Golden Gate 1.0

a bridge for instant CAD collaborations

User's Guide

Photo by Rich Niewiroski Jr.

Web : http://cao.etudes.ecp.fr/index.php?page=golden_gate.htm

Contacts : Pascal Morenton - pascal.morenton@ecp.fr
Thomas Paviot - thomas.paviot@supmeca.fr

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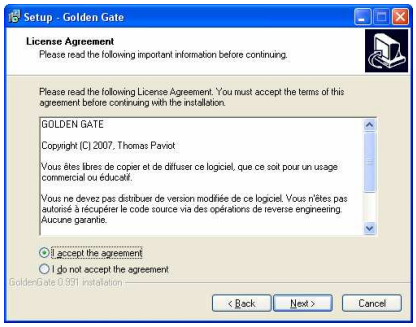
1 Installing Golden Gate

Launch the GoldenGate-1.0.exe setup program then choose your language :

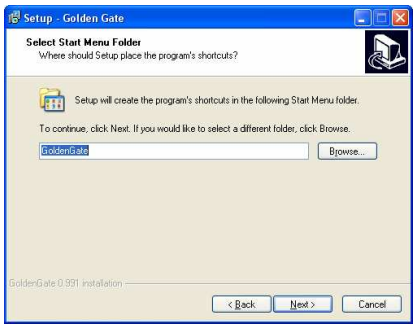
The following windows appears :

NEXT brings you to the next step :



Please, read the terms of the licence and choose "***I accept the agreement***". ***NEXT*** brings your to the next step :

Browse and choose the destination folder used for the install of Golden Gate. ***NEXT*** brings your to the next step :



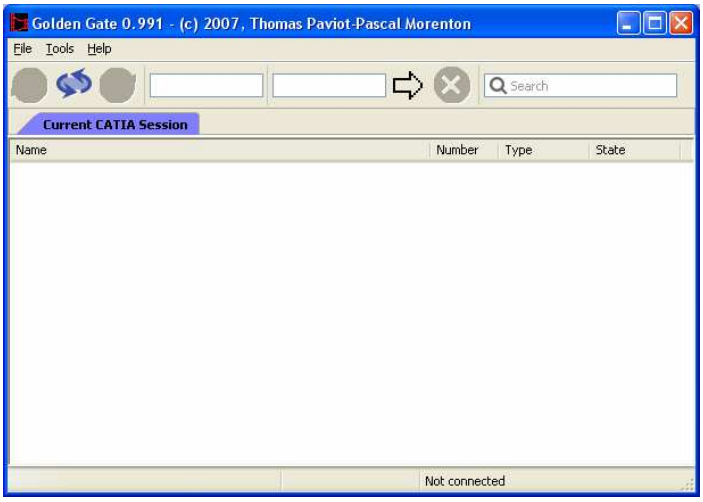
Select start menu folder. ***NEXT*** brings your to the next step :

Choose the creation or not of a shortcut on your desktop. ***NEXT*** brings your to the next step :

Setup is now ready to install. Click ***Install*** to proceed.

2 Launching and configuring Golden Gate

Start Golden Gate. The main windows will appear :



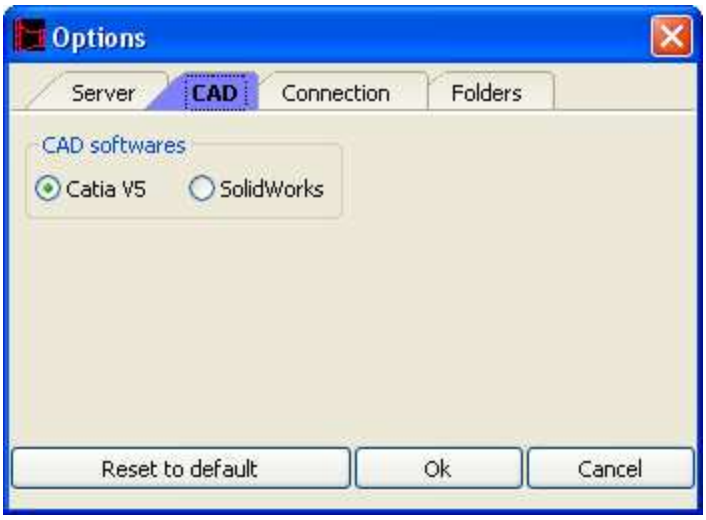
First, you have to configure Golden Gate with the command **Tools** / **Options**. If you change the previous configuration, you have to stop and restart Golden Gate.

2.1 Configuring the server

Select the PDM (product data management) system you want to use with Golden Gate : Windchill from PTC or Avitium from LASCOM. Then define the server which will be used for the CAD collaborations. You can fill in text field with the server name or IP adress. For example : windchill.somewhere.com or 255.255.255.255

Check the box ***Use HTTPS*** if you want to use such a protocol.

