

Getting Started

Before getting into the details for using CATIA - Knowledge Advisor Version 5, this section provides a step-by-step scenario demonstrating how to use Knowledge Advisor key functionalities. You should be familiar with the basic commands common to all workbenches. These are described in the *Infrastructure User's Guide*.



When working in a Japanese environment, remember to check the Surrounded by the Symbol' (Tools->Options->General->Parameters and Measure->Knowledge tab).

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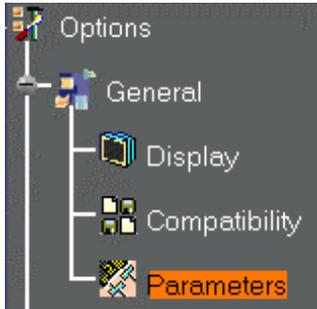
Using Parameters



This task explains how to use parameters. For a fuller outline of the parameters-related tasks, see the [Knowledgeware Infrastructure - Tips and Techniques - Summary](#) dedicated to the knowledgeware infrastructure capabilities.



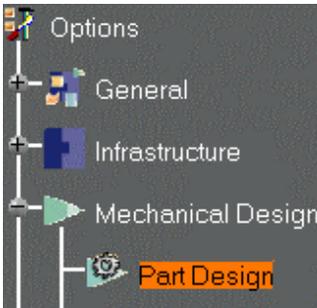
Check the settings below:



- From the `Start->Knowledgeware` menu, select the `Knowledge Advisor workbench`.
- From the `Tools` menu, select `Options->General->Parameters`.
- In the `Knowledge` tab, check the `With Value` and `With Formula` check boxes, and click `OK`.



When working in a Japanese environment, check the `Surrounded by` check box under `Parameter names`.



- From the `Tools` menu, select `Tools->Options...->Infrastructure->Part Infrastructure`
- Check at least the `Relations` and `Parameters` boxes in the `Display` tab, and click `OK`.



It is recommended to check all the options located below the `Specification tree settings`.

