NAFEMS americas events

The International Association for the Engineering Modeling, Analysis and Simulation Community

CONFERENCE PROGRAM & AGENDA

Confidence in Engineering Simulation:

The Next 10 Years of CAE in Mexico

May 23rd, 2019 | Tecnologico de Monterrey - Campus Sante Fe, Mexico City

nafems.org/americas

Keynote from the Ford Motor Company on "Development of Automotive Engineering in Mexico: New Trends" and **Invited Presentation** from GE Aviation on "*Finite Element Analysis Challenges in External Configuration Hardware*"

Three Tracks with presentations from industry, software providers, researchers, and academia

Lunch and Networking Reception Included

Contact Information

NAFEMS Americas
130 N Prospect Street, Suite K
Granville, OH 43023
Tel. 1.614.360.1922
Fax. 1.740.587.1505
americas@nafems.org

What is the future for engineering analysis and simulation in Mexico? Discover innovative engineering simulation processes and tools which are helping companies in Mexico improve production capabilities. Engage with domain experts, industry leaders, and peers in a focused, comprehensive one-day event that covers topics on engineering analysis, simulation, and systems modeling and simulation that every engineer in Mexico should know.

Join NAFEMS on May 23rd, 2019 in Mexico City for an event that will bring together leading visionaries, developers, and practitioners of CAE-related technologies in an open forum, unlike any other, to share experiences, discuss relevant trends, discover common themes, and explore future developments in Mexico. Topics will include:

- ***Driving the Design of Physical Systems, Components & Products***
- ***Innovation and Optimization through Engineering Simulation***
- ***Manufacturing Process Simulation***
- ***Implementing Simulation Governance & Democratization***
- ***Addressing Business Strategies & Challenges***

NAFEMS is the only worldwide independent association dedicated to engineering modeling, analysis & simulation. Currently, there are more than 1,400 member organizations worldwide ranging from major global corporations through small-scale engineering consultants. If you work with engineering simulation, you should be part of NAFEMS. Special thanks to Francisco Gomez (Cortina Design Engineering) for his important efforts in helping NAFEMS host its first-ever event in Mexico!

Sponsors

We would like to extend a special thanks to the sponsors of the 2019 NAFEMS Americas Conference on "Confidence in Engineering Simulation: The Next 10 Years of CAE in Mexico." Please be sure to visit and speak with each of our sponsors during the conference to see and hear about the latest advancements in their technologies.

ESSS

GrupoSSC
Ingeniería Impulsada por Simulación

Altair

intelligy

3D CAD Portal
1er Portal CAD CAM CAE en Español

cortina
DESIGN ENGINEERING

AGENDA - Thursday, May 23rd

Program subject to change.

Plenary Session: Auditorium

9:00

Welcome & Introduction
A. Wood, Americas Regional Manager, NAFEMS & F. Gomez, Cortina Design Engineering

Development of Automotive Engineering in Mexico: New Trends
A. Ayala, Ford Motor Company

Finite Element Analysis Challenges in External Configuration Hardware
L. Vidriales Escobar, GE Aviation

10:30 Break in Exhibit Area

	Auditorium	Classroom TBC	Classroom TBC
	TRACK 1: Mfr Process Simulation 1 Chair: TBC	**TRACK 2: Computational Fluid Dynamics** Chair: TBC	**TRACK 3: Optimization** Chair: TBC
11:00	**Structural, Injection Moulding and Forming Simulations During a Plastic-Metallic Infuser Development** J. Anaya, Mabe S.A. de C.V.	**Conjugated Heat Transfer Analysis of and Electric Transformer to Determine the Temperature Behaviour of Leads** J. Toledo Gonzalez, COMPLX	**Automated Pre-processing Method for BIW Mesh Creation Including Spot-Welds Using Open Source Programming Language** A. Garcia, Ford Motor Company
	Using Explicit Finite Element Code to Simulate Riveting Process Z. Yang, Valeo-Kapec NA	**Using CFD to Minimize Emissions and Combustion Instability of a GDI Engine at the Catalyst Heating Operating Point** J. Flores Mora, Robert Bosch Mexico Sistemas...	**Intrusion Car Body Optimization Combining Frontal and Side Crash Responses** F. Leonov S. López, LURI Engineering
	Analysis and Simulation of the Forging Process of an AISI 4340 Cast Ingot to Reduce Internal Defects and Energy... R. Ramirez, Frisa Forjados S.A. de C.V.	**Crude Oil Properties in CFD, Methodology and Case Studies** H. Hinojosa, Grupo SSC S.A. de C.V.	**CAE Simulation for Cost Reduction Strategies** M. E. Turanzas Forseck, Ford Motor Company
	Stamping Feasibility CAE Simulation at Early Design Stages E. Camargo, Ford Motor Company	**Beyond CFD - Powering Sustainable Innovation on an Integrated Platform** F. Dri, Dassault Systemes SIMULIA Corp.	**Leveraging Simulation to Optimize Design for 3D Printing** S. Sithambaram, SOLIDWORKS Corp.

12:40 Lunch in Cafeteria

	TRACK 1: Mfr Process Simulation 2 Chair: TBC	**TRACK 2: Machine Learning** Chair: TBC	**TRACK 3: Dynamics & Vibration** Chair: TBC
1:40	**The Relationship between Ribs Layout on a Body Exterior 'Belly Pan" Part, and its Prediction of Warping** V. Ernesto Hernández Álvarez, Ford Motor	**CAE Applied at the Right Place and at the Right Time** K. Flores Aviña, Ford Motor Company	**Seam Weld Optimization in Automotive Systems for Durability Analysis** C. Florez, Ford Motor Company
	Modeling Structural Behaviour of Metallic Safety Components R. Pérez Santiago, Joyson Safety Systems	**How the Simulation-Driven CAE Process May Be Profoundly Changed by DI & ML** V. Cook, Altair Mexico	**Design and Analysis of a Multiphase DC Motor, through Numeric Simulation** M. Ibañez, Grupo SSC S.A. de C.V.
	Influence of Second-Shot Process Conditions on the Warpage Behavior of a Two-Shot Overmolded Automotive Pillar N. Santoni, Ford Motor Company	**Statistic Modelling Approach for Front Low Speed Impact** O. Saveedra, Ford Motor Company	**Weld Points vs. Body Performance Study** C. Quiroz Garfias, Ford Motor Company
	FEA Based Design of Roll Forming Processes D. Melo, COMPLX	**Virtual Design Optimization of a Valve Train Actuator Using Computer Based Optimization Algorithms** R. Buendia, Delphi Diesel Systems Service...	**Modal Analysis and Fatigue for Bus Structure** A. Tristán-López, Instituto de Evaluación e Ingeniería Avanzada

3:20 Break in Exhibition Area

	TRACK 1: Airbag Simulation Chair: TBC	**TRACK 2: Computing & Licensing** Chair: TBC	**TRACK 3: Simulation Confidence** Chair: TBC
3:50	**SAC Folding CAE Methodology** P. Rodriguez, Ford Motor Company	**The Effect of HDR InfiniBand on CAE Simulations** G. Cisneros-Stoianowski, HPC-AI Advisory Council	**Multiobjective CAE Model Homologation** R. Singer, Ford Motor Company
	Airbag Folding Simulation Impact for Steering Wheel Design I. Juarez, Joyson Safety Systems	**How to Make The Most Of Your Analysis and Simulation Applications** F. Thomas, Open iT	**Reliable Analysis without FE Mesh** S. Nageswaran, Altair Engineering
	CAE Side Curtain Airbag Deployment for Interior Trim Integrity in the Automotive Industry H. Hernandez, Ford Motor Company		

5:05 **Networking Reception in Exhibitor Area**

Tecnologico de Monterrey - Campus Santa Fe, Mexico City Floor Plans Will Be Provided Shortly

Exhibiton Hall (Lobby Areas)

Exhibitmpq will include8

- ESSS

- GrupoSSC

- Altair Engineering

- Intelligy

Conference Venue

Tecnologico de Monterrey -
Campus Santa Fe, Mexico City
Mexico

NAFEMS

As the only non-profit international association dedicated to the analysis, simulation, and systems engineering community, NAFEMS has established itself as the leading advocate for establishing best practices in engineering simulation. Over 35 years later, industry end-users, software and hardware solutions providers, researchers, and academic institutions continue to recognize NAFEMS as a valued independent authority that operates with neutrality and integrity. NAFEMS Americas supports over 400 member companies located in the Americas region who are actively engaged in the analysis, simulation, and systems engineering community.

NAFEMS Americas Steering Committee Members

Rodney Dreisbach (The Boeing Company, Retired), Chairman
Jack Castro (The Boeing Company)
Duane Detwiler (Honda R&D Americas)
Brian Duffy (Technip USA)
Graham Elliott (Bombardier)
Mario Felice (Ford Motor Company)
Joshua Huang (Exco Engineering)
Ronald Krueger (National Institute of Aerospace)
Edward Ladzinski (E.A. Ladzinski & Associates)
Matthew Ladzinski (NAFEMS)
Rodrigo Britto Maria (Embraer S.A.)
Laura Michalske (Procter & Gamble Company)
Tina Morrison (FDA)
Dennis Nagy (Beyond CAE)
Frank Popielas (Popielas Engineering Consultancy, LLC)
Pat Prescott (Owens Corning Science & Technology)
Marcus Reis (ESSS)
Charles Roche (Western New England University)
Andrew Wood (NAFEMS)

Presenter Name: Anaya, Julio

Presenter Company: Mabe S.A. de C.V.

