Student Notes:

DASSAULT SYSTEMES

CATIA V5 Training
Foils

CATIA V5 Automation

Version 5 Release 19
January 2009

EDU_CAT_EN_VBA_AF_V5R19

Student Notes:

About this course

Objectives of the course

Upon completion of this course, you will be able to create automation scripts, programs, and macros in CATIA V5, using Visual Basic and other automation tools available in CATIA V5.

Targeted audience

Application developers

Prerequisites

Students attending this course should have knowledge of CATIA V5 Interactive and Visual Basic

8 hours

Student Notes:

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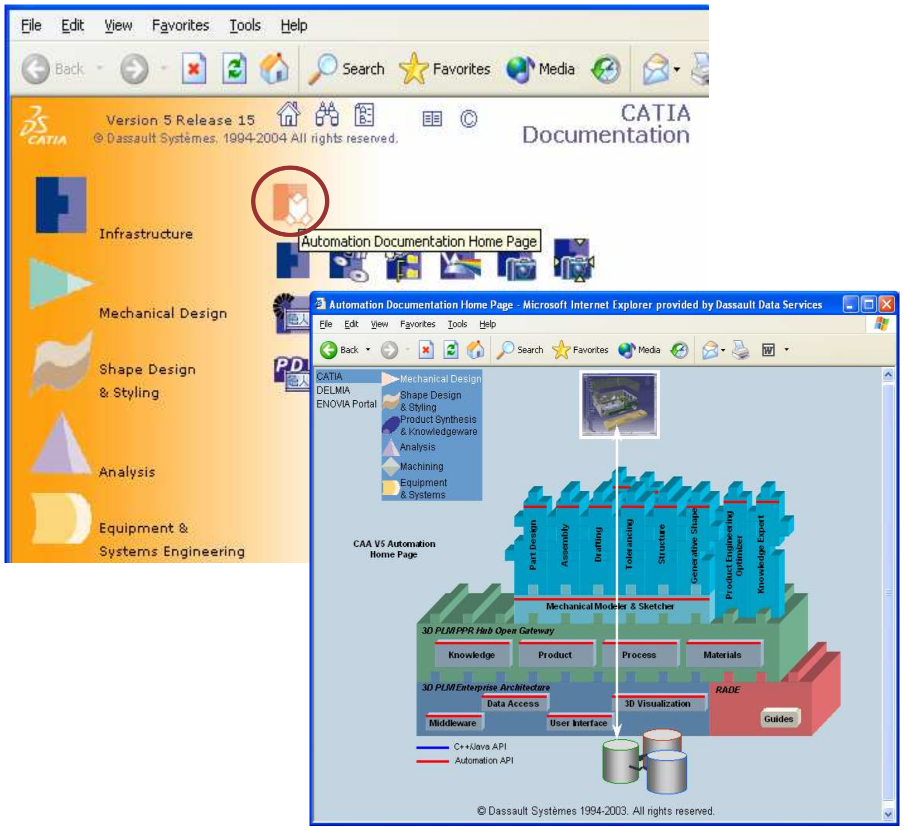
Student Notes:

Introduction to CATIA V5 Automation

You will be introduced to various tools and capabilities related to CATIA V5 Automation.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Part Design
Assembly
Drafting
Tolerancing
Structure
Generative Shape
Product Engineering Optimizer
Knowledge Expert
Mechanical Modeler & Sketcher
3D PLM PPR Hub Open Gateway
Knowledge
Product
Process
Materials
3D PLM Enterprise Architecture
Data Access
3D Visualization
Middleware
User Interface
RADE
Guides
C++/Java API
Automation API
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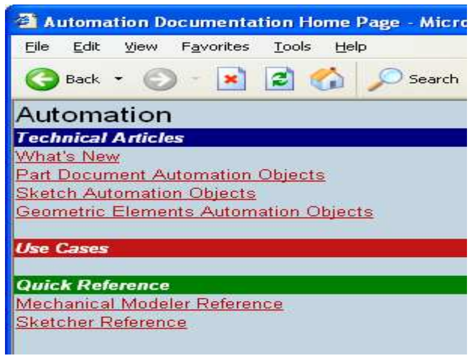
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Automation

Technical Articles

What's New

Part Document Automation Objects

Sketch Automation Objects

Geometric Elements Automation Objects

Use Cases

Quick Reference

Mechanical Modeler Reference

Sketcher Reference

CATIA V5 Automation & Scripting: Presentation

- On both Windows and UNIX
  - The scripting allows you to program CATIA in a very simple way with macros on both Windows and UNIX platforms.
  - CATIA uses the common part on MS-VBScript to have the same macros running on both the platforms.

- On Windows platforms
  - Automation allows CATIA to share objects with other external applications such as Word/Excel or Visual Basic programs.
  - CATIA can use the Word/Excel objects as well as Word/Excel can use the CATIA Objects.

Student Notes:

Knowledgeware, VB Automation, and CAA

- VB Automation
- Knowledgeware
  - Needs CATIA licenses
  - Low interaction with final user (no window interface)
  - Based on geometry
  - May be integrated to VB Automation
- CAA V5
  - Need licenses
  - High skills in programming (C++)
  - Usually for specific-domain solution

Student Notes:

Student Notes:

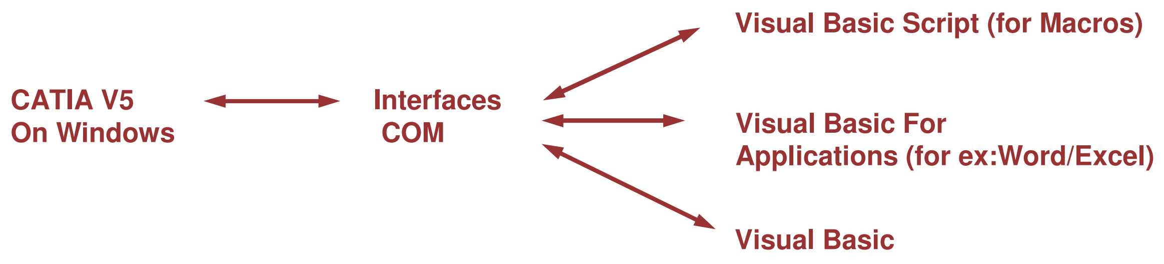
Skills Required for CATIA V5 Automation

- VBA (Visual Basic for Applications) or Visual Basic 6 basics
  - Basic language
  - Object programming
  - Event programming
- Knowledge of the CATIA workbench in which you are going to program
  - Infrastructure
  - Surfaces
  - Analysis etc.

Student Notes:

CATIA V5 Automation: Presentation (Windows only)

Automation allows communication between several process.


CATIA V5
On Windows
Interfaces
COM
Visual Basic Script (for Macros)
Visual Basic For
Applications (for ex:Word/Excel)
Visual Basic

- **COM (Component Object Model) is the "Microsoft" standard to share objects between several applications.**
- **Automation is a "Microsoft" technology to use COM objects in an interpreted environment.**
- **ActiveX component is a "Microsoft" standard to share objects between several applications even in an interpreted environment.**
- **OLE (Object Linking and Embedding) means that the document can be linked in the document of an other application OLE and can be edited in that other application (In place editing).**

Student Notes:

Language Used for Automation

- Three possible languages:
  - Visual Basic 6
  - VBA
  - VB Script / CATScript

Student Notes:

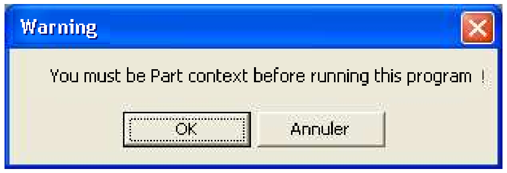
VB Script (1/2)

- VBScript (Visual Basic Script) is a subset of VBA. It is a simple interpreted Basic language
- ☺
  - Only Windows/Unix « compatible » language
  - Can call CATIA Objects
  - Directly launchable from a saved macro
- ☹
  - Weak help to programming
  - Very ‘light’ internal editor
  - No type is used. The system tries dynamically to call methods and properties of objects.
  - Sequential programming
  - Poor User Interface tools

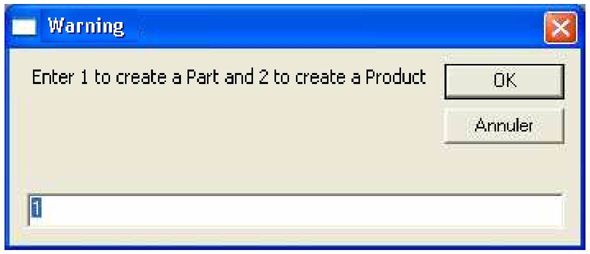
Student Notes:

VB Script (2/2)

- Poor User Interface : Examples
  - Message box
  - Input box
  - File Selection Box
  - No more …

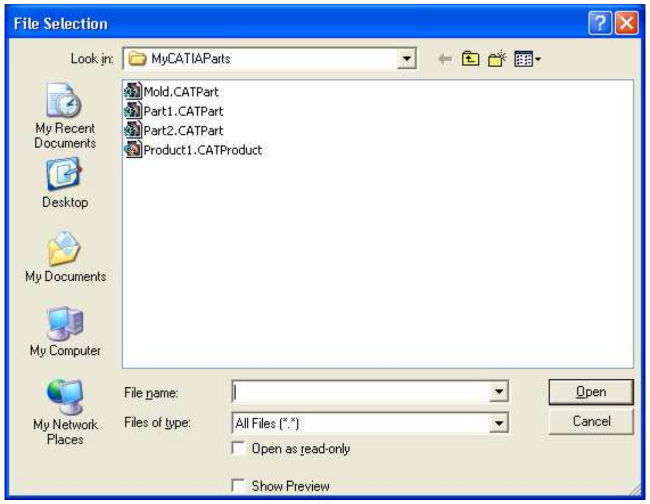


You must be Part context before running this program !



Enter 1 to create a Part and 2 to create a Product

1



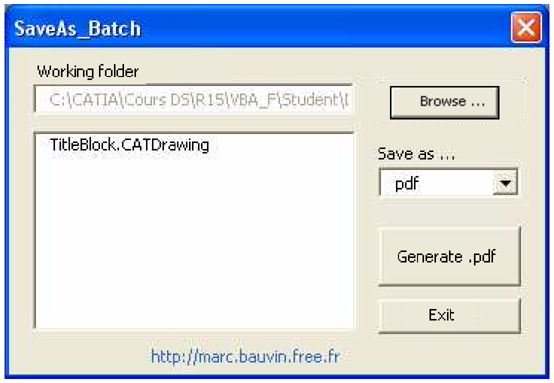
Student Notes:

VBA

VBA (Visual Basic for Applications) is a subset of Visual Basic. VBA is hosted in applications such as Word, Excel or CATIA.

- ☺
  - Programming help
  - « Event » Programming
  - CATIA drive from another VBA application (Excel, Word …)
  - Rich User Interface (buttons, lists, …)

- ☹
  - Weak protection
  - Not easy to export program
  - No « InstallShield » tool



Student Notes:

Visual Basic

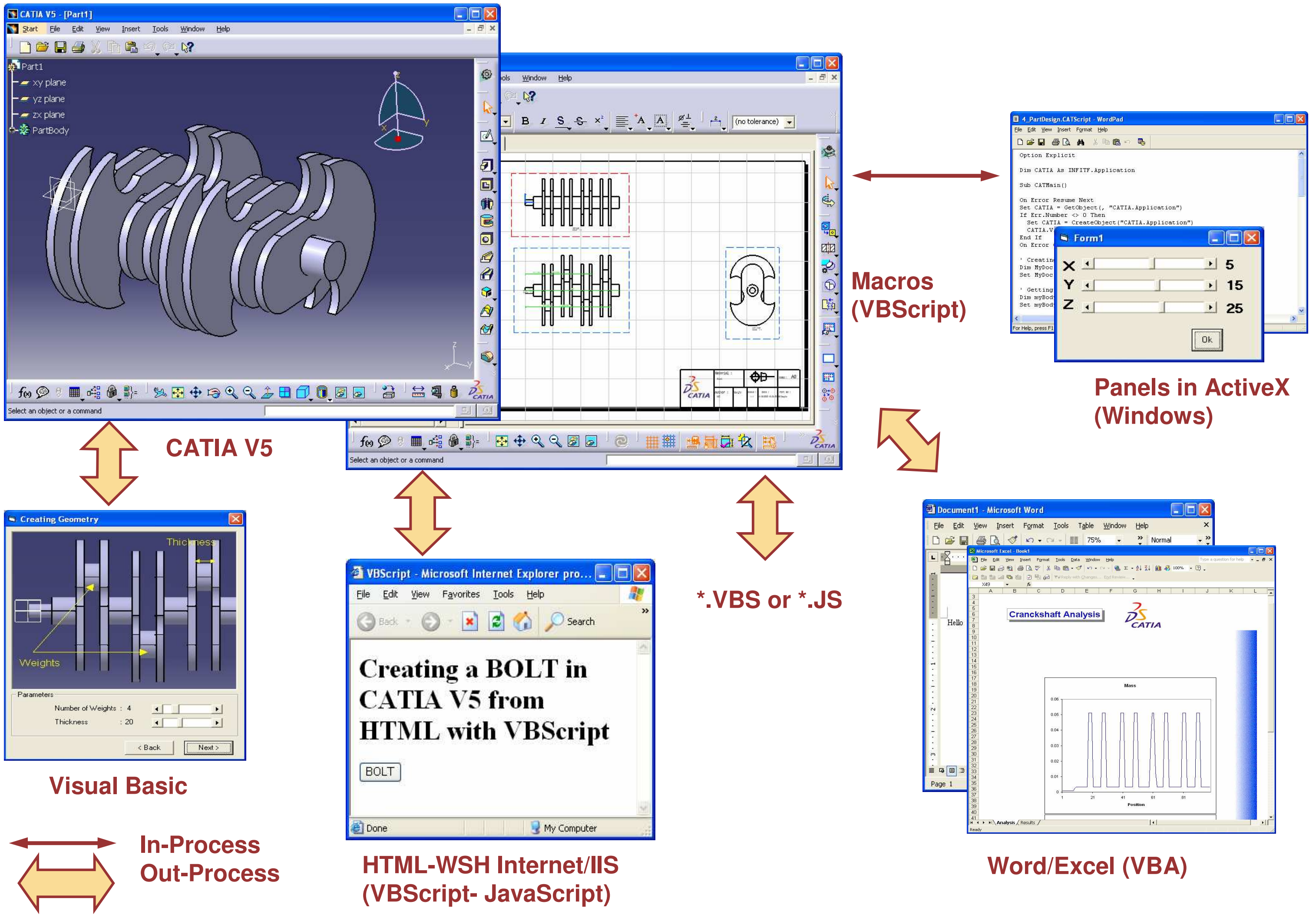
- Visual Basic is the full version
- ☺

  All the advantages of VBA + …
  - More extended set of instructions
  - Code protection (compiled program)
  - Can create ActiveX and Servers
  - provides an added documentation called “Books on line” (VB 5.0)
  - Packaging and deployment assistant

- ☹
  - Needs an additional installation
  - Microsoft licensed software

Student Notes:

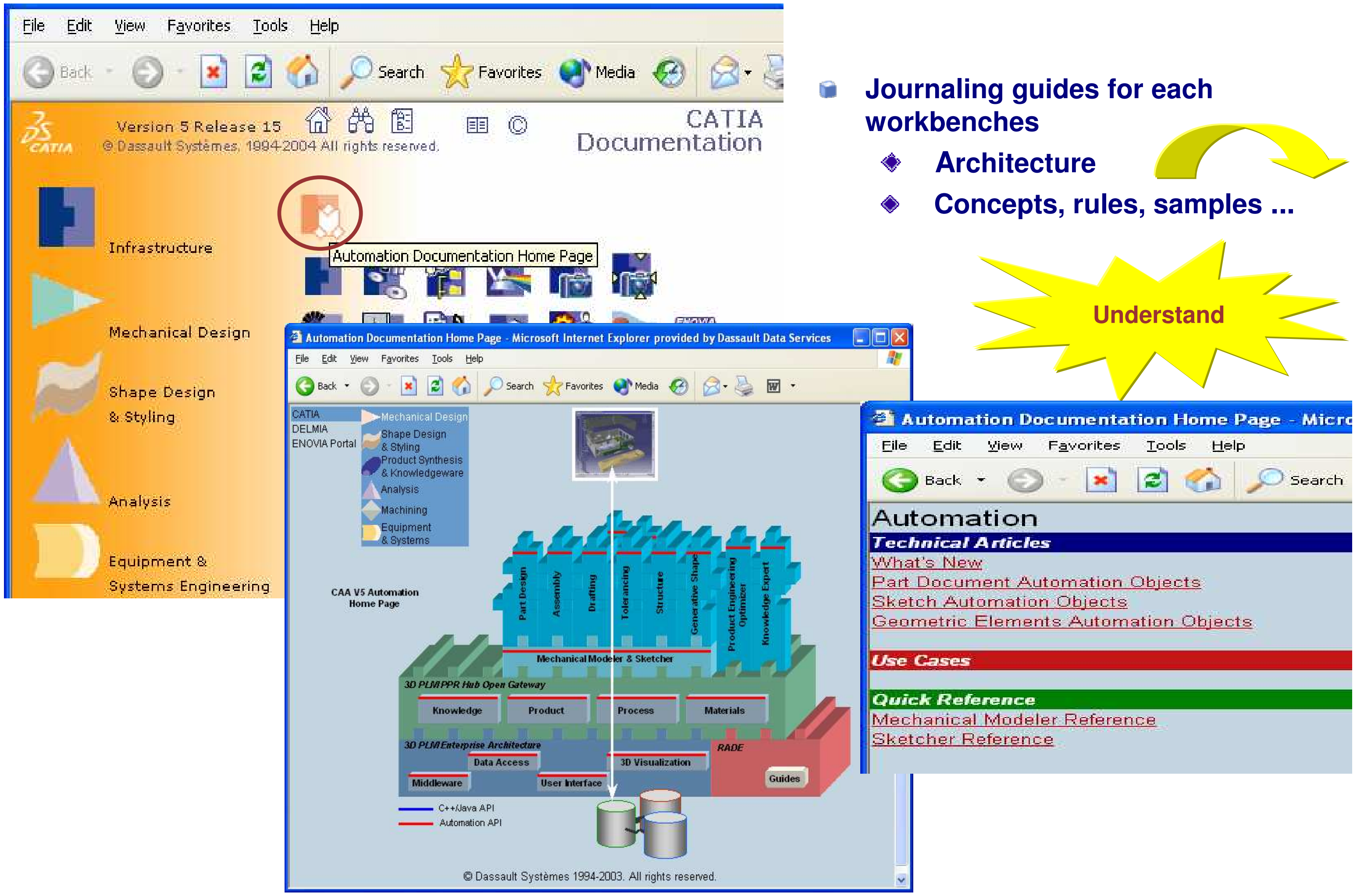
Communication Chart on Windows



```
Option Explicit

Dim CATIA As INFITF.Application

Sub CATMain()

On Error Resume Next
Set CATIA = GetObject(, "CATIA.Application")
If Err.Number <> 0 Then
  Set CATIA = CreateObject("CATIA.Application")
  CATIA.V
End If
On Error

' Creatin
Dim MyDoc
Set MyDoc

' Getting
Dim myBod
Set myBod
```

Student Notes:

Documentations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CAA V5 Automation Home Page
Mechanical Modeler & Sketcher
3D PLM PPR Hub Open Gateway
Knowledge
Product
Process
Materials
3D PLM Enterprise Architecture
Data Access
3D Visualization
Middleware
User Interface
RADE
Guides
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Automation API
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Automation
Technical Articles
What's New
Part Document Automation Objects
Sketch Automation Objects
Geometric Elements Automation Objects
Use Cases
Quick Reference
Mechanical Modeler Reference
Sketcher Reference

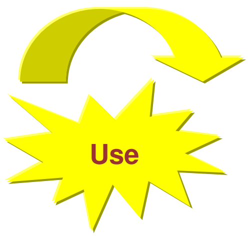
- Journaling guides for each workbenches
  - Architecture
  - Concepts, rules, samples ...

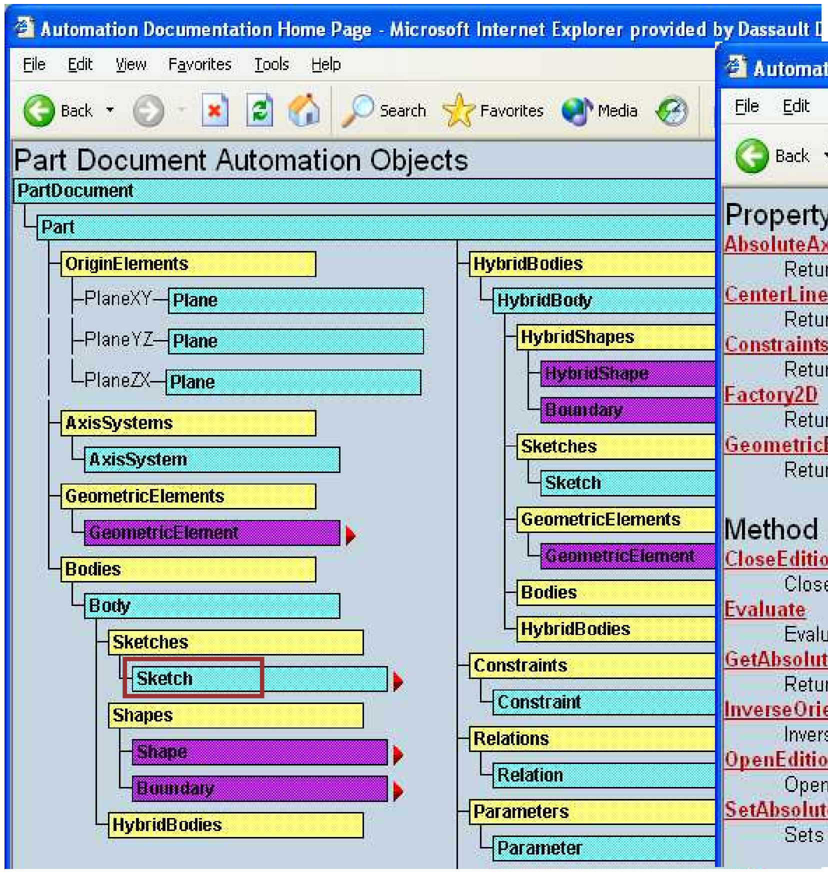
Understand

Student Notes:

Interface Documentation (1/2)

Public Interfaces

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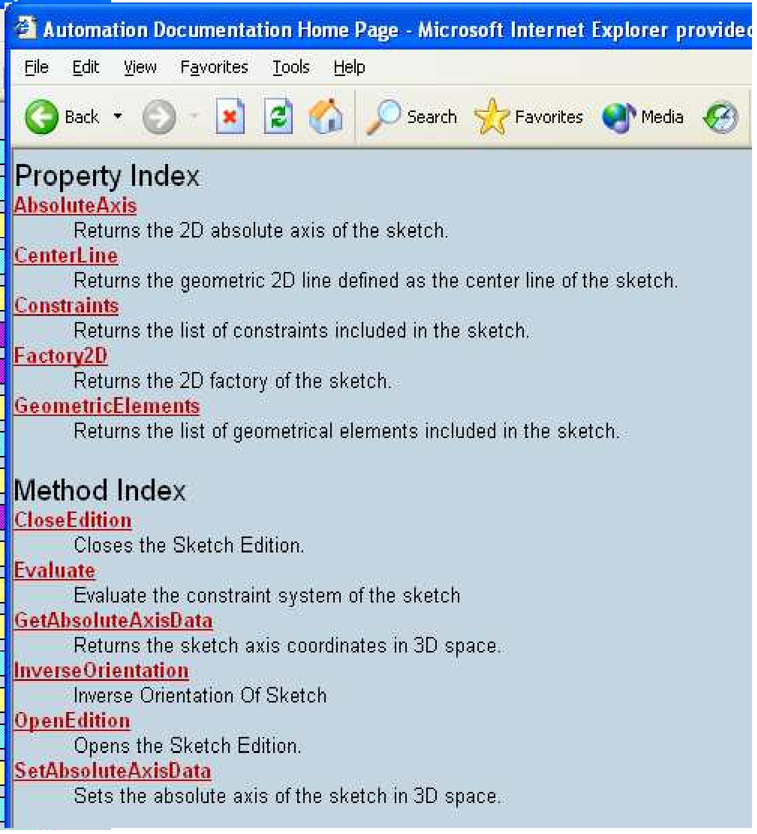
Part Document Automation Objects

- PartDocument
  - Part
    - OriginElements
      - PlaneXY — Plane
      - PlaneYZ — Plane
      - PlaneZX — Plane
    - AxisSystems
      - AxisSystem
    - GeometricElements
      - GeometricElement
    - Bodies
      - Body
        - Sketches
          - Sketch
        - Shapes
          - Shape
          - Boundary
        - HybridBodies
    - HybridBodies
      - HybridBody
        - HybridShapes
          - HybridShape
          - Boundary
        - Sketches
          - Sketch
        - GeometricElements
          - GeometricElement
        - Bodies
        - HybridBodies
    - Constraints
      - Constraint
    - Relations
      - Relation
    - Parameters
      - Parameter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Property Index

AbsoluteAxis
Returns the 2D absolute axis of the sketch.