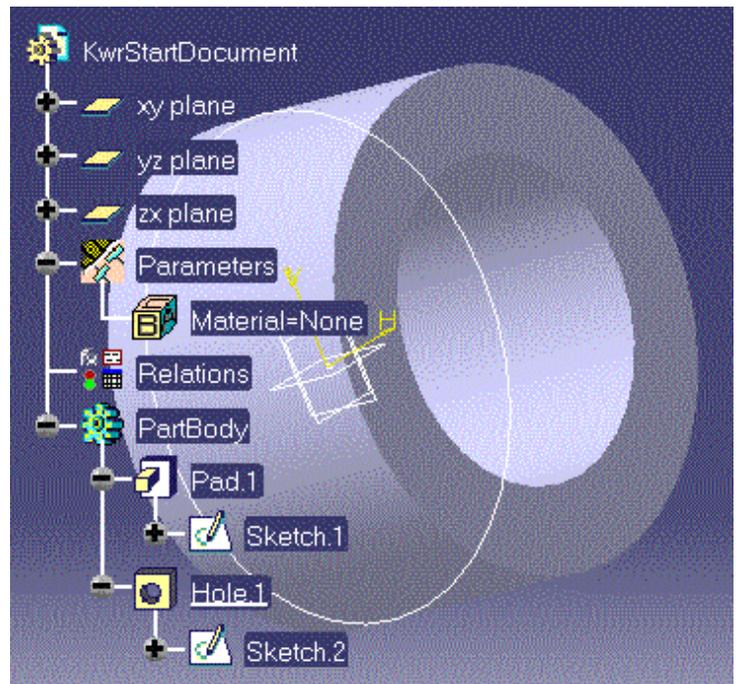


1. Open the [KwrStartDocument.CATPart](#) document.

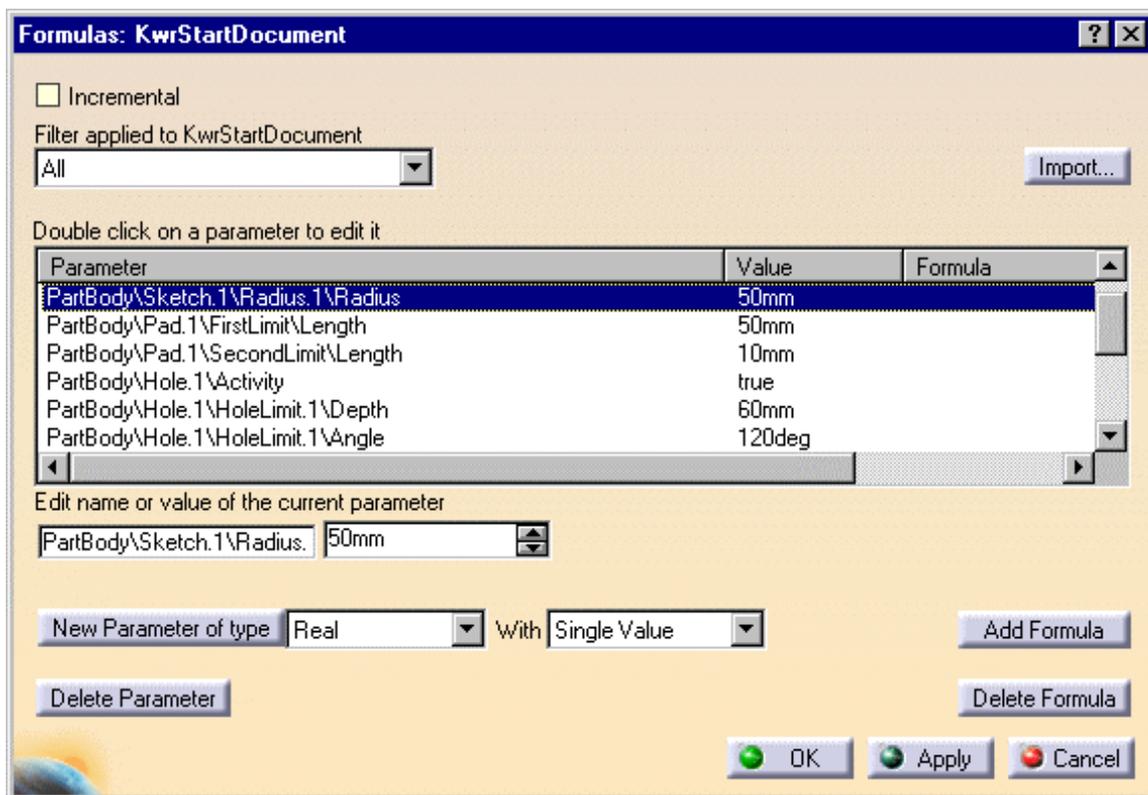
If you expand the `Parameters` node in the specification tree, the `Material` parameter is the only one displayed. At this stage of the scenario, don't pay any attention to this default parameter.



The `Relations` node can't be expanded as there is no default relation in a CATIA document.



- Click the  icon. The Formulas dialog box is displayed. The Incremental box must be unchecked.



- In the New Parameter of type list, select the Length item, then click New Parameter of type.
- In Edit name or value of the current parameter, replace the Length.1 string with PadLength, and click Apply. A new parameter is added to the document parameter list both in the Formulas dialog box and in the specification tree. You have just created a *user parameter*.

5. Click **OK** in the **Formulas** dialog box to terminate the dialog. Keep your document open and proceed to the next task.



Using Formulas



This task explains how a parameter can be constrained by a formula.

See the [Knowledgeware Infrastructure - Tips and Techniques - Summary](#) dedicated to the infrastructure knowledgeware capabilities for more information on formulas.

