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Introduction

ENOVIA VPLM Product Design

Upon completion of this course, the student should have a full understanding of the following topics:

- Create an assembly utilizing CATIA assembly tools
- Store an assembly back to ENOVIA
- Utilize ENOVIA to associate actions to define effectivity for instances and assemblies created in CATIA
- Link and unlink instances to a GCO both in CATIA and ENOVIA
- Create new instances in CATIA and ENOVIA
- Manipulate and constrain assemblies utilizing CATIA assembly design tools
- Develop a relationally based design using CATIA
- Develop a multi-tier relationally based design using CATIA
- Morph a relationally built part from one location to another

ENOVIA VPLM

As stated in the ENOVIA VPLM Basic Concepts book, ENOVIA is an object oriented PLM database. In the concepts course you learned how to navigate and utilize ENOVIA to retrieve information and send it to the various editors such as CATIA and Word. The ENOVIA VPLM Advanced Concepts course expanded on that knowledge to allow you to create and define the various objects in ENOVIA that held products, instances, and references. You also looked at the configuration management schemes and the change management system to learn how to control effectivity and specifications for a particular product. All of this will be extended upon in this course by using those concepts and applying them to use within CATIA V5.

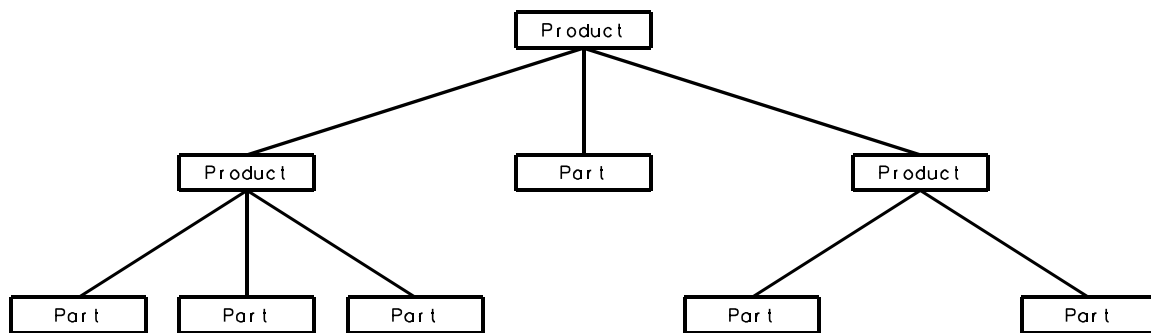
CATIA V5

CATIA V5 is the primary CAD tool integrated into ENOVIA VPLM. You will use the CATIA Assembly Design and Part Design workbenches exclusively to build and constrain your products. You should already be familiar to the various tools available to you in the CATIA Part Design and Sketcher workbenches. These tools will be critical when you begin working on the relational design aspects of this course.

ENOVIA VPLM Product Design

Very few finished designs are a single part. Usually a finished design consists of several to millions of individual parts to define them. This is where CATIA V5 assembly design is utilized. Assembly design allows parts and small assemblies of parts to be inserted to make larger, more complete products. In CATIA V5 Part Design and Sketcher, you learned how to generate parts.

It is important to understand some of the terminology that CATIA uses when working with assemblies and how they map to ENOVIA. There are basically three types of documents that are used in assembly design. They are the overall assembly, sub-assemblies and individual models. CATIA uses the word products to refer to assemblies and parts to refer to individual models. You can use parts to create products and then in turn use those products to produce other products. When you talk about these objects in ENOVIA terms, you will have products as the overall assembly, product instances for the sub assemblies, and then part instances for the individual parts. The diagram shown below represents the concept of the overall structure.

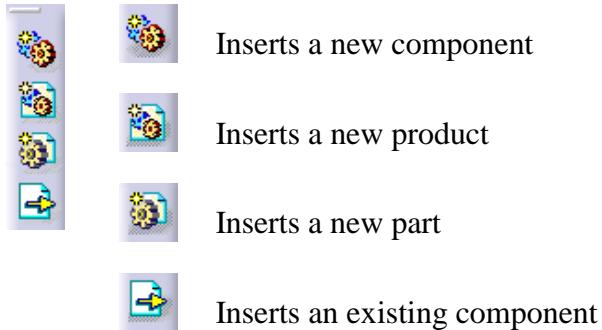


The first product at the top is generally regarded as the assembly, or product in ENOVIA, whereas the two products that are underneath are generally regarded as sub-assemblies of this assembly. ENOVIA denotes these sub-assemblies as product instances since they are a usage of a sub-assembly. With this concept in mind be aware that an assembly could be a very complex document due to its ability to have multiple levels of sub-assemblies and parts. Because of this complexity it is important that you have a plan of attack when building assemblies. There are basically two approaches that a user or company can take when building assemblies. One is to pre-determine what sub-assemblies a particular assembly is going to need. The other is to produce all of the parts and then determine what sub-assemblies are going to be created.

CATIA V5 Assembly Design

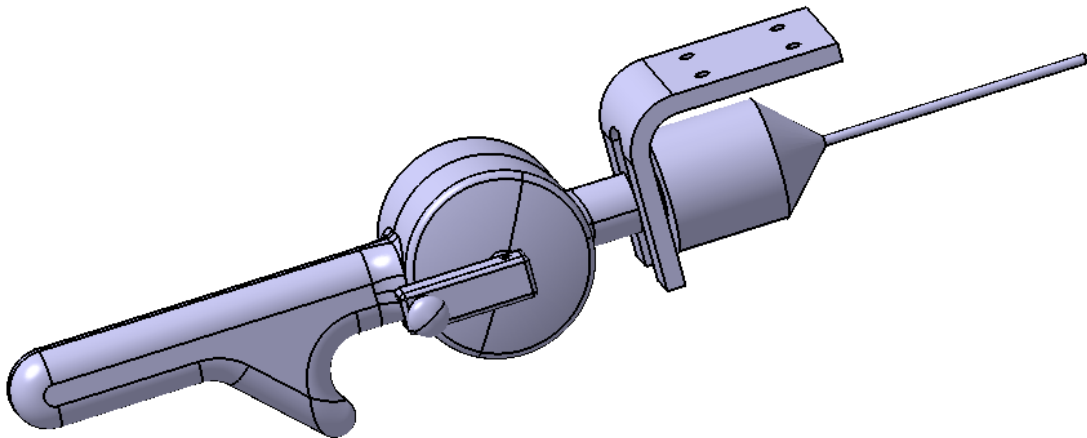
At this point, you are ready to begin creating assemblies in ENOVIA. This section is to serve as an introduction into assembly design while working in the ENOVIA environment. You should already have a good knowledge of the CATIA interface and how to manipulate the display. If you find you have difficulty working with CATIA, please refer back to your CATIA Basics books for more information.

In this exercise you will be using several Assembly Design icons to insert parts and create new subassemblies and parts. Some of those icons are:



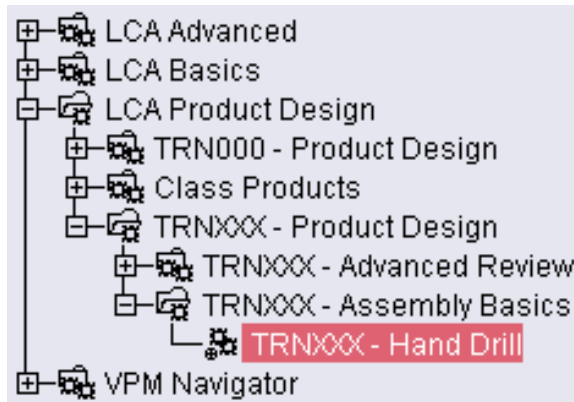
These icons can also be found under the *Insert* pull down menu and also the right click menu from a product.

For this exercise all the hand drill models, minus one, to be used are already built and positioned relative to their local axis. You will create all the product structure in CATIA and ENOVIA to complete this assembly.

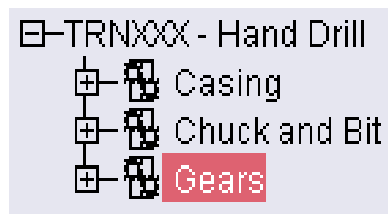


Before you get started, you will need to set up the product class, product, create a few GCOs and create an action. Be sure to save often as you go through these steps.

