ANSA & μETA
INTERNATIONAL CONGRESS

June 2-3, 2005

Athos Palace Hotel

Kassandra, Halkidiki

Greece

PROGRAMME
&
ABSTRACTS

BETA CAE Systems SA

Welcome

Dear Attendees,

Welcome to the “ANSA & μETA International Congress”.

BETA CAE Systems S.A. is honored by your participation. We assure you that we’ll try our best to make your stay a pleasant one and we hope that you will enjoy your participation.

During the congress you will have the opportunity to be informed about the latest developments in the CAE field and the products of BETA CAE Systems S.A., through the technical presentations prepared by our guests from the academic and industrial sectors.

Furthermore, you will have the opportunity to interact with the executive, development and services engineers of BETA CAE Systems S.A. to present your requirements for future developments. You can be sure that we will pay close attention to your requests and will give them a high priority in our development planning. Along with the local distributing teams from the U.S.A., Germany, Japan, China and Korea we will be at your disposal to discuss further technical and business related issues.

You are also invited to our social event, the Greek night, on Thursday evening dinner.

I, personally, and the entire BETA CAE Systems S.A. would like to thank you for coming and for contributing to the success of the congress. I wish you a productive and pleasant stay!

Please, feel free to contact us for any issue regarding your stay here and we will do all we can to assist you.

George Athanasiadis
President
BETA CAE Systems S.A.

Programme

Programme

Thursday, June 2

8:30 - 9:15	**Registration**
9:15 - 9:30	**Opening Speech** G. Athanasiadis President, BETA CAE Systems S.A., Greece
9:30 - 10:00	**Design improvement of components and structures** R. Adams[1], N. McLachlan[2], J.A. Tomas[1] [1]Advea Engineering, Melbourne, Australia [2]Australian Bell, Melbourne, Australia
10:00 - 10:30	**Coupling ANSA and optimization software for efficient design workflow – Shape and parameter optimization based on the Morphing Tool** G. Korbetis, I. Makris, G. Panagos BETA CAE Systems S.A., Greece
10:30 - 11:00	*Coffee break*
11:00 - 11:30	**Data translation from CATIA V5 to ANSA** N. Sawa Advanced CAE Div., Toyota Motor Corp. Japan
11:30 - 12:00	**Building a flexible CFD-model. Overview of surface & volume mesh generation and shape modification through a case study in external aerodynamics** E. Skaperdas, G. Panagos, C. Kolovos BETA CAE Systems S.A., Greece
12:00 - 12:30	**Numerical simulation for improving radiator efficiency by air flow optimisation** S. Chacko TATA Technologies Ltd. India
12:30 - 13:00	**ANSA as a pre-processor for CFD simulations with Fluent - Approach and Examples** M. Ehlen FLUENT Deutschland GmbH, Germany.
13:00 - 14:30	*Lunch*
14:30 - 15:00	**Design of crankshaft main bearings under uncertainty** Z.P. Mourelatos Mechanical Engineering Dept., Oakland University, USA
15:00 - 15:30	**Automated structural and multi-body dynamics: Application to linear and non-linear finite element vehicle models** G. Verros, P. Metallidis, I. Goudas[1], S.Natsiavas[2] [1]Vehicle Engineering Consultants, Thessaloniki, Greece [2] Department of Mechanical Engineering, Laboratory of Machine Dynamics Aristotle University of Thessaloniki, Greece
15:30 - 16:00	**A concept for the simulation of static and fatigue crack propagation in 3D shell structures** H. Aretz[1], S. Greuling[2], R. Weisner[2] [1]LASSO Ingenieurgesellschaft Leinfelden-Echterdingen, Germany [2]DaimlerChrysler AG, Forschung & Technologie, 70546 Stuttgart, Germany
16:00 - 16:30	*Coffee break*
16:30 - 18:00	**Technical discussions, demonstrations and meetings**
20:00 - .	**Dinner - Social Event: "Greek night"**

