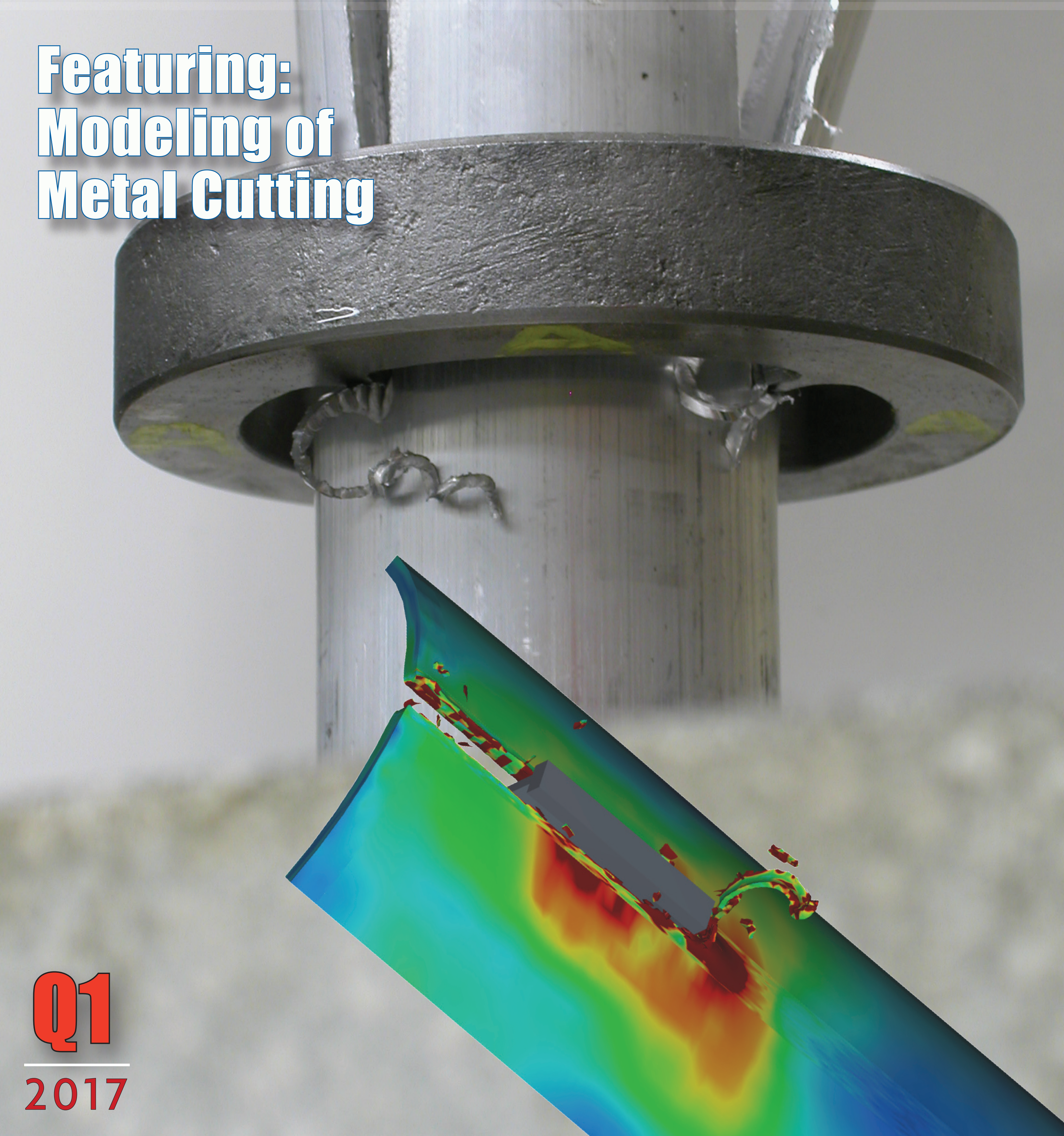
the CertaSim SOLUTION™



Featuring: Modeling of Metal Cutting

Q1

2017

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.

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

925-315-9349
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

925-315-9349
kim@certasim.com

News and Events

New Research on Cutting Deformation Applies the IMPETUS Afea Solver®

The IMPETUS Afea Solver® is being applied to many new areas of research. One example was accepted for publication in the International Journal of Impact Engineering. The work describes experiments as well as IMPETUS simulations of an energy absorbing cutting tube. The work is described in detail later in this journal.

ELSEVIER

International Journal of Impact Engineering

Available online 29 December 2016

In Press, Corrected Proof — Note to users

Finite element modeling of a novel cutting deformation mode of AA6061-T6 tubes employing higher order Lagrangian element formulations

Matthew Bondy[a], Morten Rikard Jensen[b], John Magliaro[a], William Altenhof[a],

[a] Mechanical, Automotive, and Materials Engineering, University of Windsor, 401 Sunset Ave., N9B 3P4, Windsor, Ontario, Canada

[b] CertaSIM, LLC, 4717 Sorani Way, Castro Valley, CA 94546-1316, USA

CertaSIM's CTO Visited the IMPETUS Office in Sweden

CertaSIM's CTO, Dr. Morten Rikard Jensen visited IMPETUS Research and Development Center in Huddinge just outside Stockholm, Sweden. To maintain the close connection and stay updated on new features in IMPETUS and to benefit CertaSIM's customers, our staff visits Sweden 1-2 times a year. The main focus for this two week trip was development of the IMPETUS Hybrid III 50th Percentile Male Blast ATD. The visit included discussions about recent and future features.

"It is critically important for CertaSim and our customers that we have direct access to the IMPETUS Afea development teams in Sweden, Norway and France. I really appreciate that the team in Sweden allowed me to be there such a long period of time despite their busy schedule. It is just a great group and very inspiring to visit them. And I would like to say Thank You for that!" states Dr. Jensen.