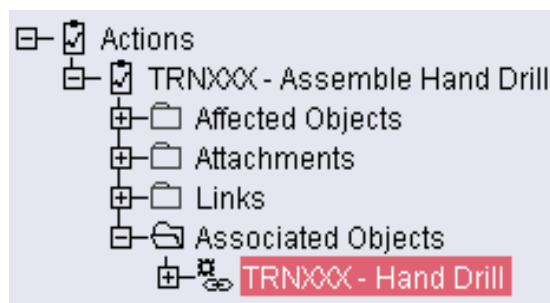
In your *Product Design* product class, create a {USERID} - Assembly Basics product class. Inside that product class create a configured product called {USERID} - Hand Drill. Remember, you MUST create every object unique to your user ID. This means that your tree will need to look like the following.



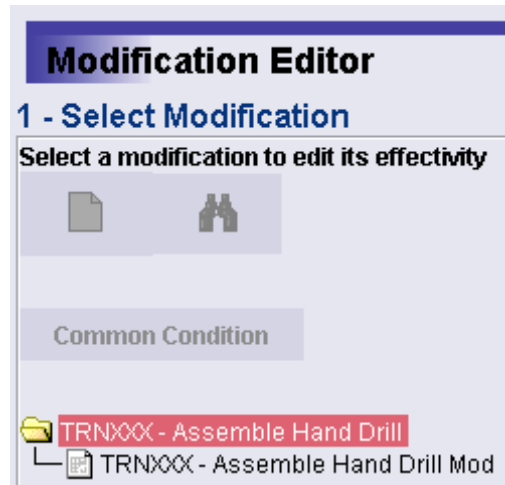
Create the following three GCOs in your Hand Drill product: *Casing, Chuck and Bit, and Gears*. All three generic components will need to be created directly below the product in the Component editor. You do not need to nest the generic components.



Create a design action called {USERID} - Assemble Hand Drill and associate it to your Hand Drill product. It should appear as shown.



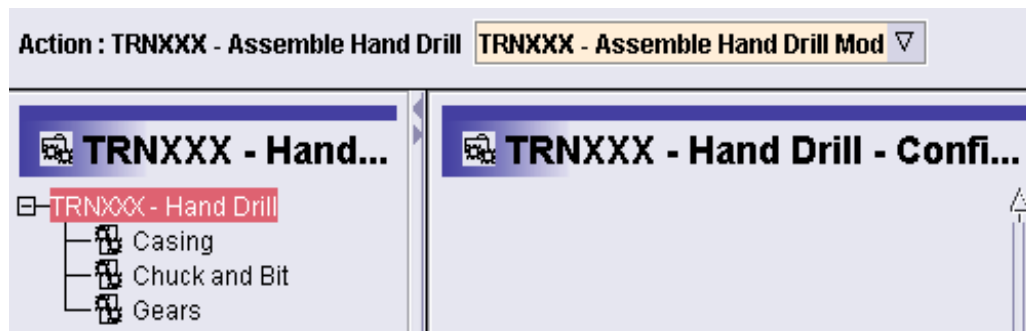
Inside the action create a modification object called `{USERID} - Assemble Hand Drill Mod` with a range effectivity from 1 to infinity. This will define the action and modification effectivity for this product.



Now you are ready for the last step, open the product and associate the action.

Open the product with your *Assemble Hand Drill* action associated in the product editor. Remember, if you send the product to the product editor from the actions editor, it will automatically have the action associated.

Once the product is open with your action associated, you are ready to begin the exercise. These steps will be required for each new exercise.

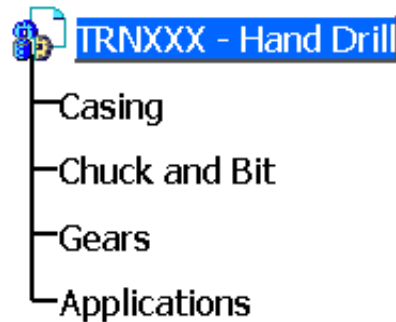


From the component pane, open the product in the instance view. Remember this can be accomplished by dragging and dropping the product or selecting *Open Assembly in Instance View* from the right click menu.

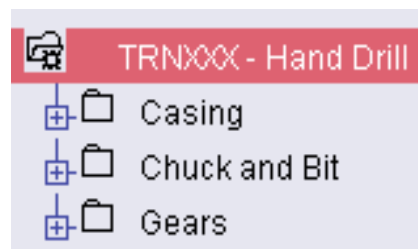


Send the {USERID} - Hand Drill product to CATIA V5 as an assembly. Although there are no instances in this product, you want to send it as an assembly to allow the GCOs to be transferred into CATIA.

This will launch CATIA and import the product and the GCO's into the session. Take a look at your product tree.



Notice that the CATIA Specification tree is very similar to that of the ENOVIA specification tree.



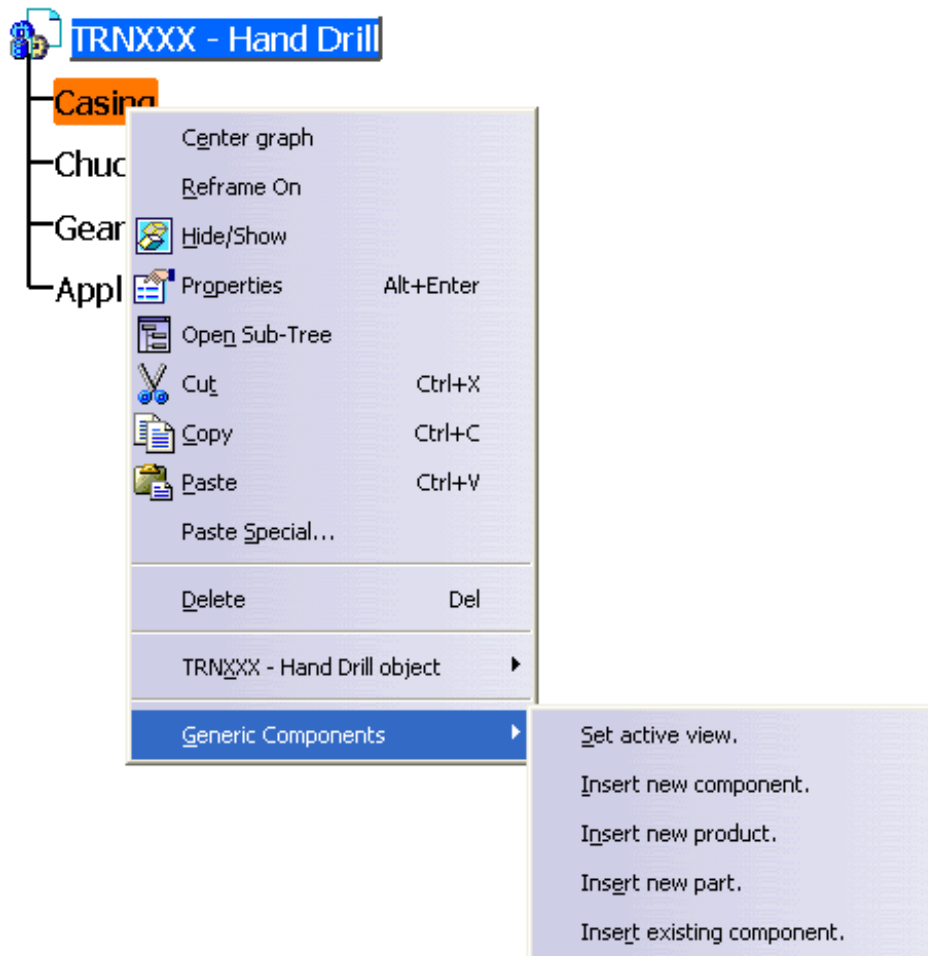
The Applications branch in the CATIA specification tree is built into CATIA and provides an area to store Kinematics, DMU, and other product related information. This information does not currently get stored into ENOVIA.

Switch over to your CATIA session if you are not already there.

Inserting Existing Components

Inserting existing components assumes that you have already created the necessary parts and they just need assembled. These models are going to exist in the *Product Design* folder under the *Assembly Basics* directory.

Right click on the *Casing* GCO in the specification tree, then select *Generic Components*. This will give you a number of insert options when working directly with the GCO.



