ANALELE UNIVERSITĂȚII
"EFTIMIE MURGU" REȘIȚA
ANUL XVII, NR. 2, 2010, ISSN 1453 - 7397

Nedelcu Dorian, Ianici Draghita, Nedeloni Marian, Daia Daniel

Numerical Behavior Reproduction of a Truss Structure and Beam, Subjected to Concentrated Load

The paper deals with a numerical reproduction of a real experiment consisting of the same restraints and concentrated load applied to a truss structure and beam; the numerical reproduction was performed with SolidWorks software, where both geometrical reconstruction and finite element analysis were made. The goals of the analysis was to compare the numerical with experimental results and to evaluate the best structure using the structural response criteria.

Keywords*: finite element, truss, beam, SolidWorks*

1. Introduction

The real experiment was performed for National Geographics Engineering Connections documentary series, in Hong Kong International Airport episode [1], for the following conditions:

- a continuous beam was fixed at the extremities and loaded with a concentrated mass 3500 kg (figure 1); the load is applied static, when the crane releases the equivalent mass of multiple concrete boxes; the beam cannot support this load and the beam failure is presented in figure 2;
- a truss structure was fixed at the extremities and loaded with the following values of the concentrated mass: 3500 kg (figure 3) and 4500 kg (figure 4); the truss structure resisted these loads, which demonstrates a better resistance behavior of the truss comparing to the beam.

This is the reason that recommends the truss structure to be used for bridge, for airplane building (figure 5) or for Hong Kong International Airport roof (figure 6).

The paper aim to reproduce the experiment using numerical simulation by finite element method. The SolidWorks software is used as tool for geometrical reconstruction of the beam and truss structure and also for the numerical analysis.

Figure 1. The beam subjected to a concentrated mass of 3500 kg

Figure 2. The beam failure for a concentrated mass of 3500 kg

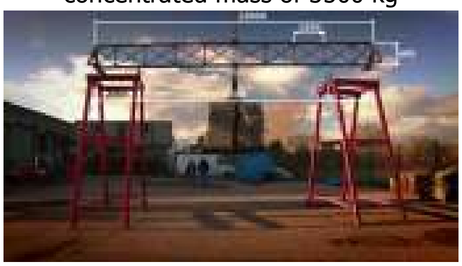

Figure 3. The truss structure subjected to a concentrated mass of 3500 kg

Figure 4. The truss structure subjected to a concentrated mass of 4500 kg

Figure 5. The truss structure used for airplane building

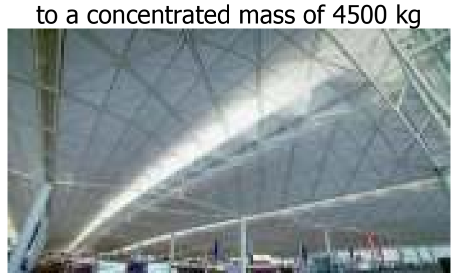

Figure 6. The truss structure used for Hong Kong International Airport roof

2. The beam analysis

For the analysis, four types of beam geometries were analysed, with the main characteristics presented in table 1 and figure 9 [2].

Table 1.

Section profile	h	b	t	d=R	r	W_z	Linear mass	Total mass
	mm	mm	mm	mm	mm	cm^3	Kg/m	kg
I12	120	58	7.52	5.1	3.1	54.7	11.2	117.6
I14	140	66	8.40	5.7	3.4	81.9	14.4	151.2
I16	160	74	9.28	6.3	3.8	117	17.9	187.9
I18	180	82	10.16	6.9	4.1	161	21.9	229.9

The beam was fixed on the two extremities 1 and 2 and loaded with a normal force of 35000 N, applied to the top central point 3 of the beam (figure 1), symmetrically disposed at L=5250 mm distance from point 1 and 2. Because of the symmetry, the V_1 and V_2 reactions are equal to the numerical value of 17500 N. The moment in central point 3 can be calculated by formula:

$$M_3 = V_1 \cdot L = 17500 \cdot 5{,}25 = 91875\ Nm \tag{1}$$

and the bending stress by formula:

$$\sigma_z = \frac{M_3}{W_z} \tag{2}$$

The beam geometry can be reproduced in SolidWorks in two ways:

