CATIA V5 Training
Foils

Knowledge Advisor

Version 5 Release 19
January 2009

EDU_CAT_EN_KWA_FI_V5R19

Instructor Notes:

About this course

Objectives of the course

Upon completion of this course you will be able to:

- Become familiar with the Knowledgeware working environment, how it can be accessed, the terminology that will be used and the Settings.

- Create a parametric part.

- Embed design knowledge in the part by creating rules, checks and reactions.

- Automate your designs modifications using various Knowledgeware Advisor tools.

Targeted audience

CATIA V5 Designers

Prerequisites

Students attending this course should have knowledge of CATIA V5 Basics

8 hours

Instructor Notes:

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Instructor Notes:

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Instructor Notes:

Knowledge Advisor Workbench Presentation

You will learn what are the main features of the Knowledge Advisor workbench as well as some infrastructure features provided with CATIA V5.

- Accessing the Workbench
- User Interface
- Knowledge User Settings

Instructor Notes:

Accessing the Workbench

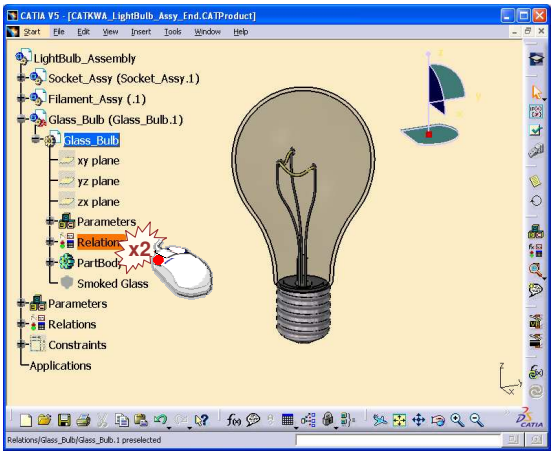
You can access the Knowledge Advisor workbench through the usual ways:

A **From the Start Menu**

B **From the workbench icon:**

Go to Tools /Customize /Start Menu to customize the content of this Welcome box

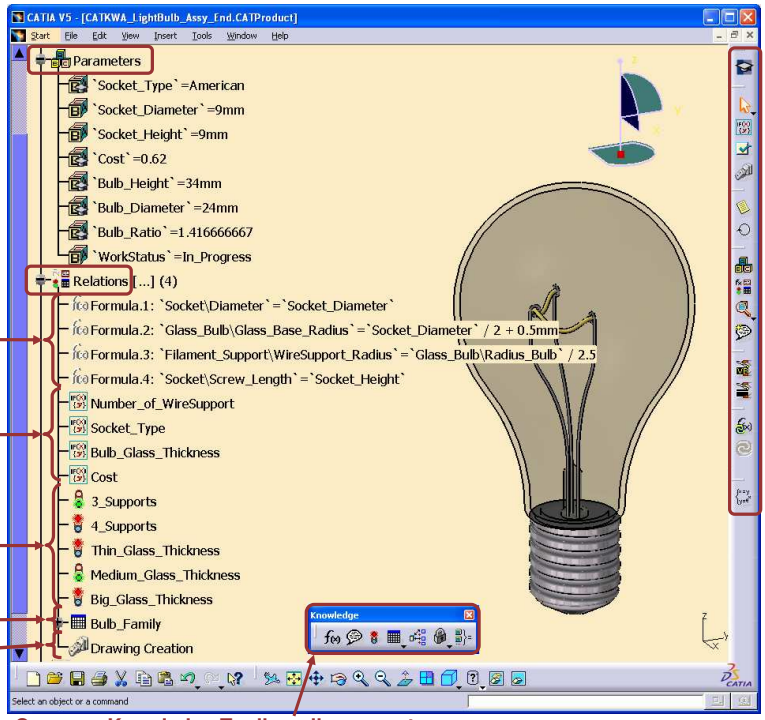
C **From a CATIA document**



If the Relations node exists in the specification tree, double-click on it to launch the Knowledge Advisor workbench.

Instructor Notes:

User Interface (1/4)



Parameters node contains User Parameters and Lists

Relations node contains:

- **Formulas**
- **Rules**
- **Checks**
- **Design Tables**
- **Reactions**

and Macro with arguments

Knowledge Advisor Workbench

Common Knowledge Toolbar allows you to access:
Formulas, Comments and URLs, Check Analysis, Design Table creation, Law creation, Knowledge Inspector, Lock/Unlock parameters, and Equivalent Dimensions

Instructor Notes:

User Interface (2/4)

Icon	Name	Definition
	Formula	Simple formulas y=f(x,y,z,…) between any V5 parameters
	Design Table	Tabulated relation of a set of parameters based on an Excel spreadsheet or a text file
	Law	y=f(x) mathematical law that can be used by geometric or analysis operators
	Knowledge Inspector	Allows to evaluate the impact of modifications (what if), and how to modify the parameters
	Lock selected parameters	Locks or unlocks the selected parameters
	EquivalentDimensions	Enables the user to apply the same value to selected Angle or Length parameters
	Comment and URLs	Searches for the URLs assigned to the user parameters or the relations

Instructor Notes:

User Interface (3/4)

Icon	Name	Definition
	Rule	Rule embedded in the design that reacts to parameter changes, and propagates parameter or geometric modifications
	Check	Check embedded in the design that reacts to parameter changes, and informs the user in case of violation
	Reaction	Feature embedded in the design that reacts to specific events, and propagates any kind of modifications
	List	List referencing a set of objects (parameters or geometric features). May compute list size, sum, min, max, etc…
	Loop	Loop similar to the loop in the languages that manage the creation, destruction or modification of a set of features. The loop is superseded by the powerful 'Knowledge Pattern' function of the Product Knowledge Template Workbench.
	Add Set of parameters	Creates a node of Parameters
	Add Set of Relations	Creates a node of Relations

Instructor Notes:

User Interface (4/4)

Icon	Name	Definition
	Parameters Explorer	Creates the user parameters stored at the feature level
	Add parameters on geometry	Adds parameters to an edge, a face or a vertex
	Comment and URLs	Adds URLs on user parameters or relations and searches for the existing URLs
	Macros with arguments	Feature to run VBScript macros with arguments. Can be called from a Rule or a Reaction
	Action	Feature that describes a function that a user can decide to execute
	Measure Update	Updates relations using measures
	Set of Equations	Mathematical set of equations and inequations that drives a set of output parameters, according to the changes in the input parameters

Instructor Notes:

Knowledge User Settings (1/5)

Display and update General Settings:

Check the corresponding option if you need:

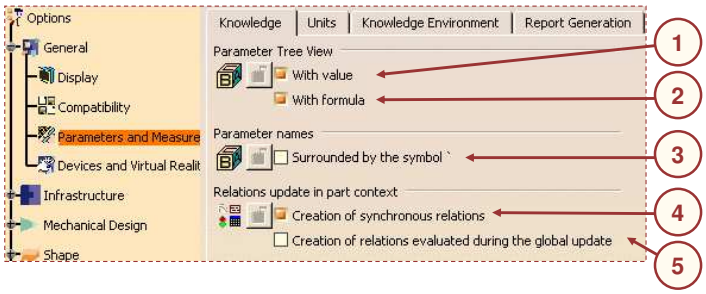
(1) The **value** of the parameter to appear in the tree.

(2) The **formula** driving the parameter to appear in the tree beside the parameter.

(3) To work with non-latin characters. Otherwise, the parameters have to be renamed in latin characters when used.

(4) To **create synchronous relations**; relations that will be immediately updated if one of their parameters is modified. Relations based on the parameters are the only ones that can be synchronous.

(5) To associate the evaluations of **asynchronous relations with the global update**. The relations can be asynchronous for two reasons: when the user wants the relations to be asynchronous or when the relation contains measures.



Instructor Notes:

Knowledge User Settings (2/5)

Design Tables General Settings:

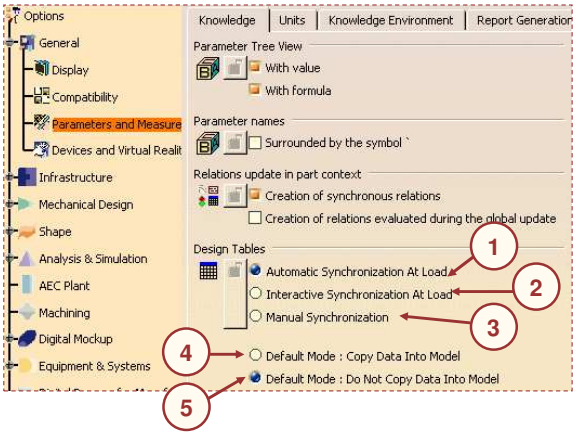
(1) **Automatic Synchronization At Load:** While loading a model containing the user design tables, if the design table files have been modified, and if the external file data is contained in the model, the design table will be synchronized automatically if this button is checked.

(2) **Interactive Synchronization At Load:** While loading a model containing the user design tables whose external source file was deleted, this option enables the user to select a new source file or to save the data contained in the design tables in a new file.

(3) **Manual Synchronization:** While loading a model containing the user design tables, if the design table files have been modified and the external file data is contained in the model, the design table will be synchronized if this radio button is checked. To synchronize both the files, right-click the design table in the specification tree and select the DesignTable object->Synchronize command or the Edit->Links command.

(4) **Default Mode: Copy Data Into Model:** If checked, the data contained in the external source file will be copied into the model.

(5) **Default Mode: Do Not Copy Data Into Model:** If checked, the data contained in the external source file will not be copied into the model.



Instructor Notes:

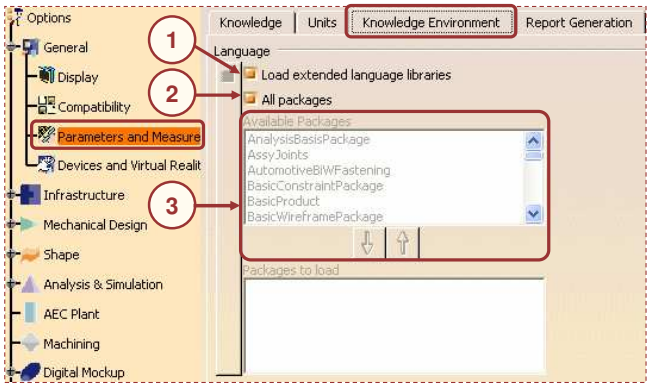
Knowledge User Settings (3/5)

Language Settings:

(1) Check this option to have access to more language libraries. That means more functions will be available for the Edition of Relations.

(2) Check this button to load ALL the available libraries.

(3) Otherwise, select the libraries packages in the list and use the arrows to add or retrieve them to the list of libraries to be loaded.



Instructor Notes:

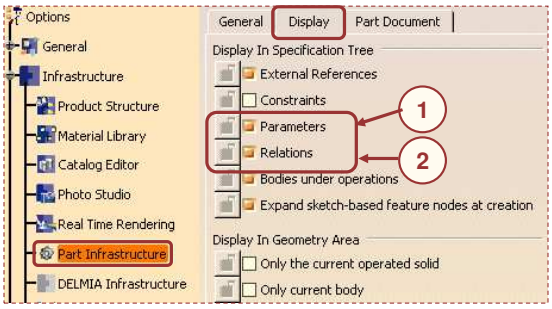
Knowledge User Settings (4/5)

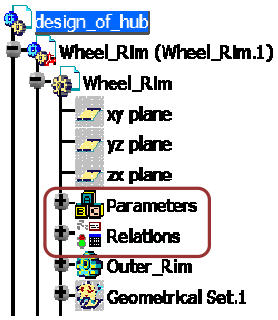
Part Infrastructure Settings:

Check the corresponding options if you need :

(1) The parameters of the part to be displayed in the specification tree.

(2) The relations of the part to be displayed in the specification tree.





Instructor Notes:

Knowledge User Settings (5/5)

Product Structure Settings:

Activate the following options if you need :

(1) The parameters of the product to appear in the specification tree.

(2) The relations of the product to appear in the specification tree.

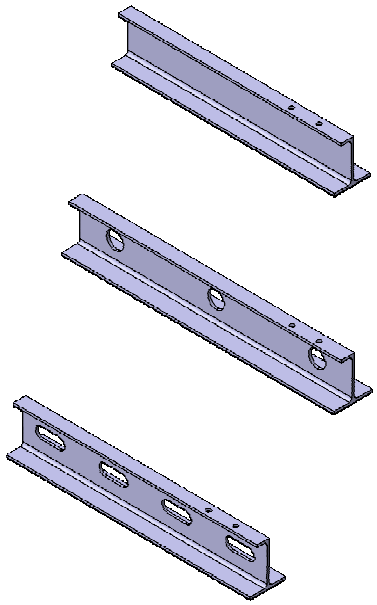
Instructor Notes:

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**

Instructor Notes:

Creating Parameters, Formulas and Lists

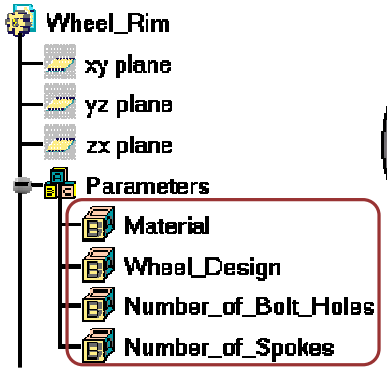
- Creating User Parameters
- Creating and Using Formulas
- Creating Lists
- Associating URLs to Parameters and Relations

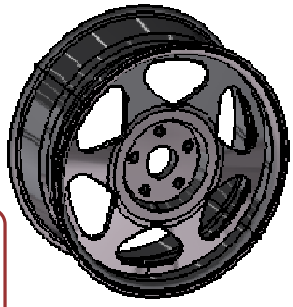
Instructor Notes:

Knowledge Advisor

Creating User Parameters

You will learn how to create and manage parameters.





Instructor Notes:

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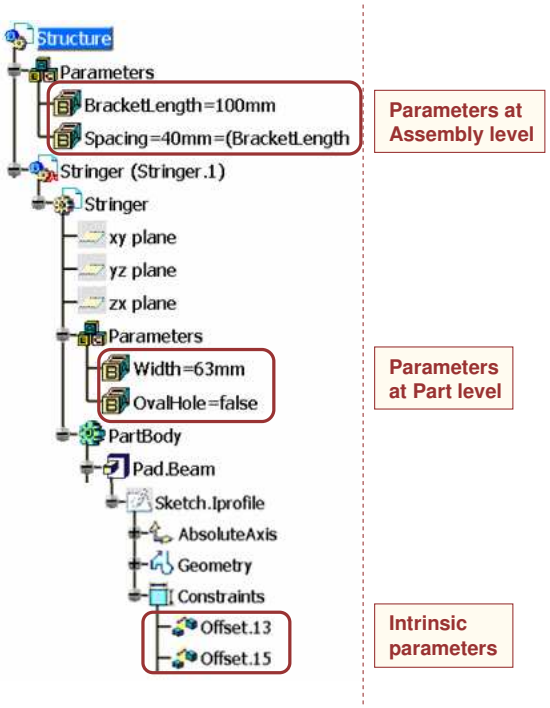
What are Parameters? (1/2)

- There are many types of parameters:
  Real, Integer, String, Boolean, Length, Mass...
- Two kinds of parameters:
  - Intrinsic Parameters are generated while creating any geometry and features. They define the intrinsic properties of the features (depth, offset, activity, …)
  - User Parameters are especially created by the user. They define the extra pieces of information added to a document. The User Parameters can be defined at different levels:
    - Part level
    - Assembly level
    - Feature level
- User Parameters can either be defined:
  - With a single value (continuous). In this case, the parameter can take any value.
  - Or with multiple values (discrete). In this case, the parameter can take only the pre-defined values given at its creation.
- Any parameter can be:
  - Defined or constrained by relations
  - Used as argument of relations



Instructor Notes:

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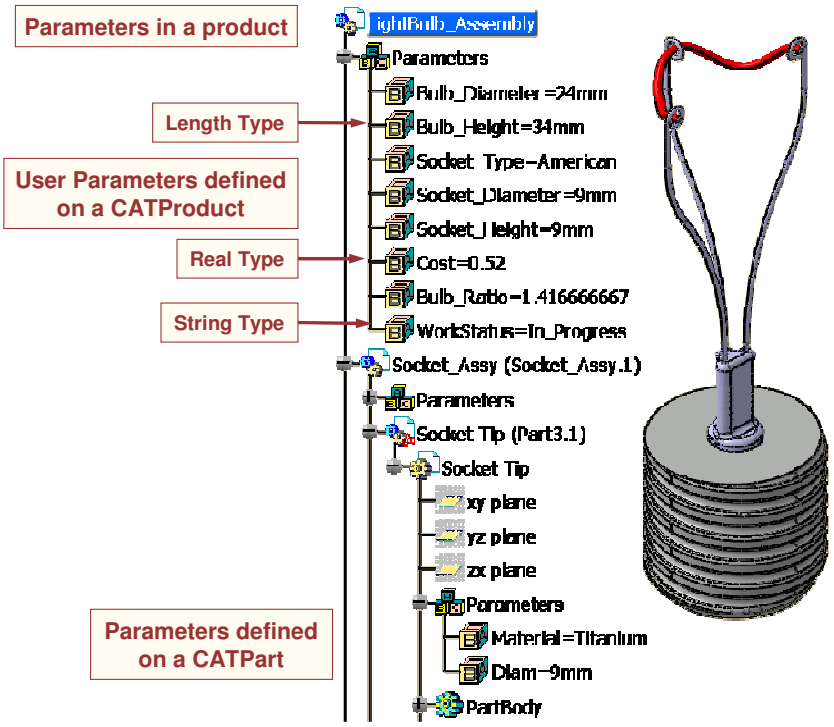
What are Parameters? (2/2)



Instructor Notes:

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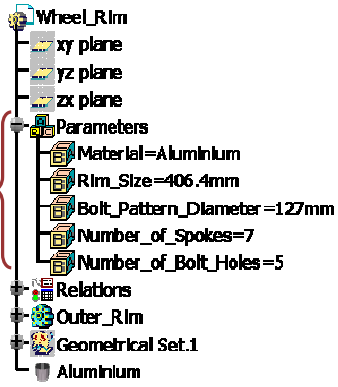
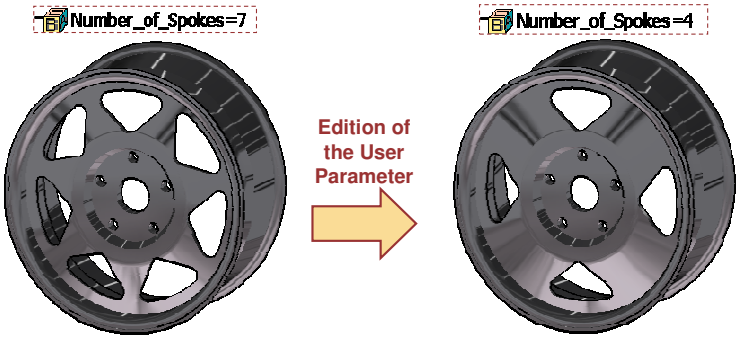
Why Use User Parameters?

