GENERIC LINKING OF FINITE ELEMENT MODELS FOR NON-LINEAR STATIC AND GLOBAL DYNAMIC ANALYSES OF AIR-CRAFT STRUCTURES

A.J. de Wit[1], D. Akçay Perdahcıoğlu[2], W.M. van den Brink[1], A. de Boer[2]

[1] *National Aerospace Laboratory - NLR, Collaborative Engineering Systems Department, the Netherlands, adewit@nlr.nl, brinkw@nlr.nl*
[2] *University of Twente, Faculty of Engineering Technology, the Netherlands, d.akcay@ctw.utwente.nl*

Abstract

Depending on the type of analysis, Finite Element(FE) models of different fidelity are necessary. Creating these models manually is a labor intensive task. This paper discusses a generic approach for generating FE models of different fidelity from a single reference FE model. These different fidelity models are created by sub-structuring and a multi-scale superposition technique. Efficiency of the developed approach is demonstrated via non-linear static and modal analysis of a carbon-fiber stiffened panel.

Introduction

During take-off, flight and landing, an aircraft structure experiences different kind of loading conditions, e.g. pressure loads, thermal loads, dynamic loads, etc. One approach to determine the structural responses due to these external loads is using the Finite Element(FE) method. Typically, each loading condition imposes different fidelity requirements to the FE model and utilizing the same model in the analyses may not be computationally efficient. In the context of non-linear static and dynamic analysis of complex aircraft structures, two different approaches can be followed to generate efficient FE models, namely sub-structuring and *multi-scale analysis*. The idea behind sub-structuring is to divide the whole structure into sub-structures. From these sub-structures reduced models are generated. In multi-scale analysis the whole system is analyzed via a coarse global model. Detailed sub-models are included in regions where critical areas are located. Hence, detailed analysis of the complete model is limited to regions where deficiencies are expected. Therefore, global responses can be evaluated more efficiently. If the multi-scale analysis consists of a hierarchy of a coarse scale and a fine scale coupled model(s), it is often referred to as global-local analysis in the literature.

The objective of this paper is to generate different fidelity models from a given structural coarse global FE model. Thereby, details can be added or extracted to/from this model depending on the analyses that are to be performed. This way, analysis times are reduced in a significant extent. The focus of this research is, currently, on nonlinear (quasi-static) stress and modal analyses. These are performed on FE models generated; using global-local analysis and sub-structuring methods, respectively; from a given coarse meshed global FE model. In the paper, a generic automated approach is presented to reduce the modeling efforts associated with inclusion of local detailed FE models into a global FE model. The coupling between the local and the global models is defined using a tight two-way coupling methodology.

In the first section of this paper, the generic global-local analysis for the non-linear stress analysis and the utilized coupling methodology is explained in detail. The second section covers the sub-structuring and the reduction methods that are taken into account for the modal analysis. Efficiency and accuracy of the introduced methods are demonstrated on a carbon-fiber stiffened panel in the third section. Finally, in the fourth section, the conclusions and the directions for future work are given.

Global-Local Analysis and Tight Two-way Coupling Methodology

A global-local analysis consists of an initial phase analyzing the entire structure discarding details that may have no significant contribution on the structures' overall behavior. Local details are included after the global analysis has identified possible critical areas that may influence global structural behavior. In this research, local FE models are inserted into the global FE model automatically. Firstly, a failure criterion to identify critical areas, e.g. maximum allowable stress, is chosen. Second, the critical areas (elements) exceeding the maximum allowable failure index are detected. These detected elements are used to determine the size and the shape of the local models. The size of the local models is at least the size of the detect critical areas. Finally, the coupling between the global and the local model is accomplished by a superposition technique.

Hierarchical multi-scale superposition techniques provide an accurate and computationally efficient means to include detailed physical behavior into a coarse scale model [1]. The present work uses a variational multi-scale framework [2] to decompose a finite element model into separate scales. Therefore, an additive split of the solution $\mathbf{u}$ into a global large scale contribution $\overline{\mathbf{u}}$ and local small scale contribution $\mathbf{u}'$ is introduced: $\mathbf{u} = \overline{\mathbf{u}} + \mathbf{u}'$. In the present paper, the global large scale is referred to as *global* and local small scale is referred to as *local*.

