the CertaSim SOLUTION™

Featuring: Modeling Crushing of Extruded Aluminum Profiles

Q4
2015

The CertaSIM Solution™ Journal is published by CertaSIM, LLC

on a quarterly basis and contains technology information related to the software products sold by CertaSIM, LLC. The content is not approved nor verified by any of the software providers. CertaSIM, LLC does not guarantee or warrant accuracy or completeness of the material contained in this publication. The IMPETUS Afea Solver® is a registered trademark of IMPETUS Afea AS, Norway. All other brands, products, services, and feature names or trademarks are the property of their respective owners.

CertaSIM, LLC is the official distributor of the IMPETUS Afea Solver® in North, Central and South America and provides technical support and training for the IMPETUS suite of software.

Sales

CertaSIM, LLC
4717 Sorani Way
Castro Valley, CA 94546-1316
510-342-9416
sales@certasim.com

Support & Training

CertaSIM, LLC
4049 First Street, Suite 233
Livermore, CA 94551-5363
925-583-5211
support@certasim.com

Editor

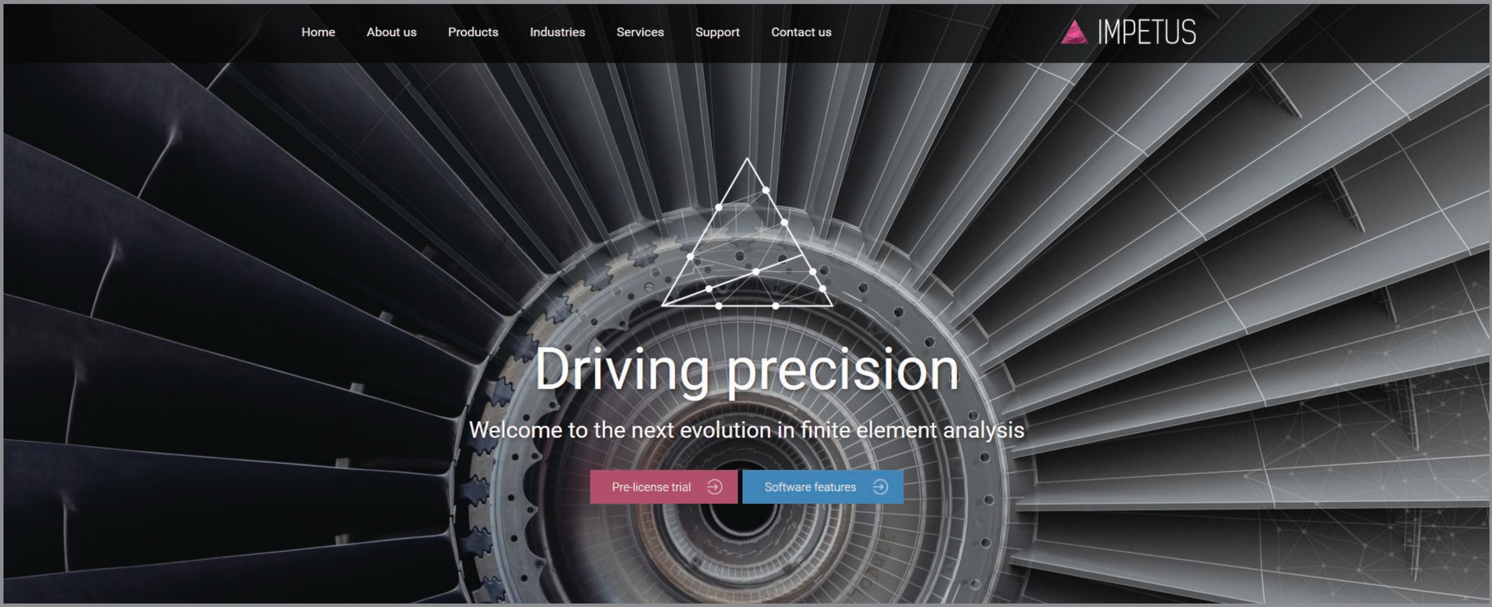
Wayne L. Mindle, Ph.D.
4717 Sorani Way
Castro Valley, CA 94546-1316
510-342-9416
wayne@certasim.com

Graphics

Kim Lauritsen
CertaSIM, LLC
4049 First Street, Suite 235
Livermore, CA 94551-5363
925-583-5211
kim@certasim.com

IMPETUS Afea - New Very Informative Website

IMPETUS Afea, the developer of the IMPETUS Afea Solver has just launched a brand new website with significant information for IMPETUS users. The support section of the website provides useful information, an online manual, Recommended Modeling Practice (RMP) reports and video tutorials, etc. The website is at:
http://www.impetus-afea.com



On the website there are also case studies under the "Industries Section".
Comments from IMPETUS users are (http://www.impetus-afea.com/industries/) :

"After an evaluation phase of IMPETUS software - we were in business."
Gard Ødegårdstuen
Manager R&D, Medium & Large Caliber Division, Nammo

"We have found IMPETUS software to be robust, you don't need to be an expert in meshing in order to produce consistent results."
Anders Artelius
Head of Aluminium Technology – Benteler Aluminium Structures Norway AS

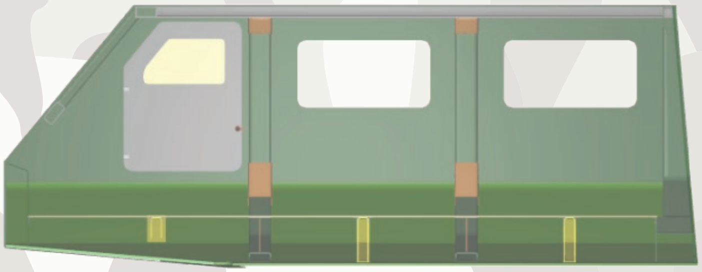
"Many will take the job, just a few are capable."
Per Erik Nilsen
Principal Engineer, Technical Safety Plant Integrity, Statoil

