MATEC Web of Conferences 45, 02005 (2016)
DOI: 10.1051/matecconf/20164502005

Stiffness Equivalent Finite Element Modelling of a Physical Assembly by Structural Optimization Method

Engin Metin Kaplan, Kamuran Kamil Yeşilkaya and Kemal Yaman

Defence Industries Research and Development Institute, Turkish Scientific and Technical Research Council, (TÜBİTAK-SAGE), 06261, Ankara, Turkey

Abstract. In this study, stiffness equivalent finite element modelling (EFEM) technique of a physical assembly which has several detailed electronic, mechanic and hydraulic parts is presented. Physical assembly can be evaluated as a subsystem. Outer geometry of the detailed subsystem is meshed by shell elements. Beam elements, of which cross sectional dimensions are found by structural optimization, are used at the inner section of the subsystem. To reach structural integrity, beam elements are connected to the outer shell with rigid elements. The deflection results of the detailed finite element model (DFEM) which is constructed by 3D solid elements are investigated under unit force laterally. The deflection results are defined as objective function in structural optimization of the EFEM. Design parameters are selected as rectangular cross section dimensions of the beam elements. The deflection results of the EFEM with optimized beam cross sections and DFEM indicate good agreement. Moreover EFEM is computationally efficient for the deflection analysis. It is shown that EFEM solution time is %1 of the DFEM.

1 Introduction

Aerospace and Defence industry is one of the most attractive market in the world [1]. Most of the worldwide and innovator companies do their designs according to their specialty, as coordinaly. Companies, which are responsible for subsytem design, do their desing about their specialty and deliver it to the system designer company. In delivery process, subsystem companies do not prefer to transfer their know-how to the system designer company. They prefer to deliver their subsytem in closed form and give information about input and output relationship of subsytem with the system. On the other hand, structural strength of the subsytem is important for the overall system strength. All detailed part of the subsytem affects the total system stiffness and must be defined as it is.

In this study a subsytem, of which outer geometry is known, and involves some detailed parts like electronic, mechanic and hydraulic are equivalently modelled with real detailed model concerning mechanical stiffness, by shell and beam elements. To reach the accurate results, firstly, real detailed 3D model of the subsytem is solved under unit forces. Deflection results of the track points are determined. These deflection result are seeked in the structural optimization process of the basic shell-beam equivalent model by sizing the beam cross section dimensions.

The solution time of EFEM and DFEM input files are also compared in this study.It is seen that EFEM is pretty short accoring to DFEM. Since two models are equal in stiffness, EFEM can be used for multiple parametric loading cases.

2 Literature Survey

Equivalent modelling by using optimization technique is useful and computationally efficient process. Various studies are available for the solution in literature.

Several researchers have worked on equaivalent modelling technique. Yu Wang et all developed three-step optimization strategy. They use NASTRAN for their application. They performed their study on the wing-type structure made of composite material. Researchers interest not only static behaviour, but also dynamic, buckling and aeroelastic behaviour. Their static displacement results of the equavalent modelling with the detalied modeling are in a good agreement. In addition to that, their modal analyses results show that first ten elastic modes are equally modelled in %5 range with detailed finite element model by optimization technique. Moreover, first two buckling mode patterns are so similar. Lastly, flutter behaviour results of the both modelling are very close. The results show that the optimization technique for the composite-wing systems is adventageous, and the equivalent finite element model prepared with the rapid modeling is appropriate for the initially structural optimization with the static and dynamic constraints [2].

Jasson Gryzagoridis et all are studied on estimation flexural rigidy of the composite panels. They also performed experimental varification about their analytical approach. Analytical approach is simply based on the euler-bernouli beam theory. They used cantilever beam in calculations. Their experimental results show that, analytical approach predicts the behaviour of the sandwich composite panel[3].

A. Moroncini et all built simple finite element model which is equivalent to the detailed model by using beam and shell elements only. They used NASTRAN both finite element analyses and structural optimization. Beam cross-section parameters are defined as design region in their analyses. The researchers emphasize that this technique is useful for initially design process, for predicting best alternative solution on vibration and acoustic behavior of the structure [4].

Fenghe Wu et all worked on optimization of the ram, which is like beam structure by using equivalent elastic modulus method. They used this method to super-heavy-duty CNC floor-type milling and boring machine. Instead of defining the complex constrains and contact parameters of the detailed finite element model, simple optimized model is used. In optimization technique, objective function is minimum volume under static loading. Furthermore, displacement and stress values are defined as constraints. Accoring to the results, basic method of the equivalent elastic modulus can effectively control the scale of optimization calculation and provide optimization going on without problem [5].

