Mechanics Transport Communications
Academic journal

ISSN 1312-3823
issue 3, 2011
article № 0574
http://www.mtc-aj.com

SUPPORTING STRUCTURE MODELING USING TECHNOLOGY OF SUBSTRUCTURE

Miomir Jovanović, Predrag Milić
miomir@masfak.ni.ac.rs, pmilic@masfak.ni.ac.rs

Full professor, assistant, Mechanical faculty University of Nis
SERBIA

Abstract: *The paper shows how to overcome the limitations of automatic model generation discrete preprocessors (in FEA) in large load-bearing structures such as stands of railway vehicles. Also in this paper can be seen how the mapping techniques can develop a mesh of finite elements which satisfy its industrial needs. This is primarily related to the determination of complex stress state of the corner seams and butt welded girders of railway vehicles. Results and graphical techniques are interpreted in the images and the practices and procedures for verification of responsible structure of the railway.*
Key words: *Locomotives stand, substructure, FEA, megamodels*

INTRODUCTION

In the practical application of the finite element method (FEM), we meet with requirements for solving tasks that accurately describe the topological objects which we design. For railway vehicles it is necessary to accurately describe all elements of the stand because its geometry is the basis for stress and strain accumulation. The orderer usualy does not accept the analysis of most approximations of geometry. On the other hand, in the static and dynamic analysis, large assemblies of railway vehicles cannot be partialy (independently) observed. This creates a classic problem of FEM analysis - a very large discrete model. This model, which often reaches millions of equations, is known as megamodel. Megamodels usually bring with them the problems of good (pure numerical) defining connections between geometry and finite elements. The problem usually occurs in the module for the spatial partition of the continuum in a regular volumes - finite elements. The problems are conditioned by demands of generating mesh of large-scale and complexity. How to solve such tasks as the stand of locomotive?

CONCEPT OF SUBSTRUCTURE

Generation of large geometric models can be made simplifying the automatic generation of orders into smaller sub-models and their respective summing. In fact, the entire continuum of compact 3D geometric model should be divided into multiple geometric entities and on each one a discrete model, should be generated.

Division of geometric entities will depend on the program for the mesh development. Separation of geometry to 3-4 parts significantly reduces the complexity required by an automatic generator. In this way, discrete structure of mesh at the 3D geometric parts are partially generated. In order to combine these substructures into single model, it is necessary to define the conditions of development of the same mesh to their mutual contact surfaces. Therefore, it is necessary to generate the same finite element mesh on connecting surfaces of substructures. This is achieved by adjusting the control parameters of the mesh (size, bias, angle, element between).

Substructure analysis [1] uses matrix reduction techniques to reduce the matrix system on a small number of degrees of freedom. Substructure (super element) can be used in any type of analysis (with some restrictions and with coupled problems). It simply represents reduced set of elements which behaves as a single element. This element (super element) can be used for the desired analysis or to form a new super element. After analysis we use detailed reconstruction of superelement solution (eg. displacements and stresses) in the so-called expansion pass.

Solving the task starts from the basic static equation as follows:

(1) $$[K]\{u\} = \{F\}$$

In equation (1), [K] is the stiffness matrix, {u} is the global movement and {F} is load vector. The equation can be divided into two groups, *master*-main (kept) degrees of freedom (index indicated „m“) and *slave*- auxillary degrees of freedom (index indicated „s“):

(2) $$\begin{bmatrix}[K_{mm}] & [K_{ms}]\\ [K_{sm}] & [K_{ss}]\end{bmatrix}\begin{Bmatrix}\{u_m\}\\ \{u_s\}\end{Bmatrix} = \begin{Bmatrix}\{F_m\}\\ \{F_s\}\end{Bmatrix}$$

arranging:

(3) $$\begin{aligned}[K_{mm}]\{u_m\} + [K_{ms}]\{u_s\} &= \{F_m\}\\ [K_{sm}]\{u_m\} + [K_{ss}]\{u_s\} &= \{F_s\}\end{aligned}$$

The main degrees of freedom must contain degrees of freedom, for all nodes, on the surfaces in contact with other parts of the structure.