Presentation Title: Structural, Injection Moulding and Forming Simulations during a Plastic-Metallic Infuser Development for an Automatic Laundry Machine

Submission Type: Presentation

Keywords: Structural, Injection Moulding, Forming, Simulation, Design, Process, Material, Characterization

Abstract:

This work deals with the iterative process between structural and manufacturing simulations needed to achieve a detailed design concept that meets the functions and requirements of a component used in a washer machine. Company internal design process is described and, therefore, injection moulding simulations, metal-forming simulations and structural simulations are described too as part of said design process. The component that provides mechanical action to the laundry in a washer machine is named infuser. A new infuser is required for a new laundry machine platform; this component is an assembly of two parts, a plastic part that is the mean body, and a metallic part that acts as a mask for the plastic mean body, this assembly has appearance, performance, structural and manufacturability functions. Plastic mean body is a polypropylene homopolymer part, its principal function is to provide stiffness to the complete assembly, by the other hand, the metallic mask is manufactured with stainless steel and its mean function is provide a good appearance inside the basket of the washer machine, finally as an assembly, both components must provide washability performance to the appliance. The challenge of this development was to find the geometries that could move the laundry to be washed, could resist the loads generated during the wash cycle, could be injected and that could be formed, all this at the same time, because of this, finite element analyses were carried out and several iterations were needed. The first objective achieved was the structural performance of the plastic mean body, stress and deflection criterion were reached, then, forming simulations were performed over the metallic mask until no wrinkles were presented and thickness reduction criteria was reached, Finally injection simulation were carried out and the warpage criteria were achieved too. Additionally, looking for confident results on the simulations, mechanical and rheological characterization for metal and plastic materials respectively were carried out. After satisfactory results, final design concept was achieve and drawings were released for tooling manufacturing. Then, at facilities, infuser components were fabricated and assembled in washer machine prototypes. Finally, a set of reliability and performance test were completed without failures. As a conclusion, in early development stages, structural and manufacturing simulations allowed a quick and effective process for design concept iteration, additionally experimental work is important, material characterization and the finite element model correlation with the real process provides to the team with major certainty at the time of design concept release.

Presenter Name: Buendia, Raul

Presenter Company: Delphi Diesel Systems Service México S.A. de C.V.

Presentation Title: Virtual Design Optimization of a Valve Train Actuator Using Computer Based Optimization (Multi-Objective Genetic) Algorithms

Submission Type: Presentation

Keywords: Design Optimization, Multi-Objective Optimization, Genetic Algorithms, Robust Engineering

Abstract:

When optimizing the performance of a component or engineering system, traditional methods suffer several drawbacks Although traditional methods can offer an “optimal” solution, it heavily depends on the decision maker’s prior knowledge. That is because they are mainly based on a trial and error strategy. Also, traditional methods are lengthy and costly processes. Thus, they are limited by the available time and budget resources. Typical engineering systems are described by a large amount of variables and they require the simultaneous optimization of more than one objective. The design engineer’s task is to specify the appropriate values for these variables. Yet, because of the size and complexity of this task, even the most skilled designer is unable to consider all of the involved variables simultaneously. In addition to that, the designer is frequently faced with a trade-off between objectives. Combining computer-aided-engineering (CAE) analysis and simulation tools with computer based optimization algorithms allows for a more realistic approach to tackle nontrivial problems. As more design parameters are considered, conflicting objectives are likely to arise. Then, the need for having a set of solutions instead of a single solution becomes natural. Obtaining a set of solutions is possible by implementing a multi-objective optimization. As such, Mutli-objective genetic algorithms (MOGAs) enable the designer to obtain a set of pareto-optimal designs. With a set of solutions, the engineer can compare and make better decisions. In essence, a MOGA will make the objective conflicts come out in a clear way to the decision maker. The purpose of this work is to apply these techniques to the study of a valve train actuator. The goal is to optimize the magnetic force which includes several design variables. The objectives concerned with the magnetic package analysis are to increase the average force while minimizing the force variation through all the actuator stroke. This is achieved by means of a Genetic Algorithm (GA) technique. In addition, to evaluate the objective response function, S/N ratio is calculated similar to the robust engineering methodology. Another aspect that this work addresses is the automatization of the process by the use of modern simulation tools. The Isight software was used as the Process Integration and Design Optimization (PIDO) tool. Isight integrates engineering CAD/CAE tools (NX & Maxwell). The resulting solution Pareto surface is then analyzed using the postprocessing tools within Isight. This allows individual designs that best meet all criteria to be chosen.

Presenter Name: Camargo, Eloy

Presenter Company: Ford Motor Company

Presentation Title: Stamping Feasibility CAE Simulation at Early Design Stages

Submission Type: Presentation

Keywords: CAE, Ford of Mexico, Stamping Simulation, Feasibility, Manufacturing, Tooling Design, Optimization

Abstract:

Ford is a global company, founded at Dearborn, Michigan in 1903. First Mexican manufacturing operations of Ford began in 1925. Subsequently, in 1964, a product development office started operating to support locally made products. Today, this PD office has more than 1500 Mexican engineers. The stamping feasibility team started operating at Ford of México around 6 years ago to support local engineering activities. Currently our capabilities allow us to support local engineering and other regions around the globe as well. The CAE simulations for the manufacturing of stamped parts help product development to detect potential issues early in the design stages. We perform virtual tests iteratively with the design group until a feasible part is obtained. The main inputs for this process are CAD geometry, the material properties and a historic process benchmark. Stamping simulation results predict the strain behaviour throughout the part and its tendency to split, thin or wrinkle. This allows for an iterative improvement of the form, which is essential in getting a feasible part and its tooling correctly on a proper time. One of Mexico's success stories is about the Transit Connect 2019. This vehicle's top hat was designed in Mexico. The challenge involved dealing with new weight and strength targets using newly introduced high strength materials. Parts including the front bumper reinforcement inner and outer passed through the above mentioned design iteration stages. The first stamping simulation results showed challenging zones that where improved to the point of having fully functional parts in production. Without this preventive work, some issues could go undetected beyond the completion of the tooling. Detecting these expensive problems early on is critical in the competitive market of today, but our activity also improves part quality and delivery time, reducing the amount of overall changes in the process, and therefore optimizing manufacturing costs. Given the trend of increasing concentration of CAE activities based in Mexico, our location is very favorable in that our direct interaction with other CAE groups makes possible the synergy needed to obtain accurate results. This proximity provides us with first hand insights into the downstream life of our results, and allows the inclusion of stamping variables in the virtual tests of other disciplines. We also constantly propose new ideas for the improvement of our processes. Our capabilities forecast for the next 10 years covers three areas: correlated virtual validation, collaborative innovation work with CAE software leaders and the integration of automation to our processes.

Presenter Name: Cisneros-Stoianowski, Gerardo

Presenter Company: HPC-AI Advisory Council

Presentation Title: The Effect of HDR InfiniBand on CAE Simulations

Submission Type:

Keywords: CAE, CFD, HDR, InfiniBand, Scalability, SHARP, In-Network

Abstract:

From concept to engineering, and from design to test and manufacturing, engineers from wide ranges of industries face ever increasing needs for complex, realistic models to analyze the most challenging industrial problems; Finite Element Analysis is performed in an effort to secure quality and speed up the development process. Powerful virtual development software is developed to tackle these needs for the finite element-based Computational Fluid Dynamics (CFD) simulations with superior robustness, speed, and accuracy. Those simulations are designed to carry out on large-scale computational High-Performance Computing (HPC) systems effectively. The latest revolution in HPC is the effort around the co-design approach, a collaborative effort to reach Exascale performance by taking a holistic system-level approach to fundamental performance improvements, is In-Network Computing. The CPU-centric approach has reached the limits of its scalability in several aspects, and In-Network Computing acting as “distributed co-processor” can handle and accelerates performance of various data algorithms, such as reductions and more. The past focus for smart interconnects development was to offload the network functions from the CPU to the network. With the new efforts in the co-design approach, the new generation of smart interconnects will also offload data algorithms that will be managed within the network, allowing users to run these algorithms as the data being transferred within the system interconnect, rather than waiting for the data to reach the CPU. This technology is being referred to as In-Network Computing, which is the leading approach to achieve performance and scalability for Exascale systems. In-Network Computing transforms the data center interconnect to become a “distributed CPU”, and “distributed memory”, enables to overcome performance walls and to enable faster and more scalable data analysis. The new generation of HDR 200G InfiniBand In-Network Computing technology includes several elements - Scalable Hierarchical Aggregation and Reduction Protocol (SHARP), a technology that enables to execute data reduction algorithm on the network devices instead of the host based processor. Other elements include smart Tag Matching and rendezvoused protocol, and more. These technologies are in use at some of the recent large scale supercomputers around the world, including the top TOP500 platforms. HPC-AI Advisory Council performed deep investigations on a few popular CFD software to evaluate its performance and scaling capabilities when using HDR InfiniBand interconnect. The study reveals the influence of the applications on runtime, scalability and performance of the simulations.