David J. Malcolm et all studied on blades modelled by equivalent beam elements for aeroelastic analysis. They use ANSYS for finite element modelling and MATLAB for equivalent model calculations. Since, their concern is dynamic equivalent modelling of the blades, their model must be also mass equivalent. Location of the elastic axes, orientation of principle axes and coupling between terms of the equivalent model should reflect 3D detailed blade parts. In conclusion, their equivalent beam modelling method demonstrates significant impression for easily involving all properties into beam elements for dynamic simulations [6].

T. R. Baumann studied on formulation of equivalent stiffness by experimental method. Researcher builds the stiffness matrices in form of two by two. He used beam elements in equivalent system and concentrated on shear and bending stiffness values. Author claims that, the equivalent model can be used in dynamic analysis. Researcher also points out that some singularity issues can occur in the model and it must be controlled. Results are acceptable for arbitrary length beams [7].

J. Sun et all worked on determination of elastic properties and lateral stiffness of multi-ribbed walls that is use in load-resistant elements in recent years. They used Mori-Tanaka method to derive analytical expression on equivalence elastic property determination. Furthermore, they verified their approach by both experimentally and numerically. They did static tests on six wall specimens, and performed finite element analysis of detailed model [8].

3 Theory

NASTRAN is used for the finite element analysis. The governing equation of the linear static finite element analysis is given below.

$$[K]\{u\} = \{P\} \tag{1}$$

Here K represents system stiffness matrix. P is load vector and u is displacement of the nodes. Displacements are unknown in the Equation 1. Displacements is obtained by inverting the stiffness matrix and multiplying it by the force value [9] .

GENESIS is used for the structural optimization. Basically, numerical optimizations solve the problem: Seek the set of the variable OF_i, i=1, N included in the vector OF that minimizes F(OF).
Subjected to;

$$g_j(OF) \leq 0 \quad j = 1, M \tag{2}$$

$$h_r(OF) = 0 \quad r = 1, S \tag{3}$$

$$OF_i^L \leq OF_i \leq OF_i^U \quad \text{i=1, N} \tag{4}$$

Here OF is the objective function. It is calculated by design variables. g_j and h_r are inequality and equality constraints, respectively. OF_i^L and OF_i^U are the lower and upper region bounds of the design variable.

In sizing optimizations, element cross sectional parameters (height, width, etc.), section parameters (area, area moment of inertia, etc.), structural parameters (length, span, etc.), and responses (strains, stresses, eigenvalues) can be studied. The cross sectional parameters, section parameters, and node locations construct the optimization model. The designer interests design variables that, generally, have some sort of nonlinear relationship with the definition of the analysis model, and constraints on the produced by the analysis model. The sum of design variables, constraints, and the objective function construct the design model. The fundamental design capabilities are used to communicate the analysis model to a basic design model [10].

4 Calculations

Syhentesis steps of the methodology is given Figure 1 below.

STEP 1
Performing finite element analysis for detailed physical system by 3D solid element and obtain deflection results

↓

STEP 2
Mesh the outer geometry of the physical system by 2D shell element. Construct 1D elements on centerline and tie them with 2D shell elements by rigid beams.

↓

STEP 3
Start structural optimization by same B.C with step 1. Select beam cross section properties as design region. Define deflection results of Step 1 as objective function.

Figure 1. Syhentesis steps of the methodology

The physical model can be evaluated as fixed-free beam problem. The degree of freedom of the root of the kit is constraint in all directions. The unit force is applied to the kit at the free end in both y and z direction. The schematic view is given in Figure 2 below.

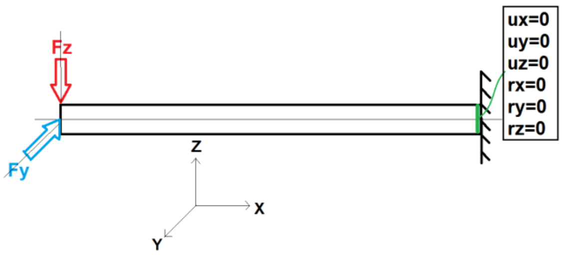

Figure 2. Boundary Conditions of the system

The number of entities of the detailed finite element analysis is given Table 1. The schematic view of the detailed finite element model is given in Figure 2.

Table 1. Number of entities of detailed finite element model

# of Nodes	# of 3D Elm's	# of MPC's
635789	365114	90

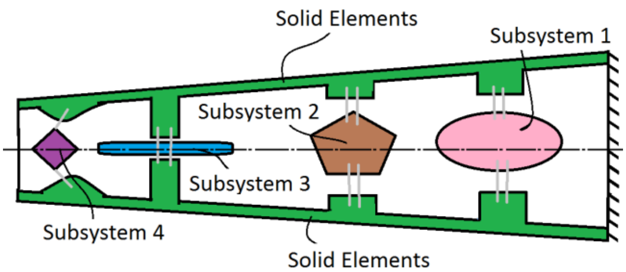

Figure 3. Schematic view of the detailed finite element model

Subsystems are modelled as dimensionless points elements which has same mass and inertial properties with real systems. The connection of the subsytem to the physical model is provided by rigid elements.

