IOP Conf. Series: Materials Science and Engineering **399** (2018) 012051 doi:10.1088/1757-899X/399/1/012051

Static linear analysis for trusses structure for supporting pipes

M Urdea[1*]

[1] Department of Manufacturing Engineering, Transilvania University of Braşov, Romania.

[*] E-mail: urdeam@unitbv.ro

Abstract: This paper is a study about truss deformations. This truss is supporting a pipe, it can be a bridge, or another structure and can be necessary for protecting rivers. The truss is a welded structure modelled with Weldment Structural Members, which is often found in industrial constructions and it can be constructed in different shapes. In order to sustain a pipeline over a damaged area, some trusses structures were modelled. The pipeline is supported by some support brackets mounted on the truss nodes. 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. The purpose of this paper is to find the best form for trusses using SolidWorks and to avoid oversizing these structures.

1. Introduction

Trusses are very common structural members; they can be used for sustaining a pipeline over a damaged area. The dictionary presents the truss as a framework, typically consisting of rafters, posts, and struts, supporting a roof, bridge, pipes or other structures. In engineering, a truss is a structure that consists of two-force members only, where the members are organized so that the assemblage as a whole behaves as a single object. A space frame truss is a three-dimensional framework of members pinned at their ends [1].

This paper presents several truss shapes. The aim of this study is to develop a truss modelling technique, to assign possible constraints and forces and to perform a linear static analysis of trusses.

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 to only axial deformation [2, 3].

2. The flat truss study

The Warren, Howe and Pratt trusses have been made of wood, iron or steel since the19th century. The first truss modelled in SolidWorks is a flat truss design after Howe truss model with beam welded elements as in figure 1. For the Howe truss the diagonal members are in compression, the vertical members are in tension, except for the centre which is a null force [4].

After the truss sketch all truss tubes are generated in SolidWorks as Structural Members. For a truss the lines generated in sketch must exactly meet at common points named joints.

Due to the Structural Members generation in Features Manager Design Tree, a Cut list folder appears in which SolidWorks groups by shape and sizes all the profiles of a welded structure.

For this study the following truss characteristics were chosen: length, L=8m; the truss height, h=0,8m; the inside diameter of the truss elements, D1=70; the outside diameter of truss pipe, D2=75. It is made of steel OL37, equivalent with ANSI 1035, from the Library Material Catalogue, with following features: Young Modulus 2,049e+011N/m2 and Poisson Ratio 0.29. The admissible tension for OL37 (S 235 JR the new symbol, after SR EN 10025) is 90 - 120 N/mm^2 (MPa). The Mass Properties function informs us about the truss weight, namely 171048.44 grams.

In order to support some pipelines for crossing a river or over a rugged terrain it is necessary to use a structure like a truss. Figure 2 shows the scheme of the truss for supporting a pipe by some support brackets mounted on the truss joints. A static analysis for the truss is required to optimize its construction.

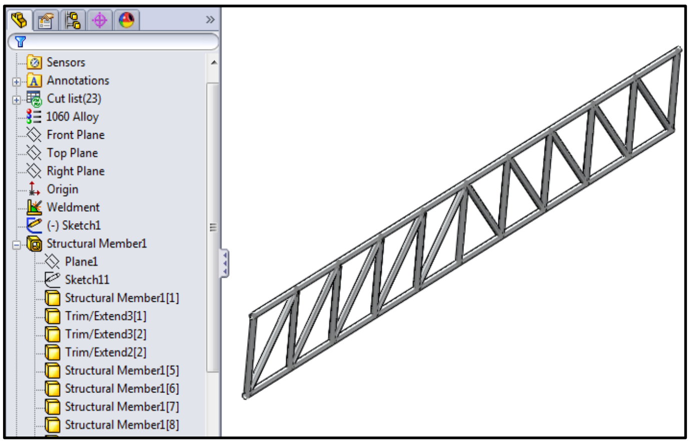

Figure 1. Flat trusses.

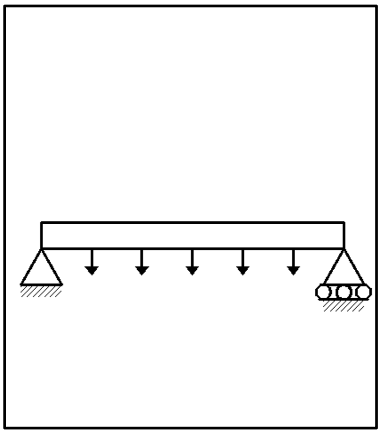

Figure 2. Truss scheme.

