

THE INDEPENDENT MAGAZINE FOR CATIA®, ENOVIA AND DELMIA USERS

CATIA

solutions

MAGAZINE

NOVEMBER/DECEMBER 2002 VOLUME 6, NUMBER 6

A CONNECTPRESS, LTD.® PUBLICATION

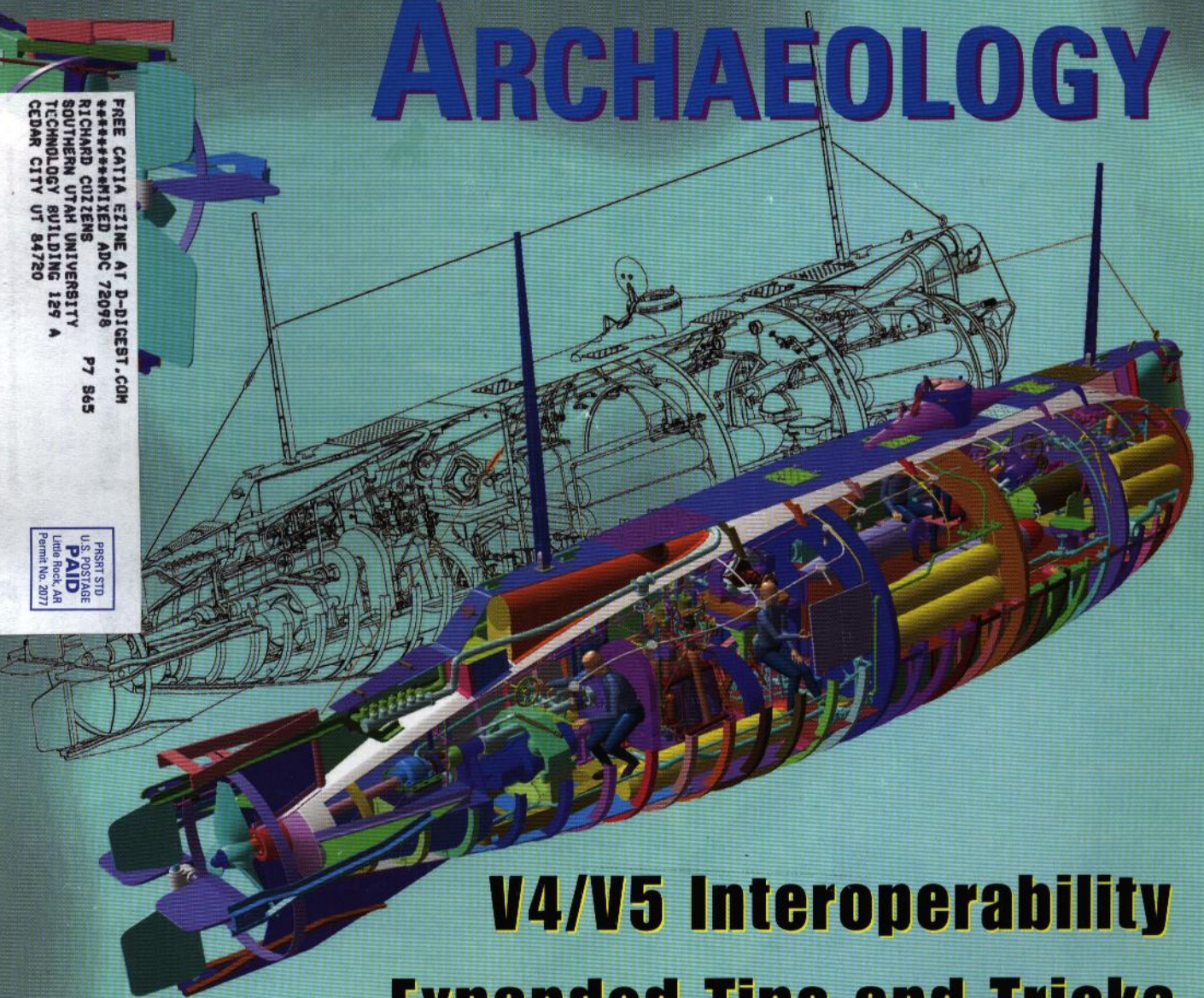
www.CatiaSolutions.com

\$19.95

DIGITAL ARCHAEOLOGY

FREE CATIA EZINE AT D-DIGEST.COM
*****MIXED ADC 72098
RICHARD COZZENS P7 865
SOUTHERN UTAH UNIVERSITY
TECHNOLOGY BUILDING 129 A
CEDAR CITY UT 84720

PSRST STD
U.S. POSTAGE
PAID
Little Rock, AR
Permit No. 2077



V4/V5 Interoperability Expanded Tips and Tricks

THE OFFICIAL MAGAZINE OF



CATIA V5 Generative Structural Analysis Workbench

by Richard Cozzens

CATIA V5's Generative Structural Analysis (GSA) Workbench is a useful and surprisingly easy workbench to operate. This lesson introduces the GSA Workbench at a basic level, but expects users to be familiar with the basic CATIA V5 workbenches as well as basic static principles. The lesson uses a simple cantilever beam, applies constraints/forces and analyzes the results.

