Abstract: This paper refers to linear static analysis of truss and beams structures modelled in SolidWorks. In SolidWorks truss and frame elements are available only for static studies. For illustration it was modelled and studied the behaviour of a support of pipe. The truss is a welded structure modelled with Weldment Structural Members and it is often found in industrial constructions. During this study, there have been investigated the following FEA steps: Pre-Processing, Processing and Post-Processing. The results of this study are images corresponding to Axial Stress and The Maximum Displacement in different situations.

Key words: SolidWorks, weldment, beams, truss, FEA, mesh.

INTRODUCTION

The beams structures are specific to metal construction. In order to sustain a pipeline over a damaged area there was modelled a truss structure. The truss element is a very common structural member. The pipeline is supported by some support brackets mounted on the truss nodes [1].

For this study it was chosen a truss consisting of three pipes connected by other pipes. The truss sizes are: the truss length, L=3m; the truss height, h=0,3m; the inside diameter of the pipe, D1=24; the outside diameter of pipe, D2=32.

To avoid any deformation of the structure due to temperature fluctuations, the beam is fixed at one end and the other end is supported and guided allowing the translation. Figure 1 presents a simplified truss representation.

2. THE STUDIED TRUSS

In SolidWorks one can create their own weldment profiles using Weldment Structural Members. These profiles are characterized by type, size and standard (ISO, ANSI și ANSI inch).

The welded structure is an assembly of components, as shown in the FeaturesManager Design Tree, only with one part. For this purpose, the functions Weldments part creates a multi body [2].

Weldments module has libraries with most profiles (angle iron, rectangular tube, pipe and c-channel) in the Structural Members function. Additionally weldment profiles are available on the Design Library tab, Structural Members folder.

In the present case it was selected by Standard - ISO, and by Type - pipe with 33.7x4 size. It may change the sizes, depending on requirements, using the sketch of the pipe. In this paper 32x4 sizes are used.

The studied truss is shown in Figure 2. All the modelling steps are presented in FeaturesManager Design Tree.

A beam structure in SolidWorks starts with a 3D sketch of the structure, in a Part type (Figure 3 a). In the first step a 3D sketch structure will be generated. Structural Member function generated the 3 pipes of 3000 mm long and the base with the same section (Figure 3 b).
Studies regarding the static analysis of truss structures

One can use segments and other solid bodies for trimming; in this case the 3 long pipes are the trimming body using End Trim and End Butt2 options. The truss base will be multiplied with Linear Pattern function (Figure 4).

Finally the truss looks like Figure 2. The final was completed with two attachment supports necessary for positioning the truss on the ground using solid body.

STATIC ANALYSIS CASE, PROCESSING

Finite Element Analysis is a process which can predict deflection and stress on a structure, here on a truss. A finite element model is a complete idealization of the entire structural part, including the node locations, the elements, material properties, loads and boundary conditions.

Finite Element Analysis essentially consists of the following 3 steps: Pre-processing, Processing, Post-processing.

After modelling the truss (pre-processing), for creating a static analysis case, the material of the truss is very important. It is made of steel OL37, equivalent with ANSI 1035, from the Library Material Catalogue, with following features: Young Modulus $2.049 \times 10^{11}$ N/m² and Poisson Ratio 0.29. The admissible tension for OL37 is $90-120$ N/mm² (MPa). Mass Properties function informs us about the truss weight, namely $66972.95$ grams.

The only inner force that develops in the member is the axial force. This is constant along its length and generates axial stress, uniform throughout the cross section. The 1D structural members are known as trusses. In finite element analysis truss members are modelled as truss elements. The truss is a special beam that can resist by only axial deformation [3].

Creation of a FEA model starts with the definition of a New Study, namely Static Study (Processing). Static studies provide tools for the linear stress analysis. The target is to find the displacements and the axial stress of the structure under 5 loads of 1000N, with two legs restrained.

Simulation verifies and displays all the joints. Edit Joints window present all joints created automatically and if it is necessary, after editing, one can generate the joint again with Calculate button (Figure 6). This information is presented in Joint group folder (Figure 5).

For fixed the truss the Fixed Geometry is used (Figure 7 a). The other end is supported and guided allowing the translation, for which the Roller/Sliding restraint is used. The Roller/Sliding is used to specify that a planar face can move freely into its plane but cannot move in the direction normal to its plane [5] (Figure 7 b).

