	95,25	7	6,5in	44,45

Planilha1

Pronto Média: 34,925 Contagem: 5 Soma: 139,7

Activity

Edit...

PartBody\Pad.1\FirstLir

- 127mm
- 139,7mm
- 152,4mm
- 165,1mm

Duplicate data in CATIA model

OK Apply Cancel



# Inserção do parâmetro dia furo.

## Associando primeiro o parâmetro “dia furo”.

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Parameters

- espessura=76,2mm = 'Geometrical Set.1\Sketch.1\espessura\Length'
- comprimento=101,6mm = 'Geometrical Set.1\Sketch.1\comprimento\Length'
- altura=127mm = PartBody\Pad.1\FirstLimit\Length
- Part\_Num = teste 1 = 'teste 1\Part Number'

Relations

- \*DesignTable.1
  - Configuration=1
  - Sheet.1
  - Formula.1: espessura = 'Geometrical Set.1\Sketch.1\espessura\Length'
  - Formula.2: comprimento = 'Geometrical Set.1\Sketch.1\comprimento\Length'
  - Formula.3: altura = PartBody\Pad.1\FirstLimit\Length
  - Formula.4: Part\_Num = 'teste 1\Part Number'
- Geometrical Set.1
  - Sketch.1
  - PartBody
    - Pad.1
    - Hole.1

Applications

\*DesignTable.1 active, configuration row : 1

Design Table Properties

Name: \*DesignTable.1

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter On teste 1

Filter Name: \*

Filter Type: All

Parameters	Columns
'Geometrical Set.1\Sketch.1\AbsoluteA	dia furo
'Geometrical Set.1\Sketch.1\Paralleli	

Associations between parameters and columns

Parameters	Columns
'Geometrical Set.1\Sketch.1\...	espessura
'Geometrical Set.1\Sketch.1\...	comprimento
PartBody\Pad.1\FirstLimit\L...	altura
'teste 1\Part Number'	Part Num

Associate Dissociate

Create parameters... Rename associated parameters

Edit table...

Duplicate data in CATIA model

OK Apply Cancel

InitialState

Pesquisar na Web e no Windows

22:21 13/10/2016

# Inserção do parâmetro dia furo. Associando primeiro o parâmetro “dia furo” – HOLE/diameter.

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Parameters

- espessura = 76,2mm = 'Geometrical Set.1\Sketch.1\espessura\Length'
- comprimento = 101,6mm = 'Geometrical Set.1\Sketch.1\comprimento'
- altura = 127mm = PartBody\Pad.1\FirstLimit\Length
- Part\_Num = teste 1 = 'teste 1\Part Number'

Relations

- \*DesignTable.1
  - Configuration = 1
  - Sheet.1
  - Formula.1: espessura = 'Geometrical Set.1\Sketch.1\espessura\Length'
  - Formula.2: comprimento = 'Geometrical Set.1\Sketch.1\comprimento'
  - Formula.3: altura = PartBody\Pad.1\FirstLimit\Length
  - Formula.4: Part\_Num = 'teste 1\Part Number'
- Geometrical Set.1
  - Sketch.1
- PartBody
  - Pad.1
  - Hole.1

Applications

\*DesignTable.1 active, configuration row : 1

Design Table Properties

Name: \*DesignTable.1

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter On teste 1

Filter Name: \*

Filter Type: Length

Parameters	Columns
'Geometrical Set.1\Sketch.1\es...	espessura
'Geometrical Set.1\Sketch.1\co...	comprimento
PartBody\Pad.1\FirstLimit\Len...	altura
teste 1\Part Number	Part Num
PartBody\Hole.1\Diameter	dia furo

PartBody\Pad.1\SecondLimit\Length

PartBody\Pad.1\ThickThin1

PartBody\Pad.1\ThickThin2

PartBody\Hole.1\HoleLimit.1\Depth

PartBody\Hole.1\HoleLimit.1\Offset

PartBody\Hole.1\Sketch.3\Offset.1\Offset

PartBody\Hole.1\Sketch.3\Offset.2\Offset

PartBody\Hole.1\Tap depth

Associate Dissociate

Create parameters... Rename associated parameters

Edit table...

Duplicate data in CATIA model

OK Apply Cancel

1 - clica

2 - seleciona Hole/diameter

3 - clica em Associate

# Inserção do parâmetro dia furo. Associando primeiro o parâmetro “dia furo” – HOLE/diameter.

DesignTable.1 active, configuration row : 1

Design Table Properties

Name: DesignTable.1

Comment: DesignTable created by Mateus 10/10/2016

Configurations Associations

Filter On teste 1

Filter Name: \*

Filter Type: Length

Parameters	Columns
'Geometrical Set.1\Sketch.1\...	espessura
'Geometrical Set.1\Sketch.1\...	comprimento
PartBody\Pad.1\FirstLimit\...	altura
'teste 1\Part Number'	Part Num
PartBody\Hole.1\Diameter	dia furo

Associate Dissociate

Create parameters... Rename associated parameters

Edit table... Duplicate data in CATIA model

OK Apply Cancel

List of associations between columns of the design table and parameters of the model



# Inserção do parâmetro dia furo.

## Associando primeiro o parâmetro “dia furo” – HOLE/diameter.

CATIA V5 - [minicurso\_exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Parameters

- espessura=76,2mm = `Geometrical Set.1\Sketch.1\espessura\Length`
- comprimento=101,6mm = `Geometrical Set.1\Sketch.1\comprimento\Length`
- altura=127mm = PartBody\Pad.1\FirstLimit\Length
- Part\_Num = teste 1 = `teste 1\Part Number`

Relations

- \*DesignTable.1
  - Configuration=1
  - Sheet.1
- Formula.1: espessura = `Geometrical Set.1\Sketch.1\espessura\Length`
- Formula.2: comprimento = `Geometrical Set.1\Sketch.1\comprimento\Length`
- Formula.3: altura = PartBody\Pad.1\FirstLimit\Length
- Formula.4: Part\_Num = `teste 1\Part Number`

Geometrical Set.1

- Sketch.1
- PartBody
  - Pad.1
  - Hole.1

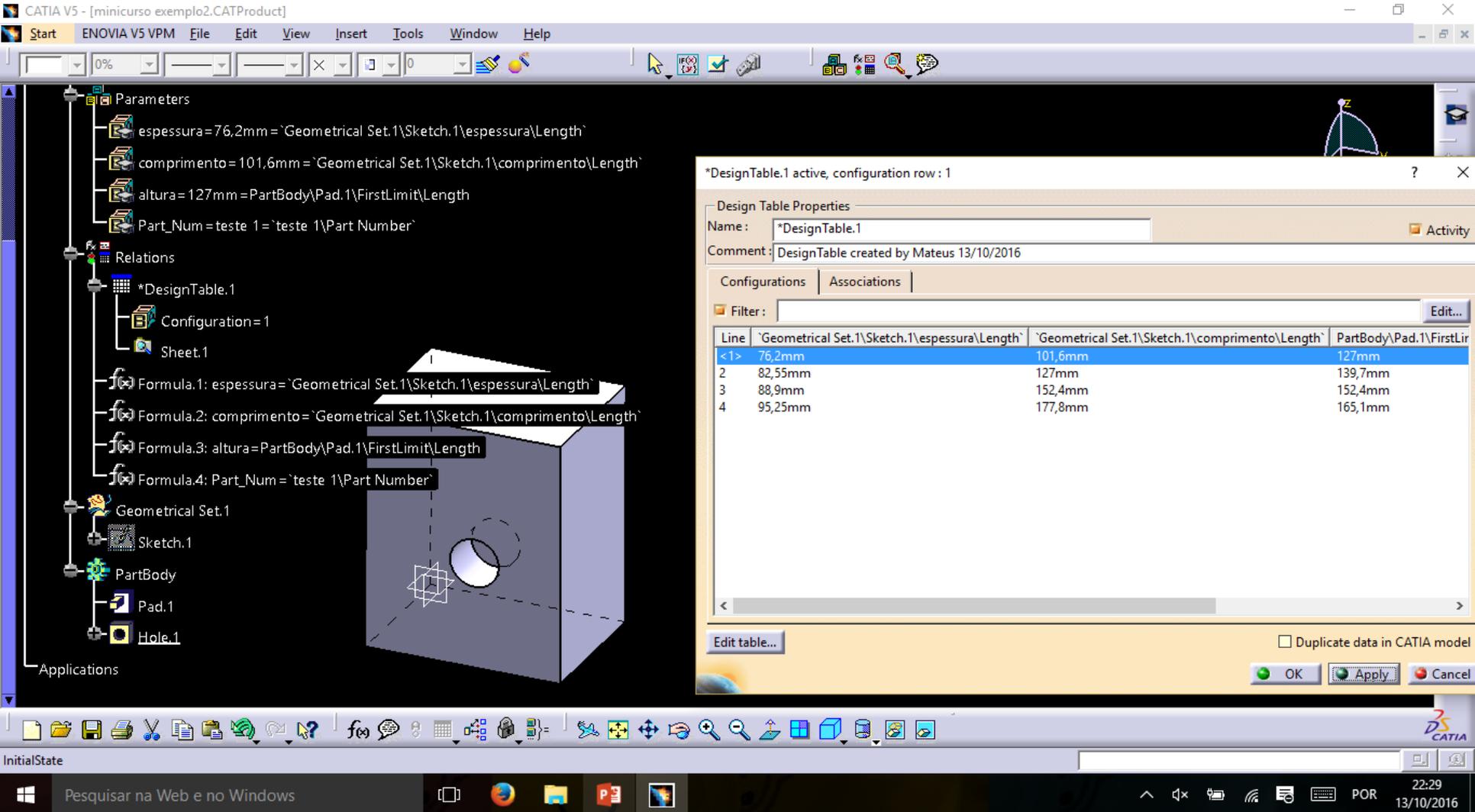
Applications

Select an object or a command

Pesquisar na Web e no Windows

22:26 13/10/2016

# Verificação do resultado usando o Design Table



CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Parameters

- espessura=76,2mm = 'Geometrical Set.1\Sketch.1\espessura\Length'
- comprimento=101,6mm = 'Geometrical Set.1\Sketch.1\comprimento\Length'
- altura=127mm = PartBody\Pad.1\FirstLimit\Length
- Part\_Num = teste 1 = 'teste 1\Part Number'

Relations

- \*DesignTable.1
  - Configuration=1
  - Sheet.1
  - Formula.1: espessura='Geometrical Set.1\Sketch.1\espessura\Length'
  - Formula.2: comprimento='Geometrical Set.1\Sketch.1\comprimento\Length'
  - Formula.3: altura=PartBody\Pad.1\FirstLimit\Length
  - Formula.4: Part\_Num='teste 1\Part Number'
- Geometrical Set.1
  - Sketch.1
  - PartBody
    - Pad.1
    - Hole.1

Applications

\*DesignTable.1 active, configuration row: 1

Design Table Properties

Name: \*DesignTable.1

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter:

Line	'Geometrical Set.1\Sketch.1\espessura\Length'	'Geometrical Set.1\Sketch.1\comprimento\Length'	PartBody\Pad.1\FirstLir
<1>	76,2mm	101,6mm	127mm
2	82,55mm	127mm	139,7mm
3	88,9mm	152,4mm	152,4mm
4	95,25mm	177,8mm	165,1mm

OK Apply Cancel

22:29 13/10/2016



# Verificação do resultado usando o Design Table

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

0%

Parameters

- espessura=82,55mm = 'Geometrical Set.1\Sketch.1\espessura\Length'
- comprimento=127mm = 'Geometrical Set.1\Sketch.1\comprimento\Length'
- altura=139,7mm = PartBody\Pad.1\FirstLimit\Length
- Part\_Num = teste 2 = 'teste 2\Part Number'

Relations

- \*DesignTable.1
  - Configuration=2
  - Sheet.1
  - Formula.1: espessura='Geometrical Set.1\Sketch.1\espessura\Length'
  - Formula.2: comprimento='Geometrical Set.1\Sketch.1\comprimento\Length'
  - Formula.3: altura=PartBody\Pad.1\FirstLimit\Length
  - Formula.4: Part\_Num='teste 2\Part Number'
- Geometrical Set.1
  - Sketch.1
  - PartBody
    - Pad.1
    - Hole.1

Applications

\*DesignTable.1 active, configuration row: 2

Design Table Properties

Name: \*DesignTable.1

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter:

Line	'Geometrical Set.1\Sketch.1\espessura\Length'	'Geometrical Set.1\Sketch.1\comprimento\Length'	PartBody\Pad.1\FirstLir
<1>	76,2mm	101,6mm	127mm
2	82,55mm	127mm	139,7mm
3	88,9mm	152,4mm	152,4mm
4	95,25mm	177,8mm	165,1mm

Edit table...

Duplicate data in CATIA model

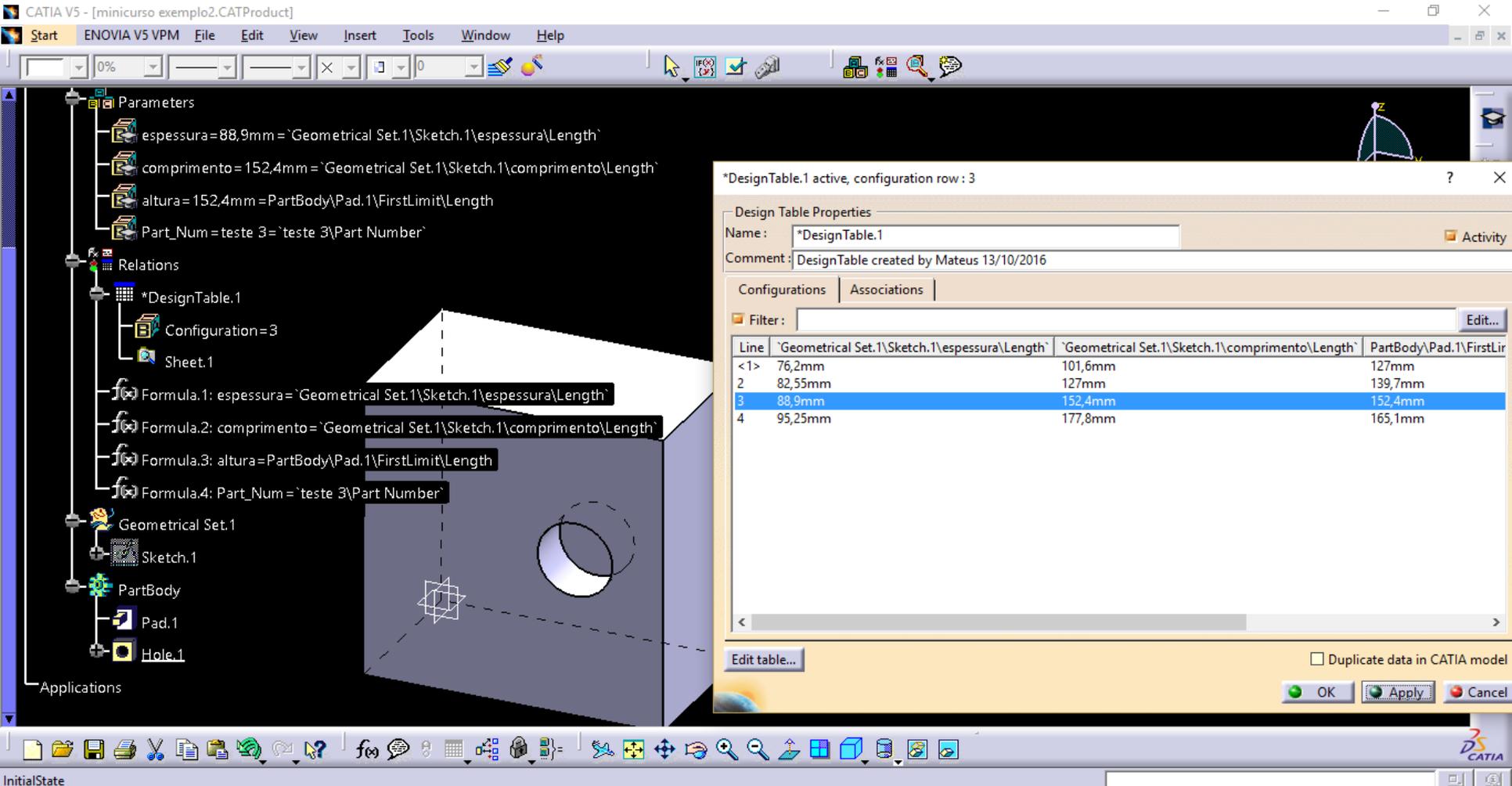
OK Apply Cancel

InitialState

Pesquisar na Web e no Windows

22:29 13/10/2016

# Verificação do resultado usando o Design Table



CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Parameters

- espessura=88,9mm = 'Geometrical Set.1\Sketch.1\espessura\Length'
- comprimento=152,4mm = 'Geometrical Set.1\Sketch.1\comprimento\Length'
- altura=152,4mm = PartBody\Pad.1\FirstLimit\Length
- Part\_Num = teste 3 = 'teste 3\Part Number'

Relations

- \*DesignTable.1
  - Configuration=3
  - Sheet.1
  - Formula.1: espessura='Geometrical Set.1\Sketch.1\espessura\Length'
  - Formula.2: comprimento='Geometrical Set.1\Sketch.1\comprimento\Length'
  - Formula.3: altura=PartBody\Pad.1\FirstLimit\Length
  - Formula.4: Part\_Num='teste 3\Part Number'

Geometrical Set.1

- Sketch.1
- PartBody
  - Pad.1
  - Hole.1

Applications

\*DesignTable.1 active, configuration row: 3

Design Table Properties

Name: \*DesignTable.1

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter:  Edit...

Line	'Geometrical Set.1\Sketch.1\espessura\Length'	'Geometrical Set.1\Sketch.1\comprimento\Length'	PartBody\Pad.1\FirstLir
<1>	76,2mm	101,6mm	127mm
2	82,55mm	127mm	139,7mm
3	88,9mm	152,4mm	152,4mm
4	95,25mm	177,8mm	165,1mm

Edit table...  Duplicate data in CATIA model

OK Apply Cancel

# Verificação do resultado usando o Design Table

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Parameters

- espessura=95,25mm = `Geometrical Set.1\Sketch.1\espessura\Length`
- comprimento=177,8mm = `Geometrical Set.1\Sketch.1\comprimento\Length`
- altura=165,1mm = `PartBody\Pad.1\FirstLimit\Length`
- Part\_Num = teste 4 = `teste 4\Part Number`

Relations

- \*DesignTable.1
  - Configuration=4
  - Sheet.1
  - Formula.1: espessura = `Geometrical Set.1\Sketch.1\espessura\Length`
  - Formula.2: comprimento = `Geometrical Set.1\Sketch.1\comprimento\Length`
  - Formula.3: altura = `PartBody\Pad.1\FirstLimit\Length`
  - Formula.4: Part\_Num = `teste 4\Part Number`
- Geometrical Set.1
  - Sketch.1
  - PartBody
    - Pad.1
    - Hole.1

Applications

\*DesignTable.1 active, configuration row : 4

Design Table Properties

Name: \*DesignTable.1

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter:

Line	`Geometrical Set.1\Sketch.1\espessura\Length`	`Geometrical Set.1\Sketch.1\comprimento\Length`	PartBody\Pad.1\FirstLir
<1>	76,2mm	101,6mm	127mm
2	82,55mm	127mm	139,7mm
3	88,9mm	152,4mm	152,4mm
4	95,25mm	177,8mm	165,1mm

This list contains the possible configurations of the design table. The selected row is the current one.

Edit table...  Duplicate data in CATIA model

OK Apply Cancel

This list contains the available configurations of the design table. The selected row is the current one.

# Linkando as constrains da posição do furo com os parâmetros

The image shows the CATIA V5 software interface. On the left, the 'Parameters' tree is visible, containing the following items:

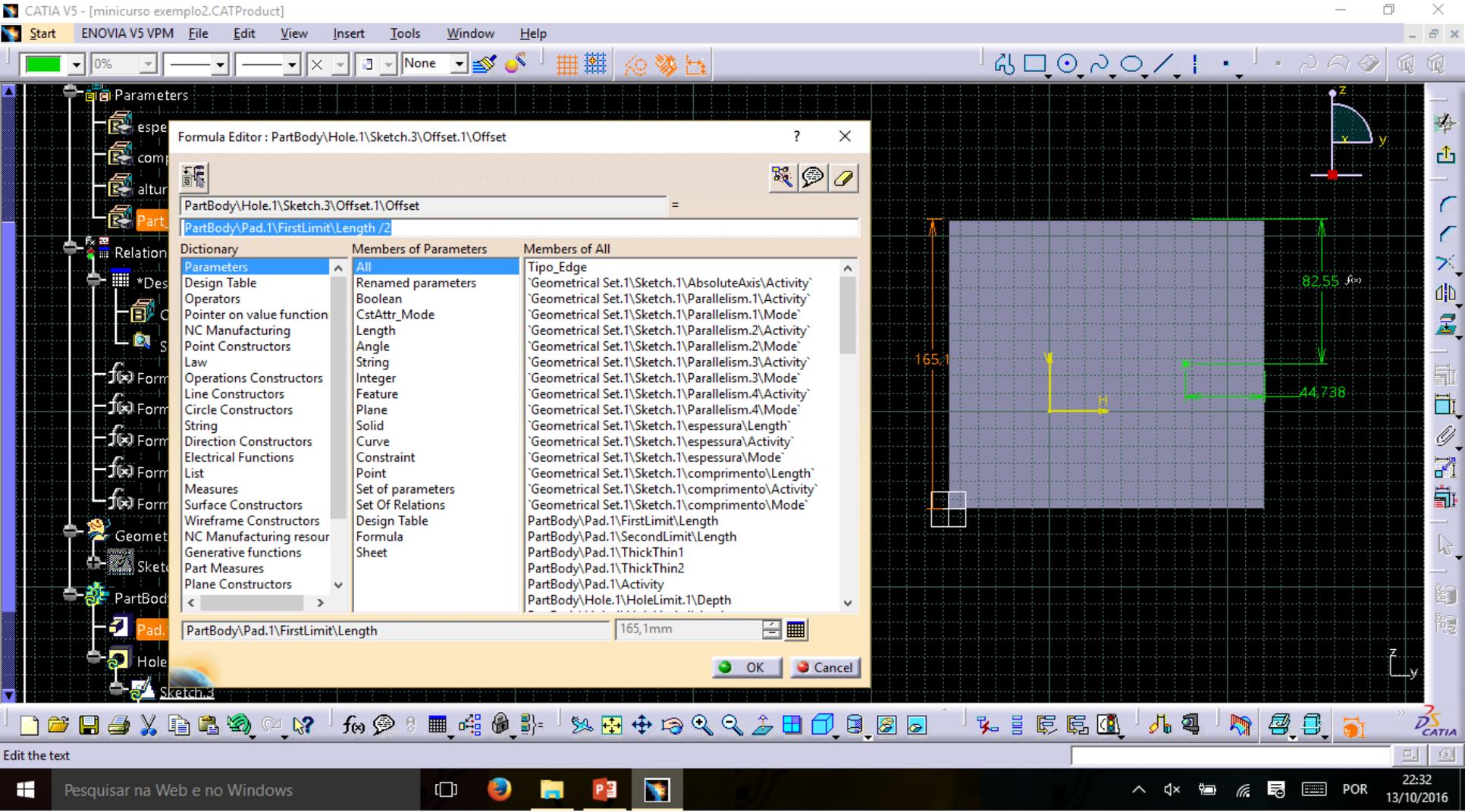
- Parameters
  - espessura=95,25mm = 'Geometrical Set.1\Sketch.1\espessura\Length'
  - comprimento=177,8mm = 'Geometrical Set.1\Sketch.1\comprimento\Length'
  - altura=165,1mm = PartBody\Pad.1\FirstLimit\Length
  - Part\_Num = teste 4 = 'teste 4\Part Number'
- Relations
  - \*DesignTable.1
    - Configuration=4
    - Sheet.1
  - Formula.1: espessura = 'Geometrical Set.1\Sketch.1\espessura\Length'
  - Formula.2: comprimento = 'Geometrical Set.1\Sketch.1\comprimento\Length'
  - Formula.3: altura = PartBody\Pad.1\FirstLimit\Length
  - Formula.4: Part\_Num = 'teste 4\Part Number'
- Geometrical Set.1
  - Sketch.1
- PartBody
  - Pad.1
  - Hole.1
  - Sketch.3
- AbsoluteAxis

In the center, a 3D model of a hole is shown with a dimension of 71.077. A right-click context menu is open over the dimension, with 'Edit Formula' selected. A red arrow points to the right-click button with the text 'Botão direito'. The context menu includes options such as 'Definition...', 'Open in New Window', 'Edit Formula', 'Lock Parameter', 'Edit Equivalent Dimensions', 'Rename Parameter', 'Deactivate', 'Display with Leader', 'Name Display', 'Name/Value Display', 'Center graph', 'Reframe On', 'Hide/Show', 'Properties', 'Other Selection...', 'Parents/Children...', 'Cut', 'Copy', 'Paste', 'Paste Special...', 'Delete', and 'Offset.1 object'.

