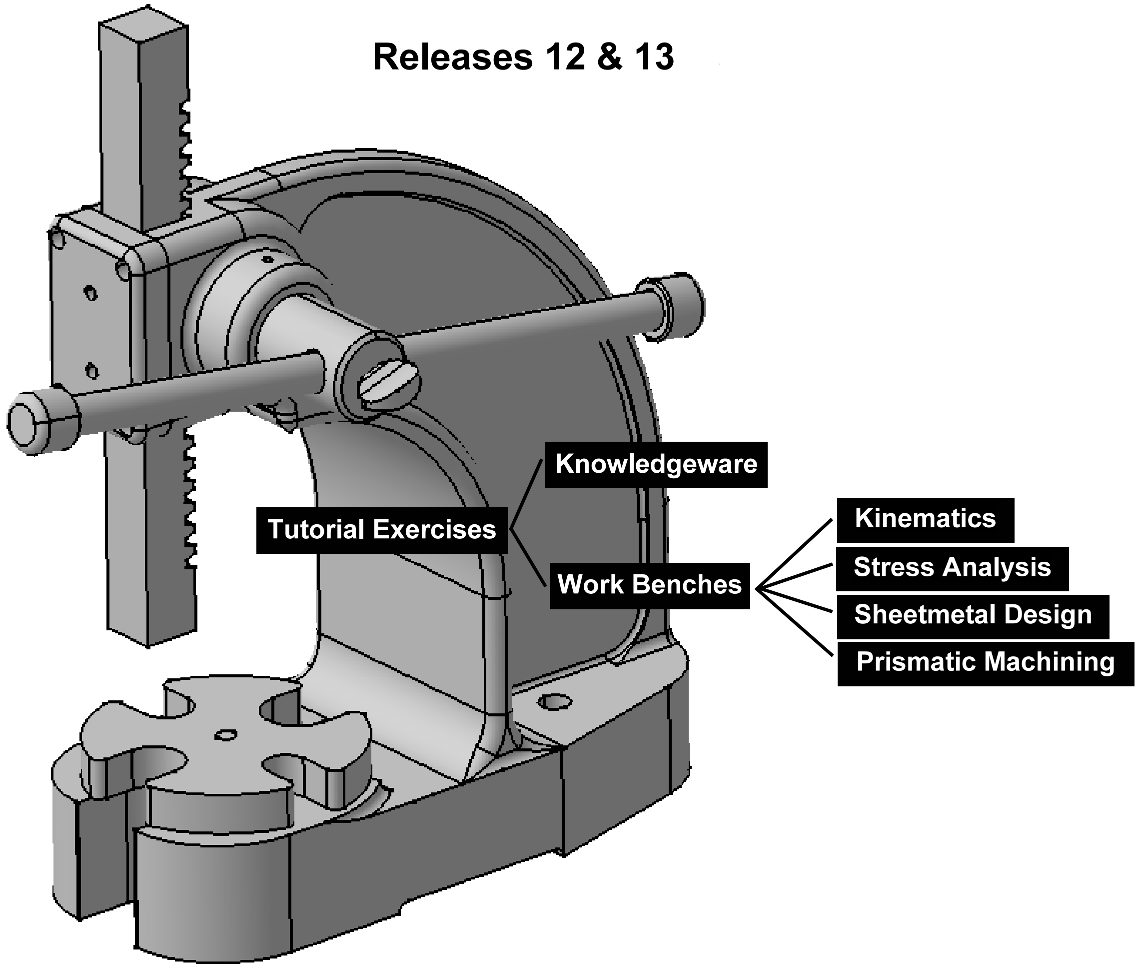
ADVANCED

CATIA V5 Workbook

Knowledgeware and Work Benches

Releases 12 & 13



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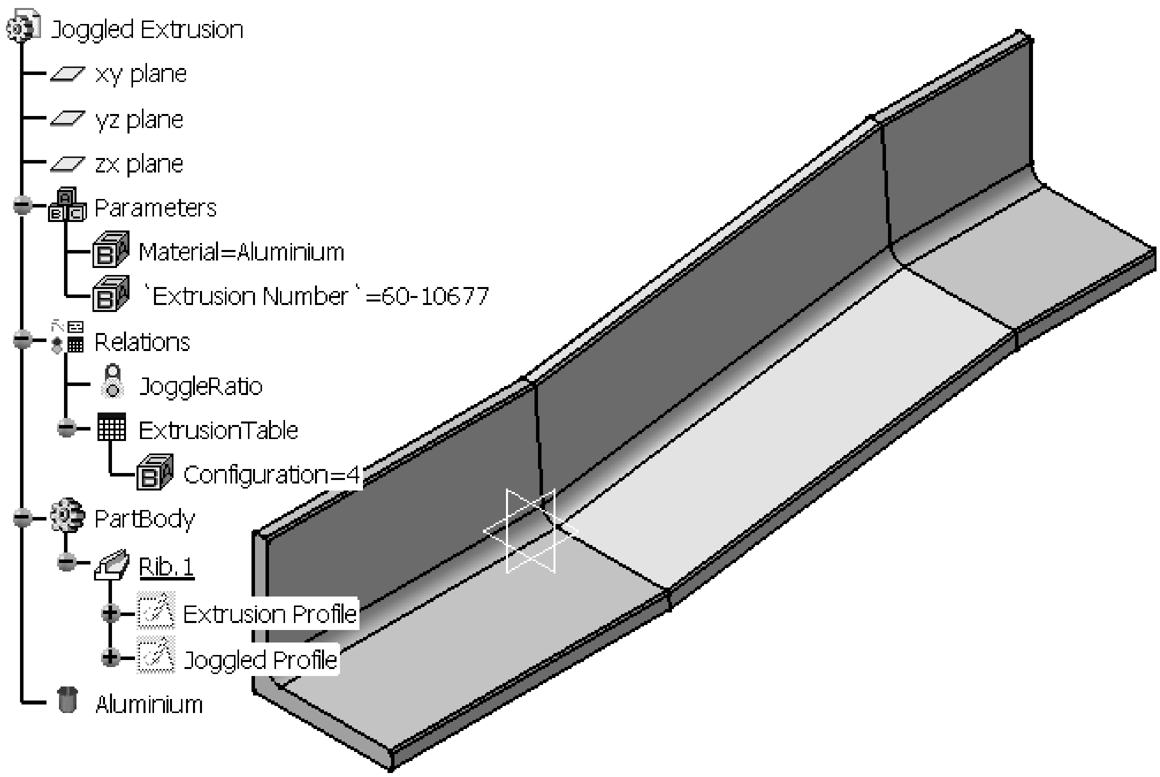
Introduction to CATIA V5 Knowledgeware

Knowledgeware is not one specific CATIA V5 work bench but several work benches. Some of the tools can be accessed in the **Standard** tool bar in the **Part Design** work bench. Simply put, **Knowledgeware** is a group of tools that allow you to create, manipulate and check your CATIA V5 creations.

Figure 1.1



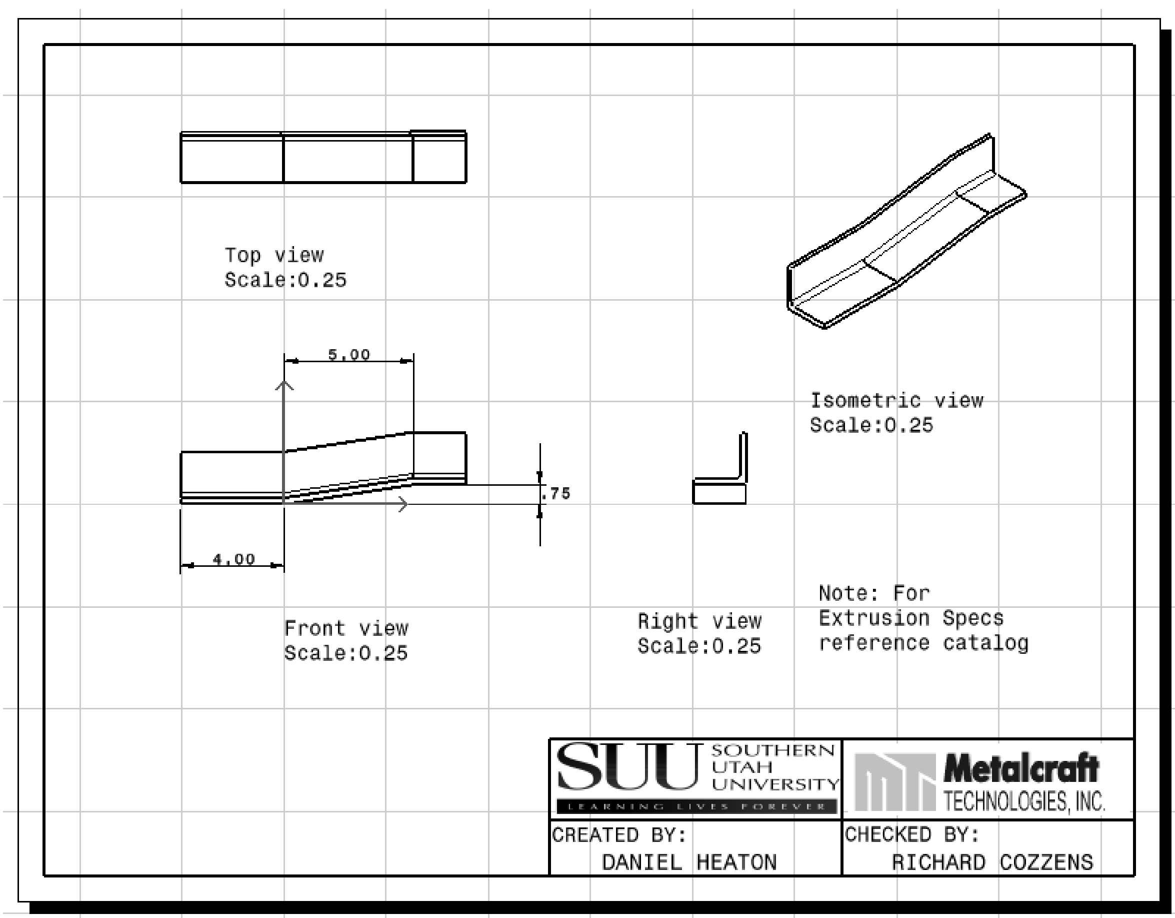
Lesson 1 Objectives

This lesson will take you through the process of automating the creation of joggled extrusions as shown in Figure 1.1. At the end of the lesson you should be able to do the following:

1. Create the **Extrusion Profile Sketch** and **Joggle Profile Sketch**.
2. Assign variable names to the required constraints.
3. Create the **Joggled Extrusion.CATPart** using the **Rib** tool.
4. Create a spreadsheet with aluminum extrusion dimensions.
5. Link the spreadsheet to the **Joggled Extrusion.CATPart**.
6. Apply the spreadsheet to update the **Joggled Extrusion.CATPart**.
7. Create a **Macro**.
8. Modify the **Macro** using **VB Script**.
9. Create prompt windows for input using **VB Script**.
10. Check for company/industry standards using the **Check** tool.
11. Implement the updated **Joggled Extrusion.CATPart** in a dimensioned drawing.

Figures 1.1 and 1.2 show examples of the **Joggled Extrusion** you will create in this lesson. Figure 1.1 shows the standard **Joggled Extrusion** along with its **Specification Tree**. Figure 1.2 shows a spreadsheet with the resultant dimensioned drawing.

Figure 1.2

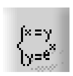


Knowledgeware Work Bench Tools and Tool Bars

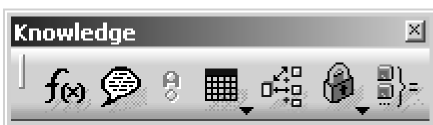
A combination of six tool bars is used in this lesson from the **Knowledgware Product**. The **Knowledgeware Product** is made up of the following work benches; **Knowledge Advisor**, **Knowledge Expert**, **Product Engineering Optimizer**, **Product Knowledge Template, Product Function Optimization** and **Product Functional Definition**. Each of these work benches has a different combination of tools in each tool bar. If you switch between any of these work benches you may see the same tool in a different tool bar. For example the **Formula** and **Design Table** tools are accessible from many workbenches in the bottom tool bar.

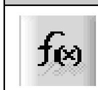
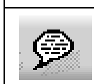
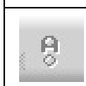
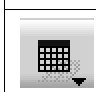
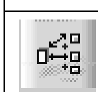
The Set of Equations Tool Bar

This tool bar contains only one tool.

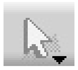
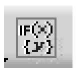
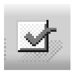
TOOL ICON	TOOL NAME	TOOL DEFINITION
	**Set Of Equations**	Solves a set of equations.

The Knowledge Tool Bar

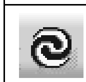


TOOL ICON	TOOL NAME	TOOL DEFINITION
	**Formula**	Creates parameters and determines the relationship between parameters.
	**Comment & URLs**	Adds URLs to the user parameters.
	**Check Analysis Toolbox**	Signals when there has been a violation in a check and/or rule.
	**Design Table**	Creates and/or imports design tables (spreadsheets).
	**Knowledge Inspector**	Queries a design to determine and preview the results of new parameters.

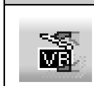
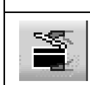
The <u>Reactive Features</u> Tool Bar

TOOL ICON	TOOL NAME	TOOL DEFINITION
	**Select**	Highlights the element you want to select.
	**Rule**	Creates a rule and applies it to your document.
	**Check**	Creates a check and applies it to your document.
	**Reactions**	Creates a script that will change feature attributes.

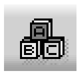
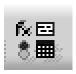
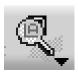
The <u>Tools</u> Tool Bar

TOOL ICON	TOOL NAME	TOOL DEFINITION
	**Measure Update**	Updates relationships.
	**Update**	Updates the CATPart and/or CATProduct.

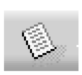
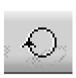
The <u>Actions</u> Tool Bar

TOOL ICON	TOOL NAME	TOOL DEFINITION
	**Macro with Arguments**	Opens a macro with arguments.
	**Actions**	Creates a script.