Latest Official Release of the IMPETUS Afea Solver

A new official QA'd version of the IMPETUS Afea Solver® has just been released. It is version 4.0.2093. The version includes many exciting new features many of which were beta tested by CertaSIM's users and found to be very useful. Some of the features are:

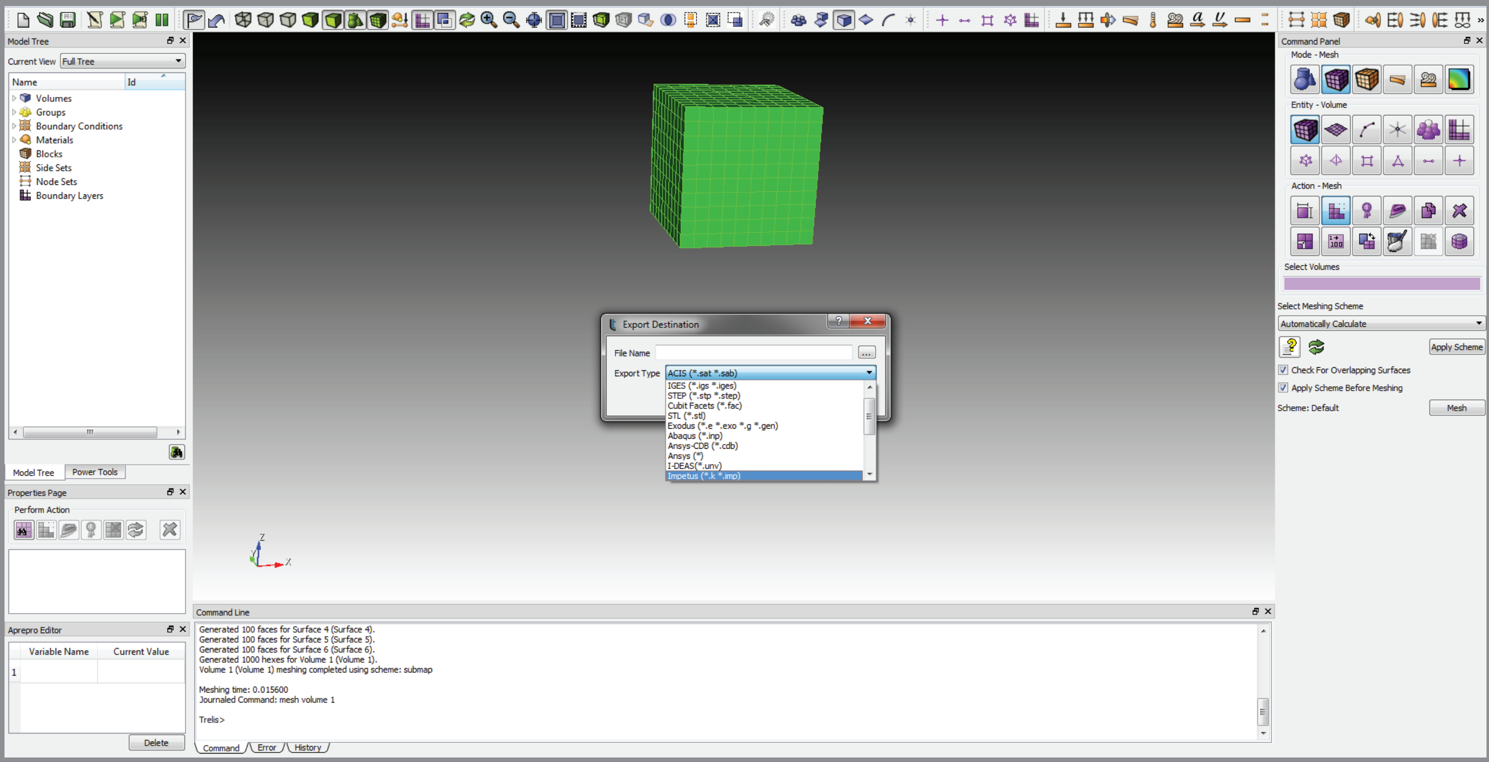
- Fixed critical contact bugs.
- *VELOCITY_CAP: a new command that defines an upper limit to node and discrete particle velocities.
- *CONTACT: improved treatment of facets in contact that are nearly orthogonal.
- More efficient GPU memory handling.
- Added GPU support for Mie-Gruneisen equation-of-state.
- *PARTICLE_DOMAIN: GPU speedup.
- Added resultants to "prescribed.out" for more convenient plotting.
- *PRESTRESS_BLIND_HOLE_BOLT: a command for prestressing blind hole bolts.
- Added GPU boost functionality for Tesla GPUs (K40, K80, P100, etc.)
- *FUNCTION: added intrinsic functions for access to joint and sensor data.
- *OUTPUT_USER: a new command for user defined ASCII output.
- *OUTPUT_SENSOR: sensors are now written to impetus_stateX.k.
- *INCLUDE: added material ID offset and a general command ID offset.
- *RIGID_BODY_JOINT: added user defined damping.

The 4.0.2093 executable can be obtained by contacting support@certasim.com.

Trelis and Bolt now Supports IMPETUS

"Creating geometry is the first step in any analysis effort and requires good software tools. As the developers of Trelis and Bolt it is our mission to provide those tools. We know that accurate geometry is critical to accurate simulations and so it was a natural step to provide support for the IMPETUS Afea Solver ASET™ Elements which include high order elements that can accurately model the geometry."

Paul Ressler, Director of Sales and Marketing, csimsoft™



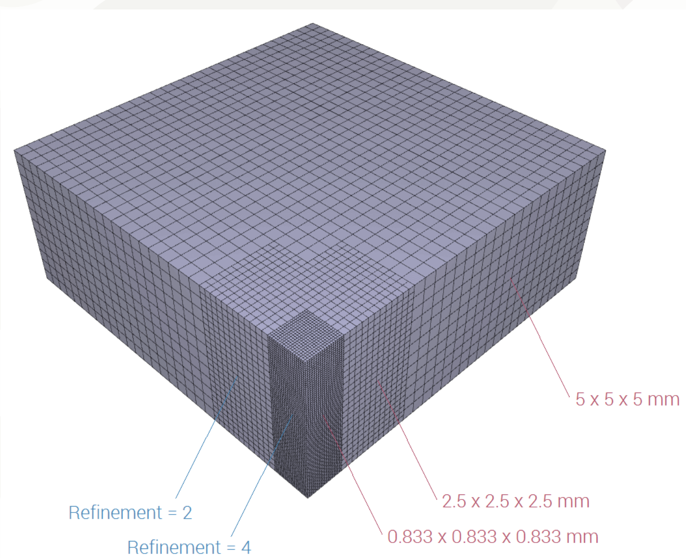
Guidelines for Modeling Ballistics

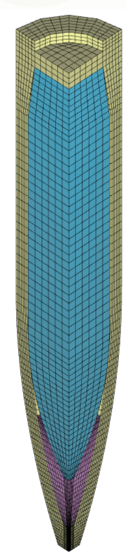
Bullet penetration of a target is very difficult to model due to the high speed of the projectile and the fragmentation of the target. The following shows the recommended way to model this event.