The contribution of the local displacements to the global displacements is assumed to be $\mathbf{u}' = \mathbf{0}$ on the boundaries of the local model to allow the static condensation of the local solution. This locality assumption is valid as long as the size of the local model is taken large enough such that the contribution of $\mathbf{u}'$ to $\overline{\mathbf{u}}$ vanishes within the region of the global-local model.

Adjustments to the superposition technique[2] are necessary to incorporate the method into the testbed B2000++ [3]. In particular, expressions that take into account the coupling between the global and the local model are considered cumbersome. The issue is implementing the procedure for generic finite elements that are used for coupling the global and the local model. All expressions depend on combining $\mathbf{B}$ matrices (second derivative of the element shape functions) of the global and the local elements to compute the element stiffness matrix. Combining $\mathbf{B}$ matrices requires a software implementation effort comparable to implementing a new finite element for each combination of the global and the local finite elements. Therefore, typically the same type of elements are used for the global and the local model [4].

To overcome the issue of combining $\mathbf{B}$ matrices, two modifications are made. First, the interpolation of the global-local area is modified. Second, the locality assumption, $\mathbf{u}' = \mathbf{0}$, is explicitly imposed on the boundary of the global-local displacement field.

Instead of interpolating the global displacements via global interpolation functions, the local interpolation functions, $\mathbf{N}'$, are applied to the global displacements. The global displacements within the two-scale region are interpolated by $\overline{\mathbf{u}} = \mathbf{N}'\hat{\overline{\mathbf{d}}}$. The vector $\hat{\overline{\mathbf{d}}}$ is obtained utilizing a displacement nodal coupling matrix $\mathbf{P}$ which links every local nodal displacement ($\mathbf{d}'$) to every global nodal displacement ($\overline{\mathbf{d}}$) within a global-local region. Hence, it contains the position of each local node within the global element.

Imposing the displacement boundary condition, $\mathbf{u}' = \mathbf{0}$, on the nodal displacements that lay on the boundary of the global-local region is accomplished using a reduction matrix $\mathbf{Q}$. Hence, $\mathbf{d}' = \mathbf{Q}\hat{\overline{\mathbf{d}}}$. This displacement boundary condition is present in the global and the local finite element equations.

Finally, to obtain additively separable global and local displacement fields the nodal degrees of freedom of the local elements that coincide with the global nodal degrees of freedom are fixed. The resulting superposition does not introduce stress increments on the global-local interfaces for small rotations of the shell elements.

Modal Analysis via Sub-structuring and Reduction

The global model is too detailed to be employed in modal analysis if an engineer is only interested in the lower natural frequencies and the corresponding mode shapes. Using sub-structuring and reduction methods to condense such global models before performing modal analysis is important to improve computational efficiency. The benefits can be summarized as: **(1):** A complex structure is divided into several substructures (components). The analysis of each component can be assigned to different groups and/or computers. Hence, parallel processing opportunities are highly supported, **(2):** The number of d.o.f. in a large FE model is reduced significantly while the accuracy of the analysis is preserved within a low frequency range, **(3):** Analyses of component models are independent from each other. When modifications are required in a certain component, only reanalyzing this part is sufficient to determine the modified matrices of the structure. The unmodified substructures do not have to be analyzed again, **(4):** It is possible to use one parametric FE model for all the similar substructures. Thus, for structures consisting of repetitive components, generating the complete model is not necessary. In this research, the *Craig-Bampton (CB) method* [5] is used for this purpose. Moreover, it is combined with an *Interface Reduction (IR) methodology* [6] to reduce the size of the CB based FE model even further.

After dividing the FE model of the complete structure into non-overlapping substructure FE models, the next step is the condensation of these models. In the CB method, this is achieved by projecting the substructure system matrices into a smaller subspace using the *fixed interface normal modes*, and the *constraint modes*. The first set of modes describes the internal dynamic behavior of a substructure. These are calculated by restraining all d.o.f. at the component interface and solving an undamped free vibration problem. The motion on the substructure interfaces, the propagation of the forces between substructures and the necessary information about the rigid body motions are defined via the constraint modes. These modes are calculated by statically imposing a unit displacement to the interface d.o.f. one by one while keeping the displacements of the other interface d.o.f. zero and assuming that there are no internal reaction forces.

