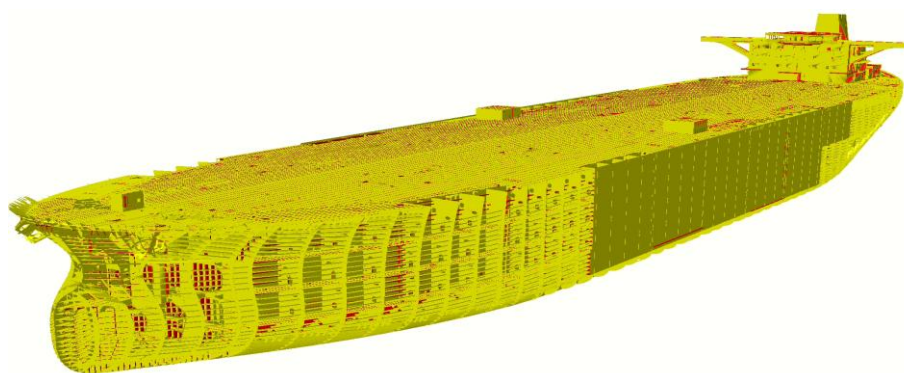


3D Tribon Importer (CT9)



User Guide

BPA Delivery 8 for V5R20 (V5.8)

Modification Tracking

Version	Date	Done by	Modification
V1	08.05.2008	OJG	Created
V2	10.01.2008	OJG	Modified
V3	05.19.2009	OJG/ZVW	Modified
V4	17.02.2010	ZVW	Modified
V5	21.05.2010	ZVW	Modified
V5	24.11.2010	ZVW	Modified
V5	07.02.2011	ZVW	Modified

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1. Introduction

This document describes the user guide for the BPA 3D Tribon Importer.

1. Related Documentation

Document acronym	Content
[DOC1]	ex: CATIA MD user guide

Table 1- Related documentation

2. Definitions

Glossary

Name	Definition
BPA	Business Process Accelerator

Table 2 – Glossary

Variables

Name	Definition
<BPA Install Directory>	C:\Program Files\Dassault Systemes\3D Tribon Interface
<CATIA Install Directory>	D:\Program Files\Dassault Systemes\B20

Table 3 - Variables

3. Prerequisite for 3D Tribon Importer

Hardware requirements

PC with Windows XP (same configurations than for MD2 CATIA V5 package)

Software requirements

 **Code Level Minimum:**

 CATIA V5R20SP5

DS Product Licensing

 MD2 or any configuration including it.

2. 3D Tribon Importer overview



This function is available in the Assembly Design and Product Structure workbench.

Icon	Function	Description
	To launch UI of CT9	Translates Tribon Data using XML files as per user requirement.

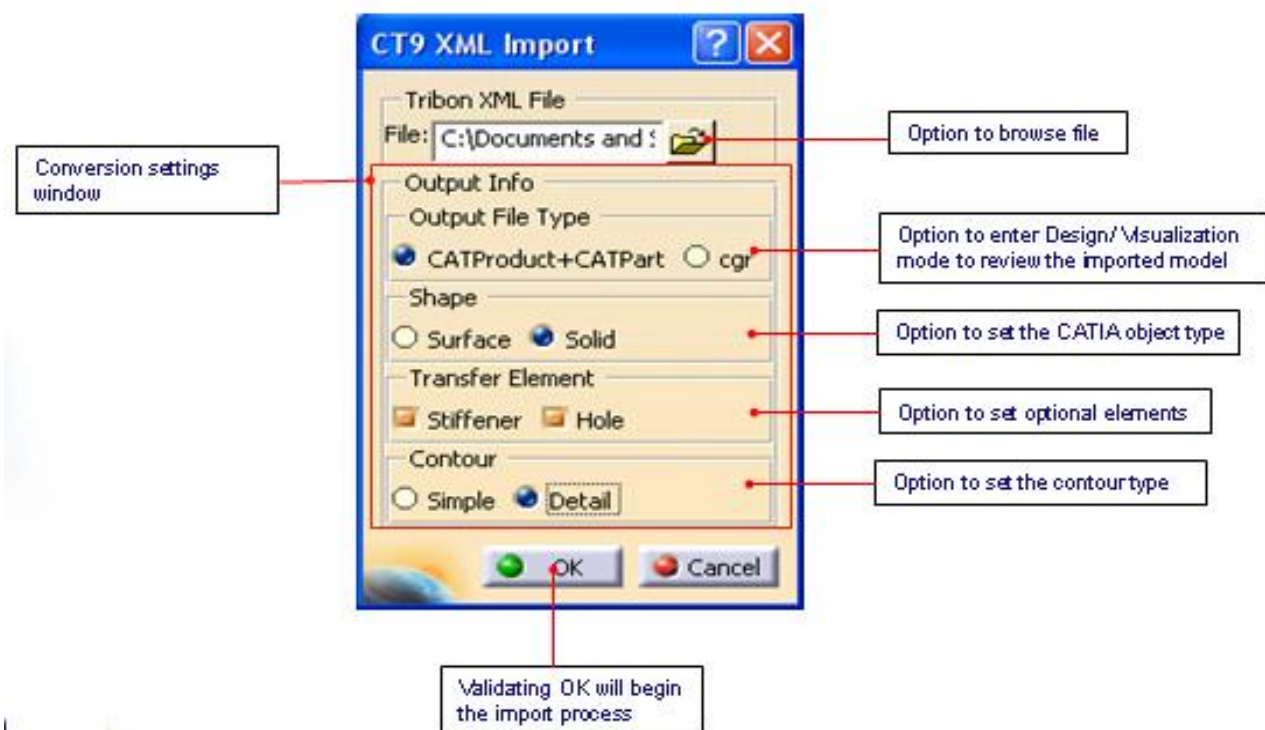
1) Tribon Interface:

Overview :

- 👤 This function is used to translate the Tribon Data with options of CATProduct+CatPart, Cgr surface, solid, stiffeners, and holes.
- 👤 It generates the hull of ship along with the stiffeners, bulkheads & holes.