Set active view Defines the active GCO for viewing items found only in that GCO through the product structure workbench

Insert new component Inserts a component into the product and automatically links it to the GCO

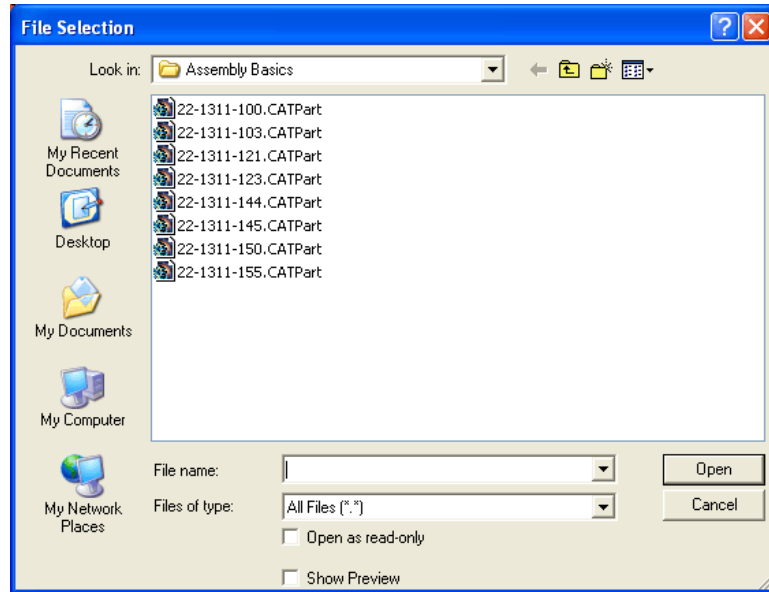
Insert new product Inserts a new product into the main assembly and automatically links it to the GCO

Insert new part Inserts a new part into the product, linking it to the GCO

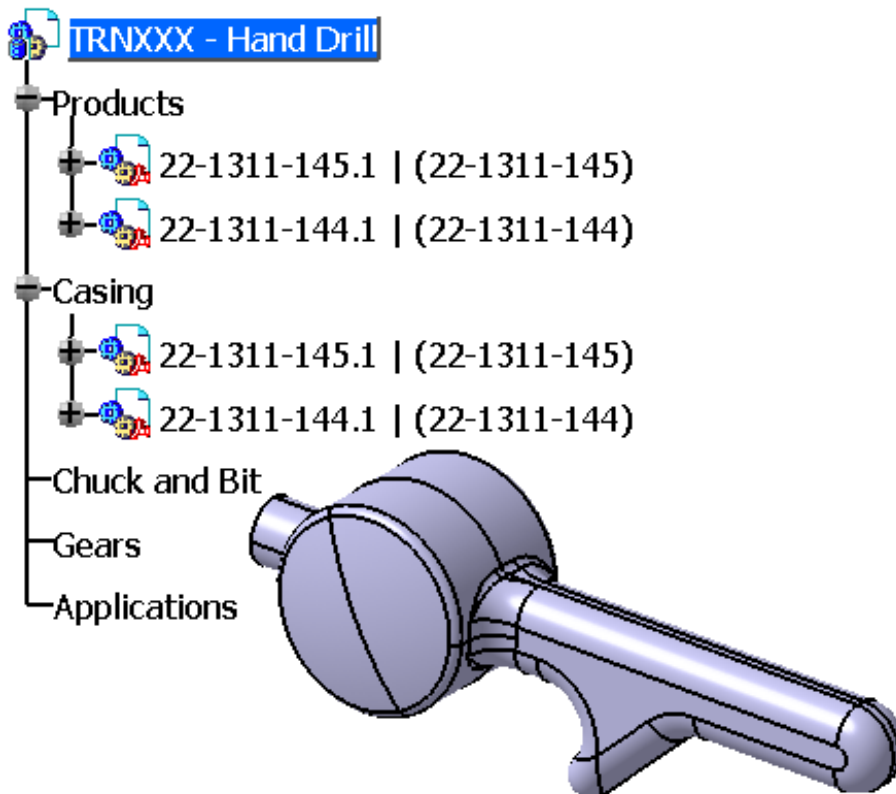
Insert existing component Inserts a part or product already existing into the assembly, linking it to the GCO

Select *Insert Existing Component*. This will open a *File Selection* window for you to browse drives on your computer and find the component you want to insert.

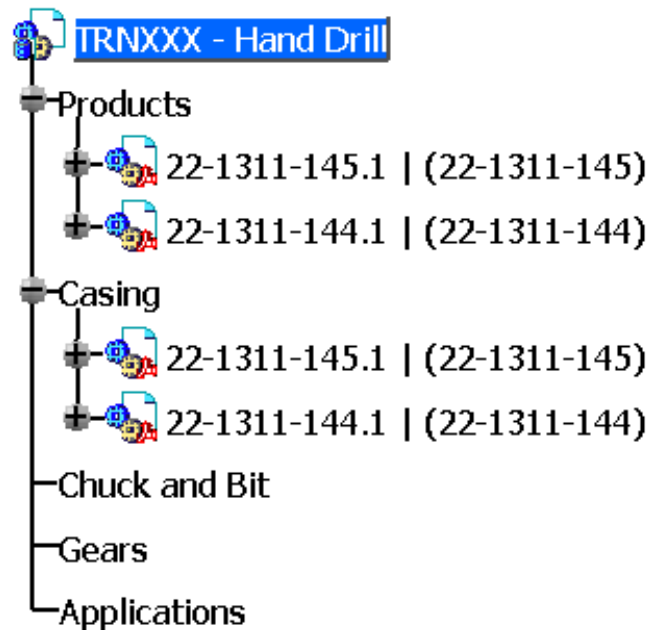
Browse to the *Assembly Basics* folder. This folder should be in the *ENOVIA Class Models* directory.



While holding the CTRL key on the keyboard, select part numbers 22-1311-144 and 22-1311-145 from the window then select *Open*. The files are loaded into CATIA.

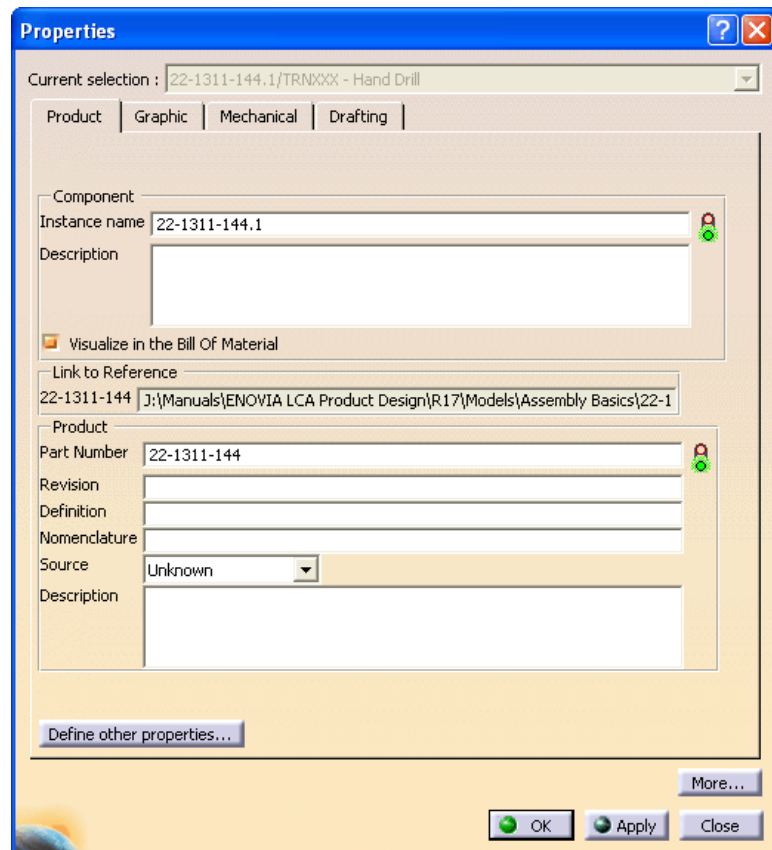


Notice the parts are inserted into CATIA and automatically linked to the GCO.



The next step, anytime you insert a part into CATIA is to rename the instance to something a little more descriptive than a part number.

