9th

BEFORE REALITY

CONFERENCE

June 14 - 16, 2023

Science Congress Center
Munich, Germany

book of abstracts

BETA
SIMULATION SOLUTIONS


Contents

Live Demonstrations


Abstracts

Session 1

The evolution of BETA CAE Systems portfolio in a nutshell

Chryssa Sferidou
BETA CAE Systems

KEYWORDS -
ANSA, EPILYSIS, META, KOMVOS, SPDRM, RETOMO, NEERE, DEEN, ANSERS, FATIQ, ML, SPH

ABSTRACT
Welcome to a selection of highlights, reflecting the evolution of our software portfolio. During this presentation we will walk you through the progress that our software has made and we will guide you through the respective presentations to be given during our event.

Building simulation capabilities in Rimac Technology

Kristina Šarović
Rimac - Technology

KEYWORDS -
FEA, Mesh generation, ANSA/META

ABSTRACT
The Simulations Department, as a part of the Components R&D at Rimac Technology, has been an integral part of the development of electric powertrain and battery systems, providing virtual prototyping capabilities for fast and accurate design insights through calculations.

Over the years, our team of experts has grown and we have continuously expanded our capabilities, databases, and toolsets. As a result, we have gone beyond our initial role of virtual validation and are now fully integrated into the R&D processes of the company, conducting material testing, validation, and verification, among other tasks.

Recently, we implemented BETA CAE software, ANSA and META, into our workflow, which has transformed our capabilities even further. By using the same pre/post-processor for different solvers and applications, we can collaborate more efficiently both within and outside the department. The advanced and customized toolsets in ANSA have also allowed us to create high-quality meshes and models, improving the accuracy of our simulations. In this presentation, we will discuss the implementation of ANSA and META and its impact on our simulation capabilities, as well as our future plans for continued growth and development in this field.

Session 2

Future directions of BETA CAE Systems product line

Dimitrios Siskos
BETA CAE Systems

KEYWORDS -

ABSTRACT
In this presentation we will talk more about our action plan for fulfilling our vision to reduce CAE turn-around time.

A new user experience in ANSA

Dimitrios Zafeiropoulos
BETA CAE Systems

KEYWORDS -
Topo, Mesh, Morph, Toolsets, Direct

ABSTRACT
Since ANSA version 22, major UI/UX design changes staring to happen, introducing new approaches for Geometry and Mesh handling. The user has available more direct and workflow oriented functionality with a clean and modern look, uniform all over the newly designed functions. The enhancements in productivity are great making ANSA easier to learn and faster to use. The classic Topo-Mesh modules will eventually become obsolete.

But the changes does not stop there, they also focus on the Analyst. The functionality in Deck Modules has been extensively revised leading to the introduction of the Analysis Tools menu, following by significant changes in Database Browser, which have greatly enhanced access to both solver data and associated functionality.

This presentation is a glance to all the changes that we made so far, focusing on what we changed and why, what we achieved so far and where we are heading to

Development processes with seamless CAD integration in ANSA and DCM

[1]Bastian Näser*, [2]Michael Tryfonidis
[1]BMW Group,
[2]BETA CAE Systems

KEYWORDS -
pre-processing, workflow, ANSA

ABSTRACT
Fast development processes are becoming increasingly important for project success, especially in volatile industries such as the automotive industry. To meet time-to-market requirements, the whole development process must be accelerated. This also applies to the simulation tool chain as the main part of the development process. In addition to pre-processing, solving and post-processing, efficient access to the geometry data (CAD data and metadata) is important, especially for large and complex products as full vehicle models for passive safety simulation.

To address this, the CAD4CAE interface was developed at BMW to provide a two-stage workflow. In the first step, the user can query the product structure of an entire or a subset of the vehicle. This structure is enriched with metadata for positioning, material, 3D preview and much more. With this lightweight information, the user can reorganize the entire structure to meet the needs of the simulation model. In a second step, the full CAD geometry is requested for the subsequent pre-processing steps such as defeaturing, midsurfaceing, meshing, etc.

In the presentation we demonstrate the seamless integration of the CAD4CAE interface in ANSA and DCM. We show how ANSA and DCM interpret the metadata to support user decisions. In addition, we will emphasize the generic approach to meet the diverse needs of different simulation disciplines to provide all users with an easy-to-use tool.

Session 3A

Automation of ANSA and META to evaluate occupant performance and seat structure for robustness

Hemant Shettar*, Yatin Kumbhar, Santosh Patil
BETA CAE Systems USA

KEYWORDS -
Safety, Kinetics, Morphing, DOE, Python

ABSTRACT
The ever-increasing power of computation and advancements in pre and postprocessors is making virtual crash test simulations more predictable. However, to produce robust results, virtual simulations must consider uncertainties of test condition, manufacturing deviations and their probabilistic effects upon the performance of the overall model.

This paper illustrates the process of setting up a DOE model using ANSA and META automation capabilities, considering the deviation caused in testing and manufacture. Here LS DYNA is used as the solver.

Two separate load cases of the same seat were built to evaluate the Robustness of:

1. Occupant performance by considering the test deviation in H point and back angle caused while dummy positioning .

2. Seat structure integrity by considering the manufacturing deviations in weld line positions.

Changes in H point and back angle of dummy at every variation creates a repetitive task to redo the dummy positioning, belt fitting and seat squash.

The altered positions of the dummy, belt and seat foam were accomplished by integrating the DOE process with dummy articulation, seat belt and seat squash tools and the required design variables were generated. Also, the ANSA Kinetic Configuration and KIN_POSITION tools were integrated with the aid of python scripting, and ultimately a solver ready model is generated. The deviations of the above parameters were evaluated on Dummy occupant injury numbers using Meta Post. Similar challenges were accompanied when modelling variation in weld line positions that is to position and realize the weld line at the defined location.

By Integrating ANSA morph DFM and box morphing technique with DOE process the required design variables and solver ready models were generated. Finally, the Seat Structure integrity was evaluated for given parameters using Meta Post as postprocessor.

Bulk positioning tool for impactors and barriers

[1]Sebastian Kreissl, [1]Dustin Draper*, [2]Berkay Atici
[1]BMW Group,
[2]ARRK Engineering GmbH

KEYWORDS -
Pedestrian Protection, Low-Speed-Crash, Positioning, Automation

ABSTRACT
With the goal of having faster car development processes, the need for automation and standardization increases. The design of a vehicle exterior has a strong interdependence with pedestrian protection and low-speed-crash load cases, a good example of which is the so-called bumper-to-bumper accident.

Pedestrian protection simulation, with its large number of required evaluations at different locations and with different vehicle configurations is particularly dependent on efficient pre- and postprocessing tools. For instance, to ensure that a new bonnet design satisfies legal requirements for head impact, usually several hundred separate finite element simulations are necessary. Besides head impact, pedestrian protection has requirements regarding lower and upper leg load cases that need to be satisfied, in addition to consumer rating (e.g., NCAP) goals. Similarly, load-speed-crash simulations require positioning different barriers (RCAR Bumper/Structure, US Part 581 and ECE R42 pendulums) for multiple vehicle configurations relative to the car as part of the preprocessing process. For a new exterior design, generally more 1000 separate finite element simulations are necessary.

Together with BETA Systems BMW has developed a tool that allows for a fully automated positioning process for both pedestrian protection and low-speed crash simulations. As inputs the tool takes a CAD file commonly used by OEMs, and finite element models of the vehicle to be evaluated, and then creates the parameters required for accurately positioning all the required impactors and barriers. What used to be a manual process for every load case separately, can now be carried out in one continuous process for all load cases. The high level of automation allows for significant time savings and since no user-interaction is required, standardized results and a reduced risk for errors.

[ONLINE] Cutting-edge FMVSS201U targeting and head positioning at General Motors

[1]Bingfeng Fan*, [1]Liwu Zhou, [1]Keshavlal Rathi, [2]Thanassis Lioras, [2]Pavlos Katsonis, [2]Thanassis Fokylidis
[1]General Motors Company,
[2]BETA CAE Systems

KEYWORDS -
FMVSS201U, Upper Interior Head Impact, ANSA

ABSTRACT
The interior safety properties of automobiles have been subject to increasingly stricter requirements under occupant protection legislation. The US laboratory test procedure known as FMVSS201U specifically focuses on head protection during impact with the upper interior components, and requires identification of critical targets, upper roof zone, and headform impact angles. To reduce the need for physical tests, manufacturers use CAE software for product performance evaluations as much as possible. However, due to the sheer number of potential load cases, an infinite number of simulations would be necessary to cover all possibilities. As a reuslt flexible tools are needed that not only allow the analysts to automate their processes, but also enable them to intervene at any stage, make modifications, and assess their impact.

General Motors, in collaboration with BETA CAE Systems, has upgraded their target marking process according to FMVSS201U. This has resulted in the achievement of fully automated identification of the desired targets in the upper interiors, as well as the ability to intervene and make modifications to auxiliary data at any step of the process. This is applicable in both FE and CAD data building, bridging the gap between analysts and designers. Additionally, a detailed and robust positioning process has been established to cover all the demanding specifications of the protocol. Finally, ready-to-run load cases for LS-DYNA are created in bulk for all target points.

The presentation provides a comprehensive explanation of the process and showcases how ANSA has met the needs of General Motors in their FMVSS201U process. The presentation also includes metrics that demonstrate the correlation between testing and simulations, offering insight into the added value that BETA and General Motors have gained from their collaboration on the ANSA tool.

[ONLINE] Virtual evaluation of system connectivity

Christopher Raprager
General Motors

KEYWORDS -
Connections, Process Automation, Python

ABSTRACT
During the model build process validation of subsystem connectivity is a large challenge. Of particular importance is determining where models are missing connectivity. We will discuss the particular difficulties to check for the missing condition as well as historic and modern solutions to this problem utilizing Ansa python API.

Session 3B

Handling complex parameterization using ANSA for structural performance optimization

[1]Sudeep Chavare*, [2]Florin Morar, [1]Eric Gowen, [2]Ravi Nimbalkar, [2]Yatin Kumbhar
[1]General Motors,
[2]BETA CAE Systems USA,

KEYWORDS -
Morphing, Optimization, Parameterization

ABSTRACT

Utilization of various morphing and parameterization techniques for design optimization is a common practice in product development. Often these parameters deal with changing height, width, shape, and size of various structural components. However, the drive towards zero prototypes and accelerating technological advances make it necessary for the CAE analysts to be able to rapidly set up complex parameters to carry out structural performance optimization for ever-evolving designs. These complex parameters are beyond simplistic changes and need sophisticated integration of software capabilities and automation to carry them out. In order to effectively address these issues, this paper presents a framework for creating rapid parametric optimization setup using ANSA and Python scripting automation. The concept and the automated parametrization process are illustrated using a benchmark study carried out by General Motors.

The benchmark study is carried out using an open-source Chevy Silverado truck model. The objective of the study is to minimize weight of the vehicle while maintaining the baseline performance criteria for the stiffness and modal performance using NVH load cases. The parameterization for design shape changes is setup using ANSA morphing and automation capabilities. ANSA Optimization Tool is used for organizing and running the optimization study using directly created Response Surface Models (RSM). The results of the study as well as the complex parameterization and responses extraction techniques using ANSA and META will be illustrated in detail.

Implementing an automated process for updating CATIA files for optimization purposes

[1]Isabella Gavarini*, [2]Zafeiria Kanellia, [2]Spyros Tzamtzis, George Pentsas, [2]Michael Tryfonidis
[1]BMW,
[2]BETA CAE Systems

KEYWORDS -
ANSA, CATIA, SPDRM. optimization, parametrization, geometry update, design table, batch meshing, automation

ABSTRACT
Achieving optimum performance for car components during multiple and complex crash scenarios, requires that simulation engineers examine various design aspects of component geometry. This is efficiently accomplished by identifying some of the parameters transferred from CAD files into the simulation model as design parameters and by varying them in a Design of Experiment (DOE). The design parameters are updated in each experiment and the task of transferring them back from the FE-environment to the CAD software has proven to be a challenging process.

While demanding and time consuming, this step is mandatory as the design parameters between simulation and CAD need to be aligned to allow automatic component design update and subsequent optimization to proceed.

This presentation shows a case at BMW of an automated process that achieves this initial update and transfer of design parameters from simulation to CAD software. This is accomplished through a process workflow that ensures the interoperability between different os-platforms.

Closing the Optimization cycle: incorporating CAD-design parameters

Michail Tryfonidis*, Zafeiria Kanellia, Spyros Tzamtzis
BETA CAE Systems

KEYWORDS -
Optimization, Catia/NX parametrization, Workflow management, Design Parameters

ABSTRACT
Improving the CAE-function of car components from a crash simulation perspective is a challenge on its own, due to the high non-linear effects that are taking place. With such pre-processing capabilities like ANSA morphing, a whole set of capabilities is available to the CAE-Engineer to investigate the shape of the geometry so and to understand how it affect its crash performance and through this knowledge to proceed in improving it. Nevertheless, in the automotive industry, the CAD and CAE departments often face communication challenges, especially from CAE back to CAD. There are obvious fundamental differences in the processes, tools and objectives used by each department, working independently of each other.

In this paper, we will go through a scenario of dealing with exactly the above, understanding and affect the CAE-Function of a car-crash simulation case by manipulating the geometrical features of the frontal rails. In order to understand how the shape of parts affects the function, Optimization studies are performed to analyze this behavior. The CAD-Design result that follows the new design change proposal is equipped with native CAD-design parameters, which allow the geometry manipulation with respect to design or manufacturing constraints. While at the same time, they enable to continue and finalize the optimization study by incorporating these true design parameters into the optimization tools and affect them directly.

The closing of the circle is achieved with an automated process that achieves the initial transfer and update of design parameters from the CAD design by ANSA toolchain. A streamlined process workflow ensures interoperability between different operating systems, making this transfer of design information efficient, ultimately leading to quicker and more accurate design updates, reducing the time and cost associated with the design and development of car components.

Latest and future developments in the Optimization Tool

Eva Ioannou
BETA CAE Systems

KEYWORDS -
Optimization Tool, DoE

ABSTRACT
Discover the latest developments of the Optimization Tool that facilitate the investigation of various model designs to improve the model's efficiency. The DoE and Optimization algorithms have been enhanced with more options, while various tools like the parallel jobs execution, the optimization log and advanced charts, among others, upgrade the user experience. Moreover, learn about the upcoming developments that will broaden even more the Optimization Tool capabilities.

Latest and future developments in Morphing and Design Toolbar

Serafim Chatzimoisiadis
BETA CAE Systems

KEYWORDS -
Topo, Mesh, Morph

ABSTRACT
In the area of Concept and Detail Design via Morphing, several new Tools and many more improvements have been released in the latest versions.The constant requirement of the CAE community of acquiring convenient tools and methods for interactive and parametric design is now met. The analyst is now able to apply minor oρ major design changes, like the creation of members, reinforcements, joints, Beads etc, without the assistance of the designers. BETA CAE Systems devoted time and effort on several tools of this area and more are about to come in the near future. The new functionality along with several relevant tools, are gathered under a new Toolbar called Design. In parallel, the Morphing Tool is continuously improving providing more specialized, precise and user-friendly methods. Contact definitions, Steady Section maintenance, spotweld density and weld chains handling, during Morphing are only some of the latest Developments.

Session 3C

Reintroducing KOMVOS as a simulation data and process management desktop client

Irene Makropoulou
BETA CAE Systems

KEYWORDS -
SPDRM, Process Management, DM

ABSTRACT
KOMVOS - SDM Console, a component of the BETA Suite, was originally introduced as an interactive console for browsing, viewing and handling CAE data. Initially focused on streamlining data handling processes for CAE engineers, KOMVOS facilitated tasks ranging from interfacing with PDM/PLM systems to the creation of Subsystems and Simulation Models.

In recent years, KOMVOS has undergone a transformative evolution by integrating process and data management functionality from SPDRM. This integration, combined with KOMVOS's already robust features, has resulted in an unparalleled solution for Process and Data Management.

This presentation reintroduces KOMVOS, now as a Simulation Data and Process Management desktop client.

Unlocking the power of process management with SPDRM

Spyros Tzamtzis
BETA CAE Systems

KEYWORDS -
HPC submission, job scheduling, notifications, execution on remote resources,

ABSTRACT
In the CAE community, the term Process Management is often used to refer to the standardization and automation of CAE workflows. Usually more focus is placed on the pre- and pro-processing workflows and their automation through Python scripts, session files and task management tools that make use of local resources to carry out the programmed tasks. But do these tools cover all the needs of contemporary engineering teams?