Activities :

- 👤 The UI which Interfaces Tribon files with CATIA has four options which can be used for the desired output.
- 👤 UI is also used to have a permutation & combination of the Eight options provided by the BPA
- The Eight options provided are:
 - 👤 The “Output File type”
 - 👤 CATProduct+CATPart
 - 👤 cgr
 - 👤 The “Shape” has two options
 - 👤 SURFACE
 - 👤 SOLID
 - 👤 The “Transfer Element” has two options
 - 👤 STIFFENER
 - 👤 HOLE
 - 👤 The “Contour” has two options
 - 👤 SIMPLE
 - 👤 DETAIL



Limitations :

- To use this function, verify that you are in Assembly Design or Product Structure Workbench.

Inputs and outputs

- It requires XML file as input in order to translate and open the part in CATIA.

Tribon I/F:



Overview :

- This Icon launches the Tribon I/F window.
- The popped up window enables you to select the desired output.

Activities: OUTPUT FILE TYPE Function

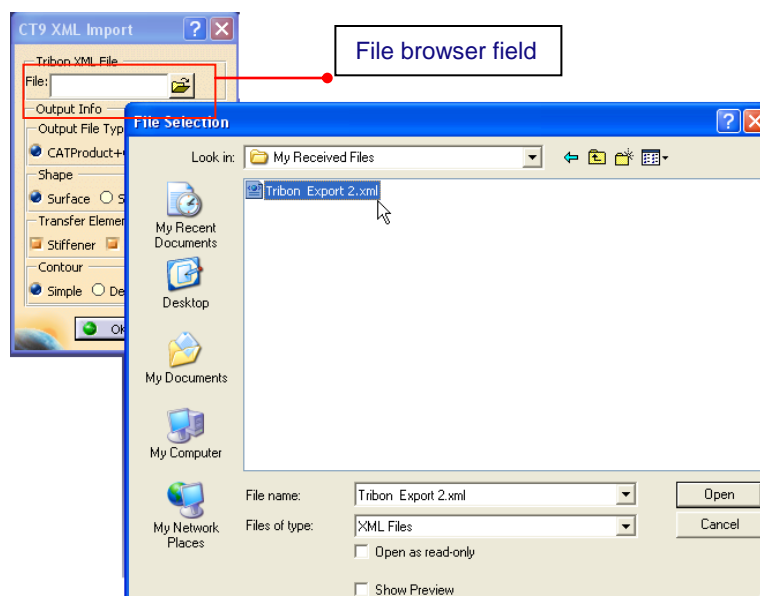
- This function allows you to select the **Output File Type**

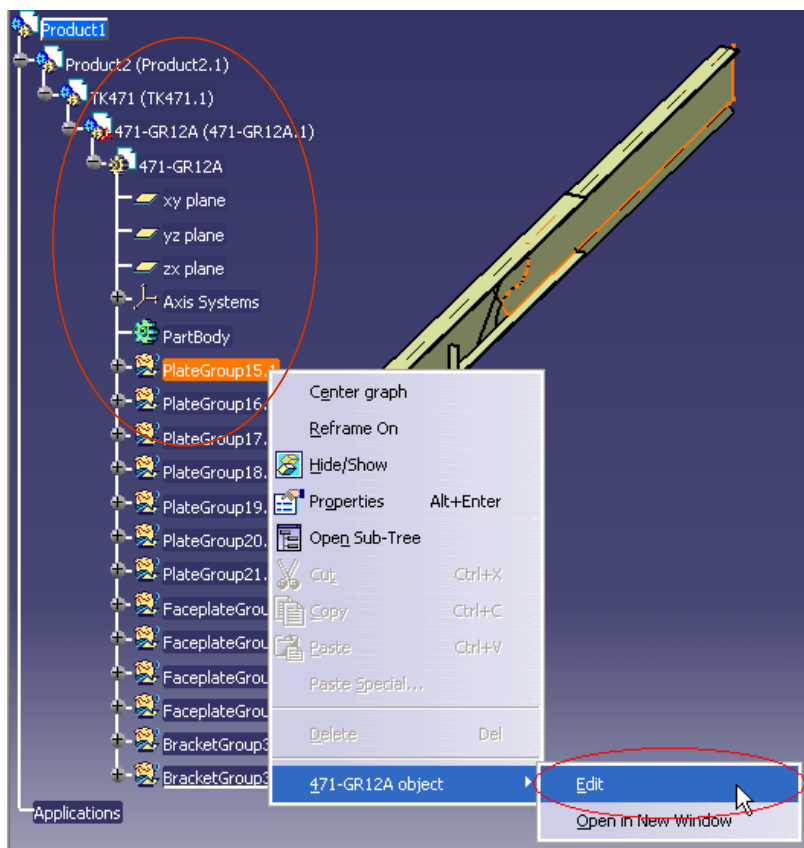
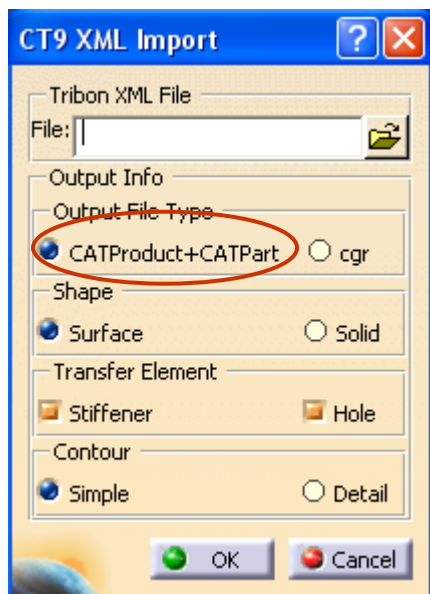
How does it work?