- To have an immediate access to the parameters that pilot the geometry and to change easily their value.
- To centralize key information so that any new user on the model can use it immediately.
- To refer easily to the same parameter when editing relations.
- With User Parameters, you can create generic models that are driven only from the User Parameter node.

All the key information of the model is accessible from this place of the part, so that you do not need to search in the PartBody to change the number of spokes, for instance.



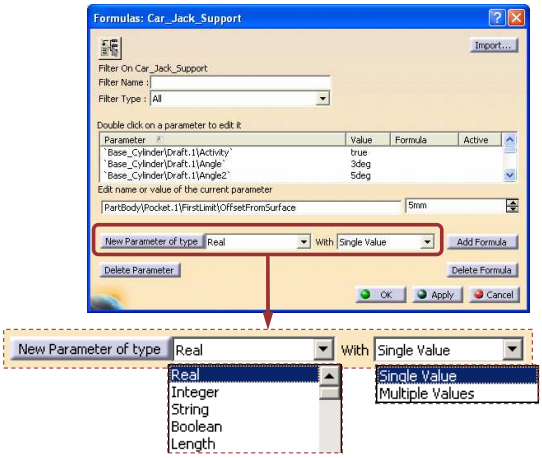


Instructor Notes:

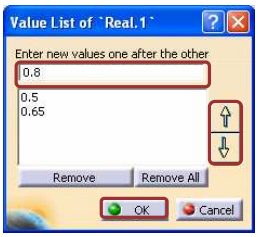
Creating User Parameters (1/2)

1. Click on the *f(x)* icon.
   The Formulas panel is displayed.

2. Select the desired type of parameter and then specify the Single Value or the Multi Values option.
3. Click the New Parameter of type button to create the parameter.



The Multiple Values option allows you to pre-define fixed values for the parameter.
In this case, you are required to enter the values of the parameter as soon as you click the New Parameter of type button. The "Value List" panel appears.



Type here the different values of the parameter. Click the Enter button to validate each value.

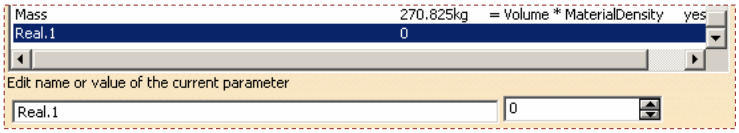
Use these arrows to reorder the values.

Click OK when finished.

Instructor Notes:

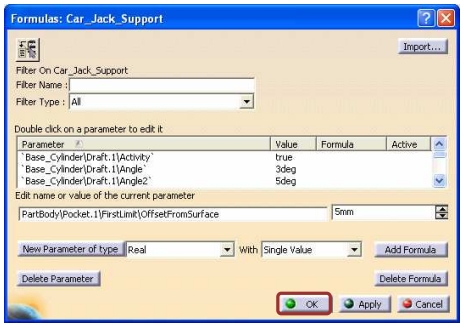
Creating User Parameters (2/2)

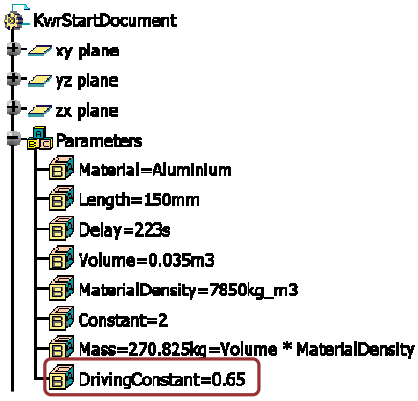
4 The new parameter appears at the end of the parameters list with a default name (here *Real.1*) and a default value *0*.



5 You can rename the parameter by typing a new name in the Edit name field; and attribute it a value by filling the Edit value field.

6 The OK button validates the creation of the parameter and closes the Formulas panel.
The new User Parameter is added to the specification tree.





Instructor Notes:

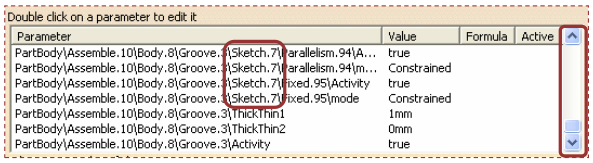
Filtering Parameters (1/2)

The Formulas panel as well as many Editor panels, in which you may use the parameters, allow you to filter parameters in order to ease their selection.

1. When the selection panel is opened, first select your selection mode: incremental or not.

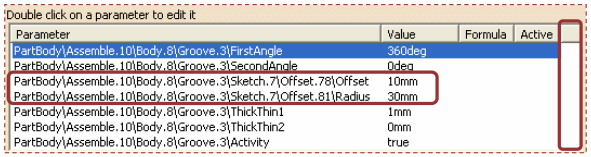
2. Then select in the specification tree the feature that contains the parameters that you want to use.

With the incremental mode unchecked, ALL the parameters of the Groove and ALL those of its definition sketch are displayed.



lots of parameters are displayed: activities, modes, etc.

With the incremental mode checked, the parameters of the Groove and ONLY the dimension parameters of its definition sketch are displayed.

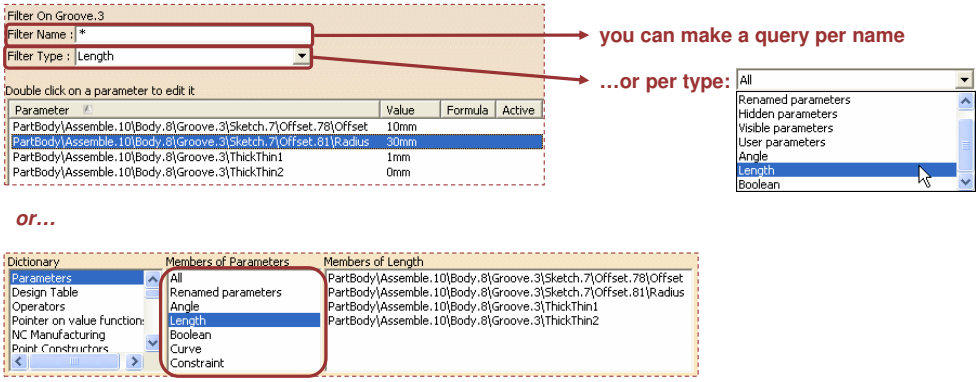


fewer parameters are displayed: only 7 where found for Groove.3

Instructor Notes:

Filtering Parameters (2/2)

3 If you still have too many parameters listed, you can use filters: you usually have the possibility to filter the parameters by types and by name.



or...

select a type in the list above

Types available in the "Filter Type" list are the types of parameters found in the current selection.

4 You should now be able to select a parameter easily.

Instructor Notes:

Adding Sets of Parameters

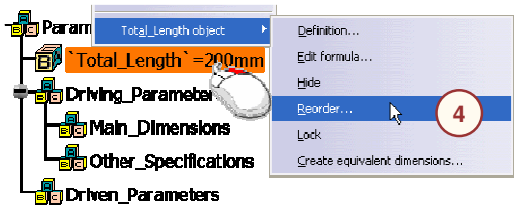
In the specification tree, you can create sets of parameters under the Parameters node in order to regroup the parameters by categories.

1. In the Knowledge Advisor workbench, click on **Add Set of Parameters** icon.

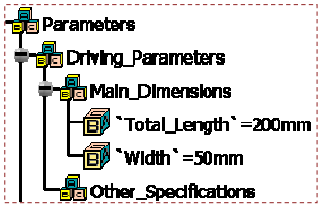
2. Select in the tree the Parameters node or an existing set of parameters under which the new Set of Parameters will be created.

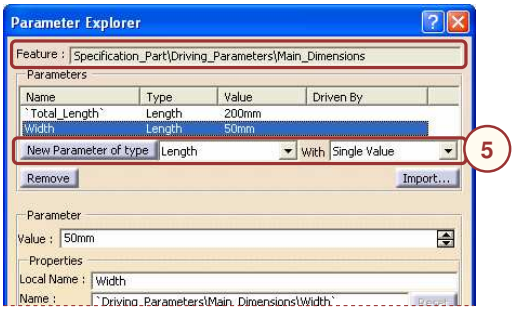
3. You can rename the Set of Parameters by editing its **Properties** (in the contextual menu).

4. You can reorder the already existing user parameters using the **Reorder** option of the parameter contextual menu. Select a Set of Parameters to place the parameter in it.



5. To create a new user parameter directly in a specific Set of Parameters, you have to use the **Parameter Explorer**. Select a Set of Parameters before clicking the New Parameter of type button.



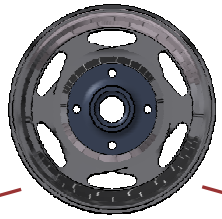


Instructor Notes:

Why Publish Parameters?

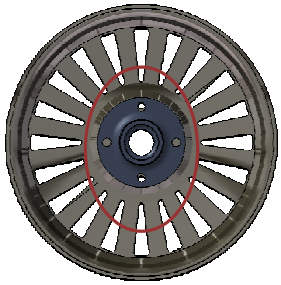
- Publication of parameters is useful when replacing in an assembly a component which contains parameters that drive other component's external parameters.
- If the exported parameters are published and if the parameters of the replacing component are published under the same name, they will inherit the control of the exported parameters.
- Otherwise, the parameters of the replaced component will keep the control.

In this example, the hub is linked to the rim: the hub reuses the number of holes and the pattern diameter of the rim.
Let us see the difference in the behavior of the hub when replacing the rim, with its parameters published or not.



The rim is replaced by a bigger one, the parameters of which are not published.

The rim is replaced by a bigger one, the parameters of which are published under the same names than the first rim.



The external parameters of the hub are still linked to the first rim. They are not updated.

The number of holes of the hub and the diameter of the pattern automatically adapt to the new rim.

Instructor Notes:

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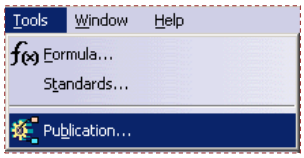
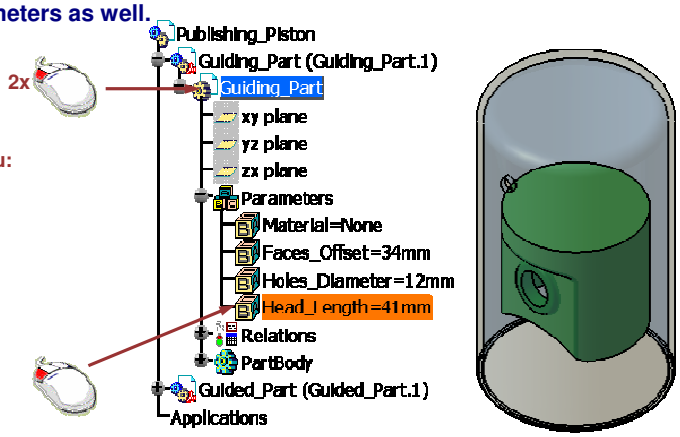
Publishing a Parameter (1/3) P2

The Publication command is available in Assembly Design and Part Design.
It publishes the geometry and the parameters as well.

1. Activate the part containing the parameter you want to publish.

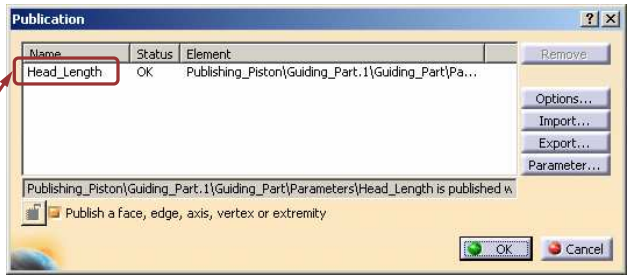
2. Select Publication... in the Tools menu:





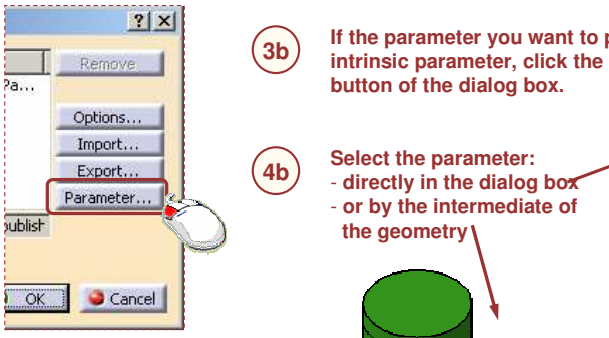
3a. If the parameter you want to publish is a user parameter, click on its icon in the tree.

4a. The user parameter now appears in the list of published elements of the Publication dialog box.



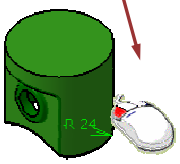
Instructor Notes:

Publishing a Parameter (2/3)



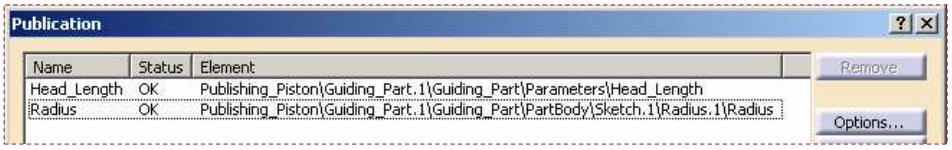
3b If the parameter you want to publish is an intrinsic parameter, click the Parameter button of the dialog box.

4b Select the parameter:
- directly in the dialog box
- or by the intermediate of the geometry



5b Click OK to validate the selection.

6b The intrinsic parameter appears in the list of published parameters:



Instructor Notes:

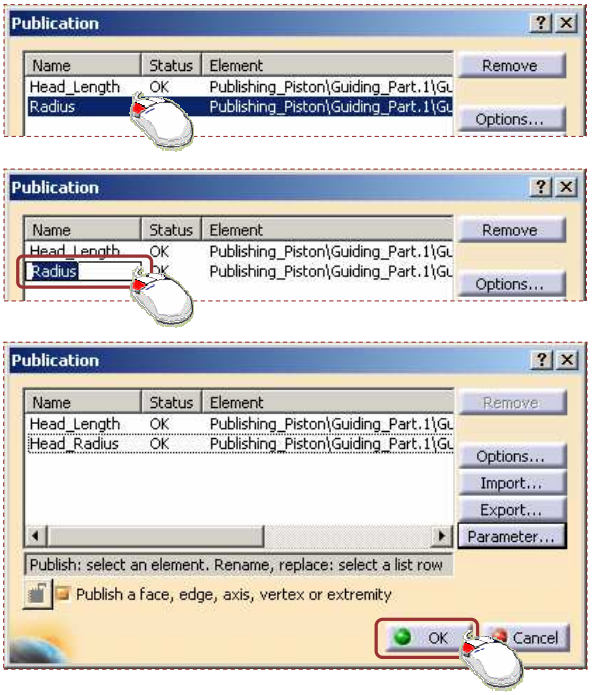
Publishing a Parameter (3/3)

7 Published Parameters appear in the list with a default publication name.

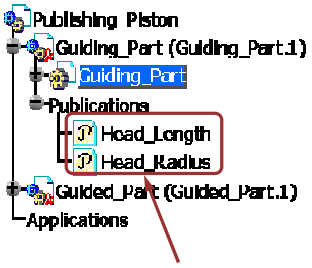
8 To modify the publication name, first select the publication.