Friday, June 3

9:00 -9:30	**Nonlinear finite element analysis of proximal femur nail (PFN) combination** U. Hindenlang[1], G. Faust[3], P. Helwig[2], A. Krieg[1] [1]LASSO Ingenieurgesellschaft Leinfelden-Echterdingen, Germany [2]Berufsgenossenschaftliche Unfallklinik, Eberhard-Karis-Universität Tubingen [3]Inst. fur Static und Dynamik der Luft- und Raumfahrtkonstruktionen Universität Stuttgart, Germany
9:30 - 10:00	**Applications of ANSA - Morphing Tool in Biomechanics modelling** E. Karatsis, I. Chalkidis Department of Mechanical Engineering, Laboratory of Machine Elements & Machine Design Aristotle University of Thessaloniki, Greece
10:00 - 10:30	**Education of engineers in design improvement methods** R. Adams, J. A. Tomas School of Aeronautical, Mechanical and Material Engineering, RMIT University, Melbourne, Australia
10:30 - 11:00	*Coffee break*
11:00 - 11:30	**Meshing and numerical simulation for vehicle development at PSA Peugeot Citroen** R. Peterschmitt , R. Khatir PSA Peugeot Citroën, France
11:30 - 12:00	**Automated simulation cycle from CAD-files to report** M. Tryfonidis, S. Chatziangelidis BETA CAE Systems S.A., Greece
12:00 - 12:30	**Legacy to common model conversion** L. Rorris, K. Skolarikis BETA CAE Systems S.A., Greece
12:30 - 13:00	**Automatic load case generation for crash and safety analysis** K. Skolarikis, L. Rorris BETA CAE Systems S.A., Greece
13:00 - 14:30	*Lunch*
14:30 - 15:00	**Towards an automated procedure of an ABAQUS standard/explicit model build-up. An innovative approach to data management and model validation** S. Seitanis, M.Giannakidis BETA CAE Systems S.A., Greece
15:00 - 15:30	**ANSA process automation tool - Future developments** D. Angelis BETA CAE Systems S.A., Greece
15:30 - 16:00	**Closing Session** G. Athanasiadis President, BETA CAE Systems S.A., Greece
16:00 - 16:30	*Coffee break*
16:30 - 18:00	**Technical discussions, demonstrations and meetings**

Abstracts

Design Improvement of Components and Structures.

R. Adams[1], N. McLachlan[2], and J. Tomas[3]

[1]Advea Engineering, Melbourne, Australia

[2]Australian Bell, Melbourne, Australia

[3]School of Aeronautical, Mechanical and Material Engineering, RMIT University, Melbourne, Australia

ABSTRACT

In recent years, various methods of optimisation have been developed, based on the finite-element methods of structural analysis. Some of these methods have been implemented in commercial computer programs such as Nastran, Optistruct, Tosca, and so on.

Some of these programs are part of a larger suite of programs, however, and not conveniently usable as an external program from the suite. This is a serious drawback for a small company like ADVEA, when projects involve not only the solution of given problems, but often also the transfer of the complete solution process to the customer. For example, MSC Nastran SOL 200 requires MSC Patran to be used efficiently; Altair Optistruct requires Altair HyperMesh to be used.

The design improvement program ReSHAPE has been developed to be completely independent from any pre- and post-processor, so that it can be used with any one of them. It was therefore an easy task to couple ReSHAPE with ANSA to efficiently and effectively solve design improvement problems, as required by customers.

The paper demonstrates the process of design improvement on selected problems. The first problem is the weight reduction of a car component by changing its shape, the second one demonstrates retuning a car component for noise reduction, and finally, the last two examples show the application of ANSA and ReSHAPE in a non-engineering field; firstly in the design of harmonic bells, and secondly the redesign of the tone arm and chassis of the Caliburn Turntable for improvement of analogue reproduction of music.

keywords: design improvement

Coupling ANSA and optimization software for efficient design workflow - Shape and parameter optimization based on the Morphing Tool.

G. Korbetis, I. Makris, and G. Panagos
BETA CAE Systems S.A., Greece

ABSTRACT

CAE technology is used to support the design process that is driven and fed back by finite element based simulation. As the complexity of the design tasks increases the computational volume and required man hours, the optimization procedure can contribute significantly in the design and calculation process. Easy and flexible definition of optimization parameters in pre-processing software is essential for the efficient model handling. ANSA is capable of interfacing with different optimizing programs to cover a wide range of cases, from early design stages to detailed model features handling.