(4) $$\{u_s\} = [K_{ss}]^{-1}\{F_s\} - [K_{ss}]^{-1}[K_{sm}]\{u_m\}$$

separating

(5) $$\left[[K_{mm}] - [K_{ms}][K_{ss}]^{-1}[K_{sm}]\right]\{u_m\} = \{F_m\} - [K_{ms}][K_{ss}]^{-1}\{F_s\}$$

(6) $$[\hat{K}]\{\hat{u}\} = \{\hat{F}\}$$

where members of the substructure equations are:

(7) $$\begin{aligned}[\hat{K}] &= \left[[K_{mm}] - [K_{ms}][K_{ss}]^{-1}[K_{sm}]\right]\\ \{\hat{u}\} &= \{u_m\}\\ \{\hat{F}\} &= \{F_m\} - [K_{ms}][K_{ss}]^{-1}\{F_s\}\end{aligned}$$

The last step is to eliminate the multiplicity of nodes and form a complex whole structure of the substructure. This is achieved by:

1. Choosing a single coordinate system.,
2. Exporting CAD parts in IGS format.,
3. Importing IGS solids in the mesh generator,
4. Mapping the contact surfaces of solids,
5. Partial generation of mesh and control,
6. The elimination of multiple nodes.

Further large-scale solution depends exclusivly specially of solver for solving algebraic system of equations.

TECHNICAL DESCRIPTION OF THE PROJECT

Locomotive Factory a.d. MIN Nis in cooperation with the Department of Transport techniques and Logistics of Faculty of Mechanical Engineering Niš, realized the project [2] to improve the quality of railway vehicles DHD 200 DK classified as diesel hydraulic trolley power 209kW, capacity of Q=8t, gross weight 28t, equipped crane 6.1/2.1 t/m hydraulic equipment for tipping load.

The project treats the complex structure of vehicles stand from the point of strength in order to improve the stress-strain state of stand. The project has impruved construction of the stand, of correcting initial design solution, by changing the position and geometry of members,and by adding and subtracting weight. The project is a set of static CAD / FEA structural analysis, which examines continuum tension of the stands at 12 nominal action and 5 alternative combinations of loads [3].

With the group of successive changes in basic technical solution (given by the project office Locomotive a.d..) improvements were introduced by bringing the stress and strain in allowable limits. Obtained solution satisfied the technical conditions no. 12 and JŽ V2.005. Structural analysis was performed by finite element method, based on the linear theory of deformation using SSAP 4.0.

Development of discrete model was performed with pre/post processor FEMAP [6]. Geometric model was performed with software SolidWorks 2005. Allowable stresses are determined based on SRPS JUS U.E7.145-150 i EC3 standards.

CAD-FEA MODEL DEVELOPMENT

Stand vehicle DHD 200 DK is made from hot rolled steel Fe 430 i Fe 510 (EN 10025). Supporting construction of stand have form of solid spatial frame with parallelepiped geometry, Figure 1. It consists of two parallel the side lattice with the top band of the UPN 240, the lateral longitudinal beams UPN 240, , front and back of the front plate, transverse open profiles of UPN 100/200, frame and plate for mounting a crane. Lattice structure is unusual, with low cross section, without nodal plates, with reinforcements in place of joints. Dimensions stand without bumpers are LxWxH = 8760 x 2800 x 1415 mm. All parts of the stand are connected by welding.

Direct generation of discrete model was not possible due to the size and complexity of the model [4]. Therefore the geometric model is divided into five individual parts with similar dimensions, Figure 1. In the standing position in the structure, substructures are exported as IGES geometric models for the exchange. These IGES geometric models are loaded into a single document of FEM modeler. Mapping and element size estimation was performed. Then on each of the five solids finite element mesh was generated. Tetrahedron was used with 12 degrees of freedom per element. Free mesh is developed. The main reason for mesh development is a high fidelity of discreet and geometric model. This means that not only the main parts of a continuum were modeled, but also holes, roundings, welds, intermediate geometry of hot rolled profiles, ribs, profile fittings, surface-supports. For the analysis of complex stress state, comparative Von Mises's stress and the maximum tangential stress is taken. Selection of the comparative stress is performed by elastic behavior, in which most of the work is spent on geometry deformation, the hypothesis Henky-Huber-Mises.

Selection of finite elements was made in the domain of solids. The basis for this is the implementation of middle thick and thick plates for beams and connecting elements because they are characterized by spatial stress state. Limitation of the model is conditioned by the size (mass) of the stand, and it is limited with the scope of numerical models. That is why this model is planned with TWO MILLION degrees of freedom which can be effectively solved with workstation based on a multiple-core. The size of the finite elements of the mesh is determined on other limiting criteria - the maximum solids geometry ratio ($D/d<5$) with an average element size of 5x4x4 mm, the thickness of the walls of the conditioning profile. Generated number of elements in the direction of the length of stand (L = 8760 mm) is determined by the ratio of length and the average size of one element and is about 2000. In this way rough finite element mesh topology is determined. Developed mesh includes all geometric details of construction and accurately describes the continuum to the level of welds, holes, fillets, ribs, butt and corner joints - the exact geometry of the structure. Figure 1 shows the geometric model of the stand and five geometric sub-models derived from the main stand model.