"IMPETUS software lets us fully utilize the potential in our Mekano channels design and structure in a precise and efficient manner."
Geir Seland
COO Øglænd System Group

"IMPETUS software lets us reduce cost and time in developing new concepts."
Eirik Enerstvedt
CEO - Wellbore AS

"IMPETUS software is an easy-to-use and efficient tool for investigating concepts of blast loaded concrete structures."
Knut Rakvåg
PhD, Senior Engineer, Special Advisor, Norwegian Defence Estates Agency

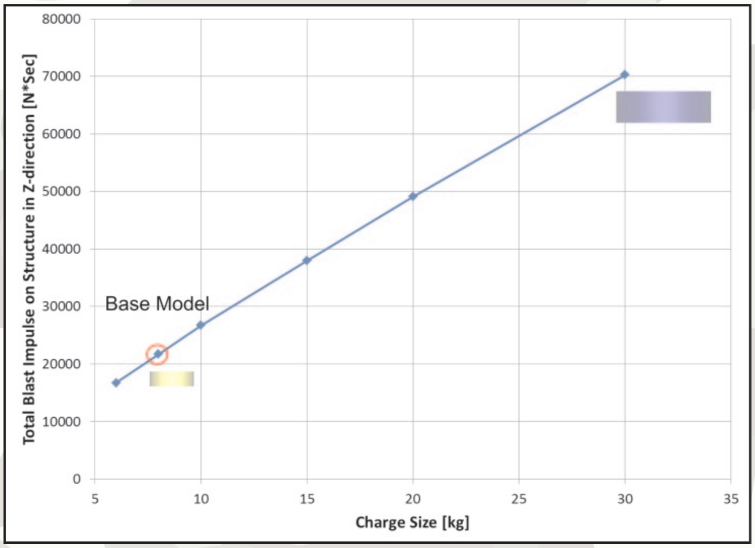
Identifying the Most Important Parameters in Buried Mine Blast

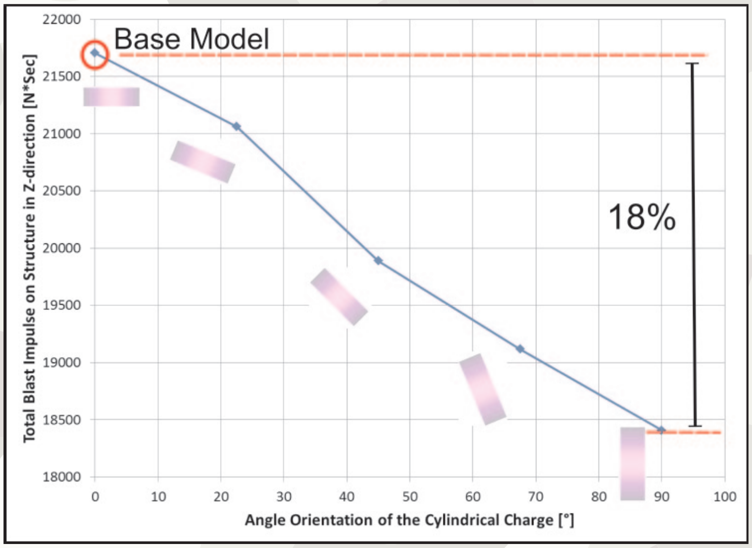


CertaSIM's, Dr. Morten Rikard Jensen presented work related to mine blast at the 2015 NDIA Ground Vechicle Systems Engineering and Technology Symposium (GVSETS), August 4-6, 2015 – Novi, Michigan. The work was a collaboration with Wilford Smith, Chief Engineer at SAIC. They decided to do a sensitivity study of a buried mine blast event both related to process parameters but also to investigate numerical parameters. With a detailed sensitivity study one gains knowledge about the numerical model and it helps in the M&S development phase of future projects with the IMPETUS Solver. Furthermore, it illustrates the stability of the software. The TARDEC Generic Vehicle Hull model was selected as a case study and the total Blast Impulse in the global Z-direction was chosen as the Response Parameter. In total 14 different parameters were considered, leading to 1000+ computational hours. The following characterization of the parameters illustrates the base for the Design Space:

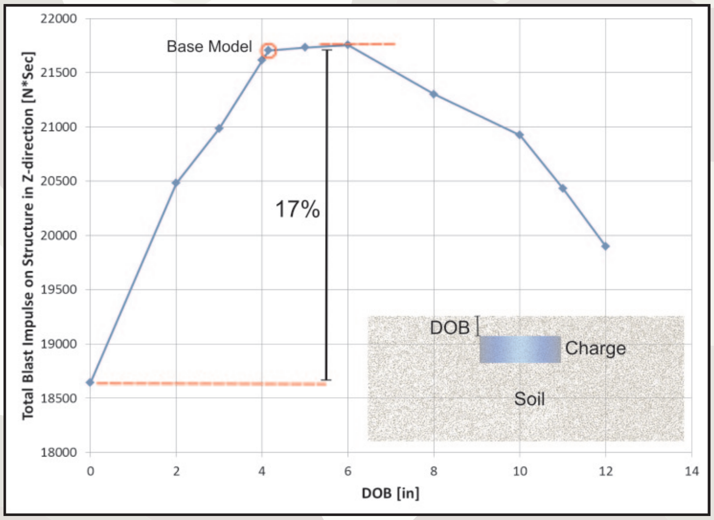
- Soil: Density, packing routine, inter particle stiffness, inter particle friction, inter particle damping, soil domain size and friction between structure and soil.
- Charge: Charge size, geometry, HE type, orientation (angle), off center location, DOB.
- General: Total number of particles.

