CATIA V5 Training

Exercises

V4 Integration

Version 5 Release 19
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EDU_CAT_EN_V4I_FX_V5R19

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Recap Exercise

Exercise 1: Presentation

20 min

In this step, you will open a CATIA V4 model file, then check the geometry for the unresolved feature, and save the component into a CATPart.

Design Intent

1. **Open a V4 Model in CATIA V5**

2. **Check the geometry**

3. **Check the Specification**

4. **Translate the V4 Solid geometry in native CATIA V5**

5. **Save your new CATPart**

Design Process

1 Open the V4 Model in CATIA V5
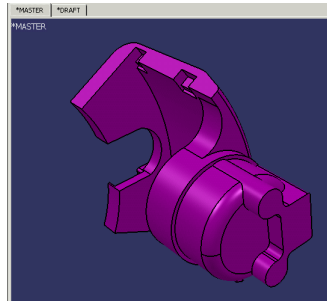
2 Check the Geometry
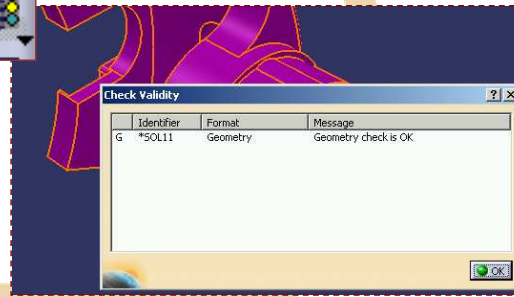
3 Check Specification
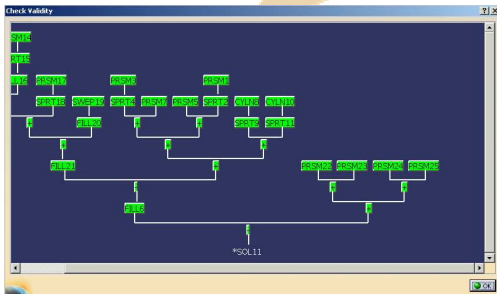
4 Copy / Paste in V5 and Save
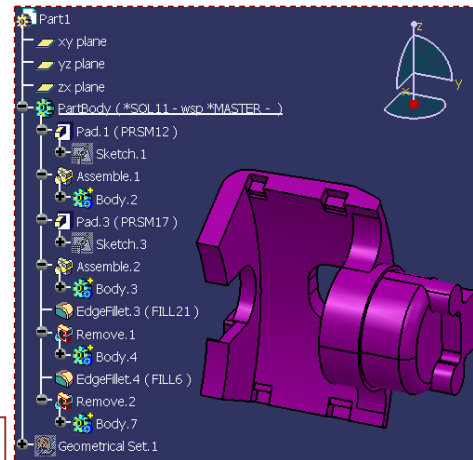
Exercise 1

Step 1: Open a V4 Model in CATIA V5
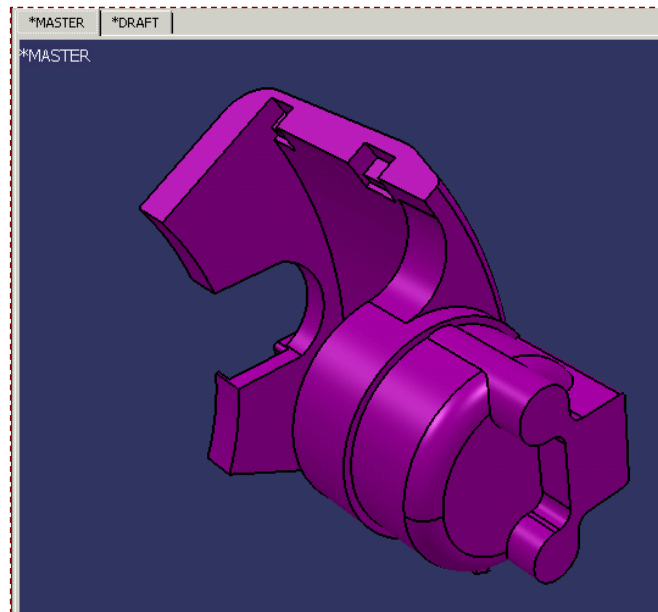
🕐 *5 min*

In this step you will open a V4 Model in CATIA V5.

Do It Yourself

1

Open a V4 model

2
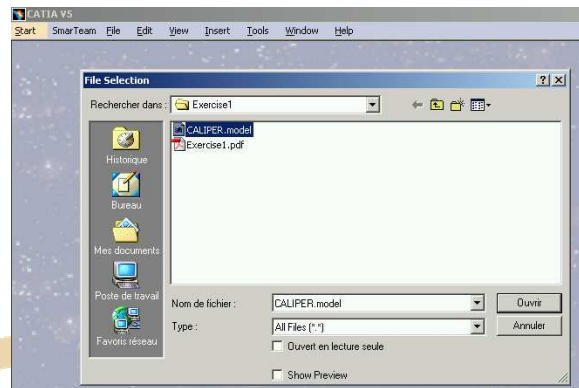
**CATIA V4
Integration**
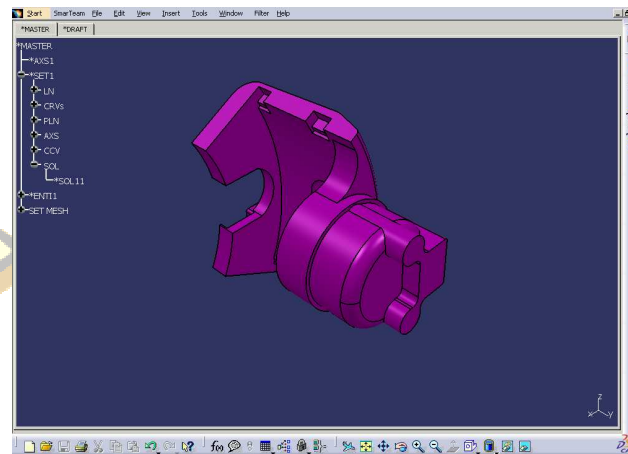
1. **Select the File > Open command and select the file "CALIPER.model".**

2. **You will automatically be in the "CATIA V4 Integration" workbench.**

Exercise 1

Step 2: Check Geometry
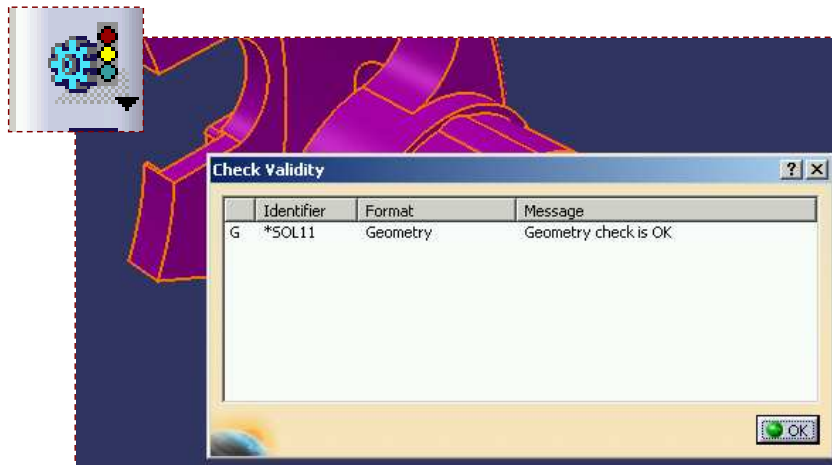
5 min

In this step you will Check the geometry of the V4 Solid.

Do It Yourself

1

Expand the tree and select a solid

```
*MASTER   *DRAFT
*MASTER
 ─*AXS1
 ─*SET1
     ─ LN
     ─ CRVs
     ─ PLN
     ─ AXS
     ─ CCV
     ─ SOL
         *SOL11
 ─*ENTI1
 ─SET MESH
```

2

Click the Geometry check button

3

The Check Validity dialog box is displayed

Check Validity

	Identifier	Format	Message
G	*SOL11	Geometry	Geometry check is OK

OK
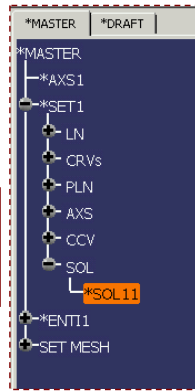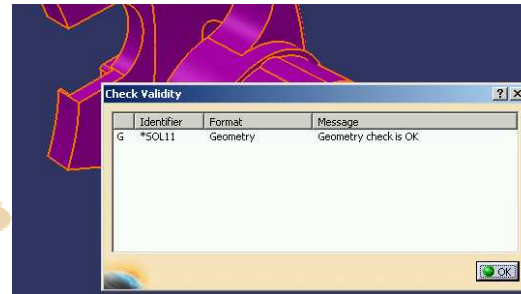
1. **Double-click on *MASTER to expand the specification tree.**
2. **As you can see, to activate the "geometry check" button, you have to select an element belonging to the specification tree : select *SOL11**
3. **Click the "Geometry check" button, the "Check Validity" dialog box is displayed.**

Exercise 1

Step 3: Check Specification

5 min

In this step you will Check the specification of the V4 Solid.