CenterLine
Returns the geometric 2D line defined as the center line of the sketch.

Constraints
Returns the list of constraints included in the sketch.

Factory2D
Returns the 2D factory of the sketch.

GeometricElements
Returns the list of geometrical elements included in the sketch.

Method Index

CloseEdition
Closes the Sketch Edition.

Evaluate
Evaluate the constraint system of the sketch

GetAbsoluteAxisData
Returns the sketch axis coordinates in 3D space.

InverseOrientation
Inverse Orientation Of Sketch

OpenEdition
Opens the Sketch Edition.

SetAbsoluteAxisData
Sets the absolute axis of the sketch in 3D space.

Student Notes:

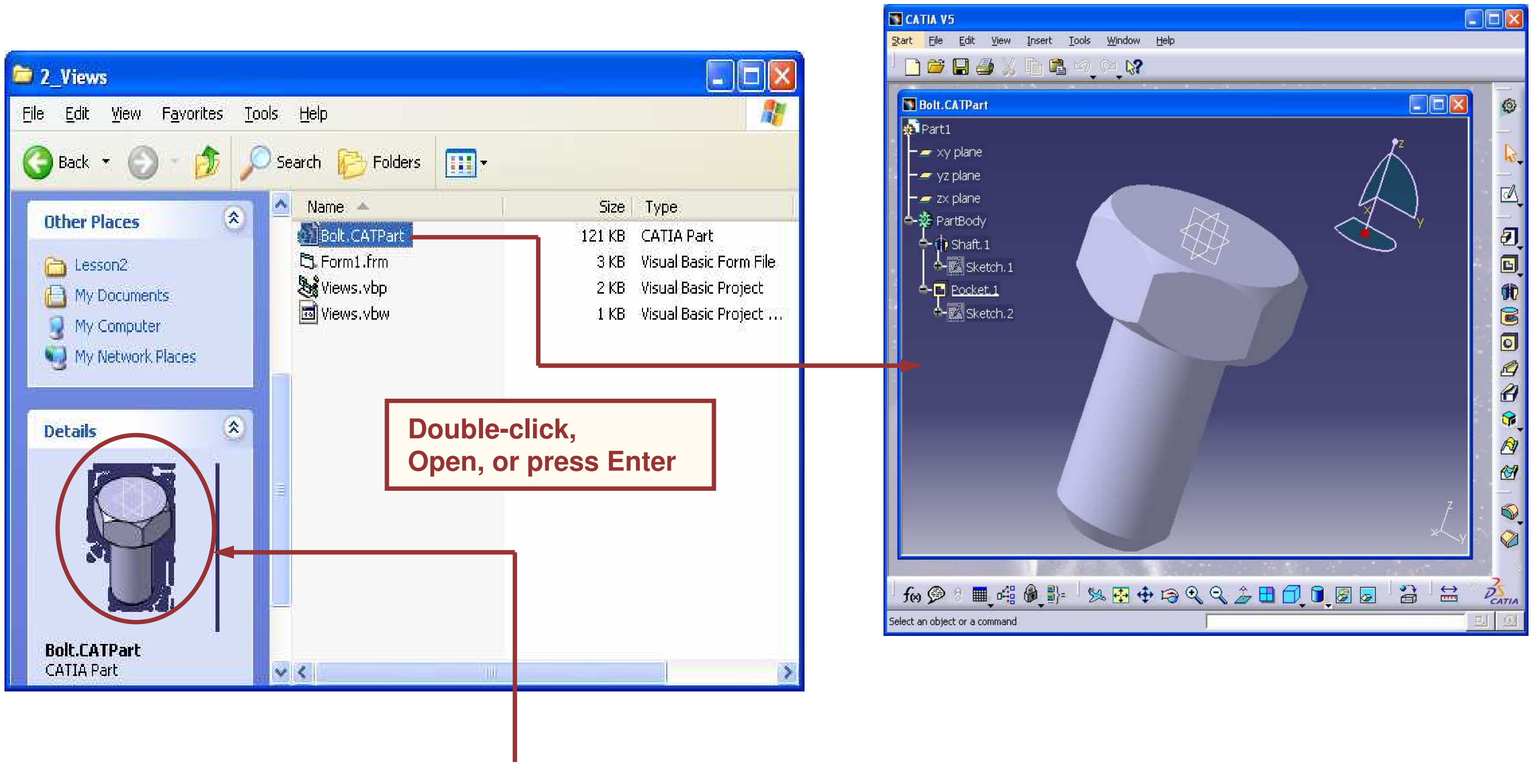
Interface Documentation (2/2)

- Additional Information
  - The documentation is also available in the CATIA code installation directory.
    Look for …\Bxx\intel_a\code\bin\V5Automation.chm
    and create a shortcut on your Desktop.
  - In the documentation, you will find helpful ‘Use cases’.

Student Notes:

CATIA V5 Association with File Extension

A double-click in Windows Explorer on a ".CATPart" file opens it in CATIA V5. On Windows, If CATIA V5 is not running, it is launched automatically.

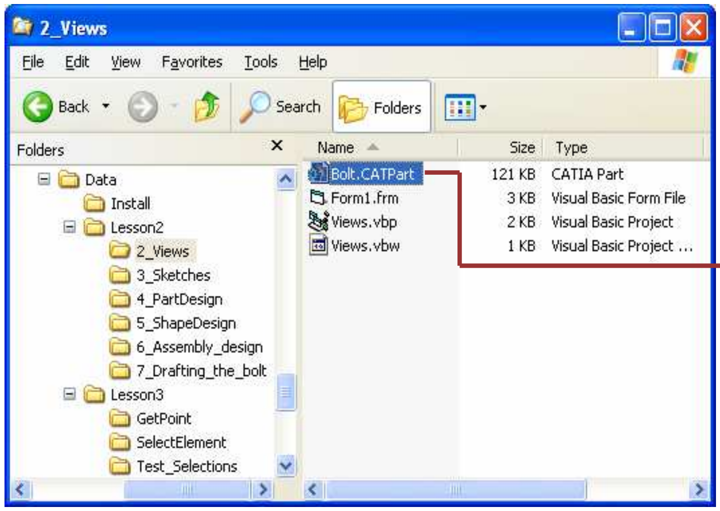


Name	Size	Type
Bolt.CATPart	121 KB	CATIA Part
Form1.frm	3 KB	Visual Basic Form File
Views.vbp	2 KB	Visual Basic Project
Views.vbw	1 KB	Visual Basic Project ...

In the Windows Explorer, the "Details" section shows a preview for all CATIA Documents.

Student Notes:

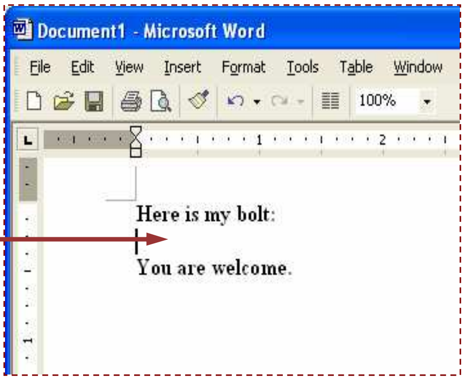
CATIA V5 OLE (Object Linking and Embedding)



Folders

- Data
  - Install
  - Lesson2
    - 2_Views
    - 3_Sketches
    - 4_PartDesign
    - 5_ShapeDesign
    - 6_Assembly_design
    - 7_Drafting_the_bolt
  - Lesson3
    - GetPoint
    - SelectElement
    - Test_Selections

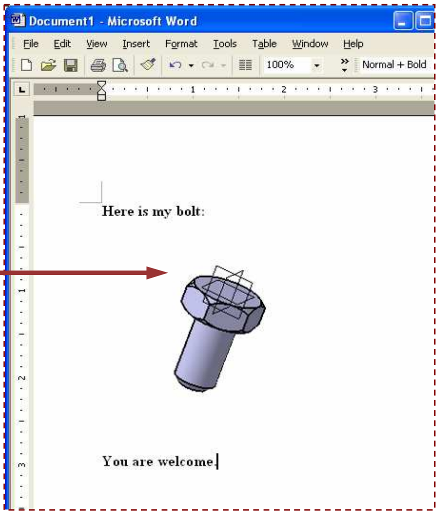
Name	Size	Type
Bolt.CATPart	121 KB	CATIA Part
Form1.frm	3 KB	Visual Basic Form File
Views.vbp	2 KB	Visual Basic Project
Views.vbw	1 KB	Visual Basic Project ...


Document1 - Microsoft Word
File Edit View Insert Format Tools Table Window
100%

Here is my bolt:

You are welcome.

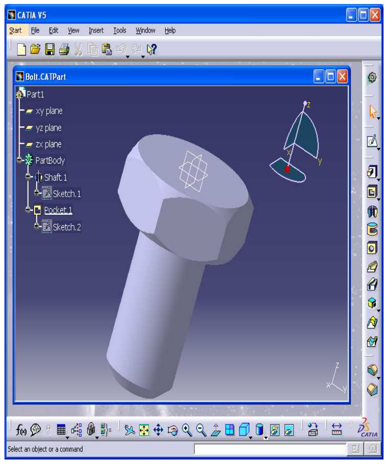
Drag and Drop in an OLE container


Document1 - Microsoft Word
File Edit View Insert Format Tools Table Window Help
100%
Normal + Bold

Here is my bolt:

You are welcome.

A double-click on the picture lets you edit the bolt in CATIA V5



Student Notes:

CATIA V5 Scripting Capabilities

IN process application:

This is done from the Tools + Macro in CATIA menu. When the Macro is running, CATIA is deactivated. You cannot store variables in a macro between two calls. Access with VBScript (Microsoft) and VBA on Windows, and with VBScript (Winsoft) on Unix.

OUT process application (Windows only)

This is based on AUTOMATION. A foreign process can call CATIA, create or modify geometry, decode, measure or pilot CATIA. CATIA is still running. For each action of the application, we have to check if CATIA is in the good state to synchronize the two processes. This kind of applications can be developed in:

Visual Basic

Windows Scripting Host (VBScript , JScript)

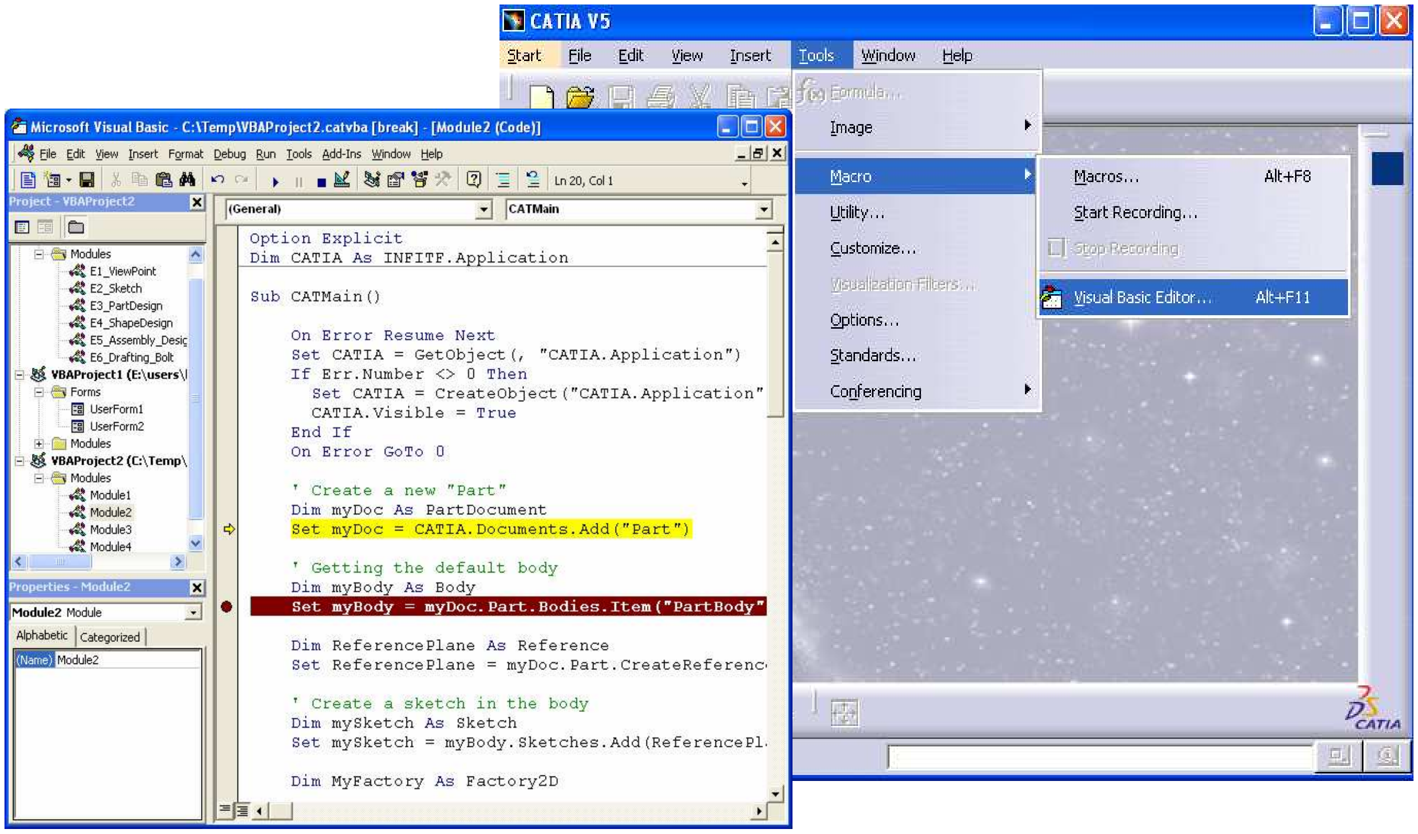
Windows Explorer HTML (VBScript , JScript)

or Any other COM applications.

Student Notes:

In-Process Macros

You will learn how to record and run the In-Process macros.

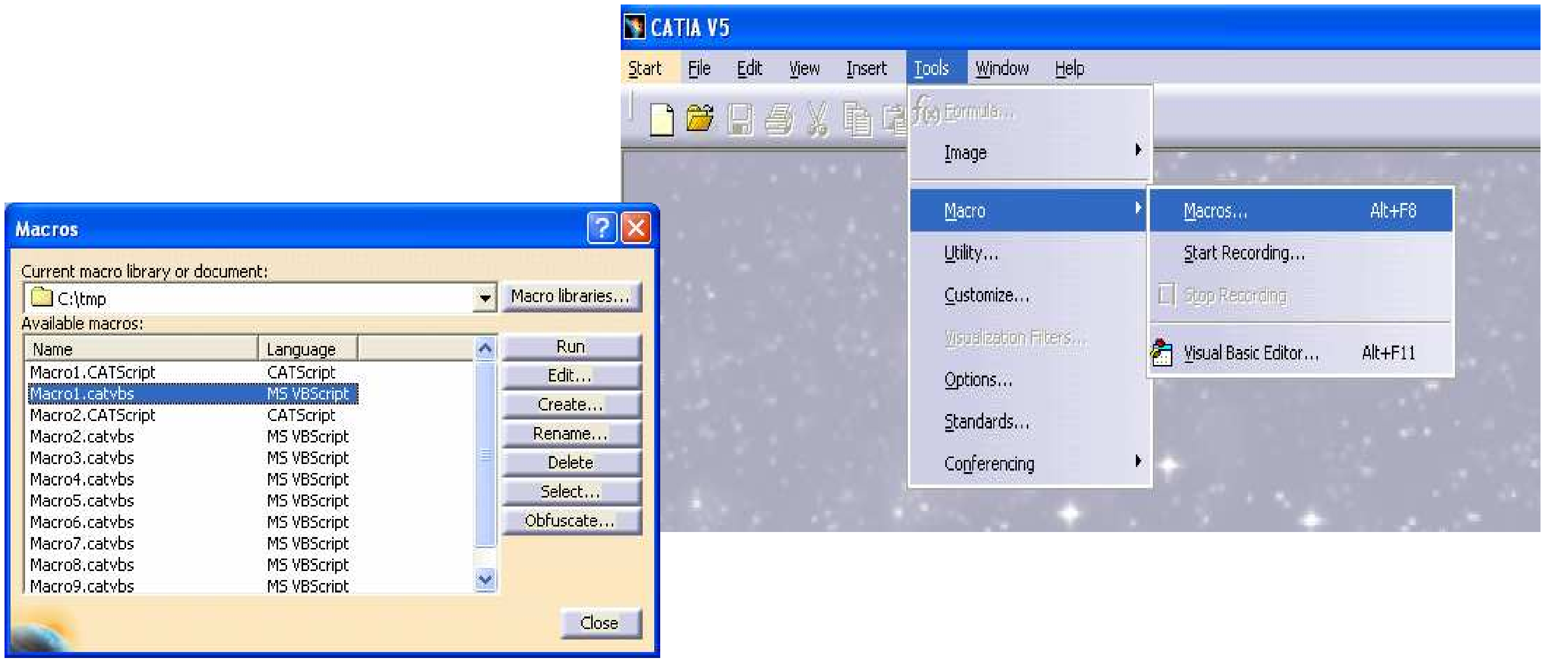


```
Option Explicit
Dim CATIA As INFITF.Application

Sub CATMain()

    On Error Resume Next
    Set CATIA = GetObject(, "CATIA.Application")
    If Err.Number <> 0 Then
      Set CATIA = CreateObject("CATIA.Application"
      CATIA.Visible = True
    End If
    On Error GoTo 0

    ' Create a new "Part"
    Dim myDoc As PartDocument
    Set myDoc = CATIA.Documents.Add("Part")

    ' Getting the default body
    Dim myBody As Body
    Set myBody = myDoc.Part.Bodies.Item("PartBody"

    Dim ReferencePlane As Reference
    Set ReferencePlane = myDoc.Part.CreateReferenc

    ' Create a sketch in the body
    Dim mySketch As Sketch
    Set mySketch = myBody.Sketches.Add(ReferencePl

    Dim MyFactory As Factory2D
```

Student Notes:

What is an ‘In-Process’ Macro

We call In-process because the script interpretation is performed in the same process as CATIA.



Name	Language
Macro1.CATScript	CATScript
Macro1.catvbs	MS VBScript
Macro2.CATScript	CATScript
Macro2.catvbs	MS VBScript
Macro3.catvbs	MS VBScript
Macro4.catvbs	MS VBScript
Macro5.catvbs	MS VBScript
Macro6.catvbs	MS VBScript
Macro7.catvbs	MS VBScript
Macro8.catvbs	MS VBScript
Macro9.catvbs	MS VBScript

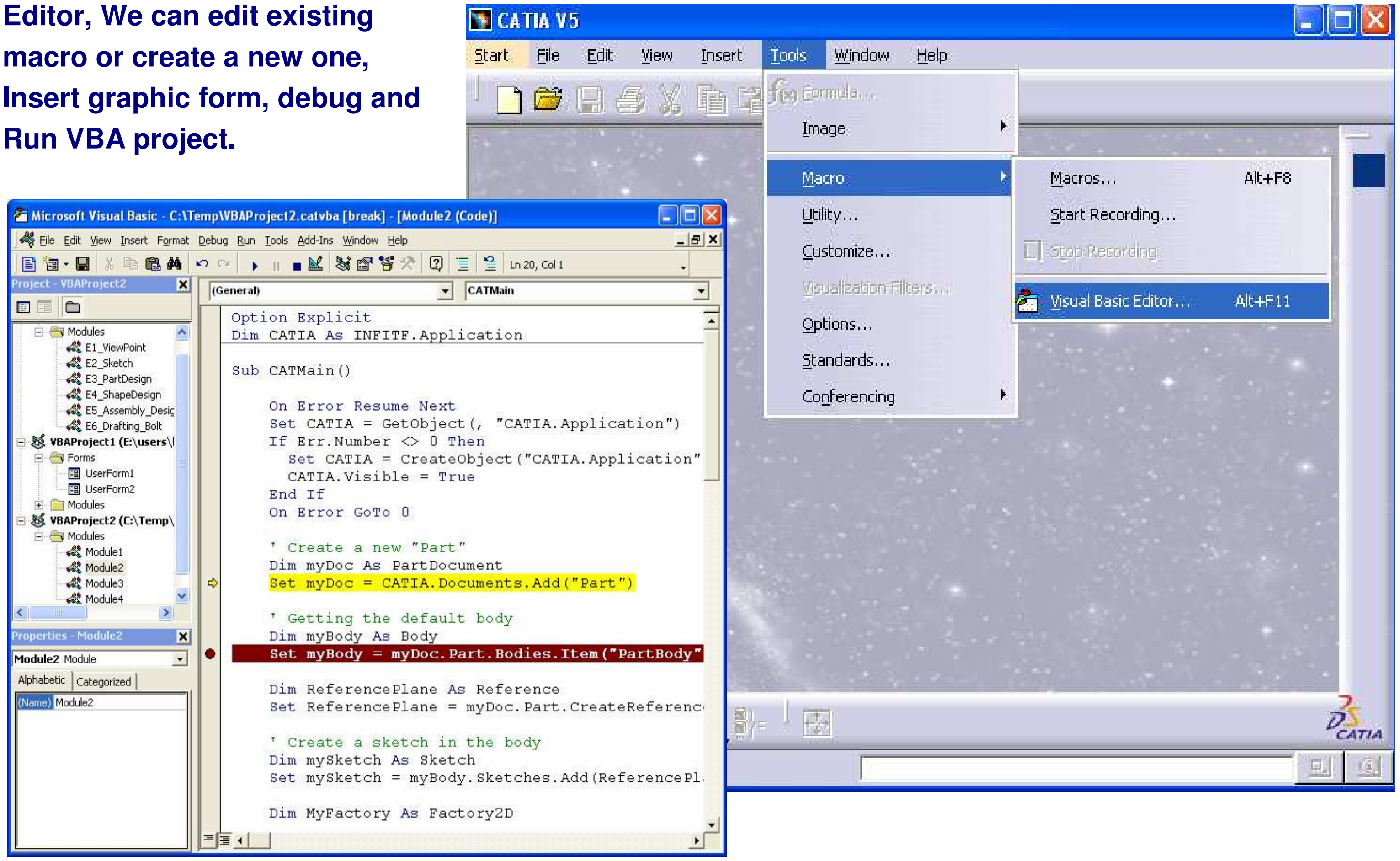
A list shows the Macro available. We can run, edit, rename or delete existing script, or create a new one.

The list of libraries of script proposed into the panel above is modifiable through the window of libraries.

Student Notes:

CATIA V5 Visual Basic Editor

With CATIA Visual Basic Editor, We can edit existing macro or create a new one, Insert graphic form, debug and Run VBA project.

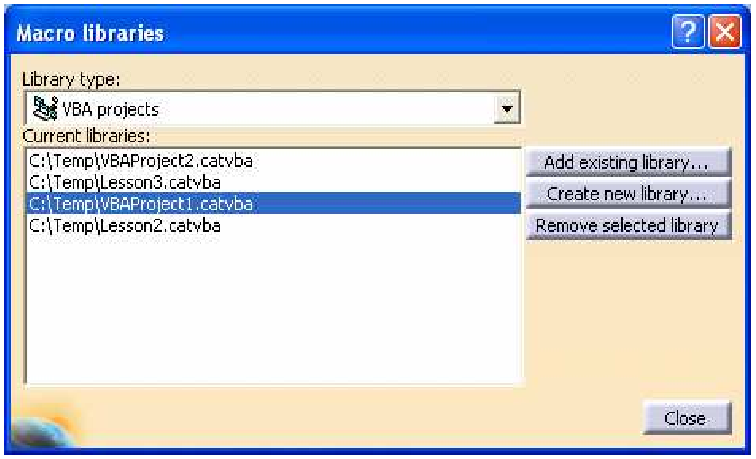


```
Option Explicit
Dim CATIA As INFITF.Application

Sub CATMain()

    On Error Resume Next
    Set CATIA = GetObject(, "CATIA.Application")
    If Err.Number <> 0 Then
      Set CATIA = CreateObject("CATIA.Application"
      CATIA.Visible = True
    End If
    On Error GoTo 0

    ' Create a new "Part"
    Dim myDoc As PartDocument
    Set myDoc = CATIA.Documents.Add("Part")

    ' Getting the default body
    Dim myBody As Body
    Set myBody = myDoc.Part.Bodies.Item("PartBody"

    Dim ReferencePlane As Reference
    Set ReferencePlane = myDoc.Part.CreateReferenc

    ' Create a sketch in the body
    Dim mySketch As Sketch
    Set mySketch = myBody.Sketches.Add(ReferencePl

    Dim MyFactory As Factory2D
```

Student Notes:

Macro Libraries

The interface of the dialog "Tools + Macro + Macros…" allows to choose a library of macro (a directory of macro or a VBA project) or a document on which we wish to work.