The number of entities of the stiffness equivalent finite element analysis is given Table 2. The schematic view of the detalied finite element model is given in Figure 4.

Table 2. Number of entities of stiffness equivalent finite element model

# of Nodes	# of Elm's		# of MPC's
	1D	2D	
23064	295	22860	7

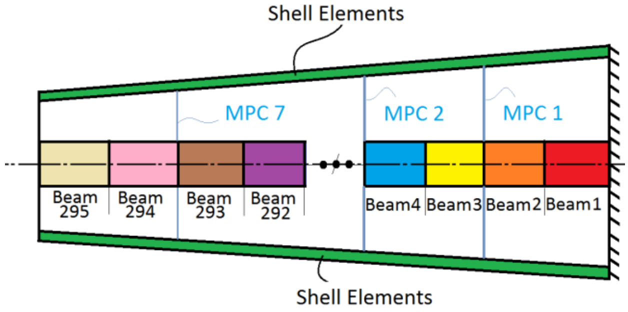

Figure 4. Schematic view of the stiffness equivalent finite element model

Material properties used in the analyses are defined in Table 3.

Table 3. Material properties of the structures used in the analyses

Material	Elastic Modulus (GPa)	Poisson Ratios	Density(kg/m^3)
Structural Steel	200	0.3	7800

In structural optimization, The design region is selected as closed rectangular cross section parameters of beam elements as shown in Figure 5.

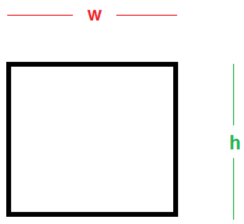

Figure 5. Design region of the beam used in structural optimization

The physical model is divided into 8 parts to catch the deflection pattern of the 3D detailed finite element analysis results. Here 9 different beam cross section parameters are optimized. Since two parameters as w and h are exist in a part, totally eigthteen design parameters are studied.

The design region bounds of the beam regions are given in Table 4.

Table 4. Optimization parameters of the beams

Parts	Initial Value	Lower Bound	Upper Bound
Beams (w,h)	(A_w), [A_h]	(%10A_w), [%10A_h]	(%1000A_w), [%1000A_h]

5 Results

There are four track points in the both 3d detailed finite element analysis and basic 1d-2d equivalent finite element analysis. These points are given in Figure 6.

According the results percentage errors of these points are given Table 5.

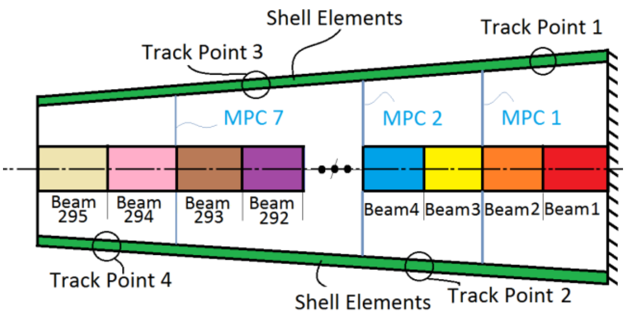

Figure 6. Schematic view of the stiffness equivalent finite element model

Table 5. Track point deflection errors of the stiffness equivalent model wrt detailed model

Track Points	Percentage Deviation of 1d-2d Equivalent Model Results wrt 3d Model	
	y axis	z axis
Point 1	%2.9	%0.12
Point 2	%1.01	%3.09
Point 3	%4.5	%3.2
Point 4	%0.9	%1.5

The cross section of the beams in y-z axes are shown in Figure 7 after structural optimization. It can be noted that, beam dimensions are changed differently. Figure 7 and Figure 8 present beam length results in y axis and z axis respectively. The results are different due to having dissimilar stiffness characteristics in each axis.