Creates or edits a formula between parameters

# Linkando as constrains da posição do furo com os parâmetros

Usar - PartBody\Pad.1\FirstLimit\Length /2



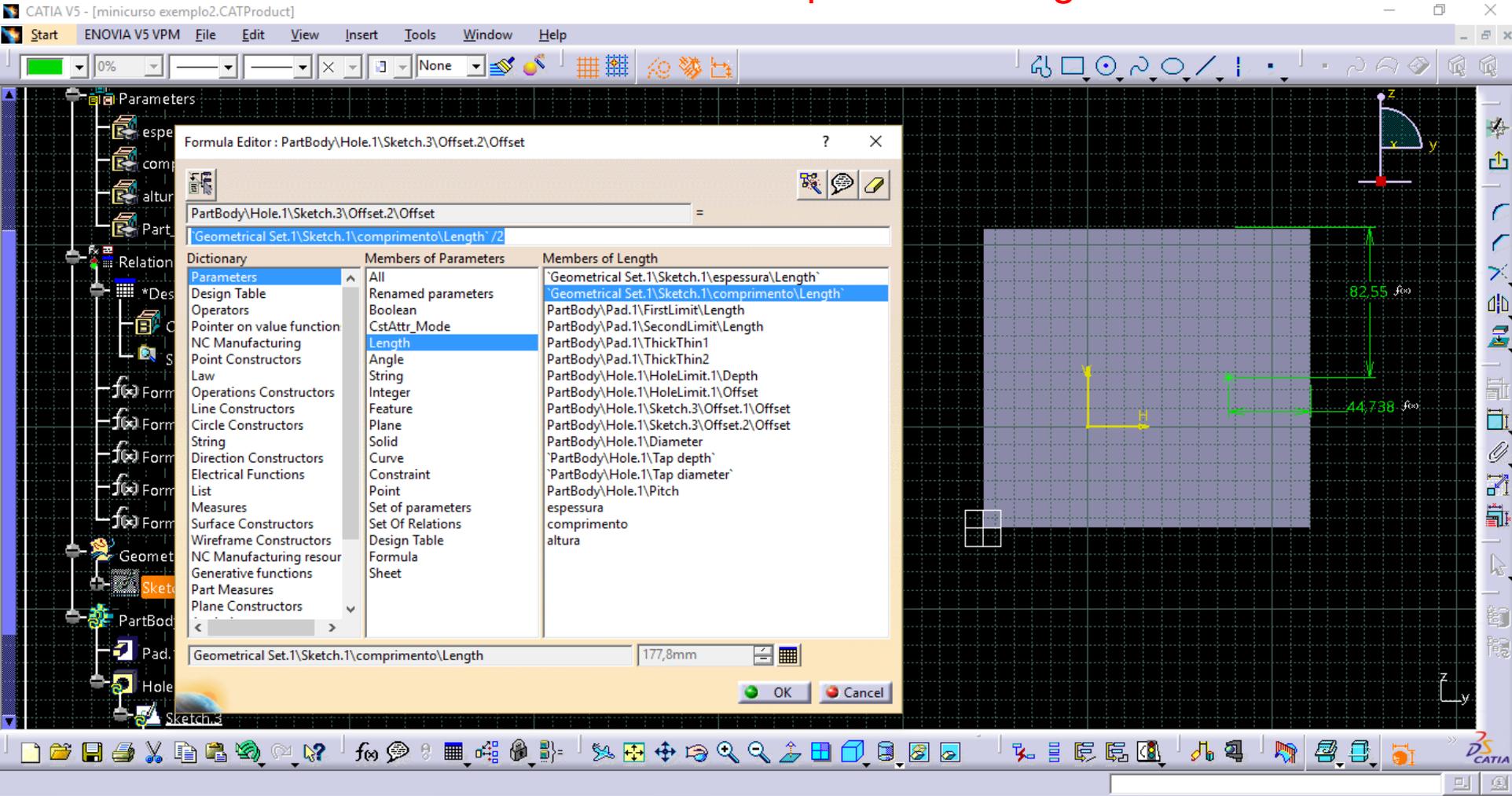
The screenshot shows the CATIA V5 interface with a Formula Editor dialog box open. The dialog box title is "Formula Editor : PartBody\Hole.1\Sketch.3\Offset.1\Offset". The formula being entered is `PartBody\Pad.1\FirstLimit\Length /2`. Below the formula input, there is a dictionary and a list of members of all parameters.

Dictionary	Members of Parameters	Members of All
Parameters	All	Tipo_Edge
Design Table	Renamed parameters	'Geometrical Set.1\Sketch.1\AbsoluteAxis\Activity'
Operators	Boolean	'Geometrical Set.1\Sketch.1\Parallelism.1\Activity'
Pointer on value function	CstAttr_Mode	'Geometrical Set.1\Sketch.1\Parallelism.1\Mode'
NC Manufacturing	Length	'Geometrical Set.1\Sketch.1\Parallelism.2\Activity'
Point Constructors	Angle	'Geometrical Set.1\Sketch.1\Parallelism.2\Mode'
Law	String	'Geometrical Set.1\Sketch.1\Parallelism.3\Activity'
Operations Constructors	Integer	'Geometrical Set.1\Sketch.1\Parallelism.3\Mode'
Line Constructors	Feature	'Geometrical Set.1\Sketch.1\Parallelism.4\Activity'
Circle Constructors	Plane	'Geometrical Set.1\Sketch.1\Parallelism.4\Mode'
String	Solid	'Geometrical Set.1\Sketch.1\espessura\Length'
Direction Constructors	Curve	'Geometrical Set.1\Sketch.1\espessura\Activity'
Electrical Functions	Constraint	'Geometrical Set.1\Sketch.1\espessura\Mode'
List	Point	'Geometrical Set.1\Sketch.1\comprimento\Length'
Measures	Set of parameters	'Geometrical Set.1\Sketch.1\comprimento\Activity'
Surface Constructors	Set Of Relations	'Geometrical Set.1\Sketch.1\comprimento\Mode'
Wireframe Constructors	Design Table	PartBody\Pad.1\FirstLimit\Length
NC Manufacturing resour	Formula	PartBody\Pad.1\SecondLimit\Length
Generative functions	Sheet	PartBody\Pad.1\ThickThin1
Part Measures		PartBody\Pad.1\ThickThin2
Plane Constructors		PartBody\Pad.1\Activity
		PartBody\Hole.1\HoleLimit.1\Depth

The 3D model in the background shows a rectangular part with a hole. Dimensions are visible: a vertical dimension of 165,1 and a horizontal dimension of 44,738. A green dimension line indicates a distance of 82,55 from the top edge to the center of the hole.

# Linkando as constrains da posição do furo com os parâmetros

Usar: `'Geometrical Set.1\Sketch.1\comprimento\Length' / 2`



The screenshot shows the CATIA V5 interface. A 'Formula Editor' dialog box is open, displaying the formula: `PartBody\Hole.1\Sketch.3\Offset.2\Offset = 'Geometrical Set.1\Sketch.1\comprimento\Length' / 2`. The dialog includes a dictionary of parameters and members, with 'Length' selected under 'Members of Parameters'. The value '177,8mm' is entered in the input field. In the background, a 3D model of a hole is shown with green dimension lines indicating a diameter of 82,55 mm and a distance of 44,738 mm from the center to the edge.

Dictionary	Members of Parameters	Members of Length
Parameters	All	'Geometrical Set.1\Sketch.1\espessura\Length'
Design Table	Renamed parameters	'Geometrical Set.1\Sketch.1\comprimento\Length'
Operators	Boolean	PartBody\Pad.1\FirstLimit\Length
Pointer on value function	CstAttr_Mode	PartBody\Pad.1\SecondLimit\Length
NC Manufacturing	Length	PartBody\Pad.1\ThickThin1
Point Constructors	Angle	PartBody\Pad.1\ThickThin2
Law	String	PartBody\Hole.1\HoleLimit.1\Depth
Operations Constructors	Integer	PartBody\Hole.1\HoleLimit.1\Offset
Line Constructors	Feature	PartBody\Hole.1\Sketch.3\Offset.1\Offset
Circle Constructors	Plane	PartBody\Hole.1\Sketch.3\Offset.2\Offset
String	Solid	PartBody\Hole.1\Diameter
Direction Constructors	Curve	'PartBody\Hole.1\Tap depth'
Electrical Functions	Constraint	'PartBody\Hole.1\Tap diameter'
List	Point	PartBody\Hole.1\Pitch
Measures	Set of parameters	espessura
Surface Constructors	Set Of Relations	comprimento
Wireframe Constructors	Design Table	altura
NC Manufacturing resour	Formula	
Generative functions	Sheet	
Part Measures		
Plane Constructors		

# Linkando as constrains da posição do furo com os parâmetros

## Resultado

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

0% None

Parameters

- espessura=95,25mm = `Geometrical Set.1\Sketch.1\espessura\Length`
- comprimento=177,8mm = `Geometrical Set.1\Sketch.1\comprimento\Length`
- altura=165,1mm = PartBody\Pad.1\FirstLimit\Length
- Part\_Num = teste 4 = `teste 4\Part Number`

Relations

- \*DesignTable.1
  - Configuration=4
  - Sheet.1
- Formula.1: espessura = `Geometrical Set.1\Sketch.1\espessura\Length`
- Formula.2: comprimento = `Geometrical Set.1\Sketch.1\comprimento\Length`
- Formula.3: altura = PartBody\Pad.1\FirstLimit\Length
- Formula.4: Part\_Num = `teste 4\Part Number`
- Formula.12: PartBody\Hole.1\Sketch.3\Offset.1\Offset = PartBody\Pad.1\FirstLimit\Length /2
- Formula.13: PartBody\Hole.1\Sketch.3\Offset.2\Offset = `Geometrical Set.1\Sketch.1\comprimento\Length` /2

Geometrical Set.1

- Sketch.1
- PartBody
  - Pad.1
  - Hole.1

2 elements selected



# Verificação do resultado usando o Design Table

## Furo centralizado

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Parameters

\*DesignTable.1 active, configuration row : 1

Design Table Properties

Name: \*DesignTable.1 Activity

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter: Edit...

Line	'Geometrical Set.1\Sketch.1\espessura\Length'	'Geometrical Set.1\Sketch.1\comprimento\Length'	PartBody\Pad.1\FirstLir
1	76,2mm	101,6mm	127mm
2	82,55mm	127mm	139,7mm
3	88,9mm	152,4mm	152,4mm
<4>	95,25mm	177,8mm	165,1mm

Edit table... Duplicate data in CATIA model

OK Apply Cancel

InitialState

Pesquisar na Web e no Windows

22:35 13/10/2016



# Verificação do resultado usando o Design Table

## Furo centralizado

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Parameters

\*DesignTable.1 active, configuration row : 3

Design Table Properties

Name: \*DesignTable.1 Activity

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter: Edit...

Sketch.1\Sketch.1\comprimento\Length	PartBody\Pad.1\FirstLimit\Length	teste 4\Part Number	PartBody\Hole.1\Diameter
127mm	139,7mm	teste 1	25,4mm
139,7mm	139,7mm	teste 2	31,75mm
152,4mm	152,4mm	teste 3	38,1mm
165,1mm	165,1mm	teste 4	44,45mm

This list contains the possible configurations of the design table. The selected row is the current one.

Edit table... Duplicate data in CATIA model

OK Apply Cancel

InitialState

Pesquisar na Web e no Windows

22:36 13/10/2016



# Inserção de Chamfro e Fillet usando a função Boleana do Knowledge

# Geração do Edge Fillet

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Product1

- teste 3 (Part1.2)
  - teste 3
    - xy plane
    - yz plane
    - zx plane
    - Parameters
      - espessura=88,9mm='Geometrical Set.1\Sketch.1\espessura\Length'
      - comprimento=152,4mm='Geometrical Set.1\Sketch.1\comprimento\Length'
      - altura=152,4mm=PartBody\Pad.1\FirstLimit\Length
      - Part\_Num=teste\_3='teste\_3\Part Number'
    - Relations
      - \*DesignTable.1
        - Configuration=3
        - Sheet.1
      - Formula.1: espessura='Geometrical Set.1\Sketch.1\espessura\Length'
      - Formula.2: comprimento='Geometrical Set.1\Sketch.1\comprimento\Length'
      - Formula.3: altura=PartBody\Pad.1\FirstLimit\Length
      - Formula.4: Part\_Num='teste\_3\Part Number'
      - Formula.12: PartBody\Hole.1\Sketch.2\Offset.1\Offset-PartBody\Pad.1\FirstLimit\Length/2

Edge Fillet Definition

Radius: 15mm

Object(s) to fillet: 4 elements

Selection mode: Tangency

Options

Conic parameter: 0,5

Trim ribbons

More>>

OK Cancel Preview

Select an edge or a face to edit the fillet.

Radius 15mm

Pesquisar na Web e no Windows

22:38 13/10/2016



# Geração do Chamfro – com chamfer

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

\*DesignTable.1  
Configuration=3  
Sheet.1  
Formula.1: espessura='Geometrical Set.1\Sketch.1\espessura\Length'  
Formula.2: comprimento='Geometrical Set.1\Sketch.1\comprimento\Length'  
Formula.3: altura=PartBody\Pad.1\FirstLimit\Length  
Formula.4: Part\_Num='teste 3\Part Number'  
Formula.12: PartBody\Hole.1\Sketch.3\Offset.1\Offset=PartBody\Pad.1\FirstLimit\Length /2  
Formula.13: PartBody\Hole.1\Sketch.3\Offset.2\Offset='Geometrical Set.1\Sketch.1\comprimento\Length' /2

Geometrical Set.1  
Sketch.1  
PartBody  
Pad.1  
Hole.1  
Sketch.3  
AbsoluteAxis  
Geometry  
Constraints  
EdgeFillet.1  
Chamfer.1

Chamfer Definition

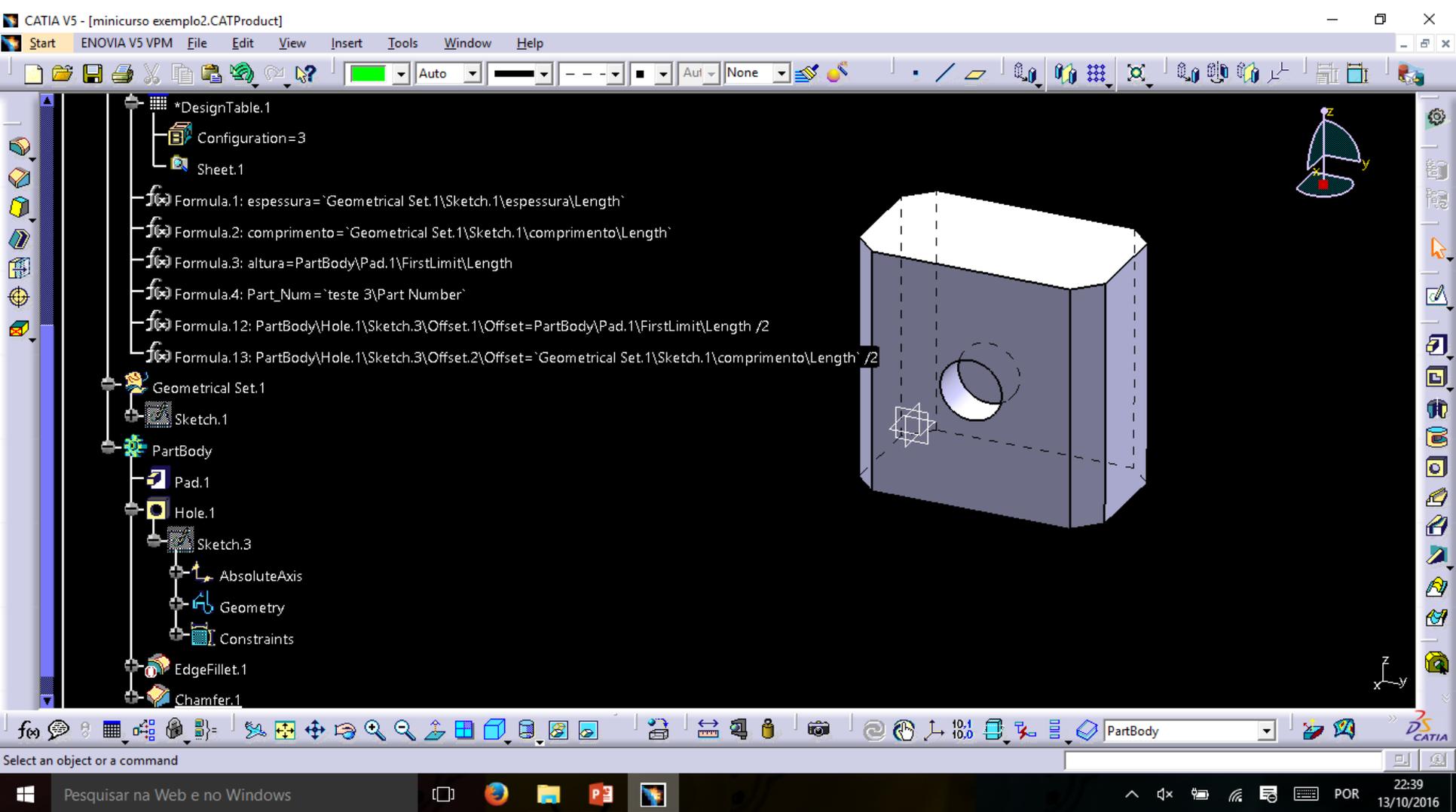
Mode: Length1/Angle  
Length 1: 15  
Angle: 45deg  
Object(s) to chamfer: 4 elements  
Propagation: Tangency  
 Reverse

OK Cancel Preview

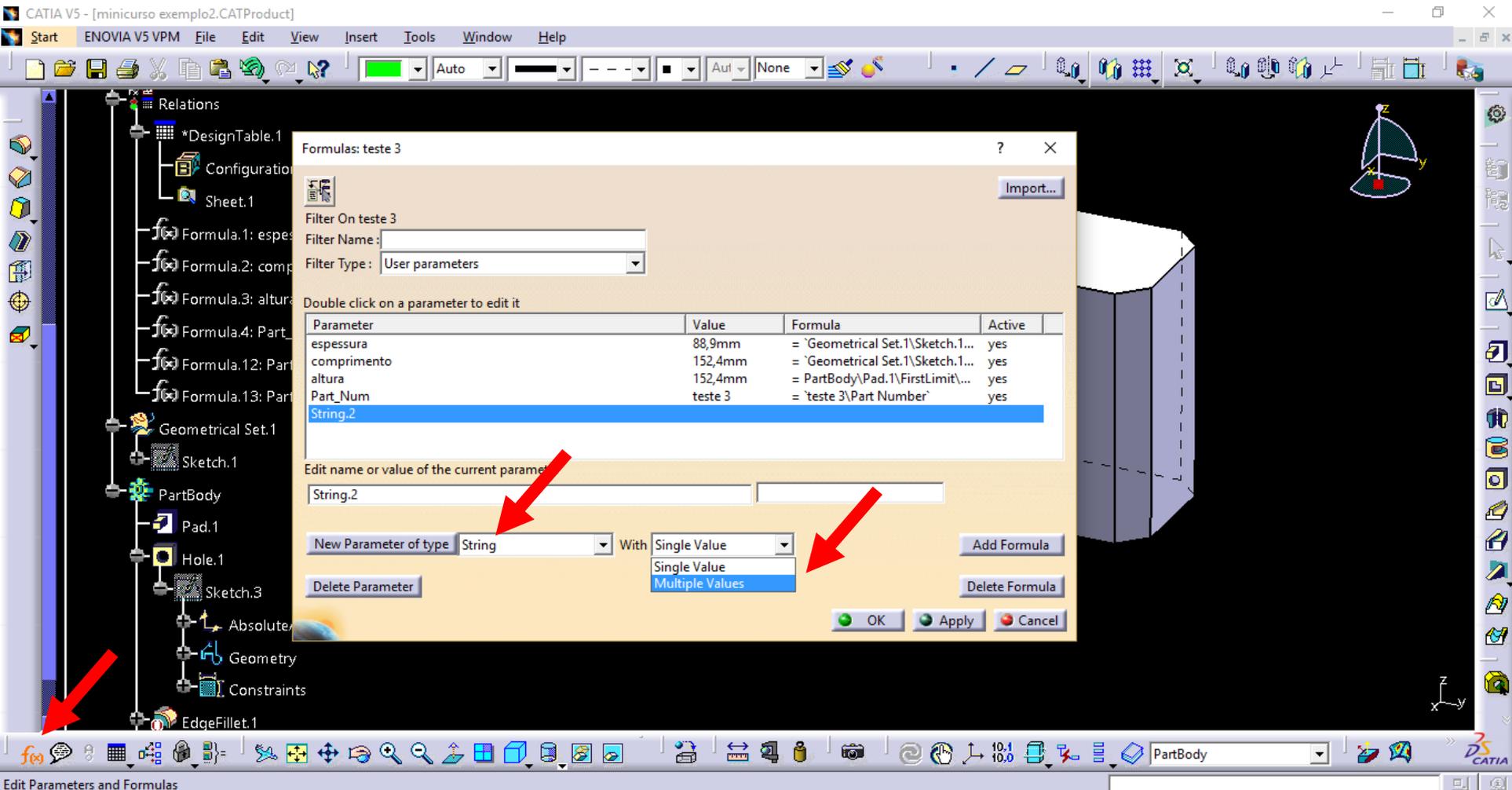
Specify the required data to define the chamfer

Spinner;Spinner | 15;45deg

# Geração do Chamfro – com chamfer



# Criação do parâmetro Edge\_type com string – multiple values



Formulas: teste 3

Filter On teste 3

Filter Name:

Filter Type: User parameters

Double click on a parameter to edit it

Parameter	Value	Formula	Active
espessura	88,9mm	= `Geometrical Set.1\Sketch.1...	yes
comprimento	152,4mm	= `Geometrical Set.1\Sketch.1...	yes
altura	152,4mm	= PartBody\Pad.1\FirstLimit...	yes
Part_Num	teste 3	= `teste 3\Part Number`	yes
String.2			

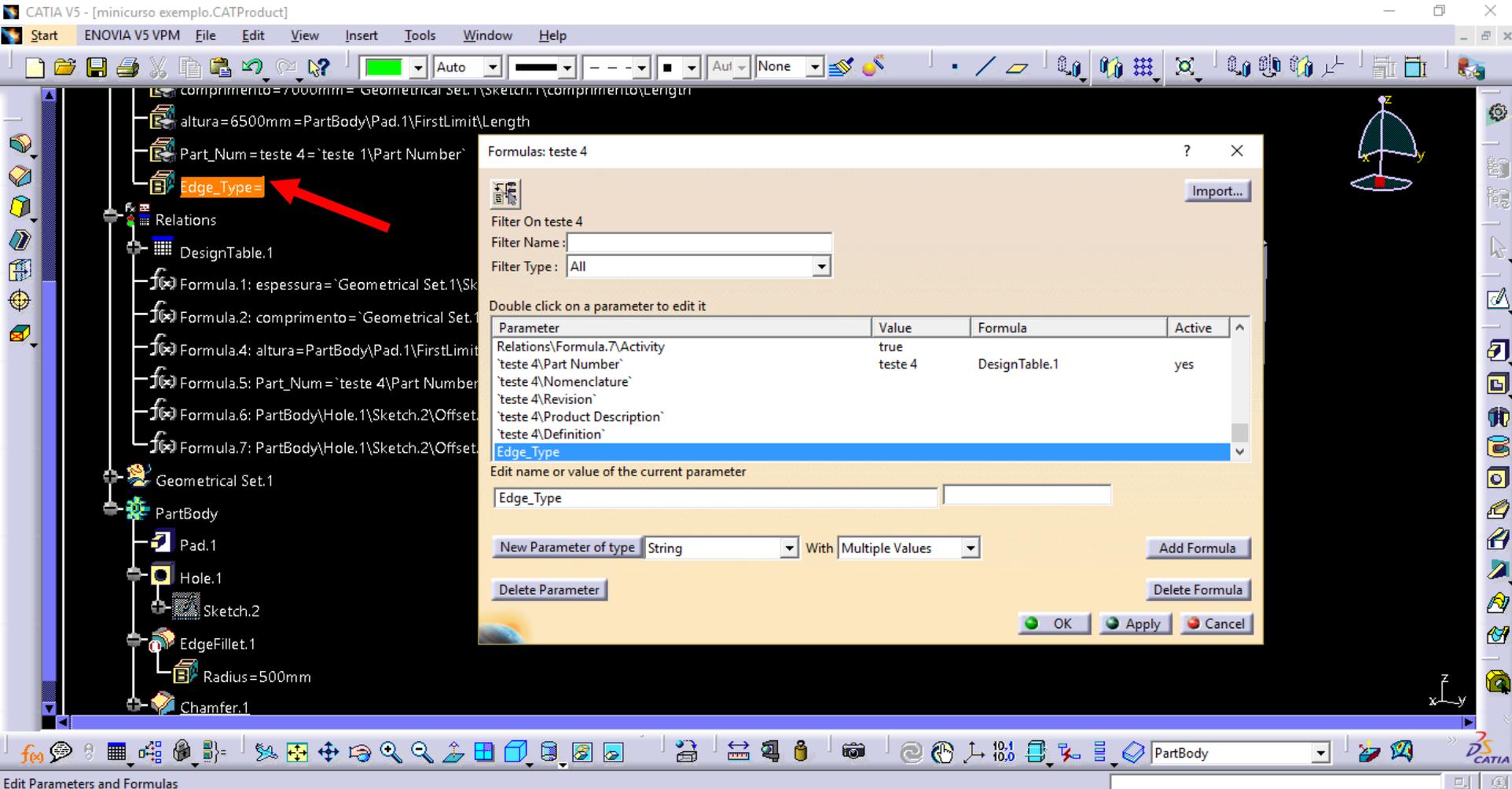
Edit name or value of the current parameter

New Parameter of type: String With: Single Value

Multiple Values

Buttons: Add Formula, Delete Formula, OK, Apply, Cancel

# Criação do parâmetro Edge\_type com string – multiple values

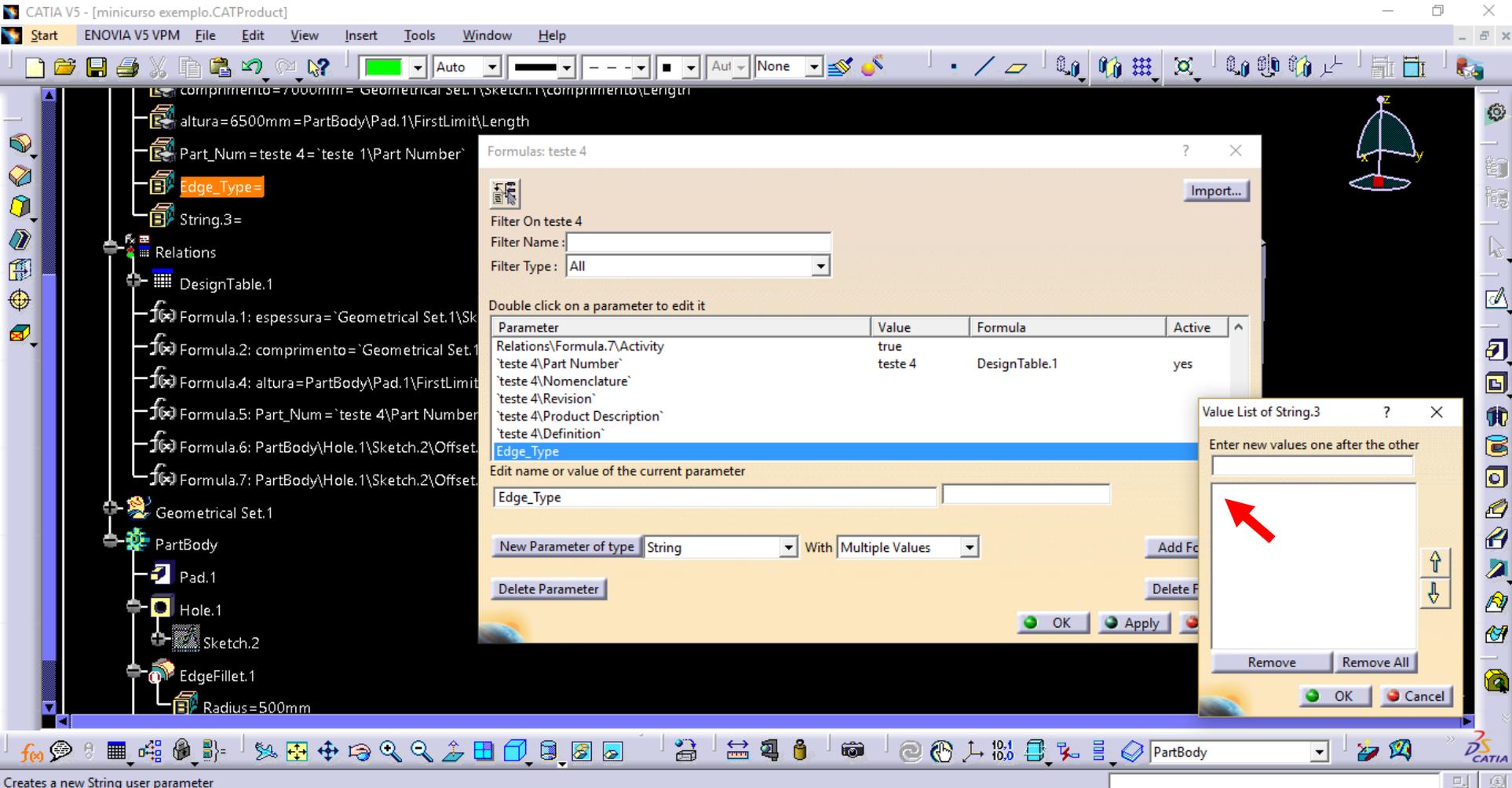


The screenshot shows the CATIA V5 interface with the 'Formulas: teste 4' dialog box open. The dialog box contains a table of parameters and a section for editing the current parameter.