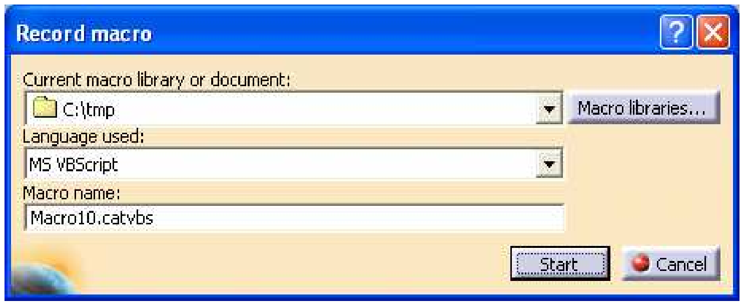
C:\Temp\VBAProject2.catvba
C:\Temp\Lesson3.catvba
C:\Temp\VBAProject1.catvba
C:\Temp\Lesson2.catvba

From this window, we can close an already opened library, open a library, or create a library. Libraries are shown by type (directory, VBA project...).
Every type of library supports different languages. VBA supports only the language MS-VBA, while in a directory we can write in MS-VBScript or in CATScript.

Student Notes:

Recording a Macro

The panel of recording VBA allows to choose the container in which we are going to record, the used language and the name of the macro to be recorded.
The syntax of the recorded script is going to differ according to the menu chosen.



Current macro library or document: C:\tmp
Language used: MS VBScript
Macro name: Macro10.catvbs

Recording a Macro generates a script corresponding to the creation or modification of the objects in the recorded sequence. After stopping the recording, we can store, edit or replay this script (called macro).

As a rule, recording a macro helps you to learn how to program something.

Student Notes:

Running a Macro (1/2)

- You can execute a macro as soon as CATIA is started :

```
CNEXT -macro E:\Users\Macros\MacroToRun.CATScript
```

- You can start CATIA in batch mode to execute a macro

```
CNEXT -batch -macro E:\Users\Macros\MacroToRun.CATScript
```

- You can run a macro interactively from the “Macro” menu
- You can run a macro interactively from an icon in a toolbar

Student Notes:

Running a Macro (2/2)

The syntax to execute macros in text files using the CNEXT -macro command line, has been expanded to allow replaying macro which are in VBA projects, in V5 documents, and are written in VBScript, CATScript or VBA.

- Here is a complete summary of the Supported Syntax:
  - Previously available options:
    - CNEXT -macro X.CATScript // Replays file X.CATScript (CATScript file)
    - CNEXT -macro X.catvbs // Replays file X.catvbs (pure VBScript file)
  - New options:
    - CNEXT -macro X.catvba Y // Replays macro Y in the VBA project X.catvba
    - CNEXT -macro X.CATPart Y // Replays macro Y in the V5 document X.CATPart

Student Notes:

Execution of a Macro from another Macro

CATIA V5 offer the capability through an Automation function to execute a Macro from another Macro. The function lets the user invoke another scripted function, contained in another macro library, potentially written in another scripting language. A support enum CATSysLibraryType is used to let the developer describe the kind of library the function to call is contained in. (Example from VBScript)

- **To call an Add function that is defined in a “E:\Macros\Math” directory library, in a file “IntegerMath.catvbs”, one would write:**

```
Dim args2(1)
args2(0) = 3
args2(1) = 2
Dim addResult
addResult = CATIA.SystemService.ExecuteScript ( "E:\Macros\Math",
catScriptLibraryTypeDirectory, "IntegerMath.catvbs", "Add", args2 )
MsgBox addResult
```

- **To call an PrintReport() sub that is defined in a “E:\Parts\Part1.CATPart” V5 document library, in a file “Statistics.catvbs”, one would write:**

```
Dim args()
CATIA.SystemService.ExecuteScript "E:\Parts\Part1.CATPart", catScriptLibraryTypeDocument,
"Statistics.catvbs", "PrintReport", args
```

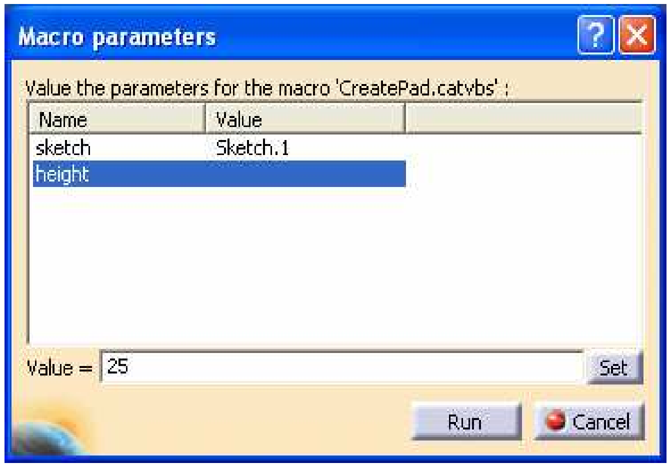
Student Notes:

Execution of a Macro which Require Parameters

CATIA V5 gives the possibility to add parameters to the CATMain function.
When the function is executed, an interactive window pops up which lets the user value the parameters by selecting V5 objects or typing the values directly. Parameters of type string, int, float, boolean and object can be entered.

If the user writes the following VBScript macro:

```
Sub CATMain(sketch, height)
    Set partDocument1 = CATIA.ActiveDocument
    Set part1 = partDocument1.Part
    Set shapeFactory1 = part1.ShapeFactory
    Set pad1 = shapeFactory1.AddNewPad(sketch, height)
    part1.Update
End Sub
```


Macro parameters

Value the parameters for the macro 'CreatePad.catvbs' :

Name	Value
sketch	Sketch.1
height	

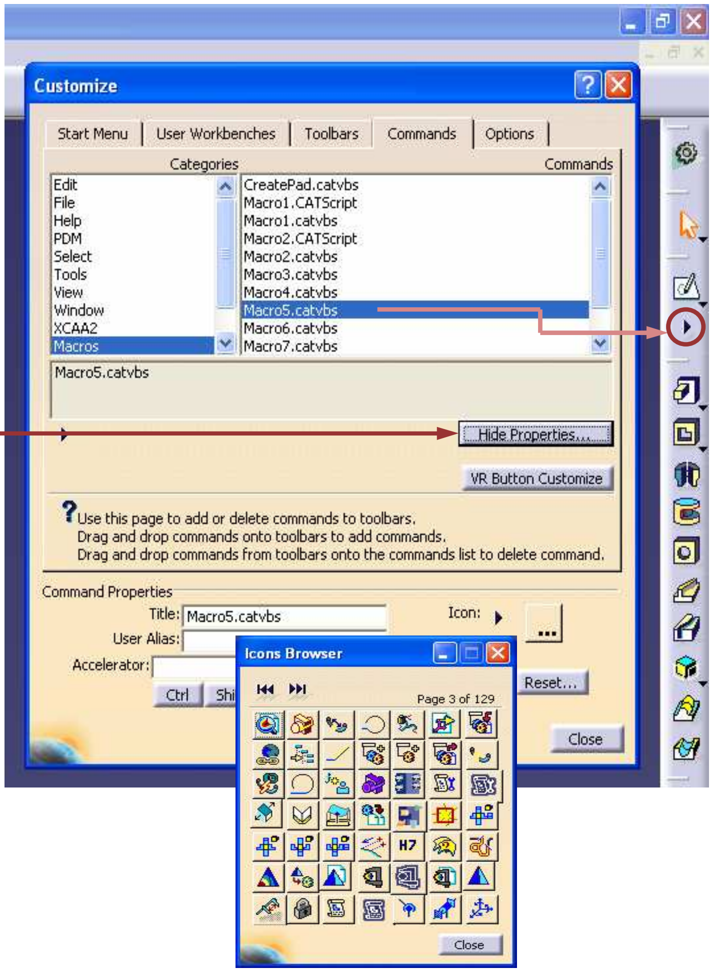
Value = 25

and runs it, the following window pop up automatically to let him/her enter the value of the sketch and height parameters.

Student Notes:

Adding a Macro as a command in a Toolbar

- Select the macro you want to add in Tools + Customize + Commands Tab page + Macros:
- Drag and Drop the macro name to the toolbar you wish :
- To select another icon than the default one, click on Show Properties...



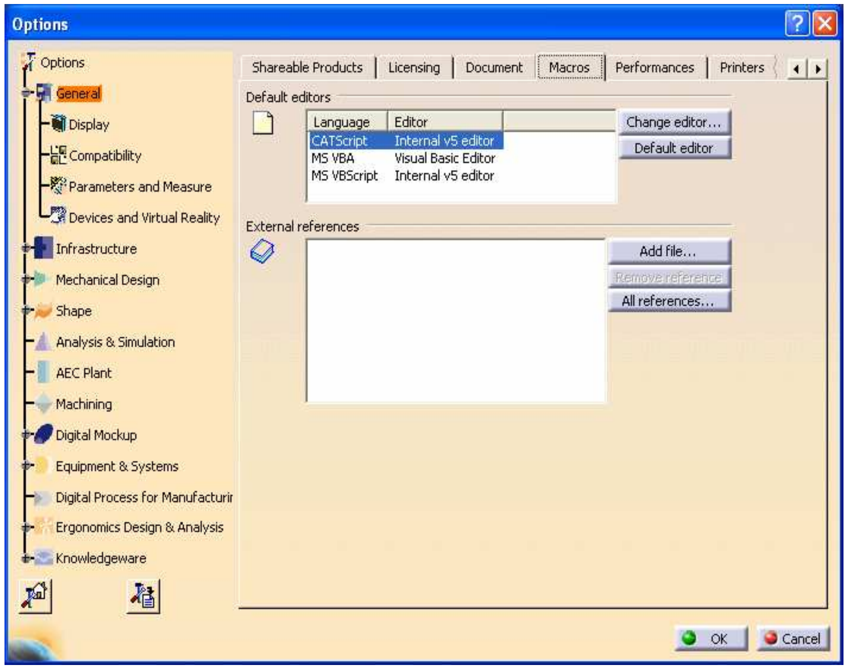
Use this page to add or delete commands to toolbars.
Drag and drop commands onto toolbars to add commands.
Drag and drop commands from toolbars onto the commands list to delete command.

Title: Macro5.catvbs

Student Notes:

Macros Options Interface

The panel of the dialog "Tools + Options + Macros" allows to choose according to the language, the Editor who will be launched to edit a macro, and to specify external typelibs to take into account in the replay of scripts.



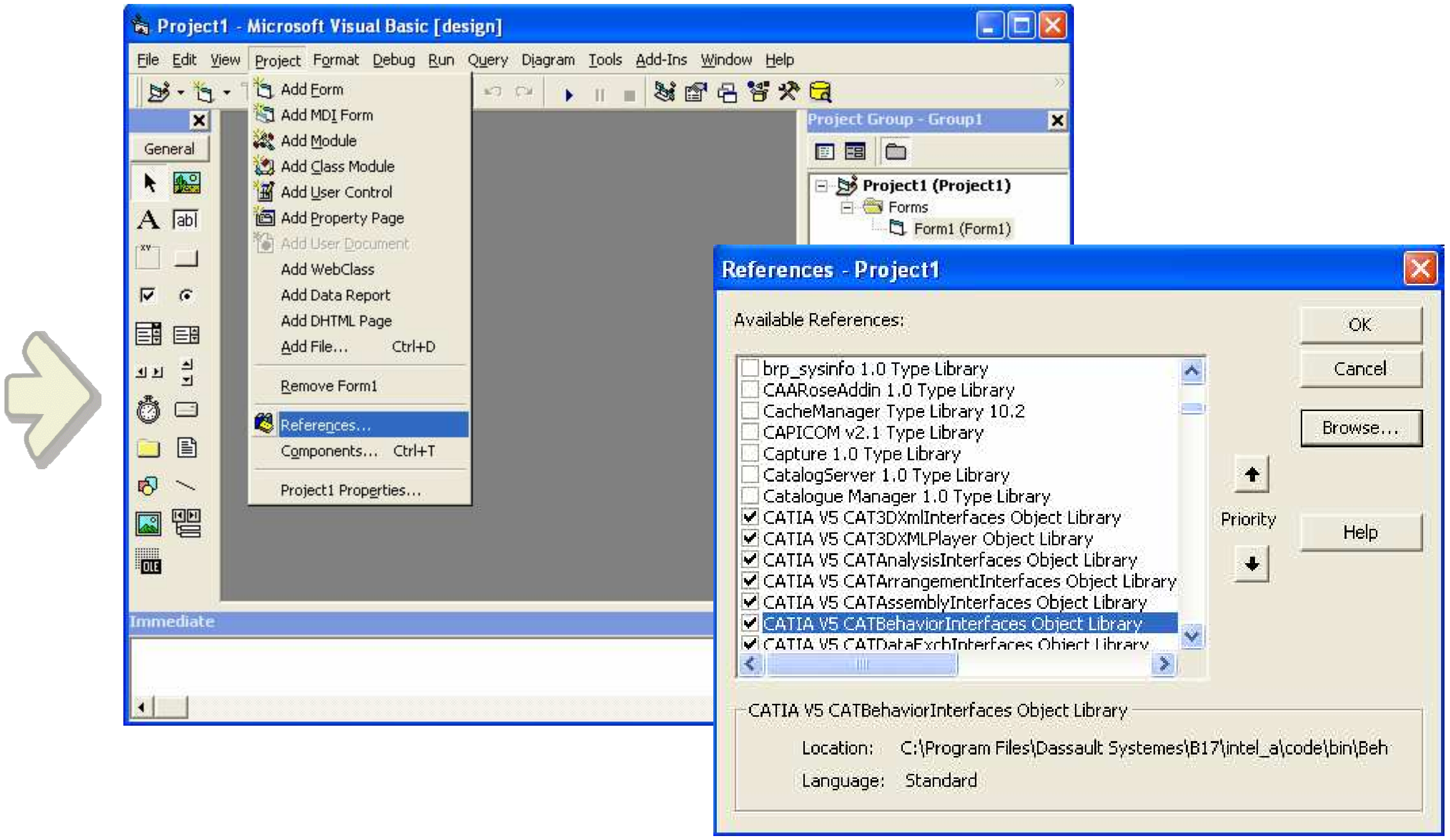
Language	Editor
CATScript	Internal v5 editor
MS VBA	Visual Basic Editor
MS VBScript	Internal v5 editor

With the new interface, all the typelibs V5 of the runtime view are automatically taken into account. This option serves only for declaring typelibs except runtime view (Excel for example).

Student Notes:

Out-Process Programs

You will learn methods to create and execute Out-Process programs for CATIA V5 Automation.



- [ ] brp_sysinfo 1.0 Type Library
- [ ] CAARoseAddin 1.0 Type Library
- [ ] CacheManager Type Library 10.2
- [ ] CAPICOM v2.1 Type Library
- [ ] Capture 1.0 Type Library
- [ ] CatalogServer 1.0 Type Library
- [ ] Catalogue Manager 1.0 Type Library
- [x] CATIA V5 CAT3DXmlInterfaces Object Library
- [x] CATIA V5 CAT3DXMLPlayer Object Library
- [x] CATIA V5 CATAnalysisInterfaces Object Library
- [x] CATIA V5 CATArrangementInterfaces Object Library
- [x] CATIA V5 CATAssemblyInterfaces Object Library
- [x] CATIA V5 CATBehaviorInterfaces Object Library
- [x] CATIA V5 CATDataExchInterfaces Object Library

CATIA V5 CATBehaviorInterfaces Object Library

Location: C:\Program Files\Dassault Systemes\B17\intel_a\code\bin\Beh

Language: Standard

Student Notes:

Running Out-Process Programs (On Windows only)

The script is running in another application running in another process, such as:

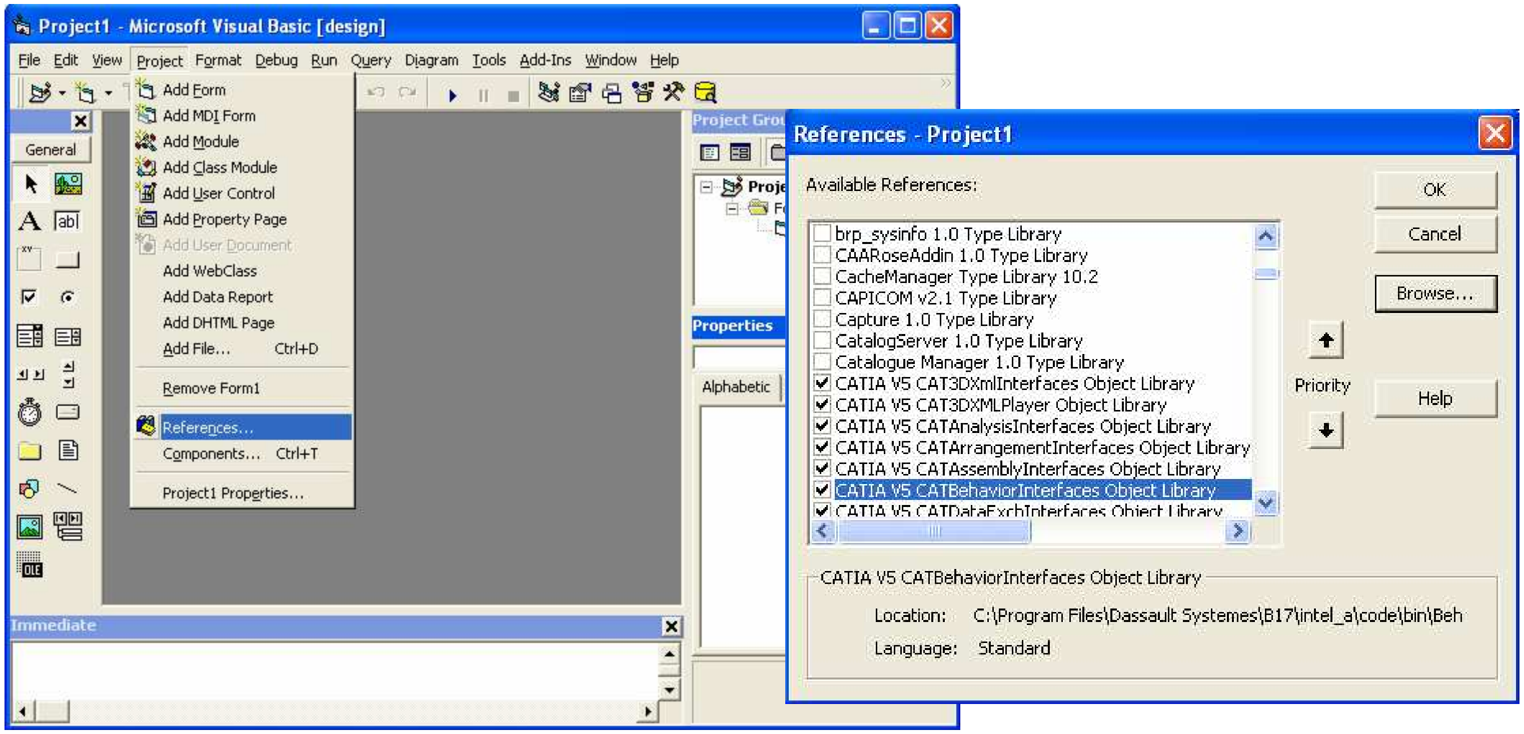
- Visual Basic
- VBA (in Excel or Word)
- Using WSH (Windows Scripting Host) with VBScript or JavaScript
- Using HTML with VBScript or JavaScript

CATIA can be scripted from any other COM Application.

Student Notes:

Running Out-Process from VBA or Visual Basic (1/3)

VBA and Visual Basic provide useful tools :
For this, you can declare all the typelib files (*.tlb) provided by Dassault Systemes.
The typelib files contain the declarations of all the objects, methods and properties of the Exposed objects.



Available References:

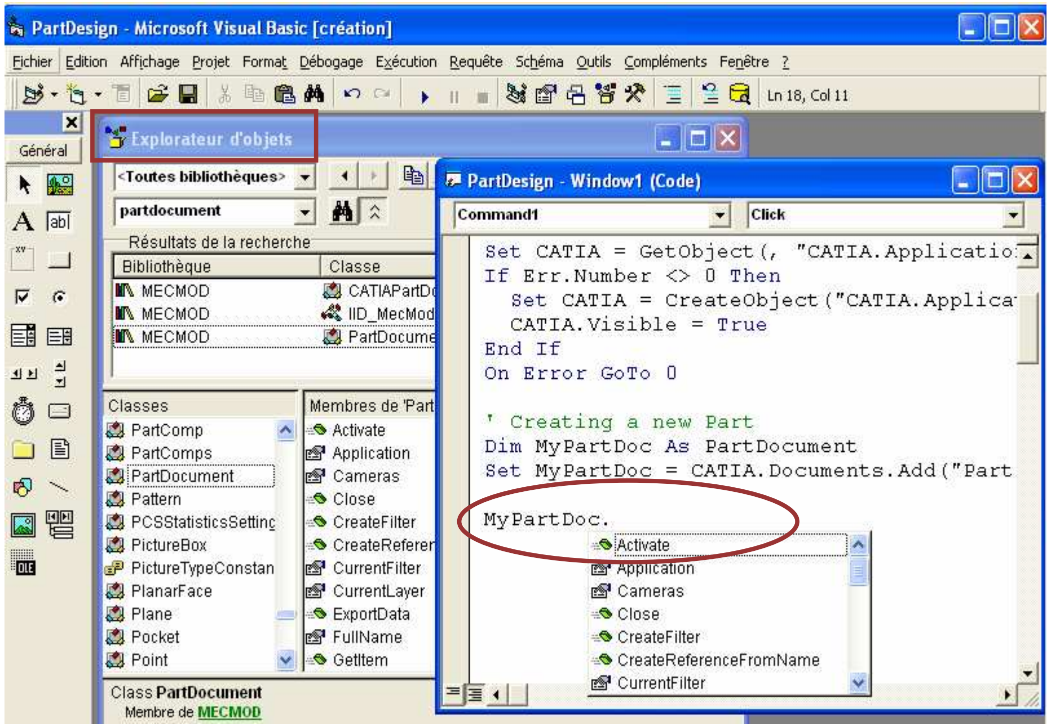
- [ ] brp_sysinfo 1.0 Type Library
- [ ] CAARoseAddin 1.0 Type Library
- [ ] CacheManager Type Library 10.2
- [ ] CAPICOM v2.1 Type Library
- [ ] Capture 1.0 Type Library
- [ ] CatalogServer 1.0 Type Library
- [ ] Catalogue Manager 1.0 Type Library
- [x] CATIA V5 CAT3DXmlInterfaces Object Library
- [x] CATIA V5 CAT3DXMLPlayer Object Library
- [x] CATIA V5 CATAnalysisInterfaces Object Library
- [x] CATIA V5 CATArrangementInterfaces Object Library
- [x] CATIA V5 CATAssemblyInterfaces Object Library
- [x] CATIA V5 CATBehaviorInterfaces Object Library
- [x] CATIA V5 CATDataExchInterfaces Object Library

CATIA V5 CATBehaviorInterfaces Object Library
Location: C:\Program Files\Dassault Systemes\B17\intel_a\code\bin\Beh
Language: Standard

Student Notes:

Running Out-Process from VBA or Visual Basic (2/3)

- **Type definition allows type checking and “early binding”.**
- **Helpful completion gives you all the properties and methods of an object and gives you the argument types of a method.**
- **Browser Object describes all the exported objects.**



```
Set CATIA = GetObject(, "CATIA.Applicatio
If Err.Number <> 0 Then
  Set CATIA = CreateObject("CATIA.Applica
  CATIA.Visible = True
End If
On Error GoTo 0

' Creating a new Part
Dim MyPartDoc As PartDocument
Set MyPartDoc = CATIA.Documents.Add("Part

MyPartDoc.
```

Student Notes:

Running Out-Process from VBA or Visual Basic (3/3)

Here are the statements to launch CATIA V5 from VBA or Visual Basic.
If CATIA is already running :

```
[...]
Dim CATIA as Object
Set CATIA = GetObject(, “CATIA.Application”)
[...]
```

First arg. is left blank

If CATIA is not already running :

```
[...]
Dim CATIA as Object
Set CATIA = CreateObject(“CATIA.Application”)
CATIA.Visible = True
[...]
```

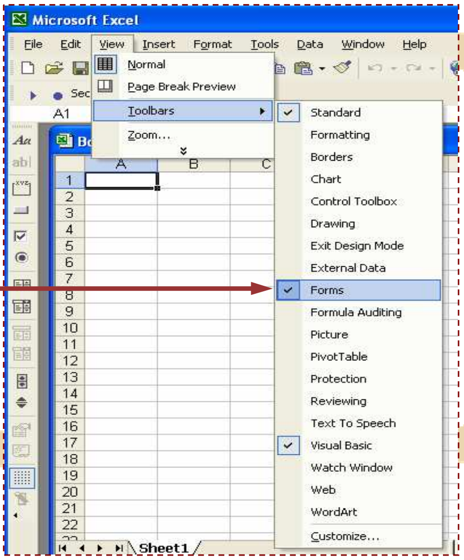
This macro starts CATIA

If the typelib INFITF.tbl is referenced, we can declare CATIA as INFITF.Application.

