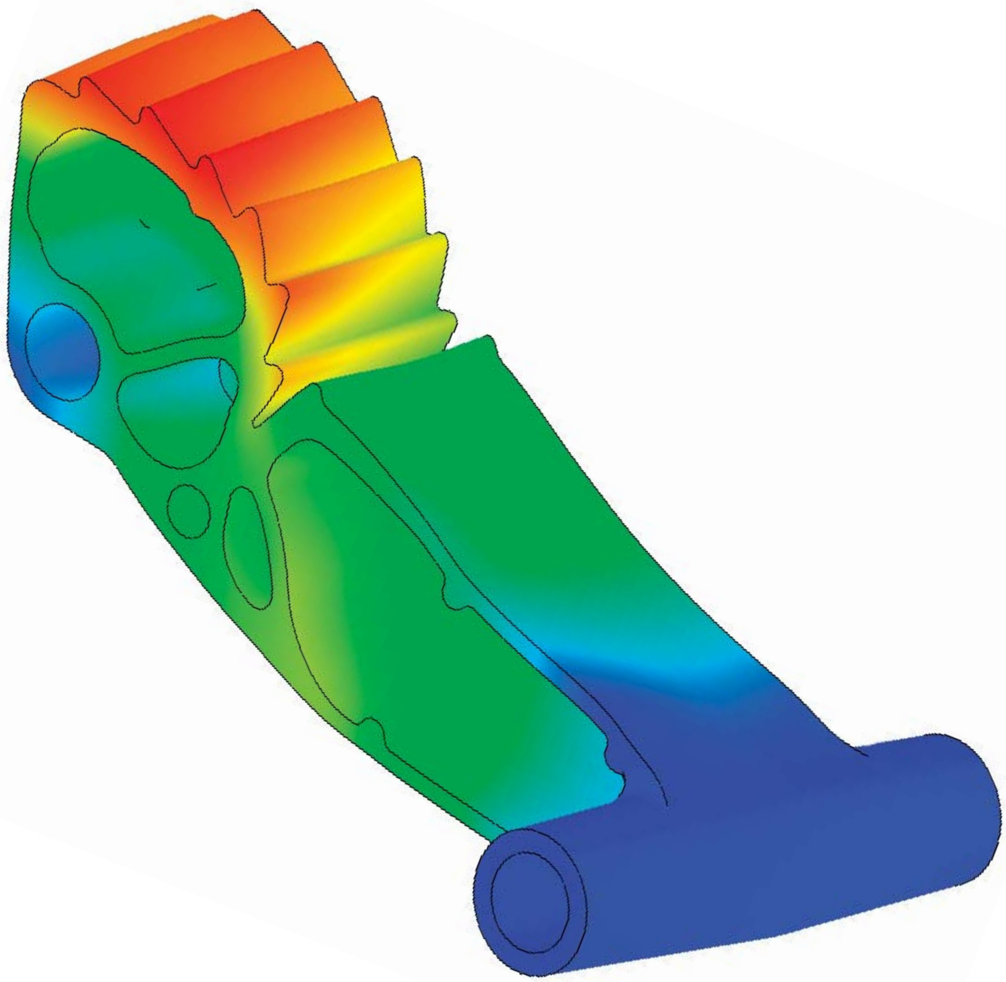
Do FEA Tools Give The Same Answers?

A Comparison of Finite Element Analysis Software

Contents



Prepared By:

Vince Adams
IMPACT Engineering Solutions, Inc.
1800 Nations Drive, Suite 121
Gurnee, IL 60031

The overall conclusion from this investigation was that, in support of the premise, each of these popular FEA tools will allow a competent user to get to approximately the same response if used with a similar amount of care and knowledge of the technology.

Executive Summary

When it comes to 'real' parts or engineering problems, concern has been expressed within the general product development industry that some Finite Element Analysis (FEA) tools are "advanced" or "expert" products while others are "basic", "first pass", or "designer level" products. The differentiation, both stated and implied, is that the latter group is acceptable for rough, seat-of-the-pants studies but if accurate results are needed, the former group of tools must be utilized.

To explore this perceived difference, IMPACT Engineering Solutions, Inc. (IMPACT) was approached by SolidWorks Corporation to perform an independent comparison of their COSMOSWorks Finite Element Analysis (FEA) tool with other, similarly marketed, FEA tools. The tools chosen for this comparison were:

COSMOSWorks Version 2005
ANSYS Version 8.1
NEi/Nastran Version 8.3
Pro/MECHANICA Wildfire 2

The primary goal of this investigation was to determine if the same or similar results would be obtained if all these tools were used with the exact same boundary conditions, properties and geometry. In nearly all the 10 stress-displacement analyses performed, COSMOSWorks was found to produce results within 10% of the average response from all the FEA tools used in this investigation. This was comparable to the consistency of the other tools.

Additionally, it should be noted that each of these tools allowed fast and easy set-up of the problems posed, allowing the user to focus on the engineering aspects of the problem versus the software interface. This represents a much-needed direction in these types of tools. Similarly, in the linear static cases, solution times were all fast enough to justify the use of these tools in an iterative design mode without fear of delay due to analysis. *When the problem called for contact conditions to be used, COSMOSWorks did complete their solution notably faster in all cases.*

The overall conclusion from this investigation was that, in support of the premise, each of these popular FEA tools will allow a competent user to get to approximately the same response if used with a similar amount of care and knowledge of the technology. In general, an engineer seeking to make a design decision regarding system performance or part failure would have come to the same conclusion with the data generated by any of these tools. It is fair to say that if these four FEA tools are used by a competent user applying the same assumption set, the results will be identical from a practical standpoint, leading to identical conclusions and design decisions.

Software Tool Details

Four software tools were evaluated in this study, COSMOSWorks, ANSYS, NEi/Nastran, and Pro/MECHANICA. IMPACT Engineering Solutions, Inc. utilizes these tools regularly and provides support and education at an advanced level for each one of them. IMPACT does not sell any of these tools. A more detailed discussion of each tool follows:

FEA SOFTWARE TESTED

- COSMOSWorks Version 2005
- ANSYS Version 8.1
- NEi/Nastran Version 8.3
- Pro/MECHANICA Wildfire 2

ANSYS

ANSYS Version 8.1, developed by ANSYS Inc, Canonsburg, PA, was used in this study. All the solutions in this study were completed using the ANSYS *Structural* configuration with the *Workbench* interface. The only exception was the Plastic Tub shell model. Due to problems importing clean geometry into *Workbench*, the part was meshed in a different tool and solved in the classic ANSYS environment. Workbench is associative with some CAD models in their native format and can import CAD files in neutral formats such as Parasolids, IGES, or STEP.

COSMOS

COSMOSWorks Version 2005, developed by SolidWorks Corporation, Concord, MA, was used in this study. COSMOSWorks is embedded inside the SolidWorks CAD tool and is available in three configurations; namely *Designer, Professional* and *Advanced Professional*. All the solutions in this study were completed using the *Designer* configuration with the exception of the modal analysis of the Muffler Guard. This required the *Professional* configuration. SolidWorks Corporation develops additional standalone COSMOS analysis products that are not exclusively linked to a single CAD software.

NEi/Nastran

NEi/Nastran for Windows Version 8.3 from Noran Engineering, Westminster, CA, was used in this study. NEi/Nastran uses *Femap v8.3* from UGS PLM Solutions as the pre- and post-processor. The *Femap* interface allows a user to import CAD models directly from their native environments as well as neutral geometry formats such as Parasolids, IGES, or STEP. Noran Engineering also develops *NeiWorks*, an FEA tool that also works within the SolidWorks environment.

Pro/MECHANICA

The *Wildfire 2* Version of Pro/MECHANICA from PTC, Needham, MA, was used in this study. All the solutions in this study were completed using the Pro/ENGINEER *Advanced Structural* and *Thermal Simulation* configuration. Pro/MECHANICA can be used as an embedded tool within Pro/ENGINEER, much like COSMOSWorks/SolidWorks or as a stand-alone analysis tool.

At the start of the investigation, the geometry for some of the parts was in native SolidWorks or Pro/ENGINEER format while others were in Parasolids, STEP, or IGES. However, the geometry for each study was brought into SolidWorks and then exported in Parasolids or Pro/ENGINEER format. This minimized the effect of translation inaccuracies between various CAD tools on the validity of the analysis comparison.

While the results from the Pro/MECHANICA solution are included in the summary tables, the average values computed consider only the h-element products, COSMOSWorks, ANSYS, and NEi/Nastran.

COSMOS, ANSYS, and NEi/Nastran use the same underlying "h-element" technology, which utilizes more, smaller elements to capture complex stress and displacement fields. Pro/MECHANICA uses "p-elements", a technology which utilizes larger elements with the internal ability to adjust the mathematical complexity of these elements to improve the solution. It is reasonable to expect a somewhat different solution based on the difference in technology. Consequently, while the results from the Pro/MECHANICA solution are included in the summary tables, the average values computed consider only the h-element products, COSMOSWorks, ANSYS, and NEi/Nastran.

Summary of Results – Comparative Tables

The following tables summarize the results of the study as described in the executive summary section of this report.