- There are two options in the Output File Type
 - CATProduct+CATPart**
 - cgr**
- CATProduct+CATPart allows:
 - This option allows the user to open the file as a product in CATIA along with the specification tree.*
 - This also allows the user to modify & Edit the product with the available options in CATIA*
- Cgr:
 - This allows the user to open the product, but without the specification.*
 - This option doesn't allow editing of part.*

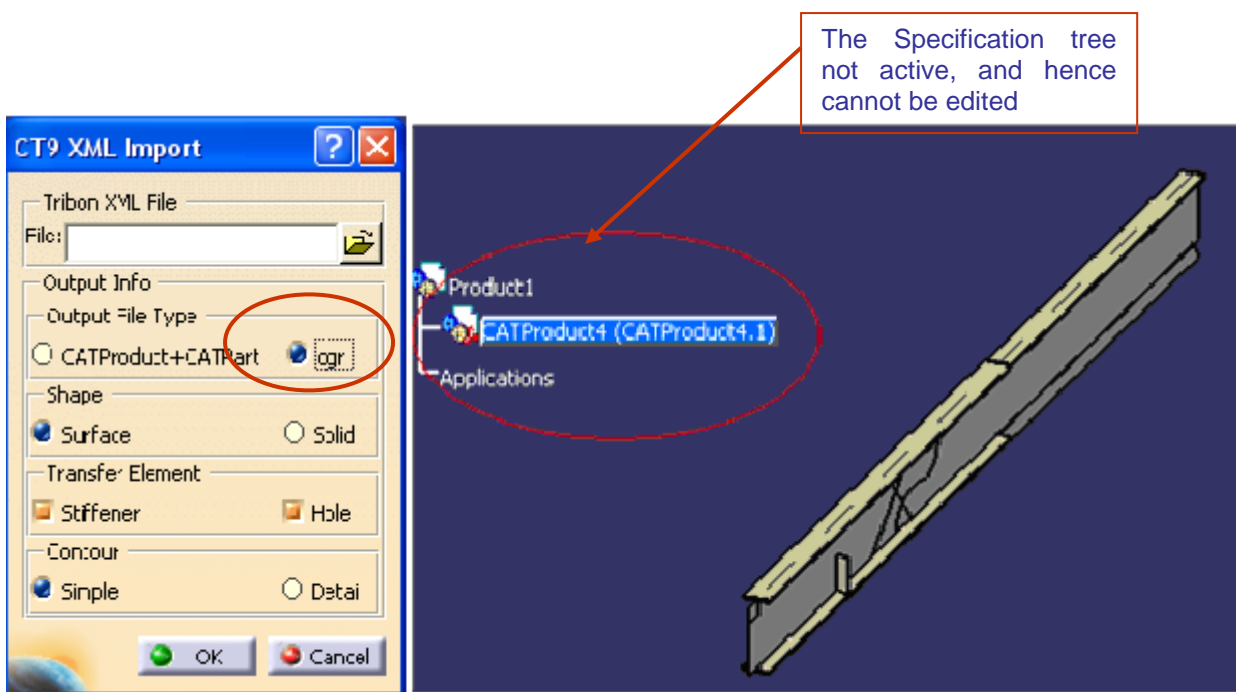
Inputs and outputs :

- The user needs to choose:
 - A XML file from the browser.
 - Any one Output File Type.





In the Specification tree the root product is Ship under which all the other sub-product in the order they are in the Xml file are getting appended in the tree and under each product there are parts which are again appended in the fashion they are in the Xml file



Activities :

- There are two functions which user has to choose one in **"Shape"** & other in **"Contour"**

How does it work?

- There are two options in the **Shape**

- Surface**

- Solid**

- Surface:

- The selection of this option allows the user to open the file as surface.

- Solid:

- This allows the user to open in the file as solid with Endcut Implementation.

- The two options in **Contour**

- Simple**

- Detail**

- Simple

- This option is used in combination with Shape options, which enhances the user to open the file with **simple** contours which do not have notches.

- Detail

- This is the second option available in the contour.

- This enables the user to open the file with **detailed** contours with the implementation of notches.

Inputs and outputs :

- The user need to choose:

- A XML file from the browser.
- Any one Output File Type.
- Any one option from the Shape options
- And one option from the Contour Field

2) Combinations with different options:

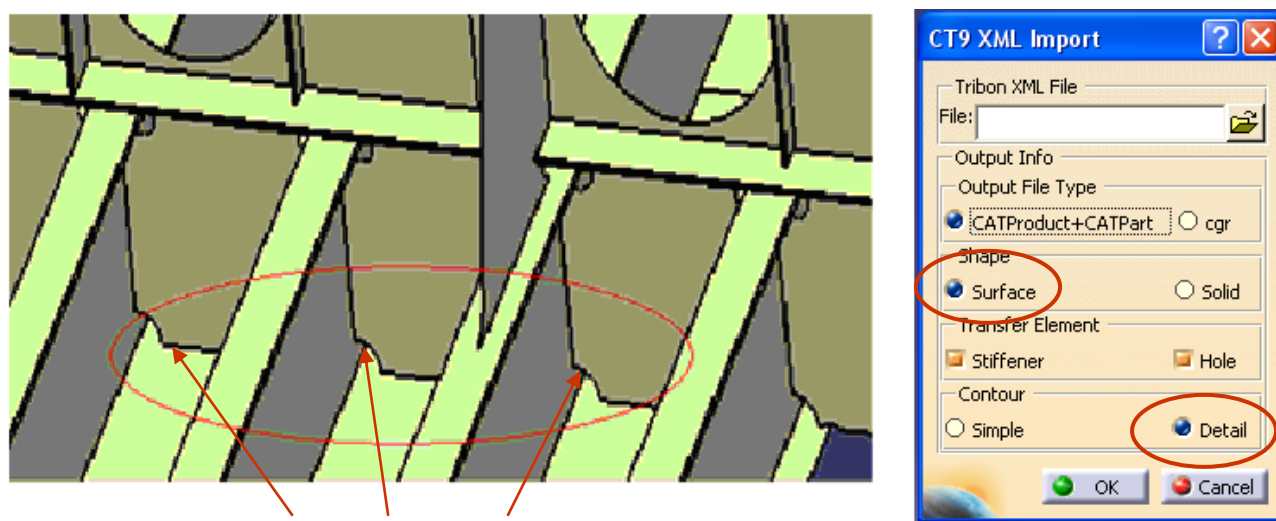
- When **surface & simple contour** is chosen result observed is as below:
 - End cuts not observed