9 Then, select the name field.

10 Edit the name and validate with Enter.



11 Validate the publication by clicking OK.



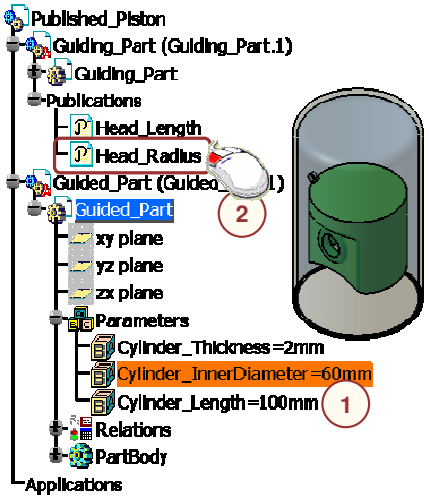
12 Your newly published parameters appear under the publications node of the active part.

Instructor Notes:

Using Published Parameters (1/5)

Published parameters are called while editing formulas.

In this example, we are going to make equal the inner cylinder diameter to the head diameter.



1. Be activated on Guided_Part level and open the formula editor panel of Cylinder_InnerDiameter parameter.

2. Edit the formula by selecting the Head_Radius parameter:
   - Under the Publications' node of Guiding_Part
   - In the External Parameters of Guided_Part, provided that it has previously been copied with link. The copy with link is already made if you have used this external parameter before, or if you have intentionally copied/pasted it Special as result with link.

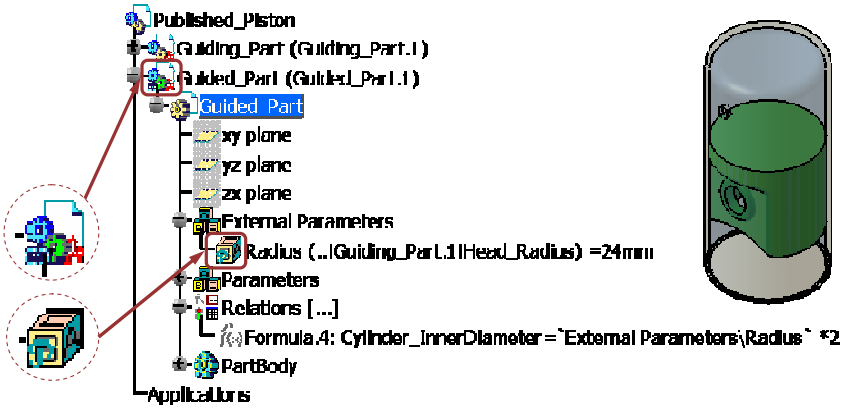
Instructor Notes:

Using Published Parameters (2/5)

Published parameters are called while editing formulas.

3a The edited part has become contextual.

3b The External Parameters linked to the published parameters appear with a green Capital P on their icon in the tree.

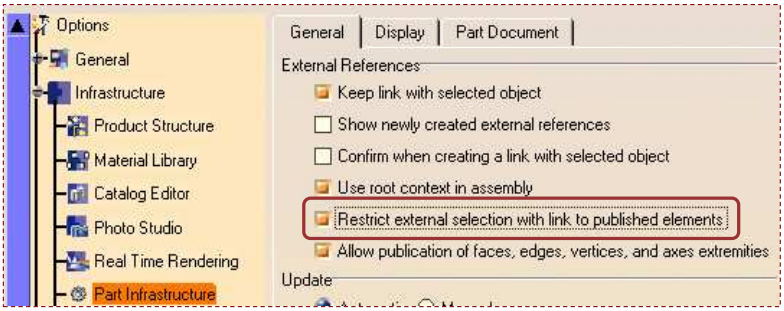


Instructor Notes:

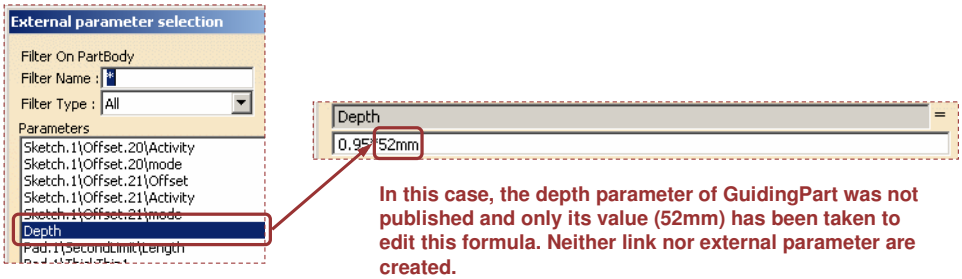
Using Published Parameters (3/5)

Some CATIA options can prevent the user from creating external parameters from the unpublished parameters.

1. **The setting preventing the use of non published geometry also works with parameters.**



2. **When this option "Restrict External selection…" is activated, and when you select an unpublished parameter in an external document, no external parameter is created and no link is kept: only the value of the parameter will be taken (as if the option 'Keep link…' was deactivated).**

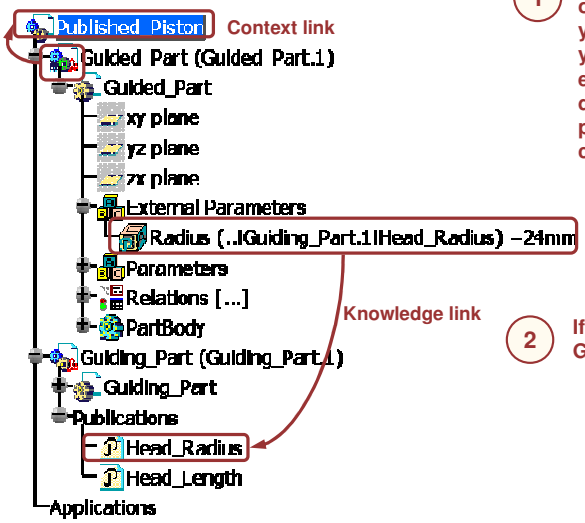


In this case, the depth parameter of GuidingPart was not published and only its value (52mm) has been taken to edit this formula. Neither link nor external parameter are created.

Instructor Notes:

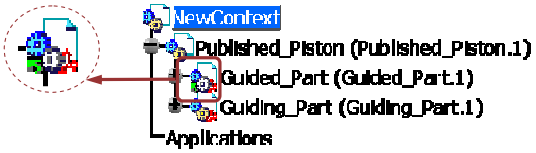
Using Published Parameters (4/5)

While using the published parameters you have to pay attention to the context assembly.



1 The first time you use an external reference or a published external parameter, not only you create links to external information, but you also define a "context" link from the edited part to the root assembly (by default). The context link is unique and the product it is connected to is called the context assembly.

2 If the root product is not anymore the context product of Guided_Part, its icon indicates it is out of context.

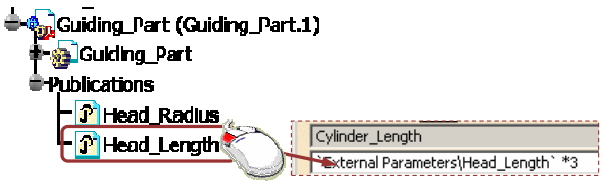


Instructor Notes:

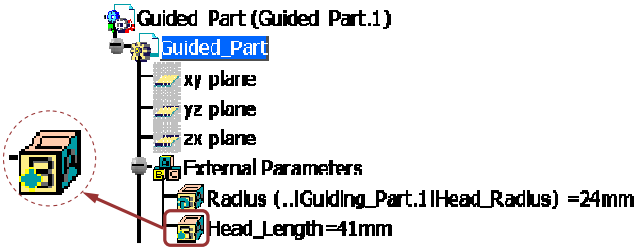
Using Published Parameters (5/5)

While using the published parameters you have to pay attention to the context assembly.

3 In this new context, try to create, in the Guided_Part, a new formula referring to another published parameter of the Guiding_Part.



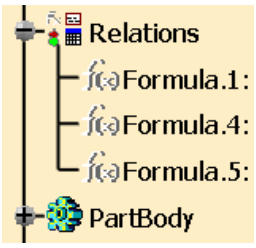
4 An external parameter which is created when the root product is not the context product will never be considered as connected to a published parameter.



Instructor Notes:

Creating and Using Formulas

You will learn how to create and use Formulas.

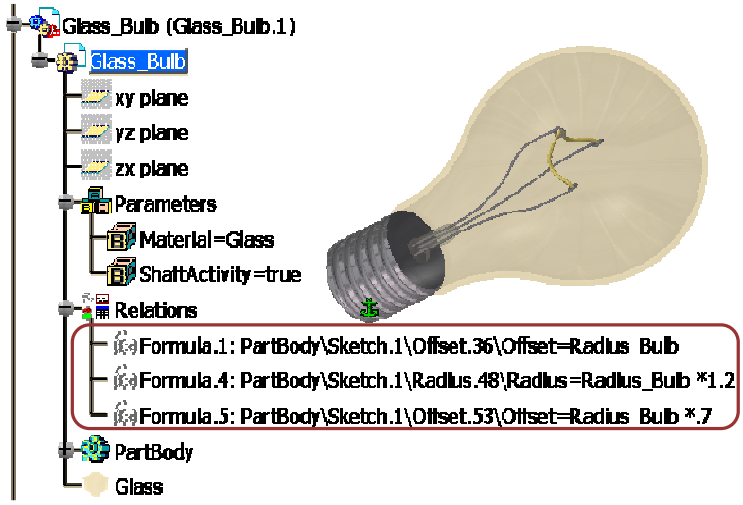


Instructor Notes:

What are Formulas?

- Formulas are relations used to define or constrain any parameter.
- Formula can be defined with parameters, operators, and functions.
- A Formula is created from the moment you attribute a user parameter to a feature, for example.
- The left part of the relation is the parameter to constrain and the right part is a statement.

- Once it has been created, a Formula can be manipulated like any other feature from its contextual menu.



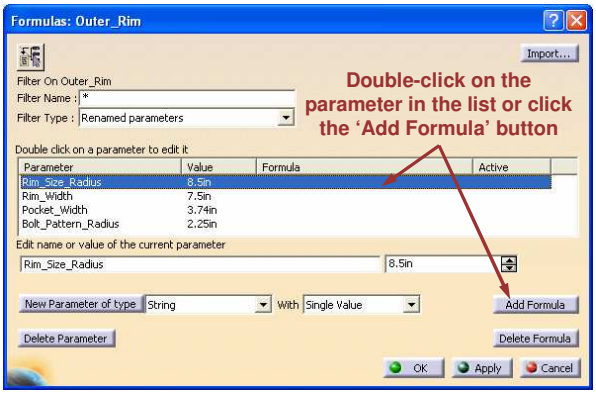
Instructor Notes:

Creating a Formula (1/2)

You can create Formulas with 'dimensions' or User Parameters.

1 **You can access the Formula Editor through different means:**

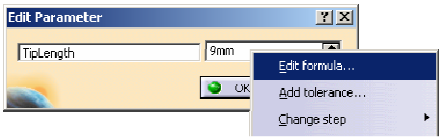
- **Click on the f(x) icon; in the Formulas panel, use the filter to select the parameter you want to edit. Either double-click on this parameter or click the Add Formula button.**



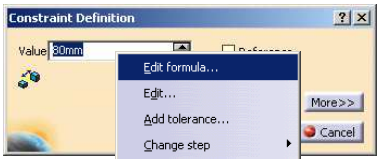
Double-click on the parameter in the list or click the 'Add Formula' button

OR

- **In the specification tree, double-click on the parameter or on the dimension you want to add a formula to. Right-click in the Value field and select 'Edit formula' in the contextual menu.**



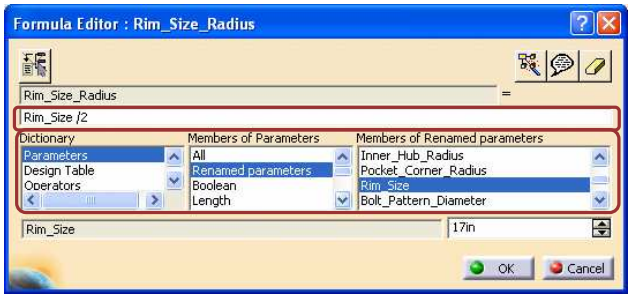
or…



Instructor Notes:

Creating a Formula (2/2)

2 The Formula Editor panel appears.
Enter the right side of the formula in the formula editor field.



Enter the formula here

Use the dictionary to select a parameter or a function


Check the Incremental mode button in order to display in the dictionary only the parameters of the feature selected in the specifications tree or in the 3D. If this option is not checked, will be displayed not only the parameters of the selected feature but also those of the features under it.


Click to open the language browser panel (see specific slides).


Click to attach a URL or a comment to the formula.


Click on the Eraser to delete all the contents of the formula field.

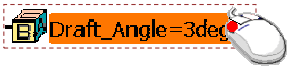
3 Click OK to validate the creation of the formula.
The Formula is added to the Relations node in the specification tree.

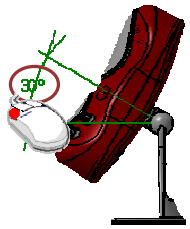
Instructor Notes:

Selecting Parameters in the Formula Editor

While creating the parametric models you often have to select a parameter to use it in a statement, in a design table, or simply to edit it. Here are different ways of selection.

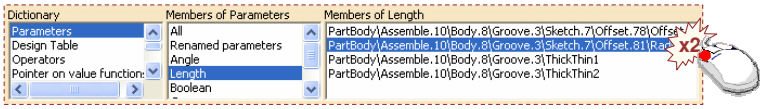
A If the parameter is displayed in the specification tree click on it.

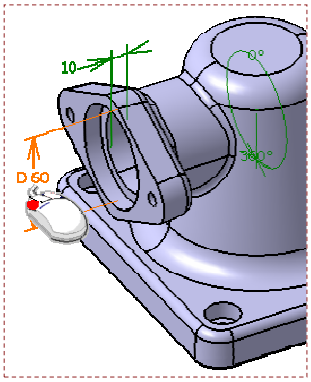




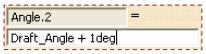
B If the parameter is displayed in the 3D (assembly constraint for instance) you can also click on it in the 3D.

C If you are using the Parameters Dictionary, you can either double-click on it in the list or click once on it in the 3D.





D If you know the exact name of the parameter you can also type it.



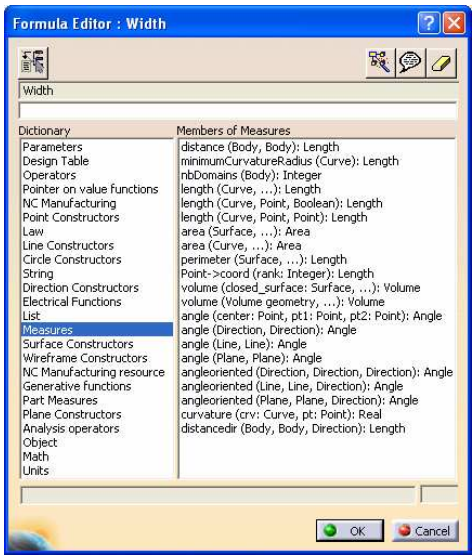
Instructor Notes:

Using Measure Functions in Formulas (1/3)

When you are editing a formula, you have the possibility to use pre-defined functions, especially measures. The functions allow you to capture values from the geometry.

For instance, the functions of the Measures dictionary allow you to define a parameter as:

- **A distance between two points**
- **The minimum radius of a curve**
- **The total length of a curve**
- **The length of a curve segment**
- **The area of a surface or a sketch**
- **The perimeter of a surface**
- **The volume of a PartBody or a closed surface**
- **An angle, oriented or not, between two lines, directions, or planes**



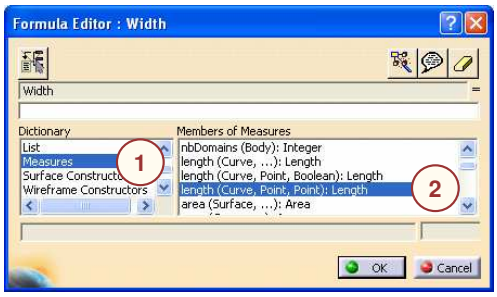
To make sure that you have access to all these functions, check that the Load extended language libraries option is selected in the Knowledge tab of General settings (Tools>Options).