In this case study four variations of a front rail are tested in crash analysis for the determination of the optimal shape for each one of them. ANSA Morphing Tool is used for the preparation of the model so as to define the shape optimization parameters. Entities of the model, such as material properties and sheet metal thickness, are accessed as design variables. ANSA scripting language and μETA-Post session files are used for the definition of input and output files containing these design variables, constraints and objective function. The problem is solved using different optimizers with Optimization and DOE methods.

A final front crash test of a BiW verifies the results attained from the partial tests. This approach also demonstrates the ability to parametrically substitute different versions of a component in an assembly.

keywords: optimization, morphing, design process

Building a flexible CFD model. Overview of surface & volume mesh generation and shape modification through a case study in external aerodynamics.

E. Skaperdas, G. Panagos, C. Kolovos
BETA CAE Systems S.A., Greece

ABSTRACT

The preparation of a mesh for CFD analysis is a complex laborious process that requires large amounts of man hours from experienced users and usually involves the combination of different software that are designed for specific tasks. All this, in combination with the huge model sizes that are currently necessary for realistic and accurate CFD simulations, and are nowadays feasible by the available hardware resources, make the process stiff and error prone. The dependence of the volume mesh to the surface mesh also make error corrections and local model modifications hard to implement without starting back from the early stages of the process.

This paper presents a streamlined and integral process for CFD model build-up, made possible by the latest development of ANSA, based on the experience and feedback from the automotive industry. The process involves CAD data interfacing with industry's most widely used CAD systems, geometry clean up and modification, automatic curvature dependant surface meshing, advanced boundary layer generation and robust volume meshing. Hence, a high mesh-quality model is achieved with minimum effort.

The resulting baseline CFD model, having modular construction, demonstrates the flexibility of performing local or global modifications through Part Replacement or Morphing. In the former case, selected parts are replaced with new ones and are integrated with the main surface and volume mesh. In the latter case, a complete volume mesh model is reshaped by controlled movements or directly to precise target shapes.

This process is an assuring solution for current industries in various engineering sectors where reduction of time-cycle and improved cost efficiency are driving forces.

keywords: CFD, pre-preprocessing, meshing, morphing

Numerical Simulation for Improving Radiator Efficiency by Air Flow Optimization.

S. Chacko
Computer Aided Engineering - EAG, Tata Technologies Ltd, India

ABSTRACT

The Efficiency of the vehicle cooling system is widely dependent on the air flow profile through the radiator .The Flow through the Radiator would also depend on the other panels in the vicinity of the radiator and these include grille, front inner panel, cowl, floor. A clear understanding of the flow happening through the radiator would help us optimize the flow, quality and improve efficiency of the radiator.

The objective of the Project was to Optimize Airflow Distribution inside the Radiator with the use of CFD as a tool. Detailed experimental measurements were also needed to validate the results provided from CFD. For achieving this test rig had been developed and experiments performed on static rig consisting of the grille, radiator, front inner panel, floor panels etc at room temperature. The data obtained from the experiments on special test-rig was compared with the numerical simulations of the similar Front-End configuration and results validated.

The CFD Analysis was done using FLUENT, while Surface Mesh & Grid generated using ANSA/TGRID Respectively. Use of ANSA helped us to reduce the total cycle time by reducing the surface meshing time for total iterations done to reach the final optimized shape of the panels in the vicinity.

The CFD Simulation Model validated were based on the following conditions

- Vehicle at Rest
- Ambient Pressure & Temperature Corresponding to Test Condition
- Pressure = 101325 Pa (Atmospheric)
- Temperature = 34 °C (Room Temperature During Testing)
- Radiator Heat Generation Neglected
- Rated Fan Flow rate considered

This paper would take one through the various phases of the project, from interpreting the basic physics behind the problem, to setting it up numerically, finding out the accuracy of the solution by validating the same with experimental data, Running the simulation, Interpreting its results, taking a note of all the possible causes, suggesting design changes, implementing those changes and predicting % improvement from the base case

ANSA as a Pre-Processor for CFD Simulations with FLUENT - Approach and Examples.

