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Wichita State University
Wichita, KS

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www.cadcamlab.org
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Introduction

ENOVIA LCA 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 LCA

As stated in the ENOVIA LCA 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 LCA 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 using CATIA V5.

CATIA V5

CATIA V5 is the primary CAD tool integrated into ENOVIA LCA. 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.
CATIA Assembly 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.

![Diagram of assembly 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 a 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 Pull Down Menus

Not all of the options will be covered since you should have already been introduced to them through previous courses. Only the new options that appear when you are in the assembly design workbench will be discussed.

**Edit**

<table>
<thead>
<tr>
<th>Option</th>
<th>Key</th>
</tr>
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<tbody>
<tr>
<td>Undo Empty selection</td>
<td>Ctrl+Z</td>
</tr>
<tr>
<td>Repeat</td>
<td>Ctrl+Y</td>
</tr>
<tr>
<td>Cut</td>
<td>Ctrl+X</td>
</tr>
<tr>
<td>Copy</td>
<td>Ctrl+C</td>
</tr>
<tr>
<td>Paste</td>
<td>Ctrl+V</td>
</tr>
<tr>
<td>Paste Special...</td>
<td></td>
</tr>
<tr>
<td>Delete</td>
<td>Del</td>
</tr>
<tr>
<td>Update</td>
<td>Ctrl+U</td>
</tr>
<tr>
<td>Move</td>
<td></td>
</tr>
<tr>
<td>Component Constraints</td>
<td></td>
</tr>
<tr>
<td>Search...</td>
<td>Ctrl+F</td>
</tr>
<tr>
<td>Selection Sets...</td>
<td>Ctrl+G</td>
</tr>
<tr>
<td>Selection Sets Edition...</td>
<td></td>
</tr>
<tr>
<td>Find Owning Selection Sets...</td>
<td></td>
</tr>
<tr>
<td>Links...</td>
<td></td>
</tr>
<tr>
<td>Properties</td>
<td>Alt+Enter</td>
</tr>
<tr>
<td>Other Selection...</td>
<td></td>
</tr>
<tr>
<td>Components</td>
<td></td>
</tr>
<tr>
<td>Representation</td>
<td></td>
</tr>
</tbody>
</table>

**Move**

Various methods of moving your components around in the assembly

**Component Constraints**

Allows you to select constraints that are linked to a component
Components
- Replace Component In Session...
- Replace Component...
- Isolate Part
- Define Contextual Links
- Load
- Unload
- Graph tree Reordering

Allows for components within the assembly to be replaced, turned on or off or reordered.

Representation
- Manage Representations...
- Associate CDM
- Design Mode
- Visualization Mode
- Activate Node
- Deactivate Node
- Activate Terminal Node
- Deactivate Terminal Node

Allows for the representation of the assembly to be changed.
Insert

- **Coincidence**... Inserts a coincidence constraint
- **Contact**... Inserts a contact constraint
- **Offset**... Inserts an offset constraint
- **Angle**... Inserts an angle constraint
- **Fix Together**... Fixes two components together
- **Fix**... Fixes a component in space
- **Quick Constraint**... Applies a quick constraint to components
- **Reuse Pattern**... Applies a pre-defined pattern to a component
<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>New Component</td>
<td>Allows the insertion of parts as details</td>
</tr>
<tr>
<td>New Product</td>
<td>Inserts a new assembly into the product</td>
</tr>
<tr>
<td>New CDM Component</td>
<td>Inserts a new CDM component</td>
</tr>
<tr>
<td>New Part</td>
<td>Inserts a new part into the assembly</td>
</tr>
<tr>
<td>Existing Component</td>
<td>Inserts an existing component</td>
</tr>
<tr>
<td>Existing Component With Positioning</td>
<td>Inserts an existing component and positions it</td>
</tr>
<tr>
<td>Document Template Creation</td>
<td>Allows the creation of a template to be stored in a catalog</td>
</tr>
<tr>
<td>Create Enhanced Scene</td>
<td>Allows the creation of a scene</td>
</tr>
<tr>
<td>Fast Multi Instantiation</td>
<td>Applies a Fast Multi Instantiation of a component</td>
</tr>
<tr>
<td>Define Multi Instantiation</td>
<td>Defines a Multi Instantiation of a component</td>
</tr>
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</table>