Sub-models are individually exported to FEM modeler where the individual generation of the finite element mesh is performed. Figure 2 shows the mapping phase of contact surfaces.

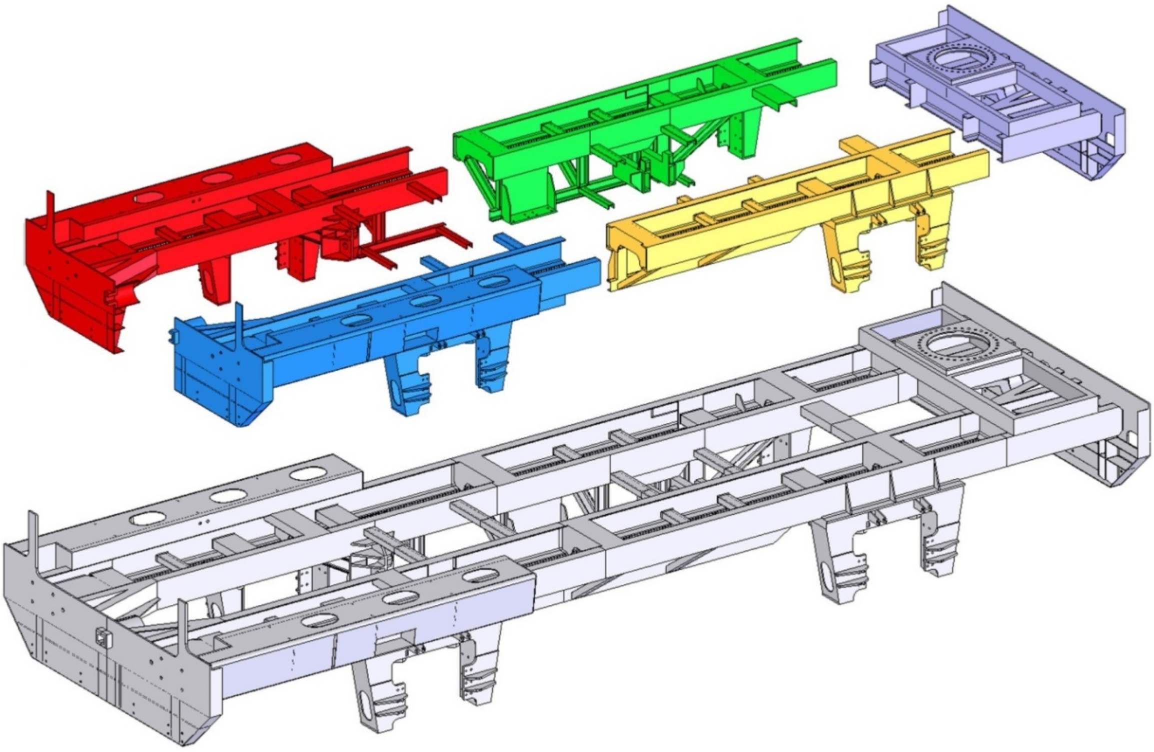

Fig. 1. *Developed geometric model of stand. DHD 200 DK* – whole with separate substructures.