2.2 Configuring your CAD system

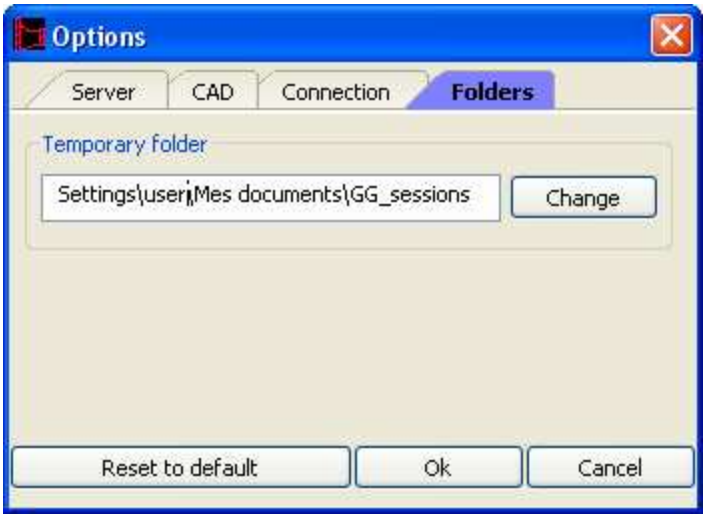


In the next tab “CAD”, select your CAD system which will be used to interact with the PDM system. You can select CATIA V5 or SOLIDWORKS. The releases supported by Golden Gate are : CATIA V5 R15 and following, SOLIDWORKS 2006 and following.

2.3 Configuring a HTTP proxy

If a http proxy is necessary for browsing the internet, declare the host and the port used for your proxy in this tab.

2.4 Configuring the temporary folder used by Golden Gate



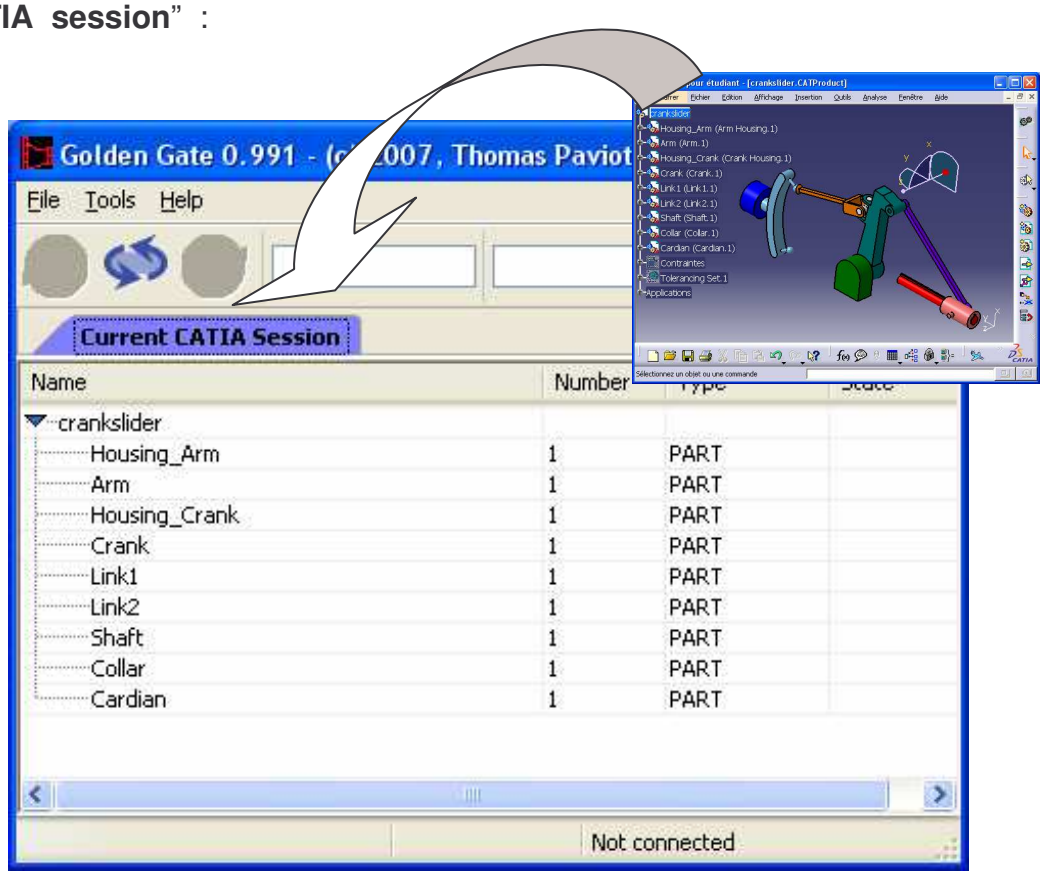
By default the temporary folder used by Golden Gate is :

C:\Documents and Settings\user\Mes documents\GG_sessions

But you can change the location of this folder by using this tab.