When **surface & simple contour** is chosen result observed is as below



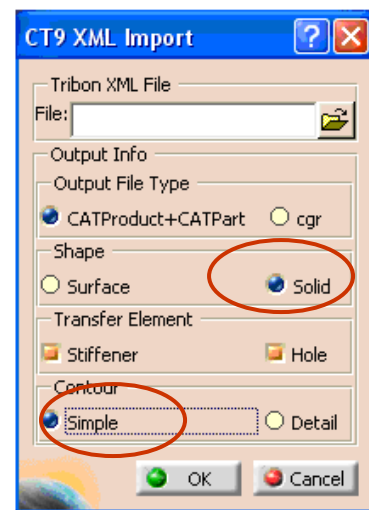
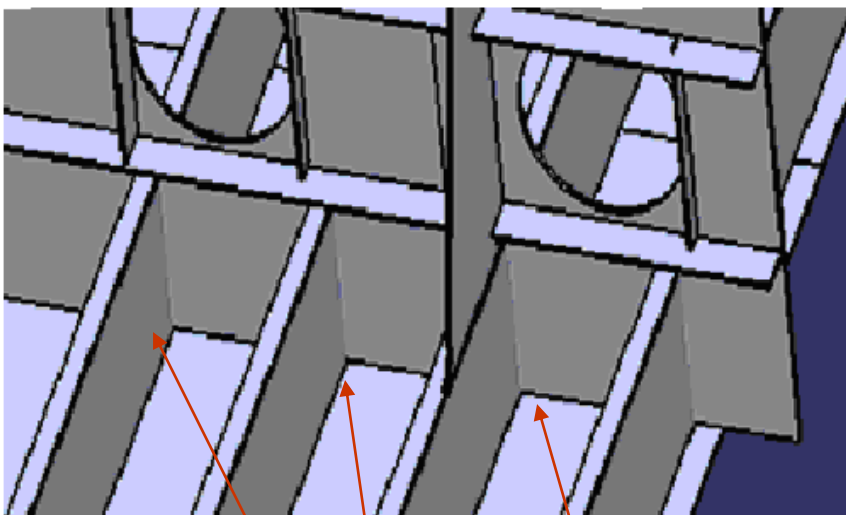
Contours and notches are not observed when **simple** option is chosen with surface

When **surface & detail contour** is chosen result observed is as below



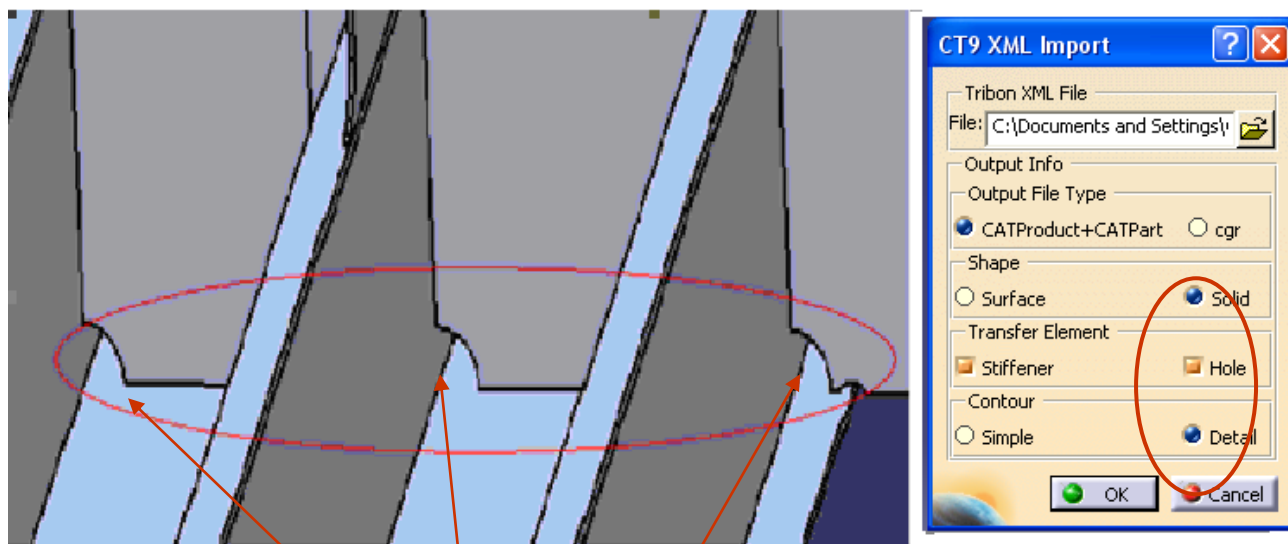
Contours and notches are observed when **detail** option is chosen with surface

When **solid & simple contour** is chosen result observed is as below



Contours and notches are not observed when **simple** option is chosen with solid

When **solid & detail contour** is chosen result observed is as below



Contours and notches are observed when detail option chosen with solid

Activities :

There are two more functions which user can choose in **Transfer Element**

How does it work?

There are two options in the **Transfer Element**

Stiffener

Hole

Stiffener:

The selection of this option opens the file with stiffeners.

And if this option is not chosen while opening the file, stiffeners will not be displayed in CATIA.

Hole:

This allows the user to open the file with holes.

When this option is not chosen holes will not be displayed in the Product.

When both the options are chosen, both the holes and stiffeners are displayed in the product when opened.

Inputs and outputs :

The user need to choose:

A XML file from the browser.