Whenever Process Management requirements cross the boundaries of different systems or need to make use of remote resources for the execution of some resource-demanding tasks or even need to employ task schedulers in order to program the execution of certain tasks, traditional Process Management tools within the pre- and post-processors offer limited options.

This work presents the advanced Process Management features of SPDRM that can take the standard process management of ANSA and META to the next level with the integration of state-of-the-art tools for job submission and monitoring, task scheduling and task delegation on remote resources.

Simulations across borders: using SPDRM's multi-site collaboration to unite global teams

Menelaos Pappas
BETA CAE Systems

KEYWORDS -
multi-site architecture, cloud integration, AWS, file transfer optimization

ABSTRACT
In today's interconnected world, the ability to effectively collaborate across borders has become essential for global teams. For CAE teams, the challenge lies in efficiently coordinating and sharing small, medium and large-scale simulation data among geographically dispersed teams.

This presentation explores the innovative solutions offered by SPDRM to enable multi-site collaboration for global teams.

Streamlining data extraction from Teamcenter: latest approaches to CAE Structure creation and maintenance

Ioannis Haralampidis
BETA CAE Systems

KEYWORDS -
Model Build, DM

ABSTRACT
Creating a robust and effective bridge between design (PDM/PLM systems) and simulation (SDM systems/CAE tools) is a longstanding challenge for CAE engineers. The difficulties mainly stem from the fact that fully utilizing available tools requires substantial expertise in both domains. Additionally, the differences in modularization between CAD models and the CAE subsystems necessitate a mapping process between the CAD BOM and the CAE subsystems, which is an additional challenge for the CAE teams.

This presentation highlights recent advancements in integrating BETA Suite products with Teamcenter, offering a user-friendly and efficient gateway to CAD structures directly from within CAE tools. It also explores the utilization of template-driven methods for creating and maintaining CAE structures. These new approaches contribute to streamlining the data extraction process and enhancing the overall efficiency of CAE workflows.

Session 3D

Mesh automation of runner systems for injection molding simulations

Christoph Mau
LEGO System A/S

KEYWORDS -
Injection moulding, Automation, Batch Mesh

ABSTRACT
The simulation of the injection moulding process plays an important role in the product development cycle of LEGO bricks. Simulations are used to fulfil the high-quality demands and standards for the products. The simulation of the filling sequence of the runner system and cavities serves the purpose to analyse the filling pattern, pressure drop and temperature distribution in the melt to ensure that all elements in the mould have similar structural and functional properties and fulfil the visual quality and product safety requirements.

There are a few hundred runner simulations carried out per year and a major issue in the process is a reliable automation mesh generation protocol. The previous setup for these simulations required a lot of manual work and time due to lack of efficient automation functionalities. To alleviate this situation, increase flexibility, reduce manual work and lead time for runner simulations, ANSA was introduced as a new mesh tool at LEGO.

Automated tools via Python scripts (Batch Mesh, mesh scenarios, etc), allow LEGO to use the 3D geometry of the runner system without oversimplifying it and to completely automate the mesh generation and facilitate the creation of a ready to run solver file. After the automation implementation, as a result, the pre-processing time for a simulation was reduced from several hours to a few minutes while meeting all solver quality criteria and various challenges like large number of layers.

The protocol and methods used to automate the mesh procedure will be presented in this article/session.

Adopt injection molding manufacturing effects to improve parts' crashworthiness

[1]Panagiotis Fotopoulos*, [2]Michael Richter*
[1]BETA CAE Systems,
[2]MATFEM Ingenieurgesellshaft mbH

KEYWORDS -
Crash worthiness, Product development, Material model MF GenYld+CrachFEM, Integrative simulation/Mapping, Fiber orientation, Weld lines

ABSTRACT
In the recent years, the pursuit of lightweight products is an important objective. Specifically in the automotive industry an increasing number of parts is replaced by fiber reinforced plastic materials in order to meet the demands with respect to lightweight and safety, independently of the propulsion type. This poses further challenges for the industrial crashworthiness as well as pedestrian safety simulations of larger components with locally varying mechanical properties caused by the injection molding manufacturing process.

The above ignites the following process loop: (i) injection molding simulation, (ii) transfer the injection results to the structural mesh for a high-quality material description including the consideration of process information (Mapping) and continue with (iii) post-process analysis. The afore mentioned process is fully covered by the harmonic connection of ANSA – modular material model MF GenYld+CrachFEM (CrachFEM) – META software products.

It is remarkable that the above solution suits as well to early design stages of product development where the design is rough and a full injection molding analysis with the standard solvers are costly in time and money. ANSA one-step molding solution provides an extremely easy setup on the currently working model (and mesh), whereas the result will be ready unexpectedly fast, without any extra knowledge, expertise or training requirements.

This minimum cost in resources opens the door to do many ‘what if’ studies or optimization loops, which can be coupled with the powerful Morphing tool of ANSA.

To cover cases of injection molding results coming from third-party software like MOLDFLOW and Moldex3D, the advanced tool of mapping results can wisely map the orientation of fibers through the thickness of the part. Furthermore, the weld lines are also transferred and handled within the target model and finally all this information is formed and exported to CrachFEM readable format.

The material model CrachFEM is then used in the structural simulation accounting for the fiber orientation, strain-rate dependency and orthotropy of fracture. For its capabilities this material model has comparatively low computational cost, rendering it suitable for industrial crashworthiness analysis. The postprocessing of the simulation results can be done in META visualizing fiber orientation and weld lines as well as identifying critical regions of plastic parts.

Efficient middle meshing: a benchmark on a headlight housing part

[1]Christos Sachanas*, [2]Simone Chirra
BETA CAE Systems,
[2]BMW Group

KEYWORDS -
Meshing, Casting, Middle mesh, Inherit Casting Meshing

ABSTRACT
Meshing of casting parts can be a time-consuming, hard task. ANSA offers functionality that helps the user in three different ways: save time, assure quality and exploit similarities between parts (updated CAD version of a part, left side - right side parts, symmetrical parts). BMW's Closures department deals with a big amount of injection moulding parts that take up the lion share of the meshing as well as the overall model preparation effort. Using a headlight housing injection moulded part from this department, a benchmark has been carried out: the time needed for meshing a with conventional methodologies was compared and with the "Align constraints" tool and workflow that ANSA has to propose. In addition, the quality of the outcome was assured with "Inspect" functionality and it was compared with the quality of the conventional methodology result. "Inherit mesh", the functionality that ANSA provides to exploit similarities of parts, was also taking into account in this benchmark. The results showed that, using the proposed ANSA functionality and workflow, the user can be benefited with great reduction of the meshing time. This meshing time reduction can be even bigger if the similarity of parts is exploited. The results also showed that ANSA functionality guides the user to achieve a high quality result, in terms of middle surface and thickness accuracy.

3D Mesh generation in ANSA for metal casting simulation with PROCAST - The Brembo automated process

[1]Federico Zaramella*, [2]Panagiotis Fotopoulos*, [3]Marcello Boschini, [3]Giacomo Valsecchi
[1]BETA CAE Italy Srl,
[2]BETA CAE Systems,
[3]Brembo S.p.A

KEYWORDS -
Metal Casting, ProCAST, Process Automation, Tet Mesh, Volumes Connection

ABSTRACT
In recent years, the pursuit of lightweight products is the main target, especially in the automotive industry. This has elevated the simulation of the metal casting processes, using CAE software products.

Following this trend, ANSA provides its strong capabilities in pre-processing, in connection with the proven casting simulation software ProCAST.

This recommended solution package is already proved by the project developed by BETA CAE Italy for Brembo SpA to streamline and speed up the generation of ProCAST's models.

Focusing on the common target of fully supporting ProCAST in the next future through a dedicated solver deck, the currently available ANSA functionalities can be used to prepare 3D meshes both manually and through fully automated custom processes. Both methods lead to a large decrease of time and effort preparation ensuring a very good elements' quality.

The project started by Brembo in collaboration with BETA CAE involves an automated process which, by exploiting the comprehensive ANSA python API, receives as inputs the geometry of the metal casting assembly (in CATIA format) and few user's parameters. Upon successful executions, the user will get a high quality Tet mesh drastically reducing time and effort when compared to manual executions.

Furthermore, the recently added libraries to read ProCAST results in *.erf format enable the usage of all the standard post-processing capabilities of META, from reading the results up to final report, fully automated.

Session 4

New equivalent static load (ESL) creation procedure for complete vehicle

[1]Jens Weber*, [2]Luiz Ricardo, [1]Jesper Bäcklund
[1]CEVT AB,
[2]Politecnico Milano

KEYWORDS -
Complete vehicle simulation, vehicle body stiffness, Opening Distortion Fingerprint ODF, Equivalent Static Load ESL, Multi Stethoscope MSS

ABSTRACT
By analyzing the dynamic distortion in all body closure openings in a complete vehicle, a better understanding of the body characteristics can be achieved compared to traditional static load cases such as static torsional body stiffness. This is particularly relevant for non-traditional vehicle layouts and electric vehicle architectures.

The body response is measured with the so-called Multi Stethoscope (MSS) when driving a vehicle on a pavé road (cobble stone). The MSS is measuring the distortion in each opening in two diagonals. During the virtual development, the distortion is described by the relative displacement in diagonal direction in time domain using a modal transient analysis. The results are shown as Opening Distortion Fingerprint ODF and used as assessment criteria within Solidity and Perceived Quality.

By applying the Principal Component Analysis (PCA) on the time history of the distortion, a Dominant Distortion Pattern (DDP) can be identified. The DDP means that more than 50 % of the body deformation states for a given pavé time history are similar to each other.

This paper presents a deeper analysis of the forces which are associated to this Dominant Distortion Pattern (DDP). The analysis includes all forces between the wheel suspension and the trimmed body. Based on the results of this force analysis, a new procedure for creating an Equivalent Static Load (ESL) was developed. Finally, a toolbar in META is shown to automize this new ESL creation procedure.

ANSERS: The single point-of-truth for simulation and physical test data

Antonis Perifanis
BETA CAE Systems

KEYWORDS -
ANSERS, SDM, SPDRM, ASAM

ABSTRACT
ANSERS is a web application designed to simplify the process of accessing and combining engineering simulation and physical test data, providing a comprehensive view for efficient engineering design decision-making. It efficiently handles large volumes of data generated by simulations and physical tests and stored in a data management system. ANSERS provides an easy-to-use interface for accessing, visualizing, correlating, and sharing simulation and physical test data, using customizable and automatically updated dashlets on a web dashboard. Data can be retrieved from various sources, while security restrictions are respected. ANSERS facilitates effective communication and decision-making, enabling effortless comparison and sharing of simulation and physical test results, and ensuring that stakeholders of all levels of expertise receive meaningful and comprehensive information.

FATIQ - The comprehensive platform for streamlined fatigue analyses

George Korbetis
BETA CAE Systems

KEYWORDS -
FATIQ, fatigue analysis, life

ABSTRACT
BETA CAE Systems presents FATIQ, its brand-new software for fatigue life prediction. FATIQ is a new tool for fatigue life prediction based on FE results. It is capable for Stress-Life and Strain-Life analysis while supporting both frequency and time domain calculations. A solver neutral application since it supports input from various FE solvers like NASTRAN and ABAQUS. It is a complete solution for fatigue analysis since it provides analysis set-up, solving and Post-processing.

FATIQ is an easy-to-use application since it provides a workflow manager to set-up the analysis in a step-wise manner that allows the use from engineers of different expertise level and avoids struggling with keyword based set-up. Enhanced tools like material assignment by rules, installed fatigue materials database and hot spot filtering facilitate the analysis set up and run.

Finally, FATIQ offers interactive Post-processing through dedicated plots like PSD plots, Event graphs, Rainflow matrices and 3D fringes. All these plots are collaborating to give the engineer a clear view of the analysis.

Features, benefits and capabilities as well as future development are shown in this presentation

Session 5A

Streamlining simulation run preparation with an end-to-end modular methodology

Irene Makropoulou
BETA CAE Systems

KEYWORDS -
MRM, DM

ABSTRACT
Modular Methodology, proposed by BETA, presents an efficient approach for the preparation and management of CAE simulations of complex structures within the Modular Run Management (MRM) environment. This methodology combines established practices such as the organization of the main deck in include files or the marking of interface locations for intermodular assembly and loadcase set-up, with cutting-edge methods for model composition, loadcase set-up and model integrity checking. The resulting solution provides a unique approach to streamline simulation preparation.

At the heart of the modular methodology, Build Processes facilitate the breakdown of the complete simulation run workflow into smaller processes, associated with specific modules. These processes can be executed independently at the module level, or combined bottom-up at the assembly level.

This presentation explores how recent advancements have expanded the capabilities of the Build Processes, allowing for their integration into a wider range of applications. These enhancements encompass various areas, such as parts processing, handling of reduced models and the generation of DOEs. As a result, more applications can now be seamlessly integrated into the Modular Environment, enabling CAE engineers to achieve greater productivity, higher quality of deliverables and faster turnaround times.

From Includes to a comprehensive CAE application environment

Carsten Hofer
Hyundai Motor Europe Technical Center GmbH

KEYWORDS -
Data Management, NVH, NVH-Console

ABSTRACT
Configuration management of today's CAE models in the automotive industry is getting more and more complex. On one hand, this is because of more sales variants e.g. due to additional powertrains or customer variants and on the other hand, CAE environments are becoming more interlinked between various disciplines. In order to approach the later, BETA CAE's Data Management has been implemented at Hyundai's European R&D centre.

In a proof-of-concept study in the NVH-CAE team focus was placed on importing CAE models, provided by the Korean R&D centre, building ready-to-run simulation models and deriving component models for the NVH-Console. The current activities are aimed at improving the usability and adding functionalities for body stiffness analyses as well as creation of flex-bodies for R&H analyses.

The presentation will elaborate on the file-based implementation of BETA CAE's Data Management, its challenges and solutions and will outline the plans to extend the system to further application areas as well as linking it to other information systems.

Addressing challenges in the BiW model build-up process in AUDI AG

[1]Thanasis Fassas*, [2]Richard Lindner
[1]BETA CAE Systems,
[2]AUDI AG

KEYWORDS -
KOMVOS, SDM, ANSA

ABSTRACT
The buildup of BiW simulation models brings up several challenges, which the engineering team must overcome. The presented workflow includes the necessary steps, starting from a CAD geometry file to the generation of a meshed BiW model, ready to be used for all the required engineering analysis disciplines, e.g. Crash, NVH, Durability etc. The challenges emerging and how these are addressed are highlighted. Starting the process, we should consider that the “design files” do not always include all the necessary information, and sometimes the provided information is wrong. Furthermore, in most cases, these files are not available at the same time, which increases the complexity of the process and the probability to introduce errors while working on the model. Users working on the model preparation often do not use the same settings, in some cases, not even the same software. All this leads to a lack of robustness. Of course, during the development phase of the BiW model, several parts get a design update and need to be inspected. Which of them includes critical geometrical changes? Is any model affected by this update? Crucial questions arise that every user must deal with, and to have them answered, appropriate workflow and software tools are needed. Among the most challenging issues are storing, browsing, querying, and updating the data. To automate the BiW model-build process, the employment of an SDM-System is essential. An effortless, trouble-free, continuous, and direct communication with such a system ensures the best quality of the process. This presentation reveals how AUDI AG has set up a workflow that supports the user throughout the process, ensures the usage of the same settings for all users, prevents the errors generation in each step, and monitors the entire process. The software streamlines the process, communicates with the user, and informs him, in a user-friendly way, about the parts of the model that are not processed each time. Especially developed software functionality supports the user during the model update. This highlights all the changes between the different versions of the part, not only at the attributes level but at the geometrical level as well. During the workflow, all processed data are stored in the SDM system. This way, the data are available for all users to work on them and set up the upcoming necessary simulation analysis scenarios. This well-established solution, through a user-friendly interface, ensures the automated and robust browsing, viewing, and updating of the data.

Closures deformation optimization considering Kinetic system

Jinsoo Shin,
Hyundai Motor Company

KEYWORDS -
Moving parts, Deformation, Kinetic system

ABSTRACT
Most of the deformation of closures is caused by Gas lift, an external force element to be mounted. Therefore, the most efficient way to improve deformation is optimizing the mounting point and reaction force of Gas lift. But in the past, It could not be applied because it was difficult to consider the force required to open and close the closures reflecting the kinetic system. Therefore, improvement plans such as simply increasing the thickness or size of parts that require rigidity reinforcement are mainly applied. This has an improvement effect, but it is inefficient because the cost is increased.

