EXERCISE BOOK

Student Notes:

CATIA V5 Training
Exercises

Knowledge Advisor

Version 5 Release 19
January 2009

EDU_CAT_EN_KWA_FX_V5R19

Student Notes:

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Student Notes:

Stringer

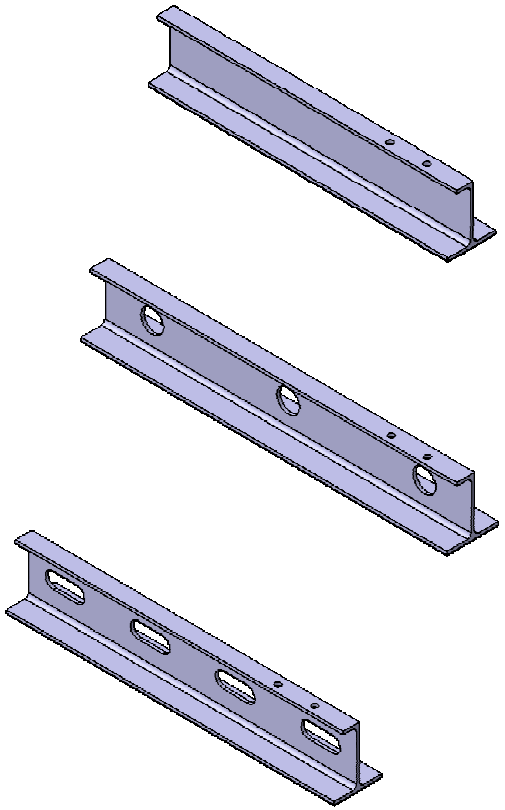
Master Exercise Presentation

65 min

In this exercise, you will design the Stringer part and control its modification using the Knowledgeware tools.

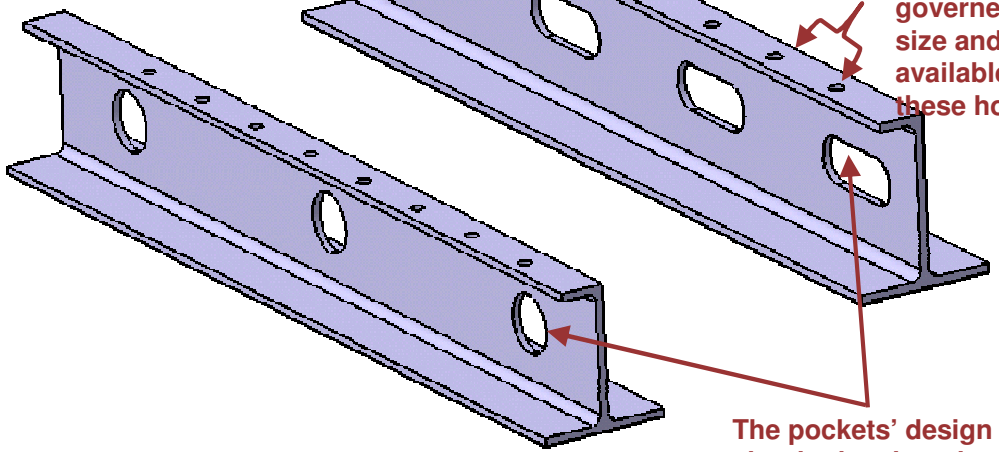
In this exercise, you will practice:

- **Creating User Parameters**
- **Creating Formulas**
- **Creating geometry using User Parameters**
- **Creating Rules and Checks for the design**
- **Creating a Design Table and changing configurations using the design table**



Student Notes:

Design Intent – Stringer



A check observes this 'Bracket Hole Pattern' and displays an error message if the holes in the pattern fall outside the Stringer length.

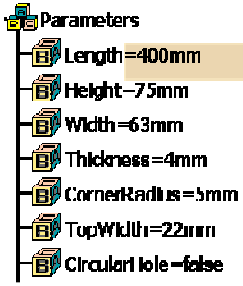
The size of the holes and its spacing is governed by a rule, which decides the size and spacing depending upon the available size of the bracket fitting into these holes.

The pockets' design can be changed from oval to circular by changing a configuration in the design table.

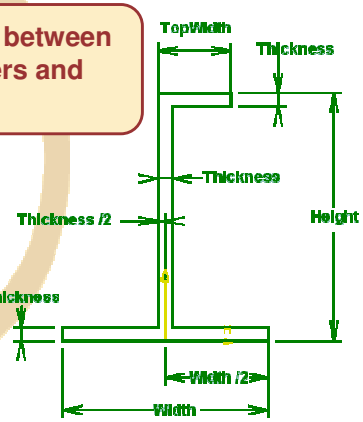
Student Notes:

Stringer - Design Process (1/2)

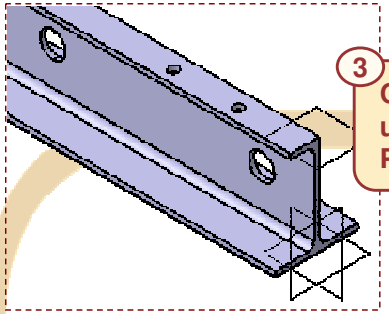
1 Creating User Parameters



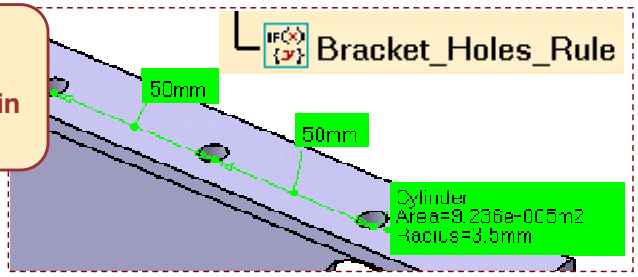
2 Creating formulas between the User Parameters and the dimensions



3 Creating geometry using the User Parameters



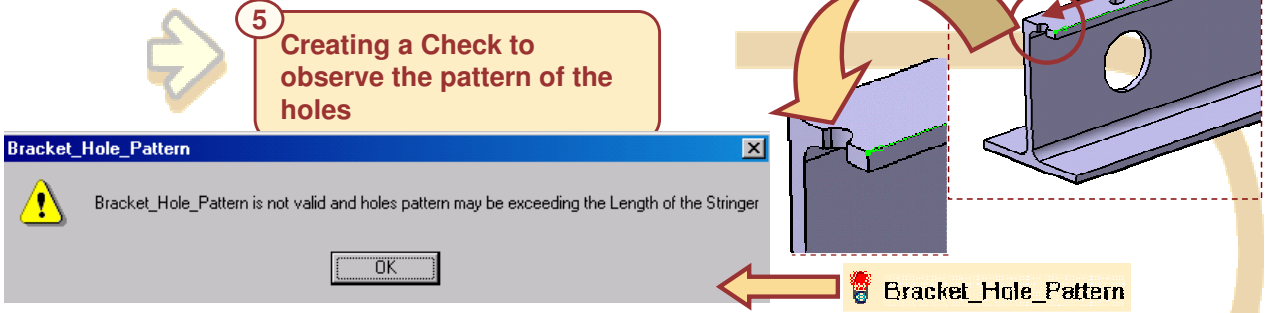
4 Creating a Rule to control the spacing and the hole diameter as per the pre-defined designs of the brackets, which will be fixed in these holes

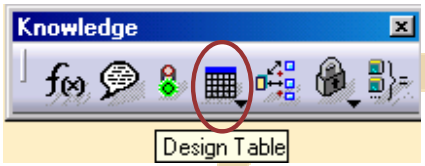


Student Notes:

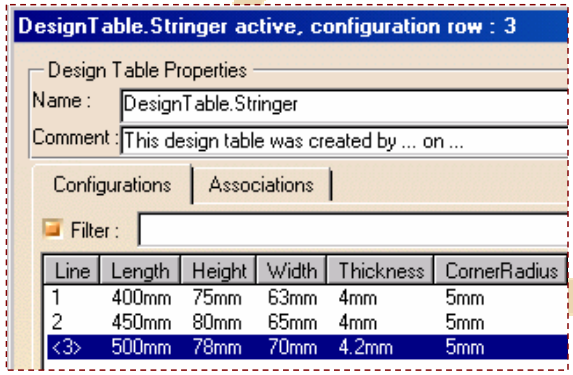
Stringer - Design Process (2/2)

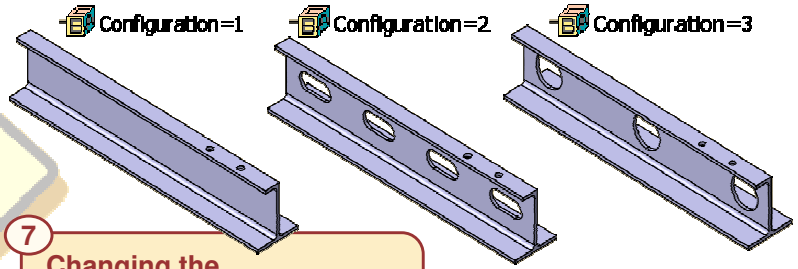
5 Creating a Check to observe the pattern of the holes





6 Creating a new Design Table from the existing User Parameters





7 Changing the configuration and updating the design

Student Notes:

Master Exercise Part 1

Design Process – Part 1

35 min

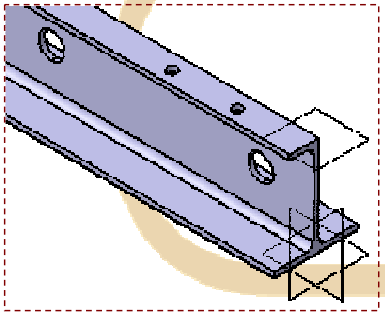
1 Creating User Parameters

Parameters
- Length=400mm
- Height=75mm
- Width=63mm
- Thickness=4mm
- CornerRadius=5mm
- TopWidth=22mm
- CircularHole=false

2 Creating formulas between the User Parameters and the dimensions

Offset.15 = 22 - f(x)=TopWidth
Offset.19 = 4 - f(x)=Thickness
Offset.17 = 4
Offset.49 = 2
Offset 29 = 75 - f(x)=Height
Offset.48 = 4 - f(x)=Thickness
Offset.22 = 31.5 - f(x)=Wicth / 2
Offset.13 = 83 - f(x)=Width

Creating geometry using the User Parameters



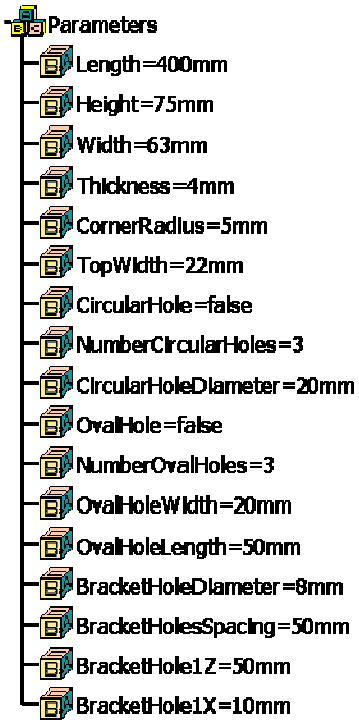
Student Notes:

Stringer

Step 1 – Creating User Parameters

10 min

In this step, you will create parameters and assign values to them.

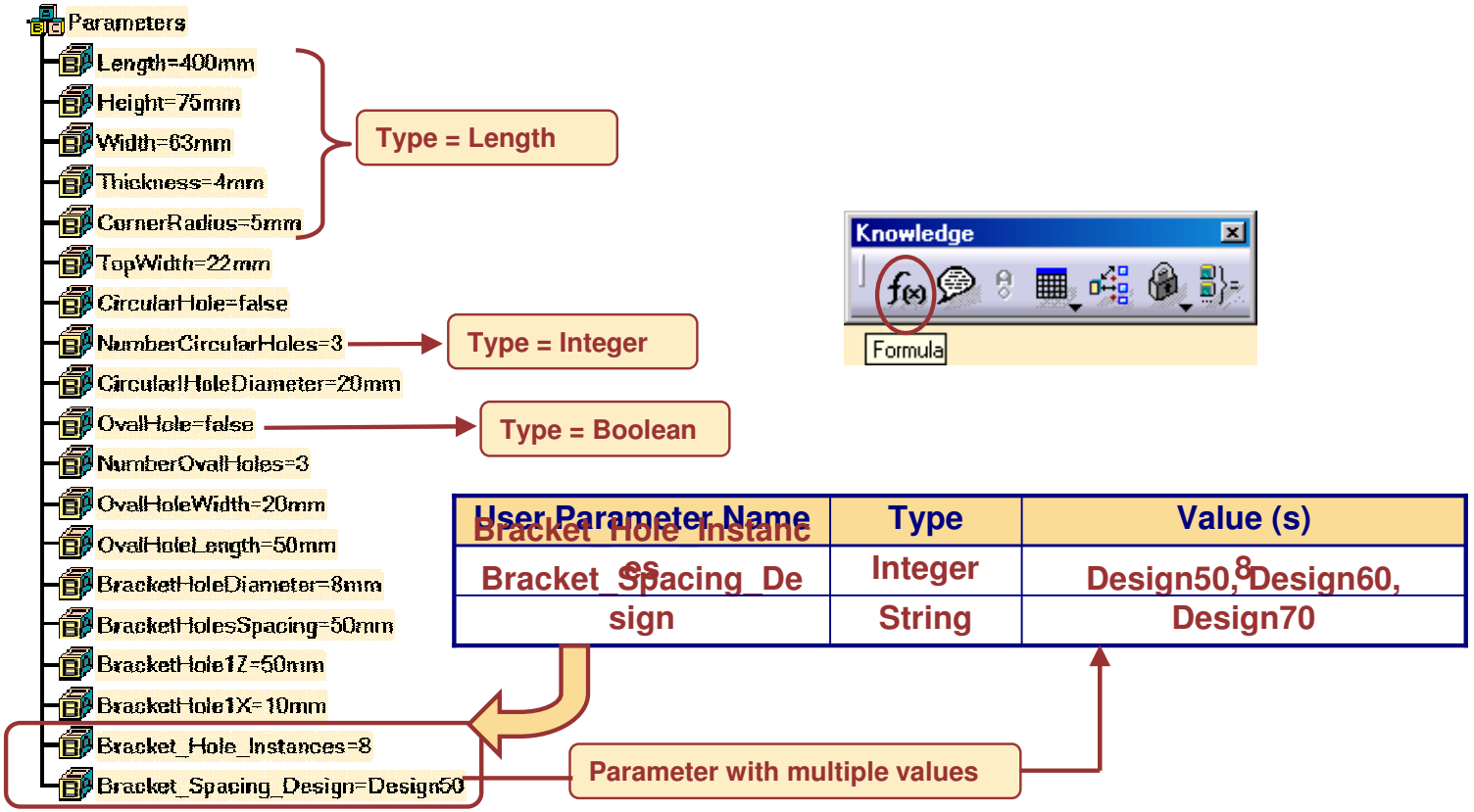


Student Notes:

Step 1: Creating User Parameters

Part used: Stringer_start.CATPart

- Open the part and note the parameters that are already created for you.
  - Create additional parameters and assign values to them as specified in the chart below.



User Parameter Name	Type	Value (s)
Bracket_Hole_Instances	Integer	8
Bracket_Spacing_Design	String	Design50, Design60, Design70

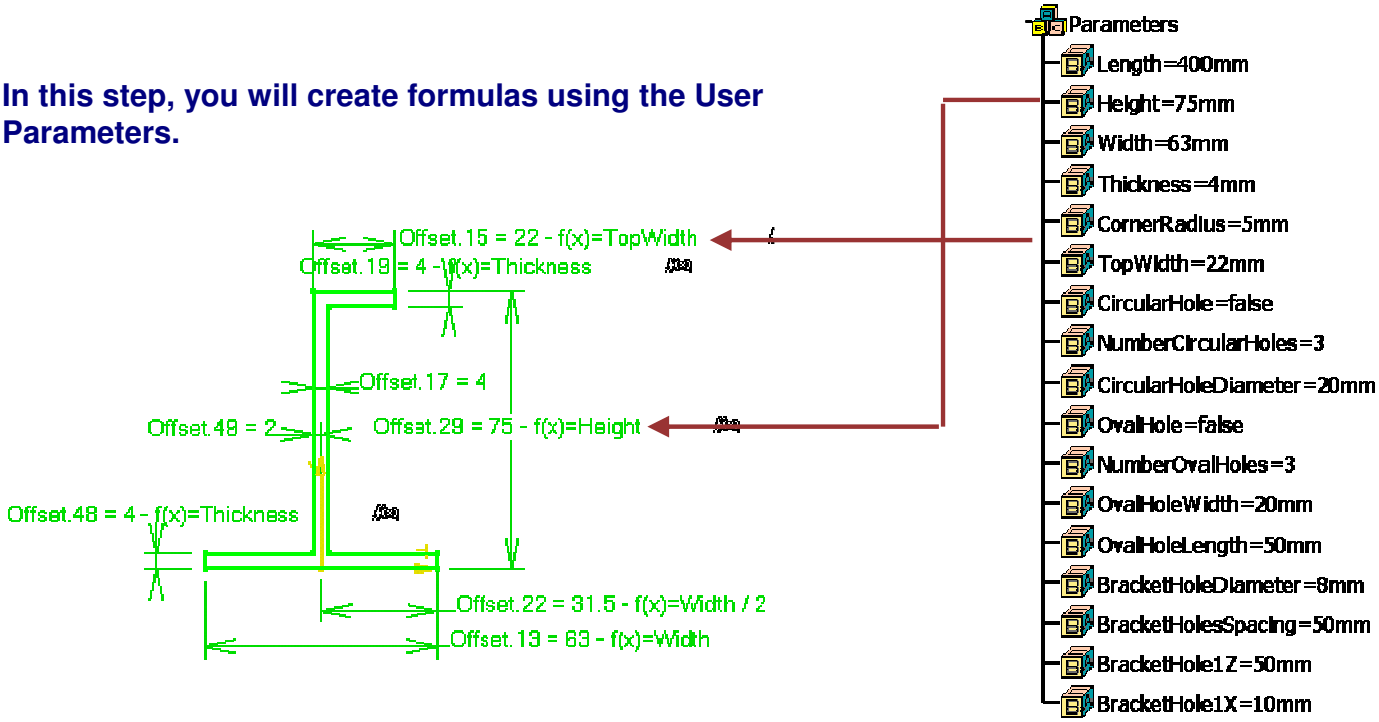
Student Notes:

Stringer

Step 2 – Creating Formulas Using User Parameters

15 min

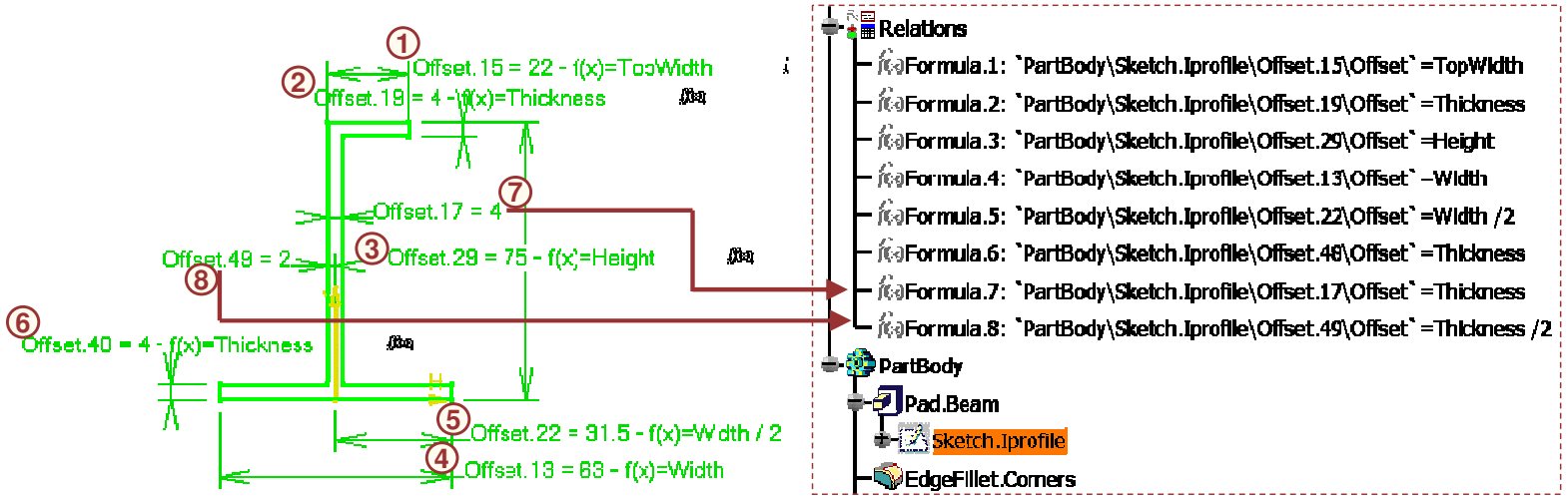
In this step, you will create formulas using the User Parameters.



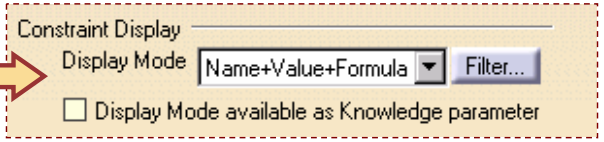
Student Notes:

Step 2: Creating Formulas Using User Parameters (1/5)

- Open the 'Sketch.Iprofile' and note the six formulas that are already created for you.



To be able to view the names of the parameters with values and formulae, select Display Mode = 'Name + Value + Formula' in Tools > Options > Parameters and Measures > Constraints and Dimensions (tab) > Constraint Display.



- Create the formulas 7 and 8 as shown in the image of 'Relations' node above.