Table 1 – Displacement Response Summary for All Cases

Displacement (in)	COSMOSWorks	ANSYS	Nei/Nastran	Pro/MECHANICA
Hinge - No Contact	0.0675	0.0674	0.0686	0.0570
Hinge - Contact	0.0889	0.0708	0.0833	0.0788
Hub-Spoke	0.0042	0.0042	0.0034	0.0039
Plastic Toy	0.0426	0.0425	0.0426	0.0424
Brass Chuck	0.0058	0.0059	0.0052	0.0057
Plastic Tub	2.0670	2.0910	2.1090	
Zinc Ratchet	0.0078	0.0078	0.0079	
Plastic Cover	0.1363	0.1360	0.1360	0.1344
Aluminum Housing	0.0276	0.0274	0.0274	0.0270
Plastic Pole Restraint	0.0282	0.0262	0.0234	0.0208

Table 2 – Von Mises Stress Response Summary for All Cases

Von Mises Stress (psi)	COSMOSWorks	ANSYS	Nei/Nastran	Pro/MECHANICA
Hinge - No Contact	54840.0	54200.0	54260.0	54200.0
Hinge - Contact	53730.0	54780.0	53980.0	46260.0
Hub-Spoke	53000.0	53000.0	50000.0	43000.0
Plastic Toy	9810.0	9815.0	9828.0	11070.0
Brass Chuck	270000.0	277000.0	307000.0	273000.0
Plastic Tub	7640.0	6450.0	6615.0	
Zinc Ratchet	97040.0	98700.0	92800.0	
Plastic Cover	5845.0	4150.0	5525.0	6730.0
Aluminum Housing	23000.0	23000.0	22500.0	21570.0
Plastic Pole Restraint	7150.0	7700.0	7180.0	6760.0

Table 3 – Modal Response Summary for Muffler Guard

Modal Response		COSMOSWorks	ANSYS	Nei/Nastran	Pro/MECHANICA
Muffer Guard	Mode 1	99	99	99	102
	Mode 2	168	167	168	173
	Mode 3	205	205	206	210
	Mode 4	381	382	381	389

Detailed Sample Problem Descriptions and Results

There were ten (10) sample problems used in this study. Nine (9) of these were linear static or static with contact solutions and one was a natural frequency (modal) solution.. The problems used are listed below:

1. Hinge Leaf
   a. No Contact
   b. With Contact
2. Bicycle Hub-Spoke System
3. Plastic Toy
4. Brass Chuck
5. Plastic Tub
6. Zinc Ratchet
7. Plastic Cover
8. Aluminum Housing
9. Plastic Pole Restraint
10. Muffler Guard

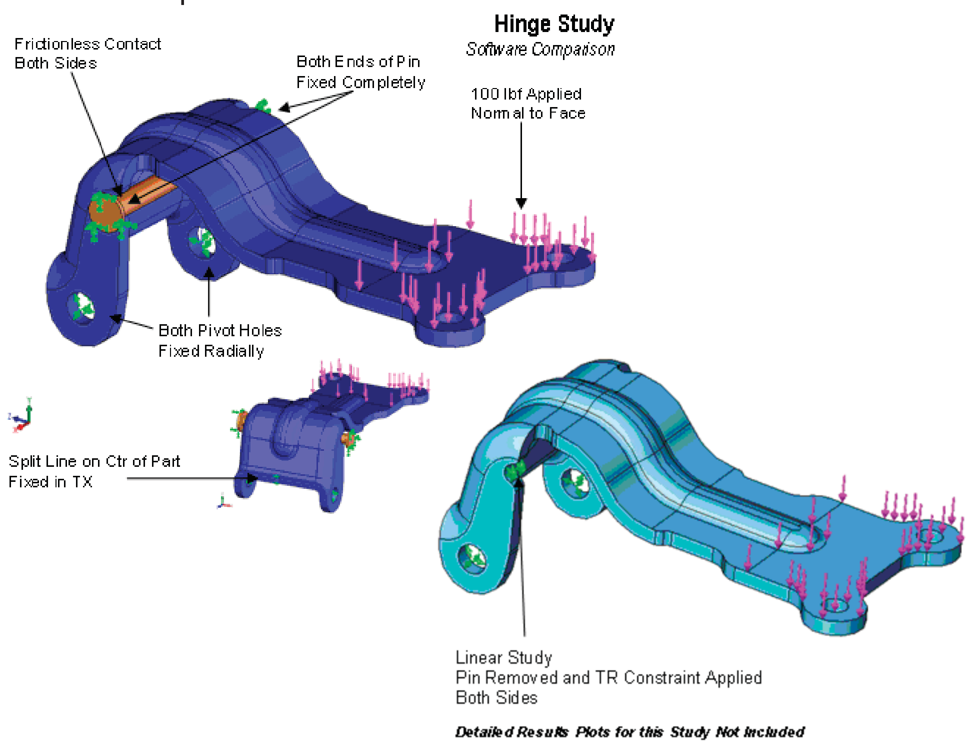
These problems are described in more detail on the following pages along with the detailed results from each analysis tool. Additionally, any observations on the problem or the way any of the software tools being studied handled that problem follow the summary. All stresses are reported in pounds per square inch (psi) and all displacements are reported in inches (in).

Hinge Leaf

This study highlights how important the way a code resolves contact is to the final results. In the "No Contact" case, the variation in the stress from one tool to the other is much less than when contact is introduced. The Hinge is intentionally forced to slide about the Pin. This will be difficult to resolve if software doesn't update contact conditions to reflect the changing geometry.

This study highlights how important the way a code resolves contact is to the final results. This will be difficult to resolve if software doesn't update contact conditions to reflect the changing geometry.

Problem Set-Up



Summary of Results

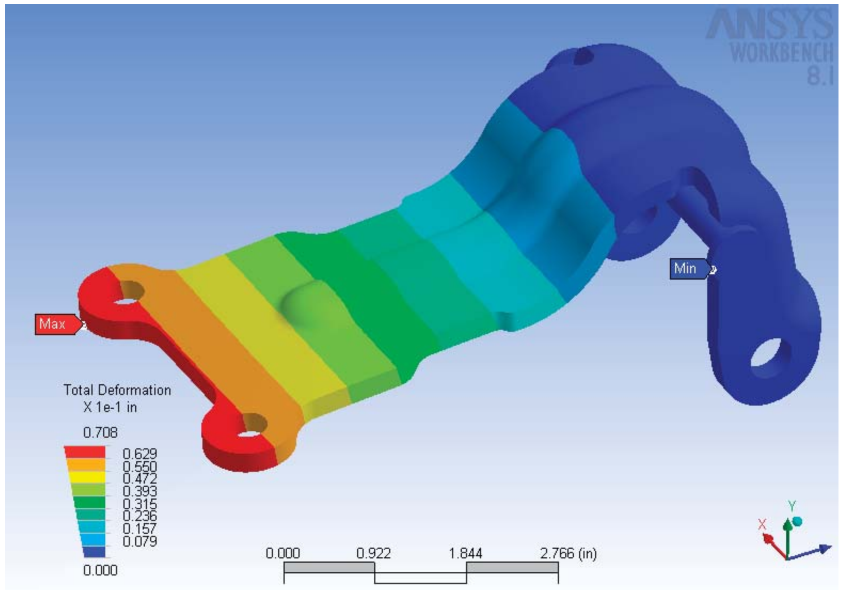
Hinge - No Contact

	Max. Displacement (in)	Max. Von Mises Stress (psi)
COSMOSWorks	0.0675	54840.0
ANSYS	0.0674	54200.0
NEi/Nastran	0.0686	54260.0
Pro/MECHANICA	0.0570	54200.0
Average	0.0678	54433.3
	Variation from Average (%)	
COSMOSWorks	-0.49%	0.75%
ANSYS	-0.64%	-0.43%
NEi/Nastran	1.13%	-0.32%

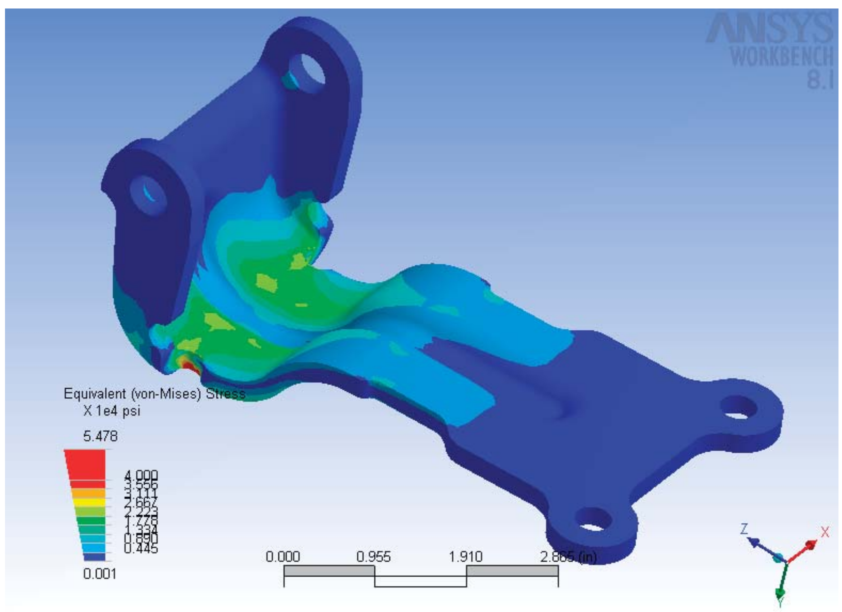
Hinge - Contact

	Max. Displacement (in)	Max. Von Mises Stress (psi)
COSMOSWorks	0.0889	53730.0
ANSYS	0.0708	54780.0
NEi/Nastran	0.0833	53980.0
Pro/MECHANICA	0.0788	46260.0
Average	0.0810	54163.3
	Variation from Average (%)	
COSMOSWorks	9.75%	-0.80%
ANSYS	-12.59%	1.14%
NEi/Nastran	2.84%	-0.34%