Presenter Name: Cook, Victor

Presenter Company: Altair Mexico

Presentation Title: How the Simulation-Driven CAE Process May Be Profoundly Changed by Data Intelligence and Machine Learning

Submission Type: Presentation

Keywords: Simulation, machine learning, data intelligence, multidisciplinary optimization, multiphysics

Abstract:

Current computing technology of all scales, together with software architecture that enables complex physics replication at increasing levels of easiness of usability and highly detailed and immersive visualization, seems closer than ever to fulfill the expectations of simulation as a faithful surrogate of the physical event -and popular, as well. The ability to estimate the outcome of a physical test without setting foot on a laboratory has made it possible to confidently use simulation to design, study and validate virtual products under a multitude of circumstances. This is well-known. What may be suspected, but not consciously even, and therefore not properly addressed by the seasoned CAE specialist is that, while faster, more intensive simulation is still needed and is increasingly finding a steady spot across design, engineering and research departments, the possibility of a virtually instant solution, true real-time simulation on par with physical phenomena, is only enough if this uniquely constrained numerical model is but one of multiple blocks comprising a validation process of a product or, speaking more generally, of a physical system to be studied. The reason is insights and decisions to be made are based on data. One runs a simulation to obtain data, and there is only so much a specialist can conclude from a single calculation, whether it takes weeks or a second to finish. Large results files do not equate plentiful, useful data, and copious historical markers and instant readings on functioning sensors often do not make sense by themselves beyond their evident form. Considering how the standards have tangibly changed for what we came to expect from computers as an aid to engineers and scientists, for instance, with applications of growing popularity in the CAE domain like stochastics and multidisciplinary optimization, or virtual factories and digital twins of systems operating on the field waiting to be timely maintained, the case for data creation, extraction, and processing at all levels -in the ways of Data Intelligence and Machine Learning- is as solid as ever, for these technologies, together with HPC, make those, among many more, applications possible. Through different examples of potential functionalities of Machine Learning algorithms and Data Intelligence methods in the context of simulation-based applications, it is discussed how current CAE processes -from model creation and set up, to methods for finding optimal, unexpected solutions- can be enhanced, if not drastically transformed, in ways we are forced to think what place simulation might occupy in the future and, therefore, what to demand from simulation itself.

Presenter Name: Dri, Fernando

Presenter Company: Dassault Systemes SIMULIA Corp

Presentation Title: Beyond CFD – Powering Sustainable Innovation on an Integrated Platform

Submission Type: Presentation

Keywords: 3DEXPERIENCE, CFD, Optimization, Realistic Simulation, Automotive

Abstract:

For years, wind tunnels have been the main tool to assess vehicle aerodynamic performance. As regulations, competition and model proliferation push the limits of vehicle design; Computational Fluid Dynamics (CFD) has gained considerable traction as an alternative method to experimental measurements. CFD is more than just a virtual wind tunnel. The use of simulation allows for conceptual investigation at early stages of the design process. Different design alternatives can be studied and compared without the need for complex prototypes. This translates into significant saving of costs and time. Realistic simulations allow for greater insights beyond what is possible with traditional experiments. CFD provides a complete picture of the flow field. This enables engineers not only to understand what is happening, but also why it is happening. In this presentation, the author demonstrates the design, analysis and optimization of a car rear wing using the integrated environment provided by the 3DEXPERIENCE Platform. The objective of this work is to obtain the most efficient geometric configuration; maximizing the downforce produced by the rear wing while minimizing the generated drag. In order to carry out this study, it has been necessary to create a fully parametrized rear wing geometry that could be used to explore the design space. A set of 28 independent parameters were used to define the shape of the rear wing (e.g. span, chord, angle of attack). A steady-state, incompressible fluid flow analysis was employed along with a K-Omega-SST turbulence model to compute the downforce and drag forces for each of the geometrical configurations. The model setup (mesh, properties, boundary conditions) was fully automated based on an initial template defined by the author. A Design of Experiments (DOE) was performed using the Optimal Latin-Hypercube sampling technique. The DOE allowed us to explore the design space and obtain a response surface to optimize the final shape of the rear wing. The seamless integration between design, analysis and automation allowed us to perform the described workflow on its entirety using the 3DEXPERIENCE Platform.

Presenter Name: E. Turanzas Forseck, Miguel

Presenter Company: Ford Motor Company

Presentation Title: CAE Simulation for Cost Reduction Strategies

Submission Type: Presentation

Keywords: Material utilization, optimization, stamping processes, sheet metal, cost reduction, simulation, product design, tooling, continuous improvement, automation, simplification, manufacturing, suppliers, CAE tools, response time, waste, Ford Motor Company, Ford of Mexico

Abstract:

Nowadays, in a very competitive market, costs are becoming a critical part of the equation to survive in the actual environment and for the next 10 years, in which we will be living lot of changes and potential closing of several companies. Around 8 years ago, Ford Motor Company has developed an area that uses CAE simulation for reducing material utilization on a vehicle, specifically for sheet metal parts. The main responsibility of the area is cost reduction in four fields: product and tooling design, process planning and cost estimating. All of the above have something in common – optimization. From predicting the best feasible process, passing through part's design improvements and simple tooling development reaching to the cost estimation itself, where all takes a fundamental role in early stages of a program by avoiding late design and process changes. In Ford of Mexico, from nearly 5 years ago, many global platforms run through this optimization process, where all starts from a data entry that is inspected and entered on an FTI's automated software (Forming Suite) that will return several process options for each part included in the BIW of a vehicle. With the simulation, material utilization engineers are able to look for opportunities to reduce raw material used to produce a part, such as: • Looking at control points on the layout to make changes accordingly. • Placing the part in an optimal way through the stamping process. • Simulate and suggest an amount of addendum. Then the MU engineer select and shares the best feasible option on a report, which shows all the part's information and process values needed to communicate with other areas and suppliers. By doing this, Ford will externally share with the manufacturing plants how a sheet metal part should optimally be stamped and, in the other hand, will internally realize if a part's design is good enough to reduce waste, increase manufacturing efficiency and be viable in terms of cost. As a conclusion, using CAE tools has helped Ford to reduce costs globally and one key point in the future will be to shorten response time, market needs will increase and customers will look for new products quicker than ever, asking for fresh designs and not the same old ones. This could be done by simplifying the process and automating sub processes, as Ford has been doing as a continuous improvement strategy. Additionally, for the upcoming years, the challenge will be the fully automation of the process, based on historical data and part's geometry description to throw, as a first pick, a best feasible scenario for stamping processes that includes all fields mentioned previously, from design and manufacturing perspectives.

Presenter Name: Ernesto Hernández Álvarez, Victor

Presenter Company: Ford Motor Company

Presentation Title: The Relationship between Ribs Layout on a Body Exterior “Belly Pan” Part, and Its Prediction of Moldflow´s Warping

Submission Type: Presentation

Keywords: CAE, Plastic simulation, Plastic molding, Ribs layout, TPO, Moldflow, Warpage, Belly pan

Abstract:

One of main goals in the automotive industry is bring products that achieve and exceed the high quality levels to the final costumer as well to the internal one in the supply chain. The plastic parts play a critical role in this chain, as they are present in an important proportion inside a car. In an assembly process, it is quite common not to reach the range of geometrical and dimensional tolerances, due to the warpage present in the parts. One possible solution is to investigate the warpage in the parts to be assembled and compare their dimensions with the tolerance of the design. Sometimes, the design proposals do not consider the relationship of the molding process conditions, the geometrical attributes and the warpage trend. Therefore, in this work is studied the influence of the four different rib layouts in the warpage trend, as well its magnitude. Other variable analysed was the thickness of the ribs. The part used in this study was a body exterior “Belly pan” part. For this analysis, the material considered for the belly pan was a thermoplastic olefine (TPO). The study of the four different geometries was made using a Computer-aided Engineering (CAE) software Autodesk Moldflow Insight. In this case, the software allows to the designer improve and optimize the part and tool design considering the warpage as an output. The molding process conditions and the tool design (feeding and cooling system) were constant in all the simulations. A Design of Experiments (DOE) and the analysis of variance (ANOVA) were develop to find the lower warpage values produced by the rib layouts and rib thickness configurations. The different results in warpage shown the importance of a good selection of rib layout, as well its thickness on a “Belly Pan” part, to avoid overdesigned parts, as well late engineering changes in the product development cycle time.