The Base Model had a buried cylindrical charge of 8 kg C4 with a height to diameter ratio of 1/3 and a DOB of 4 inches. All simulations ran to normal termination, illustrating the robustness of the IMPETUS Solver. The computational time for the Base Model is approximately 9 hours. Several interesting observations were made. A linear relationship was observed between increasing the charge size and the Blast Impulse. It was also seen that there is Blast Impulse difference of 18% from a vertically placed cylindrical charge compared with a horizontally placed charge.





It is known in the industry that an under belly charge is the most critical location when considering the deformation of the interacting structure. This was verified by placing the charge at four different locations. It was observed that the Blast Impulse difference of approximately 45% was found between an under belly blast versus one placed a distance away. The charge depth is one of the main parameters in the mine blast event. The DOB affects how much soil will impact the structure and since the soil is the major part of the Blast Impulse for a buried mine, changing DOB can significantly change the damage. The results show that for the cases investigated a maximum effect is obtained for DOB's between 4-6 inches. A smaller DOB results in less soil hitting the structure and thus a smaller impulse. For a mine where the top is flush with the ground level, air needs to be included. After the maximum range, the charge is too deep to move the soil for impact with the structure. The difference between the smallest Blast Impulse and the largest is around 17%.



On an overall level, the trend in all the results seems to match what was expected. We see a benefit from this study by current users of the IMPETUS Solver when developing new models. It shows users what parameters are important and what response is expected when changing the value of a particular parameter.

The full paper can be downloaded from:
http://files.certasim.com/download/file/tech-info/publications/Discrete_Particle_Method_is_a_Predictive_Tool_for_the_Simulation_of_Mine_Blast_-_a_Parameter_Study_of_the_Process_and_Approach.pdf

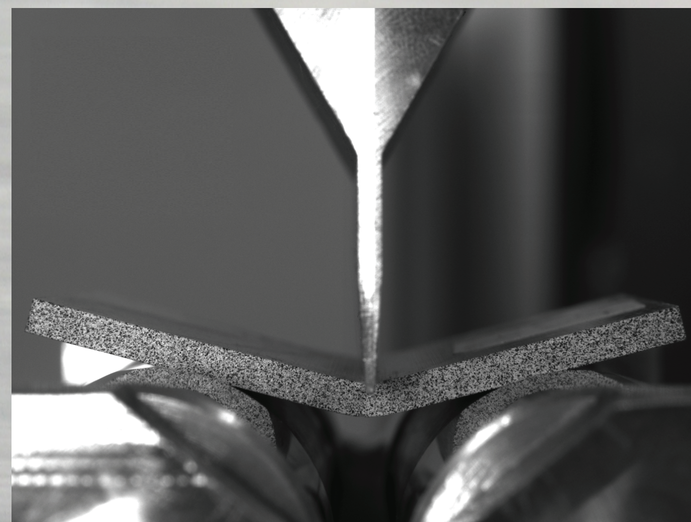
Currently, new studies have been initiated where the IED is an artillery shell placed underground to study the affect of fragmentation of the shell casing. This requires using the IMPETUS Node Splitting Algorithm to model the effect of fragmentation. Other studies cover the effect of multiple charges and IED's with various shapes, e.g., an oil or fuel container. More information on the paper and the new studies can be obtained by contacting CertaSIM, support@certasim.com.

ACKNOWLEDGEMENTS

The authors are thankful for the interest from IMPETUS Afea AB, Sweden and the support they provided.

"Many unclassified studies from past researchers have utilized fictitious vehicle geometry due to the non-availability of realistic information. Due to the sensitive nature of the work performed by the Department of Defense, data generated from testing military vehicles is usually classified, making it difficult to share data in the public domain. In order to increase the operational relevance of studies performed by the wider scientific community, the US Army Tank Automotive Research, Development and Engineering Center (RDECOM-TARDEC) has fabricated a generic vehicle hull to help evaluate blast mitigation technologies, and also shared an FEA model of the same for purposes of this research." It was necessary to develop a new model for the IMPETUS Afea Solver® and the authors would like to thank TARDEC, in particular, Dr. Ravi Thyagarajan, Mr. Madan Vunnam and Dr. Bijan Shahidi for supporting the effort to develop the model.

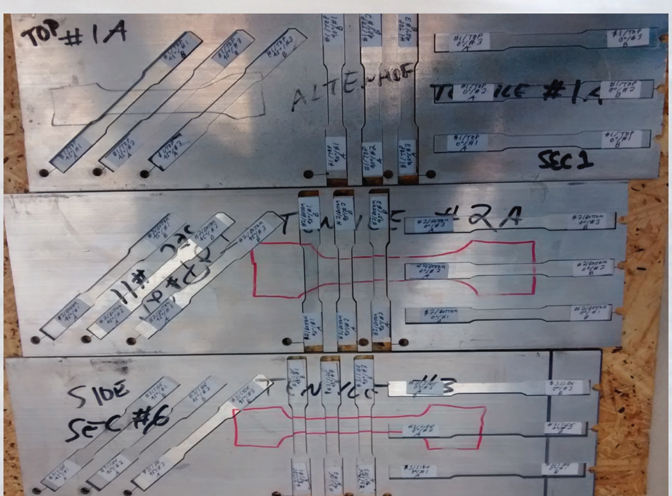
Finding the Missing Parameter for Modeling Crushing of Extruded Aluminum Profiles