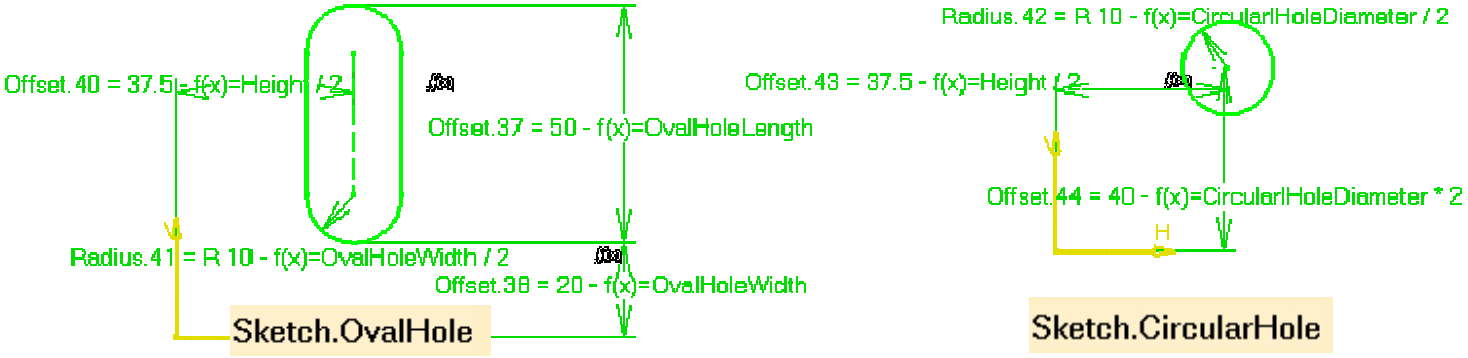
Student Notes:

Step 2: Creating Formulas Using User Parameters (2/5)

- Create ‘Formula.9’ and ‘Formula.10’ for parameters ‘Length’ and ‘CornerRadius’ respectively as shown below.

Formula.9: `PartBody\Pad.Beam\FirstLimit\Length` =Length

Formula.10: `PartBody\EdgeFillet.Corners\CstEdgeRibbon.1\Radius` =CornerRadius



- Edit both the sketches of Pocket.OvalHole and Pocket.CircularHole and create the formulas as illustrated below:

Formula.11: `PartBody\Sketch.OvalHole\Offset.37\Offset` =OvalHoleLength

Formula.12: `PartBody\Sketch.OvalHole\Radius.41\Radius` =OvalHoleWidth /2

Formula.13: `PartBody\Sketch.OvalHole\Offset.38\Offset` =OvalHoleWidth

Formula.14: `PartBody\Sketch.OvalHole\Offset.40\Offset` =Height /2

Formula.15: `PartBody\Sketch.CircularHole\Radius.42\Radius` =CircularHoleDiameter /2

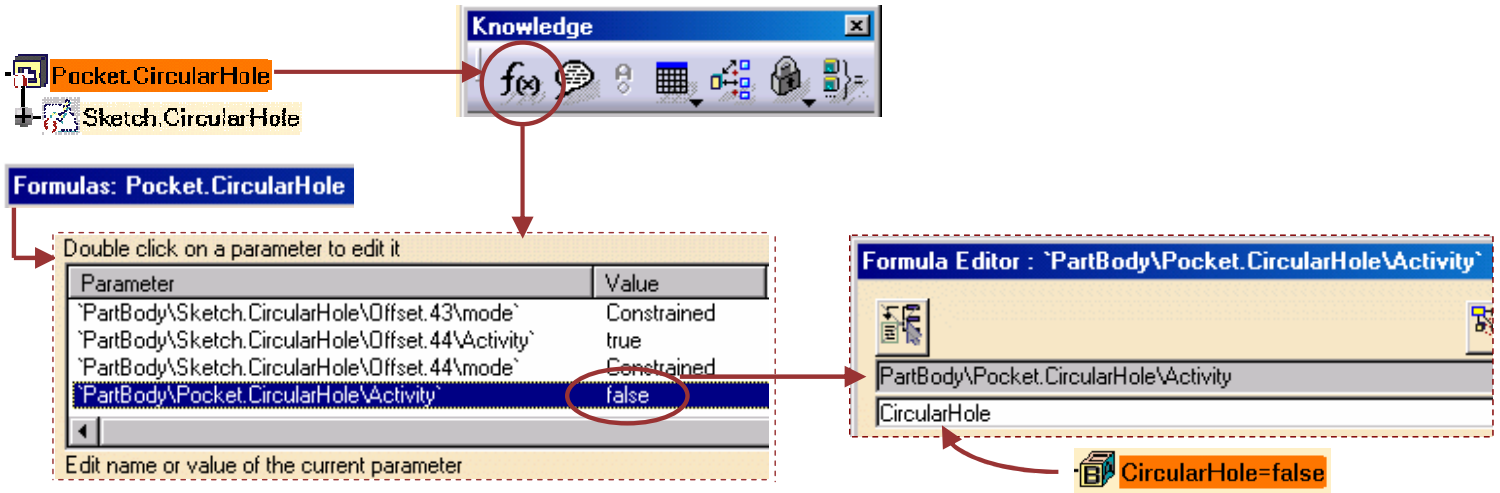
Formula.16: `PartBody\Sketch.CircularHole\Offset.43\Offset` =Height /2

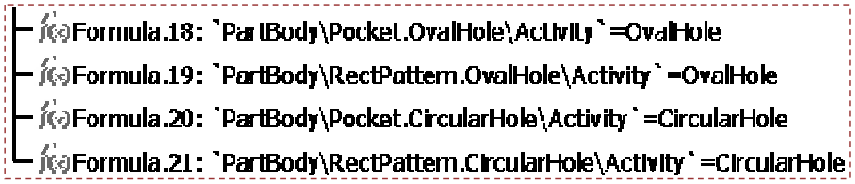
Formula.17: `PartBody\Sketch.CircularHole\Offset.44\Offset` =CircularHoleDiameter *2

Student Notes:

Step 2: Creating Formulas Using User Parameters (3/5)

- Connect the Activity parameters of the pocket features and their pattern features to both the Boolean user parameters as shown below.

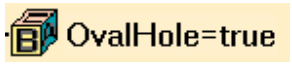




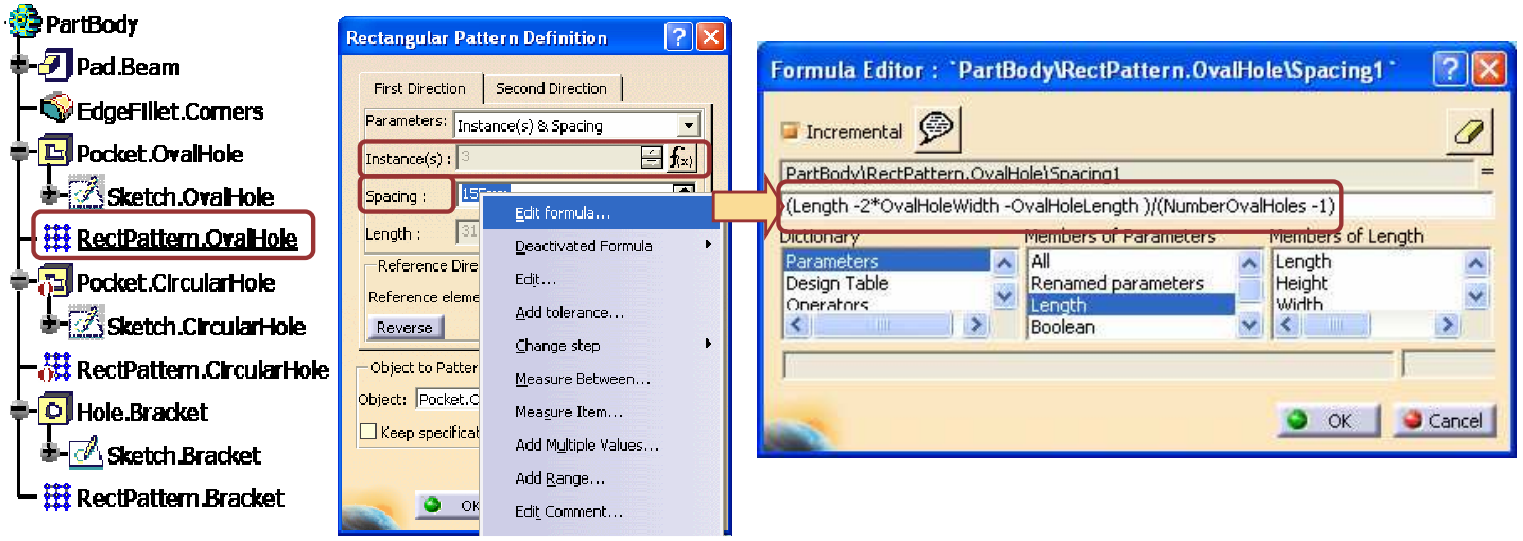
Student Notes:

Step 2: Creating Formulas Using User Parameters (4/5)

- Parameterize the pattern of 'Pocket.OvalHole'.
  - Set the activity of the 'OvalHole' parameter to 'true'.



  - Edit the 'RectPattern.OvalHole' rectangular pattern.
  - Create a formula to link the number of instances to the dedicated user parameter.
  - Create the formula as shown in the image below to define the spacing between the holes. Do not forget to use brackets to delimit the fields.



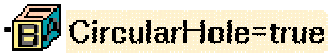
Formula.22: `PartBody\RectPattern.OvalHole\NumberInDir1`=NumberOvalHoles

Formula.23: `PartBody\RectPattern.OvalHole\Spacing1`=(Length -2*OvalHoleWidth -OvalHoleLength)/(NumberOvalHoles -1)

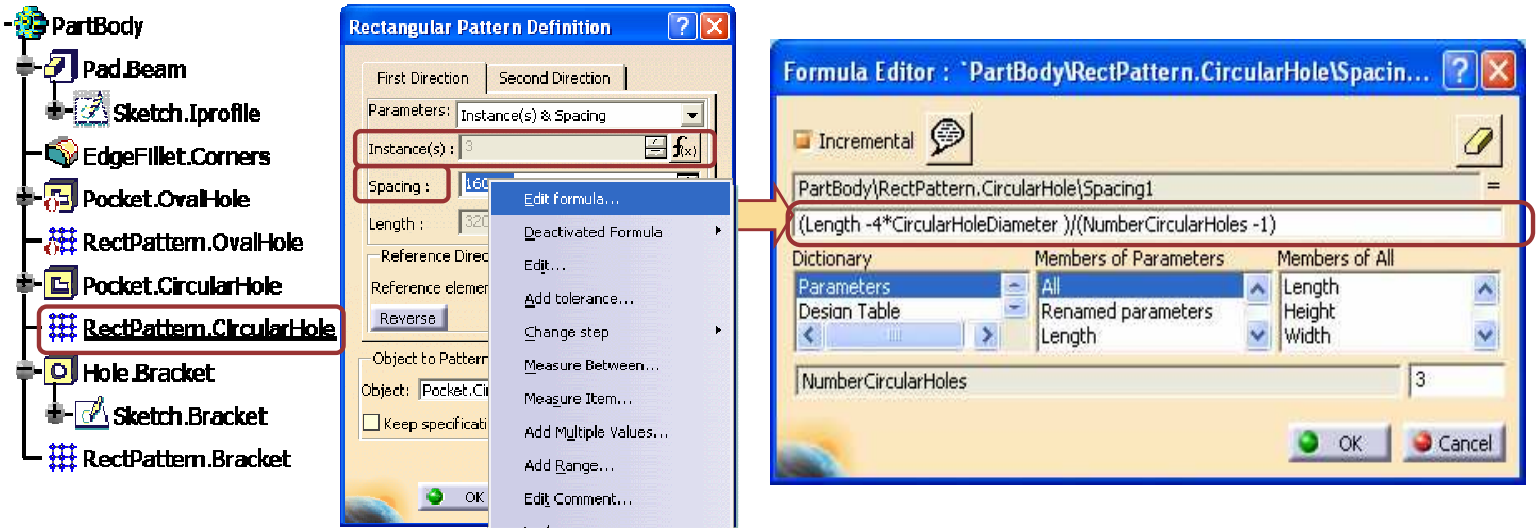
Student Notes:

Step 2: Creating Formulas Using User Parameters (5/5)

- Repeat the same process for pattern of 'Pocket.OvalHole'.
  - Set the activity of the 'OvalHole' parameter to 'false' and the activity of 'CircularHole' to 'true'.



  - The formula for the spacing in this case will be as shown below.



Formula.24: `PartBody\RectPattern.CircularHole\NumberInDir1` =NumberCircularHoles

Formula.25: `PartBody\RectPattern.CircularHole\Spacing1` =(Length -4*CircularHoleDiameter)/(NumberCircularHoles -1)

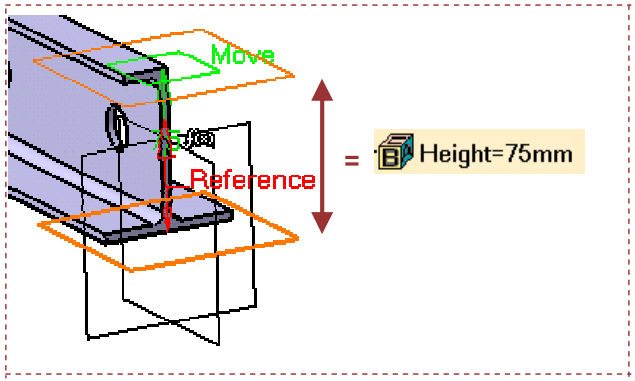
Student Notes:

Stringer

Step 3 – Creating Geometry using User Parameters

10 min

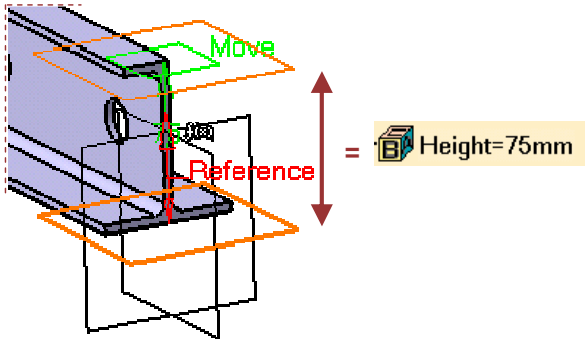
In this step, you will create geometry using User Parameters.



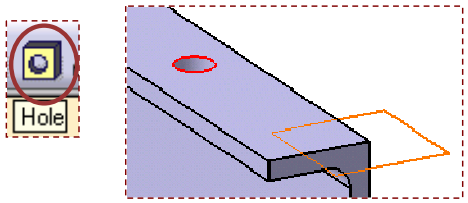
Student Notes:

Step 3: Creating Geometry using User Parameters (1/2)

- Create holes that will be used to fix a single or multiple brackets in the Stringer.
  - Create a plane offset to the zx plane at a distance equal to the height parameter.



  - Select this plane as support and create a hole as shown.



  - Open the sketch of the hole and position it with respect to the edges using parameters and formulae, as shown in the figure.

  - Associate the diameter of this hole to the parameter 'BracketHoleDiameter'.

Formula.31: PartBody\Hole.1\Diameter=BracketHoleDiameter

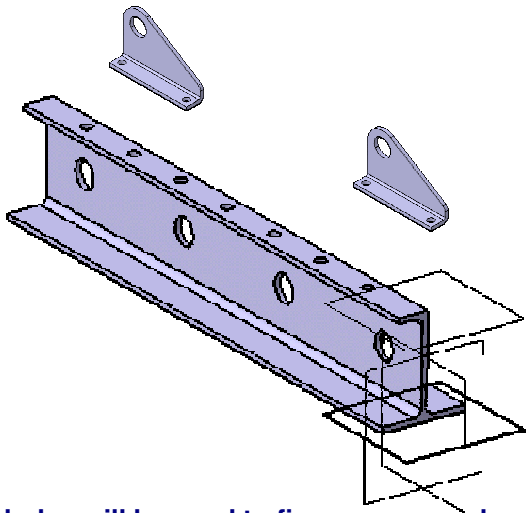
Student Notes:

Step 3: Creating Geometry using User Parameters (2/2)

- Create a pattern of this hole using parameters and values as shown.

Instance (s) = Bracket_Hole_Instances=7

Spacing = BracketHolesSpacing=50mm



These holes will be used to fix one or more brackets

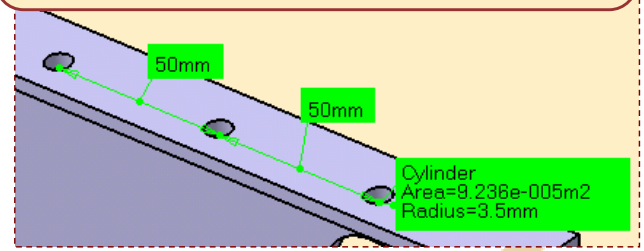
Student Notes:

Master Exercise Part 2

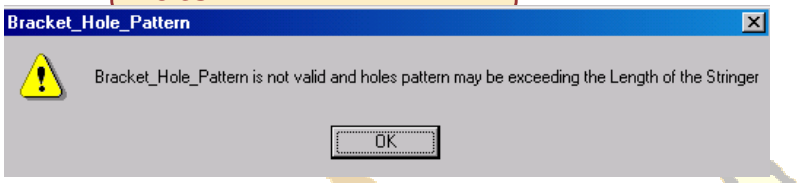
Design Process – Part 2

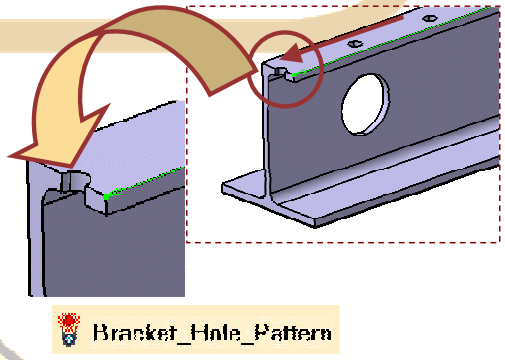
15 min

4 Creating a Rule to control the spacing and hole diameter as per the pre-defined designs of the brackets, which have to be fixed in these holes



Creating a Check to observe the pattern of the holes





EXERCISE BOOK

Student Notes:

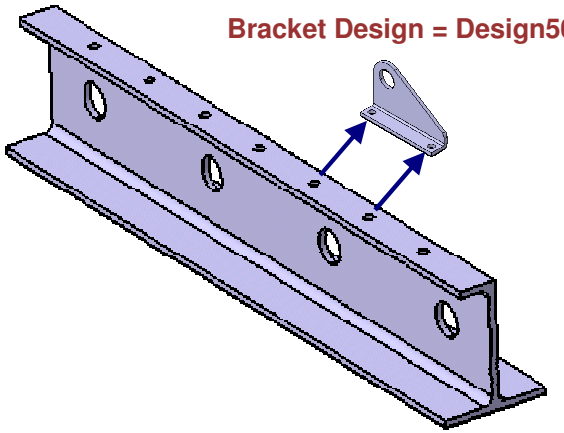
Stringer

Step 4 – Creating Rules

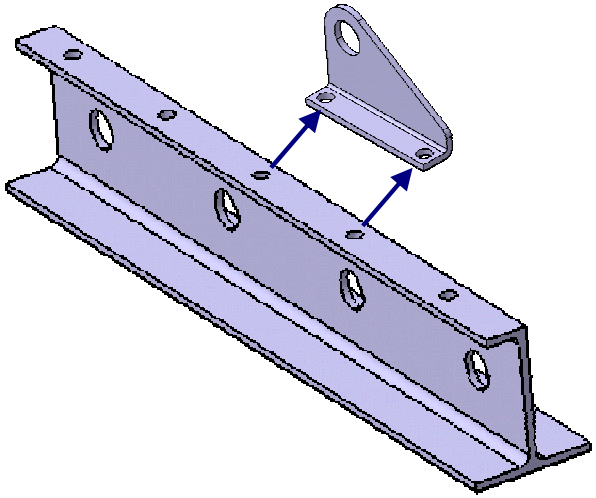
10 min

In this step, you will create rule(s) to control the design modifications of the Stringer part.

Bracket Design = Design50



Bracket Design = Design70



Student Notes:

Step 4: Creating Rules (1/2)

CATIA data used: Stringer_Part2_Start.CATPart

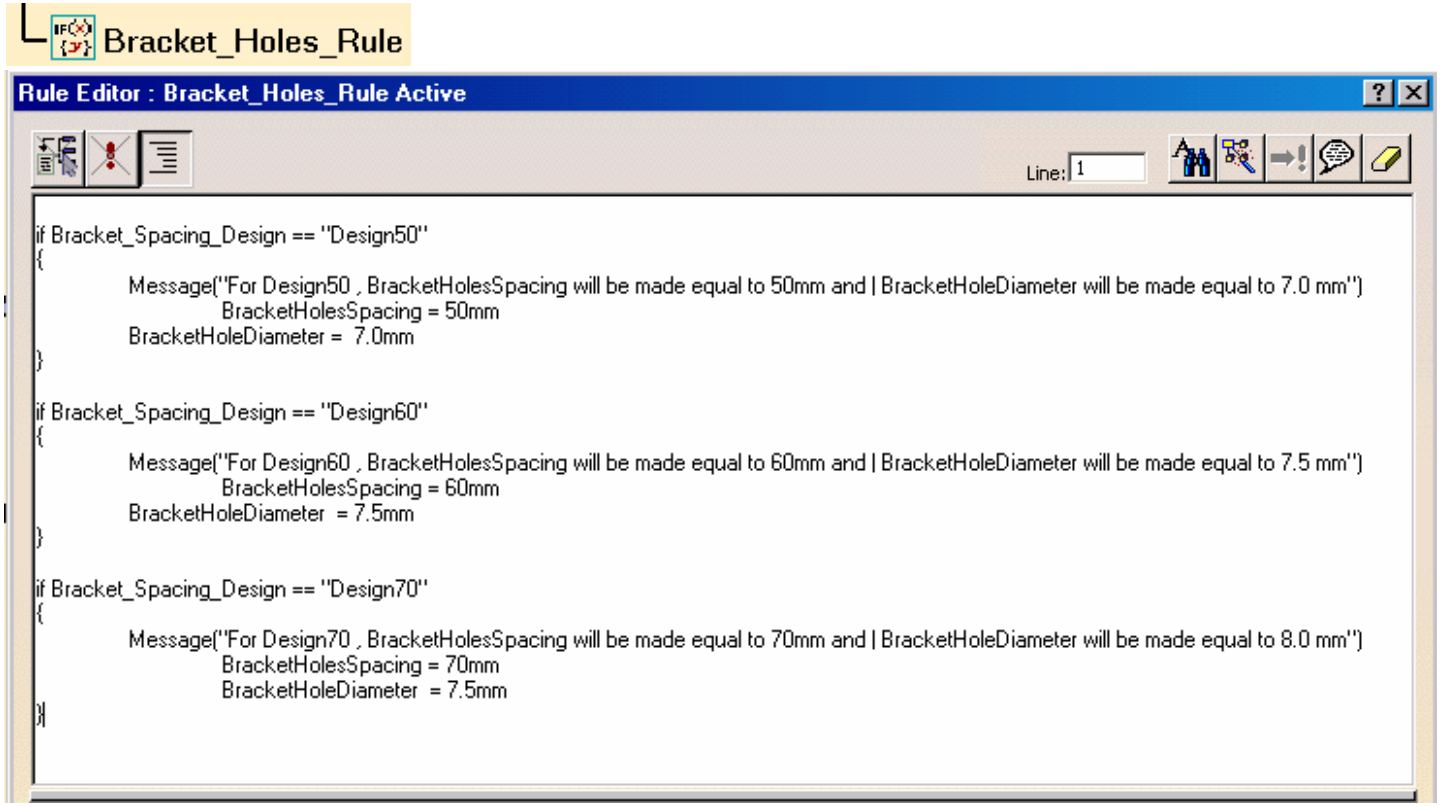
- Create a 'Rule' which will incorporate the following cases for the parameter 'Bracket_Spacing_Design'.