Do It Yourself

1
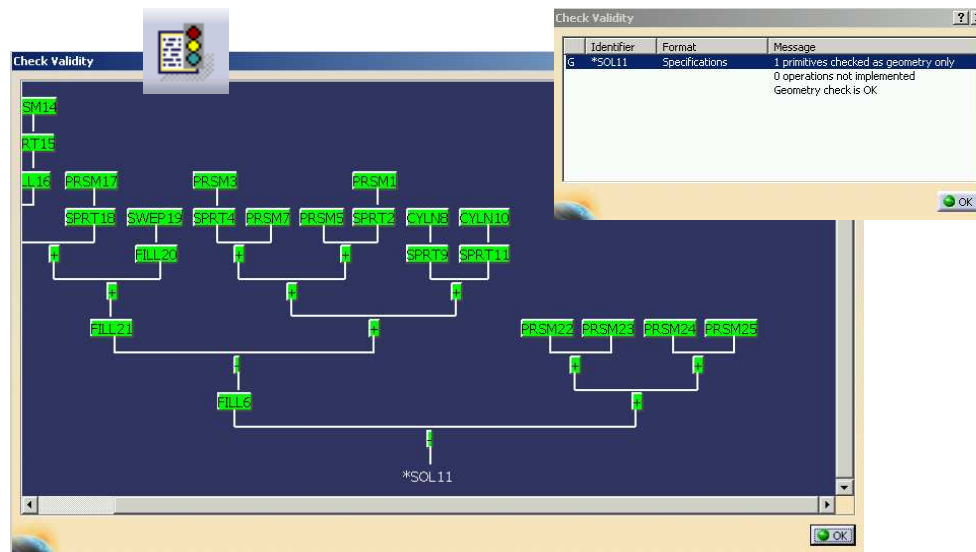
Expand the tree and select the *SOL11

```
*MASTER    *DRAFT
*MASTER
 ├─*AXS1
 ├─*SET1
 │  ├─ LN
 │  ├─ CRVs
 │  ├─ PLN
 │  ├─ AXS
 │  ├─ CCV
 │  └─ SOL
 │      └─*SOL11
 ├─*ENTI1
 └─ SET MESH
```

2
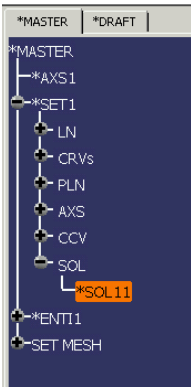
Click the specification check button

3

Double-click the line about *SOL11

	Identifier	Format	Message
G	*SOL11	Specifications	1 primitives checked as geometry only
			0 operations not implemented
			Geometry check is OK

Check Validity

OK
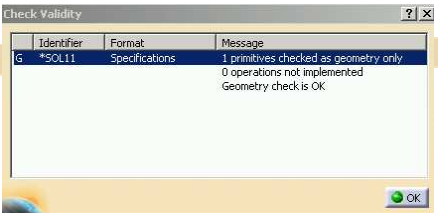
4

CSG tree of the solid

Check Validity

*SOL11

OK
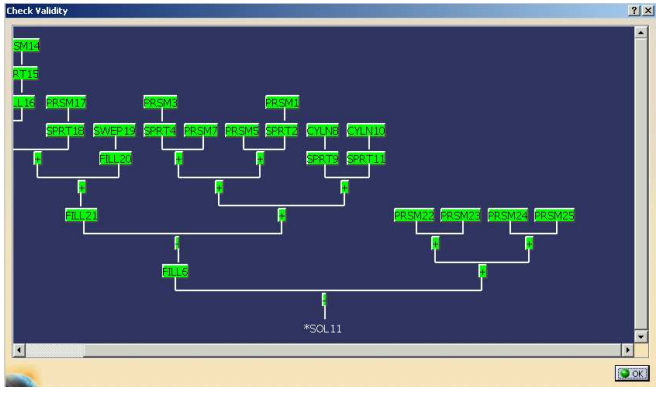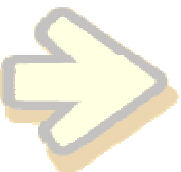
1. **Select the *SOL11 in the tree**
2. **Click the specification check button, the check validity dialog box appeared**
3. **At this step, double-click the line about the *SOL11 to display the CSG tree of the solid as it was made in CATIA V4**

Exercise 1

Step 4: Copy/Paste in V5 and Save

5 min

In this step you will copy and paste your model in CATIA V5.
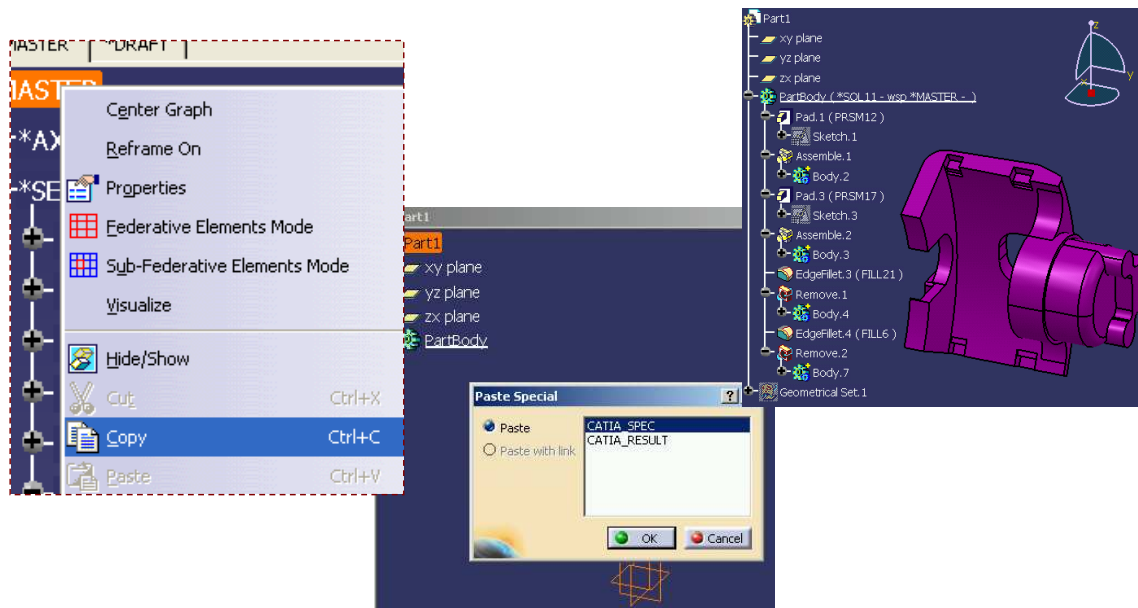
Do It Yourself
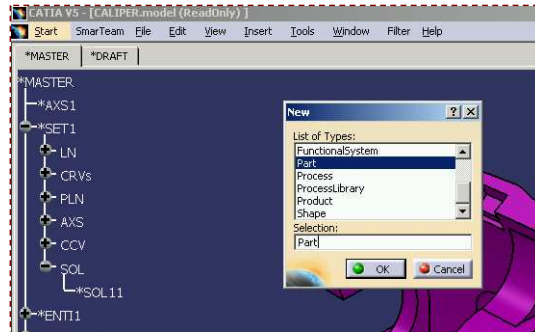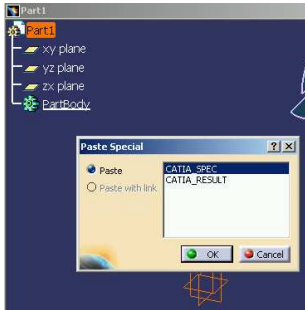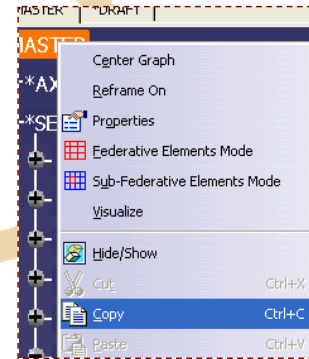
(1) Create a new CATIA V5 document

(2) Copy the model

(3) Paste and Save your CATIA V5 document
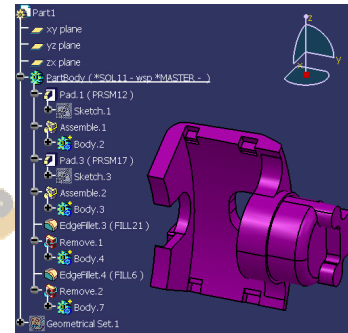
1. **Select the File -> New and choose "Part" in the dialog box (You can also do "Ctrl+N").**
2. **Activate the "Caliper.model" and with MB3 do a copy (the *MASTER is copied but you can also copy the entities you need, for example *SOL11).**
3. **Activate the new Part and do a "paste special" and choose "CATIA_SPEC" : all features have been transferred into the CATPart and need an update. You can save your CATIA V5 document.**