M. Ehlen
FLUENT Deutschland GmbH, Germany

ABSTRACT

For dealing with complex geometries coming from surface-based CAD-systems, the software packages ANSA from BETA CAE Systems S.A. and TGRID from Fluent together form a highly flexible and versatile tool. The approach is outlined, typical functionalities are explained and a variety of industrial examples are shown to illustrate "high-fidelity"-meshing. An overview on future requests for CFD meshing completes the presentation.

Design of Crankshaft Main Bearings under Uncertainty.

Z. P. Mourelatos
Mechanical Engineering Department
Oakland University
Rochester, MI 48309, USA

ABSTRACT

A probabilistic analysis is presented for studying the variation effects on the main bearing performance of an I.C. engine system, under structural dynamic conditions. The analysis is based on surrogate models (metamodels), which are developed using the kriging method. The metamodels provide an efficient and accurate substitute to the actual engine bearing simulation models. The bearing performance is based on a comprehensive engine system dynamic analysis which couples the flexible crankshaft and block dynamics with a detailed main bearing elastohydrodynamic analysis. The clearance of all main bearings and the oil viscosity comprise the random design variables. Probabilistic analyses are performed to calculate the mean, standard deviation and probability density function of the bearing performance measures. A Reliability-Based Design Optimization (RBDO) study is also conducted for optimizing the main bearing performance under uncertainty. Results from a V6 engine are presented.

keywords: I.C. engines, crankshaft, main bearing elastohydrodynamic analysis, engine system dynamics, surrogate modelling, design under uncertainty, reliability-based design optimization

Automated Structural and Multi-Body Dynamics: Application to Linear and Non-Linear Finite Element Vehicle Models.

G. Verros[1], P. Metallidis[1], I. Goudas[1] and S. Natsiavas[2]

[1]Vehicle Engineering Consultants, Thessaloniki, Greece

[2]Department of Mechanical Engineering, Aristotle University, Thessaloniki, Greece

ABSTRACT

The main objective of the present work is to develop and apply a systematic numerical process leading to an automated determination of dynamic response of complex mechanical systems. Such systems arise frequently as a result of strict requirements posed by industrial needs on the accuracy obtained in the response of various mechanical components or systems. Usually, the components of complex systems are geometrically discretized and represented by finite element models, involving a quite large number of degrees of freedom, which may sometimes reach or overcome the order of a million. The basic idea is to first reduce the dimension of the systems examined by applying appropriate methodologies in either the time domain or the frequency domain. Application of these methodologies results in an efficient reduction of the dimensions and helps the efforts towards a systematic and comprehensive study of the dynamics exhibited by large order mechanical models with nonlinear characteristics. Apart from increasing the computational efficiency and speed, the reduction of the system dimensions makes amenable the application of several numerical techniques for determining the dynamic response of the complex systems, which are applicable and efficient for low order dynamical systems.

The methodolody developed was adjusted so that it starts by reading all the necessary data from the Pre-processor ANSA in order to set up the corresponding equations of motion. After reducing the order of these equations, a number of analyses are performed, including static and kinematic analysis, eigenvalue, transient response, frequency response and random analysis as well as parametric identification and optimization. Once the analysis chosen is completed, the process is finished by directing the results obtained to the Post-processing system μETA. It is important to note that the process is developed in a general way, so that it allows hybrid modellings resulting from coupling of numerical with experimental data or results obtained from coupled mechanical and acoustics models.

The accuracy and effectiveness of the methodology applied in the code developed is illustrated by numerical results obtained for complex vehicle body and engine structures. In particular, frequency spectra of several response quantities related to model performance were constructed for motions resulting from periodic and random excitation. In cases where it is feasible, direct comparison is performed (in terms of accuracy, memory required, data transferred and numerical speed) with similar results obtained from NASTRAN and ADAMS, for structural and multi-body dynamics applications, respectively. An advantage of the code developed is the ability to determine periodic steady state response of periodically excited nonlinear dynamical systems in a direct way. This is particularly important, since the results obtained indicate that there appear substantial differences in the response diagrams of the fully nonlinear models examined and their corresponding linearized versions, which are commonly used in practice.

keywords: automated finite element analysis, structural and multi-body dynamics, linear and non-linear steady state analysis, random analysis, parametric identification, optimization

A concept for the simulation of static and fatigue crack propagation in 3D shell structures.