The Organize Knowledge Tool Bar

TOOL ICON	TOOL NAME	TOOL DEFINITION
	**Add Set of Parameters**	Creates a set of parameters.
	**Add Set of Relations**	Creates a set of relations.
	**Parameters Explorer**	Adds new parameters to a feature.
	**Comment & URLs**	Adds URLs to the user parameters.

The Control Features Tool Bar

TOOL ICON	TOOL NAME	TOOL DEFINITION
	**List**	Manage the objects you want to add to the list you are creating.
	**Loop**	Interactively apply a loop to an existing document.

The Problem:

One of the many Metalcraft Technologies Inc. (MTI) fabrication processes is fabricating a joggle in standard and non-standard extrusions. Most of the extrusion requirements are contained in large assembly mylar sheets. Most of the drawings (mylars) were created in the early 1970s. It is difficult for the engineer/planner to read and/or measure the mylar accurately. It may take the engineer/planner 10 to 30 minutes to verify he/she has found and applied the correct dimensions. It is not productive for the fabricator to also have to go through the same time consuming process. Having the drawing interpreted so many times by so many different people will inevitably introduce more chances for error. It is MTI's policy that the engineer/planner creates an individual drawing for each joggled extrusion to avoid such confusion. MTI has minimized the time required to create the individual drawings by setting up templates and standards. Yet, even with templates and standards this process is still time consuming. Each drawing is basically the same but has to be re-created because of a few simple dimensional differences and/or a different type of extrusion. The goal was to cut this time down by using the intelligence contained in the existing standard extrusion.

The Solution:

CATIA V5 **Knowledgeware** tools allow the user to capture and use the intelligence contained within the standard **Joggled Extrusion.CATPart**. CATIA V5 macro and scripting capabilities allow the user to be prompted for the critical dimensions. CATIA V5 then takes the information and updates the **Joggled Extrusion.CATPart** according to the supplied input. CATIA V5 also automatically updates the standard dimensioned drawing (CATDrawing). The dimensioned drawing is ready to be released to the production floor in a matter of minutes instead of 30 to 60 minutes.

An additional advantage to this process is adding dimensional checks. If the dimensional values do not match the company and /or industry standards the user will get a warning.

The following instructions will take you through the steps of creating the standard **Joggled Extrusion.CATPart** and then implementing the **Knowledgeware** solution described above.

Steps to Implementing the Knowledgeware Solution

A parameterized sketch/solid is a basic form of **Knowledgeware**; it contains intelligence. Prior to parametric applications you would have to create each variation of the extrusion from scratch. Parametric applications allow you to modify one constraint and the extrusion (solid) will update to that constraint.

1. Determine the Requirements

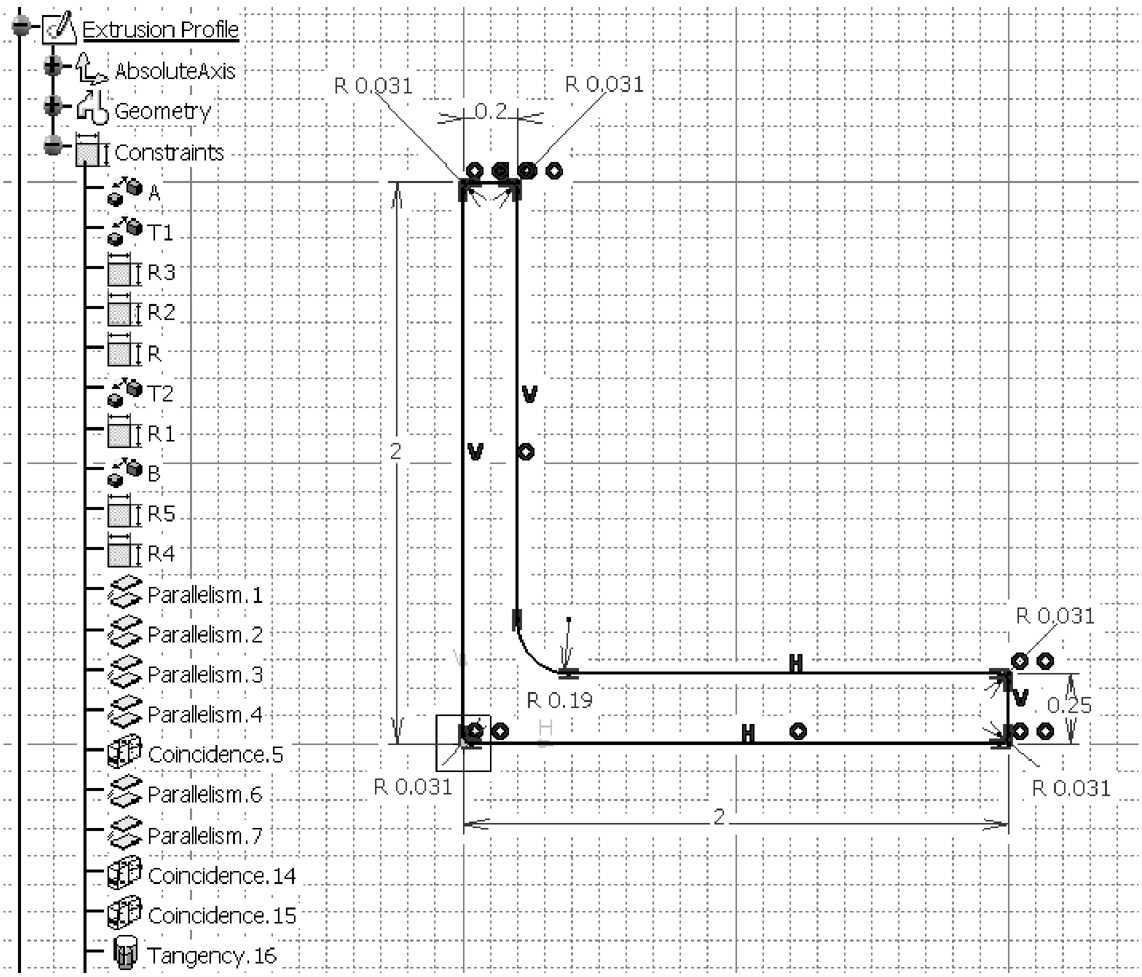
The general problem solving skills apply to implementing the **Knowledgeware** solution. You need to list all that is known and unknown and you need to list all of the variables, for example, what is known.

If you are not sure at first, manually go through the process. You must be able to create the process manually.

2. Creating the Extrusion Profile Sketch

Create an **Extrusion Profile** sketch on the **ZX Plane** as shown in Figure 1.3. The 0,0 point is located at the lower left corner of the extrusion. This sketch will be used as the standard; all other extrusions will be derived from this basic sketch. When you complete the sketch, exit the **Sketcher** work bench but do not use the **Pad** tool to create a solid. The solid will be created in Step 8 using a different tool.

Figure 1.3



3. Constraining the Extrusion Profile Sketch

After completing the rough sketch of the **Extrusion Profile** sketch as shown in Figure 1.3 you must constrain it similar to the constrains shown in Figure 1.3.

4. Modifying the Constraint Names

In this particular step it is critical that you rename the constraints. Understand that it is not absolutely necessary, but it will make this process a lot easier if you rename the constraints with a name that signifies what it is constraining. If you have problems remembering what the constraint name is, write it down; the names will be required to create the spreadsheet later in this lesson. It is suggested that you use the constraint names shown in Figure 1.4 so your information matches what you will see throughout the remaining steps into this lesson. Also, change the branch name **Sketch.1** to **Extrusion Profile**. Once you have successfully completed this lesson it is suggested that you try different variations of this process.

Figure 1.4

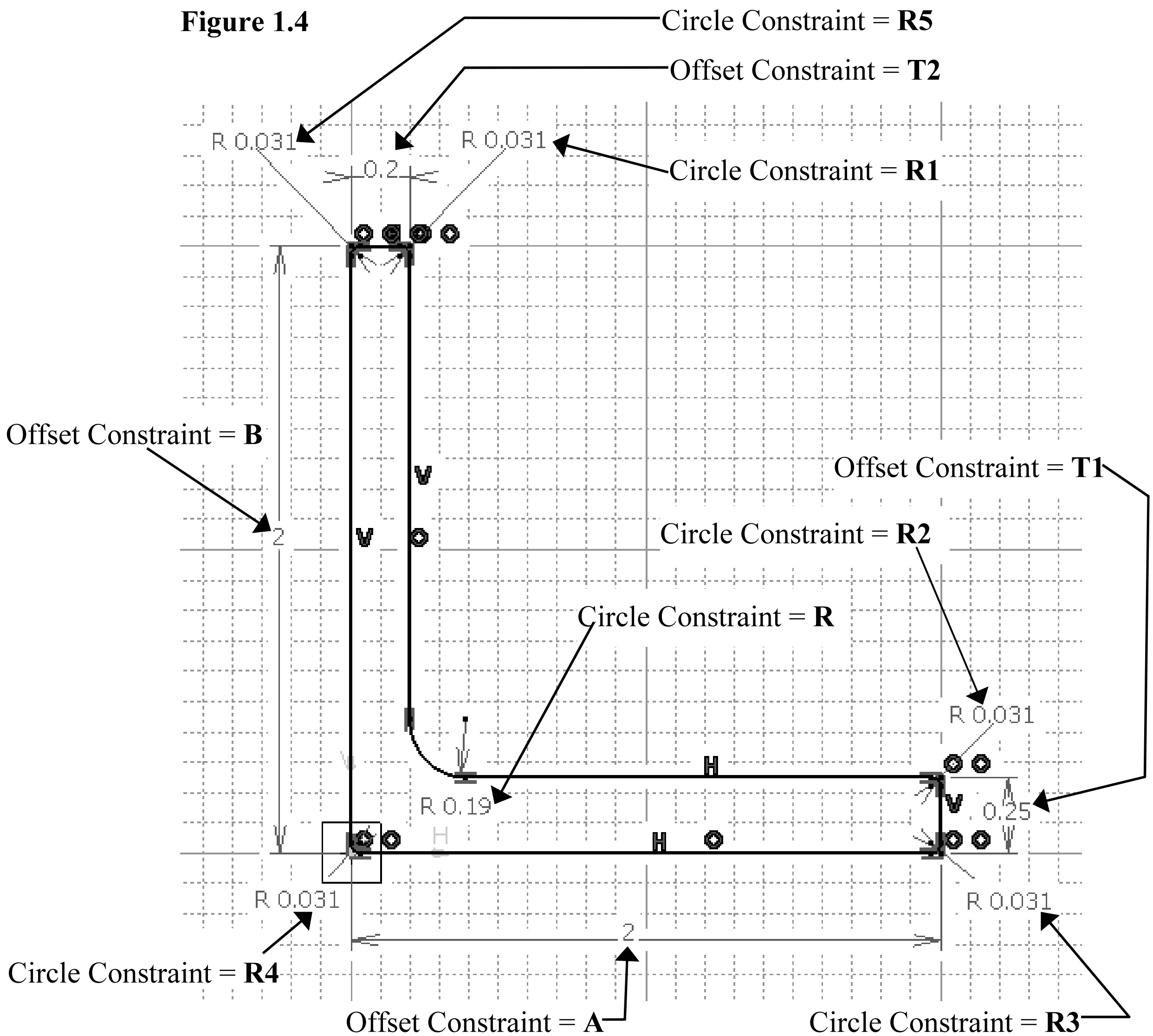
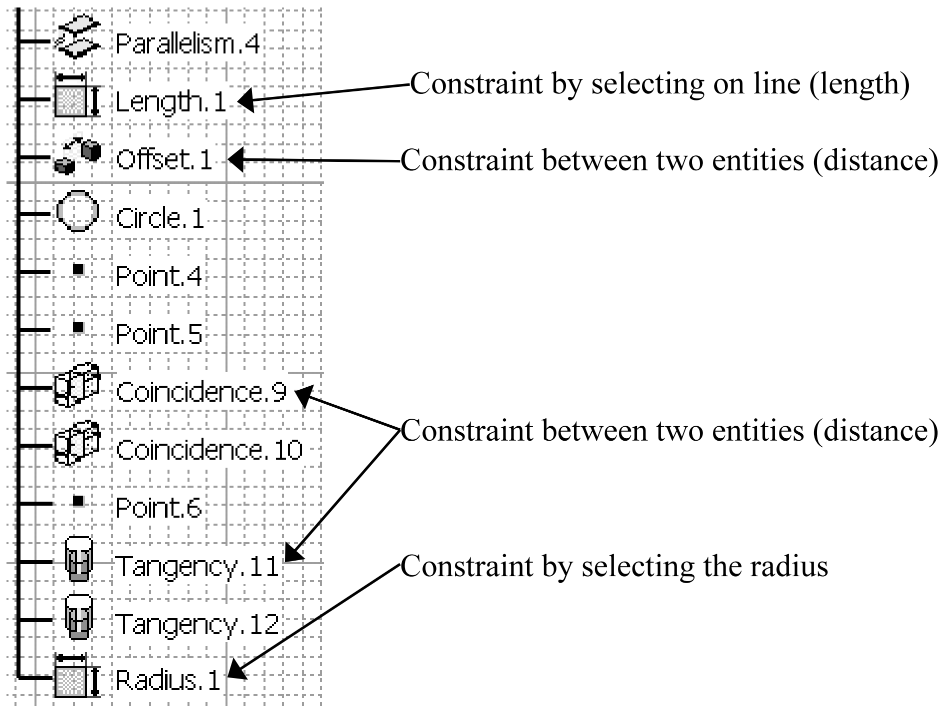


Figure 1.3 shows the constraints in the **Specification Tree** already renamed. CATIA V5 will automatically give it a name as shown in Figure 1.5 below.

Figure 1.5



Complete the following steps to rename the constraints.