Figure 3 presents the Structural Member window, the form of the tubes was selected. In these studies: Standard - iso, Type - pipe and the pipe dimensions were chosen and this type will be modified. Four Structural Member groups were generated due to the truss shape. The way of generating the model may influence the development of the finite element analysis. In this first step the static simulation recognizes the weldment geometry and anticipates that they are beam elements. A Solid Body was created for each structural member and it was placed then in the Cut-List folder (figure 4) [5].

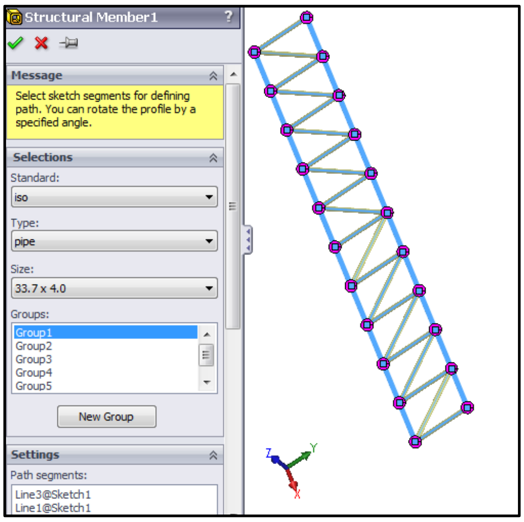

Figure 3. Structural member for flat trusses.

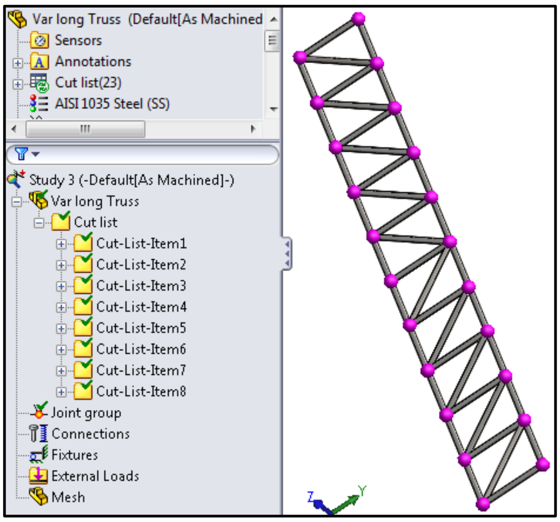

Figure 4. Cut-List folder.

In this Cut-List folder each element must be declared as a truss element, using the Apply /Edit Beam window. This activity is very important for the next step because, by default, this structure was declared with beam elements.

3. Static analysis for the flat truss

The static analysis assumes a complete idealization of the entire structural part (model), including the node truss locations, the elements, material properties, loads and boundary conditions. During this study, the following FEA steps were considered: Pre-Processing, Processing and Post-Processing. The paper includes diagrams and conclusions to illustrate the importance of this study. For this truss the external forces and the reactions are considered to act only at the nodes.

Edit Joints manages the joints with the Calculate button and all the joints are created automatically. The joints coincide with the pierce point of the weldment profile. The joints are presented in a Joint group folder (figure 4). It is noticeable that the joints transmit only forces. Moments are not present at the joints of a truss [1].

Due to the static analysis, the calculations of the forces on the every truss element can be checked by simulation, thus avoiding the complex calculation of the forces in each member of the truss and indicate whether the members are in tension or compression. It is known that the results obtained from the FEA analysis are less valid in nodal bonding areas (welds, riveting assemblies, screw assemblies) [2].

Due to this, the analysis model must have the same behaviour as the real model; the boundary conditions are defined for each of the extremes joints. To avoid any deformation of the structure due to temperature fluctuations, the truss was fixed using Fixed Geometry, namely at one end it was fouled constraints, that is Immovable (all three translations are zero on face, edge, or vertex). For the other end, using Use Reference Geometry, the translation of Oy and Oz axes was cancelled. The translation at this end can be achieved by using a translation guideline. Figure 5 and 6 present the fixtures at both ends. Every joint has six degrees of freedom per node; therefore distinguish between Fixed and Immovable restraint [6].