Step One: Set all the units of measurements to the following settings: (Tools, Options, Parameters and Measurements, Units): Length (in), Mass (lb), Volume (ft3), Density (lb_in3), Area (in2), Energy (ftlbf), Force (lbf), Inertia (in4), Massic flow (lb_in), Moment (lbfxin), Pressure (psi), Angular stiffness (lbfxin_r), Linear mass (lb_in), Linear acceleration (ft_s2), Strain energy (lbfxin), Volumetric force (lbf_in3), Surfacic mass (lb_in2) and Speed (mph).

Step Two: From the Part Design Workbench, create a rectangular beam in the yz plane that is 3.5" wide and 7.25" high. Extrude the sketch 72" using the Pad tool.

Step Three: Apply aluminum material to the beam.

Step Four: Select the GSA Workbench—Start, Analysis & Simulation, Generative Structural Analysis. Activating the GSA Workbench will bring up a New Analysis Case window as well as change the Specification Tree. (See Figure 1.)

Step Five: Select the Static Analysis option, which adds a Static Case branch to the Specification Tree. The Links Manager, which appears in the Specification Tree, contains the link

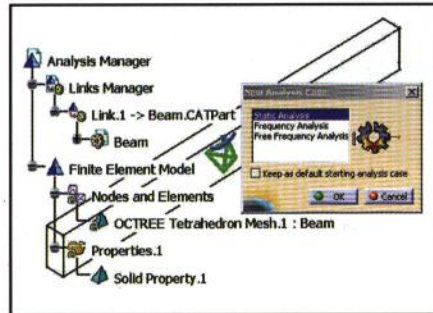


Figure 1.

to the Results and Computations files directory. Documents saved in this workbench have the *.CATAnalysis extension. The important features of a mesh are size, sag and order. The smaller the mesh size, the higher the precision will be.

Step Six: Restrain the beam. To restrain the beam as a cantilevered beam, use the **Advanced Restraint** tool. This will open the Advanced Restraint window. For the Supports section, select the front surface of the beam as shown in Figure 2. Make sure **Restrain Translation 1**, **Restrain Translation 2** and **Restrain Translation 3** options are bolded. Select OK to create the Advanced Restraint.

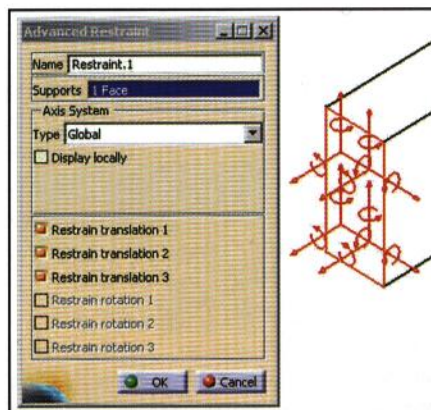


Figure 2.

Step Seven: Apply a Force. To apply a force, select the Force tool, which opens the Distributive Force window. For the Support section, select the top surface of the beam. The Support section will then read 1 Face. Type 01bf for the Force Vector X and type 100001bf for the Force Vector Z. Select OK to apply the Force. The Force symbols appear as red arrows as shown in Figure 3.

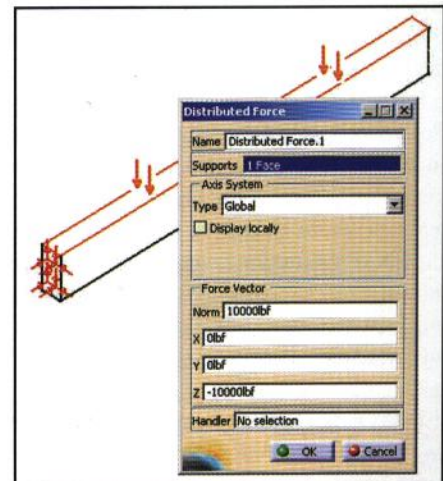


Figure 3.

