Mihaela URDEA

STRESS AND DISPLACEMENT ANALYSIS FOR A PIPES SUPPORT

Abstract: *In this paper, a structural analysis with Weldment Structural Members is performed. Halfen Company provides solutions for pipes support. The assembled support structures are characterized by increased stiffness, their elastic deformations being reduced. The identification of stress and deformation fields is done by finite element analysis. The purpose is to create a finite element model, to find the best shape of a pipes support using SolidWorks. It is very important to avoid oversizing the support.*

Key words: *Weldment Structural Members, SolidWorks, pipes support, FEA.*

1. INTRODUCTION

The aim of this paper is to model a pipes support, to assign appropriate forces and constraints and to develop a linear static analysis of welded pipe supports.

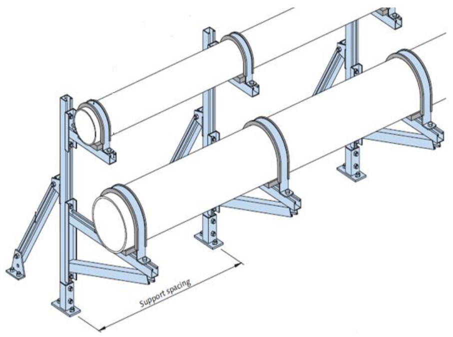
The pipes network is very common. The size of this network may depend of the served area, on the surface of a given city or district or enterprise.

The pipes sistem can be developed by heating or could pipes network. The heating network is a system of brached pipes which provide heat distribution. Its size may acording to the served area. The mechanical calculation of a steel pipe network takes into account the stress states in the pipe material generated by the loads from: internal pressure and pretension, from uneven heating of the pipes, from the storage of supports, from accidental weights (snow) [1].

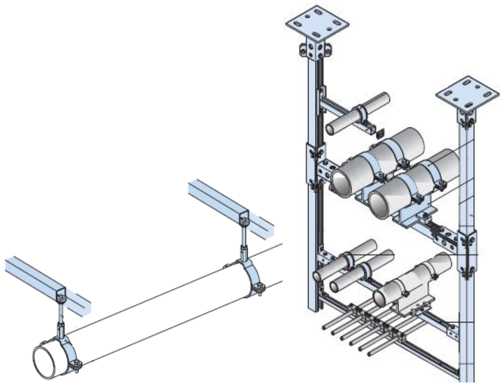
Installation of pipes equipment, requires support or their anchoring elements to the resistance structure of the building. This is made by way of elements or special subassembly, such as: columns for supporting pipes, heat pipes console supports, support bearings, compensators and clamps [2].

The pipe loads can be the pipe weight, the pipe content, all the pipe fitting, possibly the insulation. They are four main functions of a pipe support, namely to support a specified load, to guide and to anchor the pipes. Not at last to absorb shock on the all directions. Of course, pipe supports used in low or high temperature may contain insulation materials. The generally design configuration of a pipe support assembly is dependent on the loading and operating conditions [3].

Figure 1 presents some variants of pipe supports with clamps proposed by Halfen Company [4].



a



b **c**

Fig. 1 Pipes supports [4].

For supporting two pipes of different sizes, the support of figure 1a was adopted. Figure 2 presents a fragment of a pipes lines modeled in SolidWorks.

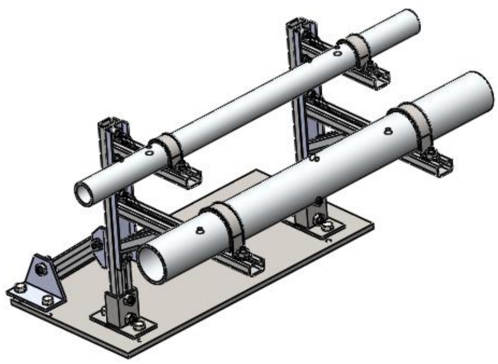


Fig. 2 Pipes support modelled in SolidWorks.

Halfen Company offers several pipe supports spacing dimensions for drain pipe, like figure 3.

The could pipes network is also very common. This paper refers to could pipes (diameter 162 and 212), supports modeled in Solidworks namely welded supports. The support spacing could be 1600 mm maximum.

2. WELDED SUPPORT

Starting from the modular support presented by Halfen Company, a somewhat similar was modelled in SolidWorks. The pipes support, studied in this paper, is a three-part assembly. The base body was modelled using Weldments components (Fig. 4).

External pipe-Ø ① [mm]	Pipe weigth (empty) [kg/m]	Pipe weigth (filled with water) [kg/m]	Support spacing max. [m]
Drain pipe, GA DIN 19 500			
60	5,300	7,505	0,60
80	7,100	11,283	0,80
112	10,300	18,791	1,15
137	13,700	26,763	1,35
162	17,300	35,437	1,60
212	32,700	64,100	2,00

Fig. 3 Distances between the pipes supports [4].

