HEAVY MACHINERY - HM 2011, 29-32
ISBN 978-86-82631-58-3
Paper accepted: 30.05.2011

# Design and optimization for truss constructions using the software package Autodesk Inventor 2011®

Mladenovic Goran, dipl.ing,[1,*] Mr Popovic Mihajlo[1]

[1] Faculty of Mechanical Engineering, Department for Production Engineering, Belgrade University, Kraljice Marije 16, 11120 Belgrade 35, Serbia

*The use of truss constructions is frequent in mechanical engineering. This paper describes the procedures of construction and optimization for truss constructions at the example of roof structure of the Hall. Frame Analysis is used to understand the structural integrity for given frame with respect to deformations and stresses, when subjected to various loading and constraints. Once when the criteria are defined, it is possible to run the simulation and view the behavior relative to the conditions which are defined. Simulations help to identify performance issues and find better design alternatives. The paper doesn't gives the analysis of load, but structure optimization is performed for defined load. In the process were used two approaches, change of beam geometry without changing the profile and change the profile without changing beam geometry.*

**Keywords: Truss constructions, Autodesk inventor 2011®, Frame Analysis, Optimization**

## 1. INTRODUCTION

The truss welded structures are commonly used in mechanical engineering. The frameworks used to support the roofs of buildings are perhaps the most common truss constructions. Modeling of these structures is different from software to software. Sometimes it is necessary to draw a sketch of the cross section, and then the sketch extrude through a trajectory. Software Package Autodesk Inventor 2011® has the possibility to select standard profiles from database; it is only necessary to define the frame - skeleton of the structure. The advantage of this type of modeling is because with the starting the Frame Analysis environment and start a new analysis, the Frame Generator assembly is automatically converted into simplified model of beams and nodes. Because of the volume, in this paper we do not address the load problem, but we only perform the structure optimization for predefined concentric load of 15KN. In the process we have used two approaches, the change in beam geometry without changing the profile and the change in the profile without changing the beam geometry.

## 2. MODELING STRUCTURES

For modeling the structure it is necessary to first sketch the skeleton. It is enough to do sketch within a one sketch, as shown on Fig. 1. In the same figure is shown places where loads are acting.

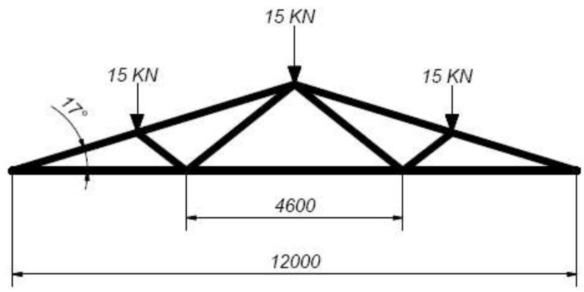


*Fig. 1. The skeleton structure with defined loads*

After defining the skeleton it is used the command Frame Generator. The above mentioned command requires the selection of standard profiles, materials, and a choice of line from the skeleton on which will be find the selected profile. Length of the selected profiles match the length of lines in the sketch and it is necessary subsequent to repair ends of the profiles using the commands Trim, Extend, Notch, and Miter. Using commands Change it is easy and quick to change a profile on construction which is of great importance for optimization process. After the implemented commands selected 3D model of the structure look like shown on the Fig. 2.

*Corr. Author's Address: FME, Belgrade University, Kraljice Marije 16, 11120 Belgrade 35, Serbia, gmladenovic@mas.bg.ac.rs

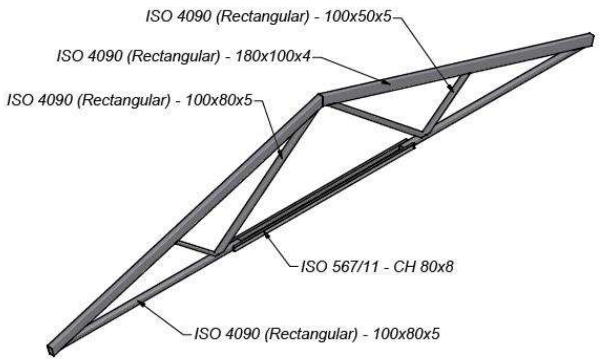


*Fig. 2. 3D model of the roof structure*

## 3. STRUCTURAL ANALYSIS

Frame Analysis is used to understand the structural integrity of a given frame with respect to deformations and stresses, when subjected to various loading and constraints. Once when the criteria are defined, it is possible to run the simulation and view the behavior relative to the conditions which are defined. Simulations help to identify performance issues and find better design alternatives. Beam elements are linear. Frame analysis does not support curved beams.