If 'Bracket_Spacing_Design' =	Then
Design50	BracketHolesSpacing = 50mm BracketHoleDiameter = 7.0mm
Design60	BracketHolesSpacing = 60mm BracketHoleDiameter = 7.5mm
Design70	BracketHolesSpacing = 70mm BracketHoleDiameter = 8.0mm

Student Notes:

Step 4: Creating Rules (2/2)

- You can use the following code in the 'Rule Editor'.



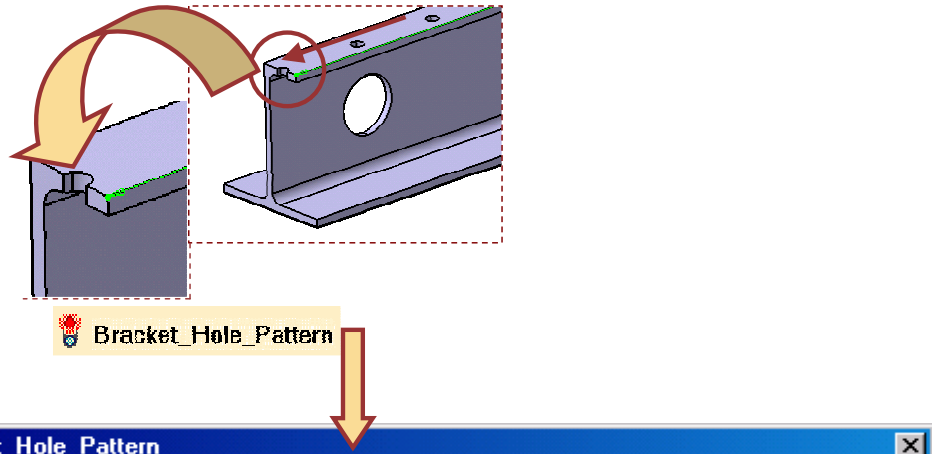
Student Notes:

Stringer

Step 5 – Creating a Check

5 min

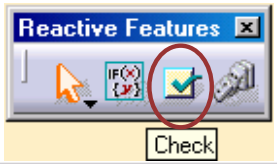
In this step, you will create a 'Check' to observe the pattern of the holes.

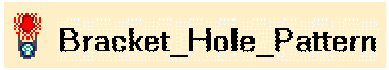


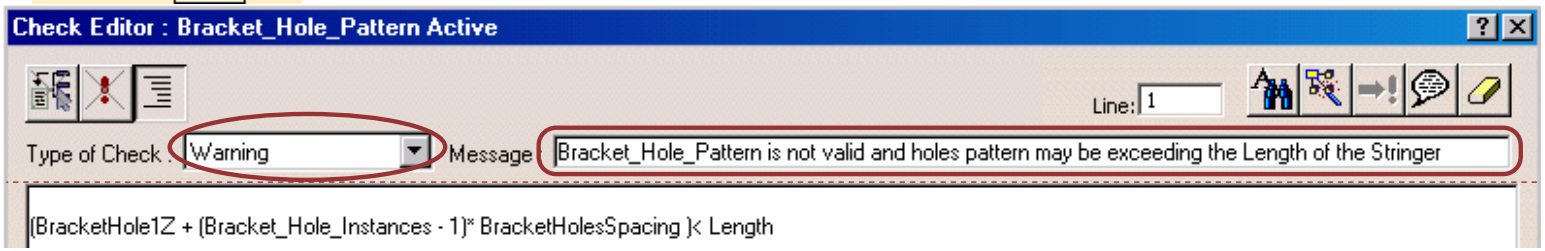
Student Notes:

Step 5: Creating Check

- Create a check to verify that the holes created by the pattern do not cross the Stringer length, resulting into invalid form of pattern.

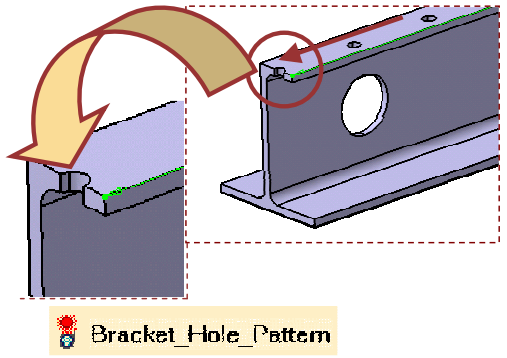






- You can use the settings and the line of code in the 'Check Editor' as shown above.

- You can also verify the warning of the 'Check' by assigning the values to the parameters as shown below.



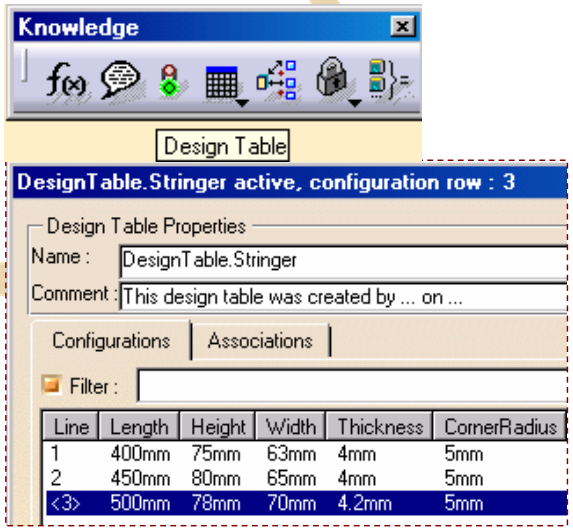
Student Notes:

Master Exercise Part 3

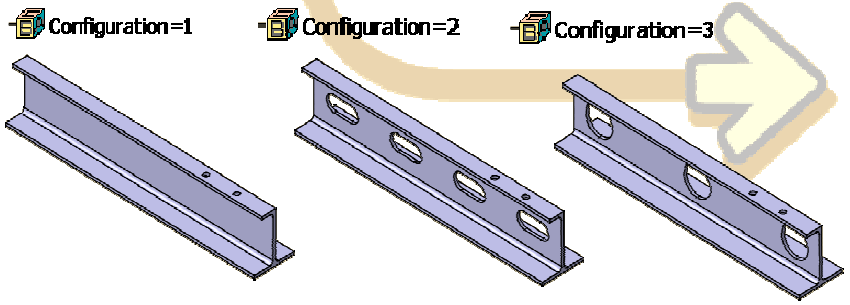
Design Process – Part 3

15 min

6 Creating a new Design Table from the existing User Parameters



Changing configuration and updating the design



EXERCISE BOOK

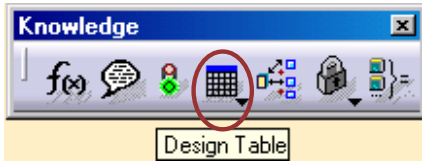
Student Notes:

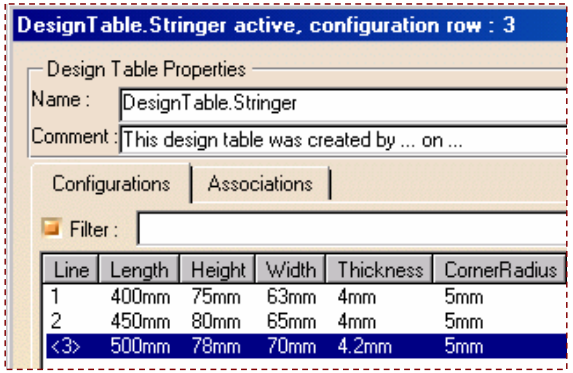
Stringer

Step 6 – Creating a 'Design Table'

10 min

In this step, you will create a new Design Table from some of the existing user parameters.





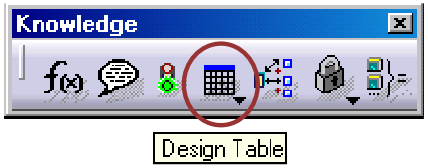
Student Notes:

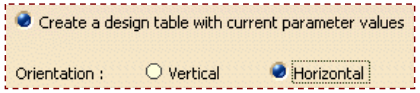
Step 6: Creating a 'Design Table'

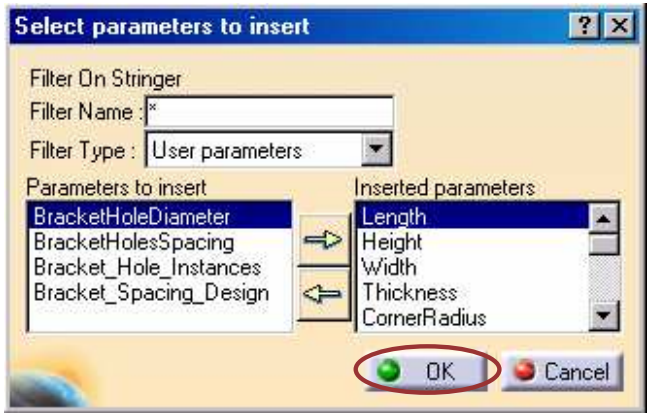
Part used: Stringer_Part3_Start.CATPart

- Create a 'Design Table' named "DesignTable.Stringer" of all the 'User Parameters' except the following:

1. BracketHoleDiameter
2. BracketHolesSpacing
3. Bracket_Hole_Instances
4. Bracket_Spacing_Design







- Click 'OK' and save the 'Design Table' as .xls or .txt file.

- Reopen the 'Design Table' and click the 'Edit Table' button to create the design configurations as shown in the adjoining table.

	A	B	C	D
1	Length (mm)	400	450	500
2	Height (mm)	75	80	78
3	Width (mm)	63	65	70
4	Thickness (mm)	4	4	4.2
5	CornerRadius (mm)	5	5	5
6	TopWidth (mm)	22	22	24
7	CircularHole	false	false	true
8	NumberCircularHoles	3	3	3
9	CircularlHoleDiameter (mm)	20	20	32
10	OvalHole	false	true	false
11	NumberOvalHoles	3	4	3
12	OvalHoleWidth (mm)	20	22	20
13	OvalHoleLength (mm)	50	55	50
14	BracketHole1Z (mm)	50	50	50
15	BracketHole1X (mm)	10	10	10

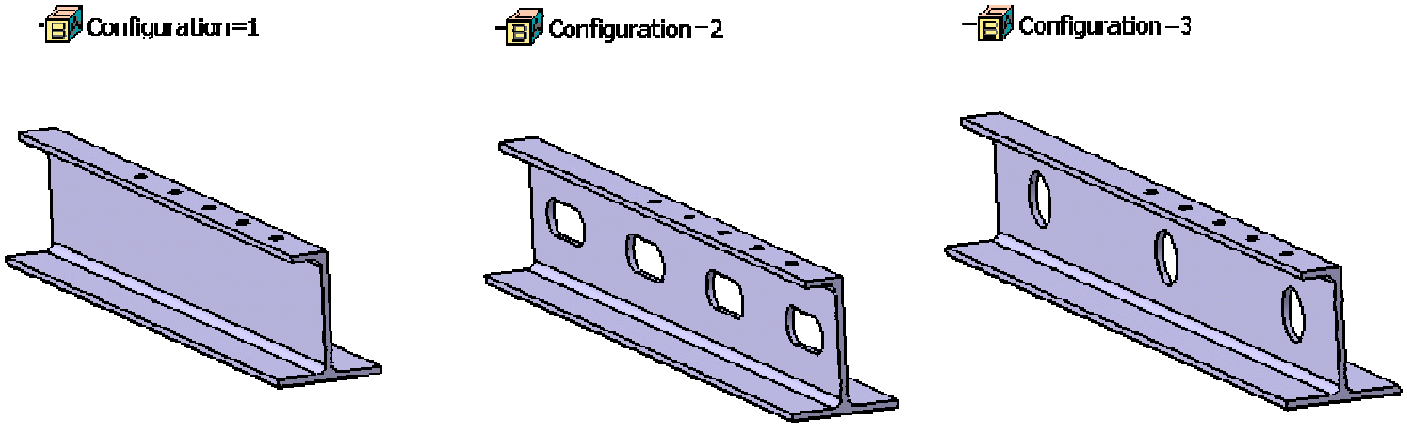
Student Notes:

Stringer

Step 7 – Changing design Configuration and Updating

5 min

In this step, you will switch between the different design configurations that you have created in the design table.



Student Notes:

Step 7 – Changing design Configuration and updating

- **Open the design table and switch between the configurations in the design table. Update if necessary.**

DesignTable.Stringer active, configuration row : 3

Design Table Properties

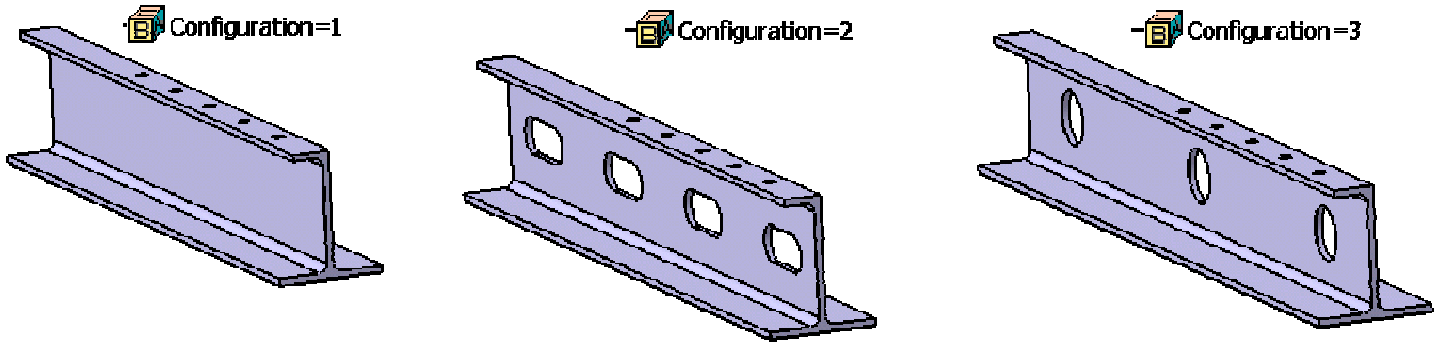
Name : DesignTable.Stringer Activity

Comment : This design table was created ....

Configurations | Associations

Filter : Edit...

Line	Length	Height	Width	Thickness	CornerRadius	TopWidth	CircularHole	Numb...	CircularHoleDiam
1	400mm	75mm	63mm	4mm	5mm	22mm	false	3	20mm
2	450mm	80mm	65mm	4mm	5mm	22mm	false	3	20mm
<3>	500mm	78mm	70mm	4.2mm	5mm	24mm	true	3	32mm



This step completes the Stringer exercise.

EXERCISE BOOK

Student Notes:

Knowledge Advisor Added Exercises

- Light Bulb Exercise
- Sheet Metal Part Exercise
- Wheel Rim Exercise

EXERCISE BOOK

Student Notes:

Light Bulb

Added Exercise Presentation

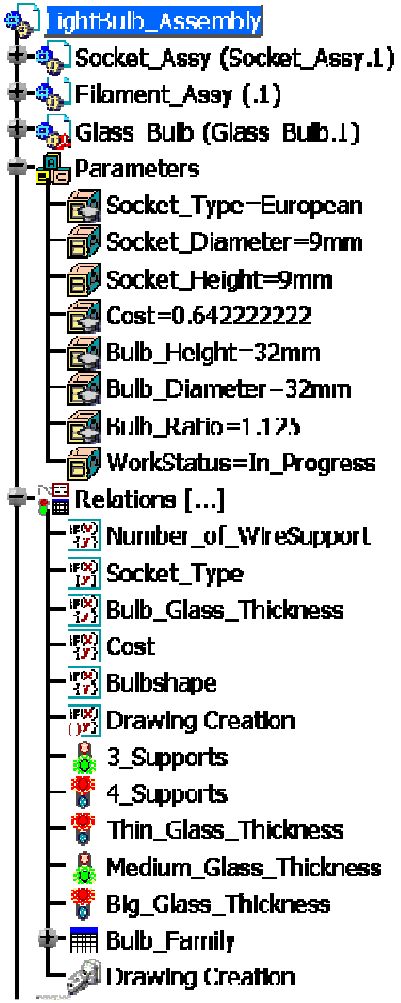
1 hour

In this exercise you will:

- Embed the knowledge within the design of a light bulb assembly using Formulas, Rules and Checks.
- Define a light bulb family using a Design Table.
- Determine the impacts and dependencies of a parameter modification using the Knowledge Inspector tool.
- Automate drawing creations using the VBscript Macros launched from rules.

Student Notes:

Design Intent: Light Bulb



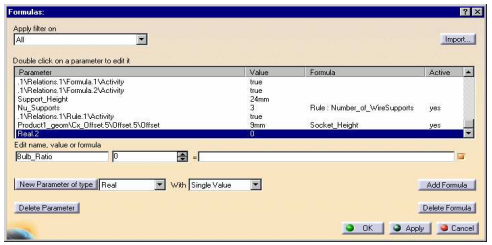
In the context of an assembly representing a light bulb:

- **Create User Parameters and formulas**
- **Create a rule to keep a specific ratio value in the bulb design**
- **Create checks to inform the user that the ratio limit values have been reached**
- **Analyze the impacts of a parameter value modification using the Knowledge Inspector**
- **Create a design table to ease the definition and the use of alternate designs for the assembly**
- **Automate the creation of drawings using macros**
- **Create and use catalogs of standard components**

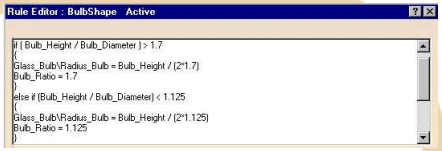
Student Notes:

Design Process: Light Bulb

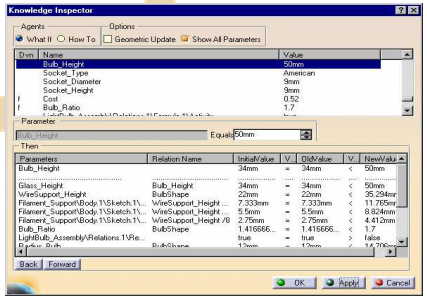
Step 1 : Create user parameters and formulas

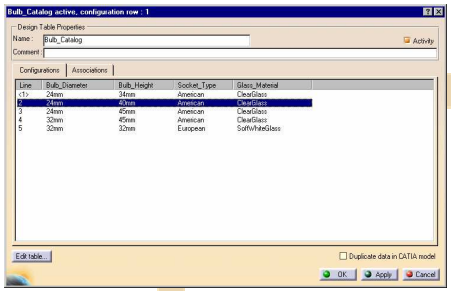


Step 2 : Create a Rule



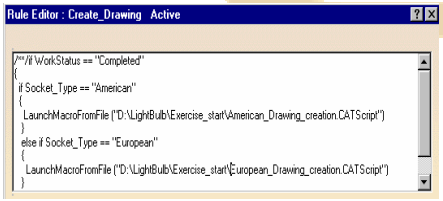
Step 3 : Use the Knowledge Inspector tool





Step 4: Create a Design Table

Step 5/6 : Generate a drawing with a Macro launched from a Rule or a Reaction



Step 7: Create a bulb glass family catalog

Student Notes:

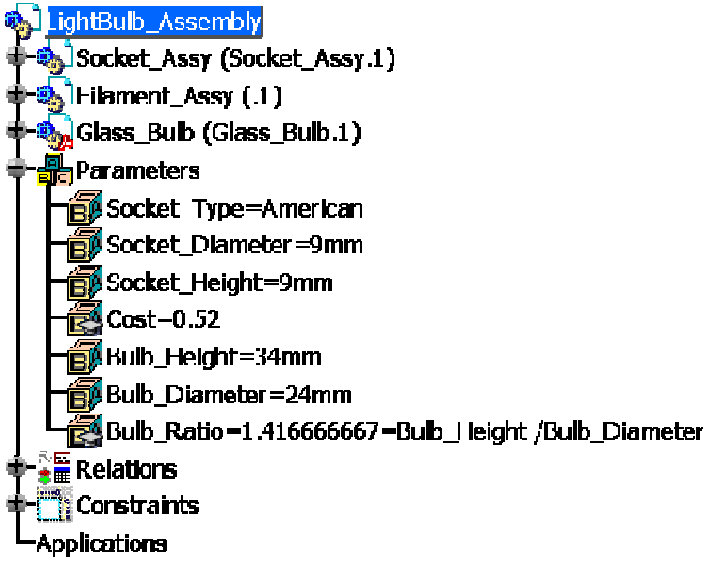
Light Bulb

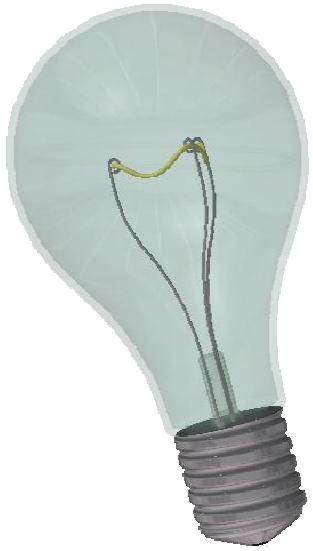
Step 1 - Create User Parameters and Formulas