- „**I**" profile sketching, according to the dimensions from figure 9 and the values found in table no. 1; creates a solid feature by **Boss-Extrude** command in one direction for 5250 mm length; multiplies the previous solid two times at 5250 distance by **Linear Pattern** command;
- sketching of the two collinear lines each measuring a length of 5250; creates two structural members using the **Structural Member** command with „**I**" profile, according to the dimensions from figure 9 and values from table no. 1; the command offer the possibility to select the standard, the type and the size of the profile and to set some option regarding corner treatment, the gap, the alignment and redefine insertion point; there are a small number of pre-defined profiles in SolidWorks; there are two possibilities, to use the right profile:
  - a new profile must be created and added to the SolidWorks library using the following steps: in a new file will be the sketched the new profile, which will be saved as library feature part („sldlfp" file) in „*C:|Program Files|SolidWorks Corp|SolidWorks|data| weldment profiles|iso|sb beam*" folder; the given name of the profile appears in the **Size** list in the **Structural Member Property Manager** when the **Structural Member** command will be launched; the origin of the sketch becomes the default insertion point, which can be later modified with **Settings → Locate Profile** option of the **Edit Feature** command applied on the created structural member; the profile creation must be applied once and then it can be used every time is needed;
  - the second option uses an existing profile from the SolidWorks library, which looks similar to the one we need, and modifies the profile to adapt or alter the dimensions according to our profile.

As a results of either two methods, two solid bodies are created. The next step is to activate the **Simulation** module in SolidWorks from the menu in the following steps: **Tools → Add-Ins → SolidWorks Simulation** and create a new Static study. If the two solid bodies were created using the **Structural Member** command they will be treated automatic as beam elements by the **Simulation** module, because the program automatically meshes structural members with beam elements. Beam elements can resist bending, shear, and torsional loads. If the two solid bodies were created using the **Boss-Extrude** & **Linear Pattern** commands, the **Simulation** module must be informed to treat them as beam elements, through the **Treat as Beam** option, selected from the right popup menu accessed on the two SolidBody geometry in study's folder.

The size of the problem and the resources required are dramatically reduced for beam elements compared to solid elements. For accurate results, the length of the beam should be 20 times larger than the longest dimension of its cross section. A joint is identified at the free end of a beam or at the intersection of two or more beams. For the two beams elements **Simulation** module will create three joints, corresponding to the points 1, 2 and 3 from figure 1. These joints can be verified and recalculated in the **Joint group** section in the study's folder. For beam elements, restraints and loads can be applied in joints.

For all the geometry, the material from the SolidWorks library must be selected; in our case the **Alloy Steel** was selected with the following properties: the Elastic Modulus $2.1\text{x}10^{11}$, Poisson's ratio 0.28, Yield strength 620.42 MPa, Tensile Strength 723.83 MPa.

The next step is to fix the geometry in joints 1 and 2, by imposing 0 displacement for two directions: normal to the beam top face (figure 9) and perpendicular on the beams length's. The force will be apllied normally to the beam top face in joints 3, with a value of 35000 N.

For I12...I18 profile, after the mesh is created and the study is runned, the results are obtained, according to table no. 2 and figure 10... 14.

SolidWorks is provided with a special tool for beam calculation, which can be activated from the menu in the following steps: **Tools → Add-Ins → SolidWorks Toolbox**; as a result, the **Toolbox** option is added to the main menu, from where the **Structural Steel** option launch the window with the same name, figure 7; the window offer the possibility to select the standard, the type and the size of the profile and to activate the **Beam Calculator** window, figure 8, where the beam deflection and stress can be calculated for the following input data: section modulus W_z, modulus of elasticity, beam length, load value and load type; also, the moment of inertia & section modulus axis and units can be selected. The load type can be selected from the following load cases: Fixed at one end, loaded at the other end, Fixed at one end, uniformly loaded, Supported at both ends, load in middle, Supported at both ends, uniformly loaded, Supported at both ends, unsymmetrical load, Supported at both ends, two symmetrical loads. The final results are presented in the **SolidWorks Toolbox Beam Calculator** column of the table no. 2.

The factor of safety FOS from table 2 is calculated by the following formula:

$$FOS = \frac{Yield\,strength}{\sigma_z} = \frac{620.42}{\sigma_z} \tag{3}$$

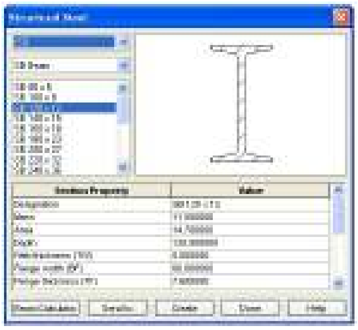

Figure 7. The **Structural Steel** window

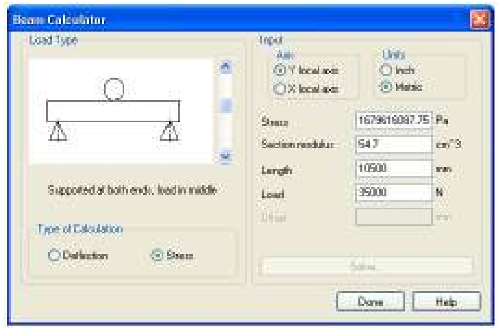

Figure 8. The **Beam Calculator** window

Table 2.