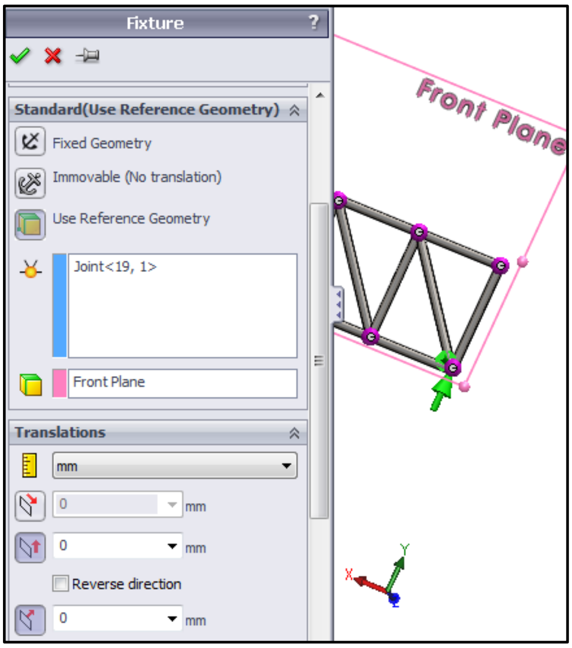

Figure 5. Fixture window - translation of Oy and Oz axes was cancelled.

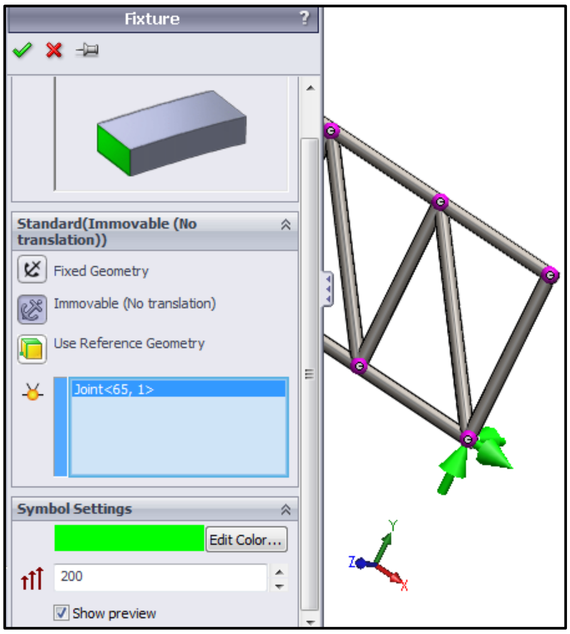

Figure 6. Fixture window - one end was Immovable.

On the other hand, all the joints are also constrained by using Use Reference Geometry. The constraints are in relation to the Front Plane and the displacement perpendicular to this plane is cancelled as in figure 7. These are restraints to prevent rigid body motion.

In this case the finite element analysis is based on a software system that assesses a material or an object by using the finite element method in order to find the applied stresses that have different effects on the design. A highly regarded advantage of the method for producers and engineers worldwide is that FEA can help determine a weakness in a design before the product is manufactured [7].

External Loads can be attached that is the Gravity and the Forces. The truss is loaded with its own forces. Its own weight is a consequence of the existence of inertial translational fields. The Forces, using External Loads, are applied to the five joints (figure 8), it is about concentrated forces at joints. The pipeline can be connected to these joints. In this study the forces are estimated to 11000 N for every 5 joints.

The next step is generating the Mesh,–by launching Mesh function. It is important to note that a truss is a special beam element that can resist only axial deformation. As in this study the beam was considered as a truss there are no mesh control options for truss frames. A straight structural member identified as a truss is represented by one truss element [8].

The post-processing step and the computing part begin with Run action which starts the solver for the active study. The simulation calculates the space truss joint locations and their displacements.

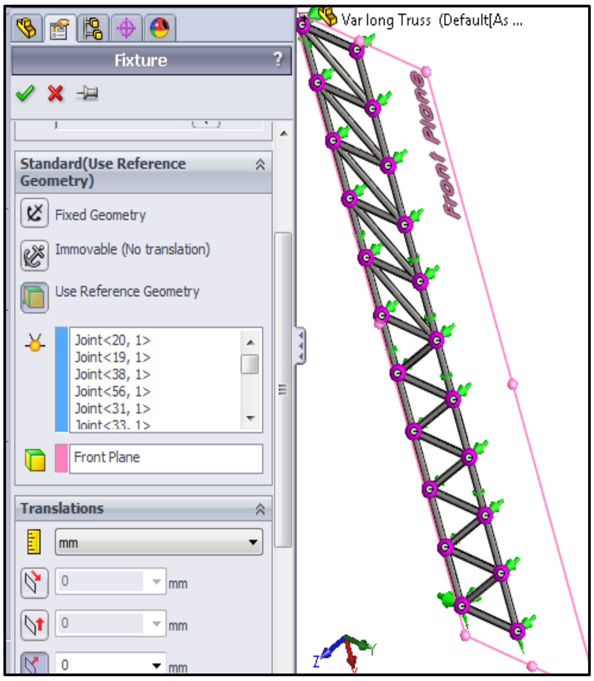

Figure 7. Reference Geometry for all the joints.