Parameter	Value	Formula	Active
Relations\Formula.7\Activity	true		
'teste 4\Part Number'	teste 4	DesignTable.1	yes
'teste 4\Nomenclature'			
'teste 4\Revision'			
'teste 4\Product Description'			
'teste 4\Definition'			
<b>Edge_Type</b>			

Below the table, the 'Edit name or value of the current parameter' section shows the parameter name 'Edge\_Type' in a text field. The 'New Parameter of type' dropdown is set to 'String', and the 'With' dropdown is set to 'Multiple Values'. Buttons for 'Add Formula', 'Delete Formula', 'Delete Parameter', 'OK', 'Apply', and 'Cancel' are visible at the bottom of the dialog.

# Criação do parâmetro Edge\_type com string – multiple values



The screenshot shows the CATIA V5 interface with the 'Formulas' dialog box open. The dialog displays a table of parameters for 'Formulas: teste 4'. The 'Edge\_Type' parameter is selected, and the 'Value List of String.3' dialog is open, showing a list of values to be entered.

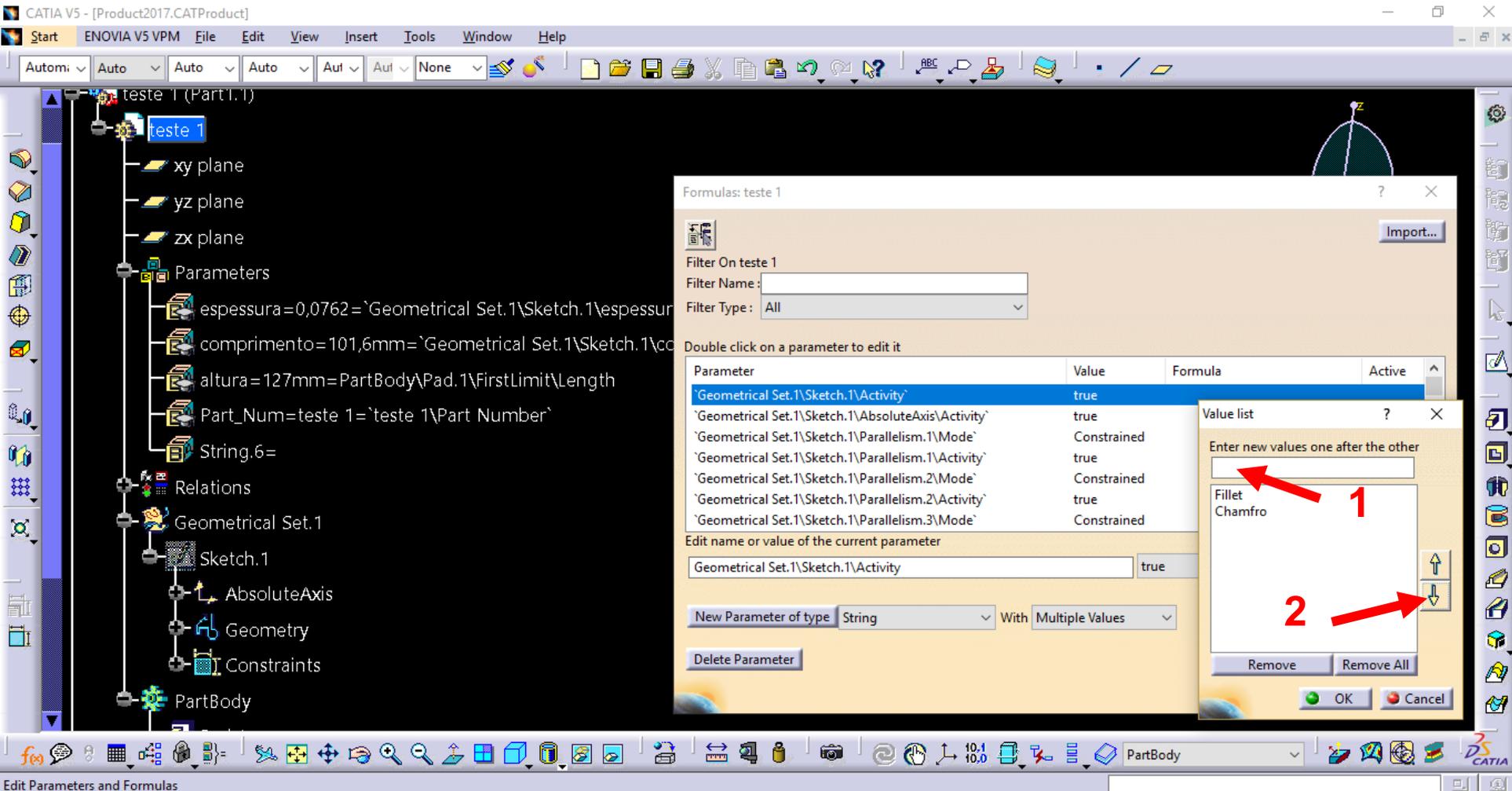
Parameter	Value	Formula	Active
Relations\Formula.7\Activity	true		
'teste 4\Part Number'	teste 4	DesignTable.1	yes
'teste 4\Nomenclature'			
'teste 4\Revision'			
'teste 4\Product Description'			
'teste 4\Definition'			
<b>Edge_Type</b>			

The 'Value List of String.3' dialog box is open, showing a list of values to be entered. A red arrow points to the input field. The dialog also includes 'Remove' and 'Remove All' buttons.

Creates a new String user parameter

# Criação do parâmetro Edge\_type com string – multiple values

## Inserindo Fillet e Chamfro no Value List



The screenshot displays the CATIA V5 interface with the 'Formulas: teste 1' dialog box open. The dialog shows a table of parameters and their formulas. The 'Value list' dialog is also open, showing a list of values 'Fillet' and 'Chamfro' with red arrows pointing to the input field and the list items.

Parameter	Value	Formula	Active
'Geometrical Set.1\Sketch.1\Activity'	true		
'Geometrical Set.1\Sketch.1\AbsoluteAxis\Activity'	true		
'Geometrical Set.1\Sketch.1\Parallelism.1\Mode'	Constrained		
'Geometrical Set.1\Sketch.1\Parallelism.1\Activity'	true		
'Geometrical Set.1\Sketch.1\Parallelism.2\Mode'	Constrained		
'Geometrical Set.1\Sketch.1\Parallelism.2\Activity'	true		
'Geometrical Set.1\Sketch.1\Parallelism.3\Mode'	Constrained		

Value list dialog content:

- Enter new values one after the other
- Fillet
- Chamfro

# Criação do parâmetro Edge\_type com string – multiple values: Fillet e Chamfro

Formulas: teste 3

Filter On teste 3  
Filter Name:   
Filter Type: All

Double click on a parameter to edit it

Parameter	Value	Formula	Active
Relations\Formula.13\Activity	true		
'teste 3\Part Number'	teste 3	*DesignTable.1	yes
'teste 3\Nomenclature'			
'teste 3\Revision'			
'teste 3\Product Description'			
'teste 3\Definition'			
Edge_Tipo	Fillet		

Edit name or value of the current parameter  
Edge\_Tipo

New Parameter of type  With

Delete Parameter

Add Formula

Delete Formula

OK Apply Cancel

Value List of String.2

Enter new values one after the other

Remove Remove All

OK Cancel

PartBody

Edit Parameters and Formulas

Pesquisar na Web e no Windows

22:48  
13/10/2016

# Criação do parâmetro Edge\_type com string – Renomeando para “Edge\_Type” e verificando os valores.

Formulas: teste 1

Filter On teste 1  
Filter Name:   
Filter Type: All

Double click on a parameter to edit it

Parameter	Value	Formula	Active
Relations\Formula.7\Activity	true		
'teste 1\Part Number'	teste 1	DesignTable.2	yes
'teste 1\Nomenclature'			
'teste 1\Revision'			
'teste 1\Product Description'			
'teste 1\Definition'			
<b>Edge_Type</b>	<b>Fillet</b>		

Edit name or value of the current parameter

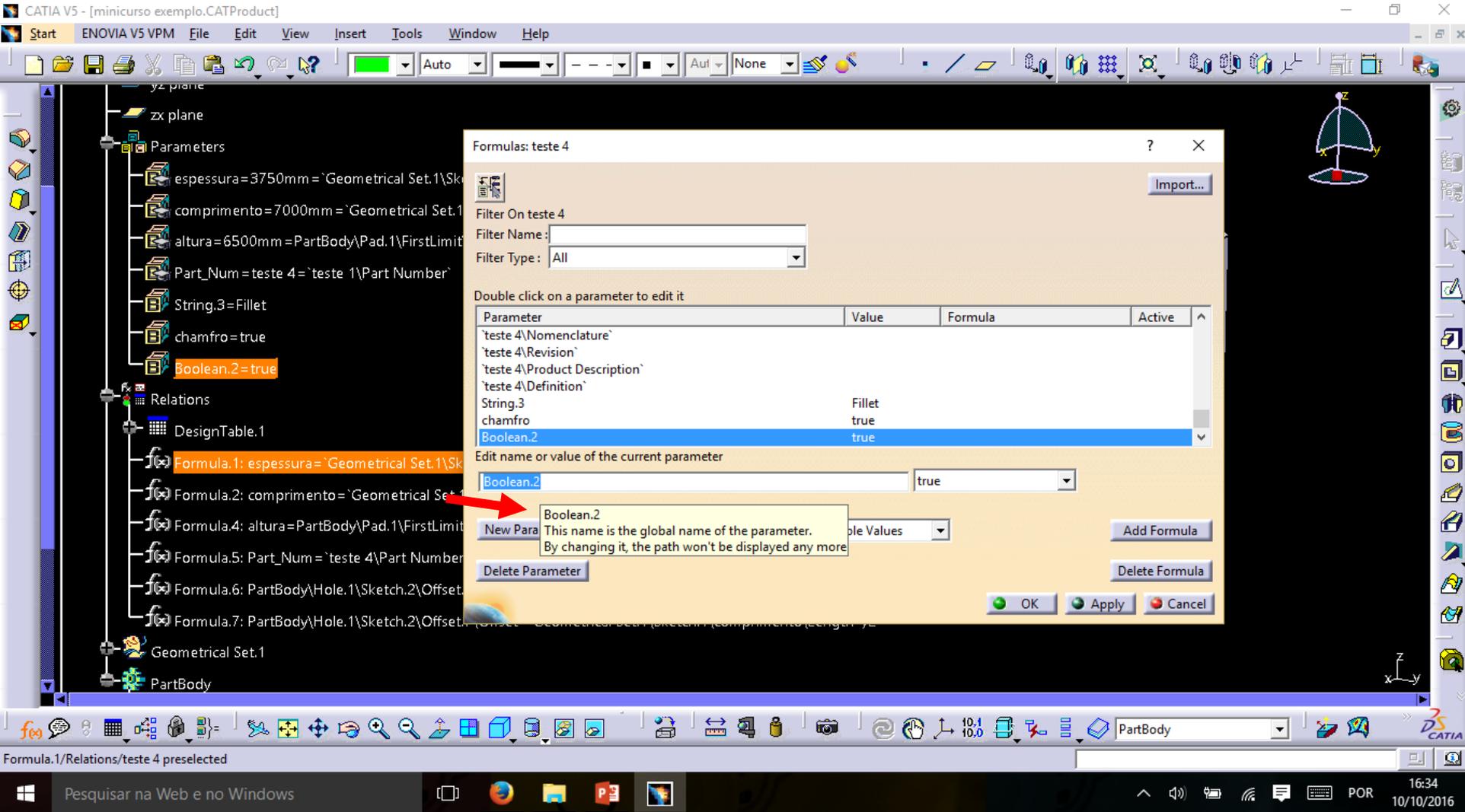
Edge\_Type

New Parameter of type String With Multiple Values

Buttons: Add Formula, Delete Formula, OK, Apply, Cancel

Change value of parameter: String.6 required.

# Criação do parâmetro Edge\_type com boolean (2X) Chamfro = True e Fillet = False



The screenshot shows the CATIA V5 interface with the 'Formulas: teste 4' dialog box open. The dialog box contains a table of parameters and their formulas. The 'Boolean.2' parameter is highlighted, and a tooltip is visible over the 'New Parameter' button.

Parameter	Value	Formula	Active
'teste 4\Nomenclature'			
'teste 4\Revision'			
'teste 4\Product Description'			
'teste 4\Definition'			
String.3	Fillet		
chamfro	true		
Boolean.2	true		

Double click on a parameter to edit it

Edit name or value of the current parameter

Boolean.2 true

New Parameter This name is the global name of the parameter. By changing it, the path won't be displayed any more

Boolean.2

Multiple Values

Add Formula

Delete Formula

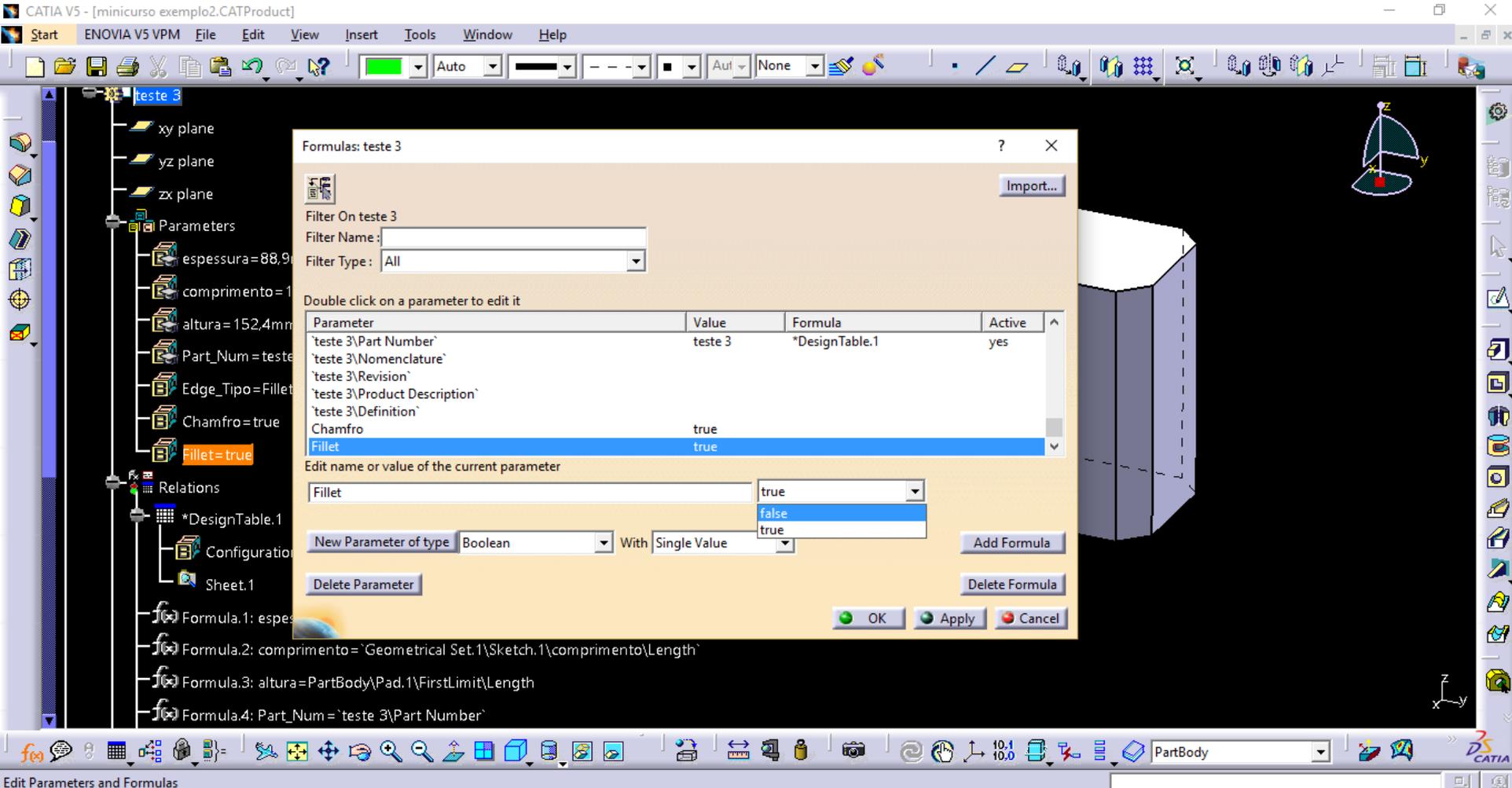
OK Apply Cancel

# Criação do parâmetro Edge\_type com boolean (2X) Chamfro = True e Fillet = False

The screenshot shows the CATIA V5 interface with the 'Formulas: teste 4' dialog box open. The dialog box contains a table of parameters and their values/formulas. The 'Boolean.2' parameter is selected, showing a value of 'false' and a formula of 'false'. The background shows the CATIA interface with a tree view containing parameters like 'chamfro=true' and 'Boolean.2=false'.

Parameter	Value	Formula	Active
'teste 4\Nomenclature'			
'teste 4\Revision'			
'teste 4\Product Description'			
'teste 4\Definition'			
String.3	Fillet		
chamfro	true		
Boolean.2	false		

# Criação do parâmetro Edge\_type com boolean (2X) Chamfro = True e Fillet = False



Formulas: teste 3

Filter On teste 3  
Filter Name:   
Filter Type: All

Double click on a parameter to edit it

Parameter	Value	Formula	Active
'teste 3\Part Number'	teste 3	*DesignTable.1	yes
'teste 3\Nomenclature'			
'teste 3\Revision'			
'teste 3\Product Description'			
'teste 3\Definition'			
Chamfro	true		
Fillet	true		

Edit name or value of the current parameter

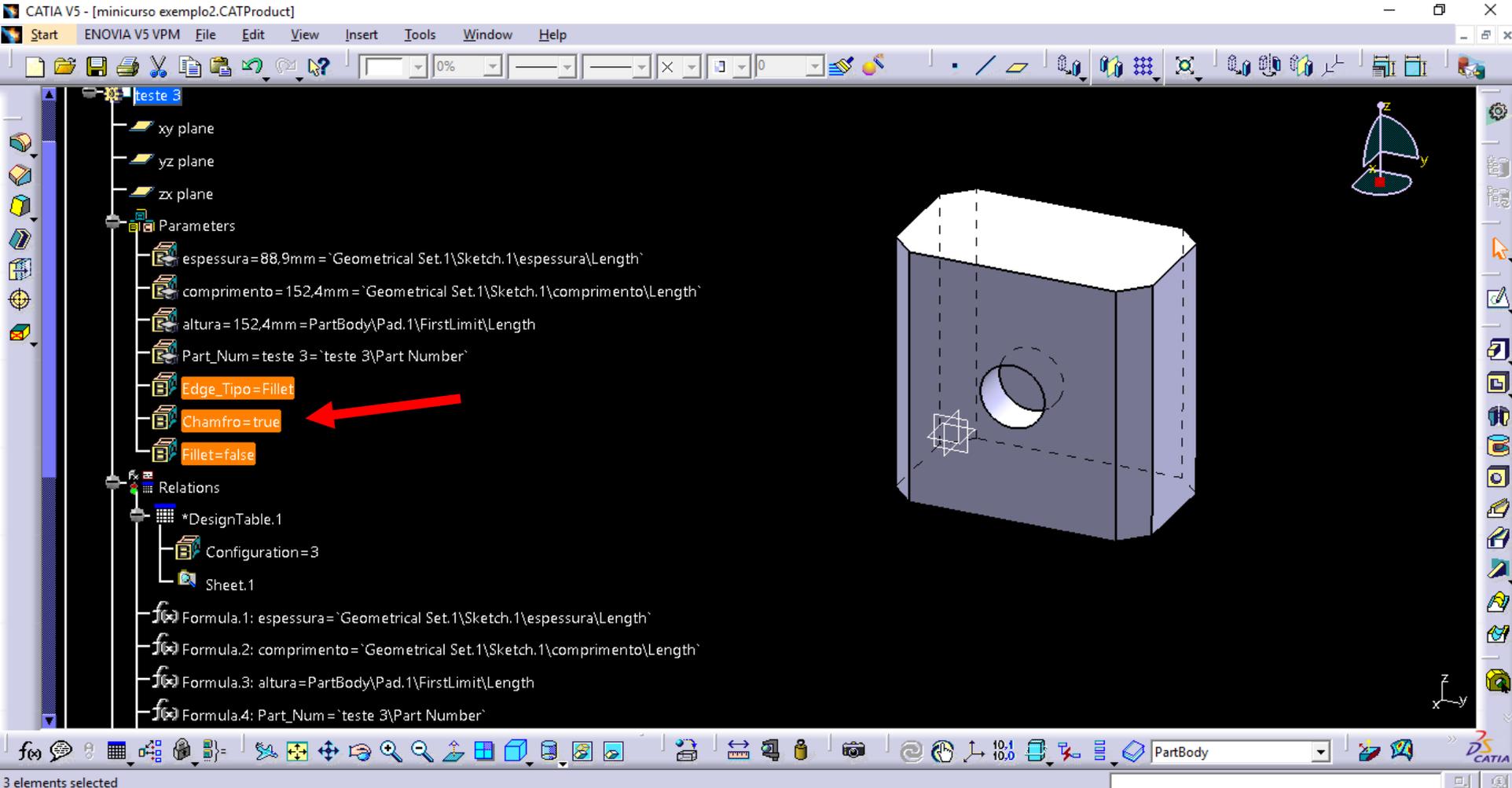
Fillet: true

New Parameter of type: Boolean With: Single Value

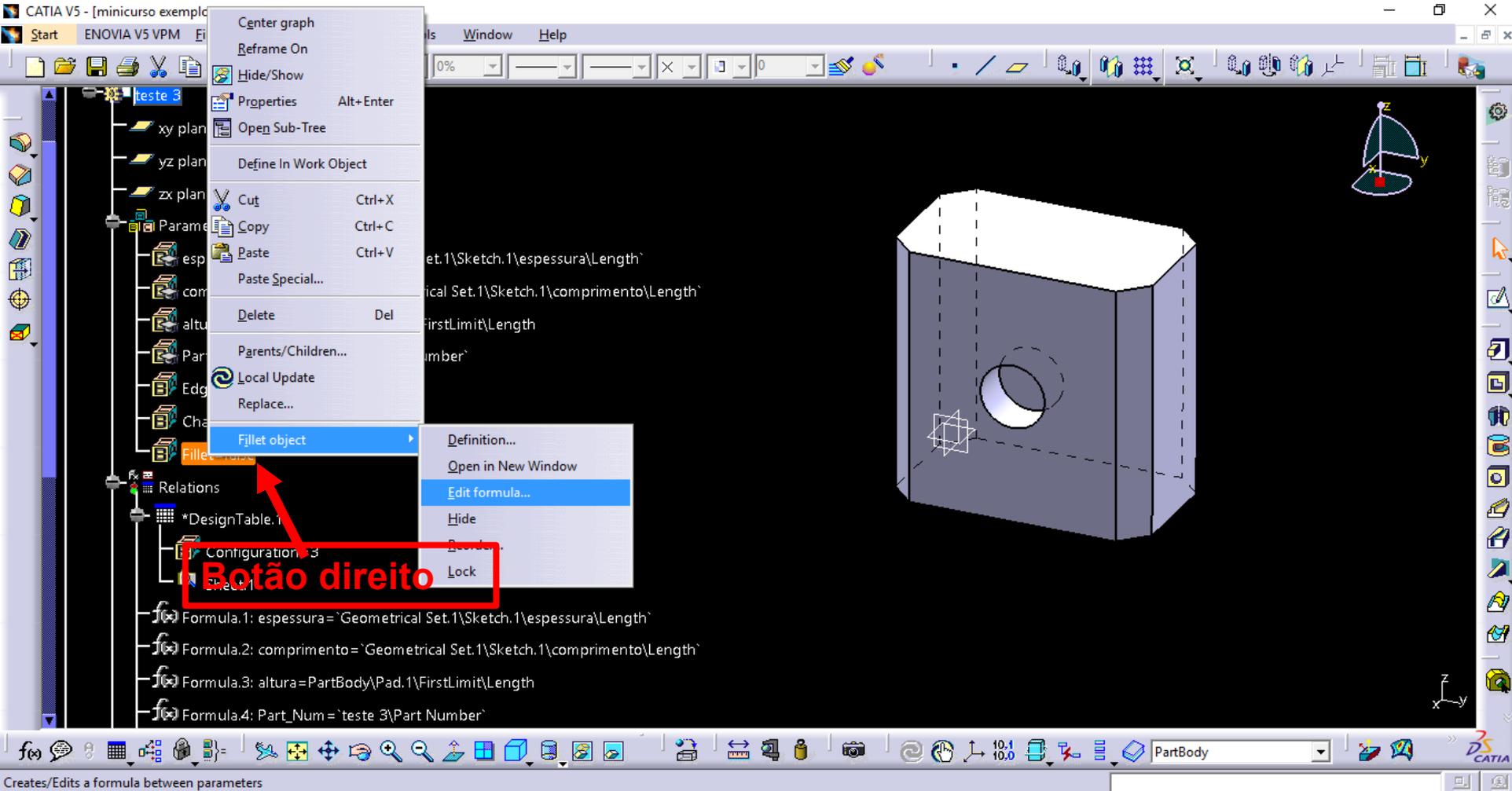
Buttons: Add Formula, Delete Formula, OK, Apply, Cancel