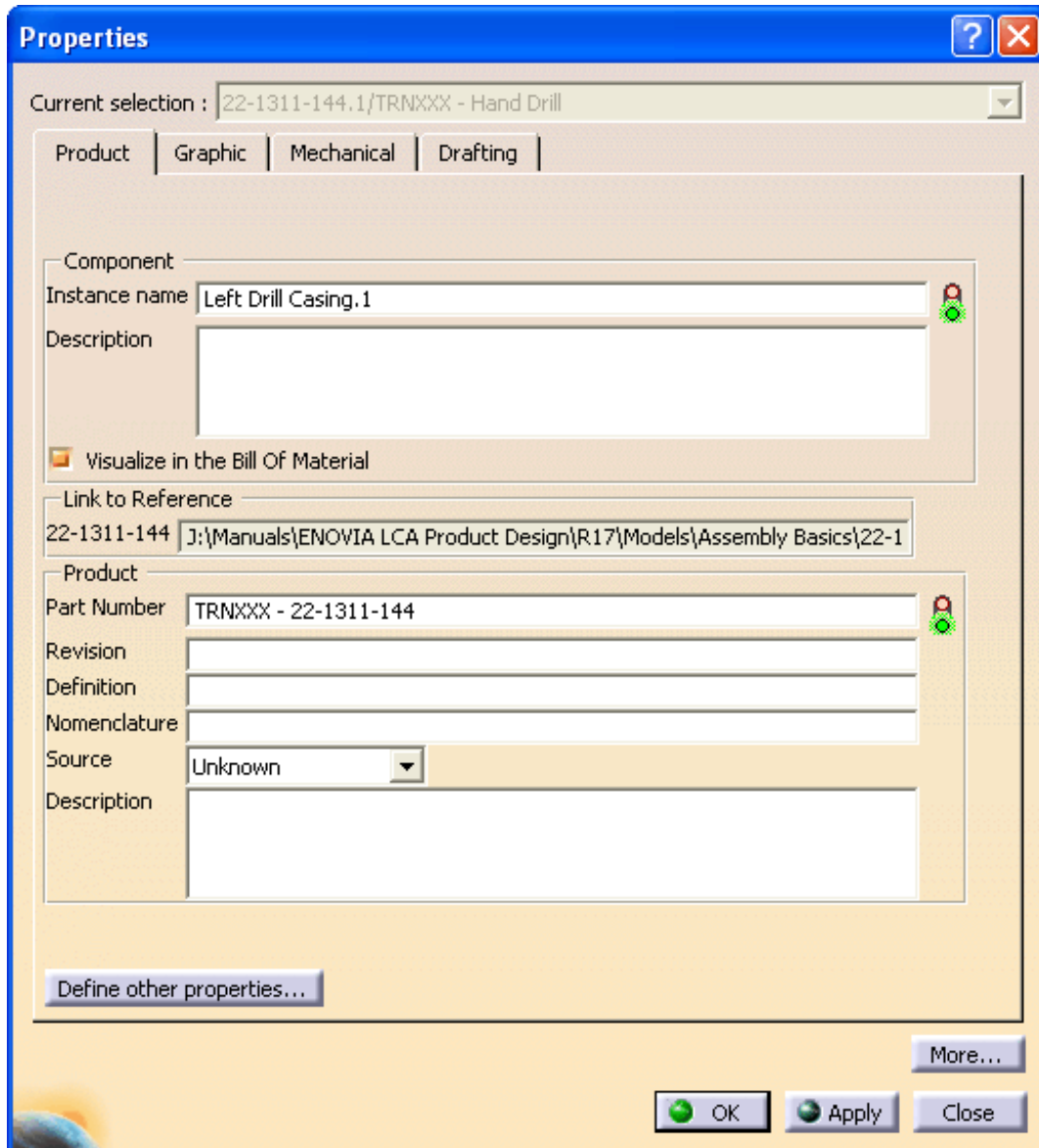
Right click on the part instance 22-1311-144.1 in the specification tree and select *Properties*. This will display the properties for the instance.



From here, you can change the instance and reference ID of the part. The instance ID is the *Instance Name* field, while the reference ID is denoted by the *Part Number* field.

Since everyone is using the same parts, you will need to make the part reference ID unique.

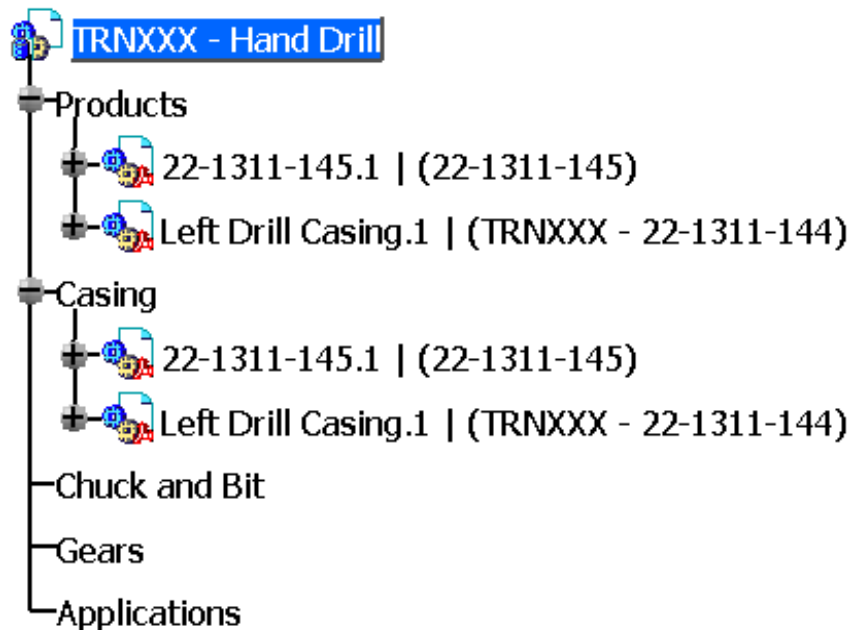
Change the *Instance Name* to Left Drill Casing.1 and the *Part Number* to {USERID} - 22-1311-144. This will give the instance a good, human readable name, as well as make the part reference unique to your user ID.



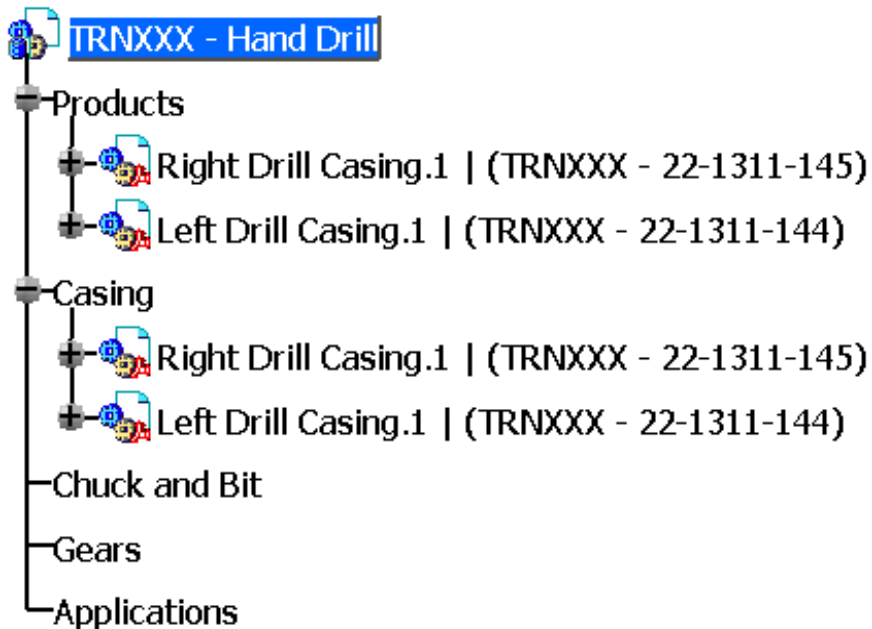
The screenshot shows the 'Properties' dialog box for a part. The 'Current selection' is '22-1311-144.1/TRNXXX - Hand Drill'. The 'Product' tab is selected. The 'Component' section has 'Instance name' set to 'Left Drill Casing.1' and a 'Description' field. There is a checkbox for 'Visualize in the Bill Of Material'. The 'Link to Reference' section shows the part number '22-1311-144' and a file path. The 'Product' section has 'Part Number' set to 'TRNXXX - 22-1311-144', 'Revision', 'Definition', and 'Nomenclature' fields, a 'Source' dropdown set to 'Unknown', and a 'Description' field. At the bottom, there are buttons for 'Define other properties...', 'More...', 'OK', 'Apply', and 'Close'.