Presenter Name: Flores Aviña, Kristian Ulyses

Presenter Company: Ford Motor Company

Presentation Title: CAE Applied at the Right Place and at the Right Time

Submission Type: Presentation

Keywords: Product Development, automotive, CAD, CAE, powertrain, powertrain mount brackets, optimization, NVH, Durability and First time right design

Abstract:

Nowadays, in a product development process throughout the automotive industry has become common to see CAE applications during all phases of the design process, and this discipline is enabling, since its introduction, to design products in a faster, cheaper and optimized ways, with improved performance. In addition, new tools in the market are allowing to optimize the components to a level we have never seen, taking into account multi objective functions in an automated way. However, we are missing an important piece of the puzzle, the CAD required by a CAE engineer to conduct the necessary assessments to the component. Although it is known that some software suites have some capabilities to connect CAD and CAE to speed up the optimization and iterative processes, the truth is that are not yet at the required level. Normally the first CAD intent of the design has no useful data to guide the systems interaction between components, besides static and dynamic clearances. Here is when the following question comes up, what if CAD and CAE rather than being considered as services areas to support the development of components, are considered as protagonists, acting at the right place and at the right moment of the design process? Designing vehicles is a very complicated affair. Consequently, this presentation is intended to show a proposed optimization process that takes advantage of available CAE capabilities embedded in CAD tools, cross-functional collaboration between CAD designers/engineers, CAE engineers and D&R engineers. It starts by using CAD software with embedded CAE Wizards to start giving guidance to CAD designers from day one then translating this information directly to CAE, in order to optimize the design by means of structural optimization that will enhance component's performance at system level. This process exemplify its application in powertrain mounts design. The reason to use these type of components is to show an example of this process' the functionality and capabilities in a very complex component, that needs to satisfy several attributes and requirements, such as, NVH or durability. This process allows the possibility to achieve these requirements since initial design stages if all the actors have useful technical information to make decision at the right place and at the right moment.

Presenter Name: Flores Mora, Joel

Presenter Company: Robert Bosch Mexico Sistemas Automotrices S.A. de C.V.

Presentation Title: Using CFD to Minimize Emissions and Combustion Instability of a GDI Engine at the Catalyst Heating Operating Point

Submission Type: Presentation

Keywords: CFD, simulation, GDI, spray, injection, combustion, catalyst, spark plug, mixture formation, wall wetting, fuel consumption

Abstract:

Due to the limited resources of fossil fuels and the increasing demand for an environmentally friendly mobility, reducing the fuel consumption and especially the particulate emission, while maintaining or even increasing the engine power, is the key point of the combustion system development for GDI engines. The understanding of the in-cylinder processes, necessary for this purpose, can only be achieved by a complementary use of experimental investigations as well as numerical simulation. This however leads to great demands on the 3D-flow-simulation, which can only be met by an appropriately detailed modelling. According to the combustion system development process for gasoline engines applied in Robert Bosch LLC, spray-targeting CFD-simulations are performed in an early phase to support different projects with different customers in North America and Europe. Since July 2018 a newly formed system engineering simulation team inside the engineering center of Bosch México in San Luis Potosi is supporting the global combustion system development team with these CFD simulations. To illustrate the complexity of those simulations of the gas exchange, injection and mixture formation in a GDI engine a current project will be described. An important part of this project was to improve the engine behaviour during engine start. By using the so-called catalyst heating operating point the time for reaching the light-off temperature of the catalyst can be reduced and thus emissions, especially during cold start, can be minimized. The main characteristic of this operating point is a retarded spark timing, enabling the generation of an increased thermal energy in the exhaust system. As this approach typically degrades the engine combustion stability it presents additional challenges to the mixture formation process. To increase the combustion stability multiple injections are typically used, with a very late injection close to spark timing. Thereby, it is desired to have a slightly fuel-rich mixture around the spark plug in order to improve ignition conditions. However, to reduce soot emissions, the overall fuel-air mixture inside the cylinder has to be as homogeneous as possible to avoid fuel-rich regions apart from the spark plug area. Another important source of soot emissions is the fuel evaporating from the cylinder walls. Thus, especially the fuel impingement of the late injection on the walls needs to be reduced. To achieve these targets, the spray layout and injection strategy need to be optimized for a particular engine geometry. Therefore, the details of the mixture formation process inside the combustion chamber have to be known. This can only be realized by using 3D-CFD simulations in addition to measurements at the test bench. In the current project several spray layouts and injection strategies were evaluated numerically. Finally, all the parameters mentioned above were improved by choosing the appropriate spray calibration.

Presenter Name: Florez, Christian

Presenter Company: Ford Motor Company

Presentation Title: Seam Weld Optimization in Automotive Systems for Durability Analysis

Submission Type: Presentation

Keywords: Durability, optimization, seam weld

Abstract:

Since weight optimization is a priority for all OEM's, there is huge value in reducing the number and size of welds while meeting all structural targets on the body system. The project seeks to minimize the number of seam welds required for an automotive system to meet durability requirements. The optimization approach is seized from noise, vibration, and harshness (NVH) analysis on spot welds resulting in an 18% decrease of these connections. Moreover, optimization approaches for fatigue analyses in spot-welds are revised to scope a suitable extrapolation to seam-weld applications. The study case is a rear-towing device or hitch system analyzed for durability. Connections between components are critical for target compliance. However, economic and manufacturing restrictions challenge the placing of seam welds efficiently. Therefore, parametric optimization combined with scripting help CAE engineers identify and propose the optimal locations to add seam welds, that otherwise would represent immense amounts of time and iterations; with no guarantee of finding a minimum. This work presents a mathematical model that can be used in different stages of product development to aid engineers in decision making to minimize seam-welds in automotive components considering durability analysis.

Presenter Name: Garcia, Alejandro

Presenter Company: Ford Motor Company

Presentation Title: Automated Pre-processing Method for BIW Mesh Creation Including Spot-Welds Using Open Source Programming Language

Submission Type: Presentation

Keywords: Pre-processing, scripting, optimization, automation, enhancement of CAE value added

Abstract:

The Pre-Processing stage of a vehicle CAE Analysis based upon numerical mesh methods demands for setting up a suitable Body In White (BIW) model. Generally, this stage is performed with the aid of computational tools specifically designed to handle such a task. Depending on the complexity of the vehicle model, the time invested in carrying out the Pre-Processing may increase gradually from some hours to a couple of days turning a crucial process tedious, very laborious, and risk prone. Broadly, during the Pre-processing a CAE Engineer must accomplish BIW Geometries' Cleaning up, Meshing, Material & Thickness assignment, and Spot-welds creation. Unfortunately, each of the steps are low value added and require tremendous effort due to the rate of time invested compared to the scheduled deliverable Milestones. Furthermore, to highlight weaknesses in a common Pre-process, the constant design updates in combination with the amount of design versions (for vehicles: 5/3 door, panoramic, steel, etc.) to be analysed, expose how the difficulty of a BIW model setting up may exponentially increase. In this paper, a novel implementation of the Pre-Processing stage through a series of scripts is shown. Each of the steps early mentioned are coded using open source programming language coupled with third party pre-processor software. Scripts execution have highlighted not only a massive reduction in time during the Pre-processing, but also have revealed some issues in PLM software (Company database from which the Geometries are taken). A thoroughly discussion about scripting optimization, evidence of improvement in time over current scheme, and PLM software dependency drawbacks are exposed. Finally, concluding remarks about success cases using scripts, and future enhancements are presented.

Presenter Name: Hernandez, Hector

Presenter Company: Ford Motor Company

Presentation Title: CAE Side Curtain Airbag Deployment for Interior Trim Integrity in the Automotive Industry

Submission Type: Presentation

Keywords: CAE, side curtain airbag deployment, Particle Method

Abstract:

Mainly during last decade the numerical simulation of the airbag deployment process has become a standard application of commercial finite element codes. Side Curtain Airbag (SAC) deployment is directly related to Safety Requirements (Full Side Impact and Ejection Mitigation), which is fundamental for the product design and final customer's safety. This phenomena is a very complex process to simulate because it includes a high vehicular integration level that commonly yields in expensive late design changes during physical validation testing. Traditionally, product deployment of side curtain airbags and its interaction with Interior Trim is driven by physical test, based in expertise of Restraints Specialist and conjunction of airbag External Suppliers. CAE alternatives have proven to be an effective option to reduce design timing and cost in automotive industry. For this reason is imperative to develop a CAE methodology that accurately predicts the Interior Trim behavior when SAC deploys. This will allow engineers to get a timely direction about how to proceed, with the expectation of improving design efficiency time and cost wise. Also this methodology can be subjected to create further knowledge among the engineering community based on analysis of results from simulations. The present study aims to describe a CAE methodology to analyze the interaction between the SAC and Interior Trim components during static deployment with Particle Method to capture the cushion´s kinematics due to the gasses flow and define a representative load case in the CAE assembly. An important portion of the SAC deployment methodology is the CAE material characterization for upper hard trim plastics and headliners which is needed to predict correct behavior of materials at very high strain rate (more than 1000 1/s). CAE Correlation and predictions compared with physical test are presented of successful cases based on analysis of testing videos and CAE. This methodology allows to predict potential failure modes such as: component fracture or crack, poor retention, trapped airbag behind hard trim and accurate SAC trajectory.