So in this study, ANSA and META were used to improve this. The Kinetic tool in ANSA was configured as automation, allowing user to convert the moving into the kinetic model with just a few clicks, and to check the opening and closing power lead through META. Through this, it is possible to find an improvement plan that can satisfy the opening and closing power of the moving part while minimizing deformation. Also, we have a plan to develop an optimization technique to find the optimal mounting point and reaction force that satisfies the opening and closing power within a given design range and minimizes deformation.

In conclusion, it is possible to derive an efficient improvement plan capable of minimizing the improvement cost while reducing the deformation of the closure.

Session 5B

Watertight simuation with SPH to preventing leakeage in design stage

Jong Hyun Baek*, Chul Hong Lee
Hyundai

KEYWORDS -
SPH, watertight

ABSTRACT
The task at hand is to develop a technique for predicting the path and drainage conditions of watertightness problems caused by manufacturing deviations. One possible approach is to use the Finite Volume Method (FVM) for water tightness prediction analysis. However, this method requires detailed analysis, which can take up to 5 months from modeling to analysis. Thus, it may not be feasible to apply in practice.

Another approach that can be used is Particle Behavior Analysis or Smoothed Particle Hydrodynamics (SPH) for the development of a Predictive Analysis Method for Watertightness. This approach has the potential to provide a visual prediction of watertight robustness in the drawing stage. The time required for this method is within 2 weeks, making it more practical for real-world applications. The SPH method can be used to secure watertight quality in the drawing stage through BIW (Body-in-White) watertight analysis. This approach can help to identify any potential issues with watertightness before the manufacturing process begins, thus saving time and resources.

Overall, the development of a predictive analysis technique using SPH can help to ensure the quality of the manufacturing process and reduce the risk of watertightness problems caused by manufacturing deviations.

The integration of ANSA into the Climate Control systems engineering CFD process at Ford

[1]Konstantinos Haliskos*, [2]Igor Golberg
[1]BETA CAE Systems,
[2]FORD WERKE Gmbh

KEYWORDS -
CFD model process, Climate control, Automotive, ANSA, kinetics

ABSTRACT
From CAD to CFD simulation model, the requirements of CCSE group at FORD are demanding both in volume of designs and number of process tasks that cause additional workflow to users. Frequently groups follow different approaches on common problems and use a variety of tools for the same tasks increasing overall complexity. The time required to extract the appropriate CAD assemblies and prepare a watertight mesh model for each process is considerable and every effort to automate this is of paramount importance. Standardization through the use of common model or common process and tools is fundamental to the success of the group in this field.

This study provides a methodology used to handle the multidisciplinary tasks of CCSE groups at geometry and mesh level with the aim to create high quality surface and volume mesh with elements of model parameterization while establishing an efficient common environment for users. The included CFD models are looking at duct pressure drop / air vents control in cabin, defrost / demist and air extraction. The data provided as input includes an assembly of cabin with ducts which is used for preliminary studies as well as for full vehicle model of sign-off CAD. From the generation of the separate model sub-assemblies within ANSA, to the use of batch mesh and advanced meshing tools the process has been simplified and standardized. Additional tools like kinetics have been employed to handle model shape modifications under operation and automation has been implemented as per requirement.

During this study a diversity of model meshing techniques and tools have been exploited and included in the verification process with the scope to increase efficiency and improve meshing turnaround time. Overall, the ability of ANSA to handle all tasks and provide solutions per subcase has been a critical aspect in the decision to implement it in the process at FORD CCSE.

Advancing the virtual geometric representation of sealings on surrounding structures for precise simulation prediction in Aerodynamics Soiling and Aero-acoustics

Mattias Hejdesten,
Volvo Group Trucks Technology

KEYWORDS -
Virtual Development, Optimization, Aerodynamics, Soiling, Aero-acoustics, Accurate Geometric Representation

ABSTRACT
Recent legal demands introduced by EU commission on reducing CO2 emissions of the heavy-duty vehicle sector, has put intense focus on the European truck OEMs to improve the energy efficiency of their respective products to stay legal and competitive for years to come.

Air resistance is one of the greater contributors inhibiting the forward motion of trucks, thus leading to fuel consumption and CO2 emissions on diesel vehicles and yet inhibiting range on battery electric vehicles. Moreover, by the much quieter electric drivelines being progressively introduced in the market, the main sources of vehicle emitted noise, the engine and gearbox, will be eliminated causing a complete re-distribution and focus on the remaining sources of noise.

Shape, and more specifically detail optimization, are the main success factors to improving the aerodynamic performance of trucks. Surprising to many, but details as small as 1 mm in the most sensitive areas on a heavy-duty truck play a vital role in fulfilling the increased demands on energy efficiency, cleanliness of surface sensors and driver environment noise. Proper fitment of sealings to cover up split lines and gaps between panels in those sensitive areas is one measure being heavily considered in the aerodynamic improvement of trucks.

Consequently, there has been a notable rise in the demand for a more assured reproduction of the geometric representation of sealings as they are assembled onto surrounding structures. This heightened need stems from the pursuit of achieving highly accurate and realistic simulation predictions. By accurately capturing the intricate details of sealings in relation to their surrounding structures, it becomes possible to enhance the precision of simulations and improve the understanding of their effects on aerodynamics, soiling, and aero-acoustics. Through this advancement, the aim is to facilitate more reliable predictions and deeper insights in these crucial domains.

Session 5C

New features and enhancements in EPILYSIS

Panagiotis Pantazidis
BETA CAE Systems

KEYWORDS -
EPILYSIS, Optimization, Topology, NVH, Modal Model, AMLS, FSI, FMU

ABSTRACT
EPILYSIS has been for a lot of time now, a steady and robust alternative to all the major FEA Solvers. The new enhancements of the latest versions provide extra features that have steadily broadened the simulation solutions that are used by the everyday user. Enhancements in performance, optimization, and NVH are some of the new features of this presentation.

The introduction of the Multifrontal Massive Parallel Sparce direct linear solver (MUMPS), the MKL64 (64bit) library, and the Message Parsing Interface (MPI) facilitates a significant performance increase.

In the optimization field, an expansion of the already existing solutions in order to incorporate ANSA Laminates tool has been added to enable new optimization applications. Furthermore, the implementation of new response keywords offers curve matching in optimization cases for frequency response solutions. Also, the "Manufacturing Constraints Check" plugin allows an easy examination of the manufacturing constraints.

Multiple enhancements were made in the field of NVH. For example, modal models are a fully developed reduction technique, which can be an alternative to Superelements. The Automatic Multi-Level Substructuring (AMLS) method, which is embedded in Epilysis, offers new features and enables short turn-around times for modal NVH solutions. Another new NVH feature is the pressure equilibrium method, which calculates the Fluid-Structure Interaction (FSI) for vibro-acoustic analysis in a very accurate way.

The calculation of Flex Bodies for Multibody Simulations (MBS) has been further enhanced and offers the output in FMU format, which enables the coupling to Model-Based Design (MBD) software.

ANSA plugin of Virtual Strain Gauge utilities for FEMFAT

[1] **Minoru Nakano,** [2] **Kazumasa Kato**
[1] BETA CAE Systems Japan Inc., [2] MAGNA INTERNATIONAL JAPAN INC., Japan

KEYWORDS –
ANSA, Plugin, FEMFAT, Fatigue, Strain Gauge

ABSTRACT –

FEMFAT is a software that enables highly reliable fatigue strength evaluation using finite element analysis results, and its extension module, FEMFAT STRAIN, allows users to confirm/validate the correlation between experimental testing signal and fatigue simulation signal by attaching "virtual strain gauges" to FE models and comparing them with load tests. However, the definition of virtual strain gauges requires additional works/setups and is not compatible with similar functions of general-purpose post processors. We have now developed a plugin that enables the definition and editing of virtual strain gages for FEMFAT STRAIN to be performed efficiently on ANSA. The output from this function can also be loaded into META for use. This session will provide an overview of this functionality. A use case will also be presented to show the usefulness of this plugin. On a fatigue simulation of a differential case under real load condition, users can assess the sensitivity of strain gauge position before testing.

1. OVERVIEW OF MULTIDISCIPLINARY FATIGUE ANALYSIS BY FEMFAT

2. DEVELOPMENT OF THE PLUGIN - HOW IT STARTED?

3. THE FUNCTIONALITIES OF THE PLUGIN

4. USE CASE

5. CONCLUSIONS

- We have developed an ANSA plugin “Virtual Strain Gauge Utilities”, which enables creation of virtual strain gauges for FEMFAT Strain.

- In this plugin, users can make all definitions required by FEMFAT Strain efficiently in ANSA, including collapsed CTRIAs and measurement files.

- Strain gauges defined by this plugin can also be imported to META.

- A use case of a fatigue simulation on a differential case under real vehicle condition is presented. With this plugin, users can evaluate how big the sensitivity of strain gauge position is before testing.

Latest and future developments in simulations for durability and composites materials-made structures

Yiannis Asaniotis
BETA CAE Systems

KEYWORDS -
Durability, Composites, Abaqus, Nastran, Marc, Ansys

ABSTRACT
The latest versions of ANSA and META leverage new significant tools and features in the Durability and Composites domain so as to update and facilitate the daily job of an analyst.

More specifically , the connection manager has been enhanced to define contacts and pretension on solid bolts and beams among other features.

The cross section tool now provides calculations for nonhomogenous and mixed thin-solid cross sections.

Furthermore, new keywords and dedicated tools are introduced for Abaqus, Ansys, Marc, Nastran and Permas.

The results mapper has been improved to couple ANSA and META with more capabilities and results.

New toolbars have been introduced within META incorporating methods such as the opening closures distortion and neuber correction

In the domain of the composites, the laminate tool has been redesigned with a more user-friendly layout and the homogenization tool can now treat external RVE and periodic boundaries.

Especially for Ansys the lamination information is now fully supported in ANSA & META through ANSA comments

Position effect of drain holes on crossmember for Side Poll Crush scenerio

Serhat Soyer
Siro Energy

KEYWORDS - -
Battery, Crossmember, Drain Holes, Crush, Safety

ABSTRACT
Crossmembers are essential parts of a battery pack. These structural parts have many roles on torsional stiffness, torsional strength, bending stiffness, modules carrying and side poll crush so From basic design of a crossmember to serial production there are too many challanges to overcome. One of the biggest challange is to define drain holes positions on crossmembers because of their weakener effect. Study is investigating the position effect of drain holes on crossmember for side poll crush scenerio with ANSA tool.

Session 5D

The application of an IGA body in white in a hybrid full car simulation

[1]Frank Bauer*, [2]Lukas Leidinger
[1]BMW,
[2]ANSYS

KEYWORDS -
IGA, crash simulation, passive safety, CAD2CAE

ABSTRACT
IGA technology is getting closer and closer to being used in industrial environments. Be it through the potential to transfer CAD to CAE more quickly or to increase prediction quality through the more detailed description of the geometry. In order to test its use in an full vehicle simulation in terms of stability, performance and process, a complete body in white was converted into IGA and installed in a hybrid full vehicle model. The presentation discusses how well geometries can be converted to IGA in Ansa and where there are still challenges for full automation, as well as presenting the results of the overall vehicle simulations in comparison with conventional FE simulations.

IGA technology validation for forefront crashworthiness CAE analysis

Pablo Cruz*, Lluis Martorell, Adrian Ros, Alejandro Dominguez
Applus+ IDIADA

KEYWORDS -
Isogeometric analysis, IGA, crashworthiness, Shells, aluminium, advanced plasticity and fracture

ABSTRACT
Since its release in 2005, Isogeometric Analysis technology has drawn significant attention for its potential to bridge the gap between the CAD and CAE worlds, allowing for a more streamlined workflow. With IGA technology, design features can be embedded in the CAE solver, making it possible to include them as optimization variables throughout the design process, from the pre-concept phases to the final product release.

This paper analyses IGA potential for producing more efficient and lightweight car body designs through improved accuracy in capturing crash deformation patterns and ductile fracture. To achieve this goal, first IGA technology needs to be validated before integrating it into actual production.

To validate IGA technology, IDIADA conducted a comparison of IGA results to coupon and component testing, as well as state-of-the-art FEA results involving crash-like deformation patterns and material fracture. These activities were carried out as part of an Industrial Doctoral PhD, in-house innovation programs, and actual production projects. Testing and FEA results courtesy of Faraday Future.

Isogeometric Analysis in LS-DYNA: pushing the frontiers of modern simulation

[1]Lukas Leidinger*, [1]Stefan Hartmann, [2]Frank Bauer, [3]Dave Benson, [3]Attila Nagy
[1]DYNAmore GmbH, an ANSYS company,

[2]BMW Group,
[3]ANSYS

KEYWORDS -
Isogeometric Analysis (IGA), Automotive Crash Simulation, Shells, Explicit Dynamics, CAD/CAE integration

ABSTRACT
Isogeometric Analysis (IGA) is a Finite Element Analysis (FEA) technology that uses spline basis functions known from Computer Aided Design (CAD). Within IGA, both the geometry and the solution field are described by (smooth) splines, for example by Non-Uniform Rational B-Splines (NURBS). This isogeometric approach may allow a tighter integration between CAD and analysis, a more accurate geometry description, improved solution quality due to higher-order and higher-continuity elements, and a larger time step size in explicit time integration schemes.

In the last few years, IGA in LS-DYNA has made huge steps forward towards productive applicability in (automotive) industry. Key enablers for this accelerated progress were the IGA model generation capabilities for shells in ANSA, and the close collaboration between the LS-DYNA developers, the ANSA developers and industry partners like the BMW Group. The current backbone of IGA in LS-DYNA is the jointly developed and CAD-inspired *IGA keyword data structure that considers both geometry and topology information of the model.

With these IGA capabilities in ANSA and LS-DYNA, users can now generate and run hybrid IGA/FEA body-in-white models with hundreds of IGA shell components (trimmed multi-patch NURBS shells) in full vehicle crash simulations.

In this contribution, we will give an overview about the latest IGA developments and applications in LS-DYNA, with a focus on trimmed (multi-patch) NURBS shells, explicit crash simulations, connection technology and material damage. Finally, we provide an outlook to trimmed IGA solids and postprocessing on NURBS geometries.

Methods and tools for IGA models for LS-DYNA

Lambros Rorris,
BETA CAE Systems

KEYWORDS -
IGA, Crash, LS-DYNA,

ABSTRACT
Isogeometric Analysis (IGA) is maturing fast and offers certain advantages over classic FEA that makes it useful for large deformation dynamic analyses such as Crash applications in the automotive industry. During the last years the capabilities of both LS-DYNA and ANSA, in the creation and analysis of IGA crash models, have been under continuous development. Although the focus of our efforts has been the efficient generation of trimmed NURBS-based multi-patch surfaces, both body - fitted methods for shells, as well as immersed methods for solids are being investigated from the point of view of model creation and pre-processing. Continuous integration with existing FE models and easy creation of Hybrid FE – IGA is another focus of our development efforts. Post processing of IGA results presents its own challenges but META is following up closely.In all of these areas we've been actively developing and releasing software to the industry, and these developments will be presented.

Session 6A

Bringing Human Body Models to life. The future in Safety simulations.

Lambros Rorris
BETA CAE Systems

KEYWORDS -
HBM, Occupant, Safety, Crash

ABSTRACT
HBMs have become a much-needed tool for the safety simulations of the automotive industry. Out-of-position load cases for passengers and drivers and simulations for other vulnerable road users, like pedestrians and cyclists are increasingly needed, and HBMs can address this need. Nevertheless positioning, pre-processing and post-processing of these models on a level suitable for industrial use is not straight-forward. During the last years we have been very active both in research and in software development providing CAE users the much needed tools to industrialize the use of HBMs.

Assessment of abdominal and skeletal loadings and kinematics during frontal impacts: employing a novel tool for HBM variants generation based on the occupant's BMI

Dimitris Zouzias*, A.Fokylidis
BETA CAE Systems

KEYWORDS -
frontal impact, obesity, crash, abdomen

ABSTRACT
A new and more biofidelic methodology for creating HBM variants is presented and utilized to investigate the loading on the abdomen area and skeleton of three different HBM variants. The study was able to verify the different loading conditions observed between occupants of different BMIs on the respective areas. The loading of the upper abdomen was reduced and exhibited more time to build-up on the obese and M95 occupant model compared to the average. On the other hand the bladder and rectum exhibited higher stresses. Furthermore the lower extremities of the obese and M95 occupant seem to have higher probability of injury with shorter time to impact onto the instrument panel (IP). Different loading profiles were observed also on the spine which align with accident statistics.

