the CertaSim SOLUTION™

Featuring:

Metal Forming Rolling

Q3–Q4

2018

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
510-963-5485
sales@certasim.com

Support & Training

CertaSIM, LLC
18809 Cox Ave., Suite 150
Saratoga, CA 95070
408-796-7488
support@certasim.com

Technical Writer

Morten Rikard Jensen, Ph.D.
CertaSIM, LLC
925-487-8561
morten@certasim.com

Editor

Wayne L. Mindle, Ph.D.
4717 Sorani Way
Castro Valley, CA 94546-1316
510-963-5485
wayne@certasim.com

Graphics

Kim Lauritsen
CertaSIM, LLC
408-796-7488
kim@certasim.com

News and Events

10th Ground Vehicle Systems Engineering And Technology Symposium (GVSETS)

The 10th Ground Vehicle Systems Engineering and Technology Symposium (GVSETS) was held August 7-9, 2018 at the Suburban Collection Showplace in Novi Michigan. www.ndia-mich.org/events/gvsets

The GVSETS Conference is very special and a must on our list of conferences to attend each year. Pictured here is Dr. Wayne Mindle, the Director of Sales & Marketing at CertaSIM. This year the technical program emphasized material modeling over blast and ballistics modeling that was the focus in the past. There were many good presentations and always a wealth of information to help our team provide better support for our customers.

New Member of the CertaSIM Staff

CertaSIM, LLC is pleased to announce an addition to our technical staff, Mr. Kshitiz Khanna. Mr. Khanna graduated from San Jose State University with a Master's Degree in Mechanical Engineering. His thesis was on the topic of additive manufacturing with a focus on Finite Element simulation of the 3D printing process. He has experience in Explicit Finite Element as well as thermal coupling. His work experience includes three years in the Gas & Oil industry working as a piping engineer on projects in various parts of the world. He will be focusing on technical support, model development for pre-sales and documentation.

Latest Scientific Articles that rely on the IMPETUS Afea Solver®.

The IMPETUS Afea Solver® provides scientists all over the world an accurate and robust tool for simulation. This is evidenced by conference papers and published articles from both Industry and Academia.

Article about metallic foam from NC

Reference: *Jacob Marxa, Marc Portanovab, Afsaneh Rabieia: "A study on blast and fragment resistance of composite metal foams through experimental and modeling approaches", Composite Structures 194 (2018) 652–661.*

In this research at North Carolina State University and Aviation Applied Technology Directorate (AATD), U.S. Army Research, Development & Engineering Center, Fort Eustis, USA, Composite Metal Foam is tested experimentally for a blast wave and fragmentation impact. IMPETUS is applied to model fragmentation impact of the CMF panels the authors concluded: "The model was able to efficiently predict the behavior and energy absorption capabilities of composite metal foam."

Modeling Deflagrations in 20-foot Container – Blind Test

Reference: *T. Skjold, H. Hisken, S. Lakshmipathy, G. Atanga, M. Carcassi, M. Schiavetti, J.R. Stewart, A. Newton, J.R. Hoyes, I.C. Tolias, A.G. Venetsanos, O.R. Hansen, J. Geng, A. Huser, S. Helland, R. Jambut, K. Ren, A. Kotchourko, T. Jordan, J. Daubech, G. Lecocq, A.G. Hanssen, C. Kumar, L. Krumenacker, S. Jallais, D. Miller, C.R. Bauwens: "Blind-prediction: Estimating the consequences of vented hydrogen deflagrations for homogeneous mixtures in 20-foot ISO containers", International Journal of Hydrogen Energy xxx (2018) I-12.*

The work is part of the project "Improving hydrogen safety for energy applications through pre-normative research on vented deflagrations" (HySEA). It is a blind test of numerical models used for estimating the reduced explosion pressure and structural response in vented hydrogen deflagrations. There were two different experimental set-ups each with three tests to ensure repeatability. IMPETUS participated in this blind test where it was coupled with FLACS v10.5 in a one-way coupling. In general good results were obtained with FLACS-IMPETUS, especially for the maximum deflection container walls. In fact only two teams submitted predictions of this important response parameter.

Additively Manufactured Penetrating Warheads

Reference: *Jérôme Limido, Paul Deconinck, Aurélien Beaucamp, Frédéric Paintendre, and Pierre-Louis Hereil: "Additively manufactured penetrating warheads", EPJ Web of Conferences 183, 04007 (2018), DYMAT 2018, https://doi.org/10.1051/epjconf/201818304007.*

The large European developer and manufacturer of missiles MBDA has together with Thiot Ingénierie and IMPETUS Afea SAS conducted research regarding penetration capabilities of an additively manufactured penetrating warhead. The work was presented at the DYNMAT 2018 conference. It is shown that the IMPETUS numerical model predicts the exit velocity within 6.5% error compared to experimental data which is a very accurate result. The authors made the following conclusion: "All numerical simulations have been performed before experiments, so no artificial tuning nor modification within the model were made and they showed good correlation with the perforation tests."

Shock Tube Testing and Modeling of Annealed Float Glass

Reference: *Karoline Osnes, Tore Børvik and Odd Sture Hopperstad: "Shock Tube Testing and Modelling of Annealed Float Glass", EPJ Web Conf. Volume 183, 2018 DYMAT 2018 - 12th International Conference on the Mechanical and Physical Behaviour of Materials under Dynamic Loading, https://doi.org/10.1051/epjconf/201818301035.*

Finite Element modeling of fracture in glass is extremely difficult especially because of the stochastic failure behavior. The SIMLab Shock Tube Facility at Norwegian University of Science and Technology (NTNU) was used to model blast loading on annealed float glass plates. A total of 12 experiments were carried out and numerical work done in order to develop a strength prediction model. Further, IMPETUS was successfully used to model one of the experiments. The model used the ASET™ high order Pentahedron elements and the Node Splitting Algorithm. It was mentioned that: "..., it is seen that the agreement between the simulation and the experiment is very good, and illustrates that the use of node splitting enables a highly realistic behaviour of fracture in glass."

An Accurate SPH Scheme for Dynamic Fragmentation Modeling

Reference: *Anthony Collé, Jérôme Limido, Jean-Paul Vila: "An Accurate SPH Scheme for Dynamic Fragmentation modeling", EPJ Web of Conferences 183, 01030 (2018) https://doi.org/10.1051/epjconf/201818301030, DYMAT 2018.*

A new improved SPH Solver has been developed in France, at the R&D Department, IMPETUS Afea SAS, France. This new Solver has increased stability, improved accuracy and even reduced computational time compared to Legacy SPH Solvers. In a recent paper the algorithm is presented and validated with experiments. One of the test cases is Hyper-Velocity where a sphere impacts a plate at a speed of 6.7 km/s. By looking at the cloud of debris characteristic it is found that very good results that match the experiments are obtained. This is the case for both normal and oblique impact.

Capturing the Behavior of Pipeline Fracture with New Fracture Model

Reference: *Martin Kristoffersen, Tore Børvik, Lars Olovsson: "Pipeline Fracture Due to Compression-Tension Loading Caused by Foreign Object Impact", Proceedings of the ASME 2018 37th International Conference on Ocean, Offshore and Arctic Engineering OMAE2018, June 17-22, 2018, Madrid, Spain.*

At the last ASME International conference on Ocean, Offshore and arctic Engineering, researches from NTNU in Norway and the R&D team at IMPETUS Afea AB, Sweden presented work related to fracture of pipelines due to impact followed by stretching. The work involved implementation of a new fracture model for anisotropic damage and calibration of this model and inverse material modeling for strength parameters as well. The damage model is *PROP_DAMAGE_IMP which is an extension of the classic Cockcroft-Latham fracture model. Several different types of experiments were carried out in this work among them a three point bending dynamic impact load followed by a quasi-static stretch process. The later was also successfully modeled in IMPETUS and the authors concluded: "In general, the new fracture criterion is able to account for the reduction in tensile failure strain caused by a compressive load prior to the tensile phase, and a proof of concept has been established."

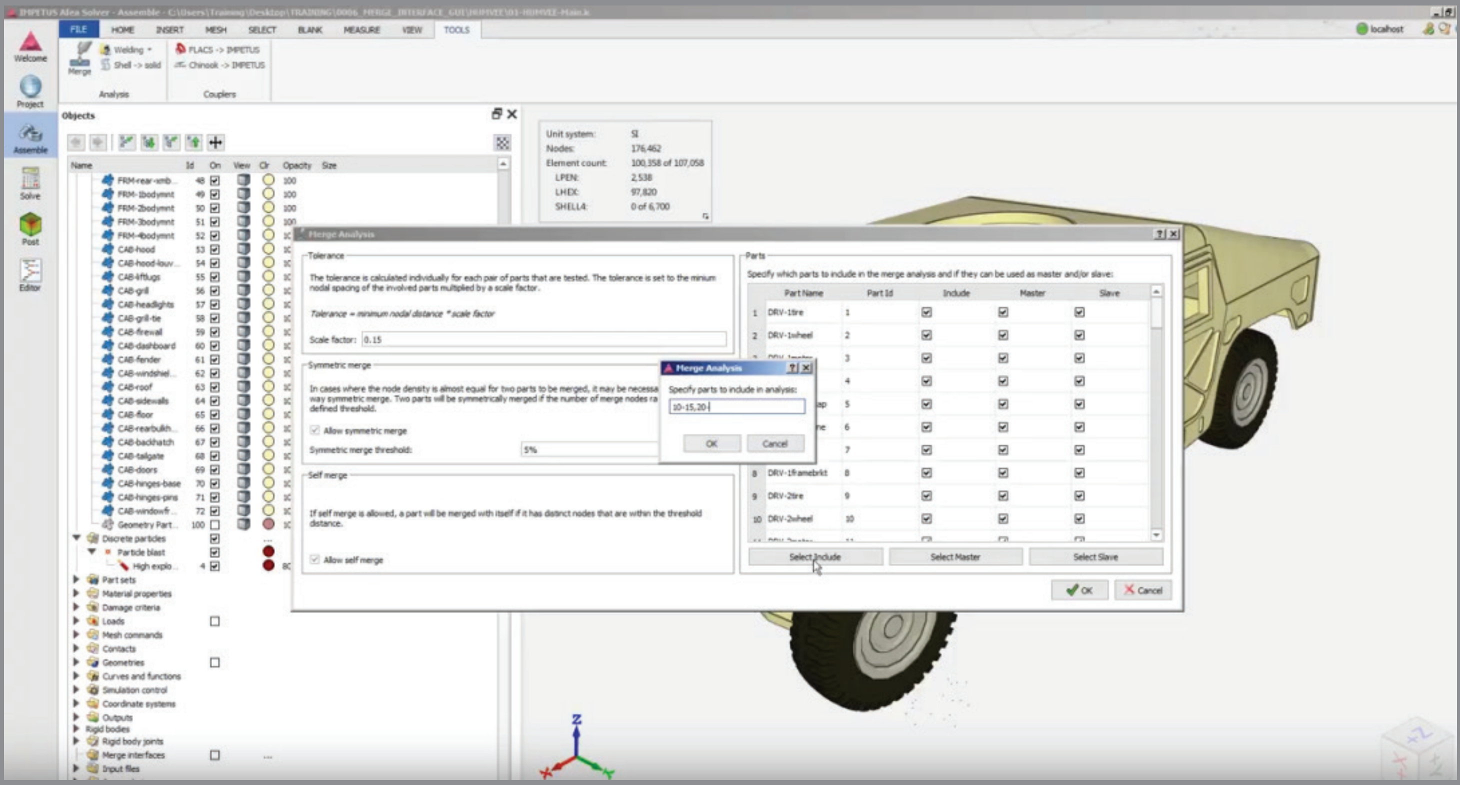
Fragmentation of a 155 mm Artillery Shell used as IED

Reference: *James G. Rasico, Morten Rikard Jensen and Craig A. Newman: "Modelling Fragmentation of a 155 mm Artillery Shell IED in a Buried Mine Blast Event", Int. J. Vehicle Performance, Vol. 4, No. 4, 2018.*

In this work significant numerical testing was performed in order to obtain knowledge about using the IMPETUS Afea Solver® to model fragmentation. To verify against experiments, an example from Lawrence Livermore National Lab of a cylinder containing high explosive was modelled. With the knowledge from this study, a 155 mm artillery shell was studied. The research was divided into three different phases, one where a high explosive was modeled inside the shell, the next phase was to place this in soil and the final phase was to place a generic hull model with an ATD above the soil and HE filled artillery shell, representing an IED. This paper shows that it is feasible to model this very complex scenario with IMPETUS.