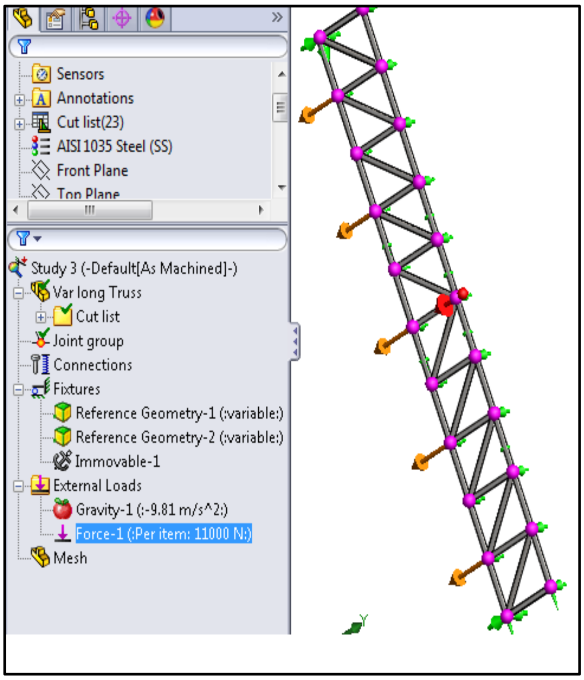

Figure 8. External loads.

The post-processing results are illustrated bellow, namely:

- upper bound axial and bending stresses (figure 9);
- displacement and deformation (figure 10).

The admissible tension of the material is 90-120 N /mm^2. The maximum stress is 104.535 N/mm^2 and the maximum displacement is 8.923mm. It is very important to define the results folder location for every FEA Study. After the post-processing the List Beam Forces and the Beam Diagrams also result.

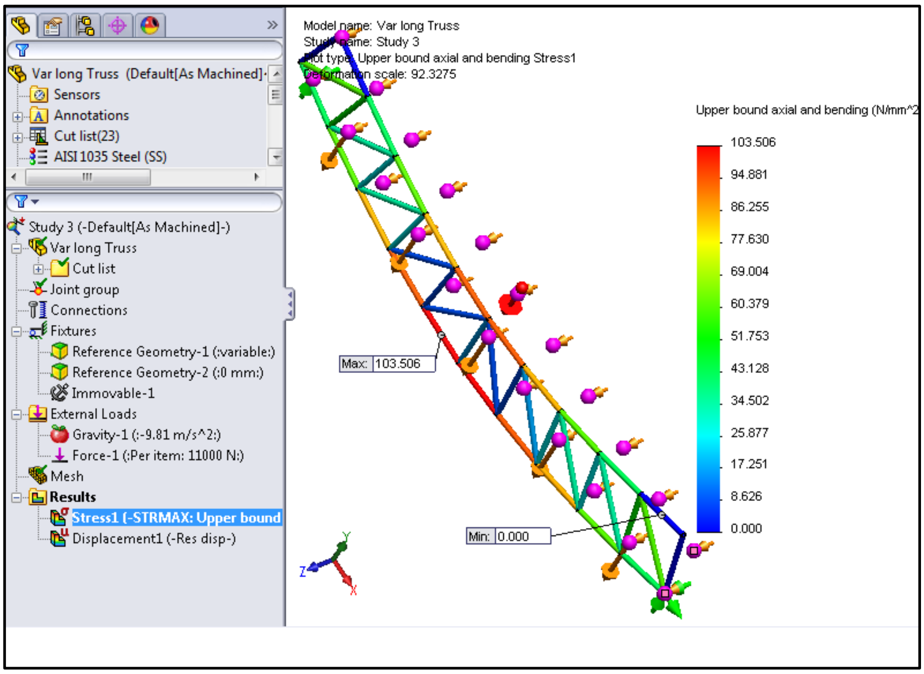

Figure 9. Upper bound axial and bending stresses for Howe truss.