3 Parsing the current CAD session

First, you can parse your current CAD session in Golden Gate with the tab “**Current CATIA session**” :

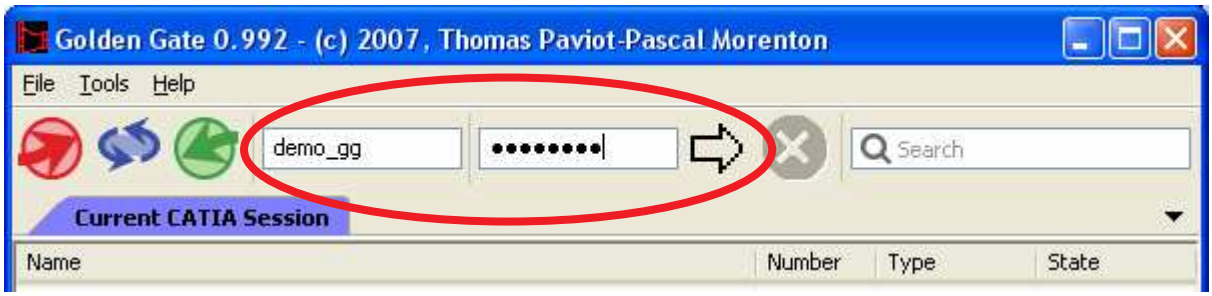


You can update the current CAD session by using this command :

Update CAD session

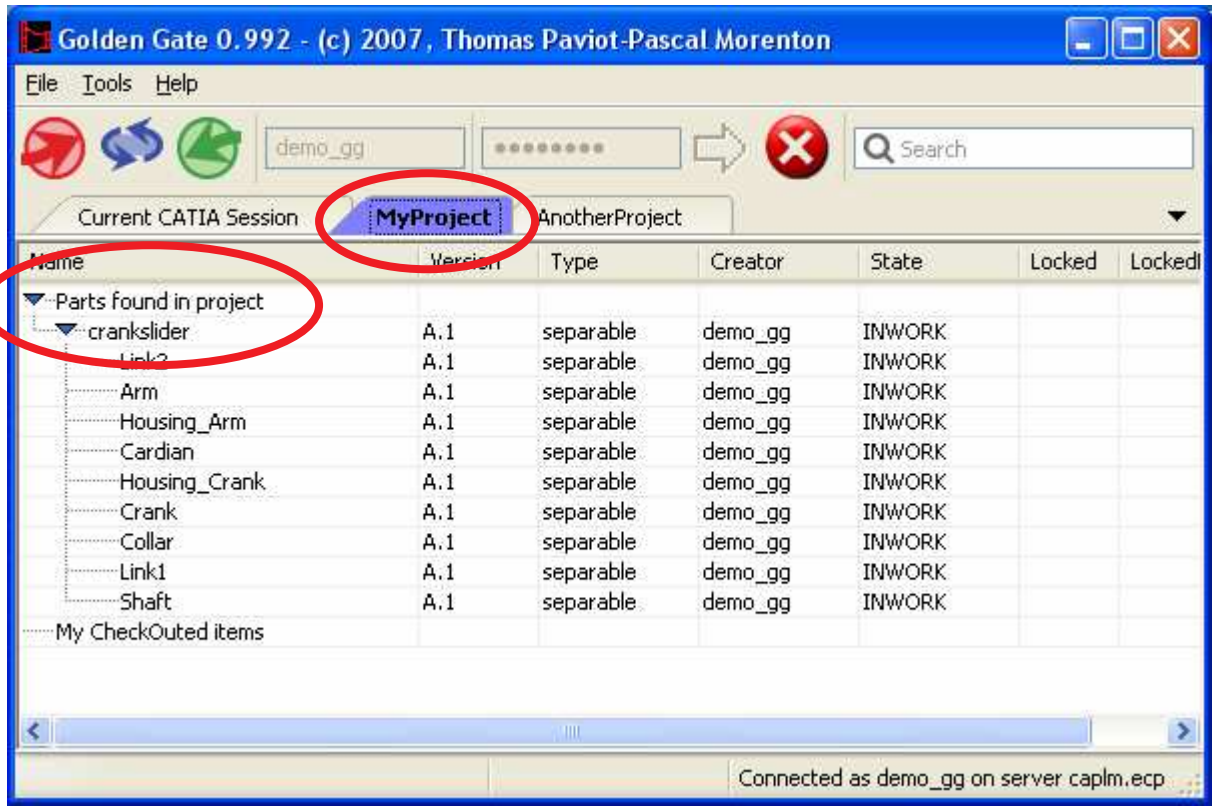
4 Connecting or disconnecting to your PDM system

If Golden Gate is not configured for your PDM system, see the section ***Configuring the server*** . Connect to your PDM system with your login / password :

