Automated Integrated Finite Element Analysis Using MSC.Nastran and SDRC/I-DEAS

Ahmed El-Bahrawy
Mattias Nyström
Luleå University of Technology Volvo Car Corporation

ABSTRACT

The present work focus on showing how MSC.Nastran, as a Finite Element solver, when integrated to Computer Aided Engineering/Finite Element Analysis-system SDRC/I-DEAS could be used for multidisciplinary analysis and optimisation. Parametric solid models are used as a geometrical base for different downstream analysis applications during the analysis and optimisation, where the optimisation software Engineous/iSIGHT is used. This automated integrated multidisciplinary analysis and optimisation environment, where SDRC/I-DEAS is the pre- and post-processor, facilitates the balancing of different, and to some extent contradicting, properties and targets. The paper presents the environment and its advantage when applied in the design process. The developed system take advantage of the application programming interface of the CAD system, which consists of C++ classes giving access to the CAD functionality and geometry database.

INTRODUCTION

Legal, computational, and marketing demands on most industry is to develop more and more comfortable, safer, as well as lighter products with less fuel consumption, lower emission levels, longer lifecycle, fewer and faster maintenance cycles, and better quality. This challenge becomes highly complicated when taking the very important requirements of constantly decreasing design, production, product, as well as maintenance costs into account [1, 2]. To fulfil all these demands and pave the way for further future improvements the product development methodologies have to be based on more effective, dynamic, flexible, and robust design platform. Thus, a design development process has to, simultaneously, take care of all targets and deal with several different analysis disciplines in an integrated environment for concept evaluation, optimisation, verification, and multiphysics integration enabling us to reach overall best solutions and very closely survey the different conceptual limitations [3].

Moreover, considering the iterative nature of the modern product development process together with the demand of reduced lead-time, the integrated virtual multiphysical simulation techniques will play an important role. Besides making it possible to develop optimal multidisciplinary products the simulation techniques are also believed to reduce the number of iterations needed.

The use of Computer Aided Design (CAD) systems in industry for geometrical modelling and representation are well established [4]. The geometry can be used in downstream applications such as packing analysis, manufacturing simulations, etc. But transferring data between different systems such as CAD and Computer Aided Engineering (CAE) is not without problems [5]. Findings point out that parametric CAD tools are not being robust enough to represent complex geometry in a multidisciplinary environment [6]. However, they are identified as having the potential to provide a sharable common solid model for several disciplines [7].

Furthermore, many CAD systems offer the capability to perform Finite Element Method (FEM) [8] pre-processing and post-processing. This feature has a great potential as the solid geometry can be used for simulation directly without first being translated. When performing multidisciplinary analysis and optimisation [9] this can be advantageous as only one parameterised geometrical model is required.

In the present article an environment for automated use of MSC.Nastran [10], as a FEM-solver, and its superelement technique [11] in integrated multidisciplinary analysis (for instance strength, structural dynamics, thermodynamics, etc.) and optimisation where the CAD/FEA-system SDRC/I-DEAS [12] is the pre- and post-processor has been developed. Parametric solid models have been used as a geometrical base for different downstream analysis applications and optimisation. The optimisation software Engineous/iSIGHT [13] has been used to deal with the interacting disparate disciplines and to balance between the different, and to some extent contradicting, targets.

ANALYSIS ENVIRONMENT

The SDRC/I-DEAS CAD/CAE system allows user developed routines to be closely integrated. Access to the system database and the creation of functionality via the CAD/CAE software's graphical user interface has been achieved through the development of a number of application programmes using C++ [14]. The application programs communicate with the CAD/CAE system using a Common Object Request Broker Architecture (CORBA) [15] based Application Programming Interface (API). CORBA is an object oriented client/server architecture that is operating system and programming language independent. It provides object technology to build distributed systems. The client and the server processes are able to run on different machines connected by a network. This makes it possible for the application program and the CAD/CAE system to communicate across different platforms and hosts.