Select *OK* when done.

The names are changed in the specification tree.



For the part 22-1311-145, change the *Instance Name* to **Right Drill Casing.1** and the *Part Number* to {USERID} - 22-1311-145. This will also make it unique in ENOVIA.



At this point, you are ready to save the two instances and references back to ENOVIA.

Saving to ENOVIA

As you might already know, saving is a very important part of working with ENOVIA. In previous CATIA courses, you generally used a *File, Save As* method of saving your parts. When saving in an ENOVIA context, you will have to do things a bit differently. Instead of using the regular save, you will have to perform an ENOVIA save. The ENOVIA save is located in the ENOVIA V5 VPM toolbar.








For these exercise you are saving CATIA files to ENOVIA from a network share.


In ENOVIA, verify the ENOVIA Portal Setting for the Product Editor *Auto lock new instances* is set to *True*. This will auto lock all the parts you save to the database to yourself.

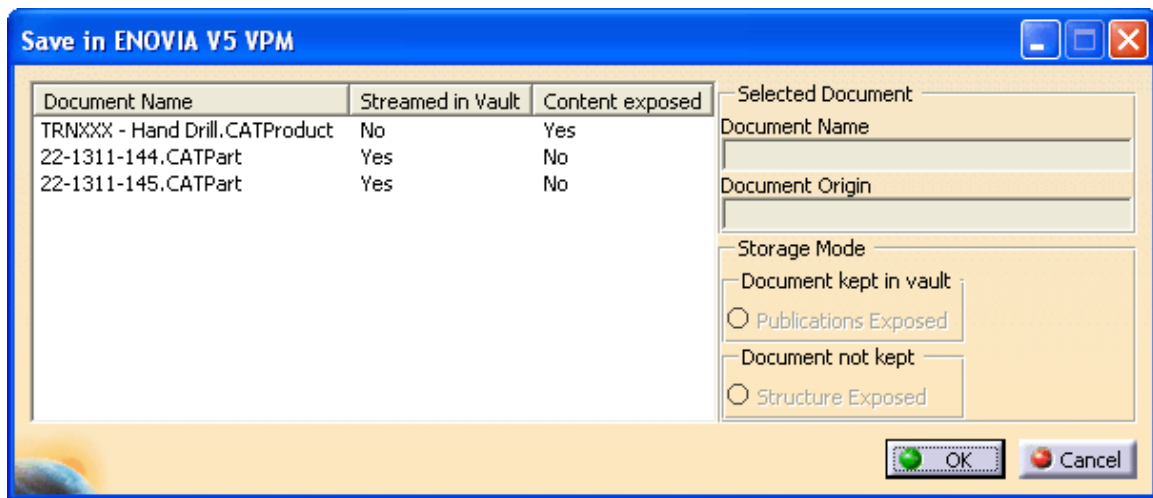


In CATIA, turn on the Enovia V5 VPM toolbar by selecting on pull down menu *View*, then select *Toolbars* and finally *Enovia V5 VPM*. This will display the *ENOVIA V5 VPM* toolbar. Take a moment to look at the various icons found here.



	<i>Connect / Disconnect Enovia V5...</i>	Launches ENOVIA VPLM from CATIA V5
	<i>Set PDM Properties</i>	Allows you to define ENOVIA save options
	<i>Save Data in Enovia V5 VPM...</i>	Saves modifications of instances and references back to the ENOVIA server
	<i>Identify in Enovia V5 VPM</i>	Identifies the selected object in ENOVIA product editor
	<i>Load Children from Enovia V5...</i>	Loads the children parts from a selected product from ENOVIA
	<i>Unload Branch from CATIA V5...</i>	Unloads a selected branch from CATIA
	<i>Refresh Documents</i>	Refreshes modified documents from ENOVIA

Select the **Save Data in ENOVIA V5 VPM Server icon**.  The *Save in ENOVIA V5 VPM* window will appear. This icon is generally called the ENOVIA Save icon.



Lets take a moment to look over this window.

Document Name This is a list of all the documents that are going to be saved to ENOVIA. This a list of documents files, hence you will not see instances in this window. The instances and references get saved automatically with the document files and product.

Streamed in Vault Indicates if an document object is created for the document in the database. CATParts will always be set to yes, CATProducts can be changed depending if you want the product to be seen as one document object or not.

Content exposed This states whether or not individual parts of a product will be saved in the database. For individual parts, *Content exposed* will always be set as *No*.

Selected Document

Document Name You can change the document name of newly created CATParts or inserted CATParts.

Document Origin Indicates if the document came from ENOVIA, VPM, or is a File.

Storage Mode

Document kept in vault

Publication Exposed

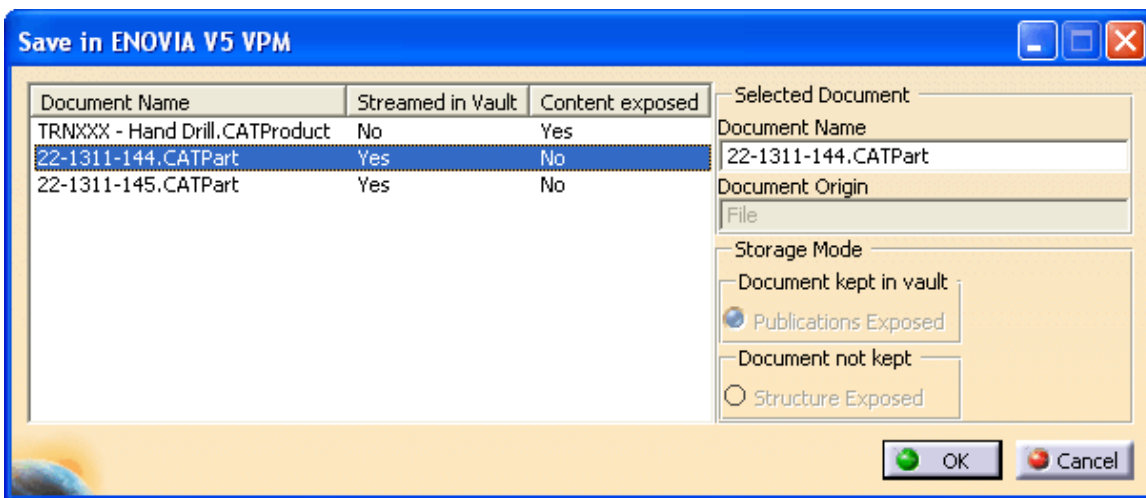
Specifies a product will be saved to the database as a Work Package Product, or in “black box mode”

Document not kept

Structure Exposed

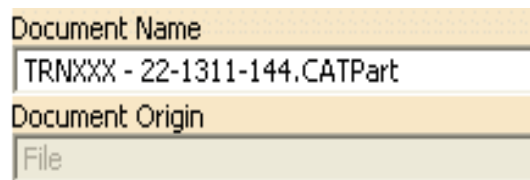
Specifies product will be saved to the database with its content exposed so you will be able to see the parts and subassemblies within it

Select the document **22-1311-144** from the **Save window**. Notice in the *Selected Document* frame you are allowed to change the *Document Name*.



Just like the references, you will need to create unique document files, otherwise you will risk creating a bracketed part.