**Views**

- Front View
- Section View/Annotation Plane
- Section Cut/Annotation Plane

*Creates annotation views*

**Annotations**

- Weld Feature
- Text with Leader
- Flag Note with Leader

*Adds a welding symbol, text with a leader or a flag note with a leader to your assembly*

**Assembly Features**

- Split
- Hole
- Pocket
- Remove
- Add

*Performs an operation on your assembly*

**Symmetry**

*Performs a symmetry operation*
Tools

Product Management  Allows the part number, and representation to be changed for a component

Publication  Allows the publication of components and elements to be modified

Generate CATPart from Product  Enables a product to be converted into a part

Catalog Browser  Access part catalogs for standard parts such as bolts, nuts, fasteners, etc.

Mechanical Standard Parts  Allows the usage of standard parts from various catalogs.

Scenes Browser  Browses the created scenes
**Analyze**

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bill Of Material</td>
<td>Generates a bill of materials from the assembly</td>
</tr>
<tr>
<td>Update</td>
<td>Updates the assembly</td>
</tr>
<tr>
<td>Constraints</td>
<td>Displays a constraint analysis window of all the constraints in the assembly</td>
</tr>
<tr>
<td>Degree(s) of freedom</td>
<td>Displays the number of movable parts and their degrees of freedom in the assembly</td>
</tr>
<tr>
<td>Dependencies</td>
<td>Displays a tree format of all the constraint dependencies of a selected component</td>
</tr>
<tr>
<td>Mechanical Structure</td>
<td>Shows the structure as the assembly sees it, this pertains to having sub-assemblies either as flexible or rigid</td>
</tr>
<tr>
<td>Compute Clash</td>
<td>Displays a Clash / Clearance computation window</td>
</tr>
</tbody>
</table>

**Note:** These are available in Part Design but they just appear in the bottom toolbar instead of a pull down menu.

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Measure Item</td>
<td>Allows you to measure a single element</td>
</tr>
<tr>
<td>Measure Between</td>
<td>Allows you to measure between elements</td>
</tr>
<tr>
<td>Measure Inertia</td>
<td>Allows you to compute an inertial analysis</td>
</tr>
<tr>
<td>Clash</td>
<td>Performs a clash analysis</td>
</tr>
<tr>
<td>Sectioning</td>
<td>Performs a section analysis</td>
</tr>
<tr>
<td>Distance and Band Analysis</td>
<td>Creates a distance analysis</td>
</tr>
</tbody>
</table>
Assembly Design Workbench

- Changes workbenches
- Selects geometry
- Product selection
- Inserts a new component
- Inserts a new product
- Inserts a new part
- Inserts an existing component
- Insets an existing component and positions it
- Replaces a component
- Allows the tree to be reordered
- Generates numbers
- Loads and unloads components
- Manages representations
- Applies a multi instantiation
- Defines a multi instantiation
- Manipulates the parts
- Snaps parts together
- Smart move
- Explodes the assembly
- Stops manipulation on clash
- Creates a scene
- Browses created scenes
- Applies a coincidence constraint
- Applies a contact constraint
- Applies an offset constraint
- Applies an angle constraint
- Fixes a component
- Fixes two components together
- Applies a quick constraint
- Flexible/rigid toggle
- Changes a constraint
- Reuses a pattern used in a part
- Creates welding symbols
- Creates text with leader
- Creates a flag note
- Splits an assembly
- Creates a hole in the assembly
- Creates a pocket in the assembly
- Performs an add operation
- Performs a remove operation
- Performs a symmetry operation
Manual Format

It is important to understand the format of the manual in order to use it most effectively. This manual is designed to be used along with an instructor; however, you will need to do a lot of reading as well, in order to fully understand ENOVIA LCA and CATIA V5. The exercises in this book will list steps for you to complete, along with explanations that try to inform you what you have just done and what you are getting ready to do. The actual steps are in bold type and the information that follows the steps is for your benefit. Anything that appears in italics refers to a message ENOVIA provides—this includes information in pull-down menus, pop-up windows and other messages.

An example of a step and its explanation is shown below (note: normally the lines will not be there):

Select a location to the right of the origin. This specifies the other end point of the line. You will continue specifying locations in order to complete your profile. It should appear similar to the diagram shown below.