# Criação do parâmetro Edge\_type com boolean (2X) Chamfro = True e Fillet = False



# Associando o parâmetro Edge\_type com boolean (2X) Chamfro = True e Fillet = False



# Associando o parâmetro Edge\_type com boolean (2X) Edge\_Tipo == "Fillet"

Formula Editor : Fillet

Fillet =

Edge\_Tipo == "Fillet"

Dictionary	Members of Parameters	Members of All
Parameters	All	espesura
Design Table	Renamed parameters	comprimento
Operators	Boolean	altura
Pointer on value function	CstAttr_Mode	Part_Num
NC Manufacturing	Length	Edge_Tipo
Point Constructors	Angle	Chamfro
Law	String	Fillet
Operations Constructors	Integer	"Relations\"DesignTable.1\Activity"
Line Constructors	Feature	"Relations\"DesignTable.1\Configuration"
Circle Constructors	Plane	Relations\Formula.1\Activity
String	Solid	Relations\Formula.2\Activity
Direction Constructors	Curve	Relations\Formula.3\Activity
Electrical Functions	Constraint	Relations\Formula.4\Activity
List	Point	Relations\Formula.12\Activity
Measures	Set of parameters	Relations\Formula.13\Activity
Surface Constructors	Set Of Relations	"teste 3\Part Number"
Wireframe Constructors	Design Table	"teste 3\Nomenclature"
NC Manufacturing resour	Formula	"teste 3\Revision"
Generative functions	Sheet	"teste 3\Product Description"
Part Measures		"teste 3\Definition"
Plane Constructors		"teste 3"
		"xy plane"

Fillet false

OK Cancel

# Associando o parâmetro Edge\_type com boolean (2X)

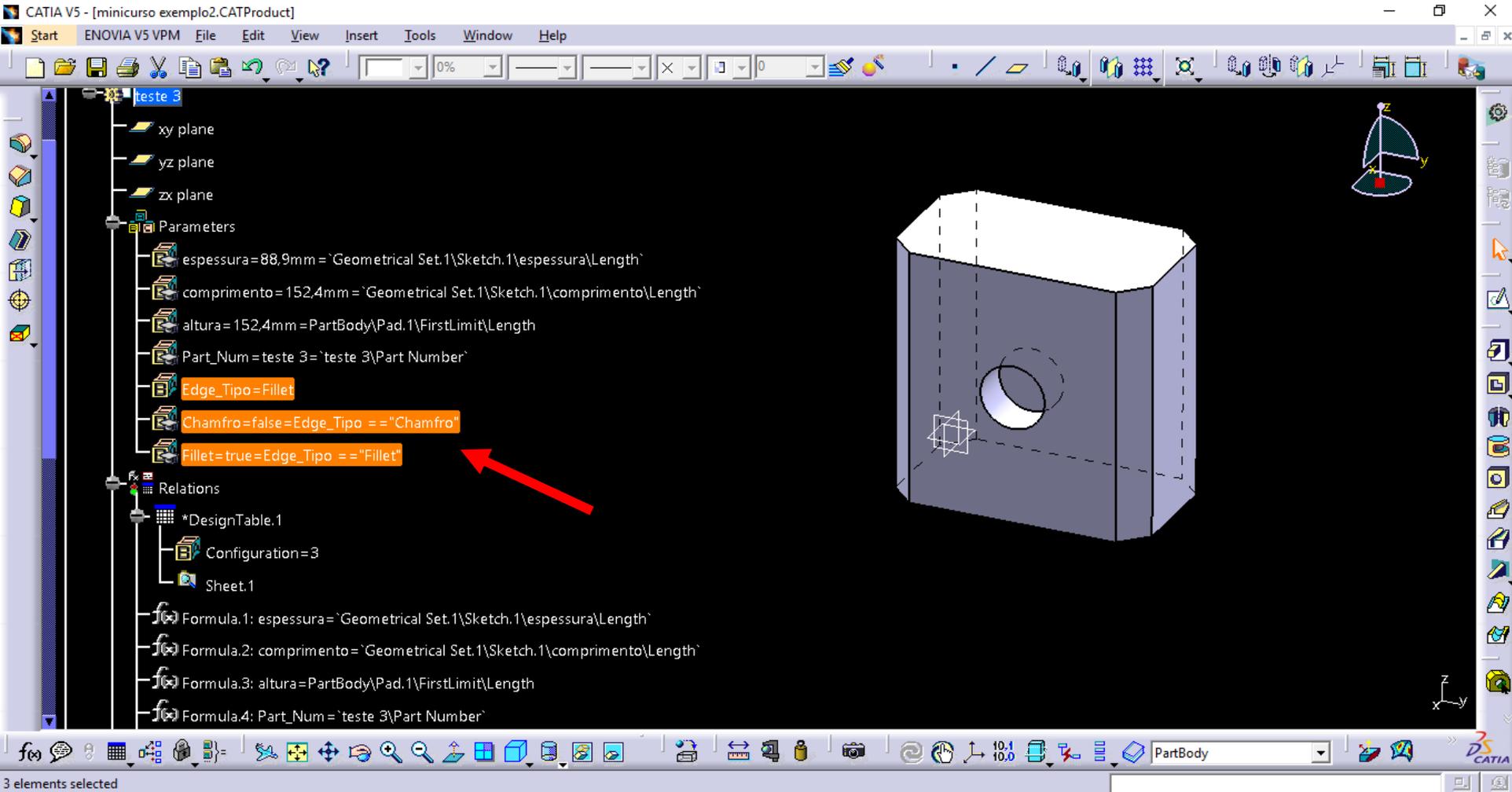
Usar: **Edge\_Tipo == "Chamfro"**

The screenshot shows the CATIA V5 interface with the Formula Editor open for a Chamfer feature. The formula being edited is `Edge_Tipo == "Chamfro"`. The editor window has three panes: Dictionary, Members of Parameters, and Members of All. The 'Members of Parameters' pane lists various parameters, with 'Edge\_Tipo' selected. The 'Members of All' pane lists all available parameters and functions, including 'Edge\_Tipo'. The background shows a 3D model of a rectangular block with a chamfered edge.

Dictionary	Members of Parameters	Members of All
Parameters	All	PartBody\Hole.1\Sketch.3\Offset.2\Activity
Design Table	Renamed parameters	PartBody\Hole.1\Diameter
Operators	Boolean	PartBody\Hole.1\Tap depth
Pointer on value function	CstAttr_Mode	PartBody\Hole.1\Tap diameter
NC Manufacturing	Length	PartBody\Hole.1\Pitch
Point Constructors	Angle	PartBody\Hole.1\Thread Description
Law	String	PartBody\Hole.1\Activity
Operations Constructors	Integer	PartBody\EdgeFillet.1\CstEdgeRibbon.1\Radius
Line Constructors	Feature	PartBody\EdgeFillet.1\Activity
Circle Constructors	Plane	PartBody\Chamfer.1\ChamferRibbon.1\Length1
String	Solid	PartBody\Chamfer.1\ChamferRibbon.1\Angle
Direction Constructors	Curve	PartBody\Chamfer.1\Activity
Electrical Functions	Constraint	espessura
List	Point	comprimento
Measures	Set of parameters	altura
Surface Constructors	Set Of Relations	Part_Num
Wireframe Constructors	Design Table	Edge_Tipo
NC Manufacturing resour	Formula	Chamfro
Generative functions	Sheet	Fillet
Part Measures		PartBody\Hole.1\Activity
Plane Constructors		Relations\*DesignTable.1\Activity
		Relations\*DesignTable.1\Configuration
		Relations\Formula.1\Activity



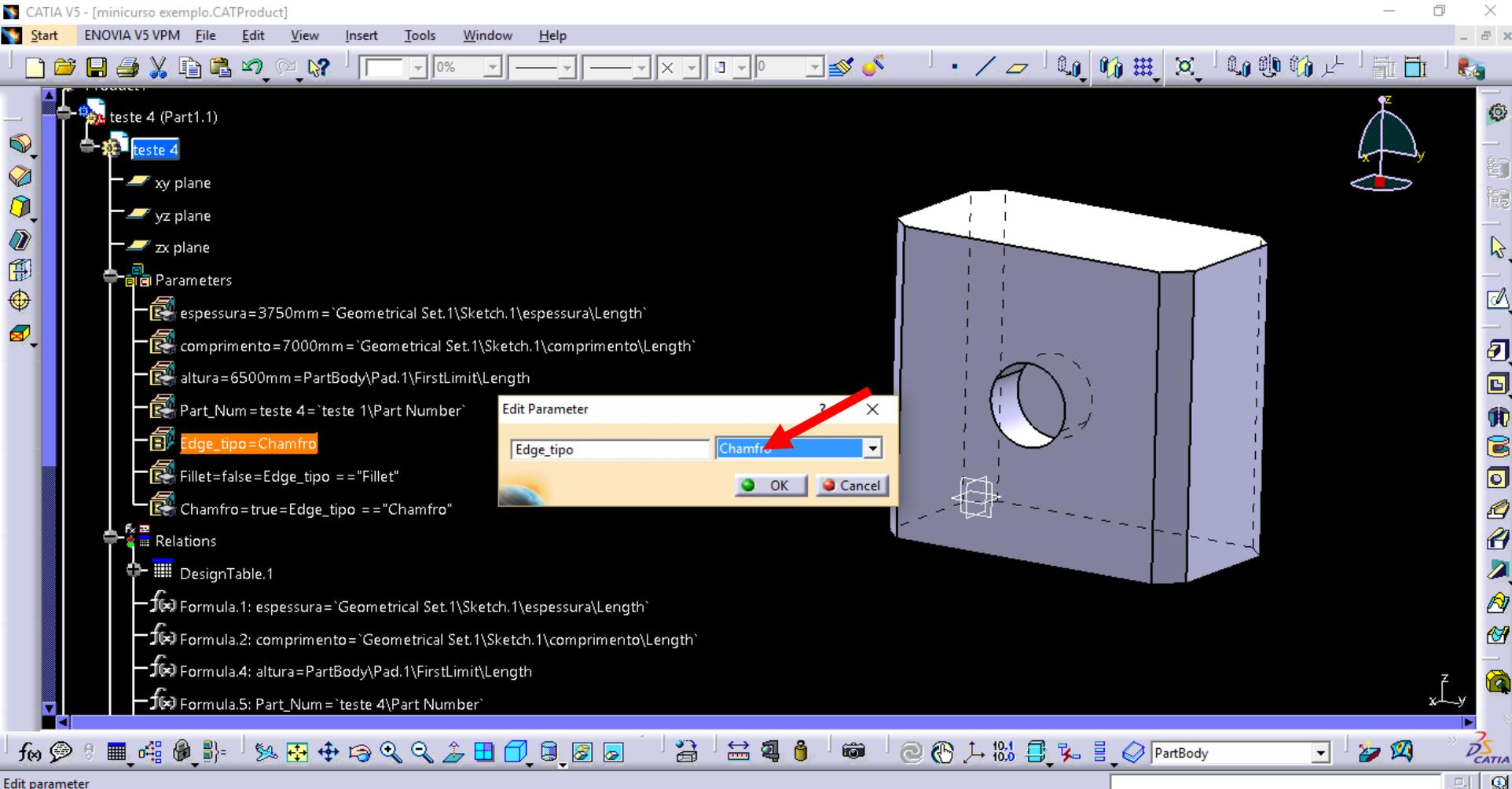
# Associando o parâmetro Edge\_type com boolean (2X) Resultado na árvore



3 elements selected

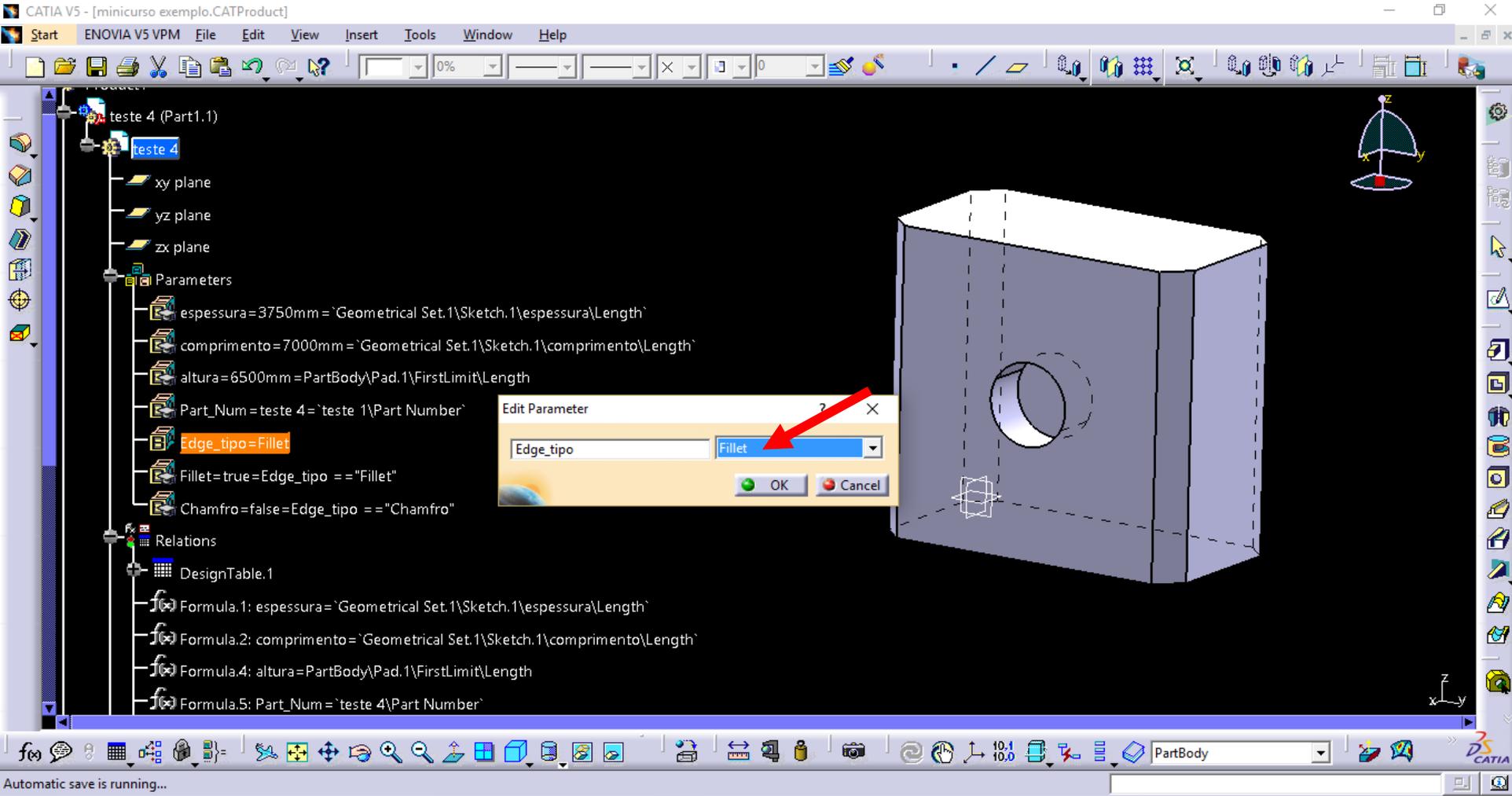


# Associando o parâmetro Edge\_type com boolean (2X) Resultado na árvore



Edit parameter

# Associando o parâmetro Edge\_type com boolean (2X) Resultado na árvore



Automatic save is running...



# Associando o parâmetro Edge\_type na fórmula

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Formulas: teste 3

Filter On teste 3  
Filter Name: \*  
Filter Type: User parameters

Double click on a parameter to edit it

Parameter	Value	Formula	Active
espessura	88,9mm	= 'Geometrical Set.1\Sketch.1\espessura\Length'	yes
comprimento	152,4mm	= 'Geometrical Set.1\Sketch.1\comprimento\Length'	yes
altura	152,4mm	= PartBody\Pad.1\FirstLimit\Height	yes
Part_Num	teste 3	= 'teste 3\Part Number'	yes
Edge_Tipo	Fillet		
Chamfro	false	= Edge_Tipo == "Chamfro"	yes
Fillet	true	= Edge_Tipo == "Fillet"	yes

Edit name or value of the current parameter

espessura 88,9mm

New Parameter of type Boolean With Single Value

Add Formula Delete Formula

OK Apply Cancel

1

2

# Associando o parâmetro Edge\_type na fórmula

Formulas: EdgeFillet.1

Filter On EdgeFillet.1

Filter Name : \*

Filter Type : All

Double click on a parameter to edit it

Parameter	Value	Formula	Active
PartBody\EdgeFillet.1\CstEdgeRibbon.1\Radius	15mm		
PartBody\EdgeFillet.1\Activity	false		

Edit name or value of the current parameter

PartBody\EdgeFillet.1\CstEdgeRibbon.1\Radius 15mm

New Parameter of type Boolean With Single Value

Delete Parameter

Add Formula

Delete Formula

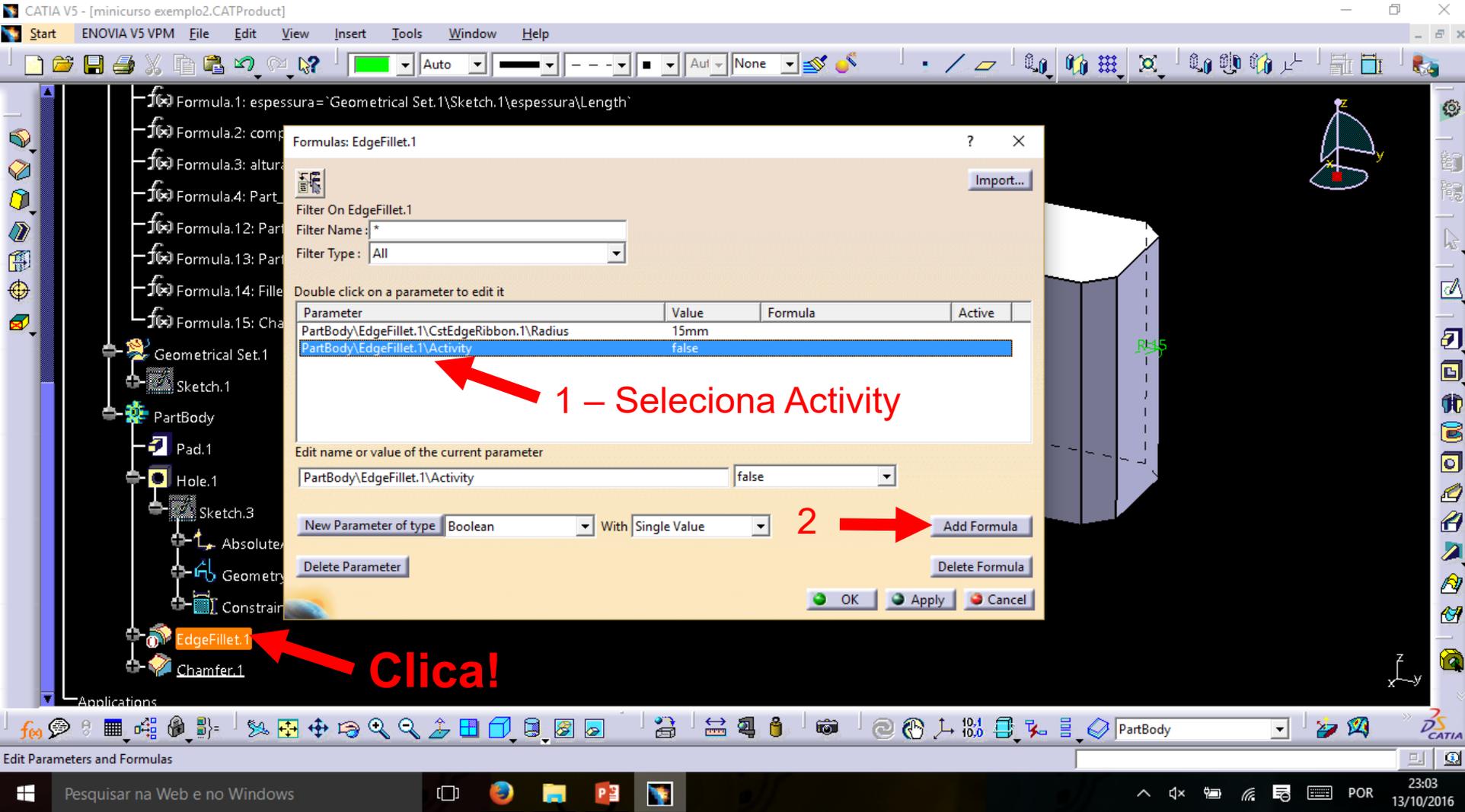
OK Apply Cancel

1 - clica

EdgeFillet.1

EdgeFillet.1

# Associando o parâmetro Edge\_type na fórmula



CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Formulas: EdgeFillet.1

Filter On EdgeFillet.1  
Filter Name: \*  
Filter Type: All

Double click on a parameter to edit it

Parameter	Value	Formula	Active
PartBody\EdgeFillet.1\CstEdgeRibbon.1\Radius	15mm		
PartBody\EdgeFillet.1\Activity	false		

Edit name or value of the current parameter  
PartBody\EdgeFillet.1\Activity false

New Parameter of type Boolean With Single Value **2** → Add Formula

Delete Parameter Delete Formula

OK Apply Cancel

**1 - Selecciona Activity**

**Clica!**

Geometrical Set.1  
Sketch.1  
PartBody  
Pad.1  
Hole.1  
Sketch.3  
Absoluter  
Geometr  
Constrai  
EdgeFillet.1  
Chamfer.1

PartBody

Pesquisar na Web e no Windows

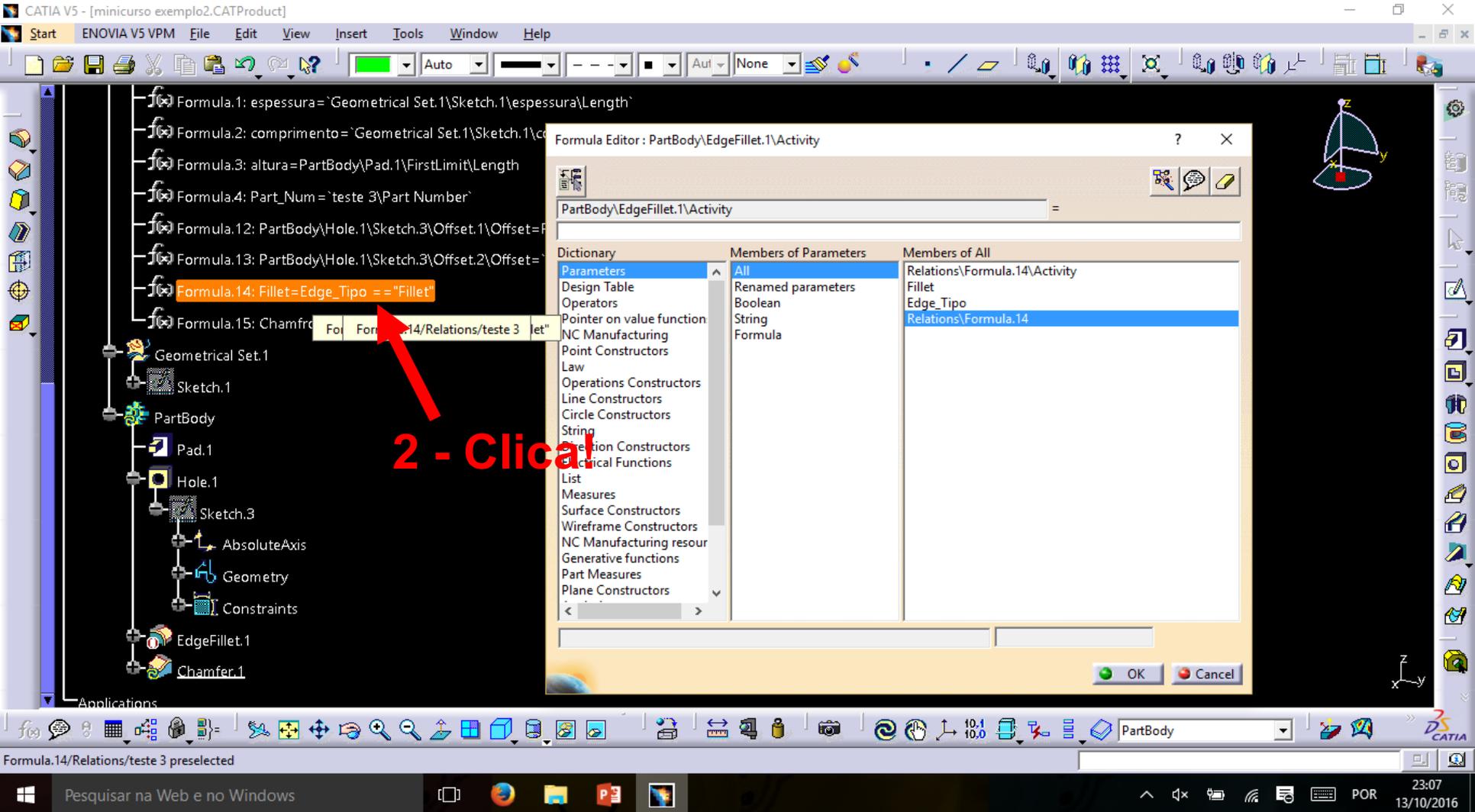
23:03  
13/10/2016

# Associando o parâmetro Edge\_type na fórmula

The screenshot shows the CATIA V5 interface with the Formula Editor dialog box open. The dialog box is titled "Formula Editor: PartBody\EdgeFillet.1\Activity" and contains a list of parameters and members. A red arrow points to the "EdgeFillet.1" feature in the tree view, with the text "1 - Clique!" next to it.

