ANALELE UNIVERSITĂȚII
"EFTIMIE MURGU" REŞIȚA
ANUL XX, NR. 1, 2013, ISSN 1453 - 7397

Cornel Iacob-Mare

Study of Geometric Model Car Body through the Finite Element Mesh

The paper proposes an analysis of a freight wagon body structure with high walls that has platform to maneuver through study recommended medium for geometric model finite element mesh type Shell63.

***Keywords**: finite element, meshing, wagon body,*

1. Introduction

We chose to study a high-walled open wagons which have certain features in the chassis. The chassis accepts only a symmetrically - longitudinal vertical - as brake platform is the full width headstocks and leaning on it and a "false beam". Variants of this car, with or without keyboard download, or different sections of the main structural elements were built over time in rolling stock manufacturing factories in Romania.

Wagons discovered high walls are widely used on railways which transport various types of goods. Since goods can be transported in bulk manipulation is difficult for coal, logs, scrap etc.., Which leads to plastic deformation resistance structure. In Romania, this type of car is probably the most common leading over time to the need to design and execution of several series of such freight cars, some of these projects with design flaws, such as the model the car chosen for the study.

At present, rail operators require life extension of freight wagons economic reasons. Thus, the life of a vehicle of this type can be increased from 30 to 48. Life extension can be justified by introducing wagon in the modernization programs involving replacement of structural elements and even change the original project. In addition, the modernization projects of some cars can be test bed for new vehicles.

For example, a new braking system can be mounted on an old wagon that by providing conclusive results as to the reliability testing thereof. Only then can move to the next stage of building a body us. Another solution is to test new technologies and joint manufacturing constituents of a wagon that can be tested by fitting the existing structure. Upgrading projects can be a stage for a project run entirely new.

2. The purpose of the paper

The calculation of the structure to be subjected to finite element analysis, in the general case is made up of lines (which are the axes of the bars of the structure), the flat and curved surfaces (which are the surfaces of the plates middle parts of the structure) and volume (which bodies are massive structure). At this stage of development, the model is continuous, with an infinite number of points as the structure of time.

Meshing is required fundamental approach and finite element method consists in passing from continuous structure (with an infinite number of points) on a discrete model with a finite number of points (nodes). This operation is covering mesh network model and is justified by the fact that in practical terms is sufficient information on the structure. Since the method usually defines the unknowns in the model nodes, also in those points calculated values. For these reasons, the mesh should be in a sufficient number so as to define all areas of interest, to approximate the structure of the boundary conditions / load [10], [13].

Points at the intersection of the mesh are called nodes. The nodes are defined primary unknowns, whose values are the results of finite element analysis. If nodes are the unknowns associated movements, the model set is called "displacement model" where the unknowns associated with nodes are efforts defined pattern is called "equilibrium model". Rarely used "mixed model" [13].

As a result of discretization, the model is divided into a certain number of parts or finished items. Finite elements are interlinked by common nodes that are vertices of the geometric elements (triangles, quadrilaterals) components (of course I did not forget the elements that have nodes on the sides). In practice, a finite element analysis is an independent component that interacts with the other elements only nodes. Following the division of the structure into finite elements, the study of the real structure is replaced by the combined study of all finite elements that make up the model [10].

2.1. Meshing the model

Since I opted for the pattern geometric areas (not volume), finite elements are used to mesh shell. Ansys library are available to the user several shell finite elements. First proposed finite element Shell63 element with four nodes (placed at the corners of the element) and the limit can have three nodes.

The literature does not provide clear indications on the average recommended shell finite elements. As a result, we have discrete geometric model

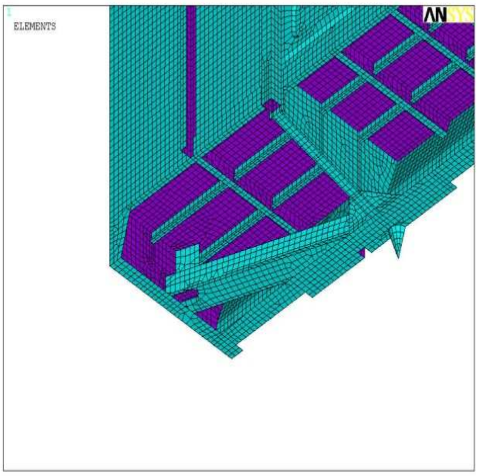
Shell63 finite elements with the average of the 50, 45, 40, 35, 30, 25 mm in order to see what size to achieve convergence of the finite element results.

The following table presents the variation of the number of nodes and elements required by the average size of the item.