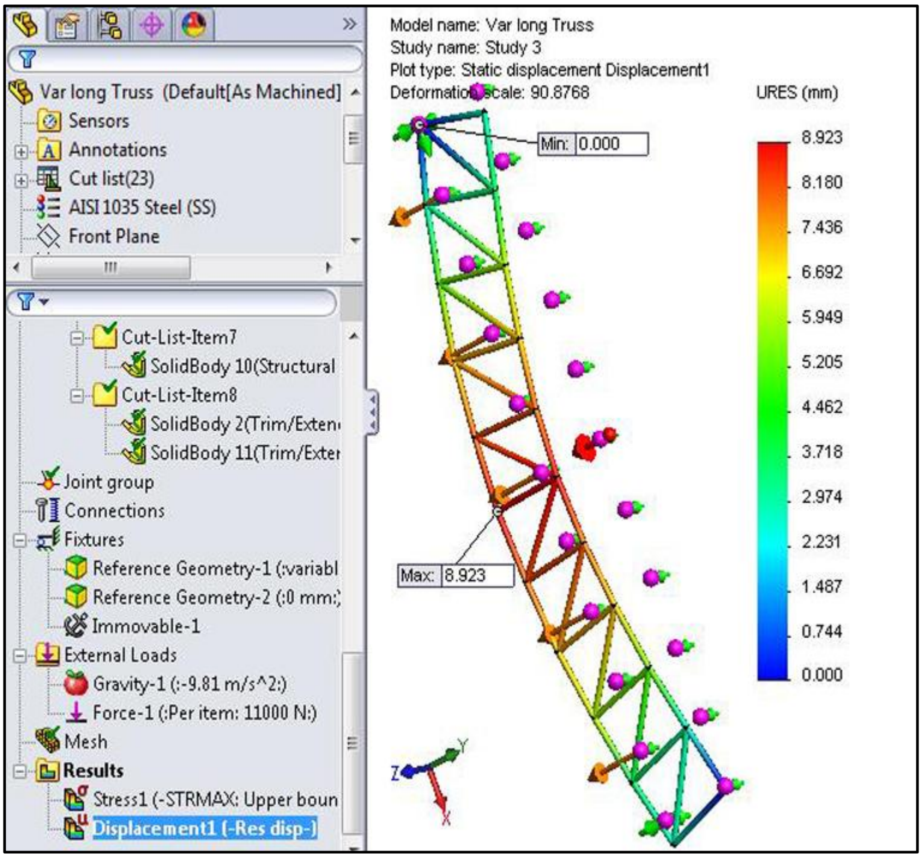

Figure 10. Displacement and deformation for Howe truss.

For the very popular Pratt truss the diagonal members are in tension, the vertical members are in compression, except the centre which is a null force. Figure 11 and figure 12 present the post-processing results for the Pratt truss. One can notice that the simulation results are very close for the Howe truss simulation results. The simulation conditions were similar.

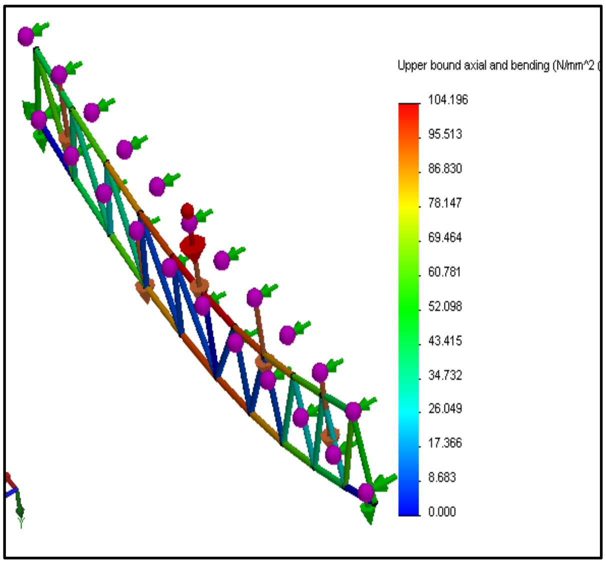

Figure 11. Upper bound axial and bending stresses for Pratt truss.

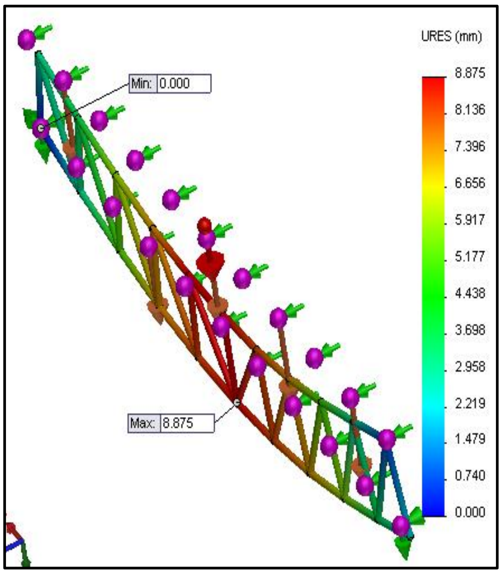

Figure 12. Displacement and deformation for Pratt truss.