IMPETUS Afea AB, Sweden has developed a suite of Recommended Modeling Practices (RMP) for various applications. Currently, five of these are available and more are under development as the Solver expands into more application areas. The fifth of these RMP's is RMP005 that discusses the topic of ballistic impact. More specifically, it relates to validation models of 12 different materials that all are included in the IMPETUS Afea Engine and GUI. These materials cover High Strength Steel, Titanium, Aluminum Alloys and Rolled Homogeneous Armor and all results are backed by experimental data. The document investigates results for different tempers of Aluminum Alloys as well as the influence from the extrusion direction.

Study	Product designation / Alloy	Comment
Borvik et al. (2009)	Weldox 500E Weldox 700E Hardox 400 Domex Protect 500	Steels produced by SSAB
Holmen et al. (2013)	AA6070-T0 AA6070-T4 AA6070-T6 AA6070-T7	Aluminium alloys
Borvik et al. (2011)	AA6082	Aluminium alloy
MIL-DTL-46077F (1998)	Ti-6Al-4V	Titanium alloy
Iqbal et al. (2016)	Armox 500T	Steel produced by SSAB
Benck (1976)	RHA	Rolled homogeneous armor

A selected group of these tests are included in the IMPETUS QA System, referred to as the "Verifier", before a new official release. The version control is also listed in the document. The various tests are discussed in detail as well are the IMPETUS models. The bullets are three parts, consisting of a core, jacket and a tip which makes it possible for the jacket to be realistically displaced from the core. In ballistics it is often a narrow area around the impact zone that highly deforms whereas the rest of the target experiences much less deformation. To accurately capture the deformation and at the same time reduce the computational time, the IMPETUS meshing technique is applied, meaning that the impact area is both meshed and refined with the IMPETUS ASET™ Element Technology that includes cubic higher order elements. The area outside the impact zone can be modeled with linear elements.

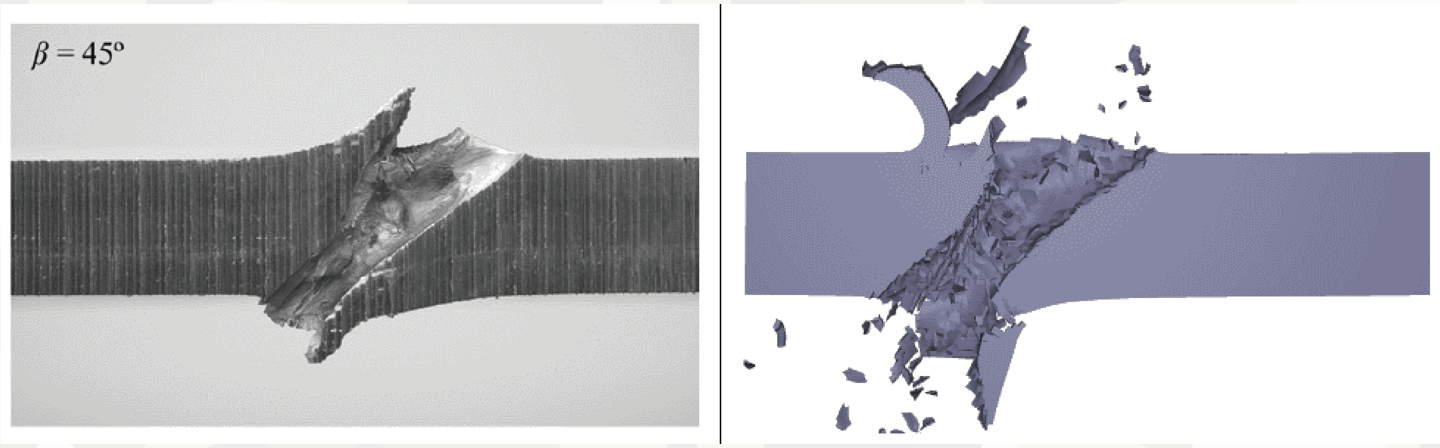




Some of the impact scenarios include oblique impact angles which have traditionally been difficult to model accurately. However, the IMPETUS Solver has shown to be very robust and accurate in the application of ballistics as this report clearly illustrates. One of the classic Response Parameters is the exit velocity or residual velocity which is the main parameter used for comparison of the results.

Simulation	Mesh	Obliquity [deg]	Material model [deg]	Result [m/s]	Target [m/s]	Error [%]
1			0	715		-1
2		0	45	728	720	1
3			90	722		0
4			0	711		-1
5		15	45	720	720	0
6			90	719		0
7			0	686		-3
8	Coarse	30	45	702	708	-1
9			90	696		-2
10			0	602		3
11		45	45	635	585	9
12			90	621		6
13			0	0 (Ricochet)		-
14		60	45	0 (Ricochet)	0 (Embedment)	-
15			90	0 (Ricochet)		-
16		0		732	720	2
17		15		716	720	-1
18	Fine	30	0	705	708	0
19		45		611	585	4
20		60		0 (Ricochet)	0 (Embedment)	-

It is not surprising that IMPETUS captures both the target and bullet deformation which includes peeling of the outer jacket.



It is strongly recommended to follow the guidelines in the report when modeling ballistics with the IMPETUS Afea Solver®. Just the information about modeling the target with both high and lower order elements will save significant computational time.

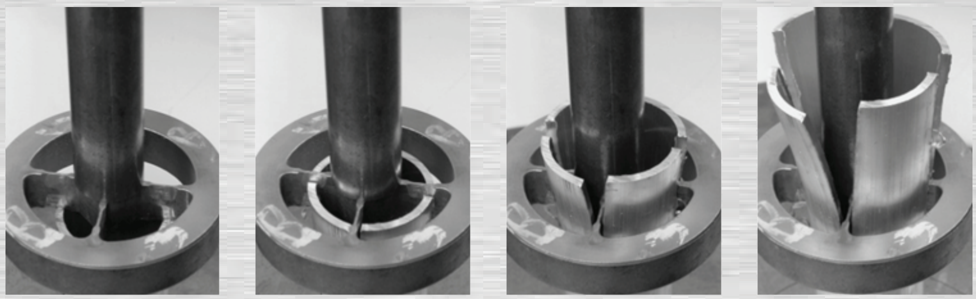
The RMP005 can be found at:
http://www.impetus-afea.com/support/documents/?doc=rmp/rmp005

Modeling Novel Cutting Deformation of AA6061-T6

Absorbing the impact energy that results from a car accident is critical to improving crashworthiness of vehicles. An efficient method has been developed at University of Windsor which center's around a tube cutting process. To help investigate the process, they have used the IMPETUS Afea Solver®.