By using the SDRC/I-DEAS's API different application programs has been built enabling the user to control and run MSC.Nastran from within I-DEAS. The application programs uses the GUI of I-DEAS where users can choose between different ways to analyse the FE-model by using menus and icons in I-DEAS, see figure 1. The FE-models are solved using a computational server where MSC.Nastran is installed. One running I-DEAS session can be used to generate, execute and monitor different jobs running in parallel on the computational server. As the jobs are going through a queue system they may also be solved sequential if there are many jobs running. The FE-models can be located in different I-DEAS's model files, and the application program finds and exports the FE-models to MSC.Nastran and imports the result back into I-DEAS when the model is solved. To generate the bulk data file the MSC.Nastran translator in I-DEAS is used. The application program just monitors and controls the job. This process is fully automated and can therefore be used in an optimisation process.

When dealing with multidisciplinary optimisation using FE-models it is preferred [7] to use a parameterised geometrical model as a base for the different downstream analysis applications.

A CAD solid model consists of geometrical data describing the shape of individual geometric elements and topological data describing the connectivity of these elements. Most modern systems also store other data including the construction operations used to create the model and are a hybrid of the classic Constructive Solid Geometry (CSG) and Boundary Representations (B-Rep) [16]. The individual geometric elements, curves and surfaces, are most commonly represented using Non Uniform Rational B-Splines (NURBS) [17]. Topological data describes how the different geometric elements are bounded and how they are connected to constitute a solid model. A surface (often called a *face*) is bounded by *edges* which are in turn connected to each other at single points or *vertices*. In the CAD/CAE database, each of these geometric elements has a specific identification number that is used to update the simulation models and also allows individual curves and surfaces to be tracked.

The use of parameterised geometry as the basis for parameterised FE-models for optimisation makes the modelling more complicated than with non-parameterised solid geometry since the FE-models are sensitive to topological changes. Topological changes occur if for instance a surface gets an additional edge. The additional edge