Recap Exercise

Exercise 2: Presentation

15 min

In this exercise, you will open CATIA V4 in CATIA V5, translation of a *SOL from the V4 model into a CATIA V5 CATPart.
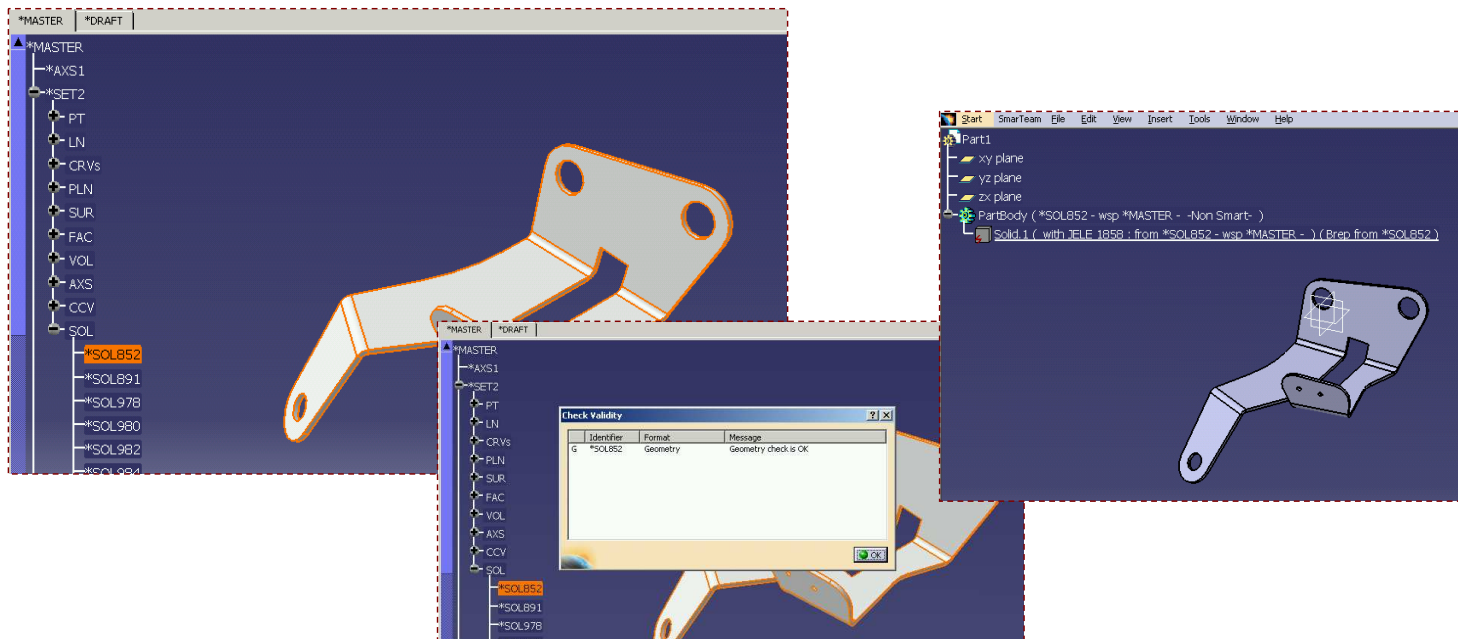
Design Intent
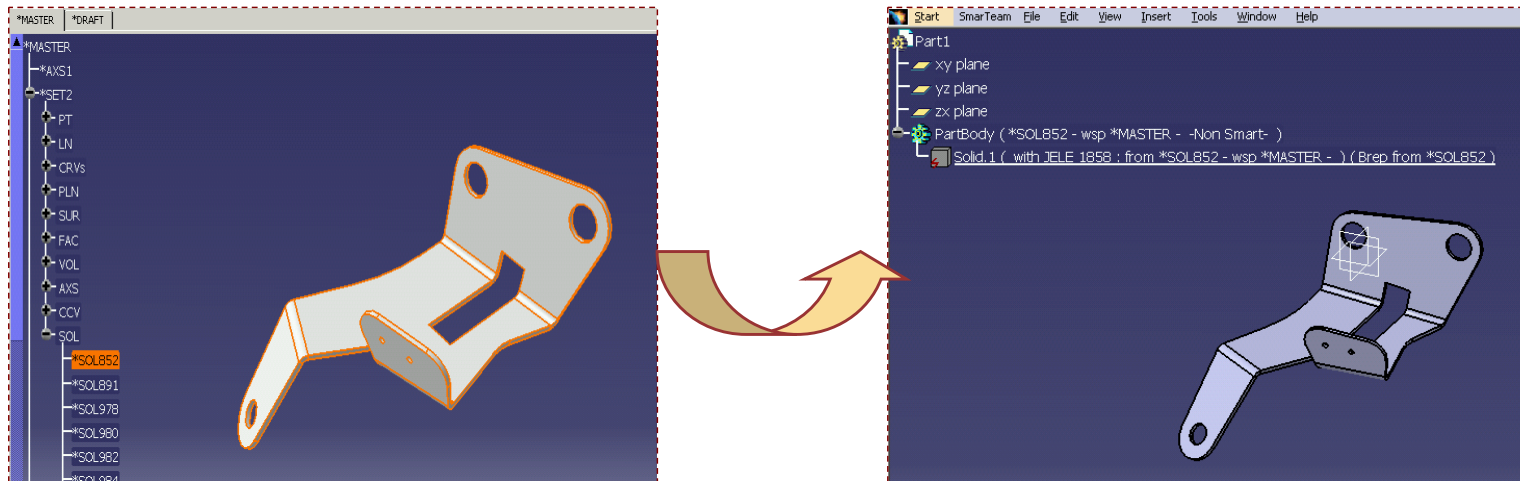
1. **Open a V4 Model in CATIA V5**

2. **Check the geometry**

3. **Translate the V4 Solid geometry in native CATIA V5 as Result**

4. **Save your new CATPart**

Design Process

1

Open the V4 Model in CATIA V5

2
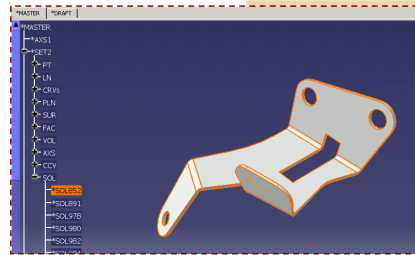
Check the Geometry

3
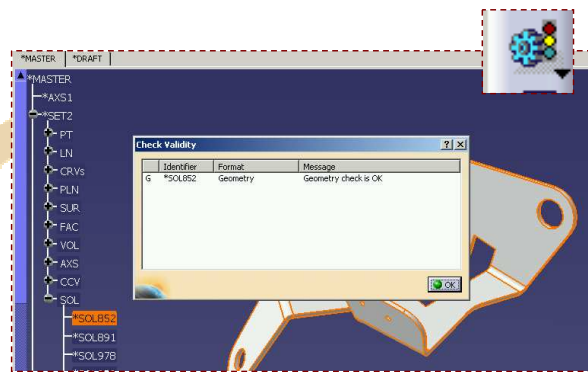
Copy / Paste in V5 and Save
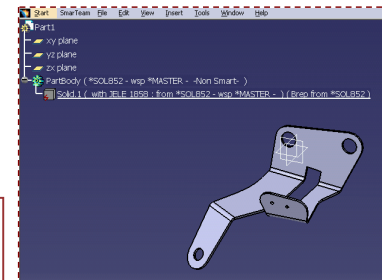
Exercise 2

Step 1: Open a V4 Model in CATIA V5
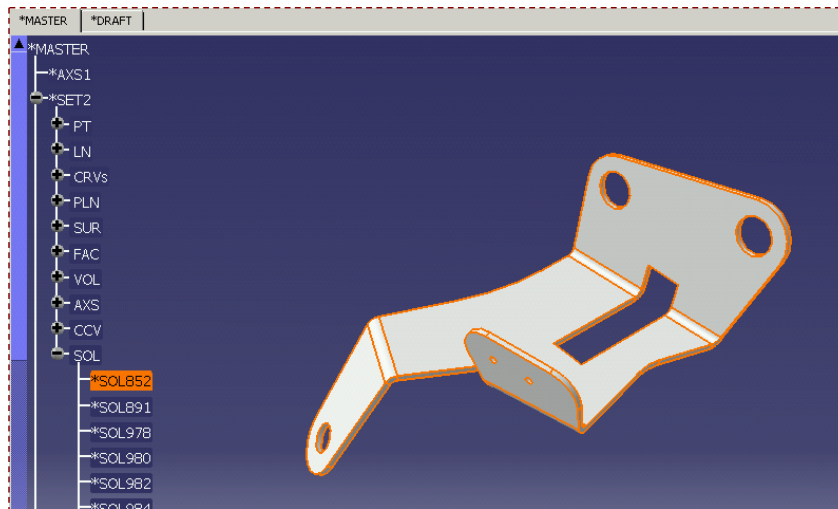
5 min

In this step you will open a V4 Model in CATIA V5.