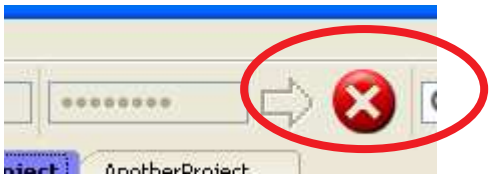


First, Golden Gate will find all your projects and will display all the Bill Of Materials (BOM) which are already stored in the workspaces of these projects :

This operation may take a while if you are member of many projects. When the search is finished, Golden Gate displays the content of the fist project :

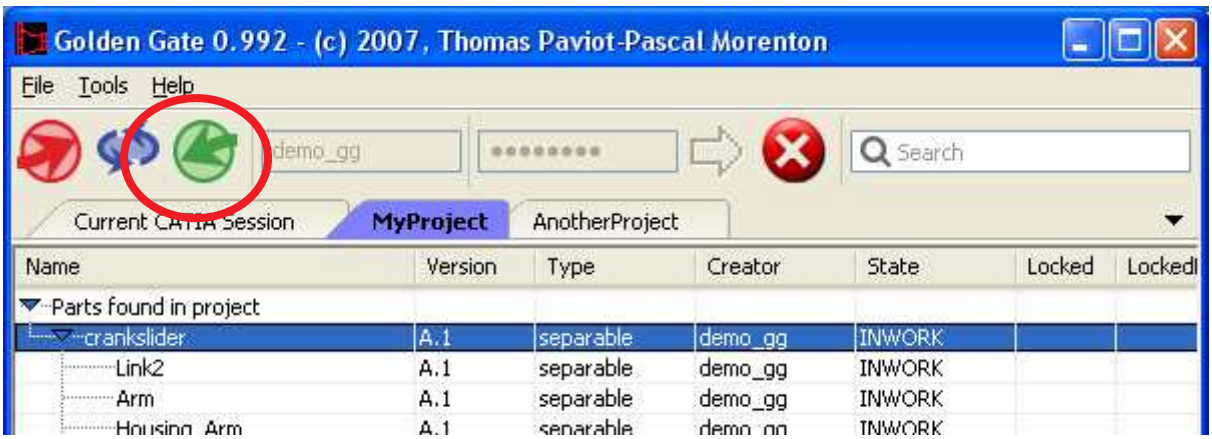


For disconnecting, use the white cross :

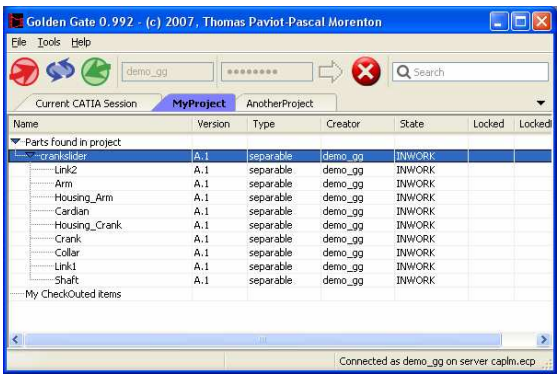


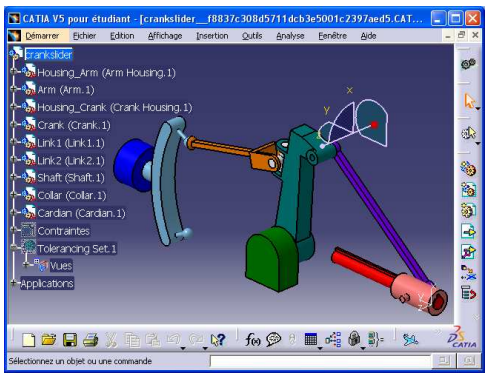
5 Downloading a model from your PDM system

Select a product or a part in the tree displayed in your project tab. Then click on the green arrow :



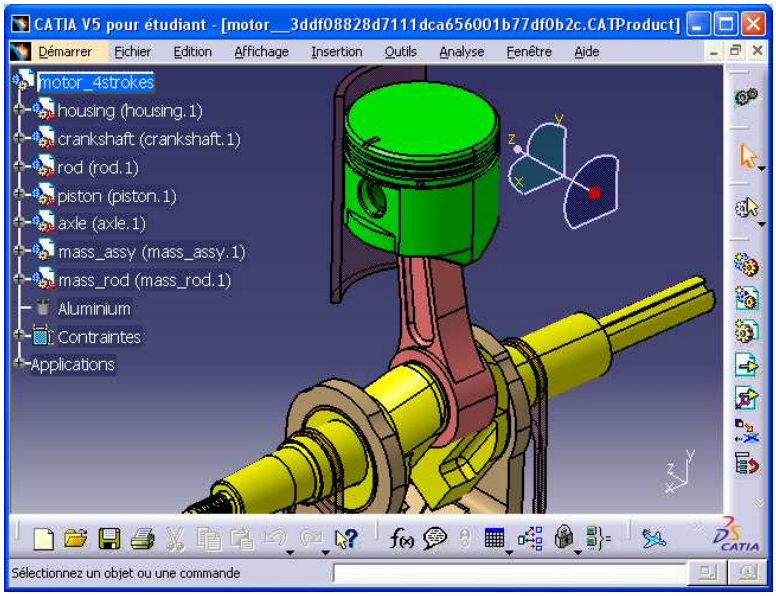
The model will be load on your current CAD session automatically :