Coupling of the CB substructure models requires both compatibility and force equilibrium to be satisfied at the interfaces of the components. The coupled system of equations of a CB model are obtained via a primal formulation. This formulation is based on defining a unique set of interface d.o.f. and eliminating the interface forces using the equilibrium on the interface. Finally, the modal analysis is performed on the coupled reduced system matrices.

While the number of internal substructure degrees of freedom (d.o.f.) can be reduced with a significant amount, the number of interface d.o.f. is preserved in the CB model. The number of interface d.o.f. is therefore a bottleneck in the computational efficiency of CB, especially when the number of substructures or the interface d.o.f. is high. The Interface Reduction (IR) method is developed for the purpose of solving this issue. The idea behind the method is statically condensing a structure on its interface, performing an eigenvalue analysis on the condensed model to compute a truncated number of modes and finally utilizing them as a basis to reduce the number of interface d.o.f. The IR method is very effective in condensing the CB model while maintaining a certain accuracy.

Demonstration of the Concepts

The introduced concepts are demonstrated on a carbon fiber reinforced panel stiffened via three I-section stringers with tapered feet. The structure is clamped at the edges parallel to the y-axes. Each stringer is manufactured from four laminates; two of these laminates form the C-sections back-to-back, the third laminate forms the tapered base and the fourth laminate is placed on the top, see Figure 1. In ad-

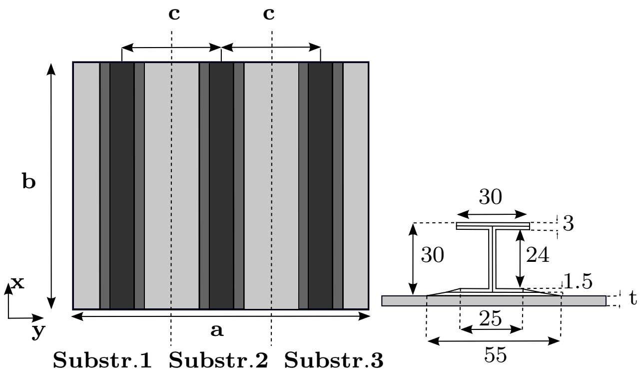


Figure 1: Test problem. Carbon fiber reinforced panel stiffened via three I-section stringers with tapered feet. The dimensions of the panel are $a = 360mm$, $b = 300mm$, $c = 120mm$, $t = 4mm$.

dition, both the shape and the dimensions of the panel and the stringer are summarized in Figure 1. The panel is manufactured from a typical aerospace material, i.e. Hexcel T800/924. The composite lay up is created from the stacking sequence $[(+45^\circ/-45^\circ/0^\circ/90^\circ)_4]_s$. The laminate lay up of each stringer is $[(-45^\circ/+45^\circ/0^\circ)_4]_s$. The orientations at 0° and 90° correspond to the x-axis and y-axis, respectively. The material properties of T800/924 are: The elastic moduli are $E_{11} = 155GPa$, $E_{22} = 8.57GPa$, Poisson's ratio is $\nu_{12} = 0.33$, Shear modulus is $G_{12} = 7.4GPa$ and the ply thickness is $0.125mm$. The remaining material properties are chosen as $E_{22} = E_{33} = 8.57GPa$, $\nu_{12} = \nu_{13} = 0.33$, $\nu_{23} = 0.052$, $G_{13} = G_{23} = 7.4GPa$ and the density $\rho = 1630kg/m^3$. The FE analyses are carried out using shell elements for the global model and 3D volume elements for the local models. The strain calculated for shell and volume models is the Green-Lagrange strain and the stress is the Cauchy stress.

The algorithms used in this study are currently available on different platforms. The non-linear static analyses are carried out within the FE program B2000++. The response of the global model is calculated using a Newton-Raphson (NR) iteration process. The response of the local model is included as a non-linear boundary constraint in the global model analysis. The detailed models are automatically created via a sub modeling program written in Python that uses the B2000++ pre-processor to generate the nodes and the finite element connectivity list. To perform modal analysis, the complete structure is divided into three identical components. Each component consists of one third of the panel and a single stringer, see Figure 1. The coarse global (full) and the reduced models are generated utilizing the model of a single substructure. This model is generated in ANSYS and its system matrices are transferred to MATLAB using the in-house code AMIPS. The CB and the IR algorithms utilized for model reduction are written in MATLAB. The generated full and reduced substructure models are also assembled and solved in MATLAB.