Instructor Notes:

Using Measure Functions in Formulas (2/3)

1. In the Formula Editor panel, select the Measures item from the dictionary list.

2. The list of measures functions appears. Select for example the ***length(Curve,Point,Boolean)*** item by double-clicking on it.



Instructor Notes:

Using Measure Functions in Formulas (3/3)

3 The length function is added to the Formula Editor.

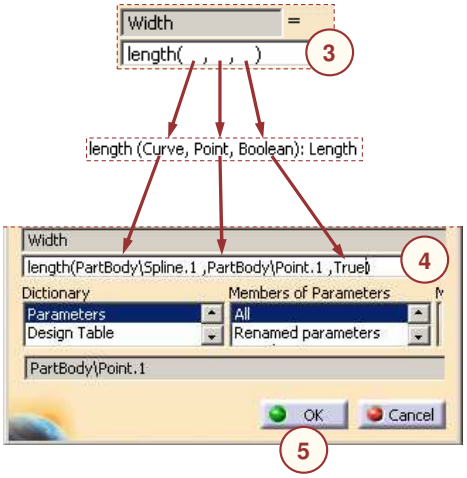
4 You now need to fill the arguments of the function. The function description informs you of the nature of the arguments.

For each argument, check that the cursor is positioned where the argument is intended to be typed, and then select the corresponding feature in the tree.

Of course, when the argument is an Integer or a Boolean, you can just type it. In our example, the third argument is a Boolean: type 'True' if the length is to be calculated from the origin, and 'False' if the length is to be calculated from the curve end.

5 Validate by clicking OK.

6 CATIA may ask you if you want the relation to be updated automatically with global update. We advise you to answer 'Yes'.



Instructor Notes:

Using the Language Browser

1. Open the Language browser panel by clicking on the following Action button in a Relation Editor.

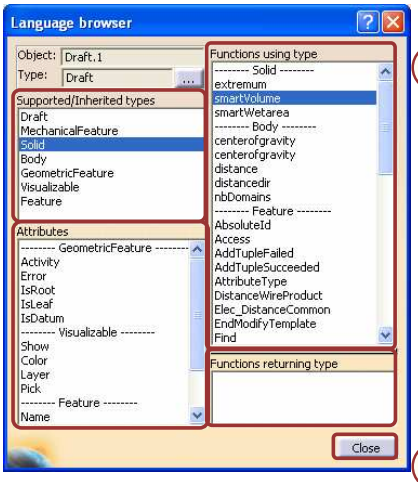
2. Select a feature in the tree or in the geometry. Its type is indicated in the Type field. You can also choose a type manually using the ... button.

3. The **Supported/Inherited types** field provides you with a list of the types supported by the selected type, and of the types that the selected type inherits from.
Double-click on the type to have it automatically declared in your relation.

Let Solid1 (Solid)

4. The **Attributes** field lists the possible attributes of the selected type, and of the supported and inherited types.
Double-click on an Attribute to have it filled in your relation.

.Activity



5. The **Functions using type** field lists the functions and methods whose first argument is a type of the Supported/Inherited types list.
The **Functions returning type** field lists the functions and methods returning the selected type.
Double-click on a function to have it added to your relation.

smartVolume()

6. Click Close to close the panel.

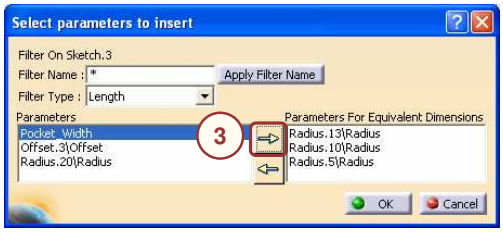
Instructor Notes:

Equivalent Dimensions Feature

The Equivalent Dimensions feature helps you to define an equality between a set of Angle or Length parameters. Its value can be modified through the editor and is propagated to all the parameters belonging to the equivalence.
This feature increases the designer's productivity and also decreases the model size.

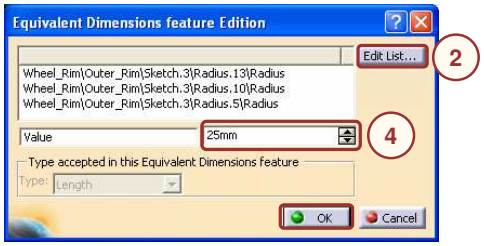
1. Click on the Equivalent Dimensions icon in the common Knowledge Toolbar. The Equivalent Dimensions Edition window displays.

2. Click the Edit List... button. A panel displays for you to select the equivalent parameters.

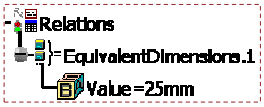


3. Select in the list the parameters that will have the same value and use the right arrow button to add them to the Equivalent Dimension feature. Click OK when all the parameters are selected.

4. Back in the Equivalent Dimensions Edition panel, check the value of the equality before validating by clicking OK.



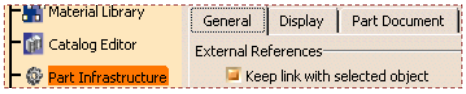
5. The Equivalent Dimensions feature is displayed in the Relations node.
Double-click on it to view the list of parameters, modify it or change the value.



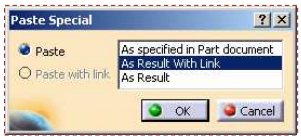
Instructor Notes:

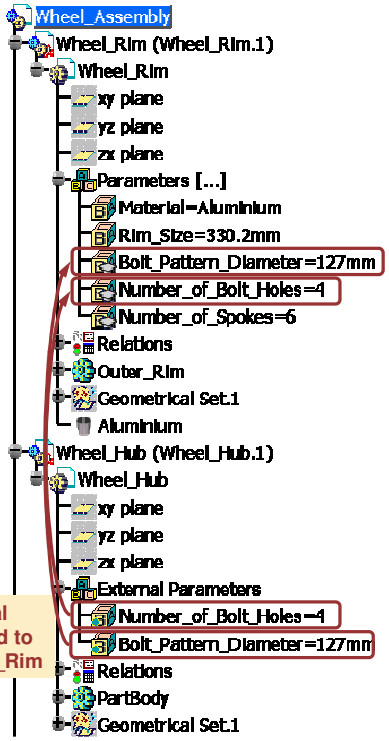
What is an External Parameter?

- External Parameters are linked copies of parameters driven in an external document.
- It is possible to create them provided that the 'Keep Link with selected object' in the Tools > Option menu is activated.



- They can be created:
  - Automatically by referring to another part's parameter in a relation
  - Manually by using the Copy/ Paste Special – As Result With Link command





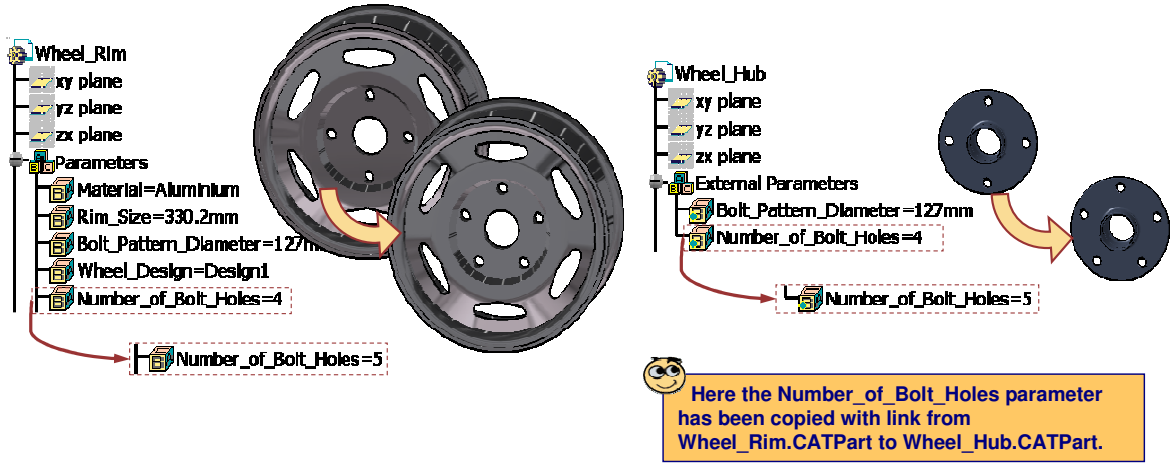
These two External Parameters are linked to their fathers in Wheel_Rim

Instructor Notes:

Why Use External Parameters?

- To reuse a parameter that drives a Part into another Part in order to link their geometries.
- To be sure that the design of the two linked parts is consistent.
- To avoid manual update of all the parameters that must have the same value in different parts.

In this example, the hub needs to adapt to the holes of the rims. External parameters have been created in order to link the number of holes and the bolt pattern diameter.



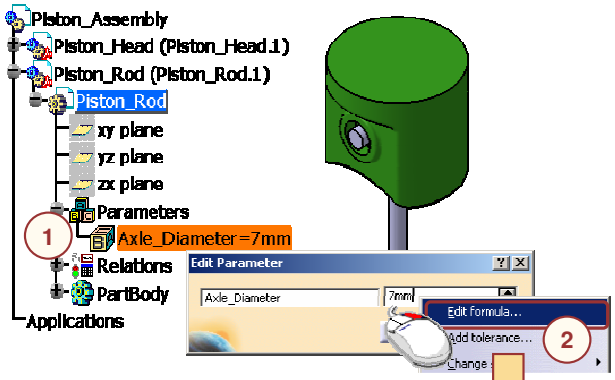
Here the Number_of_Bolt_Holes parameter has been copied with link from Wheel_Rim.CATPart to Wheel_Hub.CATPart.

Instructor Notes:

Referring to External Parameters in Formulas (1/2)

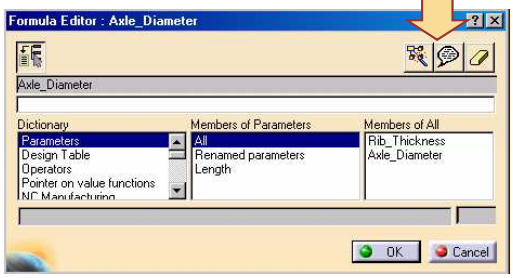
In a Formula, you can use the parameters defined in the external documents.
This is possible between any type of document.

The following Assembly contains two Parts.



1. In the specification tree, double-click on the user parameter Axle_Diameter in order to edit it.

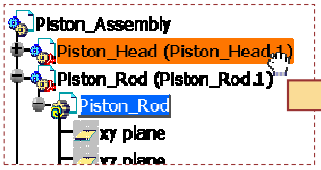
2. In the contextual menu of the parameter's value, select the *Edit formula* option.
The Formula Editor panel is displayed.

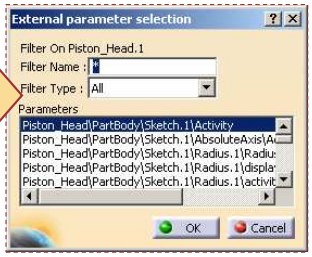


Instructor Notes:

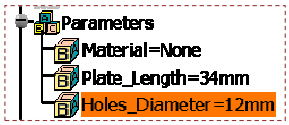
Referring to External Parameters in Formulas (2/2)

3 Select the second instance (Piston_Head).
The *External parameter selection* panel is displayed.



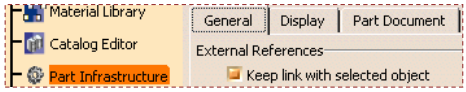


Remark:
The *External parameter selection* panel is mainly used to select intrinsic parameters. In the case of user parameters, it is possible to directly select the parameter in the tree.

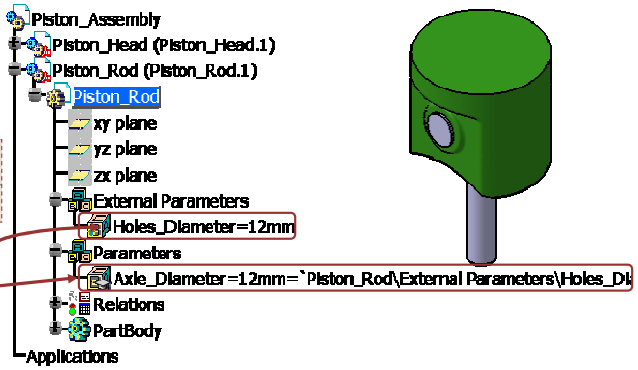


4 Select in the tree the user parameter Holes_Diameter.
Validate by clicking OK in the *External parameter selection*, in *Formula editor*, and in the *Edit Parameter* dialog box.

5 Provided this option was activated,



an external parameter has been created in the Piston_Rod.CATPart and is used in the newly created formula.



Instructor Notes:

Creating Lists

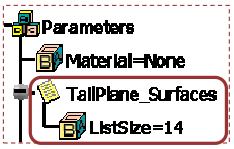
You will learn how to create lists. List features can be used to manage lists of objects or parameters.

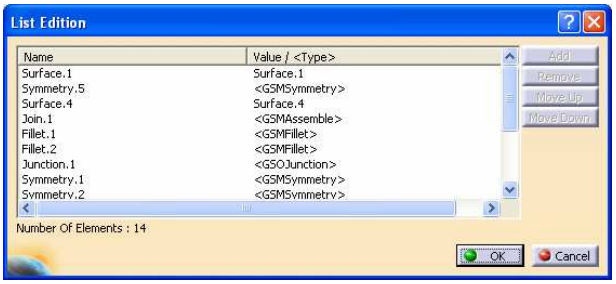
Instructor Notes:

What are Lists?

- The Knowledgeware List features are lists of ordered features or parameters.
- A list can be populated either automatically or manually.
- The items of a list can be reordered either manually or throughout functions.
- The list features are stored under the Parameters node of the specification tree and are integrated in the update mechanism.
- A ListSize integer parameter indicates the number of items that populate the list. It is computed automatically.

- Lists can be used:
  - To make a sum of parameters easily
  - To count the number of features of a given type in a document and then to calculate a cost
  - To create loops in reactions features or in loops features





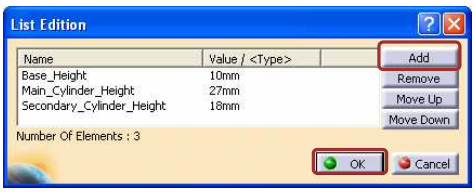
Instructor Notes:

Creating a List

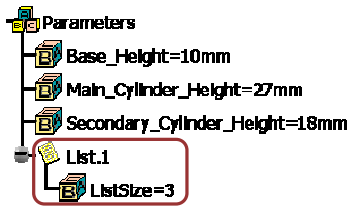
1. In the Knowledge Advisor Workbench, select the List icon.
   The List Edition panel appears.

2. Select some parameters or features in the tree and click the Add button to add them to the list.

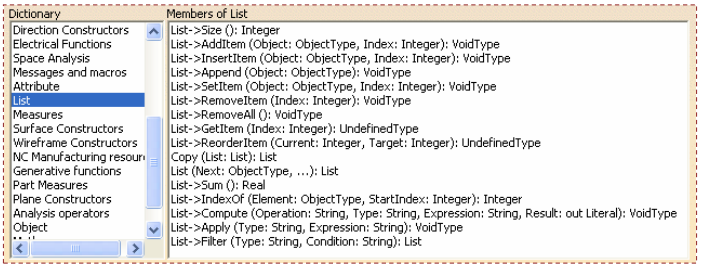
3. Validate List creation by clicking OK.



4. The List appears under the Parameters node in the tree, and a ListSize parameter is automatically created and indicates the number of items in the list.
   You can rename the List using its Properties.



The list feature can be manipulated through specific functions to:
- Add and remove elements to the list
- Get an element
- Retrieve values from the list
- Move elements of the list to another position
- Copy the content of a list into another one



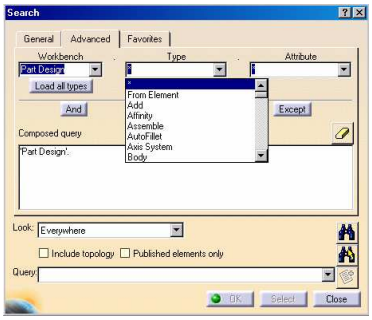
Instructor Notes:

What is Populating a List Using a Query?

- Using the Query function you can automatically populate a List whith features that verify a specified expression.
- In the example below, the result of the search will return the holes of the PartBody whose diameters are greater than 10mm:

  Example: List.1=PartBody.Query("Hole","x.Diameter>10mm")
  Where:
  - List.1 is the name of the list on which the calculation will be performed
  - PartBody is the body on which the search will be carried out
  - Hole is the Type of the searched feature
  - x.Diameter>50mm is the expression (optional). If no expression is to be verified, just write PartBody.Query("Hole","")

- To know the possible feature types and attributes that you can use in the Query function, use the Edit/Search command.