The material components is steel ANSI 1035 is considered equivalent to OL37 (the new symbol for OL37 is S 235 JR, according to SR EN 10025). ANSI 1035 has the following features: Young Modulus 2.049e+011N/m^2, Poisson Ratio 0.29. The admissible tension of ANSI 1035 is 120 N/mm2 (MPa). The elastic modulus defines the resistance to deflection of the material [5].

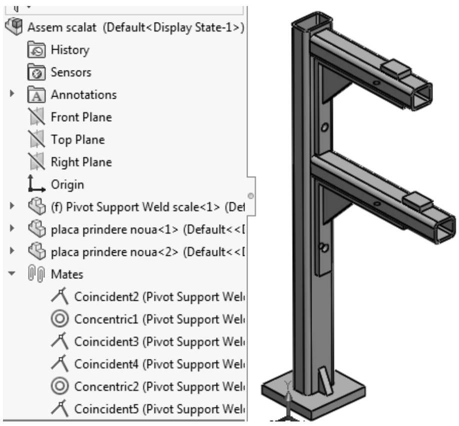


Fig. 4 Welded support.

3. STATIC ANALYSIS – FEA

Because the loads are applied slowly, the static analysis of the support is chosen. The initial situation of the ensemble is shown in figure 5. The welded components are automatically declared beams, the simulation recognizes the weldment geometry. This is observed in the three folders.

Considering the diversity of the components, the Weldments components (automatically declared Beam) have been transformed into solid.

Figure 6 presents the possibility to transform the beam in solid using a pull down menu.

The fasteners components are represented by connectors. This are mechanisms that defines how an entity is connected to another entity or perhaps to the ground. This connectors simplifies the modelling, in many cases, because they can simulate the desired behaviour, without generating the detailed geometry or defining contact conditions. The bolt connector can be used, generally, between two components, often, or between a component and the ground [5].

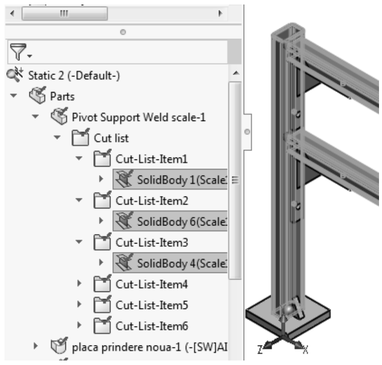


Fig. 5 Assamble support.

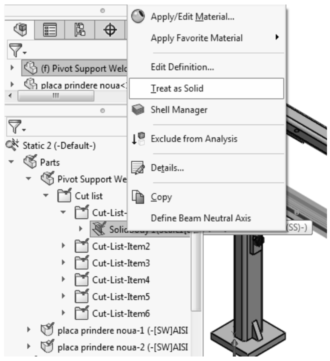


Fig. 6 Changing the component features.

For generating a connector bolt, can be defined the following steps:

- Selecting the type of the bolt, Standard or Counterbore Screw (Fig. 7);
- Selecting Calculated tensile stress area in the Strength Data section. SolidWorks calculate the tensile stress area of the bolt;
- Selecting the Pre-load section, if the axial load on the bolt is known, here 150N.

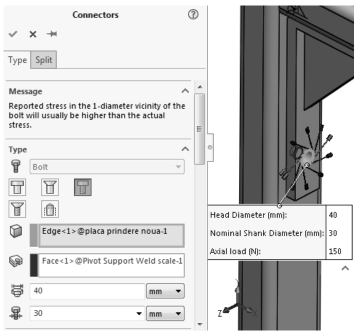


Fig. 7 Bolt connector.

SolidWorks give as information about using the bolt connector. The simulation of bolt connectors involves the internal creation of rigid links. The bolt connector is defined by a beam element which does not have any resistance to torque. This is consistent with the physical model. In reality, the slippage between the connecting parts is resisted by the frictional force provided by the clamp force (preload) [5]. In this way will be defined four visible Counterbore. Screws in the Connections folder (Fig. 8).

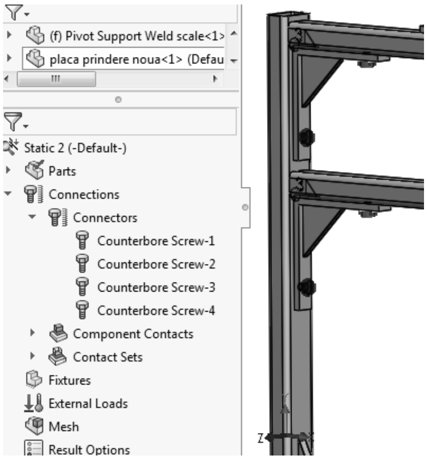


Fig. 8 Connection - Bolt connector.