When the Frame Analysis environment is opened and starts a new analysis, the Frame Generator assembly is automatically converted into simplified model of beams and nodes.

From a high-level perspective, a typical frame analysis workflow looks like the following:

1. **Set expectations**: Estimate physical behavior using a conceptual model.
2. **Pre-processing**: Enter physics into the model and define analyses to perform.
3. **Solving**: Solve the mathematical model.
4. **Post-processing**: Display and evaluate the results.
5. **Review expectations**: Compare the results with the initial expectations.
6. **Conclusion** (Improve Inputs).

Next step is to create a simulation, and when the simulation is created it contains:

- Simulation Type
- Materials
- Sections
- Loads
- Constraints
- Releases
- Rigid Links

It should be noted that the software included the weight of the structure as a particularly stressful load.

After all this defined and simulation is finished it is possible to view results of simulation. That result includes:

- Forces
- Moments
- Stresses

On the Fig. 3 and Fig. 4 is shown the calculation results, only for displacements and the maximum normal stresses. It is possible to output results in HTML, MHTML, or RTF format. Reports contain text and PNG images that represent a static snapshot of the analysis results. For the observed structure and loads, the maximum displacement was 4.363 mm, and maximum Normal Stress was 70.7 MPa, the construction weight is 468.391 kg.

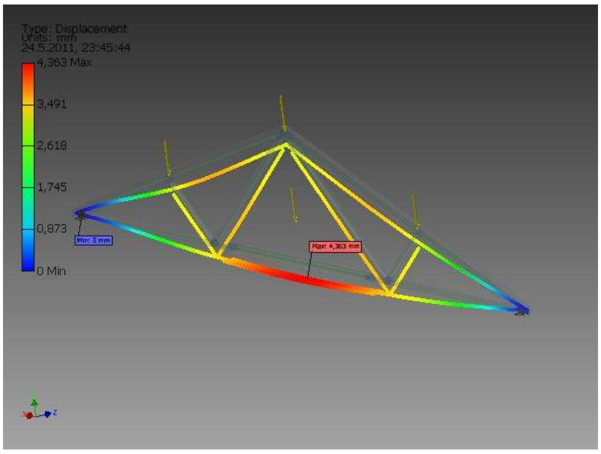


*Fig. 3. Displacement*

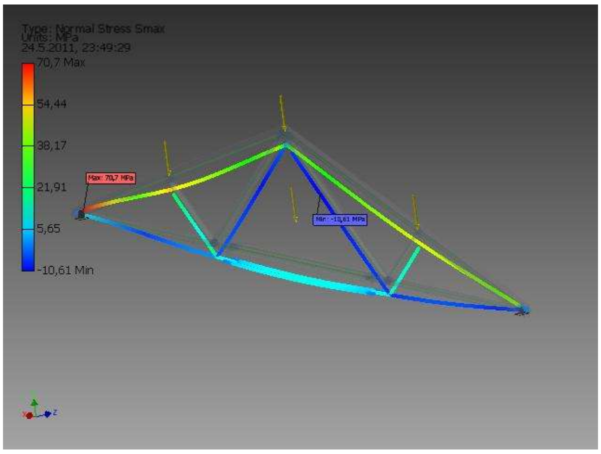


*Fig. 4. Normal Stress ($S_{MAX}$)*

## 4. STRUCTURAL OPTIMIZATION

For better visibility beams are numbered with numbers 1 – 5 as shown in Fig. 5.

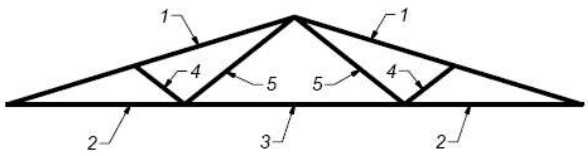


*Fig. 5. Numbered beams*

The results of optimization are presented in the table 1. Optimization goal is to obtain a uniform stress distribution, and obtain the structure which has a lower weight. In this case the design approach was to vary the profile and restart the simulation with the same load. Based on the table below, it can be concluded that the most appropriate beams form is under number 4, because it gets a smaller displacement in vertical direction and uniform stress distribution than in the first case. The design weight was not significantly different from case to case.