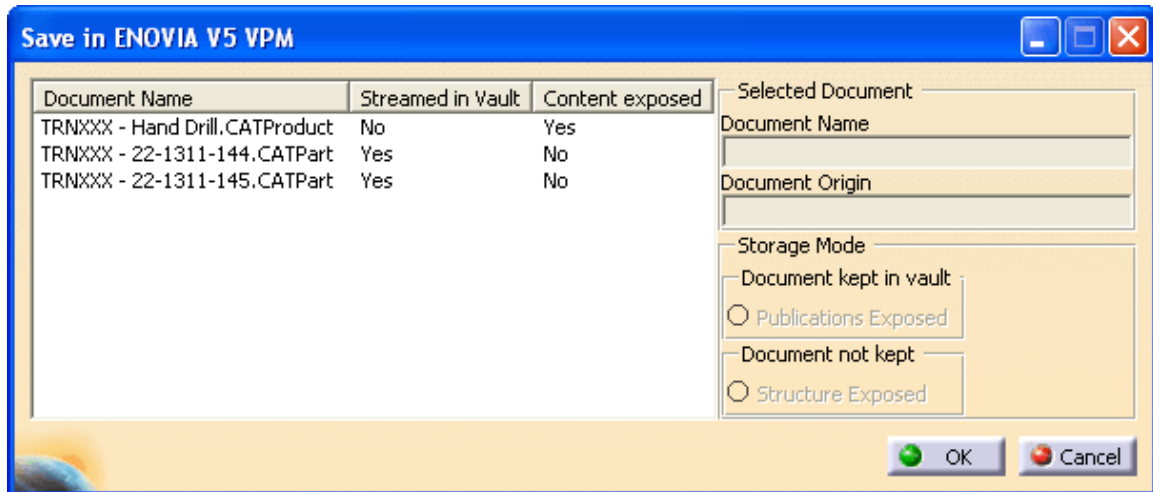
Change the *Document Name* to `{USERID} - 22-1311-144.CATPart`. Do NOT hit Enter on the keyboard when you are done. In general, if you need to change a document name that you are inserting off of a hard drive, you have to change the reference name in one location and the document name as you did here. If you create a new part in CATIA in an ENOVIA product you do not have to worry about this step.



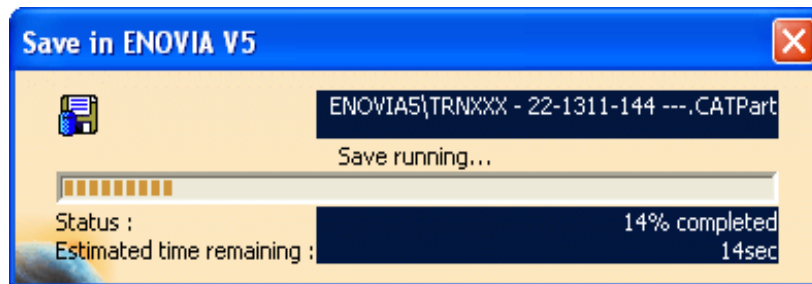
Select the document **22-1311-145** from the **Save window**. This will change the name of the previous document you changed.

Change the document name to **{USERID} - 22-1311-145.CATPart**. This will make the document name unique in ENOVIA.

Select in the blank area below the document names. This will apply the changes to the documents.



Select **OK** when done. This will display a brief window that will show the save progress.



Once this window disappears, CATIA is done saving back to ENOVIA. If no error windows appear after the save is complete you can assume a successful save.

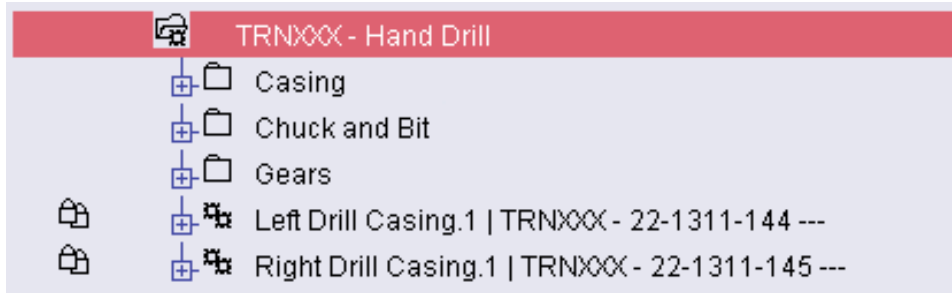
Switch over to the **ENOVIA Portal**. Use the taskbar at the bottom of your screen.

Notice your parts do not automatically show up in the product.

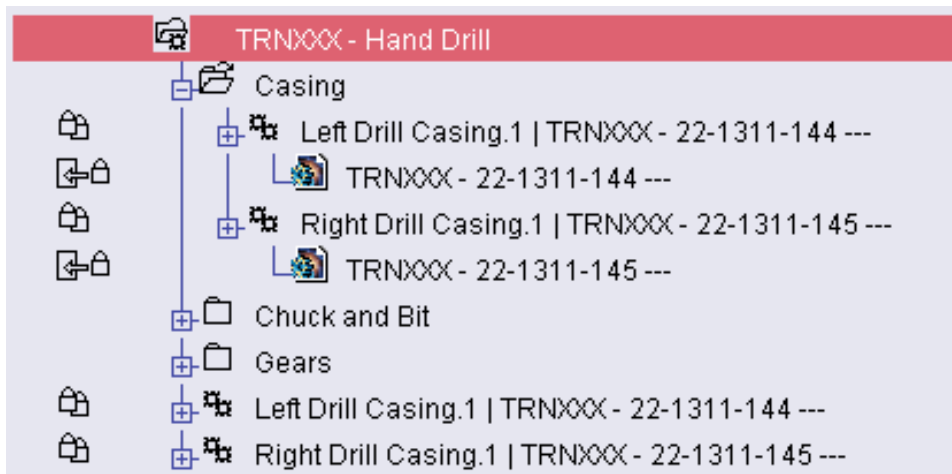


You must refresh ENOVIA in order to view the new instances and references.


Perform a local refresh. The *Left Drill Casing.1* and *Right Drill Casing.1* will appear. A database refresh is not necessary, however, it will make no difference.



Expand out the *Casing* GCO and the casing instances. Notice the ENOVIA specification tree has a similar structure as the CATIA specification tree.



Now you are ready to insert a few more parts into your CATIA assembly.

In CATIA, select the Existing Component icon.  The icon will highlight and is waiting for the user to select a product to insert the component into.

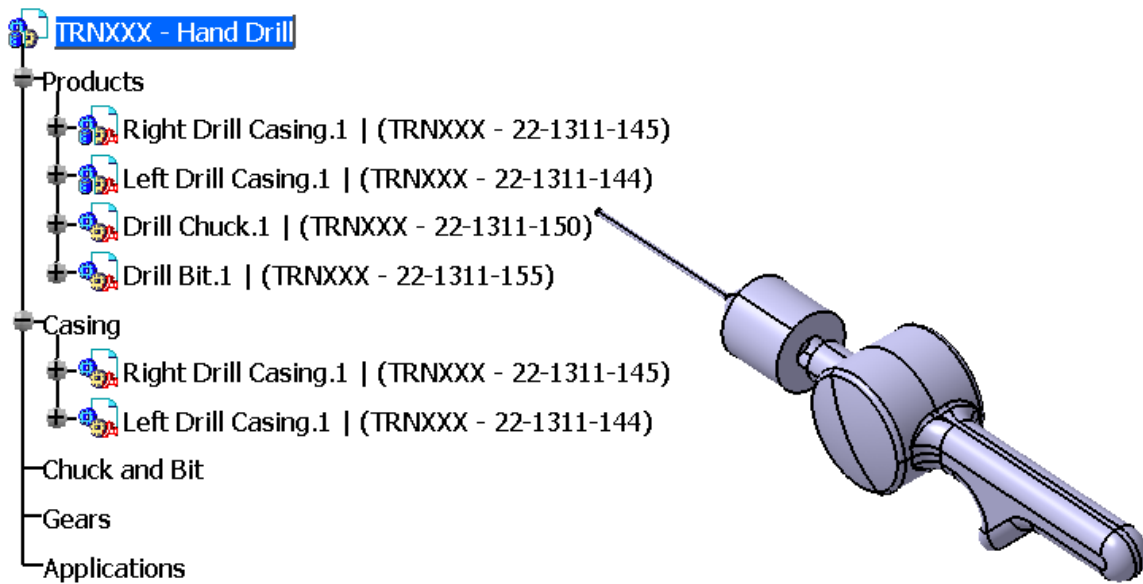
Select the *Hand Drill* product from the top of the tree. This will define what product the component will be placed into. A *File Selection* window appears.

Select the part numbers 22-1311-150 and 22-1311-155 then select *Open*. The parts will be inserted into the product.

Change the part numbers as follows. Go into each part number properties to make these changes.