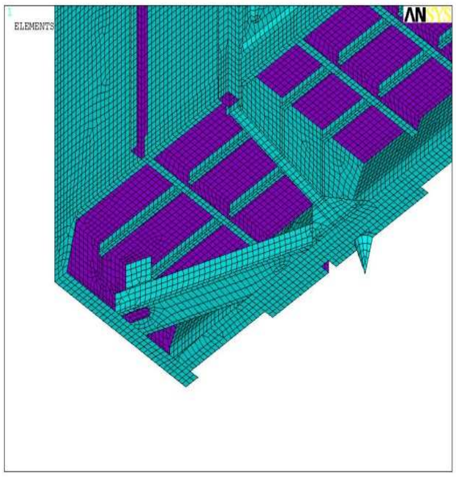
Table 1.

Average size required	Number of nodes	Number of elements
50	47565	48546
45	56768	57788
40	68612	69665
35	90160	91412
30	120820	122197
25	168668	170121

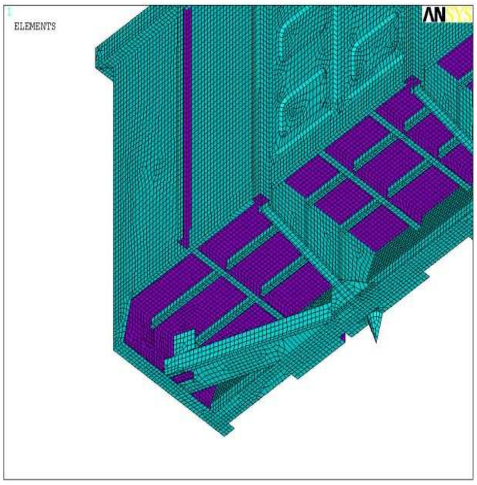
I fell element size less than 20 mm, as a size 25 mm sides already in some parts of the structural frame side elements average size is less than the thickness of the.



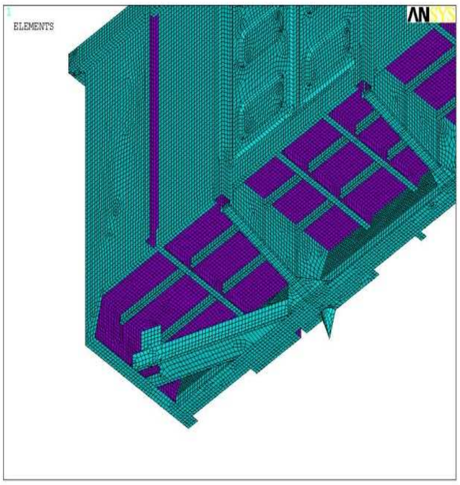
a) Average size 50 mm



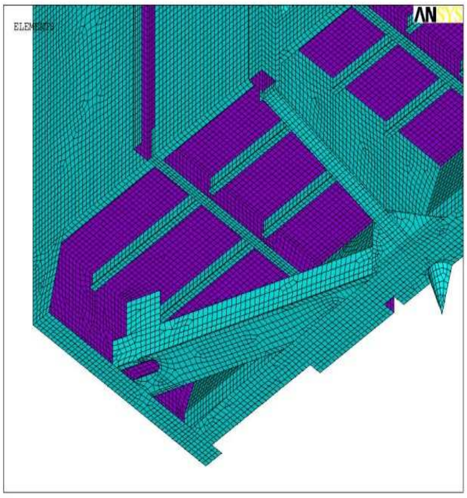
b) Average size 45 mm



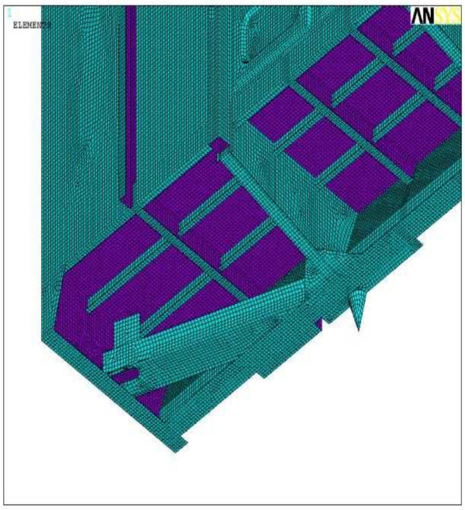
c) Average size 40 mm



d Average size 35 mm



e) Average size 30 mm



f) Average size 25 mm

Figure 1. Meshing the model

3. Conclusion

Finite element modeling of shell, provides the advantage of a smaller number of nodes (with direct implications runtime) than if the model were meshed with solid but has the disadvantage that their surfaces are generated median plane which takes us away from reality for laminated profiles (profiles U ☐) which is built wagon. In addition, in the case of profiles of different height or the same height but of different type (U ☐ for example), their intersection results in a simplification of the model in that area.