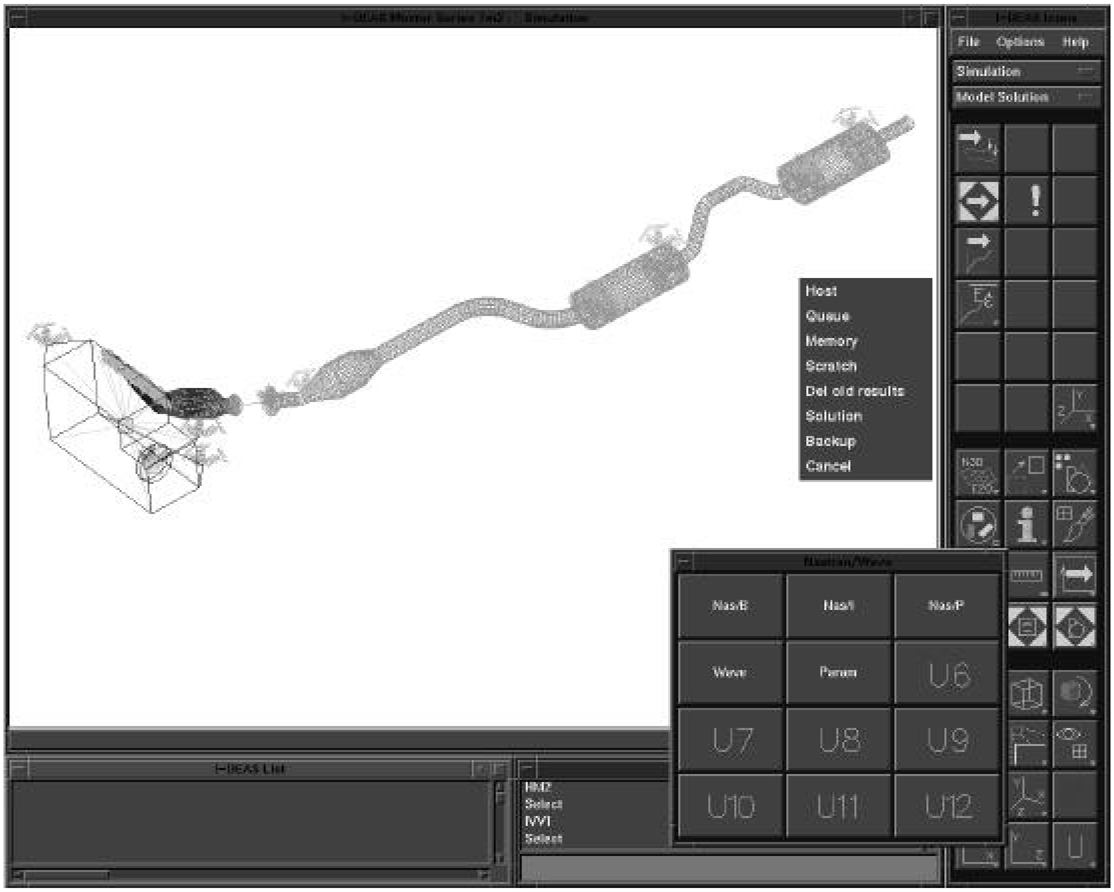

Figure 1. A part of the built interface to control and run MSC.Nastran from within I-DEAS

gives a complete different topology of the model and the FE-model based on the old topology will fail to update.

SDRC/I-DEAS can be used to create geometry based FE-models based on the parameterised solid geometry. These FE-models can be used in an optimisation process where an optimal geometry is desired. In geometry based FE-model, the boundary conditions as well as the mesh definition is related to the different geometrical elements of the solid model. When the geometry changes the mesh and the boundary conditions are automatically updated. Free meshing or mapped meshing can be used in this process.

In order to save CPU time and increase the level of detail, substructuring has been used. This divides the structure into smaller, simpler blocks called superelements [11]. When using substructuring, certain nodes are defined as the connecting nodes between the superelements so as to form a complete structure. However, when using an automated optimisation loop with free re-meshing, the node reference numbers will continuously change and even the positions of the connecting point may change. For this reason, it has been necessary to develop specific functions for calculation the new node numbers as an extension of the functionality of the pre-processor. This has enabled automated optimisation based on the use of substructures.

The functions developed take full advantage of the topological data available from the CAD/CAE database by associating the connections between the different superelements with vertices or edges. The meshing algorithm within the CAD/CAE system places nodes on the vertices. Thus if a vertex is used to define a connecting node, the node number can easily be calculated. Connecting node numbers can also be calculated by using the node closest to a given co-ordinate in the global space. Edges can also be used to define connecting nodes but here the number of nodes on an edge and the distance between the nodes must be fixed when meshing. Node numbers can also be calculated from the underlying surface of a face, using the u and v co-ordinates of points on the surface. As the u and v co-ordinates are a percentual measure of a position on the surface, they can be rather difficult to control if the length, width or other shape parameters are changed. Another limitation of using a surface is that only the node number closest to a position derived from u and v can be calculated.

The individual superelements have to be positioned in the global space so that their connecting nodes coincide. As the superelements that are to be geometrically optimised are based on the solid model, the solid model has to be translated and rotated in order to be in the proper position. This can cause problems as the local co-ordinate system of the solid model, which is also the co-ordinate system used by the FEM model, moves with the solid model. For consistency, all superelements should use the highest level co-ordinate system of the owning solid model and should be placed at a certain co-ordinate, for instance the point (0, 0, 0). Using different co-ordinate systems for the FE-model can cause errors. However, if the FEM mesh is generated and then transformed to the correct position in space it will reference the correct co-ordinate system. By building an assembly of the structure, the transformation matrices for the different instances of the solid models can be calculated. These transformations can then be applied to the FE-model. However, some limitations remain since geometry based boundary conditions cannot be moved and the mesh has to be completely deleted before generating a new mesh. The mesh definition can still be geometry based.

ANALYSIS OBJECT AND RESULTS

In this work an exhaust system is used to be analysed and optimised as a design object when investigating the developed multidisciplinary analysis environment. An optimisation with focus on structural dynamic analysis [18, 19], which could be a part of stepwise or simultaneous multidisciplinary coupled optimisation strategy, has been performed. The objective is to minimise the dynamical forces at the exhaust system suspension points. The exhaust system to be optimised is shown in figure 2.