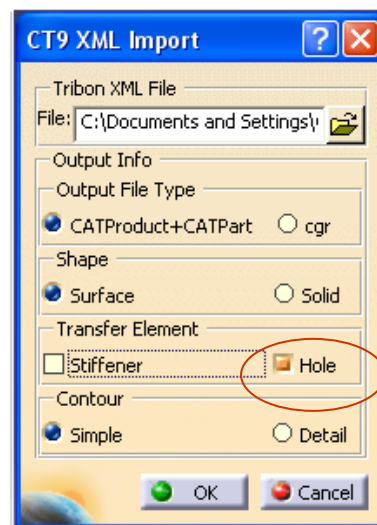
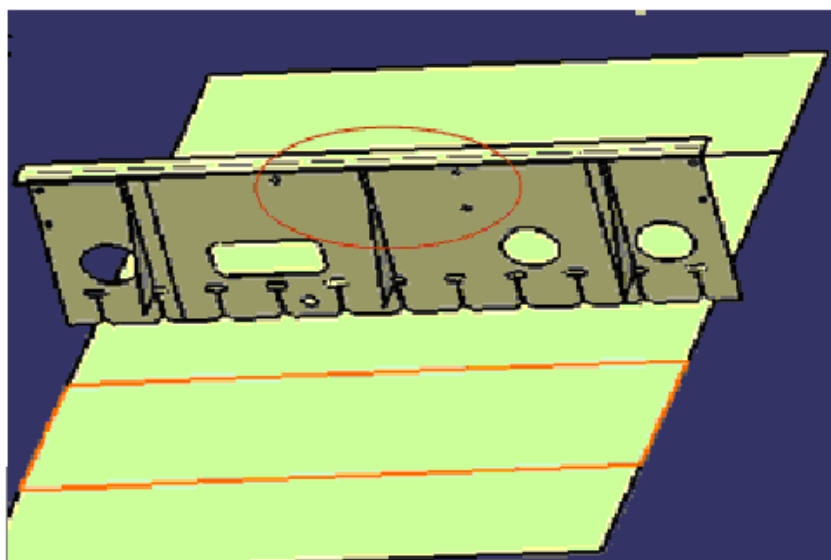
Any one Output File Type.

Any one option from the Shape options

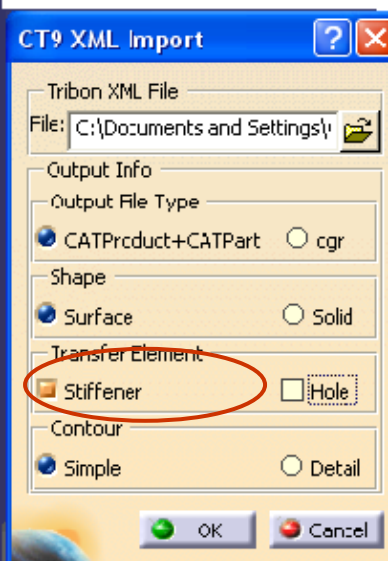
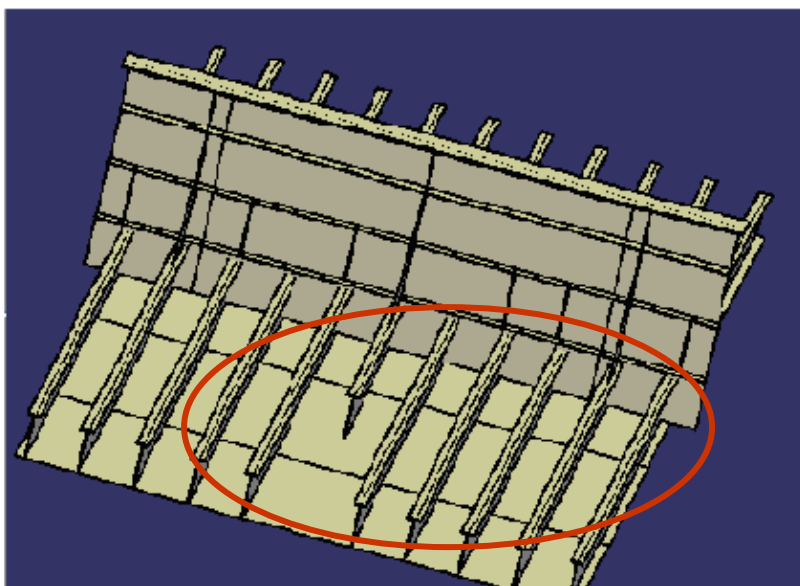
And one option from the Contour Field

Option in Transfer Element has to be chosen, both the options can be activated at a time, as per the requirement of the user.

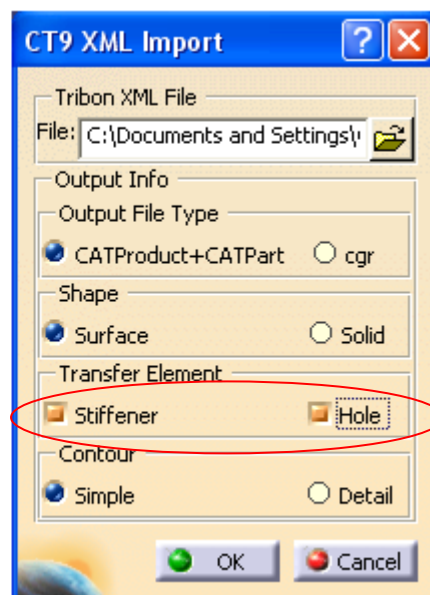
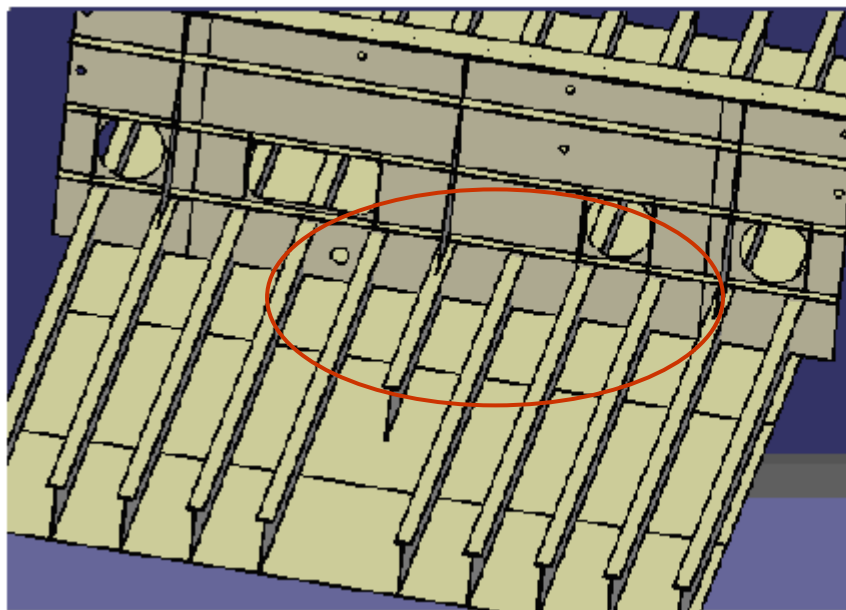
- ⤴ Part when opened with only **Hole** option.
- ⤴ This part shows only **Holes**, there are no **Stiffeners** in this product