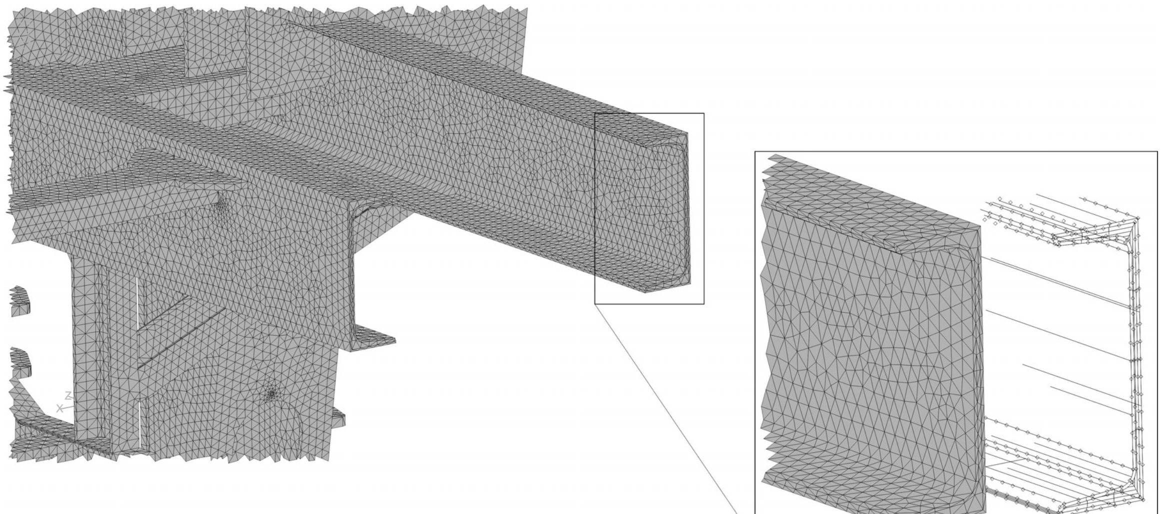

Fig. 2. *Detail of substructure mapping at the place of contact with the other substructure.* [4]

ANALYSIS OF SOLUTIONS

Solving such a large algebraic system (number of elements is 1,858,859, the number of equations 1859064th) is performed by mathematical decomposition methods.

The decomposition can be performed by two methods:

Direct (with the exact solution) and indirect (with approximate solution). Popular methods of implementation are the Gauss and Gauss-Jordan's method of the direct elimination and the LU decomposition method (where the reduction is performed on the lower or upper triangular matrix).

Indirect (approximate) methods are used in software for large algebraic systems with more than 100.000 of equation, and this is the case of this megamodel. The following Figure 3 shows the form of solutions in graphical form with continuous shadow image display form of central substructure complex stress.

Fig. 3. *Structural analysis of smash to another vehicle. Symbolic load: $1.3Fv + F_H$ (5g)*
View of the bottom center of the stand: Maximum Solid Von Mises comparative stress 388.762.432, N/m^2 .

CONCLUSION

Illustrated example of railway vehicle stand is solved with complete structural decomposition into five substructures, each partial treated. This exceedes the inability to generate a discrete model directly from the whole. Significance of represented CAD/FEA technology is in numerical setting of model in terms of flexion and torsion stiffness, static and dynamic properties of the structure.

Acknowledgment

This paper is financially supported by the Ministry of Education and Science of Republic of Serbia, Project Nr. 35049. This support is gratefully acknowledged.

REFERENCES

[1] ANSYS V11. - Theory Reference, Southpointe 275 Technology Drive Canonsburg, PA 15317, 2007.
[2] Jovanović M., Kozić P., Milić P.: STRUKTURNA ANALIZA POSTOLJA ŽELEZNIČKOG VOZILA DHD 200 DK - Lokomotiva a.d. MIN Niš, Razvojni projekat 612-22-168-2/08, Mašinski fakultet Niš, 2009.
[3] Jovanović M., Milić P.: DESIGN ANALYSIS HEADING TO BETTER DESIGN, ADEKO, FTN Novi Sad, Jurnal - Machine design, 2009,
[4] MSC NASTRAN 2004, Interactive Systems, Inc, Part No 440.401 Supergen, Pittsburgh, 2004.
[5] Saouma V.: MATRIX STRUCTURAL ANALYSIS WITH AN INTRODUCTION TO FEM, CVEN 4525/5525, University of Colorado, Boulder, 1999.
[6] Vladić J, Jovanović M, AUTOMATIZOVANO PROJEKTOVANJE, udžbenik, TU Novi Sad, 2001,

МОДЕЛИРАНЕ НА ПОДКРЕПЯЩИ КОНСТРУКЦИИ ЧРЕЗ ИЗПОЛЗВАНЕ НА ТЕХНОЛОГИЯТА НА СКЕЛЕТА

Miomir Jovanović, Predrag Milić

full professor; assitant, Mechanical faculty University of Nis
SERBIA

Ключови думи: *локомотивен стенд, конструкция, FEA, мегамодели*

Резюме: *В статията се показва как се преодоляват ограниченията на дискретните процесори (FEA)за автоматично моделиране на големи носещи конструкции, като например стендове за железопътни возила. Също така в този доклад, може да се види как с техниките за картотекиране могат се разработи на крайни елементи, които удовлетворяват промишлени нужди. Това е свързано основно с определяне на сложни напрегнати състояния на ъгловите шевове и челно заварените греди на железопътните возила. Резултатите и графичните техники се тълкуват в образи, практиките и процедури за проверка на важните железопътни конструкции.*