The optimisation program Engineous/iSIGHT is used to provide the optimisation algorithms and run the developed environment through the Mechanical Computer Aided Engineering (MCAE) system SDRC/I-DEAS. MSC.Nastran is used to analyse the dynamics of the structure using the solution 103, i.e. normal mode analysis [17]. From the MCAE system an indata file to the solver will be exported and a normal mode analysis will be performed. After the model is solved, the results file

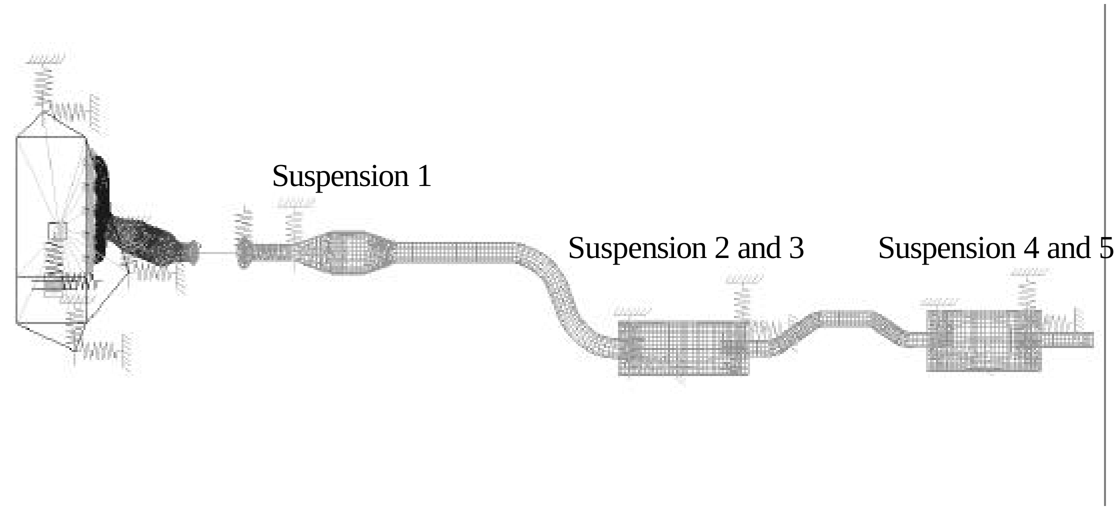

Figure 2. The exhaust system to be used as analysis object and its suspension points.

will then be imported to the MCAE system. In the MCAE system the results from the normal mode analysis will be used to perform a modal frequency response analysis. Finally, the results from the frequency response analysis will be post-processed. MSC.Nastran provides the capability to use sub-structuring which will be used to account for the dynamical interaction effects with the other subsystems such as the engine. The complete model consists of a substructure called a superelement (the engine, the engine suspension, and the de-coupler) and a residual structure (the exhaust system and it suspension), see figure 3.

The post-processed results from the frequency response analysis will be used by the optimisation program Engineous/iSIGHT when searching for an optimal design.

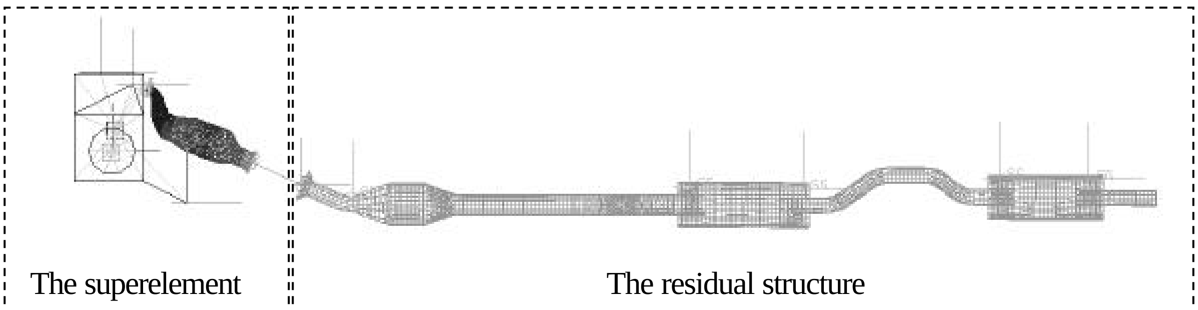

Figure 3. The superelement and residual structure.