H. Aretz[1], S. Greuling[2], R. Weisner[2]
[1]LASSO Ingenieurgesellschaft mbH, Markomannenstrasse 11,
70771 Leinfelden-Echterdingen, Germany
[2]DaimlerChrysler AG, Forschung & Technologie, 70546 Stuttgart, Germany

ABSTRACT

In this contribution an efficient and modular method is presented to simulate fatigue crack propagation within the framework of linear-elastic fracture mechanics. The FEM code ABAQUS/Standard is used to simulate the load/displacement history of the considered 3D shell structure using 6-node shell elements while the preprocessor ANSA is applied to employ remeshing. In addition, special purpose software modules have been created whose major purposes are (i) to achieve a continuous data transfer between ABAQUS and ANSA and (ii) to introduce the advancing crack tips into the finite element mesh. A governing shell script controls all software modules so that the crack propagation simulation runs automatically until all cracks arrest or unstable crack growth occurs.

In order to simulate efficiently fatigue crack propagation in large finite element models a submodel is extracted from the global model. The submodel is subjected to the kinematics given at the interface to the global model. The assignment of the kinematic boundary conditions to the interface nodes of the submodel is achieved by a special purpose program.

The stress intensity factor concept is applied. Stress intensity factors are calculated from the finite element mesh within an ABAQUS user subroutine using the displacement correlation technique. Variable amplitude loading is accounted for by using a special counting algorithm and an averaging procedure to calculate the crack extension direction. In order to calculate the number of load cycles various fatigue models have been implemented.

The focus of the present contribution is put on the applicability of the proposed concept. Various examples demonstrate the capabilities of the developed fatigue crack propagation simulation environment.

keywords: fatigue; crack propagation; fracture mechanics

Nonlinear Finite Element Analysis of proximal femur nail (PFN) combination.

U. Hindenlang[1] G. Faust[3], P. Helwig[2], A. Krieg[1]

[1]LASSO Ingenieurgesellschaft Leinfelden-Echterdingen, Germany

[2]Berufsgenossenschaftliche Unfallklinik, Eberhard-Karls-Universität Tübingen, Germany

[3]Inst. für Statik und Dynamik der Luft- und Raumfahrtkonstruktionen Universität Stuttgart, Germany

ABSTRACT

Geometrical nonlinear analysis of proximal femur nail combination under static loads including fractured parts using computer tomographic data for femur stiffness and geometry with realistic contact conditions. The mechanical behaviour of the implant is compared to clinical observations.

keywords: PFN, femur, fracture, CT-density,CT- bone-geometry, mapping , morphing

1. **Introduction:**
Nowadays modern CAE Methods are also applied in the large area of medical care. In particular nonlinear FE-analyses are used to predict the mechanical behaviour of implants and can attribute in the future to a higher reliability on the selection and functionality of the specific femur nail combinations for the surgical practice.

2**. Method**
The considered part of the skeleton is reproduced via computer-tomography. The data are imported to CAD-programs in the present work to ANSA and contain information concerning the shape of the bones as well as the distribution of densities which are used to define the repartition of the bone-stiffness.The CAD-data of the implants are blended into the bone surface and bone stiffnesses are mapped to the 3-dim mesh. One specific loadcase is analysed by geometrical nonlinear calculation, allowing the incremental analysis of the motion guaranting equibilbrium at every time step. Thus the behaviour of every part in interaction may be analysed and evaluated and the complete motion may be animated.

3. **Results**
The application of commercial software (ANSA, ABAQUS) in the area of medical area is shown. The behaviour of the proximal region of a fractured femur under a load of 2000 N [1] was analysed, showing a slip back of the femural neck screw relative to the antirotational screw. This may lead to a tilting of the proximal main fragment around a sagittal axis.