Researchers in the Mechanical, Automotive, and Material Engineering Department at the University of Windsor, Ontario, Canada have been investigating axial cutting of lightweight metal extrusions both experimentally and numerically. The mechanism is interesting since it has the ability to absorb the high energy created at impact which will result in saving lives.

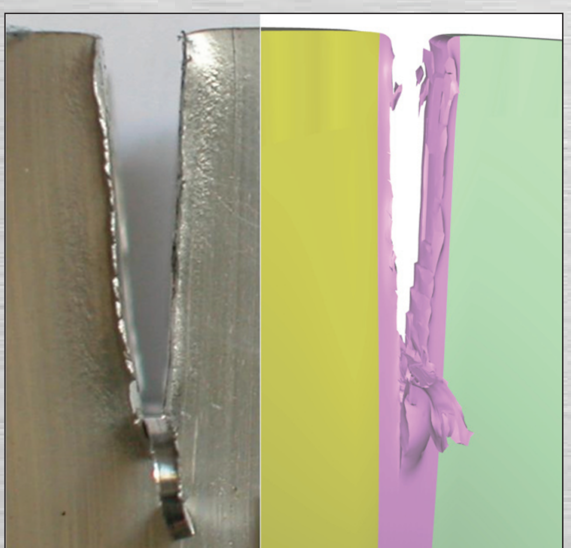
In the past the researchers have used Legacy Codes which include both ALE solvers and other methods which were found to be problematic for brittle materials. The models could simply not predict the results with the required accuracy.



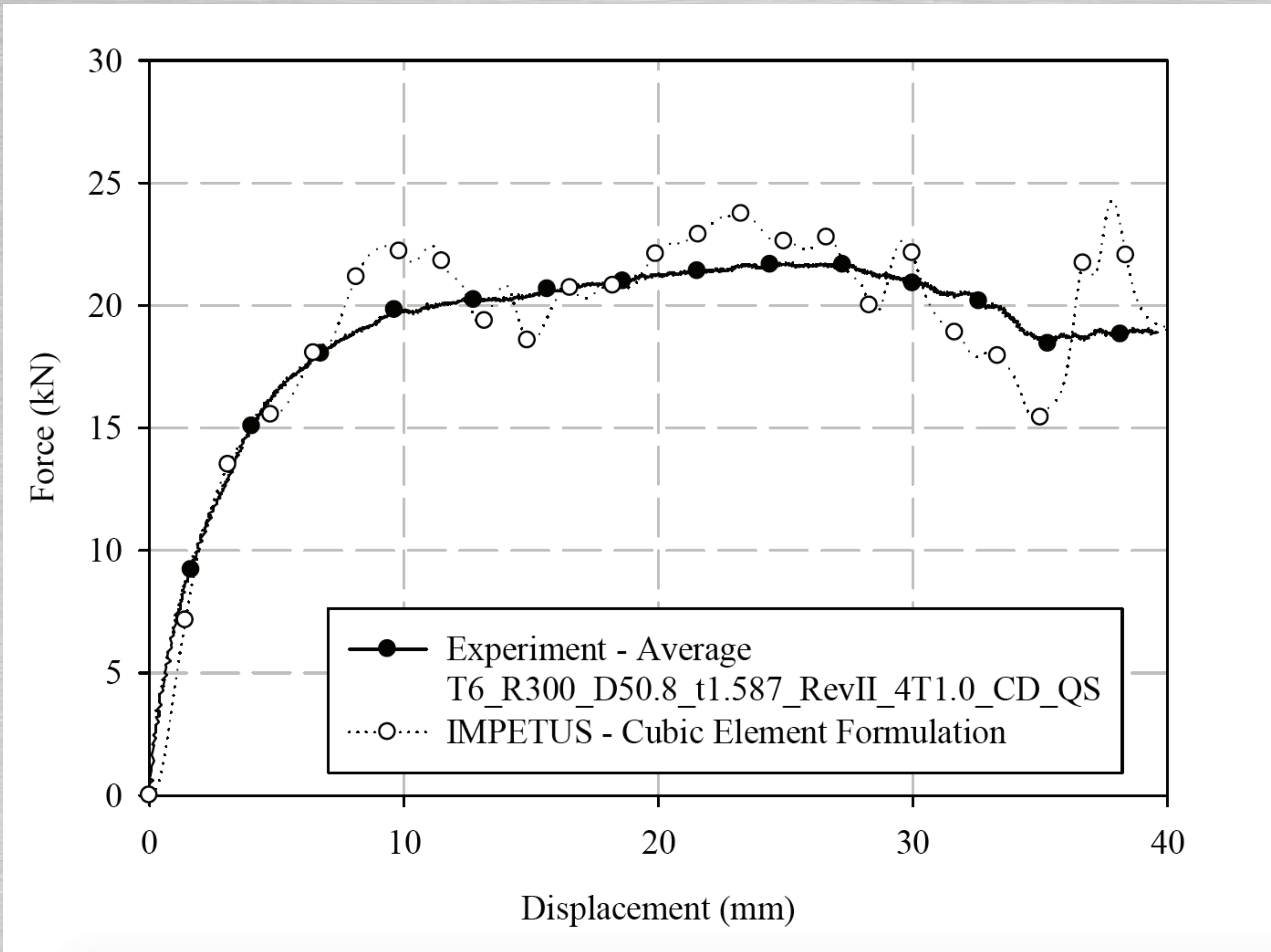
They have switched to using the IMPETUS Afea Solver® due it's accuracy and ability to apply node splitting, an accurate method to model cracking and splitting. The work is documented in [1] and [2] where the following was stated regarding the use of IMPETUS:

"...highly discretized FE models of this large deformation problem running on 1st generation Tesla hardware accurately capture the chip formation and force-deflection response." [1]

"The model is capable of accurately capturing the force versus deflection response with Oberkampf-Trucano validation metrics exceeding 0.9 demonstrating greater accuracy than previous modelling efforts with an alternative commercial solver." [2]



The tube is modeled with ASET™ cubic elements that are further refined in the cutting zone. The outer zone is modeled coarser with linear elements, following the classic meshing strategy for modeling with IMPETUS. This technique saves computational time but still keeps results in an accurate solution. The material for the tube was AA6061-T6 aluminum and the material characterization was done with a combination of uniaxial tensile tests and VDA 238-100 bending tests. These are done with specimens taken in different directions relative to the extrusion direction; 0°, 45° and 90°. An inverse material modeling technique is applied to find the damage parameters which is then modeled with the Cockcroft-Latham damage model, *PROP_DAMAGE_CL in IMPETUS. The results are very impressive when comparing experimental results for both the force-deflection curve and chip formation.