4. Double flat truss study

The second truss modelled in SolidWorks is a double flat truss as in figure 13 and it develops from the Howe truss structure. In this case, the truss dimensions are: the truss length 8m; the truss height 0,8m, the truss width 0,6m; the diameters of the truss elements are the same. This study respects similar steps. Figure 14 presents the fixtures and the external loads. The pipeline can be connected to five joints as in figure 12. At the moment the forces are estimated at 11000 N for every 5 joints.

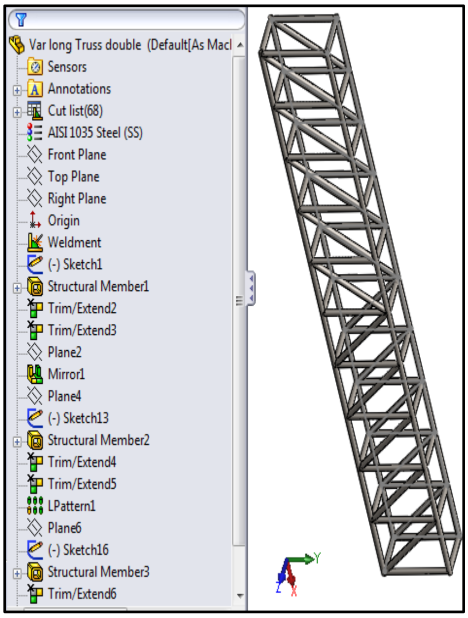

Figure 13. A double flat truss.

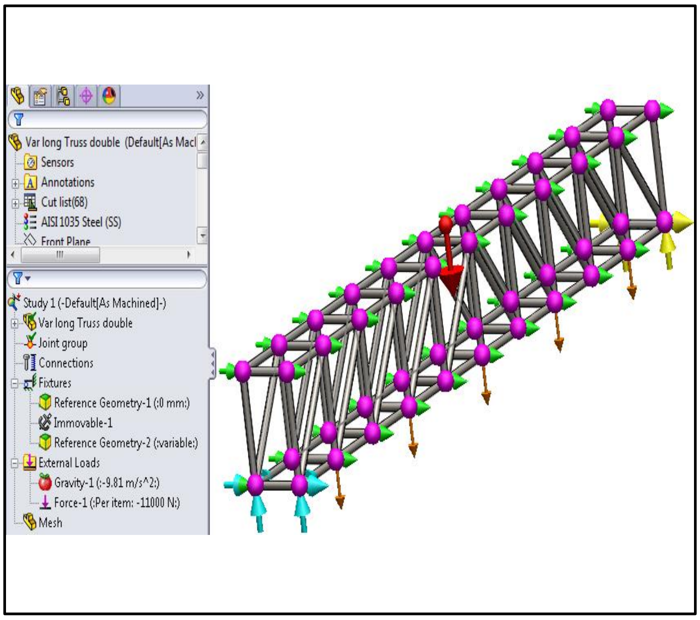

Figure 14. All fixtures and external loads.

The post-processing results are illustrated bellow, as follows:

- upper bound axial and bending stresses (figure 15);
- displacement and deformation (figure 16).

The admissible tension of the material is 90-120 N /mm^2. The maximum stress is 102.036 N/mm^2 and the maxim displacement is 8.726mm. There is a very small difference between the values of the two studies.