Results Plot



Deformation plot from ANSYS WORKBENCH 8.1



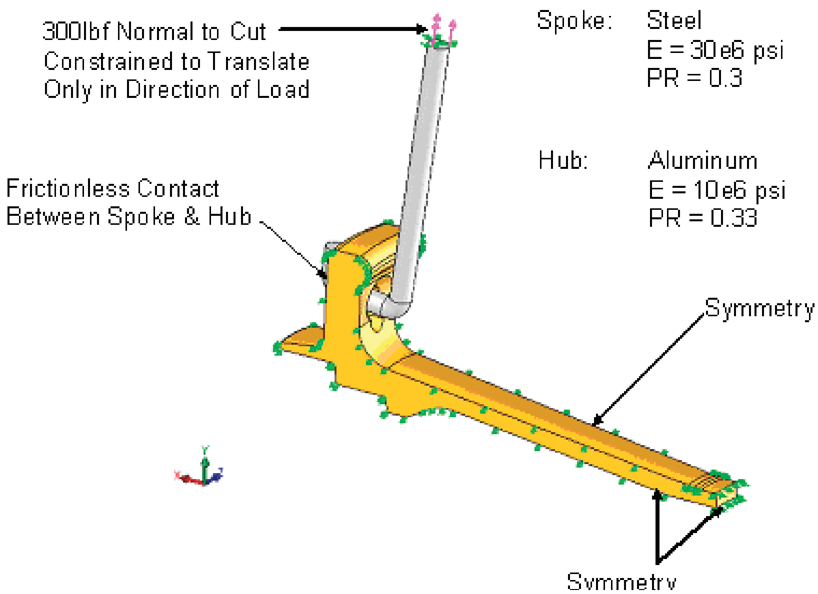
Stress plot from ANSYS WORKBENCH 8.1

Bicycle Hub-Spoke System

This turned out to be the most difficult problem in the test suite. Referring to the problem set-up image, the Spoke is initially contacting on the outer edge of the hole. However, as it loads up, it rocks due to the deformation of the shaft so that the in the final solution, it is contacting the inner edge of the hole. Note, that the Pro/MECHANICA stress solution is 20% different than the other stress solutions. One explanation for this is that COSMOSWorks, ANSYS and NEi/Nastran use a Large Displacement solution which is truly nonlinear and will update the contact conditions for changing orientation and position of the various contact pairs (created between elements and nodes of opposing contacting surfaces.) It is believed that Pro/MECHANICA does not update the contact conditions in this manner thus generating a fictitious load distribution in the system.Another observation is that while the NEi/Nastran and ANSYS solutions took 3-5 hours to solve, the COSMOSWorks Large Displacement solution took less than 15 minutes.

Another observation is that while the NEi/Nastran and ANSYS solutions took 3-5 hours to solve, the COSMOSWorks Large Displacement solution took less than 15 minutes.

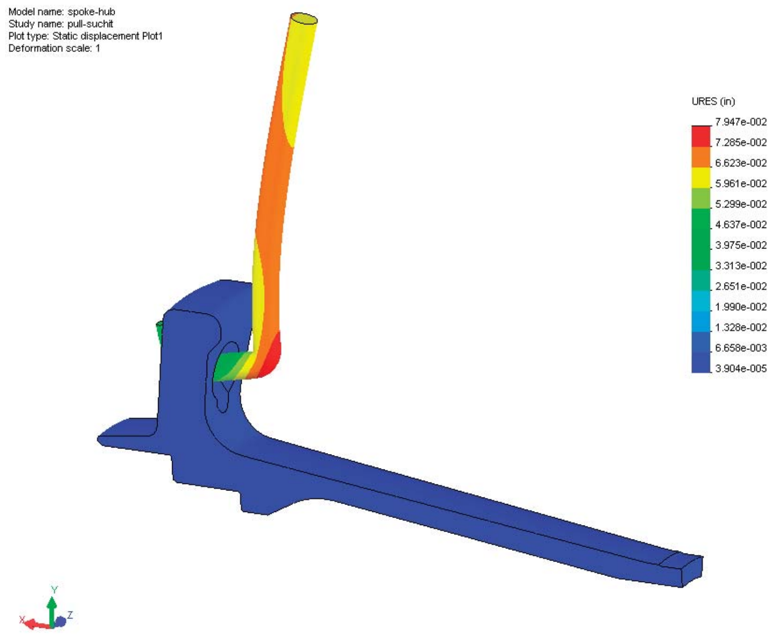
Problem Set-Up



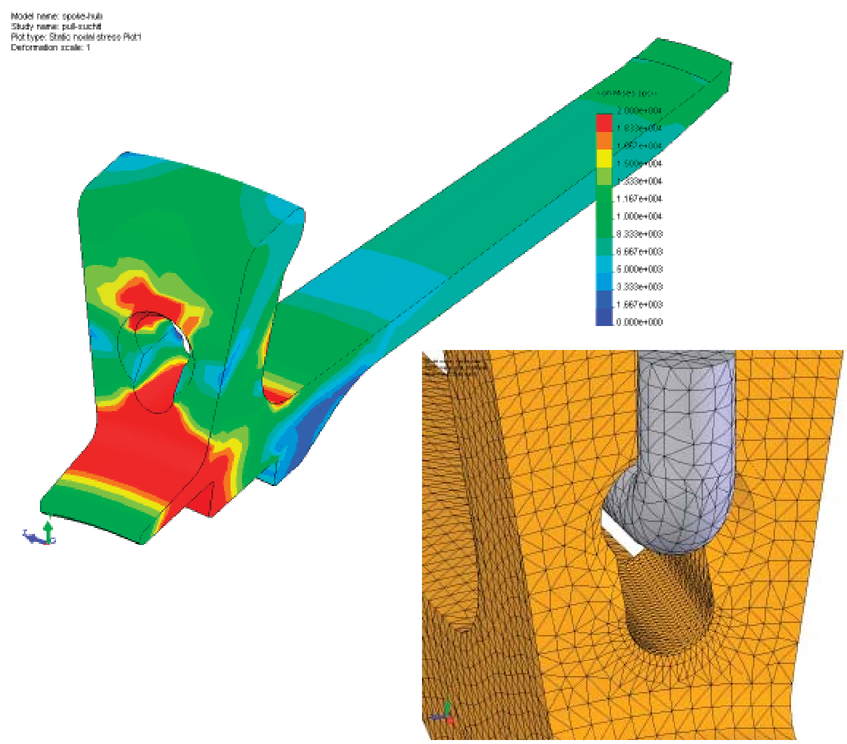
Summary of Results

	Max. Displacement (in)	Max. Von Mises Stress (psi)
COSMOSWorks	0.0042	53000.0
ANSYS	0.0042	53000.0
NEi/Nastran	0.0034	50000.0
Pro/MECHANICA	0.0039	43000.0
Average	0.0039	52000.0
	Variation from Average (%)	
COSMOSWorks	6.78%	1.92%
ANSYS	6.78%	1.92%
NEi/Nastran	-13.56%	-3.85%

Results Plot



Displacement plot from COSMOSWorks 2005

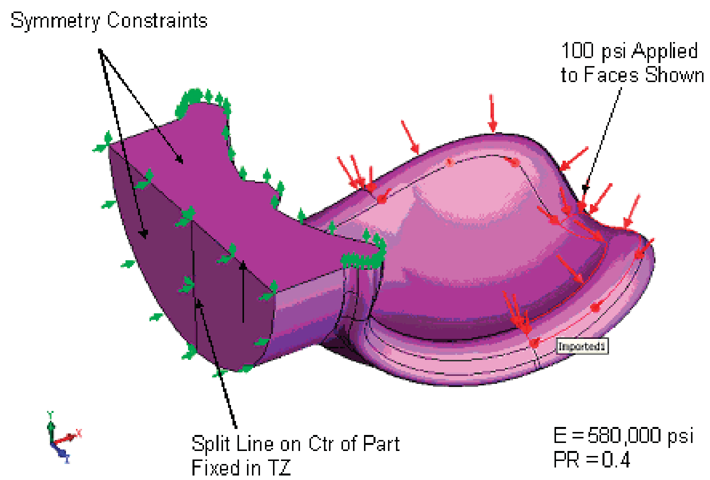


Stress plot and mesh from COSMOSWorks 2005

Plastic Toy

The noticeably higher stress solution calculated by Pro/MECHANICA is most likely due to the difference in element technologies. However, the three h-element solutions were converged over multiple iterations such that the stress presented is their final response.