Comparison of the injury risk prediction of the THOR-Reclined dummy and the THUMS HBM

[1]Pablo Lozano*, [1]Simona Roka, [1]Alessandro Gravina, [1]Genís Mensa, [2]William Marshall
[1]Applus IDIADA,
[2]CELLBOND

KEYWORDS -
Human Body Model, HBM, Anthropomorphic Test Device, ATD, dummy, reclined posture, automated vehicle, submarining, THUMS, THOR-Reclined kit, injury risk prediction, kinematics, injury criteria, comparative analysis.

ABSTRACT
Autonomous vehicles are expected to allow car occupants to position themselves in more relaxed positions inside the vehicle. These new seating positions constitute a new challenge for crash safety analysis. Therefore, new crash test protocols, adapted to this new paradigm, may be required in the future.

In the literature, most of the virtual reclined posture analysis has been performed using Human Body Models (HBMs) which are increasingly used to assess vehicle safety and injury risk, as currently regulated ATDs (Anthropomorphic Test Devices) are neither designed nor validated for reclined seating configurations. Nevertheless, these HBM simulation studies need to be correlated against repeatable physical tests that allow future cars to be rated according to regulation and consumer testing protocols. New options for crash dummies such as the THOR-Reclined kit from CELLBOND; which allows adapting the THOR ATD for these new reclined seating postures, are being developed and may enable the performance of physical tests in reclined occupant positions. However, the question of whether its performance is comparable to that of an HBM remains unanswered.

A series of simulations were then conducted comparing the behavior of the THOR-Reclined simulation model and the THUMS v4.1 by means of kinematics and injury risk prediction. Also, a series of tests using the THOR-Reclined in IDIADA's deceleration facility have been planned and the results will be shared in future publications. Injury risk prediction was then compared between the HBM and the ATD. The ATD and the HBM FE models were compared by means of kinematics, restraint system outputs, injury criteria, and injury risk prediction. The result of this comparison will be discussed in this paper.

Some differences were observed between the models. THUMS allowed to study injury risk criteria based on the strain of the rib cage, while the ATD is mainly designed for measuring displacements and accelerations. The primary limitation of this work is the lack of thorough validation data of the active HBM and the ATD model in the studied position. However, this work provides further insight into the comparability of their performance and the differences found between the studied models. Differences have been found between the two models, mainly due to their physical dissimilarities. Nevertheless, some comparisons can be made between them from a kinematic and injury criteria perspective and will be shared in this paper.

Session 6B

Sensitized-RANS simulations of complex flows in conjunction with ANSA®: towards increasing efficiency

Louis Krüger*, Maximilian Bopp, Suad Jakirlic
Technical University of Darmstadt

KEYWORDS -
CFD, vehicle aerodynamics, IC-Engine intake, Eddy-resolving URANS modelling, LES, Magnetic Resonance Velocimetry

ABSTRACT
Scale-resolving turbulence models based on the hybrid RANS/LES concept (Reynolds-Averaged Navier Stokes / Large-Eddy Simulation) have increasingly become a computational tool of engineering importance. Here, an appropriately sensitized RANS model is used in an LES-related computational procedure, thus enabling the capture of the spectral dynamics of turbulence, to an extent consistent with the underlying grid resolution and turbulence quantity equations to be solved. They combine the spectral properties of an LES method with industrially suitable RANS models aiming at achieving the practically affordable simulation tool. In the present work, two computational studies on the “Influence of Wheel Wake on Vehicle Aerodynamics” and on the “Intake Flow of an IC-engine” were performed.

The first study focuses on the influence of the wheel wake on vehicle aerodynamics considering the rotation of detailed rims. The car configuration corresponds to a full-scale (1:1) notchback configuration of the well-known ‘DrivAer’ vehicle model, Heft et al. [1]. The objective of this work is to investigate the performance of some popular turbulence models in conjunction with different methods for handling the wheel rotation – rotating wall velocity, ‘multiple reference frame’ and ‘sliding mesh algorithm’. The second study is concerned with the simulation of an intake flow of an internal combustion engine, the so-called “engine flowbench” case, representing an IC engine configuration, with opened valves and non-moving piston. Correct capturing of such complex flows requires application of sophisticated turbulence models. Accordingly, a scale-resolving method employing the sub-scale model based on the solutions of the equations governing the residual turbulence quantities is applied.

Two near-wall model versions are available. A four equation eddy-viscosity model, solving, in addition to equations governing the sub-scale kinetic energy of turbulence and its dissipation rate, the equation describing the dynamics of the normal-to-wall turbulence intensity (Krumbein et al., 2020, [2]) was applied to the wheel aerodynamics and a model governing the entire sub-scale stress tensor (Jakirlic and Maduta, 2015, [3]) was applied to the “engine flowbench”. The models’ sensitivity towards appropriate capturing of the fluctuating turbulence is enabled through inclusion of an adequately modelled production term into the relevant scale-supplying equation, proposed in line with the Scale-Adaptive Simulation concept (SAS, Menter and Egorov, 2010, [4]). Such RANS-based, anisotropy-resolving sub-scale model formulations are grid-spacing free unlike the majority of relevant hybrid RANS/LES models, representing certainly an advantage, especially if unstructured grids with arbitrary grid-cell topology are employed. All model equations are implemented into the open-source code OpenFOAM®, with which all simulations are performed in conjunction with pre-processing software ANSA®.

The computational results, including the instantaneous and time-averaged flow properties, are discussed along with the complementary LES (by using the σ-model for the subgrid-scales, [5]) and experimental reference, where the latter is provided, among others, by the magnetic Resonance Velocimetry (MRV) measurement technique, [6]. In addition, the POD (Proper Orthogonal Decomposition) analysis is applied to the wheel wake, allowing to detect most energetic spatially-independent coherent structures. Furthermore, the Reynolds-stress anisotropy analysis is performed to highlight the flow characterization within the IC engine.

References:

[1] A. Heft, T. Indinger and N. Adams, “Introduction of a New Realistic Generic Car Model for Aerodynamic Investigations,” SAE Technical Paper 2012-01-0168, 2012, https://doi.org/10.4271/2012-01-0168

[2] B. Krumbein, R. Maduta, S. Jakirlic and C. Tropea, “A Scale-Resolving Elliptic-Relaxation-Based Eddy-Viscosity Model: Development and Validation,” New Results in Numerical and Experimental Fluid Mechanics XII, Notes on Numerical Fluid Mechanics and Multidisciplinary Design (NNFM) 142 (2020): pp. 90-100.

[3] S. Jakirlic and R. Maduta, “Extending the bounds of ‘steady’rans closures: Toward an instability-sensitive reynolds stress model,” International Journal of Heat and fluid flow, vol. 51, pp. 175–194, 2015.

[4] F. Menter and Y. Egorov, “The scale-adaptive simulation method for unsteady turbulent flow predictions. part 1: theory and model description,” Flow, turbulence and combustion, vol. 85, no. 1, pp. 113–138, 2010.

[5] F. Nicoud, H. B. Toda, O. Cabrit, S. Bose, and J. Lee, “Using singular values to build a subgrid-scale model for large eddy simulations,” Physics of Fluids, vol. 23, no. 8, 2011.

[6] D. Freudenhammer, E. Baum, B. Peterson, B. Böhm, and S. Grundmann, “Towards time-resolved magnetic resonance velocimetry for ic-engine intake flows,” Exp. Fluids, vol. 55, p. 1724, 2014.

Best-in-class CFD meshing for the aerospace sector

Vangelis Skaperdas
BETA CAE Systems

KEYWORDS -
CFD, Meshing

ABSTRACT
CFD meshing for the aerospace sector is a demanding field with high standards in accurate geometry representation and mesh quality. The CFD simulations, that provide a critical foundation in the development of an aircraft, are as good as the meshes they are run on, and their fidelity cannot be compromised.

Additional challenges that have to be addressed include advanced proximity and curvature refinement with specific mesh treatments like anisotropic treatment of leading and trailing edges, demanding boundary layer resolution with extreme element aspect ratios, custom Size Field functions to resolve specific flow features, as well as very large mesh sizes pushing the hardware resources to their limits for speed and memory. In addition, the latest developments in the field of HIgh Order CFD, open up new needs for high order meshing either directly or via mesh elevation methods.

In this presentation we will demonstrate how ANSA addressed all these topics and provides the best in class CFD meshing pre-processor for aerospace application examples from the High Lift Prediction Workshops.

Session 6C

Session 6D

Isogeometric analysis in ANSA using local refinement and thb-splines

[1]Angelos Mantzaflaris*, [2]Ioannis Chalkidis, [1]Christos Karampatzakis
[1]Inria Center of the French Riviera University,
[2]BETA CAE Systems

KEYWORDS -
hierarchical splines, isogeometric analysis, Bézier extraction

ABSTRACT
The concept of hierarchical splines is a well-established approach to adaptive refinement in the isogeometric analysis (3). The approach considers a set of nested spaces that are generated by globally refining a coarse mesh. They are generated using a certain selection mechanism that ensures high global smoothness and efficient local refinenet. More recently, a slight modification of this approach, the so called truncation mechanism, was shown to provide a basis with better properties, such as the partition of unity property, better stability and robustness (4).

An open-source implementation of truncated hierarchical splines is available in the G+Smo (Geometry plus Simulation Modules) library for isogeometric analysis. G+Smo is an object oriented, templated library, that implements a generic concept for isogeometric analysis, based on abstract classes for basis, geometry, assembler, solver etc.

With the help of G+Smo we create hierarchical spline models that use local refinement to capture small features, such as small drill holes in car body. We transfer the model to ANSA (1) using Bézier extraction in order to setup and perform suitable analysis. This first experience of hierarchical spline analysis in ANSA shows the strong potential of this novel spline technology.

Isogeometric contact analysis in G+Smo with preprocessing in ANSA

[1]Christos Karampatzakis*, [2]Dr. Angelos Mantzaflaris, [3]Dr. Ioannis Chalkidis,
[4]Prof. Christopher Provatidis, [1]Prof. Athanassios Mihailidis
[1]Aristotle University of Thessaloniki,
[2]Inria Sophia Antipolis, Université Côte d'Azur,
[3]BETA CAE Systems,
[4]National Technical University of Athens

KEYWORDS -
Isogeometric Analysis, Contact, Splines, G+Smo, Interface

ABSTRACT
Geometry + Simulation Modules (G+Smo, pronounced gismo) is an open-source C++ library implementing multiple families of splines as well as providing the assembly and solution procedures required to perform simulation on these geometries. While it offers a lot of functionality, it currently has no graphical user interface (GUI), thus the creation of models is a time consuming procedure. Since ANSA offers preprocessing capabilities for B-Spline geometries in its last few versions, an ANSA / G+Smo interface is proposed and implemented, in order to ease this workflow, in the sense that files can be exported from one platform to the other.

Isogeometric analysis is a relatively new simulation paradigm, which is starting to gain traction in industrial applications. The main advantage of the method is that the simulation domain is not approximated by linear elements as in the Finite Element Method, but represented exactly, since it uses the same discretisation as in CAD systems. The exactness of geometry is especially important for contact simulation, since the problem is highly sensitive to the domain's geometry. Also since the bases most commonly used in Isogeometric analysis are of higher degree, they possess higher inter-element continuity, which ensures a continuous field of normals within a patch. This alleviates the need for smoothing procedures that are very commonly employed in the Finite Element Method.

The Gauss Point to Surface (GPTS) has been implemented in G+Smo, along with two of its variants. This algorithm has been selected due to its relative simplicity in implementation and accuracy of results. The proposed workflow is showcased through an example where the B-Spline geometry for some characteristic mechanical contact problems is prepared in ANSA and simulation is performed within G+Smo.

Session 7A

Model development and simulation process of customized numerical Human Body Models for the study and analysis of healthcare products

Alit Putra*, Mikael Almquist
ALTEN Sweden

KEYWORDS -
Finite Element, Customized Human Body Model, Healthcare

ABSTRACT
Currently, there has been an increase in using Finite Element (FE) Human Body Models (HBMs) as a tool to develop and study healthcare products. There are many advantages of using FE HBMS for healthcare product development. FE HBMs can help to gain an understanding of how the product's design interacts with the human body. Parametric studies by changing some parameters can also be possible using FE HBM due to the relatively lower cost of running simulations compared to conducting physical experiments. The results from FE HBMs simulation can also be used for marketing purposes to illustrate the product benefits. Despite these advantages, many challenges still exist nowadays when using FE HBMs in the healthcare sector, one of them being the complexity of human anatomical and physiological properties.

Humans are very complex in terms of anatomical and physiological properties. It is almost impossible to have a model that fully represents the human population. Huge variations in human size or anthropometrics have also been reported in the literature. Not only variations in gross anatomical size such as Body Mass Index (BMI) but also local anthropometric variation, such as variation in pelvic bone geometry, have created a tremendous challenge in FE human body modeling. Therefore, a customized FE HBM is needed to account for the specific purpose of each study.

The main objective of this presentation is to give an overview of the model development and simulation process of customized FE HBMs. The area of applications was emphasized in the study and analysis of various healthcare products. Several examples from real-business cases were presented, highlighting the important steps that need to be taken to be successfully conducting the simulations. In each step, the challenges and how to approach them were listed and discussed.

Geometric variation of pelvic bone in Human Body Models with focus on submarining risk

[1]Marius Rees*, [2]Prof. Dr. Steffen Peldschus, [1]Philipp Wernicke
[1]BMW AG,
[2]LMU München, Biomechanik und Unfallforschung, Institut für Rechtsmedizin

KEYWORDS -
Human Body Models (HBMs), pelvic bone, submarining risk, iliac wings

ABSTRACT
The phenomena of “Submarining” occurs when the lap-belt slips over the pelvic bone, resulting in severe abdominal and pelvic injuries. Next to the restraint systems, also the geometrical shape of the pelvic bone can influence the occurrence of submarining [1]. This study focuses on the geometrical variation of the anterior iliac wings, a small frontal part of the pelvic bone, based on real volunteer CT-scan data. For this, the image database of the Institute of Legal Medicine of LMU University Munich was used and the most diverse geometries were taken. By using the morphing tools of ANSA Preprocessor, the original pelvic bone shape of a Human-Body-Model (HBM) was modified to these new target geometries.

The now varying HBMs were integrated into the generic seat environment of Uriot et al. [2] and positioned in a reclined seating posture, as illustrated in Fig. 2. In the scope of a following parameter study, all considered geometries were compared by doing a 50 times variation of several input parameter by using a Sobol Sampling method [3]. The aim of this parameter study was to answer the question if some geometries show submarining while others do not under the same conditions.

It can be emphasized, that already minor local variations of anterior iliac wings have an impact on submarining risk. Especially more vertically oriented wings indicate a lower submarining risk, whereas more horizontally oriented geometries are more willing to submarine.

Addressing positioning challenges for ATDs and HBMs: embracing the ANSA approach in industrial safety applications

Thanassis Fokylidis*
BETA CAE Systems

KEYWORDS -
Dummies, Human body models, positioning

ABSTRACT
ATDs are a crucial component of safety simulations, widely utilized by analysts studying occupant safety in vehicles for decades. Similarly, human body models have gained popularity in recent years, offering a more realistic representation in safety simulations. Analysts require precise and effortless positioning tools for both ATDs and human body models. ANSA provides a tool for ATDs that can swiftly and accurately position the respective FE model's limbs either manually or using the marionette method with solver aid. Additionally, ANSA in conjunction with it's multi-body dynamic solver and advanced morphing algorithms allows for the straightforward positioning of human body models. ANSA can address the needs of safety analysts, with the possibility of extending existing tools for more intricate industrial applications. This presentation details the current status of the aforementioned tools and examples of industrial applications that require their extension.

Advanced FE biomechanical positioning method

[1]Francesco Cappellino*, [2]Alessandro Sias
[1]C.R.F. S.C.p.A
[2]FCA Italy SpA

KEYWORDS -
Automated, Biomechanics, Positioning, Safety

ABSTRACT
The purpose of this project is to create an automated procedure, based on the ANSA Pre-processor (BETA CAE Systems), which guides the user step by step in a customized process that allows to perform all the necessary tasks to correctly position the seat and the dummy, to create the seat belt and finally set up a complete FE model for biomechanical analysis ready to be run with LS-Dyna solver. This must be done by implementing, standardizing and automating the user's usual operations during the preparation of FE models, minimizing manual activities and errors. The final goal is to develop a dedicated graphical user interface (GUI) which is user friendly and easy to execute even for users without familiarity with ANSA Pre-processor (BETA CAE Systems) and Biomechanics applications. The presented wizard gathers all the available tools specific for interior safety setup case in one unique environment, which has been specifically developed and customized to meet our modelling needs.