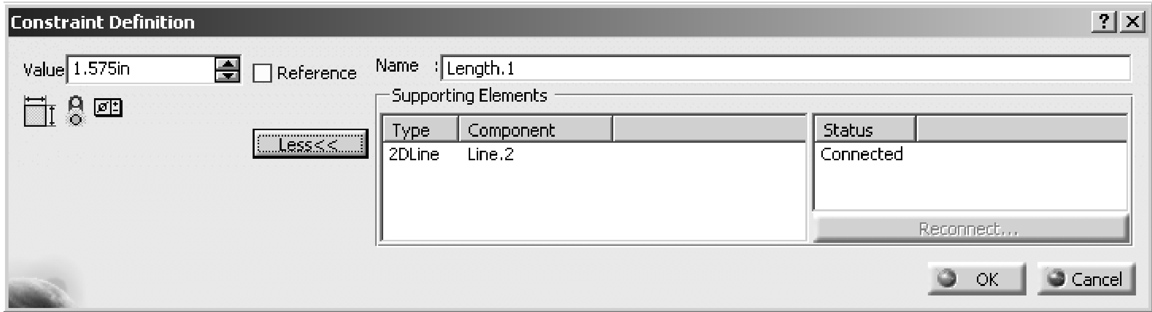
4.1 Double click on the constraint that you want to rename. This will bring up the **Constraint Definition** window with the constraint value in it.

4.2 Select the **More** button. This will bring up a **Constraint Definition** window as shown in Figure 1.6.

4.3 Edit the current constraint name in the **Name** box to what you want the new constraint to be named.

4.4 Select **OK**. The newly renamed constraint will show up in the **Specification Tree**.

Figure 1.6



5. Creating the Profile Sketch of the Joggle

This step, like Step 2, requires you to create another sketch. This sketch is created on the **YZ Plane** in the negative direction (notice where the → is located in relation to the sketch in Figure 1.7). Use the information in Figure 1.7 to create the **Joggle Profile** sketch.

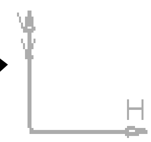
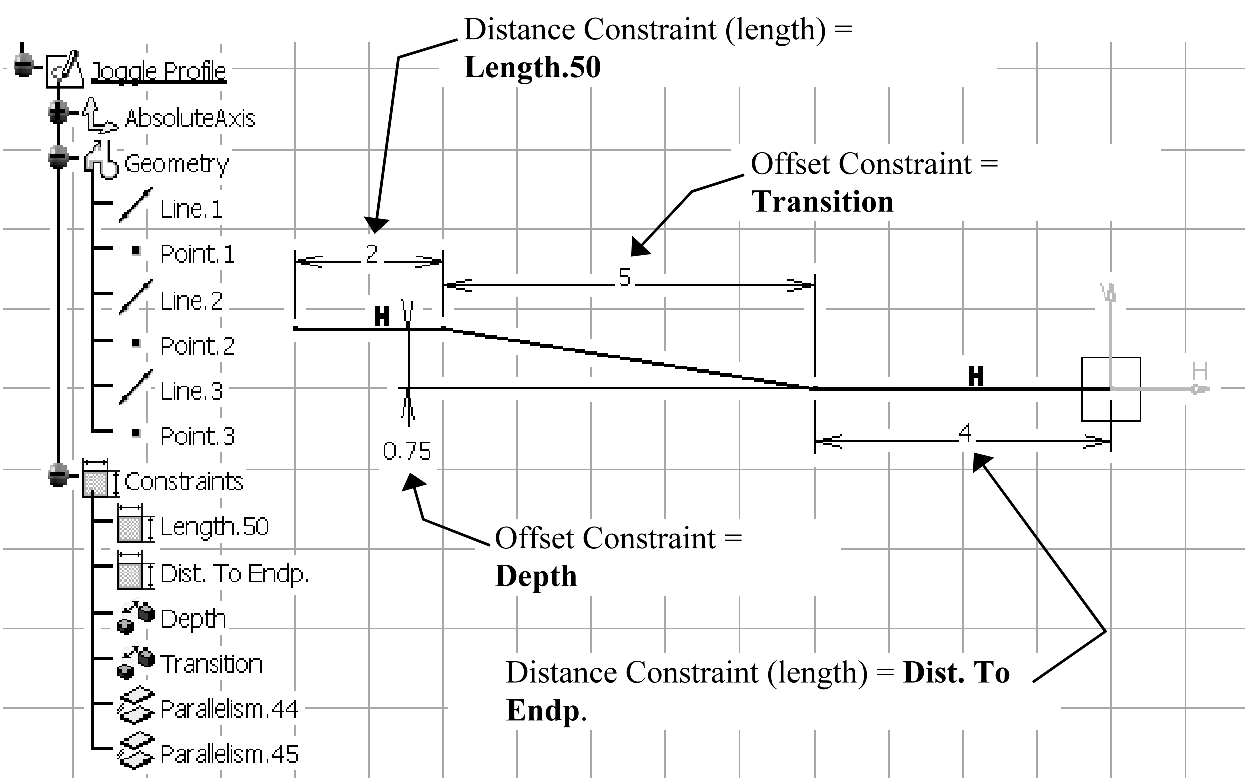


Figure 1.7



6. Constraining the Joggle Profile Sketch

Create constraints for the **Joggle Profile** sketch similar to the ones shown in Figure 1.7.

7. Modifying the Constraint Names

Modify the constraint names you created in Step 6 to match the constraint names shown in Figure 1.7. Step 4 describes the process of renaming constraints.

NOTE: It is important that the constraint names be consistent throughout this lesson. The names will be used to link the information to a table in the next few steps. If you deviate from the naming convention used in this lesson, the remaining steps will not work as described.

8. Creating a Solid of the Joggled Extrusion

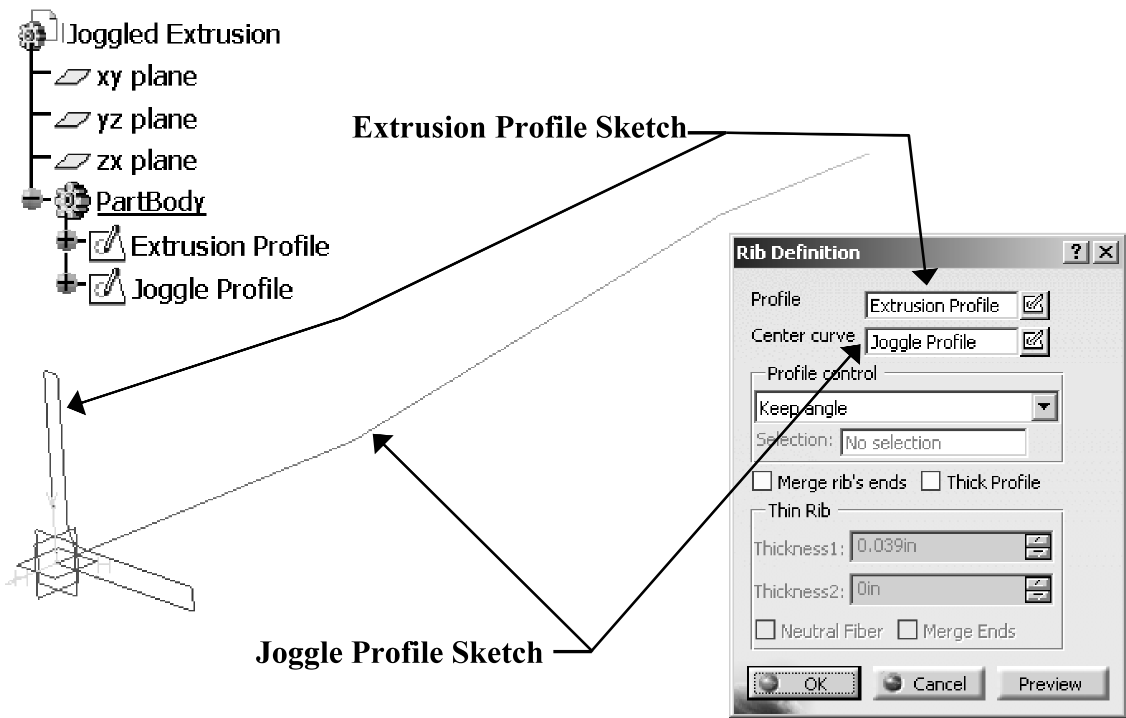
Now that both sketches are created you are ready to create the solid. This will be accomplished by using the **Rib** tool found in the **Part Design** work bench. Complete the following steps to create the solid

8.1 Select the **Extrusion Profile** sketch created in Step 2. Make sure it is highlighted.

8.2 Select the **Rib** tool found in the **Part Design** work bench. This will bring up the **Rib Definition** window as shown in Figure 1.8. The prompt zone will prompt you to **Define the center curve**. The **Extrusion Profile** will be listed in the **Profile** box.

8.3 The **Center Curve** box should be highlighted. Select the **Joggle Profile** either from the geometry or the **Specification Tree**. CATIA V5 will give you a preview of the **Extrusion Profile** being extruded along lines that define the **Joggle Profile** sketch.

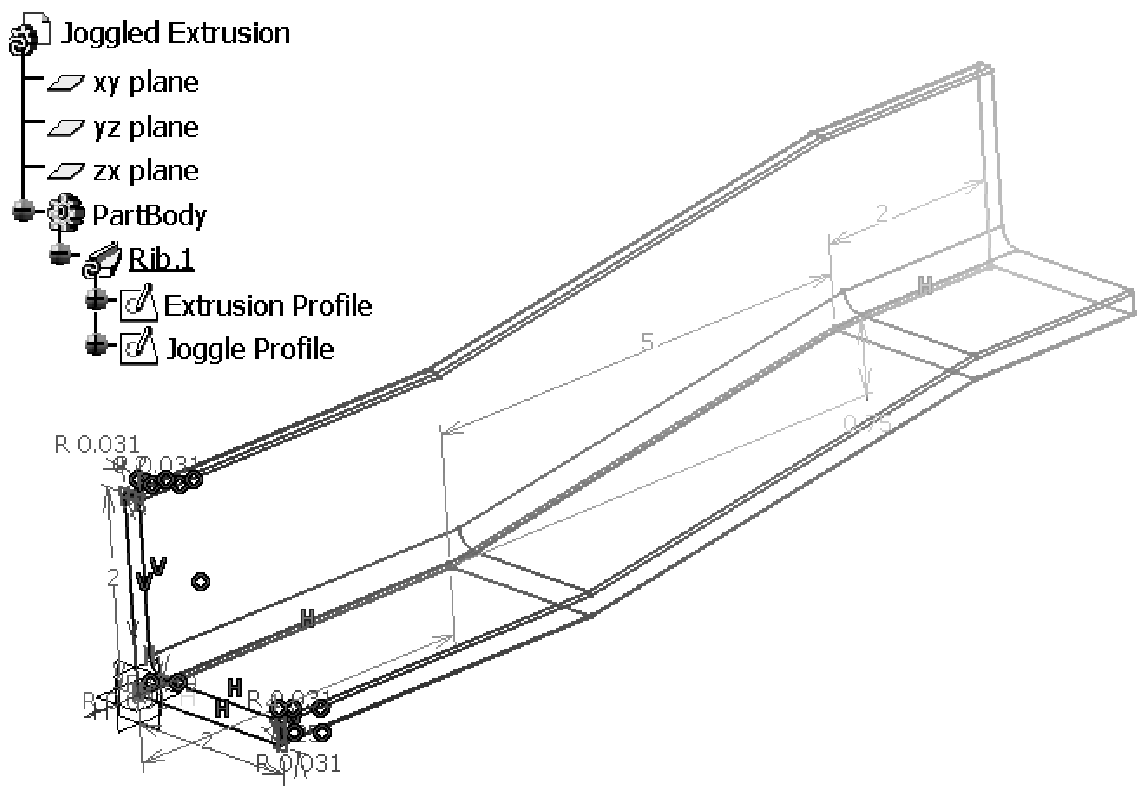
Figure 1.8



8.4 If the preview looks similar to the joggled extrusion that is shown in Figure 1.9, select the **OK** button to complete the operation. The **Joggled Extrusion** will be made into a solid.

Now that you have created a solid "Joggled Extrusion," you are ready to go on to the next step: creating a table of different types of extrusions.

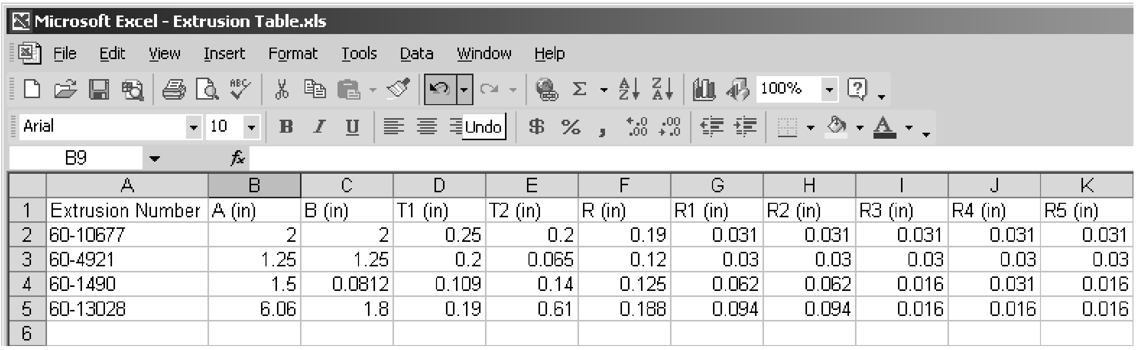
Figure 1.9



9. Creating an Extrusion Table

Figure 1.10 is an Excel (Spreadsheet) that contains the dimensions to four different types of aluminum extrusions. The extrusions and their dimensions were taken from the Tierany Metals Catalog. You might recognize the extrusion on row 5; it is the one you created in the previous steps. If you wanted to create the extrusion in row 2 you would have to start from step one again or you could go back to the **Extrusion Profile** sketch and revise the constraints. Obviously revising the constraints would be the quickest and easiest method to creating the new extrusion. CATIA V5 Knowledgeware tools can make this process even quicker and easier. This is accomplished by linking the Excel File to the CATPart.

Figure 1.10



	A	B	C	D	E	F	G	H	I	J	K
1	Extrusion Number	A (in)	B (in)	T1 (in)	T2 (in)	R (in)	R1 (in)	R2 (in)	R3 (in)	R4 (in)	R5 (in)
2	60-10677	2	2	0.25	0.2	0.19	0.031	0.031	0.031	0.031	0.031
3	60-4921	1.25	1.25	0.2	0.065	0.12	0.03	0.03	0.03	0.03	0.03
4	60-1490	1.5	0.0812	0.109	0.14	0.125	0.062	0.062	0.016	0.031	0.016
5	60-13028	6.06	1.8	0.19	0.61	0.188	0.094	0.094	0.016	0.016	0.016
6											

You can use an existing spreadsheet if it is available. If it is not available, you will have to create your own. The spreadsheet does not have to be an Excel program; any spreadsheet program will work. Each column requires a header. The header will be used as a variable link later in the lesson. Notice the column headers used in Figure 1.10 match the constraint names used in the previous steps to create the **Extrusion Profile** sketch. This is not absolutely necessary, but it does make the linking process much more intuitive.