Do It Yourself

1

Open a V4 model

2
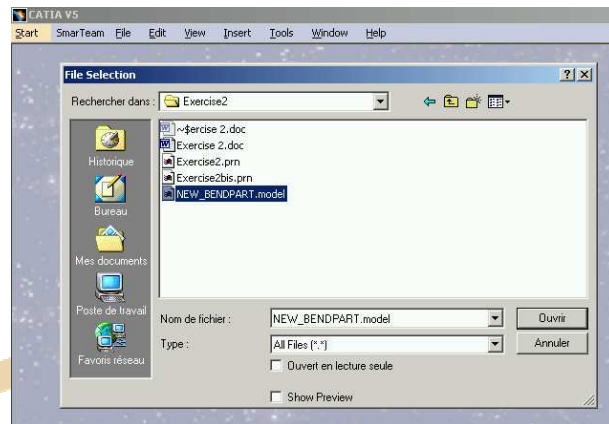
CATIA V4 Integration
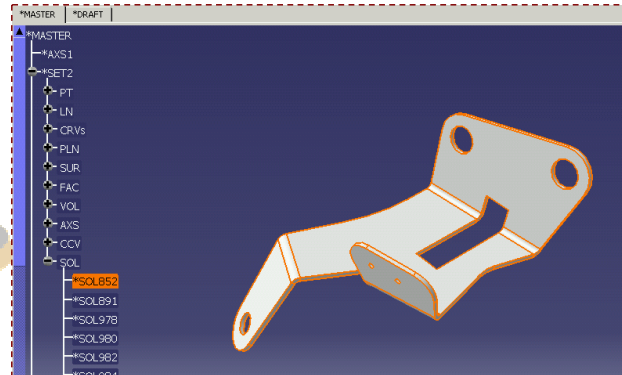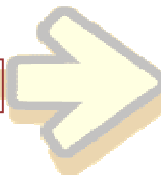
1. **Select the File > Open command and select the file "NEW_BENDPART.model.**

2. **You will automatically be in the "CATIA V4 Integration" workbench.**

Exercise 2

Step 2: Check Geometry

🕐 *5 min*

In this step you will Check the geometry of the V4 Solid.
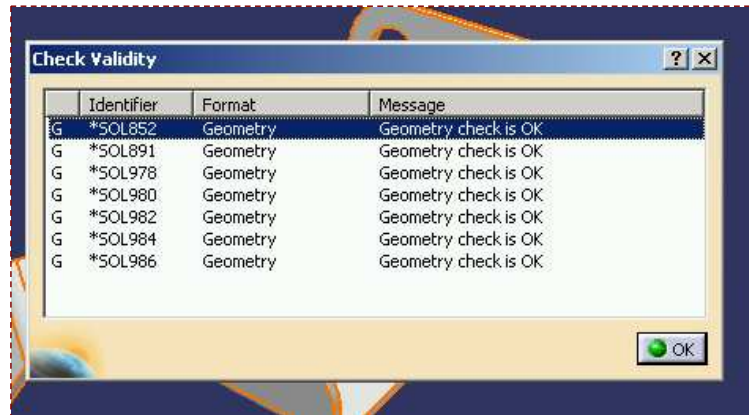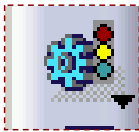
Do It Yourself

1 Expand the tree and select a solid

2 Click the Geometry check button
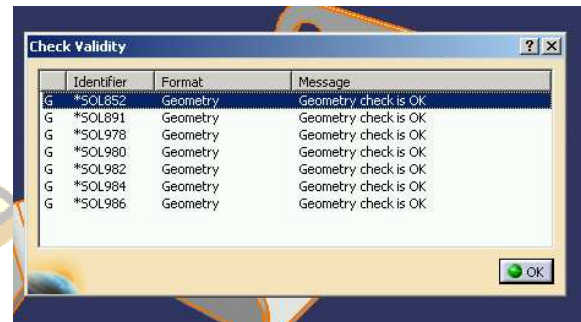
3 The Check Validity dialog box is displayed

Check Validity

	Identifier	Format	Message
G	*SOL852	Geometry	Geometry check is OK
G	*SOL891	Geometry	Geometry check is OK
G	*SOL978	Geometry	Geometry check is OK
G	*SOL980	Geometry	Geometry check is OK
G	*SOL982	Geometry	Geometry check is OK
G	*SOL984	Geometry	Geometry check is OK
G	*SOL986	Geometry	Geometry check is OK

OK

1. **Double-click on *MASTER to expand the specification tree.**

2. **As you can see, to activate the "geometry check" button, you have to select an element belonging to the specification tree: select SOL.**

3. **Click the "Geometry check" button, the "Check Validity" dialog box is displayed.**

Exercise 2

Step 3: Copy / Paste in V5 and Save

5 min

In this step, you will copy and paste (AS_RES) your model in CATIA V5 in order to reuse it later when you will learn the migration of CATIA V5 applications.
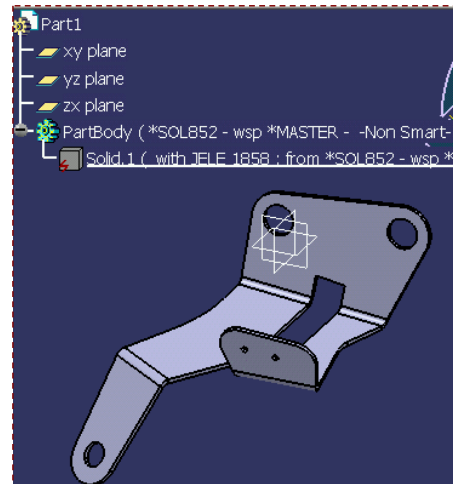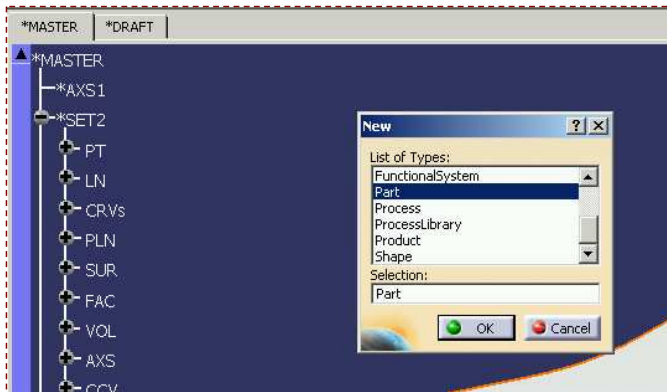
Do It Yourself

1

Create a new CATIA V5 document

2

Copy the model

3

Paste and Save your CATIA V5 document

1. **Select the File -> New and choose "Part" in the dialog box (You can also do "Ctrl+N").**

2. **Activate the "NEW_BENDPART.model", select *SOL852 and with MB3, do a copy.**

3. **Activate the new Part and do a "paste special" and choose "CATIA_RESULT" (you can also do a simple "paste"): all features have been transferred into the CATPart and need an update. You can save your CATIA V5 document.**

Recap Exercise

Exercise 3: Presentation

20 min

In this exercise, you will learn how to manipulate and use the V4 and V5 data in a V5 CATProduct.

Design Intent

1. **Open a new CATProduct in CATIA V5 and insert the existing component**

2. **Replace one of these components with another**

3. **Manipulate and apply constraints**

Design Process

1

Create a new CATProduct and insert components
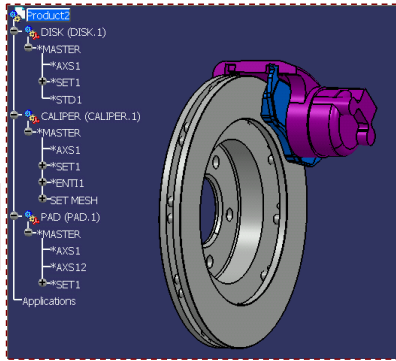
2

Replace a component
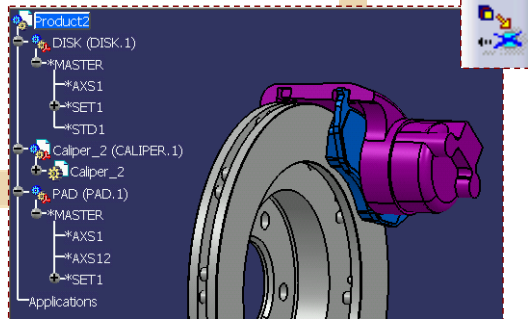
3

Manipulate and constraint the objects
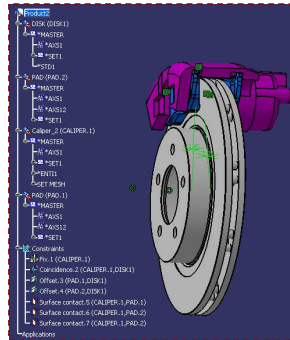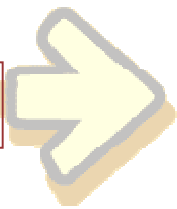
Exercise 3

Step 1: Create a CATProduct and Insert Components

5 min

In this step, you will create a new V5 CATProduct and insert the existing components.

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Do It Yourself

1 Create a new CATProduct

2 Insert the components

1. **Select the File > New and choose "Product" in the dialog box (You can also do "Ctrl+N").**
    Rename the Part number: "DISK_BRAKE_MECHANISM".

2. **In the Assembly Design workbench, insert the existing components: select**
    CALIPER.model, DISK.model, and PAD.model.

Exercise 3

Step 2: Replace a V4 component by a V5 CATPart

🕐 *5 min*

In this step, you will replace the "CALIPER.model" by the "CALIPER2.CATPart".
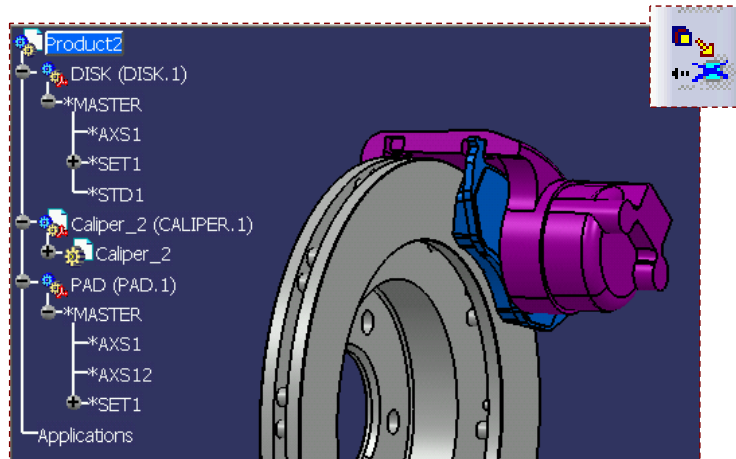
Do It Yourself

1

**Select the
CALIPER.model**

2

**Click the "Replace
component" Tool icon**

3   **Select in the dialog box the
CALIPER2.CATPart**

1. **Select in the product the "CALIPER.model".**

2. **Select the tool "replace component (you can also select with the contextual
    menu, "Components/Replace Component…").**

3. **Select in the dialog box the "CALIPER2.CATPart".**

Exercise 3

Step 3: Manipulate and Apply Constraints

10 min

In this step, you will duplicate one component and apply constraints in order to have a more precise position of the components.