10 min

In this step you will create four User Parameters:

- **Bulb_Height**
- **Bulb_Diameter**
- **Bulb_Ratio (stands for the ratio of the bulb's height to its diameter (ratio=Height/Diameter))**
- **Wet_Area**





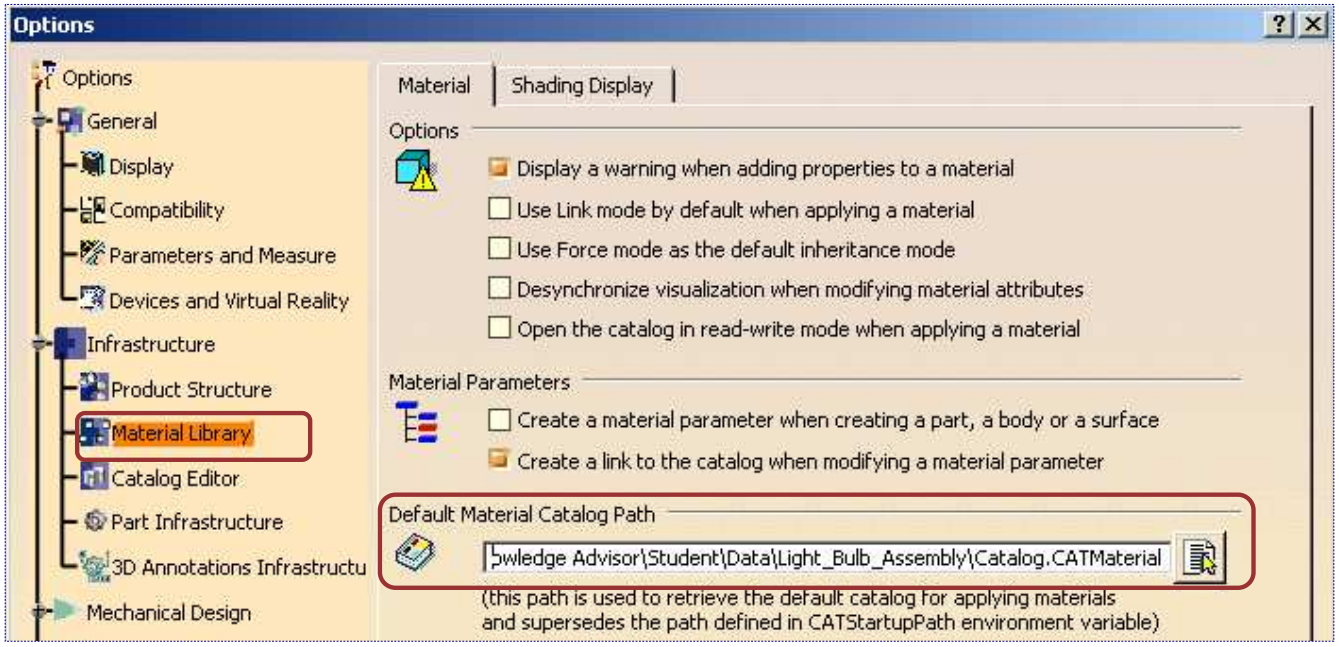
Student Notes:

Do It Yourself (1/2)

Catalog.CATMaterial

1. **In CATIA V5 Tools/Options, declare as default catalog the Material catalog furnished with the training data: …/Student/Data/Light_Bulb_Assembly/Catalog.CATMaterial. This will add a new glass material in your material library.**

2. **Choose the correct display mode to see the materials.**



Student Notes:

Do It Yourself (2/2)

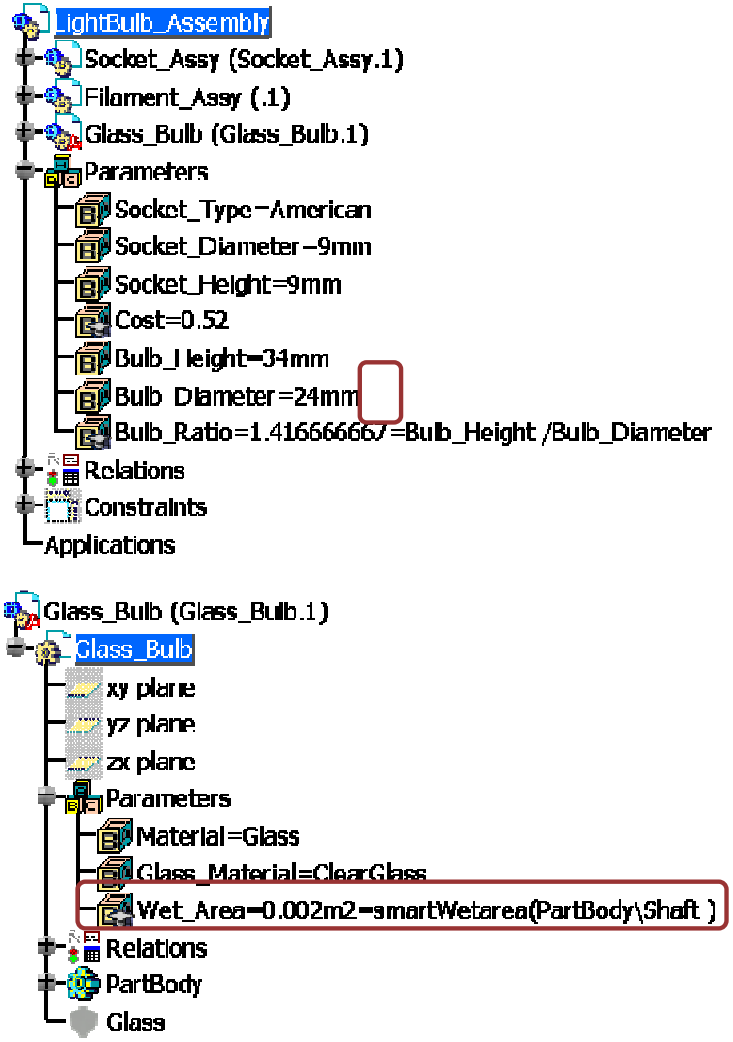
Open CATKWA_LightBulb_Assy_Start.CATProduct

1. At the root level, create two User Parameters of type Length:
   - Bulb_Height =34mm
   - Bulb_Diameter =24mm

2. Create a Parameter « Bulb_Ratio » of type Real defined by the formula:
   Bulb_Ratio=Bulb_Height /Bulb_Diameter

3. Add the following formula on Glass_Bulb\Glass_Height parameter :
   Glass_Bulb \Glass_Height =Bulb_Height

4. Activate the Glass_Bulb Component and create a parameter of type Area. Rename it « Wet_Area » and define it with the formula :
   Wet_Area=smartWetarea (PartBody\Shaft)

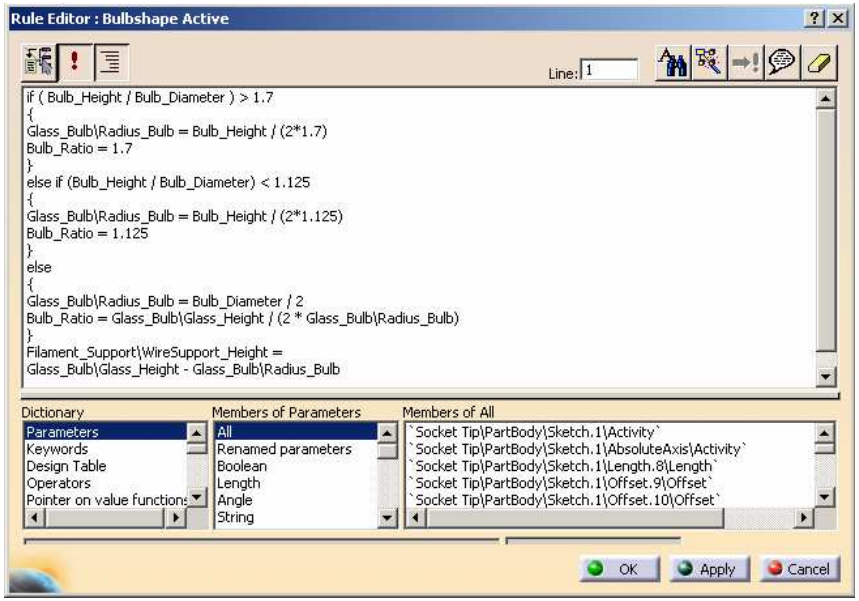


Light Bulb

Step 2 - Create a Ratio Rule

10 min

In this step, you will create a Rule that will ensure that the bulb ratio (Height/Diameter) always remains between 1.125 and 1.7 in order to avoid to get strange bulb shapes.



Student Notes:

Student Notes:

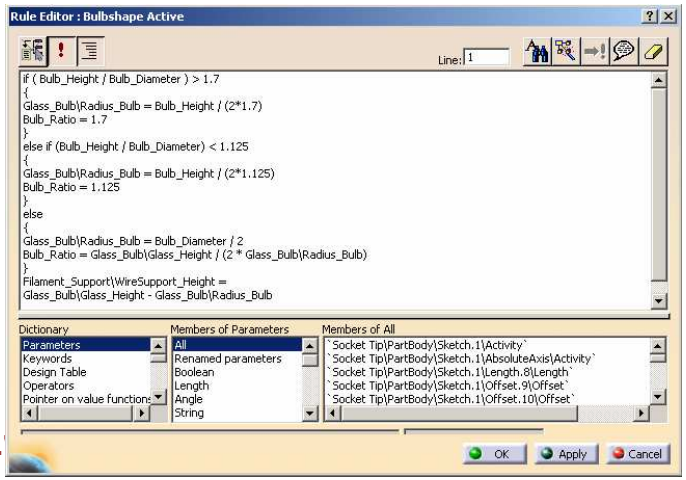
Do It Yourself

1. **At the root level, create a Rule named « Bulbshape » with the following instructions :**

```
if ( Bulb_Height / Bulb_Diameter ) > 1.7
{Glass_Bulb\Radius_Bulb = Bulb_Height / (2*1.7)
Bulb_Ratio = 1.7
Message ("The Ratio has reached its upper limit of 1.7. |
        Bulb diameter has been modified accordingly.|
        Bulb diameter is driven by ratio of 1.7 of the Bulb_Height")}

else if (Bulb_Height / Bulb_Diameter) < 1.125
{Glass_Bulb\Radius_Bulb = Bulb_Height / (2*1.125)
Bulb_Ratio = 1.125
Message("The Ratio has reached its lower limit of 1.125.|
        Bulb diameter has been modified accordingly.|
        Bulb diameter is driven by ratio of 1.125 of the Bulb_Height.
Else

{Glass_Bulb\Radius_Bulb = Bulb_Diameter / 2
Bulb_Ratio = Glass_Bulb\Glass_Height / (2 *
        Glass_Bulb\Radius_Bulb)}
Filament_Support\WireSupport_Height =
Glass_Bulb\Glass_Height - Glass_Bulb\Radius_Bulb
```



2. **Click “Yes” in the « Conflicts Warning » panel in order to avoid a valuation conflict between Formula.5 and this new Rule.**

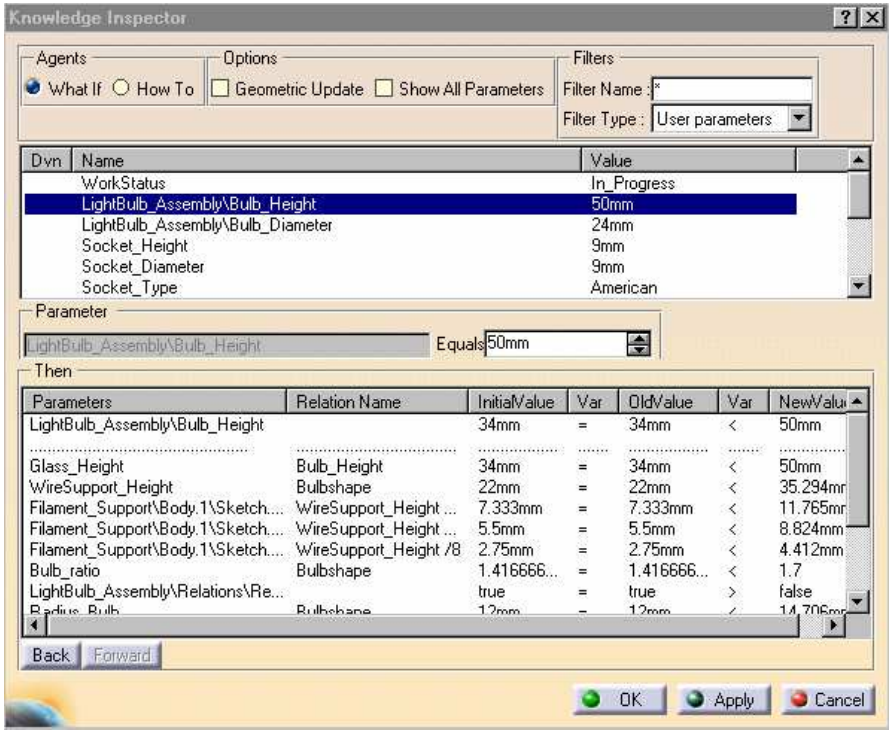
Student Notes:

Light Bulb

Step 3 - Use the Knowledge Inspector Tool

10 min

In this step, you will use the Knowledge Inspector tool to understand what are the impacts in the light bulb design of the Bulb_Height parameter modification.



Student Notes:

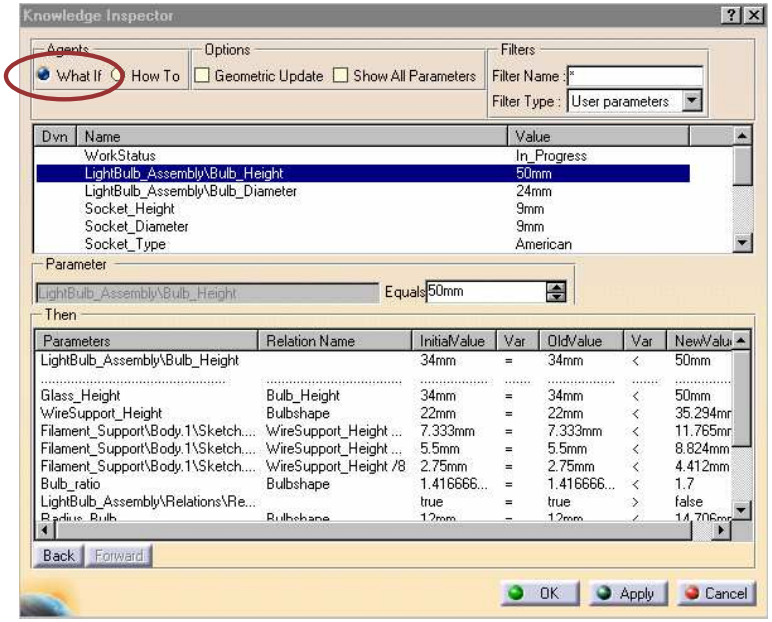
Do It Yourself

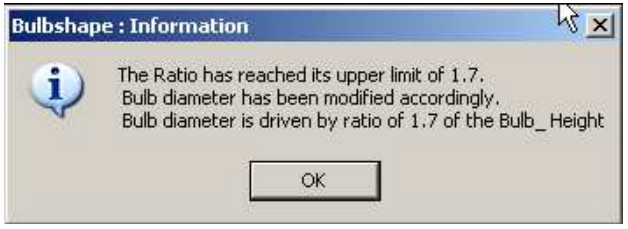
1. **In the Knowledge Inspector panel, select the « What if » mode.**

2. **Change the value of *Bulb_Height* from 34mm to 50mm and click the *Apply* button. Analyze the impacts of this modification.**

3. **Change the value of *Bulb_Height* from 50mm to 26mm and click the *Apply* button. Analyze the impacts.**

4. **Click the *Cancel* button so that the parameter modification is not taken into account.**



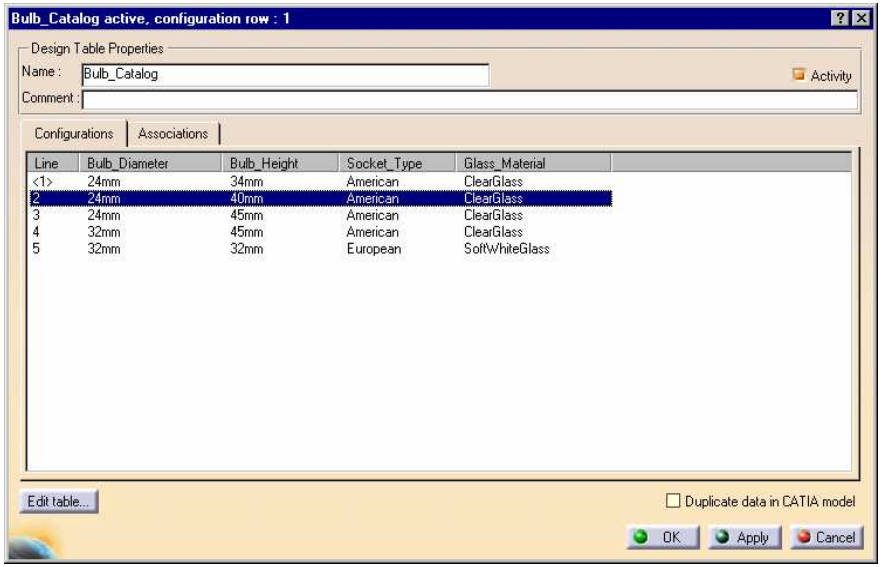


Light Bulb

Step 4 - Create a Design Table

10 min

In this step, you will create a Design Table to manage a light bulb family. Two different creation methods will be used.



Student Notes:

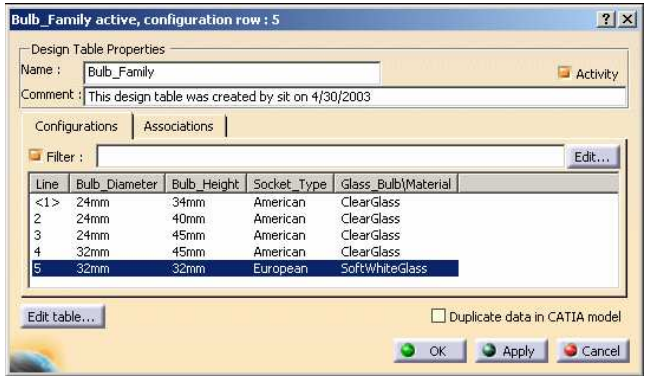
Student Notes:

Do It Yourself

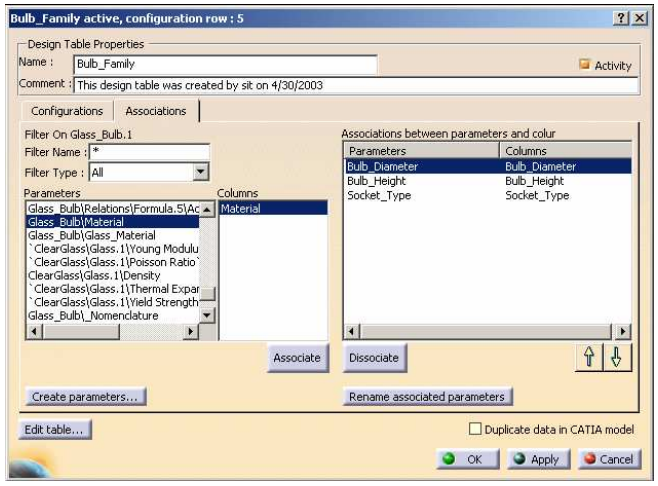
1. **Creation method N°1 :**
   Create a Design Table named « Bulb_Family » and check the option « Create a design table with the current parameter values ». Add four new configurations as shown below.

A	B	C	D
Bulb_Diameter (mm)	Bulb_Height (mm)	Socket_Type	Glass_Bulb\Material
24	34	American	ClearGlass
24	40	American	ClearGlass
24	45	American	ClearGlass
32	45	American	ClearGlass
32	32	European	SoftWhiteGlass

Change the configuration to N°5



2. **Creation method N°2 :**
   Delete the previous Design Table.
   Create a new Excel file using the data shown in the above image:
   « CATKWA_Bulb_Family.xls ». Use the Automatic association and associate manually the « Glass_Bulb\Material » parameter to the « Material » column.



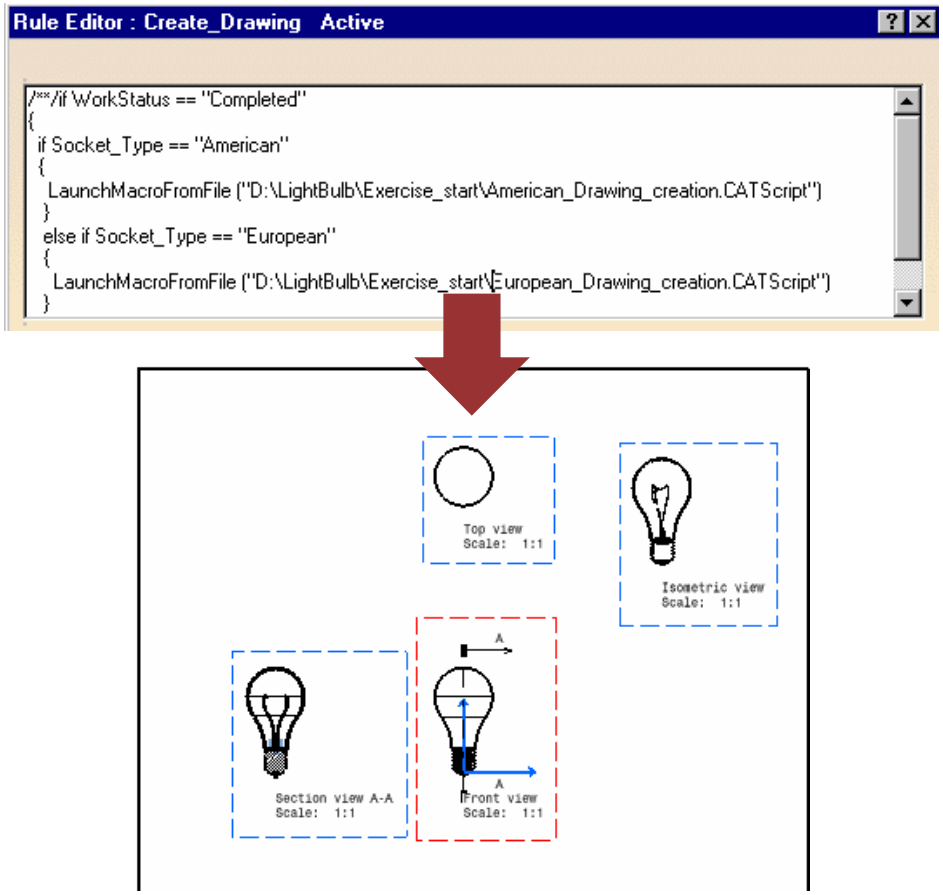
Student Notes:

Light Bulb

Step 5 - Lauching a Drawing Macro from a Rule

5 min

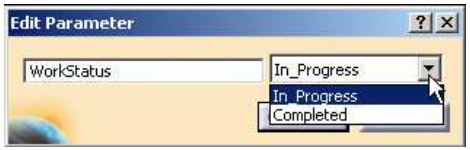
In this step, you will create a Rule that will launch macros to create the bulb drawing automatically. If your bulb is an American type, a macro will create projection views using the third angle standard and if it is an European bulb, another macro will create projection views using the first angle standard.