Training Material from CertaSIM

To affectively support our customers, CertaSIM provides comprehensive training material. This includes video tutorials on various topics. The latest video shows how to use the Merging Tool in the IMPETUS Afea Solver GUI. This is an easy to use tool to locate and generate *MERGE commands for tying components together. It is especially beneficial for very large models. The video is located at: youtu.be/DXJ40uyraJU



Questions on this feature can be sent to support@certasim.com

New Option in the IMPETUS AFEA Solver Engine

The IMPETUS development teams are constantly working on adding useful and cutting edge features. New implementations also cover well known material models that are used extensively for specific applications.

The Hosford-Coulomb ductile failure criterion is now implemented in IMPETUS as *PROP_DAMAGE_HC. In this implementation the HC criteria has been extended to include strain rate dependent ductility. The criteria can be applied to modeling Sheet Metal Forming of High Strength Steel and it includes dependence on Stress Triaxiality and the Lode Angle.

```
*PROP_DAMAGE_HC
"Optional title"
did, erode, noic
```

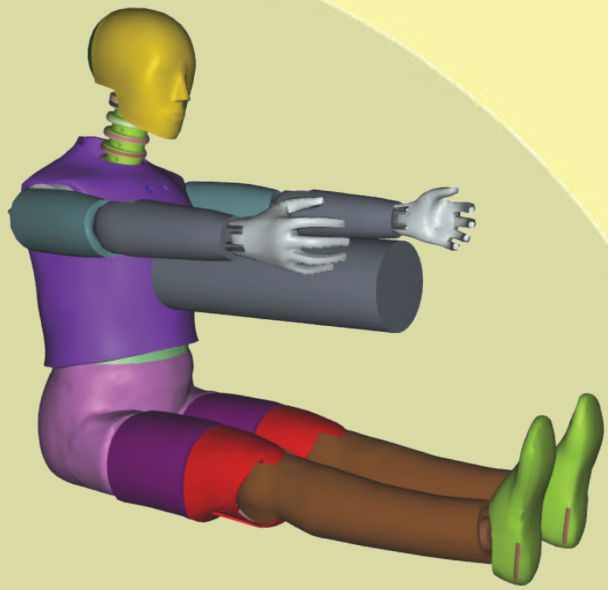
$a,\ b,\ c,\ n,\ \dot{\varepsilon}_0,\ s,\ \sigma_s,\ t_s,\ \alpha_s$

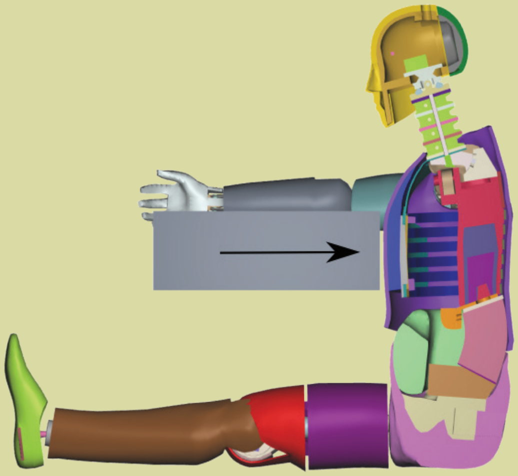
Using the parameters; a, b, c, and n, one has the original model and rate dependence is specified through the and s parameters. The online manual for IMPETUS has a detailed description of the parameters. Further information can be obtained by contacting support@certasim.com.

ATD Calibration for Crash – Thorax Impact Test

With continued military conflicts in the world one of the most dangerous situations for the warfighter is an attack from an Improvised Explosive Device (IEDs) which results in extensive damage to their vehicle and therefore a threat to their life. To develop better protection for the vehicle it is necessary to include the effect that blast loading has on the warfighters that occupy the vehicle. This is accomplished by including an Anthropomorphic Test Device (ATD) as part of the physical test. Simulation of this involves a computer model of the ATD. IMPETUS has developed a fully calibrated ATD model based upon the SAE standards but has, together with CertaSIM, extended the calibration to include the results from physical blast tests, which is something that has not been done before. This series of articles describes the different calibration requirements found in the SAE standards; the following presents the results for the Thorax Impact Test. The Thorax Impact Test is defined in [1]. The chest is impacted by a 23.36 kg rigid probe with an impact velocity of 6.71 m/s.

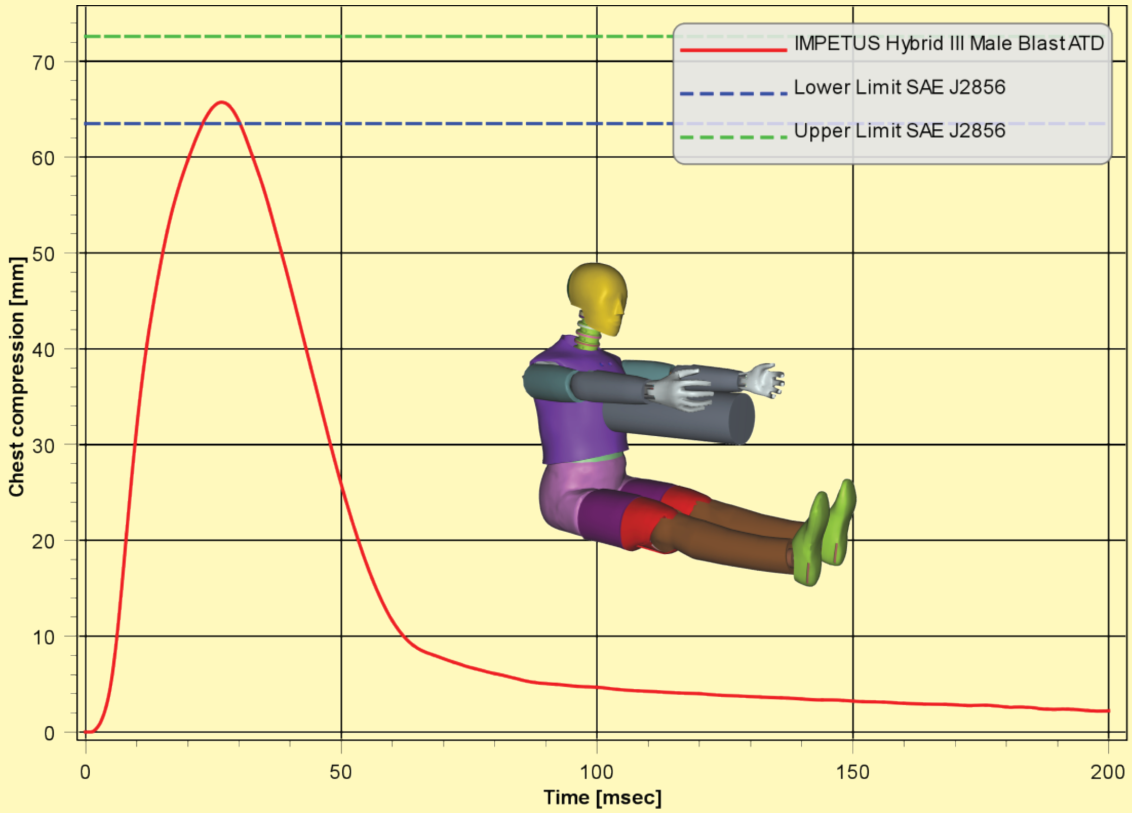
The Thorax Impact Test is the only calibration test that uses the whole ATD only excluding the shoes. It represents an impact of the chest with a 23.36 kg cylindrical shaped impactor with an impact velocity of 6.71 m/s. Details can be found in [1]. The figure shows the IMPETUS model set-up.





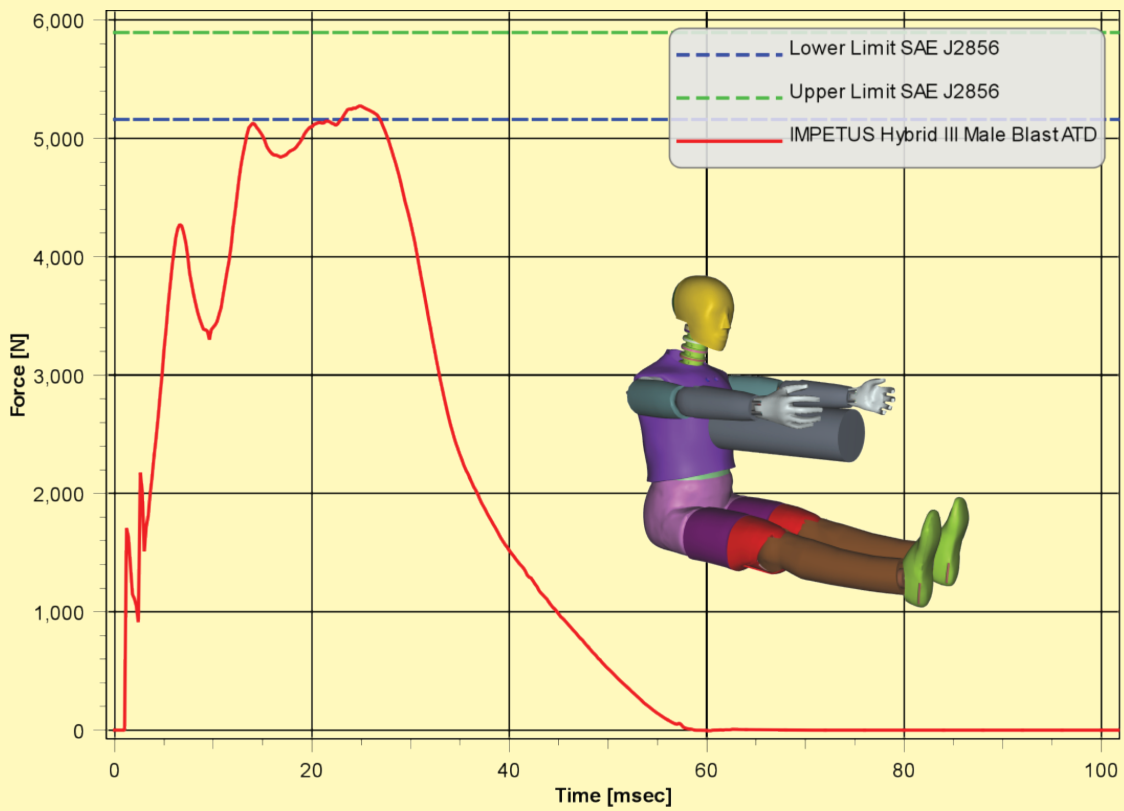
Requirements for the Thorax Impact Test are documented in [1]. The maximum sternum-to-spine deflection must be between 63.5 mm to 72.6 mm. The maximum force applied to the Thorax by the test probe must be between 5160 N and 5894 N. Furthermore, the internal hysteresis ratio must be between 69% and 85%. In the IMPETUS model the deflection is taken from the dummy_sensor_1.out file by plotting the Chest Compression component.

The result is shown in the Figure, where it is seen that the computed value is inside the required band.



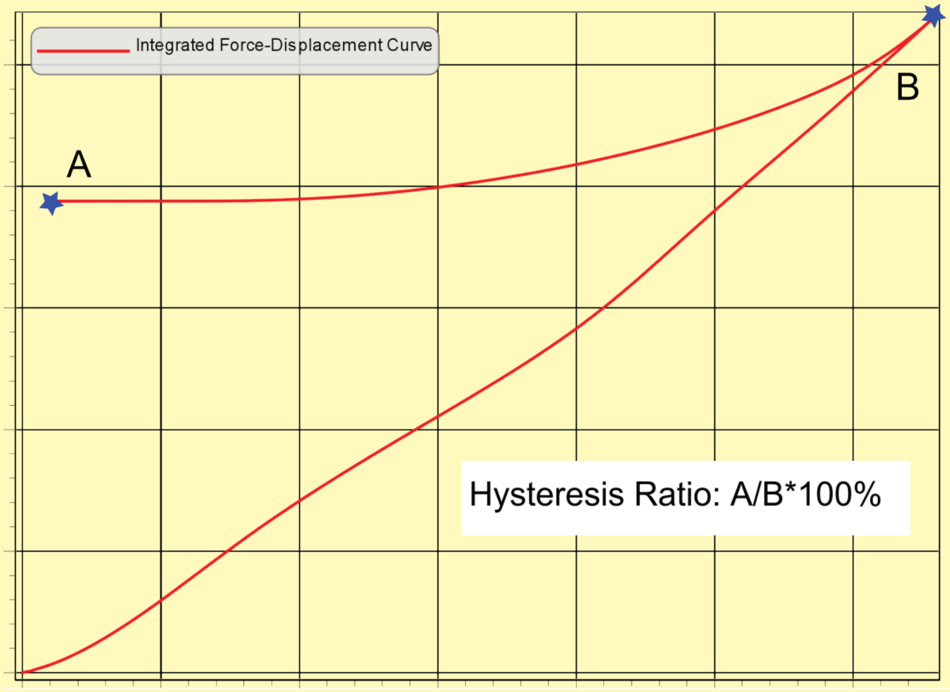
Results for the sternum-to-spine deflection from IMPETUS compared with the requirement from the SAE standard.

The maximum impact force on the Thorax is found by looking at the force related to the probe. This is found in rigid. out, plotting the force in the global X-direction for part 100. As indicated in the following figure, the numerical results are within the required force span.



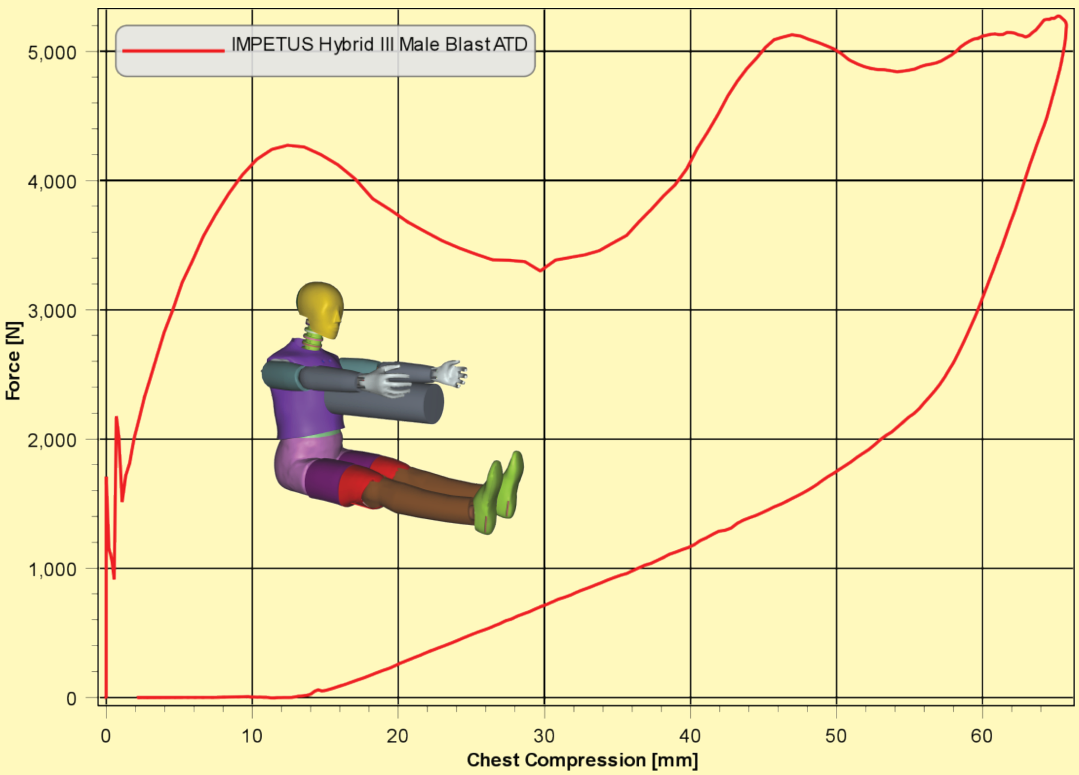
Maximum force result is within the requirements for the Thorax Impact Test.

In order to find the hysteresis ratio, the force for the probe (part 100 in rigid.out) in the global X-direction is plotted against Chest Compression found in dummy_sensor_1.out. The curve is then integrated and the second value on the X=0 axis is divided by the Y-value to the maximum X value. This is "the internal hysteresis ratio", the following figure illustrates the values to use.

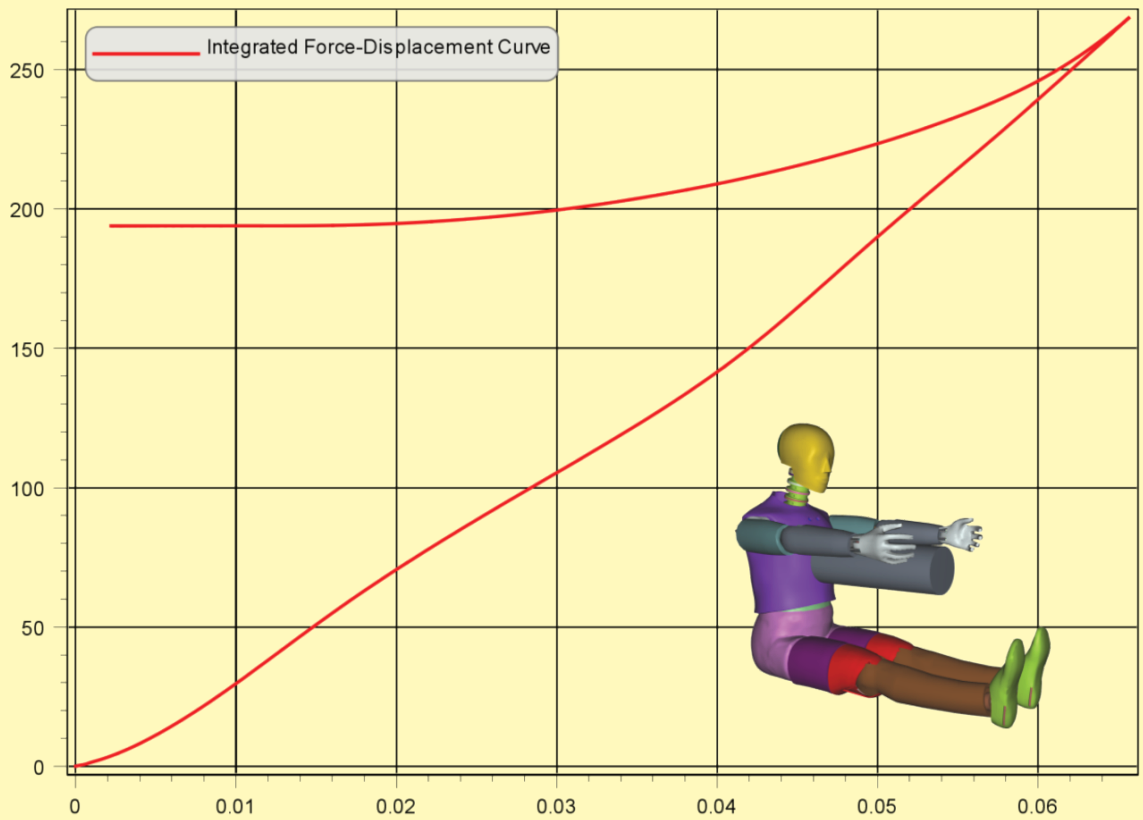


The curve as a result of integrating the force versus displacement curve. This curve is used to find "the internal hysteresis ratio" which is done based on the values of A and B as listed in the graph.

The ratio must be between 69% and 85% which is the case since the ratio is found to be (193.954/268.9)*100% = 72% in the IMPETUS model. The curve for the force versus displacement is shown in the figure below and the integrated curve is given in the following. Notice that the integration is done on the force-displacement curve with consistent units, not what is plotted in the figure.



The force versus displacement for the Thorax simulation.



Resulting curve of the integration of the force versus deflection curve given in the previous figure.
Note that the units for the integrated curves are the same units of N and mm.

Additional information about the IMPETUS Model for this test can be found in [2] which also covers the blast calibration done for the IMPETUS Afea Blast ATD.

References:

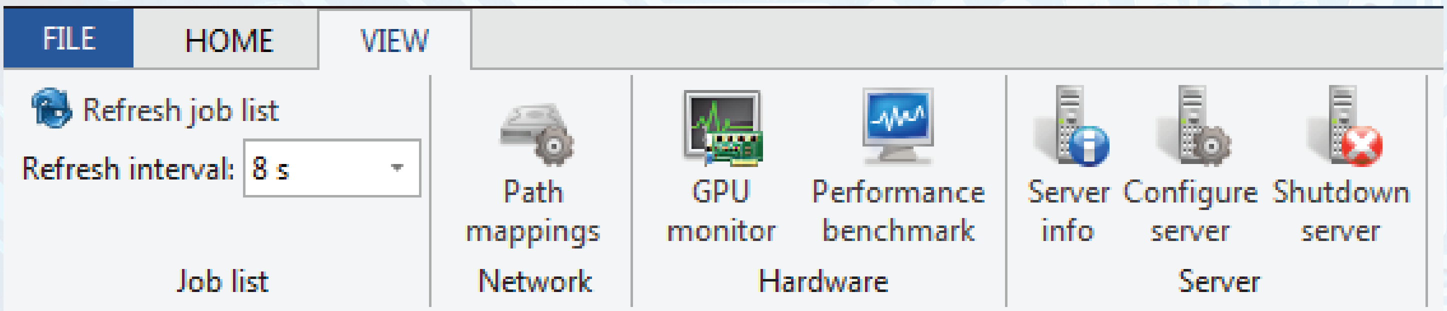
[1] SAE International J2856 September 2009, “User’s Manual for the 50th Percentile Male Hybrid III Dummy”.

[2] M. R. Jensen, “The IMPETUS Hybrid III 50th Percentile Male Blast ATD”, CertaSIM Report # CS-0052-09012017.

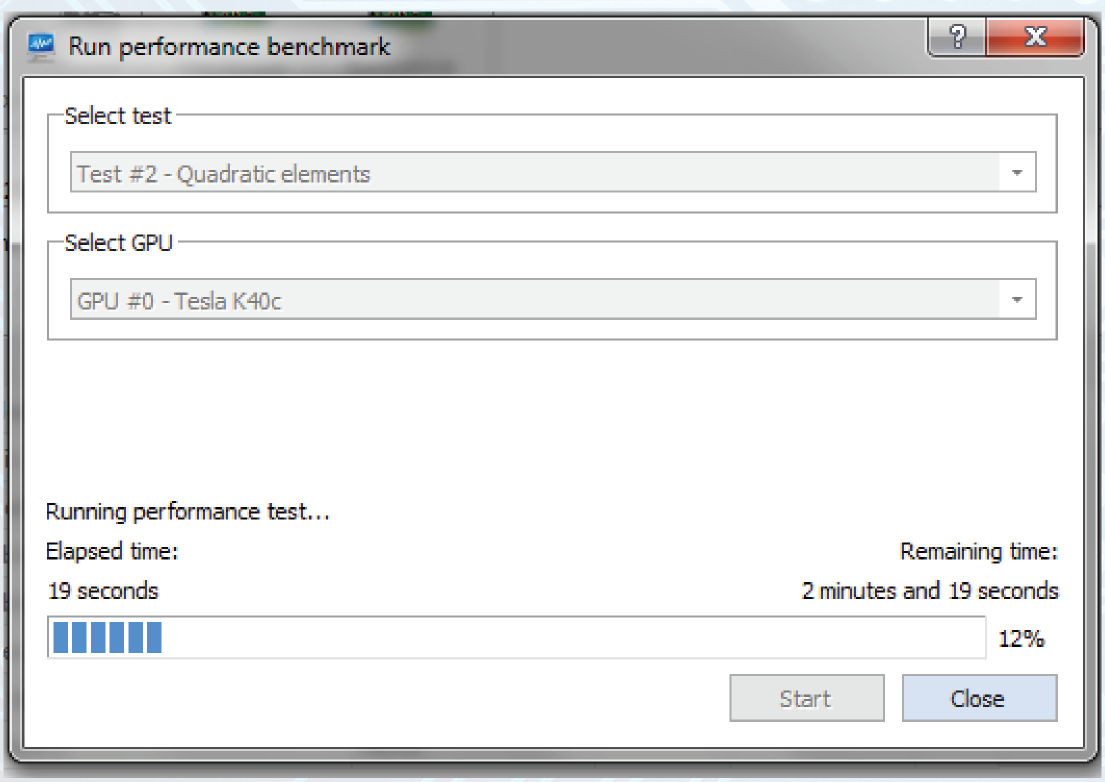
Benchmark Testing of Hardware Systems

Hardware changes happen rapidly in today's IT environment and as a result increased speed is seen with the IMPETUS Afea Solver®, especially with the frequent performance updates by NVIDIA for their GPU processors. To have a standard set of tests to compare performance of different systems, IMPETUS has included a benchmark suite of models in the GUI for customers to run and get a score that can be used to rank different systems. This article shows the interface to use this option.

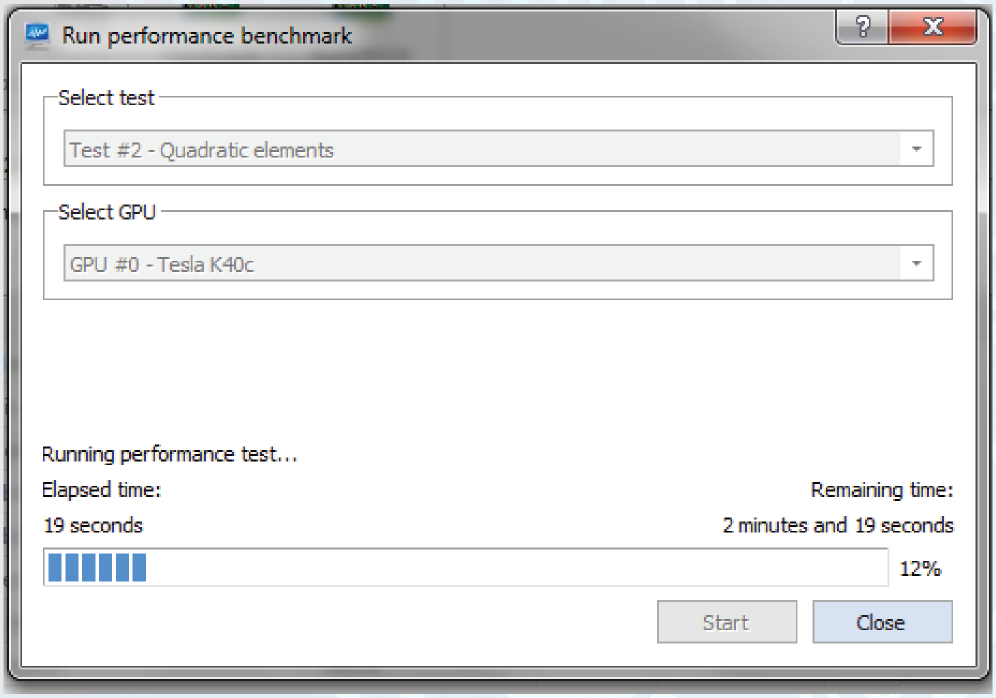
In the IMPETUS Afea Solver GUI one can test hardware performance. When selected a set of benchmark tests will be submitted and after all have finished a score for the hardware system is given. A score of 100 and over indicates a good score.
To activate the tests go to the Solve Mode and click on the VIEW button.



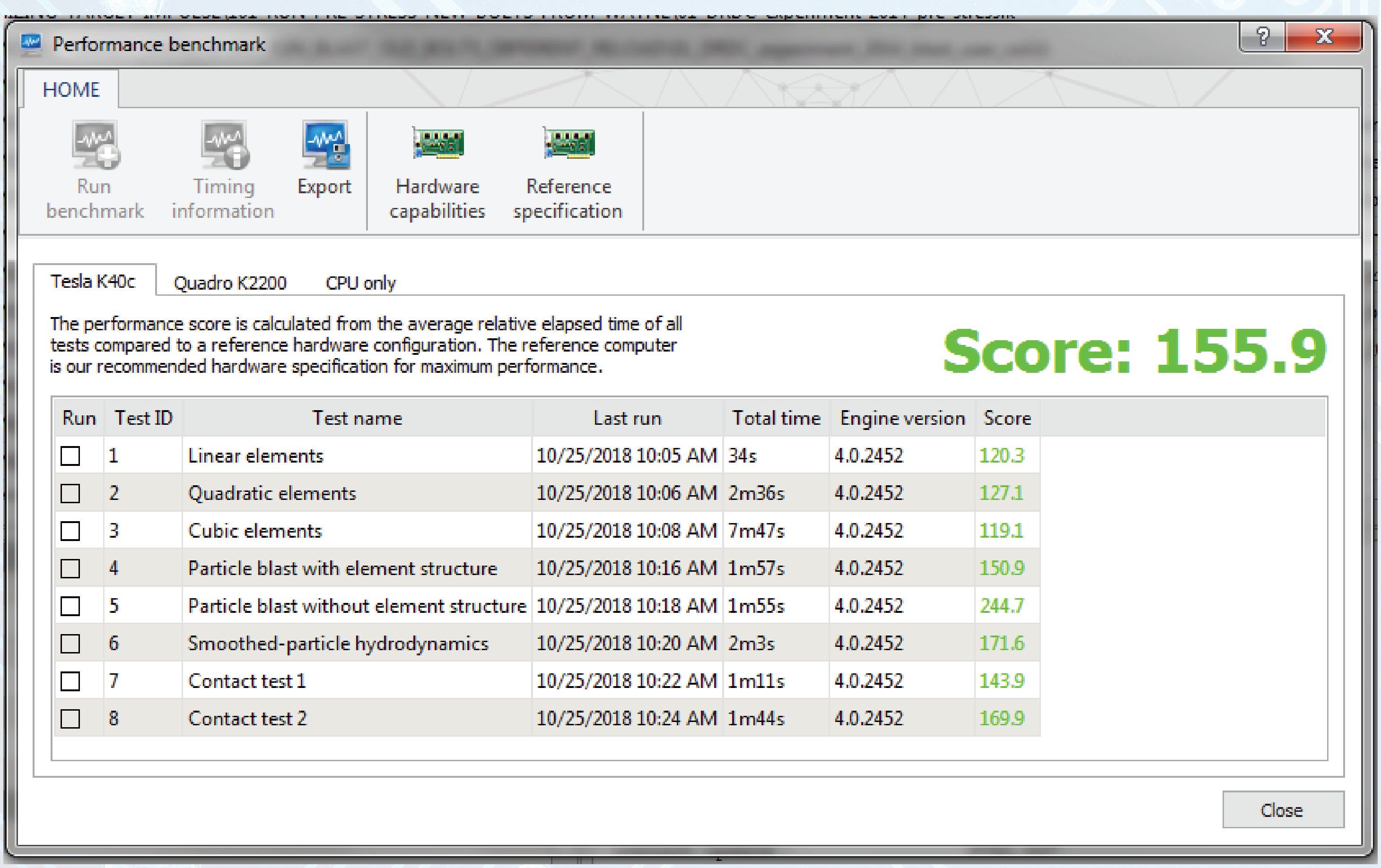
In this menu click on the Performance benchmark icon under the Hardware section.



Now it is possible to select the different tests and the running dialog box will appear.



After the tests are done the results will be listed, including the final score. In this example the score is 155.9 which indicates a good system for running IMPETUS.



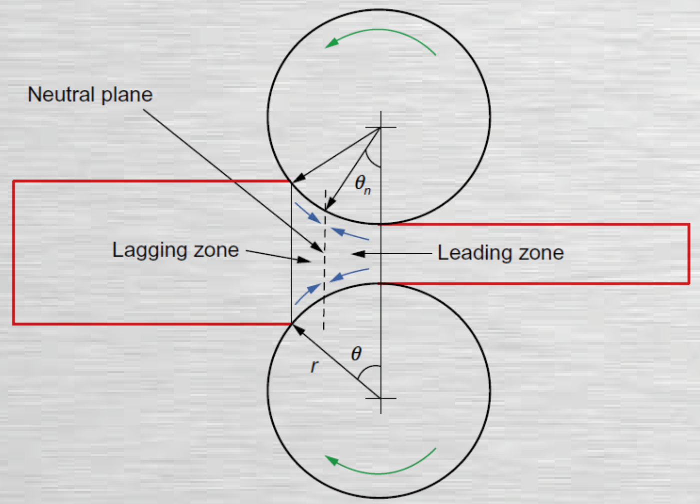
Run	Test ID	Test name	Last run	Total time	Engine version	Score
☐	1	Linear elements	10/25/2018 10:05 AM	34s	4.0.2452	120.3
☐	2	Quadratic elements	10/25/2018 10:06 AM	2m36s	4.0.2452	127.1
☐	3	Cubic elements	10/25/2018 10:08 AM	7m47s	4.0.2452	119.1
☐	4	Particle blast with element structure	10/25/2018 10:16 AM	1m57s	4.0.2452	150.9
☐	5	Particle blast without element structure	10/25/2018 10:18 AM	1m55s	4.0.2452	244.7
☐	6	Smoothed-particle hydrodynamics	10/25/2018 10:20 AM	2m3s	4.0.2452	171.6
☐	7	Contact test 1	10/25/2018 10:22 AM	1m11s	4.0.2452	143.9
☐	8	Contact test 2	10/25/2018 10:24 AM	1m44s	4.0.2452	169.9

Any questions about the performance test or hardware can be directed to CertaSIM, LLC by contacting support@certasim.com.

Modeling Hot Rolling with Finite Elements

Rolling of a work piece to a smaller thickness or a certain shape is a bulk forming process. The process can be used to manufacture many different items either directly or for material to be used in further production. Hot Rolling forming is a complex process that requires significant knowledge. This is an area where Finite Element Modeling can be extremely beneficial to optimize the tooling and the process to obtain better thickness distribution. This article summarizes a literature study on both the rolling process and to simulate it. Some of the main parameters are listed and difficult numerical issues are discussed.

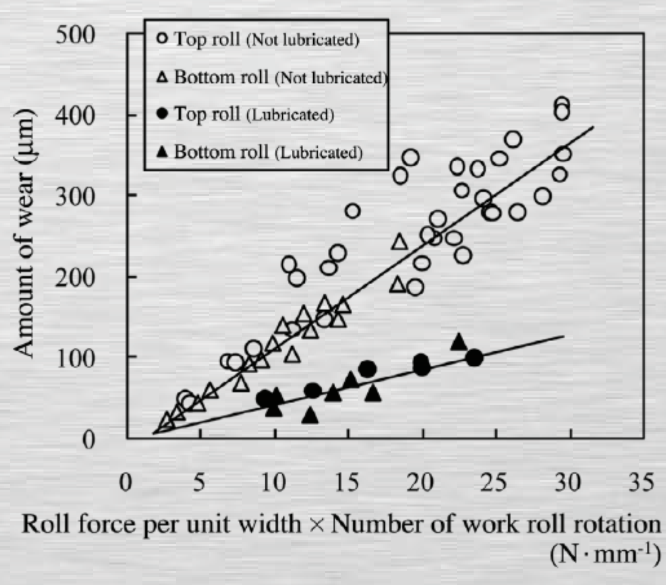
There are many different types of rolling that include Hot Rolling, Cold Rolling, Thread Rolling, Surface Rolling, Slab Rolling etc., see e.g. (Lange, 1985) for a description of the most common ones. In some applications, complex profiles are rolled and in others it is a slab rolled to very thin plates or coils. Generally speaking a work piece is deformed between two rolls and sometimes also deformed from the side with an edge roller. A sketch of the rolling process is shown below, note the definition of the various zones.



Schematic figure of the rolling process (Davim, 2017).

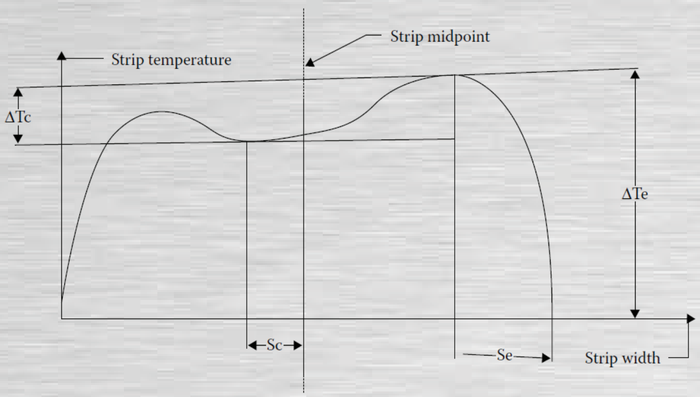
There are in general three zones specified by the velocity of the work piece. These are described in (Lange, 1985) and in (Davim, 2017): Entry Zone (Lagging Zone) where the horizontal component of the roll circumferential velocity is greater than the velocity of the work piece. Neutral Plane, where the velocity of the roll is equal to that of the work piece. In the Leading Zone (Exit Zone) the velocity of the work piece is less than the velocity of the roll. The θ angle shown in the figure is the Roll Bite Angle which is a critical parameter for rolling and it depends on the roll radius and the thickness reduction. If the rolling temperature of the work piece is above the recrystallization temperature for the metal then the process is called Hot Rolling and is typically between 1050 and 1280 °C (Moratal, 2010). The temperature is an important parameter in the rolling process and depends strongly on the material rolled. An interesting point is given in (Ogawa, 2012) pointing out that the industry is aiming to lower the process temperature to save energy and hence money. Other process parameters are friction between rollers and the speed of the roller(s).

Related to friction and wear of the rollers is the Rolling Lubrication which is used to decrease the friction to minimize the tool wear but on the other hand a certain amount of friction is needed to maintain the biting and avoid slipping. In (Ogawa, 2012) the influence of lubrication on the tool wear is investigated and discussed as illustrated in the figure.



Frictional effect from the lubrication on the tool wear (Ogawa, 2012).

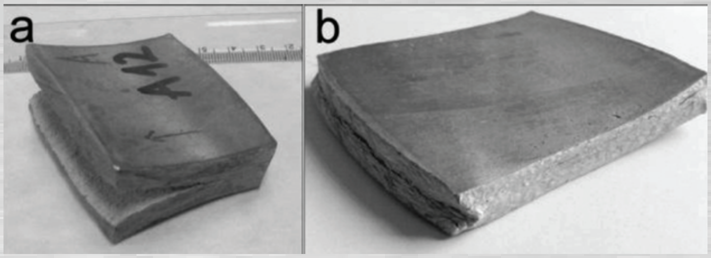
Response Parameters for the rolling process includes; rolling force, pressure profile, thickness distribution and temperature distribution. The latter is the topic of the research done in (Zhang, 2009) where the transverse temperature distribution is considered for 228 specimens and the most typical distribution is shown in this figure.



A typical strip transverse temperature distribution (Zhang, 2009).

It is seen that there is a higher temperature in the center than is observed at the edges of the specimen. The four defined parameters shown in the figure are in the article used to describe the temperature measurement. One finding is that higher strip temperature gives a smaller variation in the average transverse temperature. The rolling force is also named Roll Separation Force (Lange, 1985) and is a very often used as a Response Parameter. It is influenced by several parameters as mentioned in (Dwivedi, 2012), larger rolling velocity or larger radius of the roller will in both cases increase the rolling force. Often the rolling force increases to a plateau and at the end of each pass it then drops again. According to (Soulami, 2014) one of the most common issues associated with rolling is the non-uniform thickness. It can vary along the length of the rolled strip as well as across the width. In their work they also obtained "Dog-boning" which is thickening of the end edges of the work piece.

There are several known defects that can occur in the rolling process such as Edge Cracking, Alligatoring and Fish-Tail, etc. A good description of these, their causes and remedies are listed in (Al-Mousawi, 1992). It is mentioned that for Hot Rolling edge rolls can prevent rounded edges. The Alligatoring defect is characterized by an opening of the rolled slab ends due to a crack formation along the central horizontal plane of the slab (Romhanji, 2016). There can still be a concave front without having the Alligatoring defect which is illustrated in the figure.



Defects in rolled slaps. (a) Alligatoring effect showing the large splitting. (b) Concave end profile without the Alligatoring effects (Romhanji, 2016).

Looking through the literature it seems that the rolling process is sensitive to the mentioned process parameters and there are not a lot of conclusive, agreeable findings, at least not in the open literature. As an example consider the transverse temperature distribution where (Ginzburg, 2009) states in the conclusion: "*In considering all the simulation methods and results presented in the literature about the issue of strip transverse temperature distribution, we found a large amount of different data. This uneven amount of results clearly shows the need to fit the simulation models with some real measurements, which unfortunately are still difficult to obtain because of measuring hardware limitations and measuring difficulties*".

Based on this and due to the complexity of the process in general, Finite Element modeling of the process is not an easy task. There is large deformation which requires remeshing if many passes are to be modeled. In (Shiekh, 2006) the meshing topic was investigated, i.e. if one mesh could be used for all passes. They conclude, based on their specific model, that: "*Although this method was the easiest, the solutions achieved were not accurate and produced several overlapping … and thinning instances*". Adaptive meshing was applied in the work by (Tripathi, 2014) where attention especially is on the influence from friction between rollers and work piece.

As mentioned in (Rowe, 1991) a full FEM model should be used, though it has been common to consider the process as plane strain (Davim, 2017) and too apply either quarter or half symmetry. Modeling the work piece with solids will increase the computational time considerably. As an example Bagheripoor (2011) used the Legacy Solver Abaqus† to simulate a quarter model which took at least 11 hours to complete one pass.

Another thing to consider is modeling of the tools, i.e., the rollers. Different approaches have be used as in (Soulami, 2014) where the rolls are rigid shells which makes it easier to apply the motion. However, in (Cavaliere, 2001) it is stated: "*When developing a 3D finite element model, for simulating the hot rolling of steel plates, it is necessary to include in this model the rolls deformation, because it plays a central role in determining the resulting profile and flatness of the rolled plates.*"

The value of the friction is one of the more difficult process parameters to specify and in the literature it seems that many different values are applied. In general it should be high in order for the rollers to grip the work piece and continue the rolling process. In (Rowe, 1991) and (Shahani, 2009) a value of 0.5 is considered, in (Cavaliere, 2001) a value of 0.7 and in (Sheikh, 2006) a very low value of 0.01.

Not only the process parameters but also the numerical description of the material behavior is important which is true for all applications modeled with Finite Elements. In the Hot Rolling case the temperature influence increases the complexity of the constitutive model. In (Duan, 2004) three different commonly used models are compared with regards to temperature, strain and strain rate. In (Davim, 2017) an overview of constitutive models applied in different research publications is discussed.

In the "Modeling Hot Rolling – A Case Study" article later in this issue a case study of Hot Rolling is presented where the IMPETUS Afea Solver® is applied. The article shows the commands that are used to include sensors and functions to provide an effective and robust methodology to modelling the rolling process.

References:

(Lange, 1985) K. Lange, "Handbook of Metal Forming", Society of Manufacturing Engineers, ISBN: 0-87263-457-4, 1985.

(Davim, 2017) J. Paulo Davim, "Computational Methods and Production Engineering", Elsevier, ISBN 978-0-85709-481-0, 2017.

(Bagheripoor, 2011) M. Bagheripoor and H. Bisadi, "Effects of Rolling Parameters on Temperature Distribution on the Hot Rolling of Aluminum strips", Applied Thermal Engineering 31 (2011) 1556-1565.

(Moratal, 2010) D. Moratal, "Finite Element Analysis", Sciyo, ISBN 978-953-307-123-7, 2010.

(Zhang, 2009), J. Zhang, L. Tian, P. Patrizi and F. Miani, "Chapter 32: Analysis of the Transverse Temperature Distribution in the Hot Strip Mill of a Compact Strip Production Plant", in "Flat-Rolled Steel Process – Advanced Technology", Editor V. B. Ginzburg, CRC Press, Taylor & Francis Group, ISBN 978-1-4200-7292-1, 2009.

(Sheikh, 2006) M. A. Sheikh and R. R. Palavilayil, "An Assessment of Finite Element Software for Application to the Roll-Forming Process", Journal of Materials Processing Technology 180 (2006) 221-232.

(Romhanji, 2016) E. Romhanji, T. Radetic and M. Popovic, "Homogenization of an Al-Mg Alloy and Alligatoring Failure: Alloy Ductility and Fracture, MTAEC9, 50(3)403(2016), ISSN 1580-2949.

(Cavaliere, 2001) M. A. Cavaliere, M. B. Goldschmit, E. N Dvorkin, "Finite Element Simulation of the Steel Plates Hot Rolling Process", International Journal for Numerical Methods in Engineering 2001; 52:1411-1430.

(Rowe, 1991) G. W. Rowe, C. E. N. Sturgess, P. Hartley and I. Pillinger, "Finite-Element Plasticity and Metalforming Analysis", ISBN 10-0-521-38362-5, 1999.

(Soulami, 2014) A. Soulami, C. A. Lavender, D. M. Paxton and D. E. Burkes, "Rolling Process Modeling Report: Finite-Element Prediction of Roll-Separating Force and Rolling Defects", U.S. Department of Energy, Pacific Northwest National Laboratory, Report PNNL-23313, April 2014.

(Ogawa, 2012) S. Ogawa, Y. Miura, T. Shiraishi, T. Inoue, T. Akashi, Y. Nakamura, S. Uchida, K. Yamada, Y. Serizawa, Y. Kondo, S. Hayashi and T. Saito," Progress and Prospect of Rolling Technology", NIPPON Steel Technical Report No. 101 November 2012.

(Shahani, 2009) A. R. Shahani, S. A. Nodamaie and I. Salehinia, "Parametric Study of Hot Rolling Process by the Finite Element Method", Transaction B: Mechanical Engineering Vol. 16, No. 2, pp. 130-139, April 2009.

(Tripathi, 2014) A. Tripathi and A. Medhavi, "Finite Element Simulation of Hot Rolling for an Aluminium 2024 Plate", International Journal of Scientific and Research Publications, Volume 4, Issue 12, December 2014.

(Duan, 2004) X. Duan and T. Sheppard, "The Influence of the Constitutive Equation on the Simulation of a Hot Rolling Process", Journal of Materials Processing Technology 150 (2004), 100-106.

(Dwivedi, 2012) S. Dwivedi, G. Agnihotri and K. K. Pathak, "Parametric Study of the Hot Rolling Process Using FEM", International Journal of Engineering Research and Applications, ISSN: 2248-9622, Vol. 2, Issue 5, September-October 2012, pp. 1517-1522.

(Al-Mousawi, 1992) M. M. Al-Mousawi, A. M. Daragheh, S. K. Ghosh and D. K. Harrison, "Some Physical Defects in Metal Forming Processes and Creation of a Data Base", Journal of Materials Processing Technology, 32 (1992) 461-470.

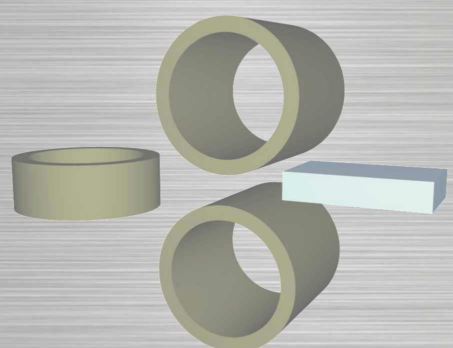
Notes:

† Abaqus is a trademark of Dassault Systèmes.

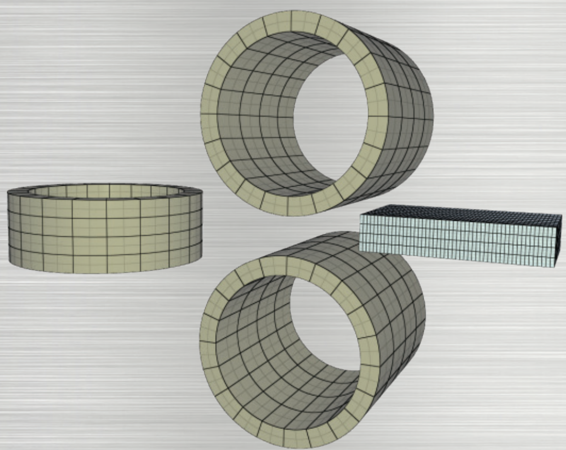
Hot Rolling in the IMPETUS Afea Solver® – Case Study

Numerical modeling of the Hot Rolling process has shown to be difficult and the knowledge is often proprietary to each company leading to a lack of information about this type of simulation. Due to the increase in customers applying IMPETUS to rolling, CertaSIM, LLC has put together with IMPETUS Afea AB, Sweden a case study that showcases the rolling process. This article discusses the numerical model used in the study.

A case study was performed for a steel rolling process using the IMPETUS Afea Solver®. The set-up is a steel slab with initial dimensions of 1 meter in length, 1.2 m in width and a thickness of 0.2 m. There are 3 rollers: bottom, upper and edge. Half symmetry is applied to the model in order to reduce computational time but a full model can easily be run.



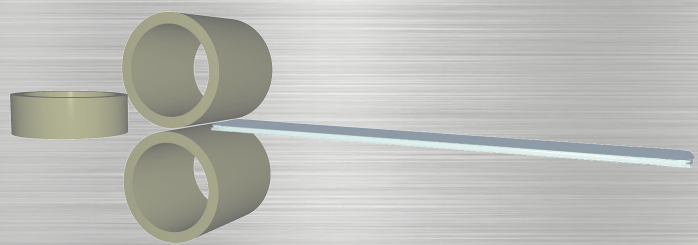
The tools are modeled as rigid parts but can be changed to deformable parts. A gradient mesh is used for the slab in the global y-direction with a finer mesh at the edge. This is easily accomplished with the *TRANSFORM_MESH_CARTESIAN command. All parts are created with the built-in mesh commands *COMPONENT_options avoiding the need for a pre-processor to generate the mesh. Material for the slab is specified with the temperature dependent material model, *MAT_CREEP. In this constitutive model the strength and creep parameters can be given as functions of temperature which was done in the current model. Six passes were used to reduce the slab thickness to around 37% of the initial thickness which is done without any remeshing! This is possible since the Aset™ cubic high order element formulation is used to model the slab. Tools are modeled with quadratic elements and smoothing is done on the rollers to obtain accurate and realistic contact surfaces.



The rollers have prescribed spin and the Upper Roller has an additional vertical displacement applied. Both motions are tabulated with the newly added *TABLE command. Gravity is also added to the whole model. Time and motion features are controlled by sensors where the positions are measured at each end of the slab. The slab reversal is done using an applied force to the slab in the horizontal direction. Magnitude of this force is determined by the mass and acceleration whereas the direction is based on the coordinates of the sensors.

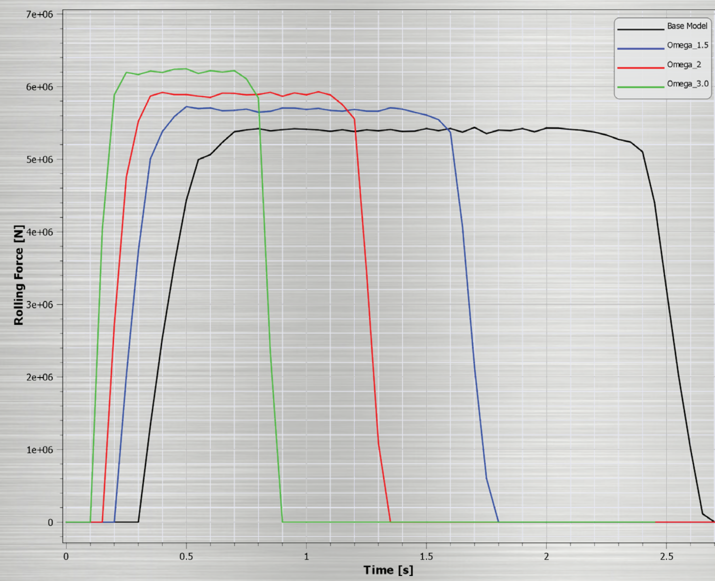
For the rolling process, the temperature field is very important as mentioned in the Rolling Literature Review Article found in this issue. Both the tools and the slab have thermal properties assigned with *PROP_THERMAL and the slab is initially heated to 1250 K which would characterize the process as hot rolling. Thermal convection is given by using the *LOAD_THERMAL_SURFACE and thermal conductivity is also given in the contact definitions.

The model was successfully simulated in IMPETUS showing the strength of the cubic elements. It took around 6 hours of computational time for the six passes and half symmetry utilizing an Nvidia Quadro P5000 GPU for massively parallel processing on a standard workstation.



Large deformation is obtained at the end of the slab though no Alligatoring is seen, which is a common defect in the hot rolling process. However, a concave front profile was seen. This behavior is also observed in the experimental work described in (Romhanji, 2016).

With a successful and stable numerical model it is now possible to research different process parameters, numerical parameters and optimize the process by changing slab geometry, etc. As an example consider the influence from the magnitude of the roller spin on the rolling force. In general the level of this force increases with increasing roller spin which is verified in the figure for different test cases where the values where changed.



Other interesting parameters to investigate are the thickness and temperature distribution, not to mention a direct comparison with experimental results. This R&D work is currently been carried out at CertaSIM, LLC.

The model presented is available from CertaSIM, LLC by contacting support@certasim.com.

References:

(Romhanji, 2016) E. Romhanji, T. Radetic and M. Popovic, "Homogenization of an Al-Mg Alloy and Alligatoring Failure: Alloy Ductility and Fracture", MTAEC9, 50(3)403(2016), ISSN 1580-2949.

Acknowledgement:

The presented model was developed by Dr. Lars Olovsson, IMPETUS Afea AB, Sweden and the release together with in-depth discussion about the model is greatly appreciated.

Component Modeling – Easy model building

This article presents the commands to use in the Editor Mode to create simple geometric components. These commands make it possible to specify the component geometry and mesh size. Using this feature avoids the use of a Pre-Processor and makes small geometric changes extremely easy and fast leading to increased productivity.

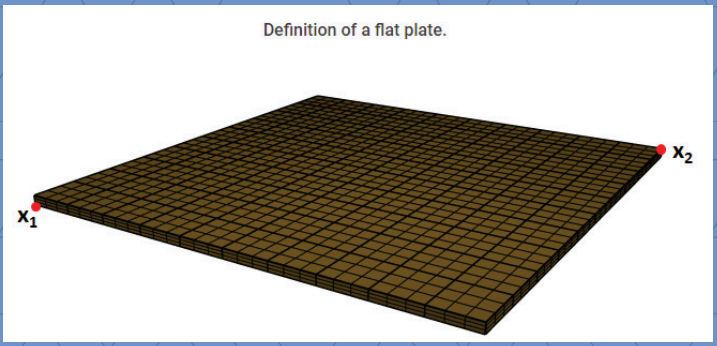
In many cases simple geometries are enough to obtain the simulation goals. A very flexible and useful option in the IMPETUS Afea Solver® is the set of *COMPONENT_ commands that makes it possible to avoid meshing in a pre-processor. The options are:

_BOLT
_BOX
_BOX_IRREGULAR
_CYLINDER
_PIPE
_REBAR
_SPHERE

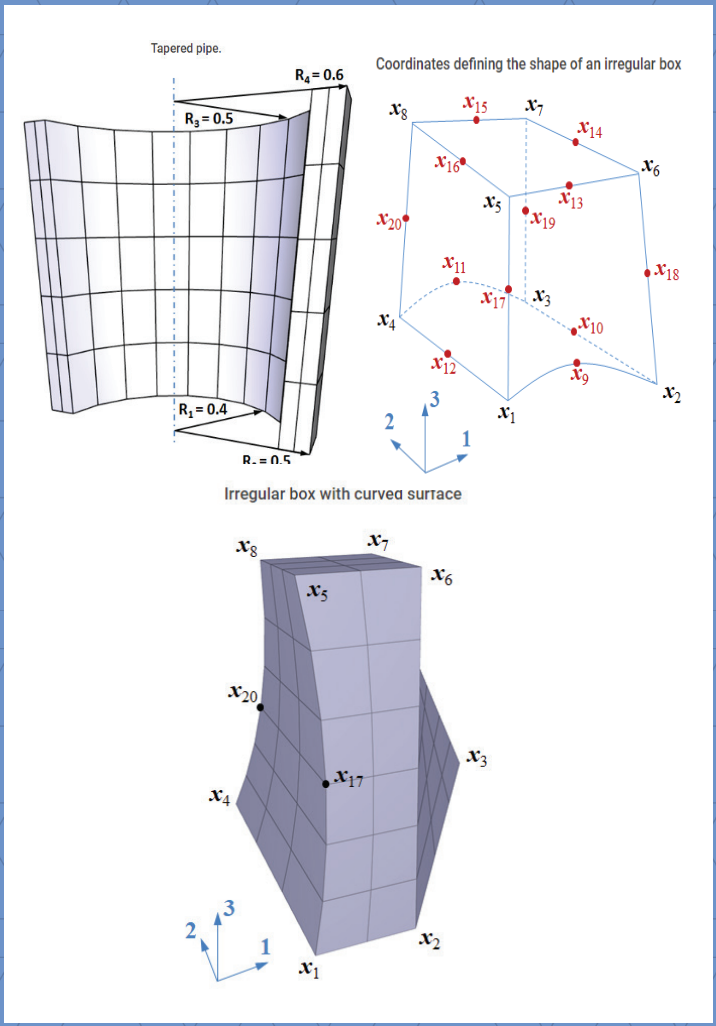
As an example consider how to build a box. A command ID (coid) is defined and a part ID as well (pid). The number of elements are controlled by N_X, N_Y and N_Z. If the box is not aligned in the global coordinate system then the csysid parameter can be used to define the component in a local coordinate system.

```
*COMPONENT_BOX
"Optional title"
coid, pid, Nx, Ny, Nz, csysid
x1, y1, z1, x2, y2, z2
```

The geometry itself is defined by two coordinate points that define opposite corners of the box. They are X_1 and X_2 in the figure.



More complicated components can also be generated, e.g., a part of a tapered pipe or the fairly detailed *COMPONENT_BOX_IRREGULAR command.



The number of coordinates needed depends on the shape of the irregular box. If the lines are straight only eight corners are needed. For curved sides, a total of 20 coordinate sets can be given as shown in the figure, leading to rather complex shapes and one should be careful with the specification not to promote a potential risk for negative volume in the element. There are no problems turning the elements into high order elements or with using mesh smoothing. The _BOLT option is also commonly used in vehicle design where bolt, washer and nut are automatically generated.

Morten Rikard Jensen, Ph.D.

Dr. Jensen graduated from Aalborg University, Denmark in 1995 with a master degree in Manufacturing Engineering and he obtained his Ph.D. and Doctor Europeus in Mechanical Engineering at the same University in 1999. In 1997 he was a research assistant at the Department of Solid Mechanics, Linköping University, Sweden.

His main area of interest is Sheet Metal Forming which involved experiments to determine tool wear and investigation of the hydro-mechanical deep drawing process. In addition to experimental work Dr. Jensen has worked extensively with numerical simulations using explicit and implicit nonlinear transient dynamic solvers. During his time at Aalborg University, Dr. Jensen taught classes in experimental sheet metal forming, manufacturing processes and how to start and plan a manufacturing company. Dr. Jensen also conducted in depth research in the area of experimental Sheet Metal Forming and numerical simulation of the same. Dr. Jensen has published over 10 papers in international journals, more than 20 papers in international conferences and 40 additional papers and reports.

From 2001 to 2012 he worked at LSTC, USA, the developer of LS-DYNA®. The last seven years as Support and Training Manager in charge of all support at LSTC. He authored the official Getting Started with LS-DYNA® book. Dr. Jensen's work at LSTC covered all main areas of the LS-DYNA® code, e.g., crash simulation, metal forming, bullet penetration, explosion, etc. Dr. Jensen has taught LS-DYNA® Introduction classes, LS-DYNA® Implicit classes and seminars on specific LS-DYNA® features such as; material models, contact algorithms and modeling drop tests of cellular phones.

Since 2012, Dr. Jensen has been the CTO at CertaSIM, LLC, the official distributor of the IMPETUS Afea Solver® in the Americas. As the CTO he is responsible for all technical issues, including product support, training and consulting. His work covers many different applications such as mine blast, ballistics, composites, car crash, metal forming, water ditching, sports applications, biomedical simulations, etc. He teaches the Getting Started class, Introduction class and many different application classes including, Mine Blast, Modeling Golf Equipment, ATD and Sheet Metal Forming, etc.

Improving productivity is a key factor in any industry to reducing cost and that certainly applies to simulation technology. We are pleased to present this discussion by Dr. Jensen regarding "Productivity when using Finite Element Modeling in a production environment."

In thinking back when I was a young man working in my Father's factory which was a production environment I can recall how different workers approached their jobs. I noticed that some workers were very efficient and others were not. Some workers took longer breaks, talked more and in general produced less. In the summers when studying to become an engineer, I worked in a metal forming factory to support my studies. Here again, productivity was an issue and the quality of the end product was integral to the defining productivity. Is working faster or smarter or a combination of the two the answer to better productivity? Productivity is defined in [1] as: "The optimum use of all resources – material, energy, capital, labor and technology – or as output per employee per hour, productivity basically measures operating efficiency". How realistic is it to measure that? It must be related to time….During classes studying manufacturing engineering in the 90's, I learned how to build factories, including assembly lines which included floor layout, positioning of tools and components in the assembly stations. Together with these a Time Sequence Study of the assembly flow had to be carried out. It was a very interesting study where one would for example add up the time it took to "Move Left Arm 45°" and "Turn Eyes to the Left". While doing this analysis I often wondered about how to factor in the extra restroom visit, the quick talk between the assembly workers or at the night shift when it was typical to take a 20 minute shower before clocking out? The Professor never discussed this, probably too practical to teach.

It's one thing to measure productivity during the manufacturing stage which is relatively straightforward to quantify, but what about productivity in the engineering environment? What about productivity amongst CAE engineers – what is it, how do we measure it and what influences it? Is it also related to time and output? Unlike a manufacturing process, developing a numerical model cannot be measured by how many elements one creates per hour. This is not an appropriate measure of productivity. Computer technology which has seen a drastic increase in processor speeds and modern GUI interfaces have changed the way models are built. In the last 30 years CAE has been integrated into all aspects of simulation technology starting with engineering designers using CAD software tools to develop the physical design of a product for production and to also be used by the engineering analyst to develop the Finite Element model that is used to validate and help design the product. The step from CAD to an FE Model is a major effort because CAD files are developed by designers for the manufacturing process. The geometric details, e.g. fillets, holes, engravings on surfaces, etc., that are necessary to produce a product may not be necessary to analyze the product for a particular function. Including unnecessary details can complicate the model and lead to longer runtimes. One cannot just push a button and convert the CAD model to an FE mesh. Simulation is as much an art as a science. Knowing when a simplification is necessary and or appropriate to obtain an accurate solution takes experience.

In the early days of the Finite Element Method interactive interfaces did not exist to build, run and post-process the data [2]. It was very tedious and the input files were on computer "cards", typed on a keyboard one at a time. It could take a considerable amount of time just to create a model of a simple structure not to mention the limitation on memory and processing speed. Today, it is a given that everything is interactive with GUI interfaces consisting of drag and drop, pop down menus, etc. that make it far easier to create and modify a model. But that does not eliminate the most fundamental part of the FE Analysis, what and how to model the physical structure. What assumptions to make: symmetry, plane stress, plane strain, and what type of elements to use.

The upside of today's computing environment is that the engineering analyst is capable of including more geometric detail with the goal of providing a better analysis but also to provide more realistic looking models. But the downside is that for an FE model more detail translates into larger models with smaller elements and thus longer runtimes. More detail also makes it more difficult to debug a model because of its complexity and the longer runtimes. Larger models also require more computing resources. However, it is a typical scenario to start with a complex model because a production environment is focused on getting the job done as quickly as possible. Any experienced analyst will tell you that nothing works the first time and so the process of debugging the model starts. As an example consider an offshore oil platform which may be subjected to impact loads all across the deck structure. Underneath the deck are a series of pipes, cables, underlying structures, etc. To assure that the design can withstand drop loads at many locations the model resolution has to have enough detail so that all locations of interest can capture the localized damage resulting from the impact.

With 25+ years of experience with simulation technology which includes Legacy solvers and the last 6 years with the IMPETUS Afea Solver® I have seen a definite improvement in productivity from our users. As a relatively new player in the simulation market our customers are very experienced with Legacy solvers so convincing them to move to IMPETUS demanded that we provide a more productive solution. Innovation almost always comes from small companies that think outside-of-the-box. This was made clear in a report released by the National Science Foundation in May 2006, "Simulation – Based Engineering Science" SBES [3]. The blue ribbon panel included giants in the field of Finite Element Analysis that included Professors, Ted Belytshko, Thomas J.R. Hughes and J. Tinsely Oden, just to name a few. The report makes many good points with two statements that are pertinent to the discussion of CAE productivity. The first one talks about the past success of Legacy software and the second one about what is needed in the future:

"At the heart of these successes, however, are simulation methodologies that are decades old, too old to meet the challenges of new technology. In many ways, the past successes of computer simulation may be its worst enemies, because the knowledge base, methods, and practices that enabled its achievements now threaten to stifle its prospects for the future."

"Tomorrow's SBES software requires extraordinary degrees of robustness, efficiency, and flexibility. The new software must not only execute simulation algorithms, but must also dynamically manage data throughput and model adaptivity and control."

It is interesting to note that the report was released in 2006 and the development of the IMPETUS Afea Solver® began in 2007 with the introduction of the first NVIDIA GPU designed for high performance computing in 2009.
The development of the IMPETUS Solver resonates with the statement from the SBES report. As an IMPETUS user you rely on our next generation element technology, the ASET™ family of elements of quadratic and cubic Hex, Tet and Pent elements and the ability to build a relatively coarse model that can be refined at runtime to add the accuracy when you need it where you need it while at the same time minimizing run times. For user's that also rely on the IMPETUS Discrete Particle and γSPH solvers, the same flexibility applies, define a geometry to fill with particles and let the solver do it at runtime which allows the analyst to adjust the particle resolution by simply changing the density of the particles. Add GPU Technology for massively parallel processing on a standard workstation and a modern GUI Interface to connect all the pieces and the result is increased productivity.

References:

[1] S. Kalpakjian, "Manufacturing Engineering and Technology", Addison-Wesley Publishing Company, 1992, Second Edition, ISBN 0-201-63093-1.

[2] W. S. Woodward and J. W. Morris, "Improving Productivity in Finite Element Analysis Through Interactive Processing", Finite Elements in Analysis and Design 1 (1985) 35-48.

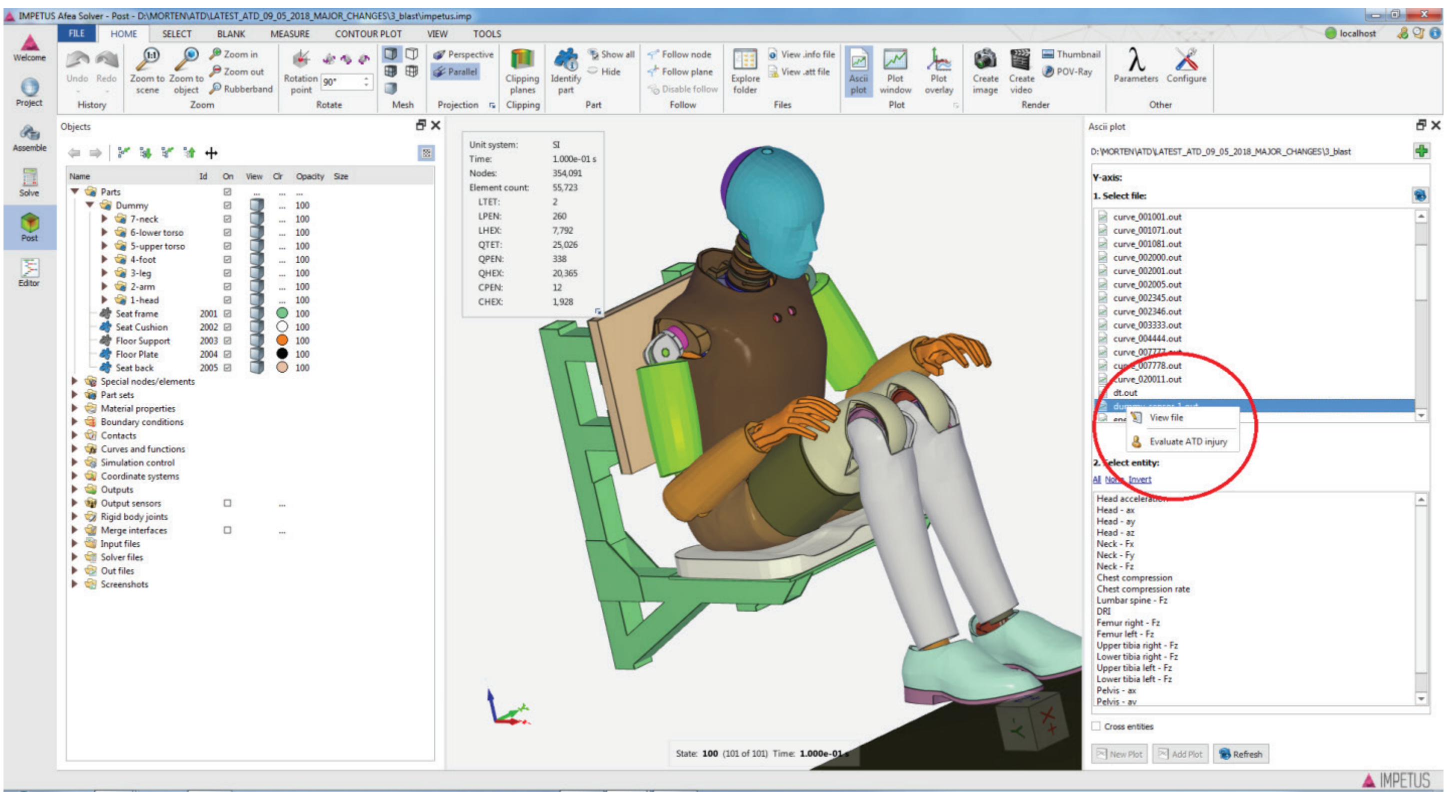
[3] NSF, "Simulation – Based Engineering Science", Report of the National Science Foundation Blue Ribbon Panel on Simulation-Based Engineering Science, May 2006.

New Features in the IMPETUS Afea Solver GUI

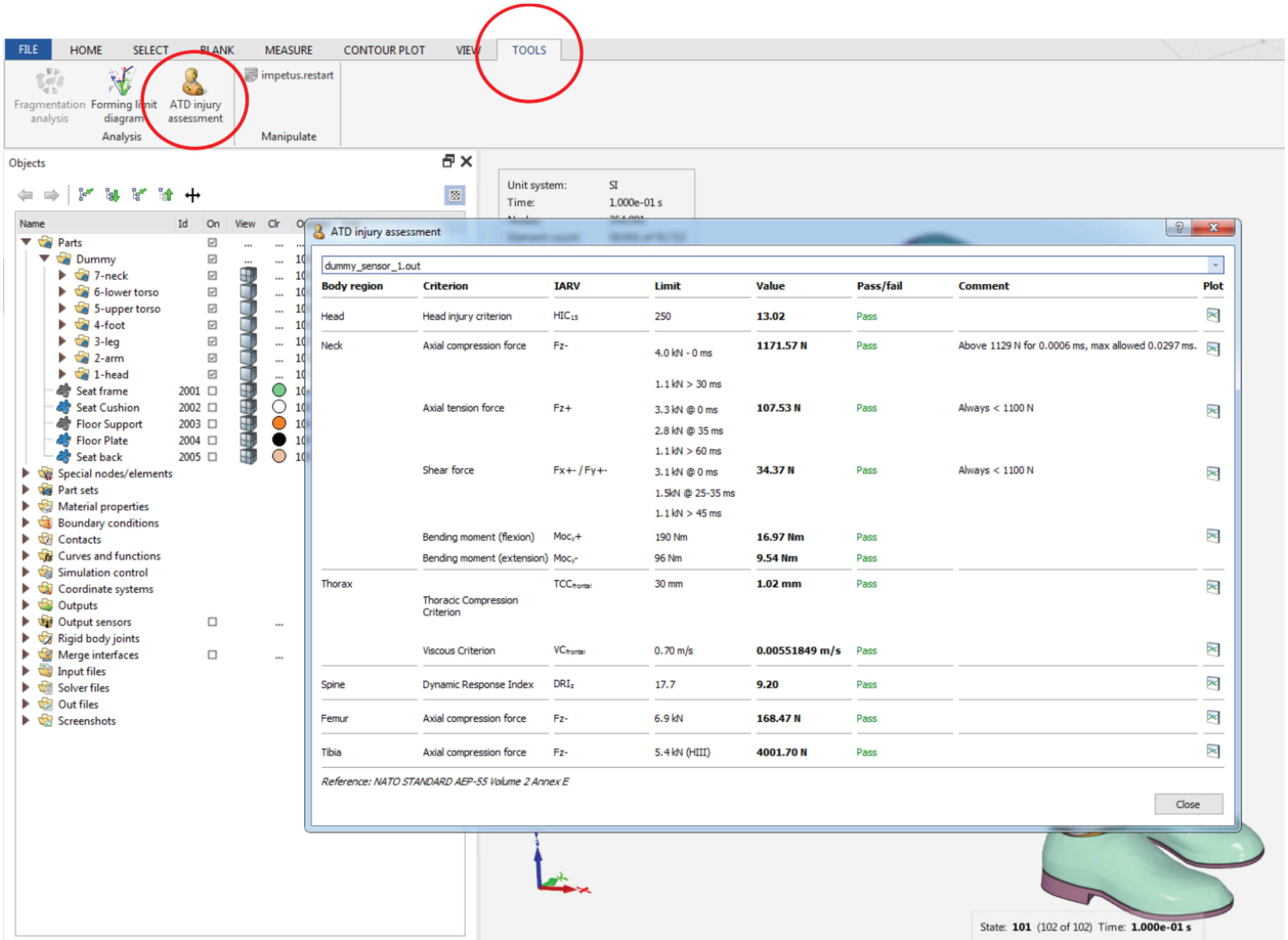
Injury Assessment in Mine Blast Simulations of Military Vehicles – Made Easy!

The IMPETUS Engine and GUI teams have made a new implementation of a streamlined interface for easy checking of the NATO AEP-55 Volume 2 Annex E standard requirements.

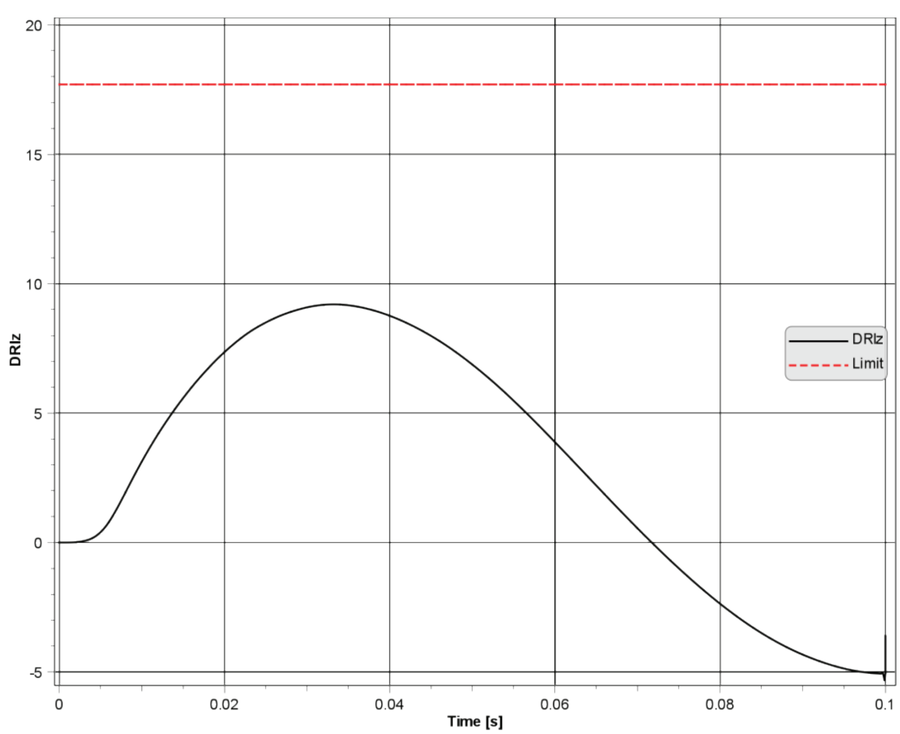
In the Post Mode one can simply right click on the dummy sensor file and then click on "Evaluate ATD Injury" which will bring up a table summarizing what has passed and what has not.



This can also be found under Tool using the ATD Injury Assessment icon.



In the table one can use the Plot icon to plot the curve which includes parameter limits, as shown below.



This new feature saves a tremendous amount of time in the Post-Processing phase since the alternative would be to find each of these components and for some of them even manipulate them to get the value for the criteria. Here one just needs to click on a plot icon or grab the numbers from the table.