It is common knowledge in industry that it is very difficult to predict fracture and failure of a side impact crash of extruded aluminum profiles as well as for the case of axial loading. Several automotive companies have pointed out to CertaSIM that experimentally they see variation in the results with materials that behave the same in uni-axial tensile tests, which often has been the base requirement for the material supplier to meet. They have also observed that it has been nearly impossible to accurately predict the failure and behavior of the profiles with Legacy Codes. The best they can do is to "tune" a model by turning numerical "knobs" to match experiments. This is typically done based on an element erosion criteria by setting an effective plastic strain parameter to define failure. It is firmly believed that with the Aset™ Element Technology, a Node Splitting Algorithm to model damage and fracture, and a Direction Dependent Failure Criteria this application can be modeled accurately [1]. In order to prove the procedure, experimental work is necessary, which includes uni-axial tensile tests, VDA 3-point bending tests [2] and crush tests.

The University of Windsor, Ontario, Canada and CertaSIM are collaborating on a joint project called "Concept for Modeling Crashworthiness of Extruded Aluminum Profiles with the IMPETUS Afea Solver®". The experimental work is being performed by Professor William Altenhof and his Ph.D. student Matthew Bondy, Department of Mechanical, Automotive and Materials Engineering. The project is funded by the R&D Department of CertaSIM, LLC. A press-release can be found at:
http://files.certasim.com/download/file/tech-info/press_releases/UWindsor_CertaSIM_News_Release_11_19_2015.pdf

The material selected for the study was AA6061-T6 which is a commonly used aluminum alloy. Specimens were taken in the extrusion direction, 45° and 90° to this direction. Three sets are taken within one profile and this is repeated for three different profiles resulting in 27 specimens for both uniaxial and VDA testing.

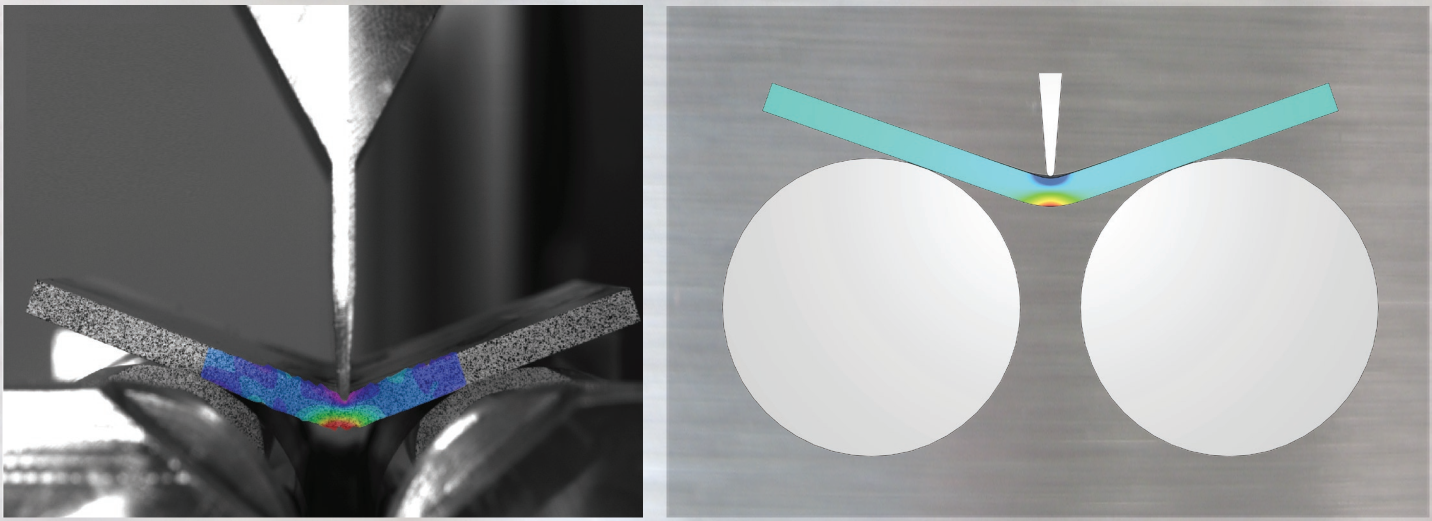


Three different directions relative to the extrusion direction are considered in the experiments.

The uni-axial tensile test gives the strength of the material and inverse material modeling is used to find the damage parameters. This is done by modeling the VDA test in the IMPETUS Afea Solver® and optimizing the damage parameters to match the experimental force deflection curve and bending angle. These parameters are then used to model an axial crush and side crush of extruded profiles. To verify the methodology the crush simulations will be compared with experimental results using the same extruded profiles.

Set-up for axial and side crush experiments.

The status of the project is that the uni-axial tensile tests are finished and the VDA tests are nearly done. In both cases very good repeatability of the experiments were found. It was seen that for the uni-axial tensile tests a difference could be seen between the results for the different directions where the specimens in 45˚ had the smallest fracture strain. For the VDA tests a large difference was found for the specimens taken in the various directions which indicates that the ductility of the AA6061-T6 profile is strongly direction dependent. This direction dependency of the ductility is the basis for the need to develop a new procedure to accurately model crushing of extruded profiles. The next step in the project is to experimentally perform axial and side crush of the extruded profiles and numerically develop models of the VDA tests and the crush tests. The latter is currently in progress but the optimization to identify the damage parameters is still to be done.