References:

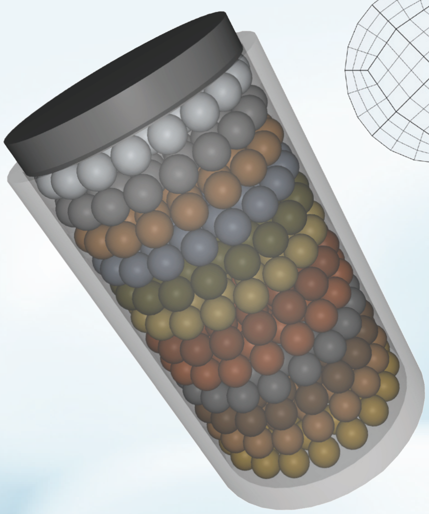
[1] M. Bondy et al., "Finite Element Modelling of a Novel Cutting Deformation Mode of AA6061-T6 Tubes Employing Higher Order Element Formulations and GPU Computing Technology", ICILSM 2016, 22-26 May 2016, Turin, Italy.

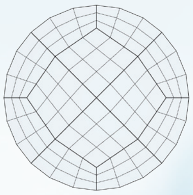
[2] M. Bondy et al., "Finite Element Modelling of a Novel Cutting Deformation Mode of AA6061-T6 Tubes Employing Higher Order Lagrangian Element Formulations", International Journal of Impact Engineering, In Press, Available Online, http://www.sciencedirect.com/science/article/pii/S0734743X16306406.

Multi-Particle Finite Element Method with the IMPETUS Afea Solver®

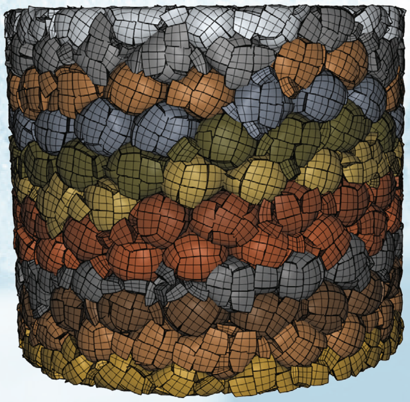
For the first time the powder compaction process has been successfully simulated in 3D with the particles individually modeled. Adding even further complexity, the particles are deformable and can break apart. The work illustrates the incredible strength of the IMPETUS Solver for this type of application.

The Multi-Particle Finite Element Method (MPFEM) is used to model the compaction process, representing each particle with a fully discretized Finite Element mesh [1]. The method has also been applied in [2] where it is compared to the use of the Discrete Element Method (DEM) shown in [3] that exhibits a softer response. So far these models have been done only in 2D. But with the computational advantages of the IMPETUS Afea Solver® combined with higher order accurate cubic Aset™ elements and node splitting, full three dimensional simulations are a reality.

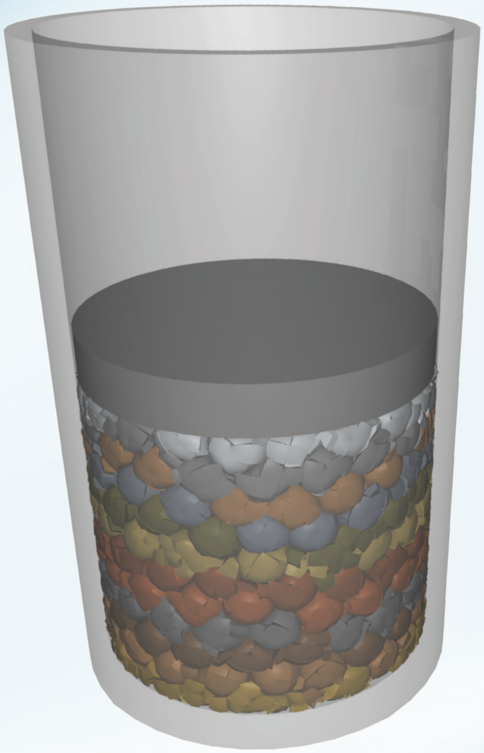




The configuration shown here consists of a movable platen and a cylinder filled with particles. The platen was moved to compress the particles half the distance from the fill line. The particles are modeled as elasto-plastic material which included a damage criteria, allowing for the particles to fracture. This was accomplished by applying the IMPETUS "Node Splitting Algorithm". Contact is specified between the structure and the particles as well as between the individual particles. Contact is also invoked between the particle fragments that are generated as part of the process.



All models ran to normal termination which shows very good element and contact behavior. As examples consider two models, one with 444 particles and a larger model with 1056 particles. Both have 32 cubic elements per particle and twelve layers but the smaller model has 37 particles in each layer. The larger model has 88 particles in each of the twelve layers. The small model, 444 particles took 12 hours and the large model with 1056 particles took 21 hours. With these models showing the robustness and efficiency of the IMPETUS Afea Solver®, it is now possible to investigate the compaction process with regard to process behavior as plastic deformation, fracture, and influence from friction are considered.



References:

[1] J. Zhang, “A Study of Compaction of Composite Particles by Multi-Particle Finite Element Method”, Composites Science and Technology 69 (2009) 2048-2053.
[2] A. T. Procopio et al., “Simulation of Multi-Axial Compaction of Granular Media from Loose to High Relative Densities”, Journal of the Mechanics and Physics of Soilds, 53 (2005) 1523-1551.
[3] P. Redanz et al., “The Compaction of a Random Distribution of Metal Cylinders by the Discrete Element Method”, Acta Mater. 49 (2001) 4325-4335.

Acknowledgement:

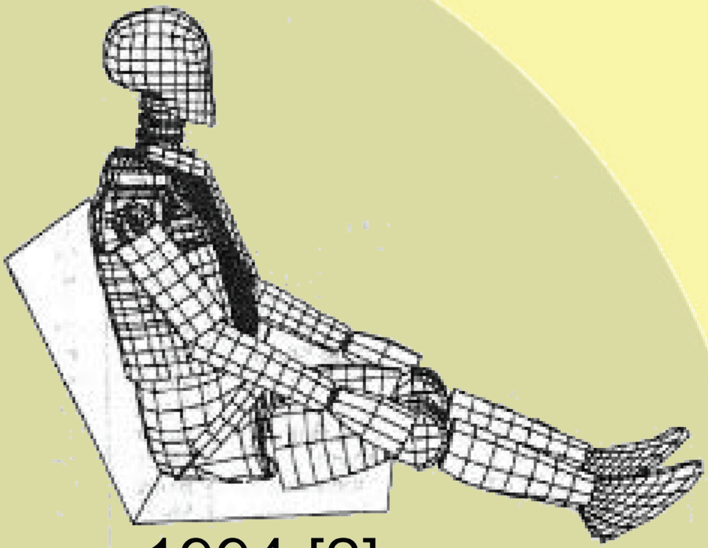
The initial model development was done by MDG Solutions, Inc. under contract with CertaSIM, LLC.