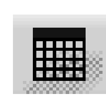
To complete this step, go into the spreadsheet program of your choice and enter the information in as shown in Figure 1.10. Save the file; preferably in the same directory that your CATPart file exists. Remember the file name and where it exists as you will need that information in the following step.

10. Importing the Extrusion Table

CATIA V5 allows you to create a design table inside CATIA V5 or import an existing design table. This step will show you how to import the design table created in Step 9. As you go through the process of importing a design table, you will be able to observe how CATIA V5 allows you the opportunity to create and modify a design table inside of CATIA V5. To import a design table, complete the following steps.

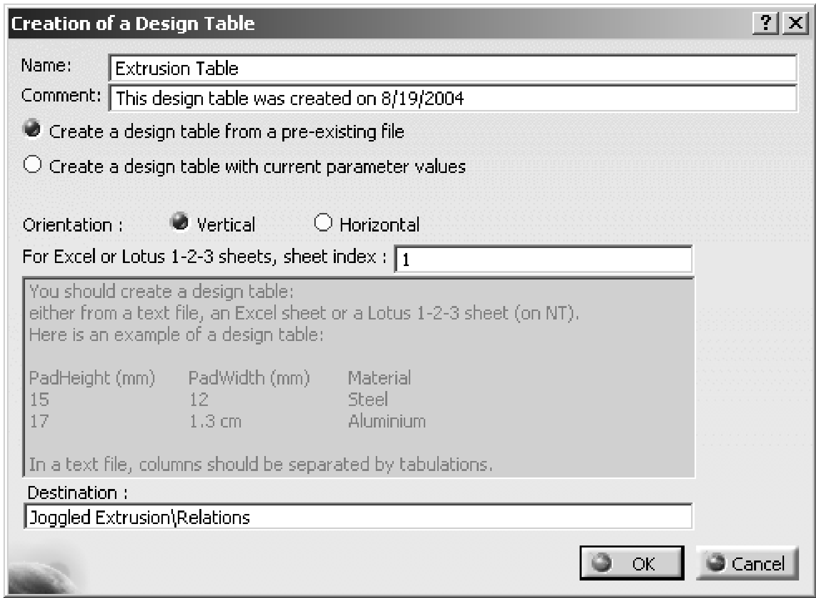
10.1 In the **Part Design** work bench, double click on the **Design Table** tool. The **Design Table** tool is located in the **Standard** tool bar at the bottom of the CATIA V5 screen. The **Design Table** tool icon is shown in Figure 1.11. This will bring up the **Creation of a Design Table** window as shown in Figure 1.12.

Figure 1.11



10.2 Name the design table "**Extrusion Table**" using the **Name** box as shown in Figure 1.12.

Figure 1.12



10.3 The **Comment** box will automatically place the date of creation. You can modify this box to any text that might help. This is just a comment box and will not have any effect on the following steps.

10.4 Select **Create a design table from a pre-existing file**. Although you will not use the other choice in this lesson it is important that you know that the other choice is available. The other choice is **Create a design table with current parameter values**. This choice allows you to create a design table inside CATIA V5.

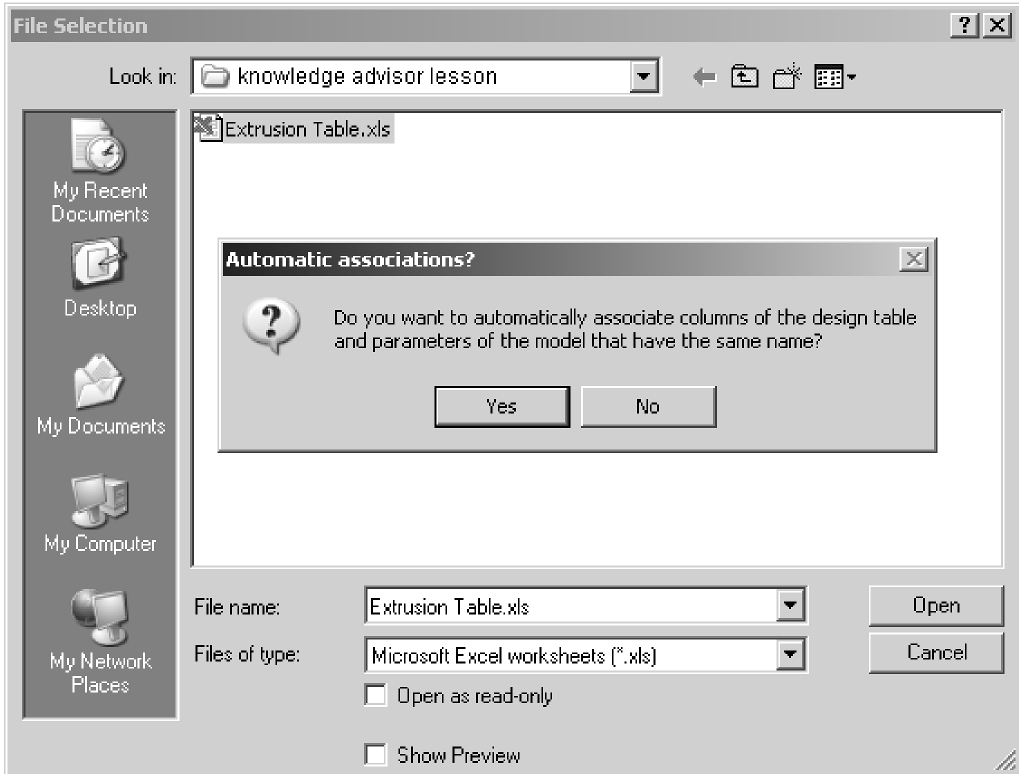
10.5 Select the **OK** button. This will bring up browser window labeled **File Selection**. This is the standard Windows file browser. Reference Figure 1.13.

10.6 Select the directory and the file that you want to import. For this step, you will want to select the **Extrusion Table** created in Step 9, as shown in Figure 1.13.

10.7 Select the **Open** button. This will bring up an **Automatic Associations?** window as shown in Figure 1.13. The prompt window asks if you want to automatically associate the parameters.

10.8 Select **Yes**. This will bring up the **Extrusion Table Active** window as shown in Figure 1.14. Note that Figure 1.14 is shown with the **Associations** tab selected, not the **Configurations** tab. If there are no associations listed in the **Configurations** box, CATIA V5 was not able to automatically associate any of the **Constraint Parameters** or **Extrusion Table Column Headings**.

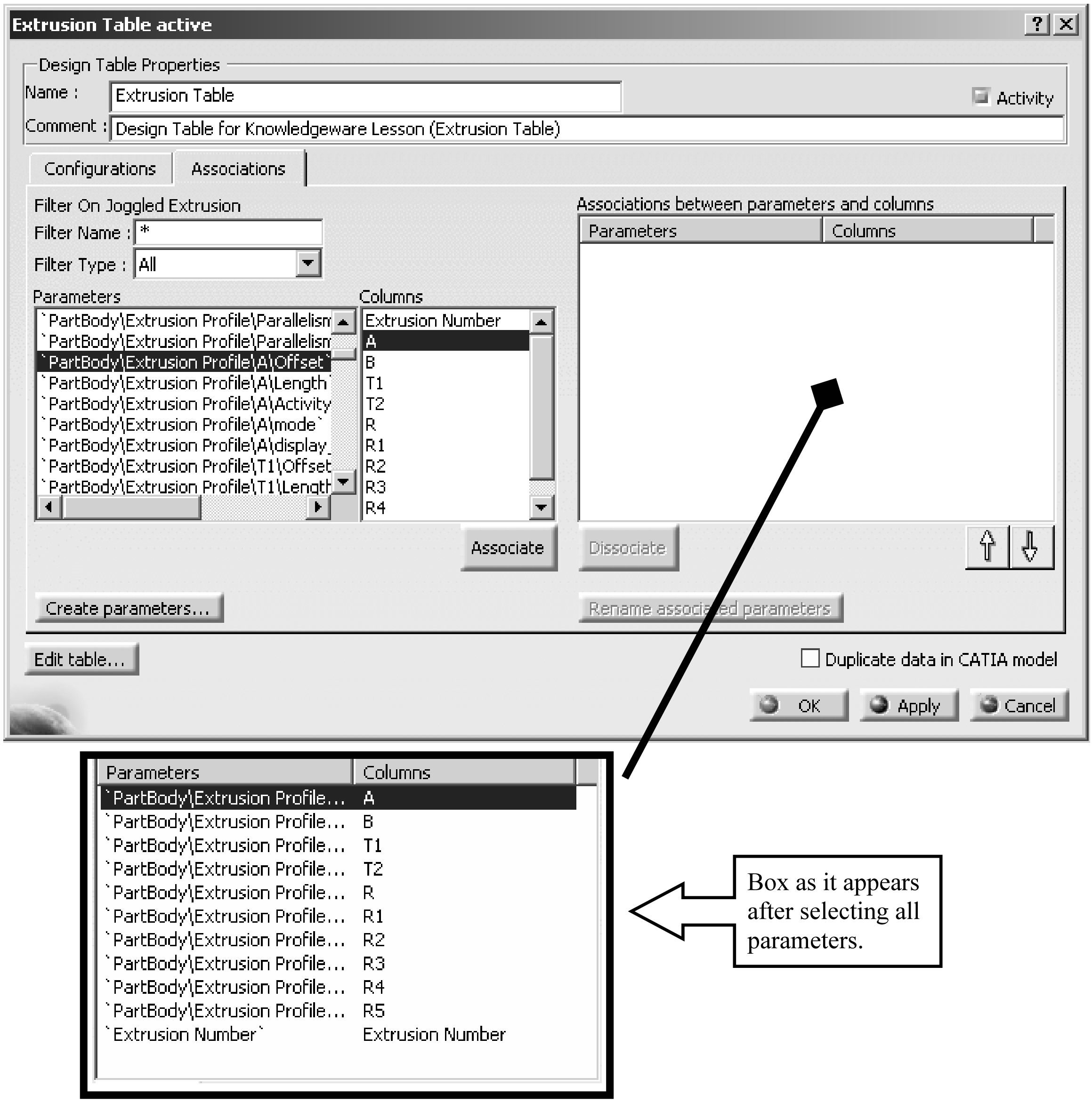
Figure 1.13



10.9 When CATIA V5 is not able to automatically associate the two together, you will have to manually associate them. To do this, select the **Associations** tab in the **Extrusion Table Active** window as shown in Figure 1.14.

10.10 The **Parameters** box lists all the parameters CATIA V5 created in the **Extrusion Profile** sketch. A CATIA V5 sketch contains a lot of parameters that the users are not usually aware of. What makes it more difficult, is the CATIA V5 naming convention. It is difficult to identify a CATIA V5 parameter listed in this box to an actual parameter in the **Extrusion Profile** sketch. This is where renaming the constraints in the previous steps will prove to be beneficial. You should be able to scroll through the **Parameters** box and identify the constraints you renamed. All the parameters are represented on two separate lines. For this lesson you will use the line that ends with a type of measurement such as **Radius**, **Offset** or **Length**. You will not use the line ending in **Activity**. For this step, scroll through the **Parameters** list; verify the constraints you renamed in Step 4 are listed.

Figure 1.14



10.11 Select **A** from the **Columns** box.

10.12 From the **Parameters** box, find and select the line '**PartBody\Extrusion Profile\A\Length**'.

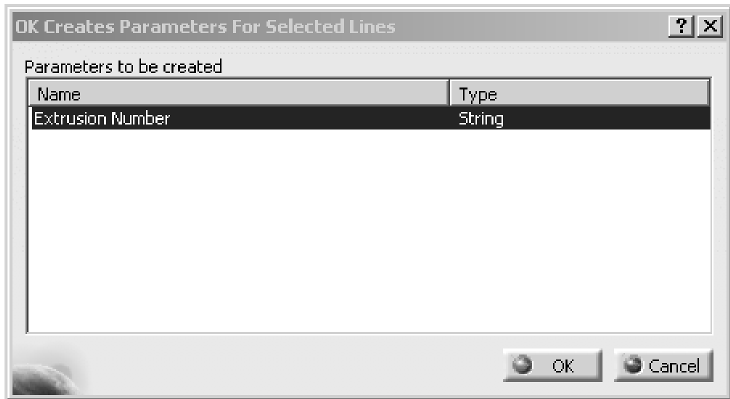
10.13 Select the **Associate** button. Your two selections from the **Parameters** and **Columns** boxes will show up in the **Associations between parameters and columns** box. This means that they were successfully associated.

10.14 Continue this process until all the variables in the **Columns** box, except for **Extrusion Number**, is matched up to the appropriate parameter. (**R**, **R1**, etc. will of course be a **Radius** rather than a **Length**).

10.15 Now you can take care of the **Extrusion Number** column heading. The **Extrusion Profile** sketch has no associative value to the **Extrusion Number** that was created in the **Extrusion Table**. You can assign it one by selecting the **Extrusion Number** in the **Columns** box.

10.16 Select the **Create Parameters…** button. This will bring up the **OK Creates Parameters for Selected Lines** window as shown in Figure 1.15.

Figure 1.15



10.17 Make sure **Extrusion Number** is selected/highlighted.

10.18 Select the **OK** button. This will create an association of a string type to the **Extrusion Number** heading. The association will be displayed in the **Extrusion Table Active** window under the **Associations** tab along with all the other associations you created in this step. What this really does for you is allows the **Specification Tree** to show the **Extrusion Number**. Figure 1.16, under the **Parameters** branch, displays **'Extrusion Number' =60-10677**. The string of numbers **60-10677** is linked from the specific row in the **Extrusion Table**. If you select another row (extrusion) from the **Extrusion Table** the **Specification Tree** will reflect the change just as the solid does.

NOTE: In order for the parameters to show up in the **Specification Tree** you must have the **Options** set correctly. Step 13 will show you how to set the correct options.