Instructor Notes:

Populating a List Using a Query

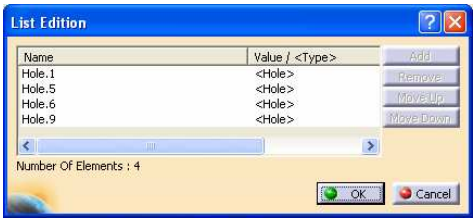
1. Create an empty List: click on the List icon and click OK without adding any item to the list.

2. Open the Formula Editor. Select the new List in the tree and click the Add Formula button.
The formula editor panel is displayed.

3. Enter the right side of the formula in the formula editor field. For instance:
PartBody.Query("Hole","x.Diameter>5mm")
In this case, the List will be populated by all the holes of diameter greater than 5mm.

4. Click OK to validate the formula creation and close Formula panel.

5. The List is automatically populated with the holes of diameter greater than 5mm.



Instructor Notes:

Associating URLs to Parameters and Relations

You will learn how to create and find URLs attached to parameters and relations.

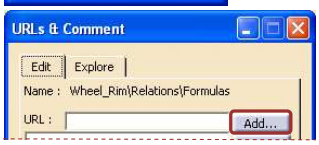
Instructor Notes:

Adding URLs

You can associate one or more URLs with user parameters and relations. This task is only meaningful when the active document contains user parameters and/or relations.

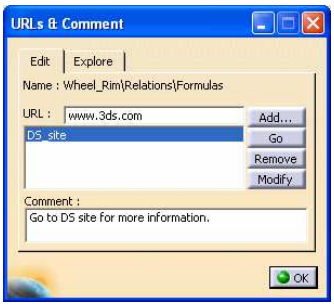
1. Select the Comment & URLs icon in the Knowledge Advisor workbench.

2. In the specification tree, select any parameter or relation (formula, rule, check, etc) to which the URL will be added. Then click the Add button.
The Add URL dialog box is displayed.



3. Enter a name for the URL and the link to it. It may be, for instance, an Internet address or a path to a document. Click OK to validate the creation of the URL.

4. Back in the main edition window, you can also add a comment to the parameter or relation.
Click OK to exit the panel.
The URL and the comment are added to the selected feature.



URLs can also be added to relations at their creation or edition.
To each parameter or relation can be added several URLs but only one comment.

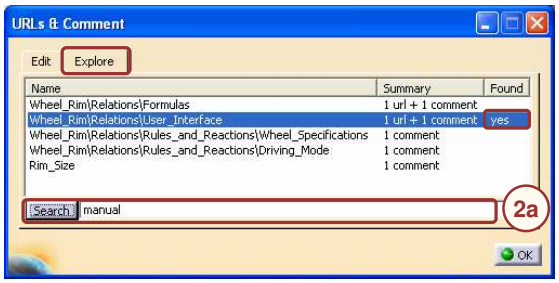
Instructor Notes:

Searching for URLs

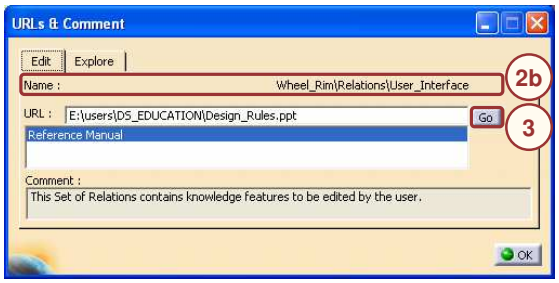
1 Click on the Comment &URLs icon.
The URLs & Comment dialog box opens.

2a From the Explore tab panel:
Enter the name of the searched URL and click the Search button.
If the specified URL is found, "yes" is displayed in the Found column. Then return to the Edit Tab.

OR:



2b From the Edit tab panel:
Select a parameter or a relation in the specification tree : the URLs and the comments of the object are displayed.



3 In the Edit tab, the URL which has been found is highlighted.
Click the Go button to display the page or document related to this URL.

Instructor Notes:

Master Exercise Part 1

Design Process – Part 1

35 min

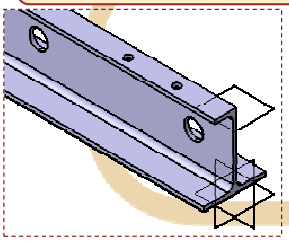
1 Creating User Parameters

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.48 = 2
Offset.28 = 75 - f(x)=Height
Offset.46 = 4 - f(x)=Thickness
Offset.22 = 31.5 - f(x)=Width / 2
Offset.13 = 89 - f(x)=Width

Creating geometry using the User Parameters



Instructor Notes:

Creating Adaptive Behaviors

- Creating Rules
- Creating Checks
- Creating Reactions

Instructor Notes:

Creating Rules

You will learn how to create and use the Rules feature.

Instructor Notes:

Comment about the pictures: Remove all the comments.

Keep only the relevant pictures; remove others; in this case the picture indicates a Slide to Project.

Key Point:

Topic to note:

Topic to discuss with the students:

Show them: (DEMO)

Questions ?

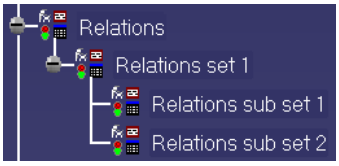
Practice:

What's Next:

Adding Sets of Relations

You can create sets of relations below the Relations node of the specification tree. Using this capability enables you to regroup the relations into categories. Formulas, design tables, rules and checks can all be created into relation sets. When no relation set has been created, the destination field of the relation editor is by default the main Relations node.

1. **To create sets and sub-sets of relations, click on the « Add Set of Relations » icon and select the Relations node in which the new set will be created. Eventually, rename the Relations sets using their Properties command (MB3).**



2. **While creating a new Relation (Check, Rule, etc), select the desired Relation set to store your new Relation.**

Instructor Notes:

What is a Rule?

- A Rule is a set of instructions, generally based on conditional statements, whereby the relationship between the parameters is controlled.
- A Rule appears in the Relations node of the current document:

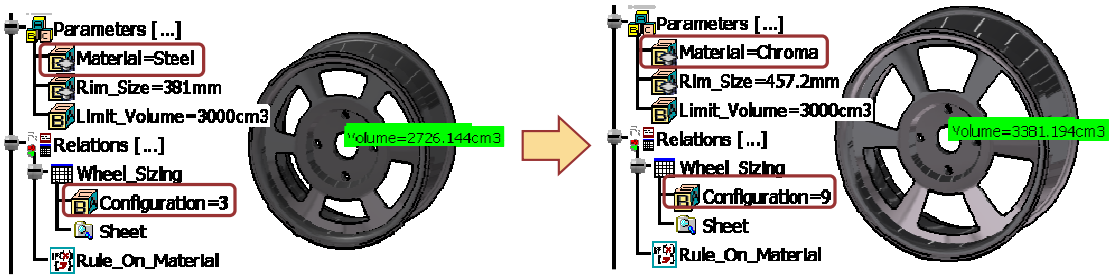
In the example below, the rule calculates the volume of the PartBody and sets the Material parameter in consequence with the result:

```
if smartVolume(PartBody)< Limit_Volume
{Material="Steel"}
```

if the volume of the PartBody is less than a limit value (here 3000cm3), the Material is set to Steel

```
else
Material="Chroma"
```

Otherwise, it is set it to Chroma



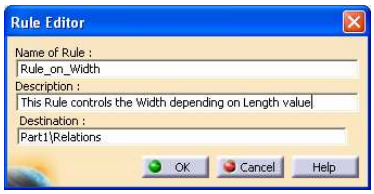
Here, we have changed the wheel's size by changing the configuration of the design table: the volume of the wheel has changed and its material has been updated automatically.

Instructor Notes:

Creating a Rule

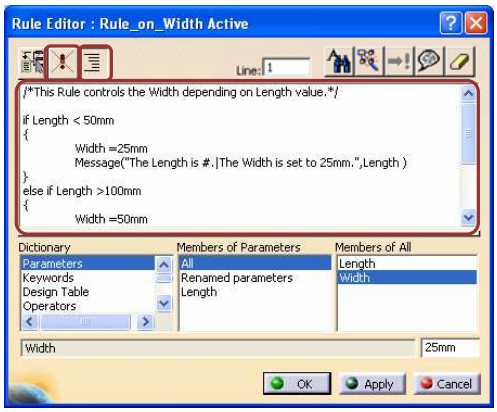
1. Open the Knowledge Advisor workbench and click on the Rule icon.

2. Enter the rule name and comments. You can also choose the relation set to which the Rule will be added.
Click OK.



3. The Rule Editor panel is displayed.
Enter the body of the Rule:
- Check the Alignment button to have an automatic text formatting.
- Write your comments between the "/*" and "*/" signs.
- Use the Dictionary to help you select the parameters and the functions.

4. Check the (!) button to have the syntax of your rule verified interactively.
You can also click the Apply button when you have finished scripting the rule to check its syntax.
Click OK to validate the Rule creation.



5. Rule feature is displayed in the tree under the selected Relations node/set.

Instructor Notes:

Using the Rule/Check/Reaction Editor Interface

The edition panel of the Knowledge Advisor Reactive Features present a few buttons intended to help the user to write the body of the relation.

Type here the feature body.

Use the Dictionary to select the parameters and the functions.

Here is a preview of the latest selected parameter and of its actual value.


Check this button to activate the incremental mode: when you select a feature in the specification tree or in the geometry area, only the first level of features right below the selected feature will be displayed in the editor, which is very useful while working with large models.


Check this button to have a dynamic verification of the body syntax.


Check this button to have the text automatically formatted and indented.


Click this button to open the Language Browser panel.


In case of syntax errors, click this button to highlight the errors.

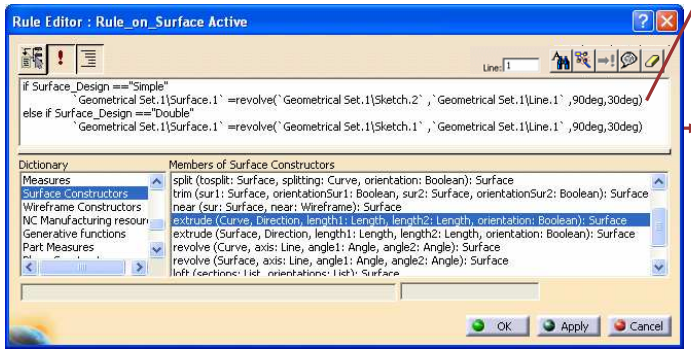

Click this button to add a URL to the relation or to change its comment.

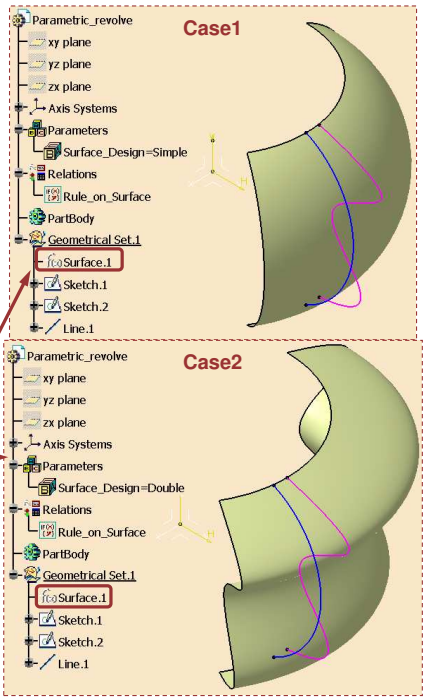

Click the Eraser button to clear the contents of the body field.

Instructor Notes:

What is Creating Geometry from Rules?

- In order to create more adaptative designs, it is sometimes useful to create geometric elements from Rules. To do so, you will use the geometrical operators available in the functions dictionary.
- The following geometric elements can be created:
  - Point
  - Plane
  - Surface
  - Line
  - Curve
  - Circle



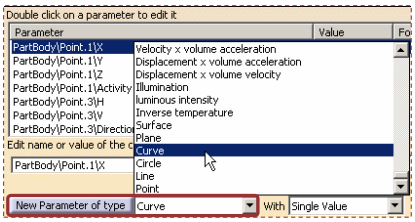


Instructor Notes:

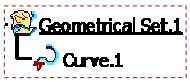
Creating Geometry from Rules

1. Click on the F(x) icon to open the formula editor.

2. Select the geometric type of element you want to create (Curve for example) and click the New parameter of type button.
Close the formula editor by clicking OK.

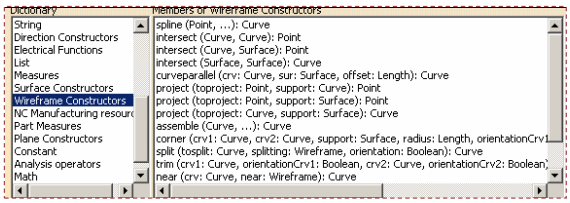


3. The new parametric feature has been added to the tree as a geometrical element.
You can rename it by using its properties (MB3).



4. Create a new Rule in order to valuate the geometric parameter created previously. Use the geometrical operators from the Dictionary.

```
if Case=="1"
{
PartBody\Curve.1 =intersect(PartBody\Extrude.1 ,PartBody\Extrude.2 )
}
else
{
PartBody\Curve.1 =intersect(PartBody\Extrude.1 ,PartBody\Offset.1 )
}
```

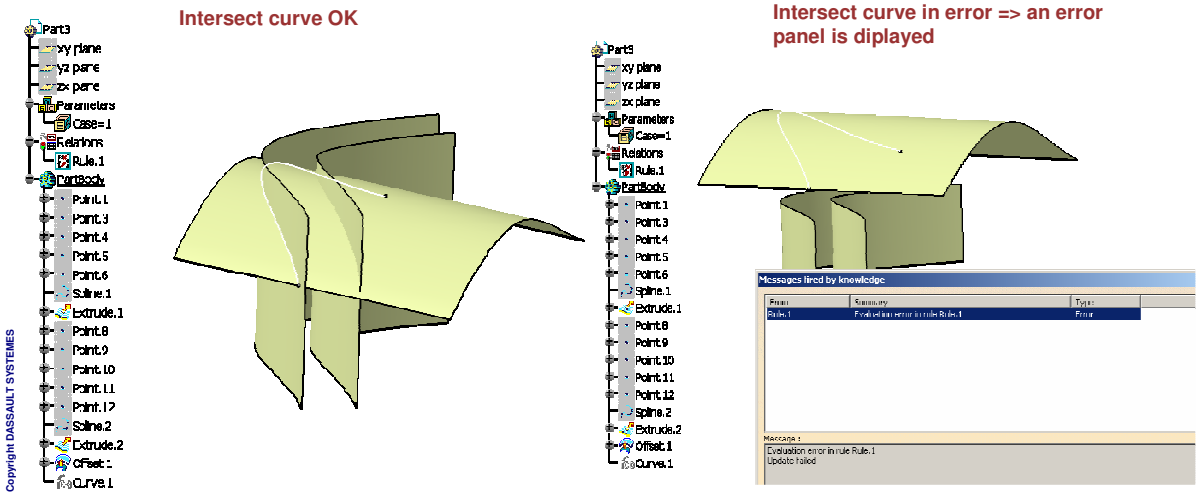


5. Once the Rule is created, the geometric element is displayed in the tree with the F(x) icon meaning that it is driven by a formula or a Rule.

Instructor Notes:

Handling Errors in Rules (1/2)

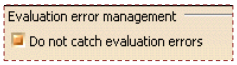
- It is possible to test a geometric feature in error while creating rules. Indeed, the use of geometrical operators to value the geometry in relations may lead to update errors in the created features.
- For example, if the user values a datum curve with the result of the intersection of two surfaces, these two surfaces may not intersect and the intersection curve is therefore in error.



Instructor Notes:

Handling Errors in Rules (2/2)

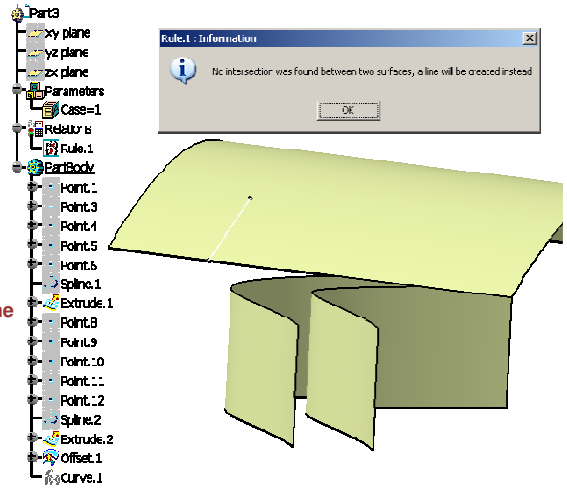
1. In the properties of the Rule (MB3), check the "Do not catch evaluation errors" option.