ATD Calibration for Crash – Head Drop Test

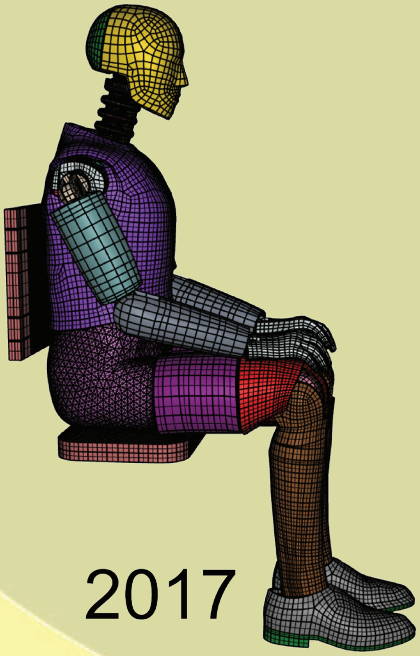
With today's continued military conflicts one of the most dangerous situations for our warfighters is the attack from an Improvised Explosive Devices (IEDs) which results in extensive damage to their vehicles. To develop better protection in the vehicles it is necessary to measure the influence on the passengers. This is accomplished by including an Anthropomorphic Test Device (ATD) as part of a physical test. For simulation this involves a computer model of the ATD. IMPETUS has developed a fully calibrated ATD model based upon the SAE standards but has extended the calibration to include the results from physical blast tests, which is something that has not been done before.

Large improvement has been made in the development of physical ATD since the first one was introduced in 1947. Many variations have been built and tested and a good historical description can be found in [1]. Within the last two decades Finite Element Models have developed which includes very limited and crude models to ones with a high level of details as seen in the IMPETUS ATD.

The IMPETUS Hybrid III 50th Percentile ATD consists of 105 parts and a total of 377,199 nodes. It is unique in the use of the accurate ASET™ elements which leads to a robust and accurate response even for large deformation as seen in mine blast events. In the following and articles to come in the next issues of the CertaSIM Solution Journal, the different crash test calibration set-ups will be highlighted, showing the results as well.



1994 [2]

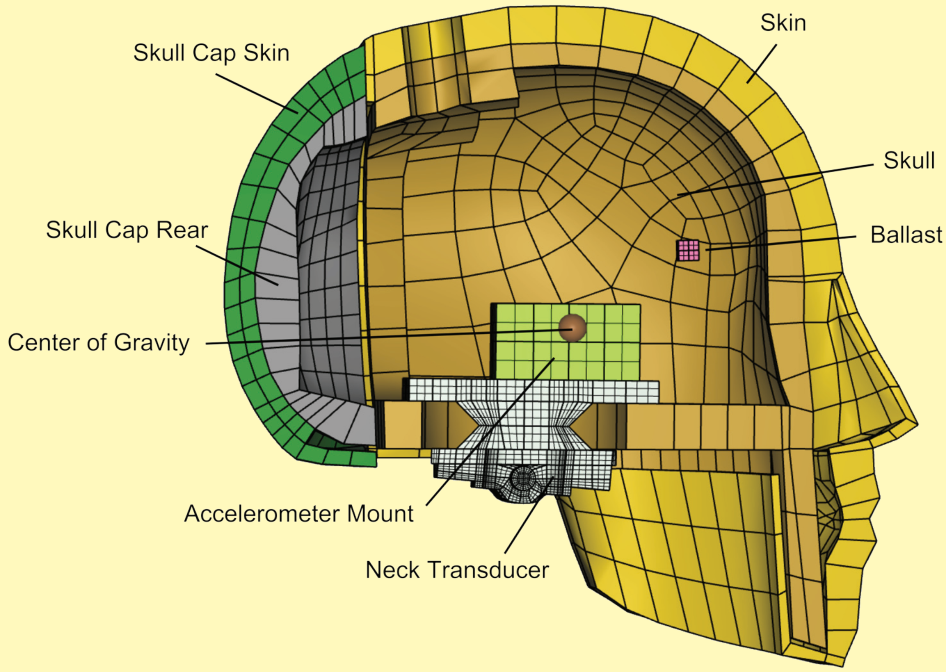


2017

In total nine different calibration tests are carried out [3]:

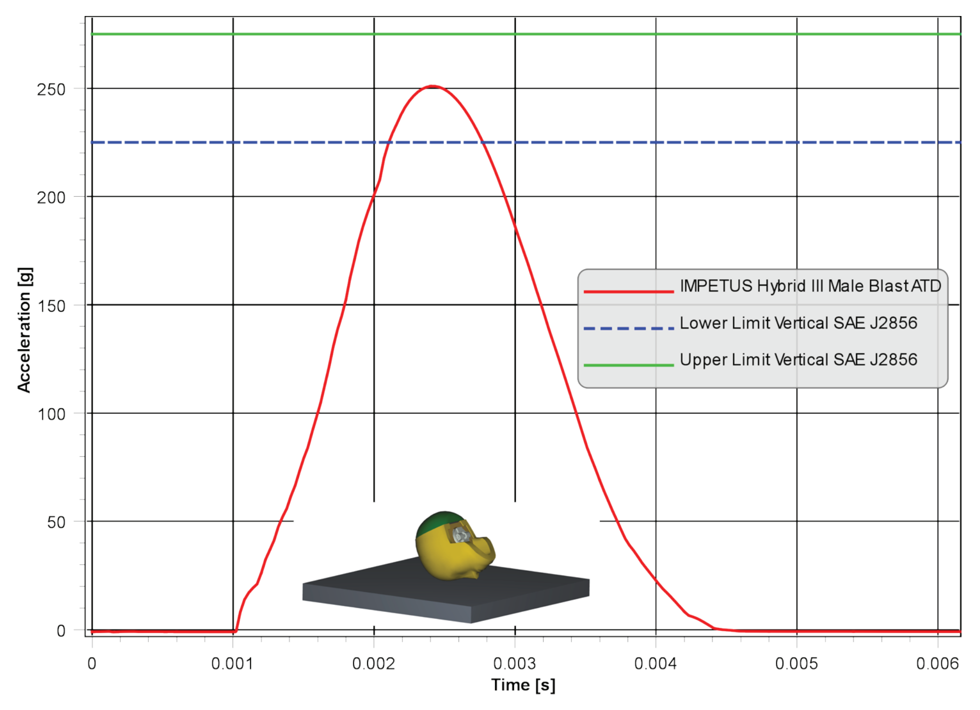
- Head Drop Test
- Neck Flexion Test
- Neck Extension Test
- Thorax Impact Test
- Knee Impact Test
- Knee Slider Test
- Upper Foot Impact Test
- Lower Foot Impact Test
- Static Foot Impact Test

The Head Drop Test makes use of seven parts of the ATD, these are the parts that are referred to as the head assembly. The test follows [4]. This assembly is dropped from a height of 376 mm onto a hard surface (assumed rigid).