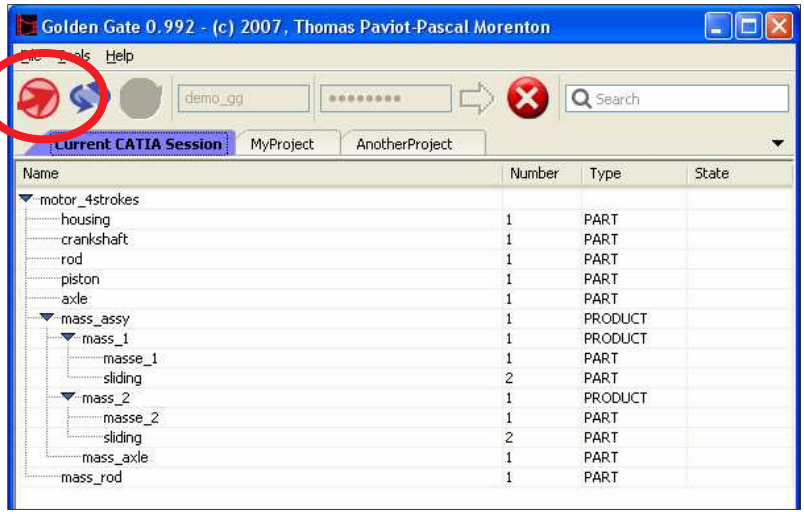


6 Uploading a model from your CAD session

Load a CATPart or a CATProduct in your CAD session :

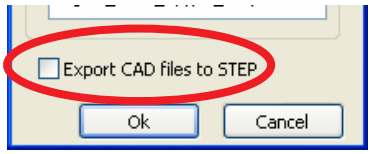


In Golden Gate, refresh the tab Current CAD session to be sure that the current model found by Golden Gate is the good one. Then click on the red arrow :



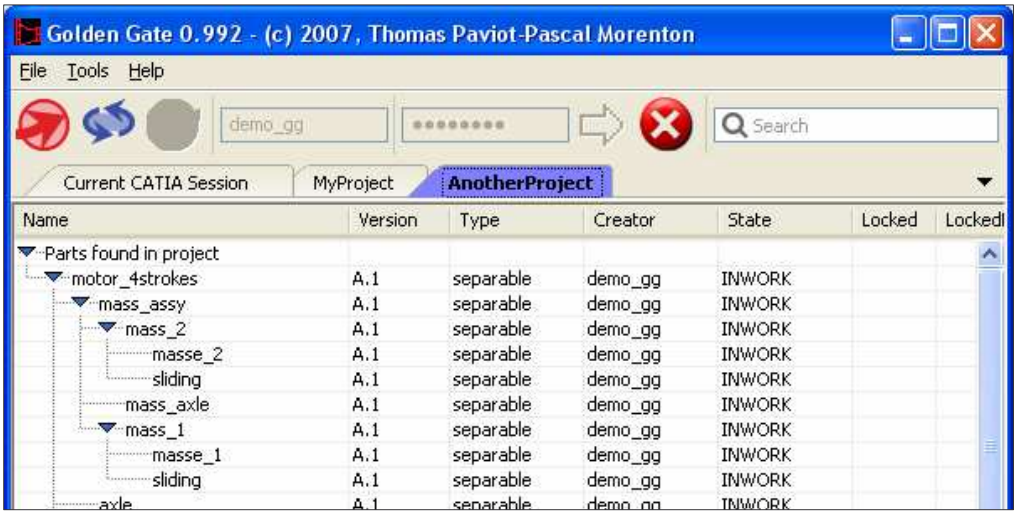
Select the project and the folder where the data will be saved in your PDM system :

You can also export the data in a STEP format for make the collaboration easier with your partners..



Click on OK and the export will begin … first Golden Gate parses the product structure of your CAD model, then the upload will begin :

At the end of the upload, the tab is refreshed :



And the following data are now stored in your PDM system :

- the product structure of your CAD model : for each part or product in your model, a part is created in the PDM system with the hierarchical links.
- the CAD files of your model
- the 3DXML views of each part of product
- if you asked for, the STEP files of each part and product of your model

Within WINDCHILL, the result is in ProjectLink :