Presenter Name: Hinojosa, Humberto

Presenter Company: Grupo SSC S.A. de C.V.

Presentation Title: Crude Oil Properties in CFD, Methodology and Case Studies

Submission Type: Presentation

Keywords: Crude Oil properties, CFD, CFX, density, viscosity, Fortran

Abstract:

With the growing complexity in oil extraction, transport and conditioning systems, engineers face the challenge of understanding and describing physical phenomena as well as designing equipment and processes associated with crude oil production in adverse situations. Currently, numerical simulation tools such as the Computational Fluid Dynamics (CFD) help the engineers to better understand the physical processes that take place inside equipment or installations where a direct visualization is not possible. It also allows the engineers to design equipment with virtual prototypes, reducing costs with respect to conventional design methods and saving resources, and it even can be used for the optimization of the equipment/processes previously designed. However, in order to build the numerical models in CFD, it is necessary to incorporate all the information of the equipment that will be used in a given process as well as the information of the properties of the fluids that will be processed. The properties of reservoir fluids are very diverse, and can change with the location of the production field and even over time. Therefore, in order to be able to construct a realistic CFD model, the implementation of the physical models that describe properties such as density and viscosity of the fluids of interest in the computational models is required. This paper describes a methodology to incorporate the physical information of reservoir fluids that consists of a previous characterization that includes experimental data such as composition, density and viscosity at different temperature and pressure conditions. This characterization procedure has the purpose of generating libraries, which contain all the physical information of the fluids of interest. These libraries can be exported to different CFD tools, one of them is ANSYS – CFX. Case studies about a transport line and a separation system will be shown, where special libraries written in Fortran are used to provide the crude oil properties to a material in ANSYS – CFX, and both will include phase change modeling.

Presenter Name: Ibañez, Mario

Presenter Company: Grupo SSC S.A. de C.V.

Presentation Title: Design and Analysis of a Multiphase DC Motor, through Numeric Simulation

Submission Type: Presentation

Keywords: DC motors, Multiphysics, numeric simulation

Abstract:

DC motors have endless applications on mechatronic systems, their electrical characteristics make them the best option for these applications especially if we compare them with other motors such as alternate current motors, servomotors, step motors, etcetera. The applications for DC motors are very diverse and can be found in robotics, biomedics or the automobile industry. In the automobile industry a lot of technology regarding the use of DC motors has been developed, some examples are the windshield cleaners, gasoline pumps, water deposit pumps and the most significant example is the DC motor used for the traction of the car. As it is well known, each year fabricants from all the different automobile companies are constantly improving their designs, from the structure to the electrical, mechanic and electronic components. DC motors that are used in the integration of automobiles, as well as all the other components that are used in a car, are constantly being modified in order to improve their efficiency. There are several techniques that are used in order to achieve the maximum efficiency. Sometimes with their experience DC motors fabricants are capable of satisfying the needs demanded for the creation of new vehicles. However, there is an additional possibility that can allow us to raise the quality. In this work an alternative for building DC motors that can be used in vehicles applications will be shown. This will be through numeric simulation and the finite element method. This technique consists on the virtual creation of a DC motor which is used for the gas pump of an automobile. This will be made considering all of its physic, mechanic and electric variables.The Ansys software as well as Multiphysics studies will be used in order to achieve this purpose. Physic test will be applied on the model in work conditions. The development of the motor will be shown through Multiphysics and structural mechanics studies; fluids dynamics and electrical studies will be applied as well. All of this will be made with the purpose of knowing the operation and efficiency of the product, before it is manufactured.

Presenter Name: Juarez, Ilse

Presenter Company: Joyson Safety Systems

Presentation Title: Airbag Folding Simulation Impact for Steering Wheel Design

Submission Type: Presentation

Keywords: Airbag folding, automotive, steering wheel, driver airbag, airbag modelling

Abstract:

Driver airbags and steering wheel components are an important part of passenger automotive safety. Their design is usually physically tested; however, predictability of airbag simulations is becoming an important tool in the design process of safety components for the automotive industry. Evaluation of the correct deployment of the airbag is part pf the design process by testing the structural integrity of the steering wheel components. During the deployment, the airbag interacts with the components and they are subjected to high speed forces, making it important to maintain their structural integrity. At the cost of physical tests, using Finite Element Analysis (FEA) for the assessment of interaction of the airbag and steering wheel components has become an important tool in the design process. Several approaches have improved the simulations of airbag deployments, such as using the airbag particle method, in addition, folding of the airbag mesh also impacts the interaction between the airbag and other components as it begins to unfold. Currently, crunching of the airbag mesh is used for simulating deployments and analyzing components of the steering wheels. This method sometimes provides intersections within the mesh of the airbag. Avoiding intersections within the airbag mesh after it has been folded is critical for correct airbag deployment during simulation. Therefore, following the physical steps of the airbag folding is crucial for a more precise correlation between a physical test and the simulation, consequently being able to predict behavior of several components after changing their design. This report summarizes the procedure followed for folding a driver airbag using Sim-Folder/ Pam-Crash according to the design team steps, and its effects on the proper correlation of the tests and design improvement implementations. Comparison between deployment of the crunched airbag and properly folded airbag are presented.

Presenter Name: Leonov S. López, Fabian

Presenter Company: LURI Engineering

Presentation Title: Intrusion Car Body Optimization Combining Frontal and Side Crash Responses

Submission Type: Presentation

Keywords: Topometry, Topographic, Large scale optimization, Equivalent Static Loads, Nonlinear

Abstract:

Automobile Manufacturers are required to design their vehicles for safety so that the occupants will survive of a variety of crash scenarios. Computer simulated crash analysis evolved over the years to help augment the crash rest programs and to give engineers to insight into the crash events. These nonlinear simulations have become commonplace during the design phase to save time and design cost. Designing an automobile for compliance with these safety standards along with fuel efficiency standards is hard because of some contradiction between these requirements. Today, to improve the design, analysts and engineers are using analyses and coupling them with general purposes optimization packages, hence, several crash conditions simultaneously can be defined to obtain the last response of the structure and utilizing optimization tools. Crash simulations typically require a significant amount of computational time and resources. While using a general purpose optimization package the cost exceed. As a result, there is an important interest in using approximate models to deal with optimization processes. An approximated model is constructed from a limited set of simulations and used during the optimization to obtain an improved design. But in normal conditions, there is a limitation on the number of independent design parameters that can be used because of the accuracy of the approximate models. This research describes a design system to optimize the non-linear responses computed from a Nonlinear software and General Purpose Optimization packages using various optimization techniques, especially with large-scale (large number of design variables) optimization. With the Nonlinear package software will be implemented to perform equivalent static loads (ESL Method) based on a Nonlinear analysis responses. The optimization package will be used to optimize the structure under frontal and side crash while decreasing intrusions at nodal outputblocks. Equivalent Static load method will require multiple iterations process of non linear structural analysis and Optimization. Large scale optimization techniques, such as, topology, topometry and topography will be implemented easily.

Presenter Name: Melo, Dulce

Presenter Company: COMPLX

Presentation Title: FEA Based Design of Roll Forming Processes

Submission Type: Presentation

Keywords: Roll forming, Design, FEM, Implicit, LS-Dyna

Abstract:

Roll forming is a process used to manufacture sheet metal products such as garage doors, metal roofing, windows frames, and countless others. The process consists in metal strips that are deformed progressively into all kind of sections by a successive set of rotating rolls. The complex process of forming generates stresses beyond the linear limit of the applied materials, generally metals. The quality of the final product depends on variables like strip thickness, material properties, roll speed, but mainly, in the correct design of the roll forming process, including the tools. Despite the common design practices, the roll forming tool design procedure still remains more of an art rather than a science. Design errors, which consist on the difference between the desired profile and the obtained geometry, can be verified by a finite element analysis, where strain distributions and sheet geometry can be predicted. Although fracture of material is unusual, defects of the process include flaring, twisting and final dimensions outside of the specified tolerances. In this study, the described simulation model allowed the validation of the process design for a new component, specifically developed according to the requirements of an industrial roll-former. An explicit analysis using LS-DYNA® was employed to simulate the rolling stations. The set up of the dynamic analysis considers the main aspects of any FEM analysis: the boundary conditions of the system, applied as prescribed motion on the rolls and sheet, the contact between surfaces, along with the elasto-plastic material definition, timestep controls, and naturally, the correct discretization of the geometries. Indeed, the dependence of the accuracy on the mesh size was proven. The development project comprised an initial stage, where an existing roll forming machine was simulated, in order to use the physical available results to validate the set-up conditions for the analysis. This was followed by the flower (a sketch structure of a successive bending of the product when passing through the rolling stations) design of the new profiles. The design of the new roll forming tools was made according to the flower design. Finally, 3D rolling simulation models for the new rolling stations were generated. A series of multiple simulations were made to correct the defects encountered in the final form. It was observed that the main variable that controls the resultant profile is the angle in which the transition of a fold is made, so the flower design went through an iterative process. The radios of the folds and longitudinal measurements of the virtual profile were inside ± 0.5 and 1 mm respectively. With the current validation process, the design errors could be detected and corrected, giving confidence to the customer for commissioning the tooling manufacturing.