Since the size of the pattern elements of 35 mm has a small number of nodes and elements. Size 35 mm sides of the elements is confirmed in [14].

Comparing the pattern to the average size of the mesh bars 50 mm and 35 mm, there is a uniformity of elements for the second.

References

[1] Burada C., Buga M., Crăsneanu A., *Elemente şi structuri portante ale vehiculelor de cale ferată,* Editura Tehnică Bucureşti 1980.

[2] Copaci I., Mănescu T., Olaru S., Creanga F., *Rezistenţa la solicitări variabile care apar în exploatarea vehiculelor feroviare*, Editura Mirton, Timişoara, 2005.

[3] David. G. I., *Calculul si construcţia structurilor portante ale vehiculelor feroviare*, Institutul Politehnic, Timişoara, 1981.

[4] Gillich, G.R., *Dinamica Maşinilor. Modelarea sistemelor tehnice*, Editura AGIR, Bucureşti, 2003.

[5] Ianici S., *Ingineria sistemelor mecanice, vol 1*, editura „Eftimie Murgu", Reşiţa, 2001.

[6] Liu Y., *Lecture Notes: Introduction to the Finite Element Method*, versiune on-line (http://faculty.ksu.edu.sa/rizwanbutt/Documents/).

[7] Mănescu T. Ş., Copaci I., Olaru S., Creangă F.Ş, *Tensometria electrică rezistivă în cercetarea experimentală,* Editura Mirton, Timişoara, 2006.

[8] Mănescu T.Ş., Jiga G.G., Zaharia N.L., Bîtea C.V., *Noţiuni fundamentale de rezistenţa materialelor şi teoria elasticităţii,* Editura Eftimie Murgu, Reşiţa, 2010.

[9] Mănescu T.Ş., Jiga G.G., Zaharia N.L., Bîtea C.V., *Noţiuni fundamentale de rezistenţa materialelor,* Editura Eftimie Murgu, Reşiţa, 2008.

[10] Mănescu T.Ş., Nedelcu D., *Analiza structurală prin metoda elementului finit,* Editura Orizonturi Universitare, Timişoara, 2005.

[11] Mocanu D.R., Buga M., Brateş M., Calcule de rezistenţă cu specific feroviar, Editura Căilor Ferate, Bucureşti 1958.

[12] Nedelcu D., *Modelarea parametrică prin Autodesk Inventor*, Editura Orizonturi Universitare, Timişoara, 2004.

[13] Sorohan Ş., *Practica modelării cu elemente finite*, versiunea on-line (http://www.resist.pub.ro/Cursuri_master/PMEF/).
[14] Zaharia N.L., *Contribuţii privind optimizarea structurilor portante ale vagoanelor cisternă de cale ferată în vederea creşterii sarcinii pe osie şi micşorarea tarei vehiculelor*, Teză de doctorat, Universitatea „Eftimie Murgu", Reşiţa, 2010.
[15] *** Fişa UIC 577 *Wagon stresses* , 2005.
[16] *** NTF 53-001 *Aplicaţii feroviare. Cerinţe de dimensionare a structurilor portante a vehiculelor feroviare. Încercări ale structurilor portante şi ale elementelor componente*, 2005.
[17] *** Raportul ERRI B12/RP17 *Programme des essais à faire subir aux wagons à châssis et superstructure en acier (aptes à recevoir l'attelage automatique de choc et traction) et à leurs bogies à châssis en acier* (8ème édition), 1997.
[18] *** Raportul ERRI B12/RP60 *Prescriptions de réalisation et contraintes limites* (2 ème édition), 2001 .
[19] *** SR EN 12663-1 *Aplicaţii feroviare – Cerinţe de dimensionare a structurilor vehiculelor feroviare, Partea 1: Locomotive şi vagoane de pasageri (şi metodă alternativă pentru vagoane de marfă)*, 2010.
[20] *** SR EN 12663-2 *Aplicaţii feroviare – Cerinţe de dimensionare a structurilor vehiculelor feroviare, Partea 2: Vagoane de marfă*, 2010.
www.ansys.com

Address:

- Drd. Eng. Cornel Iacob-Mare, "Eftimie Murgu" University of Reşiţa, Piaţa Traian Vuia, nr. 1-4, 320085, Reşiţa, c.iacob-mare@uem.ro