As you can see, the desired action blends in with the text except that it appears in bold. The information following the step explains what that step accomplished and where you are going next. It is important for you to read this information to help in your understanding of ENOVIA and CATIA.

Also, you will find that the exercises build upon themselves. Later exercises often assume you know how to do certain steps which have been covered in earlier exercises. If you did not quite pick up what you needed to know from an exercise, you will probably wish to review it several times before moving on to the more advanced sections. As you progress through the manual, it expects that you are learning and therefore you are able to do a lot more with fewer steps. Eventually, you are expected to be able to perform actions without any steps.
Logging into ENOVIA LCA

ENOVIA utilizes a username and password database that is generally independent of the username and password you log into your computer with. Before logging into ENOVIA, you must have a username and password defined. Your account will also have an associated role. A role defines what a user can and cannot do in the database. This includes not only read and write access to the database, but also functionality access such as the ability to perform tasks in the database. All three parts of your login credentials must be supplied in the login screen before you have access to the ENOVIA database.

Take a moment and look at the login screen for ENOVIA.

![Login Screen](image)

**Username**
This will be the username provided to you. The username might, or might not, be the same as the account you logged into the computer with.

**Password**
This is the password used with your username.

**Language**
This allows you to choose between various languages used in ENOVIA.

**Host**
The host field allows you to choose between various ENOVIA servers. Generally, there will only be one host set up, however, if you have multiple ENOVIA servers running on your network, all of them can be accessible though the portal.

**Check to choose role at logon**
This allows you to choose or change your role within ENOVIA.

**Enter your username and password, then click Logon.** Once you click logon, you will be prompted for a role and workbook.
Choose a Role  This will display the list of roles available with your login ID.

Choose a Workbook  This defines the workbook project that you will utilize. When used in a group setting, this will allow your group to view the same data and make things more uniform.

Generally, the first time you log into ENOVIA, you will have to define the Role and Workbook that you want to use for this session. Anytime you wish to change roles, you will have to log out of ENOVIA and log back in. The roles defined with your user account are set up by an ENOVIA administrator. If you find that you are unable to perform functions in the database that you need to have access to, you will have to have an ENOVIA administrator adjust the roles associated with your user ID.

Choose the role **TRAINEE.ENOVIA TRAINING.DEFAULT**. This will define the default training role for your work.

Change the Workbook to **DefaultProject** if not already there. For now, the default project will be used.

Click **Apply Role and Project**. This will log you into the ENOVIA portal.
ENOVIA is a Multi-User Database

As you create objects in the database it is important to remember that this is a multi-user database. This means that everyone in your class, as well as other classes are all utilizing the same database and vault for storage. This is an advantage of ENOVIA to be able to share data between all users of the database locally and across the globe.

One important aspect to draw from this is that your book may vary slightly from what you see on the screen. The book may show that there are no product classes, when in fact you may see many product classes. This will be due to other users creating objects in the database. Keep in mind, this book is written from a blank database, hence only a few objects created by the author will exist. As you, your classmates, and other ENOVIA based courses create objects in the database, it will quickly be populated with objects similar to the one you are creating.

Another important aspect is the necessity to create unique objects in the database. This is the same principle as working with files on a computer. If you place an image on your desktop called “Funny Picture”, then you save another image on your desktop as “Funny Picture”, you will only end up with the last one saved since it would have overwritten the original. This is why you would name the new image “Funny Picture 2” or a different, unique name. ENOVIA is no different. As you create objects in the database, you will want to be sure that your object is unique to you and only you. This is why the book will ask you to create an object with your user ID. All user ID’s are unique to the database, and should not be repeated in a class session. The book will utilize the user ID TRN000, a user ID not used by anyone else. Any time you wish to create an object in the database, be sure to include your unique user ID. This is not an issue at companies utilizing ENOVIA for data storage since all objects will be identified by a unique part number. This is only a concern with the training environment since everyone will be creating the same objects simultaneously.

In order to keep consistency, the book will use the following convention when creating objects utilizing your user ID:

**Key {USERID} - Object Type for the Object ID.**

Then when you look at the picture you would see:

![ID TRN000 - Object Type](image)

In this case, any time you see {USERID}, you will use your user ID. The book will always use the user ID TRN000.

The same concept will be true as you save objects back to the database from CATIA. Every object, including the CATIA documents will have to have a unique ID.
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.