Student Notes:

Do It Yourself

1. **Create a Parameter of type String with two multiple values (In_Progress and Completed) and rename it « WorkStatus ». Set it to « In_Progress ».**

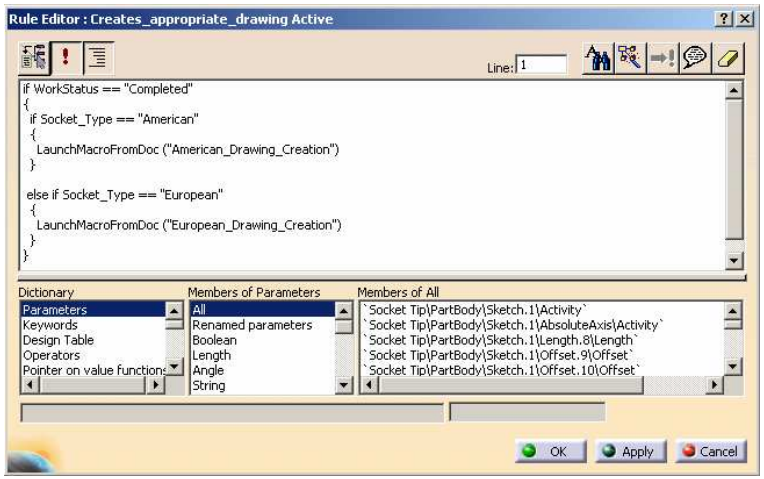


2. **Create a Rule named « Drawing Creation » with the following script :**

```
if WorkStatus == "Completed"
{  if Socket_Type == "American"
  { LaunchMacroFromDoc
("American_Drawing_Creation") }

   else if Socket_Type == "European"

{LaunchMacroFromDoc("European_Drawing_Creation")}}
```



3. **Change the WorkStatus parameter from « In_Progress » to « Completed ». It will automatically create the appropriate drawing views.**

Note: You can view both the used macros using the Tools/Macro command.

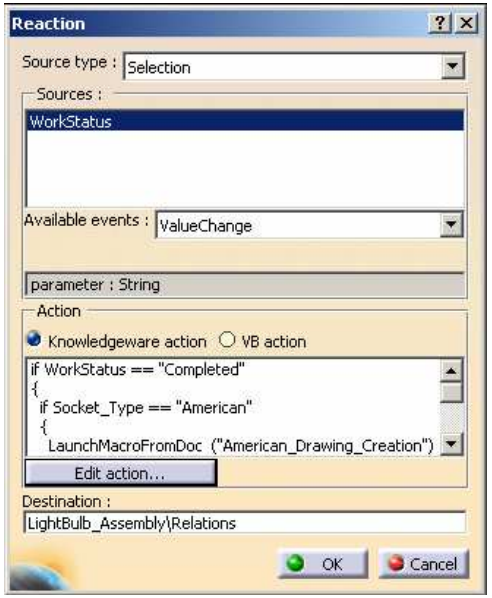
Student Notes:

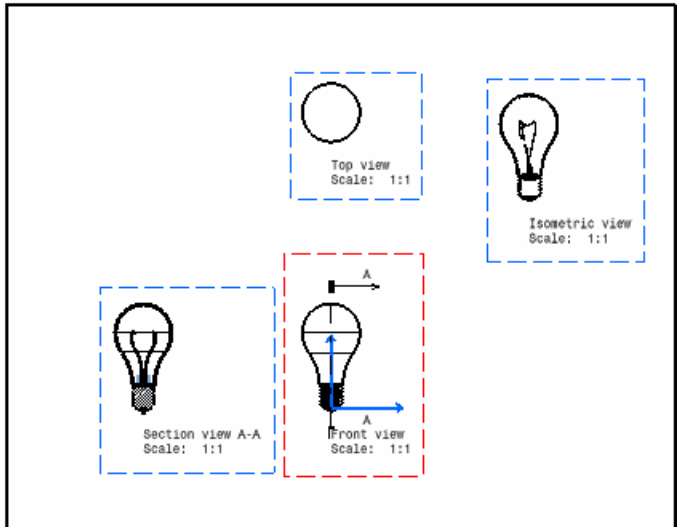
Light Bulb

Step 6 - Launching a Drawing Macro from a Reaction

5 min

In this step, you will create a Reaction that will launch macros to automatically create the bulb drawing.

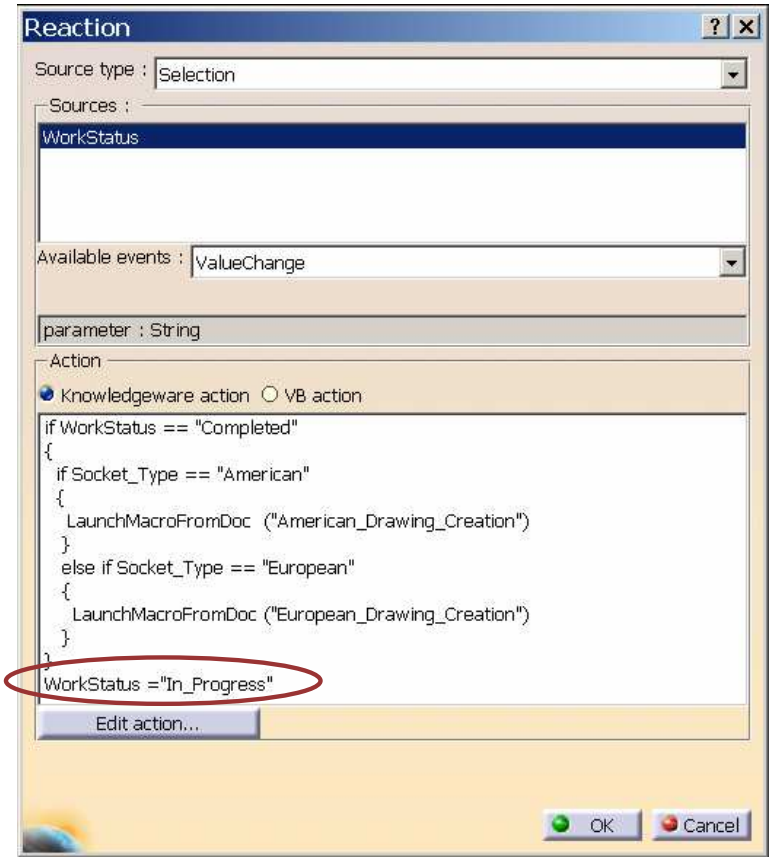




Student Notes:

Do It Yourself

1. **Deactivate the Rule that you have created in the last step.**
2. **Create a Reaction (Knowledgeware action type) which triggers the same drawing process then the « WorkStatus » parameter value is equal « Completed ». At the end of the instructions, add the line WorkStatus =« In_Progress » so that the parameter gets back its original value. Notice that is not possible in a Rule to do so.**
3. **Change the « WorkStatus » parameter from « In_Progress » to « Completed ». It will automatically create the appropriate drawing.**



Student Notes:

Light Bulb

Step 7 : Create a Bulb Glass Family Catalog

10 min

In this step you will:

- Create a bulb glass family catalog
- Reuse it in an assembly

Student Notes:

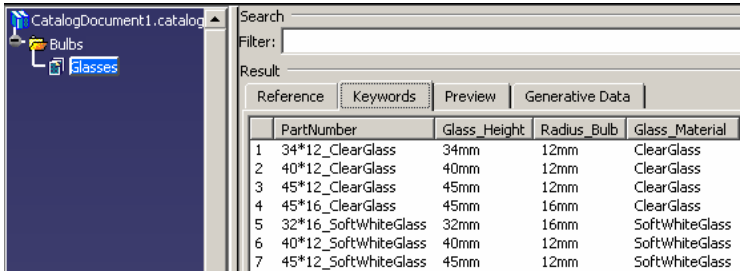
Do It Yourself: Glass Catalog Creation

Open CATKWA_GlassBulb_DT.CATPart

1. Edit DesignTable.1 and insert a column « PartNumber » in the Excel sheet. Specify a PartNumber value for each row, « height*radius_material » for example.
2. Save the CATPart and close it.
3. Create a new CatalogDocument.
4. Rename the default chapter in « Bulbs » and add a part family named « Glasses » using the CATKWA_Glass_Bulb_DT.CATPart document.
5. Resolve the new part family.
6. Save and close the Catalog document.

Glass_Bulb.xls

	A	B	C	D
1	PartNumber	Glass_Height (mm)	Radius_Bulb (mm)	Glass_Material
2	34*12_ClearGlass	34	12	ClearGlass
3	40*12_ClearGlass	40	12	ClearGlass
4	45*12_ClearGlass	45	12	ClearGlass
5	45*16_ClearGlass	45	16	ClearGlass
6	32*16_SoftWhiteGlas	32	16	SoftWhiteGlass
7	40*12_SoftWhiteGlas	40	12	SoftWhiteGlass
8	45*12_SoftWhiteGlas	45	12	SoftWhiteGlass

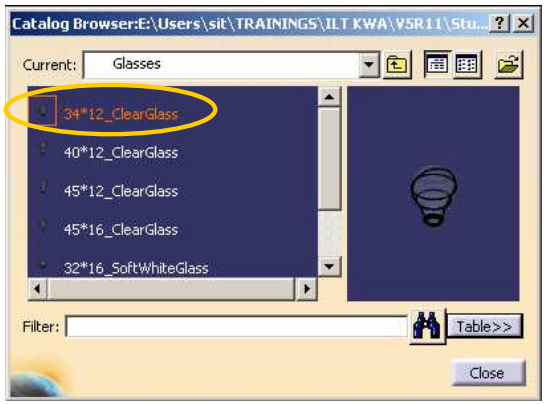


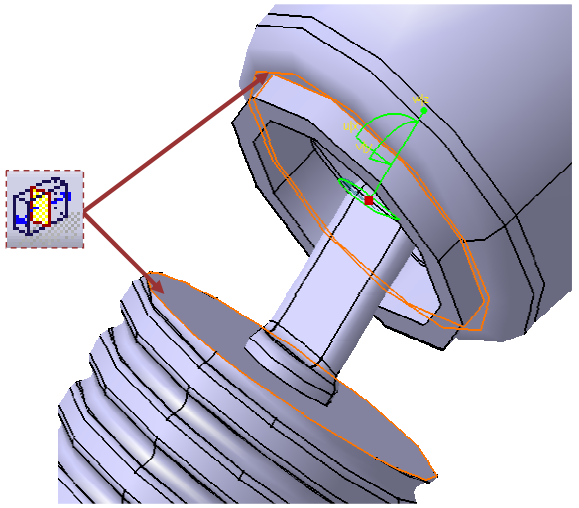
Student Notes:

Do It Yourself: Glass Catalog Reuse

Open CATKWA_LightBulb_Assy_NoGlass.CATProduct

1. In this product, instantiate the « 34*12_ClearGlass » component from your catalog browser. Change its PartNumber into « Glass_Bulb ».
2. Create one coincidence constraint between the Glass_Bulb axis and the Socket axis.
3. Create one contact constraint between the two planes as shown on the right picture.
4. Add the following formula:
   Glass_Bulb\Glass_Height =Bulb_Height





Student Notes:

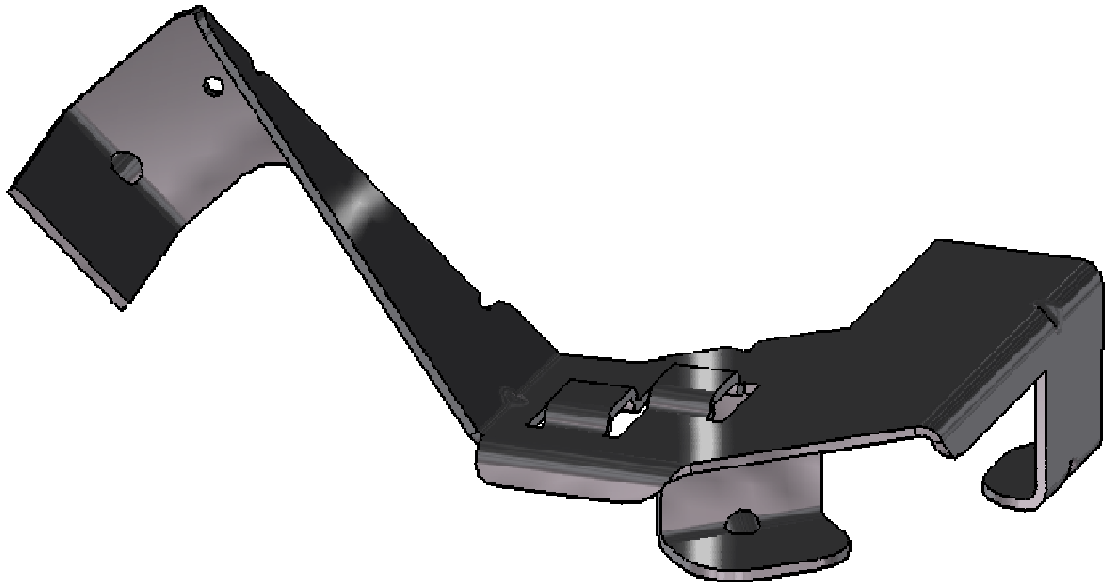
Sheetmetal Part

Sheetmetal Part Exercise: Presentation

25 min

In this exercise you will:

- Use a List to automatically get the total number of bends
- Use a Rule to compute the part's cost
- Use a Check to control the over cost



Student Notes:

Design Intent: Sheetmetal Part

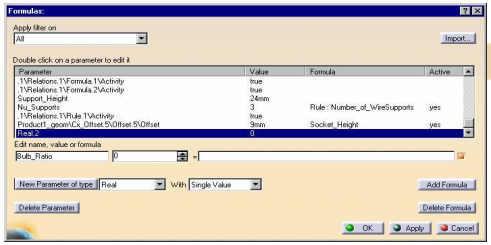
In the context of a part representing a Sheetmetal Part…

- **Creating User Parameters**
- **Creating a List**
- **Creating a Formula**
- **Creating a Rule**
- **Creating a Check**

Student Notes:

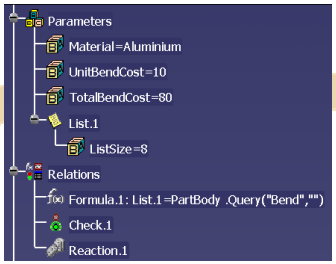
Design Process: Sheetmetal Design

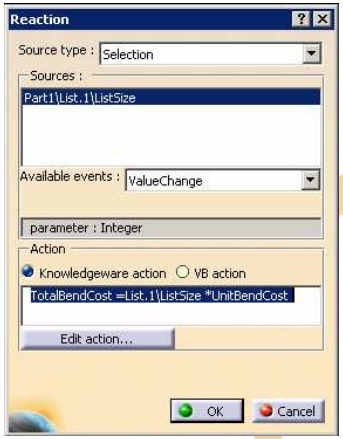
Step 1 : Create User Parameters and Formulas



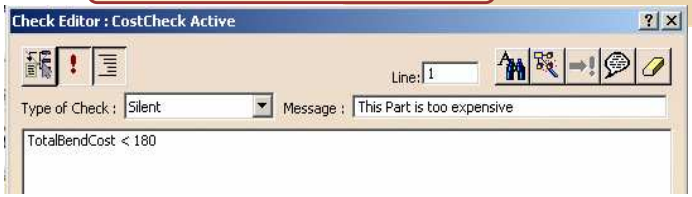
Step 2 : Create a bend List

Step 3 : Create a cost Rule

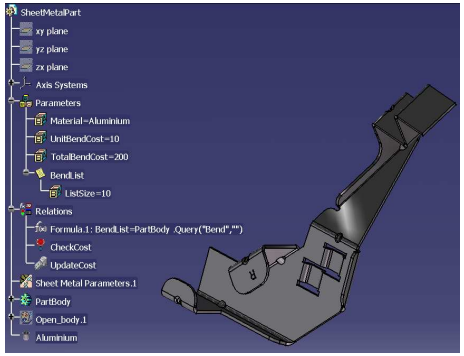




Step 4: Create a cost Check



Step 5: Create new Bends and evaluate the cost



Student Notes:

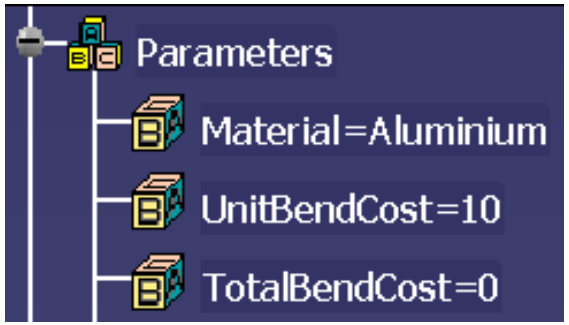
Sheetmetal Part

Step 1 - Create User Parameters

5 min

In this step, you will create two User Parameters:

- **"UnitBendCost" for the cost of one bend**
- **"TotalBendCost" for the bend's total cost**



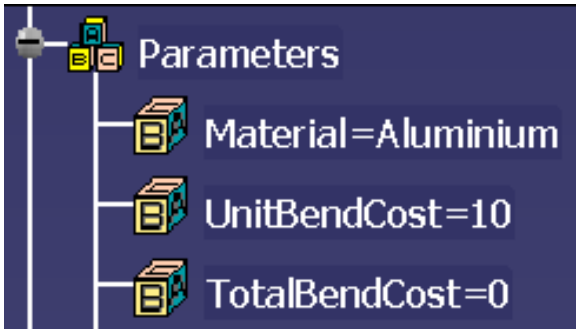
Student Notes:

Do It Yourself

Open SheetMetalPart.CATPart

Create two Parameters :

UnitBendCost = 10 as Real.
TotalBendCost = 0 as Real.



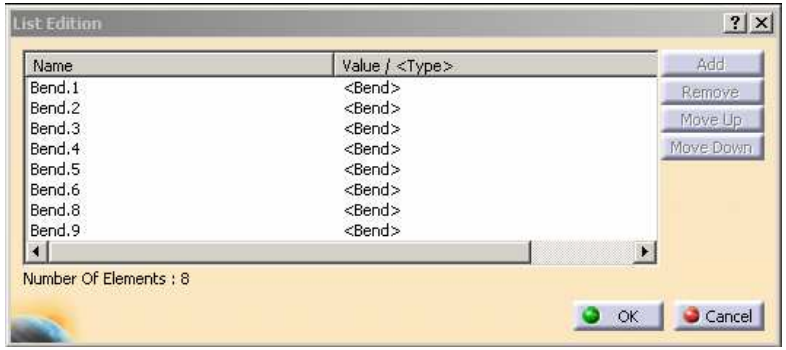
Student Notes:

Sheetmetal Part

Step 2 - Create a Bend List

5 min

In this step, you will create a List that will be automatically populated with the sheetmetal part bends.



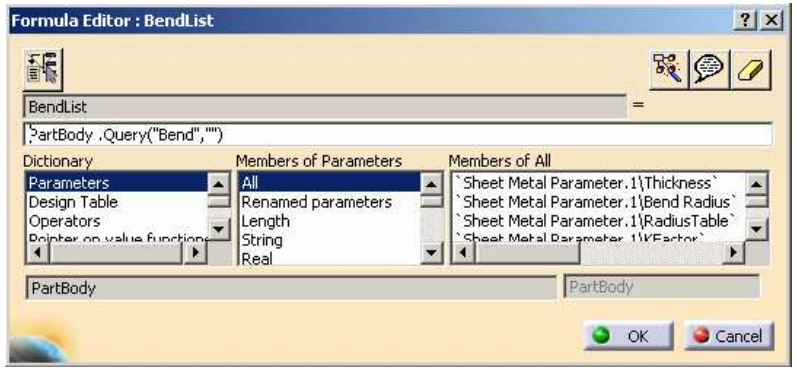
Student Notes:

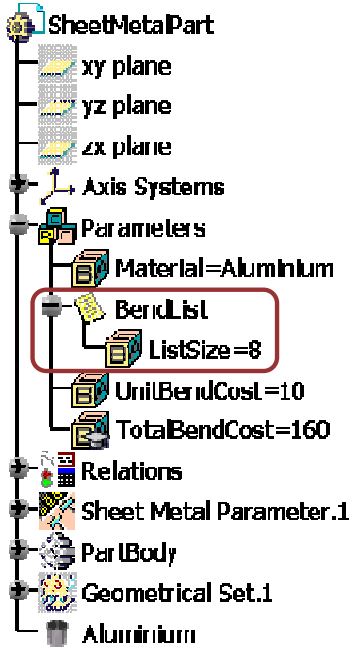
Do It Yourself

1. **Create a new empty List named « BendList »**

2. **Add the following formula on the BendList parameter to automatically populate the List with the existing bends:**
   BendList = PartBody .Query("Bend","")

Note that there is a space between "PartBody" and ".Query" …





Student Notes:

Sheet Metal Part

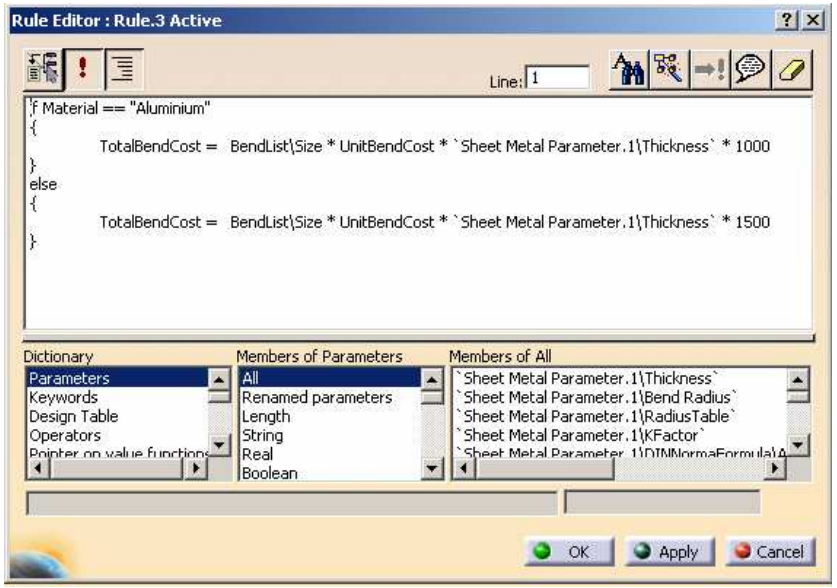
Step 3 - Create a Cost Rule

5 min

In this exercise, you will create a Rule that will compute the total cost of the bends.