Validation: Global-Local Analysis and Tight Two-way Coupling Methodology

Critical areas within the panel are determined during a non-linear static analysis. The edges of the panel parallel to the x-axis (Figure 1) are unconstrained. The structure is compression loaded via an incremental displacement load applied at the top side of the plate parallel to the y-axis (Figure 1). The displacement load is stepwise increased up to a value of $-2.00mm$. Approximately 2000 shell elements are used within the global FE model to mesh the geometry. The reference model used to compare the global-local result is meshed via 15000 3D volume elements.

Local models consisting of 3D volume elements are generated in regions of the shell element model where the failure index exceeds 0.57. Hence, 43 percent of the strength of the element in this region is left. The global-local approach is compared to a reference model created with 3D volume elements. In both models the failure index is determined via a Hashin[7] failure criterion. Additional parameters necessary for this failure criterion are presented in Table 1. The results for the reference volume model and global-local model are shown in Figure 2.

Table 1: Material properties used for the different failure criteria.

parameters	Hashin[7]
maximum tension fiber direction	1982.0 MPa
maximum tension orthogonal to fiber direction	48.69 MPa
maximum compression fiber direction	1550.0 MPa
maximum compression orthogonal to fiber direction	250.0 MPa
maximum shear strength	113.0 MPa
maximum shear strength	113.0 MPa
maximum shear strength	113.0 MPa

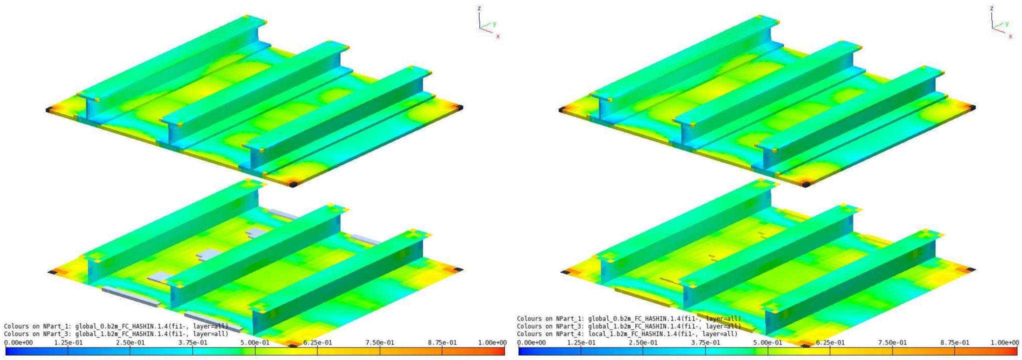


Figure 2: Failure index of a compression loaded stiffened plate modeled via volume elements (top) or modeled via local volume models superimposed onto a global shell element mesh (bottom). In the left figure the local models are grey for clarity. In the right figure the local models are colored with the computed failure index. Inclusion of local models is limited to the region inbetween the stiffeners.

Inclusion of the local models is limited to the region inbetween the stiffeners, see Figure 2(bottom). High failure indices at the clamped edges are due to introducing the loading. Between the stiffeners a non-symmetric displacement pattern typical for buckled plates is visible. Hence, the skin left to the central stiffener buckles before the right part of the plate. Here, strains are high and a 3D volume model is necessary to accurately predict the remaining strength of the material. The bottom figures show the global-local models are created in the areas where high failure indices are computed. These areas are similar to the high failure index areas in the reference model(top). Because the global shell element model uses far less elements then the reference model the region with high failure indeces is larger in the shell element model. However, the magnitude of the computed failure index in both the reference model and the global-local model are comparable.

During the global-local analysis with Hashin failure criterion no numerical difficulties were observed with respect to the convergence of the incremental loading steps. In addition, the insertion of local models at the boundaries of the global model did not pose any difficulties for the user. The boundary conditions are applied on the global model only and automatically accounted for in the local model. Furthermore, no stress jumps were observed at the interfaces of the global and global-local region. However, the numerical effort required to perform the static condensation of the local models is significant. Therefore, future research will focus on improving numerical performance to enable up-scaling of the method to larger global models, e.g. barrel sections.