Left: Experimental DIC picture of the VDA test. Right: Pre results of modeling the process in IMPETUS.

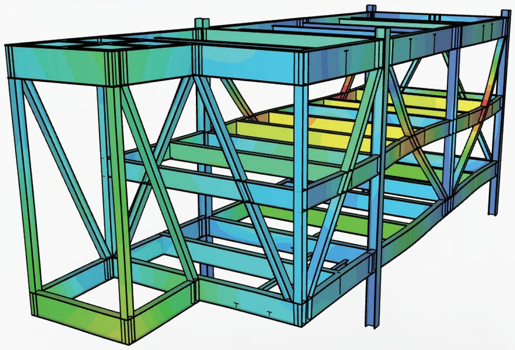
We look forward to publishing the results in the Q1 2016 Journal. More information about the project can be obtained by contacting CertaSIM at support@certasim.com. All pictures of the experiments shown in this article were taken by Matt Bondy, University of Windsor, Ontario, Canada.

[1] M. R. Jensen, "Concept for Modeling Crashworthiness of Aluminum Profiles with the IMPETUS Afea Solver®", CertaSIM Report # CS-0039-031215.

[2] Verband der Automobilindustrie, "VDA 238-100 (Test Specification) Plate Bending Test for Metallic Materials", December 2010.

IMPLICIT Module– Eigenvalues are a useful Tool

The IMPETUS Afea Solver® includes an implicit linear solver which is part of the IMPETUS BASIC license. Currently, it is possible to do linear static analysis, compute buckling modes and calculate eigenvalues. As engineers we know that natural frequencies of a system, namely the eigenvalues provide very useful information about a structure. Many structures and components of larger structures have design requirements that make it necessary to avoid exciting natural frequencies of the system to eliminate vibration modes. An eigenvalue analysis of a structure modeled with an explicit Finite Element simulation can also be useful to visualize component connections and to see if they are applied in correct locations. The IMPETUS Afea Solver® has been applied in the golf industry where eigenvalue analysis is used to tune the sound of the club head hitting the ball.



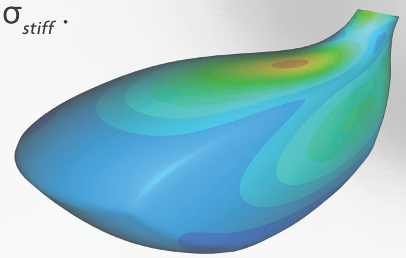
Eigen Mode for container in the offshore industry.

The Subspace iteration algorithm is used in the IMPETUS eigenvalue solver where the only input that is required from the user is to select the structural parts to analyze and request the number of eigenvalues, N, to compute. This is the N smallest eigenvalues of the system. In addition, stress stiffening can be activated, i.e., when the load or initial stresses change the eigenvalues.

The syntax is very simple, here is the command:

```
*ANALYSIS_DYNAMIC_EIGENMODE
N, psid
σstiff
```

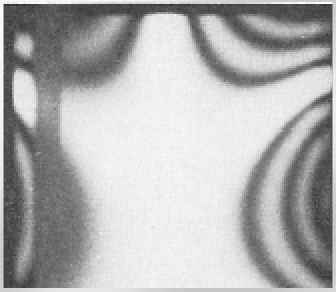
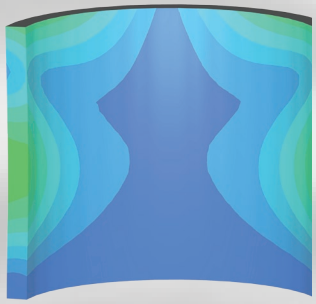
ID *psid* defines the structural parts to consider and the stress stiffening is activated by defining the parameter σ_{stiff} .



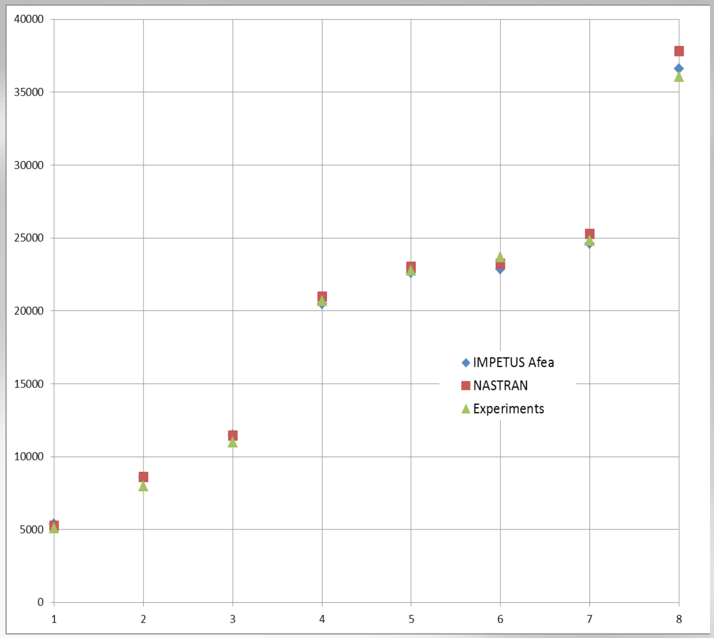
Eigen Mode for golf club driver.

Post-processing is simple, just load the *impetus.imp* file in the usual manner and select the eigenvalue menu to animate the eigenmodes and list the frequencies and the associated displacements. The procedure for Post-Processing an eigenvalue analysis is shown in the IMPETUS Afea Post-Processor Section at the end of this Journal.