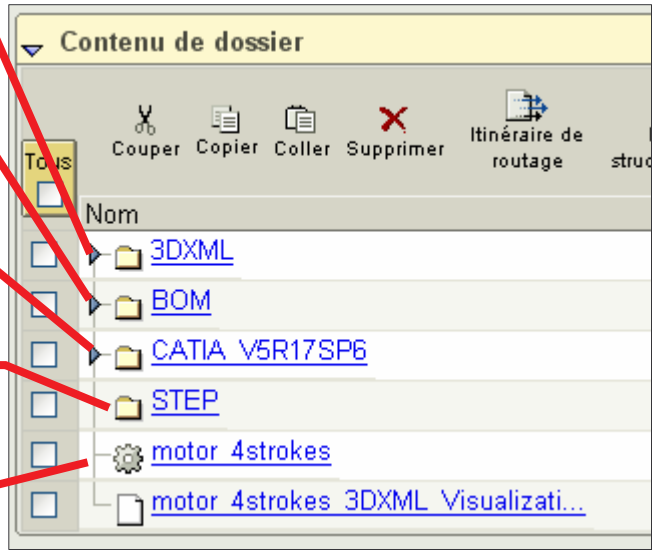
3D XML views of the model

Bill of materials

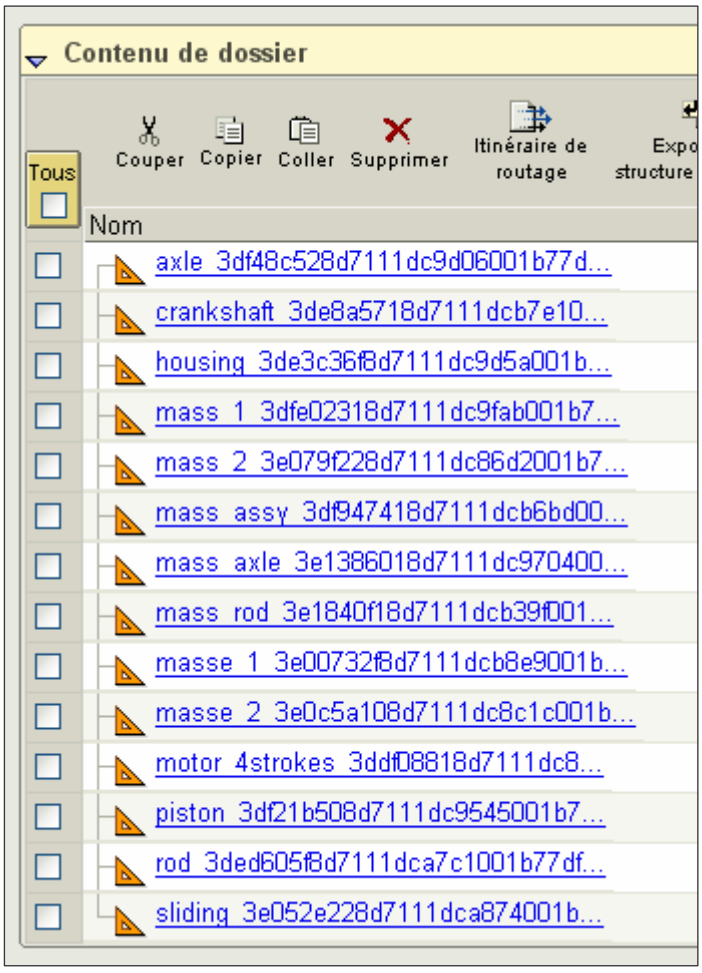
CAD Files of the model

Result of the translation of the model into a STEP format

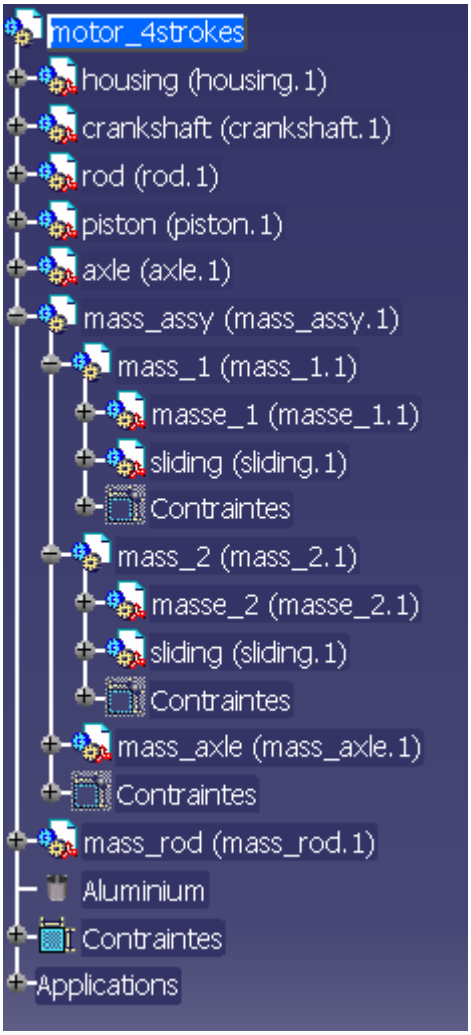
The roots of the assembly and its 3DXML views are there to be found easier