- ⤴ Part when opened with **Stiffener** option
- ⤴ This part shows only **Stiffeners** and **Holes** are not displayed

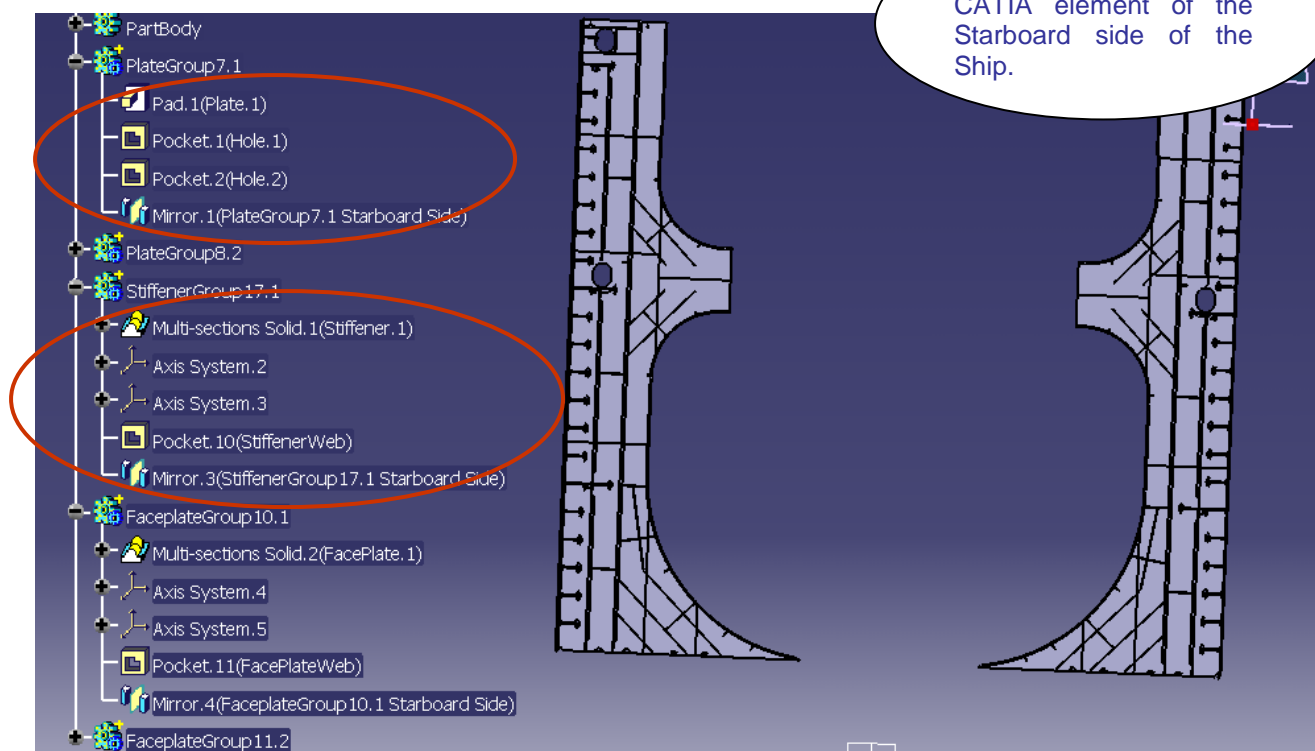
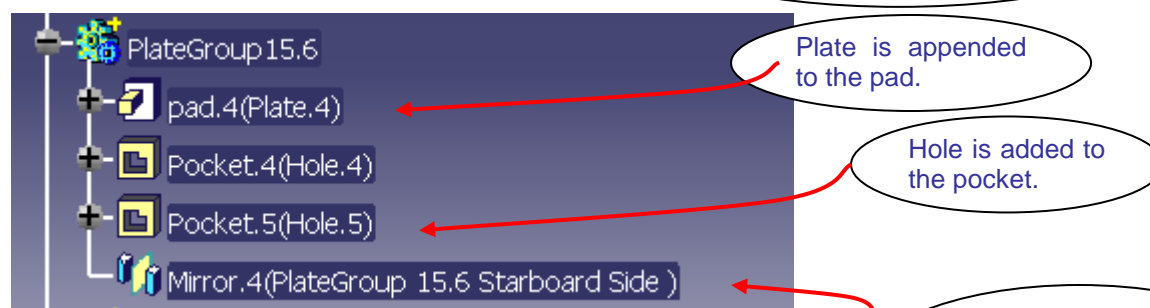
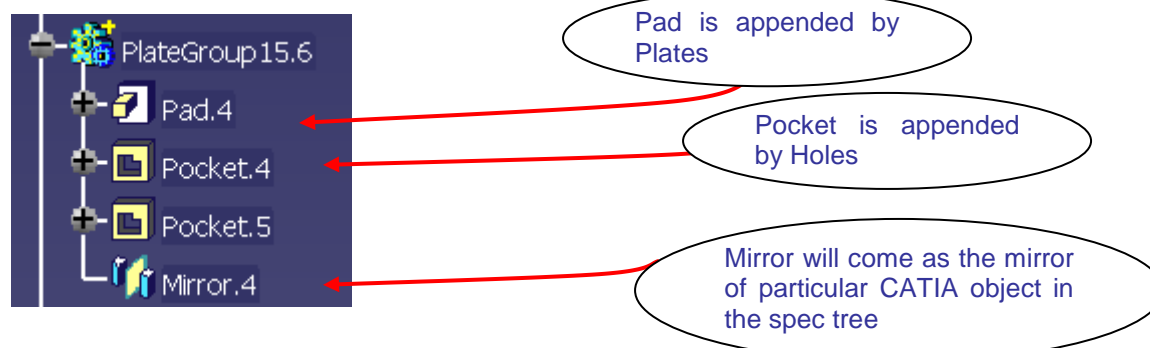