This cost depends on:

- **The number of bends**
- **The bend unit cost**
- **The sheetmetal part's thickness**
- **The sheetmetal part's material**

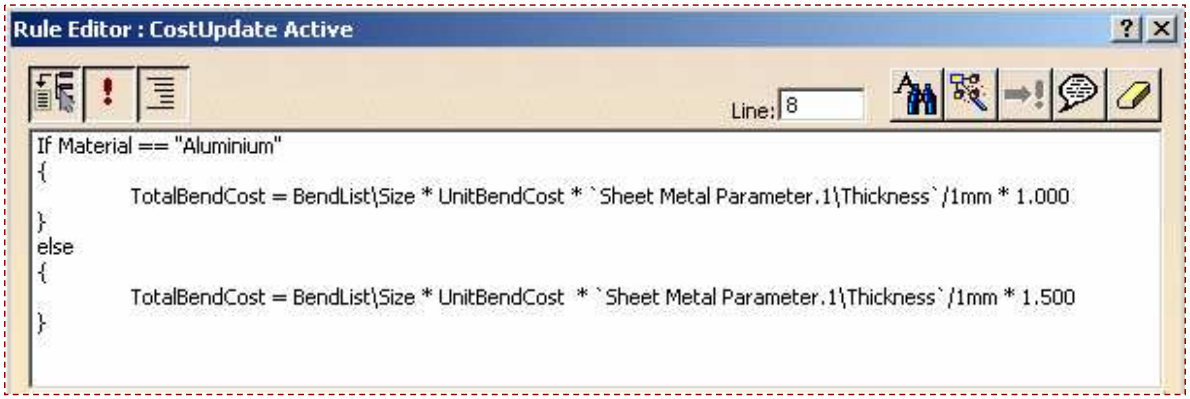


Student Notes:

Do It Yourself

1. **Create a Rule named « CostUpdate » with the following instructions :**

```
If Material == "Aluminium"
{TotalBendCost = BendList\Size * UnitBendCost * `Sheetmetal Parameter.1\Thickness`/1mm * 1.000}
else
{TotalBendCost = BendList\Size * UnitBendCost  * `Sheetmetal Parameter.1\Thickness`/1mm * 1.500}
```



Student Notes:

Sheet Metal Part

Step 4 - Create a Cost Check

5 min

In this step, you will create a Check to warn the user in case the maximum cost value is exceeded.

Student Notes:

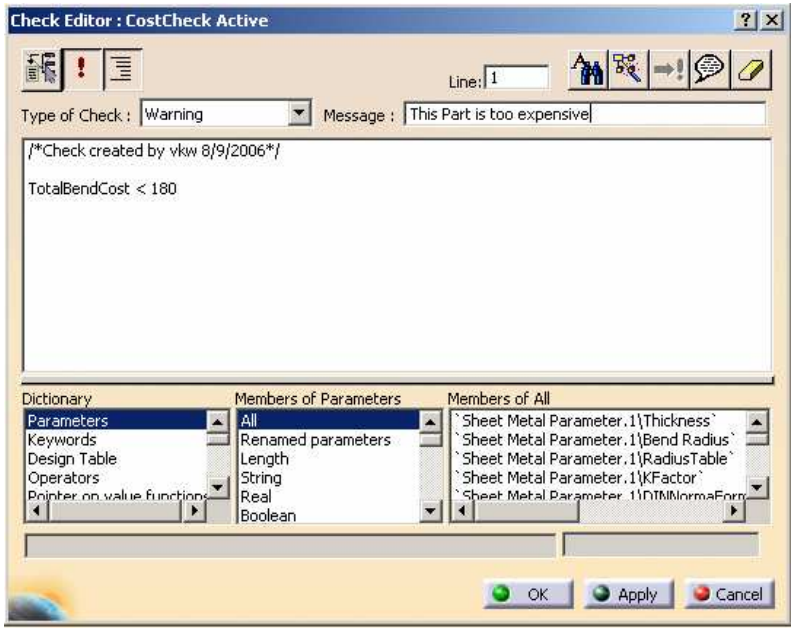
Do It Yourself

1. **Create a Check named « CostCheck » and defined by the condition :**

   TotalBendCost <180

2. **Select Warning for the Type of Check and key in the following message:**

   "*This Part is too expensive*"



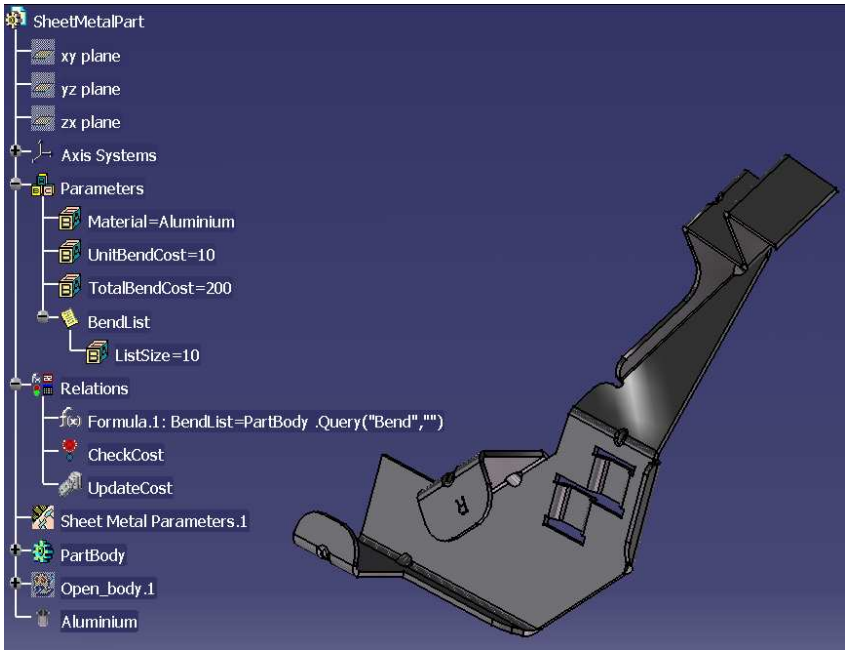
Student Notes:

Sheet Metal Part

Step 5 - Add New Bends and Evaluate the New Cost

5 min

In this step, you will create a new wall with bend.

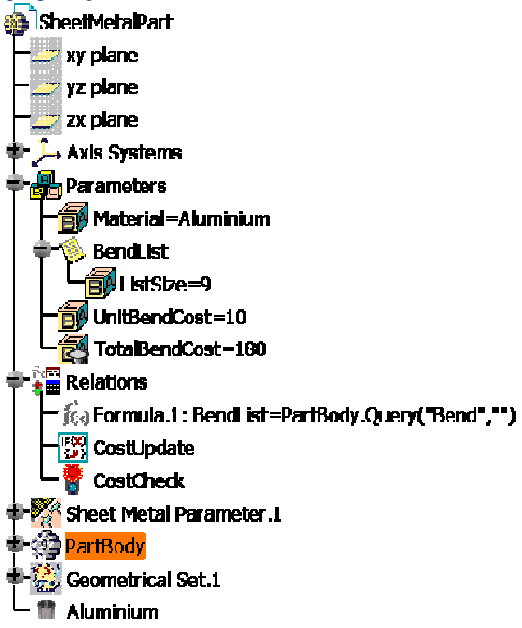


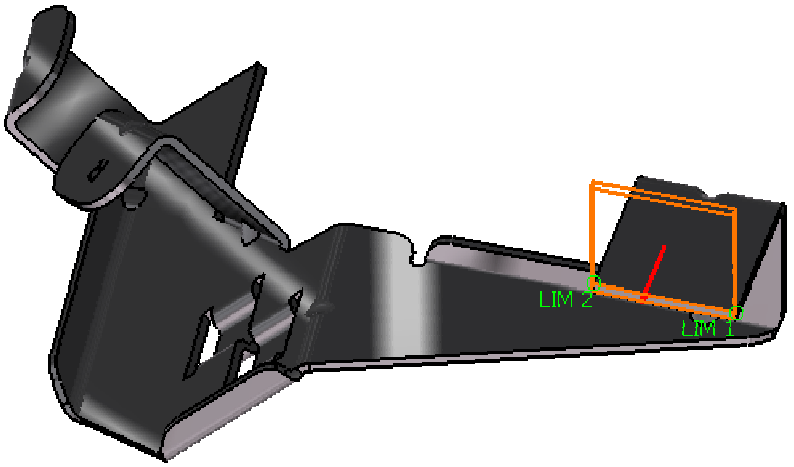
Student Notes:

Do It Yourself

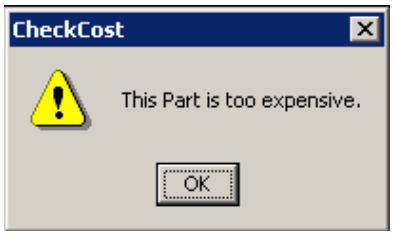
1. Enter the Sheetmetal Design workbench. (Use license SD2.slt).
2. Click the « Wall on edge icon » and check the « With Bend » option:

3. Select the edge as shown beside and create the wall.





Notice the new values of the « ListSize » and the « TotalBendCost » parameters. The Check's light turns to red and a warning message is displayed.



EXERCISE BOOK

Student Notes:

Wheel Rim Exercise

- Wheel Rim Exercise Presentation
- Wheel Rim Exercise Part 1
- Wheel Rim Exercise Part 2
- Wheel Rim Exercise Part 3

Student Notes:

Wheel Rim

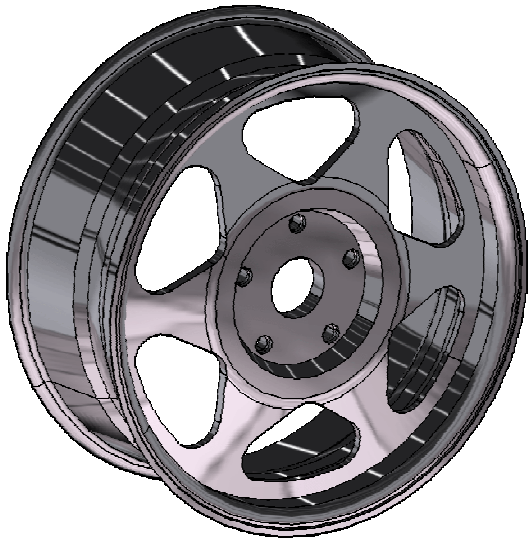
Exercise Presentation

1hr 25min

In this exercise, you will automate the design of a wheel rim and define a wheel rim family.

You will practice on:

- **Creating Parameters**
- **Creating Formulas**
- **Creating Rules and Checks**
- **Generating a Part Family through design tables**
- **Creating a Reaction**

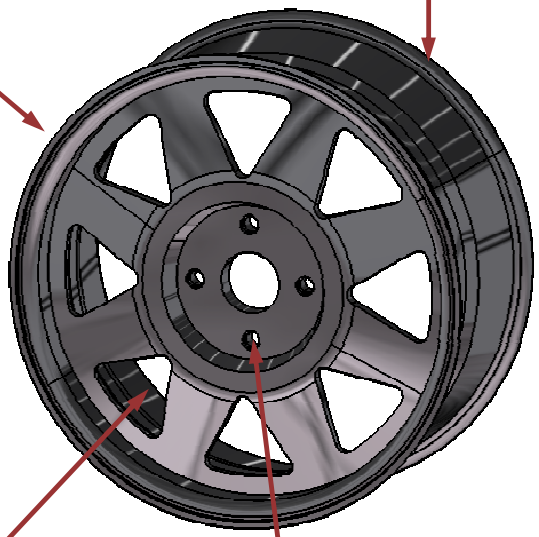


Student Notes:

Design Intent: Wheel Rim

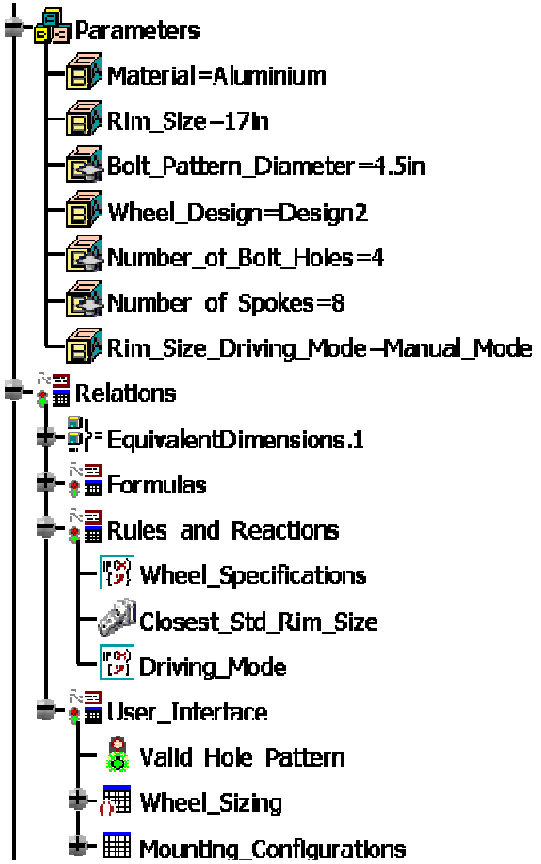
Rim:
- Its diameter can be driven either manually or through a design table
- Its diameter always has a standard value
- Its width adapts to the chosen diameter

Material:
- Depends of the rim diameter



Spokes:
- Three available designs
- Their size adapts automatically to the rim diameter

Bolts Crown:
- Six pre-defined configurations
- A Check verifies that the crown is not too large compared to the hub diameter

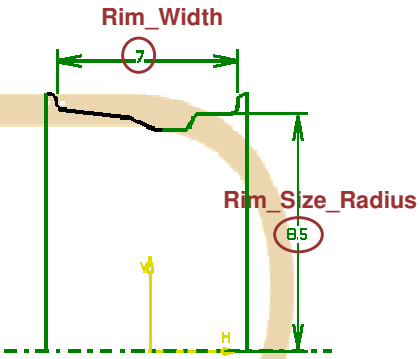


Student Notes:

Design Process: Wheel Rim - Part 1

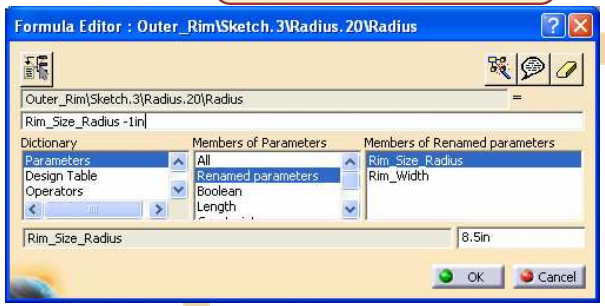
1

Rename the parameters



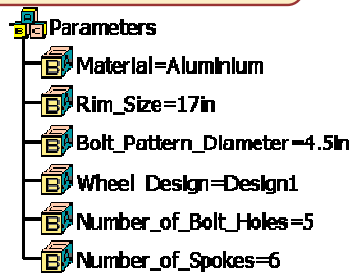
2

Assign formulas to the geometric parameters



3

Create User Parameters and drive them by formulas

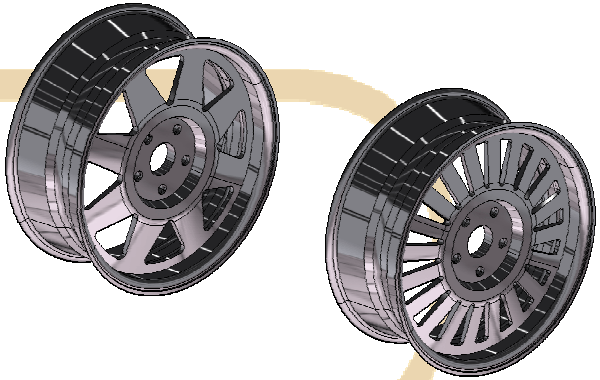


Student Notes:

Design Process: Wheel Rim - Part 2

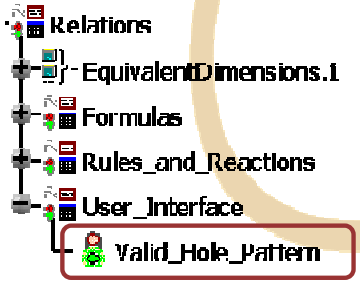
4

Create a Rule that pilots the spokes design



5

Create a Check to control the position of the bolt holes

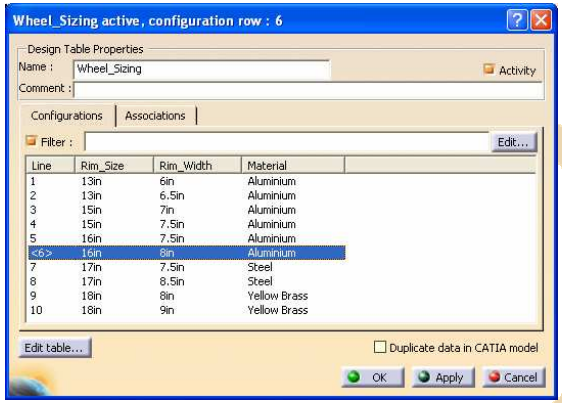


Student Notes:

Design Process: Wheel Rim - Part 3

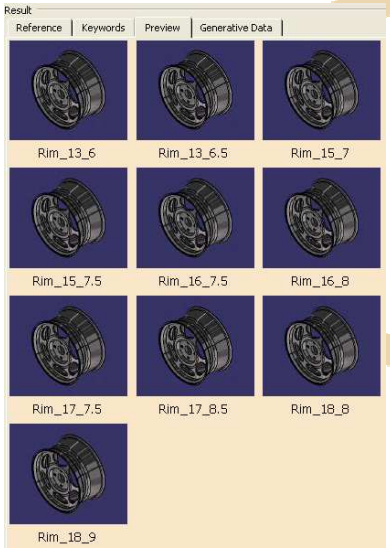
6

Create two Design Tables



7

Generate a Part Family



8

Create a Reaction and a Rule to control the Rim diameter edition mode

Student Notes:

Wheel Rim Exercise Part 1

- Step 1: Rename Parameters
- Step 2: Assign Formulas to Intrinsic Parameters
- Step 3: Create User Parameters

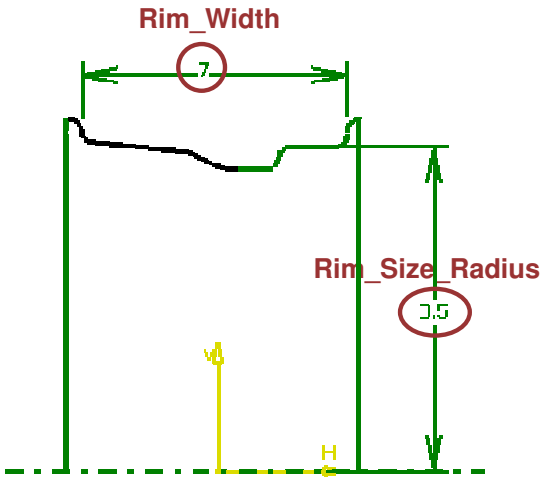
Student Notes:

Wheel Rim

Step 1 - Rename Parameters

10 min

In this step, you will rename some sketch parameters.

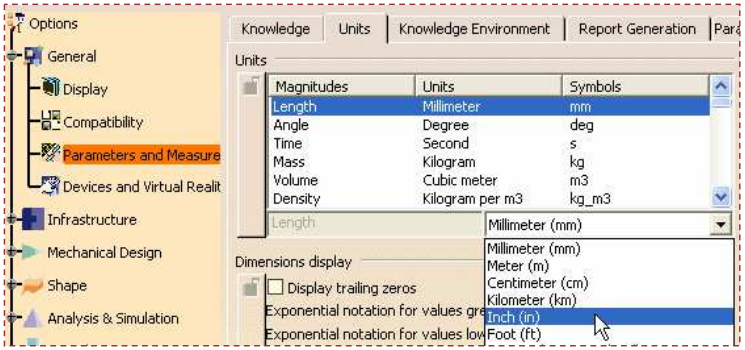


Student Notes:

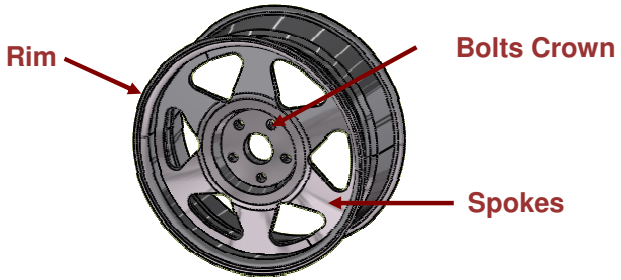
Do It Yourself

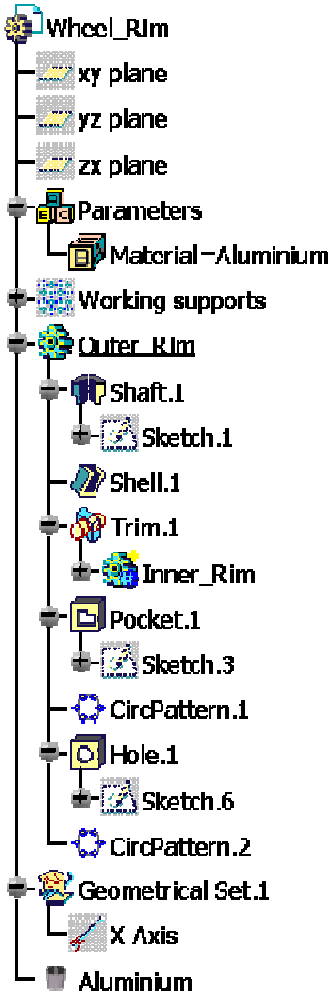
Open CATKWA_Wheel_Rim_Part1_Start.CATPart

1. Optional: Change the Length unit to Inch in Tools/Options. If you do not, the dimensions in inches will automatically be converted into mm and the values will be displayed in mm.