Validation: Sub-structuring and Reduction Methods

The size of the full (global) and the reduced FE models and, the number of the reduction components are summarized in Table 2.

Table 2: The number of d.o.f. and the reduction components

Method	♯ of Fixed Interface Modes	♯ of Constraint Modes	♯ of IR basis	♯ of d.o.f.
Full (Global) Model	-	-	-	12456
Craig-Bampton (CB)	18 (per substructure)	150 (per substructure)	-	354
CB&IR_6	18 (per substructure)	-	6	60
CB&IR_{12}	18 (per substructure)	-	12	66
CB&IR_{18}	18 (per substructure)	-	18	72
CB&IR_{20}	18 (per substructure)	-	20	74

The accuracy of the reduction methods are computed on the basis of the relative frequency error (ε_ω) and the Modal Assurance Criterion (MAC). For the calculation of ε_ω, the full FE results are compared with those of the IR and the CB methods using

$$\varepsilon_\omega = \frac{|[\mathbf{\Lambda}_{\text{Full}}]_{jj}^{\frac{1}{2}} - [\mathbf{\Lambda}_{\text{R}}]_{jj}^{\frac{1}{2}}|}{[\mathbf{\Lambda}_{\text{Full}}]_{jj}^{\frac{1}{2}}} \quad j = 1,\dots,20$$

where “R” represents the reduction method, $|\cdot|$ is the absolute value and $\mathbf{\Lambda}_{jj}$ is the jth diagonal entry of the spectral matrix $\mathbf{\Lambda}$. MAC is a scalar value between 0 and 1, and it represents the correlation number between the two mode shapes. A MAC value close to 1 indicates a high degree of correlation between two mode shapes. For its calculation, first the reduced model solutions are expanded to their full forms and then these eigenvectors are compared with the eigenvectors calculated by full FE analysis using

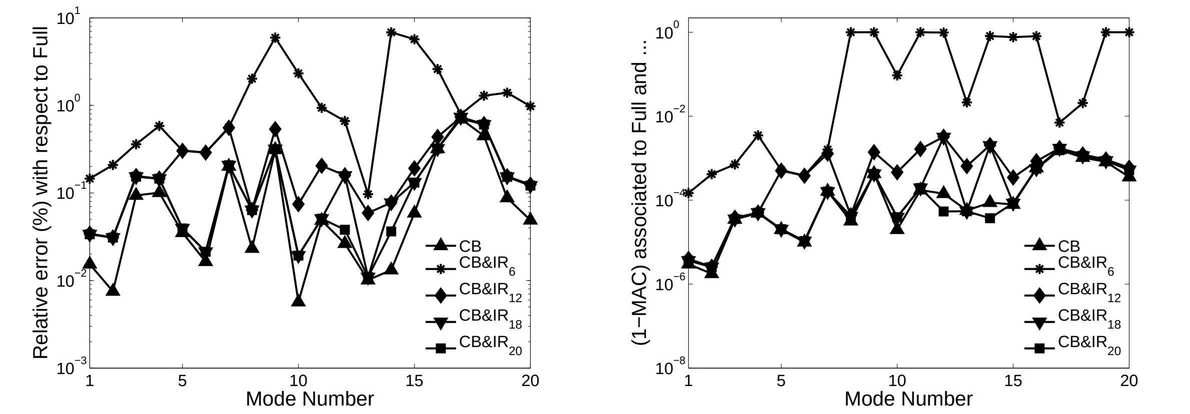
$$\text{MAC} = \frac{([\tilde{\Phi}_{\text{Full}}]_j^{\text{T}}[\tilde{\Phi}_{\text{R}}]_j)^2}{([\tilde{\Phi}_{\text{Full}}]_j^{\text{T}}[\tilde{\Phi}_{\text{Full}}]_j)([\tilde{\Phi}_{\text{R}}]_j^{\text{T}}[\tilde{\Phi}_{\text{R}}]_j)} \quad j = 1,\dots,20 \tag{1}$$

where $[\tilde{\Phi}_{\text{Full}}]_j, [\tilde{\Phi}_{\text{IR}}]_j$ are the jth eigenvectors corresponding to the full FE analysis and the reduction method, respectively.

The results of the test problem are presented in Figure 3.