Presenter Name: Nageswaran, Shan

Presenter Company: Altair Engineering

Presentation Title: Reliable Analysis without FE Mesh

Submission Type: Presentation

Keywords: Improving Confidence in Analysis without FE mesh

Abstract:

Computer Aided Design (CAD) and Computer Aided Engineering (CAE) are intrinsic parts of modern engineering. CAE simulation tools are extremely important because they allow for performance validation and optimization of a product design before the product is physically created. Today this is typically done using software based on Finite Element Analysis (FEA). The weak link in traditional FEA is the need to create an accurate and efficient finite element mesh. The meshing process would typically fail if done on original geometry, or produces excessive number of elements which would often make FEA impractical. In case of assemblies, incompatible meshes on adjacent parts make handling part connections extremely difficult. In practice, geometry model simplification is a must for a successful FEA. But the simplification is a non-trivial step which requires a highly skilled specialist in both FEA and CAD technologies. External Finite Element Method is intended to drastically streamline simulation process and increase engineering productivity and accuracy by replacing the underlying FEA technology, rather than through improvements to user interface. New technology eliminates the most failure-prone and skills/labor consuming stages of analysis process. This paper was written to provide understanding of how the technology works. Technological foundations of External FE method are discussed with respect to mathematical background, computer implementation, and positioning among other numerical methods. TECHNOLOGY SUMMARY External FE method can be used develop application which performs statics, dynamics and thermal, analyses of structural systems. External FE method always uses fully featured, not simplified, solid geometry models in the analysis and does not use a mesh. External FE method computational engine is based on breakthrough extensions to the theory of external approximations. External approximations are a generalization of Finite Element Method (FEM) in terms that: • absolutely arbitrary geometrical shapes can be used as “finite elements” • basis functions which approximate field of interest in the “element” can be of arbitrary class and are independent of the “element” shape, alternatively to strictly structured polynomials used in conventional FEA External FE method controls solution accuracy using multi-pass adaptive analysis. Adaptivity can be defined on a global or part local basis and adaptivity is always active. CONCLUSIONS In order for simulation to truly drive the design process it needs to work lock step with each geometry concept and concept modification. The complexity of traditional FEA eliminates its use in all but the most trivial of design conditions. Simulation working directly on design geometry provides a path to quick meaningful answers that guide designers and engineers to more optimal design scenarios. Only External FE Method can provide this by not only eliminating time consuming and expertise extensive geometry simplification techniques such as defeaturing and mid-planning but by also eliminating the mesh all together. The result is a simulation tool that is both: • Fast enough with respect to both model and solve time to be used every day • Accurate enough compare to tradition FE method

Presenter Name: Pérez Santiago, Rogelio

Presenter Company: Joyson Safety Systems

Presentation Title: Modeling Structural Behaviour of Metallic Safety Components

Submission Type: Presentation

Keywords: Manufacturing tolerances, material properties, boundary conditions, elastic-plastic behaviour

Abstract:

Besides its value for the final validation or root cause analysis of designs, Finite Element Analysis (FEA) can offer critical information at early stages of development. This rationale has fostered the utilization of intuitive FEA modules, with limited capabilities, embedded in most of the market-leading parametric CAD tools. Different to typical applications, where material yielding is used as the main failure criteria, the design of safety components comprises the need to control deformation beyond the elastic limit up to material failure. Thus, being able to predict the elastic-plastic behaviour of the structural components of interest may mean the difference between correct or out of order system performance. The previous requirement, in turn, demands for specialized FEA tools and closer to reality modelling procedures. While metallic failure prediction is an ongoing research topic, this presentation reviews the effect of different modelling approaches, assumptions and simplifications used to estimate the resistance of a typical steel component under pull-test conditions. Design details, geometric variation, element types and formulations, boundary conditions as well as material model and properties were explored and their effects on load bearing quantified and compared. Considering the manufacturing effects, forming in this case, of the component was also explored. After a briefly review of the current trends in Verification and Validation of FEA results, this report summarizes the procedure used to evaluate the influence of the afore-mentioned variables on the results. Data analysis allowed to identify the main effects from those parameters with low impact on the results. The final part of the document shares the lessons learned from this project and recommendations for future studies.

Presenter Name: Quiroz Garfias, Cuauhtémoc

Presenter Company: Ford Motor Company

Presentation Title: Weld Points vs. Body Performance Study

Submission Type: Presentation

Keywords: Spot Weld Optimization, Finite Element, HEEDS, TCL, NASTRAN

Abstract:

At an advanced stage of the development process where the vehicle was mature and already met the required performance, the manufacturing team informed that the capacity of specific welding stations was exceeded in one of the plants where the vehicle was going to be manufactured. Leading to decide between investing to increase the plant capacity or reducing the amount of welds in the design to fit current capacity. The first option implied a cost that must be justified and in the second there was a risk of impacting the performance. To support the decision, the team carried out a study to better understand the impact on performance due to the reduction of welds. Since the original design included several thousands of welds and the response time was short, the design space had to be bounded. Considering conditions like commonality, supplier welds and safety, the subspace to study was about 600 spot welds, each considered as a discrete and binary variable, presence or absence. Given the fact the amount of combinations was high and two load cases needed to be evaluated per combination, the subspace did not qualify for physical testing and was not manageable for one analyst. Therefore, it was decided to use virtual validation (FEA) along with automation and optimization tools (TCL for Hyperworks and HEEDS) which would help to obtain the optimal solution in the trade off curves of performance vs number of spots. After 600 iterations, the result was that 18% of spots could be eliminated with a marginal impact of 0.6% and 1.6% in torsion stiffness and diagonal distortion at liftgate opening respectively. This provided the data that supported the decision to not to invest in increasing the capacity while staying within acceptable level of performance.

Presenter Name: Ramirez, Roberto

Presenter Company: Frisa Forjados S.A. de C.V.

Presentation Title: Analysis and Simulation of the Forging Process of an AISI 4340 Cast Ingot to Reduce Internal Defects and Energy Resources

Submission Type: Presentation

Keywords: Open die forging, simulation, forge, porosity, cast ingot

Abstract:

The use of simulation in material forming processes can represent a substantial saving in scrap material and failed experiments. Having prior knowledge of the result of the process parameters can help reduce the number of physical experiments and have a better understanding of the relation of all these factors. With the combination of both simulation and experimental data, a model can be created to predict the behaviour of future processes accelerating the testing and validation phases of the developments. This advantage becomes noteworthy when dealing with large parts and expensive processes as the open die forging. Minimizing internal defects of cast ingots by giving the necessary amount of work during deformation is the main task in this kind of forging. In order to remain a competitive option for our clients, an optimization of the resources becomes of crucial importance when it comes to eliminating those defects with the least amount of energy possible. For this paper, a simulation of the open die forging process was performed using the software Forge Nxt 2.0® by Transvalor. The model was recreated based on an actual fabricated squared bar, from which data such as porosity, die forces and thermocouple information was obtained and used to validate the initial results. The Niyama criterion was used to measure the reduction of porosity and was calibrated with the available experimental data for this alloy. In addition to this, true strain and damage were monitored to observe their relation with the porosity. Once the real process was accurately represented, both the die penetration depth and the initial temperature profile of the ingot were varied to optimize the effort required to meet the porosity criteria and the best forging strategy was defined. Next steps include simulation of the casting process of the ingot, with experimental data from our steel mill facility, to have a more accurate input data and to track the defect to its origin.

Presenter Name: Rodriguez, Pablo

Presenter Company: Ford Motor Company

Presentation Title: SAC Folding CAE Methodology

Submission Type: Presentation

Keywords: CAE, SAC, folding, simulation

Abstract:

Over the last few years, CAE simulations have been substantially improved as a result of the growing need to achieve full vehicle developments in a shorter time span while also attending the demand of cost reduction in such developments. One of the most critical components regarding the passive safety features of a vehicle is the Side Airbag Curtain. The need arises to involve this crucial component in an agile product development process. Consequently, when the validation using numerical methods of such component is performed, a full deployment of the airbag is needed to be evaluated and analyzed, having as a key objective the monitoring of its dynamic behavior caused by the effect of interacting with nearby components. In view of the foregoing, the folding process of the airbag plays a key factor in its whole operation. This study describes a methodology to fold a Side Airbag Curtain by means of a simulation-based routine, which can be defined on commercial CAE pre-processors currently available for the industry. This work aims to englobe the tools and steps followed in order to obtain, within a short period of time, a CAE model of the airbag, capable of representing efficiently and accurately a deployment, which might be used for analyses such as Interior Trim integrity and safe interaction. Using this CAE methodology, a new scope of problem solving techniques opens up. Applying the novel approach described in the preceding paragraph, a folding scenario could be useful to control the dynamics of the airbag in order to achieve a faster deployment in a certain zone, to avoid an undesired interaction with the interior trim of the vehicle, or to simply evaluate the aperture time of the system overall. All this adds up to a feasible cost reduction alternative to the most common techniques that involve modifying and adapting geometries including supplementary components, that impact directly in the prime cost of a vehicle.