Step Eight: Compute a solution. The Static Case now has the minimum number of restraints and loads to compute the Static Case Solution.1. To compute the solution, select the **Compute** tool, which opens the Compute window. Select **All**, select the **Preview** option and click OK. This will open the Computation Resources Estimation window. Since this analysis is simple, it will not require a large amount of CPU time. Select **Yes** to compute.

Step Nine: Visualize the displacement. If the computation is successful, the Image toolbar will be activated. If the

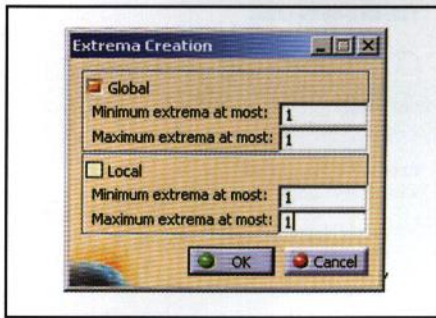


Figure 4.

computation was not successful, a singularity error most likely occurred, which generally is due to a missing restraint. Next, select the **Displacement** tool. This will activate the Analysis Tools tool bar.

Step Ten: To determine the maximum and minimum displacement of the beam, select the **Search Image Extrema** tool located in the Analysis Tools tool bar. This will open the Extrema Creation window. (See Figure 4.) Type in 1 for the Global option, Minimum extrema and Maximum extrema fields. De-select the Local option. Select the OK button to create the displacement. The Translational displacement vector Global Maximum.1 value in Figure 5 is 0.192513 inches. Your resulting value may vary slightly, depending on the defined mesh and sag.

The accuracy of the answer that the GSA Workbench computes depends directly on the values you set for the mesh and sag. If you compare 0.192513 inches (the GSA answers) with the answer derived using the traditional method, you will find they are close but not the same. To get a more accurate value, decrease the mesh and sag size. Be aware that this will require more CPU time.

The GSA Workbench is simple to apply when you know how to set up the constraints and forces. Reviewing this process using a simple beam should give users the necessary tools and knowledge to get started on more complex parts and assemblies. This brief introduction, although powerful, represents only a portion of GSA Workbench tools.

This GSA Workbench tutorial is a condensed version of the Generative Structures Analysis lesson taken from the Advanced CATIA V5 Workbook. The lesson was developed by Daniel Cozzens, an aeronautical engineer in the structural test labs at Lockheed Martin Aeronautics Co. in Marietta, GA. Richard Cozzens is a professional in residence at Southern Utah University's CAD/CAM engineering technology department and author of the CATIA V5 Workbook and Advanced CATIA V5 Workbook. For more information, visit www.catiav5workbook.com or email Cozzens at cozzens.suu.edu.

To share your comments about this article, please send an email to editorial@catiasolutions.com.

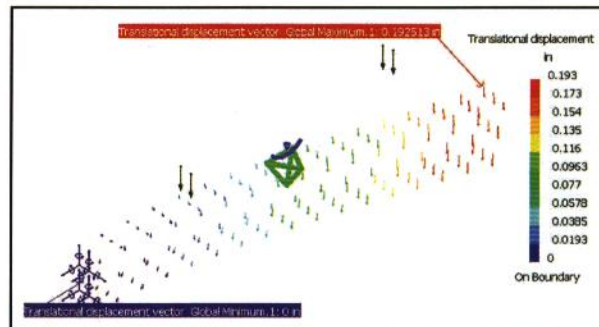


Figure 5.

Eye on the Industry

Building Bridges

(Continued from page 22)

Regarding SolidE Boolean operations: they still exist in V5, but only are used in specific cases. V5 uses a feature-based modeling approach and it is not necessary to subtract a hole, because the nature of such a feature is the removal of material. The tools in V5 inherently determine whether material is to be added or removed. It is possible to create separate Bodies and to Add, Remove and Intersect them with one another, but it is best to begin V5 with the feature-

based idea, and leave V4 methods behind.

These examples may help to explain why it would be a disservice to provide command-to-command translations. V5 is a natural transition because it utilizes the designer's feature-based thought process and is a truer reflection of real-world geometric relationships. However, it also is natural to resist change and to be apprehensive of the unknown. V5 is a new product and new practices must be learned.

Barbara Gaillard has specialized in CAD technology for 20 years, with CATIA being a main focus since 1986. Her experience spans the automotive and aerospace industries, with some exposure to consumer products and shipbuilding. She also has been a member of the advisory board for CATIA Solutions Magazine since 1999. She can be reached at bgillard@worldnet.att.net.

To share your comments about this article, please send an email to editorial@catiasolutions.com.