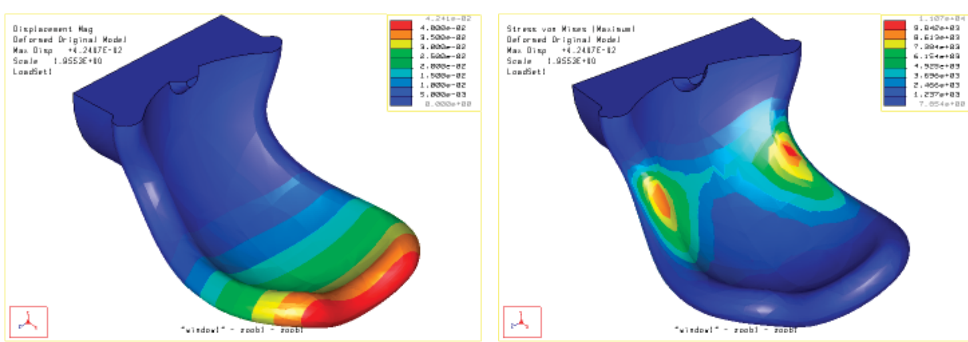
Problem Setup



Summary of Results

	Max. Displacement (in)	Max. Von Mises Stress (psi)
COSMOSWorks	0.0426	9810.0
ANSYS	0.0425	9815.0
NEi/Nastran	0.0426	9828.0
Pro/MECHANICA	0.0424	11070.0
Average	0.0426	9817.7
	Variation from Average (%)	
COSMOSWorks	0.08%	-0.08%
ANSYS	-0.16%	-0.03%
NEi/Nastran	0.08%	0.11%

Results Plot

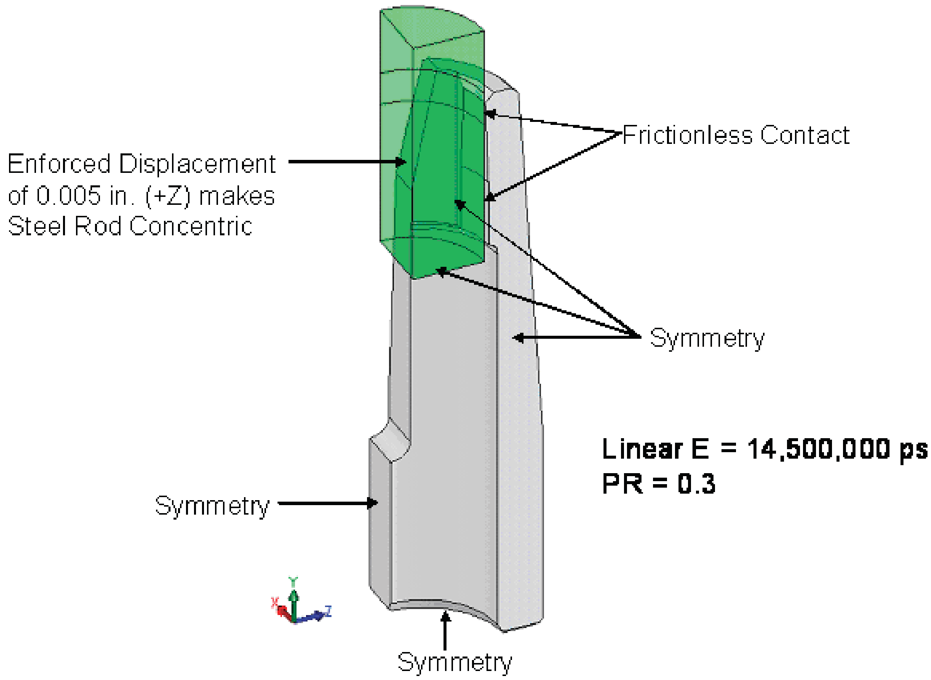


Displacement and stress plots from Pro/MECHANICA Wildfire 2

Brass Chuck

In this example, the contact initiates at the top of the Chuck but as it opens up, the bottom edge of the contact pad engages the Pin. The stress solution in the notch area required multiple convergence passes and highlighted the difference approaches each of these tools provide for implementing local mesh refinement. Albeit different, all were satisfactory for the task in this case.

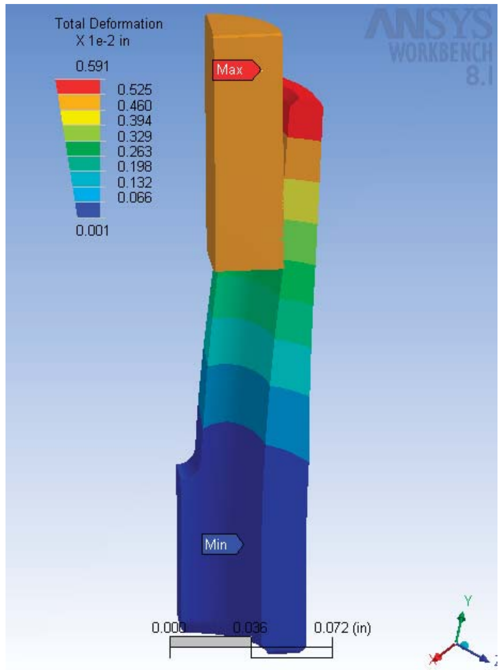
Problem Setup



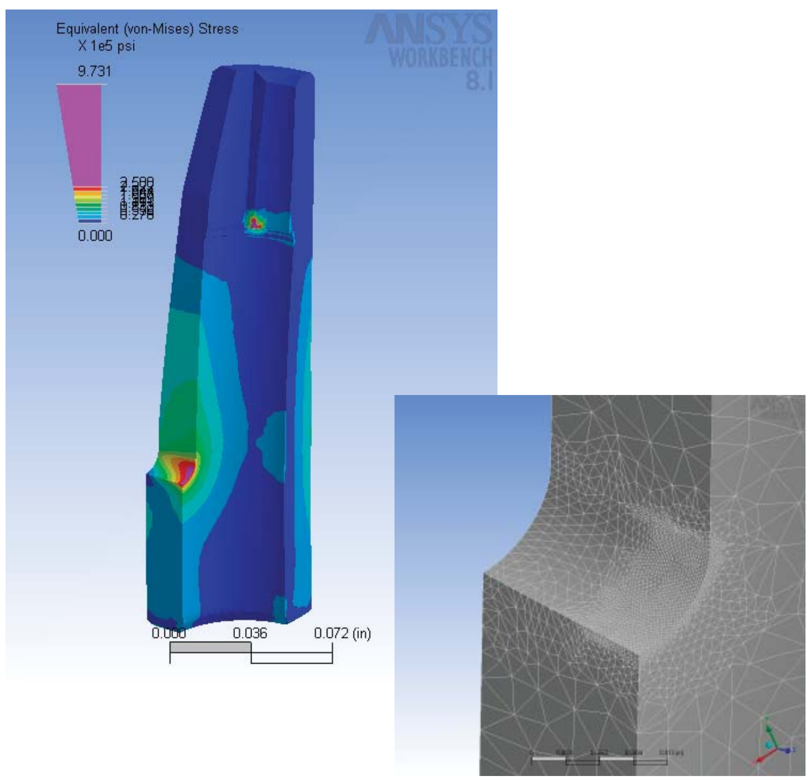
Summary of Results

	Max. Displacement (in)	Max. Von Mises Stress (psi)
COSMOSWorks	0.0058	270000.0
ANSYS	0.0059	277000.0
NEi/Nastran	0.0052	307000.0
Pro/MECHANICA	0.0057	273000.0
Average	0.0056	284666.7
	Variation from Average (%)	
COSMOSWorks	2.37%	-5.15%
ANSYS	5.22%	-2.69%
NEi/Nastran	-7.60%	7.85%

Results Plot



Displacement plot from ANSYS WORKBENCH 8.1



Stress plot and mesh from ANSYS WORKBENCH 8.1

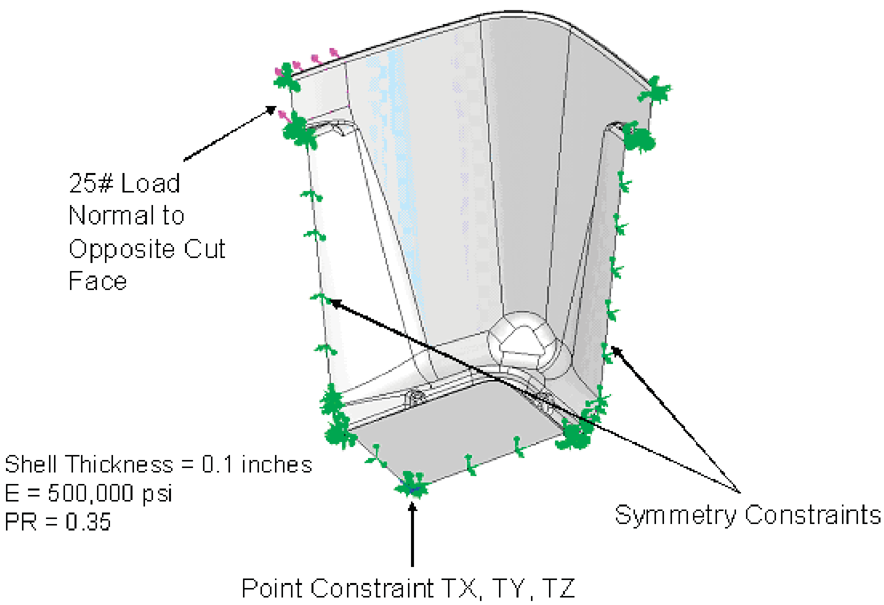
Plastic Tub

The IGES geometry that existed for this part did not read cleanly into either Pro/ENGINEER or in the standalone Pro/MECHANICA interface. A reasonable attempt was made to heal the geometry or stitch over gaps manually but it became too time consuming, therefore Pro/MECHANICA was not used for this problem

Both ANSYS and NEi/Nastran solutions used linear, or 4-noded, quadrilateral elements for the solution while COSMOSWorks used a 2nd-order, 6-noded, triangular elements. On a positive note the higher order triangular mesh in COSMOS converged to a higher stress value than the lower order quadrilateral mesh.