It is very important, for a good result using Component Contact, to select the components and to set the default contact condition between them. In the Component Contact window, it is chosen Bonded Contact Type, after that it was inserted the support components.

In this Component Contact the selected components or bodies behave during simulation as if they were welded [5]. Figure 9 presents the Component Contact window with all selected components.

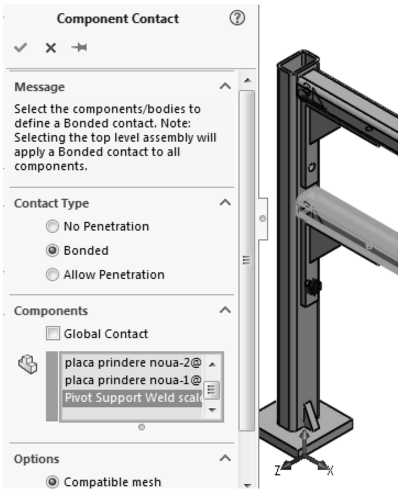


Fig. 9 Component Contact.

For Contact Sets one can apply multiple contact conditions using No Penetration option. This contact type prevents the interference between two entities but allows the formation of empty space.

The contact areas between the two support brackets and the main body have been selected manually. For this, initially, the visibility of the main body has changed. Figure 10 presents the Contact Sets window used in this case. The support is firmly fixed to the ground and in this case Fixed Geometry was used. Fixed Geometry means that the translation and rotation have been entirely restricted. The loaded forces, using External Loads, can be load in the next step. All components accelerate at the same speed under the effect of gravity, regardless of their mass. In the present case, for a single component, the model with its own gravitational force is loaded, consequence of the gravitational field, force perpendicular to Normal Plane.

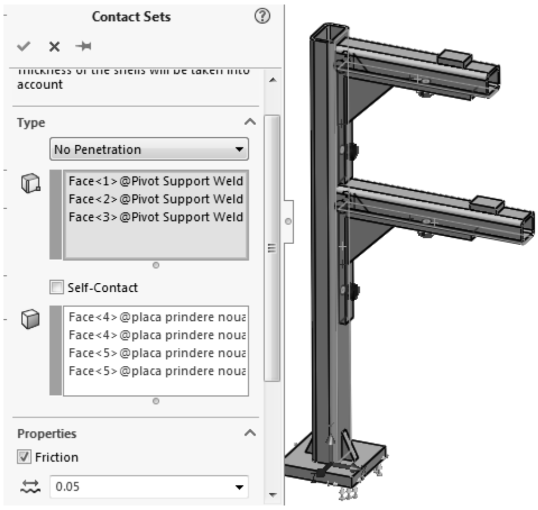


Fig. 10 Contact Sets.

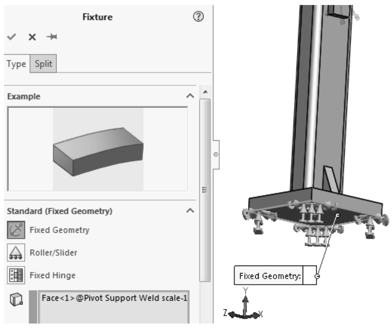


Fig. 11 Fixed Geometry.

Both arms will support loaded pipes. It is appreciated that these forces could be 7000N and 9000N (Fig. 12).

4. MESH

Mesh generates automatically the mesh network. Very important is the materiel of support, the forces and the restraints.

Due to the shape of the support, it is proposed to use Mesh Control. Mesh discretion can be changed in Standard Mesh with Automatic transition active. Automatic transition allows a special mesh in rounded areas or crossing from one wall to another.

Figure 13 presents the Mesh Control window with the Mesh parameters and the meshed support. Every change of the study support, requires a new mesh of the model.

5. RUNNING THE STUDY

The post-processing step, Run action, starts the solver. The Run function generates the FEA results. It results an expressive colour map for understanding the existing deformations. Von Mises, equivalent stresses**,** is a scalar quantity obtained from the principal stress values and it is used to measure the state of stress (Fig. 14) [6]..

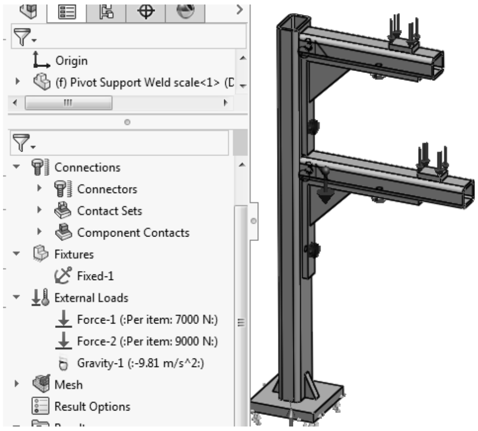


Fig. 12 External Loads.