10.19 Select the **Configurations** tab in the **Extrusion Table Active** window. If you correctly associated the **Parameters** and **Columns**, it should look similar to the table shown in Figure 1.16. If your window looks similar to the one shown in Figure 1.16, select the **OK** button to complete the association process.

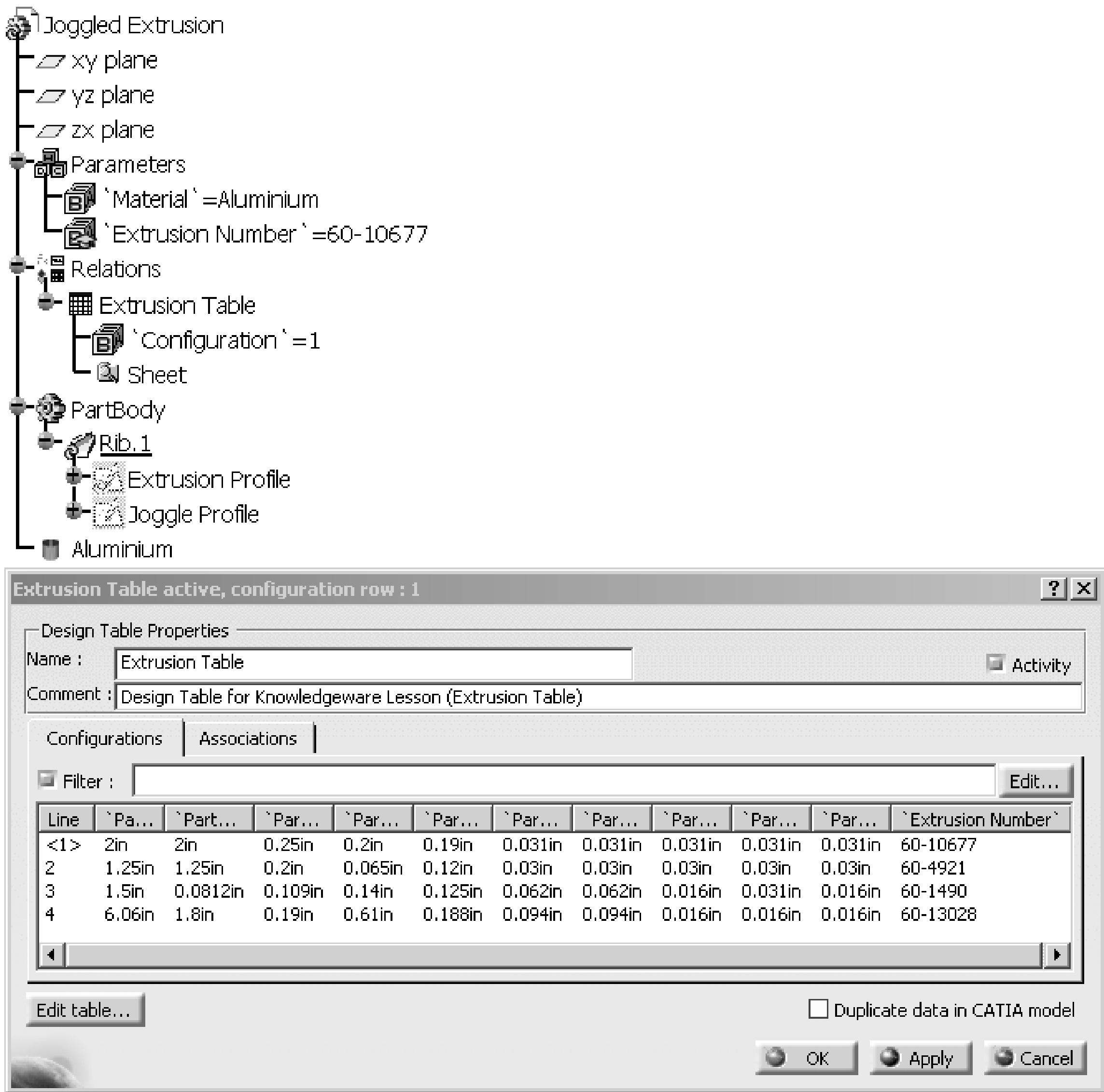
10.20 Doing this will make the window disappear and **Extrusion Table.1** shows up on your **Relations** branch of the **Specification Tree**. You may wonder what else is different. What did you just accomplish? Step 11 will show you the advantages of what you just accomplished.

11. Applying the Extrusion Table to the Joggled Extrusion

The purpose for linking a design table to the CATPart file is to update the part without having to redraw and/or revise the constraints manually. (Keep in mind that if you move your saved table, it will break the link and you will need to re-link it.) To test this, complete the following steps.

11.1 Double click on **Extrusion Table** in the **Specification Tree**. This will bring up the **Extrusion Table Active** window as shown in Figure 1.16. The data in row 1 is currently the active row. There are several methods to tell which row of data is active.

Figure 1.16



11.1.1 The window label contains the information: **Extrusion Table active, configuration row: 1**.

11.1.2 Row 1 has brackets around it [**<1>**]. The inactive lines do not have the brackets around it.

11.1.3 One other method is to check the data against actual extrusion dimensions. Figure 1.16 and the entire product in the previous steps represent the data that is contained in row 1.

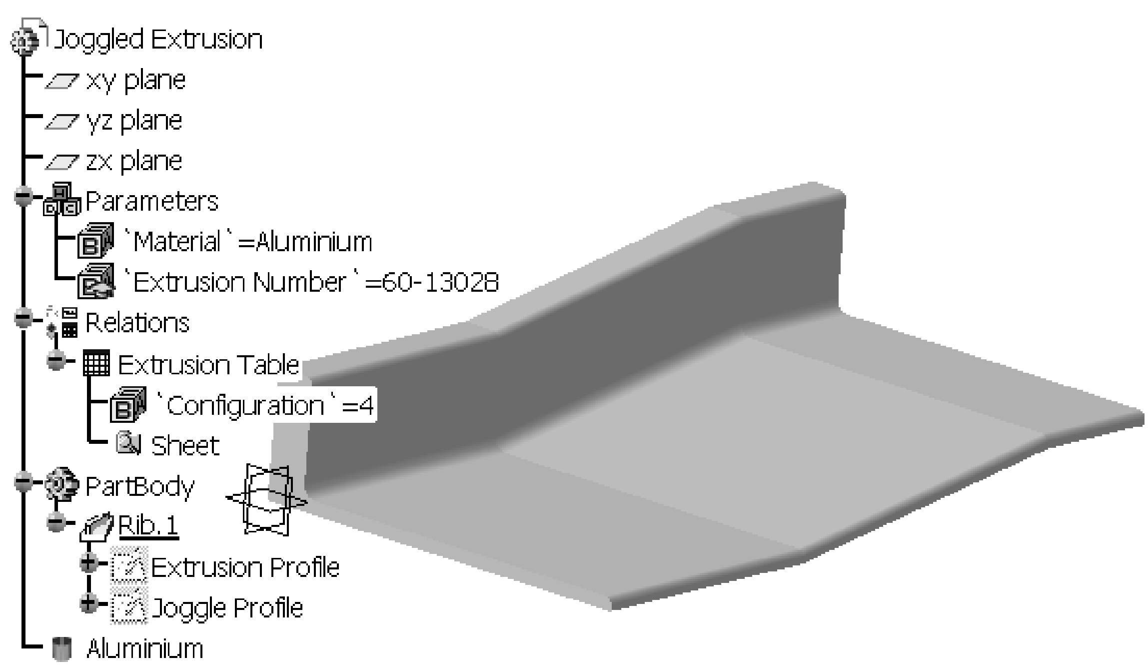
11.2 To make row 4 (Extrusion Number 60-13028) active, select the row. The existing extrusion will turn red signifying it needs to be updated.

11.3 Select the **OK** button. This will update your active extrusion to the data contained in row 4. Figure 1.17 shows the row 4 extrusion. Compare the differences between the extrusion represented in Figure 1.16 and 1.17. Verify the extrusions with the dimensions in the **Extrusion Table** (design table).

NOTE: If your extrusion does not automatically update you will have to select the **Update** button in the **Standard** tool bar section to force the solid to update. If you want CATIA V5 to automatically update select **Tools**, **Options**, **Infrastructure** branch, **Part Infrastructure** branch, **General** tab, **Update** section and select the **Automatic** button.

Once you link your **Extrusion Table** to your CATPart, updating is quite simple. Click on the **Extrusion Table** in the **Specification Tree** to bring up the design table. Select the row of data you want to apply to the CATPart and select **OK**. Be sure to select row 1 (Extrusion Number 60-10677) again before moving on to the next step.

Figure 1.17



12. Editing the Extrusion Table

You now have the **Extrusion Table** linked to the **Joggled Extrusion CATPart**. As the previous step demonstrated, creating new extrusions are only a few clicks away. Editing the **Extrusion Table** (design table) is just as easy. Modifying the **Extrusion Table** can be done in CATIA V5 or outside of CATIA V5. To modify the **Extrusion Table** inside of CATIA V5, complete the following steps.

12.1 Double click on the **Extrusion Table** in the **Specification Tree**.

12.2 This brings up the **Extrusion Table Active** window. Click on the **Edit Table** button at the bottom left of the window.

12.3 This brings up the original spreadsheet that the **Extrusion Table** was created in. Modify the number **2** in row 2 (Extrusion Number 60-10677) and column C (header B (in)) to a **4**.

12.4 Save and exit the revised spread sheet program. CATIA V5 will notify you that the **Extrusion Table** has been revised. Select **Close** to update the link.

12.5 The part will turn red because it is the active row. Select **OK** to update the **Joggled Extrusion CATPart**.

Your part is now updated to the information edited into the spreadsheet without leaving CATIA V5. You can use this method to add rows of new information, in this case additional extrusion types. You can also delete rows of information. The second method of revising the spreadsheet is editing the spreadsheet outside of CATIA V5. CATIA V5 will still give you a warning and a chance to accept or reject the revised spreadsheet.

NOTE: 2″ is correct for the 60-10677 Extrusion Number, so be sure to change it back.

13. Displaying the Extrusion Type in the Specification Tree

Figure 1.18 shows the **Specification Tree** without the value displayed, and Figure 1.19 shows the **Specification Tree** after the following process to display the value. Select **Tools**, **Options**, **Parameters and Measure** under the **General** branch, **Knowledge** tab, **Parameters Tree View** section; check the **With Value** box.

Figure 1.18

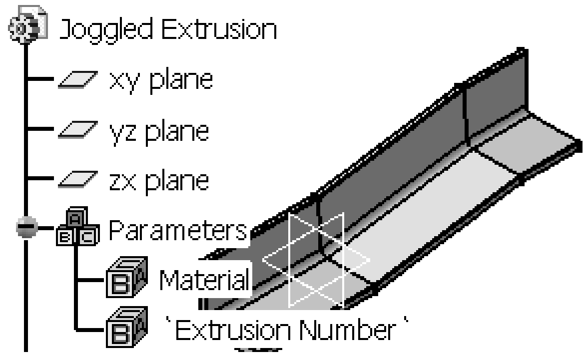
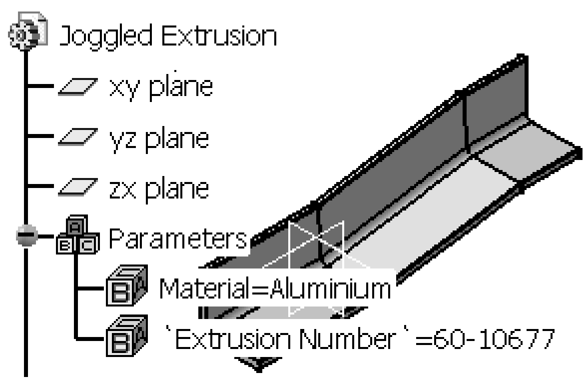


Figure 1.19



14. Modifying the Existing Joggle Profile Sketch

The previous steps showed you how to create, select and automate the creation of different types of extrusions. This was accomplished using a spreadsheet and the **Extrusion Profile** sketch. The following step will show you how to apply joggle information to the selected extrusion. This step uses/modifies the **Joggle Profile** sketch. If joggle information was standardized, you could create a spreadsheet with the required information and apply it to the **Joggle Profile** sketch as you did to the **Extrusion Profile** sketch. Joggle information is not standard; it is as varied as the parts and assemblies they are applied to. With the help of **Knowledgeware** this process can still be automated by getting information directly from the user in the place of the spreadsheet.

For this step, revise all the constraints in the **Joggle Profile** sketch to match the constraints shown in Figure 1.16.

The following steps will show you how this is accomplished. This step will start out real basic so you can better appreciate the power of CATIA V5's **Knowledgeware**. The **Joggle Profile** sketch controls the joggle of the extrusion. If you want to change the joggle depth, you could go into the **Joggle Profile** sketch and revise the constraint that controls the depth. Figure 1.7 shows that value of **Depth** is currently **.75″**. Entering the **Joggle Profile** sketch and modifying all the constraints for every individual part becomes very repetitious and time consuming. The following steps will show you how **Knowledgeware** can help you automate this process.

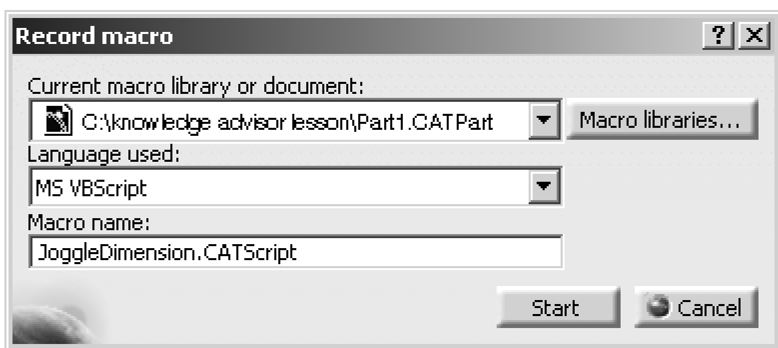
15. Automating the Modification Using a Marco

This step is similar to what was explained in Step 12. You go through the same steps except that you turn on the **Macro Recorder** to record everything you do. To accomplish this, complete the following steps.

15.1 Enter the **Joggle Profile** sketch, as shown in Figure 1.7. The first thing you need to remember is to record only what is necessary, other wise you get a lot of information that only complicates the process.