2. To test if a feature is in error, first create a local variable using **let** keyword and use the **error** keyword as shown in the example below:

```
let x(Curve)

if Case=="1"
{
x =intersect(PartBody\Extrude.1 ,PartBody\Extrude.2 )

 if (x.Error==true)
 {
Message ("No intersection was found between two surfaces, a line will be created instead");
x=line( PartBody\Point.1 ,PartBody\Point.3 )
 }

PartBody\Curve.1=x

}
else
{
PartBody\Curve.1  =intersect(PartBody\Extrude.1 ,PartBody\Offset.1  )
}
```

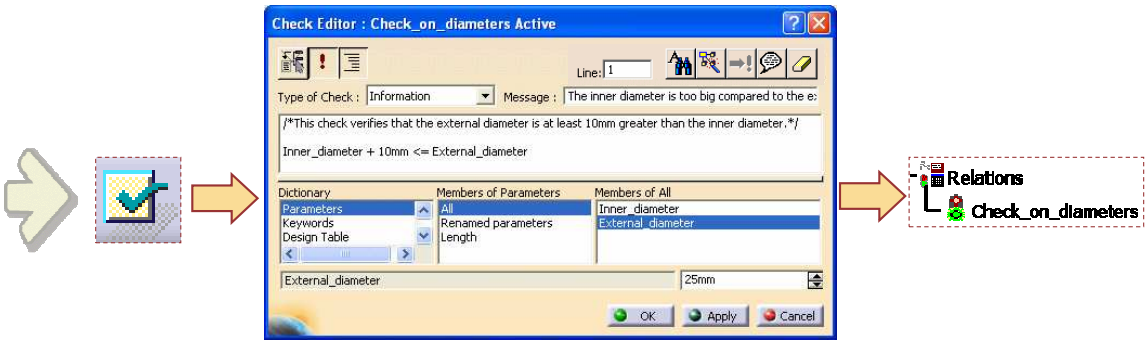
3. Now, in case there is no intersection between the surfaces, an information panel will be displayed and the intersection result will be a line.



Instructor Notes:

Creating Checks

You will learn how to create and analyze the Checks feature.



Instructor Notes:

Comment about the pictures: Remove all the comments.

Keep only the relevant pictures; remove others; in this case the picture indicates a Slide to Project.

Key Point:

Topic to discuss with the students:

Topic to note:

Show them: (DEMO)

Questions ?

Practice:

What's Next:

What is a Check?

- A Check is a set of statements intended to let the user know whether certain conditions are fulfilled or not.
- A check does not modify the document. It is applied to and just gives a design indication.
- A check usually appears in the Relations node of the specification tree with a traffic light icon, switching to red or green according to the check's status.

Check status is OK.

Check status is not OK.

- There are three types of checks:
  - Silent – the status of the check is only indicated by the feature's icon.
  - Information - the status of the check is indicated by the icon, and an Information message occurs when the check is wrong.
  - Warning - the status of the check is indicated by the icon, and a Warning message occurs when the check is wrong.

Information message

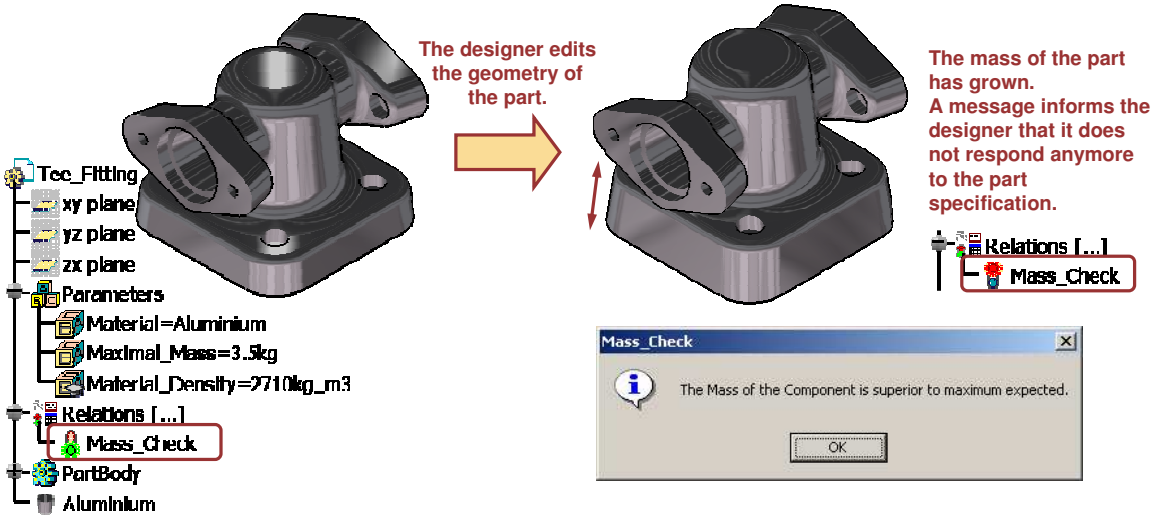
Warning message

Instructor Notes:

Why Use Checks?

- To check that a parameter or a component property responds to a technical limitation or to a set of conditions.
- To ensure compliance with the corporate design rules.
- To avoid update errors that are foreseeable. The check sends a warning message while editing the feature so that the unsuitable value can be changed before an update.

For instance, this check verifies that this mechanical part respects a maximum mass:



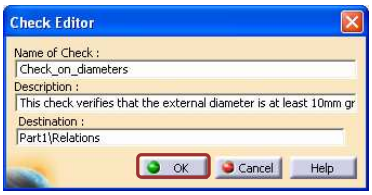
Instructor Notes:

Creating Checks

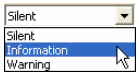
A check is a relationship between the parameters. A direct feedback on the status of the check is given in the tree, thanks to a red or a green light. In case of violation, the user can also be informed by a message panel.

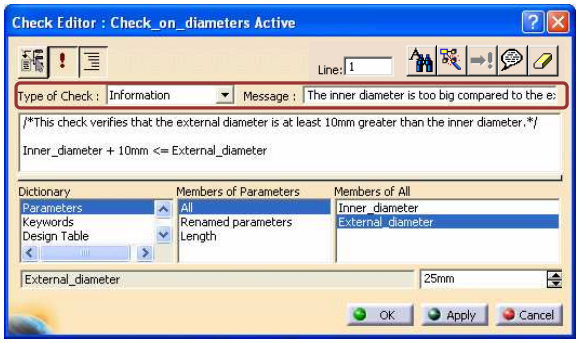
1. In the Knowledge Advisor workbench, click on the Check icon.

2. Enter the check name and a comment. You can also select the set of relations in which the check will be placed. Click OK.



3. The Check Editor panel has opened.
Select the type of check in the list and enter a message that will appear in case of failure.





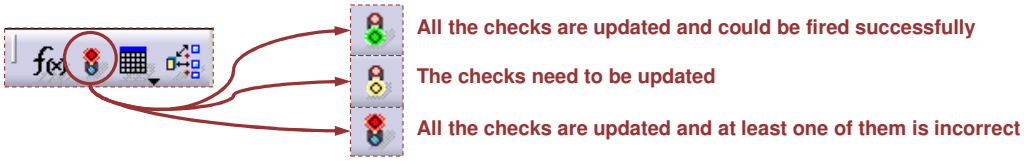
4. Type the body of the check in the main field.
A check is a statement generally based on comparison operators:
"<", "<=", "==", ">=", ">", "<>".
You can use the Dictionary to help you select the parameters.
Click OK to validate the creation of the check.

5. The Check feature is displayed in the tree under the selected Relations node/set.

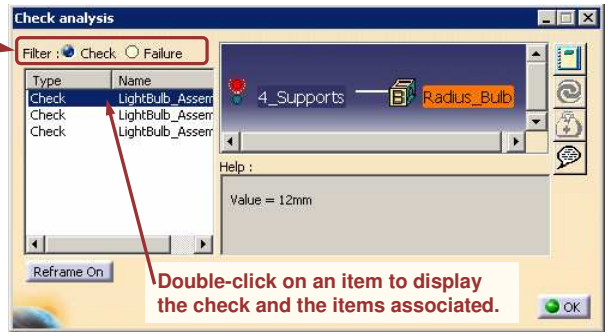
Instructor Notes:

Analyzing Checks

- The Global Analysis Tool is designed to manage the Knowledge Expert and the Knowledge Advisor Checks wherever they may be located in the specification tree. It helps to understand the validation status of the designs and allows navigation by checks or violations, and highlights failed components.
- In the Knowledge toolbar, the « Check analysis toolbox » icon light indicates the active document Checks status:



- Click on the  icon in the toolbar to accede to the Check analysis window:



The Check mode displays only the Check features that failed while updating the check report.

The Failure mode displays all the items that failed while updating the check report.

Double-click on an item to display the check and the items associated.

Click here to generate the customizable report

Click here to solve the checks created

Click here to launch correction (only available for the Knowledge Expert Checks)

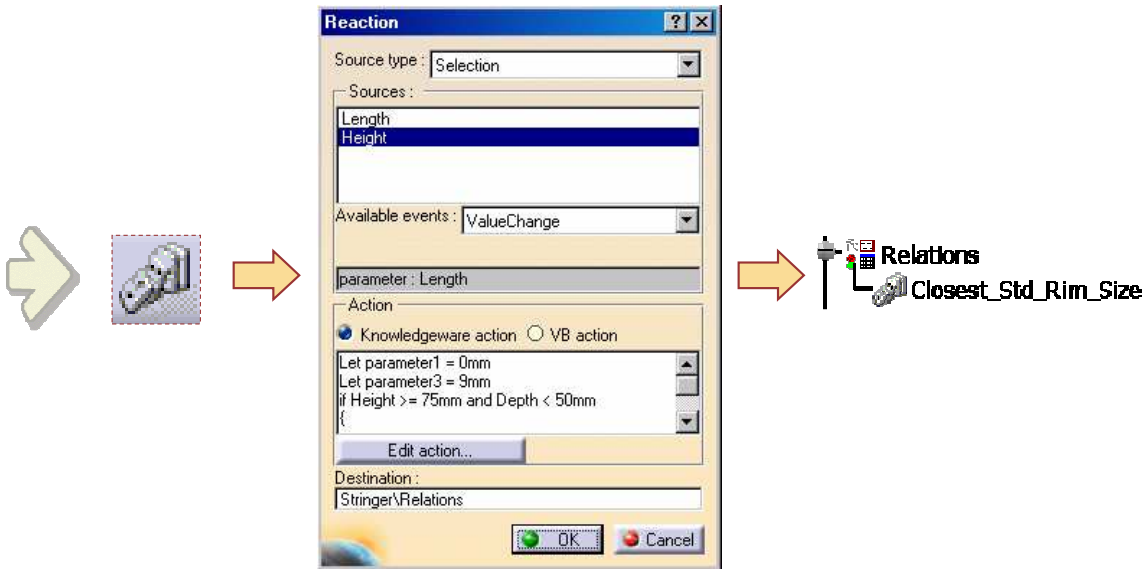
Click here to display or associate a URL

Instructor Notes:

Knowledge Advisor

Creating Reactions

You will become familiar with the Reaction feature.



Instructor Notes:

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Why Use Reactions? (1/3)

- The Knowledge Advisor rules have their own limit.
  - They react to parameter changes or feature updates
    - You cannot control exactly when they are fired
    - They may be fired several times when you would not like to
  - They are integrated to the update mechanism
    - Parameters cannot be in input and in output. For example, it is not possible to write: if x>18mm {x=18mm}

The attached part 'ForceValue.CATPart' contains a reaction which forces the value of the length.1 parameter to 50mm if it is increased above 50mm.

    - Loops and conflicts are forbidden
  - Their language is simple
    - And limited too

Instructor Notes:

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Knowledge Advisor

Why Use Reactions? (2/3)

- To cope with those limitations and to create more associative and reactive designs use the Reaction feature.
  - A reaction is similar to a rule in the fact that:
    - It is stored in the model
    - It reacts to changes and triggers modifications
    - It also references other objects and parameters in the document, and supports replace mechanism
    - It can be used for the definition of PowerCopies and user defined features
  - But:
    - It can react to a larger amount of changes
    - It can drive very complex modifications

Instructor Notes:

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76

Why Use Reactions? (3/3)

- A reaction is a feature that reacts to events on its source(s) by triggering an action
  - The source can be:
    - A selected feature (or a list)
    - A parameter (result of a test)
  - Events can be:
    - General events on objects (creation, destruction, update, attribute changes) and parameters (value change)
    - Specific events such as instantiation and update for a user defined feature
  - Action can be :
    - Written in Knowledge language to access the existing objects in the document or in the Visual Basic Script to extend the action scope
    - It can access the source object and its arguments

Instructor Notes:

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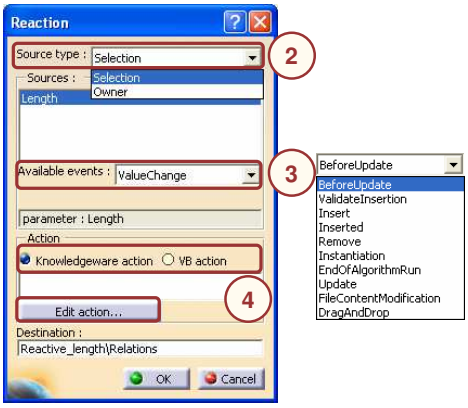
Creating Reactions

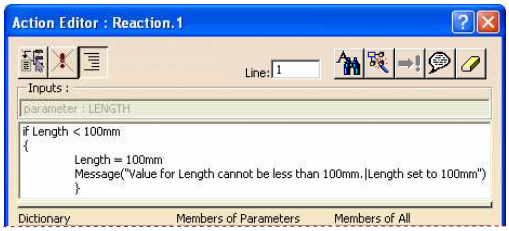
1. In the Knowledge Advisor workbench, click on the Reaction icon. The Reaction dialog box opens.

2. Select the Source type:
   - **Selection** enables you to manually select one or more items in the specification tree or in the geometrical area. These items will be displayed in the Sources field.
   - **Owner** enables you to link the action with a feature of the geometry or of the specification tree. To link the reaction with an object of the geometry, click the Destination field and select an object in the specification tree or in the geometry.
3. In the proposed list, select the **Event** which will trigger the Reaction.
4. Select the language (Knowledgeware or VBScript) in which you want to write the action triggered by the reaction. Click the Edit Action button.

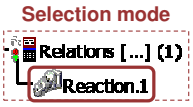


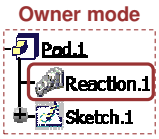
VBScript offers some additional functions and facilities. So, in such cases you can use VBScript.

5. The Action Editor dialog box has opened.
   Type the body of the Reaction in the main field.
   If you have chosen Knowledgeware language, use the Dictionary to select the parameters and the functions.



6. Reaction feature is displayed in the tree:
   - Under the Relations node in the Selection mode,
   - Under the source in the Owner mode.
   You can rename the Reaction using its Properties (MB3).





Instructor Notes:

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Knowledge Advisor

Creating a Loop in a Reaction (1/3)

- Using For statement
  - The first type of loop is a loop based on the element of a list. See syntax below:

    For x inside List
    {Body}

  - X is a variable name of a given type. It may represent an object or a value.
  - List is a variable name of type List or an expression returning a list.
  - X (like any other variable of the language) can be used in the body. It contains the Nth element of the list.
  - The body is executed Nth times, where N is the number of elements of the list.

Instructor Notes:

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Knowledge Advisor

Creating a Loop in a Reaction (2/3)

- Using For statement
  - The second type of loop executes until an expression becomes false. See syntax below:

    For x while predicate
    {Body}

  - X is a variable name of the integer type. It is incremented at the end of each execution of the body.
  - Predicate is a Boolean expression. The body is executed as long as this expression is true. This expression is evaluated before the body.
  - Note that the second for operator can lead to infinite loops.

Instructor Notes:

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Knowledge Advisor

Creating a Loop in a Reaction (3/3)

- Using While statement
  - This loop executes until an expression becomes false. See syntax below:

```
let i = 1
let x(Point)

for i while i<=parameter.Size()
{x = parameter.GetItem(i)
if (x.GetAttributeReal("Y") < 0.04)
x.SetAttributeReal("Y",0.04)}
```

  - i is a variable name of the integer type. It is incremented at the end of each execution of the body.
  - X is a variable for points.