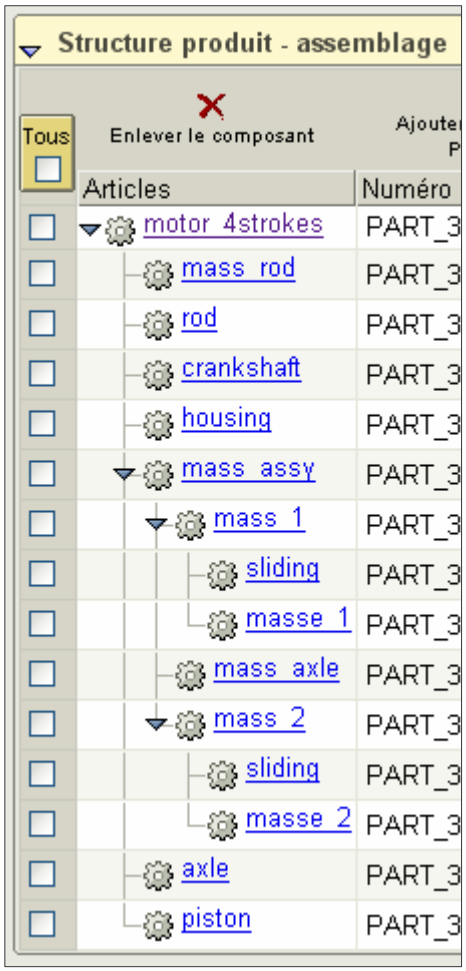


For example, the contents of the CATIA folder is :

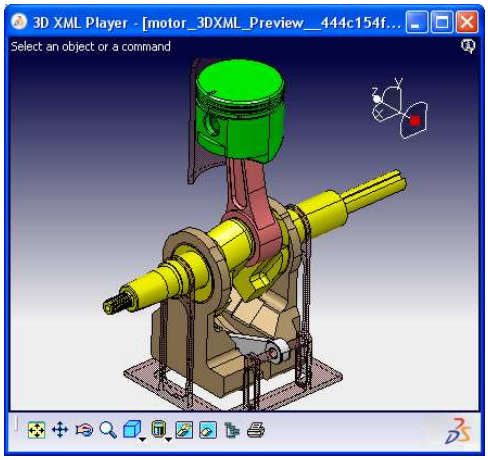


Golden Gate has created the product structure into your PDM system as a mirror of the CAD model structure found in your CAD session :





To see the 3DXML visualization, follow the link and the 3DXML view will be invoked automatically by your web browser :



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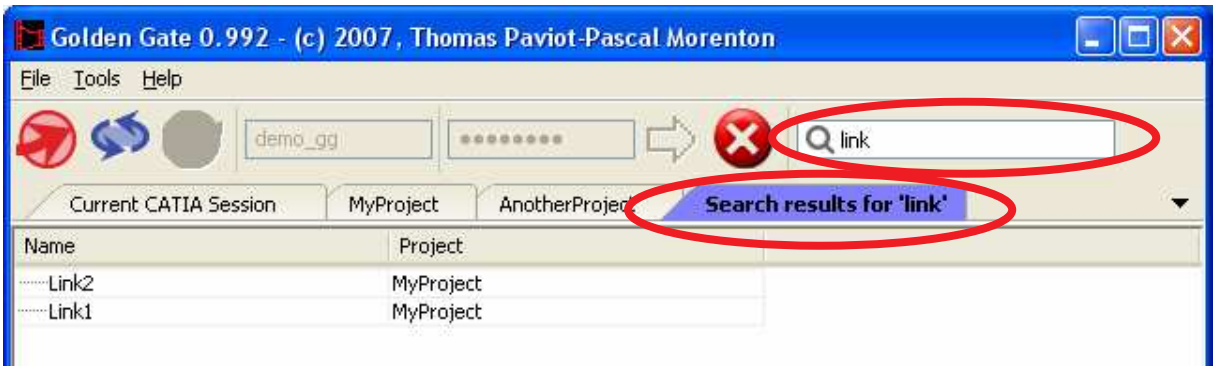
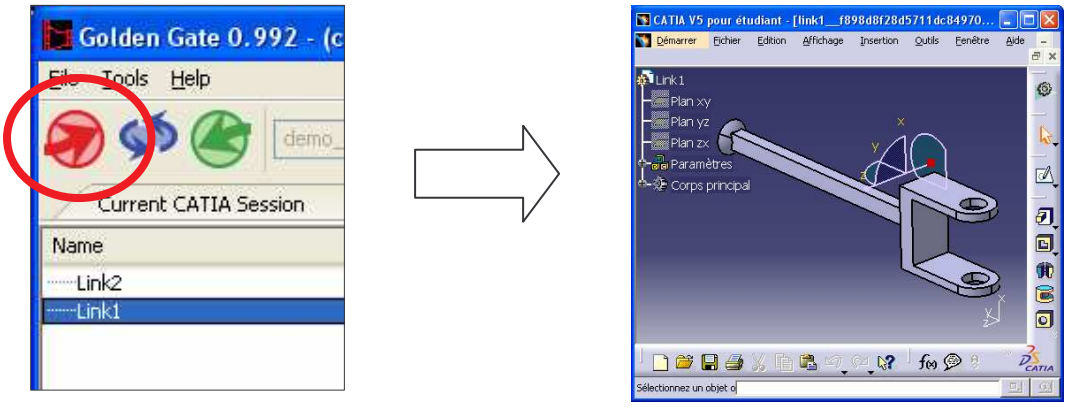
7 Searching and opening a part from your PDM system