15.2 Select **Tools**, **Macro**, **Start Recording**. This will bring up the **Record Macro** window as shown in Figure 1.20.

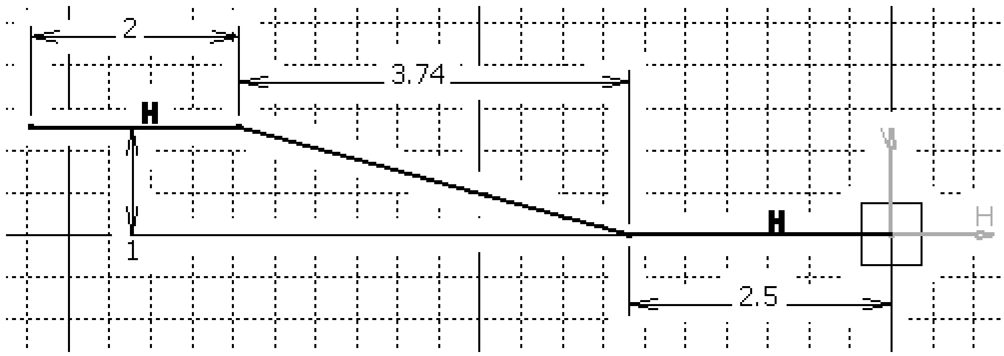
Figure 1.20



15.3 The **Current Macro Library Or Document:** should default to your CATPart at its designated saved location. Select **CATScript** for the **Language Used:** box.

15.4 Name the macro "**JoggleDimensions.CATScript**." CATIA V5 will default the name to **Macro1.catvbs** unless you specify a name. You must also add the **.CATScript** extension. Adding the extension allows CATIA V5 to save the macro externally not only as a macro but also a CATScript.

Figure 1.21



15.5 Before you start recording, make sure you know from start to finish what you are going to record. In this step you are going to modify all of the constraints in the **Joggle Profile** sketch to match the constraints shown in Figure 1.21. Select the **Start** button to start recording. Notice when you start recording, CATIA V5 creates a **Stop Recording** tool bar with a **Stop Macro Recording** tool on it.

15.6 Revise the constraints to match the constraints shown in Figure 1.21 in the following order: **.75** to **1.0** (Depth), **5** to **3.74** (Transition), and **4** to **2.5** (Dist. To Endp.).

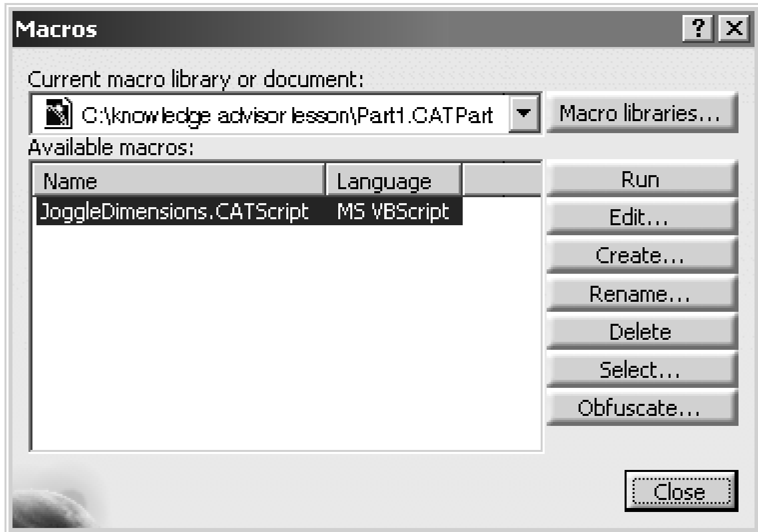
15.7 Stop the recording. You can stop the recording by selecting the **Stop Macro Recording** tool in the **Stop Recording** window explained in Step 15.4. Another method is to select **Tools**, **Macro**, **Stop Recording**.

15.8 Now go back to the **Joggle Profile** sketch and change the constraints to the previous values; the values shown in Figure 1.7.

15.9 Exit the **Sketcher** work bench.

15.10 Select **Tools**, **Macro, Macros**. This will bring up the **Macros** window as shown in Figure 1.22. Select the **JoggleDimensions.CATScript** macro.

Figure 1.22



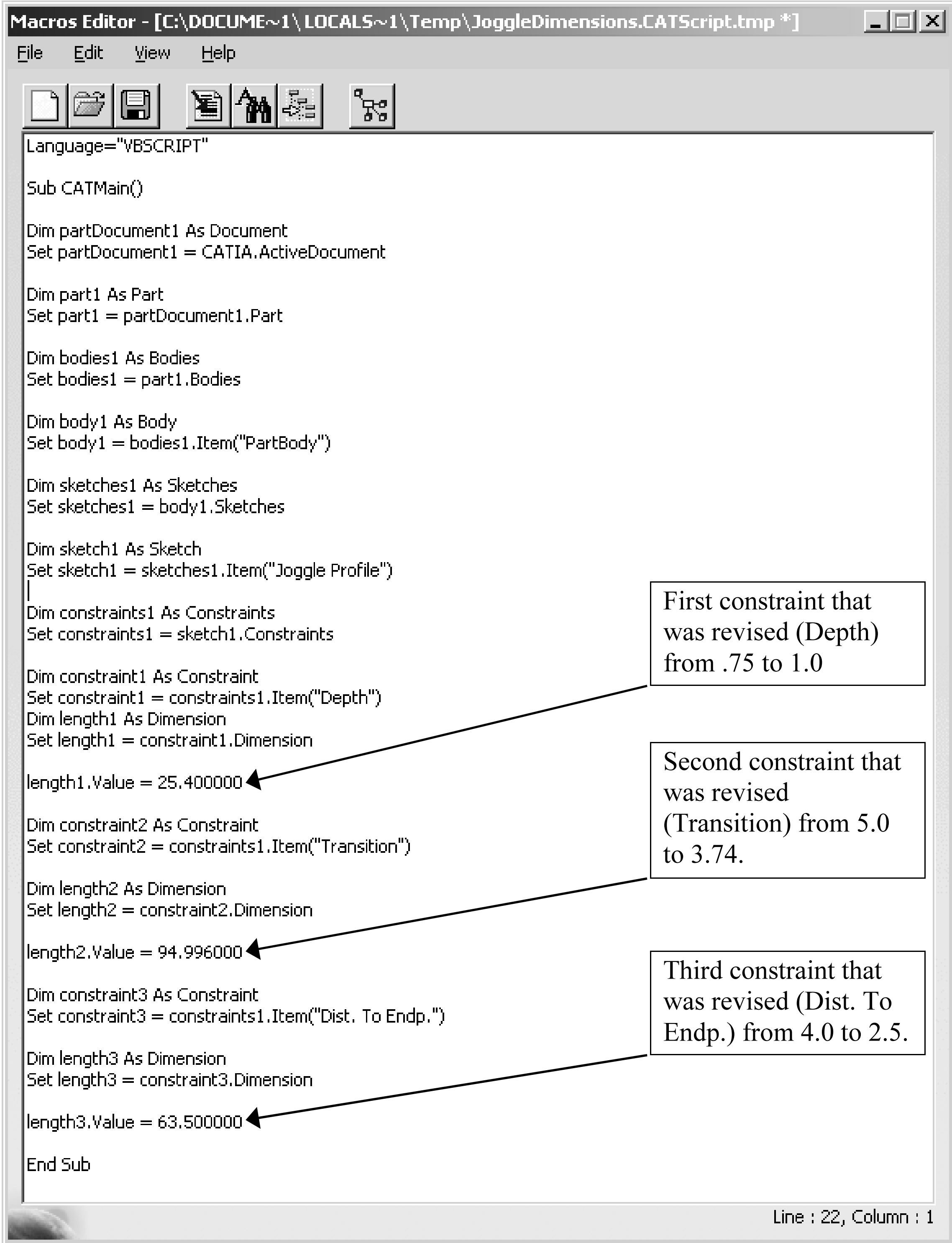
15.11 Select the **Run** button. This will run the **JoggleDimensions.CATScript** macro. Notice your **Joggled Extrusion.CATPart** will turn red and then update to the joggled dimensions you created in the macro.

15.12 The previous step demonstrates the result of the macro/script you just created. As you recorded the macro, CATIA V5 translated the action into the **VBScript Language**. CATIA V5 allows you to view and edit the scripted language. To view the **VBScript Language** you just created, select **Tools**, **Macro, Macros**, and then select the **JoggleDimensions.CATScript** file in the **Macros** window.

15.13 Select the **Edit** button. This will bring up the **Macros Editor** window shown in Figure 1.23.

The macro function is a powerful tool when it comes to accomplishing a process that is repeated over and over. The real power of the macro or CATScript you just created will be shown to you in the next step.

Figure 1.23




```
Language="VBSCRIPT"

Sub CATMain()

Dim partDocument1 As Document
Set partDocument1 = CATIA.ActiveDocument

Dim part1 As Part
Set part1 = partDocument1.Part

Dim bodies1 As Bodies
Set bodies1 = part1.Bodies

Dim body1 As Body
Set body1 = bodies1.Item("PartBody")

Dim sketches1 As Sketches
Set sketches1 = body1.Sketches

Dim sketch1 As Sketch
Set sketch1 = sketches1.Item("Joggle Profile")

Dim constraints1 As Constraints
Set constraints1 = sketch1.Constraints

Dim constraint1 As Constraint
Set constraint1 = constraints1.Item("Depth")
Dim length1 As Dimension
Set length1 = constraint1.Dimension

length1.Value = 25.400000

Dim constraint2 As Constraint
Set constraint2 = constraints1.Item("Transition")

Dim length2 As Dimension
Set length2 = constraint2.Dimension

length2.Value = 94.996000

Dim constraint3 As Constraint
Set constraint3 = constraints1.Item("Dist. To Endp.")

Dim length3 As Dimension
Set length3 = constraint3.Dimension

length3.Value = 63.500000

End Sub
```

First constraint that was revised (Depth) from .75 to 1.0

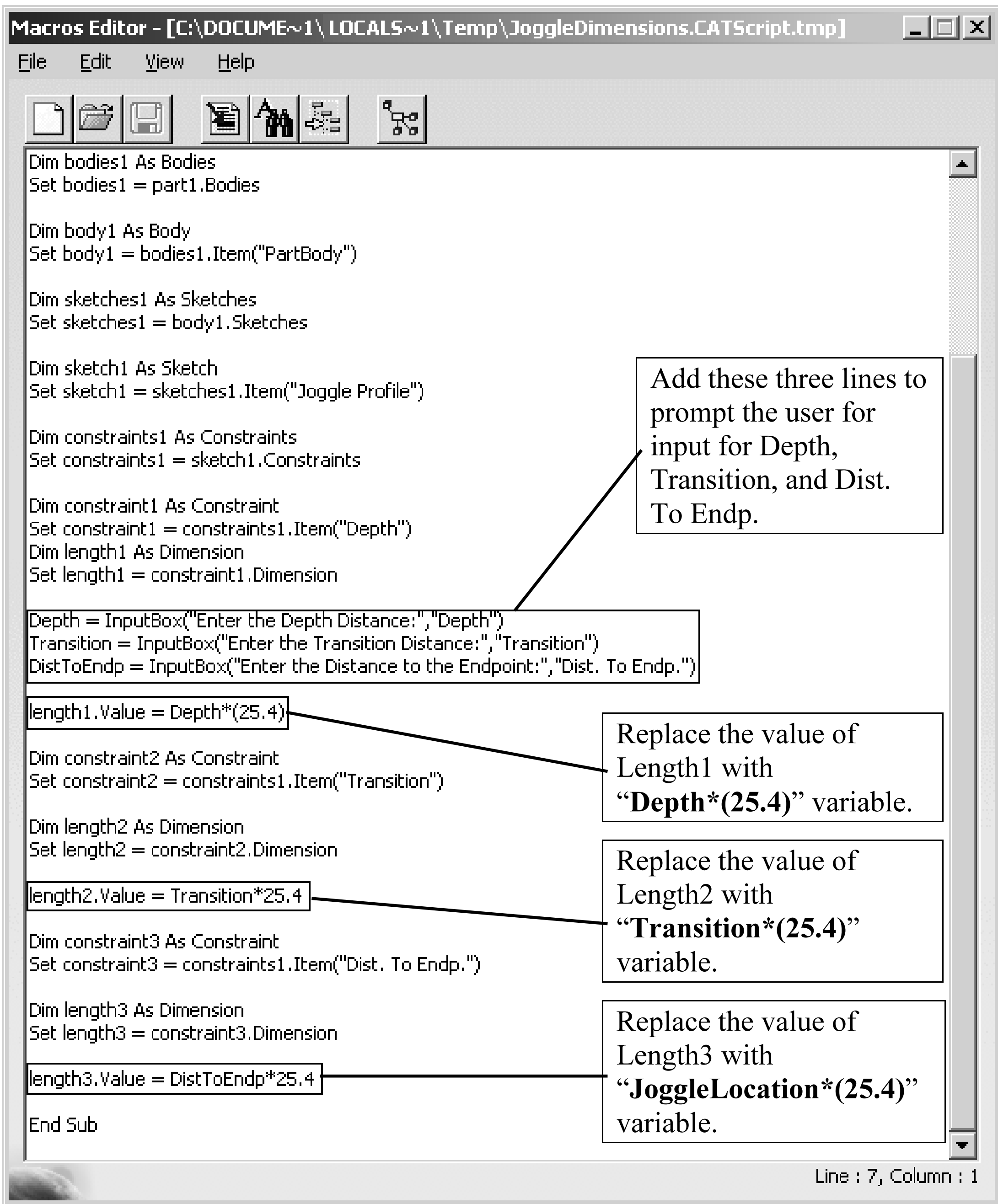
Second constraint that was revised (Transition) from 5.0 to 3.74.

Third constraint that was revised (Dist. To Endp.) from 4.0 to 2.5.

16. Customizing the Macro Using VBScript

CATIA V5 **Knowledgeware** allows you to customize the CATScript using **VBScript Language**. This customization makes the **Macro** and **Scripting** capabilities of CATIA V5 **Knowledgeware** almost limitless. You don't have to be a VBScript guru to take advantage of this tool, but obviously the more you know about it the more powerful a tool it becomes. This step will show you how to add the constraint variables you created in the **Joggle Profile** sketch in the previous steps. To accomplish this complete the following steps.