Dictionary	Members of Parameters	Members of All
Parameters	All	Tipo_Edge
Design Table	Renamed parameters	'Geometrical Set.1\Sketch.1\AbsoluteAxis\Activity'
Operators	Boolean	'Geometrical Set.1\Sketch.1\Parallelism.1\Activity'
Pointer on value function	CstAttr_Mode	'Geometrical Set.1\Sketch.1\Parallelism.1\Mode'
NC Manufacturing	Length	'Geometrical Set.1\Sketch.1\Parallelism.2\Activity'
Point Constructors	Angle	'Geometrical Set.1\Sketch.1\Parallelism.2\Mode'
Law	String	'Geometrical Set.1\Sketch.1\Parallelism.3\Activity'
Operations Constructors	Integer	'Geometrical Set.1\Sketch.1\Parallelism.3\Mode'
Line Constructors	Feature	'Geometrical Set.1\Sketch.1\Parallelism.4\Activity'
Circle Constructors	Plane	'Geometrical Set.1\Sketch.1\Parallelism.4\Mode'
String	Solid	'Geometrical Set.1\Sketch.1\espessura\Length'
Direction Constructors	Curve	'Geometrical Set.1\Sketch.1\espessura\Activity'
Electrical Functions	Constraint	'Geometrical Set.1\Sketch.1\espessura\Mode'
List	Point	'Geometrical Set.1\Sketch.1\comprimento\Length'
Measures	Set of parameters	'Geometrical Set.1\Sketch.1\comprimento\Activity'
Surface Constructors	Set Of Relations	'Geometrical Set.1\Sketch.1\comprimento\Mode'
Wireframe Constructors	Design Table	PartBody\Pad.1\FirstLimit\Length
NC Manufacturing resour	Formula	PartBody\Pad.1\SecondLimit\Length
Generative functions	Sheet	PartBody\Pad.1\ThickThin1
Part Measures		PartBody\Pad.1\ThickThin2
Plane Constructors		PartBody\Pad.1\Activity
		PartBody\Hole.1\HoleLimit.1\Depth

# Associando o parâmetro Edge\_type na fórmula

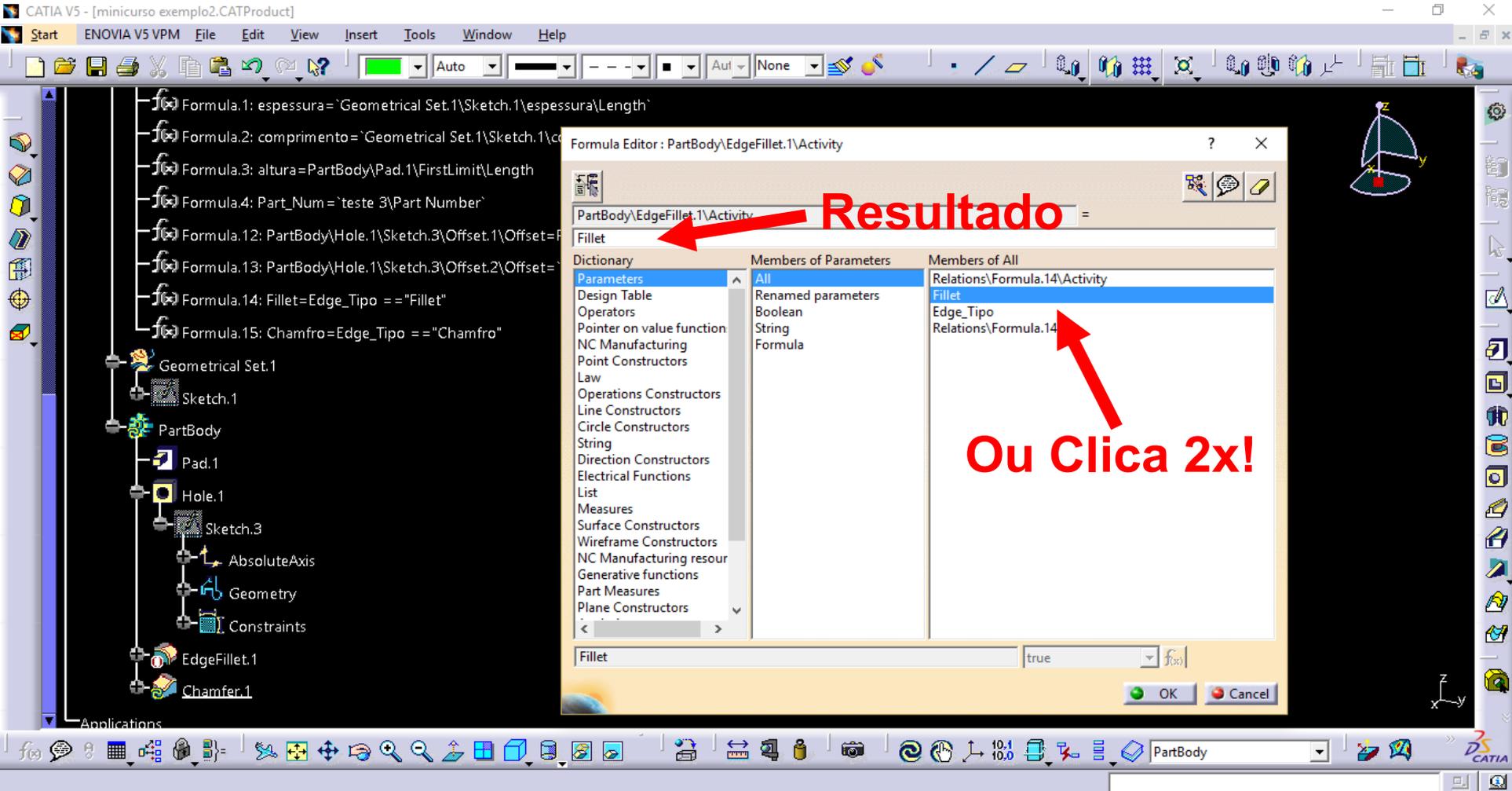


Formula Editor: PartBody\EdgeFillet.1\Activity

Dictionary	Members of Parameters	Members of All
Parameters	All	Relations\Formula.14\Activity
Design Table	Renamed parameters	Fillet
Operators	Boolean	Edge_Tipo
Pointer on value function	String	Relations\Formula.14
NC Manufacturing	Formula	
Point Constructors		
Law		
Operations Constructors		
Line Constructors		
Circle Constructors		
String		
Operation Constructors		
Mathematical Functions		
List		
Measures		
Surface Constructors		
Wireframe Constructors		
NC Manufacturing resour		
Generative functions		
Part Measures		
Plane Constructors		

**2 - Clicar**

# Associando o parâmetro Edge\_type na fórmula



Formula Editor: PartBody\EdgeFillet.1\Activity

PartBody\EdgeFillet.1\Activity =

Dictionary	Members of Parameters	Members of All
Parameters	All	Relations\Formula.14\Activity
Design Table	Renamed parameters	Fillet
Operators	Boolean	Edge_Tipo
Pointer on value function	String	Relations\Formula.14
NC Manufacturing	Formula	
Point Constructors		
Law		
Operations Constructors		
Line Constructors		
Circle Constructors		
String		
Direction Constructors		
Electrical Functions		
List		
Measures		
Surface Constructors		
Wireframe Constructors		
NC Manufacturing resour		
Generative functions		
Part Measures		
Plane Constructors		

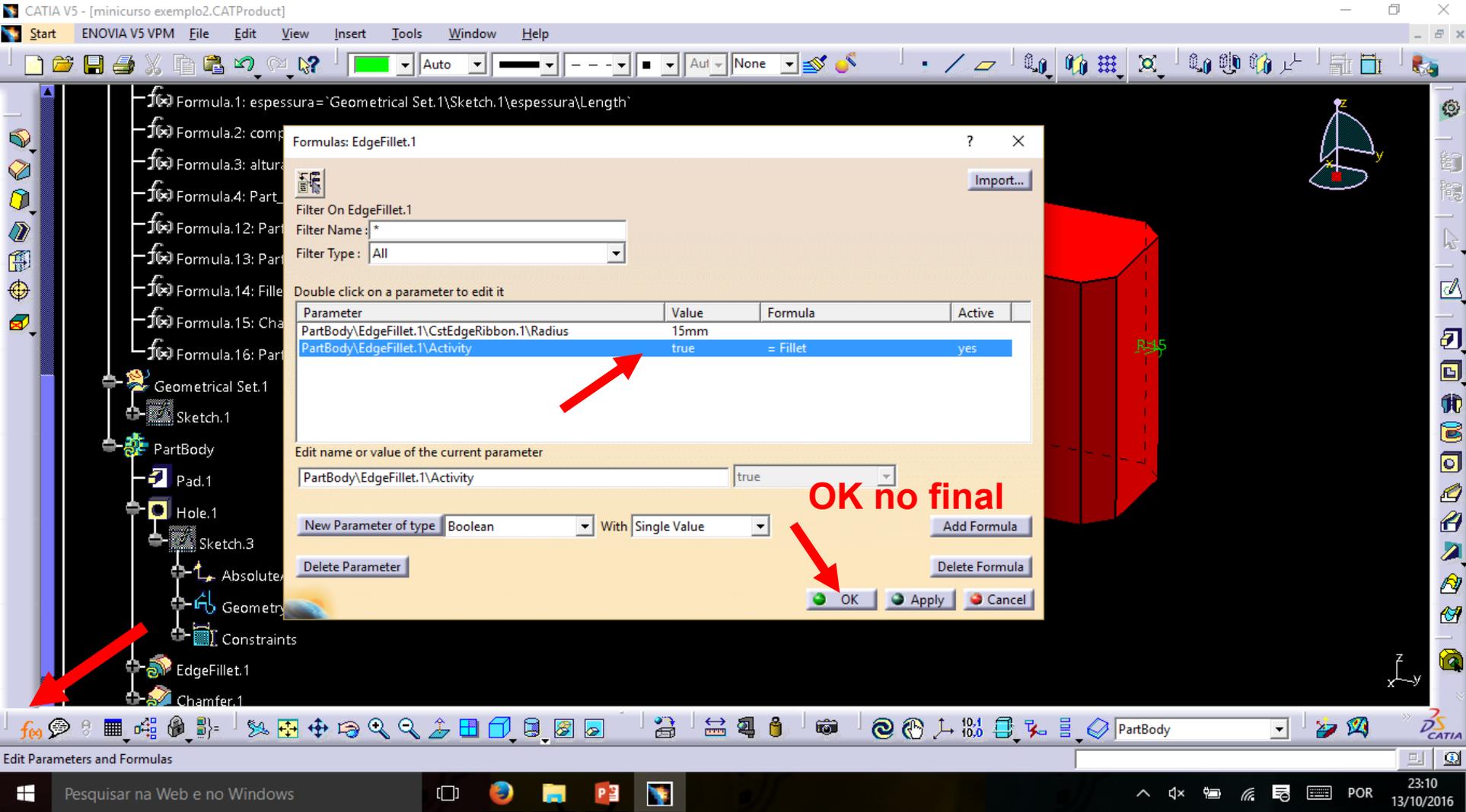
Fillet true

OK Cancel

**Resultado**

**Ou Clica 2x!**

# Associando o parâmetro Edge\_type na fórmula



CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Formulas: EdgeFillet.1

Filter On EdgeFillet.1

Filter Name: \*

Filter Type: All

Double click on a parameter to edit it

Parameter	Value	Formula	Active
PartBody\EdgeFillet.1\CstEdgeRibbon.1\Radius	15mm		
PartBody\EdgeFillet.1\Activity	true	= Fillet	yes

Edit name or value of the current parameter

PartBody\EdgeFillet.1\Activity true

New Parameter of type Boolean With Single Value

OK Apply Cancel

**OK no final**

PartBody

Pesquisar na Web e no Windows

23:10 13/10/2016



# Associando o parâmetro Edge\_type na fórmula

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

100% None

Formula.1: espessura = `Geometrical Set.1\Sketch.1\espessura\Length`  
 Formula.2: comprimento = `Geometrical Set.1\Sketch.1\comprimento\Length`  
 Formula.3: altura = PartBody\Pad.1\FirstLimit\Length  
 Formula.4: Part\_Num = "teste 3\Part Number"  
 Formula.12: PartBody\Hole.1\Sketch.3\Offset.1\Offset = PartBody\Pad.1\FirstLimit\Length /2  
 Formula.13: PartBody\Hole.1\Sketch.3\Offset.2\Offset = `Geometrical Set.1\Sketch.1\comprimento\Length` /2  
 Formula.14: Fillet = Edge\_Tipo == "Fillet"  
 Formula.15: Chamfro = Edge\_Tipo == "Chamfro"  
 Formula.16: PartBody\EdgeFillet.1\Activity = Fillet

Geometrical Set.1  
 Sketch.1  
 PartBody  
 Pad.1  
 Hole.1  
 Sketch.3  
 AbsoluteAxis  
 Geometry  
 Constraints  
 EdgeFillet.1  
 Chamfer.1

Update Diagnosis: teste 3

Feature	Diagnosis
Chamfer.1	Edges or face not found Re-select the edges or faces on current ...
Edge.5	Edge not found
Edge.6	Edge not found
Edge.7	Edge not found

Edges or face not found  
 Re-select the edges or faces on current geometry

Close

Chamfer.1/PartBody/teste 3 selected



# Associando o parâmetro Edge\_type na fórmula

CATIA V5 - [minicurso exemplo.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

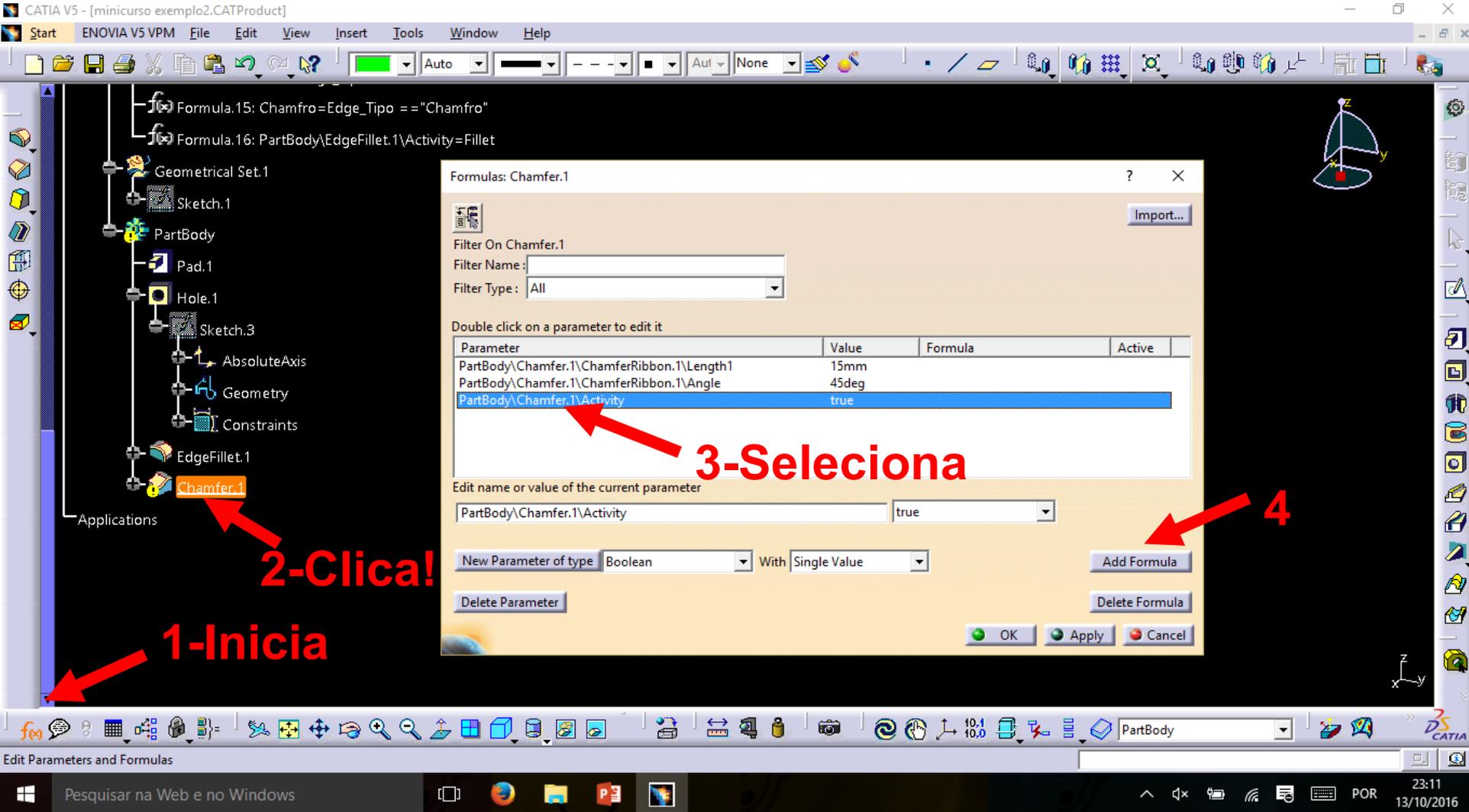
yz plane  
zx plane  
Parameters  
 espessura=3750mm = `Geometrical Set.1\Sketch.1\espessura\Length`  
 comprimento=7000mm = `Geometrical Set.1\Sketch.1\comprimento\Length`  
 altura=6500mm = PartBody\Pad.1\FirstLimit\Length  
 Part\_Num = teste 4 = `teste 1\Part Number`  
 Edge\_tipo = Fillet  
 Fillet = true = Edge\_tipo == "Fillet"  
 Chamfro = false = Edge\_tipo == "Chamfro"

Relations  
 DesignTable.1  
 Formula.1: espessura = `Geometrical Set.1\Sketch.1\espessura\Length`  
 Formula.2: comprimento = `Geometrical Set.1\Sketch.1\comprimento\Length`  
 Formula.4: altura = PartBody\Pad.1\FirstLimit\Length  
 Formula.5: Part\_Num = `teste 4\Part Number`  
 Formula.6: PartBody\Hole.1\Sketch.2\Offset.8\Offset = PartBody\Pad.1\FirstLimit\Length /2  
 Formula.7: PartBody\Hole.1\Sketch.2\Offset.7\Offset = `Geometrical Set.1\Sketch.1\comprimento\Length` /2  
 Formula.11: Fillet = Edge\_tipo == "Fillet"

PartBody

Select an object or a command

# Associando o parâmetro Edge\_type na fórmula - CHAMFRO



CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Formula.15: Chamfro=Edge\_Tipo == "Chamfro"

Formula.16: PartBody\EdgeFillet.1\Activity=Fillet

Geometrical Set.1

Sketch.1

PartBody

Pad.1

Hole.1

Sketch.3

AbsoluteAxis

Geometry

Constraints

EdgeFillet.1

Chamfer.1

Applications

Formulas: Chamfer.1

Filter On Chamfer.1

Filter Name:

Filter Type: All

Double click on a parameter to edit it

Parameter	Value	Formula	Active
PartBody\Chamfer.1\ChamferRibbon.1\Length1	15mm		
PartBody\Chamfer.1\ChamferRibbon.1\Angle	45deg		
PartBody\Chamfer.1\Activity	true		

Edit name or value of the current parameter

PartBody\Chamfer.1\Activity true

New Parameter of type Boolean With Single Value

Delete Parameter

Add Formula

Delete Formula

OK Apply Cancel

1-Inicia

2-Clica!

3-Seleciona

4

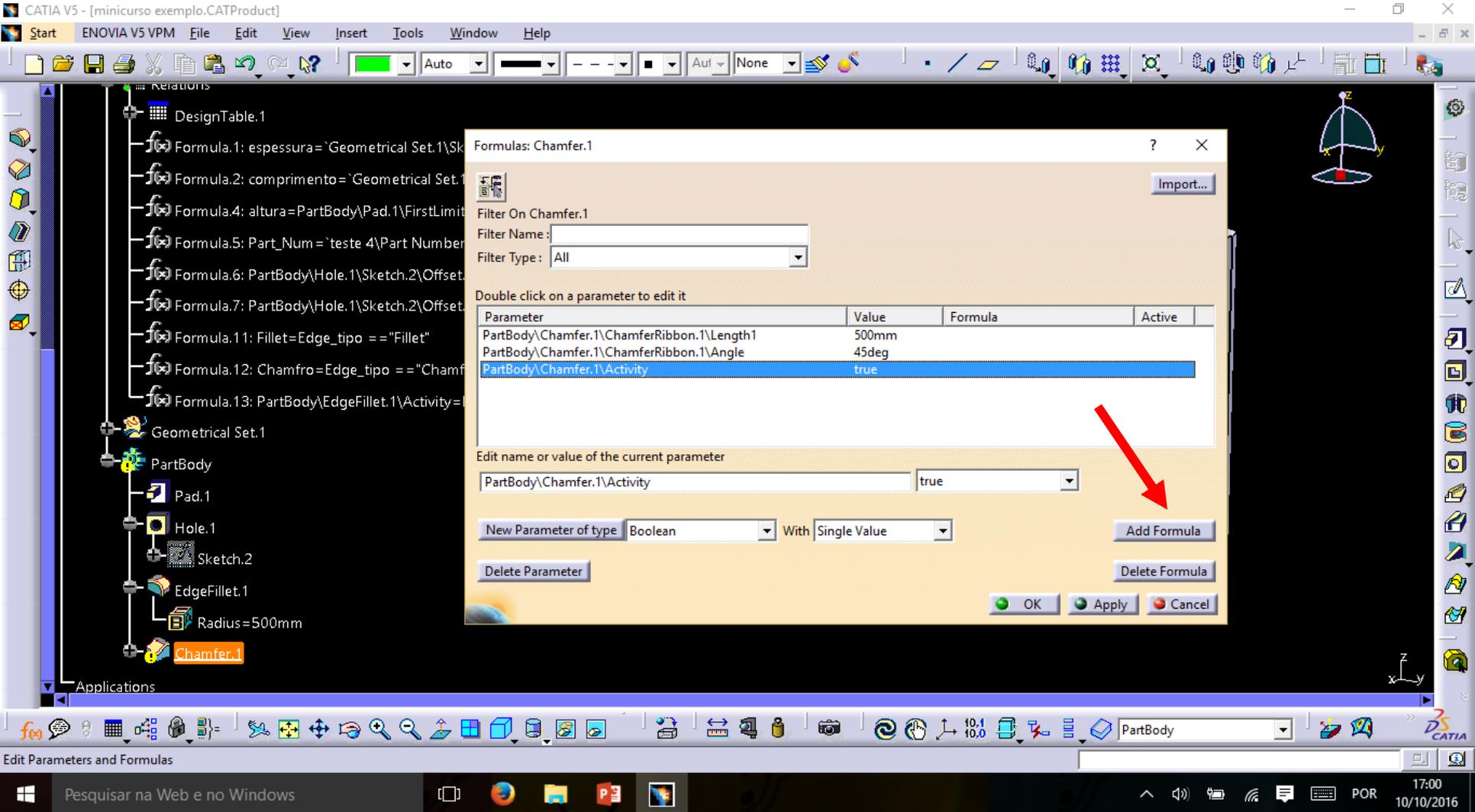
PartBody

Edit Parameters and Formulas

Pesquisar na Web e no Windows

23:11 13/10/2016

# Associando o parâmetro Edge\_type na fórmula



CATIA V5 - [minicurso exemplo.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Relations

- DesignTable.1
  - Formula.1: espessura='Geometrical Set.1\Sk...
  - Formula.2: comprimento='Geometrical Set.1...
  - Formula.4: altura=PartBody\Pad.1\FirstLimit...
  - Formula.5: Part\_Num='teste 4\Part Number...
  - Formula.6: PartBody\Hole.1\Sketch.2\Offset...
  - Formula.7: PartBody\Hole.1\Sketch.2\Offset...
  - Formula.11: Fillet=Edge\_tipo == "Fillet"
  - Formula.12: Chamfro=Edge\_tipo == "Chamf...
  - Formula.13: PartBody\EdgeFillet.1\Activity=...
- Geometrical Set.1
- PartBody
  - Pad.1
  - Hole.1
    - Sketch.2
  - EdgeFillet.1
    - Radius=500mm
    - Chamfer.1

Applications

Formulas: Chamfer.1

Filter On Chamfer.1

Filter Name:

Filter Type: All

Double click on a parameter to edit it

Parameter	Value	Formula	Active
PartBody\Chamfer.1\ChamferRibbon.1\Length1	500mm		
PartBody\Chamfer.1\ChamferRibbon.1\Angle	45deg		
PartBody\Chamfer.1\Activity	true		

Edit name or value of the current parameter

PartBody\Chamfer.1\Activity true

New Parameter of type Boolean With Single Value

Delete Parameter

Add Formula

Delete Formula

OK Apply Cancel

PartBody

Edit Parameters and Formulas

Pesquisar na Web e no Windows

17:00 10/10/2016

# Associando o parâmetro Edge\_type na fórmula

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

PartBody\Chamfer.1\Activity =

Dictionary	Members of Parameters	Members of All
Parameters	All	Relations\Formula.15\Activity
Design Table	Renamed parameters	Chamfro
	Boolean	Edge_Tipo
	String	Relations\Formula.15
	Formula	

Chamfro false

OK Cancel

1 - Clica!