You can search for a part or a product with the “search” field in Golden Gate. The results are displayed on a new tab “Search result for …” :

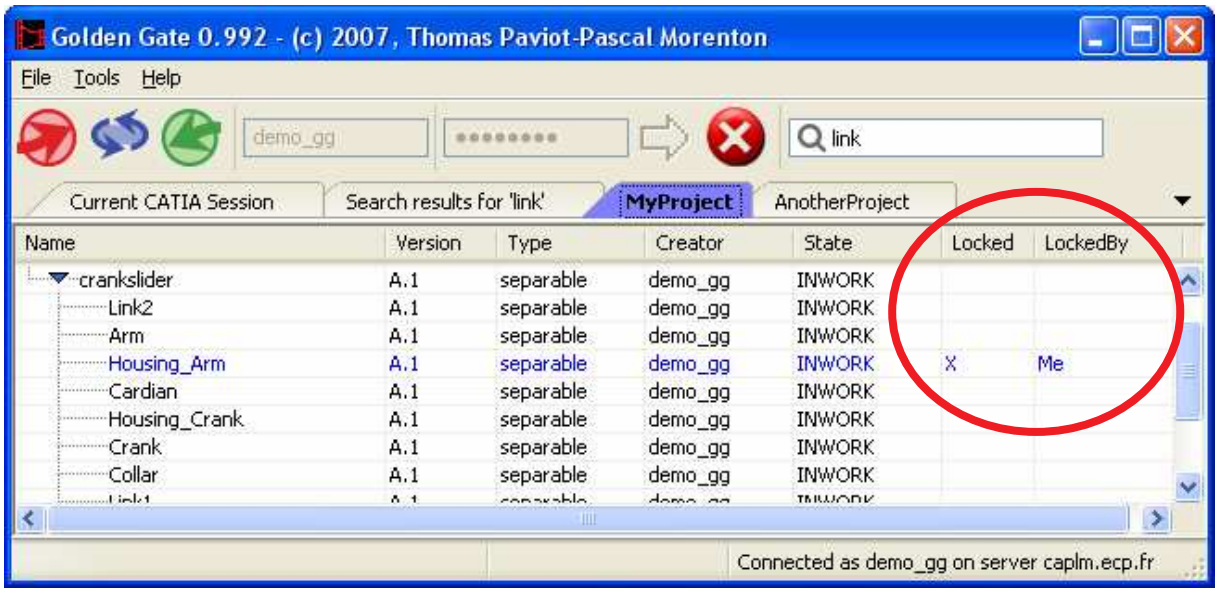


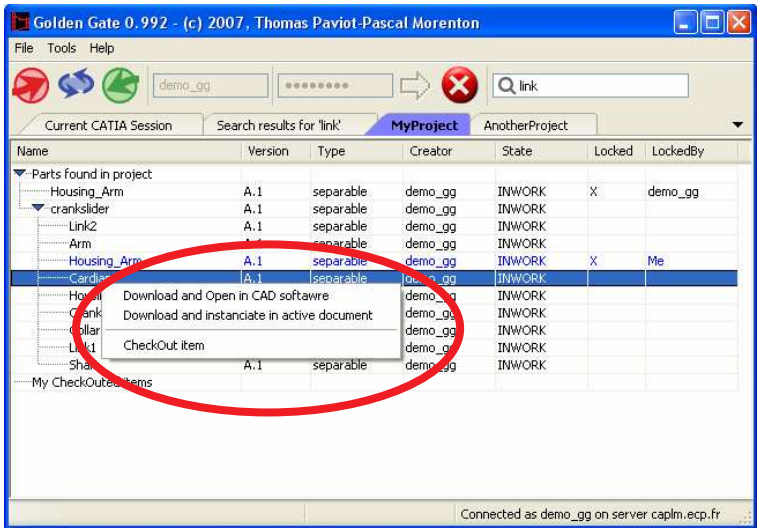
In this tab, you can select a part and open it into your CAD session with the ***download*** command :



8 Managing “check-in” / “check-out” items

When you parse a project, the items which has been “check-out” are mentioned in the locked and “locked by” column :



9 Contextual commands

--- not yet implemented ---