# TABLE OF CONTENTS

Introduction .............................................................. 1
Generative Structural Analysis ........................................ 2
Pull-down Menus .......................................................... 3
  Insert ........................................................................ 3
  Tools ....................................................................... 4
Generative Structural Analysis Workbench .......................... 5
Bottom Toolbar Changes .................................................. 8

Preview ........................................................................... 9

Meshing .......................................................................... 19

Restraints ....................................................................... 33
  Clamp Restraints ........................................................ 33
    Clamp ..................................................................... 33
  Mechanical Restraints .................................................. 34
    Surface Slider ......................................................... 34
    Slider ..................................................................... 35
    Sliding Pivot .......................................................... 36
    Ball Join .................................................................... 37
    Pivot ....................................................................... 38
  Advanced Restraints ...................................................... 39
    Advanced Restraint ................................................... 39
    Isostatic Restraint ..................................................... 40

Loads ............................................................................. 41
  Pressures ..................................................................... 41
    Pressure ..................................................................... 41
  Forces ......................................................................... 42
    Distributed Force .................................................... 42
    Moment ..................................................................... 43
    Bearing Load ........................................................... 44
    Imported Force ........................................................ 46
    Imported Moment ..................................................... 47
  Accelerations ............................................................... 48
    Acceleration ............................................................ 48
    Rotation Force ......................................................... 49

Force Density .................................................................. 50
  Line Force Density ....................................................... 50
  Surface Force Density .................................................. 51
  Body Force ................................................................... 52
  Force Density .............................................................. 53
  Enforced Displacement ................................................... 54
  Temperature Field ......................................................... 55
  Temperature Field from Thermal Solution ......................... 56
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Restraints and Loads Exercise</td>
<td>57</td>
</tr>
<tr>
<td>Results</td>
<td>63</td>
</tr>
<tr>
<td>Computing Results</td>
<td>63</td>
</tr>
<tr>
<td>Viewing Results</td>
<td>67</td>
</tr>
<tr>
<td>Imaging Tools</td>
<td>67</td>
</tr>
<tr>
<td>Visualization Tools</td>
<td>74</td>
</tr>
<tr>
<td>Creating Sensors</td>
<td>80</td>
</tr>
<tr>
<td>Adaptivity</td>
<td>83</td>
</tr>
<tr>
<td>Managing Results</td>
<td>86</td>
</tr>
<tr>
<td>Results Validation</td>
<td>99</td>
</tr>
<tr>
<td>Color Bands</td>
<td>100</td>
</tr>
<tr>
<td>Virtual Parts</td>
<td>107</td>
</tr>
<tr>
<td>Rigid Virtual Parts</td>
<td>107</td>
</tr>
<tr>
<td>Smooth Virtual Parts</td>
<td>108</td>
</tr>
<tr>
<td>Contact Virtual Part</td>
<td>109</td>
</tr>
<tr>
<td>Rigid Spring Virtual Parts</td>
<td>110</td>
</tr>
<tr>
<td>Smooth Spring Virtual Part</td>
<td>112</td>
</tr>
<tr>
<td>Periodicity Conditions</td>
<td>114</td>
</tr>
<tr>
<td>Frequency Analysis</td>
<td>123</td>
</tr>
<tr>
<td>Distributed Mass</td>
<td>125</td>
</tr>
<tr>
<td>Mass Density</td>
<td>127</td>
</tr>
<tr>
<td>Line Mass Density</td>
<td>127</td>
</tr>
<tr>
<td>Surface Mass Density</td>
<td>128</td>
</tr>
<tr>
<td>Generative Assembly Structural Analysis</td>
<td>131</td>
</tr>
<tr>
<td>Analysis Connections</td>
<td>131</td>
</tr>
<tr>
<td>General Analysis Connection</td>
<td>131</td>
</tr>
<tr>
<td>Point Analysis Connection</td>
<td>132</td>
</tr>
<tr>
<td>Point Analysis Connection within one Part</td>
<td>133</td>
</tr>
<tr>
<td>Line Analysis Connection</td>
<td>134</td>
</tr>
<tr>
<td>Line Analysis Connection within one Part</td>
<td>135</td>
</tr>
<tr>
<td>Surface Analysis Connection</td>
<td>136</td>
</tr>
<tr>
<td>Surface Analysis Connection within one Part</td>
<td>137</td>
</tr>
<tr>
<td>Connection Properties</td>
<td>138</td>
</tr>
<tr>
<td>Face/Face Connections</td>
<td>138</td>
</tr>
<tr>
<td>Distant Connections</td>
<td>144</td>
</tr>
<tr>
<td>Welding Connections</td>
<td>149</td>
</tr>
<tr>
<td>GAS Exercise</td>
<td>153</td>
</tr>
<tr>
<td>Analysis Connections and Connection Properties</td>
<td>153</td>
</tr>
<tr>
<td>Restraints and Loads</td>
<td>173</td>
</tr>
<tr>
<td>Computing and Viewing Results</td>
<td>179</td>
</tr>
<tr>
<td>Sensors</td>
<td>185</td>
</tr>
<tr>
<td>Saving</td>
<td>189</td>
</tr>
</tbody>
</table>
# Table of Contents