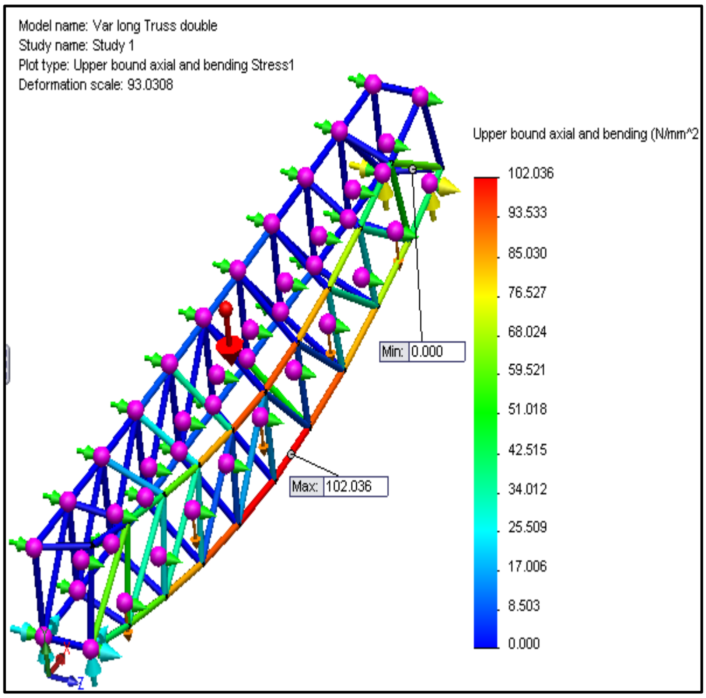

Figure 15. Upper bound axial and bending stresses for the double flat truss study.

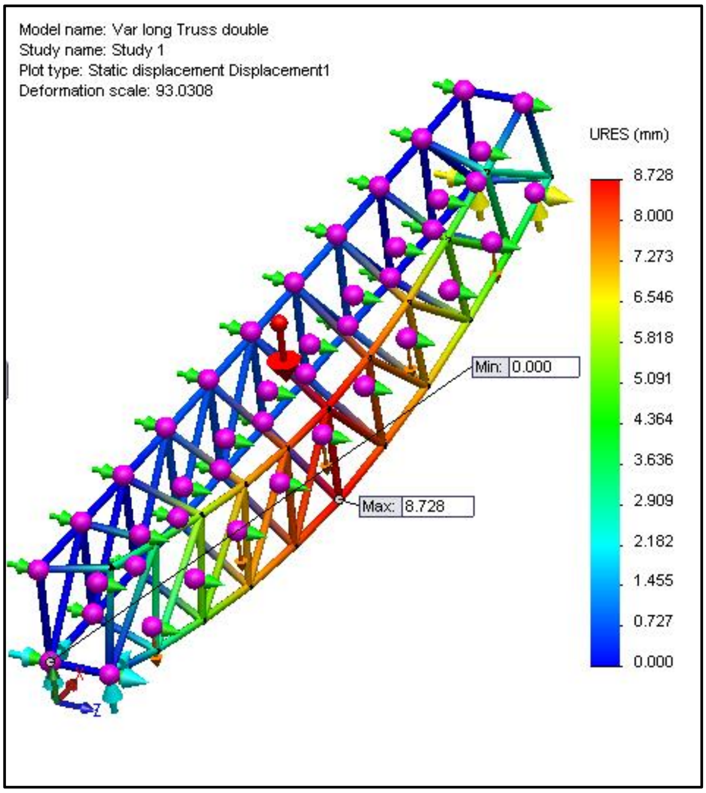

Figure 16. Displacement and deformation for the double flat truss study.

5. The third study

Considering the same second truss, a third study was conducted with another position of the pipeline; namely the possibility of introducing the pipeline into the truss was studied. In this situation the pipeline will be supported on five beam elements. The study will refer to a beam. In the Cut-List folders each element must be declared as beam element. The processing results with all the restrains and external loads are illustrated in figure 17.

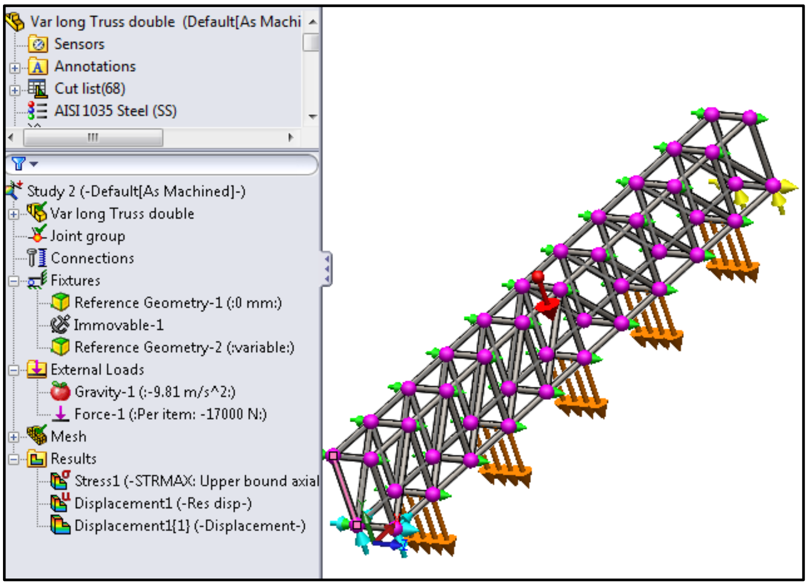

Figure 17. Results with all restrains and external loads.