In order to achieve such ambitious goal the development stage has been carried out by BETA CAE expertise in strong and constant cooperation with us. Following the ongoing deployment stage we plan to extend the usage of the tool to all Safety and Biomechanics Virtual Analysis Teams of STELLANTIS company.

Session 7B

Latest and future developments in ANSA and META for CFD

Grigoris Fotiadis
BETA CAE Systems

KEYWORDS -
Topo, Mesh, CFD

ABSTRACT
In an environment of ever-expanding needs for accurate results, high quality meshes, efficient memory management, reduced turnaround times and automation, the complexity of problems that concern CFD engineers increases continuously. The solution to these problems must be designed in a manner that ensures the highest level of quality without compromising ease of use and possibilities for automation allowing the user to focus on the physics of the simulation. This presentation summarizes the wide array of tools and solutions offered by ANSA and META that open new ways of approach to these challenging tasks. From thin wall meshing to new powerful functionalities for mesh refinement through Size and Adaptivity fields and from multi-threaded surface wrapping to new great improvements in part replacement in volume mesh enabling mesh modularity, ANSA significantly improves the meshing workflow for CFD applications.

Additional improvements include High Order meshing, new advanced Layer generation options, improvements in Hexa-dominant meshing with HexaInterior and Hextreme algorithms as well as pure-hexa meshing with Hexablock, a brand new SPH solver and better integration with CFD solvers. Similarly, the latest post-processing tools for CFD offered by META are presented, including newly supported file formats, Schlieren Field visualization, custom shape of Cut Planes,

multiple creation of Cut Planes and IsoFunctions along with great performance improvements. Finally, a summary of future developments is presented demonstrating the commitment to offer even more advanced and exciting features for CFD applications.

Suitability assessment of various meshing strategies for CFD applications

Nicholas Mitroglou
BETA CAE Systems UK

KEYWORDS -
CFD,feature treatments, quad, tria, poly

ABSTRACT
For several years the starting point for the majority of grids for flow simulations has been a triangular surface mesh. A notable exception has been the aerospace industry. The process would then include generation of prism layers and completion of the domain with a tetrahedral, or hexahedral type interior volume mesh. The pros and cons of this technique have been analysed multiple times and all analysts agree on one point: there are compromises made in this approach that sometimes are negligible, while in other cases they are seen as a “necessary evil”. The latter does not stand true for the aerospace industry and their persistence for “high quality quadrilateral surface mesh” has been remarkable over the years. Despite the fact that current meshing software globally may, or may not be at the desirable level for generating a flawless quad-surface mesh, we often wonder: is there anything to be learnt from this approach that may benefit other industries and improve their predictions? A similarly puzzling question is related to polyhedral elements; they have been faced with awe, initially, but gradually gain supporters. The current study aims to provide the information to enable stakeholders decide what is best and why. Therefore, an exhaustive list of numerous combinations of different meshing strategies have been created and simulated. Results include, but are not limited to flow fields predictions; an attempt has been made to present the overall experience any combination of meshing strategies offers to the analyst.

Introducing Flow360 - a fast and accurate CFD solver integrated with ANSA

Maciej Skarysz*, CJ Doolittle, Zongfu Yu
Flexcompute

KEYWORDS -
Flow360, Flexcompute, CFD, Integration

ABSTRACT
The accurate prediction of aerodynamic performance is crucial for the development of efficient and high-performance vehicles. In quick development environments, the integration of computational fluid dynamics (CFD) with pre-processing tools is essential for the efficient analysis of complex geometries. In this work, we present the integration of Flow360 with ANSA for accurate aerodynamic analysis.

Traditionally, performing complex flow simulations on real-world products has been both time consuming and costly. However, with the emergence of the Flow360 solver, these barriers are being eliminated. This state-of-the-art solver is based on a full-stack software-hardware design that enables steady-state flow simulation of complex products within a matter of minutes - a process that used to take several hours and require thousands of cores to achieve. The Flow360 solver can perform transient simulations of complex flows in just a few hours, and it can prepare complete aero databases in days. Thanks to its cloud-native architecture, this cutting-edge tool can be accessed through both web applications and ANSA, offering users unparalleled flexibility and ease of use.

The integration of Flow360 with ANSA allows for a seamless workflow for the meshing and preparation of simulations. ANSA provides powerful meshing capabilities that enable the creation of high-quality meshes for CFD simulations, and Flow360 provides accurate and highly efficient CFD simulations of vehicle aerodynamics. The integration of these two tools streamlines the CFD workflow and reduces the time required for analysis.

The Windsor Body and DrivAer models are benchmark test cases widely used in the automotive industry for aerodynamic analysis. Our study focuses on evaluating the accuracy of Flow360 simulations and presenting the easy-to-use workflow integrated with ANSA. We conducted a series of simulations for the Windsor Body and DrivAer test cases, comparing the results with experimental data. Flow360 results show good agreement with experimental data and other CFD codes for both the Windsor Body and DrivAer test cases. The aerodynamic coefficients obtained from the simulations are in close agreement with the experimental data, demonstrating the accuracy of the Flow360 solver. The 260M cell DrivAer simulation was completed in just 17 minutes with Flow360.

Overall, the integration of Flow360 with ANSA provides a powerful and efficient workflow for the aerodynamic analysis of complex geometries in the automotive industry.

Session 7C

Latest and future developments in NVH

Vassilis Pavlidis
BETA CAE Systems

KEYWORDS -
NVH, Acoustics, Reduced modelling, System-level analysis, MBD, Optimisation, Modal Response, FRF Assembly, NVHConsole

ABSTRACT
Following an intensive development on NVH over the last years, BETA suite has been broadened with more capabilities and this aligns with the global trend of CAE community for more accurate NVH analyses with high performance. This presentation provides the main highlights of this development effort including for ANSA the introduction of a better coupling algorithm, the enrichment of ACTRAN deck to cover more analysis types as well as the support of building models for Statistical Energy Analysis. A complete tool for DOE and Optimisation studies of bush properties and modal parameters has been added to NVHConsole allowing for the easy set-up of such analyses, therefore, boosting up productivity. The calculation of Dynamic and Equivalent Stiffness of a subsystem is now facilitated through a dedicated standard loadcase in NVHConsole thus streamlining the whole process. Significant improvements in META target at improved performance and enhanced interactive usage while new toolbars such as the Participation Analysis Study and the Damping Estimation provide a deeper insight to the results. The Modal Response calculator has been extended to include Acoustic Sources as loading points, to calculate stress/strain results in Transient analysis and to accept modal displacements as input. Furthermore, the creation of FMUs from within the Modal Response tool enables the inclusion of META in Model-Based Design (MBD) analyses. Last but not least, FRF Assembly capabilities have been augmented with element results calculation along with the FRF Assembly decoupling method. More exciting developments anticipated with future versions are also discussed.

Optimization of sound pressure and vibration levels in automobiles and response predictions using Deep Neural Networks

[1]Vinamra Gehlot*, [1]Thomas Roy, [2]Markus Herbst
[1]EDAG Engineering GmbH,
[2]BETA CAE Systems

KEYWORDS -
NVH, Optimization, Machine Learning, Neural Networks

ABSTRACT
The sound pressure level in the occupant cell is among the major components of driving comfort. An acceptable sound pressure level in the occupant area is a quality indicator of how well the car can reduce internal (engine/compressor/road induced vibrations) as well as external noises. A major contributor in reducing such noises is the choice of bushings and vibration absorption pads. The aim of this study was to implement a novel workflow to optimize the sound pressure levels and structure vibration levels in the occupant cell with a series of coupled optimization and DOE studies. Moreover to develop tools to help the user make better decisions along the way.

With the advent of modern multi-objective optimization solvers, it is often difficult to have a grasp on the decisions that lead to the final optimized part. With the lack of “What If” studies, the impact of individual components on the overall response of the car is difficult to understand, unless more iterations are carried out for the specific purpose. The new proposed workflow not only tries to minimize the number of iterations and hence the resources but also allows the ability to fully understand the dependencies of components on the vehicle’s overall response. This in turn improves the quality of engineering decisions made during the last-minute changes in the development cycle.

NVH Console from BETA CAE Systems was used to implement the frequency-based assembly workflow, which allowed us to find the optimal bush-stiffness parameters alongside running topometry optimization using Nastran’s SOL200. Further, a targeted peak reduction technique was developed allowing the user to minimize multiple peaks from different responses with different weighting factors, even allowing to change the peak of interest they wish to reduce as the optimization is running in real-time.

Further, utilizing the new workflow with NVH Console’s fast response analysis, stiffness combinations and the corresponding responses were processed and used to train a deep neural network. This tool does not only assists the user approximate the peaks in responses but also their shifts across the frequency range, allowing them to take better decisions when it comes to choosing bushings of different stiffness coefficients.

Comparison of the numerical accuracy of Superelements and FRF Assembly

Markus Herbst
BETA CAE Systems

KEYWORDS -
Substructering, Superelements, FRF-Assembly, NVH-Console

ABSTRACT
Dynamic sub-structuring (DS) has always played a significant role in the field of structural dynamics. Performing the analysis of a structural system component wise instead of solving the entire system as a whole offers several advantages as it reduces the analysis time and facilitates additional root cause analysis types.

The DS techniques can be divided in two classes, the Component Mode Synthesis (CMS) methods and the FRF based sub-structuring (FBS) methods. In the first method components are represented by their modal contend and a well-known example is the Superelement (SE) technique implemented in many FE solvers. In the latter method the components are described by their Frequency Response Functions (FRF). The FRF-Assembly tool of META deploys an FBS method and it is used in the assembly process of NVH-Console, the NVH framework offered in ANSA.

In this presentation primarily the numerical accuracy of the Superelement technique (CMS) and the FRF-Assembly method of META (FBS) is investigated and compared for typical vibro-acoustic numerical FE analyses of automotive vehicles.

One aspect of the investigation is the role of damping assumptions commonly used in FE calculations. It is demonstrated that the use of structural damping is an important enabler of achieving a good numerical accuracy for all investigated DS techniques.

It will also be shown that another important aspect is given by the type of boundary conditions used for the solution of the individual components. In the FRF assembly process implemented in META all components are unsupported. In contrast, most Superelement analyses are performed using the well-known Craig Bampton approach with fixed interface degrees of freedom. It will be presented that this is another crucial aspect for achieving a good agreement between the different DS techniques and the results obtained, if the entire system is solved in a one-shot run.

Vibroacoustic coupling: a new approach

Kostas Skolarikis
BETA CAE Systems

KEYWORDS -
NVH, Vibroacoustics, Coupling, Cavity

ABSTRACT
The calculation of the fluid-structure coupling in acoustic simulations is not a trivial problem. The nonconformity of structural and fluid meshes and the geometrical complexities usually encountered in the structural enclosure of the cavity present significant challenges for coupling algorithms. Additionally, there are not standard quality criteria that can be used for the validation of the results. To address these challenges, ANSA has developed a new fluid-structure coupling algorithm that leverages the company's extensive experience in geometrical problems and result mapping techniques. ANSA's inspection and validation tools for coupling have also been enhanced ensuring the robustness of the new algorithm.

Session 7D

ANSA and META for battery electromagnetic simulations

Athanasios Papadopoulos
BETA CAE Systems

KEYWORDS -
Topo, Mesh, post-processing, EM

ABSTRACT
Electromagnetic systems and more specifically batteries, continue to gain grounds in almost limitless applications, ranging from consumer devices and heavy equipment to the automotive vehicles. Despite their mass appeal however, reliable electromagnetic simulation of batteries remains a challenge for the engineers. Such a reliable simulations would result in game changing benefits as it would drastically assist in the development of new products while same time reducing product design time and costs. In this pursuit, this work demonstrates how electromagnetic simulations can be facilitated with the aid of the ANSA and META pre- and post processors. In this presentation a detailed analysis for two simplified yet representative and accurate examples of the most commonly used batteries is demonstrated, from the pre-processing of the geometry to solving using the LS-DYNA electromagnetic solver, and to then visualizing the results. The results are really encouraging, reflecting real electromagnetic values, such as the discharge rate, the current and the voltage of the battery, and reveal ways to speed up production process, reduce the cost and the environmental impact of real testing.

Computational electromagnetics with ASERIS-BE™

[1]Benoît Chaigne*, [2]Christos Liontas
[1]IMACS,
[2]BETA CAE Systems

KEYWORDS -
BEM, Surface Mesh, H-matrix, FMM, EMC

ABSTRACT
ASERIS-BE(TM) is a simulation software developed by Airbus and IMACS dedicated to Electromagnetics based on the resolution of Boundary Integral Equations (BIE) derived from Maxwell's equations in the frequency domain. It is used for a wide variety of industrial applications such as antenna design, antenna siting, Electromagnetic Compatibility applications, Lightning Indirect Effects (LIE) or High-Intensity Radiated Fields (HIRF)…

Approximation of the BIEs by a conforming H(div) finite element method leads to the so-called Boundary Element Method (BEM). It offers many advantages appreciated by engineers: using the Green's kernel, the numerical approximation is very accurate and only requires the meshing of the interfaces between different homogeneous domains where we look for equivalent electric and magnetic currents. Electric and magnetic fields can then be deduced using the integral representation formulas in a post-processing phase.

This approach can handle perfect or imperfect conductors, dielectric materials, resistive or composite sheets, electric wires, ground plane and symmetries, voltage sources, lumped elements, waveguides, incident plane waves, point sources… The number of degrees of freedom grows with the frequency and the complexity of the object under study. As the BEM matrix is dense, the problem can quickly become unworkable with traditional linear solvers. ASERIS-BE(TM) implements efficient parallel fast solvers like H-matrix or Fast Multipole Method (FMM). Thus, preparation of the model with the appropriate interface properties and an optimal mesh is essential for complex simulations. We will present some use cases leveraging the recently developed ASERIS plugin in ANSA for an optimised setup of the simulation.

Thermal simulation of PCBs using ANSA ECAD Importer, ANSYS Fluent and META

[1]Christos Liontas*, [2]Alexis Sanjaime
[1]BETA CAE Systems,
[2]Thales

KEYWORDS -
Electronics, Thermal, CFD, PCB, ECAD

ABSTRACT
Thermal simulations of printed circuit boards (PCBs) are becoming increasingly crucial in design and pre-deployment testing of electronic products. High temperatures cause over 50% of electronic equipment failures, according to a study by the US Airforce Avionics Integrity Program. Therefore it is very important to have robust and reliable simulation methods to assess the risk of a thermal failure and to eliminate or mitigate it. We present a novel workflow that uses ANSA for pre-processing and ANSYS Fluent as a conjugate heat transfer solver to perform accurate and efficient thermal simulations of PCBs. The main tool that is used for preparing the FE model is the ECAD Importer plugin of ANSA that can produce a high-quality structured solid mesh with the metallization of the PCB mapped onto it. This so-called "trace mapping" method reduces the simulation footprint (CPU time and memory) significantly while maintaining adequate accuracy for thermal failure predictions.

Automating meshing and thermal analysis for an EV battery pack with ANSA TAITherm and CoTherm

[1]Sacha Jelic, [2]Daniel Woodford, [2]Eli Datema
[1]ThermoAnalytics GmbH,
[2]ThermoAnalytics Inc,

KEYWORDS -
Thermal, EV, Battery Pack

ABSTRACT
Automated steps for thermal meshing and modeling will be demonstrated for rapid design studies on an electric vehicle battery pack. This approach includes automated meshing preparation steps in ANSA with quality checks, mesh remediation and thermal model creation. Thermal boundary conditions are automatically applied to the model and the process automation tool CoTherm is used to iteratively evaluate battery pack design changes and report results to the user.

Session 8

How to optimize the design of a car-body-structure by using Machine Learning

[1]Fabiola Cavaliere*, [2]Dr. Xabier Larrayoz
[1]SEAT S.A.,

KEYWORDS -
Machine Learning, Proper Generalized Decomposition, Optimization, Parametric Modeling

ABSTRACT
The future of the automotive industry is electric. Guaranteeing the perfect match between sustainability and performance is our challenge. A feature which plays a crucial role in this challenge is the weight of the car. Lighter structures mean higher battery autonomy, as well as lower production costs for the company. At the same time, weight reduction can lead to the deterioration of the car functional properties. Standard vehicle development projects are still driven by trial-and-error methods. Based on experience, car body designers propose tentative configurations of the structure, which are then tested by the simulation team. Inevitably, multiple configurations need to be tested before a satisfactory design is reached. Moreover, due to time constraints, only few configurations can be analyzed, which limits our understanding of the problem and ignores potentially better solutions.