Presenter Name: Saavedra, Oscar

Presenter Company: Ford Motor Company

Presentation Title: Statistic Modelling Approach for Front Low Speed Impact

Submission Type: Presentation

Keywords: Data miner; FEA simulation; low speed impact; machine learning; statistical methodologies

Abstract:

Due to the complexity of simulating multiple FEA configurations and limited resources, we began an exercise using a statistical methodology to identify physical test approach based on virtual crash testing for Energy Absorber design. The complexity of the design of this component lies in the interaction of multiple parts that are responsible for the dissipation of energy. In addition to this, the system must meet the opposite tests in terms of rigidity. Initially, the baseline model was correlated against physical testing then, a DOE with several inputs for the base model was performed providing variability in results, and those were used as raw data. Monitoring all outputs allow us to use dimensionality reduction for the inputs to improve the performance of the data mining. Being this the starting point, we utilized various techniques applied for data miner commonly applied for linear regressions, neuronal networks, and decision trees. Approaches combine several machine learning techniques into the predictive model for decrease the variance (bagging), bias (boosting) to improve the prediction, however, the results were not as expected due the nature of the data (continuous) and inaccurate mining algorithm. Stepping back, using linear regression, neural networks, and decision trees mining algorithms with raw data, the prediction model was built and lead us to obtain a mean predicted which has a statistical representation around 90% of fidelity by applying using three inputs and one output at each time. This prediction was verified with the prediction model. Being these results as a starting point we need to add new specialized learning systems with different environments to consider more design variables, for example, materials, geometric changes, among others This whole study is focused on simulations of impact at low speed but considering the potential of statistical algorithms and building a broad-spectrum predictive model can be used to support other areas focused on simulation.

Presenter Name: Santoni, Nelson

Presenter Company: Ford Motor Company

Presentation Title: Influence of Second-Shot Process Conditions on the Warpage Behavior of a Two-Shot Overmolded Automotive Pillar

Submission Type: Presentation

Keywords: Plastic injection Molding, Overmolding, Warpage, Moldflow, Taguchi, ANOVA

Abstract:

Warpage is a common and unavoidable flaw of injection-molded parts, mainly associated with non-uniform shrinkage phenomena, which is inherent of the macromolecular structure of polymeric materials. This defect is influenced, in more or less extent, by part/tool design, material properties and injection-molding process conditions. The control of these parameters becomes important towards warpage minimization to improve part quality and dimensional stability, factors of capital importance to fulfil tolerance requirements in assemblies. Plastic parts with low warpage are a constant goal for plastic transformers, who look for practical and feasible ways to manage warpage issues. Computer-aided Engineering (CAE) software Autodesk Moldflow Insight can be used to simulate different scenarios of injection molding process, in order to minimize warpage in plastic parts. Warpage control becomes more challenging when dual-shot overmolding is used. This process is a variant of injection molding process that allows a secondary material to be injected partially or fully over another material previously injected (known as substrate or polymeric insert) to manufacture one single part. This is required when different properties are required (mechanical, ergonomical and aesthetical as the most important properties). In the case of overmolded plastic parts, the warpage behavior becomes more complex, due to the interaction of two plastic materials with different properties, each one with its own processing history. In this study, a 3D mesh model of a dual-shot automotive pillar was analysed. The first shot, considered as a polymeric insert for the purposes of this study, was simulated with Poly(carbonate)/Acrylonitrile-Butadiene-Styrene (PC/ABS), while the second shot was Poly(methylmetacrylate) (PMMA). The 3D mesh model was made considering the same mesh on shared surfaces between both shots. A 5-level 6-factor 25-experiments (L25 56) Taguchi orthogonal array design of experiment (DOE) was used to analyse the warpage dependence on second shot molding conditions. The molding conditions considered were: melt temperature, mold temperature, packing pressure, packing time and cooling time. In the DOE was also considered the temperature of the plastic insert as a factor. The experiments suggested by DOE were simulated in Moldflow to obtain warpage results as an output. An analysis of variance (ANOVA) was used to statistically find the relative effect of studied process parameters on warpage.

Presenter Name: Singer, Ricardo

Presenter Company: Ford Motor Company

Presentation Title: Multiobjective CAE Model Homologation

Submission Type: Presentation

Keywords: Implicit, explicit, thermomechanical, multiobjective, static, dynamic, multiphysics, frequency, requirements, compliance, multitarget

Abstract:

Over the past years it has being a common practice and sometimes almost a workflow rule to create a dedicated CAE model for a specific type of analysis, hence it involves the construction of several Finite Element Analysis setups, each one with a different finite element model, therefore, implying over simplifications, many assumptions, under or over constraining, missing interactions, missing contacts between parts, not representing exactly the physical test set up, etc., all the before mentioned circumstances lead to very hard to correlate models, that in most of the cases the simulations performs with a low or a medium/low confidence level, as well as being a highly time consuming process since it implies to obtain each separate analysis result from each type of dedicated model. This situation remains due to the complexity required to couple the mathematical theories by which the Finite Element Method is build, in which many cases are incompatible. It is a need in the professional environment to reduce costs, deliverables time and to correctly represent the physical behaviour of the system to achieve good correlations. The present paper introduces a modelling construction technic that allows to greatly improve time and performance of the simulations due to the creation of one complete homologated finite element control model. The difficulties in the homologation processes were passed, by carefully coupling the Explicit and Implicit finite element mathematical solving methodologies, allowing the interaction between every type of entity (shell elements, solid elements, beams, springs, masses, spotwelds, weldlines, etc), the elements formulations interaction, considering all the contact definitions between them, and the precise selection of the material laws including user defined material laws, that can describe the physical behaviour in both type of solving methods. This modelling technique as before mentioned was applied in one single control model without any simplification nor assumption (every vehicle component including attachments, rubbers, joints, bolts, spotwelds, fixtures, adhesives, glues, etc.) from which eleven different types of analysis were executed at the same time, among them: modal analysis, low speed damageability, step pad, customer loads, isolated thermomechanical, RCAR and thermomechanical plus pressure loading and gravity sag; all simulations conducing to satisfying results reaching medium to high confidence correlation levels, it also allows to perform multitarget or multiobjective analysis to comply between different requirements at the same time. Some metrics within the results are the following: ‘Step pad’ simulation reached in most cases above 85% correlation confidence level, in some cases the measured displacements differ by tenths of millimetre. ‘Firmness and feel’ simulation increases to medium high and high confidence levels, meaning simulations can predict physical results in a range from 70% to 95% of confidence. ‘Customer load appliance’ simulation reached a medium to high 83% of confidence. ‘Low speed damageability’ in frontal impact reached a high confidence level with 95% of certainty. Homologation in one control model allowed decreasing the payed software licenses from 3 to 1 which significantly reduces error generation and error inherit, concurrently reducing engineering time since it is no need to transform models between software element definitions and fix the loose of contacts, weldings, materials, attachments, etc. Initially 4 engineers were required to perform three different types of analysis, currently only 1 engineer is able to perform eleven different types of analysis running in explicit, implicit or thermomechanical methods according to each type of analysis with the before mentioned homologated control model, and the study definition (model set up) was reduced from taking three days down to four hours.

Presenter Name: Sithambaram, Sasi

Presenter Company: SOLIDWORKS Corp.

Presentation Title: Leveraging Simulation to Optimize Design for 3D Printing

Submission Type: Presentation

Keywords: 3D Printing, Additive Manufacturing, Topology Optimization, Lightweight Designs, Generative Design

Abstract:

Design and Manufacturing is changing in this new era of industry renaissance. The engineering companies are becoming more innovative and creative with their products. Competition is stronger than ever before and so a fundamental transformation in design and manufacture is now on the horizon. Additive Manufacturing (AM) is maturing fast enough to replace traditional manufacturing techniques. 3D Printing technology is now a viable means of custom production, shaping up the future of manufacturing across many industries. As industries begin to adopt AM, some key elements of traditional manufacturing are being transformed – beginning with the basics of how parts are designed. 3D Printing enables creation of not only complex and organic shapes, but provides a method of assembly part consolidation. While this provides the desired design freedom, the many variables involved makes this process challenging to design the most optimized shape. Furthermore, addressing product stiffness, lightweight designs, real life performance considerations and identifying the best partner to go from design to manufactured parts, is key to seeing AM as a valuable tool for overall success of your business. This paper aims to address some of these key challenges. An example model is used to discuss workflows and processes to put a sketch idea from concept to a 3D printed finished part. To illustrate this, the paper will examine how to optimize 3D CAD part design for 3D printers. This involves the discussing the main requirements for the design process and compete workflow to finalize the design for 3D printing. Once an initial design is drafted, ideal ways to use design validation tools such as topology optimization and linear static analysis will be examined to explore 3D modelling design for AM. After the pre-final design is established using design validation tools, further simulation analysis such as Fatigue, Non Linear and Vibration will be performed to predict and have confidence in overall performance of the product in operation. Once the final design is approved based on the analysis results, the paper will conclude on methods to digitize the part to production process.