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 an Assembly Basics product class, with a configured product called {USERID} - Assembly Basics. 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 Assembly Basics product: Casing, Chuck and Bit, Gears. All three generic components will need to be created directly below the product. You do not need to nest the generic components.

Create a design action and associate it to your Assembly Basics product. The action should be called {USERID} - Assembly Basics, with a range modification effectivity of 1 to infinity, with the modification object called {USERID} - Assembly Basics Mod. 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 Assembly Basics action associated.** Remember, if you send the product to the product editor from the actions editor, it will automatically have the product 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 instance 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 Assembly Basics 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.

**Inserting Existing Components**

Inserting existing components assumes that you have already created the necessary parts an 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*  
This defines the active GCO for viewing items found only in that GCO through the product structure workbench.

*Insert new component*  
This 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*  
This inserts a new part into the product, linking it to the GCO.

*Insert existing component*  
This 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 **Product Design** folder of your **Class Models**.

While holding CTRL on the keyboard, select the part number **22-1311-144** and **22-1311-145** from the window then select *Open*. The file is 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 bit more readable.

**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.

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 to anything you do. 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 LCA toolbar.

**Turn on the EnoviaLCA toolbar by selecting on pull down menu View, then select Toolbars and finally EnoviaLCA.** This will display the ENOVIA LCA toolbar. Take a moment to look at the various icons found here.

- **Connect / Disconnect Enovia LCA** Launches ENOVIA LCA from CATIA V5
- **Set PDM Properties** Allows you to define ENOVIA save options
- **Save Data in Enovia LCA Server...** Saves modifications of instances and references back to the ENOVIA server
- **Identify in Enovia LCA...** Identifies the selected object in ENOVIA product editor
Load Children from Enovia LCA... Loads the children parts from a selected product from ENOVIA

Unload Branch from CATIA V5... Unloads a selected branch from CATIA

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

![Save in ENOVIA V5 window](image)

Let's 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*  
Checked if document product or part document is going to be saved in the database

*Structure Exposed*  
This function is disabled in ENOVIA LCA until future releases.

**Document not kept**

*Structure Exposed*  
The structure of products is saved in ENOVIA.

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.l and Right Drill Casing.l 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.

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 Assembly Basics 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 22-1311-155 and 22-1311-150 then select Open. The part will be inserted into the Assembly Basics product.

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

<table>
<thead>
<tr>
<th>Original Part Number</th>
<th>New Part Number</th>
<th>Instance Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>22-1311-155</td>
<td>{USERID}-22-1311-155</td>
<td>Drill Bit.1</td>
</tr>
<tr>
<td>22-1311-150</td>
<td>{USERID}-22-1311-150</td>
<td>Drill Chuck.1</td>
</tr>
</tbody>
</table>
Notice the Drill Chuck.1 and the Drill Bit.1 are just in 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 Chuck.1 then drag it down to the GCO Chuck and Bit. 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 Bit.1 instance into the Chuck and Bit GCO. Another link is created. These links have to be create one at a time.
Select the **Save Data in ENOVIA LCA Server** icon. Do NOT hit **OK** just yet, you need to make some changes to the document names.

<table>
<thead>
<tr>
<th>Original Document Name</th>
<th>New Document Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>22-1311-150.CATPart</td>
<td>{USERID}-22-1311-150.CATPart</td>
</tr>
<tr>
<td>22-1311-155.CATPart</td>
<td>{USERID}-22-1311-155.CATPart</td>
</tr>
</tbody>
</table>

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

**Change the document names to the following.** Select the document name then change it on the field on the right, do NOT hit Enter after each name change.
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.
Other available courses

CATIA V5 and ENOVIA

- CATIA Basic Concepts
- CATIA Part Design & Sketcher
- CATIA Assembly Design
- CATIA Drafting
- CATIA Wireframe & Surfaces
- CATIA Prismatic Machining
- CATIA Surface Machining
- CATIA Fitting Simulation & Kinematics
- CATIA Functional Tolerancing & Annotation
- CATIA Stress Analysis
- ENOVIA DMU Viewer
- ENOVIA LCA Basic Concepts
- ENOVIA LCA Advanced Concepts
- ENOVIA LCA Product Design

To enroll in any of the above courses, contact us at: (316) 978-3283
toll-free at: 1-800-NIARWSU or email: info@cadcamlab.org