**Advanced Meshing Tools** .......................................................... 191  
Advanced Meshing Tools Workbench ............................................. 192  

**Advanced Meshing Tools** .......................................................... 193  
Meshing Methods ............................................................................. 193  
  Beam Mesher .................................................................................. 193  
  Advanced Surface Mesher .............................................................. 194  
  Octree Triangle Mesher .................................................................. 196  
  Tetrahedron Filler ........................................................................... 199  
  Global Parameters ......................................................................... 202  
  Local Specifications ........................................................................ 204  
  Global Specifications ....................................................................... 211  
  Execution Tools .............................................................................. 212  

**Mesh Edition** ................................................................................ 215  
  Clean Holes ..................................................................................... 215  
  Edit Simplification .......................................................................... 215  
  Imposed Elements ............................................................................ 217  
  Remesh Domain ............................................................................... 217  
  Remove Mesh by Domain .................................................................. 218  
  Lock Domain ................................................................................... 218  
  Edit Mesh ......................................................................................... 218  
  Split Quadrangles ............................................................................ 220  

**Mesh Analysis Tools** .................................................................... 227  
  Extrude mesher with Translation .................................................... 245  
  Extrude mesher with Rotation ......................................................... 246  
  Extrude mesher with Symmetry ......................................................... 247  
  Extrude mesher along a spline ......................................................... 248  

**Welding Meshing Methods** ............................................................. 251  
  Spot Welding Connections .............................................................. 251  
  Seam Welding Connections .............................................................. 252  
  Surface Welding Connections .......................................................... 253  
  Import/Export Mesh ......................................................................... 254  

**FMS Exercise** ............................................................................... 255  

**Product Engineering Optimizer** .................................................... 283  

**Miscellaneous** ............................................................................... 293  
  Data Mapping .................................................................................. 293  
  Periodic Conditions ......................................................................... 296  
  Grouping ........................................................................................ 298  
  Thermo-Mechanical Loads .............................................................. 302  
  Visualization Transferred onto Mesh ............................................. 303  
  Self-balancing on Load set ............................................................. 308  

© Wichita State University

Table of Contents, Page iii
<table>
<thead>
<tr>
<th>Practice Problems</th>
<th>309</th>
</tr>
</thead>
<tbody>
<tr>
<td>Problem #1</td>
<td>309</td>
</tr>
<tr>
<td>Problem #2</td>
<td>311</td>
</tr>
<tr>
<td>Problem #3</td>
<td>313</td>
</tr>
<tr>
<td>Problem #4</td>
<td>315</td>
</tr>
<tr>
<td>Problem #5</td>
<td>317</td>
</tr>
<tr>
<td>Problem #6</td>
<td>318</td>
</tr>
<tr>
<td>Problem #7</td>
<td>319</td>
</tr>
<tr>
<td>Problem #8</td>
<td>320</td>
</tr>
<tr>
<td>Appendix A - Options</td>
<td>321</td>
</tr>
<tr>
<td>General - Parameters and Measure - Knowledge</td>
<td>321</td>
</tr>
<tr>
<td>Analysis &amp; Simulation - External Storage</td>
<td>322</td>
</tr>
<tr>
<td>Analysis and Simulation - General</td>
<td>323</td>
</tr>
<tr>
<td>Analysis and Simulation - Graphics</td>
<td>324</td>
</tr>
<tr>
<td>Analysis and Simulation - Post Processing</td>
<td>325</td>
</tr>
<tr>
<td>Analysis and Simulation - Quality</td>
<td>326</td>
</tr>
</tbody>
</table>
Viewing Results

The viewing results options are divided into two main parts; the imaging tools and the visualization tools.