4.**Discussion**
The employed FEM-Anlyses reproduce various phenomena that are also observed in pratice. The presented approach shall be extended to a preoperative tool allowing the selection of an optimal implant for the indivudual case. For this aim a catalogue of

different sizes of standarized bones modelled with a large number of implants will be established allowing a prompt response using an automatized morphing procedure (ANSA), that accounts for the density information via the individual CT data of the patient.

REFERENCES

[1] Huber-Wagner, S., Dissertation *Chirurgische Klinik und Poliklinik*. 2002, Ludwigs-Maximilians-Universität: München. p. 149.

Applications of ANSA – Morphing Tool in Biomechanics modelling.

E. Karatsis, I. Chalkidis

Department of Mechanical Engineering, Aristotle University of Thessaloniki, Greece

ABSTRACT

The Finite Element Analysis of a human body member, as far as its strength is concerned, requires a time-consuming process to prepare the geometrical data and depends on the software that will be used. Usually, the existing geometrical data result from a computer tomography of human body members and are 2D images. These 2D images should somehow construct 3D views, which will be used to generate 3D geometry.

The aim of this paper is to define how the CT Scan data are translated to 3D geometry, as well as how to prepare the resulted geometry for further analysis, using a finite element software. To avoid the repentance of this process each time that appears a new tomography of the same member, it is recommended to create a general geometric model for the studied human body member, called template-model. The template-model is modified according to the feature parameters coming from the new human body member, called examinant-model. This modification decreases the duration of the whole process of getting the final model, since it is not required the translation of the examinant-model CT Scan data to geometrical.

In the current project, in order to reach this achievement, ANSA pre-processor has been used and more specifically the ANSA – Morphing Tool. The ANSA – Morphing Tool can modify the shape of a FE – Model with a flexible way allowing the convergence between the template – model and the examinant – model.

Although the so far ANSA – Morphing Tool functionality is developed based on the demands of technical structural applications, it could be expanded in biomechanical field, investigating its demands for further development of new functions.

Keywords: template-model, examinant-model, ANSA – Morphing Tool, feature parameters, FE – Model, FEM, CT scan, biomechanics, mandible, teeth, vertebra, segmentation

Education of Engineers in Design Improvement Methods.

R. Adams[2] , J. A. Tomas[1]

[1] School of Aeronautical, Mechanical and Material Engineering,
RMIT University, Melbourne, Australia
[2]Advea Engineering, Melbourne, Australia

ABSTRACT

The history of objects, as invented and produced by man, is the history of incremental design improvements, aiming, for example, at lighter, yet more durable components, or at creating less noisy structures, or even structures that resonate at a pleasant tone.

Even today, when powerful computational methods are available for structural analysis, the usual procedure for design improvement is still based upon the results from the analysis, with the decision about design changes being left to analysts and designers, based on their knowledge, experience, and, more often than not, on their “engineering feel”.

The course in design improvement at RMIT Melbourne aims at training students in modern design improvement methods and to teach them practical skills for selecting the method appropriate for a given task. The course is offered in the sixth semester, after the students were introduced into linear FEM in semester five.

The theoretical background of selected mathematical methods of optimization is reviewed first. Then the concept of design sensitivity is introduced and demonstrated on truss and beam structures. Finally, the problem of “reconciling” the discreteness of finite-element meshes with the smoothness of CAD designs is presented and solution methods exercised.

The software ReSHAPE, which has implemented most of the available methods for design improvement, is used, together with pre- and post-processors ANSA, HyperMesh and Patran. Each lesson is accompanied with solved examples and tutorial problems. Students are required to validate the solution of each problem by any alternative method available to them. At the end of the course, students solve one practical problem of their own choice, to demonstrate the level of their competence.

With this initiative, the industry will be provided with structural engineers, who are at the fore-front of the application of finite-element methods and who will be able to contribute more effectively to designing better products.

keywords: design improvement

Automated Simulation cycle from CAD files to Report.

M. Tryfonidis, S. Chatziangelidis
BETA CAE Systems S.A., Greece

ABSTRACT

Nowadays FEA simulation results have reached a level of accuracy comparable to physical testing. The cost of the above fact is that models complexity has increased, as the tremendous progress of information and computer technology allowed so. Throughout product design cycle, engineers often have to judge the performance of many design variants by going through numerous Finite Element Analyses. This necessity leads to the need for the automation of the pre- and post-processing of FEA so that errors and turnaround times are reduced and quality and productivity are drastically improved.