In order to demonstrate the accuracy of the eigenvalue solver, CertaSIM's R&D Group has been running different test cases to exercise the Solver. One of the test examples includes experimental data and results from Cosmic NASTRAN simulations that were performed at the Air Force Institute of Technology, Wright-Patterson Air Force Base, Ohio, USA by CertaSIM's, Dr. Wayne L. Mindle when he was employed at the Institute. He performed both the experimental and numerical work [1], [2]. The research involved experimental determination of the eigenvalues of three steel blades using holography and corresponding numerical simulations using Cosmic NASTRAN. The blades were modeled with triangular shell elements referred to as CTRIA2 elements. All blades had the same length of 25.4 mm, inner radius of 11.43 mm and were 142° cylindrical cuts. The blades only differed in the wall thickness; Blade 1 had a thickness of 1.59 mm, Blade 2 was 3.18 mm and Blade 3 was 4.57 mm. Each blade was cut from a cylindrical piece of steel so the uncut end could be mounted in a steel base and held into place with set screws. The fit was so tight that a hole had to be drilled from the side of the fixture into the bottom of the shaft where the blade was inserted to allow the blades to be slipped into place. Eight modes were experimentally determined for each blade. Because the blade is fully constrained there are no Rigid Body modes. For the IMPETUS simulations, Blade 1 was studied with different types of hexahedron elements and it was found that for the same mesh density, the element that performed closes to the experiments is the cubic formulation but the quadratic element also gave good results. Experience has shown that quadratic elements are more than sufficient and they are the recommended elements for larger structures. The mode shape is also well represented when comparing the numerical results with the holographic experiments. All three blades were modeled and in all cases the IMPETUS results were consistent with the COSMIC NASTRAN results.

Second Torsional Mode. Good agreement between the experimental and numerical determined eigenmodes.



Frequencies for Blade 1. Comparing Experiments, Cosmic NASTRAN and IMPETUS results for the first 8 modes.

In conclusion, the eigenvalue solver has shown to be accurate and is a great addition to the other solvers that are part of the IMPETUS Suite of Solvers: FE, SPH and DPM. The documentation for this case study and others, together with relevant models can be obtained by contacting Dr. Morten Rikard Jensen at morten@certasim.com.

[1] W. L. Mindle and P. J. Torvik, “Multiple Modes in the Vibration of Cantilevered Shells”, Journal of Sound and Vibration (1987) 115(2), 289-301.

[2] W. L. Mindle, “The Multiple Mode Phenomenon in the Vibration of Curved Cantilevered Blades”, Air Force Institute of Technology, Wright-Patterson Airforce Base, Ohio, Report AFIT-TR-EN-85-5, 1985.

Dr. Morten Rikard Jensen is the CTO of CertaSIM, LLC which is located in Castro Valley and Livermore, California. His background is in Sheet Metal Forming, both as an experimentalist and as a numerical analyst. He joined CertaSIM, LLC in 2012 and previously worked at LSTC (LS-DYNA®) for twelve years, the last seven years as Support and Training Manager. During that time he developed training material, including the official "Getting Started with LS-DYNA®" book. He has used Legacy Codes since 1993 and we have asked him to describe what he sees as the main differences between those Solvers and the IMPETUS Afea Solver®, a "Next Generation Solver".

"There are many features in the IMPETUS Afea Solver® that I believe distance it from Legacy Codes, those solvers that were developed 30 years ago and still rely on classic finite element formulations and techniques. The development of the IMPETUS Solver started with the concept that 'New Solid Element Technology' is the key to accuracy in simulation. The result was the ASET™ Family of High Order elements, accurate and robust solid elements that include quadratic and cubic: Hex, Tet and Pent elements. Yes, we can finally say there are really good Tetrahedron elements that perform well in both bending and plasticity, something that cannot be said for classic solid elements. And cubic Hexahedron elements that do not require perfect aspect ratios to be accurate.

In my experience one problem that often makes Explicit Transient FE Solvers difficult to use is the occurrence of hourglass modes. Users of Legacy Codes really struggle with this problem because it is often difficult to detect and if observed requires the user to select one of the many methods that were developed to control the 'problem'. In my opinion it is a time consuming Trial & Error process that requires the analyst to spend even more time to carefully scrutinize the results to make sure the solution does not cause 'another problem.' Hourglass control is basically the same as changing the material properties because it adds stiffness to the model to prevent hourglass modes. The elements available in the IMPETUS Solver are fully integrated elements so there are no hourglass modes at all. The elements can withstand very large deformations without numerical instabilities. Since there are only solid elements it eliminates the problems related to contact when beams and shells are present in a model. Most importantly, 3D solids remove the need to assume a plane stress state that is inherent in shell formulations and can be an incorrect assumption in many problems. I have also observed that when using the ASET™ elements we obtain accurate results with simpler material models and less complicated damage criterion. This is particularly telling since we know that Legacy Codes can only rely on more complicated material models with more parameters, commonly referred to as 'knob turning' to dial in better results.

For modeling of fragmentation the traditional approach has been element erosion. This is not physical and leads to early elimination of structural mass and strength. In IMPETUS, element erosion is kept to a bare minimum because of the Node Splitting Algorithm that was developed to handle fracture.

Whether to use 'Double Precision' has always been a question and the only way to really know is to run both single and double. Another decision that was made at the beginning of the development process was to insist that only double precision be used for FE Calculations. This removes another potential risk for error since some models simply need the extra precision. Double precision and higher order elements affect runtime and the IMPETUS Afea Solver is the only commercially available Explicit FE code that takes full advantage of GPU Technology for parallelization. GPU's are the ultimate shared memory processing technology, with 2880 cores per GPU with 12 GB of DDR5 memory, a standard workstation can house 2 NVIDIA Tesla K40 processing units and allow for multiple jobs to be run at the same time.