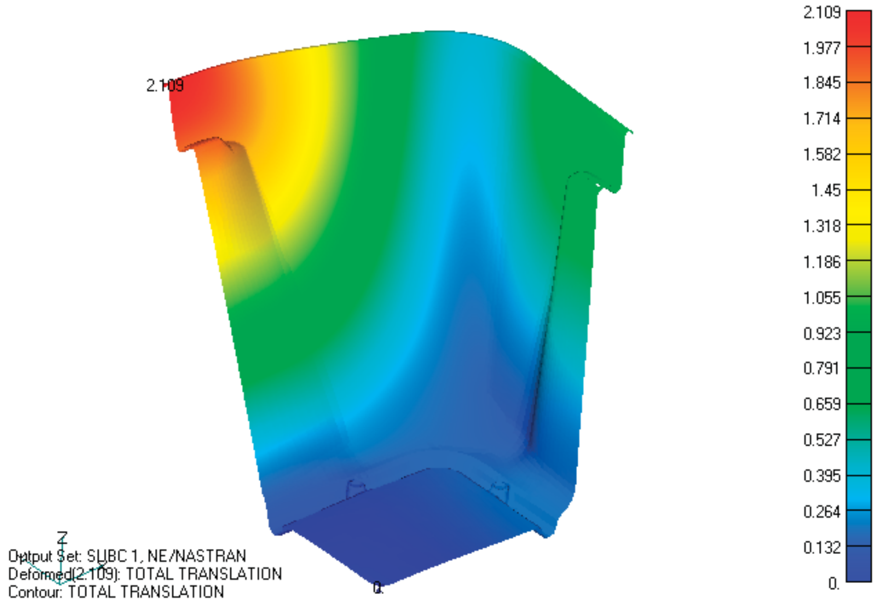
On a positive note the higher order triangular mesh in COSMOS converged to a higher stress value than the lower order quadrilateral mesh.

Problem Setup

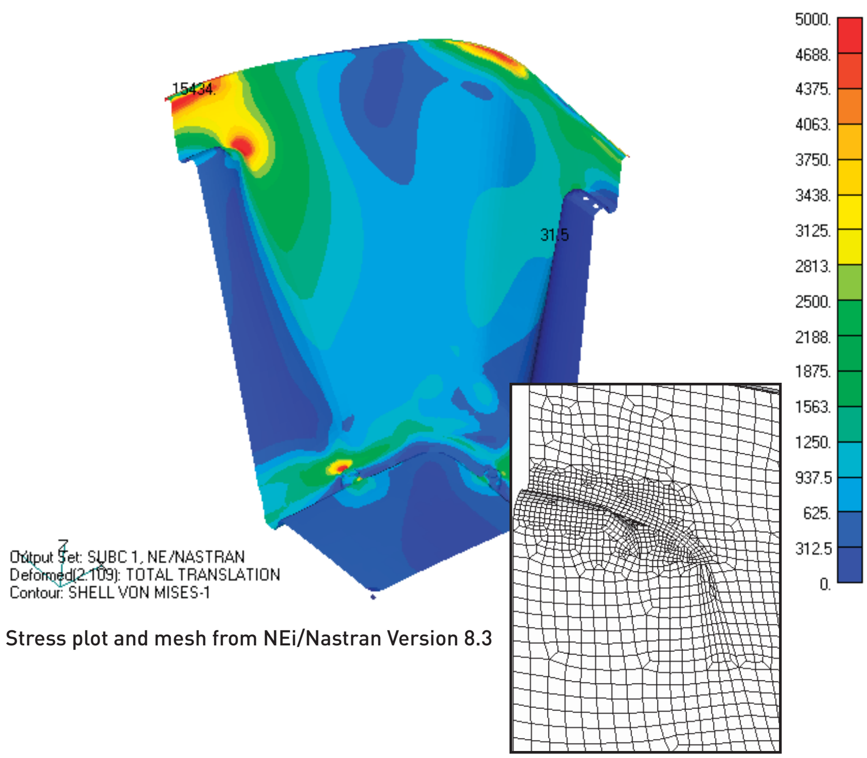


Summary of Results

	Max. Displacement (in)	Max. Von Mises Stress (psi)
COSMOSWorks	**2.0670**	**7640.0**
ANSYS	**2.0910**	**6450.0**
NEi/Nastran	**2.1090**	**6615.0**
Pro/MECHANICA		
Average	2.0890	6901.7
	Variation from Average (%)	
COSMOSWorks	-1.05%	10.70%
ANSYS	0.10%	-6.54%
NEi/Nastran	0.96%	-4.15%



Displacement plot from NEi/Nastran Version 8.3



Stress plot and mesh from NEi/Nastran Version 8.3

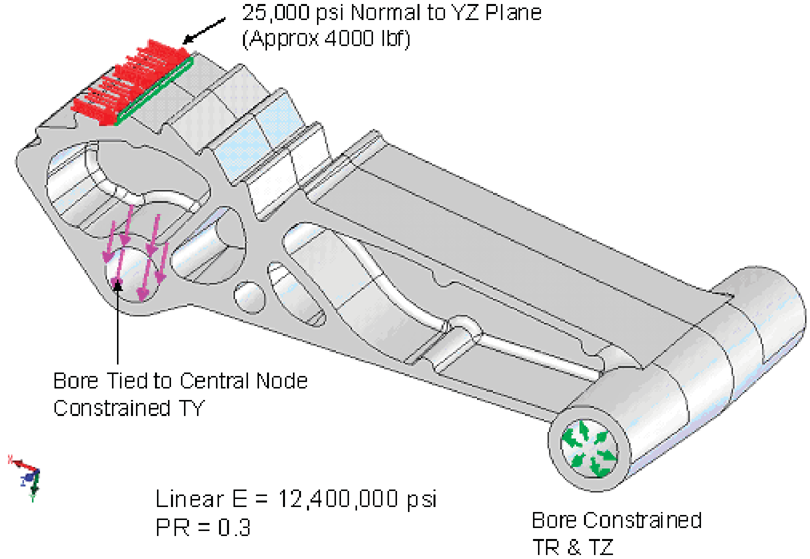
Zinc Ratchet

Again, the IGES geometry that existed for this part did not read cleanly into either Pro/ENGINEER or the standalone Pro/MECHANICA interface. A reasonable attempt was made to heal the geometry or stitch over gaps manually but it became too time consuming. Therefore Pro/MECHANICA was not used for this problem

An interesting boundary condition in this problem was tying the front bore to a central point constrained in just Y-direction. This was extremely easy to do in COSMOSWorks using the Remote Load/Constraint option. It was fairly easy in NEi/Nastran using a single rigid RBE3 element. However, ANSYS version 8.1 or Workbench did not have an easy way to apply this constraint in the user interface. A few lines of classic ANSYS pre-processing commands were used. ANSYS version 9.0 now supports this constraint directly in the user interface.

An interesting boundary condition in this problem was tying the front bore to a central point constrained in just Y-direction. This was extremely easy to do in COSMOSWorks using the Remote Load/Constraint option. It was fairly easy in NEi/Nastran using a single rigid RBE3 element.

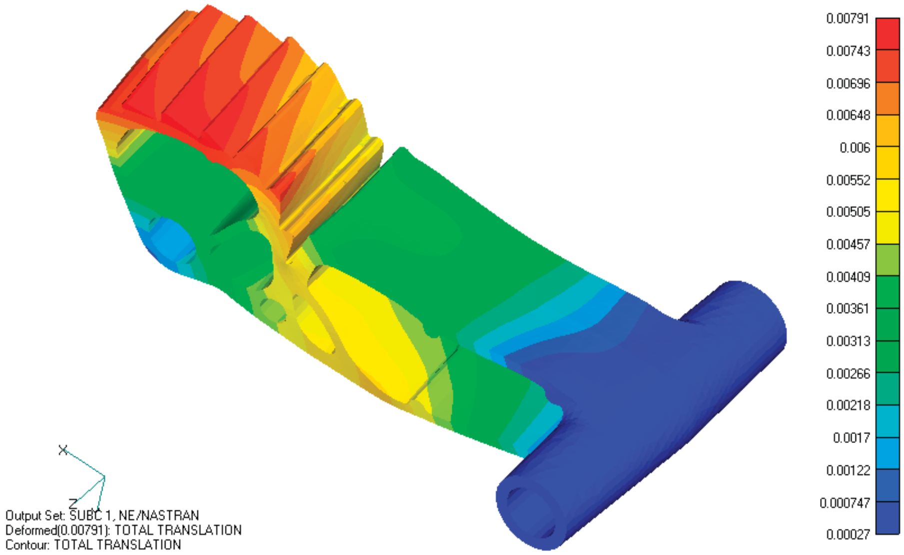
Problem Setup



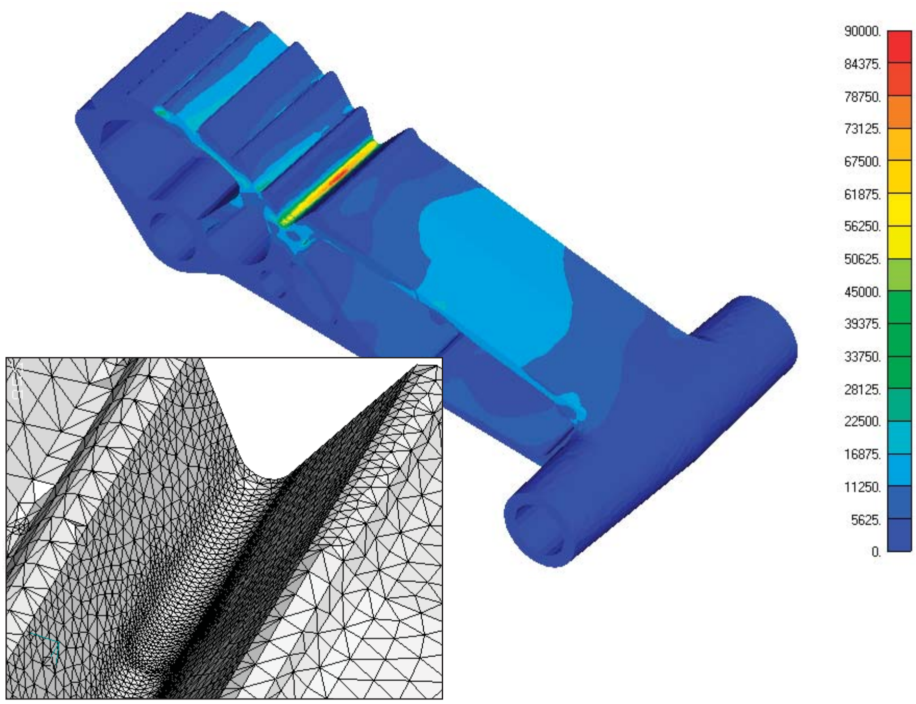
Summary of Results

	Max. Displacement (in)	Max. Von Mises Stress (psi)
COSMOSWorks	0.0078	97040.0
ANSYS	0.0078	98700.0
NEi/Nastran	0.0079	92800.0
Pro/MECHANICA		
Average	0.0078	96180.0
	Variation from Average (%)	
COSMOSWorks	-0.43%	0.89%
ANSYS	-0.43%	2.62%
NEi/Nastran	0.85%	-3.51%