During the structural dynamic optimisation of the system the length, width, and height of the first and second muffler as well as the position of the suspension points 2 – 5, see figure 2, are used as design variables. The objective is to minimise the maximum dynamic forces in the five suspension points in the frequency band 0.01-200 Hz. The first suspension point is at a static position just before the catalyst, e. i. the beginning of exhaust system in figure 2. The results of optimisation are shown in figures 4 – 8 as the acceleration levels, which are proportional to the dynamic forces, at the suspension points. The full line presents the reference levels and the dashed shows the optimised levels.

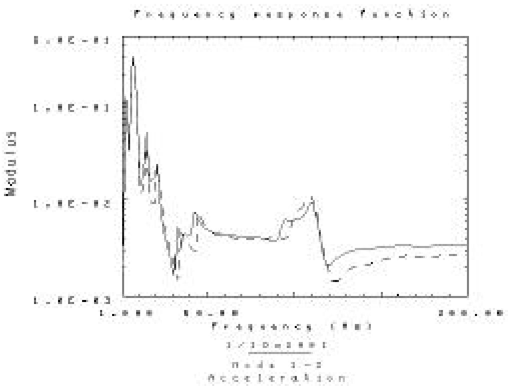

Figure 4. Acceleration at suspension point 1.

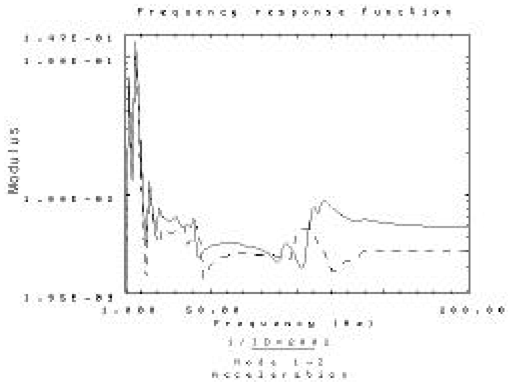

Figure 5. Acceleration at suspension point 3.

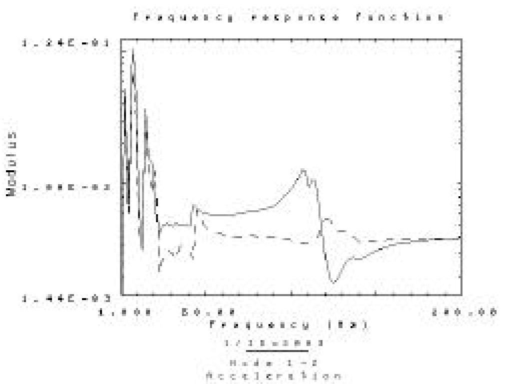

Figure 6. Acceleration at suspension point 3.

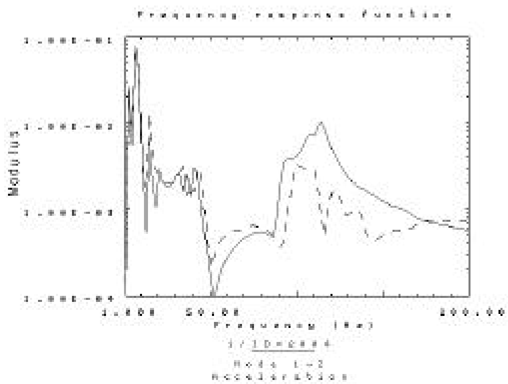

Figure 7. Acceleration at suspension point 4.

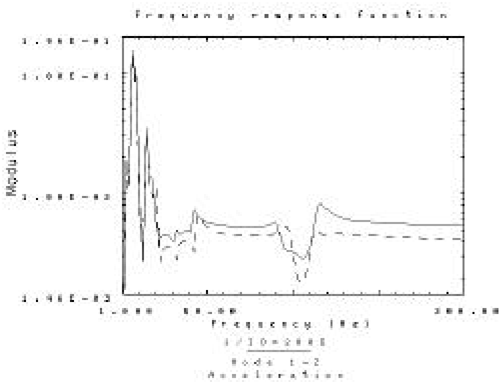

Figure 8. Acceleration at suspension point 5.