Section profile	Theoretical Formula (2)	SolidWorks Toolbox Beam Calculator	SolidWorks FEM analyse 35000 N Load Force		
	σ_z	σ_z	M_3	σ_z	FOS
	MPa	MPa	Nm	MPa	-
I12	1679.62	1679.62	91875	1696.61	0.37
I14	1121.79	1121.79	91875	1130.89	0.55
I16	785.26	785.26	91875	791.98	0.78
I18	570.65	570.65	91875	576.20	1.08

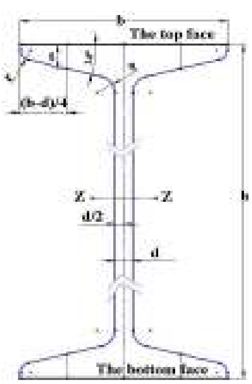

Figure 9. The main dimensions of the beam section

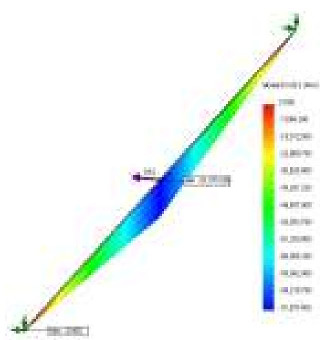

Figure 10. The moment M_3 distribution for „**I**" profiles $M_{3\ max}$= 91875 Nm

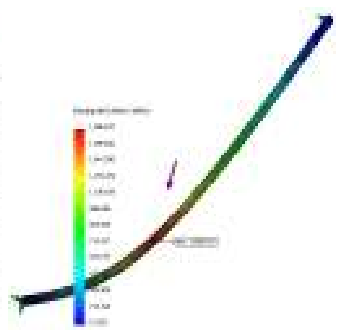

Figure 11. The bending stress σ_z distribution for I12 profile $\sigma_{z\ max}$= 1696.61 MPa

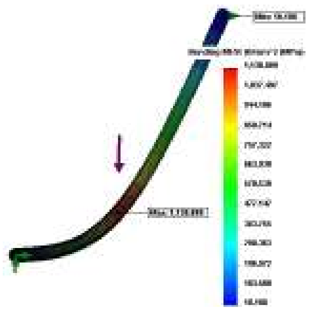

Figure 12. The bending stress σ_z distribution for I14 profile $\sigma_{z\ max}$= 1130.89 MPa

Figure 13. The bending stress σ_z distribution for I12 profile $\sigma_{z\ max}$= 791.98 MPa

Figure 14. The bending stress σ_z distribution for I12 profile $\sigma_{z\ max}$= 576.20 MPa

The numerical results confirm the beam failure presented in figure 2, for I12 to I16 profile; also, the FOS of the I18 profile is close to the 1 value, while the normal values are greather than 2.

The σ_z values resulted from **SolidWorks Beam Calculator** tool are identically with those resulted from theoretical formula (2).

3. The truss structure analysis

For the truss structure analysis, the main characteristics are presented in figure 15. The elements of the truss structure are:

- the top beam with a profile of 30 x 30 x 2 x R4 mm;
- the vertical beams with a profile of 30 x 30 x 2 x R4 mm;
- the diagonal beams with a profile of 20 x 3 mm;
- the bottom beam with a profile of 40 x 40 x 2 x R4 mm.

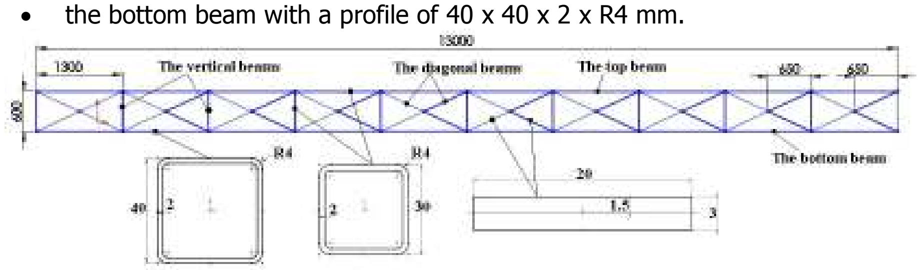

Figure 15. The main dimensions of the truss structure

The steps to reproduce a truss structure geometry in SolidWorks are:

- sketching of the truss structure, according to figure 15;
- creation of five structural members by **Structural Member** command, using an existing profile from the SolidWorks library, looking similar to the one we need;
- the modification of the profile to adapt to the dimensions of our profiles (figure 15).

The truss structure has a final mass of 76.34 kg, much smaller than any beam mass from table no. 1.