This paper demonstrates how ANSA pre-processor and μETA post-processor of BETA CAE Systems S.A. are efficiently used in the Automotive Industry for the automation of analysis processes, all the way from CAD data input to CAE results report generation.

It is demonstrated how part of the above automation is achieved with ANSA pre-processor scripting language that is simple and easy to learn and yet has the sophistication of contemporary object oriented languages. In addition to this, it is shown how the post-processing functionality of μETA is automated by its own session files. These two integrate the pre- and post-processing operation into a complete process.

Briefly, the automated process steps that will be presented are:

- CAD data collection from various sources with simultaneous parts and properties management.
- Import geometry from CAD files
  - Automatic geometry clean-up in a single step,
  - Extraction to middle surface of sheet metal parts,
- Batch Meshing,
- BiW panels welding,
- Model's sub-systems assembly,
- Load case and analysis scenario definition,
- Quality checking and automatic improvement,
- Analysis solving,
- Post-processing of analysis results for model and results validation,
- Reports generation.

keywords: Batch Meshing, Pre-processing, Post-processing, BiW Assembly, Report generation

Legacy to Common Model Conversion.

L. Rorris, K. Skolarikis
BETA CAE Systems S.A., Greece

ABSTRACT

The continuous need for sorter design time cycles puts the pressure on CAE departments
for rapid and robust model preparation for numerous types of analysis. This challenging task especially in the early stages of design can be facilitated by the reuse of validated and trusted FE Models. Thus a transfer of data can be achieved in two directions, from old to new models and across disciplines.

This transfer of data from a legacy model is done by its conversion to a common model.
A common model represents a model striped of solver and analysis specific data, being properly parametrized so as to have the flexibility to adopt any form needed by a variety of load cases and analysis types.

The methodology followed for such a conversion will be presented, demonstrating the use of existing and newly developed ANSA functions in conjunction with ANSA scripting language for the extraction and handling of data such as load case specific data, geometry data (Mesh), assembly information (spotwelds, seamwelds, adhesives etc.),trim mass information etc.

Automatic load case generation for crash and safety analysis.

K. Skolarikis, L. Rorris
BETA CAE Systems S.A., Greece

ABSTRACT

The continuously increasing complexity of FE-models and the surfeit of government regulations and consumer tests that demand always new load-cases and parameters to be taken into consideration have led to an urgent need for maximum possible automation in the stage of pre-processing. The main objectives are the saving of valuable time and the avoidance of errors that emerge from the tedious repetition of similar tasks. The afore-mentioned need for automation can now be effectively addressed with the new enhancements of ANSA.

The most important tool that has been newly redesigned to support this project is the ANSA scripting language. The new build-in functions give access to the properties of almost every entity within the ANSA database. They enable calculations based on these properties, their modification and even the creation of new entities. Together with the common functionality that every scripting language offer, it aims to be an integrated solution.

Of great importance it is also the full support of 'include' keywords. This keyword is of great use among CAE engineers because it is the only means provided by solvers which enables them to build their models in a flexible modular fashion.

With the employment of the afore-mentioned features, in this presentation we suggestively follow some steps in the preparation of a model for the cases of front and side impact tests. These steps are namely:

-The positioning of dummy, barrier and rigidwalls.

-The imposition of initial velocity and gravity forces.

-The appropriate modification of sets used by contacts, rigidwalls and initial conditions.

-The assembly of accelerometers and vehicle instrumentation.

keywords: load-case, scripting language, include , crash, safety

Towards an automated procedure of an ABAQUS Standard/Explicit model build-up. An innovative approach to data management and model validation.

S. Seitanis, M. Giannakidis
BETA CAE Systems S.A., Greece

ABSTRACT

ANSA and μETA PostProcessor offer unique modelling techniques and solutions to CAE problems in a seamless and efficient way.

ANSA is capable of producing an ABAQUS model of the highest standard, giving accurate and reliable control over model entities with minimal effort. Data handling and model verification tasks are undertaken by the software, allowing the engineer to focus on problem solving and decision making – yet giving the ability to control every model detail.

