Dr. Peter Stamerjohanns **Stars Acoustics UG**

Development of a Tool for the Acoustic-Calculation of Mechanical Systems

Abstract:

The paper will announce the development of a new tool to optimize the calculation and evaluation of the acoustic behaviour of mechanical systems, especially drive-trains. The tool will help the designers to check the dynamic values of their models in a quick, easy and very accurate manner. The tool will combine features of Ansa, Epilysis and Meta via a central database. The database will allow to build a complex drive-train models in short period of time and to calculate and analyze the models in an automated procedure. Even for big models accurate results will be available in acceptable time, stored in small data files. The designers will get results from resonance-frequencies, radiated noise up to load-based stress values of interesting parts of their structure. Result-procedures will be available to create reports and to compare results with older or other models.

Introduction

Mechanical systems have to be analyzed concerning first for stress and second for acoustical problems. But even by acoustical problems systems may fail. Often there is no chance to analyze the mechanical system in its complete range of applied parts, although all simplified models may get wrong results. The reasons are besides computer-problems – models to big – the availability of all applied parts. Furthermore even if all parts are available and calculated the result files are too big to store, e.g. for restarts. The paper presents a new process to be created within the next months to optimize the calculation and the process for the calculation.

Basic principle

The basics of the procedure are the creation of a show-model of all parts. The show-model is represented by a coated surface for all volume-parts. These shell-surfaces are first-order-elements, created on second order volume-elements. Normally for a 10-node-tetra-element it is only necessary to take care for a 3-node-tria-element. The surface-elements and their nodes are renumbered in a specified range of numbers. All other remaining nodes of the structure such as the volume-elements themselves are renumbered into a higher range of numbers. All nodes of each individually part of the structure will be found in specified ranges of nodes. The output of results can be reduced to the complete range of specified nodes for the show-model. Therefore only the results of the nodes of the complete surface-modes have to be stored, leading to small results file. An example of a cylinder-block containing 334384 volume elements and 602049 Nodes has only 154640 shell elements with 77242 nodes. In this example the show-model has only 13% of the nodes of the volume-model.

All parts of the structure will be individually calculated as super-elements. The connecting elements of the super-elements are created by the script and numbered in a specific range of nodes and elements, helping the user to identify and control all connections.

In order in decrease the process-overall-time the script is designed to help the user to avoid mistakes. As designed for frequency-analysis big problems are errors in mass and stiffness of the structure. For error in mass the script shows the mass of all parts of the structure, allowing

the user to chance the mass (instead of the density!). Therefore each part has a specified PID such as a specified MID with the same number. Errors in stiffness are mostly determined by missing connections between two parts of the structure. To help the user to find the missing connections the script allows to expand the structure. Inspecting the expanded structure the user will be able to control all connections by visually check of all RBEs.

The script renumbers all nodes and elements, creates a report-file and outputs data decks for each part to be calculated as super-element. The super-elements are calculated individually, getting the modal parameters of each super-element. Next the complete model will be calculated to get the modal values of the complete model, which will be stored and can be used for resonance-results and surface vibrations. For the reduced number of nodes the result-files are small. Therefore results of many models can easily be stored for later analysis. By use of test- or measured input-functions characteristic or specific results of the dynamic models will easily be created.

Process

The process works on four different levels within an database:

- Model base level
- Calculation level
- Result level
- Report level

Within the process a lot of automated processes are used:

- Batch Meshing
- Batch Preparation
- Batch Calculation
- Batch Reporting

Batch Meshing: creating and updating the model, storing in Model base level

Batch Preparation: creation the show-model, storing in Calculation level

Batch Calculation: creating the results, storing in Result level

Batch Reporting: creating the report, storing in Report level

Batch Meshing:

For the Batch Meshing procedure practical mesh-sets will be available. Based upon the complete CAD-model the single parts and part-groups will be separated, meshed and stored. The connecting points will be identified and specific elements positioned. The procedure will be tuned to create all meshes in a short period of time. All elements will be stored in special storage-levels, depending on already available models. All part-names include the date of meshing.

Batch Preparation:

Using the Batch Preparation procedure a model-update will be done to store additional information within the mesh-file to be available for the correct renumbering during the main preparation procedure. The additional information will be:

- Nodes for results,
- Nodes for loads,
- Nodes representing Surface elements for radiation and stress

To locate and control nodes for results and loads additional mass-elements with no-mass will be added. The surface-coating elements, created by use of the preparation procedure have to be deleted before storing the updated model. By using the preparation procedure again – e.g. for the complete structure – the structure will be updated with all information stored before. Therefore all information stored are available for each complete model using the single prepared parts.

Batch Calculation:

The script will export each part or group of parts as a data file of a super-element ready to run for Nastran or Epilysis. Within a second step all super-elements are calculated. In a third step the values of the modal analysis of the complete model are calculated, based on the results of the super-elements. In the last step some results are created based upon the modal values of the complete model. Next individually results will be created by the user. These results will be collected and compressed into graphical presentations by use of script-files.

Batch Reporting:

Script-files will help the user to create the desired reports quickly.

Results to be gathered:

For single parts such as engine-blocks a wide range of frequencies, e.g. up to 10 kHz, can be calculated. Except the lowest frequencies the vibrations of the structure are local vibrations. By test-functions the importance of the vibrations can be evaluated and stored for comparison with older or different models.

For complete models the calculation procedure will limit the available range of frequencies. A realistic frequency-limit will be 1 kHz, based upon the number of show-nodes. The calculation time will be about one day. The result-files will be limited to the nodes of the show-model and therefore rather small, allowing the storage of many different results.

Results to be created will be:

- Vibration modes for single parts like crank-cases or cylinder-heads,
- Elastic curves for the complete models
- Resonance frequencies for individually parts of the structure
- Velocity distribution on sides of different parts – summed in different frequency ranges
- Stress values of different parts of the structure, corresponding to loads and frequencies

Benefits for the user of the new procedure:

- The meshing procedure will be quick and easy
- The updated models will be available within a parts-database to be used for nearly each new structure
- All specific information such as nodes for results, loads, elements for radiation or stress are available in each new structure

- The calculation model will be an individually renumbered model with specific ranges of nodes and elements.
- The ranges of used nodes and elements are documented within a file.
- The numbers of load- and result-nodes do not vary much between different models
- The calculation procedure runs automatically, but gives the user process control
- The calculation procedure will be very fast.
- The result files are small due to the limited number of nodes used for the show-model
- Preparing the results will be easy by use of script- or session-files

The script will help the user with interest in acoustics to analyze the structures quick and accurate. It will help the user to avoid a lot of possible errors and to run multiple models even in small calculation departments.