

DETERMINATION OF THE STRESS AND DEFORMATIONS THROUGH A FEM AND INVENTOR ANALYSIS OF A TRUSS BAR

Anisoara - Gabriela Cristea
"Dunarea de Jos" University of Galati,
Faculty of Naval Architecture, Galati, 47
Domneasca Street, 800008, Romania,
E-mail:anisoara.cristea@ugal.ro

George Ciprian Iatan
"Dunarea de Jos" University of Galati,
Faculty of Naval Architecture, Galati, 47
Domneasca Street, 800008, Romania,
E-mail:ciprianiatan@yahoo.com

ABSTRACT

This papers aims mainly at presenting the effects of loads acting upon the structure of a truss bar and at pointing out the positioning modes of stress concentrators. Moreover, the papers highlights the way in which the factor of stress concentration is determined and how the results are established in order to determine the best software to analyse the beam for further possible optimization.

Keywords: finite element method, deformations, static analysis, stress concentration, normal stress.

1. INTRODUCTION

Femap is a simulation software that uses finite element analysis in order to generate the behaviour of structures in different environments or after several loadings. In this case, Femap's capabilities were used to highlight the stress and deformations inflicted by the three forces added on the truss.

Inventor is a computer-aided design application launched by Autodesk. It creates 3D models and can be used for simulations due its integrated calculation module. The user can select particular physical and mechanical specifications and see how a model will act, using finite element analysis, as we intend to show below.

The truss bar has a simple 3D geometry, generated both in FEM and Inventor Software Programs. They are used to perform finite element analysis and to determine the stress and deformations induced in the bar by several loads. For maximal precision in the results, some aspects must be taken into consideration.

- Meshing: it must be done accurately enough, so as the stress effect in the model to be showed eloquently;
- The type of element: for truss bar it should be rod;
- The dimension of the element: in this case, the truss bar is fairly simple, so the elements can be equal, although generally the stress concentrators or the "hot spots" are evaluated with a higher density of elements.

The static analysis requires to perform specific steps, such as material selection and defining its properties, drawing the nodes and the truss, adding loads and constraints.

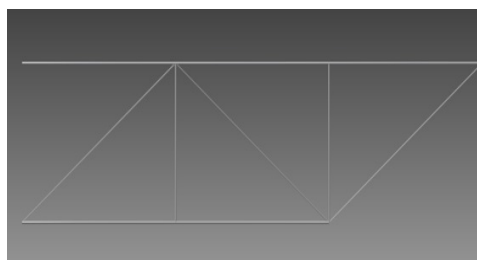


Fig. 1 3D Geometry of the truss bar

2. DESCRIPTION OF THE MODEL

The truss bar taken into consideration can be regarded as a structural element of a crane or like a reinforcing element for the roof of a building.

2.1 Main dimensions and characteristics

As entry data for the FEM and Inventor analysis, the following specifications were used:

Table 1 Physical values

Material	Iron, Gray Cast ASTM A48 Grade 20
Density	7.872 g/cm ³
Mass	54.1874 kg
Area	1285310 mm ²
Volume	6883570 mm ³
Centre of Gravity	x=0.000112691 mm y=-643.405 mm z=-2715.25 mm

Name	Iron, Gray Cast ASTM A48 Grade 20	
General	Mass Density	7.872 g/cm ³
	Yield Strength	151.684 MPa
	Ultimate Tensile Strength	179.263 MPa
Stress	Young's Modulus	20 GPa
	Poisson's Ratio	0.29
	Shear Modulus	7.75194 GPa

The analysis was made using the following units:

- Length: millimetres (mm);
- Pressure: Megapascals (N/mm²);
- Stress: N/mm² (MPa).

3. MAIN STEPS FOR OBTAINING RESULTS

The operations are being developed in FEM and Inventor in the same time. In order to follow simplicity, the 3D geometrical construction will be presented only in FEM, aiming at adding directly the Inventor model.

3.1 Defining material properties

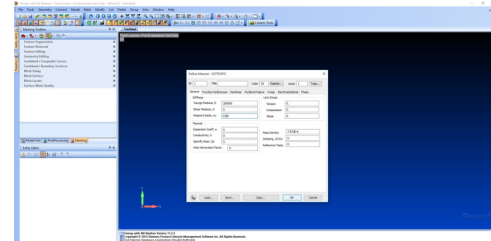


Fig. 2 Material properties selection

This is the step where we define the material specifications, as we need. Also, a predefined material can be easily chosen.