Imaging Tools

The imaging tools will allow you to view physical changes in the model as well as providing some actual data to view. Much of the data that we are trying to obtain will not be present in the display at this time. Later, reports will be used to obtain all of the data that was created from each option that was completed.

Select the deformation icon. The part is deformed according to the restraints and forces that were placed on it. Keep in mind that the deformation is exaggerated in order to make visualizing it easier.

The deformation icon basically just displays how the part would deform based on the given restraints and loads. The actual deformation of the part based on the analysis will be grossly exaggerated. This is only a visualization tool.
Double select anywhere on the deformed model. The *Image Edition* window appears.

- **On deformed mesh**: Toggles whether or not the mesh appears deformed
- **Display free nodes**: Displays the free nodes
- **Display nodes of elements**: Displays the nodes of the elements
- **Display small elements**: Displays small elements
- **Shrink Coefficient**: Determines the amount of shrink applied to the elements
- **Selections**: Allows you to select what displays

Select *Cancel*. 
Select the von mises stress icon. The model is still deformed, but it now shows the variation in Von Mises stress patterns over the entire part.

The color scale provides the range of the stress values that are present within the model for the given conditions. This is an easy way to determine the region of maximum or minimum stress in the model.

The Von Mises stress option allows you to view Von Mises stress field patterns. These patterns represent scalar field quantities which are based on the volume distortion energy density. They are used to measure the state of stress. This option allows you to check the structural integrity of the model using Von Mises criterion.

Double select on the color scale. The Color Map Edition window appears.

- **On boundary**: Toggles whether the colors are computed based on the boundary or the overall model.
- **Number of colors**: Allows you to change the number of colors represented. The more colors that are represented, the greater the accuracy.
- **Smooth**: Determines whether the color scale transitions smoothly or not.
- **Inverse**: Inverses the color scale.
- **Imposed max/min**: Allows you to impose a minimum or maximum value.
Change the **Number of colors to 15 and select OK.** The color scale expands to fifteen colors.

**Double select on the model.** The *Image Edition* window appears.

![Image Edition window](image)

- **Type** Specifies the visualization type
- **Criteria** Specifies the criteria used for computation
- **Options**

![Visualization Options](image)

Take a look at each different *Type* of visualization.

Select *Cancel* when finished.
Select the displacement icon. The model is displayed in color coded displacement vectors.

Again, the color scale represents the amount of displacement present at a given point on the model. Notice that the displacement values in the scale are very small. This illustrates the amount of exaggeration present in the visualization.

The displacement option allows you to view the translational displacement of the model based on the loads that were applied upon it. This option is important for visualizing how the part will behave under given load conditions.

Double select on the model. The Image Edition window appears.

Try the different Type options. Select Cancel when finished.
Select the principal stress icon. It is located under the displacement icon. The model is now represented by principal stress tensor symbols.

The stress tensor symbols are also color coded to represent the principal stress at each region on the model.

The principal stress option allows you to view the principal stress tensor that is created based on the loading. The principal stress tensor shows you the directions in which the model is in a pure state of tension or compression. This allows you to visualize the directions in which the loads are acting throughout the model.

Double select on the model. The Image Edition window appears.

Look at each of the different Type options. Notice that some of the different Type options have different Criteria available as well.

Select Cancel when finished.
Select the precision icon. It is located under the principal stress or displacement icon. The model now shows the estimated local error.

The precision option allows you to visualize how accurate the results are based on distribution of energy error normal estimates for the given computation. This option provides an estimate of the validity of the computation and maps the distribution of the error on the model. In essence, this option will tell you where the greatest errors in the computation exist.

Double select on the model. The Image Edition window appears.

Look at the different Type options.

Select Cancel when done.
Visualization Tools

These options will deal with customizing how the model appears on the display based on the imaging tools that have already been applied. The visualization tools allow for much clearer visualization of the part and how it has reacted to the various restraints and loads applied to it.

Expand the Static Case Solution.1 branch in the specification tree as shown.