Artificial Intelligence (AI) and Machine Learning (ML) techniques offer new ways to push our boundaries. They are based on the idea that we do not need complex and time-consuming models to identify patterns in the behavior of a structure. The method developed in the context of this work uses this principle to maximize the stiffness and comfort behavior of a car while minimizing its weight. Based on a machine learning approach known as Proper Generalized Decomposition method (PGD), the tool self-learns how to approximate the solution of a complex problem depending on a set of design parameters (material/geometry properties of car components). It consists of three main phases. First, it parametrizes the model. Next, with only one computation, it automatically computes a parametric solution which contains the results for every possible combination of predefined design variables. This parametric solution is then used to perform fast optimization analysis. All results are uploaded to an interactive app, where users, both technical and non-technical, can explore in real-time how changes in the design variables affect the car performance and make decision accordingly, thus drastically reducing the repetitive iterations in the development process and improving the quality of the final solution.

Machine Learning Tools for accelerating CAE and unleashing Design Exploration

Dimitris Drougkas*, Nikolaos Tzolas
BETA CAE Systems

KEYWORDS -
Optimization, Optimization Tool, Machine Learning, Results prediction

ABSTRACT
Machine learning tools have become increasingly prevalent in various software applications, offering numerous advantages and transforming the way we interact with technology. The challenge in the CAE-domain is to adapt and learn from the huge amount of Simulation data plus demanding load cases and complex physics. The ML-workflow which involves data collection and preparation, ML-model training and evaluation, and finally Prediction, is integrated into the BETA Products since v21.0.0.

The Prediction of scalar, 2d and 3d Simulation results give us the opportunity to use Machine learning for variety of applications.

The ML-based isolation of complex design components such as fasteners and clips can significantly reduce the modelling time.

Predictors as the ML-object in BETA Products enable CAE engineers to rapidly explore various design configurations without relying on time-consuming simulations.

Engineers can now optimize their processes, streamline workflows and expedite the realization of innovative designs. In this presentation we will present the latest features along with some of the most important applications.

Session 9A

Latest & future developments in Crash and Safety

Thanassis Fokylidis*, Thanassis Lioras
BETA CAE Systems

KEYWORDS -
Checks, Numbering, Highlight-Drawing, Includes, Ids handling, Pedestrian, Occupant, Interior

ABSTRACT
Explore the latest and upcoming advancements in Crash and Safety features within ANSA and META. Numerous improvements have been made in configuring and managing Crash models. Witness the enhancements made to the Checks Manager, numbering tools, Database Browser, Includes handling, drawings, highlight effects, the merging of deck files but also all the ones regarding post processing of crash models. Additionally, we will showcase all the recent updates incorporated into our suite of safety tools for Occupant, Pedestrian, and Interior impact. Simultaneously, we will unveil our future plans and goals for the aforementioned tools.

Optimization and quick verification of an electric vehicle side-frame design using Machine Learning methods

Christina Chatzigeorgiadou*, Athanasios Papadopoulos
BETA CAE Systems

KEYWORDS -
Machine Learning, ML, EM, Electromagnetics

ABSTRACT
Before being released to the market, all vehicle prototypes are validated in terms of their crashworthiness. Meeting the safety standards, while same time avoiding compromises in other essential design parameters, requires a very meticulous engineering simulation approach during product design. These processes become even more complicated, and time-consuming, with electric vehicles, such as Lithium-ion battery-powered cars. In many cases to accomplish the safety aims during product design, while also meeting time limitations and deadlines, sophisticated simulation tools need to be employed. Such tools are those that enable optimization studies and that take advantage of Machine Learning capabilities.

In this study, an optimization and a quick verification of an electric vehicle side frame design were performed with the aid of an Optimization tool and Machine Learning methods. Through this Optimization tool, several Design Experiments have been created and then, by training a Machine Learning model (referred to as a “Predictor”), the optimal design parameters were approached for the given objective and constraints. Since during the designing stages, the geometry can be often modified, the proposed approach saves a considerable amount of time, as it avoids repeating the complete ML Optimization process or solving each updated model individually.

To accomplish this, Transfer Learning methods were utilized in order to employ the already trained ML predictive model to verify and optimize the updated geometry. In this way, the optimal design for the updated model was calculated, avoiding re-training an updated Predictor by producing new data sets. The use of this already trained Predictor extends also to the field of a quick verification of the newly updated designs. The several design modifications were quickly tested without needing to solve the model again.

To further reduce simulation time and modeling effort, a macroscopic battery model was used. This way, the ML Predictor was able to also consider the electromagnetic phenomena related to damaged batteries without increasing the solution time of the side-crash simulation. All in all, using the Machine Learning based Optimization tool, and the Transfer Learning related functionality, an already trained predictive model was able to estimate the optimal design of the vehicle with updated components and verified the updated designs without having to re-run the complete optimization and solution processes.

Prediction of Occupant safety utilizing Machine Learning and CARLA Autonomous drive simulation software (CARLA Integration)

Dimitris Drougkas*, Panagiota Kagioglou
BETA CAE Systems

KEYWORDS -
Optimization, Optimization Tool, Machine Learning, Results prediction

ABSTRACT
Currently simulations of crash tests, as physical crash tests, are based on “lab” environment, prescribed scenarios, and regulations. Load-cases and boundary conditions are carefully and specifically defined to “assure” safety to the occupants and pedestrians up to acceptable limits, and to certify the vehicle for road use. Although these design methods and safety protocols have been constantly improving vehicle safety, no “real-case” crash scenarios are being tested, which could contribute to the improvement of the vehicle’s safety.

In the presented work, “real-case” data from an autonomous driving software were used for the crashworthiness simulation, beyond the regulated scenarios, with the employment of Machine Learning.

CARLA software is a simulator for autonomous driving that can reconstruct and simulate real-world traffic accident scenarios, with various vehicle types, and provide pre-crash data such as: speed, position, and angle. Such data from reconstructed accident scenarios are used as input in Finite Element Crash analysis to provide results concerning occupant injury. Datasets with various FE models and various crash scenarios measuring occupant safety are created to train Machine Learning models that will be able to predict the occupant injury, having as input the pre-crash data of a simulated accident scenario. These Machine Learning models are used to optimize the control of occupant safety parameters such as, airbag deployment time, and seatbelt trigger, to achieve lower occupant injury criteria.

In this study, CARLA provided data of a specific vehicle on a rear crash scenario, one of the most common accidents. The input parameters were speed, velocity, and the relative position of the two vehicles. FE analyses run for several variations of the crash, measuring the occupant injury, and creating an adequate dataset to train a Machine Learning model.

The trained Machine Learning model was then used to predict the occupant injury criterion based on various inputs from CARLA, and to optimize the safety systems parameters to achieve safer designs.

Currently simulations of crash tests, as physical crash tests, are based on “lab” environment, prescribed scenarios, and regulations. Load-cases and boundary conditions are carefully and specifically defined to “assure” safety to the occupants and pedestrians up to acceptable limits, and to certify the vehicle for road use. Although these design methods and safety protocols have been constantly improving vehicle safety, no “real-case” crash scenarios are being tested, which could contribute to the improvement of the vehicle’s safety.

In the presented work, “real-case” data from an autonomous driving software were used for the crashworthiness simulation, beyond the regulated scenarios, with the employment of Machine Learning.

CARLA software is a simulator for autonomous driving that can reconstruct and simulate real-world traffic accident scenarios, with various vehicle types, and provide pre-crash data such as: speed, position, and angle. Such data from reconstructed accident scenarios are used as input in Finite Element Crash analysis to provide results concerning occupant injury. Datasets with various FE models and various crash scenarios measuring occupant safety are created to train Machine Learning models that will be able to predict the occupant injury, having as input the pre-crash data of a simulated accident scenario. These Machine Learning models are used to optimize the control of occupant safety parameters such as, airbag deployment time, and seatbelt trigger, to achieve lower occupant injury criteria.

In this study, CARLA provided data of a specific vehicle on a rear crash scenario, one of the most common accidents. The input parameters were speed, velocity, and the relative position of the two vehicles. FE analyses run for several variations of the crash, measuring the occupant injury, and creating an adequate dataset to train a Machine Learning model.

The trained Machine Learning model was then used to predict the occupant injury criterion based on various inputs from CARLA, and to optimize the safety systems parameters to achieve safer designs.

Session 9B

Structural optimization of wing elements of an unmanned air vehicle

Andreas Psarros*, Georgios Savaidis
[1]Lab of Machine Elements and Machine Design, AUTH

KEYWORDS -
Lightweight structures, Composite materials, Optimization, Blended wing body, UAV

ABSTRACT
The significance of lightweighting in aerospace industry is undeniable. Besides lowering the material costs, lightweight aerospace structures result in increased flight time and lower impact on the environment. Unmanned aerial vehicles (UAVs) display several advantages, largely because of the absence of crew on-board reduced design requirements, reduced operational cost, ability to operate under hazardous conditions and increased flight endurance. Modern aerospace structures are consisted of multilayered laminated surfaces which are fabricated easily due to the evolution of the manufacturing methods.

In this study, the wing module of an UAV made of composite materials has been chosen. The main goal is weight minimization while maintaining the highest stiffness possible. Air pressure distribution and acceleration loads, from CFD simulations, have been included in the procedure, describing loads acting to the structure during flight. During the optimization task, for each layer, both thickness and orientation are used as design variables. Compliance, stiffness described by displacements and failure index values are used as constrains during the procedure. Results are obtained by utilizing extensive use of automatization tools involving Python scripting in both ANSA pre-processor and META post-processor. They include both thickness and failure index distribution over the surfaces.

FE analysis of closed cell aluminum foam under impact loading

[1]Nikolaos Michailidis*, [2]Christos Tegos, [1]Emanouil Smyrnaios, [1]Fani Stergioudi, [3]Georgios Maliaris
[1]Physical Metallurgy Laboratory, AUTH,
[2]BETA CAE Systems, [3]Additive Manufacturing Laboratory, International Hellenic University

KEYWORDS -
Closed-cell aluminium foams, 3D Voronoi model, Deshpande-Fleck model, Finite element method, Impact test

ABSTRACT
Metal foams have been intensively explored for possible application in many industry domains like automotive, naval, aerospace, and biomedical due to superior mechanical energy absorption combined with their lightweight design. It is still challenging to predict the mechanical response of metal foams under impact loads, and their practical use in engineering often involves lengthy experimental testing. Numerical models based on the finite element method (FEM) can contribute to minimizing the experimentation effort.

In this work, FEM models were built that explain the acting mechanisms that take place during impact, study the yield properties as well as the energy absorption during the impact of closed-cell aluminium foams. The simulation results are compared with the ones derived from respective experimental uniaxial tests. Two different modelling approaches were applied thus creating two models. The first model relies on a cell-based method where the initial geometry of the foam was generated based on the Voronoi tessellation algorithm. The generation of the solid elements for this model was optimized using automated tools of advanced pre-processor ANSA software package by setting appropriate mesh parameters and quality criteria in order to keep the exact shape of cells. The second one relies on the isotropic, strain-hardening, and continuum-based model developed by Deshpande-Fleck. The FE results were visualized and processed in META post-processor. The outcome of the investigation sheds light on the metal foam behaviour under impact by explaining macro- and micro-structural phenomena that develop during impact.

Simulation of shot peening implementing stochastic aspects

[1]Christos Gakias*, Georgios Savaidis
[1]Laboratory of Machine Elements and Machine Design, AUTH

KEYWORDS -
shot peening, finite element method, residual stresses, stochasticity

ABSTRACT
Shot peening (SP) is a widely used surface treatment process that involves the use of small spheres to impact the surface of a component, resulting in a localized deformation and the development of a compressive stress field within the surface layer. This compressive stress field can extend up to 300-400 μm in depth and significantly improve the fatigue life of the component. However, the effectiveness of the SP process can be influenced by various stochastic parameters such as the variability of shot diameters and shot velocity.

In this study, a structured modelling approach based on the Finite Element Method (FEM) is introduced to account for these stochastic parameters alongside with elastic-plastic behavior and accurately simulate the SP process. The modelling process is semi-automated, with discrete stages including problem definition, setup, simulation, and results extraction using ANSA and META software.

The validation of the model is also presented and includes comparison with experimentally determined data. The results demonstrate the effectiveness of the proposed approach in capturing important process parameters and highlighting the impact of stochasticity on the SP process. Moreover, the model was shown to be capable of examining the effect of various process parameters on the resulting surface residual stress field, which is crucial for the fatigue life of the component.

Overall, the structured modelling approach introduced in this study can provide valuable insights into the SP process and aid in the optimization of SP parameters for different applications, leading to improved fatigue life and durability of components.

Session 9C

NEERE: Beyond maturity, as a communications and training platform

Gerasimos Skouvaklis
BETA CAE Systems

KEYWORDS -
NEERE, colloboration, training

ABSTRACT
Effective and secure communication between team members became paramount during the COVID-19 outbreak, as physical isolation was imposed. NEERE matured not only in terms of user experience, but also in training capabilities, stability, communication productivity features and more.

This presentation deals with the most significant developments in NEERE and how these can help you improve your training sessions and boost your team's productivity by minimizing the time needed to efficiently:

- transmit applications settings to collaborators
- share files with roommates
- organize the conversation flow
- manage the room's access privileges
- allow external users to join a room
- follow along multiple trainees' screens
- protect a roommate's privacy
- focus on the presenter's video
- configure the features specific user groups may access
- eliminate browser compatibility issues

Eliminating physical clamping processes with the aid of Engineering Simulation

Panagiotis Pantazidis
BETA CAE Systems

KEYWORDS -
Virtual Clamping, Epilysis

ABSTRACT
During stamping, the quality of produced parts is usually not identical to the designed models. A standard procedure, performed by most OEMs to measure the quality of produced parts, is to clamp the part in predetermined measuring mounts, and then measure it using 3D scanning. Subsequently, a comparison is made between CAD and measured data.

In this process, the initial step is the positioning of the sheet-metal part in a dedicated device. After the part is clamped in predefined positions, a scanner creates its digital footprint. Then, the digital part is compared to the original CAD part to evaluate the differences. This approach requires high level off expertise due to the complexity in parts' geometry, in the assembly process when positioning the part, and in the calibration of the tool as well.

This process can be replaced with engineering simulation. This way, both time requirements and complexity are considerably reduced, removing also the expensive devices required. In this presentation, such a simulation technique is demonstrated. In this new technique, named Virtual Clamping, the stamped part is scanned in a free state and the clamping process is generated through a CAE simulation. The comparison is then run between the virtual results and the CAD data.

For this new technique, a new tool has been developed to make use of the capabilities of the ANSA pre-processor, EPILYSIS solver, and META post-processor. This tool evaluates the estimated displacements between the nominal CAD part and the clamped manufactured part. It can extract several pieces of information from the CAD model, load the scanned model, and through a user-friendly wizard prompt all needed selections to run the virtual clamping process. Among available options, the user can also select whether to take gravity under consideration.

Breaking down the interoperability barrier among different FEA software

Thanasis Fassas*, Georgios Mokios
BETA CAE Systems

KEYWORDS -
VMAP, interoperability, ANSA

ABSTRACT
Multidisciplinary engineering simulation workflows involve several analysis types and FEA solvers. Most of these analyses share a common geometric model, while in many cases, the results of an analysis are required as an input for a subsequent one. Such a landscape may require the repetition of some modeling work for different analysis types. This brings up significant interoperability issues among different solvers. A way to address the above, is to employ advanced pre-processing capabilities to build and maintain a single analysis model for several FEA solvers at the same time, and to export solver-specific input files for each analysis. A further suggested step, which eliminates the necessity of exporting and maintaining solver-specific files, is the employment of a standard neutral file-format. Files in this standard neutral file-format, exported by the pre-processor and then properly supported and interpreted by the different FEA solvers, can reduce the repeating modeling burden and the interoperability issues. Such a file-format, if capable to migrate the results of one analysis as an input to the next one, also reduces the effort of the additional analysis model preparation effort. Notably, model consistency can be assured. Model updates, applied upon a single unified file, can be directly propagated to the different solvers, reducing error-prone multiple analysis model updates. Furthermore, the use of a single unified model, in a neutral format, facilitates the co-operation between engineering teams. As expected, the overall digital product development procedure is accelerated, and its quality is improved. VMAP is such a standard format for CAE data storage, based on HDF5 (High-performance data management and storage suite), with the aim to incorporate, in a solver-neutral manner, user data, meta-data, geometry and discretization, coordinate and unit systems, results and state variables, parameters of material models etc. This presentation, demonstrates the setup of an interoperable solver-neutral file in VMAP standard format, and the employment of it through complex simulation workflows. Moreover, the presentation examines how a multidisciplinary pre-processor, with multiple FEA solver interoperability capabilities, can accommodate the requirements of this approach.