3.2 Defining the elements

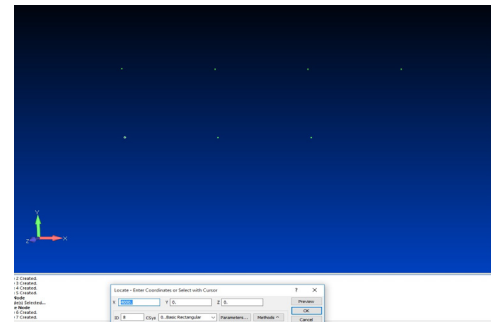


Fig. 3 Positioning the nodes

Here we define the three types of elements which take part in the model. The difference between them is the diameter of the rod, taken at 50mm, 40mm and 30 mm.

3.3 Drawing the nodes

In FEM we use a triorthogonal system in order to define the nodes. The first dot is defined, and the others are placed depending on it.

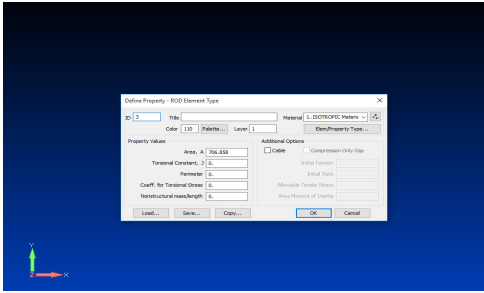


Fig. 4 Introducing the type of elements

3.4 Drawing the truss

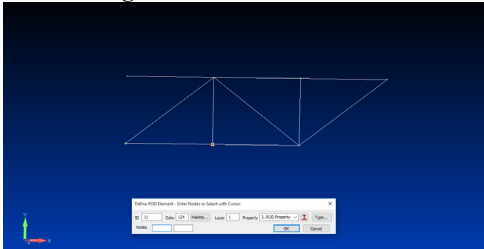


Fig. 5 Positioning the truss elements

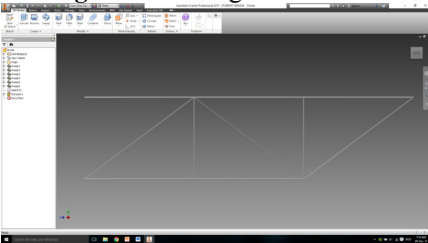


Fig. 6 Truss bar in Inventor

Step by step, we add the elements between the nodes, according to our model. The horizontal bars have the diameter of 50mm, the vertical ones have 40 mm in diameter, and the diagonal bars - 30mm. In the same step, the Inventor model is also added.

3.5. Adding loads over the truss

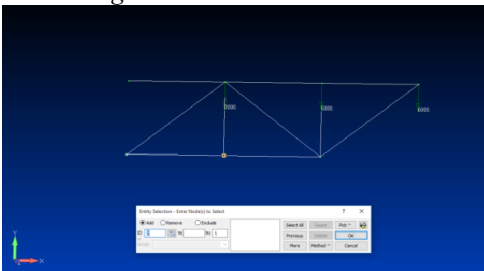


Fig.7 FEM loaded truss

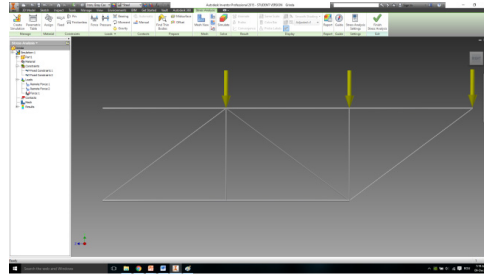


Fig. 8 Inventor loaded truss

The truss bar is loaded with three forces in the top three nodes. From right to left, the forces are equal to 6000 N, 5000 N, 3000 N.

3.6 Constraints

The truss is fixed on both left margins, so the model has zero degrees of freedom relative to itself.