Notice that all but the Estimated local error case are deactivated.

Right select on Von Mises Stress in the specification tree. The following contextual menu appears.
**Select Activate/Deactivate.** The Von Mises case is activated. Now you have two cases activated and they both appear in the visualization. This can be a good way of comparing two views or simply just switching from one visualization to another.

**Deactivate the Estimated local error case in the same manner.** Only the Von Mises Stress case should be active now.

**Select the information icon and select the Von Mises case from the tree.** It is located in the bottom toolbar. The *Information* window appears.

This window displays important information about the analysis.

**Select Close.**

**Activate the Translational displacement vector case in the tree.**
Select the images layout icon from the bottom toolbar. The Images Layout window appears.

This window determines how the images will be displayed. Only the active images will be displayed.

Select **OK** to tile the images along the X axis. Both images are displayed. You may have to move one of the color tables to view everything more clearly. You may also need to adjust the **Distance** so that the two images are closer together.

Select on the *Translation displacement vector* color table to activate it. Drag it with the second mouse button to position it as shown. This allows for better positioning of active images in your display.

Select on the color table again to deactivate it.

Select the images layout icon again. Select **Default** and select **OK**. The translational displacement vector case is repositioned on the Von Mises case. The color trees stay positioned as they were.
Deactivate the *Translational displacement vector* case in the tree. Only the Von Mises case is displayed now.

Select the animate icon in the bottom toolbar. The *Animate* window appears and the part is animated from its original position through the deformation and back.

- Jumps to start
- Plays backward
- Steps backward
- Pauses
- Steps forward
- Plays forward
- Jumps to end
- Plays back and forth continuously
- Plays in one direction continuously
- Plays in one direction from start to finish

*Number of steps* Specifies how many steps will be made from start to finish

*Speed* Adjusts how fast the animation will play

Select the pause button. The animation is paused.
Change the loop mode to be in one direction from start to finish. This will make the animation play straight through and then stop at the end.

Select the play forward icon. The animation will play from the paused location to the end.

Change the Number of steps to 20. This will show twice as many steps in the animation. The animation should return to the beginning.

Select the play forward icon again. Notice the difference in the number of steps.

Adjust the speed to allow better visualization of the animation. Sometimes the animation goes too slow and some times it is too fast. The speed option allows you to adjust the animation so that it may be viewed as clearly as possible.

Select Close to end the animation.

Select the simplified representation icon and select the Von Mises case from the specification tree. The Simplified Representation window is displayed. The simplified representation option will allow you represent the model in a simplified form whenever you are manipulating it. This in turn will allow you to move things faster because CATIA does not have to calculate the representation as exactly as it normally would. This option will normally be more valuable on large complex models.

None Displays no simplified representation

Bounding Box Displays a bounding box around the model during manipulation

Compressed Reduces the accuracy of the representation

Select the Bounding Box option and select OK. Now anytime you manipulate the model, it will be represented as a bounding box.
Rotate the model around. The model should be represented as shown when you move it. As soon as you finish manipulating it, it should return to the normal representation.

Select the simplified representation icon and the Von Mises case again.

Change the option to None and select OK. This is a simple model and so the simplified representation doesn’t really help much here.
Creating Sensors

Sensors can be applied to the analysis in order to monitor certain results factors. They can be applied at any time during the analysis.

Right select on Sensor.1 in the specification tree.

A contextual menu appears.

Select the Create Global Sensor option. The Create Sensors window appears.

Select Maximum Displacement and select OK. A sensor is created to measure the maximum displacement that occurs during the analysis.
Notice that the maximum displacement during the analysis is 0.00032 inches.

Right select on Sensor.1 again. The contextual menu appears.

Select Create Global Sensor. The Create Sensors window appears.

Select Global Error Rate and select OK. The sensor is created.

Notice that the Global Error Rate of the analysis is 38.14%.

It is very important to keep track of the error of the analysis. The error is due to the many approximations that are involved in the analysis process. One of the things that you can modify to try to improve the error is the mesh itself since this is probably the biggest approximation in the process.

Double select on OCTREE Tetrahedron Mesh.1 in the specification tree as shown.
The OCTREE Tetrahedron Mesh window appears.

Change the Size to 0.25 and the Absolute sag to 0.025 and select OK. This will refine the mesh. You will need to recompute the results.