In SolidWorks a structural member can contain groups of sketch segments. A group is a collection of related segments in a structural member. Configure a group will affect all its segments without affecting other segments or groups in the structural member. It is possible to define a group in a single plane or in multiple planes. A group can contain one or more segments. A structural member can contain one or more groups. After the group is defined, it is possible to operate on it as a single unit. Types of groups are:

- contiguous - a continuous contour of segments joined end-to-end; it is possible to control how the segments join to each other; the end point of the group can optionally connect to its beginning point;
- parallel - a discontinuous collection of parallel segments; segments in the group cannot touch each other.

For all the geometry the same material (**Alloy Steel**) was selected from the SolidWorks library. The geometry is fixed in joints 1, 2, 3, 4, 5, 6, figure 16, by imposing 0 displacement for all directions. The forces will be apllied normally to the top beam in joint 7, with a value of 35000 N and 45000 N, figure 16.

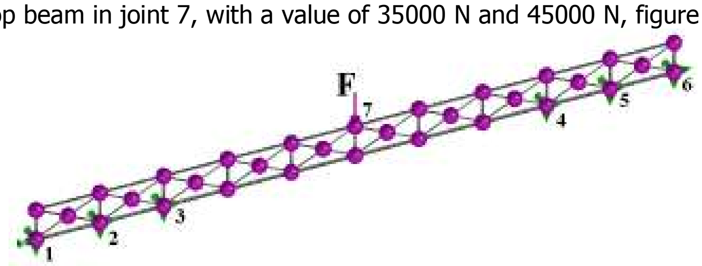

Figure 16. The joints, restraints and force applied to the truss structure

After the mesh is created and the study is runned, the obtained results are presented in table 3 and figure 17 and 18.

Table 3

Load Force [N]	SolidWorks FEM analyse	
	σ_{max}	Factor of safety FOS
	MPa	-
35000	442.49	1.40
45000	568.91	1.09

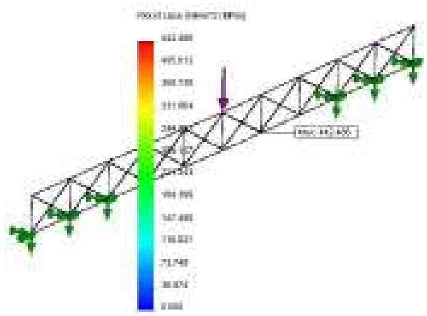

Figure 17. The bending stress σ distribution for F=35000 N

$\sigma_{\max} = 442.49 MPa$

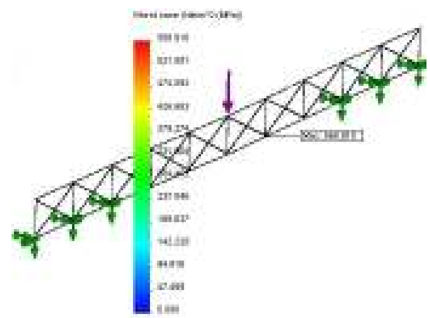

Figure 18. The bending stress σ distribution for F=45000 N

$\sigma_{\max} = 568.91 MPa$

4. Conclusion

The SolidWorks software is provided in the same interface with tools for geometrical design and structural analyze. The numerical results of the safety factor FOS confirm the real experiment results, showing that a truss structure is more capable than a beam, even though the mass of the truss structure (76.34 kg) is much smaller than the beam mass (117.6 Kg...229.9 Kg). For the same force F=35000 N, the FOS value of the truss structure is 1.4 compared with values 0.37..1.08 of the beam geometry.

References

[1] http://natgeotv.com/uk/engineering-connections/videos/hong-kong-international-airport

[2] Buzdugan, Gh., Voinea, R., Beles, A., Petre, A., Blumenfeld, M., Constantinescu, I., *Culegere de probleme de rezistenta materialelor*, Editia a VI-a, Editura Didactica si Pedagogica, Bucuresti, Anexa 9.

Addresses:

- Prof. Ph. D. Nedelcu Dorian, „Eftimie Murgu" University of Reşiţa, Traian Vuia Square, no. 1-4, 320085, Reşiţa, d.nedelcu@uem.ro
- Assist. Eng. Ianici Draghita, „Eftimie Murgu" University of Reşiţa, Traian Vuia Square, no. 1-4, 320085, Reşiţa, d.ianici@uem.ro
- Eng. Nedeloni Marian, „Eftimie Murgu" University of Reşiţa, Traian Vuia Square, no. 1-4, 320085, Reşiţa, m.nedeloni@uem.ro
- Eng. Daia Daniel, „Eftimie Murgu" University of Reşiţa, Traian Vuia Square, no. 1-4, 320085, Reşiţa, d.daniel@uem.ro