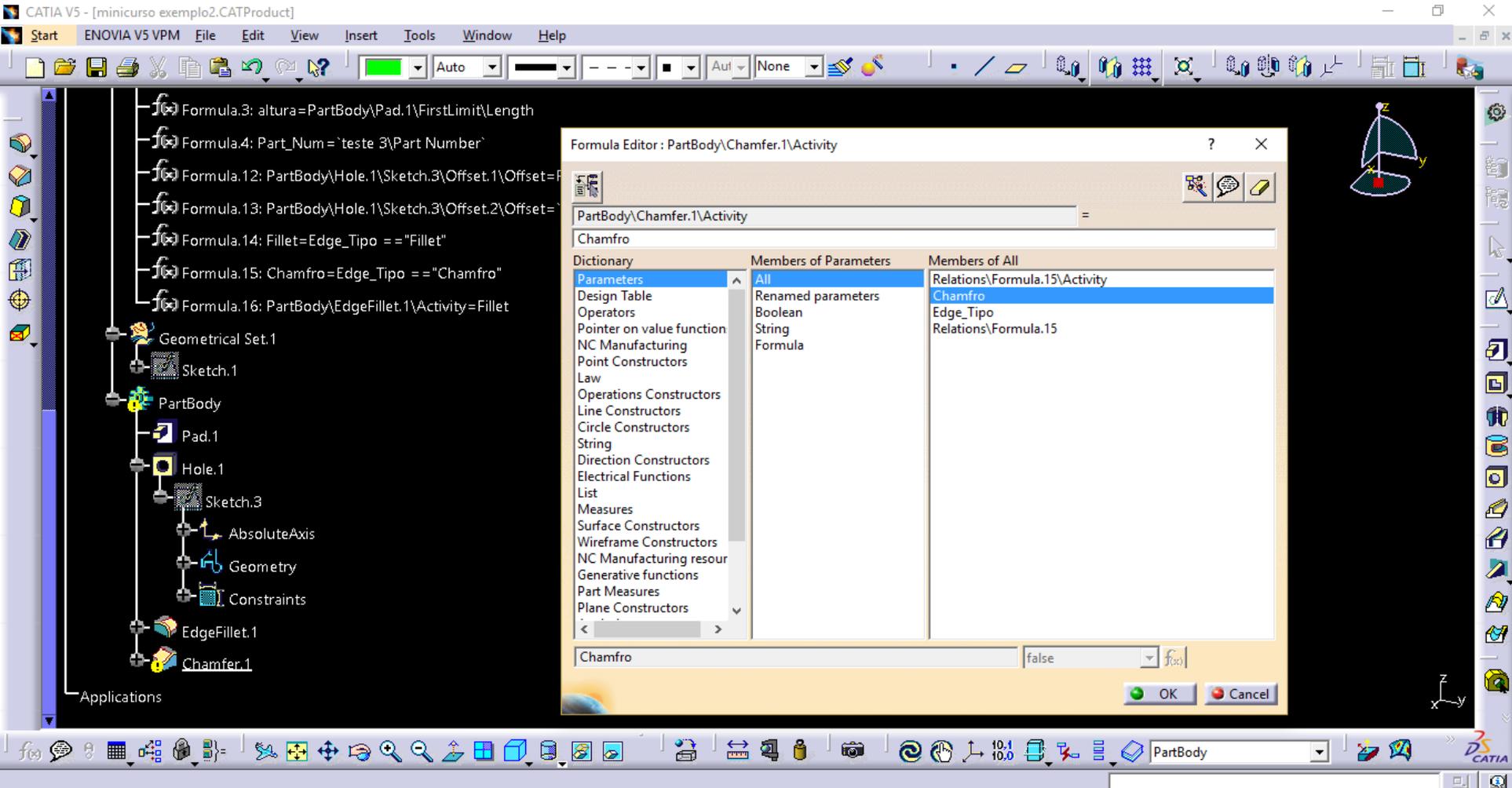
Ou Clica 2x!

Formula.15/Relations/teste 3 preselected

Pesquisar na Web e no Windows

23:13 13/10/2016

# Associando o parâmetro Edge\_type na fórmula



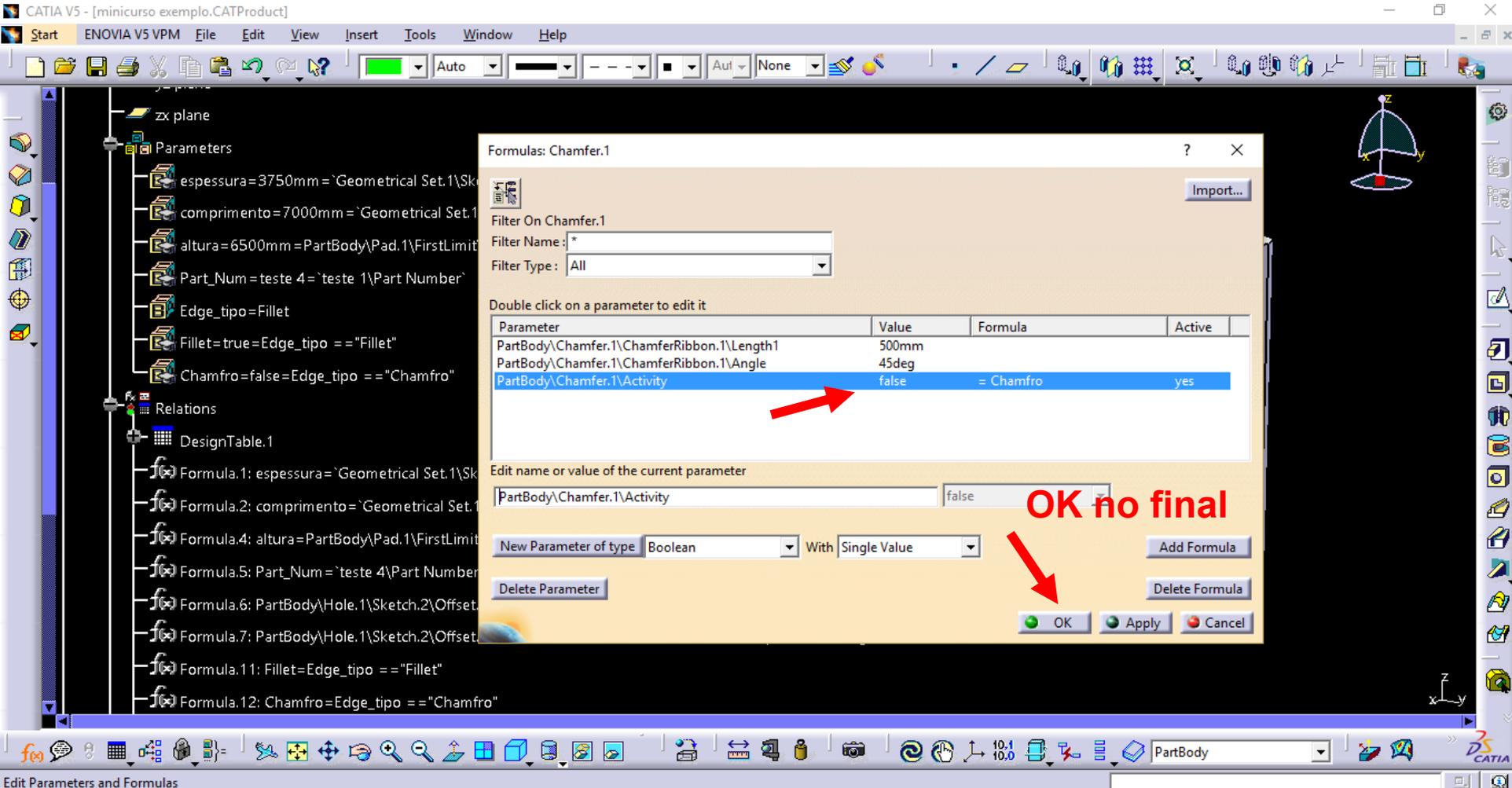
The screenshot displays the CATIA V5 interface with the Formula Editor dialog box open for the 'Chamfer.1' activity. The dialog box contains the following information:

- Formula Editor:** PartBody\Chamfer.1\Activity
- Formula:** Chamfro =
- Members of Parameters:** All, Renamed parameters, Boolean, String, Formula
- Members of All:** Relations\Formula.15\Activity, Chamfro, Edge\_Tipo, Relations\Formula.15
- Current Value:** false

The background shows the CAD tree with the following structure:

- Formula.3: altura=PartBody\Pad.1\FirstLimit\Length
- Formula.4: Part\_Num = 'teste 3\Part Number'
- Formula.12: PartBody\Hole.1\Sketch.3\Offset.1\Offset=...
- Formula.13: PartBody\Hole.1\Sketch.3\Offset.2\Offset=...
- Formula.14: Fillet=Edge\_Tipo == "Fillet"
- Formula.15: Chamfro=Edge\_Tipo == "Chamfro"
- Formula.16: PartBody\EdgeFillet.1\Activity=Fillet
- Geometrical Set.1
  - Sketch.1
  - PartBody
    - Pad.1
    - Hole.1
      - Sketch.3
        - AbsoluteAxis
        - Geometry
        - Constraints
    - EdgeFillet.1
    - Chamfer.1

# Associando o parâmetro Edge\_type na fórmula



Formulas: Chamfer.1

Filter On Chamfer.1

Filter Name: \*

Filter Type: All

Double click on a parameter to edit it

Parameter	Value	Formula	Active
PartBody\Chamfer.1\ChamferRibbon.1\Length1	500mm		
PartBody\Chamfer.1\ChamferRibbon.1\Angle	45deg		
PartBody\Chamfer.1\Activity	false	= Chamfro	yes

Edit name or value of the current parameter

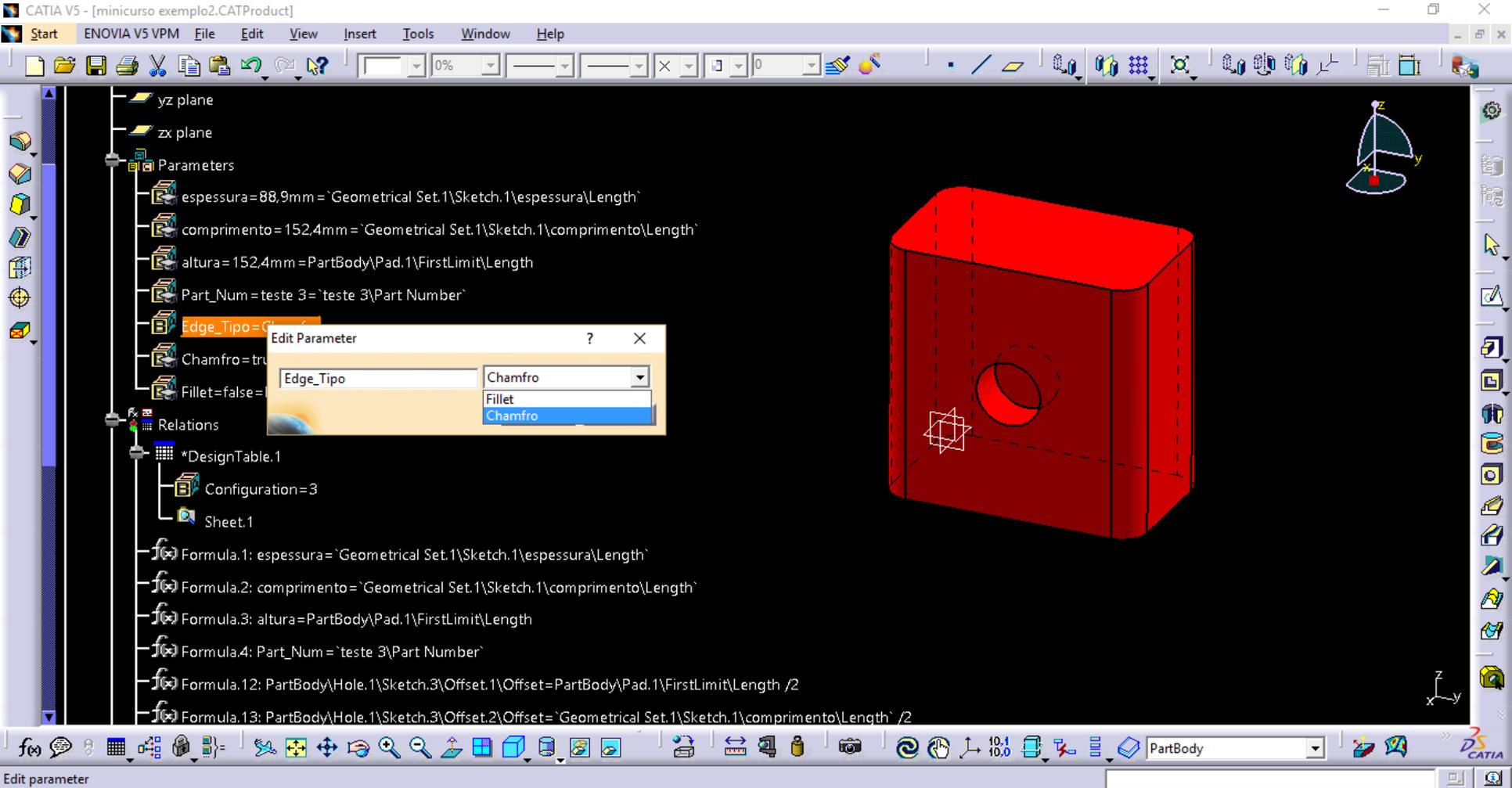
PartBody\Chamfer.1\Activity false

New Parameter of type Boolean With Single Value

OK Apply Cancel

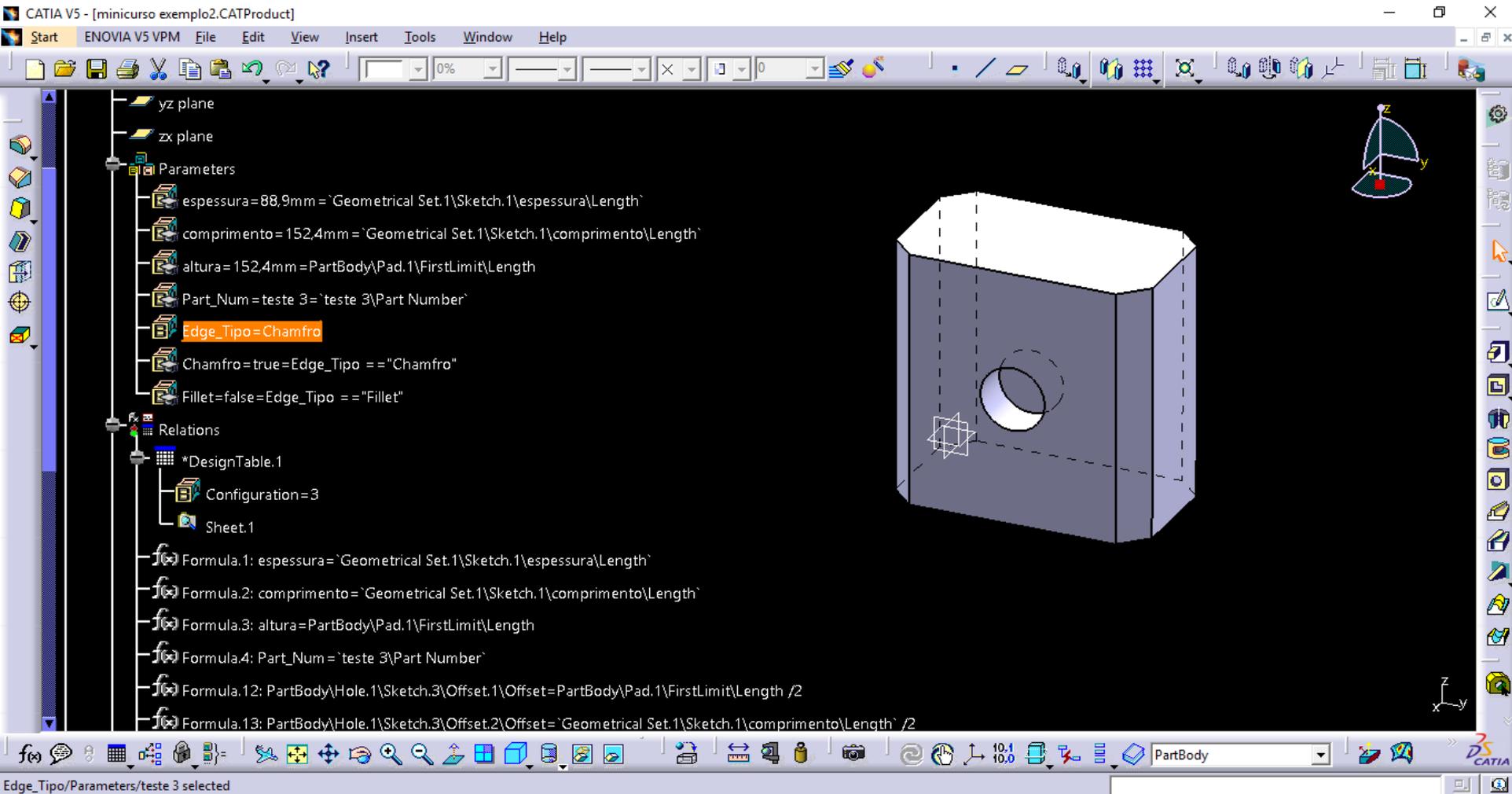
**OK no final**

# Verificando e fazendo a mudança de fillet p/ chanfro





# Verificando e fazendo a mudança de fillet p/ chanfro



# Fazendo a mudança do modelo usando Design Table

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

yz plane

\*DesignTable.1 active, configuration row : 4

Design Table Properties

Name: \*DesignTable.1 Activity

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter: Edit...

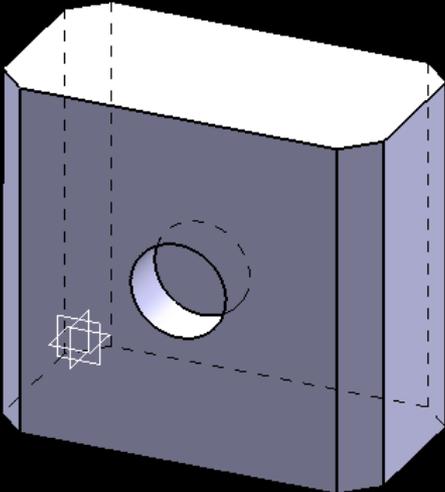
Line	Geometrical Set.1\Sk...	Geometrical Set.1\S...	PartBody\Pad.1\FirstLimit\Length	teste 1\Part ...	PartBody\Hole.1\Diam
<1>	76,2mm	101,6mm	127mm	teste 1	25,4mm
2	82,55mm	127mm	139,7mm	teste 2	31,75mm
3	88,9mm	152,4mm	152,4mm	teste 3	38,1mm
4	95,25mm	177,8mm	165,1mm	teste 4	44,45mm

Edit table...  Duplicate data in CATIA model

OK Apply Cancel

Formula.12: PartBody\Hole.1\Sketch.3\Offset.1\Offset=PartBody\Pad.1\FirstLimit\Length /2

Formula.13: PartBody\Hole.1\Sketch.3\Offset.2\Offset='Geometrical Set.1\Sketch.1\comprimento'\Length' /2



InitialState

Pesquisar na Web e no Windows

23:16 13/10/2016

# Fazendo a mudança do modelo usando Design Table

CATIA V5 - [minicurso exemplo2.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

yz plane

\*DesignTable.1 active, configuration row : 1

Design Table Properties

Name: \*DesignTable.1 Activity

Comment: DesignTable created by Mateus 13/10/2016

Configurations Associations

Filter: Edit...

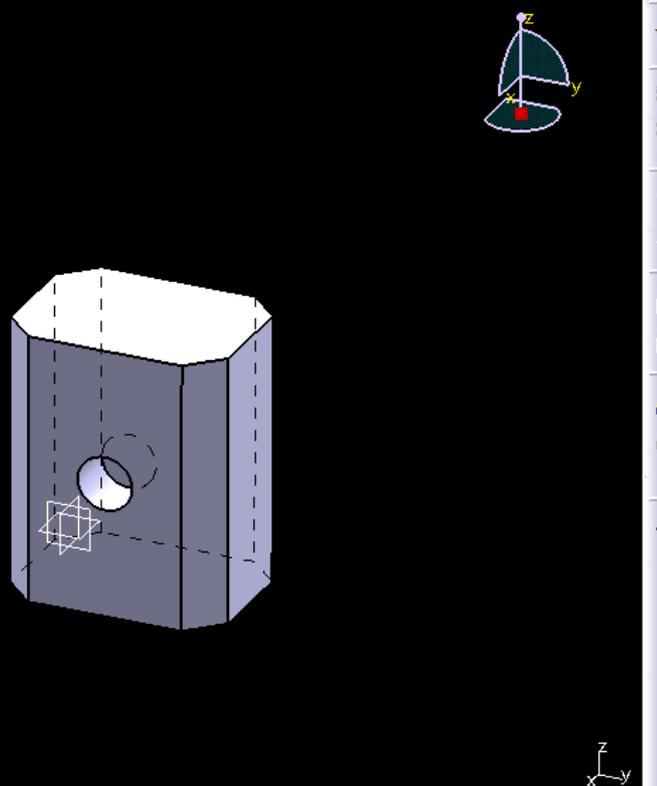
Line	'Geometrical Set.1\Sk...	'Geometrical Set.1\S...	PartBody\Pad.1\FirstLimit\Length	'teste 1\Part ...	PartBody\Hole.1\Diarr
<1>	76,2mm	101,6mm	127mm	teste 1	25,4mm
2	82,55mm	127mm	139,7mm	teste 2	31,75mm
3	88,9mm	152,4mm	152,4mm	teste 3	38,1mm
4	95,25mm	177,8mm	165,1mm	teste 4	44,45mm

Edit table...  Duplicate data in CATIA model

OK Apply Cancel

Formula.12: PartBody\Hole.1\Sketch.3\Offset.1\Offset='PartBody\Pad.1\FirstLimit\Length' /2

Formula.13: PartBody\Hole.1\Sketch.3\Offset.2\Offset='Geometrical Set.1\Sketch.1\comprimento\Length' /2

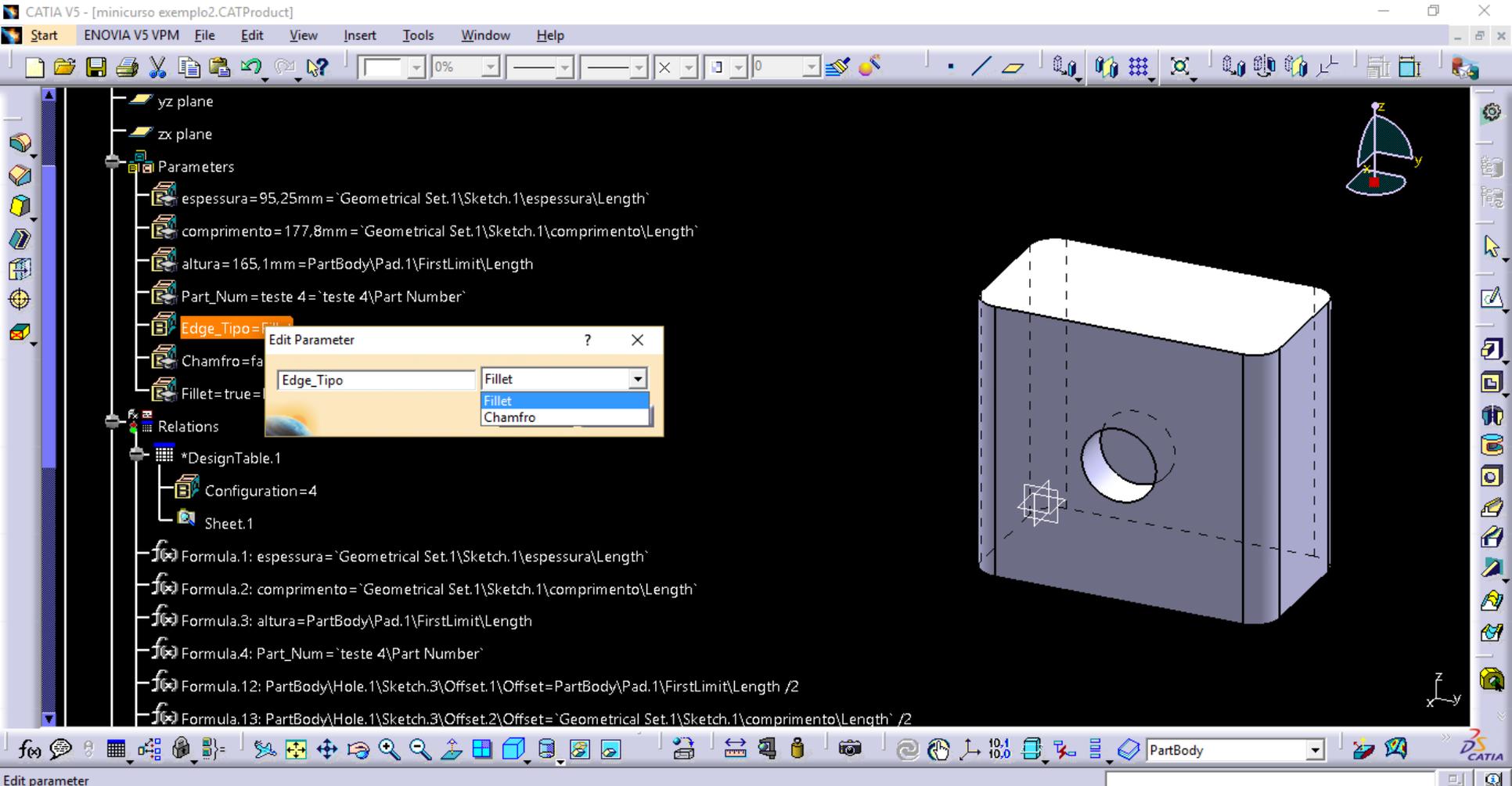


This list contains the available configurations of the design table. The selected row is the current one.