Results Plot



Displacement plot from NEi/Nastran Version 8.3

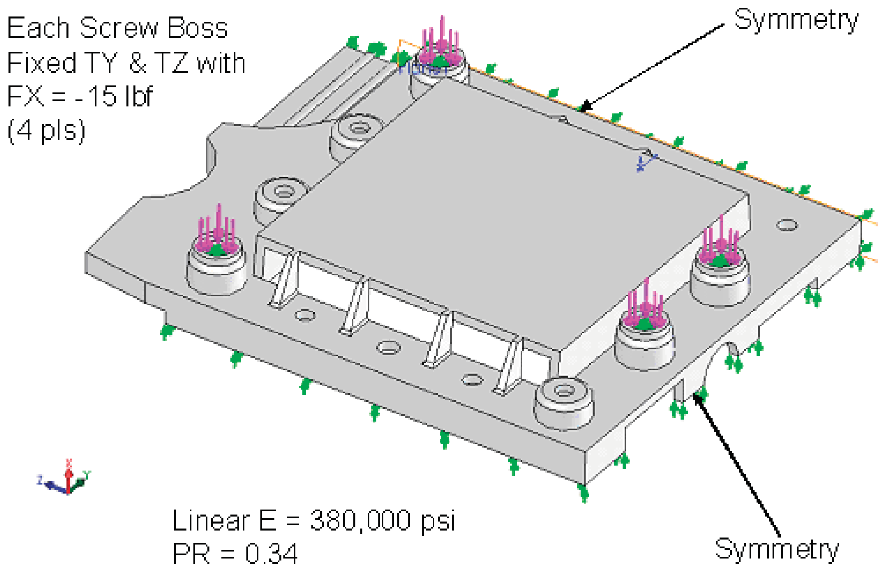


Stress plot and mesh from NEi/Nastran Version 8.3

Plastic Cover

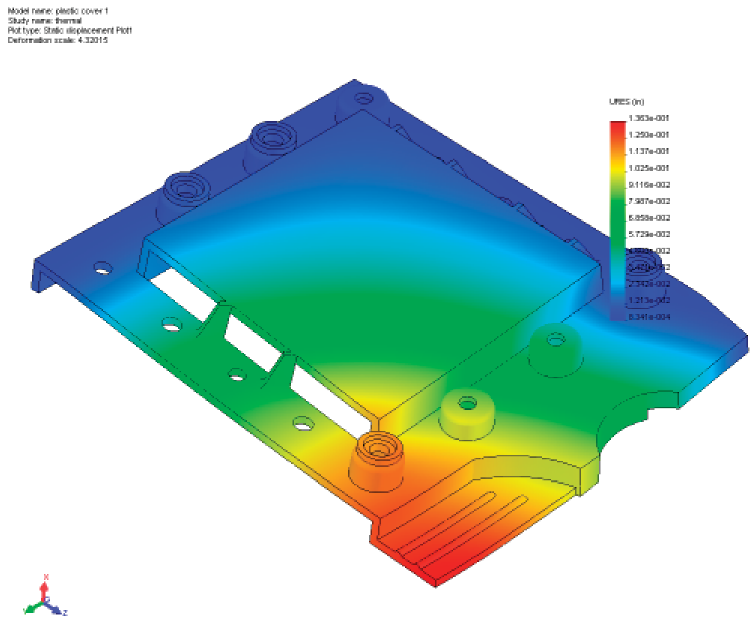
This case warranted the most convergence passes of all the problems in the study.

Problem Setup

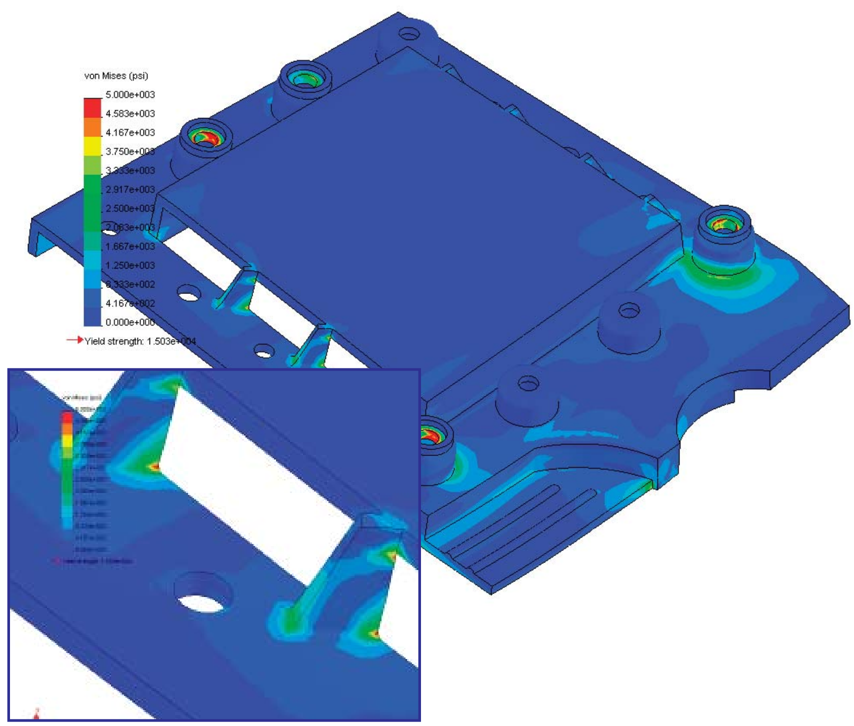


Summary Results

	Max. Displacement (in)	Max. Von Mises Stress (psi)
COSMOSWorks	**0.1363**	**5845.0**
ANSYS	**0.1360**	**4150.0**
NEi/Nastran	**0.1360**	**5525.0**
Pro/MECHANICA	**0.1344**	**6730.0**
Average	0.1361	5173.3
	Variation from Average (%)	
COSMOSWorks	0.15%	12.98%
ANSYS	-0.07%	-19.78%
NEi/Nastran	-0.07%	6.80%



Displacement plot from COSMOSWorks 2005

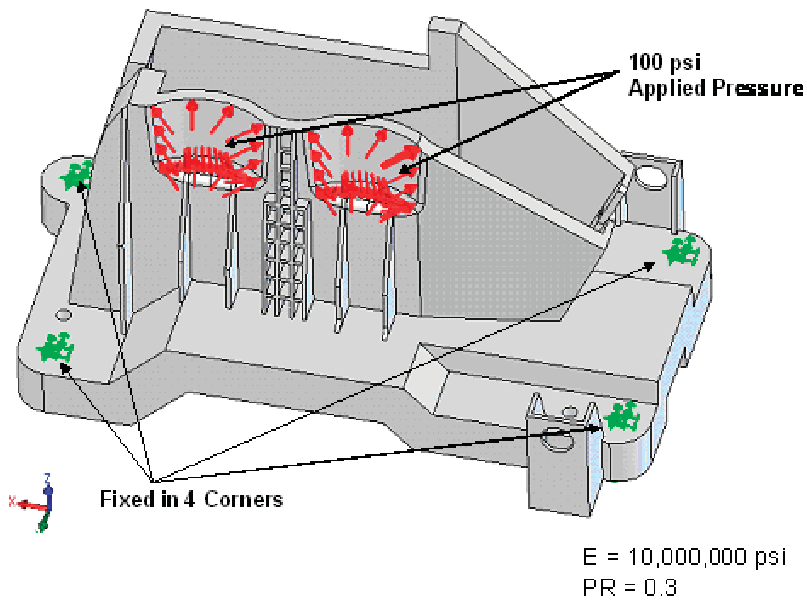


Stress plot and detail from COSMOSWorks 2005

Aluminum Housing

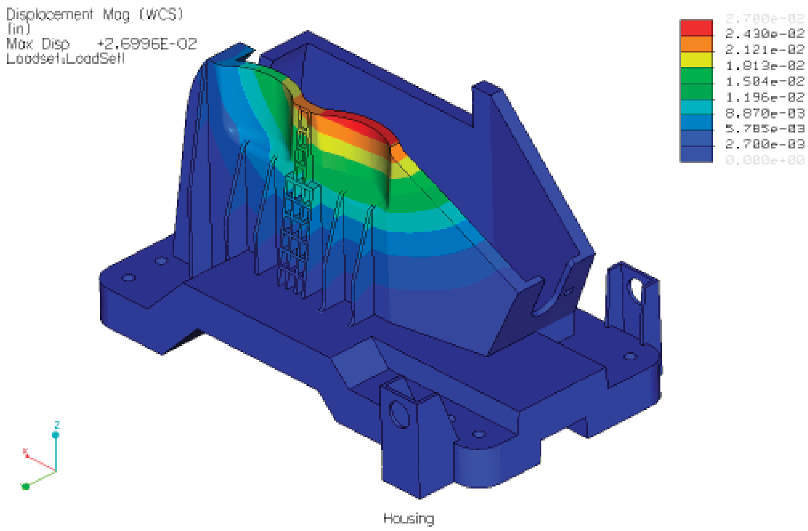
As evidenced by the consistency of the results, this was one of the most straightforward problems in the study.