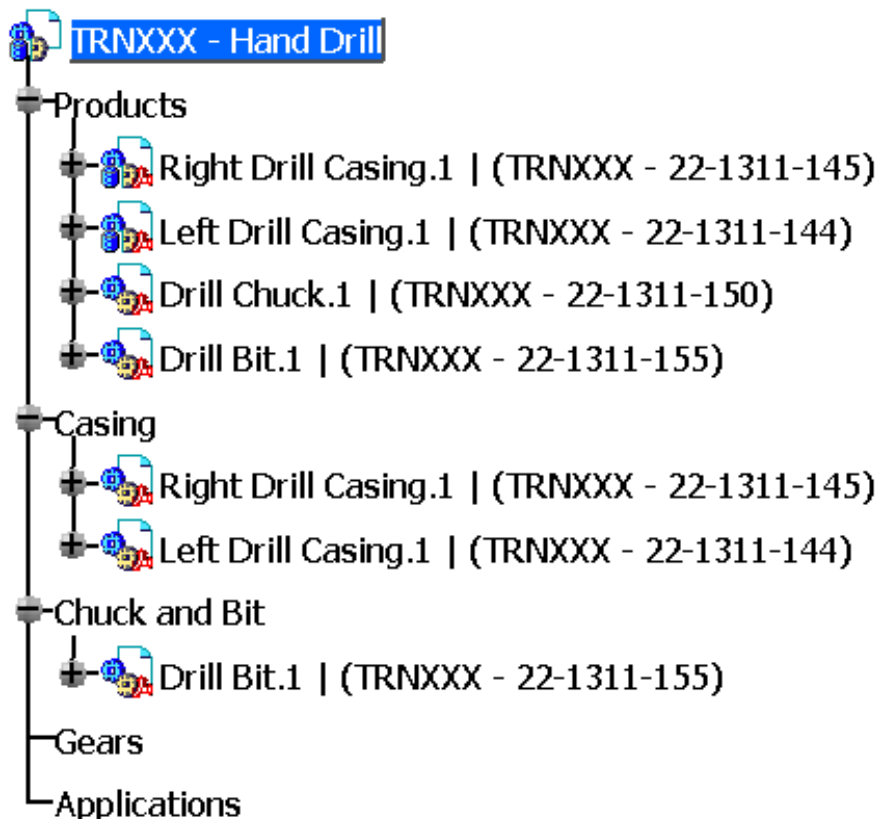
Original Part Number	New Part Number	Instance Name
22-1311-155	{USERID} - 22-1311-155	Drill Bit.1
22-1311-150	{USERID} - 22-1311-150	Drill Chuck.1

It should appear as shown.

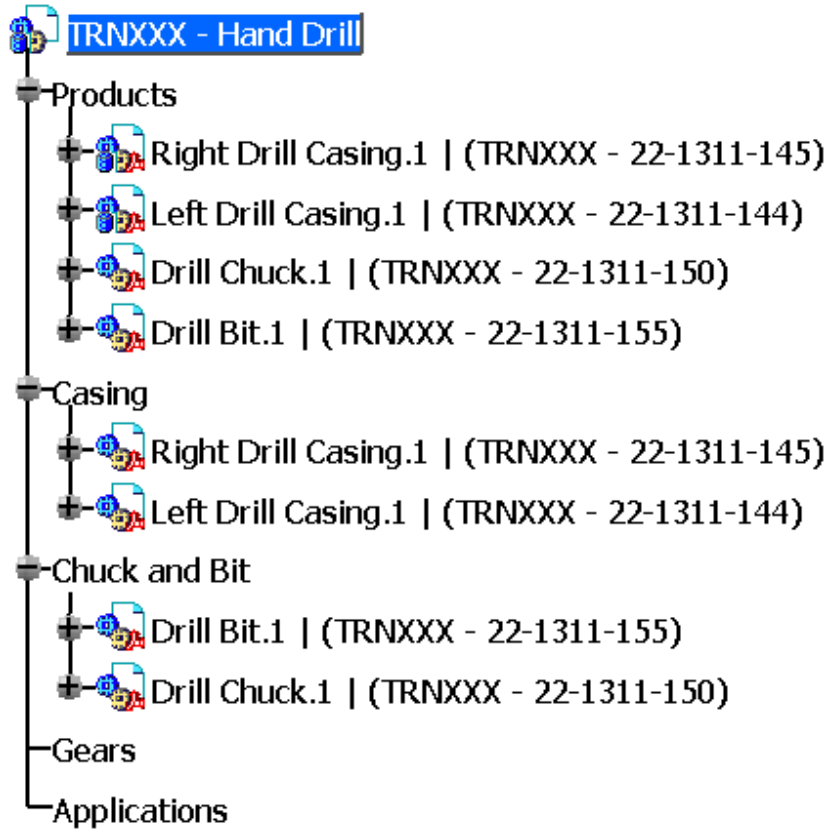


Notice the *Drill Chuck.1* and the *Drill Bit.1* are just in the parts list below the product branch. Anytime you insert parts into the product, and not through the GCO, they will be inserted into the product and not linked to a GCO. Keep in mind, you do not have to link all parts to a GCO. You are going to create this link manually with a drag and drop method.

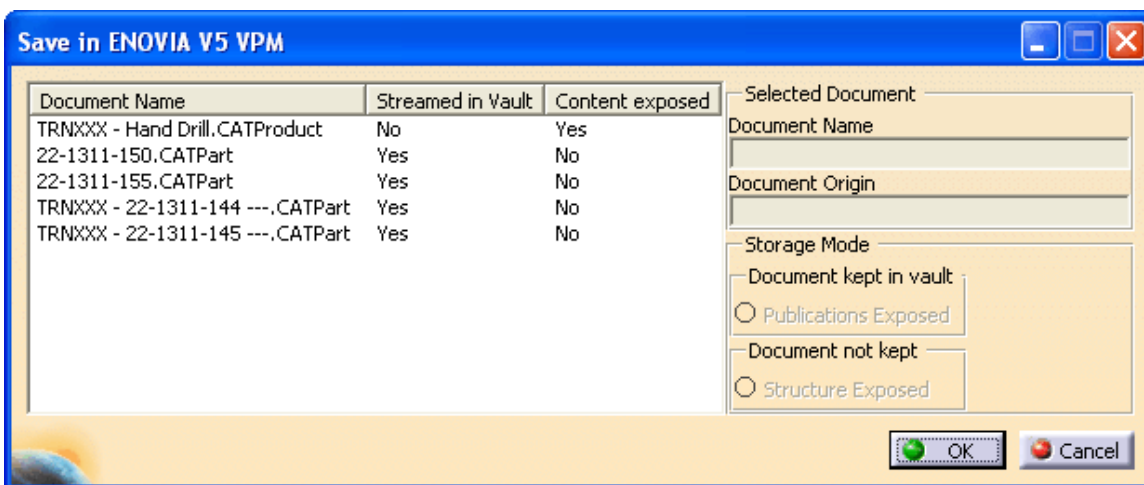
With the first mouse button, select and hold on *Drill Bit.1* then drag it down to the *Chuck and Bit* GCO. Release the mouse button once over the GCO. This will create a link in the GCO to the instance listed directly under the product.



Drag and drop the *Drill Chuck.1* instance into the *Chuck and Bit GCO*. Another link is created. These links have to be created one at a time.



Select the **Save Data in ENOVIA V5 VPM Server icon**. Do NOT hit *OK* just yet, you need to make some changes to the document names.