The accelerations are measured in g's and the peak resultant acceleration must lie between 225g and 275g. Furthermore, the time history curve shall be unimodal with the requirement that the oscillations after the main peak are less than 10% of the peak. Lastly, the lateral acceleration vector cannot exceed 15g. The head assembly impacts the surface, deforms and rebounds. The model ran to normal termination.

The accelerations can be found in the output file for rigid parts, rigid.out file. The accelerations are measured at the Center of Gravity (COG). The resultant maximum acceleration is found by plotting the acceleration in the global Z direction. The IMPETUS response is within the required band as the peak value is 251g. The specification of a unimodal curve is satisfied as well.



Lateral acceleration must be below 15g. This is found by plotting the acceleration for COG in global X and Y directions. As before the value is found in the rigid.out file. The numerical results from IMPETUS are well below the acceptable maximum value of 15g. The numerical maximum value is 10.7g obtained in the global Y-direction.

References:

[1] AGARD, "Anthropomorphic Dummies for Crash and Escape System Testing", AGARD Advisory Report 330, 1996, North Atlantic Treaty Organization.

[2] Khalil, T. B. and Lin, T. C., "Simulation of the Hybrid III Dummy Response to Impact by Nonlinear Finite Element Analysis", S.A.E. transactions 103(6): 1868-1886, 1994.

[3] M. R. Jensen, "The IMPETUS Hybrid III 50th Percentile Male Blast ATD, CertaSIM, LLC Report, CS# 005209012017 (In Work).

[4] SAE International J2856 September 2009, "User's Manual for the 50th Percentile Male Hybrid III Dummy".

Dr. Lars Olovsson, CTO at IMPETUS AB, Sweden

In a recent interview with Dr. Lars Olovsson, CTO at IMPETUS AB, Sweden, he looks at the future of the software for 2017. Dr. Olovsson is the chief strategist behind the IMPETUS Solver and his staff in Stockholm, Sweden develops the main part of IMPETUS, the structural FE module including the DPM module.

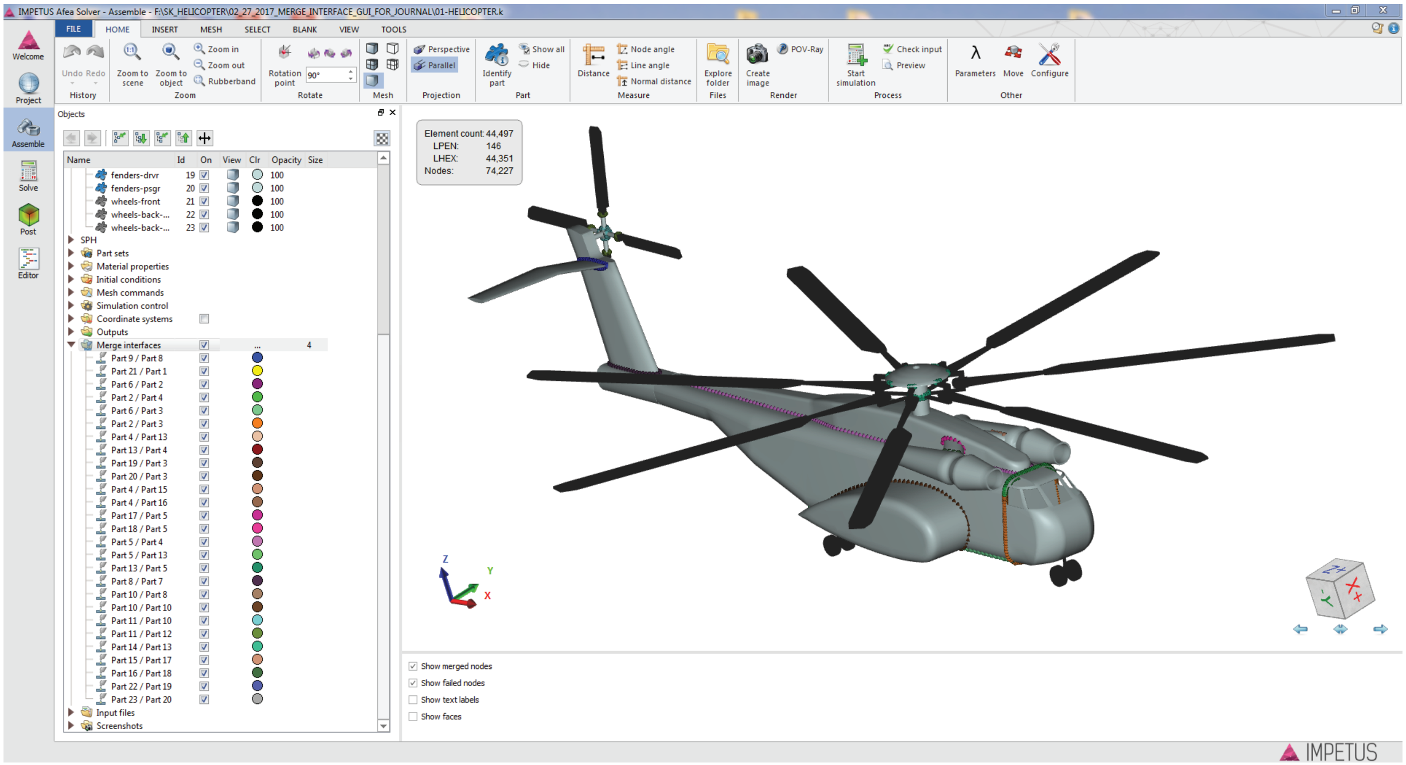
“2017 is an exciting year. We are about to launch a major rewrite of the GUI 3D engine. Besides improved performance and much better rendering of our higher order elements the new framework will facilitate the development of new features.