Problem Setup

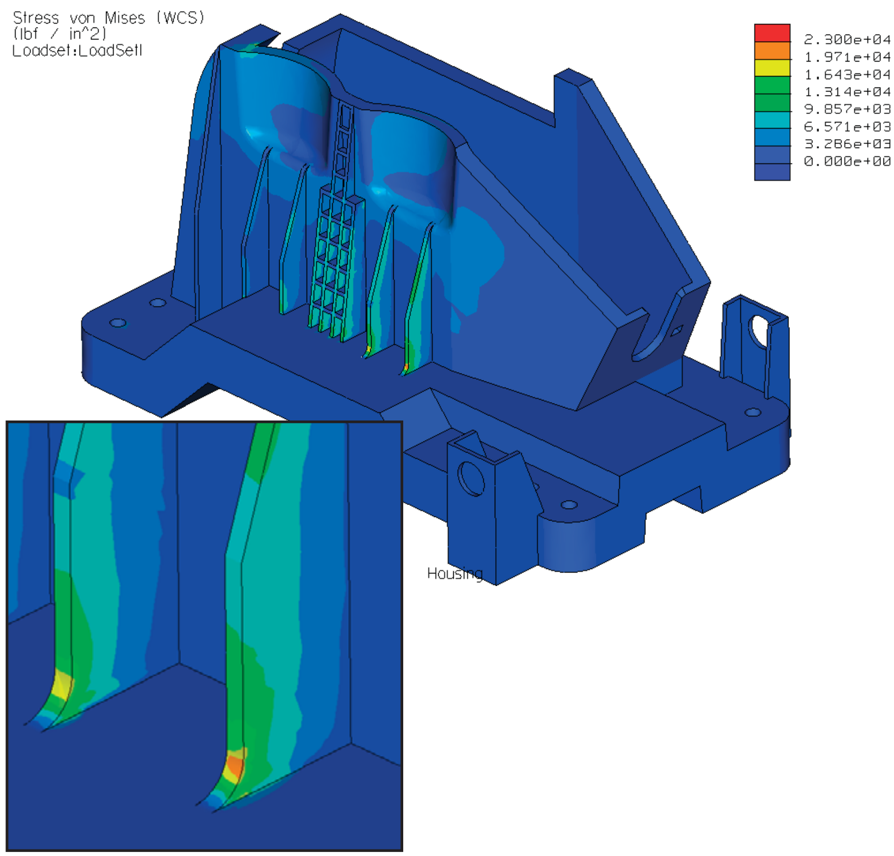


Summary of Results

	Max. Displacement (in)	Max. Von Mises Stress (psi)
COSMOSWorks	0.0276	23000.0
ANSYS	0.0274	23000.0
NEi/Nastran	0.0274	22500.0
Pro/MECHANICA	0.0270	21570.0
Average	0.0275	22833.3
	Variation from Average (%)	
COSMOSWorks	0.49%	0.73%
ANSYS	-0.24%	0.73%
NEi/Nastran	-0.24%	-1.46%



Displacement plot from NEi/Nastran Version 8.3

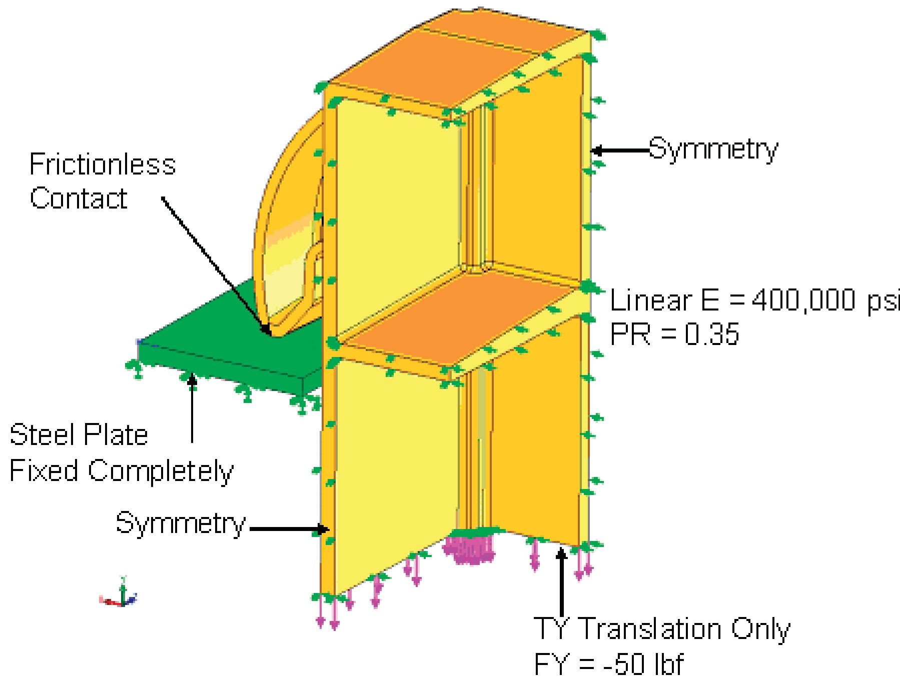


Stress plot from NEi/Nastran Version 8.3

Plastic Pole Restraint

This was a 'well behaved' contact problem. Regions that come into contact initially stay in contact without much sliding.

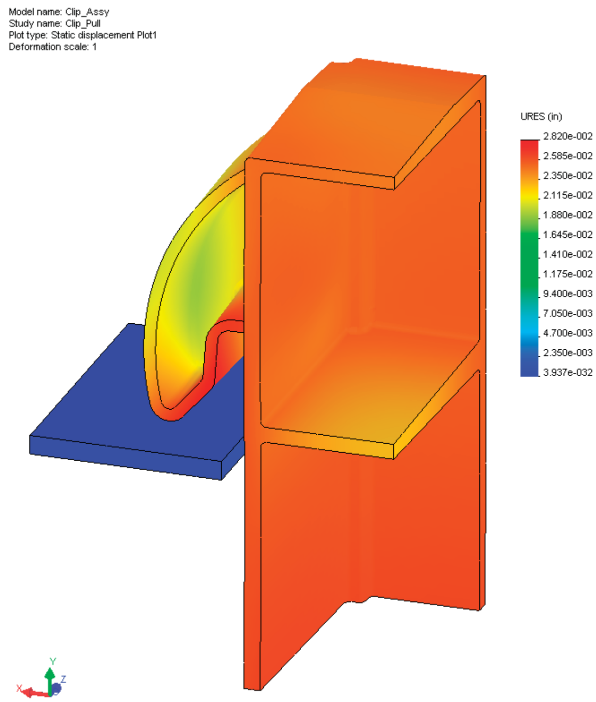
Problem Setup



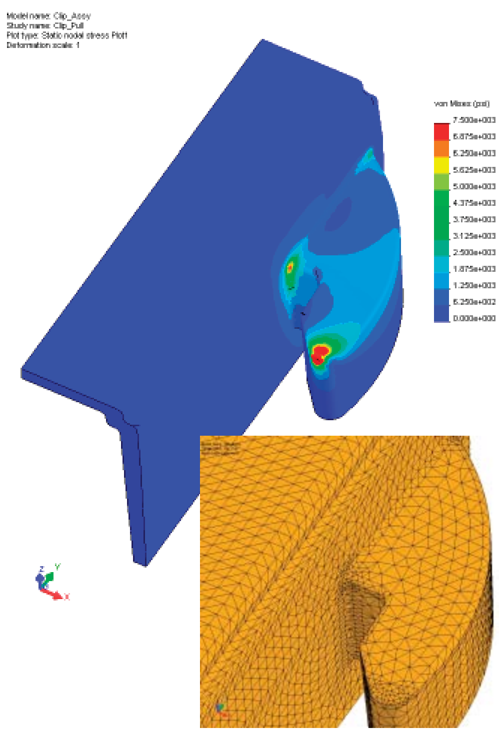
Summary Results

	Max. Displacement (in)	Max. Von Mises Stress (psi)
COSMOSWorks	0.0282	7150.0
ANSYS	0.0262	7700.0
NEi/Nastran	0.0234	7180.0
Pro/MECHANICA	0.0208	6760.0
Average	0.0259	7343.3
	Variation from Average (%)	
COSMOSWorks	8.74%	-2.63%
ANSYS	1.03%	4.86%
NEi/Nastran	-9.77%	-2.22%

Results Plot



Displacement plot from COSMOSWorks 2005

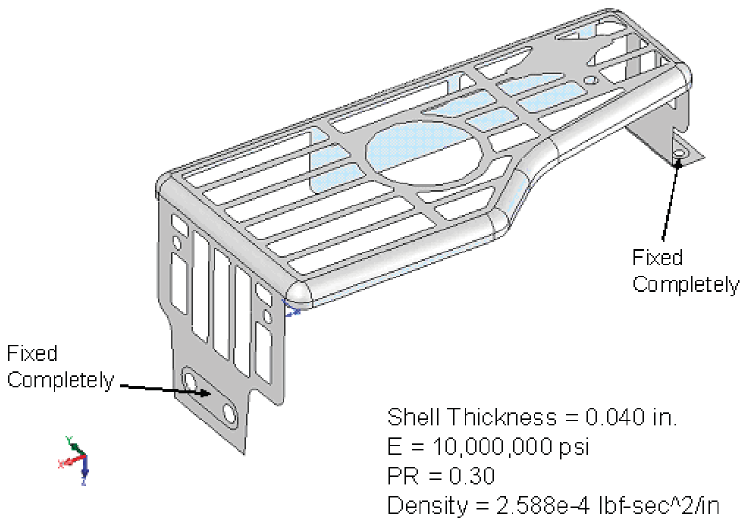


Stress plot and mesh from COSMOSWorks 2005

Muffler Guard

All mode shapes, across the four software packages, were identical. There was very little variation in the problem set-up or response.