***Table 1**. Results of optimization*

| Number | Characteristics of the beam | | Mass [kg] | Maximum displacement [mm] | Normal Stress ($S_{MAX}$) [MPa] |
|---|---|---|---|---|---|
| | Beam | Standard profile | | | |
| 1 | 1 | ISO 4090 (Rectangular) - 200x100x4 | 477.687 | 4.158 | 68.99 |
| | 2 | ISO 4090 (Rectangular) - 100x80x5 | | | |
| | 3 | ISO 567/11 CH 80x8 | | | |
| | 4 | ISO 4090 (Rectangular) - 100x60x5 | | | |
| | 5 | ISO 4090 (Rectangular) - 100x60x5 | | | |
| 2 | 1 | ISO 4090 (Rectangular) - 180x100x4 | 464.942 | 4.235 | 72.1 |
| | 2 | ISO 4090 (Rectangular) - 120x60x4 | | | |
| | 3 | ISO 567/11 CH 120x12 | | | |
| | 4 | ISO 4090 (Rectangular) - 80x60x5 | | | |
| | 5 | ISO 4090 (Rectangular) - 80x60x5 | | | |
| 3 | 1 | ISO 4090 (Rectangular) - 160x80x6 | 508.901 | 3.77 | 56.49 |
| | 2 | ISO 4090 (Rectangular) - 120x60x4 | | | |
| | 3 | ISO 567/11 CH 120x12 | | | |
| | 4 | ISO 4090 (Rectangular) - 80x60x5 | | | |
| | 5 | ISO 4090 (Rectangular) - 80x40x5 | | | |
| 4 | 1 | ISO 4090 (Rectangular) - 160x80x6 | 471.309 | 3.843 | 57.52 |
| | 2 | ISO 4090 (Rectangular) - 100x80x3 | | | |
| | 3 | ISO 567/11 CH 100x10 | | | |
| | 4 | ISO 4090 (Rectangular) - 80x60x5 | | | |
| | 5 | ISO 4090 (Rectangular) - 80x40x5 | | | |

It was considered another case of the structure solution. The profiles are the same as for variant solutions under number 4, with inserted another vertical profile with dimensions 80x40x3. By inserting this profile, the maximum displacement was reduced to 3.13 mm and it was received a uniform stress distribution (maximum stress was 57.14 MPa) what is shown on Fig. 6. It should be noted that the weight of the structure is not significantly increased and in this case it was 480.65 kg.

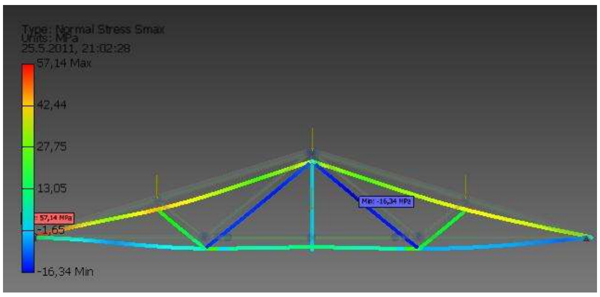


*Fig. 6. Stress distribution for the case of added profile*

## 5. CONCLUSION

This paper describes the procedures of construction and optimization of a truss construction at the example of roof structure of the Hall with the range of 12m. It was demonstrated the procedure using the command Frame Generator of the Software Package Autodesk Inventor 2011®, which use reduces the total time of time modeling the structure. Also was shown the process of analysis and design that was executed by command Frame Analysis. Structural analysis led to the optimum design which has the smallest displacement and the most uniform stress distribution for the same load, all in order to reduce the total weight of the construction. The paper doesn't gives the analysis of loads, but structure optimization is performed for defined load which acting in the three points as shown in Fig. 1.

## 6. REFERENCES

[1] Kalajdžić M., "Metod konačnih elemenata, osnovi statike, statika nosećih struktura, dinamika nosećih struktura, primene", IAMA, Beograd, 1978

[2] Sekulović, M., "Metod konačnih elemenata", Građevinska knjiga, Beograd, 1988

[3] Rašković, D., "Otpornost materijala", Građevinska knjiga, Beograd, 1985

[4] Spyrakos, C. C., "Finite Element Modeling in Engineering Practice", West Virginia University Press, Morgantown, WV

[5] Mattheck, C., "Design in der Natur", der Baum als Lehrmeister, Rombach, Freiburg, 1997

[6] Autodesk Inventor 2011® Profesional - Help

[7] http://usa.autodesk.com/