Figure 1.24



16.1 The macro you created in Step 15 assigned the constant value of **25.400000** to the first constraint you modified when recording the macro. The macro recorded the value as **length1.Value**. You will need to find the line where **length1.Value** is assigned the length. Figure 1.24 points out the approximate location of this line.

16.2 Insert the three lines indicated in Figure 1.24 above the line with the value as **length1.Value**. The purpose of doing this is to create prompt windows for the variables you are about to assign in place of the constant values that are assigned manually. Just adding variables would do you no good; you need some method of entering a value for the variables that you will create. The prompt window will allow the user to enter a value for the variable. (Make sure you type in the syntax exactly as it is shown.)

NOTE: It is obvious that each line represents a specific constraint variable. **Depth** is the variable. **InputBox** is VBScript syntax that creates a prompt window. **Enter the Depth Distance** is the text that will show up in the prompt window header. **Depth** at the end of the syntax creates a value input box.

16.3 Now back to the line that has the value as **length1.Value**. The macro converted the constant value to metric (mm). You will want to keep the units in inches so multiply the **Depth** value by **25.4**. The variable name used for the first constraint is **Depth*(25.4)**. This will convert the value back to inches. Reference Figure 1.24. Your modified line should look like similar to the line that is referenced.

16.4 Find the line that assigns **length2.Value**, the value you changed the constraint to in Step 15. Change the value to the variable constraint named "**Transition*(25.4)**." This variable needs to be converted back to inches as the one in Step 16.3 did. Reference Figure 1.24 to find the approximate location of the linen and for the way the revised line should look.

16.5 Find the line that assigns **length3.Value**, the value you changed the constraint to in Step 15. Change the value to the variable constraint named "**DistToEndp*(25.4)**." This variable also needs to be converted back to inches in the same method used in Steps 16.3 and 16.4. Reference Figure 1.24 to find the approximate location of the line and for the way the revised line should look.

16.6 Save the changes and then close the **Macros Editor** window. The **Macros** window will still be available - don't close it.

It is important that you understand the relationship between the constraint that you renamed in Step 5 and the variable name that you are editing into the VBScript file. If you get them mixed up you could be changing things in ways you didn't expect. You must be sure and follow the VBScript syntax or it will not work. When editing the lines, make sure they match the lines pointed out in Figure 1.24 exactly. Reference the CATIA V5 online help and/or a VBScript manual for more in-depth information on VBScript syntax.

17. Testing the Customized Macro

This step will take you through the process of updating your **Joggled Extrusion** by running the macro. This will be a good test to see if you have entered all the syntax as required.

17.1 The **Macros** window should still be on the screen. Select the **Run** button.

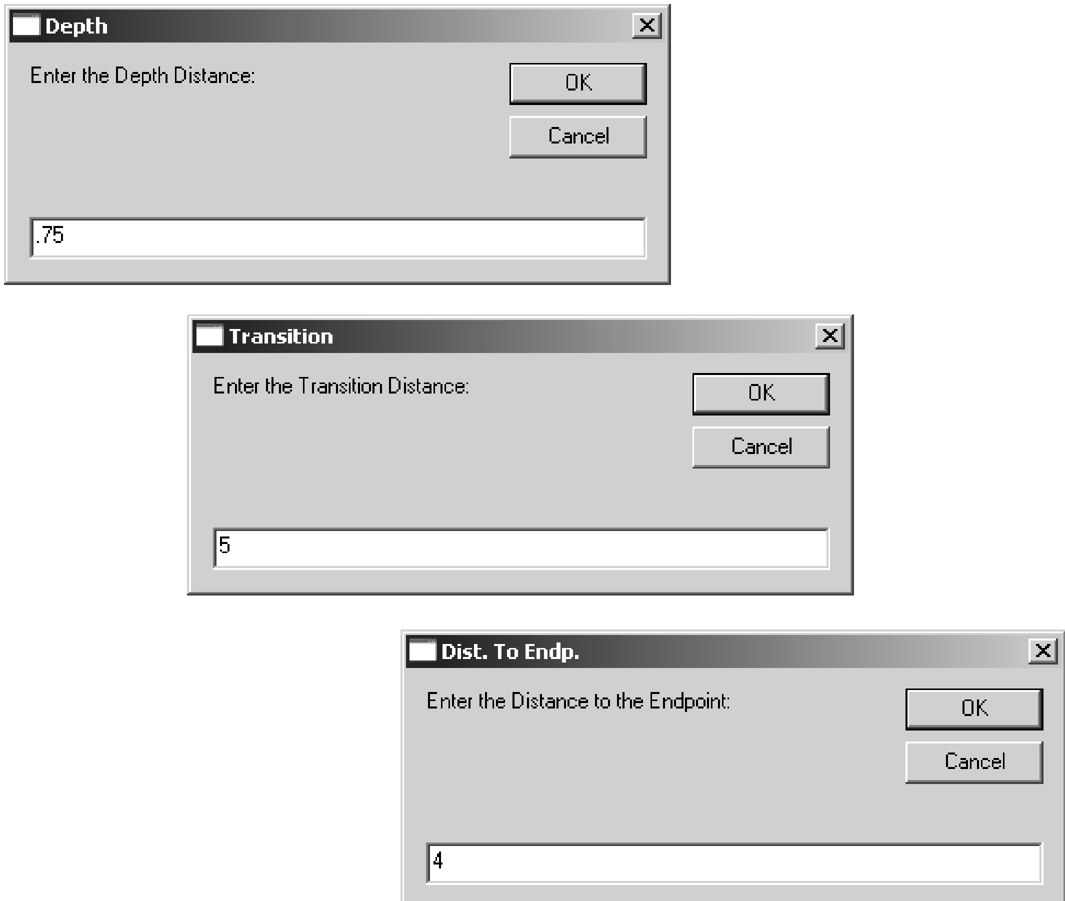
17.2 This should bring up the first of the three prompt windows that were created previously (**Depth**). Reference Figure 1.25. Type in the original value assigned in Step 5 (**.75**), then select **OK**.

17.3 This will take you to the next prompt window (**Transition**). Again type in the original value (**5**) and then select **OK**.

17.4 The last prompt window created (**Dist. To Endp.**) will appear. Type in the original value (**4**) and select **OK**.

17.5 If the syntax was set up correctly, your extrusion should update automatically or turn red to indicate updating is needed. Your extrusion should be back to its original configuration.

Figure 1.25



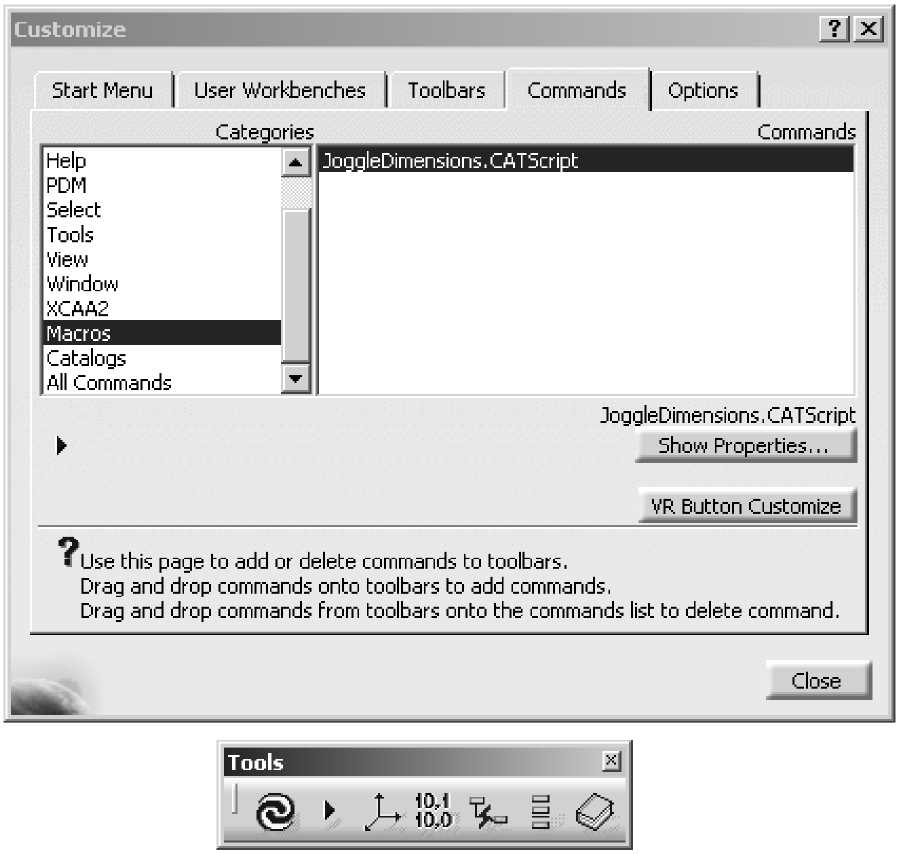
18. Creating a Tool Icon for the Macro

The macro (**JoggleDimensions.CATScript**) created and modified in the previous steps is a very powerful tool. CATIA V5 has developed another powerful tool that will save you additional time. This tool allows you to customize your CATIA V5 work environment by creating your own tool icons. Every time you wanted to run the **JoggleDimensions.CATScript** macro you could go through the same process of selecting **Tools**, **Macro**, **Macros,** select the macro, select **Run**, and finally be ready to run the macro; or, you could assign the macro a tool icon and just select the tool icon. Creating a tool icon for the macro would save five steps every time. The following steps show you how to assign a tool icon to the macro.

18.1 Select **Tools** and then select the **Customize** option

18.2 This will bring up the **Customize** window as shown in Figure 1.26.

Figure 1.26



18.3 Select the **Commands** tab in the **Customize** window.

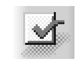
18.4 Select **Macros** from the **Categories** box. This will bring up all the macros that were created and saved with the ***.CATScript** extension.

18.5 Select the **JoggleDimensions.CATScript** located in the **Commands** box.

18.6 With the **JoggleDimensions.CATScript** file highlighted, drag it to the **Tools** tool bar. Drop the **JoggleDimensions.CATScript** on the tool bar as shown in Figure 1.26.

18.7 Close the **Customize** window and click on the newly created tool icon. This will start the **JoggleDimensions.CATScript** macro.

As you can see on the **Customization** window, CATIA V5 allows you many different ways to customize your CATIA V5 work environment. This step has shown you only one. Finding and using the tool from the tool bar is easier than going into the **Macro** option and searching for the macro.

19. Applying Correct Processes and Standards Using the Check Tool 

Currently the **JoggleDimensions.CATScript** macro will accept the value of **1″** for the **Transition** dimension and **5″** for the **Depth** dimension. Any experienced joggle operator would tell you that is not a reasonable ratio for an aluminum extrusion. A safe standard for aluminum extrusions is about **4** (run or **Transition**) to **1** (rise or **Depth**). This is a basic standard, but not every one is aware of it. It is very possible that the engineer/planner creating the drawing is not aware of the standard, thus could violate the standard. The engineer/planner could spend time planning and drawing the **Joggled Extrusion**. The part could use up time and resources being prepped for the joggle operation. Only after the extrusion gets to the joggle process would it be discovered that the joggle dimensions are not within company and/or industry standards. All of the time, material and resources have gone to waste. All of this could have been avoided if the engineer/planner was aware of the standard. One sure way to safeguard yourself and/or company from such mistakes is by incorporating the standard into the intelligence of the part. CATIA V5 offers you the tools to capture the knowledge and/or standard and apply it to your CATParts. The following step explains how to incorporate the **JoggleRatio** standard to your **Joggled Extrusion.CATPart**.

19.1 Double click on the **Relations** branch of the **Specification Tree**. This will bring up the **Knowledge Advisor** work bench.

19.2 Select the **Check** tool. This will bring up the **Check Editor** window as shown in Figure 1.27.

19.3 Label the check "**JoggleRatio**."

19.4 Select **OK**. This will bring up the **Check Editor : JoggleRatio Active** window as shown in Figure 1.28.

Figure 1.27

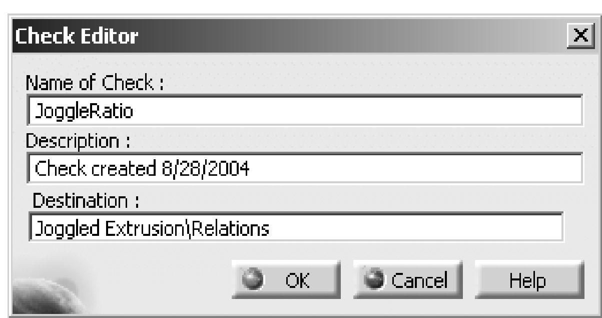
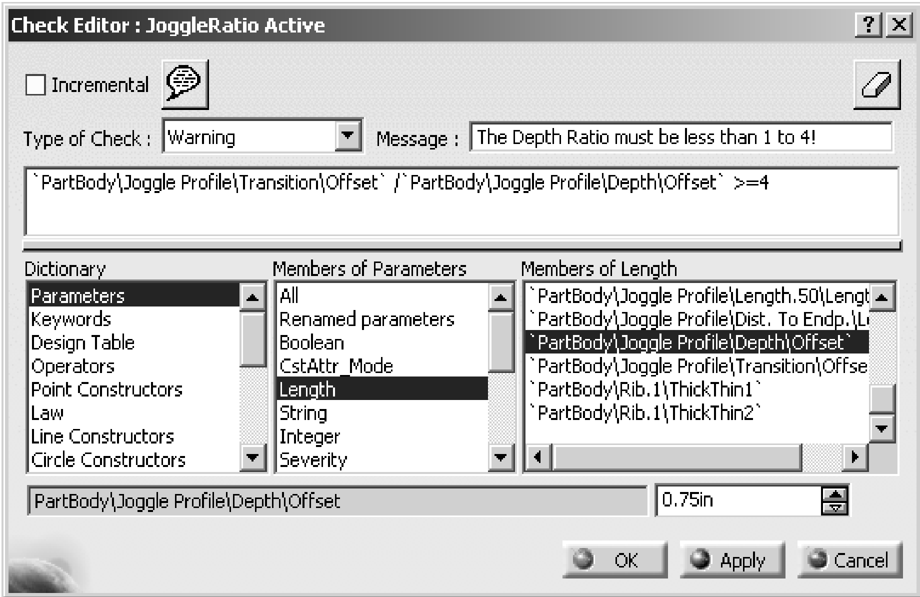