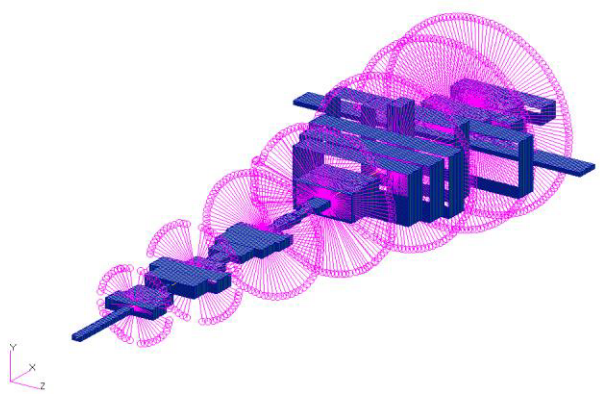

Figure 7. Beam cross sections after optimization

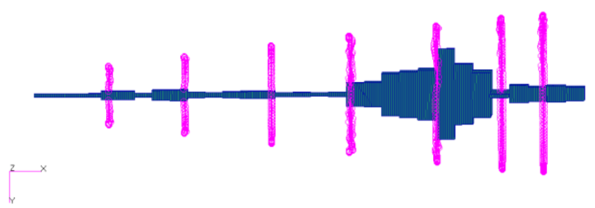

Figure 8. Beam dimensions in "y axis" after optimization

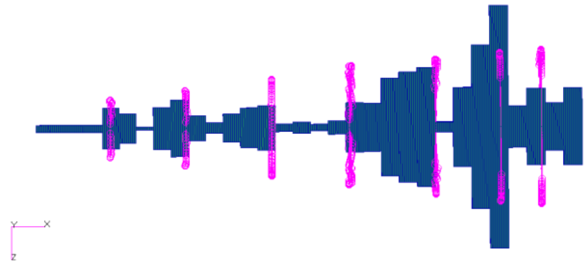

Figure 9. Beam dimensions in "z axis" after optimization

EFEM is also computaionally cheaper than DFEM. Total analysis times are compared and it is found that solution of EFEM takes 1 minutes. Moreover, solution of the DFEM takes about 98 minutes.

6 Conclusion

Equivalent stiffness finite element model of the subsystem which is constructed by 1d beam elements and 2d shell elements is investigated in this study. Firstly, linear static finite element analysis of the subsytem is performed with all detail parts. Deflection results of the several points in both y and z directions are noted. Then, equivalent finite element model is prepared by beam and shell elements. Beam elements are placed in the center of the sections along the part. Beam elements are connected in several regions to the shell elements by rigid beams.

Structural optimization is used in the second stage of the technique. Deflection results of the DFEM is defined as objective function both on y and z directions. Cross section dimensions of the beam is selected as design region. Lastly optimization algorithm determines the beam cross section dimensions.

As it can be seen from the Table 5, percentage error of the EFEM technique is so small with respect to DFEM. It can be said that EFEM of the subsytem can be used in the linear static analysis of the system instead of DFEM Thus, no detailed information about electronics, mechanics, hydraulics etc. of subsytem is delivered to the system design authority.

EFEM is also computaionally efficient. As it can be seen from Table 1 and Table 2, finite element quantity of the EFEM is lower than DFEM. Thus, computationally time solution of EFEM input file is approximately %1 of the DFEM. EFEM can be used for deflection analysis for several loading cases efficiently.

7 Future Work

Numerical studies mentioned above will be checked with the experimental setups using load cells, strain gages and laser displacement sensors to measure deflection values.

References

1. Vision 2023 Project, TUBITAK Conference Report, Defence and Aerospace Industry Report-Additional Part 3, (2003)

2. Y. Wang, X. Ouyang, H. Yin, X. Yu, Structural-Optimization Strategy for Composite Wing Based on Equivalent Finite Element Model, Nanjing University of Aeronautics and Astronautics, 210016 Nanjing, People's Republic of China, (2014)
3. J. Gryzagoridis, G. Oliver, D. Findeis, On the Equivalent Flexural Rigidity of Sandwich Composite Panels Cape Peninsula University of Technology, South Africa
4. A. Moroncini, Dr. L. Cremers, Dr. M. Kroiss, NVH Structural Optimization using Beams and Shells FE Concepts Models in the early Car Development Phase at BMW, Isma (2010)
5. F. Wu, X. Xu2, W. Jun, J. Fan, Static Stiffness Optimization of Large Ram Based on Equivalent Elastic Modulus, Key Engineering Materials Vol. **621** (2014)
6. D. J. Malcolm, D. L. Laird, Modelling of Blades as Equivalent Beams for Aeroelastic Analysis, AIAA Reno Jan, **#0870**, (2003)
7. T. R. Baumann, Formulation of Equivalent Uniform Beam Elements from Structual Stiffness Matrices, AIAA-92**-#2418**-CP(1992)
8. J. Sun, Y. Jia Y. L.Mo, Evaluation of Elastic Properties and Lateral Stiffness of Multi-Ribbed Walls Based on Equivalent Elastic Model., Elsevier Engineering Structures, **#72** 92-101, (2014)
9. MSC Software, Linear Static Analysis User's Guide, MD/MSC Nastran (2010)
10. GENESIS Design Manuel, Volume 2, Vanderplaats Research & Development, Inc., Colarado-USA(2012)