Instructor Notes:

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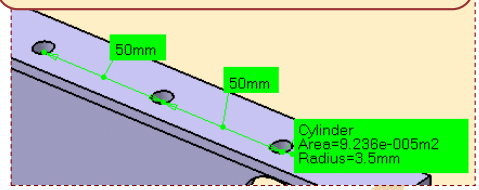
Master Exercise Part 2

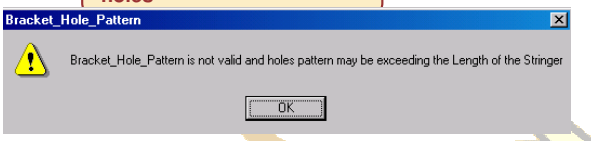
Design Process – Part 2

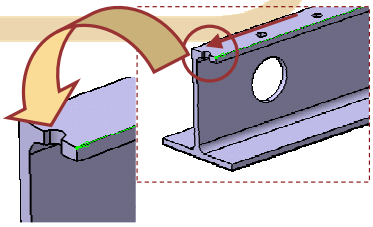
15 min

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





Bracket_Hole_Pattern

Instructor Notes:

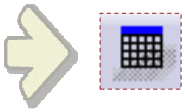
Creating Design Tables and Part Families

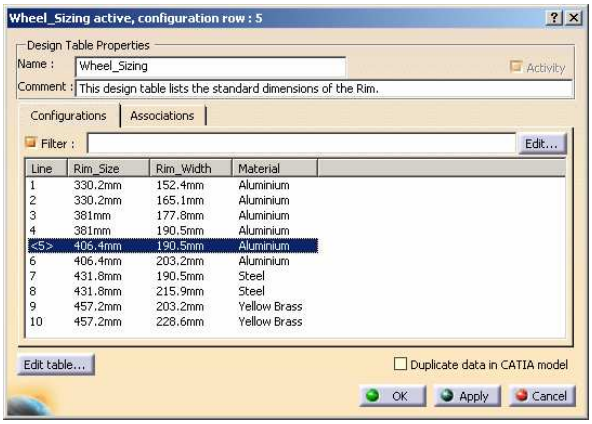
- Creating Design Tables
- Creating a Part Family Catalog

Instructor Notes:

Creating and Using Design Tables

You will learn how to create Design Tables using the document parameters.





Instructor Notes:

What is a Design Table?

- The purpose of the Design Table is to drive the parameters of a CATIA document from external values.
- The Design Table allows to create and manage component families. These components can, for example, be mechanical parts just differing in their parameter values.
- A configuration is a set of parameter value and corresponds to a row.

- A Design Table can be created:
  - From the CATIA document parameters
  - From an external file
- The values are stored either in a Microsoft ® Excel file on Windows™ or in a tabulated text file.

Design Table icon in the Knowledge Toolbar

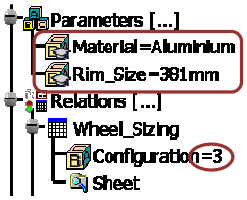
If you create the design from an existing file, it is possible to indicate the sheet number where the table is found.

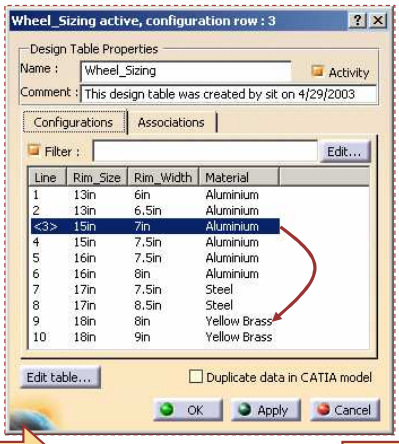
Instructor Notes:

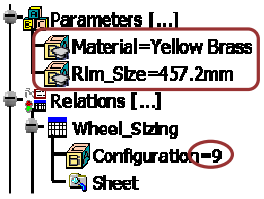
Why Use Design Tables?

- To pre-define possible configurations of the model and to ease the modifications of the dimensions.
- To select only the realistic configurations of the component.
- To link the parameter values that cannot be expressed with a mathematical relation.
- To create part families.

Here is a part whose main dimensions are driven by a design table.







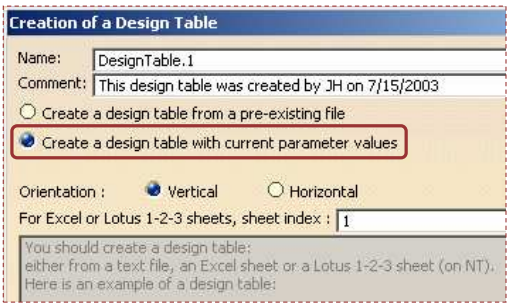
When you change its configuration, three parameters are updated at a time, including an intrinsic parameter (the access of which is not easy).

Instructor Notes:

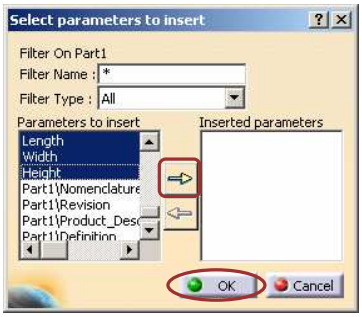
Creating a Design Table from Document Parameters (1/2)

1. Click on the Design Table icon.

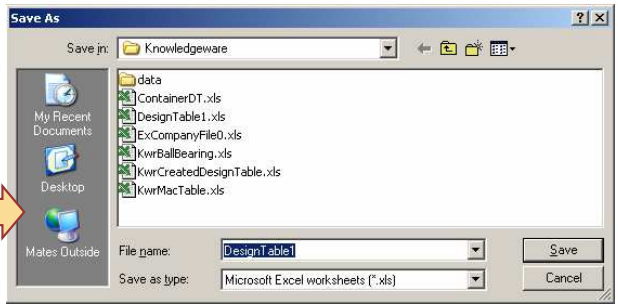
2. The Design Table creation panel is opened. Select the option Create a design table with current parameter values. Click OK.



3. Select the parameters to add to the design table and use the arrows to add them to the list. Click OK.



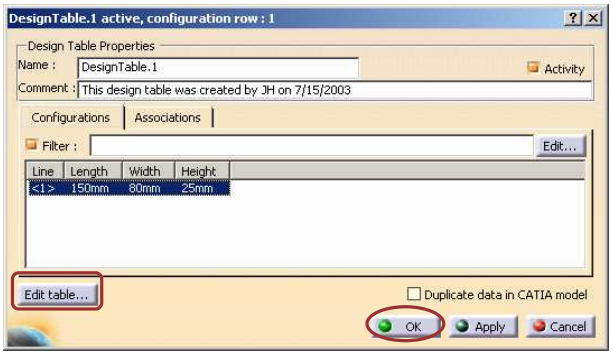
4. Specify the folder and the file name where the data are stored. Click the Save button.



Instructor Notes:

Creating a Design Table from Document Parameters (2/2)

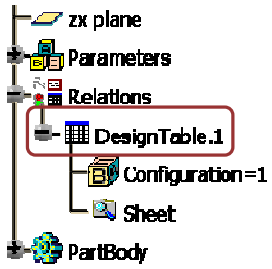
5



The Design Table dialog box has appeared. The Design Table contains only one configuration: the current one.
If you want to add more configurations, click the Edit table button.
Click OK to confirm the Table creation.

6

The Design Table feature appears in the specification tree within the Relations node.



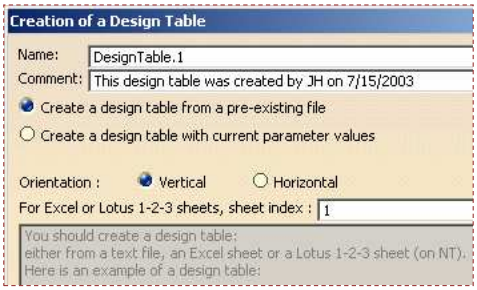
Instructor Notes:

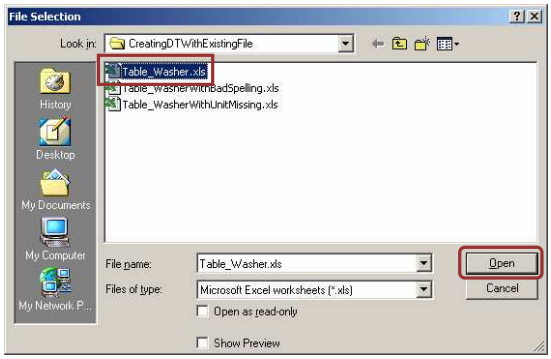
Creating a Design Table with an Existing File (1/2)

You can also create a design table from an already existing file.

1. Select the Design Table icon.

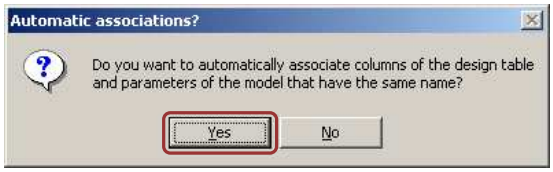
2. The Design Table creation panel is open. Select the Create a design table from a pre-existing file option; Click OK.





3. Specify the external file containing data of your design table; Click the Open button.

4. Click yes if you want an automatic association between the columns of the external file and the parameters of the CATIA document.

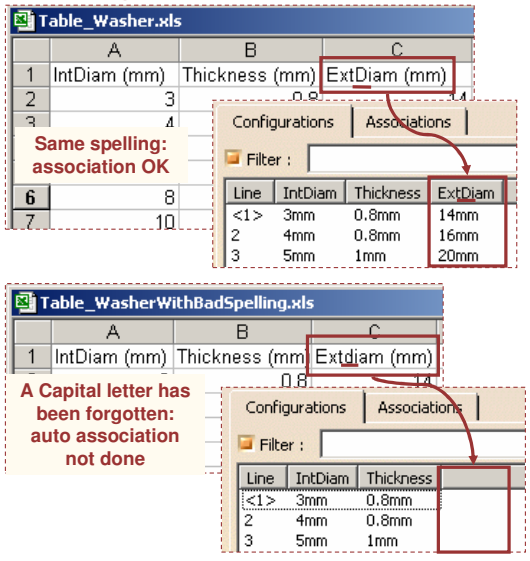


Instructor Notes:

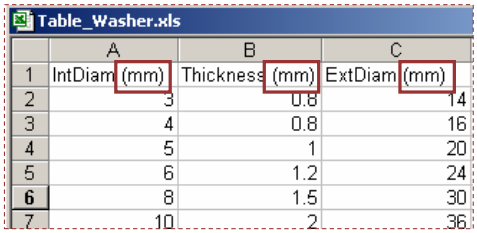
Creating a Design Table with an Existing File (2/2)

When using an existing file, you have to manage the associations between the columns and the parameters. Here are a few pieces of advice to have them automatically made.

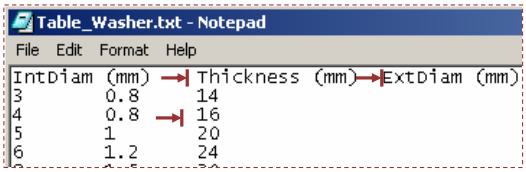
1 **Automatic association occurs between the parameters and the columns having exactly the same spelling (take care of blank space and capital letters).**



2 **In the external file, be careful to specify the units of the values in the top case of the column. If not done, CATIA considers they have the international system (meter for length etc...).**



3 **If the external file is a text file, take care of having only one tab space between the titles and between the values.**

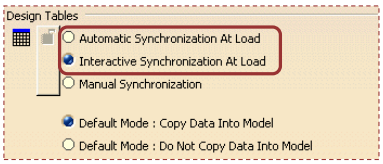


Instructor Notes:

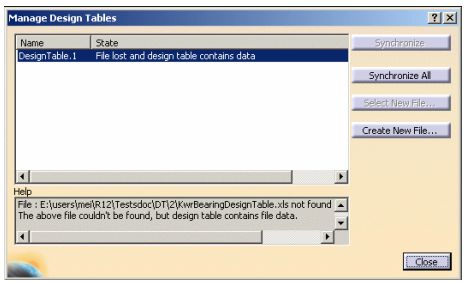
Generating a File From a Design Table

It is possible to regenerate an external file (.XLS or .txt format) using the data contained in the model. The data contained in the model comes from an external file that was previously deleted.
The design Table has to be created with the Duplicate data in the CATIA model option.

1. From the **Tools->Options...->Parameters and Measure** command, access the Knowledge tab and make sure the **Interactive Synchronization At Load** is checked.



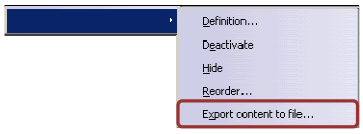
2. Open the CATPart document of which the Design Table file has been deleted or renamed without CATIA. The Manage Design Tables window displays indicating that the external file has been deleted.



3. Click the Create New File... button to generate a file from the data contained in the .CATPart document. The Save As dialog box displays

   OR...

   If you are working with the option **Automatic Synchronization At Load**, right-click the DesignTable in the specification tree and select the DesignTable.x object->Export content to file... command.



4. Enter the name of the file that you want to create: .XLS is the default file type. The text format is also available. Click Save and Close when done. The file containing the design table data is created.

Instructor Notes:

Design Table Functions (1/3)

Various Design Table methods are available to find / set values and configurations in the design tables. These functions can be used in Rules and Reactions. The explanation for a few functions is given below.

CloserSupConfig()

This function applies to a design table sheet. It returns the configuration which contains the values closest to those given in the arguments.

When several configurations meet this condition, the method sorts out the possible configurations with respect to the column order as it is specified in the argument list.

Syntax of the function is given below:
sheet.CloserSupConfig(columnName: String, minValue: Literal, ...): Integer

No.	SketchRadius(mm)	Pad_Limit_1(mm)	Pad_Limit_2(mm)
1	120	60	10
2	130	50	30
3	120	60	25
4	140	50	40

For the design table shown above, an example of the use of 'CloserSupConfig' is given below.

Relations\DesignTable1\sheet_name.CloserSupConfig("SketchRadius", 120mm, "PadLim1", 60mm, "PadLim2", 20mm)

The above function will return configuration number '3' ('third' configuration).

Instructor Notes:

Design Table Functions (2/3)

CellAsReal()

This function applies to a design table sheet. It returns the contents of a cell (intended for real values). Returns zero if the cell does not contain a real value or if the method arguments are not properly specified.

Syntax
sheet.CellAsReal(rowIndex: Integer, columnIndex: Integer): Real

In the above syntax, the rowIndex is the configuration number (integer from 1 to n) and columnIndex is the column number.

No.	SketchRadius(mm)	Pad_Limit_1(mm)	Pad_Limit_2(mm)
1	120	60	10
2	130	50	30
3	120	60	25
4	140	50	40

Relations\DesignTable1\sheet_name.CellAsReal(3, 2)

The above function will return 60.

Instructor Notes:

Design Table Functions (3/3)

SetCell()
Enables you to fill in a cell at a given position in an Excel file or a tab file.
Note: the index must start at 1 for the (1,1) cell to be located at the left top corner.

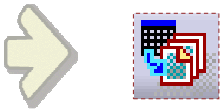
Syntax:
sheet.SetCell(IndexRow:Integer, IndexColumn:Integer, CellValue:Literal): Void
Example:
Sheet.SetCell(2, 2, 45)

No.	Radius(mm)	Pad_Limit_1(mm)
1	120	60
2	130	45
3	120	60
4	140	50

Instructor Notes:

Creating a Part Family Catalog

You will learn how to create a Part Family Catalog from a Part containing a Design Table.



Instructor Notes:

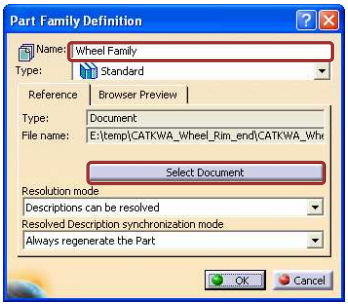
Creating a Part Family Catalog

1. Edit the Part's Design Table and insert a column called "PartNumber". Fill in this column with the names that will be given to the parts that are going to be generated.

	A	B	C	D
1	PartNumber	Rim_Size (in)	Rim_Width (in)	Material
2	Rim_13_6	13	6	Aluminium
3	Rim_13_6.5	13	6.5	Aluminium
4	Rim_15_7	15	7	Aluminium
5	Rim_15_7.5	15	7.5	Aluminium
6	Rim_16_7.5	16	7.5	Aluminium
7	Rim_16_8	16	8	Aluminium

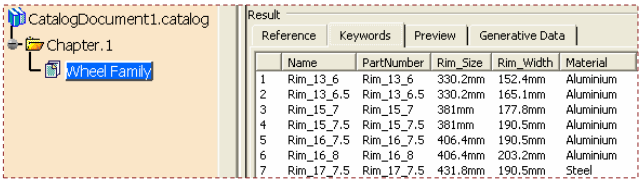
2. Create a new CatalogDocument (File>New). Activate a chapter and click on the Add Part Family icon.

3. Click the Select Document button to browse the CATPart definition document. The CATPart must contain at least one Design Table with a PartNumber column.
Enter a name for the Family in the top field.



4. In the Browser preview tab, click the Select an external preview file button to preview an external file in the .jpg, .bmp., etc. format (optional).

5. The part family is created and displayed in the specification tree. It contains a component per line of the design table.
Save the new Catalog document.