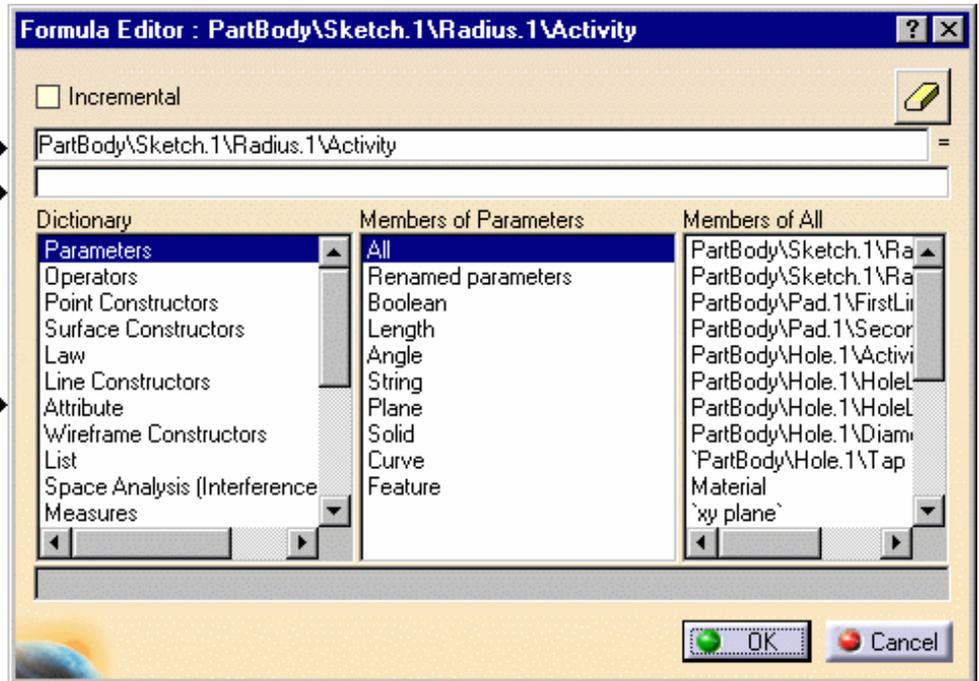


1. Click the  icon. The **Formulas** dialog box is displayed.
2. In the parameter list, select the `PartBody\Sketch.1\Radius.1\Radius` item, then click **Add Formula**. The **Formula editor** displays.

The upper field of the editor displays the parameter to be constrained.

The empty field is for the formula.

The dictionary is a comprehensive list of the items that can be used in a formula. The items displayed may slightly differ from those

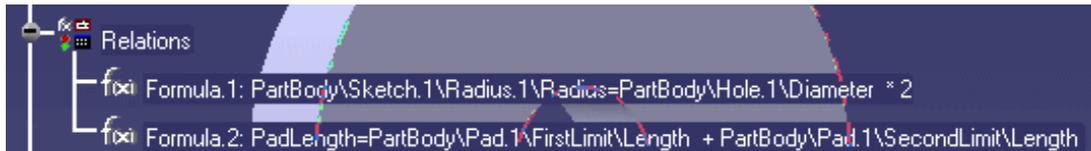


The icon located on the right is simply a rubber you can use to erase the formula.

3. Enter the $2 * \text{PartBody}\backslash\text{Hole.1}\backslash\text{Diameter}$ relation.
4. Click **OK** in the **Formula Editor** once you have typed your relation. The **Formula.1** relation is added to the specification tree. In the parameter list of the dialog box, a formula is now associated with the pad radius.
5. In the parameter list, select the **PadLength** item, click **Add Formula** to create the formula below:

$\text{PadLength} = \text{PartBody}\backslash\text{Pad.1}\backslash\text{FirstLimit}\backslash\text{Length} + \text{PartBody}\backslash\text{Pad.1}\backslash\text{SecondLimit}\backslash\text{Length}$

In the parameter list, the **Formula.2** relation is now associated with the **PadLength** user parameter. In the specification tree, **PadLength** is also displayed with the value resulting from **Formula.2**. This is now what you should see in the specification tree under "Relations":