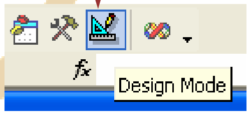
Student Notes:

Example: Bolt from Excel (1/3)

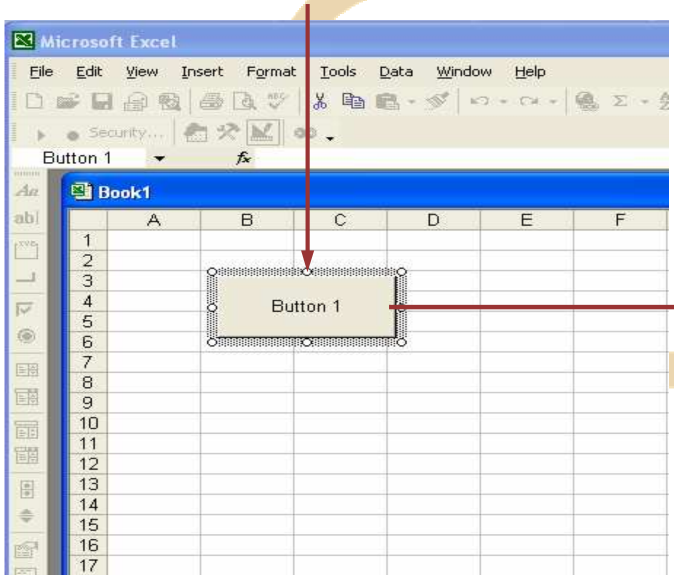
1 Start a New Excel Document and activate the toolbars: Forms and Visual Basic.

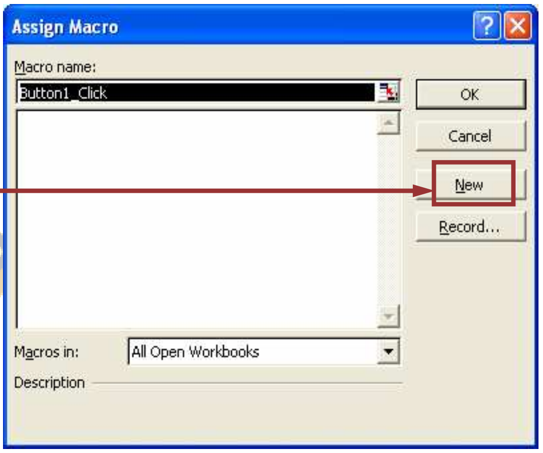


2 Activate the Design mode.



3 Create a new Button and click on “New” button to “Assign Macro”.



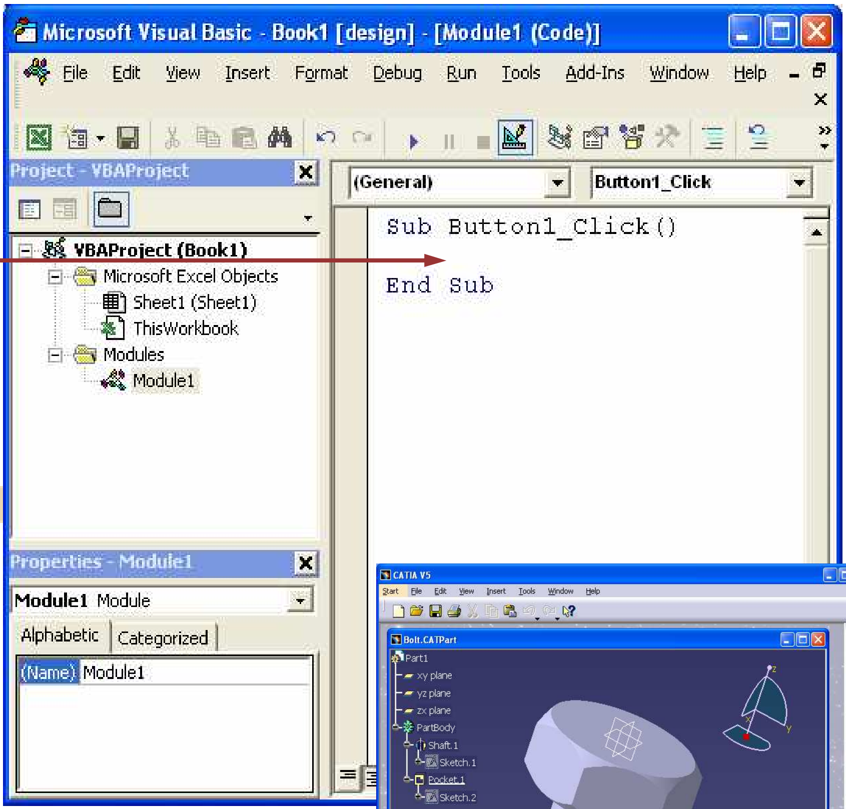


Student Notes:

Example: Bolt from Excel (2/3)

1

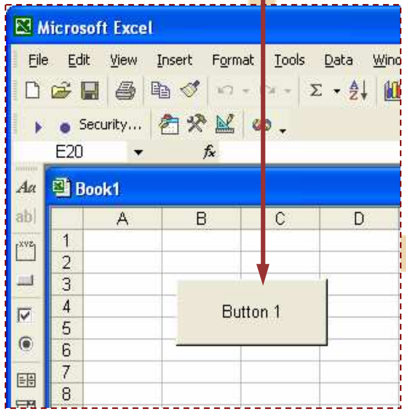
You go automatically in Visual Basic.
Insert the Code from the file "BoltFromExcel.txt"

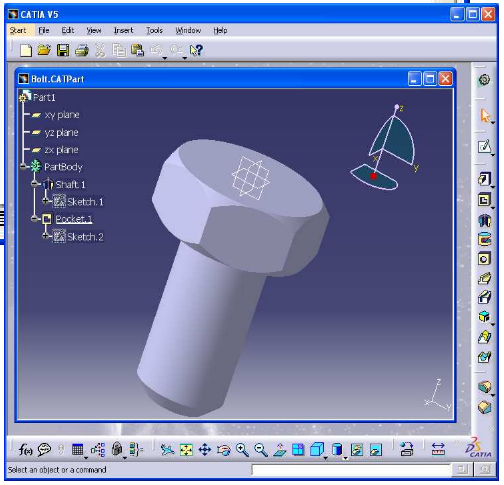


```
Sub Button1_Click()

End Sub
```

2

Close Visual Basic and return to Excel.
Click on Button1
CATIA will be launched and a bolt will be created.





Student Notes:

Example: Bolt from Excel (3/3)

Have a look on the code:

This tries to retrieve CATIA and if not successful starts it.

```
Dim CATIA As Object
  'Get CATIA or start it if necessary.
  On Error Resume Next
  Set CATIA = GetObject(, "CATIA.Application")
  If Err.Number <> 0 Then
     Set CATIA = CreateObject("CATIA.Application")
     CATIA.Visible = True
  End If
  On Error GoTo 0
```

Both functions GetObject and CreateObject are provided by VBA (as well as Visual Basic).

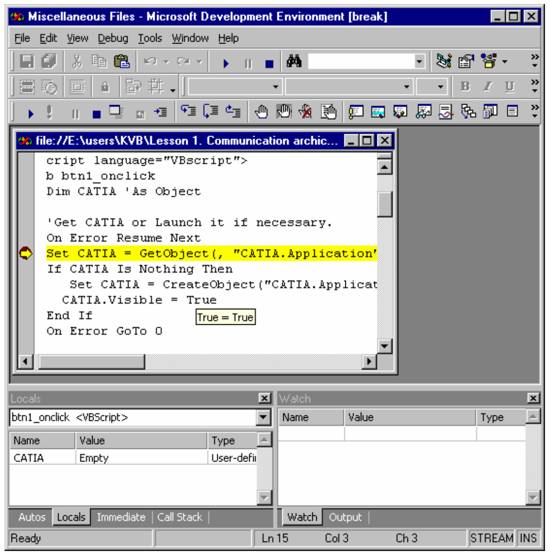
Student Notes:

Running Out-Process using the Windows Scripting Host

WSH enables scripts written in different languages such as VBScript and JavaScript.

You can use a standard Editor to write your application. If your program fails, you will be able to use the Windows Development environment.
It is possible to develop an application in Visual Basic, then put the types in comment and run it with WSH.

To run a VBScript (*.vbs) or JavaScript (*.js) file, you can use cscript or wscript commands.
Usually, wscript.exe is already associated with those extensions.


Miscellaneous Files - Microsoft Development Environment [break]
File Edit View Debug Tools Window Help
file://E:\users\KVB\Lesson 1. Communication archic...
True = True
Locals
btn1_onclick <VBScript>
Watch
Name Value Type
Autos Locals Immediate Call Stack
Watch Output
Ready Ln 15 Col 3 Ch 3 STREAM INS

```
cript language="VBscript">
b btn1_onclick
Dim CATIA 'As Object

'Get CATIA or Launch it if necessary.
On Error Resume Next
Set CATIA = GetObject(, "CATIA.Application'
If CATIA Is Nothing Then
   Set CATIA = CreateObject("CATIA.Applicat
  CATIA.Visible = True
End If
On Error GoTo 0
```

Name	Value	Type
CATIA	Empty	User-defi

Student Notes:

VBScript under Windows Scripting Host

You can write a VBScript program in a simple ASCII text file with the extension ".vbs" for Visual Basic Script.
Under WSH, GetObject and CreateObject are provided by the object Wscript.
The following code launches CATIA V5.

```
On Error Resume Next
 Set CATIA = Wscript.GetObject(,"CATIA.Application")
 If Err.Number <> 0 Then
    Set CATIA = Wscript.CreateObject("CATIA.Application")
    CATIA.Visible = True
  End If
```

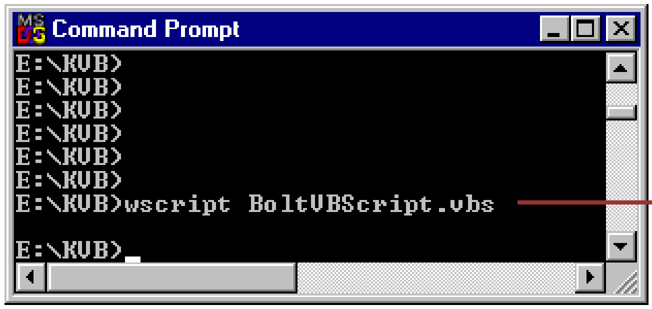
VBScript !

Student Notes:

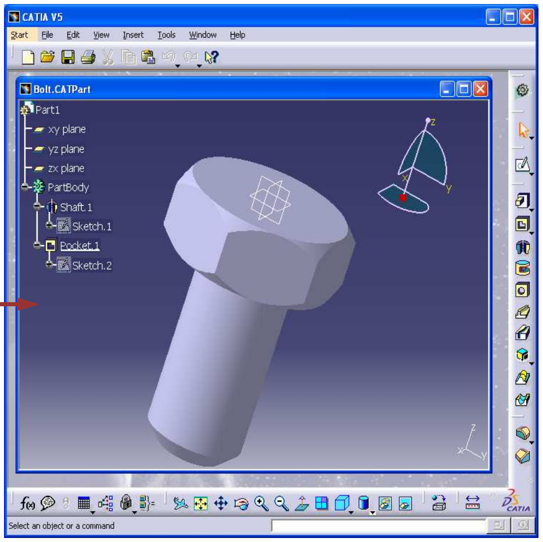
BoltVBSript.vbs Example

Have a look on the source code.
To run the “BoltVBScript.vbs” example , you can double–click on it.
To run the example on a Command prompt Window you can also use :

```
Cscript BoltVBScript.vbs
Or wscript BoltVBScript.vbs
```



```
E:\KVB>
E:\KVB>
E:\KVB>
E:\KVB>
E:\KVB>
E:\KVB>
E:\KVB>wscript BoltVBScript.vbs

E:\KVB>_
```



Student Notes:

JavaScript under Windows Scripting Host

In the same way as VBScript, you can write a JavaScript program in a simple ASCII text file with the extension ".js" for JavaScript.
Under WSH, GetObject and CreateObject are provided by the object Wscript.
The following code launches CATIA V5.

```
CATIA = WScript.GetObject(,"CATIA.Application");
 if ( CATIA == null) {
   CATIA = WScript.CreateObject("CATIA.Application");
   CATIA.Visible = true;
 }
```

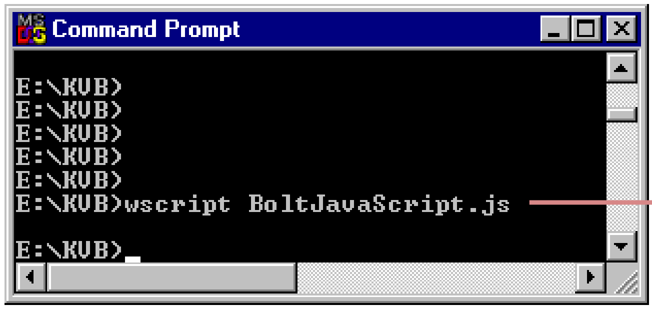
Java Script !

Student Notes:

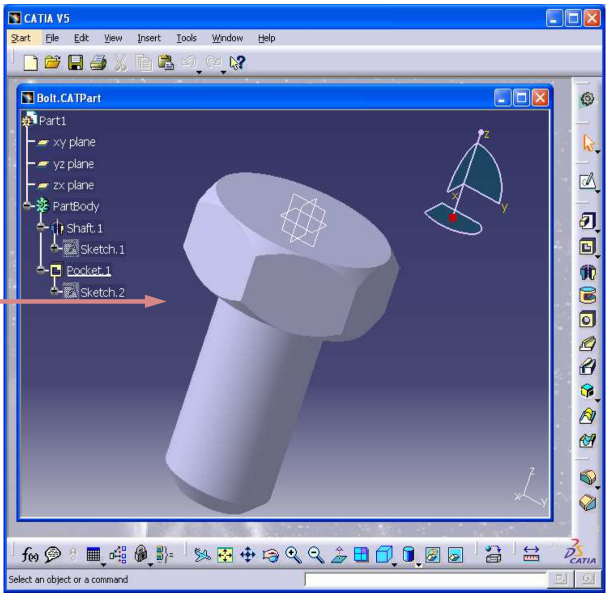
BoltJavaSript.js Example

Have a look on the source code.
To run the "BoltJavaScript.js" example, you can double–click on it.
To run the example on a Command prompt Window you can also use :

```
Cscript BoltJavaScript.js
Or wscript BoltJavaScript.js
```



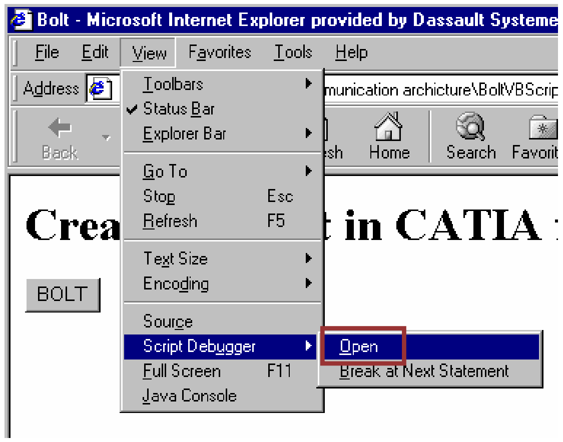
```
E:\KVB>
E:\KVB>
E:\KVB>
E:\KVB>
E:\KVB>
E:\KVB>wscript BoltJavaScript.js

E:\KVB>_
```

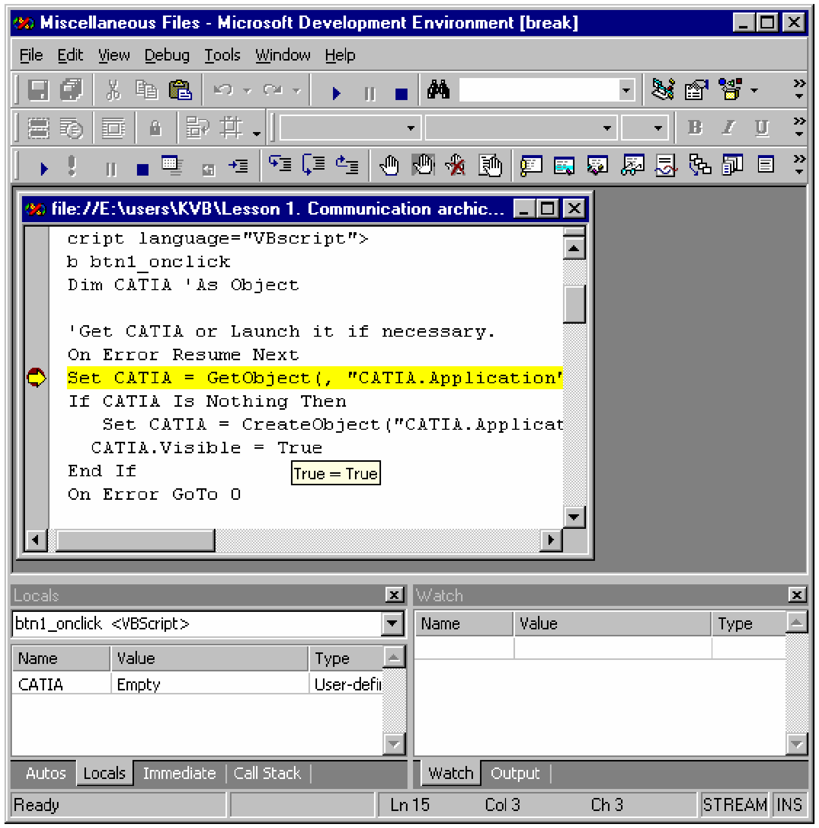


Student Notes:

Out-Process in HTML (Windows & Microsoft IE Only)

In HTML, you can include VBScript or JavaScript routines.
Microsoft explorer provides ActiveXObject (in JavaScript) and CreateObject
(in VBScript) functions to Launch Automation Servers.
So, it is possible to call CATIA V5 this way.
You can access to the Microsoft Development Environment to debug your Application.
You can launch the "Microsoft Development Environment from Explorer by :
"Script Debugger" " Open"




Miscellaneous Files - Microsoft Development Environment [break]
File Edit View Debug Tools Window Help
file://E:\users\KVB\Lesson 1. Communication archic...
True = True
Locals
btn1_onclick <VBScript>
Watch
Autos Locals Immediate Call Stack
Watch Output
Ready Ln 15 Col 3 Ch 3 STREAM INS

```
cript language="VBscript">
b btn1_onclick
Dim CATIA 'As Object

'Get CATIA or Launch it if necessary.
On Error Resume Next
Set CATIA = GetObject(, "CATIA.Application"
If CATIA Is Nothing Then
   Set CATIA = CreateObject("CATIA.Applicat
  CATIA.Visible = True
End If
On Error GoTo 0
```

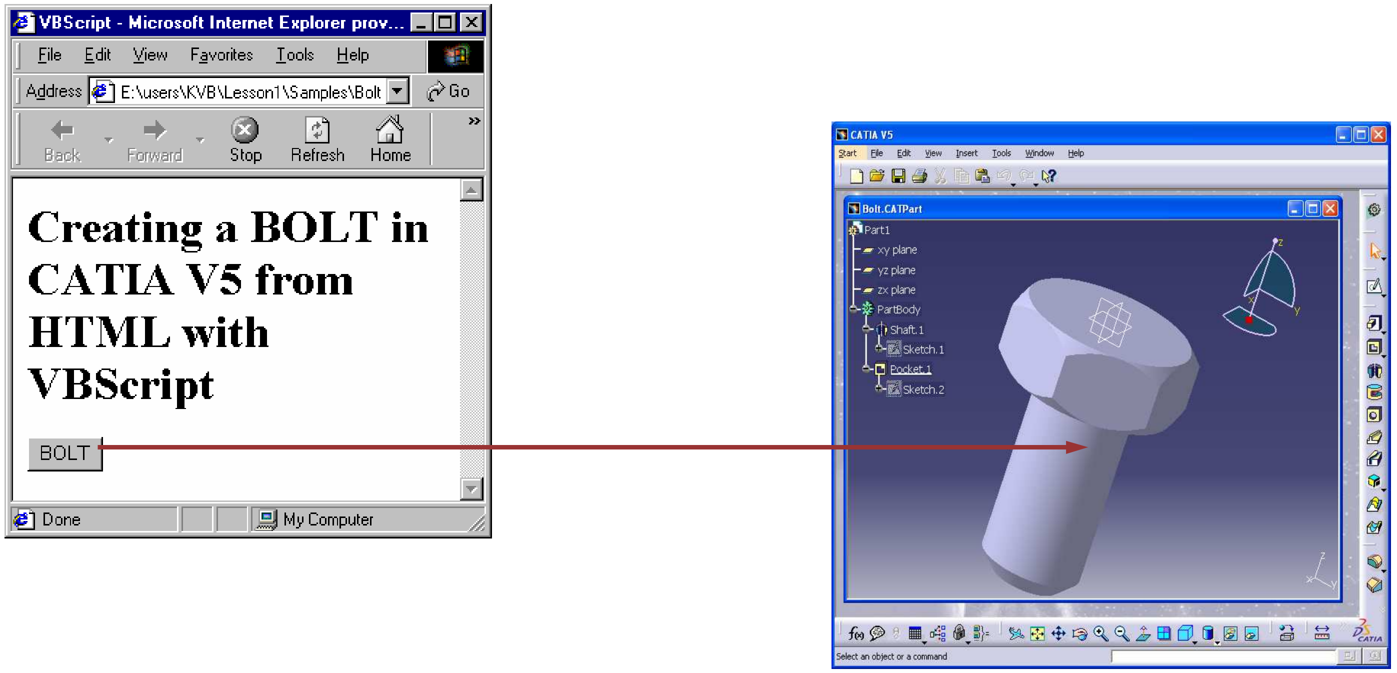
Name	Value	Type
CATIA	Empty	User-defi

Name	Value	Type

Student Notes:

Out-Process in HTML with VBScript

Load the “BoltVBScript.html” example in Microsoft Internet Explorer. You will be able to Click on the “Bolt” button to create a Bolt in CATIA V5.
You can view the source code by <View>+<Source> menus.



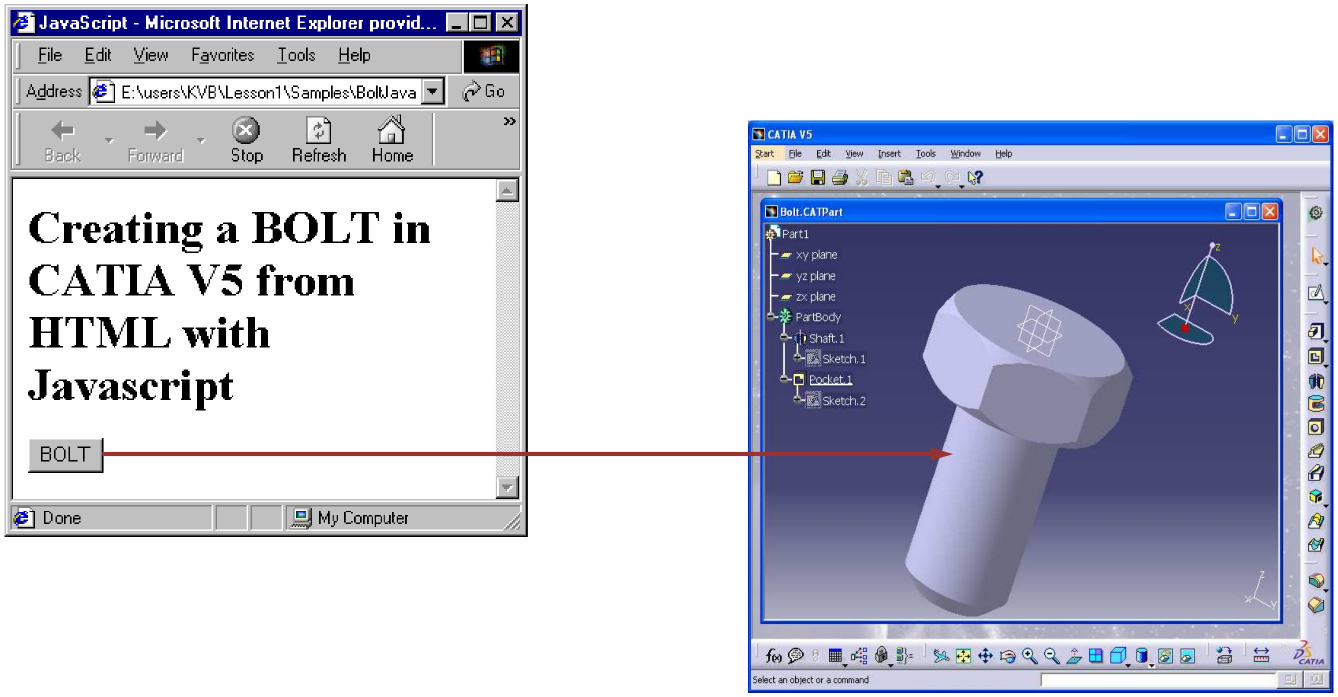
Creating a BOLT in CATIA V5 from HTML with VBScript

Due to security reasons, some warning messages for running the script, may appear depending on the configuration and if the server is local or not.