Session 9D

Exploring the latest developments in RETOMO

Karatsis Evangelos
BETA CAE Systems

KEYWORDS -
RETOMO, CT Scans, Computed Tomography

ABSTRACT
The introduction of real-world objects into the Computer-Aided Engineering (CAE) domain is becoming increasingly crucial across industries. Computed Tomography (CT) emerges as a powerful non-destructive method to address this need, even for the most intricate objects. Our solution, RETOMO, continually evolves to tackle new challenges, providing enhanced functionality for various applications.

This presentation aims to showcase the latest advancements and future prospects of RETOMO. The talk will focus into key areas of improvement, including handling extra-large CT scans, mapping non-homogeneous material properties, detecting assembled parts (flanges), and introducing advanced semi-automatic segmentation tools. Moreover, the algorithms powering RETOMO have been optimized to enhance efficiency and resource management.

Attendees will have the unique opportunity to explore the cutting-edge developments within RETOMO. The presentation will offer an in-depth examination of the latest features, demonstrating how RETOMO addresses industry demands and paves the way for future advancements.

Latest developments and future roadmap of the BETA Development Platform

Yianni Kolokythas
BETA CAE Systems

KEYWORDS -
Platform, Scripting, Python, API, Extensions

ABSTRACT
CAE tools need to provide advanced open programming capabilities where specialists can provide rapid solutions. Continuing the effort to further improve the capabilities of the development platform, improvements and new functionality is introduced. A new Python version is embedded in the BETA Suite that brings faster code execution, better debugging capabilities and improved programming paradigms. The latest APIs, recently developed functionality, tools for code development and debugging, and the future development roadmap will be presented.


Demo Sessions

Session 3E

DEEN for simulation driven design

Gerasimos Skouvaklis*, Dimitris Chanopoulos

The exchange of information between CAD and CAE engineers has always been a challenge, leading to lengthy design cycles and increased project costs. To address this issue, BETA CAE Systems developed DEEN, a collaboration tool that bridges the gap between CAD and CAE worlds, offering an easy-to-use application for designers to understand the proposals coming from their CAE colleagues.

In this live demo, the expert engineers behind DEEN, will provide a hands-on demonstration of its capabilities, showcasing how it can help designers and engineers communicate and collaborate more effectively. Participants will get familiar on how to use DEEN's tools for asynchronous and concurrent collaboration, allowing them to provide feedback, comments, and views on design proposals.

Machine Learning Design Variable and Feature-based training and simulation results prediction

Dimitris Drougkas

This demo will be a detailed description of data collection and preparation, Machine Learning Training and Prediction using BETA CAE products. The data will be created in ANSA or imported directly in KOMVOS. Data driven Design variable based ML Training will be demonstrated, providing predictions for scalar, 2d, 3d simulation results. Feature based ML will be demonstrated for rediction of elastic mode frequency values on CAE data as well as Mode classification prediction on CAE data.

Session 3F

Latest developments in Kinetics

Apostolos Paraschoudis

Multibody dynamics simulations (MBD) have always been the standard approach to predict the behavior and evaluate the overall dynamic performance of a system. Given that systems become more and more complex with many interconnected parts, MBD simulations become more demanding while the requirements for quick solutions, design validations and what-if scenarios remain. This demo session will mainly focus on topics such as: -Linearizing a system in order to determine its eigenmodes and evaluate the dynamic characteristics

-Performing optimization studies to improve a design according to the imposed design constraints

-Collaborating with FATIQ software to provide a seamless integrated durability solution from scratch

Simulation coupling and transferring of results across calculations

Ioannis Nerantzis

This demo showcases the capabilities of the Results Mapper tool in ANSA.

The general concept of this tool is to pass information from source to target model with the use of interpolation methods. Organizing this process leads to significant time reduction of loading conditions definition for several field result types.

Applications of this tool extend to multiple data types including: thickness, stress-strain initial conditions, pressures, temperatures, material orientation and lamination, and across disciplines such as CFD, NVH, Durability, Stamping and Crash.

Transcoding data is also applicable during the mapping process which enables the analyst to easily pass data across different solvers, from one simulation model to another. During this process a number of solver file formats is supported. Arbitrary ASCII file formats may also be treated by utilizing the ANSA python scripting API.

Matching or similarly shaped geometry is the only pre-requisite to pass data across different mesh types (quad/tria, tetra/penta-hexa), and entity types (nodes/shells/solids). Auto-positioning, symmetry and alignment, units conversion, tensor results transformation, element normal realignment, anisotropic material orientation and homogenization are some of the data transformations that can be performed during the process to condition and conform data between source and target. Three interpolation methods: Shape functions, Common Area , RBF and raw transfer of results are applicable depending on the type of mapping that is performed.

The whole process is organized in a generic entity in ANSA which, once defined, it can be easily applied/erased and allows for quick re-application. This achieves an overall CAE turnaround time reduction by preparing numerous mapping scenarios with the minimum effort and avoiding redefinition of the similar mappings on different models.

Dedicated plugins built on top of the results mapper tool are available to aid a. perform mapping from data stored in a META results database, b. review of SOL200 design experiments by updating the model from a specific optimization cycles and c. perform mapping of the lamination information of composite parts directly from CAD formats.

Session 3G

META Automation: best practices and proven techniques

Emmanouil Kastrinakis

Join us for a 30-minute demonstration where we explore the features and functionalities of META for automating your work. This interactive session will cover various aspects of automation, providing practical insights and useful tips. We will start by addressing frequently asked questions about automation, including how to pass arguments from the command line. Then, we will gradually introduce you to the tools and techniques already available in the software. You'll learn about the Template Manager, View Best commands, handling Annotations, and exporting high-quality images.

As we progress, we will delve into dynamic variables and built-in functions, showcasing their usefulness in automating tasks. We will also explore session files and how to use image placeholders in presentations. To create intuitive interfaces, we will discuss when to use User Toolbars or the guitk module. And, we will address how to decide to use a session file or Python.

When it comes to scripting, we will cover essential topics like executing scripts and disabling redraw safely. You'll discover how to leverage the power of NumPy for data collection and calculations. We will also explain the advantages of using META's Python API functions instead of session commands, particularly for performance analysis using tools like cProfile.

Additionally, we will touch upon using attributes to store information efficiently and discuss the potential of storing themes and styles for reusability.

This demonstration aims to provide you with practical guidance and insights into automating your tasks with META Post Processor Software.

Join us to enhance your automation skills and optimize your workflow effectively.

Script development and package management using Virtual Environments and VSCode.

Klearchos Thomopoulos

This session will focus on the capabilities of VSCode, which has been integrated into the BETA Suite, as well as the advantages of creating a Virtual Environment, in order to manage the Python libraries for different projects.

Session 5E

A new approach for Geometry and Mesh handling

Dimitris Mavropoulos

Since v23 a new approach for geometry and mesh handling has been introduced that allows the user to directly work on the geometry / mesh without the need to continuously activate functions from the module menus. The suggested way of working with ANSA will be demonstrated that includes the direct manipulation of geometry, creation of middle surface and the generation and review of the shell mesh.

Understand and exploit the capabilities of Batch Meshing

Nikolas Drivakos

ANSA Batch meshing has proven to be one of the most effective meshing tools available for the CAE engineer. Many OEMs have set up automatic processes that batch process tenths of thousands of parts overnight. The results of these processes are meshed assemblies that can be used with no or minimum interaction for setting up a simulation model. At the same time, batch meshing is the suggested tool to manually generate a mesh of high quality for any type of analysis, from crash and durability, to NVH and CFD. Since each analysis had different requirements the quality of the results depends solely on the combination of mesh parameters and quality criteria used for the meshing.

In this demo you will be able to understand better the capabilities of ANSA Batch mesh, be updated with our latest developments and have a glimpse into our future plans

Efficient middle meshing

Dimitrios Christias

Rapid technological progress leads to new challenges, like tackling with different scopes of meshing strategies for parts of different design characteristics such as aluminum casting parts, injection molding or over-molding, tailor rolled and tailor welded blanks, extrusion profiles and many others. Moreover, complete subsystems of sheet metal parts are being replaced by complex aluminum megacast parts. ANSA is well known for the best in class middle meshing both in terms of quality and efficiency. In this demo we will show the full capabilities of ANSA in middle meshing and how the user may benefit in three different ways: save time, assure quality and exploit similarities between parts (updated CAD version of a part, left side - right side parts, symmetrical parts). Along with our latest developments we will also present you our future plans for “middle” meshing.

Structured meshing: Challenges and solutions

Dimitris Mavropoulos

Over the years the request for more detailed simulations is increasing. The analysts, in their effort to better simulate reality, try to include more parts and at the same time keep as much detail as possible. This is encountered more in assemblies of electronic devices that nearly all components, even screws and bolts are included and are solid meshed. For this reason, the demand for fast and easy hexa and penta meshes has been significantly increased since, compared to tetrahedral mesh, the user gets very good results with significantly less elements.

In this demo the full capabilities of ANSA for structured meshing will be demonstrated. This includes tools like MapBlock that automatically identifies and meshes mappable volumes, tools to automatically decompose volumes as well as the best practices to create efficiently a hexa/penta mesh. For axisymmetric parts, how to generate a revolute mesh, with special core pattern and how to edit and modify existing revolute meshes. Along with our latest developments we will also present you our future plans for “hexa/penta” meshing.

Session 5F

ANSERS use cases

Antonis Perifanis

This demo showcases the ANSERS web application's features, which offer an easy and efficient way to access, visualize, correlate, and share engineering simulation and physical test data. The demo includes various dashlet cases that demonstrate the tool's functionality, such as generic viewing of models, results, or test data, loadcase templates for comparing crash, NVH, and stiffness analyses and overview dashlets about the status of a project.

ANSERS is a web application designed to simplify the process of accessing and combining engineering simulation and physical test data, providing a comprehensive view for efficient engineering design decision-making. It efficiently handles large volumes of data generated by simulations and physical tests stored in a data management system.

ANSA and META for Occupant Safety

Manos Dagdilelis

During the vehicle's development and design phase, ensuring the safety of occupants is a crucial aspect that undergoes standard analysis. At the same time, the number of relevant load cases has significantly risen due to the continuous introduction of new legal tests and regulations. ANSA and META provide the appropriate tools for efficiently managing the dummy-seat system.

The ongoing demonstration session primarily focuses on various aspects, including seat positioning in accordance with different regulations, placing the dummy on the seat, and effectively restraining the entire system using seatbelt entities. Lastly, the META Occupant Injury tool will be showcased to illustrate how injury criteria can be evaluated easily and reliably.

Modular Run Management for Pedestrian / FMVSS201U analysis

Antonis Perifanis

This demo workshop showcases BETA CAE Systems' new modular run management approach for pedestrian and interior safety analysis. The approach provides better visual control of different variants and their results. The data management system interacts closely with the pre-processing pedestrian and FMVSS 201U marking tool to offer seamless simulation generation. The tool streamlines the extraction of key performance indicators, such as the Head Injury Criterion, their storage in the data system and their overview visualization through META's pedestrian or FMVSS 201U evaluation tools. The tools include dedicated functionality for comparing variants while retaining links to their respective runs, models, and subsystems in the data management system. This demo showcases the enhanced efficiency and ease of use of the modular run management approach, improving the overall quality of pedestrian and interior safety analysis.

Session 5G

FMI/FMU: Making detailed CAE simulation models accessible to all Engineers

Dimitrios Daniil

BETA SUITE enters Model-Based Design (MBD). This demo showcases how to combine our software products with multi-disciplinary simulation and MBD. Exploit our advanced FEA tools of high level engineering detail and Artificial Intelligent technology by incorporating them into system-level processes, via the Functional Mock-up Units (FMUs) capabilities for Co-Simulation or Model Exchange.

ANSA's Graphview Tool

Eleftherios Daskalakis

ANSA's Graphview is a powerful tool for illustrating connectivity in complex systems as an intuitive and customizable graph. With support for a wide range of components, from higher level containers like Includes and Subsystems to generic containers that group entities at a lower connectivity level, Graphview makes it easy to identify and understand even the most intricate connections in your model.

Graphview is fully integrated in ANSA, featuring synchronized selection and visibility controls. It can be used either as your main standalone working tool or boost your usual selection and visibility controls, allowing much easier highlighting and isolating specific areas of interest.

Graphview is more than just a powerful visualization tool. It is constantly supported with new features and abilities that aim to assist the user with complex and what would otherwise be time-consuming tasks. Some of our latest features include:

•Interactive exploration of a model in a step-by-step fashion, with progressive clustering and history support, making it easy not only to check connectivity on specific areas of interest, but also navigate through consecutive exploration states.

•Partitioning automation of a finite element model to Includes. Clustering in Graphview is not used only for visualization purposes. Users are now able to convert these groups to Includes and partition their model to their liking in no time.

•Pattern recognition for analyzing and classifying element formations into structural or connecting, helping users gain a deeper understanding of the underlying structure of their model.

•Automated connection checks with validity indicators, status reports and more.

In short, with its intuitive interface, powerful visualization capabilities, and advanced analysis features, ANSA's Graphview is an essential tool for any user looking to streamline their workflow and gain deeper insights into their models.

Design tools for the Analysts

Serafim Chatzimoisiadis

The new "Design" module, among others, provides the analyst with interactive tools to modify models without going back to the Designers.

Building new members, reinforcements, joints, depressions, ribs or openings are now simple actions within the pre-processor.

Create new Members either based on standard or user-defined cross sections or starting from pre-existing ones.

In a fully interactive User Interface, define precisely both the location and the dimensions of the new component, even from the preview .

Generating and adapting new Flanges, spreading SpotWelds, specifying mesh parameters etc. are some of the several attributes and powerful applications included in the same environment.

Furthermore, the generated components are stored as a stand alone ANSA entity.

Thus, it is easy to parameterize their attributes and handle them through the ANSA Optimization Tool.

Session 6E

Key features and use cases of FATIQ

Christos Tegos

FATIQ is a new fatigue analysis software built on a versatile solver-neutral framework, designed to simplify and streamline fatigue assessment processes. In this conference demo presentation, we showcase the software's capabilities through two widely recognized industrial use cases. The first use case focuses on vibration fatigue analysis, where a single Power Spectral Density (PSD) loading is applied. The second use case delves into a pseudo-static scenario, where a complex loading pattern is represented by a duty cycle. By showcasing these examples, will gain insights into the process of load definition, materials assignment, and comprehensive post-processing functionalities.

Introducing the redesigned Representative Volume Element (RVE) Generator Tool

Vangelis Palaiokastritis

Introducing the redesigned RVE Generation Tool: a powerful tool for digital material characterization. Explore and examine various microstructure types of Composites Materials, generate Representative Volume Element (RVE) models and perform FE Homogenization. Join us to learn about the new layout and enhanced features of this tool.

Session 6F

New optimization capabilities in the NVH-Console

Grigorios Kalampoukas

Optimization in general is an important and effective tool used in the development process of automotive structures.

The optimization capabilities in NVH-Console offer a wide range of functionalities that enable users to streamline and improve the NVH characteristics of a vehicle. These capabilities encompass the use of advanced algorithms, simulation techniques, and data analysis tools to identify and address noise and vibration issues effectively. By utilizing these optimization features, engineers can systematically refine vehicle components, systems, and overall vehicle design to achieve optimal NVH performance.

In this demonstration it will be shown how within NVH-Console the physical properties of connecting elements between different components of a vehicle, e.g. bushes or even hydro mounts, as well as modal properties of components can be used as design variables (DV). The design responses and the design objective consist of acoustic and tactile transfer functions. Due to the assembly process and certain reduction methods used in NVH-Console this kind of optimization can be run very effectively. The optimization process itself or alternatively a Design of Experiment (DOE) is controlled and driven by the ANSA optimization tool. Also the result evaluation is being performed in the ANSA optimization tool, whose enhanced analysis tools will be showcased.