- ⤴ In this case both the options are active.
- ⤴ Hence **stiffeners** as well as **holes** are displayed in this part



3) Naming of the Elements in the CATIA Spec Tree

After the translation of all the elements, Plates are named as Pad (Plate) and Holes as Pockets (Hole) in the spec tree.



4) Limitations:

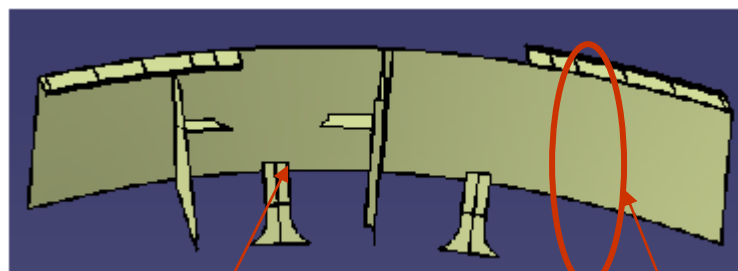
- There is limitation in Surface option as compare to Solid option which is explained below
- The thick surface will be created in the normal direction of the input surface as there is no information about the direction in the xml.

Solid:

- In solid option the all the instance i.e. both, reflected and as defined is taken in account.
- Endcut is also implemented

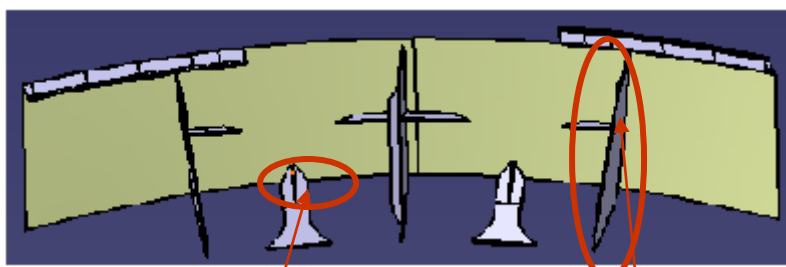
Surface:

- In Surface option instance with tag name Reflected is not taken in account were as both and as defined is taken in account.
- Endcut is not implemented.



Endcut is not implemented

Reflected is not taken in account



Endcut is implemented

Reflected is taken in account

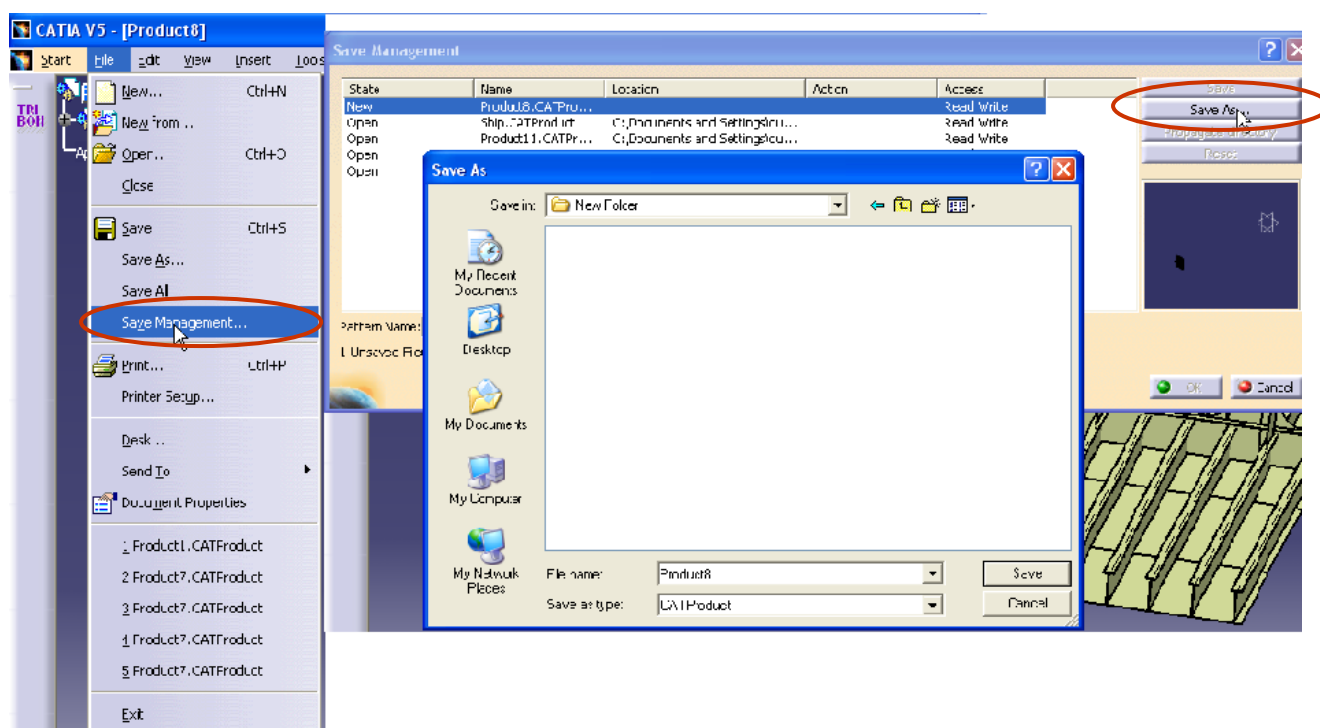
3. SAVE MANAGEMENT

Save Opened File:

- ✎ After the part is opened with the desired options, we need to save this product in CATIA.
- ✎ This enables user to make changes or modification in the product and save it in CATIA.