Student Notes:

Out-Process in HTML with JavaScript

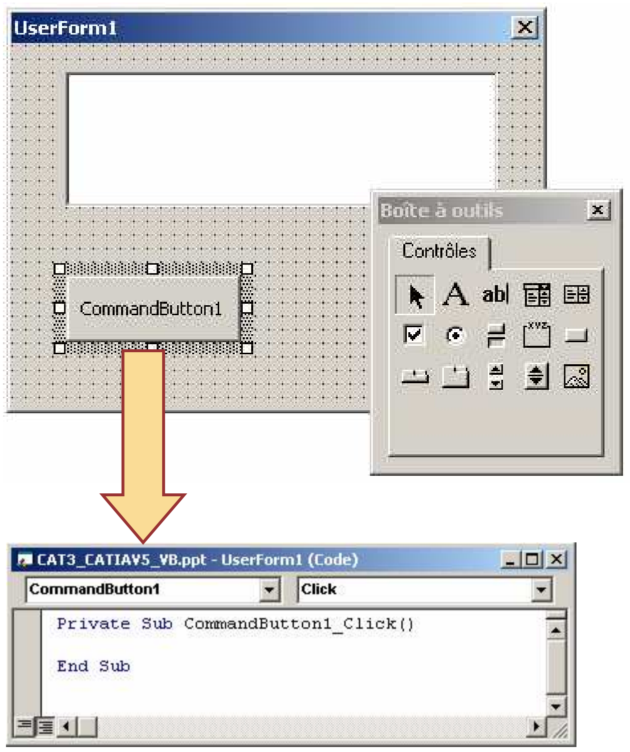
Load the “BoltJavaScript.html” example in Microsoft Internet Explorer. You will be able to Click on the “Bolt” button to create a Bolt in CATIA V5.
You can view the source code by <View>+<Source> menus.


Creating a BOLT in CATIA V5 from HTML with Javascript
JavaScript - Microsoft Internet Explorer provid...
File Edit View Favorites Tools Help
Address E:\users\KVB\Lesson1\Samples\BoltJava
Go
Back Forward Stop Refresh Home
BOLT
Done
My Computer
CATIA V5
Start File Edit View Insert Tools Window Help
Bolt.CATPart
Part1
xy plane
yz plane
zx plane
PartBody
Shaft.1
Sketch.1
Pocket.1
Sketch.2
Select an object or a command

Student Notes:

VBA / VB Programming Basics

You will be introduced to the Visual Basic and Visual Basic for Automation programming techniques.

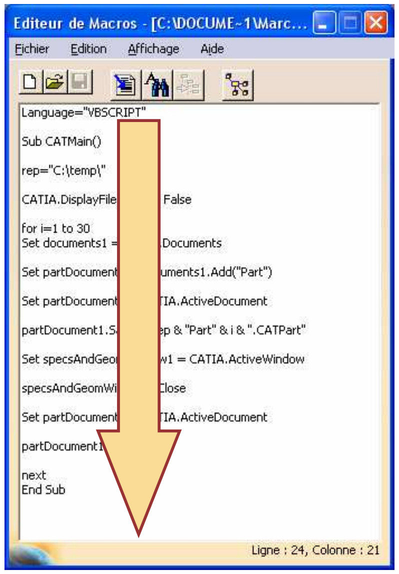


```
Private Sub CommandButton1_Click()

End Sub
```

Student Notes:

Programming Generalities (1/5)

- Sequential Programming
  - The program is read from top to bottom, sometimes with loops and tests.
    - 12 : If A=O then GOTO 67
  - User interface (windows, buttons, …) has to be « manually » built, with programming instructions
  - Typical of « old » programming languages
    - Fortran, Basic, GWBasic
  - VBScript may be considered as a sequential language



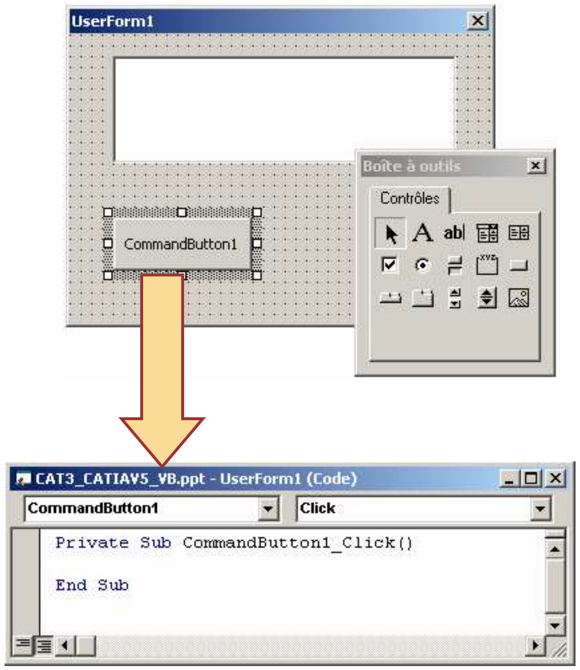
```
Language="VBSCRIPT"

Sub CATMain()

rep="C:\temp\"

CATIA.DisplayFile         False

for i=1 to 30
Set documents1 =          .Documents

Set partDocument          uments1.Add("Part")

Set partDocument          TIA.ActiveDocument

partDocument1.S           ep & "Part" & i & ".CATPart"

Set specsAndGeo           w1 = CATIA.ActiveWindow

specsAndGeomWi            Close

Set partDocument          TIA.ActiveDocument

partDocument1

next
End Sub
```

Ligne : 24, Colonne : 21

Student Notes:

Programming Generalities (2/5)

- **Event-driven Programming**
  - **The program execution will depend of events (mouse click on a button, typing characters in a text form…)**
  - **Graphic interface is drawn using a toolbox. This interface is the program «skeleton»**
  - **Nowadays programming languages are event-based**
    - **VBA, Visual Basic, Visual C++ …**



```
Private Sub CommandButton1_Click()

End Sub
```

Programming Generalities (3/5)

- Object Programming
  - In the application, all is Object
  - These Objects have Properties that you can read or set
  - These Objects have Methods which will modify their behavior
  - An example …

Student Notes:

Student Notes:

Programming Generalities (4/5)

- An example : Bernard owns a garage in Suresnes …
- As we have to work with its garage, we need to assign it to a «type», and a «variable name» :
  - Dim myGarage As Garage
  - set myGarage = Bernard.Garage
- To get its address (Proprerty of myGarage Object), I can get this :
  - myGarage.Address
- Bernard has a Collection of cars
  To get one special car :
  - Set myCar = myGarage.Cars.Item(2)
  - Set myCar = myGarage.Cars.Item('Golf_2')

Student Notes:

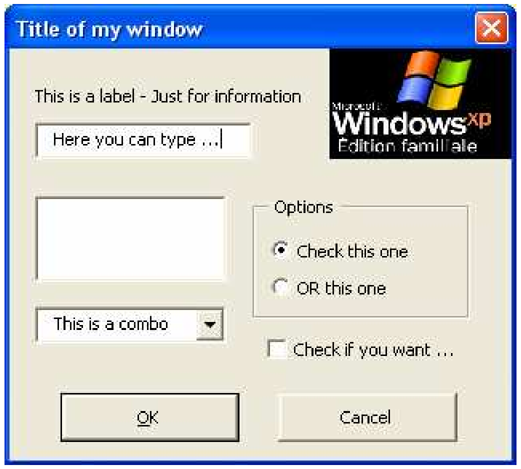
Programming Generalities (5/5)

- To select a fuel, I get a Property :
  - myCar.Fuel = ‘Petrol’
- If I do not remember its colour :
  - Colour = myCar.Colour
- To accelerate, I have a method which has the speed I want to reach (the speed is an argument) :
  - myCar.Accelerate (110)
- The same method may return an object or an information (here the fuel consumption) :
  - myConsumption = myCar.Accelerate (110)

Student Notes:

VBA / VB Common Controls

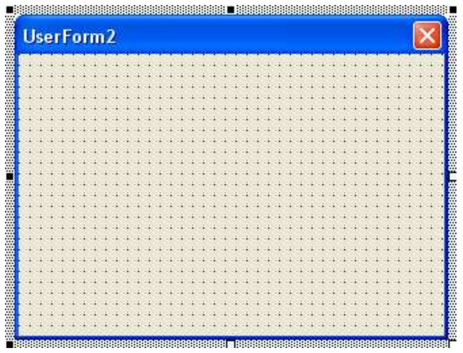
You will learn the user-interface related controls of Visual Basic and Visual Basic for Application.



Student Notes:

The « userform » Control

- Icon : -
- Interest :
  - It is your window, on which you are going to insert controls



- Useful Properties / Methods

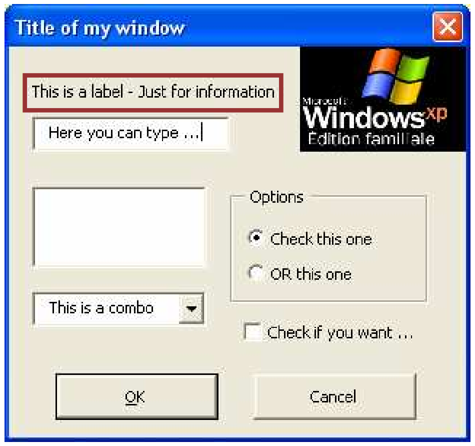
frmForm.Caption	to inform the title
frmForm.StartUpPosition	to control its position

  - UserForm_Initialize() is read when project is run (VBA)
  - UserForm_Load() is read when project is run (VB)

Student Notes:

The « label » Control

- Icon : A
- Interest :
  - The text label allows you to display a text that cannot be modified by user



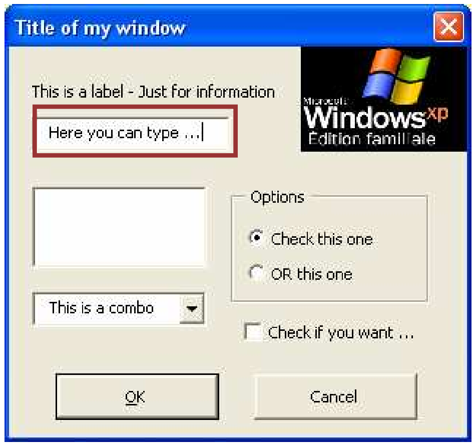
- Useful Properties / Methods

lblText.Caption	to control the text
lblText.Font	to control its font
lblText.Autosize	to control its dimensions
lblText.TextAlign	to control its alignment

Student Notes:

The « textbox » Control

- Icon : ab|
- Interest :
  - The textbox allows user to enter a text



- Useful Properties / Methods

txtBox.Text	to get the text
txtBox.Font	to control its font

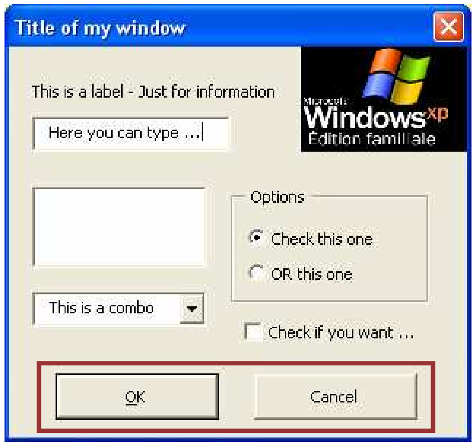
Student Notes:

The « commandbutton » Control

- Icon :

- Interest :
  - The command button allows user to run an action by selecting a button

- Useful properties / methods
  - cmdButton.Caption to control the button text

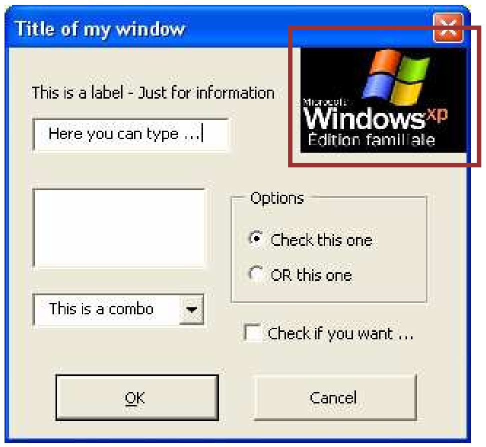


Student Notes:

The « image » Control

- Icon :

- Interest :
  - The image control allows you to display a bitmap image



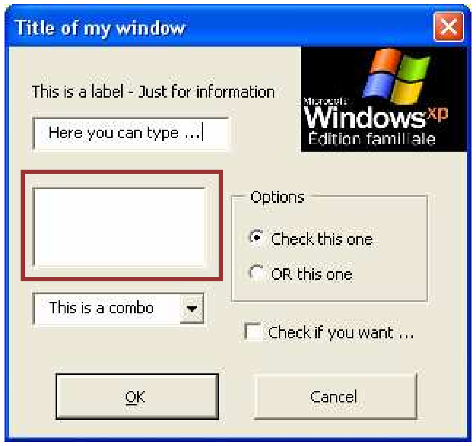
- Useful properties / Methods

imgImage.Picture	to select the image to display
imgImage.Autosize	to resize it automatically

Student Notes:

The « listbox » Control

- Icon :
-
- Interest :
  - The listbox control allows you to display a list, from which an element can be selected



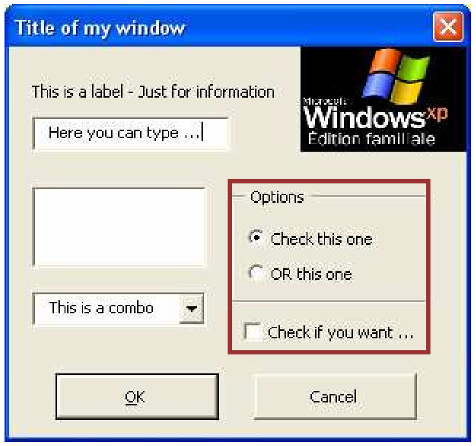
- Useful Properties / Methods

lstBox.Additem “Text”	to add an item
lstBox.Clear	to clear the list
lstBox.RemoveItem (lstBox.ListIndex)	to remove an item
lstBox.ListCount	to get the item count
lstBox.Text	to get the selected item
lstBox.List(i)	to get item #i

Student Notes:

The «checkbox» and «optionbutton» Controls

- Icons : ☑ ◉
- Interest :
  - The checkbox allows the user to pick an option. The optionbutton allows the user to pick only one among a set of options.



- Useful Properties / Methods
  - chkBox.Caption to control the text
  - lstBox.Value to get its value (true/false)

Student Notes:

Managing Data/Variables (1/3)

- Naming variables
  - It must be less than 255 characters
  - Special characters are forbidden : space '.' & ...
  - It must not begin with a number
    - 'myCar' and 'intRadius3' are ok
    - 'my Car' and '2Radius' are not ok

- Declaring variables with "Dim" is helpful for programming (it allows 'completion'), and saves memory at running
    - Dim myCar as String
    - Dim myCar$
    - Dim A1 as Integer
    - Dim A1%

Student Notes:

Managing Data/Variables (2/3)

- **Main types of Data**

Type	Non/Numeric	Range	Decl. Suffix
Boolean	Non-numeric	True or False	
Integer	Numeric	-32768 to 32767	%
Long	Numeric	-2 147 483 648 to 2 147 483 648 (integer)	&
Double	Numeric	-1.79769313486232e+308 to -4.94065645841247E-324 for ‘-’ values 4.94065645841247E-324 to 1.79769313486232e+308 for ‘+’ values	#
String	Non-numeric	Variable	$
Variant	both	-	

Managing Data/Variables (3/3)

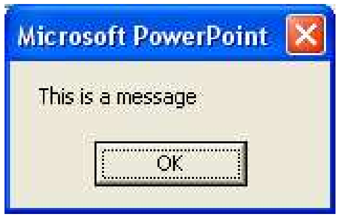
- Initializing Variables
  - Variable = expression
  - Set variable = expression
    - (you need set if 'variable' is an object)
  - Examples
    - A = 150
    - myCar = "Scenic"
    - A = B + 100
    - A = A + 100
    - Set myObject = CATIA.ActiveDocument

Student Notes:

Student Notes:

Standard VBA/VB window: « msgbox »

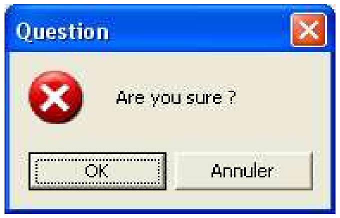
- Syntax #1
  - msgbox message
  - To display a single message (or variable)
    - example :
    - msgbox “This is a message”



Microsoft PowerPoint

This is a message

- Syntax #2
  - value = msgbox (message,[param.],[Title])
  - To ask a question, and get the result
    - example :
    - a1=msgBox("Are you sure?",vbOKCancel+vbCritical,"Question")



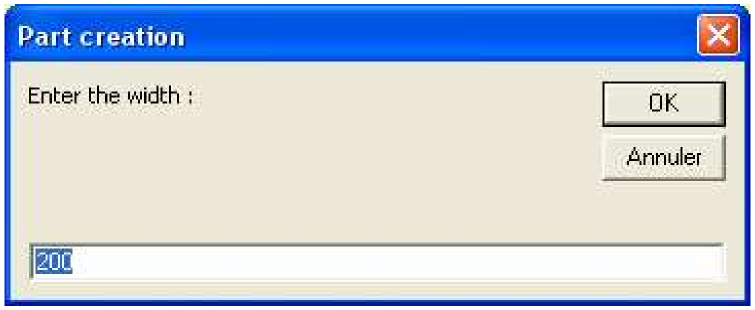
Question

Are you sure ?

Student Notes:

Standard VBA/VB window: « inputbox »

- Syntax
  - Value = Inputbox (message,title,def. val.)
  - To ask a question, and get an information (text or value)
    - Example :
    - Width = InputBox("Enter the width :", "Part creation", 200)



Enter the width :

200

Student Notes:

VB Programming Techniques

You will learn Visual Basic Programming Techniques which are required for CATIA V5 Automation and Scripting.

Student Notes:

Testing with 'If … then … else ... End if'

- Syntax #1
  - If (condition is true) Then
    ...
    End If
    - Example
      If Int_Diam >= Ext_Diam Then
      msgbox "Wrong values"
      End If
- Syntax #2
  - If (condition is true) Then
    ...
    Else
    ...
    End If

Student Notes:

Testing with ‘Select Case’

- Syntax
  - Select Case variable
    Case value1 (or expression1)
    …
    Case value2 (or expression2)
    …
    …
    End Select

    - Example :
      Select myResult.Text
      Case “A”
      lblResult.Caption = “xxx”

      Case “B”
      lblResult.Caption = “yyy”
      End Select

Student Notes:

Loops with 'For… Next and Do'…Loop

- Syntax
  - For i = x To y [Step z]
    …
    Next [i]
    - Example :
      For incr = 0 to 10
      msgbox "Increment is " + incr
      Next

- Syntax
  - Do While (condition is true)
    …
    Loop
    - Example :
      Do While i <10
      lblOut.Caption=i
      i=i+1
      Next

Student Notes:

Working with Strings of Characters

- A fixed string has to be surrounded by "
- To concatenate strings : « + » or « & »
    - Txt = "Hello, my name is " + myName
- To get a part of a string
    - left(str,nb) , right(str,nb), mid(str,pos,nb)
        - left(txt,5) (returns « Hello »)
- To get the length of a string
    - len(str)
- To know if a string contains another, and its position
    - Instr()
- To convert numeric to string, and reverse
    - str(num), val(str)

Student Notes:

Working with text files (Open Mode)

- To open a file and read its content
  - Open "filename" For Input As #id
    - To open the file
  - Line Input #id string
    - To read a line
  - Close #id
    - To close the file
    - Example :

```
Open “C:\temp\test.txt” For Input As #1
While Not EOF(1)
Line Input #1, line
msgbox line
Wend
Close #1
```

Student Notes:

Working with text files (Write Mode)

- To open a new file and write
    - Open "filename" For Output As #id
        - To open the file
    - Print #id ,string
        - To write a line
    - Close #id
        - To close the file
        - Example :

```
Open “C:\temp\testw.txt” For Output As #2
For i = 1 to 100
Print #2, i
Next
Close #2
```

Student Notes:

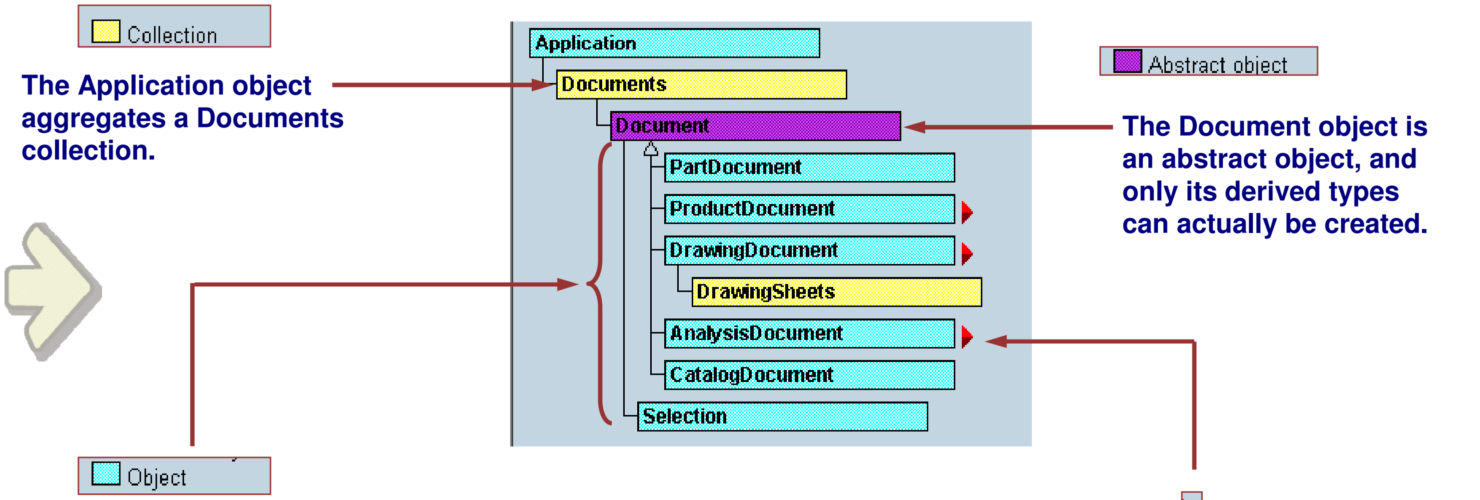
Scripting with CATIA V5

In this lesson you will learn various CATIA V5 scripting techniques. You will also see some examples of scripting with some CATIA Features.

Student Notes:

CATIA V5 Object Architecture

In this lesson, you will learn the Object Architecture and methodologies required for scripting with CATIA V5.



The Application object aggregates a Documents collection.

The Document object is an abstract object, and only its derived types can actually be created.

The ▶ symbol shows that the object is the root of an object structure expanded in another object diagram.

Student Notes:

Introduction to objects

Architecture of the Exposed Objects

CATIA V5 exposes models to create documents, parts, surfaces, wireframes, products, drawings and many other objects available in the CATIA workbenches.
Those exposed objects provide properties and methods to read and modify all existing objects.
All exposed objects seen in interactive, can be scripted from macros or from out-process programs.

- Table of contents

Student Notes:

About Objects, Collections, Properties, and Methods

Scripting languages such as Visual Basic rely on objects. With CATIA, documents, windows, viewers, parts, sketches, pads, … even lines and curves are represented as objects in Visual Basic.