Do It Yourself

1 Duplicate the "Pad" in the product

2 Apply constraints to position it

1. **Copy / Paste the "Pad" in the CATProduct to have two pads.**

2. **Move the second instance of the "Pad" with compass and rotate to get a mirror of the first. Fix the "CALIPER" with the "Fix Component" tool, put a coincidence constraint between the "CALIPER" and the "DISK", then put an offset constraint between the disk and the two pads. The value of the offset is 1.5mm. You will see that pads are clashing the caliper. The face contacts can be defined with the "Contact constraint" tool.**

Master Exercise

Welding Gun

3 hours

In this exercise you will:

- **See the integration of V4 through a V5 process**
- **Create a CATProduct containing V4 .models**
- **Create assembly constraints, transfer a V4 solid in a V5 CATPart**
- **Design a CATPart in context, generate a drawing**
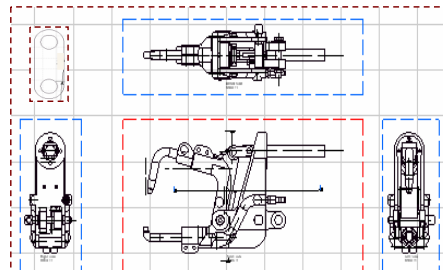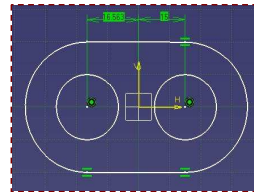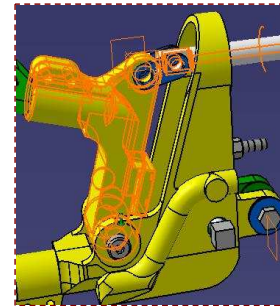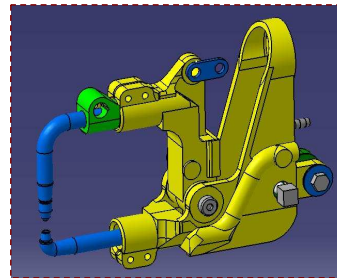
Design Intent
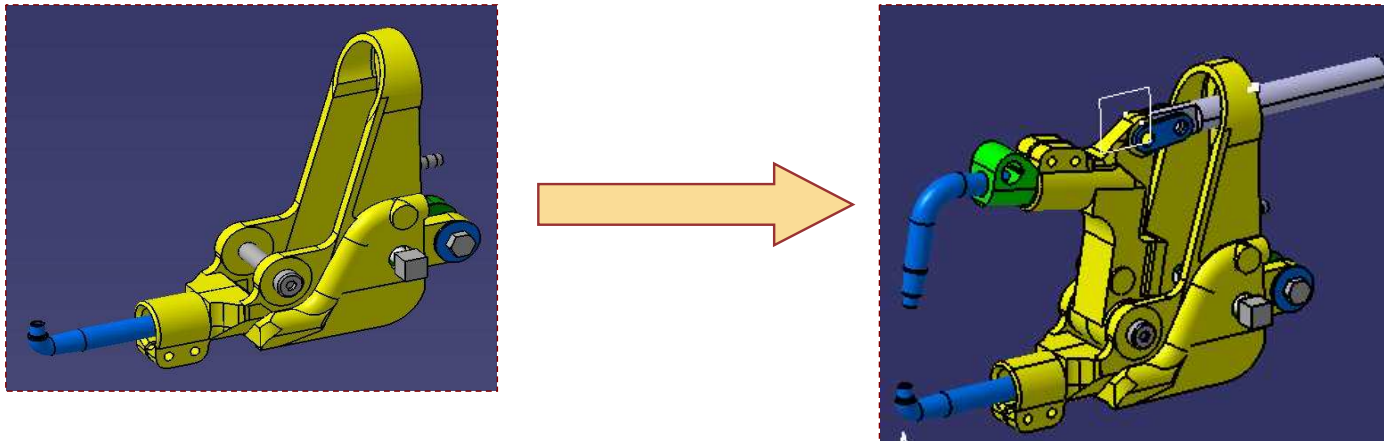
1. **Create a product from a V4 .session and .model**

2. **Creation of assembly constraints**

3. **Conversion of V4 data in a V5 part**

4. **Design a V5 part in context**

5. **Generate a drawing**

6. **DMU Space Analysis**

7. **Modification of a .model in V4 and see the result in V5**

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Design Process

1 Creation of a CATProduct with V4 data
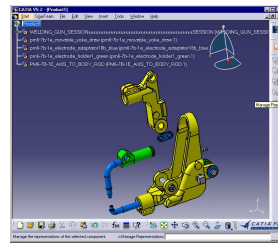
2 Assembly of V4 models

4 Design in context
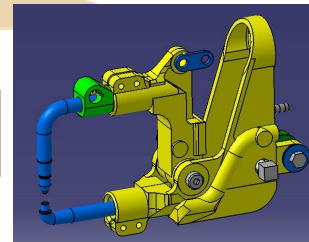
5 Generative Drafting
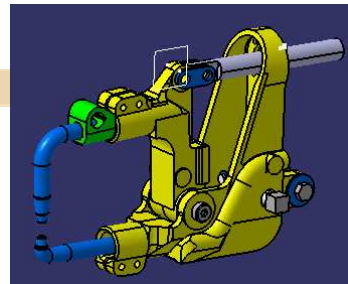
3 Transfer of V4 data in a V5 part
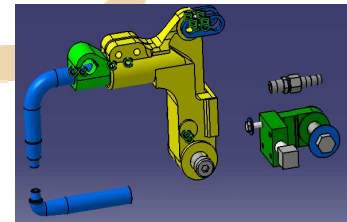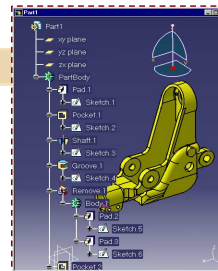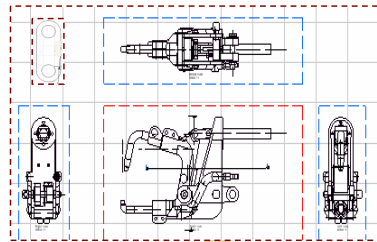
6 DMU Space Analysis
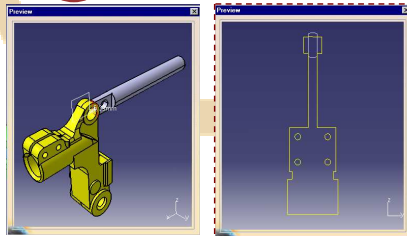
7 Modification of V4 model and impact in V5 document
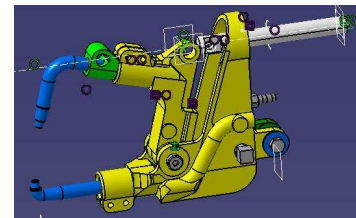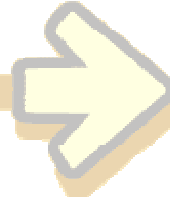
Master Exercise

Step 1: Creating a CATProduct with V4 data

10 min

In this step, you will open / insert V4 data into a product structure.
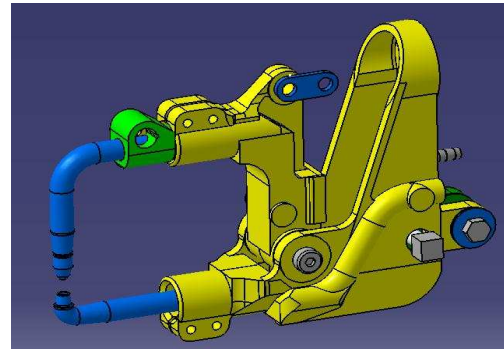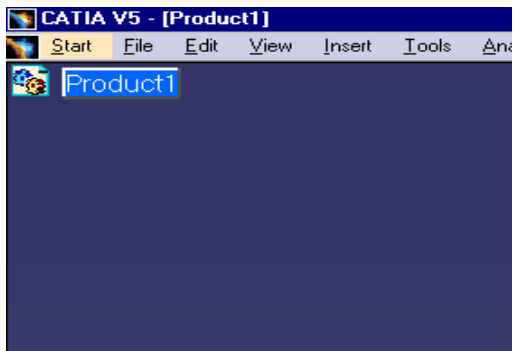
Do It Yourself

1

Go under the right path and select the two directories: Welding Gun & Session models

| General | Automation | Document | Licensing | Performances |

Document Environments

Document Environment	State	
Folder	Current	Current
DLName	Not allowed	Allowed
		Not allowed
		Configure...