Pesquisar na Web e no Windows

23:16 13/10/2016

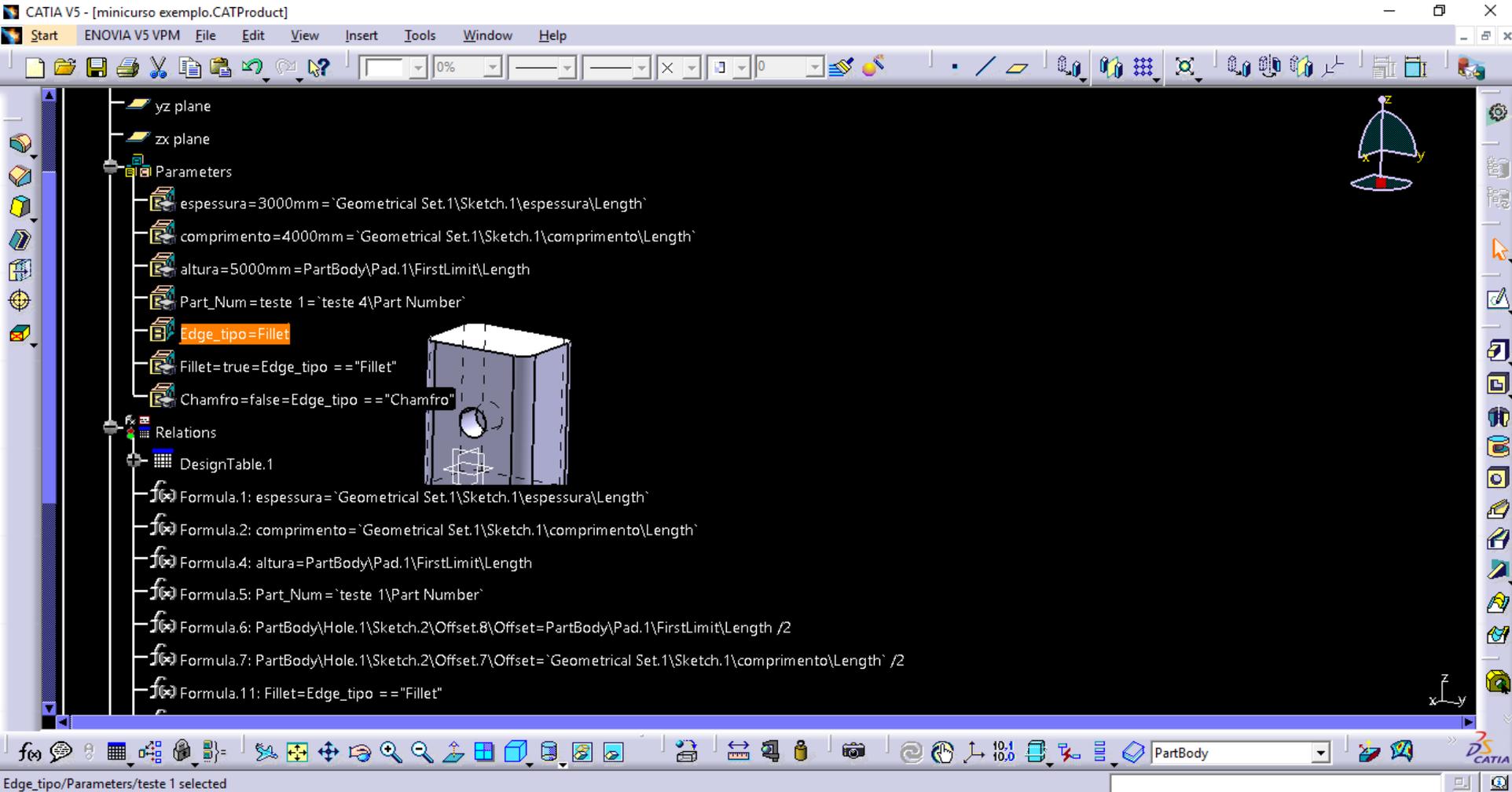
# Fazendo a mudança do modelo usando Design Table



Edit parameter

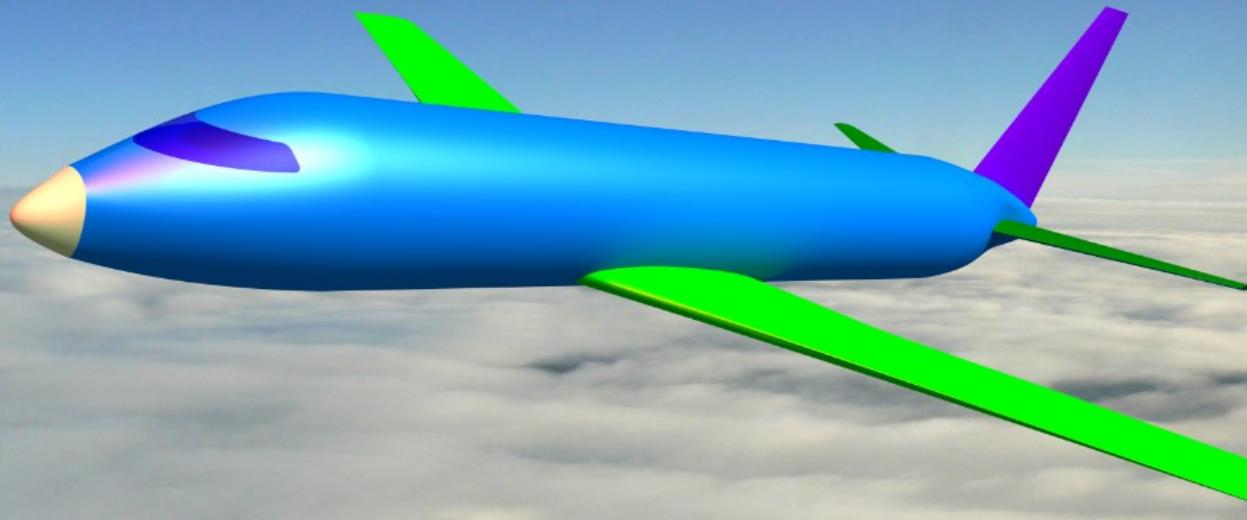


# Fazendo a mudança do modelo usando Design Table



## Parte 2

# Inserindo os Parâmetros para um projeto de aeronave



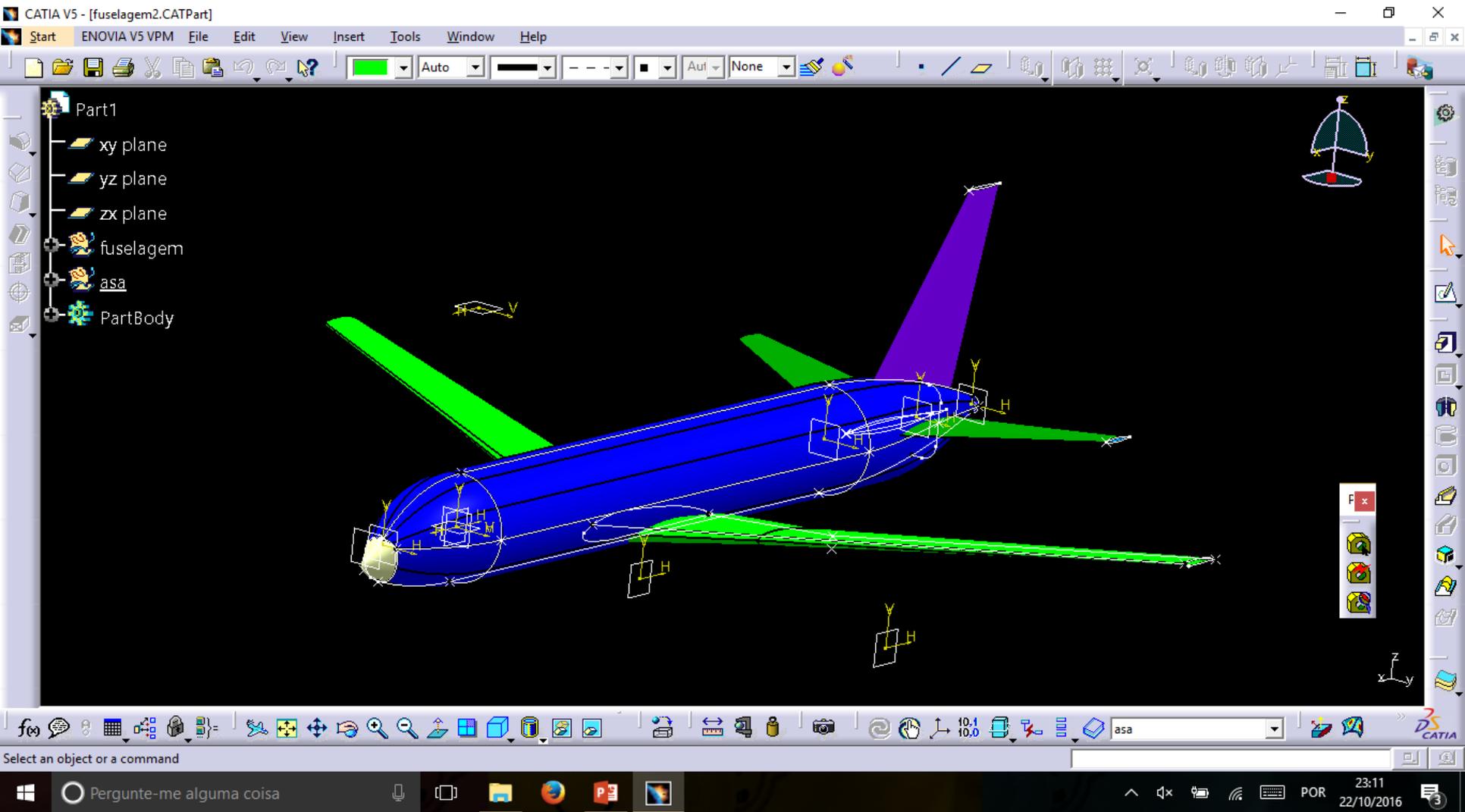
## Procedimentos iniciais:

- 1 - Fazer o download do modelo da aeronave no moodle.
- 2 – Gerar a tabela no MS Excel
- 3 – Inserir os parâmetros segundos os passos do tutorial anterior.
- 4 – Testar a mudança dos conceitos.

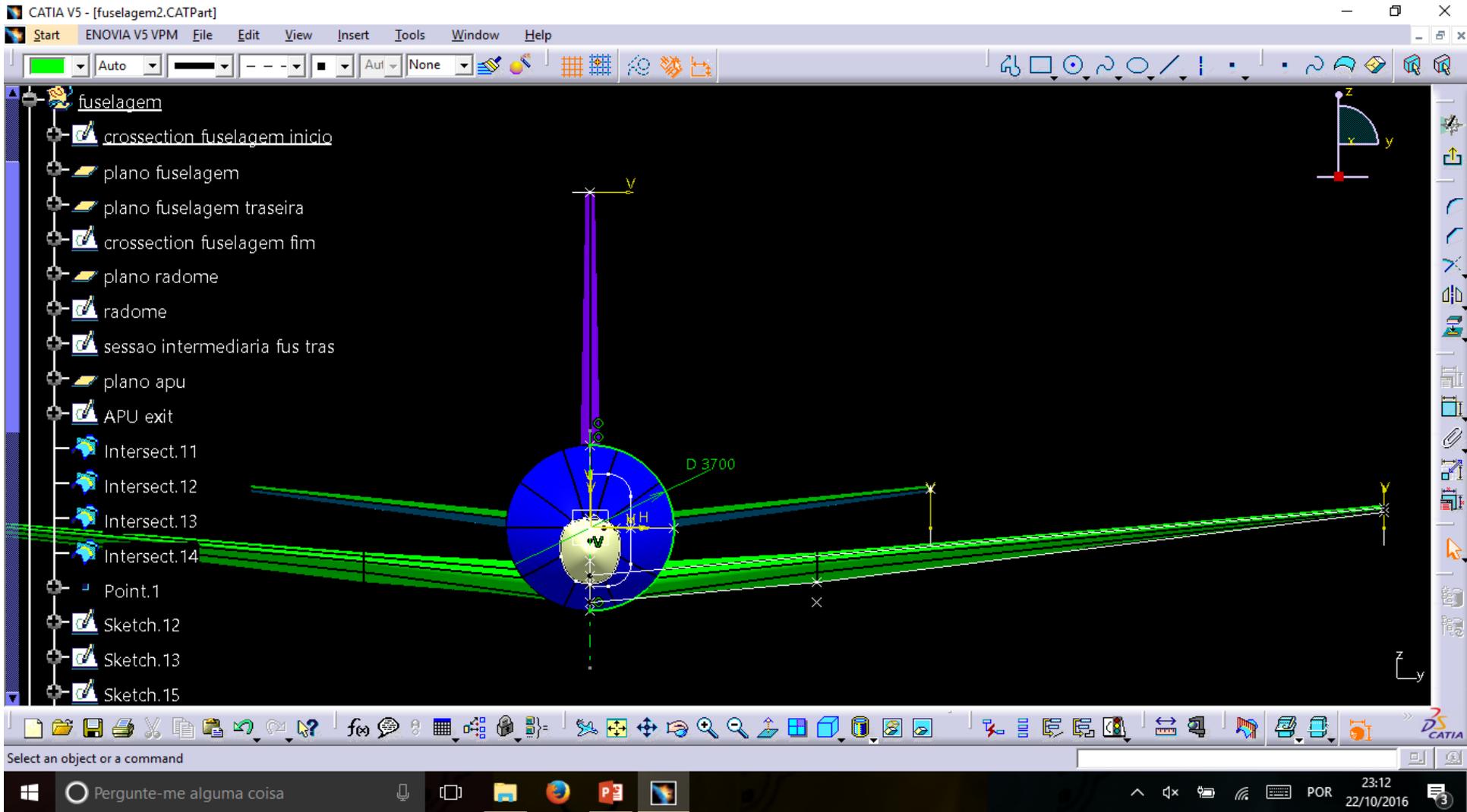
## Tabela em MS Excel dos parâmetros de projeto das aeronaves.

Conceito Aeronave	comp fuselagem section central (mm)	posicao inicio asa (mm)	comp cockpit section (mm)	raio fuselagem (mm)	comp fuselagem section traseira (mm)	evergadura (mm)
conceito 1	20000	12000	4000	3700	5000	17500
conceito 2	25000	13000	4000	3700	5000	17500
conceito 3	30000	17000	4900	4500	6000	20000
conceito 4	35000	20000	4900	4500	6000	20000

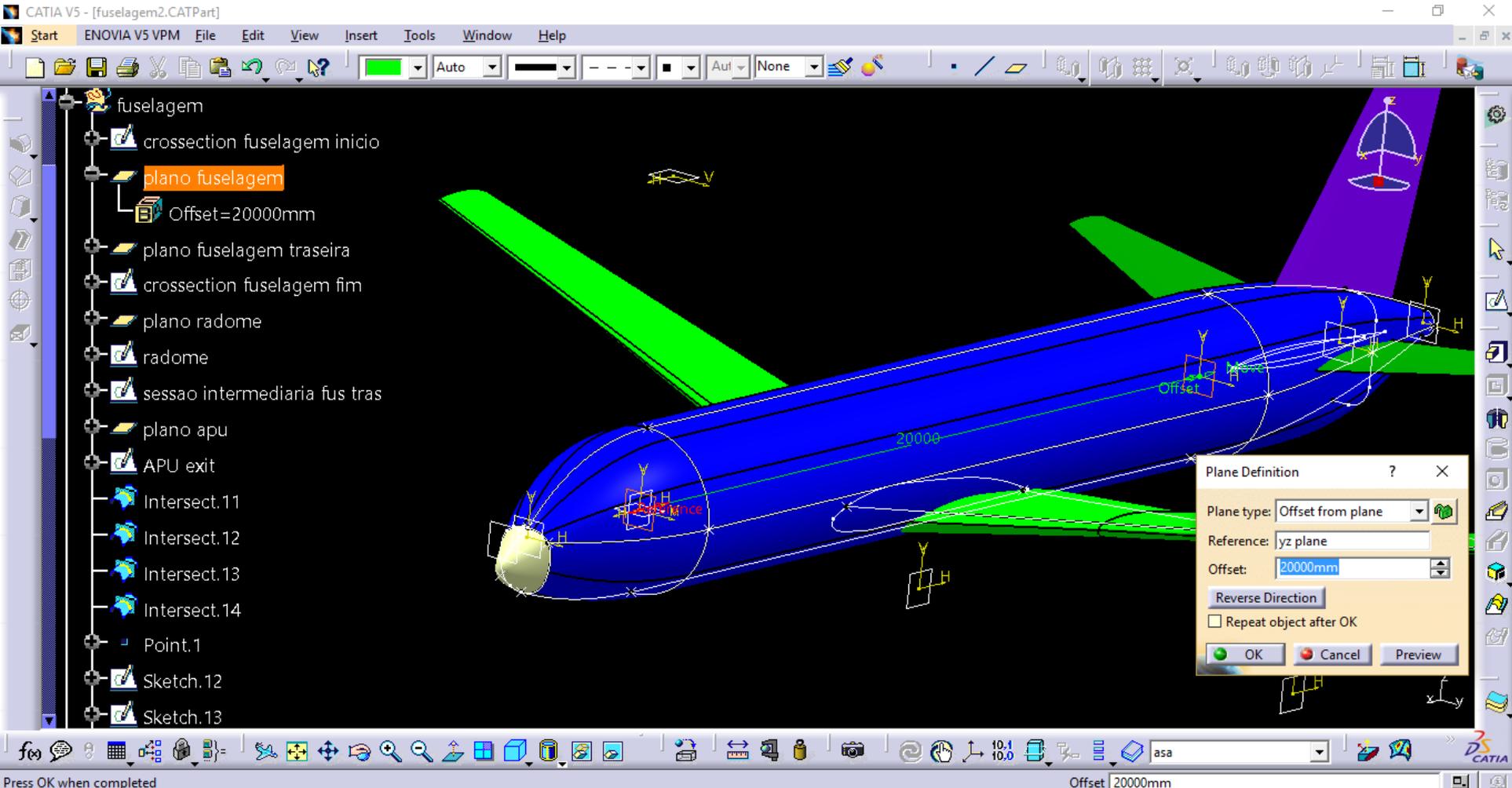
# Projeto da Aeronave no Catia



# Sketches e constrains a serem parametrizadas. Diâmetro da fuselagem

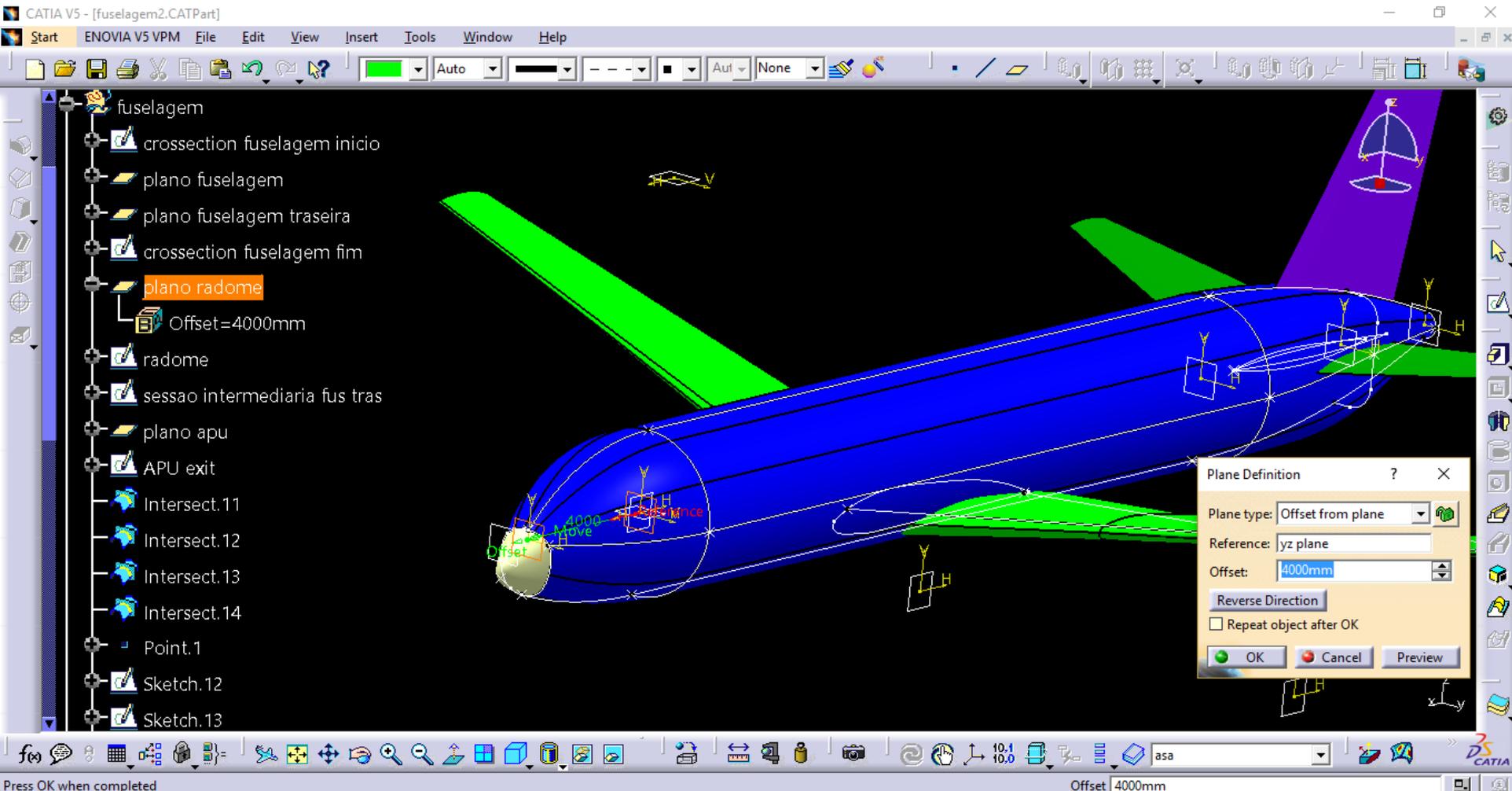


# Sketches e constrains a serem parametrizadas. Comprimento da fuselagem principal



Press OK when completed

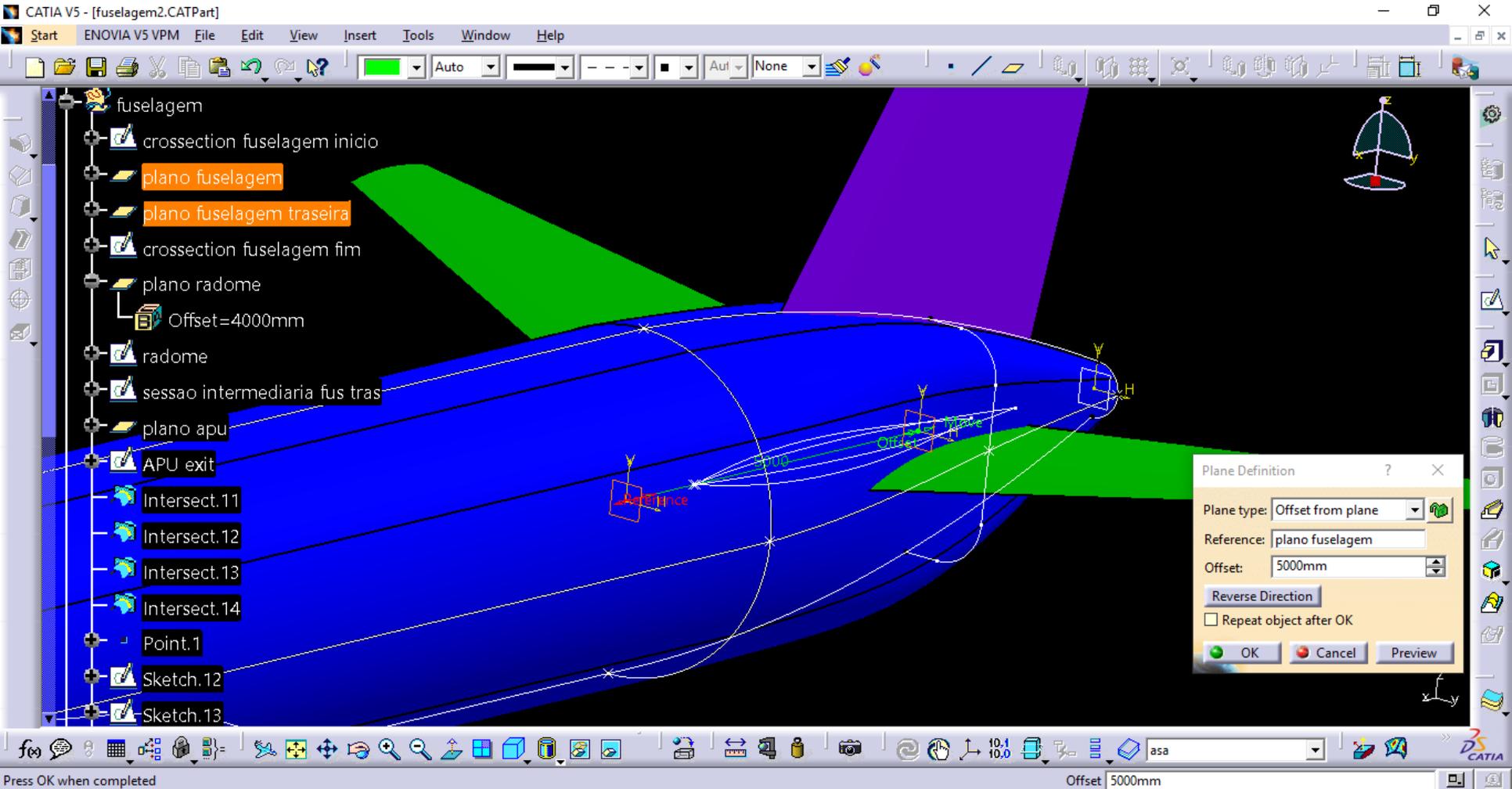
# Sketches e constrains a serem parametrizadas. Comprimento do cockpit