- **Object: an entity.**
  Ex: Document, Pad, Line, …

- **Property: a characteristic of an object.**
  DocName = CATIA.ActiveDocument.FullName

- **Method: an action on an object**
  CATIA.ActiveDocument.SaveAs “MyNewName”

- **Collection: a list of objects.**
  The CATIA collections index begins at 1, and not 0 :

```
For i=1 to CATIA.Documents.Count
msgbox CATIA.Documents.Item(i).Name
Next
```

  We can also access a collection with the name of the object:
  Example: msgbox CATIA.Documents.Item("Product1.CATProduct").Name
  We add objects to the collections with the method add :
  set myPartDoc = CATIA.Documents.add("Part")

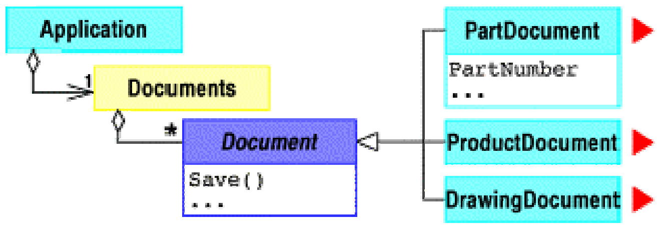
Student Notes:

About Inheritance and Aggregation

- There are two kinds of relations between objects :
  - Inheritance helps you to specialize objects while gathering common properties and methods
  - Aggregation is the ability of an object to contain another one

In the following diagram, the Application aggregates the Documents collection.

The PartDocument object is a specialized document that is dedicated to parts and which inherits the properties and the methods of the Document object.



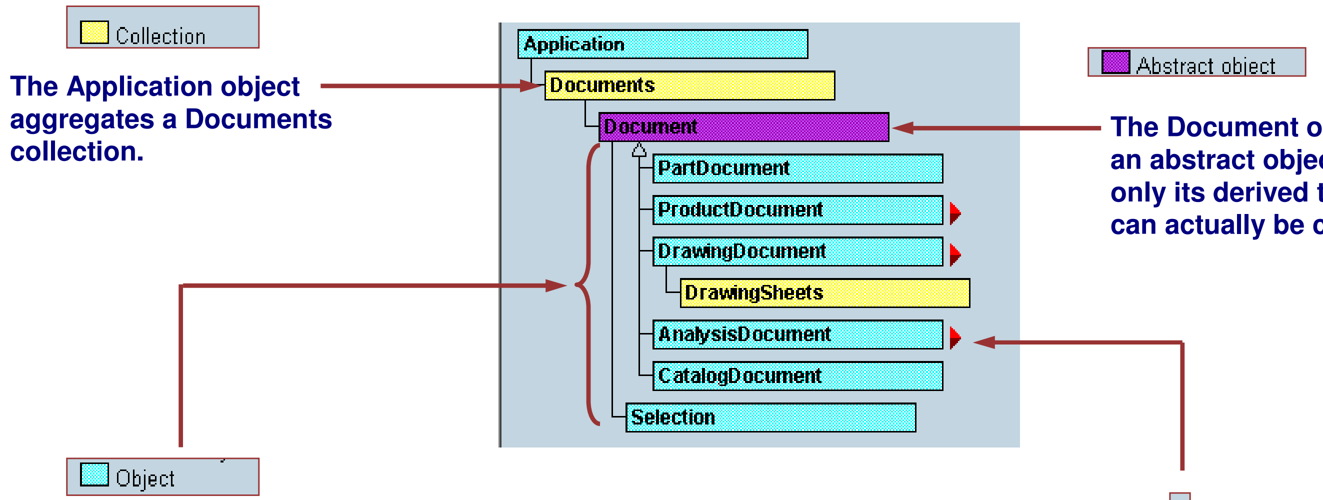
UML Notation

<u>*Student Notes:*</u>

Object Diagrams

To describe object, their properties and methods, and the relashionships between objects we use object diagrams and object descriptions.

Object diagrams describes the overall structure of how objects are linked together:



The Application object aggregates a Documents collection.

The Document object is an abstract object, and only its derived types can actually be created.

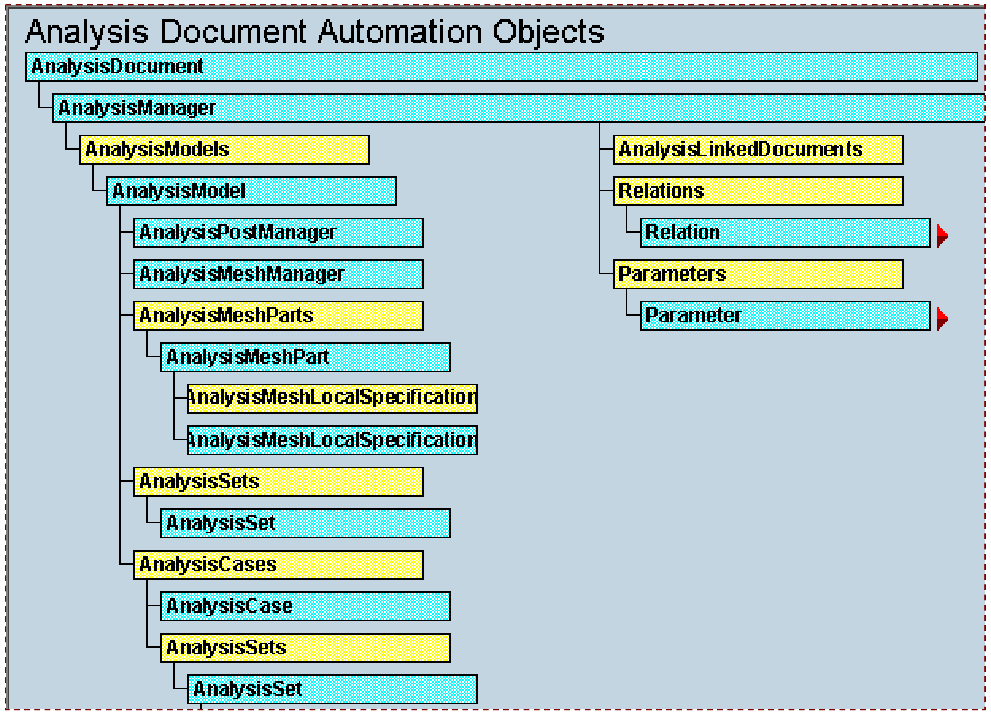
PartDocument, ProductDocument, DrawingDocument, AnalysisDocument and CatalogDocument documents inherit the aggregated Selection object. But only the DrawingDocument has a DrawingSheets collection.

The ▶ symbol shows that the object is the root of an object structure expanded in another object diagram.

Student Notes:

Analysis Object Model

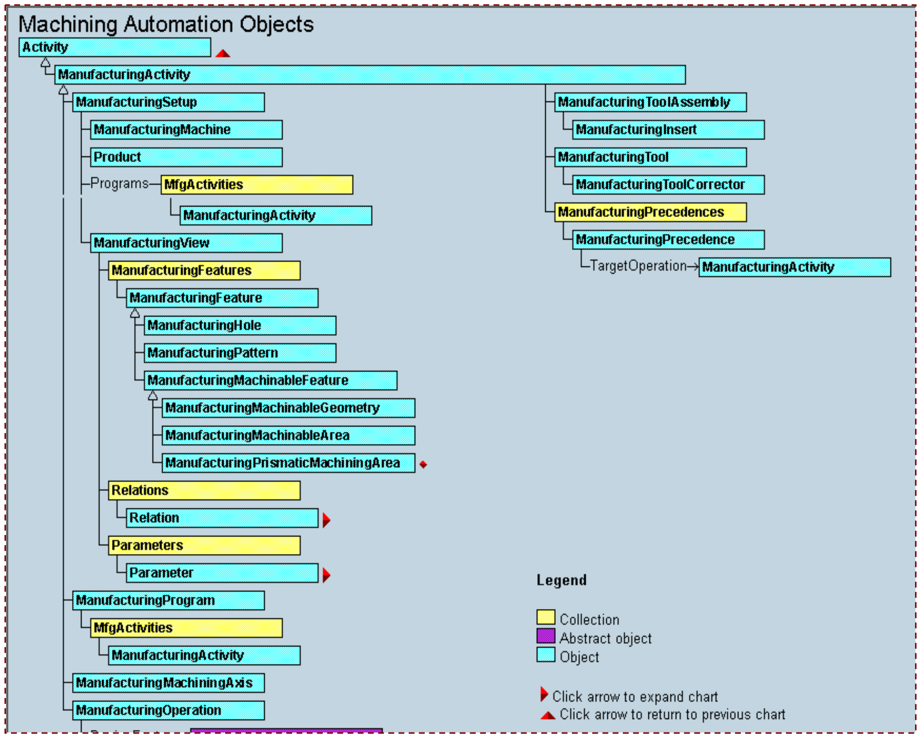
- Manage Analysis documents
- Manage linked documents
- Create AnalysisSet, AnalysisEntity …
- Integrated with Knowledgeware.
- Extract visualization Images
- Launch Computation



Student Notes:

Machining Object Model

- Full read/write access to Machining Objects
  - Manufacturing Activities
  - Manufacturing Resources
  - Manufacturing View, Manufacturing Features
- Manufacturing Activities creation in connexion with Knowledgeware



Student Notes:

Programming Method

- **Some tips about Sub and Function**
- **About the use of “Set” and “Call”**
- **About SafeArray variant**
- **Resolution of object**

Student Notes:

Some Tips about Sub and Function

- A method exposed by an objects is called by Visual Basic:
  - A Sub if it does not return any value
  - A Function if it does

To pass arguments to a Sub with Visual Basic Script, do not use parentheses:

```
Object.Sub arg1, arg2, arg3
```

But use parentheses with a Function:

```
Dim ReturnedObject As AnyObject
Set ReturnedObject = Object.Function (arg1, arg2, arg3)
```

Student Notes:

About the use of “Set”

- In Visual Basic, we use *Set* keyword to assign an object to a variable.

```
Dim myDocument As PartDocument
Set myDocument = CATIA.ActiveDocument
```

- We do not use *Set* in two cases:
  - If we do not assign an object to a variable but to an object property:

```
MyViewer3D.ViewPoint3D = MyCamera3D.ViewPoint3D
```

  - If we use a variable of intrinsic type:

```
Dim myString As String
mystring= ”directory”
```

Student Notes:

About the use of “Call”

In Visual Basic, a method can be a function or a subroutine.
A function returns something:

```
Set myReturnedObject = myObj.Function1(arg1,arg2)
or
    myReturnedValue = myObj.Function2(arg1,arg2)
```

A Subroutine returns nothing. Two syntaxes are possible:

```
Call myObj.Subroutine1(arg1,arg2)
or
    myObj.Subroutine1 arg1,arg2
```

If a subroutine has arguments, it is better to use the first syntax for readable reasons.

Student Notes:

About SafeArray Variant (1/2)

Many methods return an array of values instead of a single value.
In this case, you should use SafeArray of Variant.
A SafeArray is an Array declared with the "ReDim" instruction (instead of Dim):

- **Use ReDim functions to declare and assign a size to an array:**

```
ReDim MyArray(2) ☞ Declare a SafeArray and set a size
```

- **A SafeArray Variant begins at 0:**

```
MyArray(0) = 200 ☞ Set the value of the first element to 200
```

- **When you don't know the size of an array, use the Ubound property**

```
HighestRank = MyArray.UBound ☞ returns 2 in our example
```

Student Notes:

About SafeArray Variant (2/2)

Many methods need an array of values as argument instead of a single value.
In this case, you should use an untyped intermediary variable.

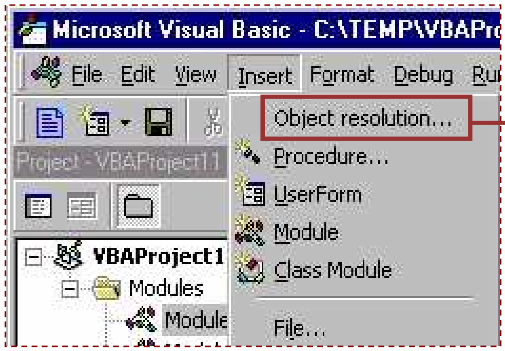
```
Dim MyFactory As Factory2D
Set MyFactory = mySketch.OpenEdition()

ReDim MyArray(3)
Dim MySpline1 As Spline2D

Dim myFactory2    ☞ untyped intermediary variable
Set myFactory2 = MyFactory

Set MySpline1 = myFactory2.CreateSpline(MyArray)
```

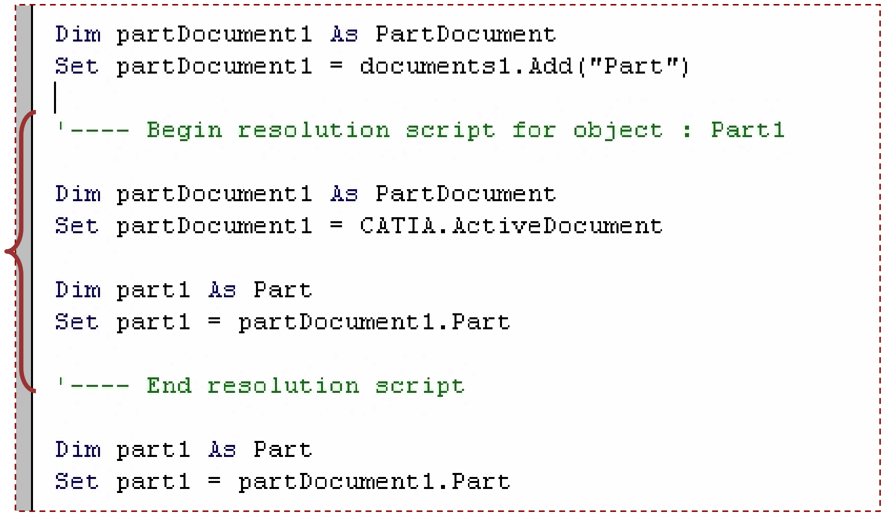
Student Notes:

Resolution of Object References (1/2)

When writing a script, one of the questions the developer frequently asks himself is: how do I programmatically retrieve the object I am interested to work with ?
It is now possible to select interactively the object from VBA or the V5 script editor and have to sequence of instruction to resolve the object directly inserted into the editor.

Object resolution in VBA

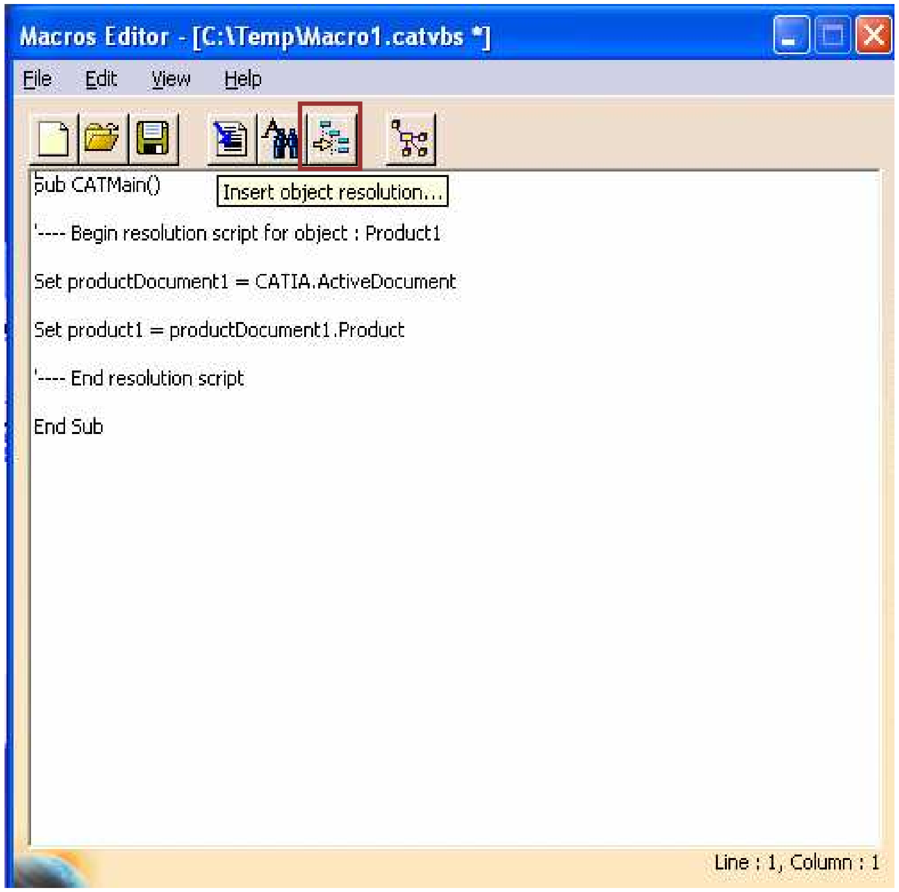




```
Dim partDocument1 As PartDocument
Set partDocument1 = documents1.Add("Part")

'---- Begin resolution script for object : Part1

Dim partDocument1 As PartDocument
Set partDocument1 = CATIA.ActiveDocument

Dim part1 As Part
Set part1 = partDocument1.Part

'---- End resolution script

Dim part1 As Part
Set part1 = partDocument1.Part
```

Resolution of Object References (2/2)

Object resolution with the V5 Script Editor


Macros Editor - [C:\Temp\Macro1.catvbs *]
File
Edit
View
Help
Insert object resolution...
Line : 1, Column : 1

```
Sub CATMain()

'---- Begin resolution script for object : Product1

Set productDocument1 = CATIA.ActiveDocument

Set product1 = productDocument1.Product

'---- End resolution script

End Sub
```

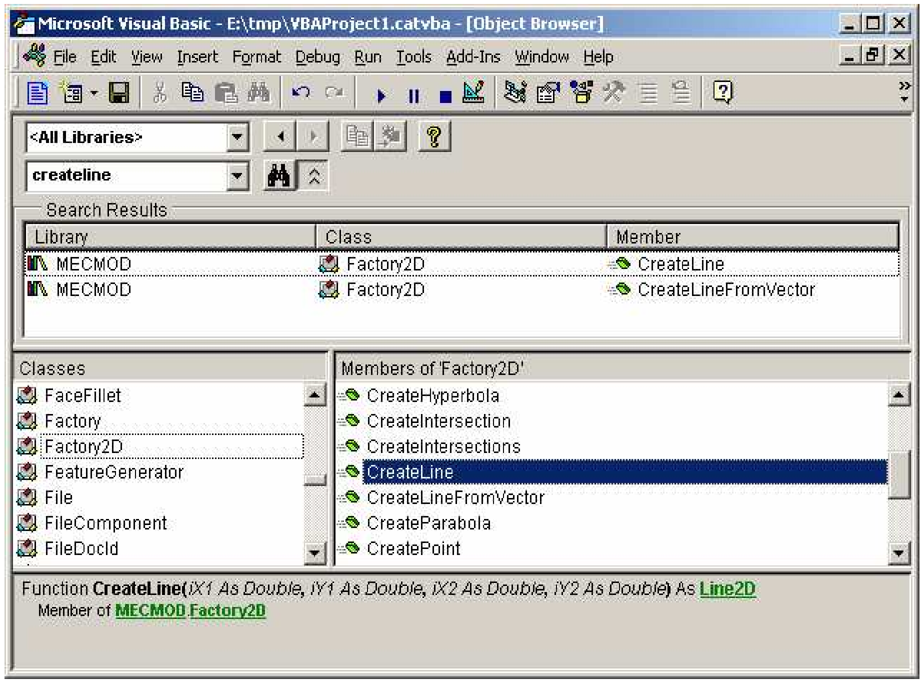
Student Notes:

Student Notes:

Object Browser Help (1/2)

- It can be found in the internal VBScript editor and in the VBA/VB editor

- It describes all objects/methods


Microsoft Visual Basic - E:\tmp\VBAProject1.catvba - [Object Browser]
File Edit View Insert Format Debug Run Tools Add-Ins Window Help
<All Libraries>
createline
Search Results

Library	Class	Member
MECMOD	Factory2D	CreateLine
MECMOD	Factory2D	CreateLineFromVector

Function **CreateLine**(*iX1 As Double, iY1 As Double, iX2 As Double, iY2 As Double*) As Line2D
Member of MECMOD.Factory2D

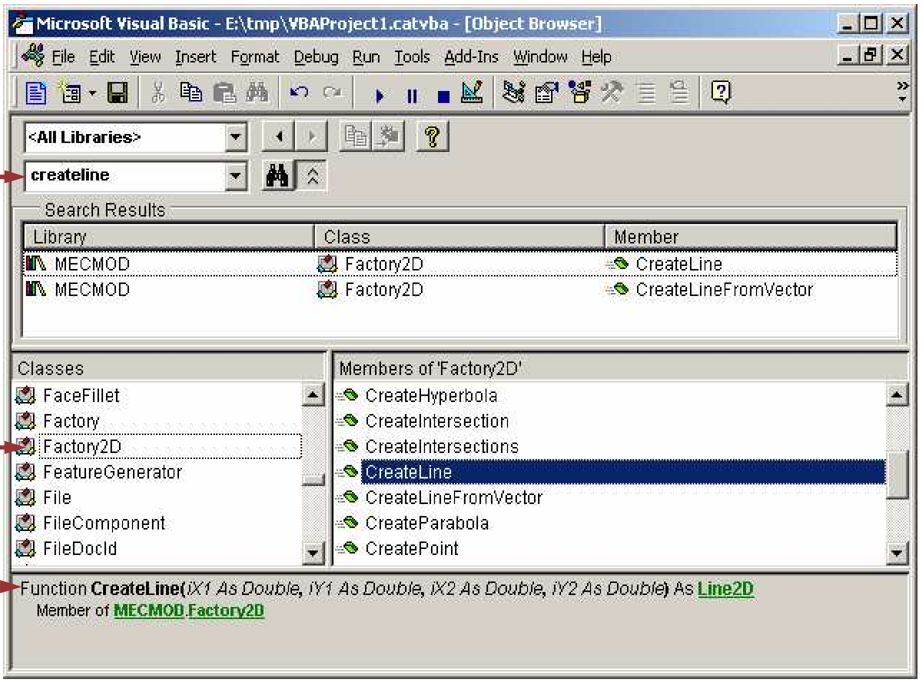
Student Notes:

Object Browser Help (2/2)

The Method that you are looking for.

The Method is member of a Class.

The Method has some arguments and returns an Object.


Microsoft Visual Basic - E:\tmp\VBAProject1.catvba - [Object Browser]
File Edit View Insert Format Debug Run Tools Add-Ins Window Help
<All Libraries>
createline
Search Results
Classes
Members of 'Factory2D'

Library	Class	Member
MECMOD	Factory2D	CreateLine
MECMOD	Factory2D	CreateLineFromVector

Classes	Members of 'Factory2D'
FaceFillet	CreateHyperbola
Factory	CreateIntersection
Factory2D	CreateIntersections
FeatureGenerator	CreateLine
File	CreateLineFromVector
FileComponent	CreateParabola
FileDocId	CreatePoint

Function **CreateLine**(*iX1 As Double, iY1 As Double, iX2 As Double, iY2 As Double*) As Line2D
Member of MECMOD.Factory2D

```
Dim MyFactory As Factory2D
…
Dim MyLine As Line2D
Set MyLine = MyFactory.CreateLine(10#, 10#, 10#, 30#)
```

Student Notes:

Scripting Infrastructure Features

You will learn scripting of Infrastructure Features.

```
Dim Doc as Document
Set Doc = CATIA.Documents.Add("Part") ☞ create a PartDocument
```

```
Dim Doc as Document
Set Doc = CATIA.Documents.Open("E:\Parts\DocumentToOpen.CATPart")
```

To close the active document : CATIA.ActiveDocument.Close
To close a Document assigned to a Doc variable : Doc.Close
To close a Document assigned to a name :
CATIA.Documents.Item("TheName").Close

To save the active document : CATIA.ActiveDocument.Save

To save a Document assigned to a Doc variable : Doc.Save

To save a Document assigned to a name :

CATIA.Documents.Item("TheName").Save

Student Notes:

Scripting Infrastructure Features Overview

Documents and Windows

The CATIA Application object aggregates two main collection objects: Documents and Windows.