The results show that the accuracy of the CB method is in good agreement with the reference values and, the utilized d.o.f. are significantly less than those of the global FE method. The maximum relative error among the first twenty natural frequencies with respect to the full FE results is less than 1%. In addition, the computed mode shapes show a good correlation with the global FE results.

Further reduction in the CB model is obtained by employing Interface Reduction (IR) methodology. The IR model with 6 basis vectors is in good agreement with the reference values up to the 7th natural frequency. The results show that increasing the number of basis vectors makes the solutions converge to the CB results. The accuracy of the computed first 20 eigenvalues and the corresponding frequencies are in good agreement with the reference values for the IR model with 18 basis vectors. The number of d.o.f.



(a) Relative errors of frequencies with respect to the Full FE results

(b) Correlation of the mode shapes with the Full FE results

Figure 3: Results of the CB and the IR methods

in this model is 173 times less than that of the global model. The results clearly show it is possible to accomplish a significant reduction of the size of the global model while preserving an acceptable degree of accuracy for modal analysis problems.

Conclusions and Future Work

A global-local analysis method and, sub-structuring and reduction methods are applied to generate different fidelity models from a given coarse meshed global FE model. These methods are studied for their accuracy and efficiency when applying them to nonlinear static and modal analyses of aircraft structures. The current research presents application of these methods on a carbon-fiber stiffened panel. The aim is to extended the current implementations to a barrel section of a fuselage in the future.

Applying a multi-scale approach to couple global and local models of different fidelity was found to be accurate in terms of displacements and stresses along the global and global-local region. In addition, the method was found efficient when automating the creation and insertion of the local models with respect to boundary conditions applied to the model. Local models are inserted at the boundaries of the global model without additional effort to translate the boundary conditions to the local model. However, static condensation of the local models was found to be computationally demanding. Therefore, applying the current implementation to larger models such as barrel sections seems challenging. Further research to overcome this limitation will focus on different procedures for computing the static condensation step.

Sub-structuring and reduction methods , namely Craig-Bampton (CB) and Interface Reduction (IR) are very promising methods for efficient modal analysis of aircraft structures. The size of the global FE model can be reduced significantly while the accuracy of the lower natural frequencies are being preserved. Currently these methods are implemented into MATLAB which will be soon available in the commercial FE package B2000++. Moreover, the CB method will be extended for the analysis of structures where its substructures have non-matching interfaces. For efficient modal analysis, FETI method will also be implemented into CB.

The research presented in this paper is part of a larger ongoing effort to automatically generate different FE fidelity models within a single design platform. This platform handles the communication between the different fidelity models and the exchange of analysis results. Therefore, user interaction to analyze a structure for different analysis cases is reduced to a minimum. The carbon-fiber stiffened panel is a suitable example in this context. It contains all the relevant characteristics of an industrial use case.

Acknowledgement

The authors wish to acknowledge Mathias Doreille and Thomas Ludwig from SMR for their suggestions and support to include the developed approaches into the program B2000++. Part of the research leading to these results was carried out within the project "generic linking of Finite Element based Models - glFEM" which has received funding from the European Community’s Seventh Framework Programme (FP7/2007-2013) under grant agreement no 234147.

References

[1] J. Fish and K. Shek. Multiscale analysis of large scale nonlinear structures and materials. *International Journal for Computational Civil and Structural Engineering*, 1:79–90, 2000.

[2] Thomas J.R. Hughes, Gonzalo R. Feijóo, Luca Mazzei, and Jean-Baptiste Quincy. The variational multiscale method - a paradigm for computational mechanics. *Computer methods in applied mechanics and engineering*, 166:3–24, 1998.

[3] SMR Engineering & Development SR. B2000++, 2011.

[4] A. Hund. *Hierarchische mehrskalenmodellierung des versagens von werkstoffen mit mikrostruktur*. PhD thesis, Universitaet Stuttgart, 2007.

[5] R. R. Craig and M. C. C. Bampton. Coupling of substructures for dynamic analysis. *AIAA*, 6(7):1313–1319, 1968.

[6] R. R. Craig and C. J. Chang. Substructure coupling for dynamic analysis and testing. Technical report, NASA, CR-2781, 1977.

[7] Zvi Hashin. Failure criteria for unidirectional fiber composites. *Journal of Applied Mechanics*, 47:329–334, 1980.