6. Click OK in the `Formulas` dialog box to terminate this task. Keep your document open and proceed to the next task.



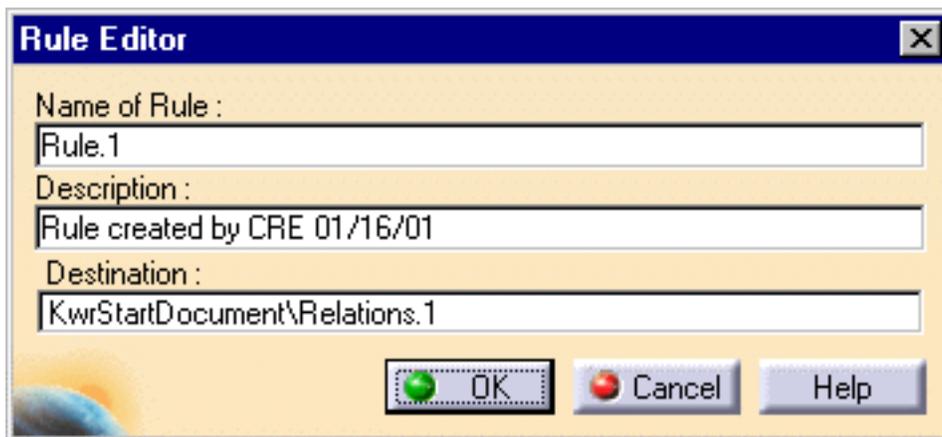
Using Rules

-  This task introduces the Knowledge Advisor rules without getting bogged down in details. Unlike the parameter and formula capabilities which are available to all CATIA users, the rule and check capabilities require the Knowledge Advisor product. For more information on rule-related tasks, refer to [Rule and Check Tasks](#).

-  • For more information about Rules, see [Working with the Rule Feature](#).

To know more about the Rule Editor, see [Using the Rule Editor](#).

-  1. Select the KwrStartDocument item in the specification tree
2. Access the Knowledge Advisor workbench from the Start->Knowledgeware menu.
3. Click the  rule icon. The following dialog box is displayed:



The dialog box fields display default values that can be modified:

- a** - The rule name: Rule.i. The first rule created in a document is Rule.1 by default. This name is the one displayed in the specification tree unless you modify the default name at creation.
- b** - The user and the date of creation.
- c** - The destination, i.e. the feature you are going to add the rule to. By default, in this scenario, the destination is the Relations feature

(the Relations node in the specification tree). But a rule could be added to another feature, then only apply to this feature.

4. Replace the Rule.1 string with Cylinder_Rule, if need be modify the comments but don't modify the destination. Click OK. The Rule Editor is displayed (see below).



5. Type the code below into the edition box or copy/paste it from your browser to the edition box.

```
PartBody\Hole.1\Activity = true
if PadLength <= 50mm and PadLength > 20mm
{
PartBody\Hole.1\Diameter = 20mm
Message("PadLength is: # | Internal
Diameter is: #",
PadLength,PartBody\Hole.1\Diameter)
}
else if PadLength > 50mm and PadLength <
100mm
{
PartBody\Hole.1\Diameter = 50mm
Message("PadLength is: # | Internal
Diameter is: #",
PadLength,PartBody\Hole.1\Diameter)
}
else if PadLength >= 100mm
{
PartBody\Hole.1\Diameter = 80mm
Message("PadLength is: # | Internal
Diameter is: #",
PadLength,PartBody\Hole.1\Diameter)
}
else
{
PartBody\Hole.1\Activity = false
Message("PadLength is: # | Internal
Diameter is: #",
PadLength,PartBody\Hole.1\Diameter)
}
```

Users working in a Japanese environment should use the script below:

```
`PartBody\Hole.1\Activity` = true
if `PadLength` <= 50mm and
`PadLength` > 20mm
{
`PartBody\Hole.1\Diameter` = 20mm
Message("PadLength is: # | Internal
Diameter is: #",
`PadLength`,`PartBody\Hole.1
\Diameter`)
}
else if `PadLength` > 50mm and
`PadLength` < 100mm
{
`PartBody\Hole.1\Diameter` = 50mm
Message("PadLength is: # | Internal
Diameter is: #",
`PadLength`,`PartBody\Hole.1
\Diameter`)
}
else if `PadLength` >= 100mm
{
`PartBody\Hole.1\Diameter` = 80mm
Message("PadLength is: # | Internal
Diameter is: #",
`PadLength`,`PartBody\Hole.1
\Diameter`)
}
else
{
`PartBody\Hole.1\Activity` = false
Message("PadLength is: # | Internal
Diameter is: #",
`PadLength`,`PartBody\Hole.1
\Diameter`)
}
```

6. Click **Apply**. An information window displays the **PadLength** and **Pad internal diameter** values. Click **OK** in the Information window.