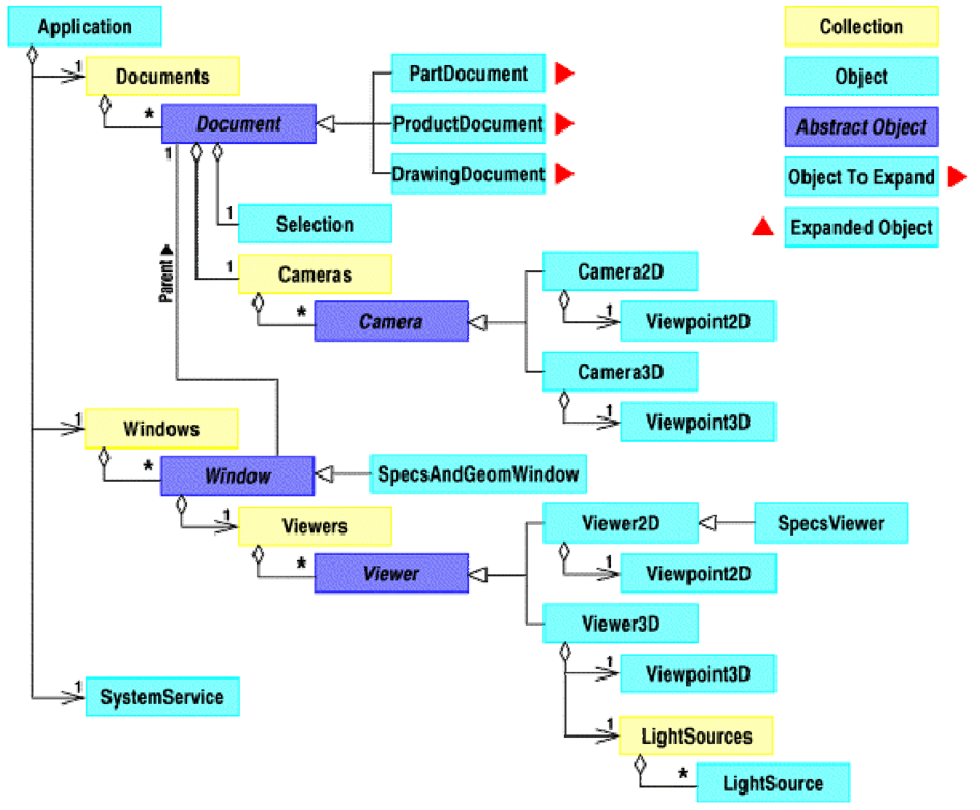
The Documents collection aggregates any number of Document(s) since the multiplicity is set to *. The Document object is an abstract object, and only its derived type can actually be created, that is the PartDocument, ProductDocument and DrawingDocument.

The Windows collection aggregates number of Window(s), which themselves aggregate one Viewers collection which aggregates any number of Viewer(s). The binary association between the Window and the Document objects means that the Document object is the parent of the Window object.

Student Notes:

Infrastructure Object Architecture

These objects are shared by all CATIA products. The root object is the Application, which aggregates or includes Documents and Windows.



Student Notes:

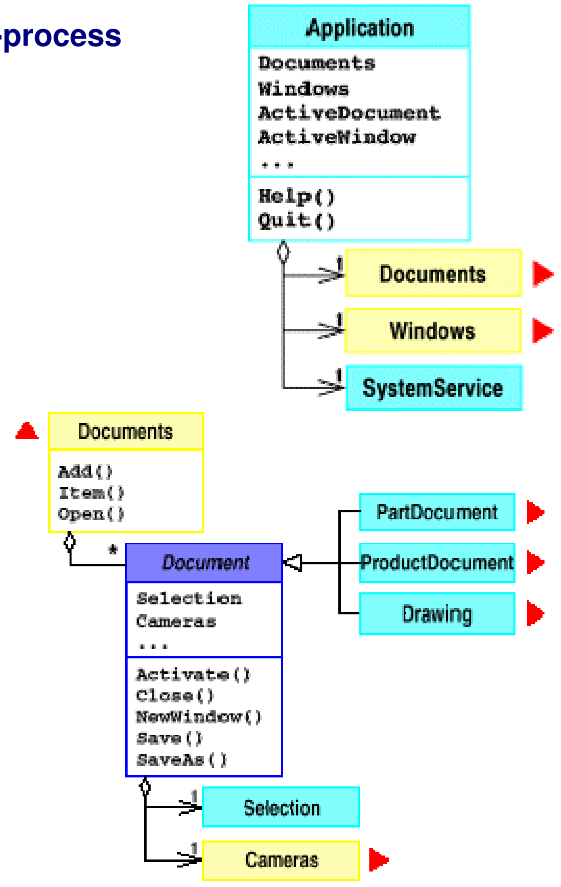
Application and Document

Application is the Root object for all CATIA macros. This correspond to the CATIA frame window.
The CATIA application is always named CATIA for in-process macro.

The Documents collection provide a method to Add a new document and another to Open an existing document.

A Document can be a PartDocument, ProductDocument or a DrawingDocument.

Each Document provide Its own methods to Save or SaveAs Itself.



Student Notes:

Creating and Opening a Document

Creating a New Document:

```
Dim Doc as Document
Set Doc = CATIA.Documents.Add(“Part”) ☞ create a PartDocument
```

Instead of “Part”, Use “Product” for a Product Document or “Drawing” for a Drawing Document

Opening a New Document:

```
Dim Doc as Document
Set Doc = CATIA.Documents.Open(“E:\Parts\DocumentToOpen.CATPart”)
```

Both Add and Open functions add the Document in the Documents Collection

Student Notes:

Closing and Saving a Document

Closing a Document:

To close the active document : CATIA.ActiveDocument.Close
To close a Document assigned to a Doc variable : Doc.Close
To close a Document assigned to a name :
CATIA.Documents.Item("TheName").Close

The Close function removes the Document from the Documents Collection

Saving a Document:

To save the active document : CATIA.ActiveDocument.Save
To save a Document assigned to a Doc variable : Doc.Save
To save a Document assigned to a name :
CATIA.Documents.Item("TheName").Save

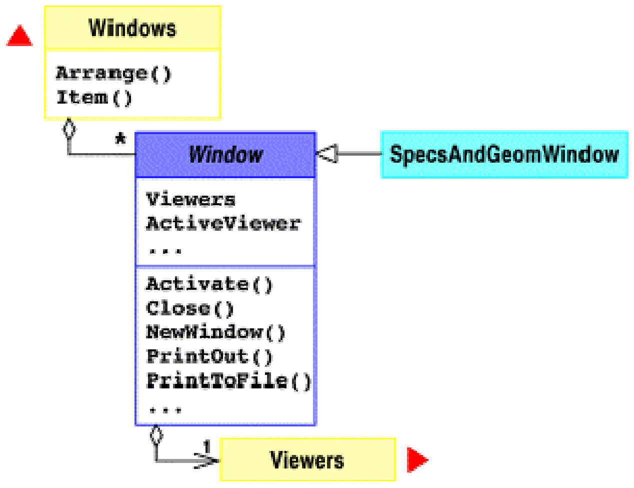
Student Notes:

Windows Collection

The Windows collection gathers Window objects that make the link with the windowing system and display documents in a viewable form, mainly in 3D or 2D modes, or as a specification tree.

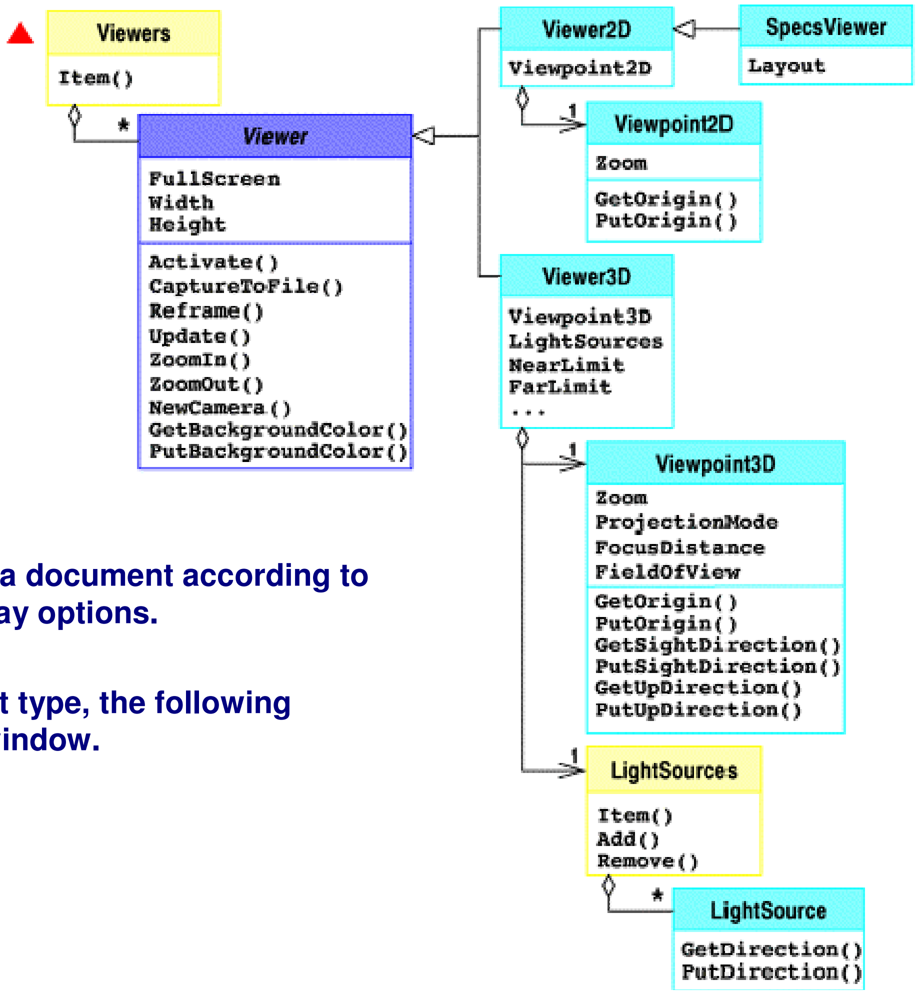
A Viewers collection enables the window to display the application data in the appropriate modes.

A SpecsAndGeomWindow object features altogether a 2D or a 3D viewer and a specification tree viewer.



Student Notes:

Viewers and Viewpoints



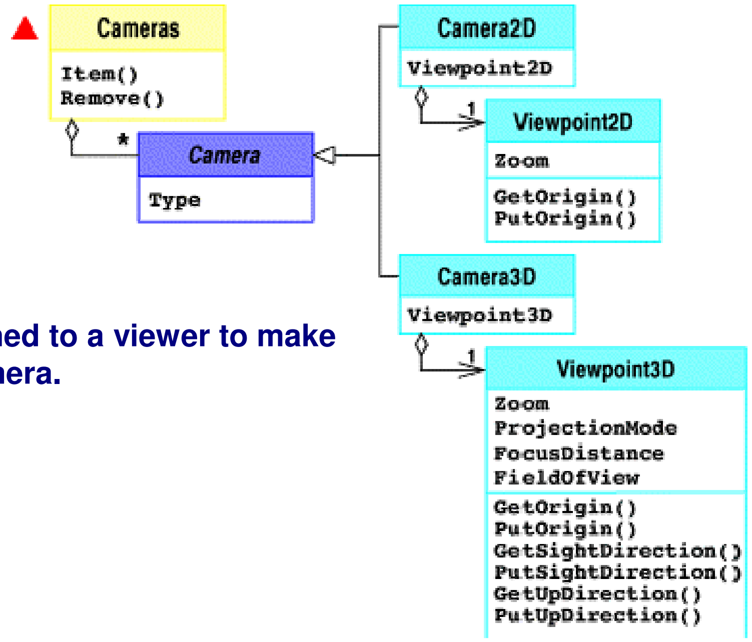
A Viewer is used to display a document according to a given viewpoint and display options.

Depending on the document type, the following viewers can be found in a window.

Student Notes:

Cameras

A Camera is the persistent form of a viewpoint. You can create a camera from the current viewpoint using the NewCamera method of the Viewer object.

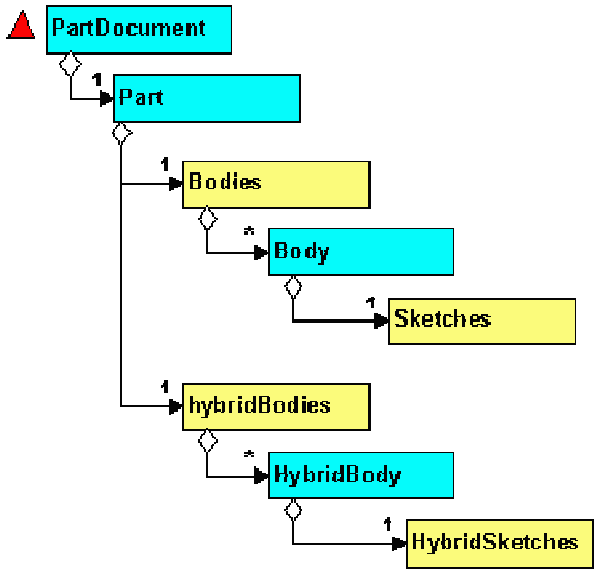


A created camera can then be assigned to a viewer to make its own viewpoint change to this camera.

Student Notes:

Scripting Sketches

You will learn scripting of Sketches.



Student Notes:

Scripting Sketches Overview

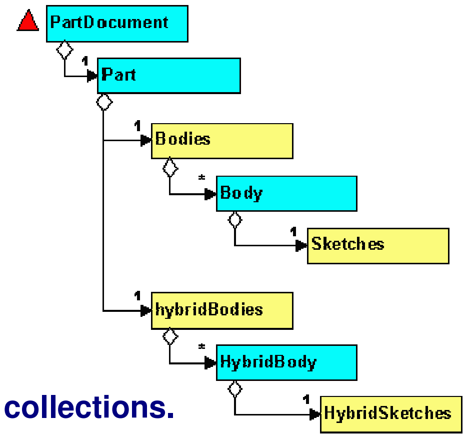
Bodies and HybridBodies Collections

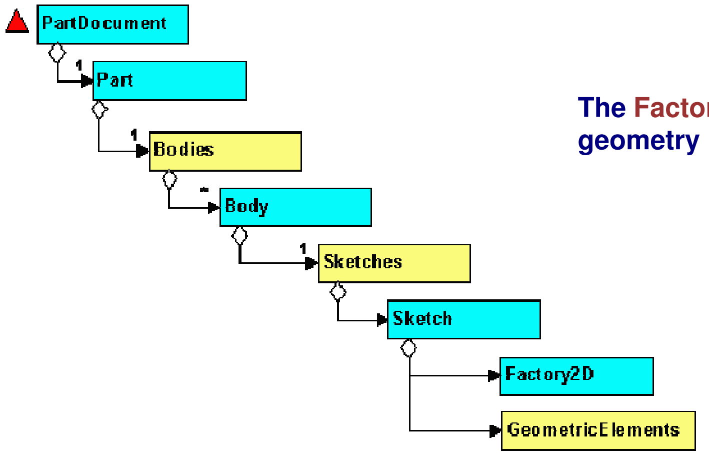
In the "part" of a PartDocument, there are 2 collections:

- **Bodies aggregate all the solid features**
- **HybridBodies aggregate all the surfaces, wireframe and 3D points (called OpenBody in interactive)**

Two different containers for sketches

Sketches can be created in both Bodies and HybridBodies collections.





The Factory2D contains methods to create geometry in the GeometricElements collection.

Student Notes:

Scripting Sketch Geometries

Opening and Closing a Sketch.

To create 2d elements in a sketch, we have to open it and get the factory2d using:

set myFactory2D = mySketch.OpenEdition

A Sketch has its own Factory2D to create all 2D objects available in it.

When you have finished to create geometry, you must close the sketch using mySketch.CloseEdition

Creating and Editing Geometry

Generally we have only one simple constructor for each element type, for example:

To create a circle, we have:

Function CreateClosedCircle(iCenterX As Double, iCenterY As Double, iRadius As Double) As Circle2D

If you want the circle centre to be an existing point, you can modify it with the “CenterPoint” property.

Easy methods to create Constraints

The Constraints collection allows you to add constraints between 2D elements.

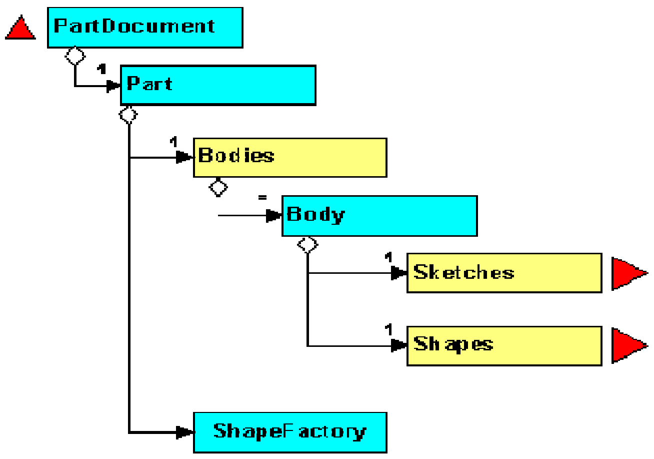
Recording a sketch creation in a macro

You have to exit the sketch before stopping the record.

Student Notes:

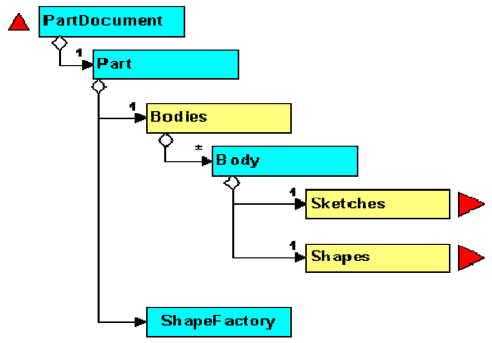
Scripting Part Design Features

You will learn scripting of 'Part Design Features'



Student Notes:

Scripting Part Design Features Overview



Scripting Part Design features means to create shapes in the bodies of a PartDocument.

Getting or creating a Part Document

Use ActiveDocument or Documents.Item(“Part1.CATPart”) or Documents.Add(“Part”)

Getting or creating a “PartBody”

When we create a PartDocument, a PartBody is created automatically.

We can get it using:

Set MyBody = MyDocument.Part.Bodies.Item("PartBody")

Or we can add a new PartBody using:

Set MyNewBody = MyDocument.Part.Bodies.Add()

Student Notes:

Scripting Functions for Part Design Features

The ShapeFactory

The ShapeFactory object is in the Part.

It allows you to create Pads, Shafts, Grooves, Holes, etc in the “active” Body.

You can redefine the “active” body using the InWorkObject property.

Boolean Operations

If you need to make a boolean operation with another Body, you can add a new body with:

MyDocument.Part.Bodies.Add

Then call AddNewAdd, AddNewIntersect, AddNewTrim or AddNewRemove method.

References

Some functions need references to objects instead of the object itself. References are used to bind an object to an other.

You can use:

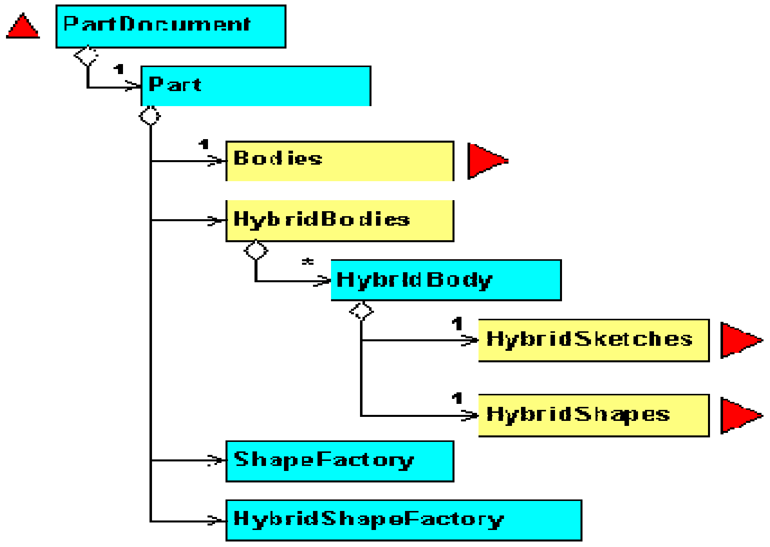
Dim myRef as Reference

Set myRef = Part.CreateReferenceFromGeometry(myGeometry) to create a reference on myGeometry object.

Student Notes:

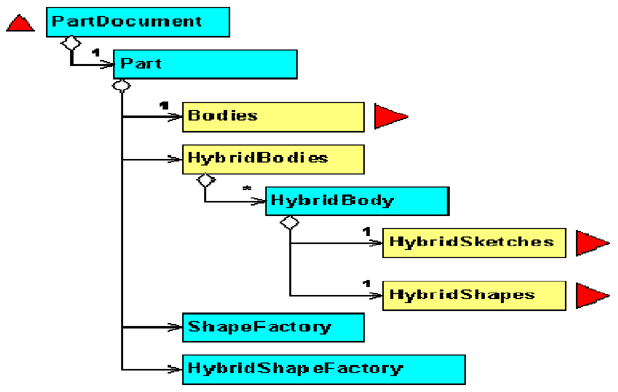
Scripting Shape Design Features

You will learn scripting of Shape Design Features



Student Notes:

Scripting Shape Design Features Overview



Scripting Shape Design features means to create HybridShapes in the HybridBodies of a PartDocument.

Why Hybrid ?

In Generative Shape Design, some elements are called “hybrid” because they can be surfaces, lines or points.

The HybridBody is called OpenBody in interactive.

Creating an HybridBody.

When we create a PartDocument in Shape Design, an HybridBody is created automatically.

If we are in a Part Design context, no Open_Body is created.

Student Notes:

Scripting Functions for Shape Design features

The HybridShapeFactory

The HybridShapeFactory is in the Part object.

It allows you to create in the “active” HybridBody” 3D points, 3D lines, surfaces such as extrusions, sweeps, etc.

You can redefine the “active” HybridBody using the InWorkObject property.

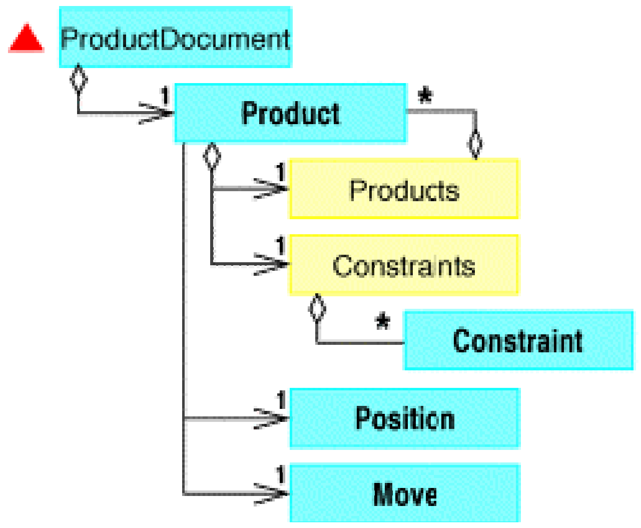
AppendHybridShape

To make your 3D lines or shapes visible, you have to use the AppendHybridShape method onto the HybridBody.

Student Notes:

Scripting Assembly Design Features

You will learn scripting of Assembly Design Features.



Student Notes:

Assembly Design – Object Model

Each Product contains a Products Collection

Each product of this collection can itself include a collection of products, and so forth. This defines a tree structure.

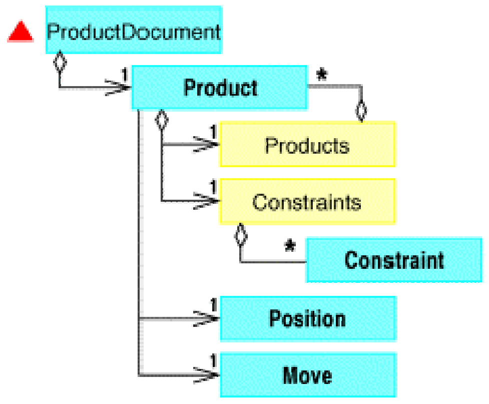
The RootProduct

The ProductDocument object aggregates a product tree structure starting with a single product, named the Root Product, that includes a collection of products.

Positioning the Sub-Assemblies

Each product has a position defined by a 3D matrix relatively to the position of its father.
Move.Apply allows a relative move.

Geometric constraints (coincidence, contact, etc) can be applied between product components.



Student Notes:

Assembly Design - Definitions

Definitions

ReferenceProduct : Each product owns a ReferenceProduct that can be another Product or the hidden Product of an other document.(for example a PartDocument). (eg: a wheel).

Component : Is an instance of a ReferenceProduct, (eg: the front wheel of car). A component owns its own characteristics related to how it is integrated in the assembly (for example: its relative location).

Product.PartNumber : The name of the Reference.

Product.Name : Name of the Instance.

Student Notes:

Functions for Scripting Assembly Design Features (1/2)

Adding Products

- Product collections can be built with following methods:
  - AddNewProduct("PartNumber")
    Creates both a Reference Product to the product and a component in the Product Document
  - AddNewComponent("DocumentType", "PartNumber")
    Creates both a Reference Product to the product and a component in the Product Document. The Reference Product is a new document
  - AddComponent(ReferenceProduct)
    Creates a product component from a Reference Product already existing in the Product Document
  - AddExternalComponent(ProductDocument)
    Creates a product component from a product reference already existing as root product of another Product Document
  - AddComponentsFromFiles("Document"(), "DocumentType")
    Creates a product component using an external file.