Recompute the results. This may take a little time.

Select the von mises stress icon. The part is deformed again and displays the von mises stress.

Notice that the error improved to 33.7%. This is still a significant amount of error but you see how you can modify the analysis to minimize the error.
Adaptivity

Adaptivity will allow you to improve the quality of your results by allowing the computer to compute iterations of the analysis in order to reach a defined error percentage.

Select the new adaptivity entity icon. The Global Adaptivity window appears.

Name Allows you to define the name of the adaptivity
Supports Allows you to define the supports of the operation
Solution States the name of the solution
Objective Error Allows you to define the desired error
Current Error Defines the current error

Select OCTREE Tetrahedron Mesh.1 in the specification tree to define the Supports for the adaptivity. The adaptivity will be performed on the mesh.

Key in 30 for the Objective Error. We will try to get the error down under 30%.

Select OK. Now we need to compute the analysis using adaptivity.
Select the compute with adaptivity icon. It is located under the compute icon. The Adaptivity Process Parameters window appears.

Name
- Allows you to name the computation

Iterations Number
- Specifies the maximum number of iterations allowed

Allow unrefinement
- Specifies whether or not the global sizes of the mesh parts can be modified

Deactivate global sags
- Specifies whether or not the global sags will be ignored

Minimum Size
- Allows you to specify a minimum mesh size

Sensor stop criteria
- Allows you to use a sensor to limit the computation

Sensor parameter
- Defines the sensor(s)

Tolerance
- Allows you to define a tolerance value
Key in 3 for the Iterations Number. This will limit the computation to a maximum of three iterations.

Toggle the Allow unrefinement option on. This will allow the global size of the mesh to be modified to improve quality.

Key in 0.1 for the Minimum Size. This will specify that the minimum size of the mesh cannot go lower than 0.1 inches.

Select OK. The computation with adaptivity will commence. This may take a while.

A Warnings window appears.

![Warnings Window]

This window is simply telling you that the global size of the mesh is going to be modified from 6.35mm to 18.61mm. In other words, the mesh size will be modified from 0.25in to about 0.73in.

Select OK. Now the entire analysis needs to be recomputed.

Recompute the analysis using the compute icon.

The error has been reduced to 25.8%. This is substantially less than the 38.4% that we started with. There are many extra tools to allow you to customize the analysis based on the accuracy needs.

Activate the Von Mises Stress in the specification tree.
Managing Results

The results management tools will allow you to adjust the way your results appear both in and out of CATIA. The most important aspect of the results management tools is that they enable you to export your results in an organized form.

Select the cut plane analysis icon from the bottom toolbar. The Cutting Plane window appears as well as the cutting plane on the model with the compass attached to it.

View section only  Toggles whether or not the model is shown along with the section view

Show cutting plane  Toggles whether or not the cutting plane appears or not

Your representation may appear differently than the one shown. The compass is attached to the cutting plane to allow you to manipulate the plane wherever you want to.
Using the compass, drag and rotate the cutting plane to show the following view.

The loads have been hidden in order to clarify the picture. Keep in mind that your representation won’t match this one exactly since none of the movement can be exact.

**Turn on the View section only option.** The representation should appear similar.
**Turn off both the View section only and Show cutting plane options.** The representation should appear similar.

This option may be used to better visualize any deformations that occur based on the given restraint and load conditions.

**Select Close.** Notice that the compass remains in the same position on the model. You may leave the compass there so that you will not have to move it to get this same representation next time you wish to use the cutting plane option. Otherwise, you may drag the compass down to the xyz axis in the bottom right hand of the display and release it there to properly reset it.

**Select the amplification magnitude icon in the bottom toolbar.** The Amplification Magnitude window appears.
Slide the scaling factor bar all the way to the right. This will exaggerate the amount of deformation even more by increasing the scaling factor. The model should appear as shown.

This is a way to control the amount of exaggeration in present in the display.

Select the Maximum amplitude option and key in 3 for the Length. This will deform the model even further. You may have to select in space or select the Tab key to update the selection.

Key in 0.25 for the Length. Now it is much less obvious that the model is deformed at all.

Select the Default button to return the amplitude to the default settings and select OK. This option is very helpful in order to better visualize exactly which regions of the model are deforming as well as how much they are deforming comparatively.