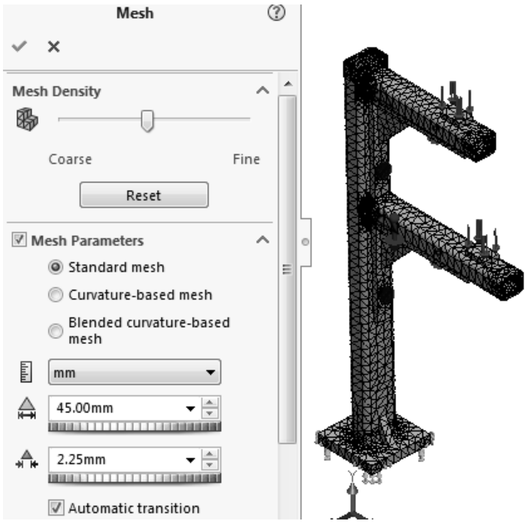


Fig. 13 Mesh.

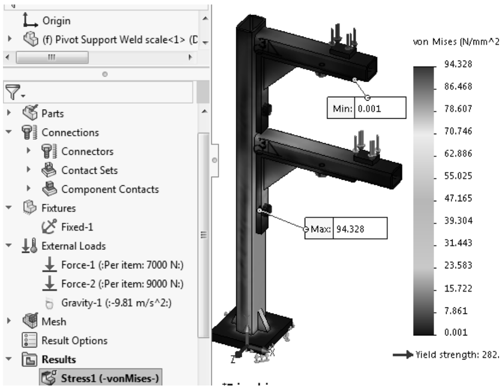


Fig. 14 Von Mises.

Generally the von Mises effective stress must compared to the material yield stress for metals. The Von Mises results are shown in N/mm^2 (94.3 N/mm^2). The maximum stress is noticeable in the longer arm area, namely at the bottom of the corner, in the screw area.

Displacement measures the maximum deflection. The simulation calculates also the displacements like figure below. The results are shown in mm (6.7 mm). The maximum displacement is noticeable at the end of the shorter arm.

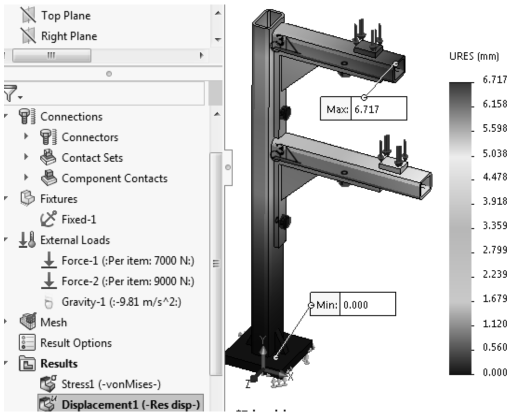


Fig. 15 Displacements.

6. CONCLUSION

The support shape assumed several transformations to the first form. The modular structure is less resistant. For this reason the presented structure was preferred. This structure can be used also for overloaded pipes.

The maximum stress is 94.3 N/mm^2, it is under the admissible tension of the material (90-120 N / mm^2).

For reducing von Mises tensions and displacements, the design of the support has had many transformations, the support brackets were shortened and the corners were also extended.

REFERENCES

[1] Scheibner, E. (2010) *Dimensioning of heating networks pipes*, Review of the Air Force Academy Brasov, Ed. Technical Sciences and Applied Mathematics, pp. 51-55.

[2] Urdea, M. (2012) *CAD software package, for designing and assembling heating pipes*, Proceedings of The 16th International Conference Modern Technologies, Ed. ModTech, pp. 1005-1009, ISSN 2069-6736, Sinaia, Romania.

[3] Pipes Supports, available at: *https://www.enggcyclopedia.com/2012/06/pipesupports/ Accessed:* 2018-10-02.

[4] Catalog HILTI (2010). *Technical information. Halfen framing channels and accessories*. Halfen-Deha.

[5] SolidWorks Help (2017): available at: *http://help.solidworks.com/2017/english/solidworks/cworks/idc_help_rigidconnection.htm, Accessed:* 2018-08-02.

[6] Urdea, M. (2017) *Studies regarding the processing of small thickness containers*, COSME'16, Transilvania University of Brașov. Pub MATEC Web of Conferences 9, 01013.

Author:

Associate Prof. Mihaela URDEA, Transilvania University of Brasov, Department of Manufacturing Engineering, Romania. E-mail: urdeam@unitbv.ro