Student Notes:

Functions for Scripting Assembly Design Features (2/2)

Master Shape Representation

Each product can have a different Master Shape Representation that defines its geometry, material properties and used to display or print it

This representation is shared by all the components pointing to this product reference. It can be a :

- CATIA V4 model
- CATIA V5 Part
- VRML File
- or whatever format supported by CATIA V5

Bill Of Material

We can generate a Bill Of Material directly in a text or HTML file using the ExtractBOM method.

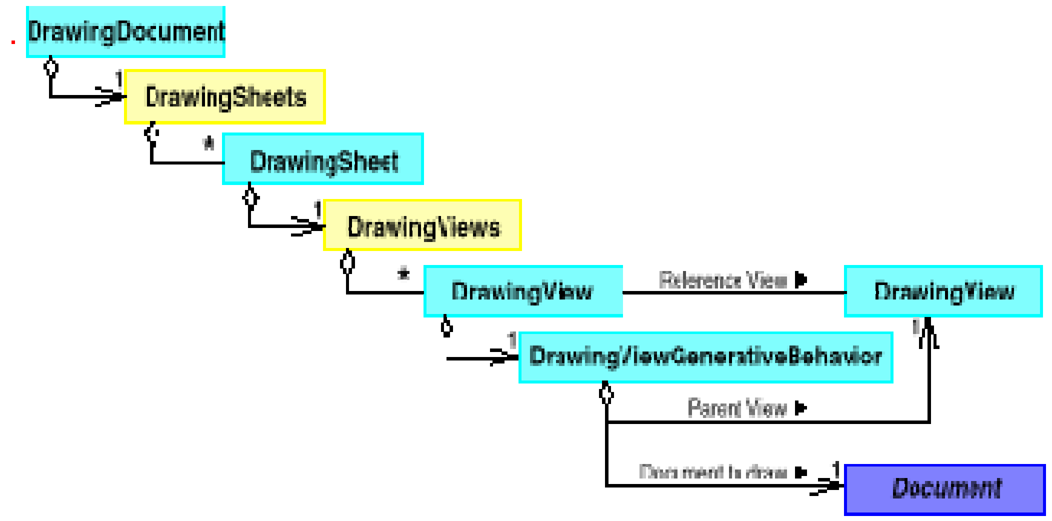
Instantiating a Product

To instantiate a product, you have to use AddComponent() using the ReferenceProduct of the existing product as argument.

Student Notes:

Scripting Drafting Features

You will learn scripting of Drafting Features.



Student Notes:

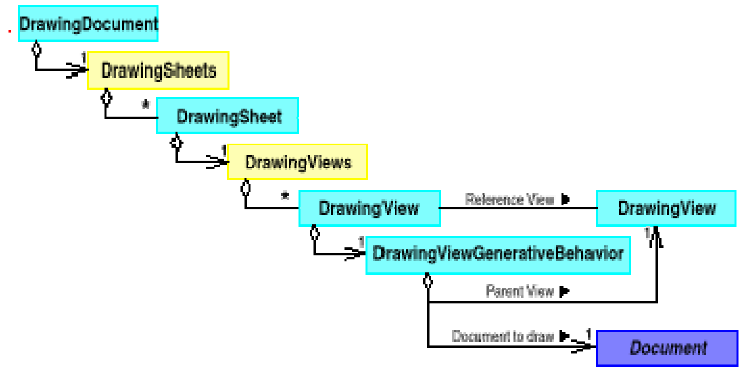
Scripting Drafting Features Overview

Drafting features regroup all the 2D features.
Generative Drafting represent 3D objects in the drawing and Interactive Drafting represent 2D features directly created in the 2D views.

DrawingSheets and DrawingViews

The DrawingDocument object aggregate a DrawingSheets collection.

Each DrawingSheet of this collection aggregates a DrawingViews collection.



Main View and Background View.

A drawing contains at least 2 views: The background view and the Main View that is the view in which you can draw just after creating a drawing.

Student Notes:

Function for Scripting Drafting Features (1/2)

The ActiveSheet

The active sheet can be found in the DrawingSheets collection

The ActiveView

The active view can be found in the DrawingViews collection

The Front View and the Others

A DrawingView can be the “Front View” or an other view relatively to the “Front View”. DrawingViewGenerativeBehavior can define both behaviour:

DefineFrontView() defines the “Front View” and its direction.

DefineProjectionView() defines an other view relatively to the “Front View”

Other methods allow to create other types of views such as Section Views, Detail Views, etc.

Student Notes:

Function for Scripting Drafting Features (2/2)

Generative Views
DrawingView owns a “Document” property that allows you to specify the associated 3D document from which the “generative view” is created.

Interactive Views
DrawingView owns a “Factory2D” property that allows you to create 2D features. This kind of view is called an “Interactive View”.

Filling the Title block
The Texts collection allows you to create 2D texts in the drawing

Extract Dimensions
A DrawingSheet has the method called GenerateDimension() that generates dimensions automatically in all the views from the 3D geometry.

Student Notes:

Accessing Elements

You will learn methods of accessing the geometric elements using scripts.

Student Notes:

Automatic Selections

Sometimes, graphic selections are not necessary and we just want to get all objects of a certain type using some selection filters.

We can do that just by scanning the containers and testing some criteria

We can scan for instance:

- All the documents or a specific document.
- All the sketches or a specific sketch
- All the geometric elements of a sketch
- All the Shapes
- All the HybridShapes

Student Notes:

Scanning the Containers

We can scan all the collections with a simple loop

```
For i = 1 to myDocuments.Count
 … myDocuments.Item(i)…

Next
```

Every container like Documents, Shapes, HybridShapes, Sketches, GeometricElements, Texts (in a drawing) … can be scanned like this.

Student Notes:

Access to a Specific Element

Access in a collection with the name.

We can access to a specific element of a collection by its name.

… Documents.item("ItsName") …

Or it can be the Father of an element that we know.

myElem.Parent.Parent… ' We can get all its parents until CATIA.Application.

For example, if you have a 2D point in a sketch:

Point2D.Parent	is the GeometricElements Collection
Point2D.Parent.Parent	is the sketcher

Access with Selection.Search()

We can access to a specific element using Search() of the "Selection" object.

The results will be added to the selected objects.

We can use Selection.FindObject() to scan the results.

Graphic Selections

Macros and Out-Process applications need a service so that the user can make Graphic selections.

There are two ways to make graphic selections.

1. **We can selected the object before entering the macro (or program).**
2. **We can call a function inside a macro (or program) to ask for an acquisition.**

Student Notes:

Graphic Selection using “Selection” (1/2)

- We can read the selection Object but the geometric element must be selected before starting the command.

In application like Word or Excel, the selections always precede the commands.

- If we don’t care about the order of the selected elements,
  the user can make only one selection.
  Then we can call FindObject several times for each type of object.

- If we care about the order of the selected elements with the same type,
  we must call several FindObject with the same type to get the first element then the others…
  We will ask for several acquisitions.

Student Notes:

Student Notes:

Graphic Selections using “Selection” (2/2)

We can scan this object using FindObject() function as follow. You can loop until the selection is empty.

```
    Do
          Set xObj = xSel.FindObject(MyType)
          If (Err.Number <> 0) Then
             Err.Clear
             Exit Do
          Else
            … xObj is selected
          End If
     Loop
```

Limitations :

All the objects must be selected before calling the macro (or an out-process function).

Student Notes:

Graphic Selections using SelectElement2

During a program, we can call the SelectElement2 method to ask to the user for elements of different types.
SelectElement2(iFilterType() As Variant, iMessage As String, iInterwindowSelection As Boolean) As String

- We can ask for several types (Multi-Type)
- We can loop on this method to get several elements (Multi-Selection)

It will create a specific list of selected elements (different than the selected elements accessible with FindObject() in the standard Selection)
To read this list, you can use Selection.Count and Selection.Item(I).

This is an example which asks the user to select a circle or a face in the current window:

```
Dim InputObjectType(1)
InputObjectType(0)="HybridShapeCircle"
InputObjectType(1)="Face"
Status= CATIA.ActiveDocument.Selection.SelectElement2(InputObjectType, "Select a circle or a face",
True)
```

Student Notes:

Creating Panels from Macros

With Out-Process, we can use an external user interface depending of the context. It can be Visual Basic, HTML or something else.

The problem appears for In-Process macros. We can use:

A. Very simple panels with “InputBox” and “MsgBox” functions

B. More complex panels creating an ActiveX using Visual Basic.

Student Notes:

Input box

InputBox function allows the user to enter a numeric or an alphanumeric value in a simple panel.

Limitations:

- Only one value at a time.
- You need as many InputBox as parameters you want to get.
- You have to force the order of acquisition and to manage the Escape button in your macro.

```
Language="VBSCRIPT"
Sub CATMain()
' Ask for a name and rename the curent Window
resu = InputBox("Enter the new name of the active window", "Acquisition Using InputBox", "MyWindow")
if resu <> "" then
  On Error Resume Next
  Set MyWin = CATIA.ActiveWindow
  If MyWin Is Nothing Then
    msgbox "No Active Window"
  else
    MyWin.Caption=resu
  End If
End If
End Sub
```

Student Notes:

Panels in an ActiveX component (Windows Only)

An other solution is to create panels in an ActiveX component.

An ActiveX can be called from a CATIA Macro.

This ActiveX can display panels.

The Macro calls a method which displays the panel and waits for the end of the acquisition. The method returns a status and the parameter values.

a) The ActiveX can get the CATIA session and call CATIA objects.
   We can, by this way, display or compute intermediate results or we can reuse a selection for several actions.
b) Several panels of your application(s) can be gathered in one ActiveX component
c) For a large user application, an out-process model is recommended

Student Notes:

Miscellaneous Information (1/4)

About Numbers, Literals, and Units

Numerical values stored in internally system units. Nevertheless, the user interface can be set to display and get values from the end user according to another unit system which better matches its needs or habits.

All the objects that inherit of ‘Dimension’ such as Length, are called “Literal”.

On theses objects, you can use:

- ValueAsString to read a system value within the user system coordinates.
- ValuateFromString allows the value stored in your object to be valuated using a user unit.

```
Call MyFaceFillet.Radius.ValuateFromString(“5.08mm”)
Call MyHole.Diameter.ValuateFromString(“2in”)
```

Student Notes:

Miscellaneous Information (2/4)

Resizing CATIA Window

```
CATIA.Width  = 500
CATIA.Height = 400
CATIA.Left = 150
CATIA.Top  = 50
```

☞ Dimensions in pixels.

☞ The position of a Window is determined by the position of its top left corner.

Opening and printing a Document on the Default Printer

```
Dim Doc as Document
Set Doc = CATIA.Documents.Open(“E:\Parts\PartToPrint.CATPart”)
CATIA.ActiveWindow.PrintOut                ☞ Print the Document
CATIA.ActiveDocument.Close                 ☞ Close the Document
```

Student Notes:

Miscellaneous Information (3/4)

Opening and Converting a Document in VRML

```
    Dim Doc as Document
    Set Doc = CATIA.Documents.Open(“E:\Parts\PartToConvert.CATPart”)
  Call Doc.ExportData(“E:\DocName", "wrl")
```

Scripting Knowledgeware Features

It is possible to create:

Parameters

Rules

Check

Design Tables and Select Configurations

Student Notes:

Miscellaneous Information (4/4)

Measures

The measures are provided by the "Analyze" object of a Product.
We have to create a product

How to measure the Centre of Gravity

Analyze.GetGravityCenter() method get the coordinates of the gravity centre.

How to measure the Mass

Analyze.Mass return the Mass of the Product

Student Notes:

Obfuscating and Converting Macros

You will learn to hide the source code of the macros and convert them into an encrypted format.

Student Notes:

Obfuscate VBScript Macros (1/4)

CATIA V5 includes an infrastructure to obfuscate scripts based on the MS Script Encoder.
The main characteristic of this component are as follows:

- It lets V5 scripts interoperate with other scripts which have chosen the same obfuscation scheme.
- It is developed and marketed by Microsoft, so it is widely available and well known.
- The encoding algorithm is quite weak though. However, Microsoft says that the encoder is not as much a technical protection as a legal protection.

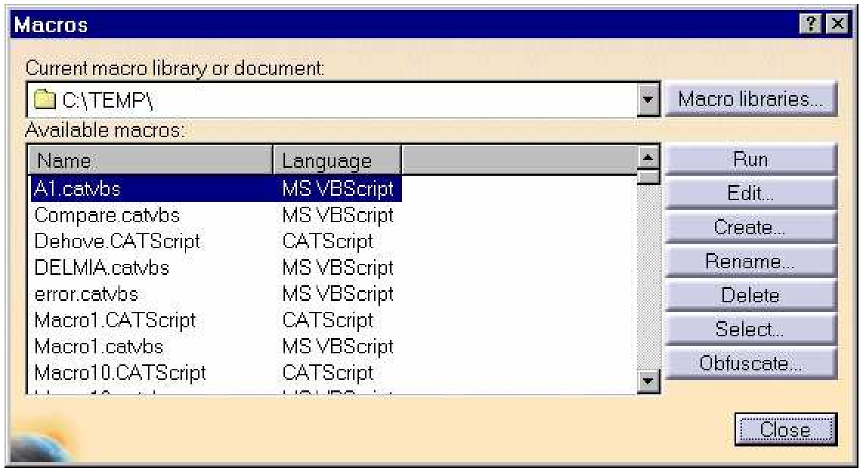
The Scripts can be encoded on Windows only. However, encoded scripts can be replayed on the UNIX and Windows.
Only VBScript scripts can be encoded. For other languages, there are other schemes Available (project passwords for VBA, ...)

For the obfuscation to be available, the MS Script encode must have been installed.

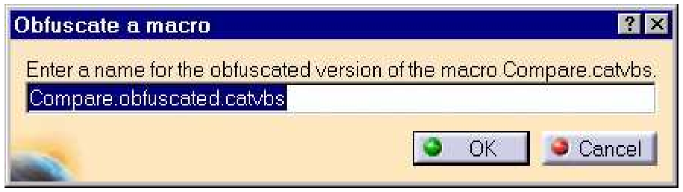
Student Notes:

Obfuscate VBScript Macros (2/4)

- **To obfuscate a Macro:**
  - **Select the Tools->Macros->Macros menu item.**
  - **In the macro panel, select the macro you want to obfuscate (for instance, Compare.catvbs) and click the "obfuscate..." button.**



Name	Language
A1.catvbs	MS VBScript
Compare.catvbs	MS VBScript
Dehove.CATScript	CATScript
DELMIA.catvbs	MS VBScript
error.catvbs	MS VBScript
Macro1.CATScript	CATScript
Macro1.catvbs	MS VBScript
Macro10.CATScript	CATScript



Enter a name for the obfuscated version of the macro Compare.catvbs.

Compare.obfuscated.catvbs

  - **In the obfuscate window, click the "Ok" button.**
  - **A new "Compare.obfuscated.catvbs" macro is created.**
  - **This macro can be replayed on Windows or UNIX without the MS Script Encoder installed.**

Student Notes:

Obfuscate VBScript Macros (3/4)

Here is the text for the "Compare.catvbs" macro before obfuscation:

```
Sub CATMain()
Set doc1 = CATIA.Documents.Item(1)
Set doc2 = CATIA.Documents.Item(1)
Set doc3 = CATIA.Documents.Item(2)
If doc1 Is doc2 Then
        MsgBox "doc1 == doc2"
End If
If doc1 Is doc3 Then
        MsgBox "Gasp"
End If
End Sub
```

Here is the text of the obfuscated macro:

```
#@~^CgEAAA==j!4~;b:Hlbxvb@#@&?nY,NG^8P',/b:qbc9Gm!:□xO/ (D+h`8b@#@&?nO,NW^+,'~ZzP&b
    GW1E:□xDdR&Yn:vFb@#@&?+D~NKm&,x~ZzK&b fG^!:nxDdR&Ynhv
    #@#@&@#@&(0,[Kmq,qkPNKmy~K4+U@#@&d\dTAWX~J9Wm8~x',NKm+J@#@&Ax[P&W@#@&
    @#@&(6PNG^8P(/,[Kmf,K4+x@#@&i\/TAG6,J!CkwJ@#@&2
    NP&W@#@&@#@&Ax[Pj;(@#@&n0IAAA==^#~@
```

Student Notes:

Obfuscate VBScript Macros (4/4)

Advanced Features

A special 'Start Encode** tag can be placed in the script to tell the obfuscator to only Partially encode a script. This can be useful to leave a copyright notice unencrypted. For instance:**

before obfuscation:

```
' Copyright 2004 MyCompany. All rights reserved
'**Start Encode**
Sub CATMain()
' My secret algorithm goes here
End Sub
```

Here is the text after obfuscation:

```
' Copyright 2004 MyCompany. All rights reserved
'**Start Encode**
      #@~^PQAAAA==@&@&?;(P;bKtlbU`*@&vPtX~d□mD□OPmVoK.rY4:,oG+d~4+.+@&3x9Pj;(@&txE
      AAA==^#~@
```

Student Notes:

Multi-Product Support (1/2)

DS is now a multi-product company. However, until V5 R9 all the products were registered under the same keys in the Windows registry as far as the scripting was concerned. This made it impossible to develop solutions which relied on several V5 products glued together by scripting code. From the V5R9 scripting infrastructure changes this.

The root object (an instance of the VB Application class) used to be always registered under the name CATIA, a confusing name for DELMIA or DMU users. Since V5R9, when recording a script, the root object name has the same name as the application. For instance, if you record a script which creates a part in DELMIA, you will get:

```
Sub CATMain()
      Set documents1 = DELMIA.Documents
      Set partDocument1 = documents1.Add("Part")
End Sub
```

Multi-Product Support (2/2)

For compatibility with previously recorded scripts, and interoperability of scripts recorded in other DS products, the names "CATIA", "DELMIA" and "DMU" all refer to the root object (an instance of the VB Application class) and can be used in place of each other.

Out-process: Each of the main V5 applications which use scripting (CATIA, DELMIA, ENOVIA) now have a separate entry in the windows registry. The command:

DMU /RegServer
Will create an DMU.Application entry in the registry.

In the same way,
DELMIA /RegServer
will create a DELMIA.Application in the registry.

To instantiate a DMU and a DELMIA application from a CATIA VBScript, one would write:

```
Dim DMUApp
Set DMUApp = CreateObject ("DMU.Application")
Dim DELMIAApp
Set DELMIAApp = CreateObject ("DELMIA.Application")
```

Student Notes:

Student Notes:

Registering CATIA and Type Libraries

A. When CATIA V5 is installed, it is registered in the "registry" of "Windows", so that it is possible to use this instruction:

```
Set CATIA= CreateObject("CATIA.Application")
```

If you have several versions of CATIA and If you want to change of version, you can unregister CATIA in a command window typing :

```
> cnext /UnregServer
```

To register an other version, you can type:

```
> cnext /RegServer
```

These commands must be typed in a command window and in the folder: ".\intel_a\code\bin" of the chosen CATIA version.

B. To register the corresponding CATIA Type Libraries, you must first run a macro. Then, you will be able to check in the "Project/References " menu of Visual Basic (or VBA) that the location of the *.tlb files has changed.

Student Notes:

Converting a Macro to a Visual Basic Program

At first you need to start CATIA from your VB program.

A. If you don't want to use the type definitions (Like Macros)

   a) Comment all the CATIA objects' types.2)

```
Dim myDocument 'as Document
```

B. If you want to use the type definitions (for completion)

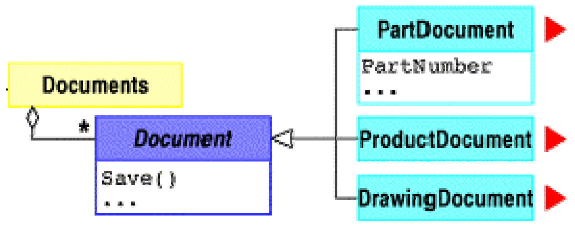
   a) Rename the variables to have better understanding names.

   b) Give the right type for objects if necessary:

```
Dim myDocument as PartDocument
Instead of
Dim myDocument as AnyObject
```

   a) If we have inheritance, we must create an intermediate variable to define the correct type for the "Inheritance branch" we choose.

```
Dim myPartDocument as PartDocument
Set myPartDocument = CATIA.ActiveDocument
Set myPart = myPartDocument.Part
Instead of
Set myPart = CATIA.ActiveDocument.Part
```



Because Part is a property of a PartDocument but not of a Document.

Student Notes:

Converting a Visual Basic programm to a CATIA Macro

A. **Be Careful**
From a Visual Basic program, you cannot reuse the graphical Forms.
The user interface is reduced to MsgBox and InputBox panels. (ActiveX on Windows)

B. **Don't care about the objects' type definitions**
They are not used in macros and they are automatically commented at run time.

C. **Review some Syntaxes**
   a) Don't use # for constants. Replace 10# by 10.0
   b) Prefer the syntax with Call keyword for subroutines which use arguments:

```
Call mySubroutine(Arg1,Arg2)
```

   c) Use simply Next in stead of Next I for loops
   d) Don't use the type definitions in the arguments of your own subroutines.
   e) Don't use For each loop syntax on Basic Script (UNIX).

You can easily develop your program using Microsoft Visual Basic, then convert it into a CATIA macro when it is finished.

Student Notes:

Macros on UNIX / Windows

A. Windows Macros (VBScript) don't use the Object Types
The types are automatically commented at run time.

B. Visual Basic programs can use types
We can define the real type or an abstract type.
It depends on the kind of properties we want to access.
To access to myDocument.Part, we must define myDocument as PartDocument.

For example:

```
Dim myDocument as Document
Or
Dim myDocument as PartDocument
```

C. UNIX Macros (Basic Script) use types
These are not always the same types as those used in Visual Basic.
On UNIX, we must define the object type returned by the function or property used to assign the variable.
Example: "CATIA.ActiveDocument" returns a "Document". So we must type:

```
Dim myDocument as Document.
Set myDocument = CATIA.ActiveDocument
```

Student Notes:

Calling an other Automation Server

You will learn to call other automation servers like 'MS – Word, Excel, PowerPoint' etc using your automation scripts.

Student Notes:

Calling WSH from VBScript

In VBScript macros, you are not able to use file system functions such as open, close and print #n. To access file system functions on Windows, you can use WSH (Windows Scripting Host).

The Attached Macro “read.CATScript”, read coordinates in the file “E:\Test.DAT” and create 3D points.
The service is created with the follwing code:
Set o = CreateObject("Scripting.FileSystemObject")
Then we can open a file using:
Set fil = o.GetFile("e:\test.dat")
The file is read using a TextStream:
Set ts = fil.OpenAsTextStream(1)' (For Reading)
We can read each lines with:
ts.ReadLine
And close the stream with:
ts.Close

CATIA provides an object called FileSystem to be able to do the same on both UNIX and Windows Platforms.

Student Notes:

Calling Word

You can call Word from VBScript as well as from VBA or Visual Basic.

The best way to develop Word functions is to try it in Word itself. You will be able to use the Visual Basic Editor with its help (“F1”), completion, Object Browser and debugger.

You can also record macros to see what functions you have to use.

Here is a sample that opens a new document and writes “Hello Word!”.

```
Dim Wr As Word.Application
On Error Resume Next
Set Wr = GetObject(, "Word.Application")
If Err <> o Then
  Set Wr = CreateObject("Word.Application")
  Wr.Visible = True
End If
Set d = Wr.Documents.Add (DocumentType:=wdNewBlankDocument)
d.Range.Text = "Hello Word!"
```

Student Notes:

Calling Excel

You can call Excel from VBScript as well as from VBA or Visual Basic.

The best way to develop Excel functions is to try it inside Excel. You will be able to use the Visual Basic Editor with its help (“F1”), completion, Object Browser and debugger.

You can also record macros to see which functions you have to use.

Here is a sample that opens a new sheet and writes “Hello” in the cell “B2”.

```
Dim xl As Excel.Application
On Error Resume Next
Set xl = GetObject(, "Excel.Application")
If Err <> o Then
  Set xl = CreateObject("Excel.Application")
  xl.Visible = True
End If
xl.Workbooks.Add                 ‘ Add a new WorkBook
Range("B2").Select               ‘ Select the cell “B2”
ActiveCell.FormulaR1C1 = "Hello"
```

Student Notes:

Drawing a Chart in EXCEL

- To draw a chart in Excel, you have to create a chart using:
  Charts.Add
- Fill cells with values and associate them to the chart using:
  ActiveChart.SetSourceData
- Then position the chart in one sheet using:
  ActiveChart.Location
  Then, you can modify the properties of the chart.

See the example .\Student\Data\Lesson4\SampleTestWordExcel\Project1.vbp for more details.

Student Notes:

To Sum Up

In this course you have seen:

- **In-Process and Out-Process Macros.**
- **Recording / Creating of macros using various scripting languages like VB / VBA etc.**
- **Scripting various features like Sketches, Part Design Features, Surfaces, Drawings etc.**
- **Calling other automation servers.**