2. Open the CATPart and study its features already created for you.





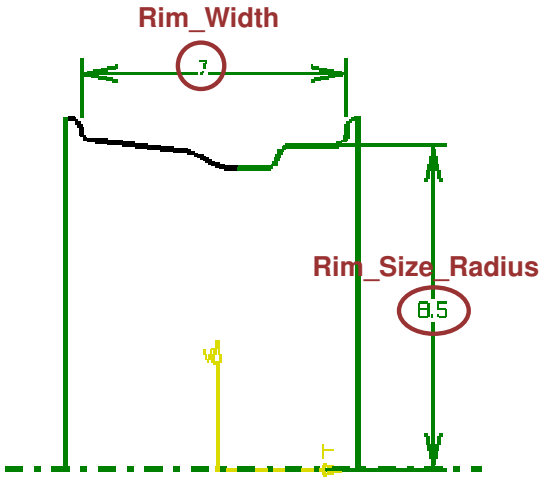
Student Notes:

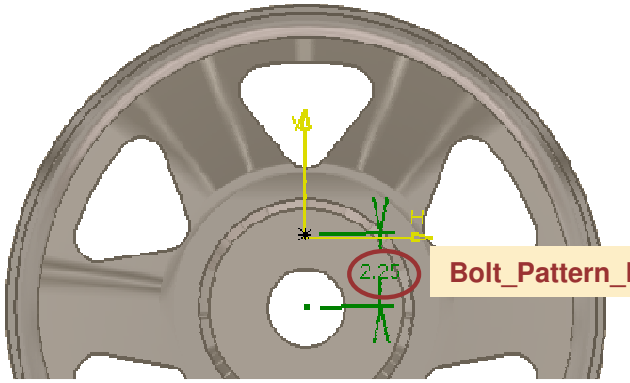
Do It Yourself

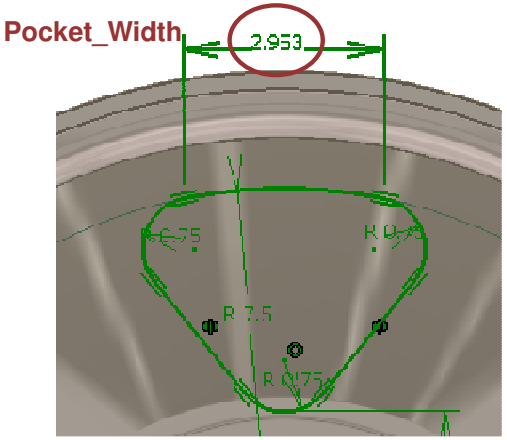
3. **In Sketch.1, rename the following parameters using their contextual menu and the "Rename parameter" option:**
   - **"Outer_Rim\Sketch.1\Offset.40\Offset" in "Rim_Width"**
   - **"Outer_Rim\Sketch.1\Offset.39\Offset" in "Rim_Size_Radius"**

4. **Create a distance constraint of 2.25in between the Hole.1 center point and the central cylindrical surface. Rename this distance parameter "Bolt_Pattern_Radius".**

5. **In Sketch.3, rename the pocket width dimension "Outer_Rim\Sketch.3\Offset.11\Offset" by "Pocket_Width".**







Student Notes:

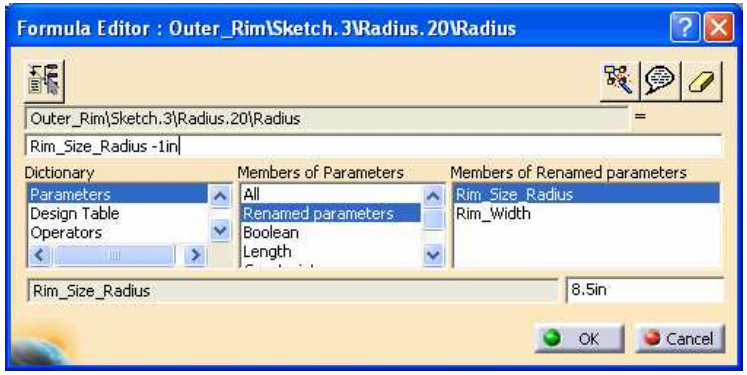
Wheel Rim

Step 2 - Assign Formulas to Intrinsic Parameters

10 min

In this step you will:

- **Assign formulas to intrinsic Parameters**
- **Define Equivalent Dimensions**



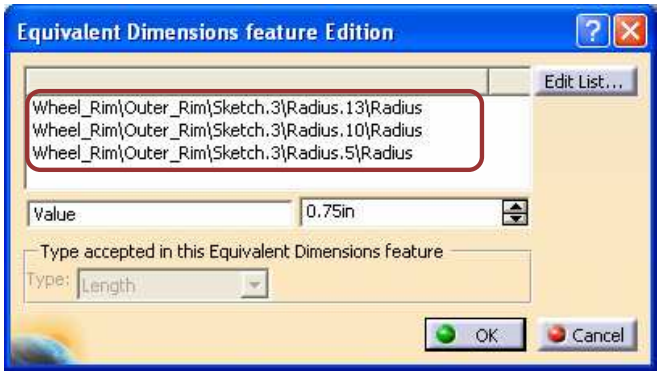
Student Notes:

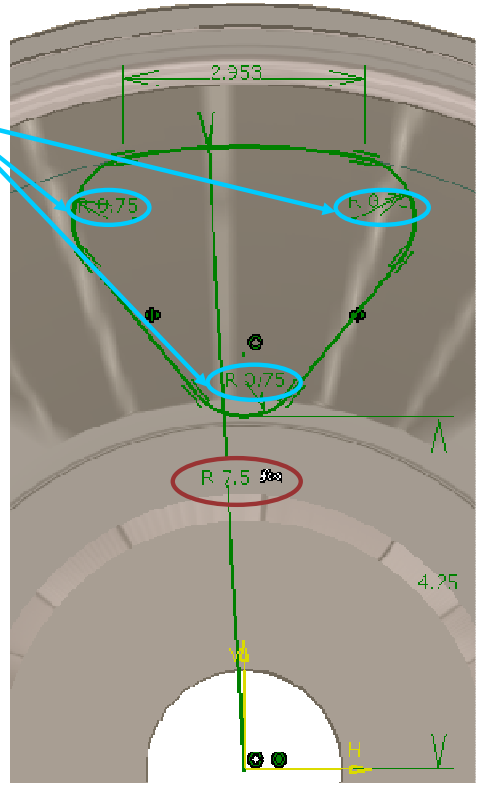
Do It Yourself

Open CATKWA_Wheel_Rim_Part1_Step2_Start.CATPart

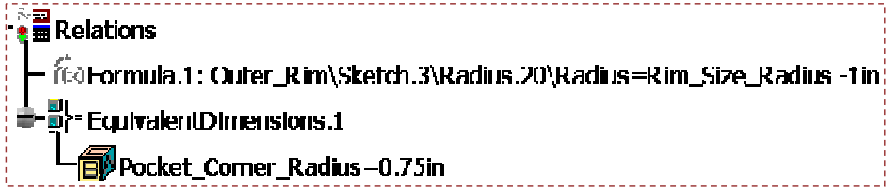
1. In Sketch.3, assign the following formula to the pocket radius dimension:
   Outer_Rim\Sketch.3\Radius.20\Radius=Rim_Size_Radius -1in

2. Then create a new *EquivalentDimensions* feature with the value 0.75in and select the three radius parameters:





3. Rename the EquivalentDimensions parameter as "Pocket_Corner_Radius".



Student Notes:

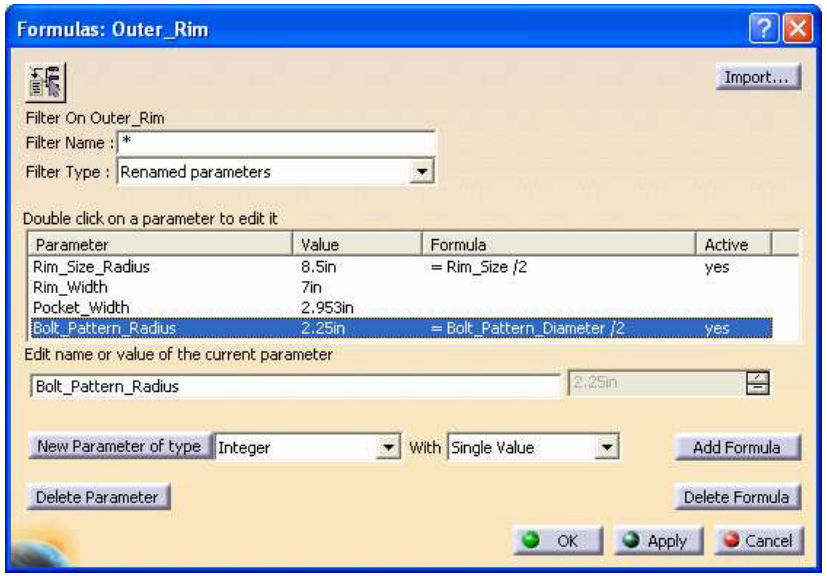
Wheel Rim

Step 3 - Create User Parameters and Use them in Formulas

10 min

In this step you will:

- **Create User Parameters**
- **Create formulas in order to make these User Parameters drive the geometry**

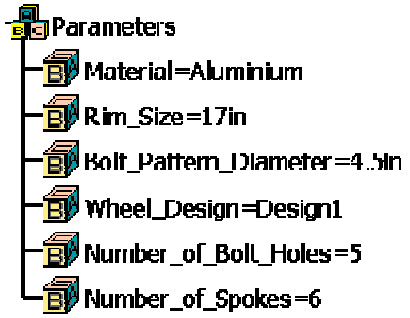


Student Notes:

Do It Yourself

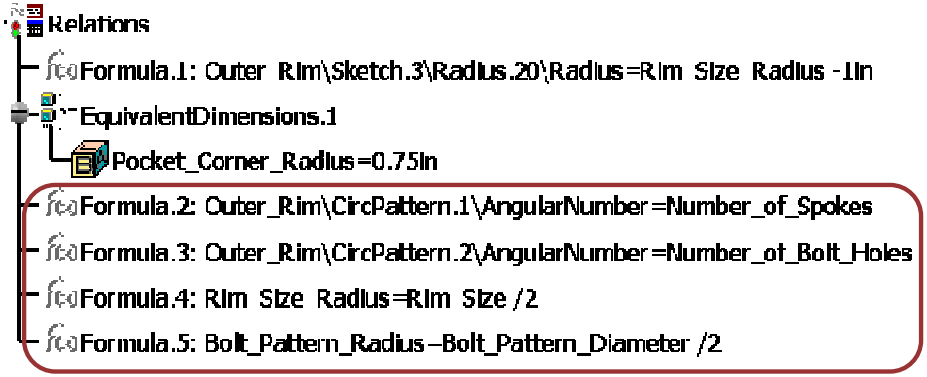
Open CATKWA_Wheel_Rim_Part1_End.CATPart to check your result.

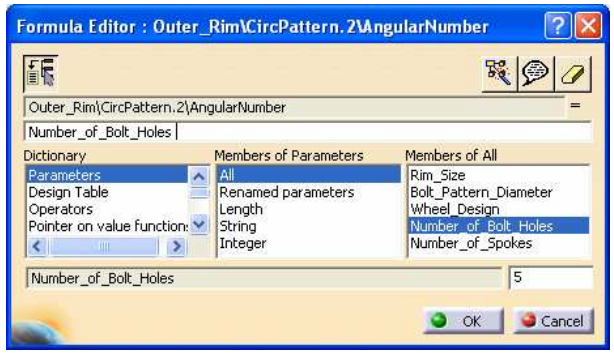
1. Create five new User Parameters:
   - *Rim_Size = 17 in*
   - *Bolt_Pattern_Diameter = 4.5 in*
   - *Wheel_Design = [« Design1 », « Design2 », « Design3 »]*
   - *Number_of_Bolt_Holes = 5*
   - *Number_of_Spokes = 6*



2. Create the following formulas:

Outer_Rim\CircPattern.1\AngularNumber=Number_of_Spokes
Outer_Rim\CircPattern.2\AngularNumber=Number_of_Bolt_Holes
Rim_Size_Radius = Rim_Size / 2
Bolt_Pattern_Radius = Bolt_Pattern_Diameter/2





Student Notes:

Wheel Rim Exercise Part 2

- Step 4: Create a Design Rule
- Step 5: Create a Check on the Bolts Crown

Student Notes:

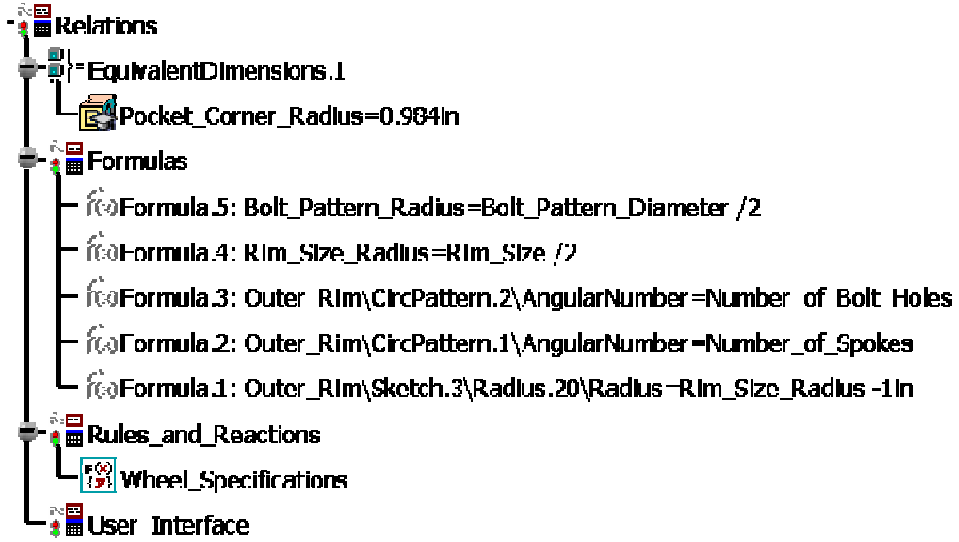
Wheel Rim

Step 4 - Create a Design Rule

15 min

In this step you will:

- Create Sets of Relations and reorganize the Relations node
- Create a Rule that automatically modifies the style of the wheel rim spokes depending on a parameter value

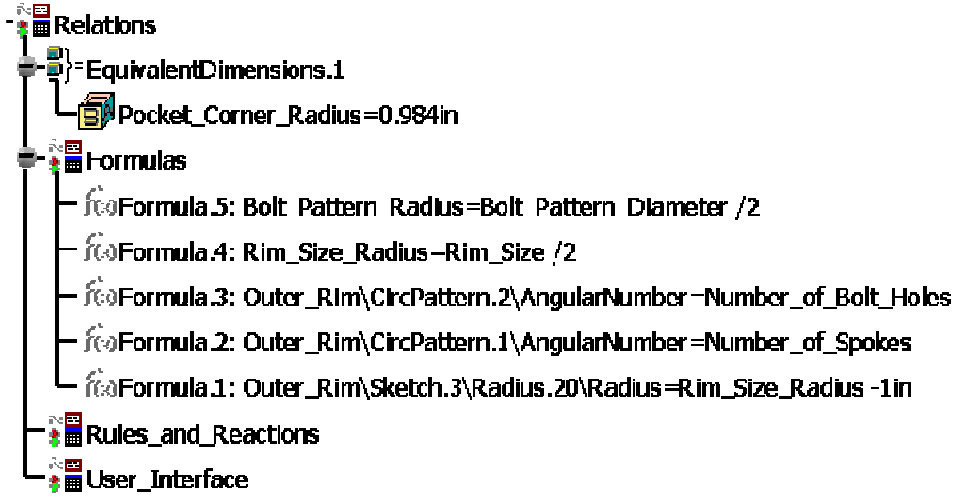


Student Notes:

Do It Yourself (1/3)

Open CATKWA_Wheel_Rim_Part2.CATPart.

1. In the Relations node, create three sets of Relations named as on the picture below. In a set called “Formulas” place the five existing formulas.



Student Notes:

Do It Yourself (2/3)

2. In the "Rules_and_Reactions" set, create a Rule named "Wheel_Specifications".

   This rule will modify:
   - The dimensions of the spokes (Pocket_Corner_Radius and Pocket_Width)
   - The number of the spokes (Number_of_Spokes)

   depending on:
   - The design solution chosen by the user (trough Wheel_Design)
   - The size of the rim (given by Rim_Size).

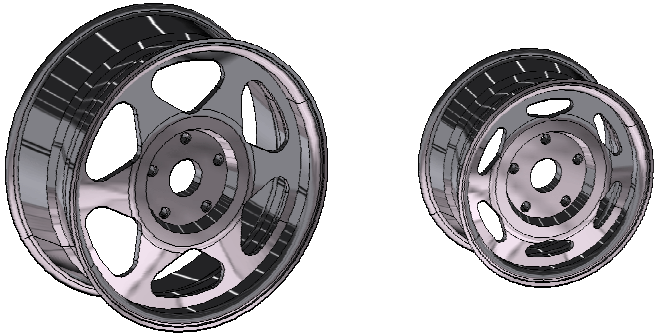
   The table below gives you a view of all the configurations and the result values provided by the rule.

Wheel_Design value:		Design1	Design2	Design3
Rim_ Size value:	< 14in	Pocket_Width=75mm Pocket_Corner_Radius=10mm Number_of_Spokes=6	Pocket_Width=75mm Pocket_Corner_Radius=10mm Number_of_Spokes=8	Pocket_Width=20mm Pocket_Corner_Radius=5mm Number_of_Spokes=20
	< 16in	Pocket_Width=95mm Pocket_Corner_Radius=15mm Number_of_Spokes=6		
	≥ 16in	Pocket_Width=95mm Pocket_Corner_Radius=25mm Number_of_Spokes=6		

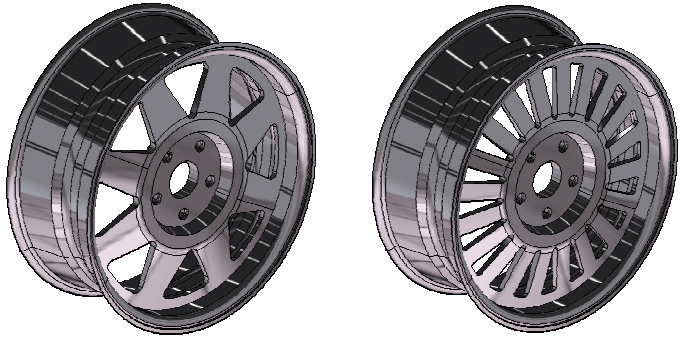
Student Notes:

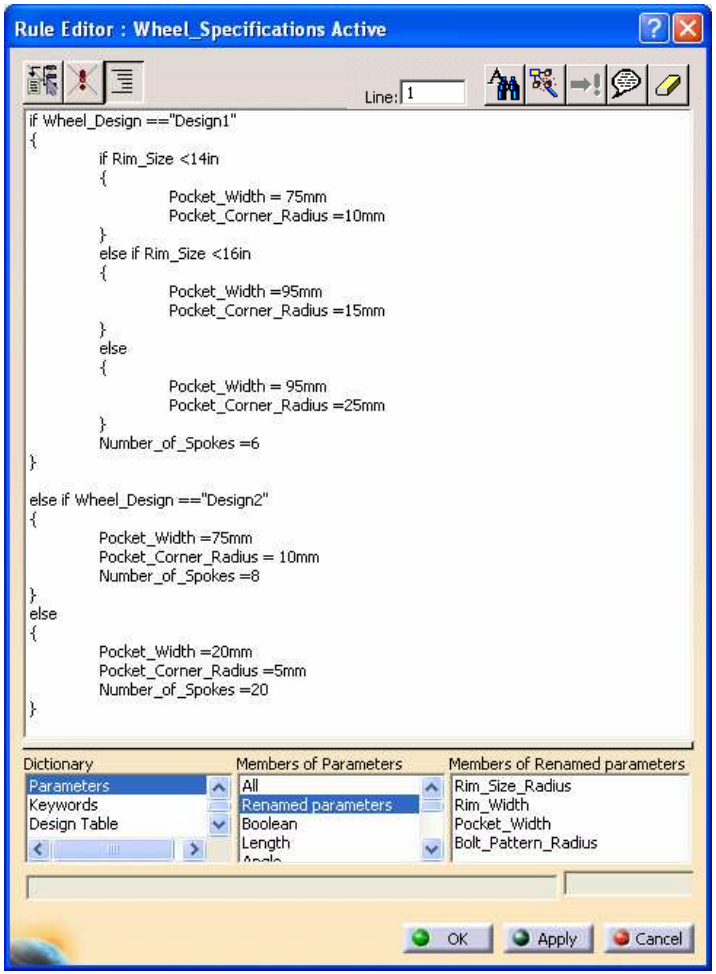
Do It Yourself (3/3)

3. Change the Rim_Size parameter from 17in to 13in and set it back to 17in.



4. Set the Wheel_Design parameter to “Design2”, then “Design3”, then “Design1”.





Student Notes:

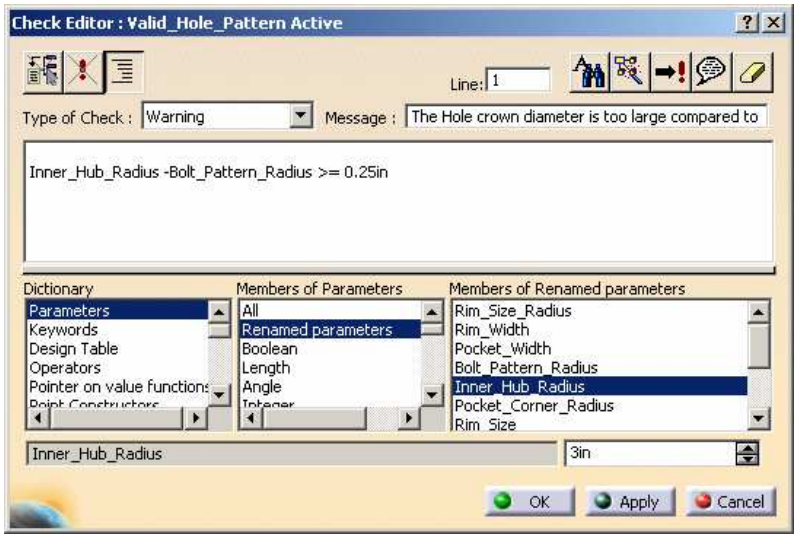
Wheel Rim

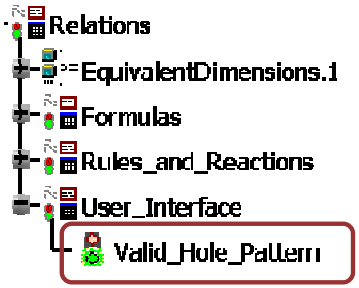
Step 5 - Create a Check on the Bolts Crown

5 min

In this step you will:

- **Create a Check to ensure that the bolt holes crown does not clash with the hub.**





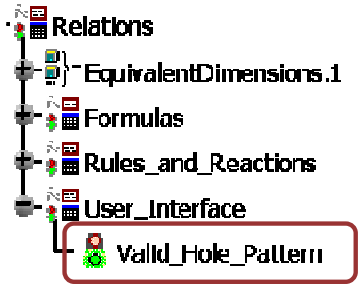
Student Notes:

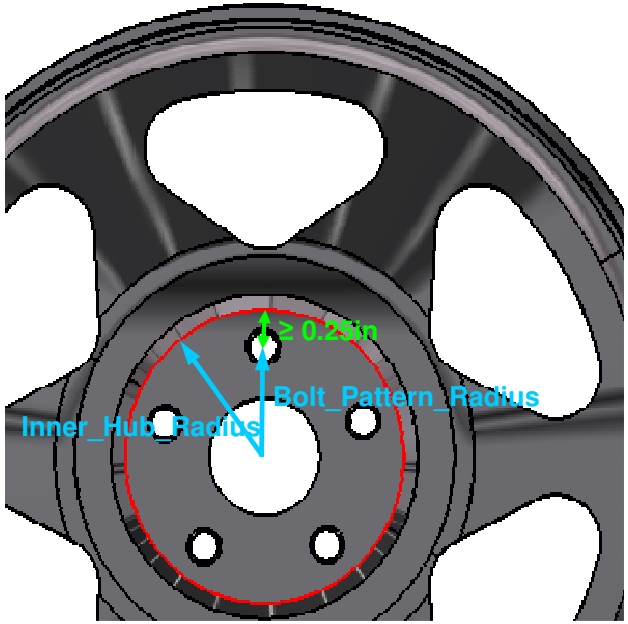
Do It Yourself

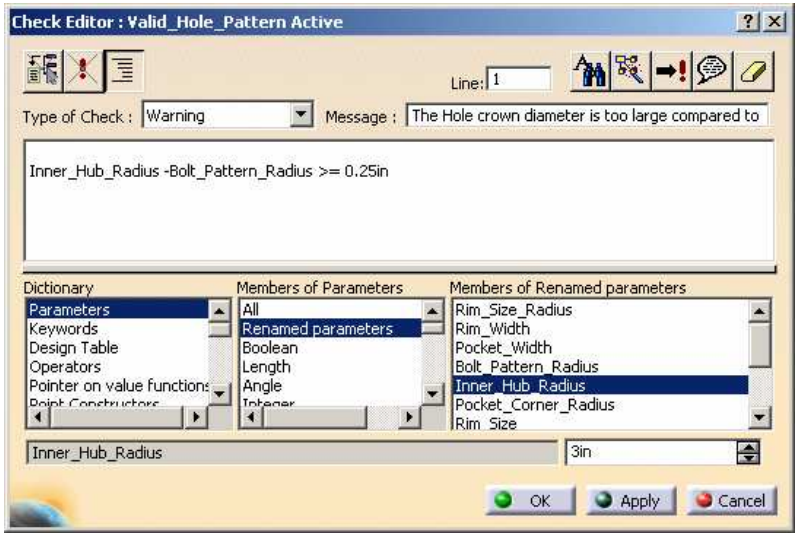
Open CATKWA_Wheel_Rim_Part2_End.CATPart to check your result.

1. In the "User Interface" set of Relations, create a new Check called « Valid_Hole_Pattern ».
   This Check will verify that the radius of the bolt holes crown is at least 0.25in less than the radius of the inner hub.

   Define a Warning message. For instance: *"The Hole crown diameter is too large compared to the hub diameter"*.







Student Notes:

Wheel Rim Exercise Part 3

- Step 6: Create Two Design Tables
- Step 7: Generate a Part Family
- Step 8: Create a Reaction to Control the Wheel Rim Diameter

Student Notes:

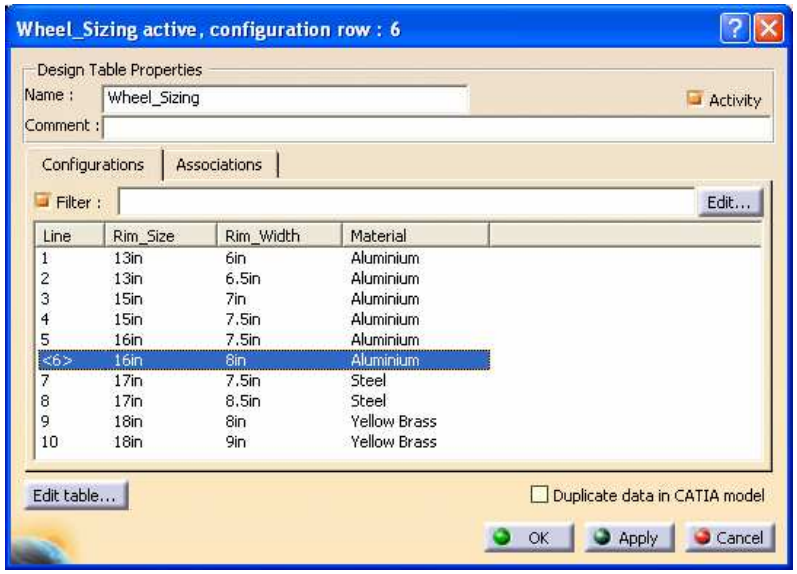
Wheel Rim

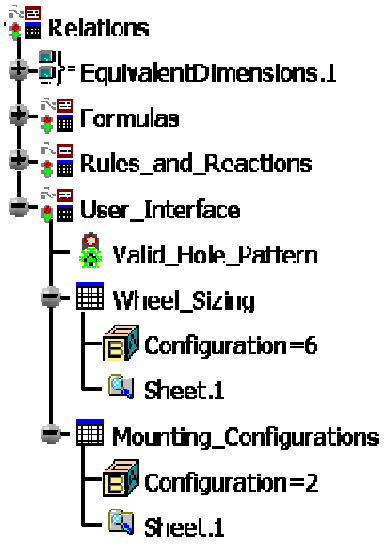
Step 6 - Create two Design Tables

15 min

In this step, you will create two design tables in order to drive:

- **Different wheel dimension configurations**
- **Different wheel mounting configurations (bolt holes)**





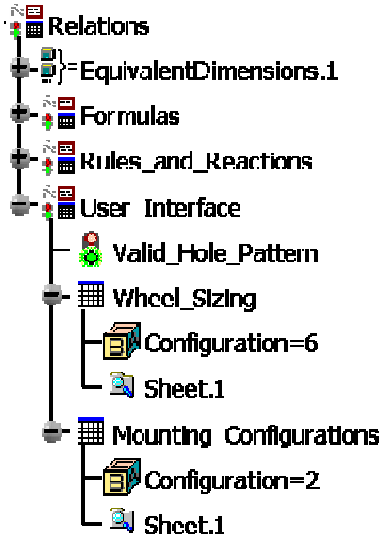
Student Notes:

Do It Yourself

Open CATKWA_Wheel_Rim_Part3.CATPart.

1. In the "User Interface" set of Relations, create a new Design Table named "Wheel_Sizing" using the "Rim_Size", "Rim_Width", and "Material" parameters of the document.
2. Add the following configurations to the design table and apply configuration 6:

Line	Rim_Size	Rim_Width	Material
1	13in	6in	Aluminium
2	13in	6.5in	Aluminium
3	15in	7in	Aluminium
4	15in	7.5in	Aluminium
5	16in	7.5in	Aluminium
<6>	16in	8in	Aluminium
7	17in	7.5in	Steel
8	17in	8.5in	Steel
9	18in	8in	Yellow Brass
10	18in	9in	Yellow Brass



3. In the "User Interface" set of Relations, create a second Design Table named "Mounting_Configuration" using the "Number_of_Bolt_Holes" and "Bolt_Pattern_Diameter" parameters of the document.
4. Add the following configurations to the design table and apply configuration 2:

Line	Number_of_Bolt_Holes	Bolt_Pattern_Diameter
1	4	4in
<2>	4	4.5in
3	5	4.5in
4	5	5in
5	5	5.5in
6	6	5in

5. Try configuration 5 of "Mounting_Configurations" and different configurations of both the design tables. Then come back to the initial values.

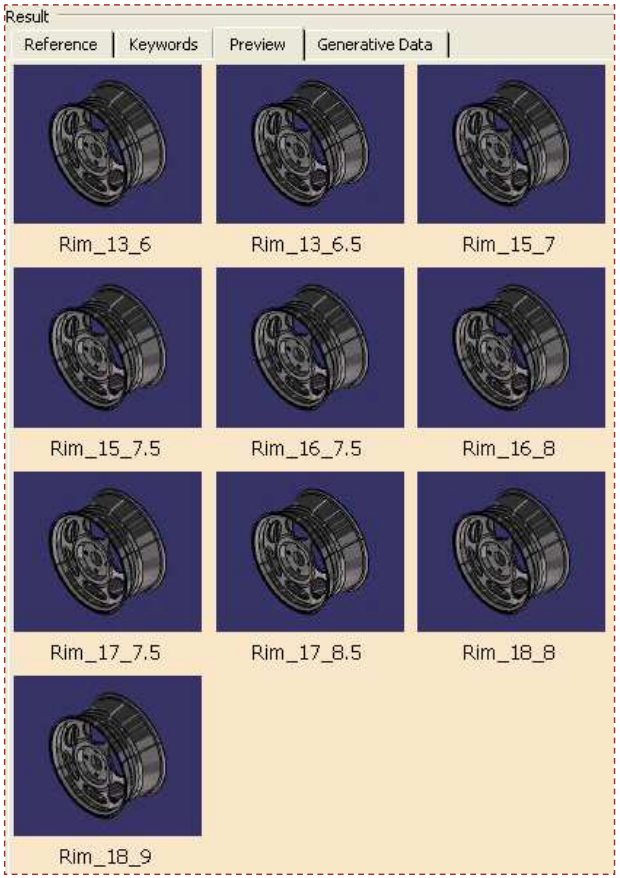
Student Notes:

Wheel Rim

Step 7 – Generate a Part Family

10 min

In this step, you will generate a Part Family from a Design Table.



Student Notes:

Do It Yourself (1/3)

1. Edit “Wheel_Sizing” Design Table in order to add a column called “PartNumber”. This column contains the names that will be given to the CATParts that will be generated.

PartNumber	Rim_Size (in)	Rim_Width (in)	Material
Rim_13_6	13	6	Aluminium
Rim_13_6.5	13	6.5	Aluminium
Rim_15_7	15	7	Aluminium
Rim_15_7.5	15	7.5	Aluminium
Rim_16_7.5	16	7.5	Aluminium
Rim_16_8	16	8	Aluminium
Rim_17_7.5	17	7.5	Steel
Rim_17_8.5	17	8.5	Steel
Rim_18_8	18	8	Yellow Brass
Rim_18_9	18	9	Yellow Brass

2. Save and close the CATPart.

3. Create a new CatalogDocument.

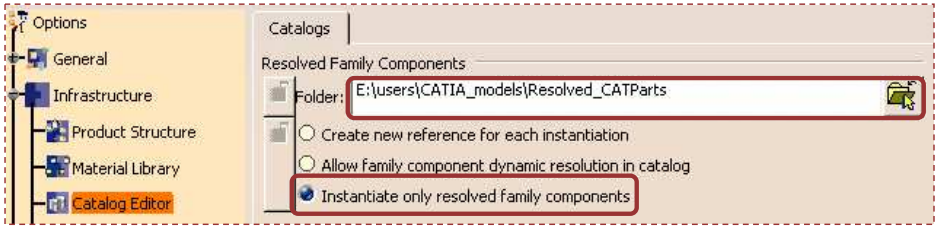
Student Notes:

Do It Yourself (2/3)

4. In a chapter, add a PartFamily. Name it “Generic Wheel Rim” and select the document containing the Wheel Rim. Click OK.

5. The family and its components are added to the catalog.

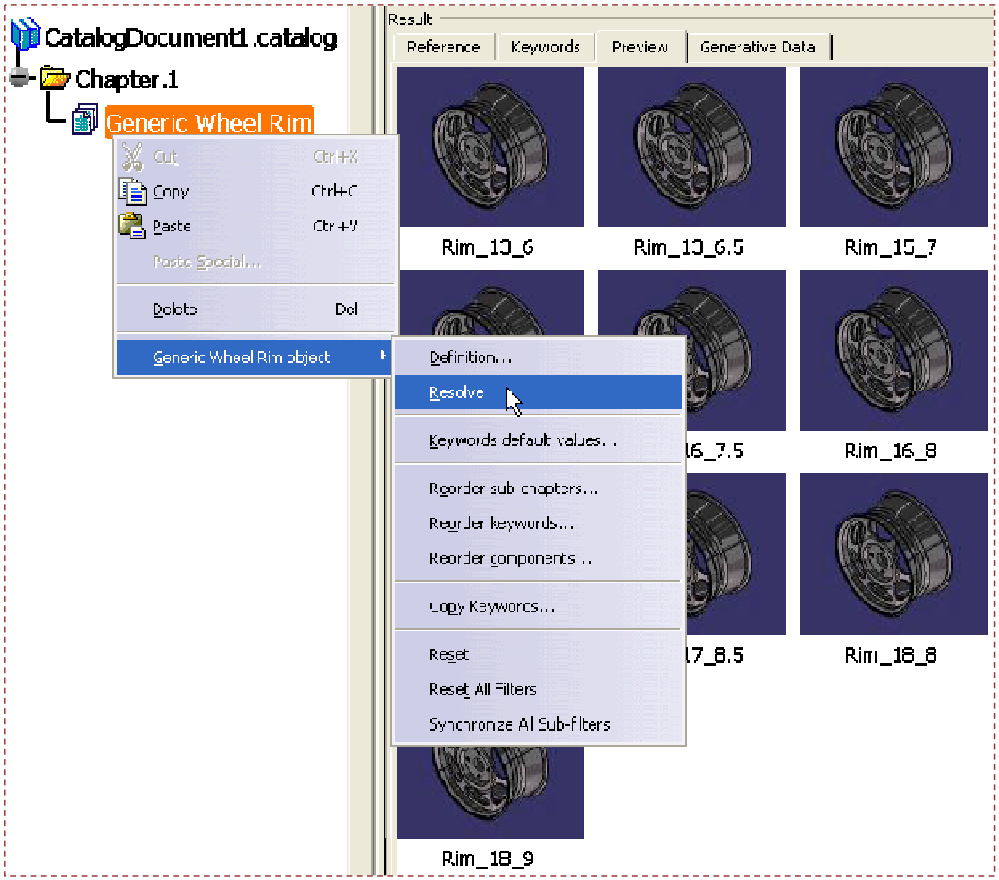
6. Check your Catalog Editor settings in Tools>Options.
Check the “Instantiate only resolved family components” option and choose the folder in which the resolved CATParts will be created.

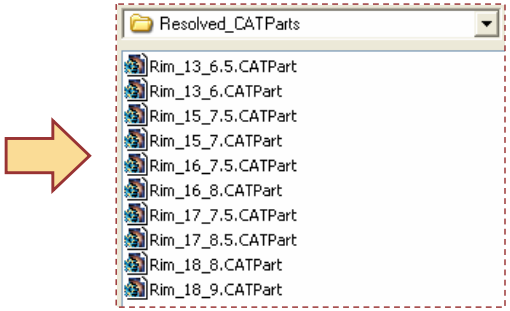


Student Notes:

Do It Yourself (3/3)

7. Open the contextual menu of the family to resolve it.
   A CATPart corresponding to each configuration of the initial design table is created in the directory you have indicated in the settings.





Student Notes:

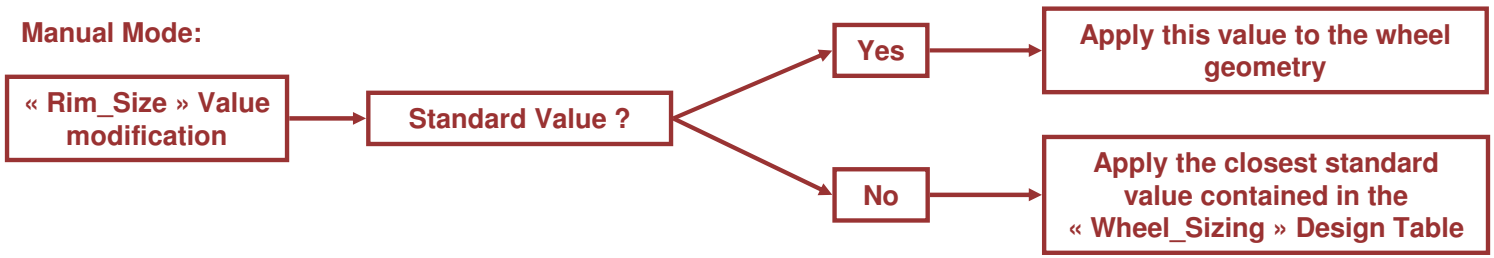
Wheel Rim

Step 8 - Create a Reaction to Control the Wheel Rim Diameter

10 min

In this step you will:

- **Create a Reaction to ensure that the current wheel radius is always one of the standard value contained in the "Wheel_Sizing" Design Table. Thanks to this reaction, this standard value will be the closest value from the one specified by the user while modifying the "Rim_Size" parameter value (see diagram below).**
- **Create a Rule to activate/de-activate some Relations depending on the wheel dimensions modification mode:**
  - **By changing the Design Table active configuration ("Design Table mode")**
  - **By changing individually the rim parameter values ("Manual Mode")**



Student Notes:

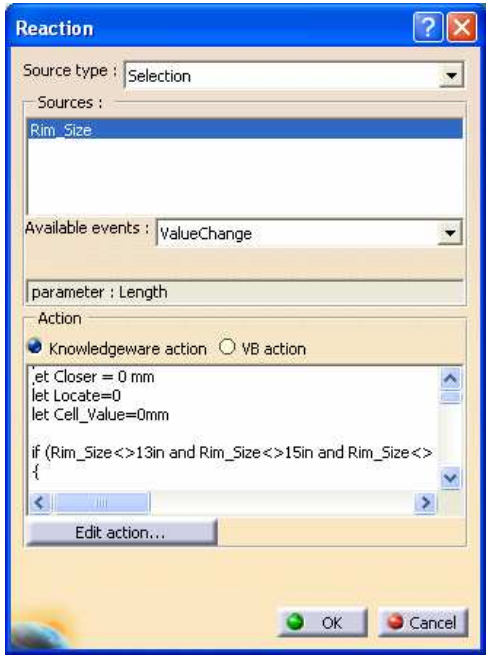
Do It Yourself (1/3)

1. **Create a new string User Parameter named Rim_Size_Driving_Mode with two values: "Design_Table_Mode" and "Manual_Mode"**

2. **In "Rules_and_Reactions" create a Reaction named "Closest_Std_Rim_Size".**
   This Reaction will react to the "Rim_Size" parameter value changes and will ensure that this parameter has a standard value contained in the "Wheel_Sizing" design table.

To be able to select the Rim_Size as the source parameter, you will have to deactivate the design table before defining the reaction.

3. **Select the Rim_Size parameter as the source and copy the script contained in "CATKWA_Wheel_Rim_Reaction.doc" in the Action editor. You will find this document in the same directory than the initial Wheel Rim CATPart.**



Student Notes:

Do It Yourself (2/3)

4. In “Rules_and_Reactions” create a Rule named “Driving_Mode”.
   This rule will deactivate either the Reaction or the “Wheel_Sizing” design table depending on the Rim_Size_Driving_Mode parameter value:

Closest_Std_Rim_Size Value	“Closest_Std_Rim_Size” Reaction Activity	“Wheel_Sizing” Design Table Activity
“Manual_Mode”	True	False
“Design_Table_Mode”	False	True

Student Notes:

Do It Yourself (3/3)

Open CATKWA_Wheel_Rim_End.CATPart to check your result.

5. Test the “Manual_Mode” driving mode:

   - Valuate the Rim_Size_Driving_Mode parameter to “Manual_Mode”
   - Modify the Rim_Size parameter value to 13in, 12.9in, 17.2in and 18.5in
   - Notice that when the specified value for the Rim_Size parameter is not a standard one, the Reaction forces the parameter value to the closest standard value

6. Test the “Design_Table_Mode” driving mode:

   - Valuate the Rim_Size_Driving_Mode parameter to “Design_Table_Mode”
   - Change the “Wheel_Sizing” Design Table active configuration