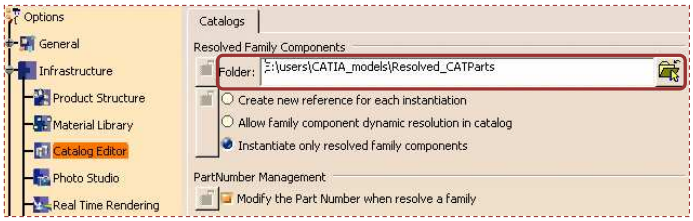


Instructor Notes:

Part Family Resolution (1/2)

Resolving a Part Family means that you generate the .CATPart documents referred to by the Part Family.
These documents are generated in a specific place, and each generated document is a copy of the generative part configured with the matching row in the design table.

1. **In Tools>Options indicate the folder where the CATParts associated to the resolved components will be generated.**

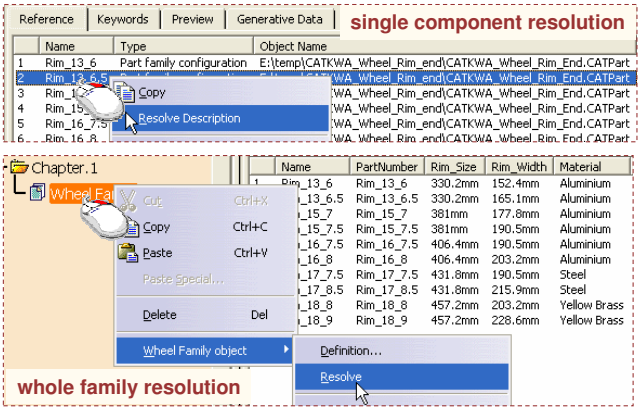


2. **If not already opened, open the catalog containing the Part Family. Activate the Part Family.**

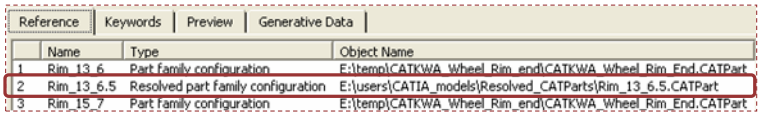
Instructor Notes:

Part Family Resolution (2/2)

3 You can resolve either the entire Part Family or a single Part Family component.
In both cases, use the Resolve option in the contextual menu.



4 The resolved component(s) can be identified in the Part Family description.



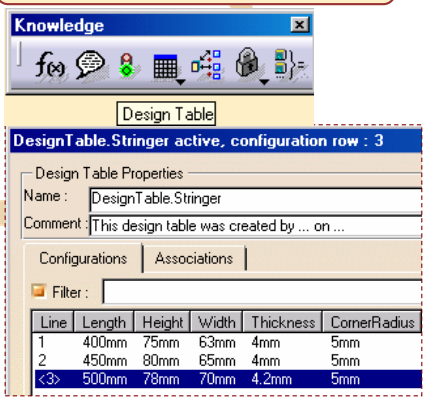
Instructor Notes:

Master Exercise Part 3

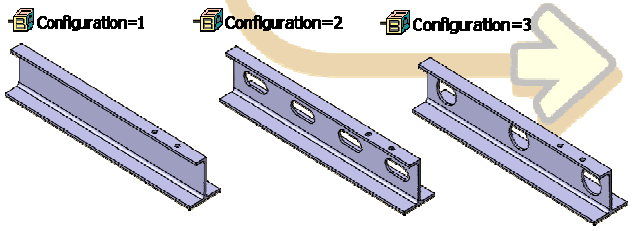
Design Process – Part 3

15 min

6 Creating a new Design Table from the existing User Parameters



Changing configuration and updating the design



Instructor Notes:

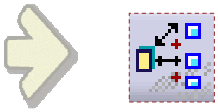
Using Knowledge Advisor Tools

- Using the Knowledge Inspector Tool
- Using the Set of Equations Tool
- Creating and Using Laws

Instructor Notes:

Knowledge Inspector Tool

You will learn how to use the Knowledge Inspector tool in order to analyze modifications, impacts, and dependencies.



Instructor Notes:

Using Knowledge Inspector: "What if" Mode (Impacts)

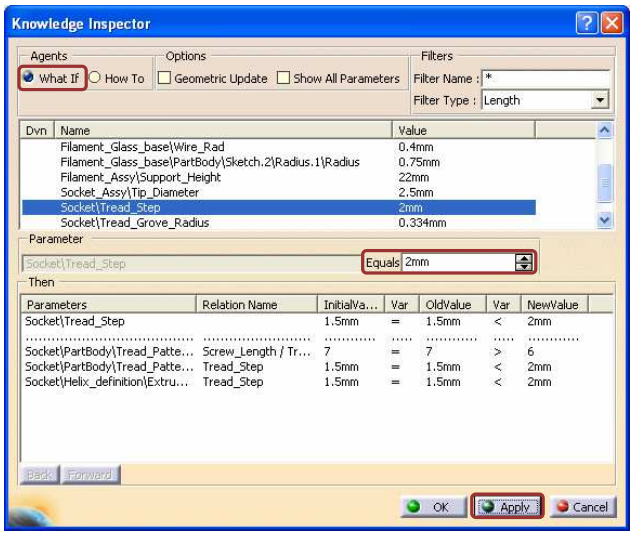
This mode helps you to understand to what extent changing any parameter of your design (such as a dimensional parameter or a material) changes the operation or design of the product on which you are working. It can be used to examine interactions of parameters with each other, and with the rules that make up the product's specifications.

1. Click on the Knowledge Inspector icon in the common knowledge toolbar.

2. Check the "What If" option. All the driving parameters are displayed in the top parameters list.
   Check the "Show All Parameters" option to display all the parameters of the document.
   Check the "Geometric Update" if you want to visualize the result of your modification in the geometry area.

3. Select in the list the parameter whose impacts are to be analyzed.

4. Use the Equals field to modify the selected parameter value.
   Click on Apply or Enter to display the values of the impacted elements in the "Then" area.



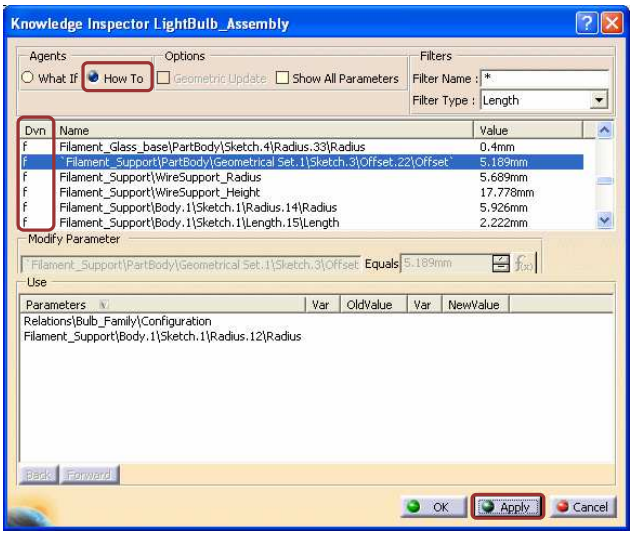
Instructor Notes:

Using Knowledge Inspector: "How to" Mode (Dependencies)

Helps you to determine how your design can be changed to achieve a desired result.

1. Click on the Knowledge Inspector icon in the Common Knowledge toolbar.

2. Check the "How to" option. The list of all the parameters of the document that are driven by a relation is displayed.
   Check "Show all Parameters" to have a list of all the parameters of the document. The driven parameters are identified by an "f" in the left column.



3. Select the parameters whose dependencies are to be analyzed.

4. Click on Apply or Enter.
   The list of impacting parameters is displayed in the use area.

Instructor Notes:

Using the Set of Equations tool

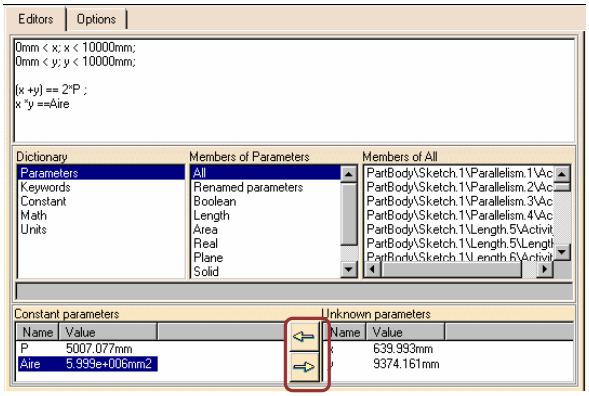
You will learn how to use the Set of Equations tool to solve the engineering problems.

Instructor Notes:

Using the Set of Equations Tool

1. Click on the « Set of Equations » icon.

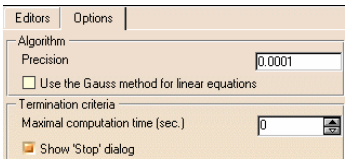
2. Define your set of equations in the editor, using the existing parameters.



3. Use the arrow button to define which parameters are Constant parameters or Unknown parameters (to be solved).

   Constant parameters can be modified by using the formula editor.

4. Select the solve options.



   Precision option defines the precision of the result.

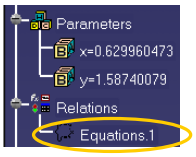
   The Gauss method accelerates the solve operation while working with the linear equations.

   Maximal computation time enables you to indicate the computation time (if 0, the computation will last until a solution is found)

   The Show 'Stop' dialog option displays a 'Stop' dialog box that will enable you to interrupt the computation.

5. Click « Apply » to check the syntax.

6. Click « OK » to exit the editor and solve the equation system.



Instructor Notes:

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Creating and Using Laws

You will learn how to create and use the Knowledge Advisor Laws and how to combine the Knowledge Advisor (KWA) and the Generative Shape Design (GSD) Laws.

Knowledge Advisor Law

Generative Shape Design Law

Instructor Notes:

Creating a Knowledge Advisor Law

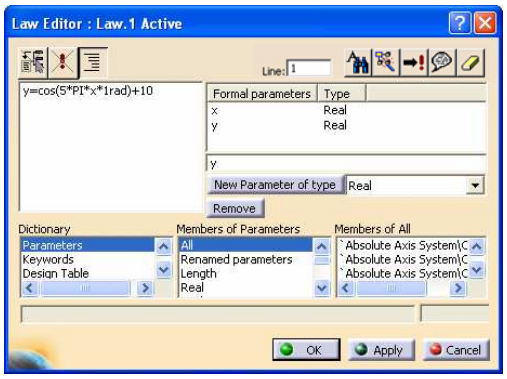
A Knowledge Advisor law is a relation whereby a parameter is defined with respect to another single parameter. Both the parameters involved in a law are called formal parameters. The formal parameters and laws are specifically designed to be used in the creation of shape design parallel curves.

1. Click on the Law icon.

2. Select a destination and give a name to the law.

3. Use the New Parameter of type button to create the formal parameters that will be used to define the law.

4. Enter the law definition, for example: y=cos(5*PI*x*1rad)+10



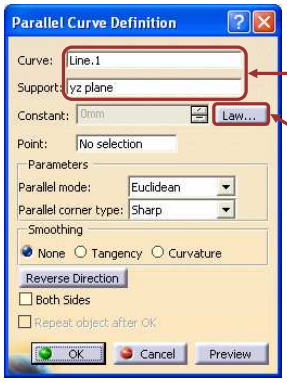
5. The Law feature is created under the Relations node.

Instructor Notes:

Using a Knowledge Advisor Law for Parallel Curves Definition

1. Create a Line as the reference curve.

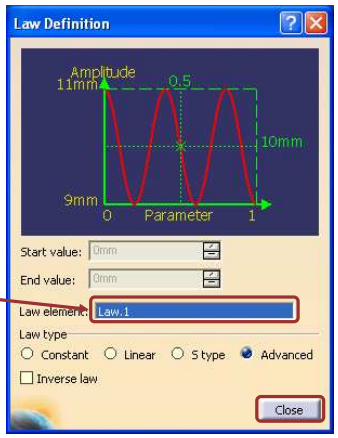
2. Click on the Parallel curve icon to create a curve parallel to the previous line:



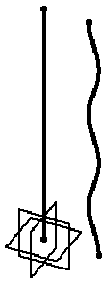
Select the reference line and the support plane.

Click the Law button:

The Law Definition panel appears, select a Knowledge Advisor law in the tree and click Close.



3. The parallel curve is created according to the law definition:

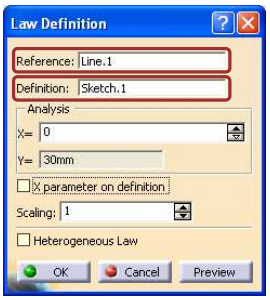


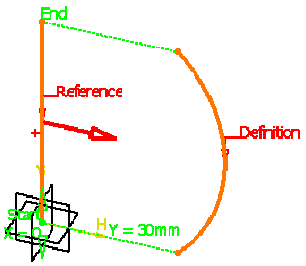
Instructor Notes:

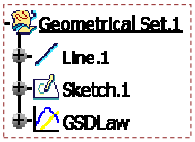
Combining Knowledge Advisor Laws and GSD Laws

You can use a combination of a Generative Shape Design law and a Knowledge Advisor law in the same relation.

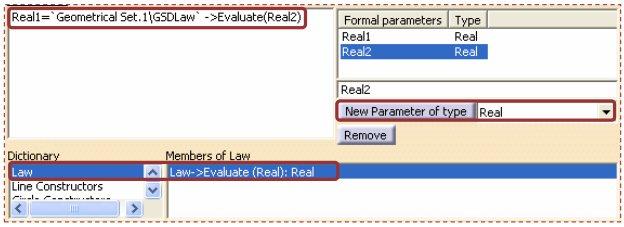
1. **Create a GSD law using a reference and a definition curve.**







2. **Create a new Knowledge Advisor law.**
   Use the GSD law with Evaluate method to define it:



Instructor Notes:

Knowledge Advisor Added Exercises

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

Instructor Notes:

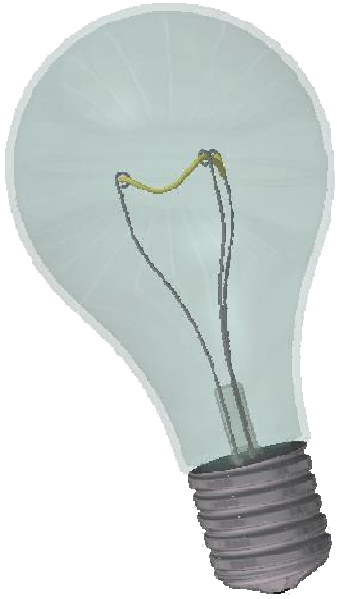
Light Bulb

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.**



Instructor Notes:

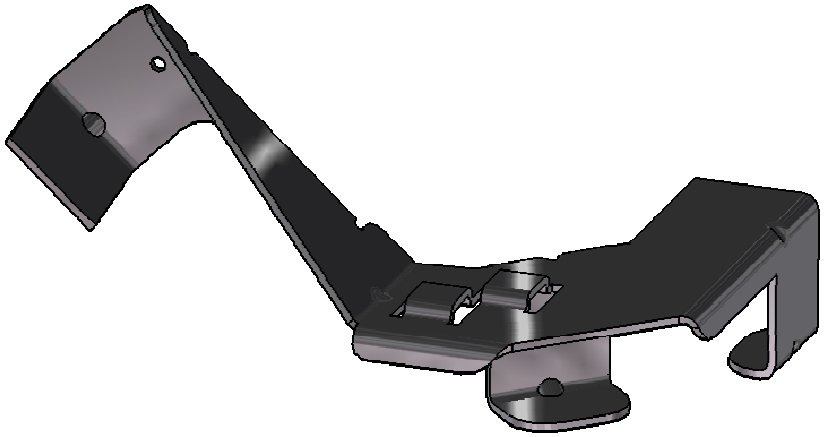
Sheetmetal Part

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**



Instructor Notes:

Wheel Rim Exercise

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

Instructor Notes:

Wheel Rim

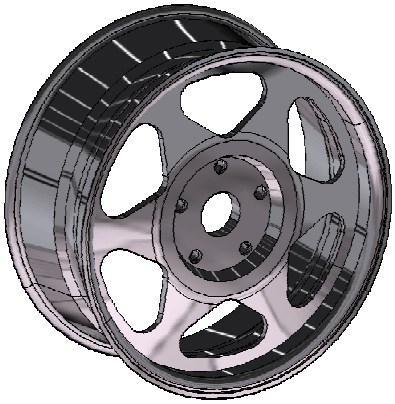
Exercise Presentation

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**



Instructor Notes:

Wheel Rim Exercise Part 1

Instructor Notes:

Wheel Rim Exercise Part 2

Instructor Notes:

Wheel Rim Exercise Part 3

Instructor Notes: