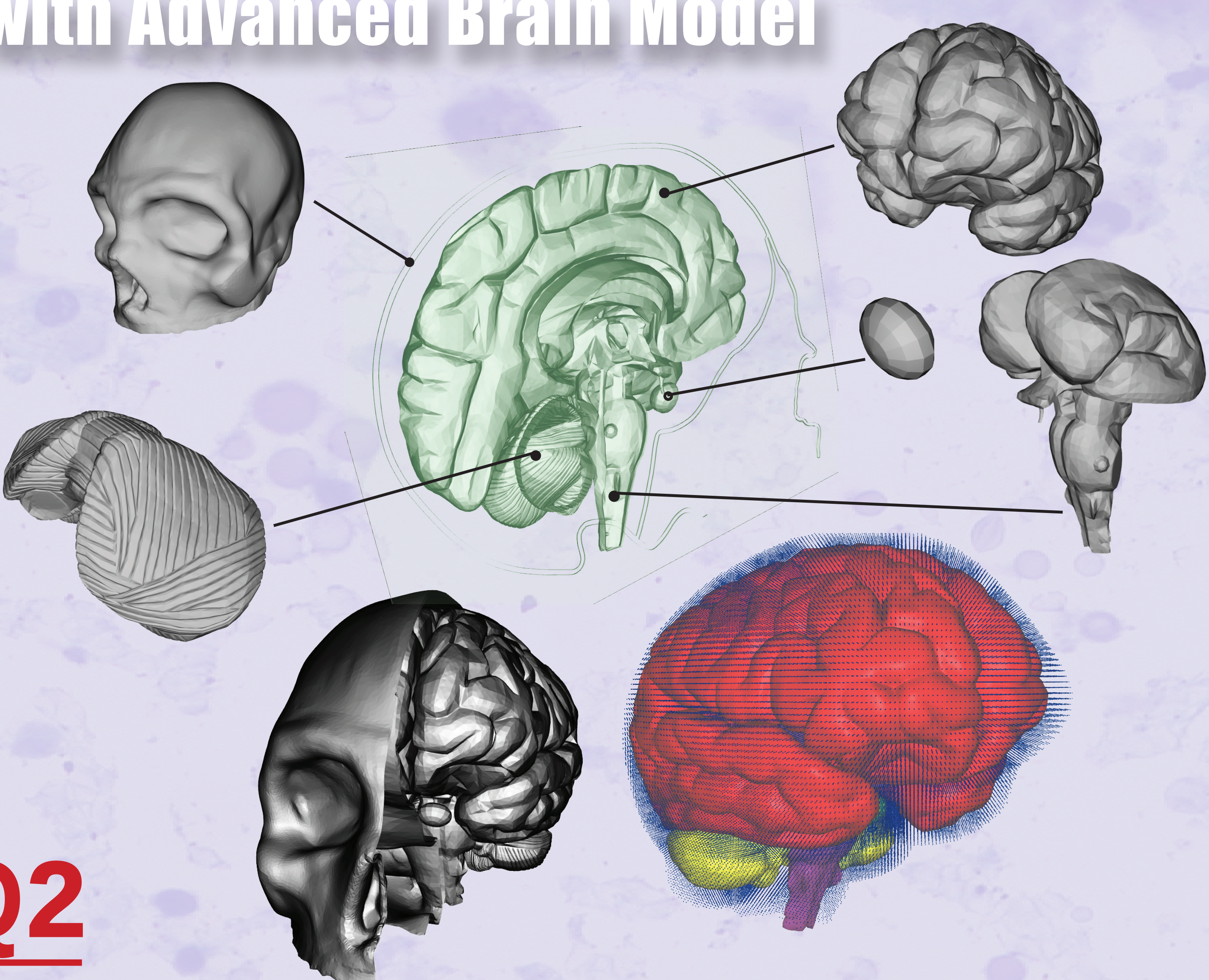
the CertaSim SOLUTION™

# Featuring:
# Simulation of Whiplash with Advanced Brain Model



Q2
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. 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

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

## 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.*

### New Research on Brain Modeling Applies the IMPETUS Afea Solver®

Dr. Milan Toma from New York Institute of Technology recently attended the 5th International Conference on Computational and Mathematical Biomedical Engineering (CMBE2017). He presented results from a recent study of car crash related whiplash. The work was co-authored with Dr. Paul Nguyen. The FSI simulation took advantage of advanced finite element technology at the heart of the IMPETUS solver and the direct coupling of the IMPETUS γSPH solver to accurately model the physics.

5th International Conference on Computational and Mathematical Biomedical Engineering - CMBE2017
10–12 April 2017, United States
P. Nithiarasu, A.M. Robertson (Eds.)

**FLUID-STRUCTURE INTERACTION ANALYSIS OF CEREBRAL SPINAL FLUID WITH A COMPREHENSIVE HEAD MODEL SUBJECT TO A CAR CRASH-RELATED WHIPLASH**

**Milan Toma[1] and Paul Nguyen[2]**

The work is described later in this issue of the CertaSIM Solution Journal.

### Experimental and Numerical Investigation of Soil Impact under High Velocity

Researchers at University of Virginia, Cambridge University and Newtec Services Group, Inc., have applied the IMPETUS Afea Solver® to model impact of sand on a 304 Stainless steel plate. A high explosive charge accelerated the sand front to impact with speeds that exceeded 1200 m/s. The work is published in the International Journal of Impact Engineering.

ELSEVIER

International Journal of Impact Engineering

Available online 17 February 2017

In Press, Corrected Proof — Note to users

High intensity impulsive loading by explosively accelerated granular matter

Anne Kyner[a], Kumar Dharmasena[a], Keith Williams[b], Vikram Deshpande[c], Haydn Wadley[a],

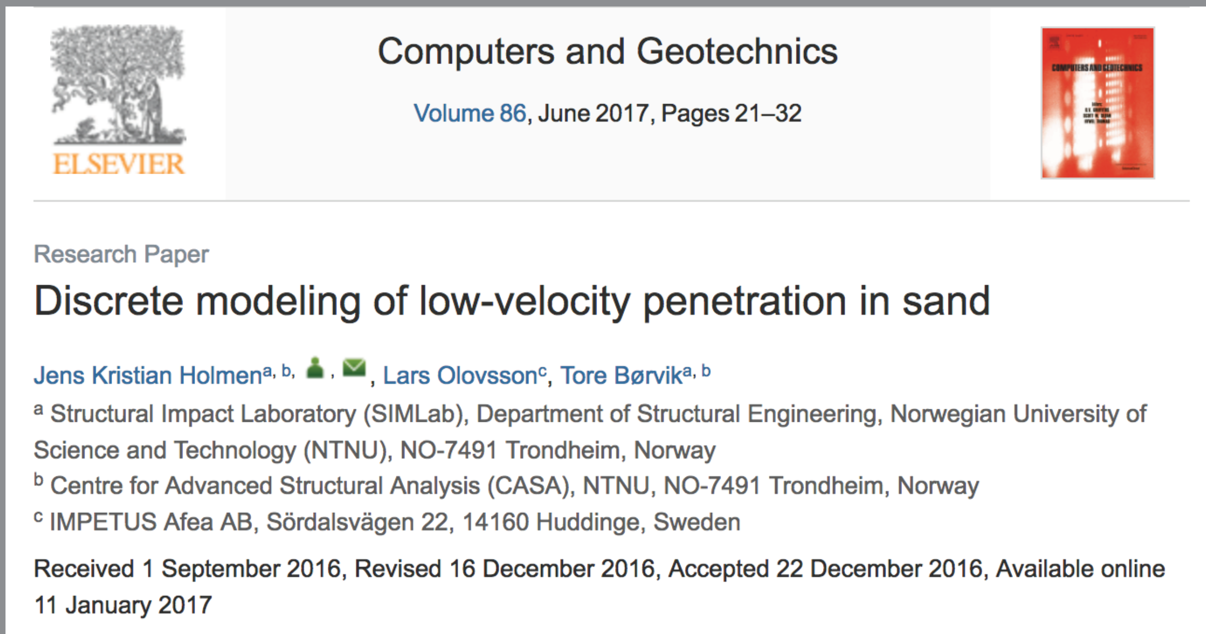
Show more

https://doi.org/10.1016/j.ijimpeng.2017.02.009

Get rights and content

## Discrete Modeling of Low-Velocity Penetration in Sand

Researchers in Sweden and Norway have investigated impact of sand under low velocity, in this case below 5 m/s. Impactors with different shapes have been used and experimental data included the resisting force from the sand. The IMPETUS Discrete Particle Method (iDPM) was applied to model the sand and it was concluded that: "...the method can also be used to describe the overall response of sand subjected to low-velocity penetration".



ELSEVIER

Computers and Geotechnics

Volume 86, June 2017, Pages 21–32

Research Paper

Discrete modeling of low-velocity penetration in sand

Jens Kristian Holmen[a, b], Lars Olovsson[c], Tore Børvik[a, b]

[a] Structural Impact Laboratory (SIMLab), Department of Structural Engineering, Norwegian University of Science and Technology (NTNU), NO-7491 Trondheim, Norway

[b] Centre for Advanced Structural Analysis (CASA), NTNU, NO-7491 Trondheim, Norway

[c] IMPETUS Afea AB, Sördalsvägen 22, 14160 Huddinge, Sweden

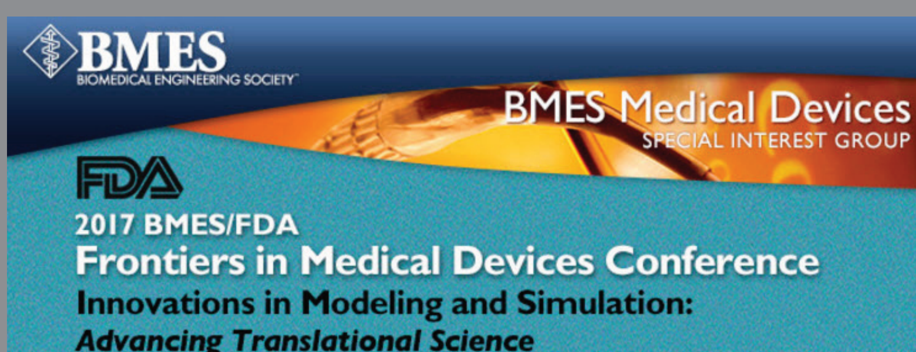
Received 1 September 2016, Revised 16 December 2016, Accepted 22 December 2016, Available online 11 January 2017

## GPU Technology Conference (GTC) 2017

The IMPETUS Afea Solver® takes full advantage of GPU Technology for massively parallel processing. Every year NVIDIA holds their GTC Conference where the latest GPU Technology is showcased to over four thousand attendees. To keep up with this technology CertaSIM attends the conference and 2017 was no exception. The conference was held in San Jose, California, May 8-11, 2017. Dr. Wayne Mindle of CertaSIM made a presentation entitled "CAE Productivity and GPU Technology." The paper focuses on the benefits of GPU Technology to improve productivity.

## 3rd Annual BMES/FDA Frontiers in Medical Devices Conference

The "Frontiers in Medical Devices Conference" is a medical device conference sponsored by the Biomedical Engineering Society (BMES) and the Federal Drug Administration (FDA). The conference is held in Washington DC, May 16-18. The IMPETUS Afea Solver® has been successfully applied to various areas in the Biomedical Industry as well as in the Medical Device Industry. CertaSIM, LLC will have a booth at the conference, together with csimsoft, the developer of Trelis and Bolt (http://www.csimsoft.com).

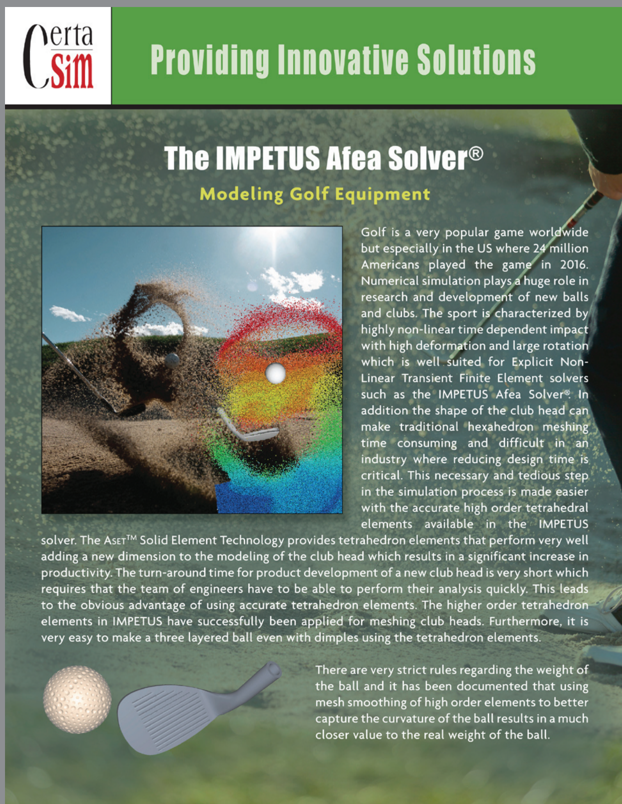



csimsoft is a CertaSIM partner and their suite of pre-processors are world reknowned for building quality solid element meshes. csimsoft and CertaSIM are currently working together on a project for generating optimal meshing for medical devices and biomedical parts to be modeled with the IMPETUS Afea Solver®.

*Please come and visit us at our booth to see our results. Read more about the conference:* http://www.bmes.org/medicaldevices

## New Brochures

If you have not already received our latest brochures, please contact us! They highlight four rather different applications; Modeling of golf equipment, the benefit of using IMPETUS for modeling the Sheet Metal Forming process, Fluid-Structure Interaction applied to modeling the function of Mitral Valves and simulations of Cerebrospinal Fluid and the human brain.

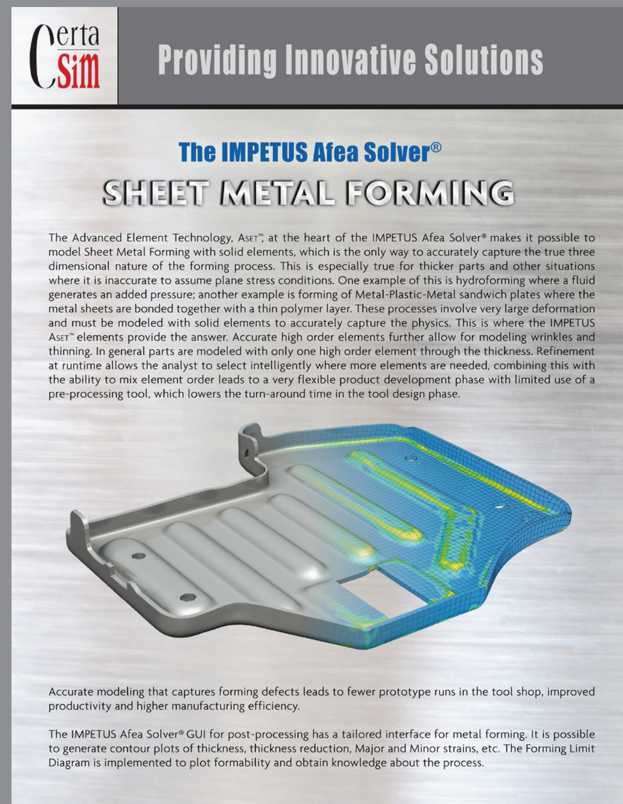

CertaSim Providing Innovative Solutions

### The IMPETUS Afea Solver®

**Modeling Golf Equipment**

Golf is a very popular game worldwide but especially in the US where 24 million Americans played the game in 2016. Numerical simulation plays a huge role in research and development of new balls and clubs. The sport is characterized by highly non-linear time dependent impact with high deformation and large rotation which is well suited for Explicit Non-Linear Transient Finite Element solvers such as the IMPETUS Afea Solver®. In addition the shape of the club head can make traditional hexahedron meshing time consuming and difficult in an industry where reducing design time is critical. This necessary and tedious step in the simulation process is made easier with the accurate high order tetrahedral elements available in the IMPETUS solver. The ASET™ Solid Element Technology provides tetrahedron elements that perform very well adding a new dimension to the modeling of the club head which results in a significant increase in productivity. The turn-around time for product development of a new club head is very short which requires that the team of engineers have to be able to perform their analysis quickly. This leads to the obvious advantage of using accurate tetrahedron elements. The higher order tetrahedron elements in IMPETUS have successfully been applied for meshing club heads. Furthermore, it is very easy to make a three layered ball even with dimples using the tetrahedron elements.

There are very strict rules regarding the weight of the ball and it has been documented that using mesh smoothing of high order elements to better capture the curvature of the ball results in a much closer value to the real weight of the ball.


CertaSim Providing Innovative Solutions

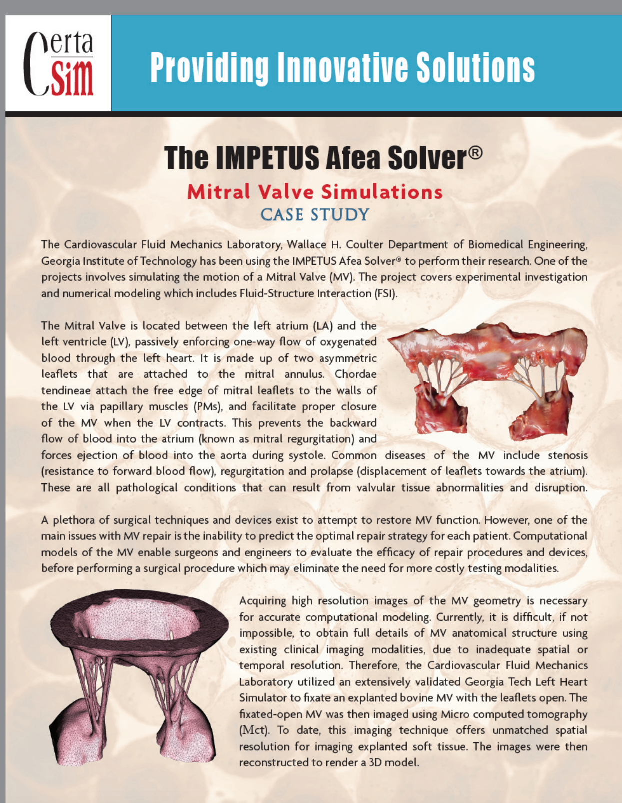
### The IMPETUS Afea Solver®

**SHEET METAL FORMING**

The Advanced Element Technology, ASET™, at the heart of the IMPETUS Afea Solver® makes it possible to model Sheet Metal Forming with solid elements, which is the only way to accurately capture the true three dimensional nature of the forming process. This is especially true for thicker parts and other situations where it is inaccurate to assume plane stress conditions. One example of this is hydroforming where a fluid generates an added pressure; another example is forming of Metal-Plastic-Metal sandwich plates where the metal sheets are bonded together with a thin polymer layer. These processes involve very large deformation and must be modeled with solid elements to accurately capture the physics. This is where the IMPETUS ASET™ elements provide the answer. Accurate high order elements further allow for modeling wrinkles and thinning. In general parts are modeled with only one high order element through the thickness. Refinement at runtime allows the analyst to select intelligently where more elements are needed, combining this with the ability to mix element order leads to a very flexible product development phase with limited use of a pre-processing tool, which lowers the turn-around time in the tool design phase.

Accurate modeling that captures forming defects leads to fewer prototype runs in the tool shop, improved productivity and higher manufacturing efficiency.

The IMPETUS Afea Solver® GUI for post-processing has a tailored interface for metal forming. It is possible to generate contour plots of thickness, thickness reduction, Major and Minor strains, etc. The Forming Limit Diagram is implemented to plot formability and obtain knowledge about the process.


CertaSim Providing Innovative Solutions

### The IMPETUS Afea Solver®

**Mitral Valve Simulations**

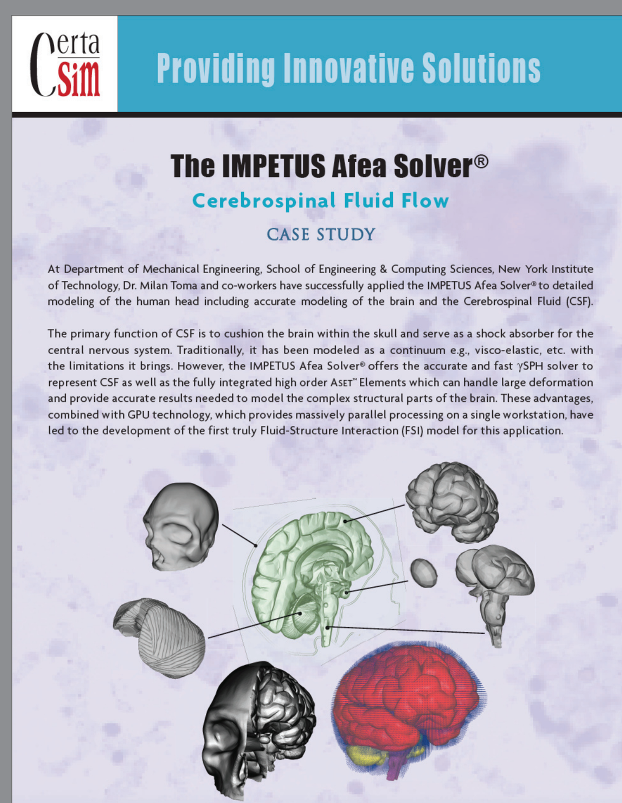
CASE STUDY

The Cardiovascular Fluid Mechanics Laboratory, Wallace H. Coulter Department of Biomedical Engineering, Georgia Institute of Technology has been using the IMPETUS Afea Solver® to perform their research. One of the projects involves simulating the motion of a Mitral Valve (MV). The project covers experimental investigation and numerical modeling which includes Fluid-Structure Interaction (FSI).

The Mitral Valve is located between the left atrium (LA) and the left ventricle (LV), passively enforcing one-way flow of oxygenated blood through the left heart. It is made up of two asymmetric leaflets that are attached to the mitral annulus. Chordae tendineae attach the free edge of mitral leaflets to the walls of the LV via papillary muscles (PMs), and facilitate proper closure of the MV when the LV contracts. This prevents the backward flow of blood into the atrium (known as mitral regurgitation) and forces ejection of blood into the aorta during systole. Common diseases of the MV include stenosis (resistance to forward blood flow), regurgitation and prolapse (displacement of leaflets towards the atrium). These are all pathological conditions that can result from valvular tissue abnormalities and disruption.

A plethora of surgical techniques and devices exist to attempt to restore MV function. However, one of the main issues with MV repair is the inability to predict the optimal repair strategy for each patient. Computational models of the MV enable surgeons and engineers to evaluate the efficacy of repair procedures and devices, before performing a surgical procedure which may eliminate the need for more costly testing modalities.

Acquiring high resolution images of the MV geometry is necessary for accurate computational modeling. Currently, it is difficult, if not impossible, to obtain full details of MV anatomical structure using existing clinical imaging modalities, due to inadequate spatial or temporal resolution. Therefore, the Cardiovascular Fluid Mechanics Laboratory utilized an extensively validated Georgia Tech Left Heart Simulator to fixate an explanted bovine MV with the leaflets open. The fixated-open MV was then imaged using Micro computed tomography (Mct). To date, this imaging technique offers unmatched spatial resolution for imaging explanted soft tissue. The images were then reconstructed to render a 3D model.


CertaSim Providing Innovative Solutions

### The IMPETUS Afea Solver®

**Cerebrospinal Fluid Flow**

CASE STUDY

At Department of Mechanical Engineering, School of Engineering & Computing Sciences, New York Institute of Technology, Dr. Milan Toma and co-workers have successfully applied the IMPETUS Afea Solver® to detailed modeling of the human head including accurate modeling of the brain and the Cerebrospinal Fluid (CSF).

The primary function of CSF is to cushion the brain within the skull and serve as a shock absorber for the central nervous system. Traditionally, it has been modeled as a continuum e.g., visco-elastic, etc. with the limitations it brings. However, the IMPETUS Afea Solver® offers the accurate and fast $\gamma$SPH solver to represent CSF as well as the fully integrated high order ASET™ Elements which can handle large deformation and provide accurate results needed to model the complex structural parts of the brain. These advantages, combined with GPU technology, which provides massively parallel processing on a single workstation, have led to the development of the first truly Fluid-Structure Interaction (FSI) model for this application.

# Simulation Helps Understand Mitral Valve Function

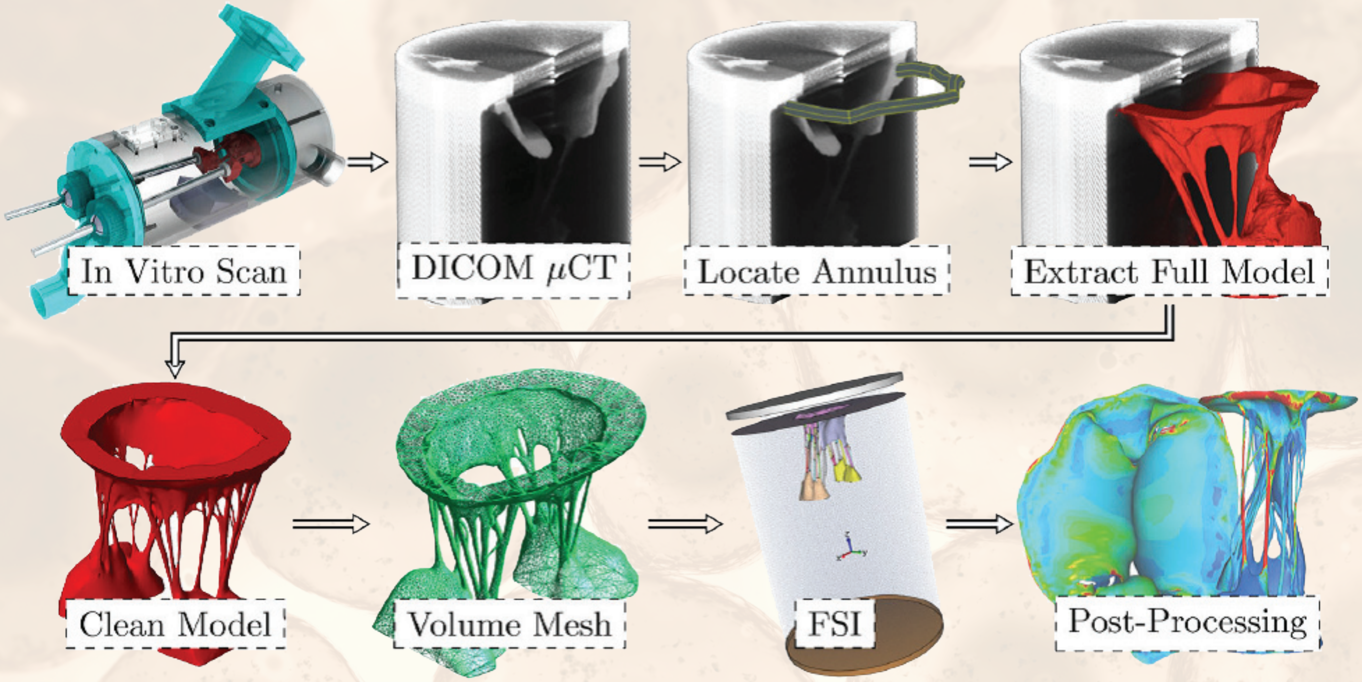
*If surgeons and engineers work together and make it possible to use patient specific numerical models, the optimal solution can be applied to increase the quality of the treatment. As an example, consider the Mitral Valve, which is of vital importance to the function of your heart. The structure and behavior of the Mitral Valve has been successfully modeled with the IMPETUS Afea Solver®. Georgia Institute of Technology and Emory University have developed procedures for both experimental and numerical procedures to analyze the function of the valve.*

The Cardiovascular Fluid Mechanics Laboratory, Wallace H. Coulter Department of Biomedical Engineering, Georgia Institute of Technology has been using the IMPETUS Afea Solver® since 2013 as their explicit Finite Element Solver. One of the projects where IMPETUS has been successfully applied is in the simulation of the Mitral Valve (MV). The project covers both experimental as well as numerical work as seen in the following where the various steps are explained. It shows the coupling of the various experimental work with results from the FSI simulations performed with the IMPETUS Afea Solver®.

The Mitral Valve is located between the left atrium (LA) and left ventricle (LV), passively enforcing one-way flow of oxygenated blood through the left heart. It is made up of two asymmetric leaflets that are attached to the mitral annulus. Chordae tendineae attach the free edge of mitral leaflets to the walls of the LV via papillary muscles (PMs), and facilitate proper closure of the MV when the LV contracts. This prevents the backward flow of blood into the atrium (known as mitral regurgitation) and forces ejection of blood into the aorta during systole. Common diseases of the MV include stenosis (resistance to forward blood flow), regurgitation and prolapse (displacement of leaflets towards the atrium). These are all pathological conditions that can result from valvular tissue abnormalities and disruption.

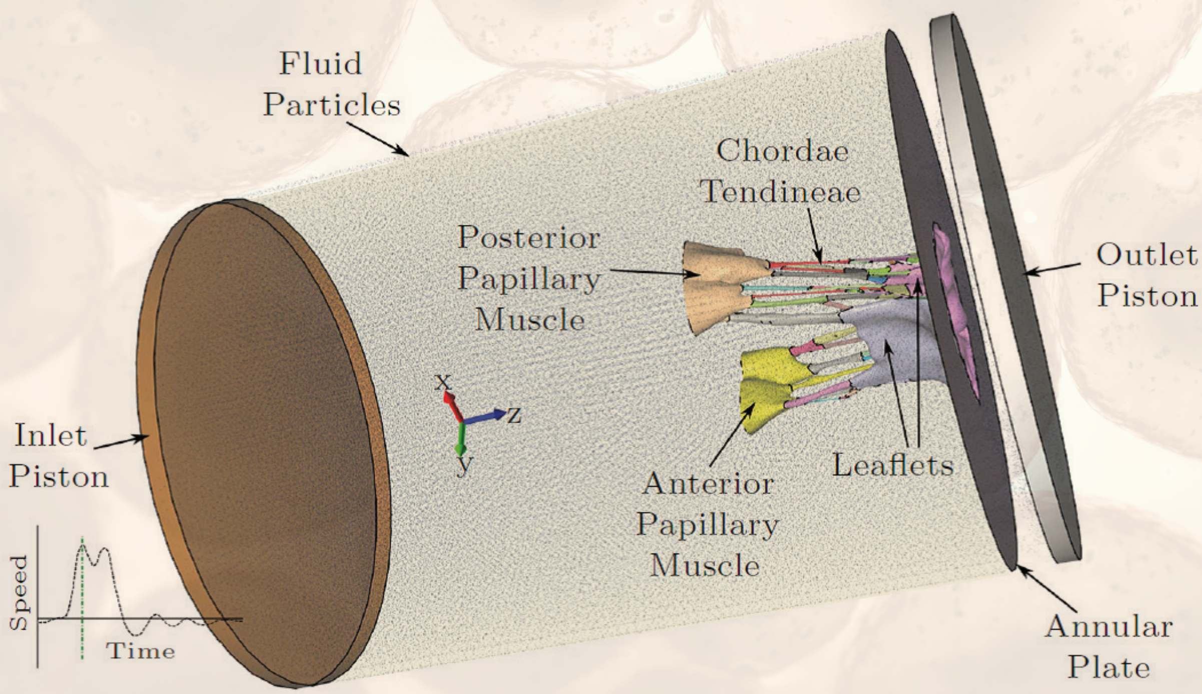
A plethora of surgical techniques and devices exist to attempt to restore MV function. However, one of the main issues with MV repair is the ability to predict the optimal repair strategy for each patient. Computational models of the MV enable surgeons and engineers to evaluate the efficacy of repair procedures and devices, before performing them on patients and moving on to more costly testing modalities. The following outlines the steps used in developing fluid-structure interaction models of a subject-specific MV.

Acquiring high resolution images of the MV geometry is necessary for accurate computational modeling. Currently, it is difficult, if not impossible, to obtain full details of MV anatomical structure using existing clinical imaging modalities, due to inadequate spatial or temporal resolution. Therefore, the Cardiovascular Fluid Mechanics (CFM) Laboratory utilized an extensively validated Georgia Tech Left Heart Simulator to fixate an explanted bovine MV with the leaflets open. The fixated-open MV was then imaged using Micro computed tomography (Mct). To date, this imaging technique offers unmatched spatial resolution for imaging explanted soft tissue. The images were then reconstructed to render a 3D model.



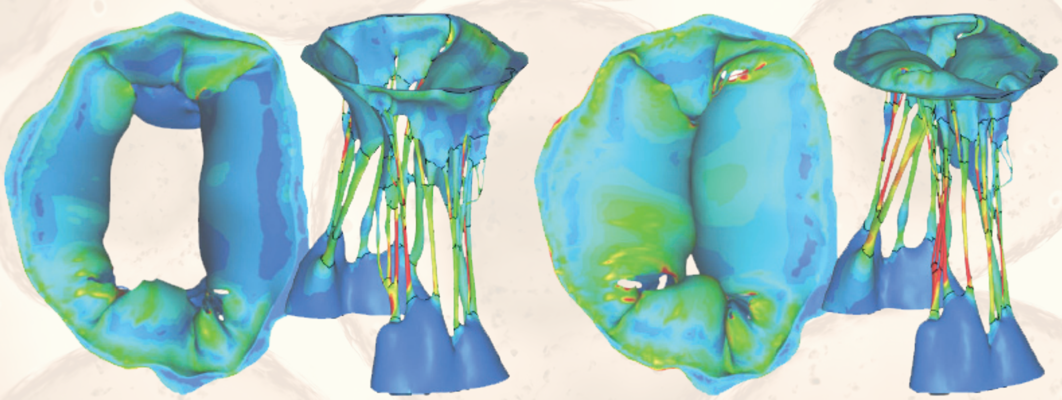

Next, a high quality robust mesh of this 3D model is created, typically using tetrahedral elements. The classic shortcomings of using tetrahedron elements in explicit Finite Element are avoided with the use of the ASET™ Elements in IMPETUS, which can accurately capture the large deformation of the MV parts. Fluid motion and boundary interaction are solved with the IMPETUS Afea γSPH Solver and the IMPETUS Lagrangian Solver, thus the fluid is represented by particles which contact the structural parts using a penalty contact. Mesh sensitivity studies are performed to ensure the final computational results are independent of the mesh size.

Both the solvers employ GPU Technology for massively parallel processing on a single workstation making it computationally fast, thus avoiding the need for a large computer cluster.



In the model set-up, the fluid particles were confined in a pipe-like rigid structure surrounding the model, and two pistons moved within the pipe at a prescribed velocity. Zero-displacement constraints in x, y and z-directions were applied at the bottom of the papillary muscles, as well as the nodes on the annular attachment.

Constructing, tuning, and validating these computational models rely upon extensive in vitro characterization of valve structure, function, and response due to diseases. μCT images of the closed MV were also obtained and reconstructed for validation of the FSI results. Further validation included comparison of the simulated forces on the chordae tendineae throughout the cardiac cycle with the experimental data [2]. It was an important observation that a fully resolved 3D model of the mitral valve requires an FSI analysis to correctly load the valve [3]. This fully validated MV model can now be used to analyze the biomechanics of various chordae related pathologies and mechanical implications of various repair techniques [4].



**References:**

[1] Toma et al., "High Resolution Subject-Specific Mitral Valve Imaging and Modeling: Experimental & Computational Methods", International Journal for Biomechanics and Modeling in Mechanobiology, 2016, 15(6):1619-1630.
[2] Toma et al., "Fluid-Structure Interaction Analysis of Papillary Muscle Forces Using a Comprehensive Mitral Valve Model with 3D Chordal Structure", Annals of Biomedical Engineering, 2016, 44(4): 942-53.
[3 ] Toma et al., "Fluid-Structure Interaction and Structural Analyses using a Comprehensive Mitral Valve Model with 3D Chordal Structure", International Journal for Numerical Methods in Biomedical Engineering, Online ahead of print, 2016.
[4] Toma et al., "Fluid-Structure Interaction Analysis of Ruptured Mitral Chordae Tendineae", Annals of Biomedical Engineering, 2017, 45(3): 619–631.

# Stent Modeling with the IMPETUS Afea Solver®

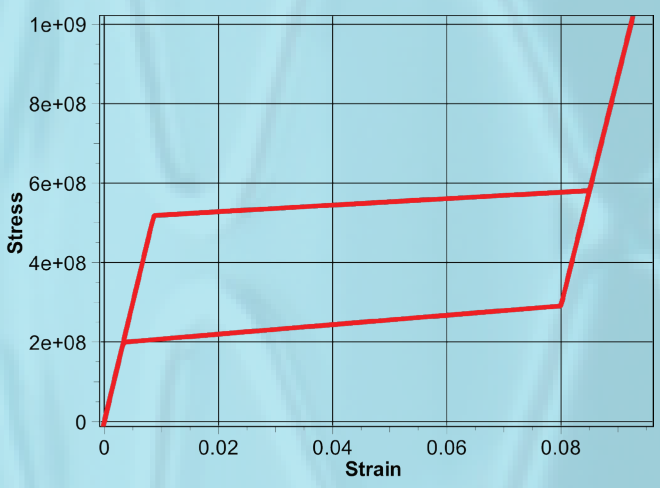
*The most common cause of death in the world is heart disease caused by plaque buildup in the walls of the arteries. One treatment of this is to insert an object at the critical position that will expand and open the artery leading to better blood flow. This object is called a stent and the design optimization and evaluation of its effectiveness is a topic that relies on Finite Element Analysis. The materials used to fabricate the stent are very sophisticated and require advanced numerical models. Recently, such a model has been implemented in the IMPETUS Afea Solver®, which will provide users a robust and accurate tool for advanced modeling in the Biomedical Device Industry.*


The first vascular stent was used the 16th of January, 1964 in Oregon, USA where Dr. C. T. Dotter used it to save the leg of an 82 year old female patient from amputation [1], [2]. The patient showed immediate improvement and was able to walk.

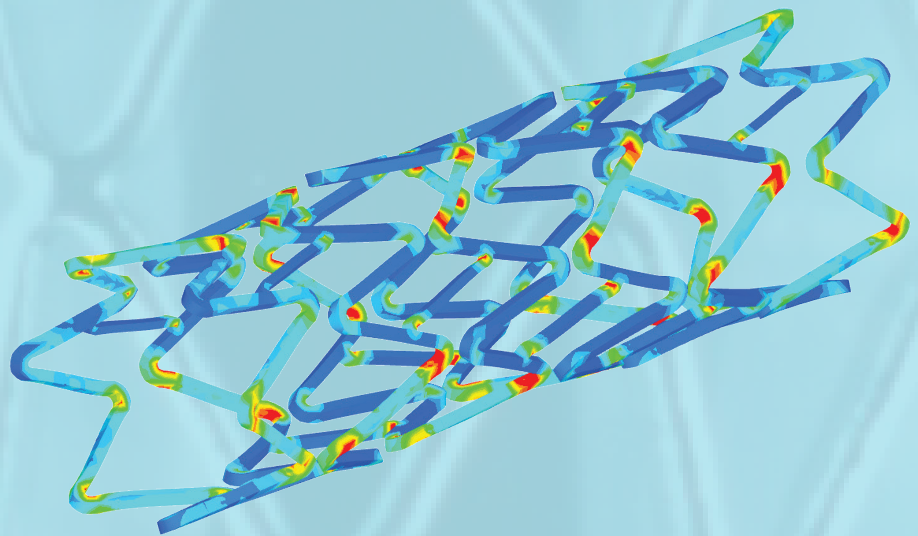
Since then extensive research has been carried out which includes Finite Element Simulations. The need for accurate and robust fully integrated 3D solid elements to capture the true 3D stress state of a material is particularly important in the medical device industry. High order elements also accurately capture the true geometry, which is critical for good contact between parts. This is echoed by the ASTM standard on this topic [3] which states: "Care must be taken to ensure that the chosen element can be adequately applied within the simulation". By using the very accurate ASET™ Elements at the heart of the IMPETUS Afea Solver® the requirements in the standard are easily met.

There are two different types of stents; balloon expandable stents where the stent is expanded by a balloon and self-expanding stents where the expansion at the treatment site is done without mechanical assistance. The first type of stent is typically fabricated with stainless steel. The latter case requires a more sophisticated material which is referred to as a Shape Memory Alloy.

One material that falls in the Shape Memory category is called Nitinol which is a complex super-elastic material which undergoes a phase transformation between austenite and martensite during the loading process. Thus a key parameter to monitor is the fraction of martensite in the structure as the stent is loaded. A Nitinol material model has been developed for the IMPETUS Solver based on the work by Auricchio, et al. [4], [5].




The model has been tested with a single element, tube bending and stent models. Validation of the model will continue based upon comparison with experimental data.



The implementation of the model was a necessary step to provide the IMPETUS community the added functionality to model these structures.

More information about the new implementation and the research can be obtained by contacting support@certasim.com.

**References:**

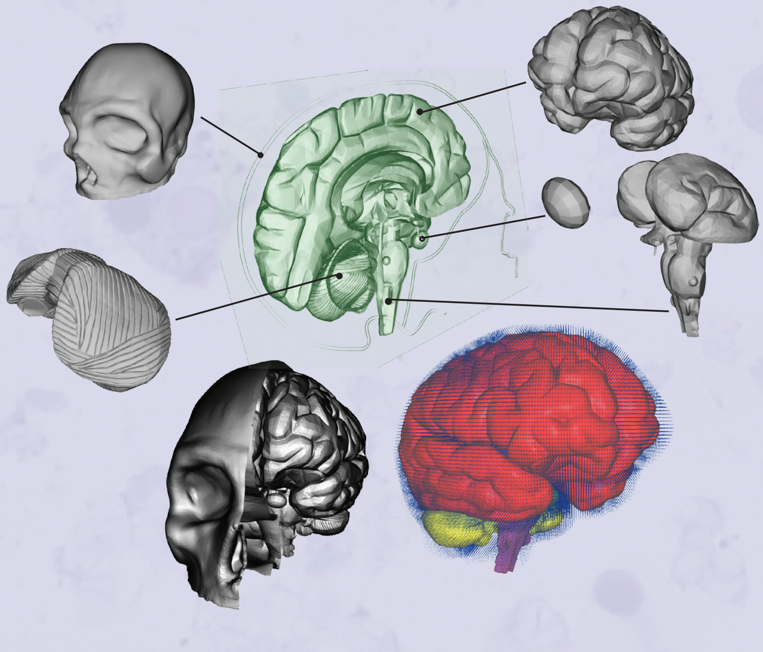
[1] C. T. Dotter and M. P. Judkins, "Transluminal Treatment of Arteriosclerotic Obstruction – Description of a New Technic and a Preliminary Report of its Application", Circulation, Volume XXX, November 1964, p. 654-670.
[2] M. M. Payne, "Charles Theodore Dotter – The Father of Intervention", Texas Heart Institute Journal, Volume 28, November 1, 2001, p. 28-38.
[3] ASTM Standard F 2514-08, "Standard Guide for Finite Element Analysis (FEA) of Metallic Vascular Stents Subjected to Uniform Radial Loading", ASTM International.
[4] F. Auricchio and R. L. Taylor, "Shape Memory Alloys: Modeling and Numerical Simulations of the Finite-Strain Superelastic Behavior", Computer Methods in Applied Mechanics and Engineering 143 (1997) 175-194.
[5] F. Auricchio et al., "Shape Memory Alloys: Macromodelling and Numerical Simulations of the Superelastic Behavior", Computer Methods in Applied Mechanics and Engineering 146 (1997) 281-312.

# Modeling Damage to the Human Brain

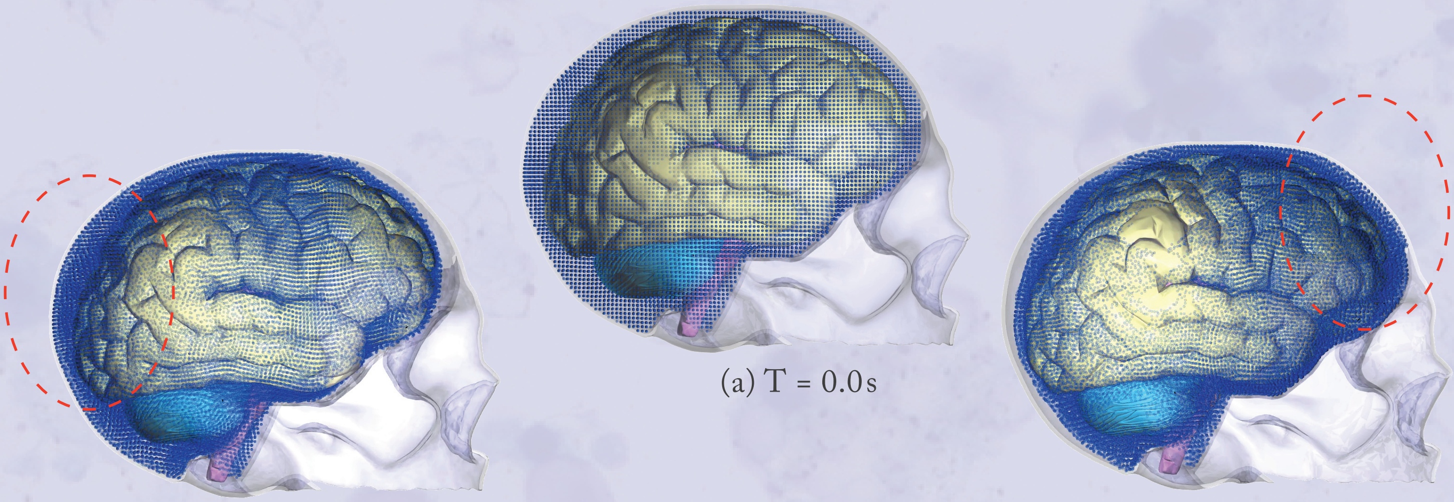
*The function of the brain is vital for human survival and obtaining knowledge about it's behavior in different interactions is extremely important. How will it react during severe impact? What are the risks and sensitive areas? The Finite Element method has been applied to help answer these questions using computer models. It is not an easy engineering task and significant research has been done. This article presents some recent work at the New York Institute of Technology where whiplash has successfully been modeled.*

*We asked Dr. Milan Toma at the Department of Mechanical Engineering, School of Engineering & Computing Sciences New York Institute of Technology, to discuss their recently published paper titled "Fluid-Structure Interaction Analysis of Cerebral Spinal Fluid with a Comprehensive Head Model Subject to a Car Crash-Related Whiplash." The benefit of the IMPETUS Afea Solver® ASET™ Element Technology, in particular accurate and robust high order tetrahedron elements, is evident for this type of structure which is geometrically complicated and extremely delicate in nature. The IMPETUS Afea γSPH Solver provided the last piece of the puzzle to model the cerebrospinal fluid (CSF). Because γSPH is a meshless continuum based method it is a perfect match for this application as it can easily follow the intricate movement of the brain structure as the contact is purely node to surface and so redirection of the flow does not require any remeshing. Finally, as Dr. Toma points out, "Ours is the first truly Fluid-Structure Interaction (FSI) model."*

The primary function of cerebrospinal fluid (CSF) is to cushion the brain within the skull and serve as a shock absorber for the central nervous system. There are published constitutive models to govern the CSF behavior, the most common models, i.e. solid-like, viscoelastic and fluid-like CSF, are all solid material models with different material properties. The solid-like CSF is linear, nearly incompressible, elastic solid with bulk modulus much larger than the shear modulus. The viscoelastic CSF constitutive model is a linear viscoelastic model with shear relaxation behavior. And, the fluid-like CSF is modeled using an elastic solid with fluid-like constitutive behavior using an equation of state constitutive model. This mean that there can be element instability like hourglassing in the single point integrated elements or simply element inversion. It also means that there is no "real flow behavior". The presented model in this work is the first truly Fluid-Structure Interaction (FSI) model.



The brain alone is of complicated structure, it consists of three main parts, namely the cerebrum, cerebellum, and brainstem. Moreover, the cerebrum consists of two cerebral hemispheres, equal halves of the brain, and is positioned over and around most other brain structures. Each cerebral hemisphere is divided into four lobes by sulci and gyri. The sulci are the grooves and the gyri are the bumps that can be seen on the surface of the brain. The CSF fills a system of cavities at the center of the brain, known as ventricles, and the subarachnoid space surrounding the brain and the spinal cord. The developed model is very comprehensive and captures most of these features. The model of the head, namely skull, cerebrum, cerebellum, brainstem and pituitary gland, is shown above. In the figure, the blue dots surrounding the brain model represent the fluid particles. The number of fluid particles filling the subarachnoid space between the skull and brain, and other cavities, is 94,690. This FSI model is then exposed to rapid acceleration and subsequently to rapid deceleration to observe the cushioning effect and dynamics of the CSF.



(a) T = 0.0s

(b) T = 0.025s

(c) T = 0.05s

In the figure above, during the rapid acceleration, the fluid particles can be seen to concentrate at the back of the head, thus preventing the brain from going backwards relative to the skull (b). Then, during the rapid deceleration, the fluid particles concentrate at the frontal lobe, again preventing the brain from moving forward and thus stopping it from hitting the skull (c). This demonstrates the dumping effect of the cerebrospinal fluid.

Currently, work is continuing at NYIT and the model is going to be used for other impact scenarios, documentation is expected to be published in the near future.

**References:**

Milan Toma and Paul Nguyen, “Fluid-Structure Interaction Analysis of Cerebral Spinal Fluid with a Comprehensive Head Model Subject to a Car Crash-Related Whiplash”, 5th International Conference on Computational and Mathematical Biomedical Engineering - CMBE2017, Pittsburgh, Pennsylvania, April, 2017.

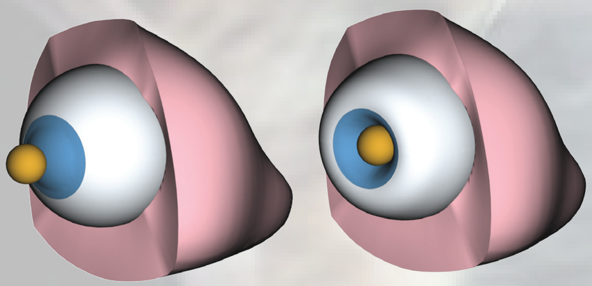
**Contacts:**

Dr. Milan Toma: mtoma@nyit.edu
CertaSIM, LLC: support@certasim.com

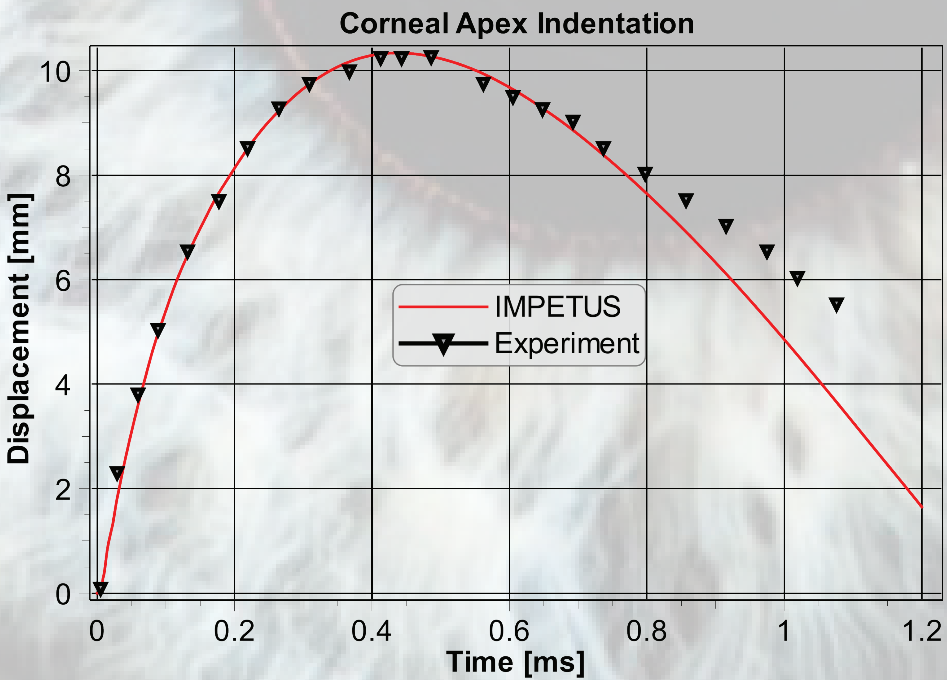
# Blunt Impact on Eye

*The eye is a complex structure and simulating the response to a blunt impact can be challenging. Blunt impact to the eye can happen in various ways and is a common occurrence. The difficulty in developing an accurate model is common to all problems, how to represent the materials in question. For a model of the eye this is particularly challenging. The IMPETUS Afea Solver® has been successfully applied to this application and compares well with experimental results.*

Many eye injuries come from blunt impacts where the damage depends on the size of the object but also on the impact velocity. To better understand the complexity and response of the eye, Finite Element simulation can be a useful tool. However, the materials that make up the eye are very soft and some of them nearly incompressible like the eye tissue which makes modeling difficult. Research in Italy and France has applied the IMPETUS Afea Solver® to blunt impact of an eye and compared with experimental results [1]. The set-up represents the impact of a 6 mm 0.2 g air soft gun bullet with an eye at a velocity of 86 m/s. In total ten experiments were carried out on fresh enucleated porcine eyes using high speed camera and a miniature transducer to gather velocity, indention and pressure.



The simulation was successful and shows it is possible to model blunt impact of an eye with the use of the IMPETUS Afea Solver®. In [1] are described some of the shortcomings and problems obtained in earlier work using MSC/Dytran such as hourglassing and the use of hourglass control that result in too stiff of a response. As indicated in the article this is not a problem with the fully integrated high order Aset™ Elements at the heart of the IMPETUS solver. The experimental corneal apex indentation and the bullet rebound speed results are compared with the IMPETUS response and were found to be in very good agreement.




The work shows that it is possible to model impact of eyes with good correlation to experimental data and further knowledge can be gained from simulation results.

**References:**

[1] C. Clemente et al., “Traumatic Eye Injuries as a Result of a Blunt Impact: Computational Issues”, 18th APS-SCCM and 24th AIRAPT, Journal of Physics: Conference Series 500 (2014) 102003.

**Professor John Leicester Williams, Biomedical Engineering, University of Memphis**

John Leicester Williams is a professor of Biomedical Engineering at the University of Memphis. He teaches Biomechanics and Biomedical Engineering design courses and collaborates on research projects with the University of Tennessee Health Sciences faculty in Orthopaedic Surgery and Physical Therapy and with scientists at Oak Ridge National Laboratory. His areas of interest include joint replacement design, dynamical modeling of the human body and of joint replacements, orthopaedic and dental implants, physical properties of bone and growth plate cartilage, and methods for stimulating cells and tissues in culture. He holds 13 US and 30 affiliated international patents related to total knee replacement designs. He is a member of the American Society of Mechanical Engineers and the Orthopaedic Research Society. He is an Academic Editor for PLOS ONE and serves on the editorial boards of the Journal of Prosthodontics and Advances in Biomechanics and Applications.

Professor Williams has spent his 35+ year career in both academia and industry and we are honored that he has taken the time to participate in our "In Review" section of the journal. We have asked him to describe his current research within the biomedical field.

"*We are using multiscale finite element modeling to understand the mechanical factors involved in bone growth. The growth of long bones such as the femur occurs within a thin (0.5-2 mm) layer of cartilage, known as the growth plate, found near each end of the bone. Growth plate cartilage consists mostly of water by mass (80%) and is extremely cellular* *with more of its volume occupied by cartilage cells than by matrix and it is subjected to strains of up to 20%. It is attached to and sandwiched between much stiffer porous bone on both sides creating two interfaces of dissimilar materials.*

*Nature has developed an interesting solution to this problem of joining two bony parts while maintaining a separation with a relatively softer material to support cell proliferation and hypertrophy, which together produce all of the growth in length. Nature does this by forming an undulating pattern of hills and valleys along the interface while at the same time allowing for the cartilage tissue joining the two bone parts to produce more bone. The cartilage cells are organized into functional and structural units called chondrons, which are tube-like structures containing individual compartments for each cell.*

*The process by which growth plate cartilage cells produce bone is one many believe requires the right mechanical cues to the cartilage cells to control cartilage cell proliferation, hypertrophy and production of matrix material used as a scaffold for bone. Our interest is in modeling the features of the interface and including cell-level structural and material property details. Due to the high fluid content and highly organized arrangement of cells of varying sizes and shapes and the variation in material properties through the plate thickness this poses computational challenges.*

*Despite these challenges there is a strong interest in developing multiscale finite element models to examine the effects of macroscale loading on the local mechanical environment of individual cells in many tissues including articular and growth plate cartilage. Understanding the environment around these cartilage cells that produce the matrix for bone formation may be useful for establishing conditions for growing bone tissue in the laboratory."*