CONCLUDING REMARKS

The complexity of successfully developing and optimising technical systems with multidisciplinary physics (for instance, vehicles or mechatronic systems) necessitates the use of an integrated virtual product development environment. Besides making it possible to do it 'right first time' and develop optimal multidisciplinary products, such environment is also believed to reduce the number of iterations needed and reach overall best solutions in shorter lead-time when being used very early in the design process. That will have high impact on product, design process, and production costs.

An automated integrated multidisciplinary design environment using well-established conventional software, e. g. MSC.Nastran, for different physics will enable us, on neutral bases, to perform the conceptual analysis and achieve optimum compromises when balancing different, and to some extent contradicting, targets and properties that are closely associated to one or several interactive subsystems.

A virtual design environment that is totally based on well-established conventional software will have high flexibility and ability to be extended to cover more physics and processes, e.g. the production and manufacturing processes, Digital Make Up (DMU), and Project Data Management (PDM).

By integrating different commercial software using their API, information can be stored in its native format and still be shared between different programs. API built in client/server style makes it possible to access information at run time using the different programs distributed and connected by a network.

Integrated virtual product development based on common geometry models that are able to be shared between different disciplines is very useful in the design process, specially when performing multidisciplinary optimisation.

When using substructuring and dividing a system into subsystems, the solid models can be less complex. It is easier to control a smaller parametric solid model than a larger. This can be helpful as several errors in the optimisation process could be related to topological changes of the parameterised geometry. The sensitivity for topological changes is a limitation for some applications.

REFERENCES

1. Isaksson, O., "*Computational Support in Product Development*", Doctoral Thesis, Luleå Univerity of Technology 1998.

2. Gartmeier, O., "*The Role of Structural Optimisation in Automotive Design-Past, Present, and Future*", 32nd ISATA proceedings Simulation, Virtual Reality, and Supercomputing, Automotive Applications, 1999.

3. Nyström, M., "*Multidisciplinary Optimisation with Application to Exhaust System Design*", Proceedings of AIAA Conference 2000.

4. LaCourse, D. E, "*Handbook of Solid Modeling*", MaGraw-Hill 1995, ISBN 0-07 -

035788-9.

5. Peak, R. S., Fulton, R. E., Nishigaki, I., Okamoto, N., "*Integrating Engineering Design and Analysis Using a Multi-representation Approach*", Engineering with Computers (1998) 14:93-114.

6. Röhl, P. J., He, B., Finnigan, P. M., "*A Collaborative Optimization Environment for Turbine Engine Development*", AIAA-98-4734.

7. Giesing, J G., Barthelemy, J. M, "*A Summary of Industry MDO Applications and Needs*", AIAA-98.

8. Cook, R. D., Malkus, D. S., Plesha, M. E., "*Concepts and Applications of Finite Element Analysis*", John Wiley & Sons 1989, ISBN 0-471-84788-7.

9. Vanderplaats, G. N., "*Numerical Optimization Techniques for Engineering Design*", MaGraw-Hill 1984, ISBN 0-07-066964-3.

10. MSC/NASTRAN, "*Quick Reference Guide*", Version 70.5, The MacNeal-Schwendler Corporation 1998.

11. MSC/NASTRAN, "*Superelement Analysis*", The MacNeal-Schwendler Corporation 1997.

12. I-DEAS Master Series, "*Smartview*", Version 7.0, SDRC 1999.

13. iSIGHT Version 5.0, "*User Manual*", Engineous Software 1999.

14. Stroustrup, B., "*The C++ Programming Language*", 3rd ed., Addison-Wesley 1997, ISBN 0201889544.

15. Siegel, J., "*CORBA Fundamentals and Programming*", Wiley 1996, ISBN 0-471-12148-7.

16. Mortenson, M. E., "*Geometric Modeling*", Wiley 1997, ISBN 0-471-12957-7.

17. Piegl, L., Tiller, W., "*The NURBS Book*", 2nd ed., Springer-Verlag 1997, ISBN 3540615458.

18. MSC/NASTRAN, "*Basic Dynamic Analysis User's Guide*", Version 69, The MacNeal-Schwendler Corporation 1997.

19. MSC/NASTRAN, "*Advanced Dynamic Analysis User's Guide*", Version 70, The MacNeal-Schwendler Corporation 1997.