On the Solver Engine side we are working hard on extending our built-in material library. Limited access to accurate material data is the number one obstacle in our customer’s pursuit of achieving reliable simulation results. We constantly work on identifying and strengthening the weakest links in the modeling chain.

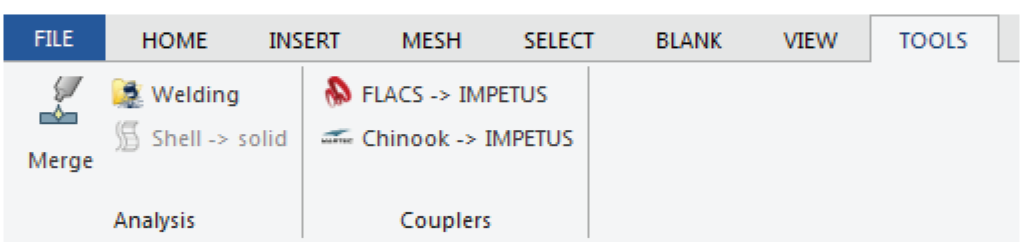
We are also about to release our 50th percentile HIII ATD model. The calibration process has taken longer than expected but a necessary step to provide an accurate model for our customers. However, with great help from the Norwegian Defense Research Establishment (FFI) and CertaSIM the final model will be a very robust and accurate product.”

Merge Interface in the IMPETUS Afea Solver GUI

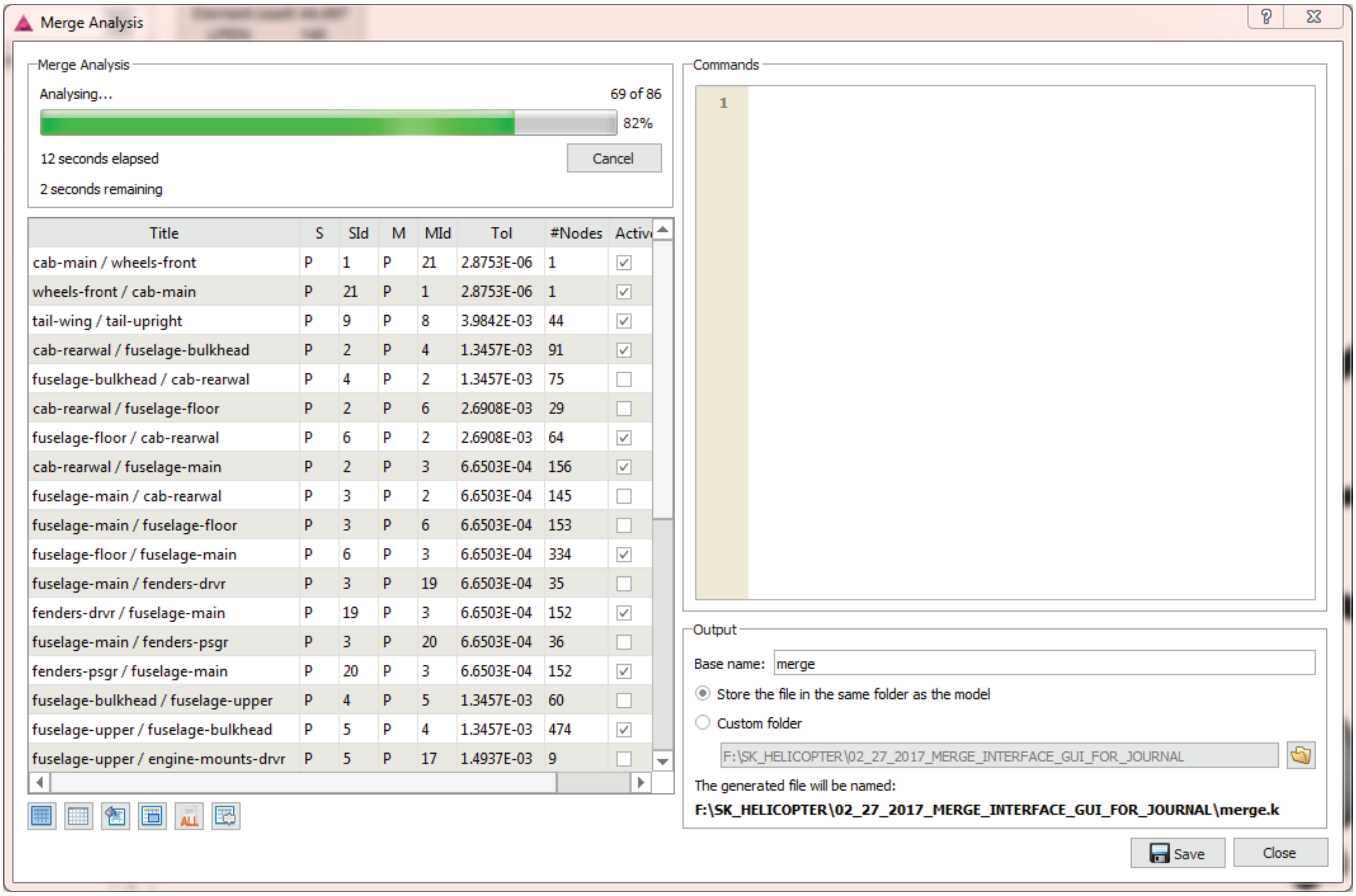
In many models it is very convenient to use the *MERGE feature which will connect parts together without sharing nodes. This means that parts of dissimilar mesh can be connected together using this constraint based method. A failure law can also be added which is commonly used in modeling delamination for composite structures. The different merge commands can be shown visually in the Assemble Mode. Notice that the Size (here shown to be 4) can be clicked on and changed. This adjusts the size of the spheres representing the nodes in the MERGE interface.



Generating these connections can be done by specifying each command but a more powerful method is implemented in the IMPETUS Afea Solver GUI. In the File – Tools Menu there exists a Merge Analysis algorithm that will automatically find free surfaces and generate the file, merge.k, which includes the connection commands including the tolerance.



When activated, the Merge Analysis Selection Interface appears. Here it is possible to select a scale factor for the minimum nodal spacing used to define tolerance applied to identify the merged parts. One can also select symmetric or self-merging. Finally, one selects the parts to include in the Merge Analysis and then click OK to start the process. The progress is shown graphically.



By default all the *MERGE commands will be written to the file merge.k file which can be included into the model with *INCLUDE command. The user can specify another name or folder to save to if desired. As seen, the Merge Analysis feature is fairly simple to use but extremely useful when working with large complicated models.