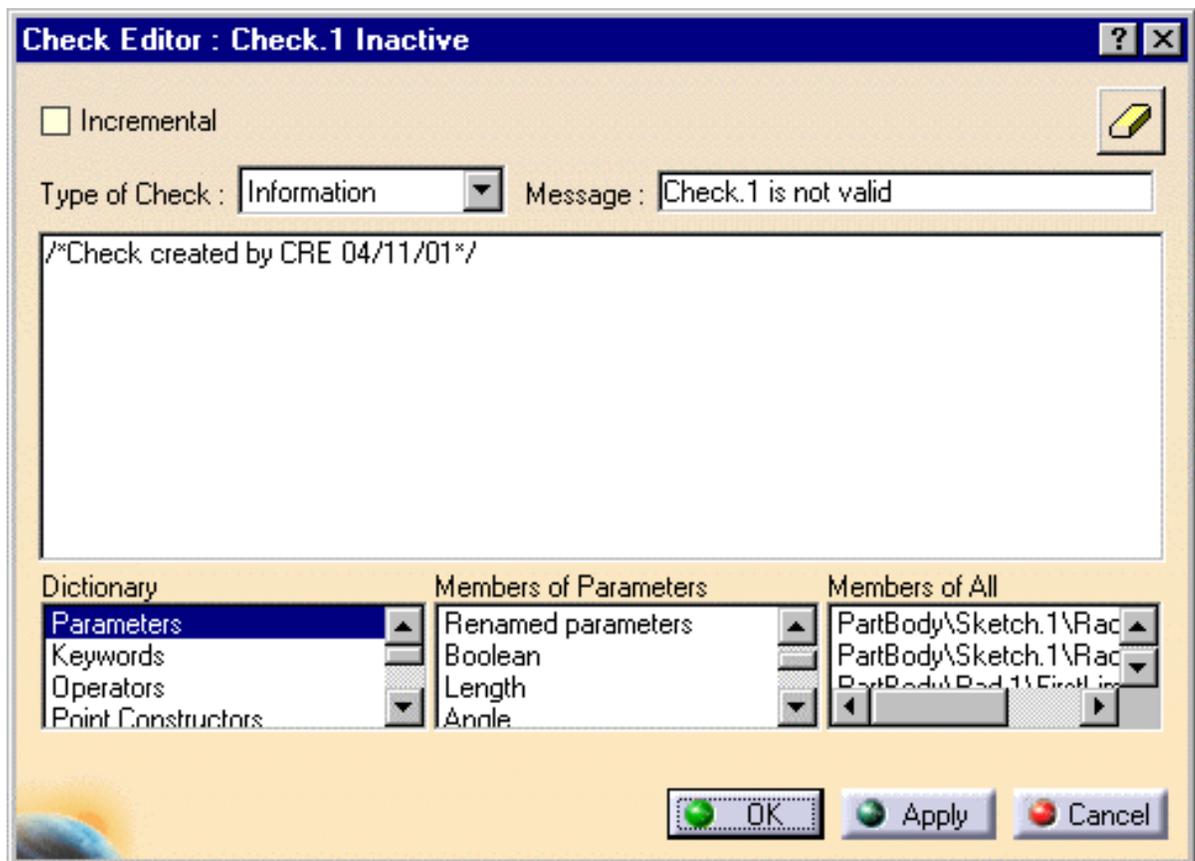
Using Checks



This task explains how to create a check and what happens when you add a check to a document. The Knowledge Advisor product is required for this task. See the [Rule and Check Tasks](#) for more information on check-related tasks.



1. Click the  icon. The first "Check Editor" dialog box is displayed.
2. Replace the Check.1 default name with Cylinder_Check, then click OK. The Check Editor box is displayed. It is similar to the Rule Editor. The Incremental box must be unchecked.



3. Select the Information item in the Type of Check list.
4. Enter a string in the message field (for example: Pad too short). This message is to be displayed whenever the statement specified by the check is not fulfilled.
5. Enter the following statement into the edition box: `PadLength > 20mm`
6. Click OK to confirm the check creation. The Cylinder_Check relation is added to the specification tree. A green icon in the specification tree means that the check is fulfilled. No message is displayed.

7. Change the Pad limits so that `PadLength <= 20mm`. The `Cylinder_Rule` relation is re-applied. An information window displays the new `PadLength` and `Pad` internal diameter values. Then, you are warned by another window ("Pad too short") that the check is no longer valid. The check icon in the specification tree turns to red.