Linked Document Localization

Document Location	Active	
Other folders		Activate
Folder of the link	Yes	Deactivate
Relative folder	Yes	Up
Folder of the pointing document	Yes	Down

or File + Open

Welding_gun±±±±±

session(*.session)

2

Open the session

4

Insert the 4 indicated models

| Mechanical | Product | Graphic |

Product

Part Number WELDING_GUN_SESSION±±±±±±±

3

Modify the Welding product name

WELDING_GUN

PM6-7B-1E_ELEC

model(*.model)

- PM6-7B-1E_AXIS_TO_BODY_ROD.model
- pm6-7b-1e_electrode_adaptator18b_blue.model
- pm6-7b-1e_electrode_holder1_green.model
- pm6-7b-1e_movable_yoke_draw.model

5

Copy / Paste the blue part in the product

6

Move both the indicated parts

Product1

Copy	Ctrl+C
Paste	Ctrl+V
Paste Special...	

Creating a CATProduct with V4 data:

1. **Customization: define correctly your search order. Tools / Options / General / Document.**
2. **Open the session.**
3. **Rename the Product component through the properties item.**
4. **In your product, insert the existing components by selecting the four model files.**
5. **Copy / Paste the Axis_to_Body_Rod.model in the product.**
6. **Move the overlapped Axis_to_Body_Rod and the Movable_Yoke_Draw with the compass.**

Master Exercise

Step 2: Assembly of V4 data in a V5 CATProduct

15 min

In this step, you will create assembly constraints between V4 data.
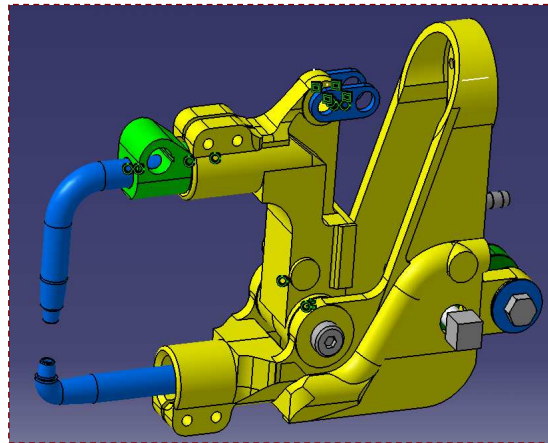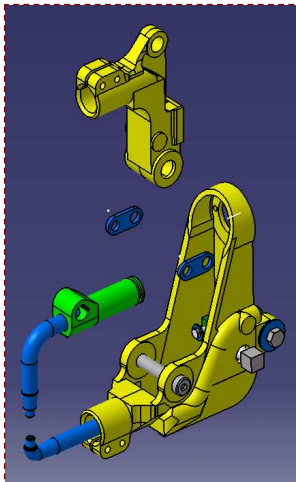
Do It Yourself

1

Fix the Session

3

Fix these three parts together

3

4,5

2

Create coincidence constraints

Fix Together

Name: FixTogether.1

Components

pm6-7b-1e_movable_yoke_draw (pm6-7b-1e_movable_yoke_dr...
pm6-7b-1e_electrode_holder1_green (pm6-7b-1e_electrode_hol...
pm6-7b-1e_electrode_adaptator18b_blue (pm6-7b-1e_electrode...

Constraints
Fix.2 (W...
Coincide
Coincide
Coincide
Coincide
Coincide
Surface
Surface
FixToge

5

Create Offset constraint

2   1

4

Create Contact constraints

2'   1'

1.   **Fix the session by creating a Fix constraint on Gun Body Draw.**
2.   **Coincidence constraints between: 1 - Movable_Yoke_Draw § Gun_Body_Draw**
                                       2 - Electrode_Adaptator18 § Electrode_Holder1
                                       3 - Electrode_Holder1 § Movable_Yoke
                                       4 - Axis_to_Body_Rod § Movable_Yoke
                                       5 - Idem with the copy of Axis_to_Body_Rod
3.   **Fix with the Ctrl Key the three V4 parts as shown on the picture.**
4.   **Contact Constraint between Axis_to_Body_Rod & Movable_Yoke, idem with its copy.**
5.   **Offset constraint between Gun_Body_Draw and Yoke body.**

Master Exercise

Step 3: Copy / Paste a V4 solid in a V5 CATPart

20 min

In this step, you will transfer a V4 solid in a V5 CATPart and replace the original model by the CATPart in the CATProduct.

Do It Yourself

pm6-7b-1e_gun_body_draw

1 Delete the Gun Body Draw

Delete

2 **Insert a new part**

WELDING_GUN

pm6-7b-1e_body   New Part...

Part1 (Part1.1)
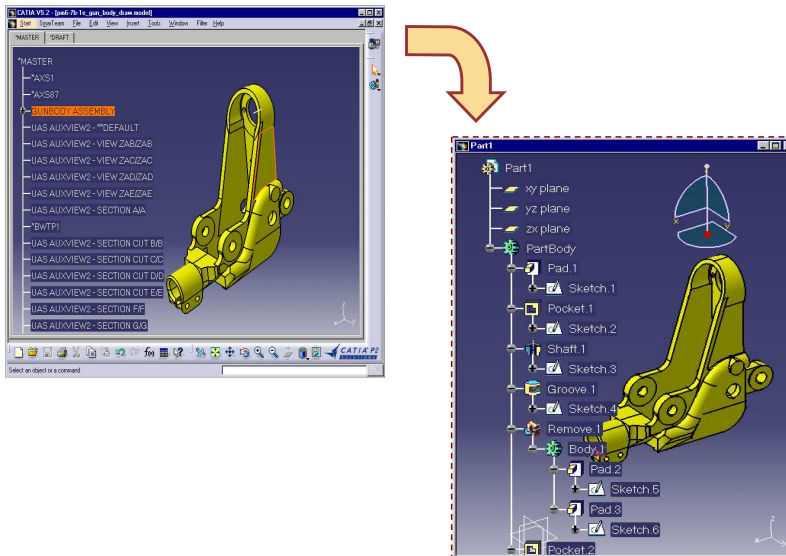
Part1

3 Open the Gun Body Draw with File / Open. Check the solid

```
*MASTER   *DRAFT
*MASTER
 *AXS1
 *AXSB7
 GUNBODY ASSEMBLY
  LN
  CRVs
  SOL
   *SOL5B1
 UAS AUXVIEW2 - **DEFAULT
 UAS AUXVIEW2 - VIEW ZAB/ZAB
 UAS AUXVIEW2 - VIEW ZAC/ZAC
 UAS AUXVIEW2 - VIEW ZAD/ZAD
 UAS AUXVIEW2 - VIEW ZAE/ZAE
 UAS AUXVIEW2 - SECTION A/A
```

Paste Special...

Part1 (Part1.1)

Part1

4 Paste it as SPEC in the new Part

Paste Special

● Paste   CATIA_RESULT
○ Paste with link   CATIA_SPEC

Warnings

Object	Message
Pocket.15 (PRSM50)	The operation performed on the feature is unnecessary.

Close

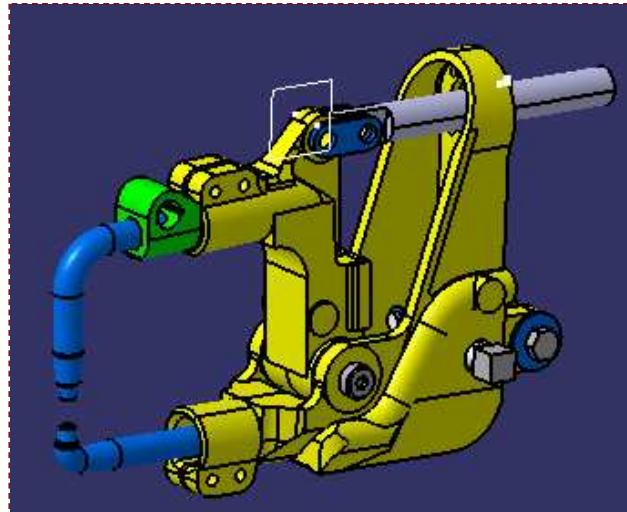1. **Delete the Gun Body Draw model (children constraints can be deleted at the same time).**
2. **Insert a new part. Access the Part Design workbench.**
3. **In Sessions_models directory, open the gun_body_draw model and check the solid.**
4. **Paste it As_Spec under Part1 and Update it.**
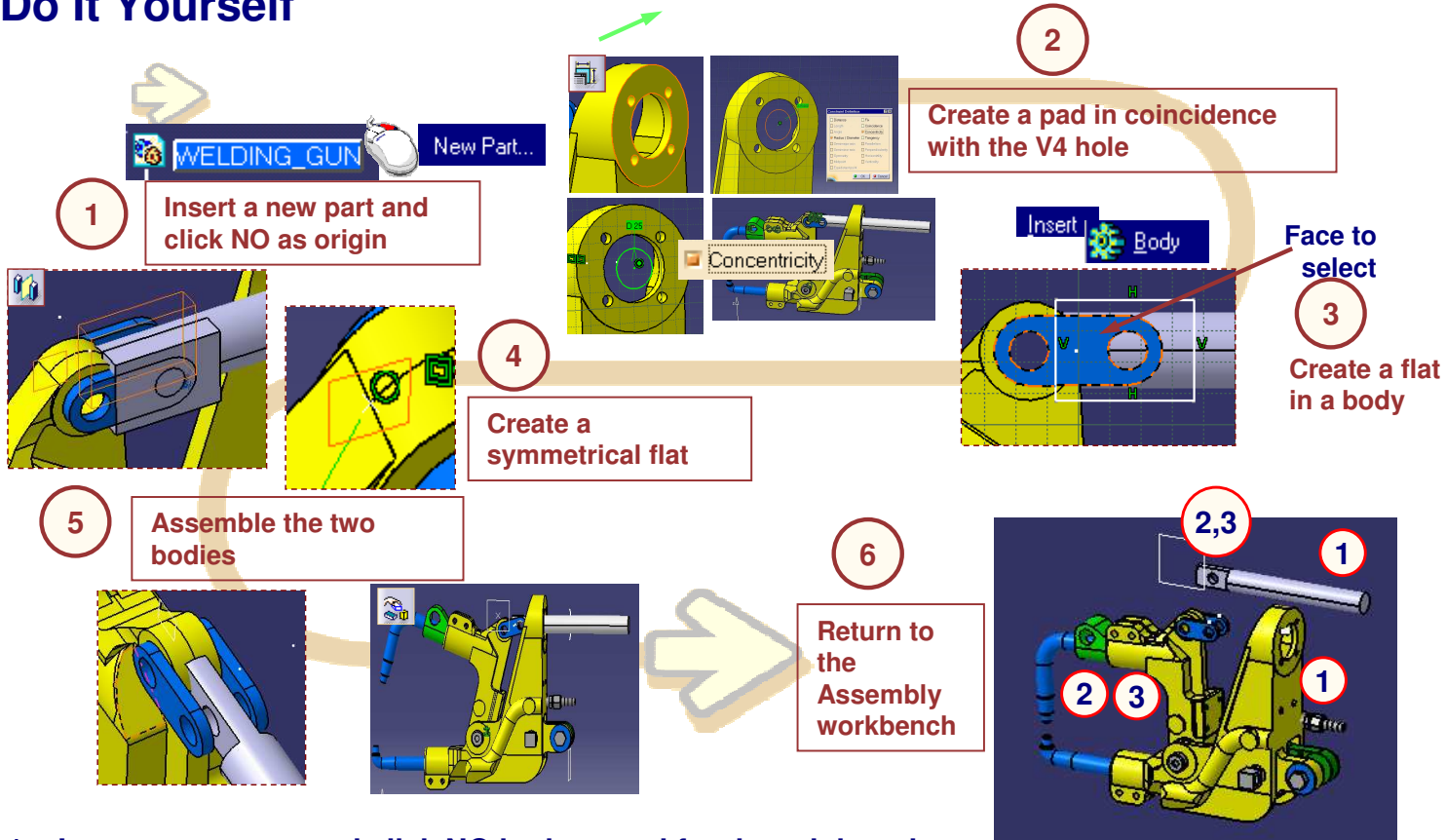
Master Exercise