Introducing Statistical Energy Analysis support in ANSA

Iraklis Gkountas

Statistical Energy Analysis (SEA) is a simulation method that is particularly fitting to High Frequency Vibroacoustics. SEA overcomes the limitations of Finite Element Analysis (FEA), due to element size and mode count requirements through the use of idealized analytical formulations for the various system components, such as plates or beams. Furthermore, the required results are averaged over the span of a component and over a range of frequencies, usually octave bands. As several research and commercial SEA solvers are available, ANSA incorporates support for the creation and editing of SEA entities, input and output, as well as, various tools for the review and checking of such models. In particular, 3D and 2D SEA components can be created using existing shell meshes, or by direct boundary definition. The respective property and material entities can also be created and assigned to SEA model entities. Furthermore, area, line and point junctions can be defined by selecting the components to be joined. Existing SEA legacy models can be imported through popular file formats used by the industry. Additionally, standard functionality supports the process of creating SEA models through the modification of existing FEA models. In particular, mesh functions, such as ISOLATE, MESH TOPO, FILL HOLE, e.t.c. can be used to transform an existing NVH FE model to SEA subsystems. Existing properties and materials are properly carried to the new model. The required SEA junctions are then created from the model connectivity. ANSA provides several SEA model review and check tools. In particular, graph visualization can reveal errors in the model connectivity by visual inspection and comparison with other models. Finally, standard SEA quantities, like Modal Densities and Coupling Loss Factors can be calculated. For further model review and testing, simple simulations can be performed for the calculation of energy levels under simple power inputs.

Session 6G

Find optimal designs of your parametric ANSA model using the embedded Optimizers in the Optimization tool

Eva Ioannou

Explore the Optimization tool's capabilities to define a ready to run parametric model. Perform a DOE and post process the result data using charts, histograms and correlation matrixes. Set up Optimization Studies using various optimization algorithms, find the optimum designs and compare the results. Use Machine Learning or other methods to create Response Surface Models (RSM), use them in the optimization and reach the optimum design faster.

An application example of the next generation approach to Crash Simulation Optimization, incorporating CAD-Design parameters

Zafeiria Kanellia

Improving the CAE-function of car components from a crash simulation perspective is a challenge on its own, due to the high non-linear effects that are taking place. With such pre-processing capabilities like ANSA morphing, a whole set of capabilities is available to the CAE-Engineer to investigate the shape of the geometry so and to understand how it affect its crash performance and through this knowledge to proceed in improving it. Nevertheless, in the automotive industry, the CAD and CAE departments often face communication challenges, especially from CAE back to CAD. There are obvious fundamental differences in the processes, tools and objectives used by each department, working independently of each other.

The purpose of this demo is to showcase a practical application example within the framework of the presentation titled "Next generation (democratized) approach to Crash Simulation Optimization, incorporating CAD-Design parameters." During this demonstration, the primary focus will be on exploring the intricate details of the deployed optimization approach, including a seamless exchange of information with the CAD department. The workflow covering this approach will be examined, by highlighting each process step involved. Concluding, light will be shed on the various background operations and functions that drive the overall execution of this application.

Session 7E

Revamping Model Build: AI and Part Build at the service of CAE modeling teams

Pavlos Floratos

To build a modern, midsized vehicle more than 30000 parts are needed. These parts are of different types, (casted, stamped, extruded, cables), size (door, panels, bolts, nuts, rivets, bearing,) and of course of different material steel, aluminum, plastic. The CAE engineer, depending on the type of analysis, faces two challenges. First, he needs to identify which parts should participate in the analysis and then according to type and material, and second prepare these parts for the specific analysis.

Identification is one of the most important operations since this allows the user to automate his model build process, avoid errors in the treatment of the parts and significantly speed up the development.

In this demo, the latest developments in the automatic identification and categorization of parts based on their geometric definition will be demonstrated. Furthermore, the newly introduced build actions for parts, that allows for the correct and effortless preparation and mesh of any part type will also be shown. With these build actions, the user can capitalize on collective experience gathered by BETA and are now available for the treatment of parts.

The demo will conclude focusing on the treatment of casting parts and how to take advantage of the machine learning (ML) functionality of ANSA to identify and treat embedded clips and features.

Design Optimization in the Modular Environment: from setup to HPC submission and post-processing

Maria Papastavropoulou

Creating Optimization Studies poses significant challenges in effectively managing associated data and centrally controlling processes such as job submission and response extraction in the post-processor. Fortunately, the BETA Suite has already addressed these challenges for numerous applications within the framework of the Modular Environment, also leveraging the Process Management capabilities of SPDRM.

In this live demo, we present the latest advances in the Optimization Tool that can now harness the capabilities of the Modular Environment and the Process Management in SPDRM for the creation and management of Optimization Studies. Through an end-to-end example, we will showcase the latest solution for generating a Design of Experiments (DOE) on a modular model, featuring seamless management of all related data and streamlined submission of simulation jobs to the central HPC system.

Data insights: Keeping track, discovering, process reviewing, and accelerating decision making

Panagiotis Mantzopoulos

The primary driver for the adoption of an SDM system has always been traceability. An SDM system can track information about the author of the data, their creation date and other key attributes, as well as information about the relationships between them. This way, it is possible to not only search quickly among vast amounts of data based on their characteristics, but also trace the link of model data with simulation results and CAD.

In this demo, we showcase how the SDM client applications of the BETA Suite, making use of this unique capability of SDM systems to keep track of all this information, facilitate the discovery of data insights, streamlining the data review process and accelerating decision making. Through live examples, we will demonstrate how the SDM client can be utilized by analysts and model build engineers to serve daily tasks, eliminating data and information hunting.

Session 7F

PERMAS from within BETA's CAE Suite

Kosmas Gourgounis

This demonstration will showcase new and improved functionality specific to PERMAS user's unique requirements, available within ANSA.

Demo topics include:

Automatic generation of contact, pretension, pressfit definitions inclusive of corresponding loads and Zero-force springs to constrain rigid body motion, where needed.

PERMAS integration within ANSA's Loadcase Manager, in order to setup multiple Situations in a generic way.

PERMAS "solver integration", to run PERMAS jobs monitored from within ANSA, including Model integrity checks performed by the PERMAS MQA module.

Extended Load Variant and LPAT/NLLOAD management, to easily define linear and non linear loading history

From complex meshes to smooth surfaces: the new SubD approach

Konstantinos Kafkalas

Subdivision surface modeling is a powerful and versatile tool that can produce complex smooth surfaces using a coarser “control” mesh. It has been used extensively in 3D modeling workflows due to the high level of control it provides and the flexibility to use finer or coarser control mesh depending on the complexity of each modeled area, while at the same time producing a smooth and high-resolution modeling result. By introducing this methodology in ANSA, we provide a new way of handling complex meshes (for example coming from topology optimization results or 3D scanners), giving the ability to quickly and easily create a smooth surface from them and then potentially export it to other modeling software. We will showcase a brand-new tool aimed at intuitively creating the control mesh for subdivision surfaces, and give some examples where this new technique can provide a fast and high-quality modeling solution.

Session 7G

The pillar-functionality of the "stamp-crash" process

Klearchos Thomopoulos, [2]Marcel Meder
[2]BMW Group

The precision of crash simulations in the automotive sector can be considerably improved by taking into account the manufacturing process of the body-in-white components. This entails utilizing the history of deep-drawn body parts to refine the virtual models for both the body-in-white and the full vehicle. The pioneering approach called the "stamp-crash" process links the Process simulation data of deep-drawing with the design and development of body parts in a systematic manner.

The stamp-crash process was presented in a previous publication, so in this demonstration we will focus on the functionalities that act as pillars of this process. In particular:

•Finding Candidates: lining the crash and the Process simulation “worlds”.

•Auto-Positioning: Auto-detect the transformation matrix so that Process simulation matches the design position of the target part.

•Compare: Identify the Geom-changes at feature level among different design versions of parts

•Inverse Forming: In case of missing Process simulation data in the Source database

A productive way to replace areas with beam elements

Nikolaos Efstathakis

The representation of indicated areas of a structure using beam elements is a common industry practice, since it can significantly shrink the solution time of large models, providing simultaneously possibility for optimization studies on cross sectional engineering parameters.

Focused on a productive and flexible way to achieve this, a newly development for auto identification of “beam-like” areas using minimal user input will be demonstrated, combined with the latest developments in Cross Module, that allows the accurate representation of complex structures comprised from both shell and solid elements, anisotropic behavior and layered structures. Before solving the “simplified beam” model, the beam Cross Sections are extensively inspected for their correctness, utilizing the latest developments in Cross Module, like the new results window, the enhanced visualization of solid areas and the 2D mesh used by the cross section solvers. Finally, both the initial and the “simplified beam” models are solved to verify the performance gain, indicating the preserve of accuracy.

Connections navigator: a new approach to assembly validation

Zafeiria Kanellia

Designing the welds for a car is a multi-step process that involves careful analysis and planning. The design team must first determine the ideal placement of the welds to ensure that the car's structure is strong and durable. This involves examining the various components of the car and identifying areas that require additional support through welding. Once the optimal placement of the welds has been determined, the specifications for each weld are created. These specifications include detailed information about the precise location, type, and size of each weld, as well as specific welding parameters that must be used to maintain consistency in quality. The CAE responsible team will check these specifications to ensure that they meet the required standards.

However, there are challenges associated with this process. For instance, there is a time gap of a few weeks between when the CAD data freeze and when the welds becoming available. This can lead to design immaturity and differences among vehicle variants. As a result, weld examination proofs a necessary step to ensure that the welds are worth the investment. Furthermore, there are many non-expert ANSA users involved in the process, who require a simple and user-friendly way to check the model's welds.

This demo introduces a novel tool that has been developed to facilitate the navigation of complex connections in ANSA pre-processor. The design and CAE processes often encounter challenges in managing connections between various elements, which can result in errors and inefficiencies. Therefore, ANSA has recently undergone special development efforts to address these issues and provide users with a time-saving solution. The proposed tool offers an easy-to-use interface that allows users to navigate between connections, thereby reducing the time and effort required to manage these connections. Moreover, users can customize the tool to meet their specific needs and preferences.

Effortless hydrodynamic simulations with the ANSA SPH solver

Liana Vasiladioti

The need for an easy and fast simulation of complex problems of fluid flows has been increased. The CFD approach is in many cases a demanding process in terms of model preparation and a time-consuming solution.

In this demo the Smoothed-particle hydrodynamics (SPH) solver, an integrated solver within ANSA software which allows dealing with problems with large displacements and free surface flows will be presented. SPH guarantees conservation of mass without extra computation since the particles themselves represent mass. A highly tuned GPU implementation of the method allows almost real-time simulation of moderately sized problems and great acceleration for bigger ones. During execution, sophisticated visualization algorithms convey all the necessary information to the user. Avoiding complicated input/output files, SPH is accessible within ANSA pre-processor through a single window, which requires minimal input from the user.

Session 9E

Hexablock meshing for CFD

Antonis Karasavvidis

Hexablock demo: Demonstration of A) Turbomachinery. Automatic template meshing for axial machines, to be expanded to radial geometries in future versions. The user specifies certain pid names which define the domain and sets up the meshing parameters. The template then automatically creates all hexa boxes and generates a periodic structured mesh for the blade passage. B) Tubular Modeling. For tubular models with junctions the user first specifies the main flow direction and the function automatically identifies all cross sections and feature lines. This allows for a very quick and easy hexa box creation at all junction points. In a second step the user selects the rest of the geometry and the function again automatically creates all necessary boxes and topologically connects them.

Volume modularity in CFD models: a complete workflow

Nicholas Mitroglou

CFD volume models are famous for their extreme element counts. Their size grows, as ever increasing and powerful computing resources become available to analysts. Volume meshed models for CFD usually reach tens of millions of elements and often sum up to few hundred millions, especially if drag and lift coefficients are to be predicted accurately. It is evident that meshing times for such models may reach hours, therefore creation of several design iterations or model variants can be cumbersome. To simplify the analyst's workflow and enable them to spend time analysing results, rather than meshing, we propose a new concept of volume modular models. The aim is to speed up model variant creation and only treat/mesh models once and reuse them as building blocks of a jigsaw puzzle.

Fiber reinforced material CT Scan processing with RETOMO

Petros Michailidis*, Evangelos Karatsis

Fiber reinforced composite materials have become increasingly important in a variety of industries due to their impressive combination of strength and lightness. These materials are used in aerospace, automotive, construction, and many other applications, where they offer significant advantages over traditional materials. However, designing and developing fiber reinforced composites is a complex and challenging process that requires a deep understanding of their microstructure.

Computed Tomography (CT) scanning has emerged as a powerful tool for non-destructive imaging and digitizing of the microstructure of fiber reinforced composite materials. By generating a detailed 3D image of the material's internal structure, CT scans enable researchers and engineers to explore and analyze the material's microscale features, including the arrangement, distribution, and orientation of the fibers and matrix.

In this workshop we will demonstrate the latest tools of RETOMO for processing CT scans of fiber reinforced composite materials. These tools can generate either a highly detailed surface mesh or a simplified volume mesh capturing the underlying fiber directions, which could be used to virtually test and develop key material properties. In the latter case, the mesh can be exported to ANSA, where it can be further processed and analysed with ease.

Session 9F

ECAD Importer: A gateway to FEA simulations for PCBs

Magdalini Androniki Vasiladioti

Electronic components are becoming more complicated, and engineers run an increasing number of simulations to meet performance requirements. Most of the information about the PCBs is included in its ECAD. ECAD is used for design, fabrication, and simulation of PCBs and they contain crucial data for lossless information flow between designers, manufacturers, and simulation engineers. ECAD designs need to be translated into reliable PCB finite element models and then prepared for simulation runs.

This demo showcases the capabilities of the ECAD Importer plugin tool of ANSA that automatically generates FE models from ECAD files of IPC-2581 (Rev.B) format. Various models can be generated using the ECAD Import wizard plugin depending on the desired level of detail. Different import options enable the generation of a wide variety of models from a single ECAD file. The import wizard offers the capability to choose between different levels of board detail and import components with or without pins. Choosing the appropriate set-up, the engineer can achieve the required accuracy of results with the minimum modeling effort.

Session 9G

ANSA modelling for ACTRAN

Dimitrios Tasoulas

This demo showcases the latest developments and capabilities of ANSA regarding the support for Actran Acoustics solver. ANSA provides comprehensive support for a wide range of acoustics analysis areas including internal vibroacoustics, exterior acoustics, and sound radiation.

By using ANSA, users can benefit from an efficient and accurate simulation process, leading to improved product designs and reduced development time. For instance in interior vibroacoustics, the pre-processor offers a range of tools to analyze sound transmission and noise reduction within the vehicle cabin. Moreover, radiation analysis can be supported, including the modeling of sound propagation in open spaces and sound radiation from vibrating surfaces. These are just a few examples of the diverse range of acoustics analysis areas that ANSA supports, making it a versatile tool for a variety of applications, such as automotive and aerospace industries.

In particular, this demo illustrates:

-The most important parts of a typical Actran solver file structure and how they are supported and implemented in ANSA.

-The various supported keywords and tools which are supported by ANSA and comprise a comprehensive toolset that can be used for a broad scope of acoustics analysis domains such as internal vibroacoustics, exterior acoustics and sound radiation.

-The enriched capabilities of the Loadcase Setup Assistant regarding the setup of analysis types such as Time Domain, Modal Extraction and Direct Frequency Response analysis along with Loadcase entities such as Boundary Conditions and Output Requests and other options.

ANSA for NASTRAN Aeroelasticity

Dimitrios Tasoulas

The field of aeroelasticity has always been an essential aspect of engineering in the aerospace industry. This session highlights the new ANSA capabilities on support of Nastran Aeroelastic solutions. With our latest developments, we have made it easier for engineers in the aerospace industry to perform aeroelastic analysis and design using Nastran. Our pre-processor's capabilities for static aeroelasticity, flutter, and dynamic aeroelasticity analysis using Nastran sol 144, sol 145, and sol 146 are now more robust, which will undoubtedly improve the accuracy and efficiency of aeroelasticity analysis and design.

In particular, this demo illustrates:

- the generation of aerodynamic elements, either surfaces or bodies, for the supported aerodynamic theories along with the automatic creation and numbering of the contained aerodynamic boxes.

- how to specify the interconnection between structural and aerodynamic model with the use of linear or surface Splines.

- the definition of global aerodynamic data such as the aerodynamic coordinate system and the mach number-reduced frequency pairs used for generation of the aerodynamic matrix.

- how the define Boundary conditions, such as steady-state trim flight conditions or gust loads for dynamic aeroelatic solution.

- the Load Case set-up for steady and unsteady solutions with the use of the Loadcase Assistant's enhanced interface. More specifically, we will emphasize on how to create new or add already existing Boundary Conditions and Output Requests, as well as how to include any other structural or aerodynamic data required for the analysis.

Latest developments and employment of NEERE as a training platform

Gerasimos Skouvaklis

NEERE training capabilities arise by the seamless collaboration via ANSA, META or VR. The ability to easily configure and monitor the progress of multiple collaborators or trainees will be demonstrated in this session. Additionally, conversation flow enhancements, privacy and usability developments will be showcased together with other improvements.

BETA
SIMULATION SOLUTIONS

physics on screen

www.beta-cae.com