Figure 1.28



19.5 Select **Warning** as the **Type of Check**.

19.6 In the **Message** box, type "**The Depth Ratio must be less than 1 to 4!**"

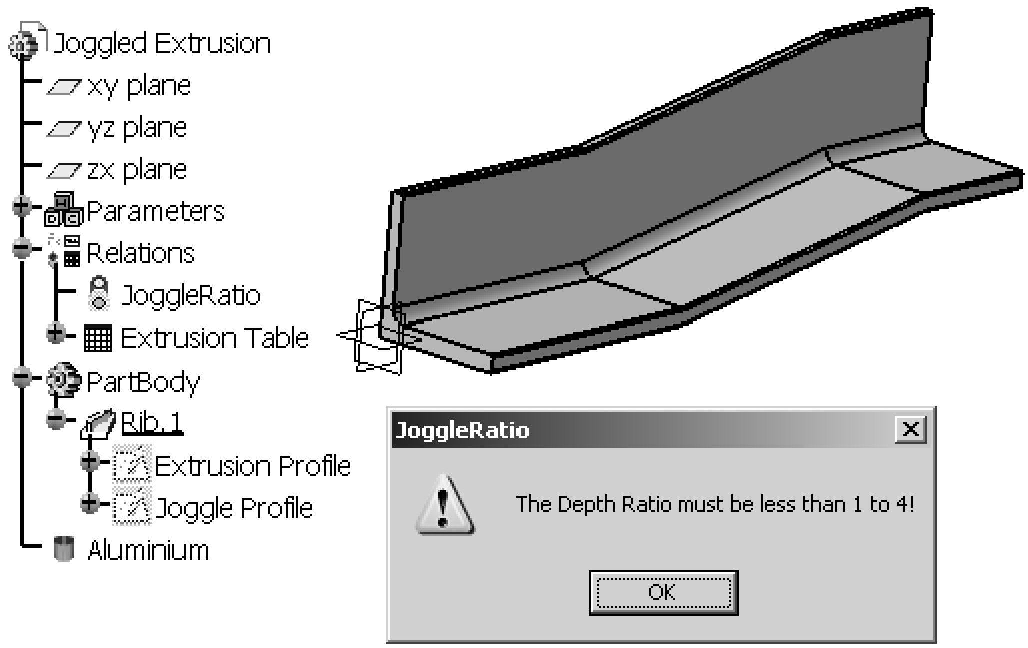
19.7 Under **Dictionary**, select **Parameters**.

19.8 In the **Members of Parameters** box, select **Length**.

19.9 In the **Members of Length** box, double click on the '**PartBody Joggle Profile\Transition\Offset**' parameter. This will copy it to the input box above it.

19.10 Type in the symbol for divide (/) after the inserted line. Reference Figure 1.28.

Figure 1.29



19.11 In the **Members of Length** box, double click the '**PartBody\Joggle Profile\Depth\Offset**' parameter. This will copy it to the end of the line you have been creating in the input box above it.

19.12 Type in **>=4** following the '**…\Depth\Offset**' parameter. Steps 19.9 through this step created a formula that tests the values the user enters when running the **JoggleDimensions.CATScript** macro. The formula needs to be the exact format as seen in Figure 1.28. If the **Transition** value divided by the **Depth** value is **>= 4**, everything is ok. If the value is not **>=4**, then a **Warning** window will appear on screen stating the message you created in Step 19.6, "**The Depth Ratio must be less then 1 to 4!**"

19.13 Select the **OK** button. Notice that CATIA V5 adds a **Check** branch labeled **JoggleRatio** on the **Specification Tree** under the **Relations** branch. When the conditions of the check are met, the **JoggleRatio** branch will show a **Green** light. When the conditions are not met, the **JoggleRatio** branch will show a **Red** light.

19.14 If the values you enter for the **JoggleDimensions.CATScript** macro are not **>=4**, the **JoggleRatio** warning window will appear as shown in Figure 1.29. In this particular **Check** the **Joggled Extrusion** will still be updated even though it did not pass the check. The **Type Of Check** was a **Warning**. CATIA V5 let you know that it did not pass the check.

Even though this is a simplified application of the CATIA V5 **Check** tool, it is very useful. It also gives you a glimpse of how powerful this tool can be. This step should give you enough information to start building on more complex checks.

20. Practical Application… Creating an Up-to-Date Production Drawing Automatically

So far this lesson has shown you some powerful **Knowledgeware** tools. This is the step that brings it all together. The objective from the beginning was to develop an automated process of creating **Detailed Production Drawings**. To accomplish this, complete the following steps.

NOTE: This lesson assumes that you know how to use the CATIA V5 **Drafting** work bench.

20.1 Using the **Drafting** work bench, create a basic **Production Drawing** of the **Joggled Extrusion**. Use the **Orthographic** views and one **Isometric** view as shown in Figure 1.30.

20.2 Dimension the characteristics of the **Joggled Extrusion**. The characteristics of the joggle are:

20.2.1 End of part to start of joggle.

20.2.2 Joggle transition.

20.2.3 Joggle depth.

There is no need to dimension the characteristics of the extrusion because the Tierany Metals Catalog contains all of the extrusion dimensions. The Joggle Operator determines the joggle characteristics, not the extrusion characteristics. The production drawing only need contain the information that is pertinent to the process it is designed for. The dimensions required are shown in Figure 1.30.

20.3 Add a title block and production notes as required, similar to what is shown in Figure 1.30.

20.4 Save the **Production Drawing** as "**Joggled Extrusion.CATDrawing**."

20.5 Print and/or plot as required.

20.6 Run the **JoggleDimensions.CATScript**. Change the joggle dimensions as shown:

20.6.1 Depth = **.40″**

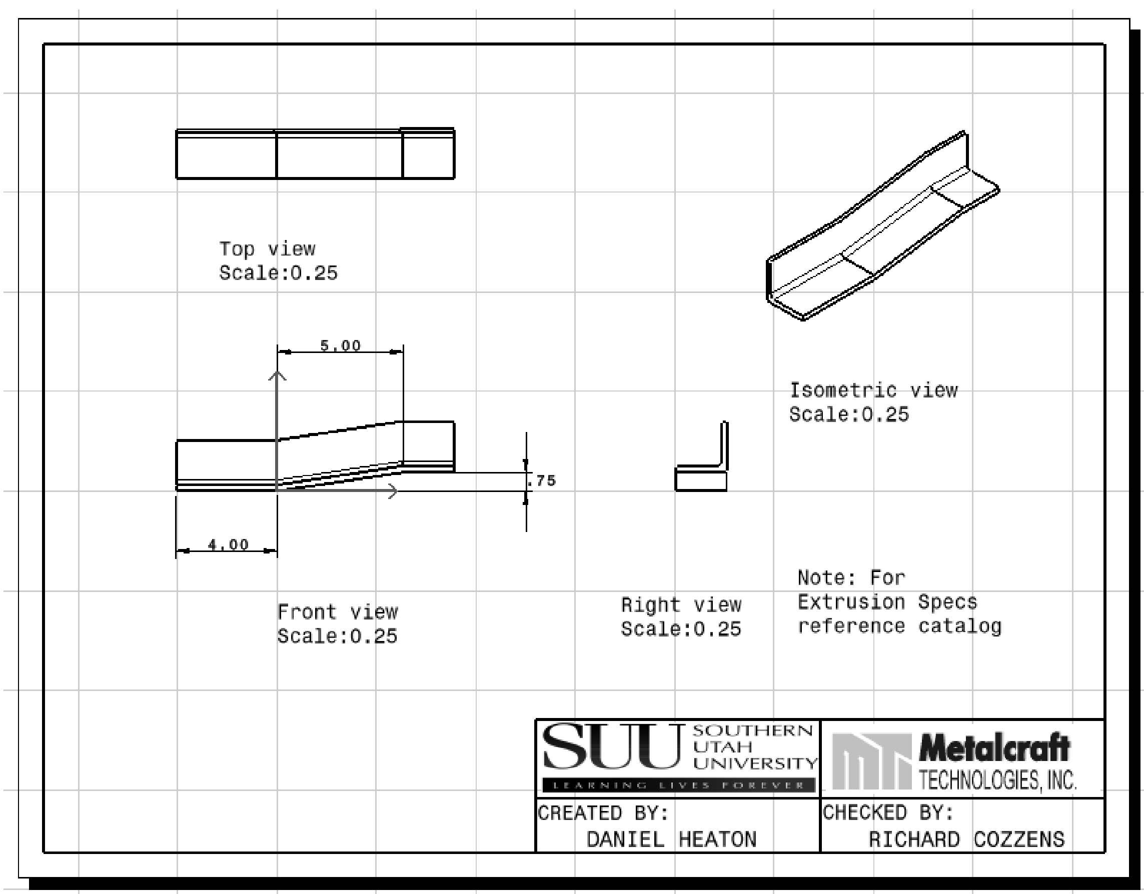
20.6.2 Transition = **3″**

20.6.3 Dist. To Endp. = **2″**

20.7 Bring up the **Joggled Extrusion.CATDrawing**. Update the drawing using the **Update** tool. Notice the view and dimensions automatically update to the newly selected extrusion and joggle dimensions.

This is where CATIA V5 Knowledgeware really saves time. The user can continue to automatically create production drawings for unique **Joggled Extrusion Parts**.

Figure 1.30



Lesson 1 Summary

The **Knowledgeware** product is made up of the following work benches: **Knowledge Advisor, Knowledge Expert, Product Engineering Optimizer, Product Knowledge Template, Product Function Optimization and Product Functional Definition**. This lesson used a small portion of the tools contained in each individual work bench. Even though only a few tools were used, a great deal of time was saved. The remaining tools offer similar opportunities for significant time saving. This lesson has supplied you with enough information to get you started. Your challenge is to find ways to implement them into your business and processes.

Lesson 1 Review

After completing this lesson, you should be able to answer the questions and explain the concepts listed below.

1. The name of a constraint can be changed by double clicking on:

   A. The constraint symbol in the **Specification Tree**.
   B. The geometry being constrained.
   C. The actual constraint in the sketch.
   D. Both A and C.

2. Where do you turn on the **Parameters** and **Relations** so that they appear in the **Specification Tree**?

   A. Tools, Options, General, Parameters and Measures.
   B. Start, Infrastructure, Product Structure.
   C. Tools, Customize, Knowledge Advisor.
   D. Tools, Options, Display, Tree Appearance.

3. How can associations be created between the column headings in an Excel table, and CATIA parameters?

   A. By giving the column headings the same name as CATIA gives to the desired parameter and allowing CATIA to automatically associate the two.
   B. By going into the **Associations** tab of the **Extrusion Table** and clicking on the **Create Parameters** button.
   C. Manually select the column heading under the **Associations** tab then also highlighting the desired parameter in the **Parameters** box and click the **Associate** button.
   D. Both A and B.
   E. Both B and C.
   F. You can't; it is impossible unless you are a computer genius.

4. How do you access an **Extrusion Table** that you have created in CATIA so that you can change the configuration row?

   A. Click on the **Extrusion Table** icon in the bottom tool bar of any work bench.
   B. Go into **Knowledge Advisor** and click on the **Extrusion Table** icon in the bottom tool bar.
   C. Double click on the **Extrusion Table** symbol in the **Specification Tree** which will take you directly into the table regardless of which work bench you are currently in.
   D. Double click on the **Extrusion Table** symbol in the **Specification Tree** which will first take you into the **Knowledge Advisor** work bench, if you are not already there, then you must double click again on the symbol to open the table.
   E. Press **Ctrl + T**.

5. How do you edit an existing **Extrusion Table** to give it different values or add new configuration rows?

   A. Open the spreadsheet outside of CATIA and make the changes, then save them and close the file. When you open the table in CATIA, it will be automatically updated for you.
   B. Make the changes directly to the **Extrusion Table** by highlighting the rows to be changed and clicking the **Edit** button.
   C. Open the design table and click the **Edit table** button and then make the changes to the spreadsheet that comes up then save the changes and close the spreadsheet.
   D. Both A and C.
   E. Both B and C.

6. What must be added to a macro name in order to access it externally or link it to an icon?

   A. .CATPart
   B. .CATDrawing
   C. .CATKnowledge
   D. .CATScript
   E. .VBScript
   F. .com

7. (True/False) An icon can be added to a tool bar that can be used to activate a macro that is stored either within the CATPart file or in an external file, as long as it has the proper extension.

   A. True
   B. False

8. Which work bench contains a tool that allows you to create a **Check** that will monitor the parameter relations in the CATPart?

   A. Product Engineering Optimizer
   B. Generative Shape Design
   C. Generative Knowledge
   D. Assembly Design
   E. Knowledge Advisor

9. Once a check has been created, CATIA indicates whether the part passes the check by:

   A. Turning the part red or grey.
   B. Bringing up the information or warning box that you programmed into the check.
   C. Making the part disappear.
   D. Not allowing you to make changes that do not pass the check.
   E. Highlighting a red or green light next to the check in the specification tree.
   F. Both A and B.
   G. Both B and E.

10. (True/False) Each time you use the Knowledgeware functions to make changes to a CATPart, you must create a new CATDrawing if you want to show the dimension changes in a new production drawing.

    A. True
    B. False

Lesson 1 Practice Exercises

Complete the following practice exercises using the information and experienced gained by completing this lesson.

1. Create a new spreadsheet using the same format used in this lesson. Link the spread sheet with the CATPart document. Update the drawing documents with each new extrusion defined in the new spreadsheet. Save the CATDrawing documents as "**Lesson1 Exercise1 Part1.CATDrawing**," "**…Part 2.CATDrawing**" and so on.

2. Modify the **JoggleRatio Check** to **1** to **3**. Save the document as "**Lesson1 Exercise2 .CATDrawing**."

3. Modify the **Joggle Depth** window prompt so it displays a warning about entering a number that violates the depth ratio 1/3 used in Exercise 2. Save the document as "**Lesson 1 Exercise 3**."

4. Modify the existing "**L Extrusion**" to a "**T Extrusion**." Modify the sketches, spreadsheet and other parameters as required. Save all related documents.