Step 4: Design in Context

20 min

In this step, you will design the welding axis in context of the assembly.

Do It Yourself

2 Create a pad in coincidence with the V4 hole

1 Insert a new part and click NO as origin

Concentricity

Insert Body

Face to select

3 Create a flat in a body

4 Create a symmetrical flat

5 Assemble the two bodies

6 Return to the Assembly workbench
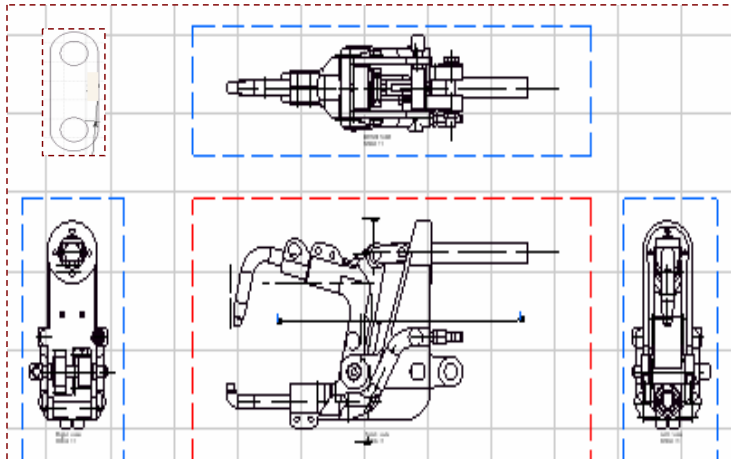
2,3 **1** **2** **3** **1**

1. Insert a new part and click NO in the panel for the origin point.
2. Create a circle (25mm) concentric to the hole and a pad (105mm,110mm).
3. Insert a body to create a flat. Create a rectangle and then pocket.
4. Create the symmetry plane. Do the mirror transformation.
5. Assemble the pockets. Design in context is finished. Save the part.
6. Go to the Assembly workbench and put coincidence constraints on the V5 welding axis.
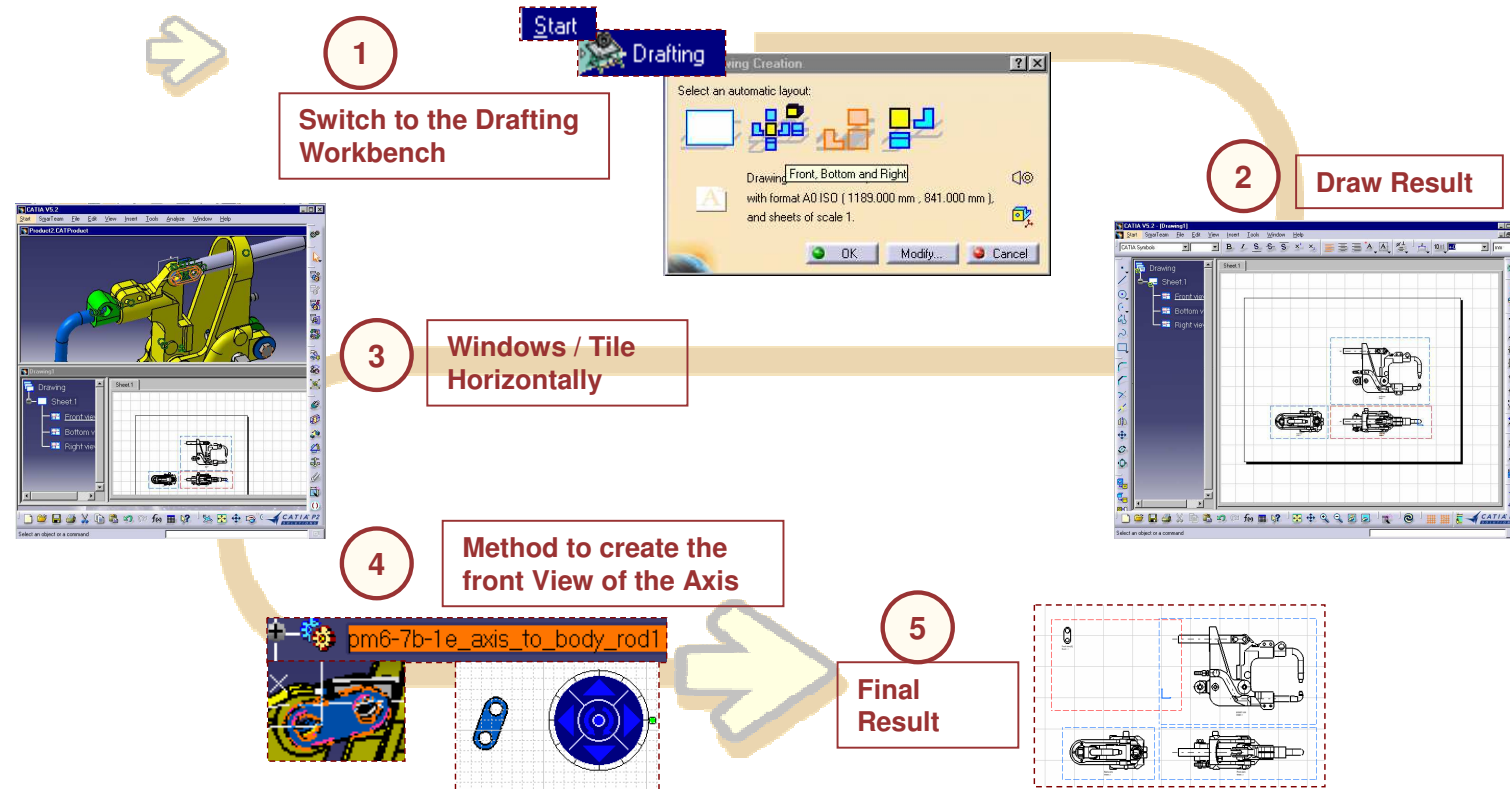   You can also simulate the movement of the Welding Gun. Save your work.

Master Exercise

Step 5: Generative Drafting

20 min

In this step, you will generate a drawing from a CATProduct containing V4 / V5 data.

Do It Yourself

1 Switch to the Drafting Workbench

Start

Drafting

Drawing Creation

Select an automatic layout:

Drawing Front, Bottom and Right
with format A0 ISO (1189.000 mm ; 841.000 mm),
and sheets of scale 1.

