Circuit Board Design

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Preface

The CATIA Version 5 Circuit Board Design is a new generation product which addresses circuit board design in a mechanical design context and interfaces with external ECAD systems.

The CATIA - Circuit Board Design offers the following main functions:

- Data can be imported from ECAD Systems through IDF Files
- Data can be exported to ECAD Systems through IDF Files
- Circuit board geometry is designed within the global assembly product definition taking into account the mechanical context
- Electronic properties such as top and bottom faces can be defined and edited on any parts and assemblies
- Spatial and technological constraint areas can be created and managed
- Circuit board definitions can be analyzed in the context of the full Digital Mock-Up definition

- Creation and management of electronic part catalogs are available

As a scalable product, CATIA Version 5 Circuit Board Design can be used in cooperation with other current or future companion products of the next CATIA generation such as CATIA Version 5 Assembly Design and ENOVIA-DMU Navigator Version 5.

Using This Guide   More Information

Where to Find More Information

Prior to reading this book, we recommend that you read the *CATIA - Infrastructure User's Guide.*

The *CATIA - Part Design User's Guide,* the *CATIA - Assembly Design User's Guide* and *the CATIA - Sketcher Version 5* may prove useful.

Up   Using This Guide   More Information

What's New?

New	[Creating Components](#)
Enhanced	[Component Properties](#)
New	[Creating a Catalog](#)
New	[Query from Catalog](#)
New	[Import using a Catalog](#)
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Getting Started

Before getting into the detailed instructions for using CATIA - Circuit Board Design Version 5, the following tutorial provides a step-by-step scenario demonstrating how to use key functionalities.

Before starting this scenario, you should be familiar with the basic commands common to all workbenches. These are described in the *CATIA - Infrastructure User's Guide*.

The main tasks proposed in this section are:

Tasks

Accessing the Workbench

Creating the Board

Pocket and Holes

Constraint Areas

Exporting Data

Importing Data

All together, these tasks should take about 15 minutes to complete.

# Accessing the Circuit Board Workbench in an Assembly Context

This task explains how to set up the environment to design in context.

To design in context, you need to load the CATProduct document containing the context; in this case the bottom part of a CD player.
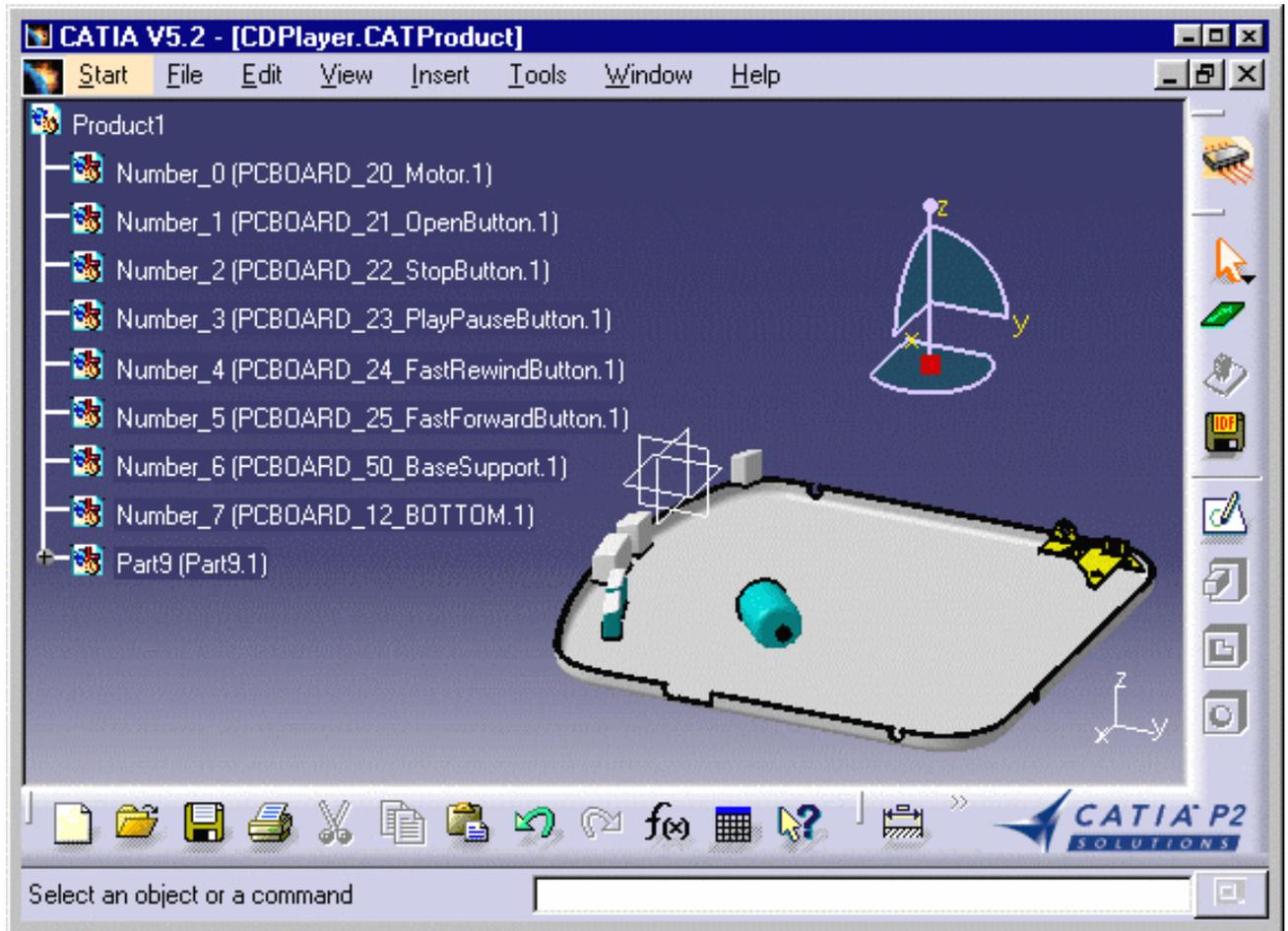
Open the CDPlayer.CATProduct document from the \online\samples\circuitboard directory.

Make sure the Product1 feature is highlighted before changing workbench.

1. Choose the Circuit Board Design item from the Start -> Equipments & Systems menu.
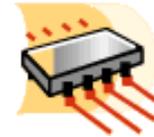
The Circuit Board Design workbench is displayed and ready to use.

Note that a new part Part9 has been automatically added to the specification tree. It will be used to design the board.

You can add the Circuit Board Design workbench to your Favorites, using the Tools -> Customize item. For more information, refer to *CATIA V5 - Infrastructure User's Guide*.

Up

Accessing the Workbench

Creating a Board

Pocket and Holes

Constraint Areas

Exporting Data

Importing Data

Creating a Board

This task explains how to create a board then design its geometry.

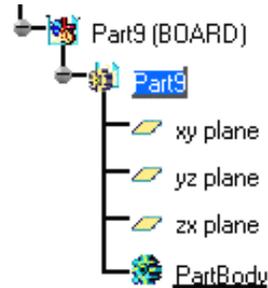Circuit Board Design functions are available when you are in the Part Design environment. Several functions are integrated from the Part Design workbench.

You are in the CDPlayer.CATProduct document.
The Circuit Board Design workbench is displayed.
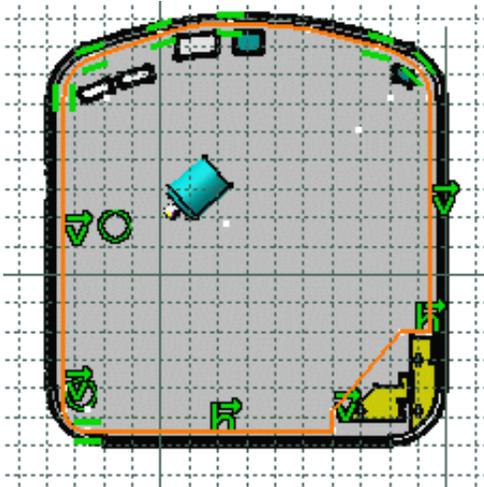
1. Select the Board icon ⬦.

2. Select the Sketcher icon ⬦.

3. Select the bottom plane of the CD box.
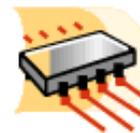The Sketcher workbench is launched.

4. Click the Profile icon ⬦.

5. Sketch the contour of the Board.
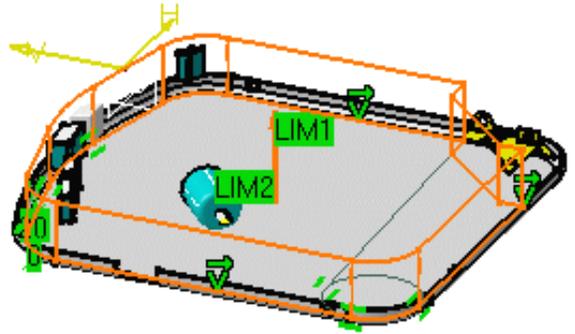The profile looks like this:

6. Quit the Sketcher using the Exit icon ⬦:
You are now in the Circuit Board Design workbench.

For more information about the Sketcher, refer to the *CATIA V5 - Sketcher User's Guide*.

7. Click the Pad icon .

The Pad Definition dialog box appears.

Pad Definition

First Limit

Type: Dimension

Length: 2mm

Limit: No selection

☐ Mirrored extent

Reverse Direction

More>>

OK   Apply   Cancel

8. Enter 2mm in the Length field.

9. Click OK.

The pad turns green, this means the board has been created.
The board has been created from the active part: Part9(BOARD)

CATIA recognizes the first pad created as the Board.
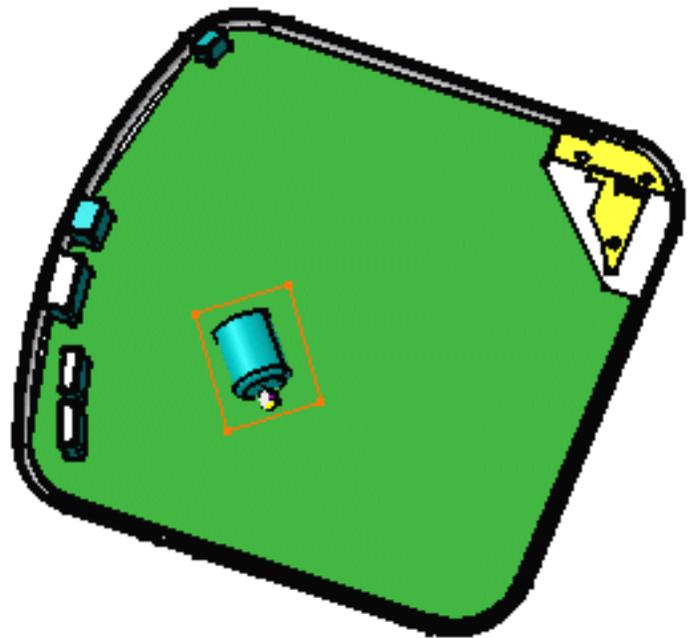
Creating a Pocket and Holes

This task shows how to create a pocket and two holes, with technological properties.
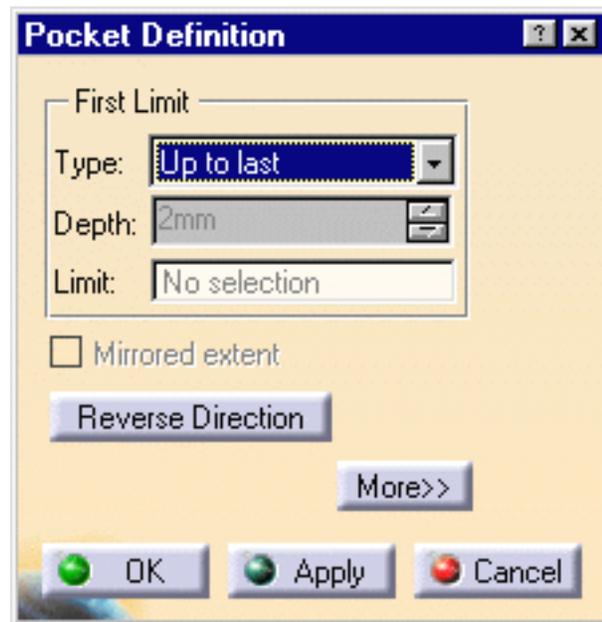
1. Sketch the contour of the pocket.

To do so:

- click the Sketcher icon
- select the board to define the working plane
- click the Profile icon and draw the contour around the motor
- then click the Exit Sketcher icon to return to the 3D world.

2. Click the Pocket icon .
The Pocket Definition dialog box is displayed and CATIA previews a pocket with default parameters.

3. Set the Up to last option to define the pocket limit.
This means that the application will extend the pocket to the last possible face, that is the pad bottom.

Pocket Definition

First Limit

Type: Up to last

Depth: 2mm

Limit: No selection

☐ Mirrored extent

Reverse Direction

More>>

OK   Apply   Cancel

4. Click OK.

5. Click the Hole icon.

6. Select the board to define the working plane.
The Hole Definition dialog box appears.
CATIA previews the hole to be created with default parameters.
7. Enter 4mm for the diameter and click OK.
The hole is created.

8. Repeat the steps 5, 6 and 7 for the second hole.

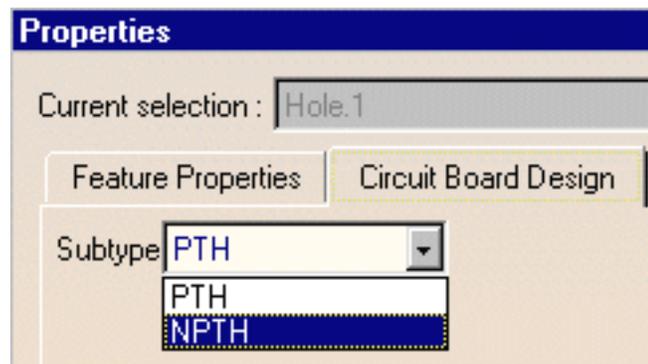The pocket and holes are added to the specification tree.
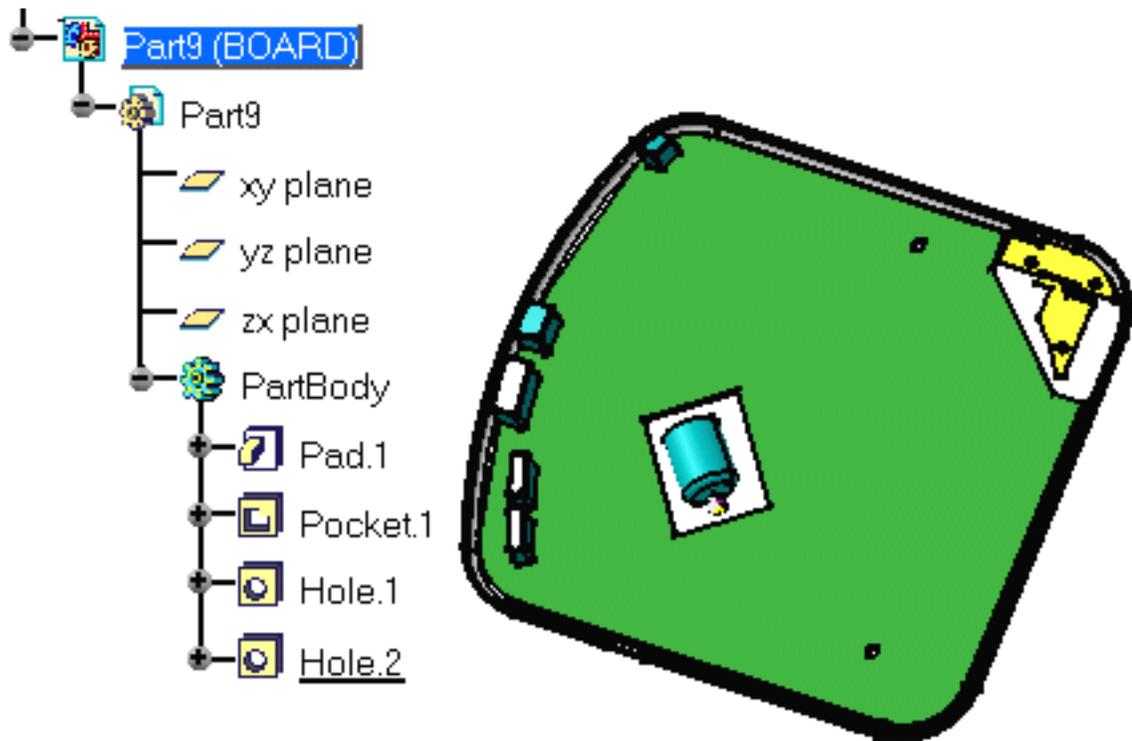
9. Right-click the Hole.1 in the specification tree to display the contextual menu.

10. Select Properties.
The Properties dialog box opens.

11. Click the More... button.

12. Select the Circuit Board tab.

13. Switch the Subtype to NPTH (non conductive) and click OK.

The hole is more than a simple hole: it has a technological property.

14. Repeat the steps 9 to 13 for the second hole.
These holes are used for mounting purposes.

Creating a Constraint Area

This task shows you how to create a constraint area.

A constraint area is a space reservation used for different purposes such as allowing/forbidding the routing or placing of components etc.
Each constraint area belongs to a part, this is to say that you need to create a new part before creating a constraint area.

Create a new part in your CDPlayer.CATProduct document.
To do so:

- Double-click Product1 in the specification tree
- Right-click the Product1 feature and select New Part in the contextual menu
- Enter ConstArea for the Part Number and click OK
- Click No to define the origin point of the assembly as the new part origin point.

1. Select the Constraint Area icon .
The Constraint Areas dialog box opens.
The constraint area types are:

- ROUTE_OUTLINE
- PLACE_OUTLINE
- OTHER_OUTLINE
- VIA_KEEPOUT
- PLACE_KEEPOUT
- PLACE_REGION
- ROUTE_KEEPOUT

Constraint Areas

ROUTE_OUTLINE
PLACE_OUTLINE
OTHER_OUTLINE
VIA_KEEPOUT
PLACE_KEEPOUT
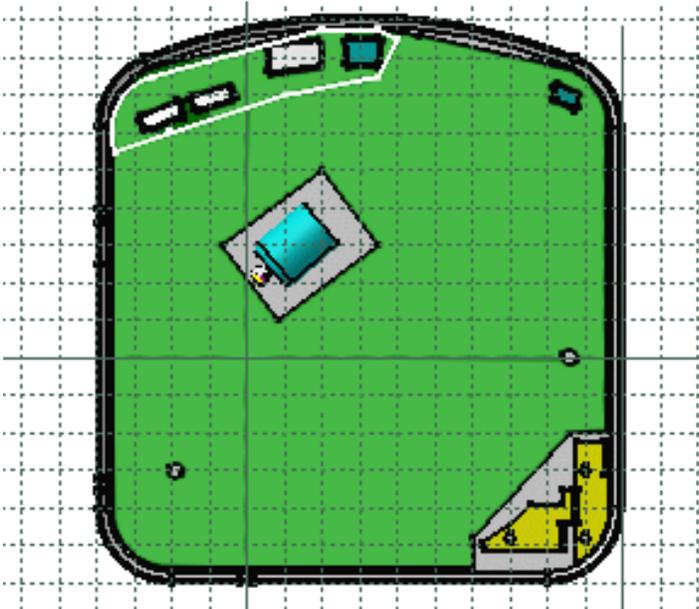PLACE_REGION
ROUTE_KEEPOUT

OK   Cancel
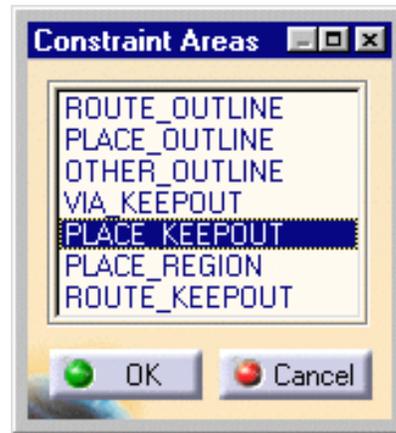
2. Choose the PLACE_KEEPOUT type and click OK.

3. Select the Sketcher icon .

4. Select the board to define the working plane.

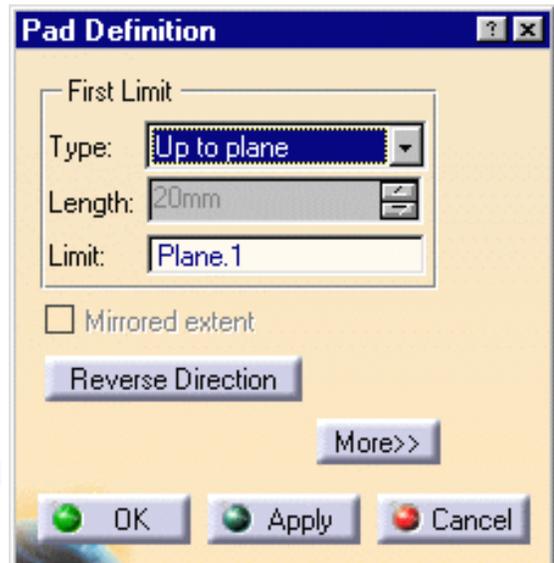5. Click the Profile icon and draw the contour around the CD buttons.

6. Click the Exit Sketcher icon ⬆ to return to the 3D world.

7. Click the Pad icon ▱ .

The Pad Definition dialog box appears.

8. Switch the type to Up to plane.

9. Select the plane on which the CD buttons are placed to define the bottom limit of the pad and click OK.

The PLACE_KEEPOUT area is displayed in transparency.

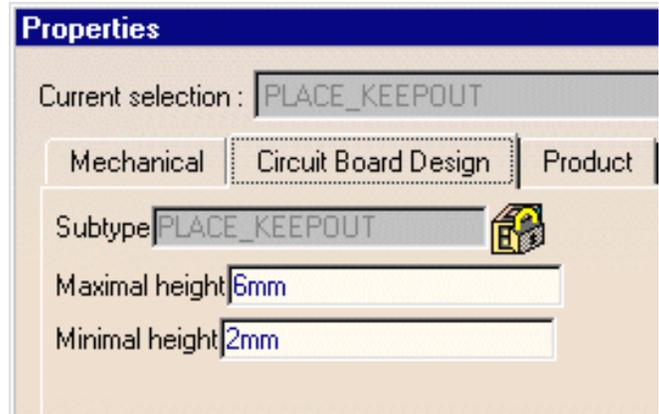ℹ️ The pad is more than a simple pad: it has technological properties.

10. Right-click the PLACE_KEEPOUT.1 in the specification tree to display the contextual menu.

11. Select Properties.
The Properties dialog box opens.

12. Click the More... button.

13. Select the Circuit Board tab.

14. Enter 6mm in the Maximal height field and 2mm in the Minimal height field then click OK.

Properties

Current selection : PLACE_KEEPOUT

| Mechanical | Circuit Board Design | Product |

Subtype PLACE_KEEPOUT

Maximal height 6mm

Minimal height 2mm

🔑 The pad is updated: the maximal and minimal height are linked to the pad limits.
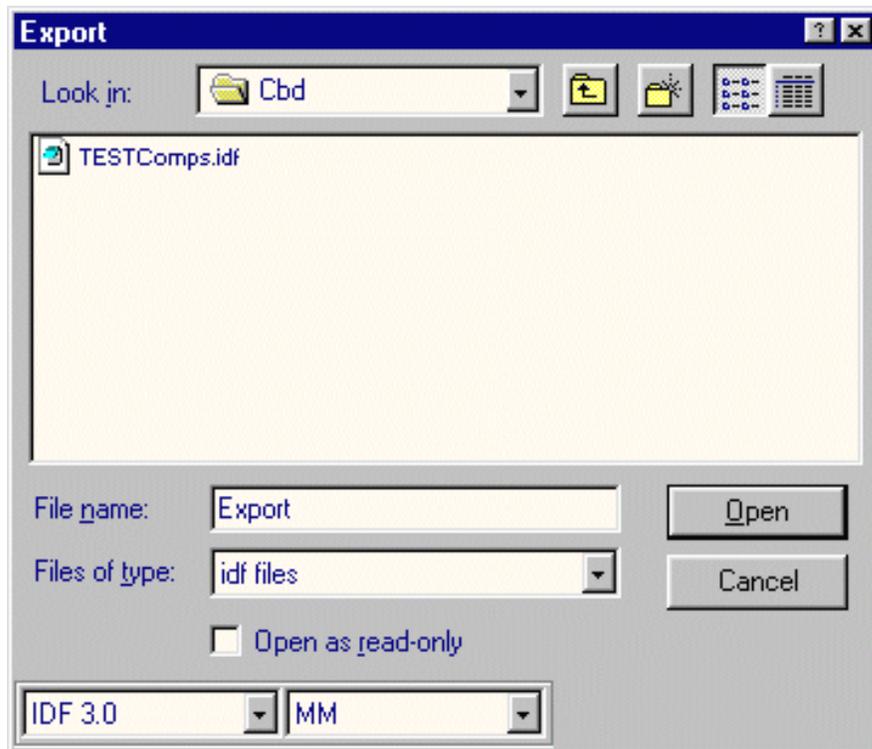It represents the overall dimensions of the constraint area.

🏁

Exporting Data (as IDF)

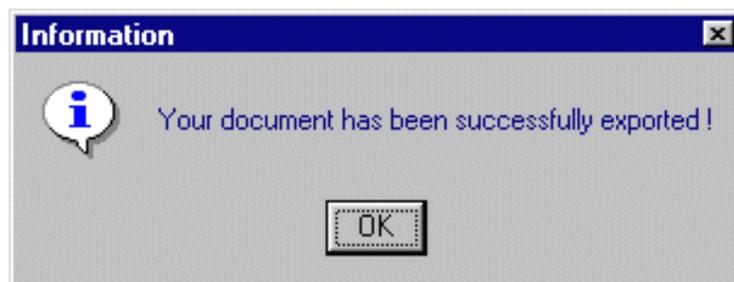This task shows how to export data to an IDF file.

1. Click the Export icon ⬛.
The Export dialog box opens.

2. Enter Export in the File name field.

3. Press Open.

An Information message is displayed:

Importing the IDF File

This task shows how to import the IDF data previously saved to a new document. You can also import IDF data from an ECAD system.
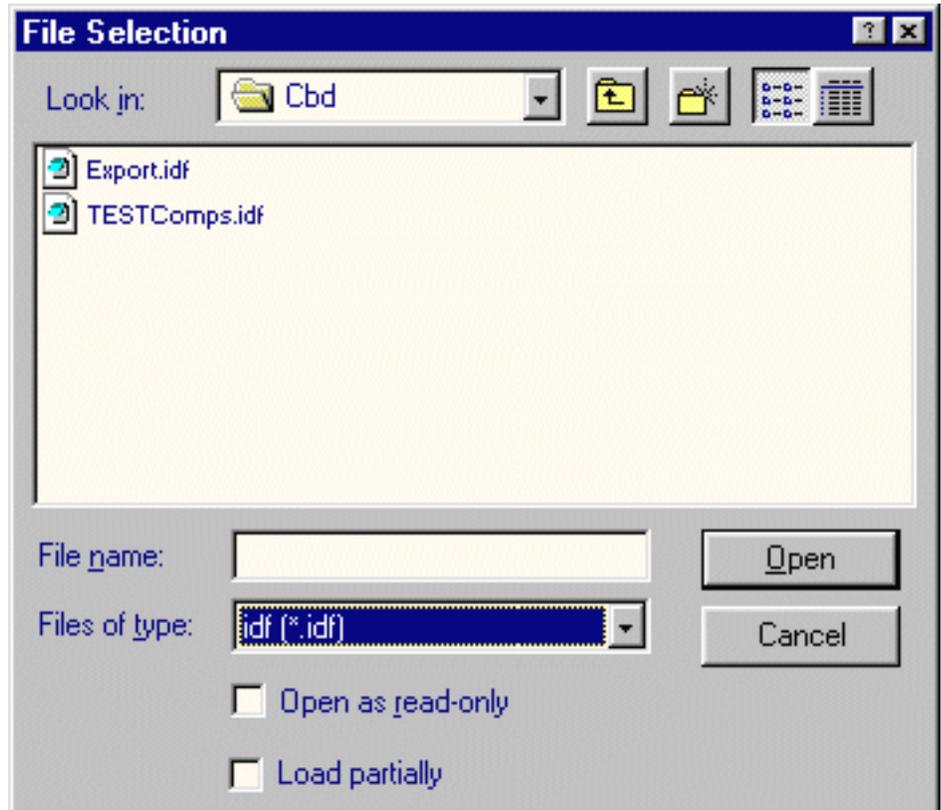
You are in a new Product document.

1. Select File -> Open in the menu bar.
The File Selection dialog box opens.

2. Select .idf in the File of type field.

File Selection

Look in: Cbd

Export.idf
TESTComps.idf

File name:

Files of type: idf (*.idf)

☐ Open as read-only
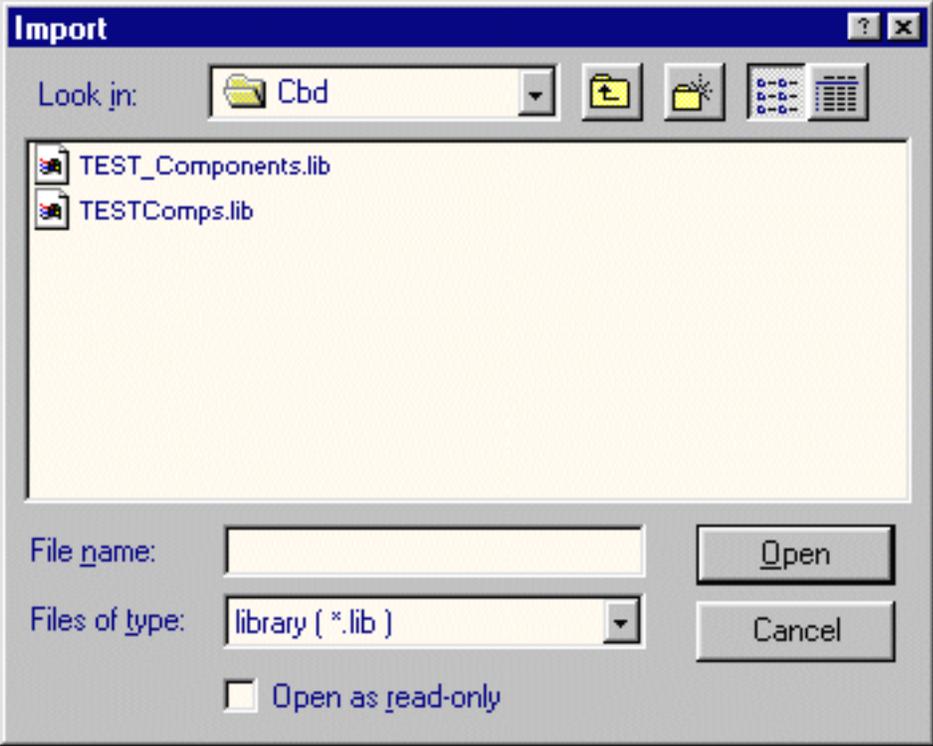
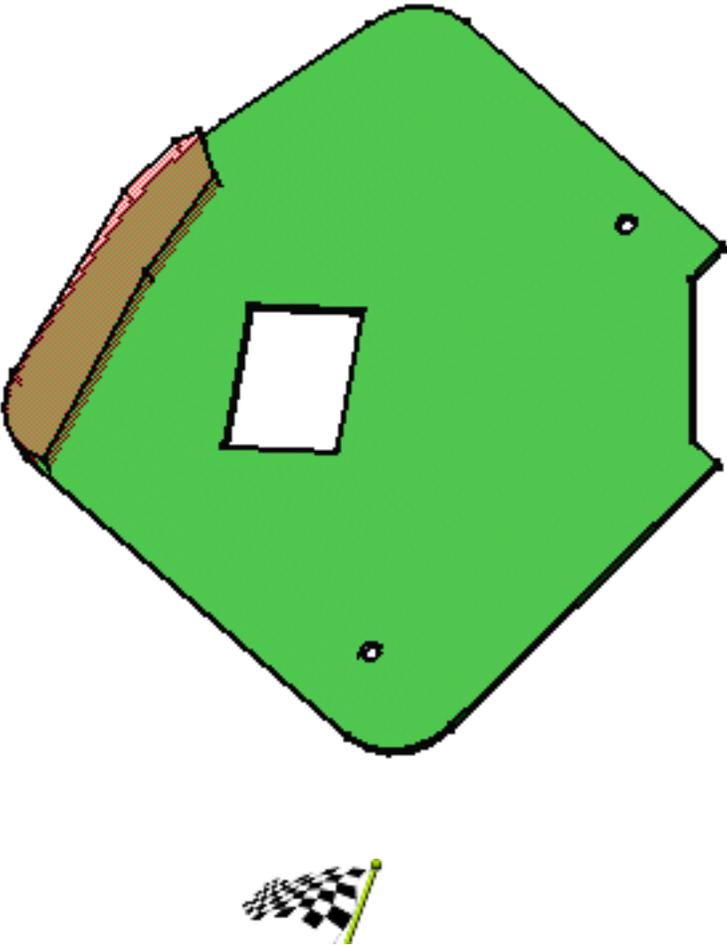☐ Load partially

Open

Cancel

3. Select the Export.idf file.
4. Press Open.

If the file contains components, an Import dialog box opens.

5. Select the library of interest.

6. Press Open.

Import

Look in: Cbd

TEST_Components.lib
TESTComps.lib

File name:

Files of type: library (*.lib)

Open as read-only

Open

Cancel

The result should look like this:

Basic Tasks

The Basic Tasks section explains and illustrates how to create various kinds of features.
The table below lists the information you will find.

Themes

Creating a Board

Managing Components

Working with Catalogs

Importing

Workbench Integration

Creating a Board

This section explains and illustrates different methods to create and/or modify a board and its components.
The table below lists the information you will find.

From Previously Created Part
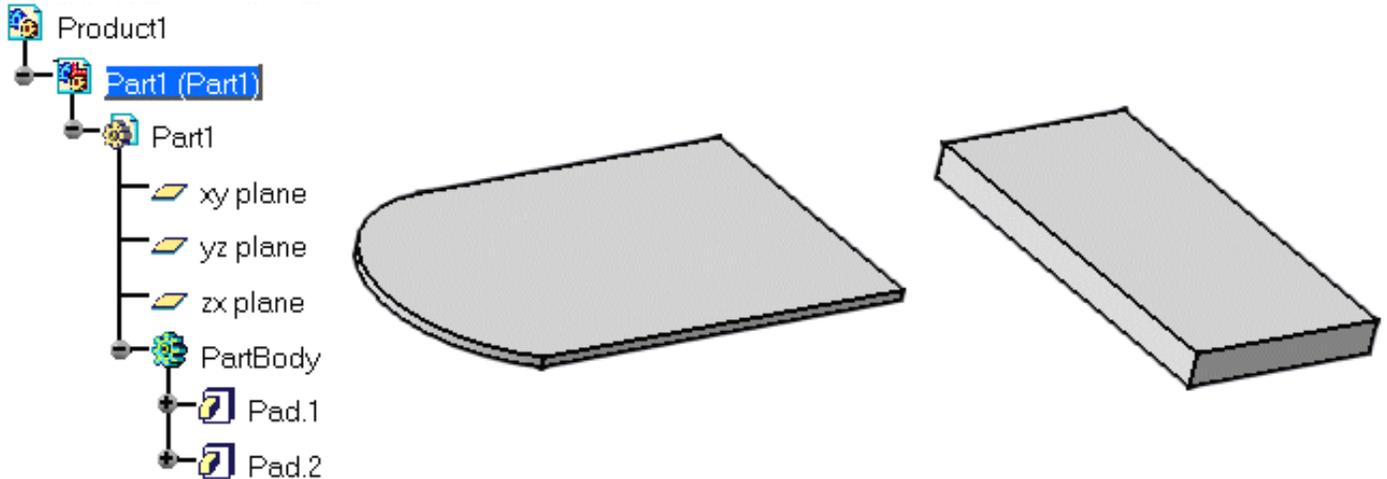
Automatic Recognition

Up

Creating a Board

Managing Components

Working with Catalogs

Importing

Workbench Integration
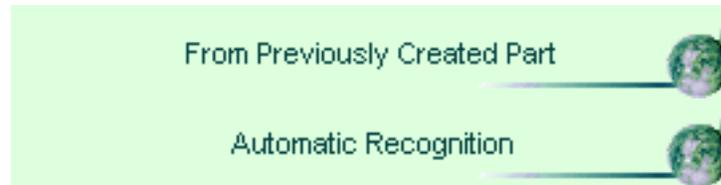
From a Previously Created Part

This task shows how to create a board from an existing part.

Open the Scenario1.CATProduct document from the Samples/circuit board directory.
It contains a part with two pads.

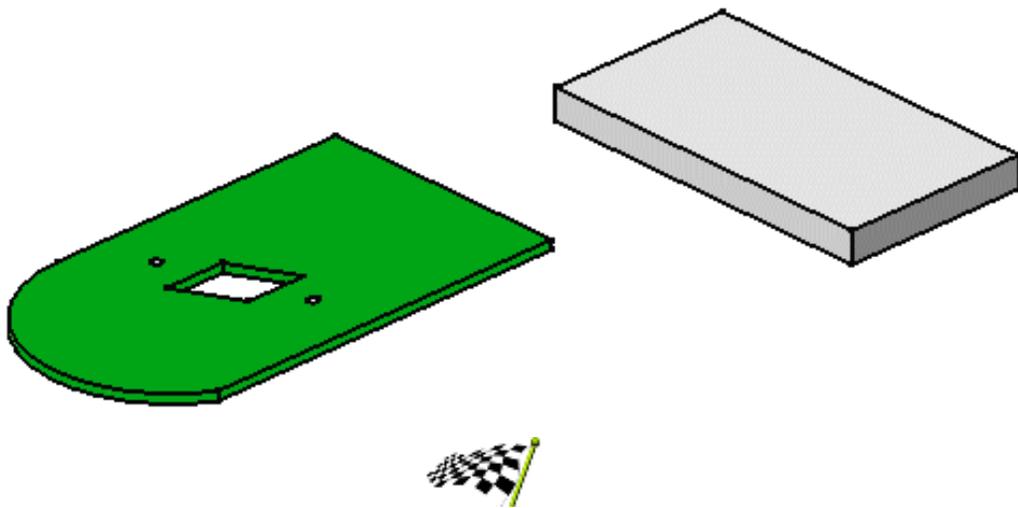1. Click the Board icon.

The PartBody contains two pads, so you have to select the pad which will represent the board.

2. Select Pad.1.

CATIA recognizes the board and displays it in green.
You can add holes, pockets and constraint areas, as well as technological properties.

Automatic Recognition
in Circuit Board Workbench

This task illustrates how Circuit Board features are automatically recognized.

Open the Scenario2.CATProduct document from the Samples/circuit board directory. It contains a part with a pad, holes and a pocket.
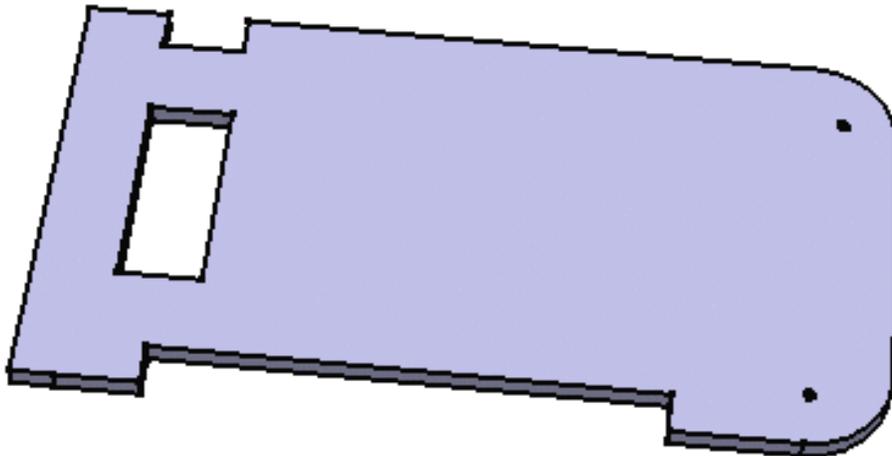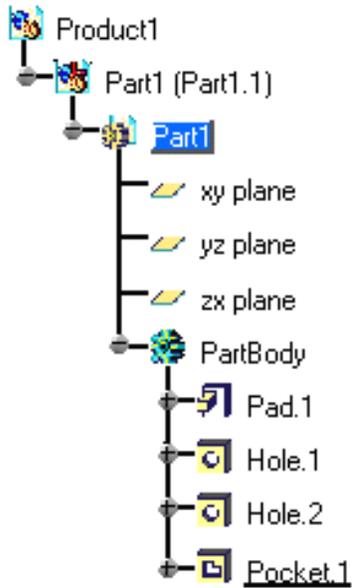
1. Click the Board icon .

Since there is only one pad, it is recognized to be the board representation.

The objects (board and holes) have automatically acquired technological properties:
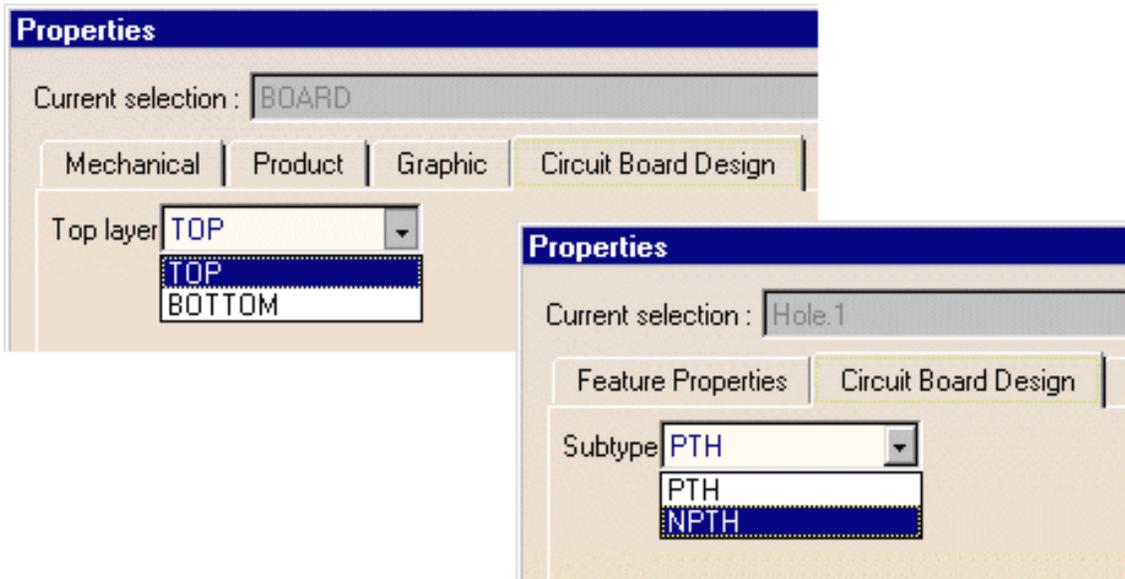
2. Select the Export icon .

A board is recognized, all the related features are taken into account and exported to the IDF file.

3. Open the .idf file as a text file. It looks like this:

```
.HEADER
BOARD_FILE 2.0 "CATIA V5R2" 5/21/99.14:59:32 1
Part1 MM
.END_HEADER
.BOARD_OUTLINE
5,000000
0        -100,000000        -80,000000        0,000000
0        -80,000000         -80,000000        0,000000
0        -80,000000         -70,000000        0,000000
0        40,000000          -70,000000        0,000000
0        40,000000          -80,000000        0,000000
0        70,000000          -80,000000        0,000000
0        90,000000          -60,000000        90,000000
0        90,000000          10,000000         0,000000
0        70,000000          30,000000         90,000000
0        -70,000000         30,000000         0,000000
0        -70,000000         20,000000         0,000000
0        -90,000000         20,000000         0,000000
0        -90,000000         30,000000         0,000000
0        -110,000000        30,000000         0,000000
0        -110,000000        -80,000000        0,000000
0        -100,000000        -80,000000        0,000000
1        -90,000000         0,000000          0,000000
1        -70,000000         0,000000          0,000000
1        -70,000000         -50,000000        0,000000
1        -90,000000         -50,000000        0,000000
1        -90,000000         0,000000          0,000000
.END_BOARD_OUTLINE
.DRILLED_HOLES
2,000000        77,296326        12,748661        PTH        BOARD
2,000000        71,181679        -66,700829       PTH
BOARD
.END_DRILLED_HOLES
```

Up From Previously Created Par Automatic Recognition

Components Management

This section explains and illustrates different methods to create, use and/or modify components.
The table below lists the information you will find.

Component Creation

Component Properties

Components and Constraints

Query From Catalog

Up

Creating a Board

Managing Components

Working with Catalogs

Importing

Workbench Integration
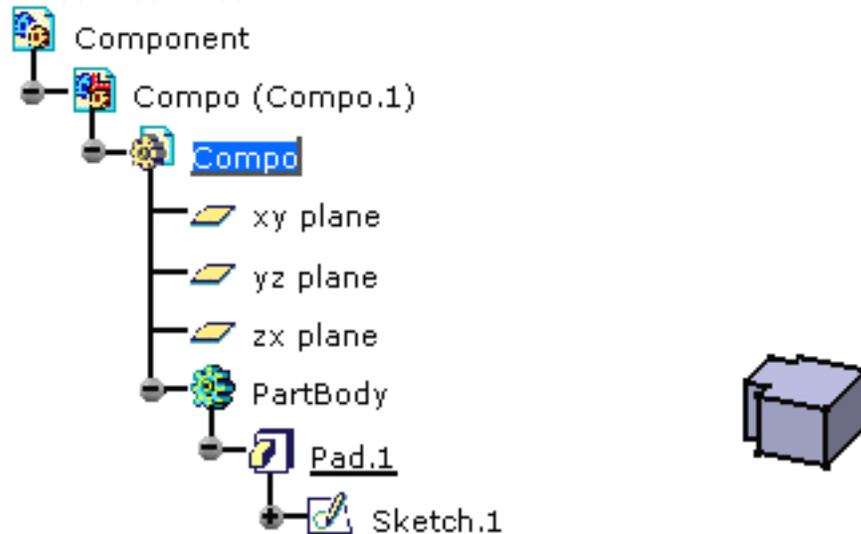
new Creating a Component

This task explains how to create and manage components.

Open the Component.CATProduct document from the \online\samples\circuitboard directory.
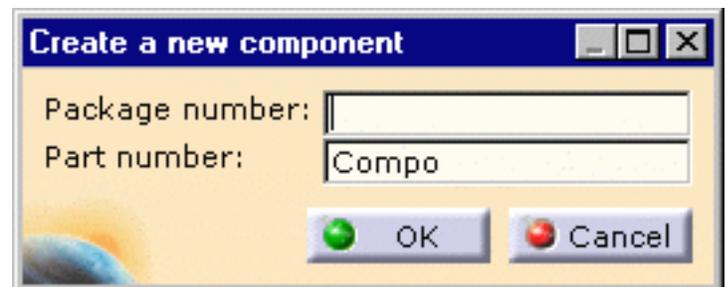
The window looks like this:

1. Select the Component icon
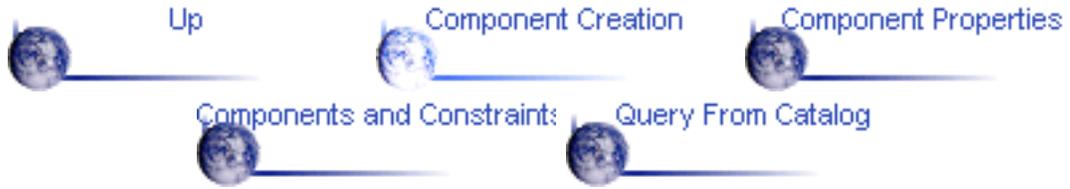
.

The New Component dialog box opens.

2. Enter comp1 in the Package number field.

3. Click OK to validate.

4. Select Pad.1 in the specification tree. This is the geometrical representation of the new component.

The component is created.
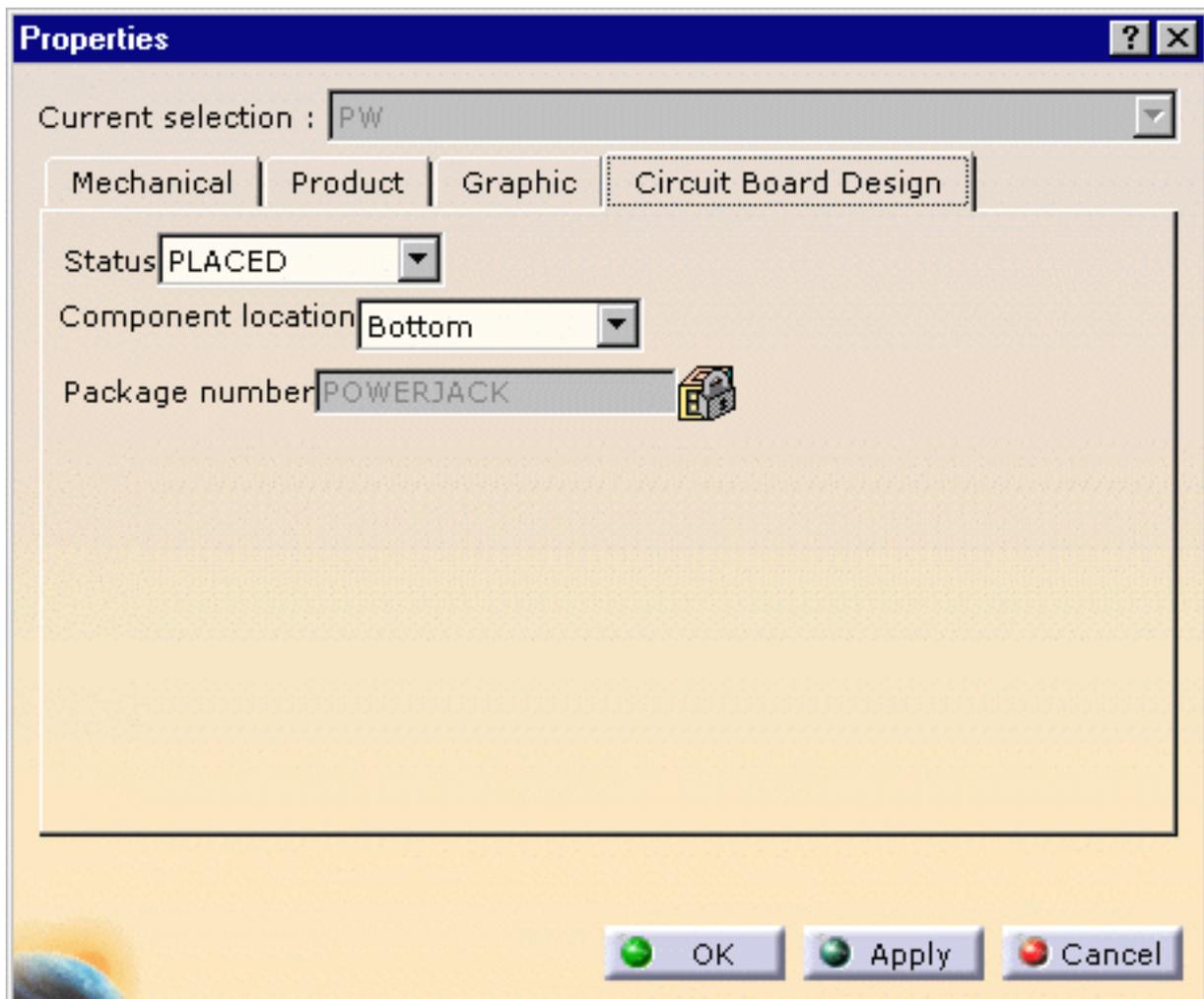
new Component Properties

This task explains how to browse the technological properties of a component.

Open the Scenario3.idf file and the corresponding Scenario3.lib file from the \online\samples\circuitboard directory. An Assembly Product document is generated.

1. Select the power jack PWJK (PW.1).

2. Using the right mouse button, open the Properties dialog box.

3. Click the More... button then select the Circuit Board Design tab.
The component is located on the Top face and its Status is Unplaced.

4. Change the Component location to Bottom and the Status to Placed.

Properties	? ☒
Current selection : PW	▼

| Mechanical | Product | Graphic | **Circuit Board Design** |

Status `PLACED` ▼

Component location `Bottom` ▼

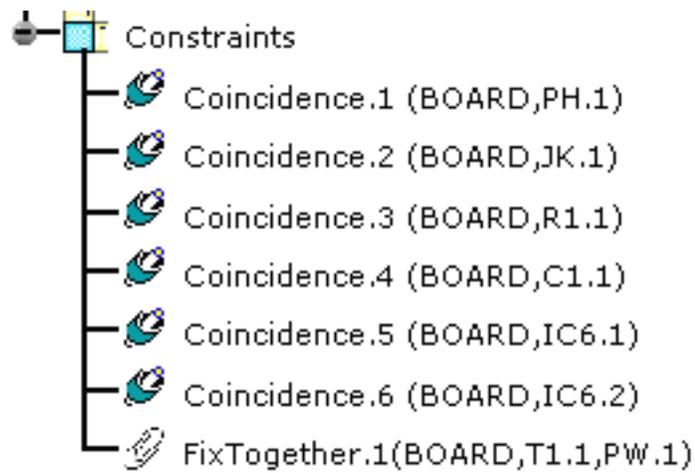Package number `POWERJACK` 🔒

| ● OK | ● Apply | ● Cancel |

CATIA projects the component onto the bottom board surface (in this case). You can still move the component to the desired location on the surface of the board.

5. Change the Status to Fixed

The FixTogether Constraint is created between the board and the power jack PW.1.

Constraints
- Coincidence.1 (BOARD,PH.1)
- Coincidence.2 (BOARD,JK.1)
- Coincidence.3 (BOARD,R1.1)
- Coincidence.4 (BOARD,C1.1)
- Coincidence.5 (BOARD,IC6.1)
- Coincidence.6 (BOARD,IC6.2)
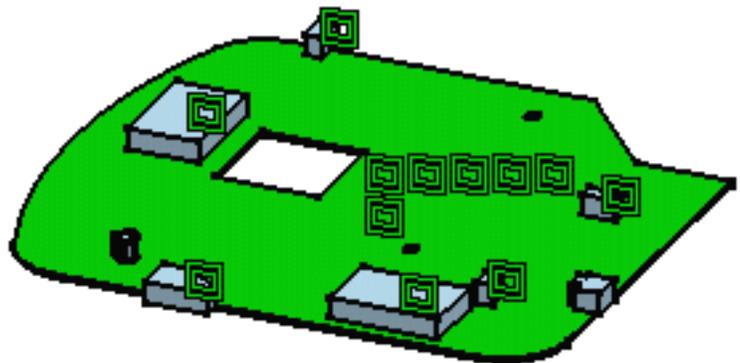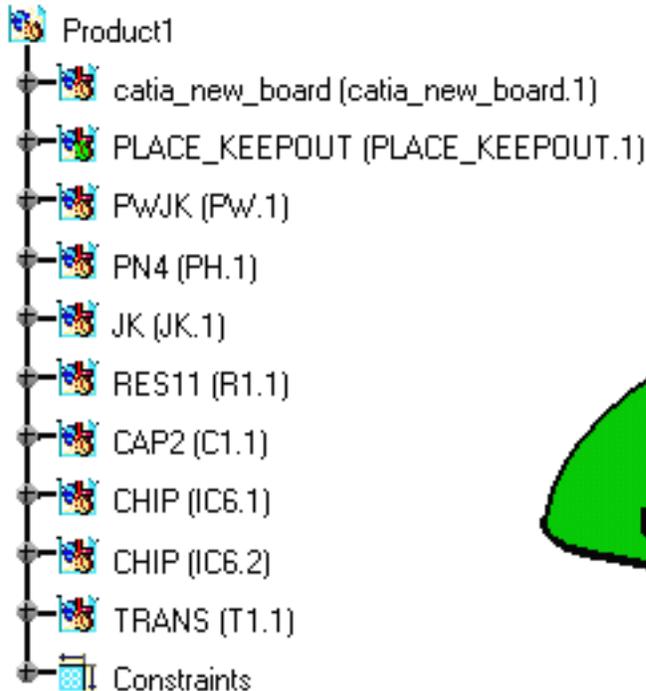- FixTogether.1(BOARD,T1.1,PW.1)

Components and Constraints

This task explains how to manage components given the component constraint type.

Open the Scenario3.idf file and the corresponding Scenario3.lib file from the \online\samples\circuitboard directory. An Assembly Product document is generated.
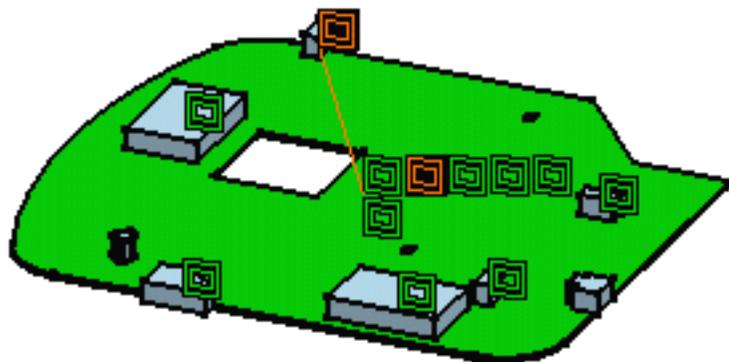

If the configuration you are using provides access to the Assembly workbench, geometrical constraints are created between the board and the components, and are displayed.
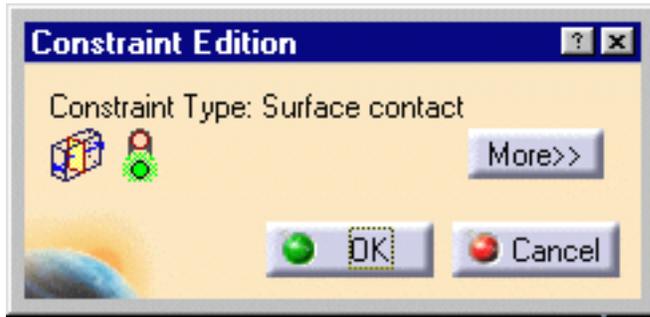Within the assembly, the board need to be anchored.

1. Double-click the power-jack constraint shown here in red.

The Constraint Edition dialog box opens.
The constraint type is either Surface Contact (placed component) or Fix Together (fixed component).

Constraint Edition

Constraint Type: Surface contact

More>>

OK   Cancel

The Assembly constraints (geometrical constraints) linking the components to the board depend on the component status.

Component Status	Assembly Constraint
Unplaced	No Constraint
Placed	Plane to Plane Constraint
Fixed	Fixity Constraint

This information is transmitted through the import file (.idf), the component capability to change location is then driven by the constraint.

Under certain circumstances, a component status can be modified:
- from unplaced to placed or fixed
- from placed to fixed

but never from fixed to placed or unplaced, nor from placed to unplaced (in order to keep the information defined in the ECAD world).

new Query From Catalog

This task explains how to insert a component resulting from a catalog query, in a product.

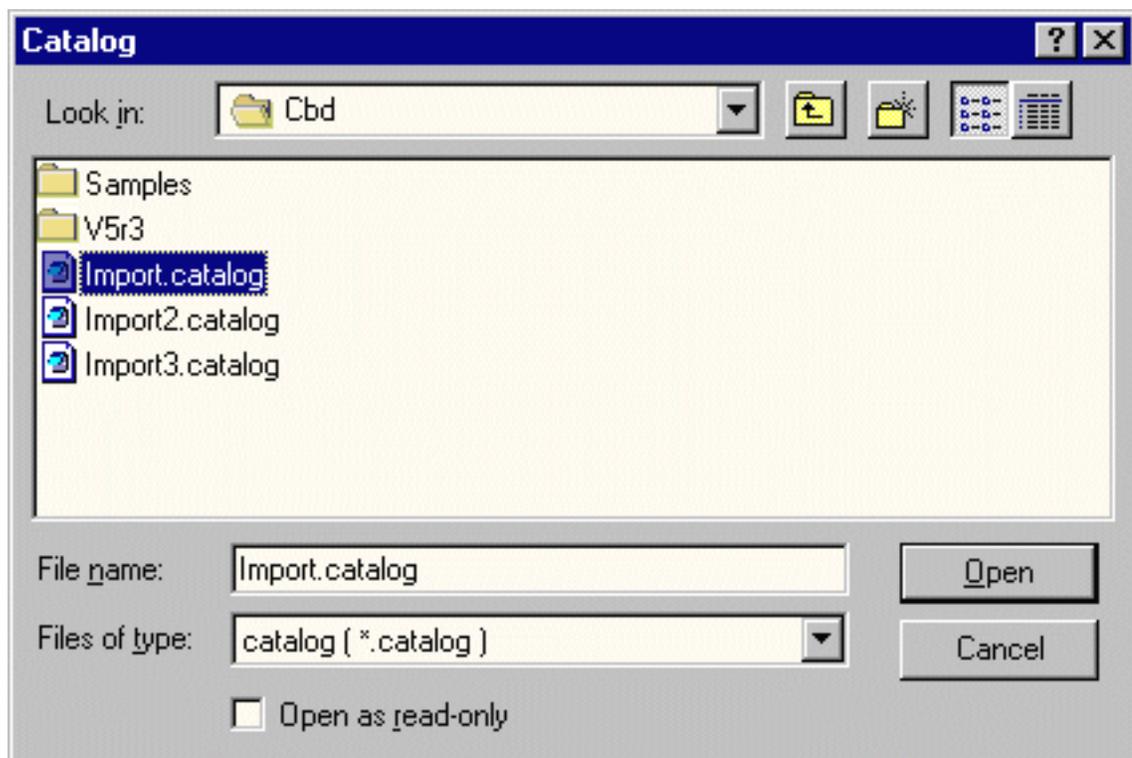Open a new Product document.
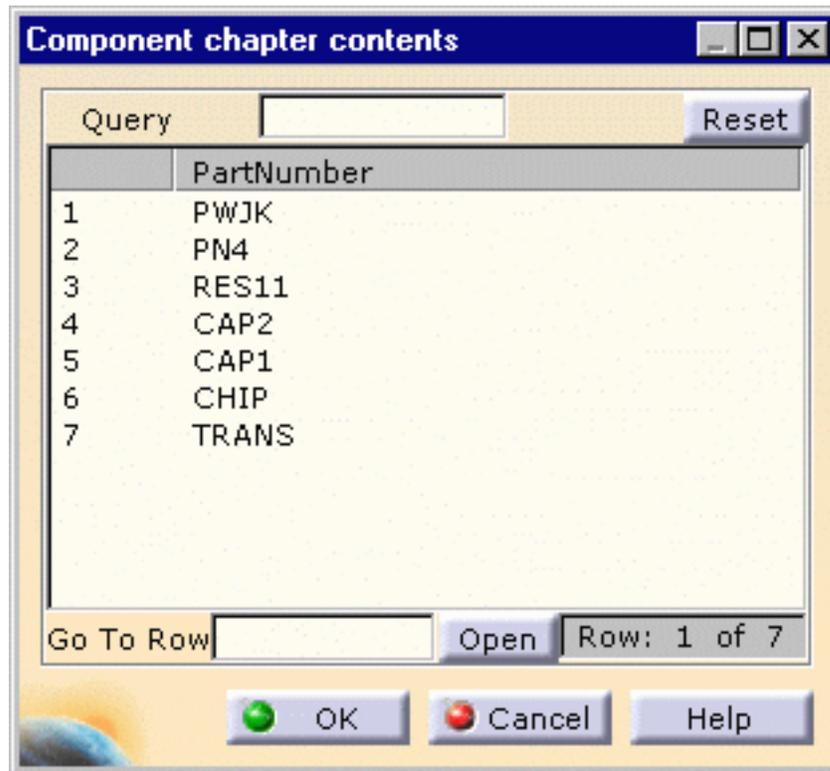
1. Click the Query from Catalog icon -c .

The Catalog dialog box opens.

2. Select the catalog of interest and validate.

Catalog

| Look in: | Cbd |

- Samples
- V5r3
- Import.catalog
- Import2.catalog
- Import3.catalog

File name: Import.catalog

Files of type: catalog (*.catalog)

☐ Open as read-only

Open

Cancel

The Component chapter contents window is displayed.

Component chapter contents

	PartNumber
1	PWJK
2	PN4
3	RES11
4	CAP2
5	CAP1
6	CHIP
7	TRANS

Query · Reset · Go To Row · Open · Row: 1 of 7 · OK · Cancel · Help

3. Select a component, for example RES11

4. Copy it (Edit -> Copy or Ctrl+C)

5. Paste it (Edit -> Paste or Ctrl+V) in your document.
You can also display it using the Open button.
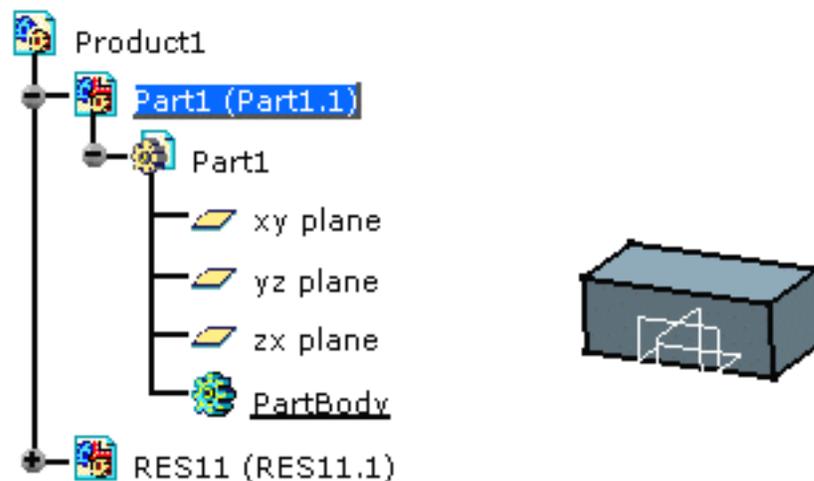
The component is now present in your Product document.

new Working With Catalogs

This section explains and illustrates different methods to create and/or modify the board and its components.
The table below lists the information you will find.

Catalog Creation

Import with Catalog

Up

Creating a Board

Managing Components

Working with Catalogs

Importing

Workbench Integration

new Creating a Catalog

This task explains how to create a catalog for circuit board components.

Open the Component2.CATProduct document from the \online\samples\circuitboard directory.
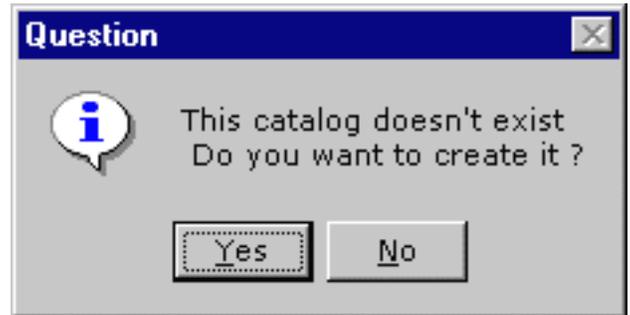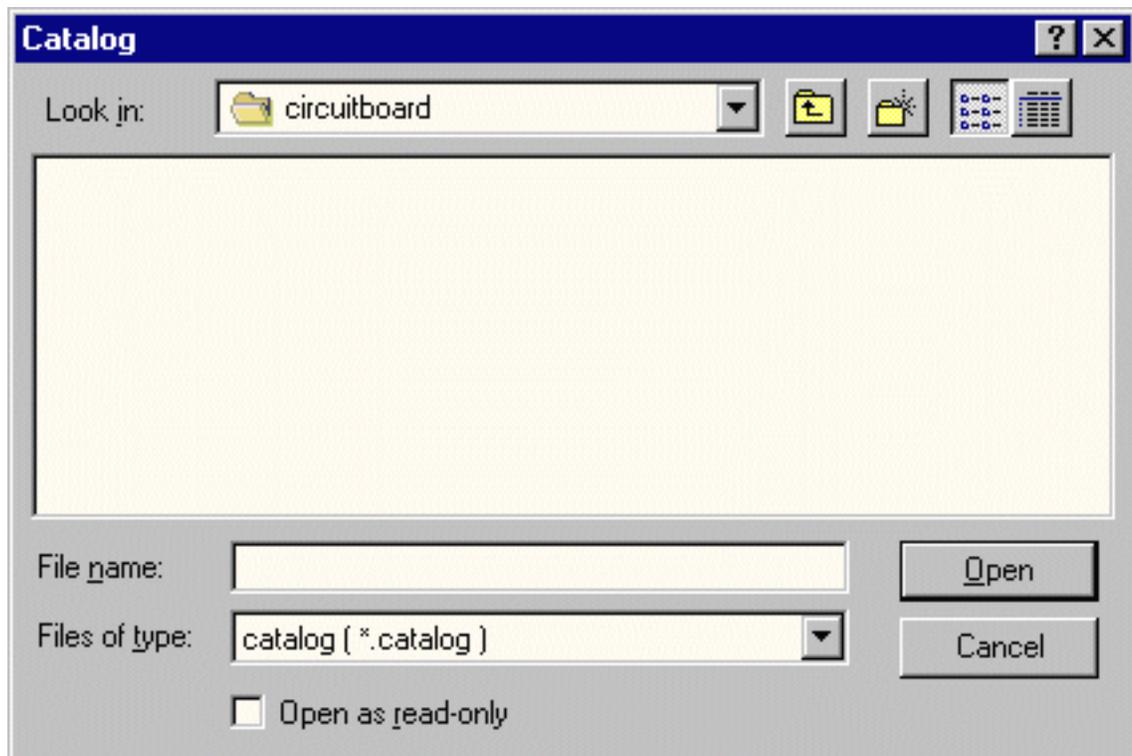The component is selected.

1. Select the Export a Component icon

.

The catalog doesn't exist, you are prompted to create it.

2. Click Yes.

Question

This catalog doesn't exist
Do you want to create it ?

Yes   No

3. Enter a name for the catalog.

Catalog

Look in: circuitboard

File name:

Files of type: catalog (*.catalog)

Open as read-only

Open

Cancel

4. Validate.

A message informs you that the catalog has been created.

Information

Your catalog has been successfully saved !

OK

Note that you can store several components at the same time in the catalog.

Up   Catalog Creation   Import with Catalog

n⊕w Import with Catalog

This task explains how to import data, where the component 3D geometry is imported from a catalog.

1. Open the Import.idf document from the \online\samples\circuitboard directory.
The .idf file contains the location of the components in the circuit board assembly.

You can also use library and/or catalog to get a better geometry of the components:
- the .lib file contains the 2D footprint geometry of the components.
- the .catalog contains the 3D (exact) geometry of the components.

However, none of these files are mandatory.

You can choose between several alternatives:

- open a library only: the 3D geometry is extruded from the 2D geometry
- open a library and use an existing catalog:
  - the component exists in the catalog, its 3D geometry is imported from the catalog
  - the component does not exist in the catalog, it is extruded from the 2D geometry and stored in the catalog
- open a library and a new catalog: the 3D geometry is extruded from the 2D geometry and stored in the newly created catalog
- open no library, but a catalog:
  - the component exists in the catalog, its 3D geometry is imported from the catalog
  - the component does not exist in the catalog, it is created with no associated geometry
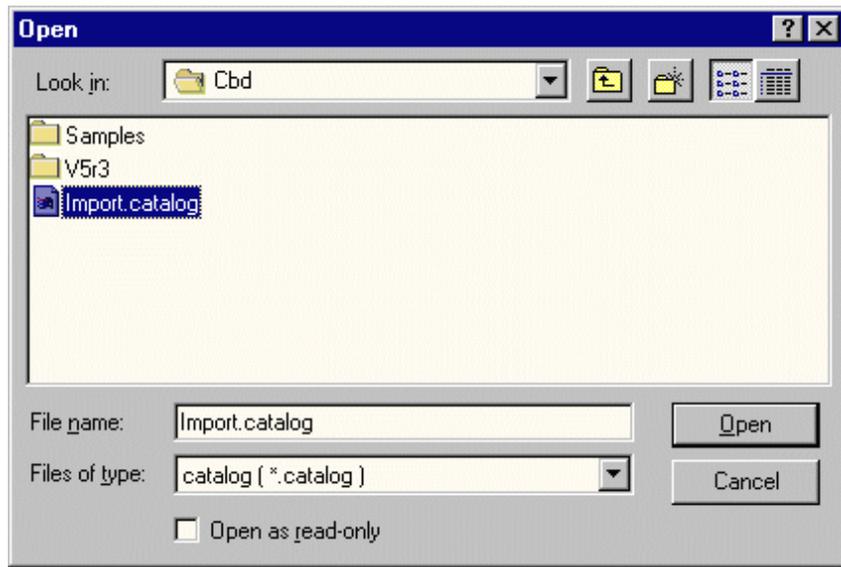- open no library and no catalog, the component has no associated geometry.

2. Select the ImportComps.lib file.

3. Check the box: Using a catalog
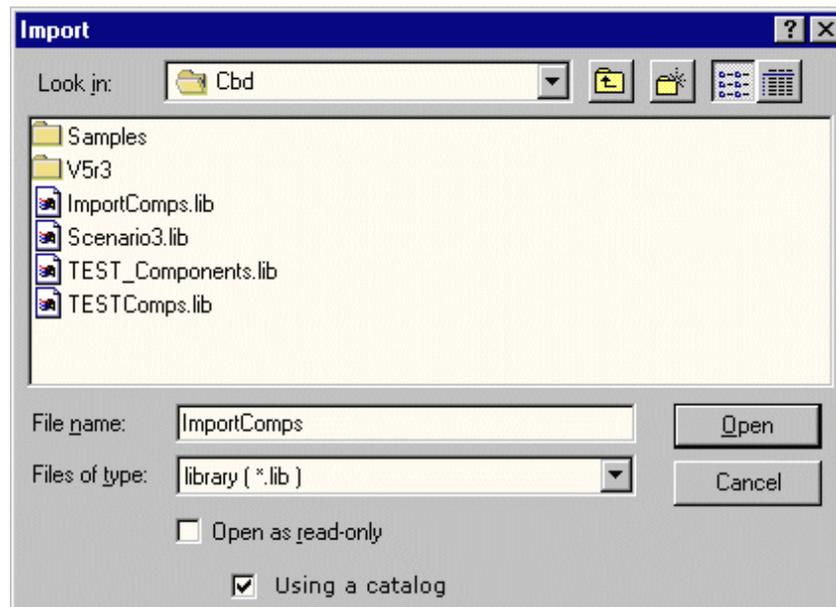
4. Click Open.

5. Select the catalog and validate.

Up   Catalog Creation   Import with Catalog

new Importing

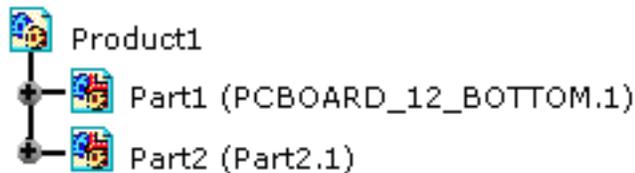See also

Importing in Context

This task explains how to import circuit board elements and place them in context.

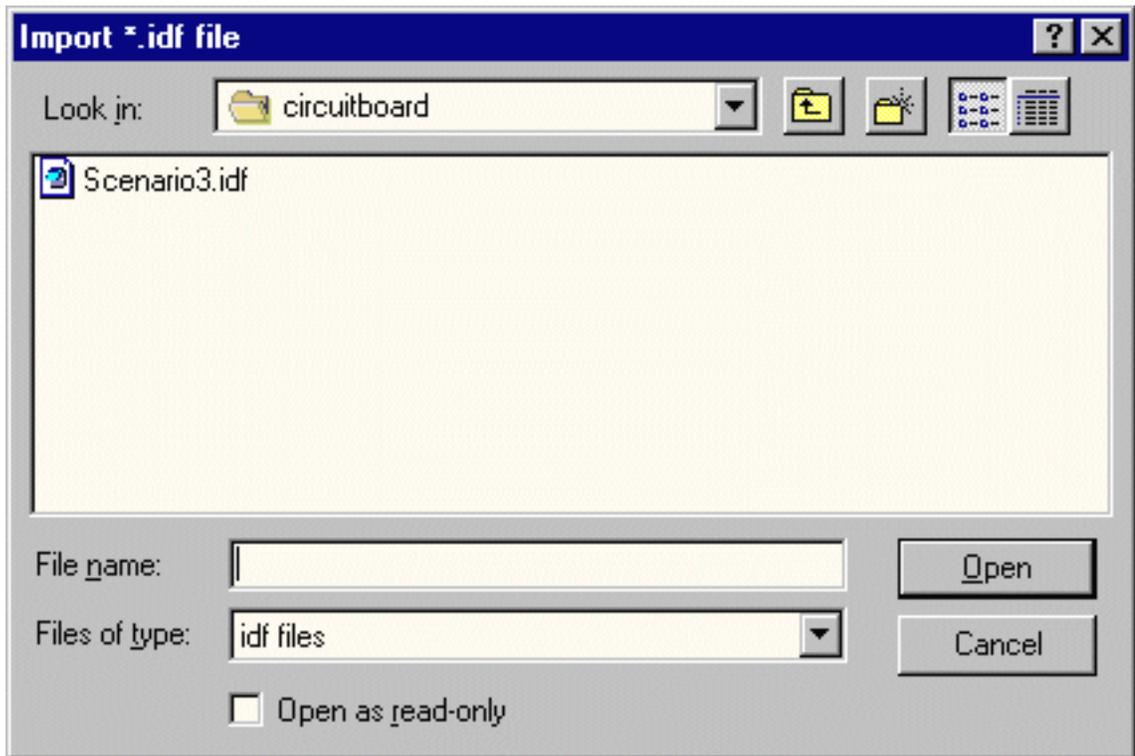Open the Context.CATProduct document from the \online\samples\circuitboard directory.

1. Enter Circuit Board Design workbench.
A new part is created.

2. Select the Import in Context icon .

3. Select the plane where you want to place the board and components, that is to say the CD bottom plane.
The Import *.idf file dialog box is displayed.

Import *.idf file

Look in: circuitboard

Scenario3.idf

File name:

Files of type: idf files

☐ Open as read-only

Open

Cancel

4. Choose Scenario3.idf file, Scenario3.lib file and click Open.
The result should look like this:
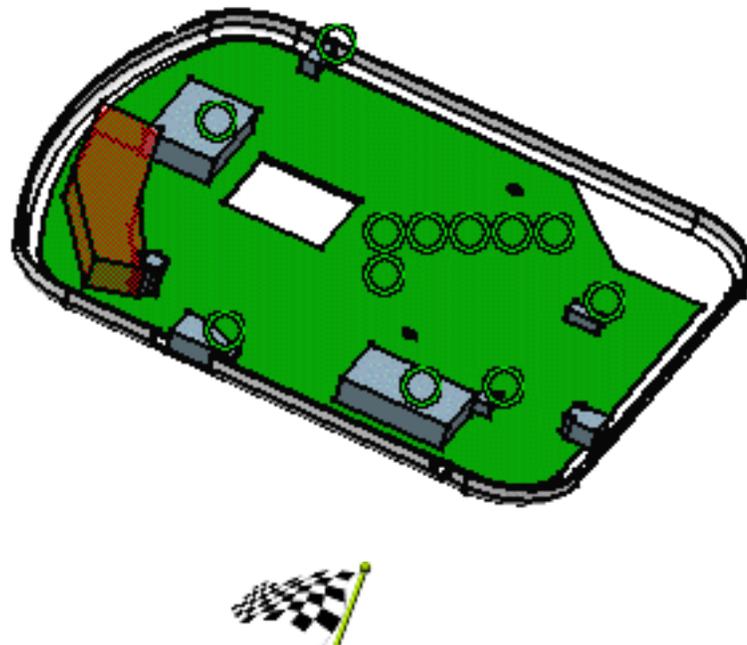
Workbench Integration

This section explains and illustrates how to integrate other workbenches
The table below lists the information you will find.

Assembly

DMU Space Analysis

Up

Creating a Board

Managing Components

Working with Catalogs

Importing

Workbench Integration

Assembly

The integration with the Assembly Design workbench lets you to move components on the board, provided you are allowed to do so by the related constraint, in the context of the full digital product definition. Refer to the Component Management section for more information.
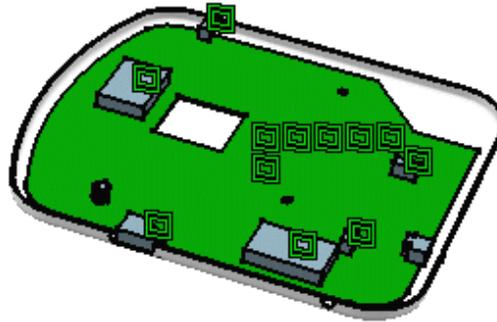
ENOVIA-DMU Space Analysis

The strong integration with ENOVIA-DMU V5 Space Analysis allows you to complete the analysis of the electronic assembly in the context of the full digital product definition.

Open the Scenario3.idf file and the corresponding Scenario3.lib file from the \online\samples\circuitboard directory.
An Assembly Product document is generated.
Then add the existing component (BOTTOM).

1. Choose the DMU Space Analysis item from the Start menu.

2. Perform a Check Clash using this icon.

The Preview window shows the clash:
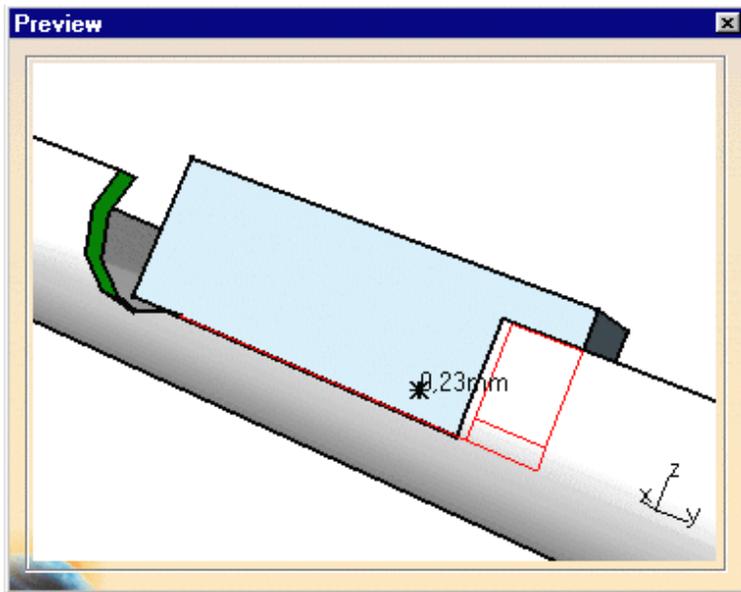
The analysis reveals a clash. You can:

- either, change the location of the component respecting the geometrical constraints
- or create a mechanical constraint area taking the clash into account; then transmit the information to the ECAD system in order to integrate it into the placing/routing algorithm.

For more information about DMU Space Analysis capabilities, refer to ENOVIA-DMU Space Analysis User's Guide.

Workbench Description

The CATIA - Circuit Board Design Version 5 application window looks like this:
Click the hotspots to display the related documentation.
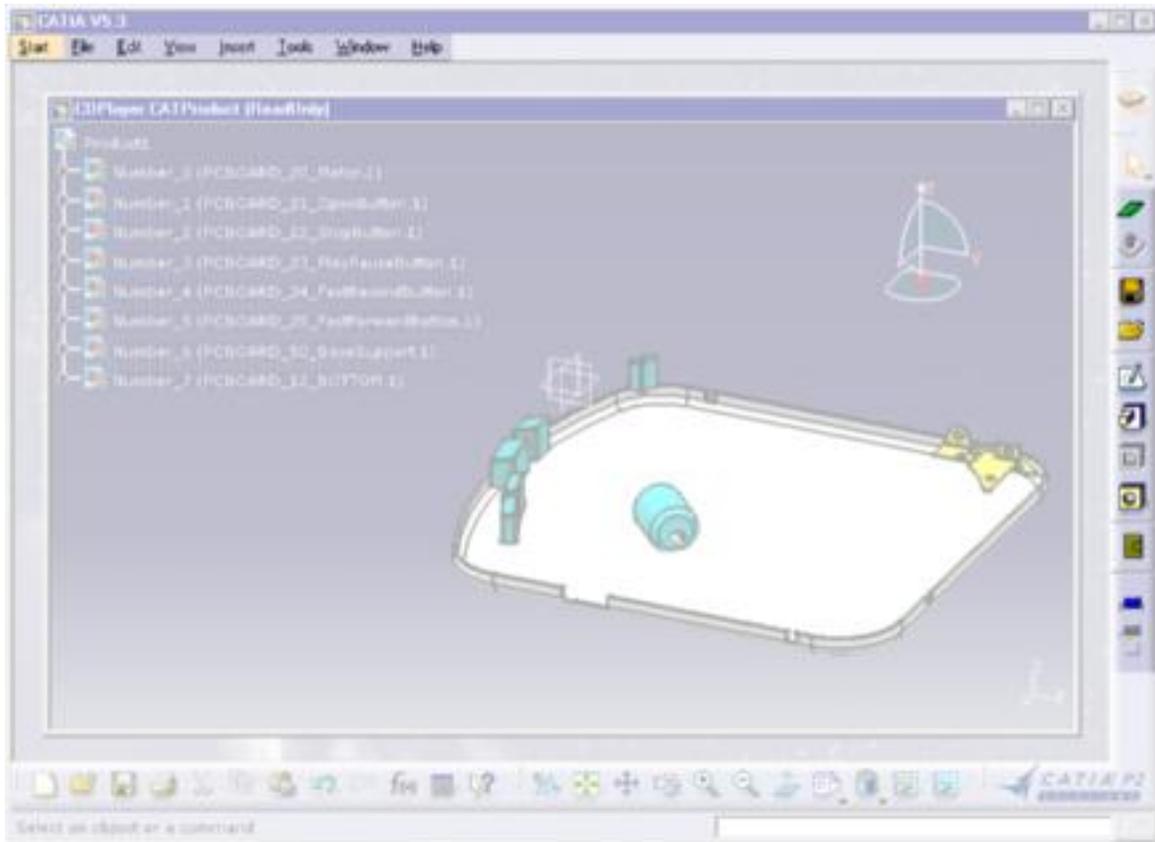
Menu Bar

Toolbar

Menu Bar

The Menu Bar and items which are available in CATIA - Circuit Board Design workbench are the standard ones. The different commands and tools are described in CATIA - Infrastructure Version 5.

For more information, refer to the CATIA Menu Bar section.

Up   Menu Bar   Toolbar

Circuit Board Toolbar

This section describes the various icons available in the Circuit Board Design workbench.
The toolbar is located on the right in the default set-up.

Circuit Board Design

See [Creating a Board](#)

See [Creating a Constraint Area](#)

See [Exporting Data in an IDF File](#)

See [Import in Context](#)

See [Sketching a Profile](#)

See [Creating a Pad](#)

See [Creating a Pocket](#)

See [Creating a Hole](#)

See [Query from Catalog](#)

See [Creating a Component](#)

See [Creating a Catalog](#)

Up   Menu Bar   Toolbar

Glossary

B

board — A pad with technological properties

BOTTOM — The technological property of a board face which makes it the copper plated face

C

component — An electronic part defined in the electronic schematic and stored in the ECAD component library

component status — Describes the position of a component on a board:
unplaced: there is no constraint between the component and the board
placed: there is a surface contact between the component and the board
fixed: there is a fixity constraint between the component and the board
Dedicated geometrical constraints are also associated with components and determine whether or not you can move them.

constraint area — A space reservation used for different purposes such as allowing/forbidding the routing or placing of components etc.
Each constraint area belongs to a part, this is to say that you need to create a new part before creating a constraint area.

cutout — A hole or a pocket on a board

D

drilled hole — A hole used to mount a board in its case

E

ECAD — Electrical Computer Aided Design

F

fixed — See component status

I

IDF — Intermediate Data Format

N

NPTH Non plated: technological property of a hole which is not conductive (mounting hole)

O

other outline Other area type defined by an outline, cutouts and a thickness

P

placed See component status

place keepout A mechanical space reservation where the component height is limited

place outline An area where you are allowed to place components

place region An area where placing is limited to component of certain types

PTH Plated: technological property of a hole which is conductive

R

route keepout An area where routing of electrical connections is forbidden

route outline An area where routing of electrical connections is allowed

S

status See component status

T

TOP The technological property of a board face which means it is not the copper plated face

U

unplaced See component status

V

via keepout An area where placing via is forbidden