The loaded forces, using External Loads, are applied to the 5 joints like in Figure 8. The pipe is connected to these joints. On the moment the forces are estimated to 1000N for every 5 joints. The forces direction is defined in the Selected direction section. One edge from the end support was selected.
Studies regarding the static analysis of truss structures

The next step is generating the Mesh, launching Mesh only function. FEA uses a complex system of points, nodes, which make a grid named mesh. There are no mesh control options for trusses, only for Mesh Density. The number of uniform elements mesh can be defined automatically or can be managed using Mesh Control.

Figure 9 presents, the meshed truss. It is important that the solid geometry of Weldment is not meshed when beam elements are used. It is meshed the underlying wireframe geometry [4].

![Meshed Truss Diagram]

**FIG. 9** The meshed truss.

**STATIC ANALYSIS CASE, POST-PROCESSING**

The post-processing step, named also the computing part, begin with Run action, this starts the solver for the active study. The result is the Simulation menu presented in Figure 10.

![Simulation Menu]

**FIG. 10** The simulation menu.

One can get tables with measurements of forces on each node; these tables can be also plotted, as all obtained information (Figure 11).

![Beam Forces Table]

**FIG. 11** List of beam forces.

The post-processing results are illustrated in Figure 12, namely:
- Axial stresses for truss elements (Figure 12 a);
- Displacement and deformation plot (Figure 12 b).

The admissible tension of the material is 90-120 N/mm². The maximum stress is 19.084 N/mm² and the maxim displacement is 0.697.

![Post-processing Results (1000N)]

**FIG. 12** The post-processing results (1000N).

As this truss is oversized, one can also study the situation in which the pipe is heavier acting on every 5 nodes with 5000N. Figure 13 presents the results.

![Post-processing Results (5000N)]

**FIG. 13** The post-processing results (5000N).

![Beam Diagrams]

**FIG. 14** Beam Diagrams.
A successful run of this study will create a large output results. Figure 14 presents a Beam Diagram with axial forces for the second situation (5000 N).

**USING THE OWN GRAVITY FORCE**

In the next step the beam is loaded also with its own forces. Its own weight is as a consequence of the existence of inertial fields translational. One can use the Gravity Property Manager to apply gravity loads to a part or assembly document for use in structural and nonlinear analyses [2].

In order for using the gravity force the Gravity load from External Loads will be access. Figure 15 a presents the Gravity window accessed from External Loads. After meshing and computing it was obtained the Results like as Figure 15 b.

![Fig. 15](image)

The simulation menu using the Gravity force.

The post-processing results for a stressed truss with 5 forces every with 5000N in 5 joints and with the Gravity load is presented in Figures 16 a, b. Because the admissible tension of the material is 90-120 N/m² result that the truss is good dimensioned.

![Fig. 16](image)

The truss diagrams with gravity load.

8. CONCLUSION

This paper is a study about the truss deformations. Truss is a framework consisting of rafters, posts, and struts, supporting a roof, bridge, or other structure. Truss is a special beam element that can resist only axial deformation. It can be modelled using solid elements, thin-shell elements, or beam elements, depending on the desired result. In this situation, for good results, the weldments structure with Structural Members was used.

The analysis of the truss worked properly with the fixing parts modelled as solid parts. It was also studied the variant in which the solid parts, needed for mounting, were excluded from analysis. The Fixed Geometry and Roller/Sliding were applied to the free end joints. In this case, after computing, errors resulted in the analysis.

SolidWorks meshes each member of truss creating a number of beams FEs. Each beam element after mesh is represented by cylinders regardless. In the truss structure the underlying wireframe geometry is meshed.

Finally the truss was loaded with 5 forces, everyone with 5000N, in 5 joints and also it was considered the Gravity force. Gravity is the own truss force, it is a consequence of the existence of inertial translational fields. Analyzing the fields of displacements and tensions lead to the result that the truss is a rigid structure (Figure 16). The maximum displacement (3.471 mm) is in the central area where the maximum stress tension is 95.419 N/m².

**REFERENCES**


Author:

Assoc. Prof. eng. Mihaela URDEA, Department of Manufacturing Engineering, Transilvania University of Brașov, Romania. E-mail: urdeam@unitbv.ro, tel. 00472346055.