Select the image extrema icon in the bottom toolbar. The Extrema Creation window appears.

Global

Minimum extrema at most  Specifies the number of minimum extrema that will be selected in the entire model
Maximum extrema at most  Specifies the number of maximum extrema that will be selected in the entire model

Local

Minimum extrema at most  Specifies the number of minimum extrema that will be selected when comparing the values to the neighboring two-leveled entities

Maximum extrema at most  Specifies the number of maximum extrema that will be selected when comparing the values to the neighboring two-leveled entities

Change both of the Global values to be 2 and select OK. Both a minimum and a maximum value should appear as shown.

This tells you where the maximum and minimum Von Mises stress values were calculated as well as those max and min values.
Reports in html format may be created in order to organize all of the information present in the analysis. This is the primary way of collecting all of the data from your analysis.

Select the basic analysis report icon in the bottom toolbar. The Reporting Options window appears.

- **Output directory** specifies where the report will be saved.
- **Title of the report** allows you to title the report.
- **Add created images** allows you to add all of the images that were created for the specified analysis case.
- **Choose the analysis case(s)** allows you to select the analysis case that the report will be created for.

Specify the Results folder that you created earlier for the Output directory by selecting the browse button and defining the correct path. This will ensure that the report is saved in your Results folder.

**Key in Results Report for the Title of the report.**

**Select OK.** The report is created.
A browser will open displaying the report that was created. Be sure to scroll through it to see all of the information present.

Close the browser and bring your CATIA session back up.

Select the advanced report icon. Again, the Reporting Options window appears.
Since we just created a report, the *Output directory* is still pointing to where we saved the previous report. This is fine. A title still needs to be defined.

**Key in Advanced Results Report for the Title of the report and select OK.** The *Advanced reporting options* window appears.
The Advanced reporting options window allows you to specify which information you would like to be extracted into the report. All of the data that appears in the specification tree is listed in the left-hand window. You must move data from the left-hand window to the right in order to specify information to appear in the report.

**Double select on Static Case in the left-hand window.** The Static Case expands as shown.

Expand the Static Case Solution.1 branch. It should appear similar.

Expand the Stress branch. It should appear as shown.

Select the Stress full tensor text and select the button to move your selection into the right-hand window. This will move the selection to the right-hand window.
Move all of the data that you wish to appear in the report to the right-hand window and select the Launch Browser button. A browser will display showing all of the information that was specified to display.

Be sure to scroll through the report to see everything. Notice that the order that you moved data over to the right-hand window defined the order that the information appears in the report. This is the best way to get all of the information that you need from the analysis.

Using the File pulldown in the browser window, save the report in your Results folder as Advanced Report. By using save as in the browser, all of the files will be stored in together as one file instead of as multiple individual files. This also allows you to name the html file. By default, the html file will be named index. Take note that when you created the advanced report, it would have over written the basic report that was saved in the Results directory if you would not have saved it in the browser window. You must use this process if you want to keep both reports or you could also save them in separate folders.

Close the browser when finished and select OK in the Advanced reporting options window. The report is created and saved in your Results folder. The report can be viewed anytime by opening the html file from your Results folder. When you select OK, another browser may be launched. Simply close it.
The next thing that will be discussed is the historic of computations option. This option should not be available right now since you must make more than one computation for it to be available. Therefore, another computation needs to be done.

Select the historic of computations icon in the bottom toolbar. The Convergence of computation visualization window appears.

Notice that there are six components in the graph. The energy, the global adaptivity, the global error rate, the maximum displacement, the number of elements and the number of nodes. To change the axis for each component, simply select on the graph that you would like to view either on the graph or in the legend.

You may edit any of the graphed elements.

Double select on the energy line in the graph. The EditPopup window appears.
Graphic Attributes

*Function Name* Allows you to change the name of the function

*Show* Allows you to specify whether points, lines or both are displayed

*Line* Allows you to customize the line

*Point* Allows you to customize the points

Y Axis

*Draw with* Specifies whether there is a secondary axis or not

*Legend* Allows you to rename the function in the legend

Select Cancel and close the *Convergence of computation visualization* window.

This finishes the results section of the generative structural analysis workbench.

Save and close your document. Be sure to use *Save Management* in order to get all of the necessary components saved in your *Results* folder.
This page is intentionally left blank.
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