Presenter Name: Thomas, Francisco

Presenter Company: Open iT, Inc.

Presentation Title: How to Make The Most Of Your Analysis and Simulation Applications

Submission Type: Presentation

Keywords: Engineering software, Analysis and simulation, Optimization, Reduce software costs, Cost allocation, Usage metering

Abstract:

The engineering industry has advanced with the use of analysis and simulation software. Engineering companies can optimize the entire design, prototype, test and production of bringing a new product to market with these software applications. Access to the latest and most innovative software tools is sometimes difficult due to the overwhelming costs of these packages. In today's session, we will not focus on optimizing designs but more specifically, optimizing the software licenses to enable: reduced software costs, identifying unused software to free up budget to be spent on new apps and features, understanding cost allocation by actual usage by department or project and understanding usage for vendor negotiations. The best way to achieve optimization of your expensive licenses is through usage metering tools. Software usage tools measure how much and how often applications are used and allow managers to quickly and easily analyze true needs. Some usage metering tools also go further by not only providing historical data but by simulating various types of agreements and scenarios to help managers make more informed decisions. This presentation will provide real engineering customer data, report examples as well as case studies where usage metering saved significant money. Business and IT Managers are applying usage data to optimize their resources and save their companies real dollars while creating a competitive edge. Whether you're managing analysis or other simulation applications, usage metering allows you to provide the right software to the right person at the right time thereby optimizing your costs.

Presenter Name: Toledo Gonzalez, Juan Pablo

Presenter Company: COMPLX

Presentation Title: Conjugated Heat Transfer Analysis of and Electric Transformer to Determine the Temperature Behaviour of Leads

Submission Type: Presentation

Keywords: CHT, CFD, Heat Transfer, Electric Transformer, Leads, Radiation, FEM

Abstract:

Dry electric transformers are key components on many industries. They are mainly used as step-up or step-down applications to supply a specific voltage to machinery when a source that provides that specific voltage is not found where the device would be installed. Temperature is one of the main limiting variables for the transformer capacity. The winding part of the transformer has been analysed extensively over the past decades and many formulations have been developed in order to calculate, within an acceptable margin of error, the winding temperature. However, little has been done in terms of modelling the temperature behaviour of other transformer components that also affects the efficiency of the system. In the present case, a Conjugated Heat Transfer (CHT) analysis, using AUTODESK CFD®, is performed to a complete 3D dry electric transformer in order to obtain the temperature distribution in the internal components, more specific, in the leads. The results were meant to be compared with lab measurement values. A 60 million elements model was created. The analysis takes into account conduction, convection and radiation within the internal components as well as the environment. The definition of the turbulence model, buoyancy forces, mesh density and wall formulation were the primary concern for the model validation. The temperature results for the leads, obtained in the model for a steady state condition, yield an error of ± 1.5 °C within the laboratory conditions. The model provided a good agreement between the winding temperature prediction and the thermal performance of the coils as a function of the average temperature and hot-spots. The main difficulty that arise when performing a CFD analysis, is the validation part. However, with the current fabrication practice, mapping the variables that could decrease the mechanism efficiency is part of the process. The later helped the engineering team to create a virtual prototype an validate a specific model, which in turn can be modified and yet still obtain results without the need of intermediate validations. The validated model will help develop new solutions in order to control the temperature within the leads.

Presenter Name: Tristán-López, Agustín

Presenter Company: Instituto de Evaluación e Ingeniería Avanzada

Presentation Title: Modal Analysis and Fatigue for Bus Structure

Submission Type: Presentation

Keywords: Boundary conditions, bus, computational laboratory, cumulative damage, fatigue, modal analysis, modal damping, natural frequencies, vehicle

Abstract:

The finite element analysis (FEA) can be used not only to calculate the stress and strain distribution in a structure or a mechanical part, it can also be a computational lab when experimental results are not available due to cost, sustainability or schedule conditions. It is a suitable method for static, thermo-mechanics or modal analysis and it also helps to solve inverse problems where boundary conditions are not previously known, in particular when there is a lack of information of some mechanical parts and also when the structure is only at the stage of design. Limitations of budget and time defined the need of the computational lab to study the structure of a passenger bus from a Mexican factory. The finite elements library of a commercial software permits to simulate beams and plates of the structure, as well as the elastic properties of the materials. Despite string and dampers are available in the library, the properties of the shock absorbers were unknown, in fact, the catalogue of the producer does not provide technical data of the shock absorbers to be used as input for FEA and the commercial representatives do not have this kind of information. The computational lab helps to solve this problem, simulating the behaviour of the vehicle under several boundary conditions of modal damping and elastic properties for the shock absorbers, prescribed as properties of the elements in the FEA software. The computational lab is used when no experimental results are available, then it is a theoretical analysis based on the designer's previous experiences from other projects where experimental results were available. A detailed explanation will be available in the final presentation. A convergent series of more than 50 iterations of modal simulations did provide a final reasonable set of values for the damping and string elements. With these parameters it was possible to calculate modal shapes and frequencies as well as the worse stress conditions for the vehicle structure under static and dynamic conditions. The FEA modal dynamics simulation of the new vehicle provided the response under various dynamic conditions. It is possible to compare the results of the new structure to those from previous vehicles where experimental modal analysis were available. Some differences between the new vehicle and previous ones were found, in particular in the range of frequencies. To reduce these discrepancies some more iterations were needed. The final model was used to simulate impact dynamics. The resultant principal stress state was used as input in fatigue analysis according to a cumulative damage model. The structure was redesigned for the beams and structural supports if: (a) Von Mises criterion exceeded yield stress or (b) fatigue simulation provided a finite life. This project is based on the experience in modal analysis and fatigue since 1989, both in experimental and computational techniques. The buses are actually in use in Mexico.

Presenter Name: Vidriales Escobar, Luis

Presenter Company: GE Aviation

Presentation Title: Finite Element Analysis Challenges in External Configuration Hardware

Submission Type: Presentation

Keywords:

Abstract:

By 2018, GE Aviation had more than 33,000 engines in service, being the world leader in jet engine manufacturing. GE Aviation has manufacturing, engineering and service business units spread around the world, being GEIQ the engineering design campus in Mexico. Among several engineering services GEIQ offers to GE Aviation, there is rotor and turbine life assessment and External Hardware ownership and design, just to mention a few. This paper summarizes the challenges that GEIQ faces when analyzes such external components by Low-Cycle and High Cycle fatigue loads, elastoplastic loads such Fan Blade Out or Burst Pressure, buckling, and the validation process from a simple modal analysis to ping/engine test. It also walks through the FEA evolution from the reliable CFM56 to LEAP and how that 30% efficiency improvement was possible, and gives an insight about some tools that are going to be used in the future such as additive manufacturing and design optimization.

Presenter Name: Yang, Zane

Presenter Company: Valeo-Kapec NA

Presentation Title: Using Explicit Finite Element Code to Simulate Riveting Process

Submission Type: Presentation

Keywords: FEA, explicit code, riveting, large plasticity, upsetting load, residual stress

Abstract:

Using rivets to join work pieces (or plates) together is still very popular in automotive industry, due to its easy setup, low cost in capital investment. A riveted assembly is completed when a pre-formed rivet is upset on its tail with to-be-joined plates being securely sandwiched in between. Although upsetting the rivet has been used for very long time in production lines, Numerically simulating its behavior has not been brought about until recently because of many challenges involved. One of challenges is that the rivet material is severely undergoing large plastic deformation as the tail of rivet quickly flows to be shaped into a rivet head. Presented in this work will be a riveting simulation with the use of an explicit finite element code generally available in the market. The model for the simulation is 3D-based in order to make it more versatile in a variety of riveted assembly. Rather than a single rivet to join multiple plates face-to-face (without any intentional gaps), a spacer-type rivet will be used, as an example in this simulation, to joint four plates with 2 inner plates being spaced at a given distance. Thus, instead of upsetting each rivet tail to form two heads one by one, we have simulated a single process such that two shop heads are formed simultaneously to cut the production time by more than half. Basically, as the moving press head pushes one tail of rivet with the other being supported on the stationary base, both rivet heads are actually formed. The rivet shanks at both end do not have to be equal in diameter or length to be able to form typical heads, even though the forces applied at the one tail must be equal to the force reacted at the other. The blind holes at the center of shanks are found to be effective in tuning both rivet heads to their desired sizes, as will be demonstrated in this simulation work. The required force to upset a rivet joint can be easily available as the result of simulation, which certainly becomes a important piece of info for planning in manufacturing plant. By the means of simulation, we can easily and quickly assess the amount of material filling the radial gap originally existing between the shank and hole. There is no question that the stress and strain state on the rivet and the work pieces (or plates) can be displayed to identify the potential troublesome areas susceptible for cracks during the riveting. More importantly, the stress that remains within a rivet or work piece, or residual stress, is very vital for their continued capability to handle service loads, once the joined assembly is put in service. In short, virtual riveting through simulation can be an important arena of FEA simulation to make our simulation more valuable to product/process design and integration.