As in this study the forces are estimated at 17000 N/mm^2 for every 5 beam elements, the post-processing results are illustrated in figures 18 and 19. The analysis of the fields of axial and bending stresses and displacements lead us to the result that the beam is a rigid structure. The maximum displacement (7.475mm) is in the central area where the maximum stress tension is 98.15 N/m2.

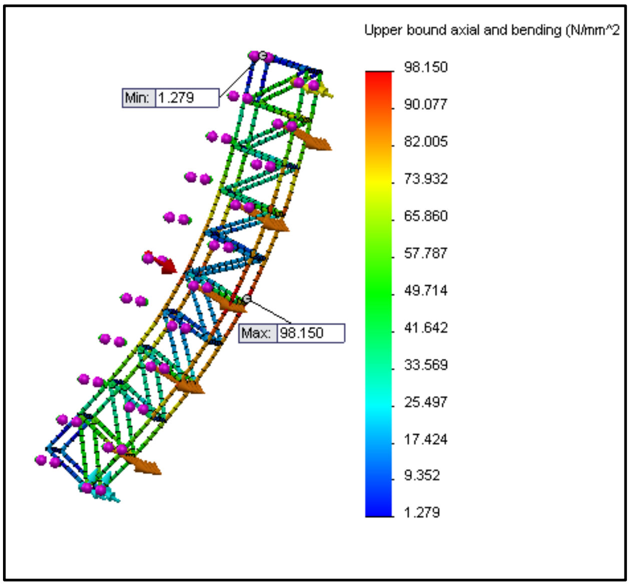

Figure 18. Upper bound axial and bending stresses for the third study.

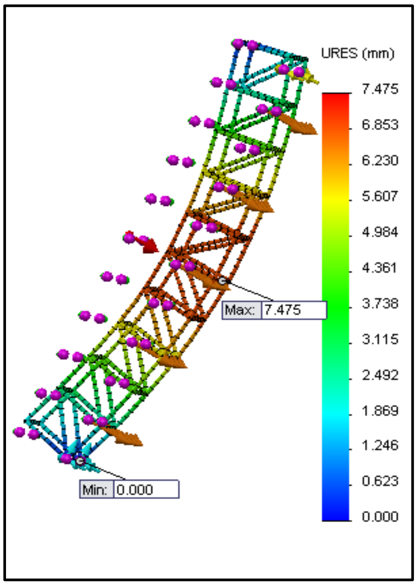

Figure 19. Displacement and deformation for the third study.

6. Conclusions

This paper is a study about the truss and beam deformations and the purpose is to find the best form for trusses using SolidWorks and to avoid oversizing these structures. The truss can be modelled by 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 studded trusses are the Pratt truss and the Howe truss. The verticals elements which are included in these trusses are for raising the span length of the truss bridge.

Three different trusses designs were considered. After the first studies, Axial stresses, Displacement and Deformation in the same simulation conditions resulted with similar values namely, there is a very small difference between the values of the two studies. For these trusses, loads can be applied only to the joints. In the second study the double truss has a more complex structure and yet the results are almost similar.

In this situation, the third study refers to a beam and in the simulation the truss elements into beam elements ware changed. By analyzing the fields of displacements and tensions the result is that the truss is a rigid structure. For the third study, the use of the beam simulation method and higher forces led to the maximum displacement (7.475 mm) and the maximum axial stress tension (98.15 N/m^2) which are lower than in the first variants.

References

[1] SolidWorks Help http://help.solidworks.com/2017/english/solidworks/cworks/c_Beams_Trusses.htm
[2] Mogan G L and Butnariu S L 2007 *Finite element analysis in engineering* (Braşov: Transilvania University of Brașov) p 197
[3] Radostina V P 2014 *Introduction to Static Analysis Using SolidWorks Simulation* (Boca Raton: CRC Press) p 353
[4] Garrett Bridges https://www.garrettsbridges.com/design/
[5] Akin J Ed 2009 *Finite Element Analysis Concepts via SolidWorks* (Houston: Rice University) p 303
[6] Urdea M 2017 *Int. Conf. on Engineering Graphics and Design ICEGD* vol 1 (Constanța: JIDEG) pp 155-158
[7] Croitoru E I, Magurian D and Oancea G 2016 Finite Element Analysis of Heating-Cooling Systems Used for Laminating Tools *Transactions of FAMENA* **40(1)** 111
[8] Kurowski P M 2011 *Engineering Analysis with SolidWorks Simulation 2011* (Mission KS: SDC publication) p 455