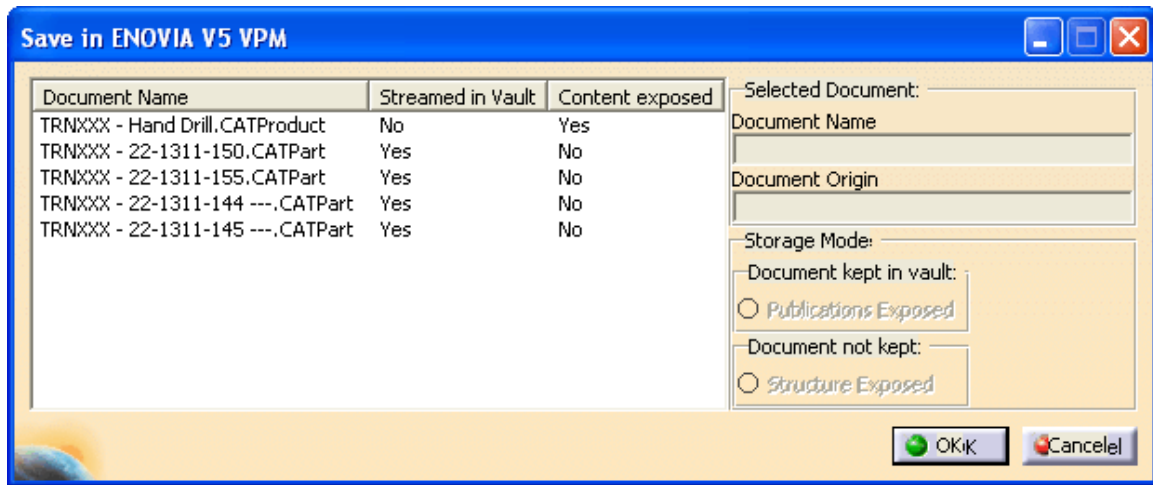


Notice the new documents that do not have your user ID in front of the document name.

Change the document names as shown in the table below. Select the document name then change it on the field on the right, do NOT hit Enter after each name change.

Original Document Name	New Document Name
22-1311-150.CATPart	{USERID} - 22-1311-150.CATPart
22-1311-155.CATPart	{USERID} - 22-1311-155.CATPart

The window should appear as shown.



Select OK. The parts will be saved back to ENOVIA.

As usual, it is always a good idea to make sure they save back to ENOVIA by opening ENOVIA and performing a refresh.

In ENOVIA, perform a local refresh. The additional parts will appear.


Select the Save icon in ENOVIA. When prompted, select No Unlocks. This ensures that everything is saved in the database. Although this step may be unnecessary, it provides the added security that it is saved. Since you are still working on this product, you do not want to unlock anything.

New Components

Next you are going to look at creating a new component in CATIA. A new component is a way to represent sub-assemblies in CATIA and ENOVIA, but there is no document associated with the component. When you create a new product as a sub-assembly, you will find that it generates a new document that wants to save with the assembly.

Using components as sub-assemblies is fairly straight forward and easy to use but they have a couple of down sides. Components do not have a document associated with them therefore you can not search for a component as a document. Also with components you can not instantiate the sub-assembly in other product of ENOVIA.

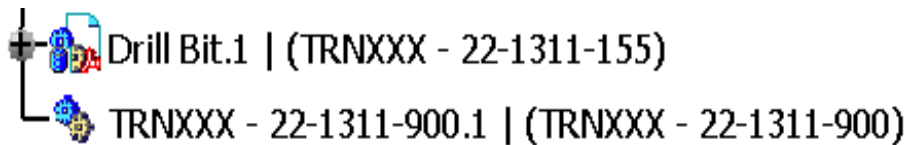
Switch back to CATIA.

Select the Component icon, then select the *Hand Drill* product.  This will insert a new component into your specification tree.

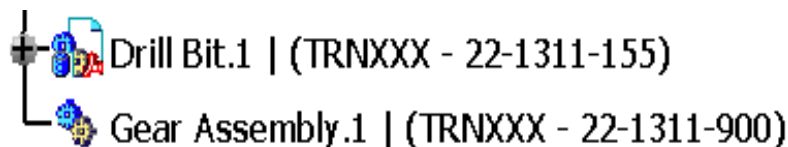



If you do not get the new *Part Number* window this can be changed under *Tools, Options*. Inside *Options*, go *Infrastructure, Product Structure, Product Structure* tab, *Part Number*, and check *Manual Input*.

Key in the part number of the new component to {USERID} - 22-1311-900 and select *OK*. This will give the part reference an unique name.

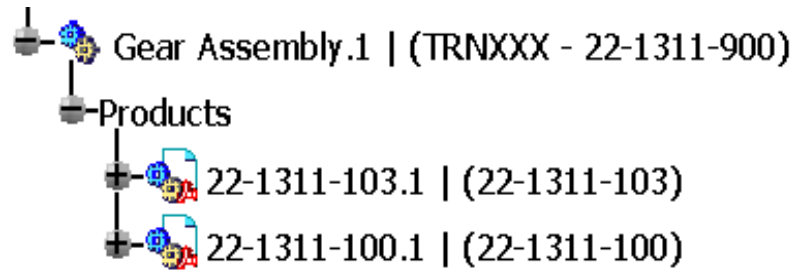


Go into the properties for the new component and change the *Instance Name* to Gear Assembly.1. This will give the instance name a more descriptive name.



Select the Existing Component icon, then select *Gear Assembly.1*.  This time, you will insert components into the gears component. The existing component icon can also be found by right clicking on *Gear Assembly* then *Components*.

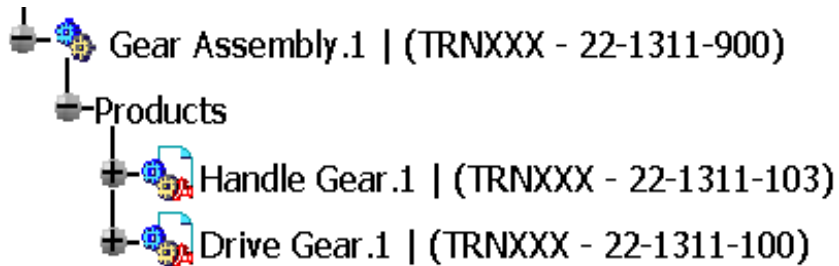
Insert both the -100 and the -103 parts into the component. This will insert the two gears into the component.



Rename the two new instances to the following:

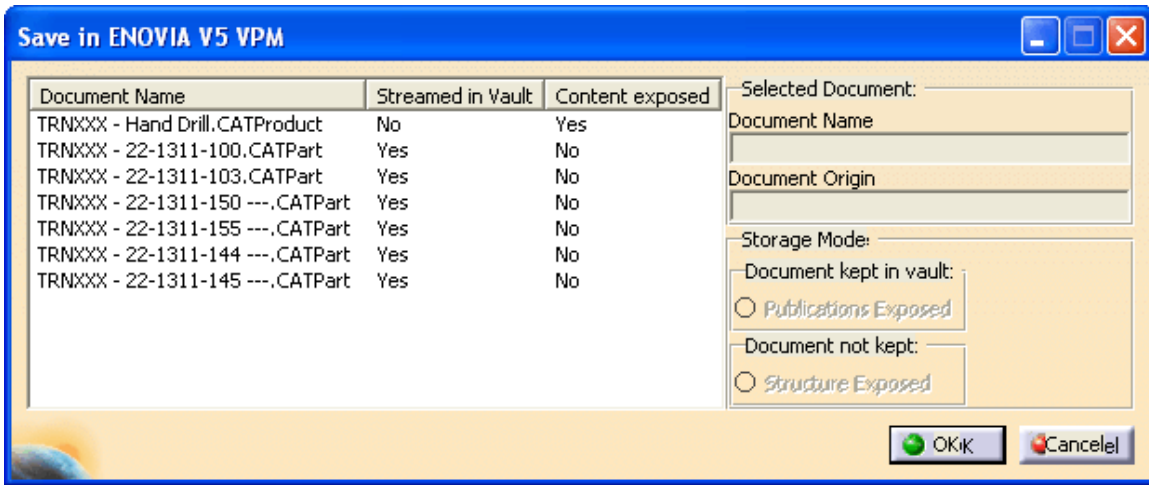
Original Part Number	New Part Number	Instance Name
22-1311-100	{USERID} - 22-1311-100	Drive Gear.1
22-1311-103	{USERID} - 22-1311-103	Handle Gear.1

They should appear as shown.



Instead of creating the link to the GCO in CATIA, you will now look at creating the link to the GCO in ENOVIA. Before you can create the link, you will need to save.

Select the ENOVIA Save icon. Rename the new documents to have your user ID in the same fashion as the other documents. Take a close look at the save window.



Notice anything missing? The {USERID} - 22-1311-900 document is missing. When you create a component, the component is kept inside the assembly product. It is not a separate file and does not get saved separately. In this case, the *Gears* component is stored in the *Hand Drill* product.

Select **OK** to save the product back to ENOVIA. The additional parts plus your changes are saved to ENOVIA.

Now switch back to ENOVIA and create the link.