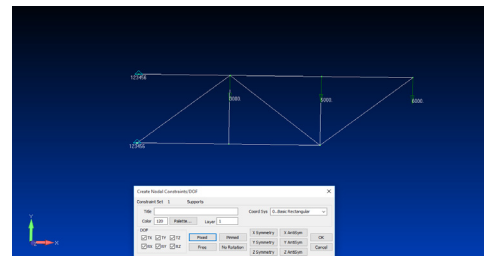


Fig. 9 FEM constrained truss

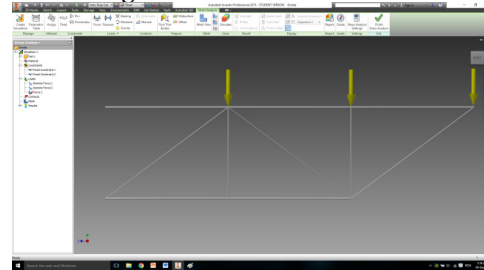


Fig. 10 Inventor constrained truss

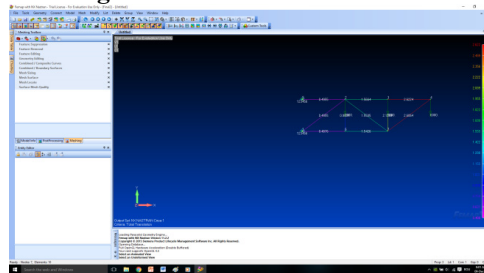


Fig. 11 FEM total translation

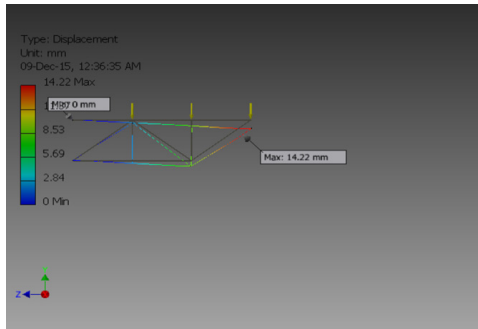


Fig.12 Inventor total translation

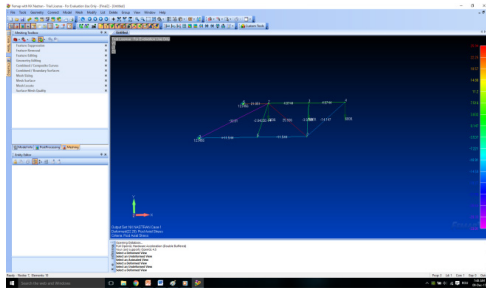


Fig. 13 FEM axial stress

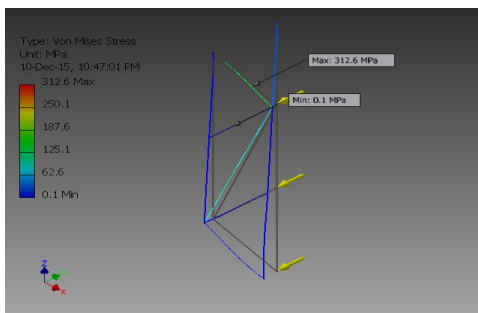


Fig. 14 Inventor Von Mises Stress

3.7 Simulation and results

In both programs, the static analysis will start by using the "Run" command. The processing time depends on the precision selected.

The results obtained after the analysis are now available in both programs. Most important of them are shown below:

Maximal translation	FEM	0.48 - 2.62 mm
	Inventor	0 - 14.22 mm
Maximal stress	FEM	33MPa at stretch and 26MPa at compression

Inventor	227MPa at stretch and 313MPa at compression
----------	---

4. CONCLUSIONS

The finite element analysis remains one of the most useful tools for engineers to approximate accurately the stress and the deformations induced by loads in any type of structure. By performing parallel analyses in different programs, the chance of obtaining accurate results is higher.

The two programs allow users to access the results in different ways, but Inventor offers a full report with images and centralized data. Furthermore, the interface in Inventor is very "user-friendly", fact that makes this software much more accessible for beginners than FEM.

Regarding the accuracy of data, in order to receive better information, in the "hot spot" of the model, there must be used a higher density of elements. For this structure, the most affected part is the top right corner of the truss, so it is recommended to increase the number of the elements there.

REFERENCES

- [1]. **Domnisoru L., Gavan E., Popovici O.,** "The ships' structures analysis by the finite element method", Didactic and Pedagogic Publishing House, Bucharest, 2005.
- [2]. **Domnisoru L.,** "The ships' structures analysis by the finite element method. Numerical applications", The Foundation of "Dunărea de Jos" University Publishing House, Galati, 2009.
- [3]. **Hadăr A., Constantinescu I.N., Gheorghiu H., Coteș C.E.,** "Modelling and models for calculations in mechanical engineering", PRINTECH Publishing House, Bucharest, 2007.
- [4]. **x x x - User's Guide Inventor 2015**
- [5]. **x x x - User's Guide Femap v9.3.1**
- [6]. **x x x - NX NASTRAN 5.0 User's Guide,** UGS Corporation / Siemens PLM Software Inc., 2007

Paper received on December 18th, 2015