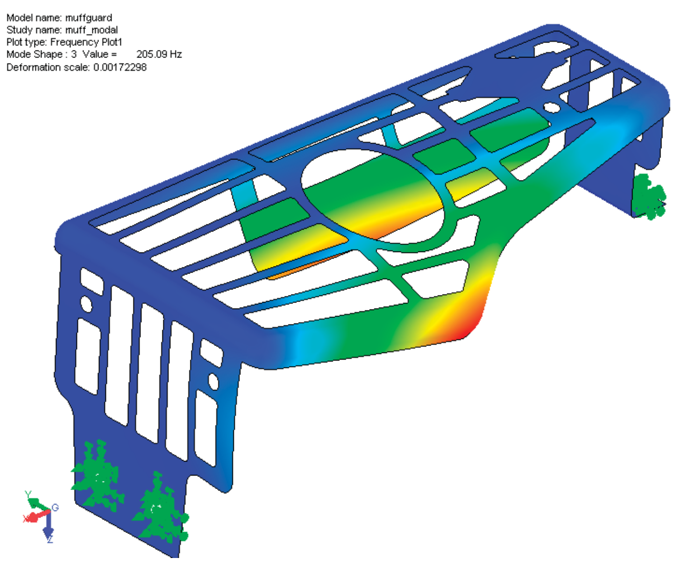
Product Setup



Summary Results

	Mode 1	Mode 2	Mode 3	Mode 4
COSMOSWorks	99	168	205	381
ANSYS	99	167	205	382
NEi/Nastran	99	168	206	381
Pro/MECHANICA	102	173	210	389
Average	99.0	167.7	205.3	381.3
	Variation from Average (%)			
COSMOSWorks	0.00%	0.20%	-0.16%	-0.09%
ANSYS	0.00%	-0.40%	-0.16%	0.17%
NEi/Nastran	0.00%	0.20%	0.32%	-0.09%

Results Plot



Frequency plot from COSMOSWorks 2005

Appendix – Additional Study Details and Rationale

IMPACT accepted this investigation as part of our on-going program to provide informative, objective and educational material to the FEA user base and to debunk the myths, if they are in fact myths, regarding the difference in accuracy between the various hierarchies of analysis tools. This appendix provides some additional detail on the study and the rationale behind the choices made.

Background

The author's experience with these benchmarks and the software used in this study is that they all perform reasonably well. Readers are encouraged to try these on their own.

Many investigations of this nature have been undertaken in the past using textbook or highly simplified examples, both by independent sources and by the developers of the software themselves. Software benchmarks, most notably those developed by NAFEMS, typically indicate that tools in question will yield acceptable levels of accuracy on targeted problems with closed-form or well-documented solutions. The author's experience with these benchmarks and the software used in this study is that they all perform reasonably well. Readers are encouraged to try these on their own.

Our experience working with engineering companies and analysis groups around the world has been that most popular FEA tools, if used correctly and within the limits of their solver, element technologies, and physics capabilities will generate valid results for most linear static solutions and many nonlinear or dynamic problems. Similarly, while some of the "basic" category tools may lack the breadth of features to provide accurate results in some more complex problems, all of the "advanced" tools provide a multitude of ways to generate inaccurate results to even basic problems in the hands of an under-trained, unsupervised user.

Where variation between tools or techniques becomes dangerous is when a user might draw erroneous conclusions regarding part acceptability from FE data that otherwise looks reasonable. It is always important to remind users and managers who are active in this technology that the validity of the results goes far beyond the accuracy or consistency of the tool being used. If the other assumptions that define a problem are invalid, yet held consistent across multiple software products, consistent incorrect answers could foster a false sense of confidence in the data.

Consistency versus Correctness

The intent of this study was not to document, or even attempt to obtain, a 'correct' solution. Simply defining 'correctness' is beyond the scope of this investigation. Additionally, a study of a tool's ability to calculate a correct solution is best performed on simple, well-defined benchmark problems as described previously when the validity of a target solution is beyond question. Instead, the intent of this study was to utilize each of the subject FEA tools on actual, manufactured or manufacturable parts with properties and boundary conditions that mimic actual or reasonable use to determine if the results differ and by how much. Armed with this information and being cognizant of studies on accuracy performed in the preferred manner described previously, a potential user of this technology can be assured that failure to achieve desired results with any of these tools rests in the quality of use, not the tool itself. This is critical because users have, within their control, the ability to improve their skill level and quality of use but few have access to multiple analysis tools and even fewer can affect changes in the software algorithms themselves.

Representative, Reasonable, & Repeatable Conditions

It was a guiding premise behind this investigation that the problems chosen, while actual parts from a variety of sources, not instigate a debate on "correct" or "proper" boundary conditions or properties for a particular application. For example, one of the problems used in this study is one leaf of a stamped hinge engaging a stop pin. Engineers familiar with hinge design could, quite rightly, take exception to the choice of constraints used. However, it is unlikely that any attempt to develop a more "realistic" set of boundary conditions would have satisfied all of the critics, thus obscuring the intent of the study. Consequently, the boundary conditions chosen for these problems will provide a representative and reasonable response in the part or assembly that is indicative of the intended use, not to attempt to validate the design or robustness of the part itself.

In some cases, the boundary conditions were chosen for repeatability across all tools to ensure set-up consistency. In other cases, they were chosen arbitrarily to produce a non-trivial response in an area that might be of concern to a developer of similar parts. Similarly, the choice of material properties was either arbitrary or restricted to the narrow focus of this study which was primarily linear static analyses. Consequently, some of the results show stress levels that most likely would flag an engineer to consider nonlinear material behavior. This was ignored in favor of, again, results consistency versus accuracy. No attempt to "engineer" these parts was made so the concept of "failure" from a materials perspective is moot. A subsequent study of the consistency of these and/or other tools in a nonlinear material or advanced dynamic scenario may lead to conclusions that differ from those found in this study but any attempt to qualify this performance was beyond the scope of this investigation.

Scope of Report

With a primary goal of reporting consistency in results, detailed model information, such as mesh size, problem set-up time, & run time, was not diligently recorded. It is our belief that there is so much variation in the way people work, even within the same tool, that precise reporting of such details could diminish the more salient points of the study. Therefore, the reported data will focus on output values that are representative of the results generated by each software product. An indicative location for displacement and stress, Von Mises Stress in all cases, was chosen based on the performance of the part under the given boundary conditions and then reported consistently in all cases. As with the discussion of the rationale behind the problem set-up, the output locations selected were not necessarily chosen based on where an engineer might need to look to validate the design. They were chosen, first and foremost, for the repeatability of the results at that location and then in an area of stress concentration so that any differences would not be trivial. However, throughout the course of the investigation, general relative observations were made about ease of setup, solution time, total problem duration, etc... that, when one of the software products stood out from the crowd, in either a positive or negative way, will be noted.

About the Author - Vince Adams

Over the past 18 years, Vince Adams has established a reputation as an expert in the use of Finite Element Analysis as an integral part of the product development process. Mr. Adams is co-author of "Building Better Products with Finite Element Analysis" from OnWord Press and numerous articles on FEA. As an invited speaker at conferences on FEA and product design around the world, Mr. Adams brings a fresh perspective to both the use and management of the technology in a product design environment.

Mr. Adams has been recognized by his peer group of engineering professionals both locally and internationally. He was invited by NAFEMS, the international organization for FEA quality and education, to serve as the inaugural chairman for their North American Steering Committee in 1999. His career in product development includes multiple successes as a Product Design Engineer and Project Manager, accumulating several US & international patents. Mr. Adams is currently the Director of Analysis Services at IMPACT Engineering Solutions, Inc., a leading provider of engineering support solutions since 1987, providing flexible and scalable solutions that are tailored to meet the unique needs of progressive manufacturers in product design and development.

IMPACT Engineering Solutions, Inc.

Milwaukee Office
3315 N. 124th Street
Suite N
Brookfield, WI. 53005
Phone 262-317-8100
Fax 262-317-8101
www.impactengsol.com

Chicago Office
Hawkston Hall Office Center 1
1800 Nations Drive, Suite 121
Gurnee, IL. 60031
Phone 847-599-5635
Fax 847-599-6035

IMPACT Engineering Solutions serves YOU, the progressive manufacturer, with a diverse, vertically integrated line of services. Since our inception in 1987, IMPACT has utilized the latest CAD, CAE and collaboration technologies to help you reduce product development time and become more productive in the process. By working with IMPACT, you benefit from the wide range of clientele and products we've been involved with over the years. Capitalize on that experience.