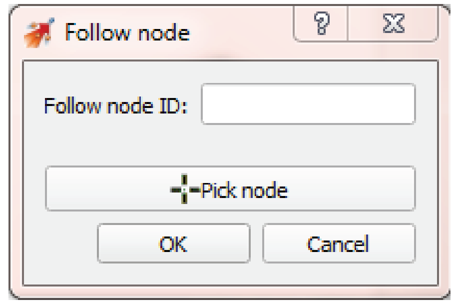
I really enjoy how the IMPETUS solver has changed the Work Flow. We start with a coarse linear mesh with enough resolution to define the geometry and during the initialization phase (RUNTIME) we can tell the solver to change any or all of the elements to quadratic or cubic. This includes the ability to also refine the mesh, again at runtime, anywhere in the model. This is really the only way to affectively use high order elements, refine only where necessary and let the software do the work not the engineer.

A question I often hear is "How steep is the learning curve?" That really depends on where you stand in your knowledge of explicit FE. For those engineers that have been working with linear static analysis only, there is a lot to learn. However, if one is an experienced user of explicit FE, e.g., been using one of the Legacy Codes for 5+ years, then it is not difficult at all to get started with the IMPETUS Solver. As an example we had a customer that had 10+ experience and he was up running his own industrial models in 2 days! I see the IMPETUS Solver as a Tool that has brought joy back to using explicit FE. The Solver is on the edge of technology where it is fun to be. It is really time for the engineer to get back to engineering and design and to spend less time babysitting a solver to just get an answer!"

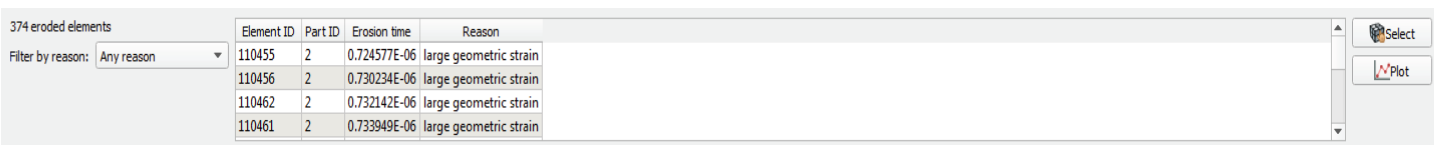
New Features in the IMPETUS Afea Post-Processor

In Post-Processing of some models it is beneficial to be able to select a node that is defined as the center for the animation of the event. This is the case in ballistic models if the impact area is the focus. This option is implemented in the IMPETUS Afea Post-Processor and called “Camera Follows a Node”. The icon for it is located in the middle of the Top Tool Bar. 

A menu appears and one selects a node to follow, this can be done by simply picking a node and clicking OK. The animation will now be locked to this node.

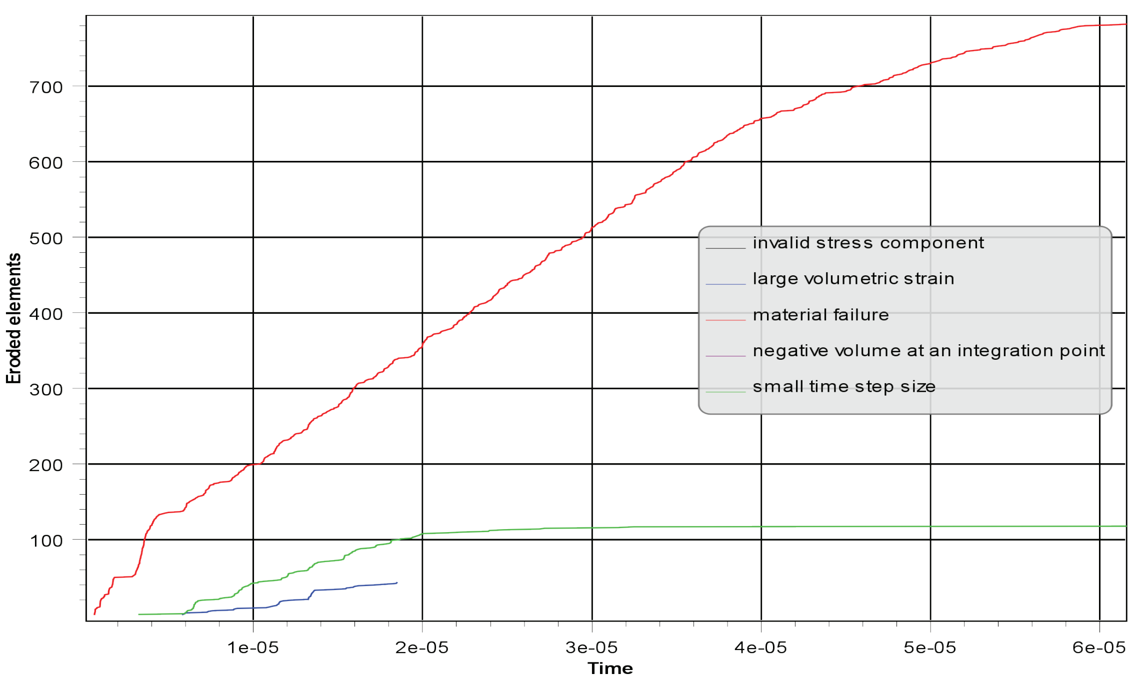


Another very useful feature when modeling ballistic impact is to be able to document the reason for element erosion. In ballistic tests with target ductile materials, element eroding is often used whereas for brittle materials node splitting should be used. In the case of eroded elements it is necessary to investigate the reason for the erosion, is it related to the time step, material damage or ? In IMPETUS the file ***eroded_elements.info*** contains information about the reason for the erosion, time stamp, element ID and part ID. It is now possible to plot this information in the Pre-Processor. In the Object Tree to the left click on the Special node/elements item and select Eroded elements. A menu appears under the Graphic Area with a table showing the information. The information can then be plotted.