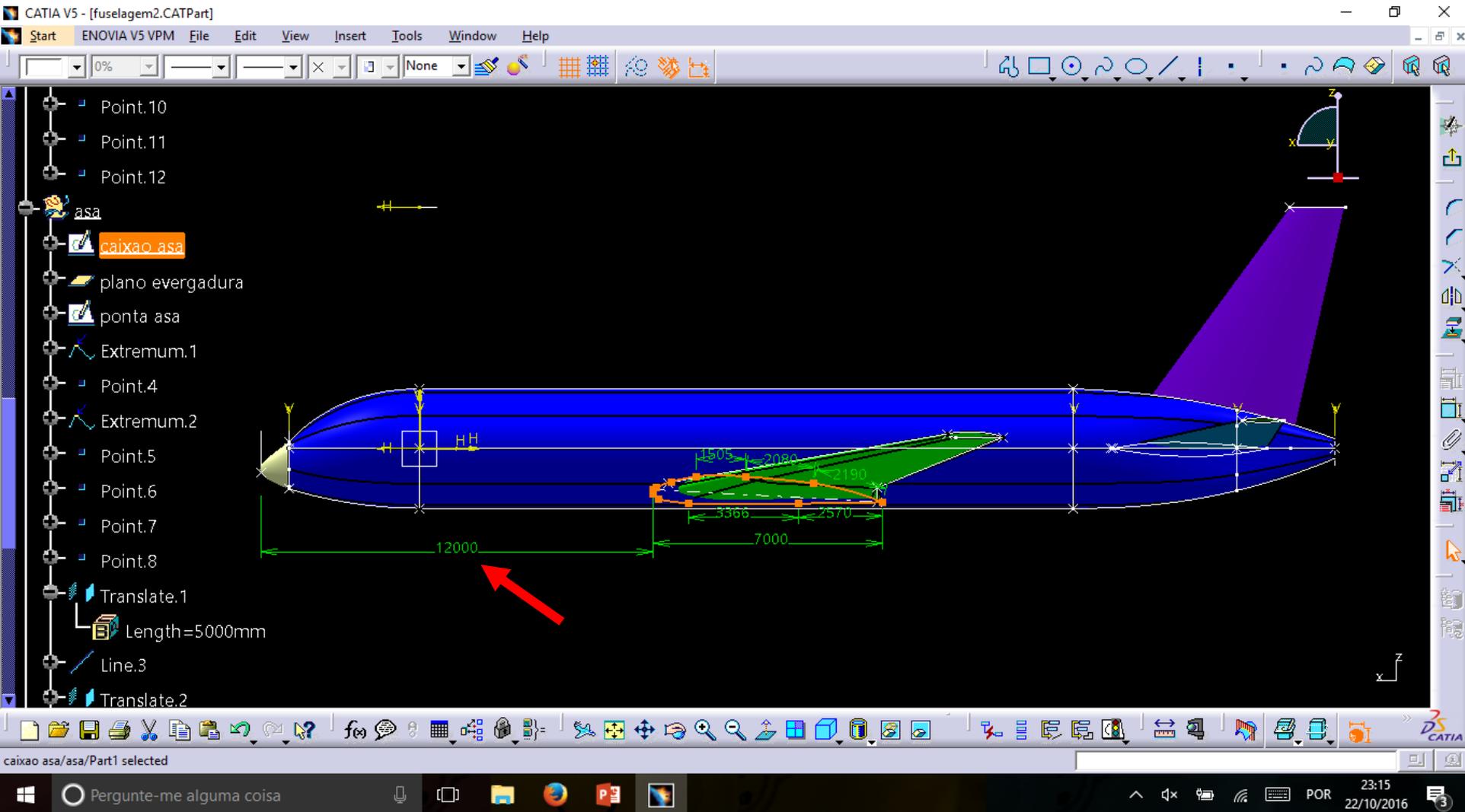
Press OK when completed

Offset | 4000mm

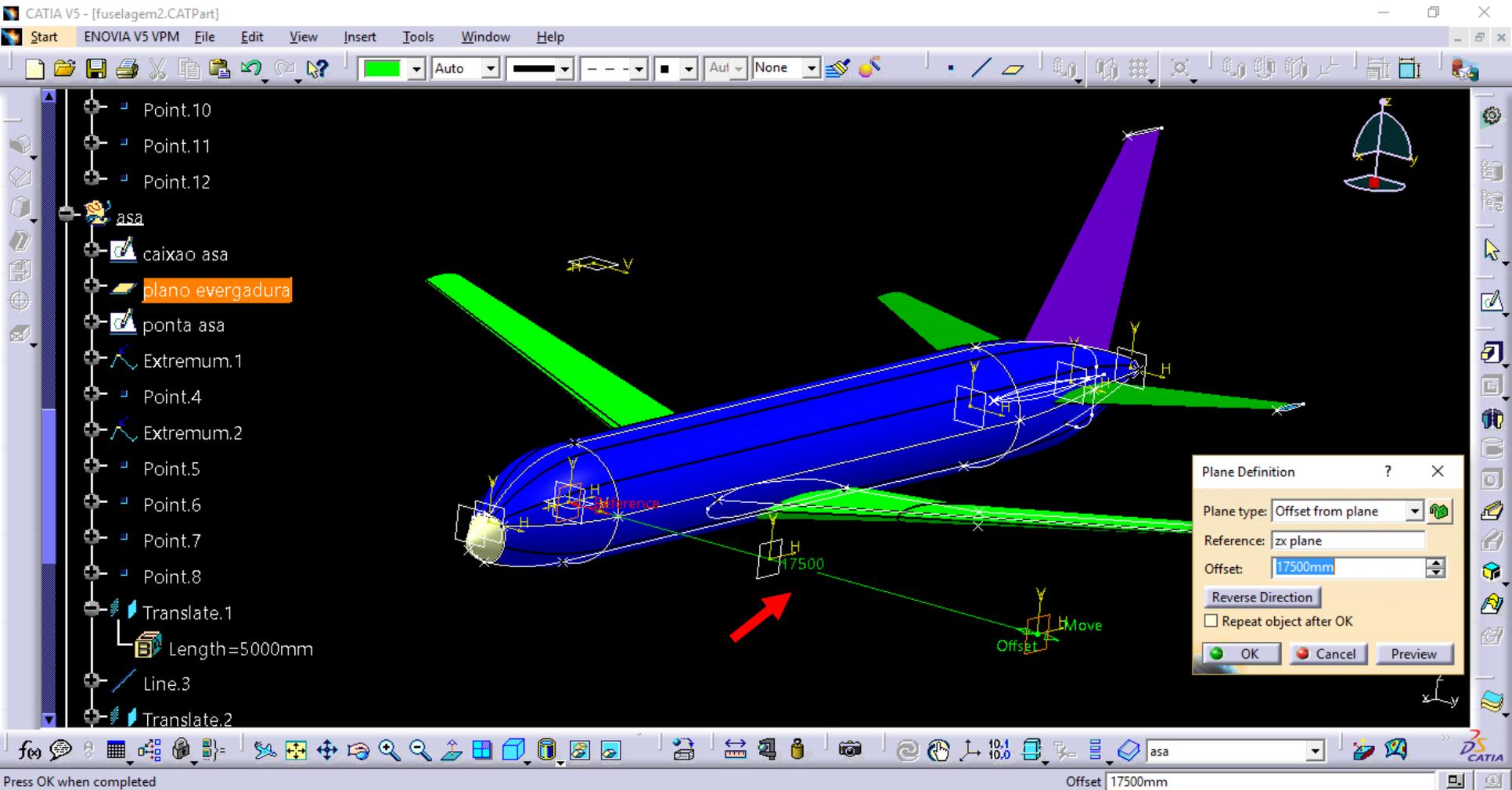
# Sketches e constrains a serem parametrizadas. Comprimento da calda



# Sketches e constrains a serem parametrizadas. Posição inicial da asa



# Sketches e constrains a serem parametrizadas. Evergadura da asa



Press OK when completed



# Parâmetros usados

CATIA V5 - [aeronave teste knowledge exercicio.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

Parameters

- comp\_fuselagem\_section\_central=20000mm
- posicao\_inicio\_asa=12000mm
- comp\_cockpit\_section=4000mm
- raio\_fuselagem=3700mm
- comp\_fuselagem\_section\_traseira=5000mm
- evergadura=17500mm
- Conceito=conceito 2

Relations

- Formula.2: `asa\caixao asa\Offset.215\Offset`=posicao\_inicio\_asa
- Formula.3: `fuselagem\plano radome\Offset`=comp\_cockpit\_section
- Formula.5: `fuselagem\crosssection fuselagem inicio\Radius.1\Radius`=raio\_fuselagem /2
- conceito aeronave
- Formula.6: `fuselagem\plano fuselagem traseira\Offset`=comp\_fuselagem\_section\_traseira
- Formula.7: `asa\plano evergadura\Offset`=evergadura
- Formula.8: `fuselagem\plano fuselagem\Offset`=comp\_fuselagem\_section\_central

fuselagem

asa

conceito aeronave/Relations/conceito 1 selected

Pergunte-me alguma coisa

23:21 22/10/2016



# Resultado final – conceito 1

CATIA V5 - [aeronave teste knowledge exercicio.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

conceito aeronave active, configuration row : 1

Design Table Properties

Name: conceito aeronave

Comment: DesignTable created by Mateus 22/10/2016

Configurations Associations

Filter:

Line	evergadura	comp_fuselagem_section_central	posicao_inicio_asa	comp_cockpit_section	raio_fuselagem	comp_fuselagem_section_traseira
< 1 >	17500mm	20000mm	12000mm	4000mm	3700mm	5000mm
2	17500mm	25000mm	13000mm	4000mm	3700mm	5000mm
3	20000mm	30000mm	17000mm	4900mm	4500mm	6000mm
4	20000mm	35000mm	20000mm	4900mm	4500mm	6000mm

OK Apply Cancel

Duplicate data in CATIA model

Formula.3: `fuselagem\plano radome\Offset`=comp\_cockpit\_section

Formula.5: `fuselagem\crosssection fuselagem inicio\Radius.1\Radius`=raio\_fuselagem /2

conceito aeronave

Formula.6: `fuselagem\plano fuselagem traseira\Offset`=comp\_fuselagem\_section\_traseira

Formula.7: `asa\plano evergadura\Offset`=evergadura

Formula.8: `fuselagem\plano fuselagem\Offset`=comp\_fuselagem\_section\_central

fuselagem

InitialState

Pergunte-me alguma coisa

23:19 22/10/2016



# Resultado final – conceito 2

CATIA V5 - [aeronave teste knowledge exercicio.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

conceito aeronave active, configuration row : 2

Design Table Properties

Name: conceito aeronave

Comment: DesignTable created by Mateus 22/10/2016

Configurations Associations

Filter:

Line	evergadura	comp_fuselagem_section_central	posicao_inicio_asa	comp_cockpit_section	raio_fuselagem	comp_fuselagem_section_traseira
<1>	17500mm	20000mm	12000mm	4000mm	3700mm	5000mm
2	17500mm	25000mm	13000mm	4000mm	3700mm	5000mm
3	20000mm	30000mm	17000mm	4900mm	4500mm	6000mm
4	20000mm	35000mm	20000mm	4900mm	4500mm	6000mm

OK Apply Cancel

Duplicate data in CATIA model

Formula.3: `fuselagem\plano radome\Offset`=comp\_cockpit\_section

Formula.5: `fuselagem\crosssection fuselagem inicio\Radius.1\Radius`=raio\_fuselagem /2

conceito aeronave

Formula.6: `fuselagem\plano fuselagem traseira\Offset`=comp\_fuselagem\_section\_traseira

Formula.7: `asa\plano evergadura\Offset`=evergadura

Formula.8: `fuselagem\plano fuselagem\Offset`=comp\_fuselagem\_section\_central

fuselagem

InitialState

Pergunte-me alguma coisa

23:19 22/10/2016

# Resultado final – conceito 3

CATIA V5 - [aeronave teste knowledge exercicio.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

conceito aeronave active, configuration row : 3

Design Table Properties

Name : conceito aeronave Activity

Comment : DesignTable created by Mateus 22/10/2016

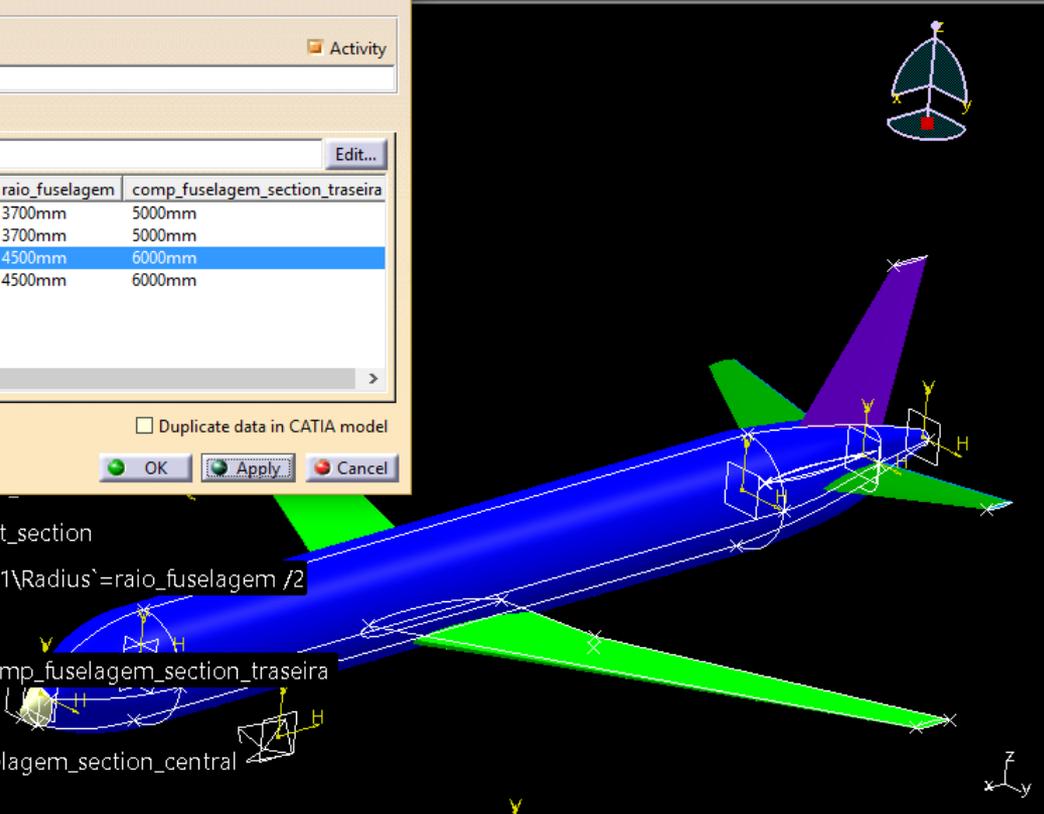
Configurations Associations

Filter : Edit...

Line	evergadura	comp_fuselagem_section_central	posicao_inicio_asa	comp_cockpit_section	raio_fuselagem	comp_fuselagem_section_traseira
<1>	17500mm	20000mm	12000mm	4000mm	3700mm	5000mm
2	17500mm	25000mm	13000mm	4000mm	3700mm	5000mm
3	20000mm	30000mm	17000mm	4900mm	4500mm	6000mm
4	20000mm	35000mm	20000mm	4900mm	4500mm	6000mm

Edit table...  Duplicate data in CATIA model OK Apply Cancel

- Formula.3: `fuselagem\plano radome\Offset`=comp\_cockpit\_section
- Formula.5: `fuselagem\crosssection fuselagem inicio\Radius.1\Radius`=raio\_fuselagem /2
- conceito aeronave
- Formula.6: `fuselagem\plano fuselagem traseira\Offset`=comp\_fuselagem\_section\_traseira
- Formula.7: `asa\plano evergadura\Offset`=evergadura
- Formula.8: `fuselagem\plano fuselagem\Offset`=comp\_fuselagem\_section\_central
- fuselagem



InitialState

Pergunte-me alguma coisa

23:20 22/10/2016



# Resultado final – conceito 4

CATIA V5 - [aeronave teste knowledge exercicio.CATProduct]

Start ENOVIA V5 VPM File Edit View Insert Tools Window Help

conceito aeronave active, configuration row : 4

Design Table Properties

Name: conceito aeronave

Comment: DesignTable created by Mateus 22/10/2016

Configurations Associations

Filter: Edit...

Line	evergadura	comp_fuselagem_section_central	posicao_inicio_asa	comp_cockpit_section	raio_fuselagem	comp_fuselagem_section_traseira
<1>	17500mm	20000mm	12000mm	4000mm	3700mm	5000mm
2	17500mm	25000mm	13000mm	4000mm	3700mm	5000mm
3	20000mm	30000mm	17000mm	4900mm	4500mm	6000mm
4	20000mm	35000mm	20000mm	4900mm	4500mm	6000mm

Edit table...  Duplicate data in CATIA model

OK Apply Cancel

Formula.3: `fuselagem\plano radome\Offset`=comp\_cockpit\_section

Formula.5: `fuselagem\crosssection fuselagem inicio\Radius.1\Radius`=raio\_fuselagem /2

conceito aeronave

Formula.6: `fuselagem\plano fuselagem traseira\Offset`=comp\_fuselagem\_section\_traseira

Formula.7: `asa\plano evergadura\Offset`=evergadura

Formula.8: `fuselagem\plano fuselagem\Offset`=comp\_fuselagem\_section\_central

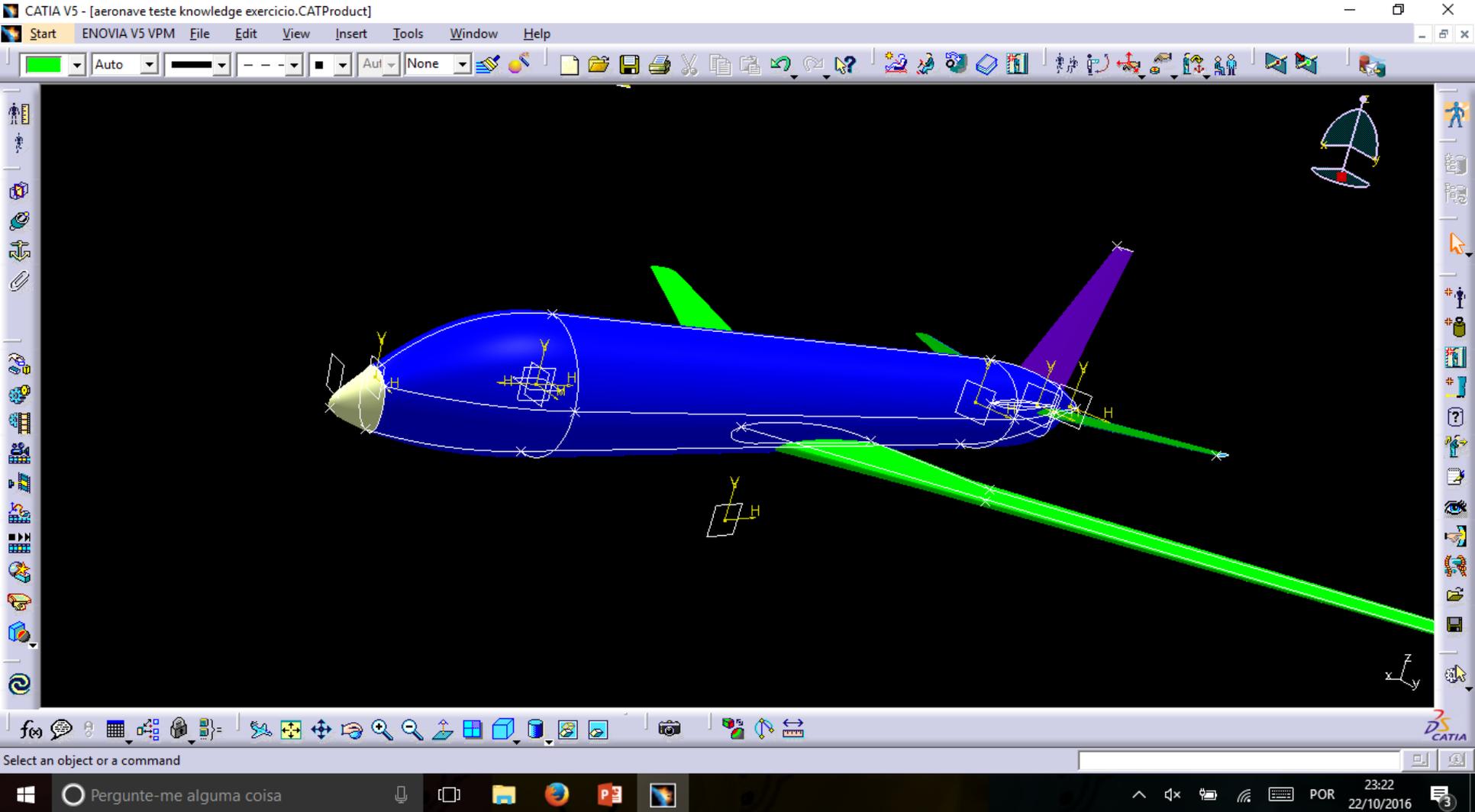
fuselagem

InitialState

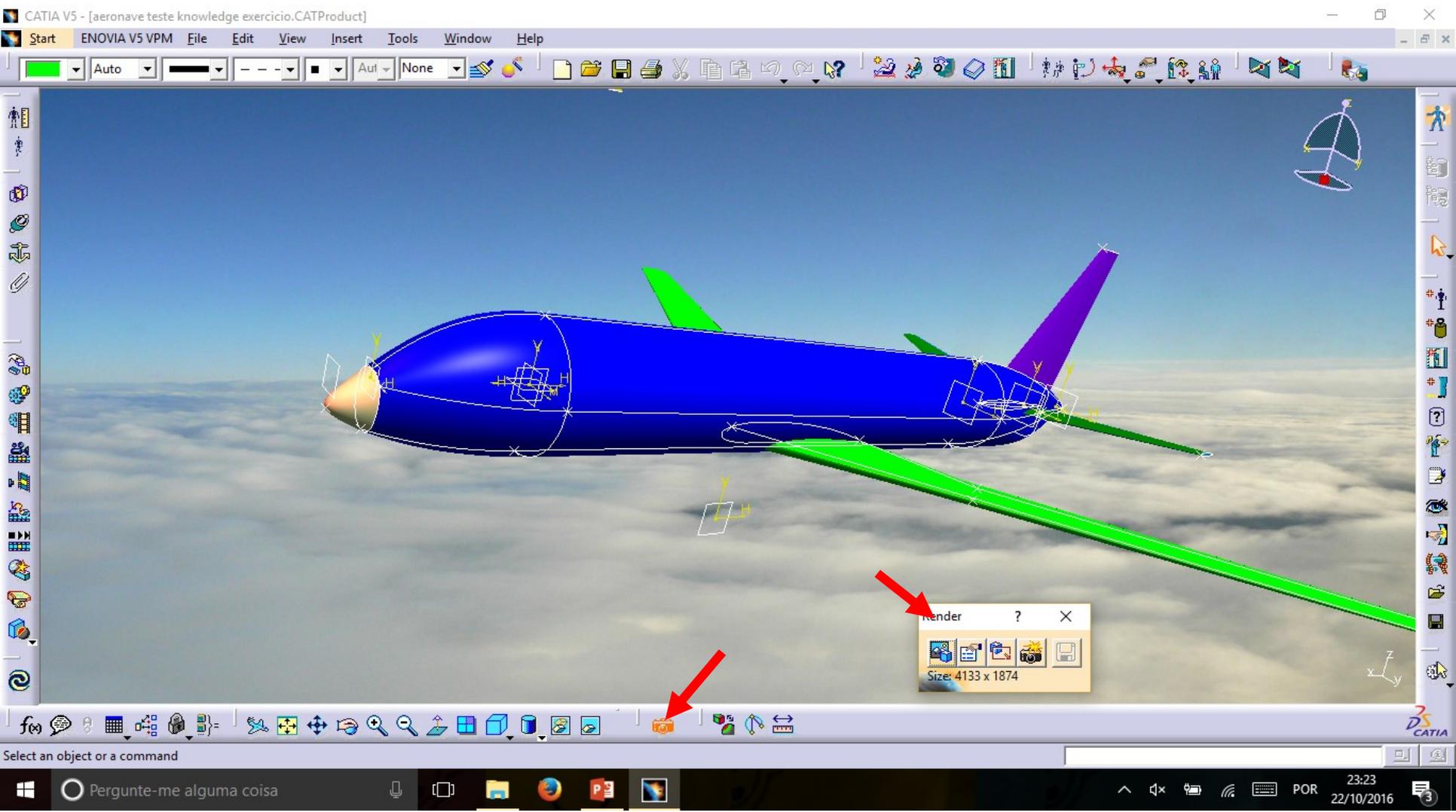
Pergunte-me alguma coisa

23:20 22/10/2016

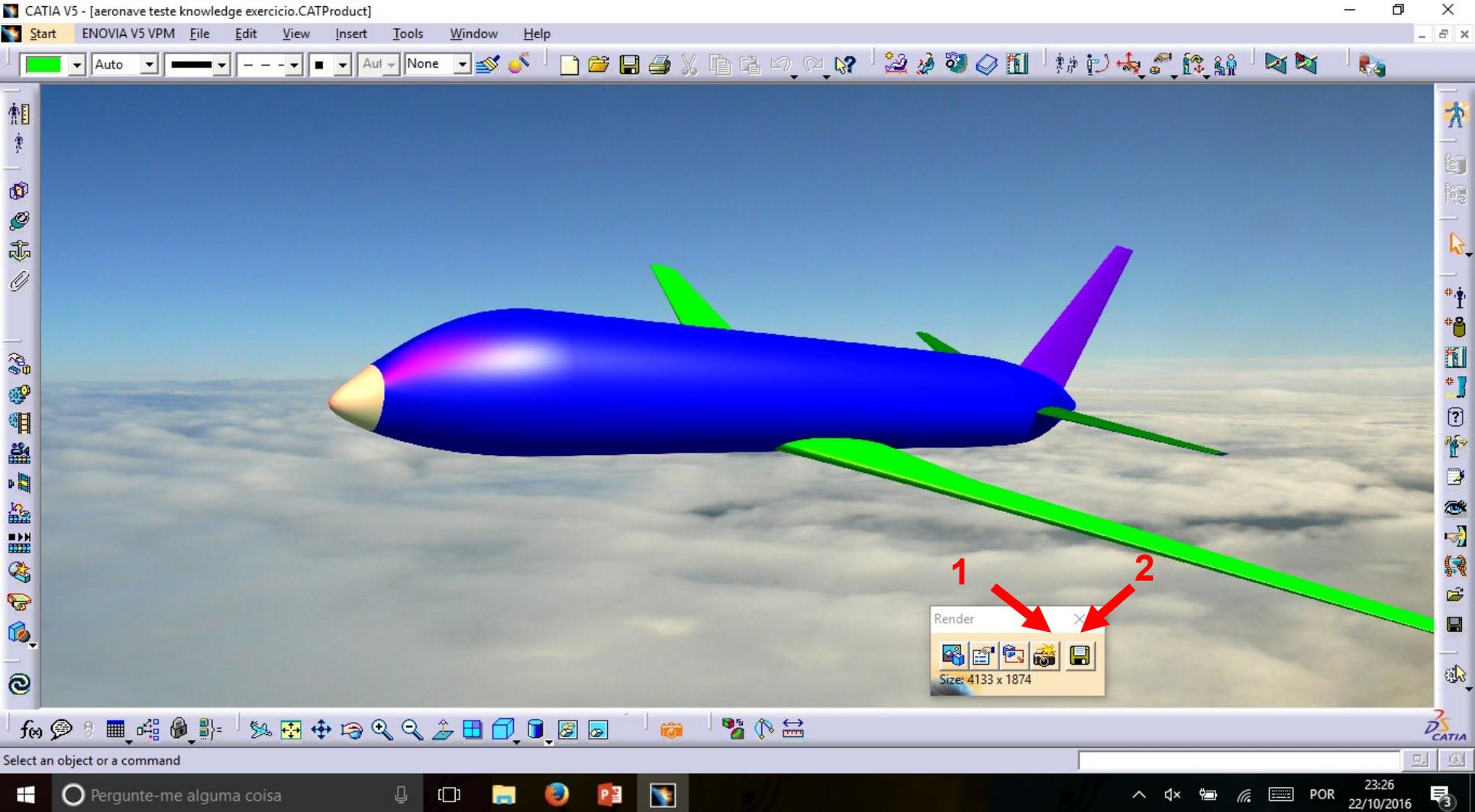
# Resultado final – Renders finais



# Resultado final – Renders finais (PhotoStudio Easy Tools)



# Resultado final – Renders finais





# Obrigado a todos!!!

Equipe Dragonfly e  
Prof. Mateus Miranda

Dúvidas: [mateusmiranda@unb.br](mailto:mateusmiranda@unb.br)