This automated procedure includes the extraction of non-geometrical data from CAD-files, as well as part attribute assignment and defeaturing. Meshing is performed through batch mesh sessions using analysis specific parameters and quality criteria. Assembly process is achieved using a BiW Connections' Manager that allows the generation of mesh dependent or independent connection types, as well as the conversion between different welding representations.

ABAQUS oriented definition and use of SETs allow the definition of initial and boundary conditions, contacts and output requests, thereby making the model easy to manipulate and update. Change or replacement of the elements automatically update the SET contents.

Material data are loaded and handled through a material database. Moreover, the materials of the model can be updated with new material values at any time during pre-processing, while a versatile 2D-plot utility allows the GUI-based management of material data tables.

Model sanitation and integrity checks are automatically performed as per ABAQUS and the user is prompted to apply a fix. Finally, model output options allow model to be broken down to a number of self-complete substructures to generate test-runs and reduce the time of the final run.

μETA PostProcessor allows engineers to rapidly validate the analysis results and produce the project's report.

Automatic retrieval of available results and discipline specific data visualization is fully automated with the use of session files.

keywords: ABAQUS Standard / Explicit, FE-model, automation, validation, post-processing

ANSA process automation tool - Future developments.

D. Angelis
BETA CAE Systems S.A., Greece

ABSTRACT

In modern Automotive Industry, the "success" of a vehicle model is not confined on its market acceptance; the routes of this success are quite deep, reaching down to the first CAE concept. The distance between conceptual design and production is covered by an ever increasing number of CAE-cycles that involve collecting and manipulating huge amount of divert data. In addition, the growing complexity of simulation and analyses leads to error-prone procedures, tends to delay crucial decisions and prolongs the CAE turnaround time. The remedy is "process automation" and the challenge is to achieve this by keeping the procedure cost-efficient and error proof.

Within this framework, BETA CAE Systems S.A. introduces an integrated process workflow manager that promotes knowledge transfer and exploits the cumulative CAE experience and expertise. By providing a high degree of automation, our vision is to decrease human error factor, promote model reusability and liberate CAE experts from tedious procedures so they can focus on problem solving.

In the heart of this integrated workflow manager lies a tool where all individual tasks of the development of a vehicle simulation model are included. This task manager is built up from the CAE expert who sets the boundaries between distinct modelling actions and assigns all modelling parameters that must be respected, leaving to the inexperienced user a minimum degree of intervention and limited (or none) decision making. In a typical modelling scenario the task manager integrates the following procedures: batch meshing, assembly, trimming, analysis dependent pre-processing and post-processing of results. Guiding the user to these procedures, the workflow manager assures that the user does not skip any step, marks completed tasks on a check list and provides the respective alternative ways to proceed. This approach reduces the CAE-cycle, safeguards the procedure and makes the CAE expertise inherent in the process.

keywords: process workflow, task manager, pre-processing steps, CAE expert

Meshing and numerical simulation for vehicle development at PSA Peugeot Citroen

R. Peterschmitt , R. Khatir
PSA Peugeot Citroën, France

ABSTRACT

This presentation is an opportunity to demonstrate that the ongoing collaboration and close working relationship between BETA CAE Systems S.A. and PSA Peugeot Citroën Group which has started many years ago has created a platform and opportunities to improve both numerical simulations as well as meshing.

Topological meshing is a crucial component in the development process of our numerical prototypes and is an integral part of our platform strategy.

Our approach can be applied to the following: crash analysis, electro-magnetic analysis, internal and external aerodynamic and body engineering.

Please note that automotive programs are facing new challenges in term of time, cost and quality. It would be therefore essential for us to possess numerical tools with even more advanced and sophisticated functionalities as well as an enhanced interface and “integration” with our design softwares, to allow fast updates of numerical models.

BETA CAE Systems S.A.
Kato Scholari, Thessaloniki, GR - 57500, Epanomi, Greece
Tel: +30-2392-021420 Fax: +30-2392-021417
Email: ansa@beta-cae.gr
URL: http: //www.beta-cae.gr