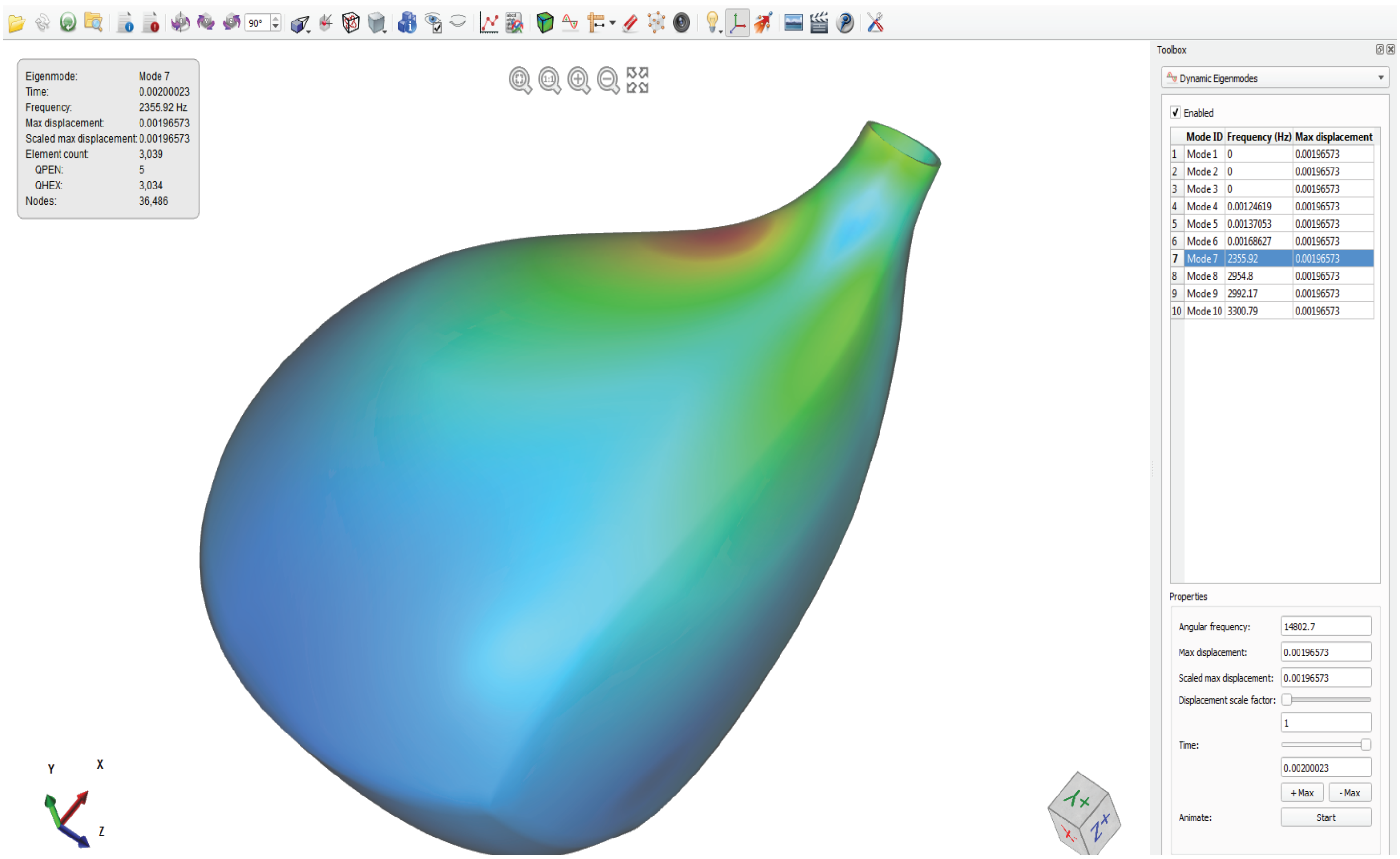
Element ID	Part ID	Erosion time	Reason
110455	2	0.724577E-06	large geometric strain
110456	2	0.730234E-06	large geometric strain
110462	2	0.732142E-06	large geometric strain
110461	2	0.733949E-06	large geometric strain

This plot shows the number of eroded elements versus time. This is very useful in order to understand if elements are being eroded for purely numerical reasons, like the time step because of element distortion, or it is due to a user defined material erosion criteria which is very important in modeling ballistics.



Post-Processing Eigen Modes in the IMPETUS Afea Post-Processor

Post-Processing results from an Implicit Eigenvalue Analysis in IMPETUS Afea Solver® is integrated into the interface for the IMPETUS Afea Post-Processor. The main IMPETUS binary file, impetus.imp, is read into the Post-Processor and the Dynamic eigenmodes icon in the Top Tool Menu is clicked. The Dynamic eigenmodes menu appears on the right side (Tool Box area).



In this menu it is possible to animate the modes, see the frequencies and displacements and set a displacement scale factor. Each individual mode can be selected and investigated. The animation of the modes can render in either regular color mode or apply displacement contours.