OK   Modify...   Cancel

2 Draw Result

3 Windows / Tile Horizontally

4 Method to create the front View of the Axis

pm6-7b-1e_axis_to_body_rod1

5 Final Result

1. **Create a new drawing. Choose the Front / Bottom / Right pre-defined view.**
2. **See the result.**
3. **Create the front View of the Axis to Body Rod. Select Window / Tile horizontally.**
4. **Select Front View function and select in the product, the Axis to Body Rod.**
5. **Click on your sheet to fix the new front view. Save the CATDrawing document.**

Master Exercise

Step 6: DMU Space Analysis

20 min

In this step, you will run the DMU Space Analysis functionalities on the V4/V5 geometry.
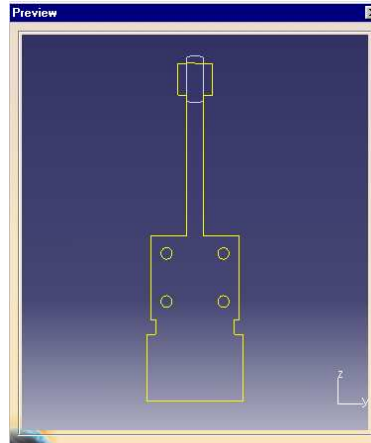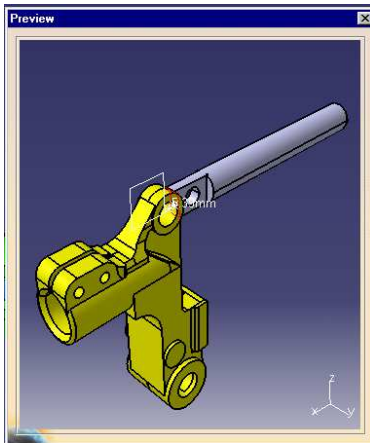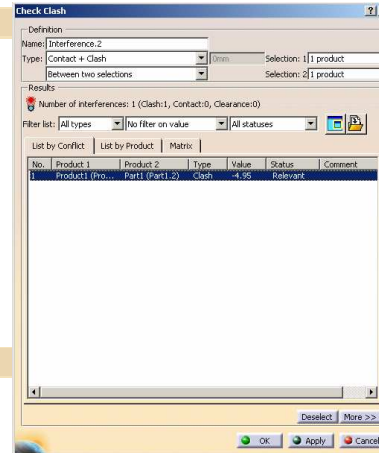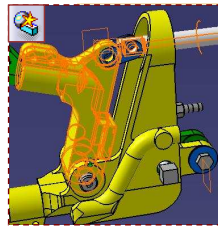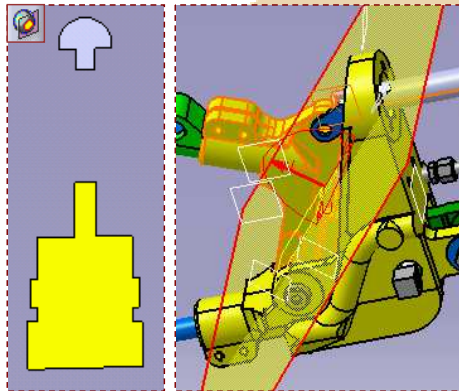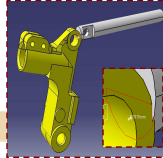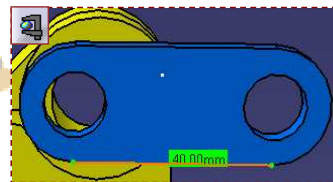
Do It Yourself

1

**Analyse of clash.
One Interference is
found**

2 **Section of the same
parts**

3 **Measure of edge
distance**

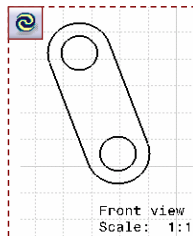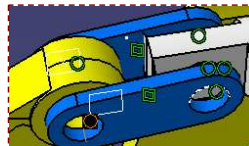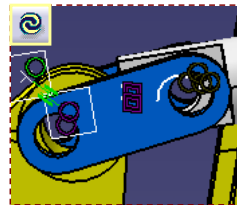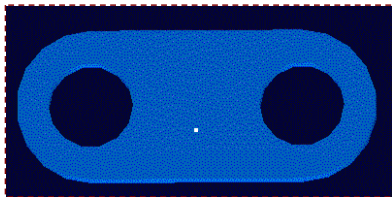1. **Analyse of clash between the V5 Welding Axis and the Movable Yoke.**
2. **Sectioning of the same parts. Use the CTRL key to multi-select.**
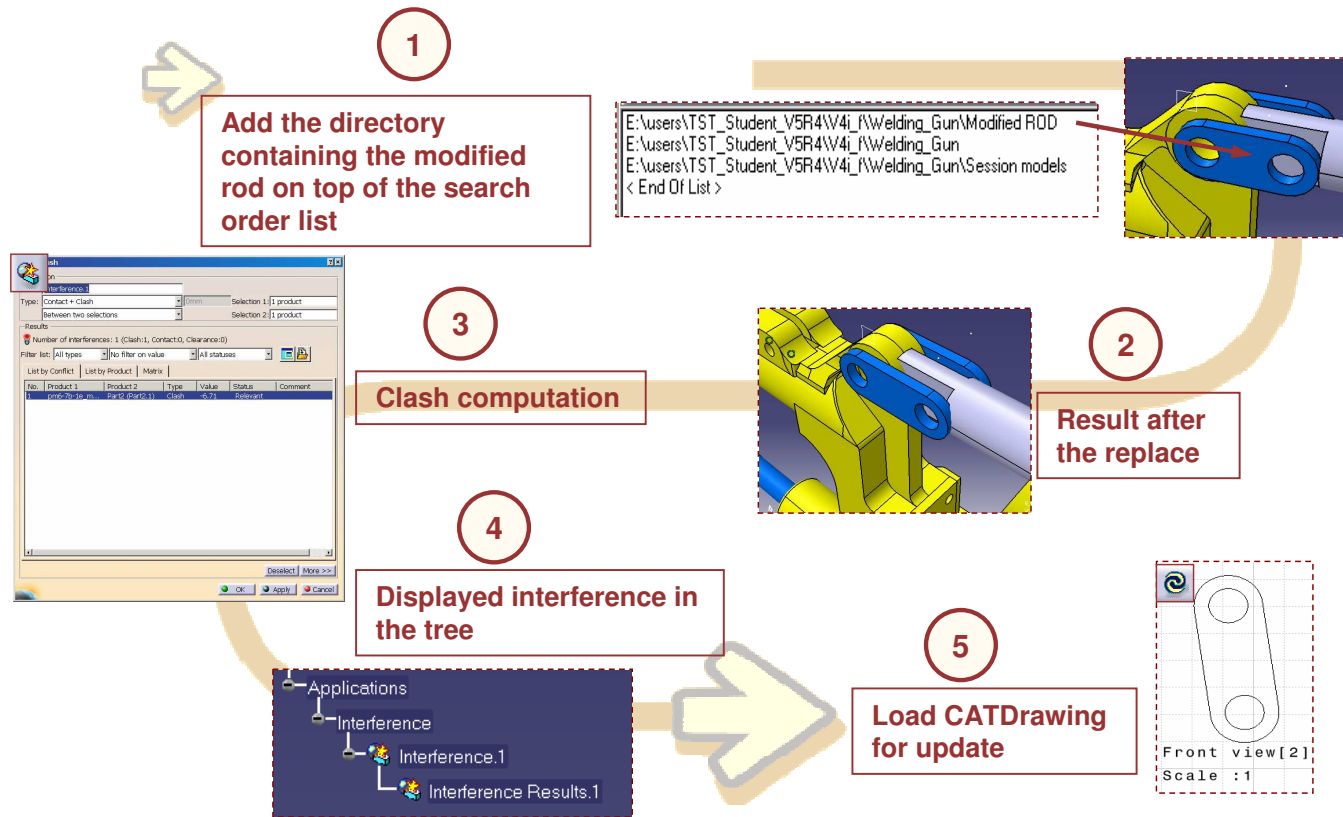3. **Measure the edge of the Axis to Body Rod. Save the CATProduct.**

Master Exercise

Step 7: V4 Modifications and Update in V5

20 min

In this step, you will simulate the modification of a V4 model and see the impact in CATIA V5.

Do It Yourself

1

Add the directory containing the modified rod on top of the search order list

```
E:\users\TST_Student_V5R4\V4i_f\Welding_Gun\Modified ROD
E:\users\TST_Student_V5R4\V4i_f\Welding_Gun
E:\users\TST_Student_V5R4\V4i_f\Welding_Gun\Session models
< End Of List >
```

3

Clash computation

2

Result after the replace

4

Displayed interference in the tree

5

Load CATDrawing for update

Front view[2]
Scale : 1

Applications
  Interference
    Interference.1
      Interference Results.1
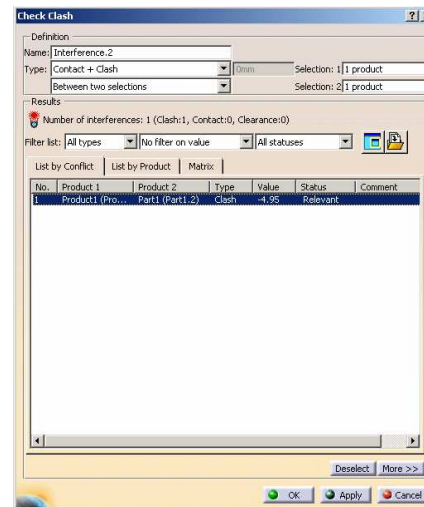
1. **Using the Search Order, reference the AXIS_TO_BODY_ROD file used in the "Modified Rod" directory. Put that directory on top of the search order list.**
2. **Result on the digital mock-up.**
3. **Update the interference which is previously computed.**
4. **Expand the Applications box of the tree to display the interference.**
5. **Load the CATDrawing document and update it.**