Inputs and outputs:

- ✎ The user needs to choose:
 - ✎ File option on the toolbar
 - ✎ Save Management
 - ✎ In the SAVE MANAGEMENT window choose a path where the file has to be saved
 - ✎ Then click on “propagate directory” in the window for all the files to be saved in that folder which are linked with the product.



4. Batch mode for CT9

Batch Mode Options

In batch mode, make a *.bat file with the following content in the specified manner.

Example:

“<CATSTART.exe path from CATIA installation folder >” – run “ISPCT9TRBT2CBatch –f “<input xml file path>” –d “<output folder path>” –s Solid –m Model –c Simple –e Stiffener –t True” –env <CT9 environment file name> -direnv “<CATEnv path>”

- CATSTART.exe <path of the CATSTART.exe from CATIA installation folder>
E.g. “...\intel_a\code\bin\CATSTART.exe”

- -run “name of batch file with all the options” (E:g ISPCT9TRBT2CBatch)
 - -f <filename>
 - -d <dirname>
 - -c Simple/ Detail
 - -e Stiffener Hole
 - -m Model/ CGR
 - -s Surface/ Solid
 - -t True/FalseBig XML Option

E.g. -run “ISPCT9TRBT2CBatch -f “...\Ship.xml” -d “...\Output” -s Solid -m CGR -c Simple -e Stiffener”

- -env <name of environment file>

E.g. “-env CT9_Env”

- -direnv <CATEnv path of the CATIA installation>

E.g. –direnv “...\Documents and Settings\User\Application Data\Dassault Systemes\CATEnv”

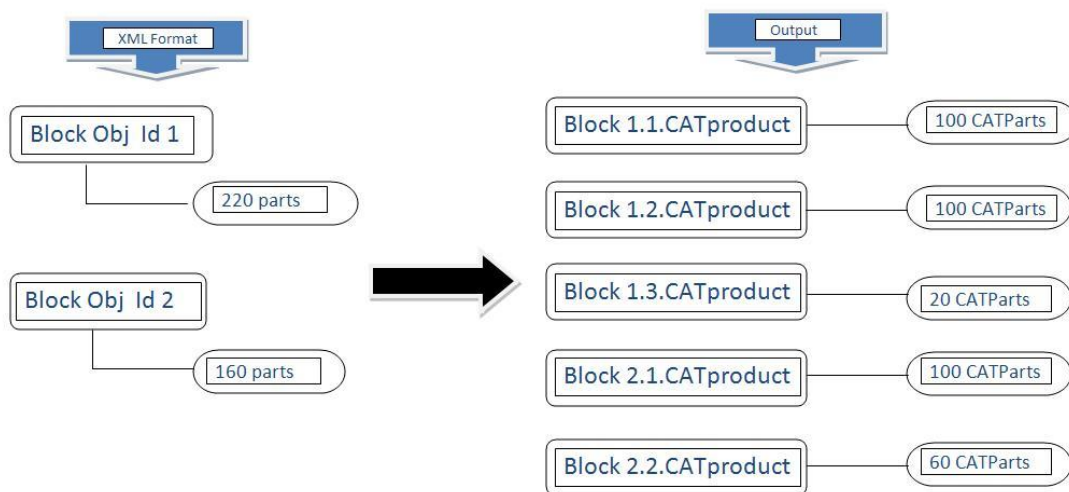
If the size of Tribon xml is big, then CATIA crashes while translating it because of memory shortage.

So, we are providing an extra option in batch mode to translate the big xml. This will create one root product (Ship.CATProduct) and child products for each 100 CATParts as explained below.

Big XML Option

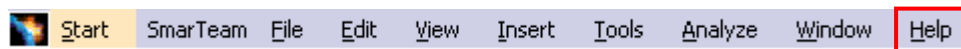
Exception: “This option is not available for cgr mode”

- -t True / False
- -t False - creation will be done in old way.
- -t True - xml is treated as a big xml and creation will be done as shown below.

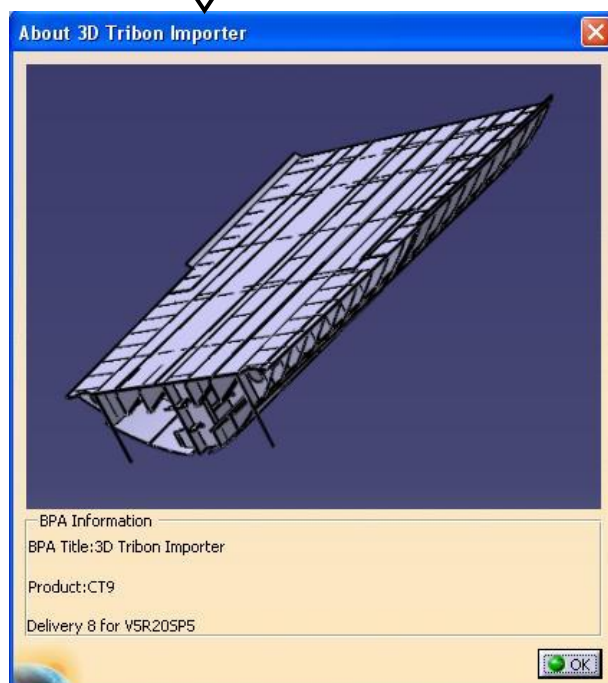


5. Help About – 3D CATIA Tribon Importer

1. Click on Help in the Menu bar.



2. Now click on About 3D Tribon Importer.



6. *New in 3D Tribon Importer (CT9)* ***– v5.8***

- FacetSurface is converted into solid data with 'Solid' as the shape option in the command.

Appendix B - Keyboard shortcuts


Function name	Icon	ShortCut
3D Tribon Importer		CTRL + T

Table 4 –keyboard shortcuts

Example:

