BULETINUL Universităţii Petrol – Gaze din Ploieşti	Vol. LXVII No. 1/2015	20 – 24	Seria Tehnică

The Use of 2D Models in Relation to 3D Models in Numerical Analysis

Alexandru Pupăzescu

Universitatea Petrol-Gaze din Ploieşti, Bd. Bucureşti 39, Ploieşti
e-mail: pal@upg-ploieşti.ro

Abstract

In this paper it is presented the design based on the numerical analysis of the main elements of the auxiliary device for pre-stressing strands of wire ropes used to bridges trough. The analysis was carried out on 3D models-in which we used tetrahedral finite elements and on 2D models- for which we have used elements SHELL. The analysis has singled out both concordances and the differences between the two models.

Key words: *model, numerical analysis, FEM.*

Introduction

With the development of strong numeric processors the use of the finite element method (FEM) has known (fewer and fewer limitations) me few limitations. This fact, in conjunction with CAD design has led to the addition of numerical analysis programs in the design software. In this way the designer can assess rapidly the state of stresses and deformations in the elements designed with direct consequences upon their design.

In this paper it is presented the design based on the numerical analysis of the main elements of the auxiliary device for pre-stressing strands of wire ropes used to bridges trough. The analysis was carried out on 3D models-in which we used tetrahedral finite elements and on 2D models- for which we have used elements SHELL.

The bridge troughs of the railway bridge are made from reinforced concrete pre-stressed strands. In this way they can take larger loads. The technological process of bridge troughs involves the pre-stressing of wire ropes to significant axial loads that must be maintained throughout the period of drying cement. Such a device is shown in Figure 1. Most requested items are: blocking gears (*1* and *2*) and the tensile anchor (3) through which will be make the pre-stressing. The two devices are shown in Figures 2 and 3.

The 3D models of two devices were created with SOLIDWORKS and meshed using 3D tetrahedral finite elements. The two model were load with maximum force of 300 tf and they had the constrains presented in Figures 2 and 3.

After the finite elements analysis has been established for (fig. 4):
- for blocking gear the maximum stress is σ_{max} = 370.5 MPa;
- for tensile anchor the maximum stress is σ_{max} = 404.6 MPa.

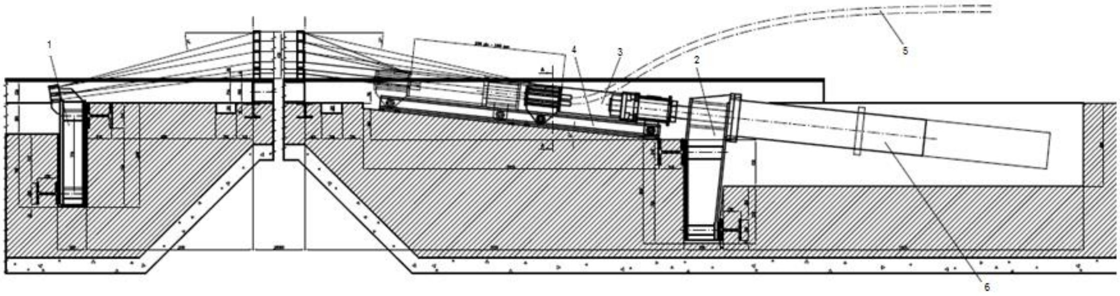

Fig. 1. Pre-stressing wire rope device:
1, *2* - Blocking gear; *3* – tensile anchor; *4* – longitudinal slide; *5* – rope strand; *6* - hydraulic cylinder.

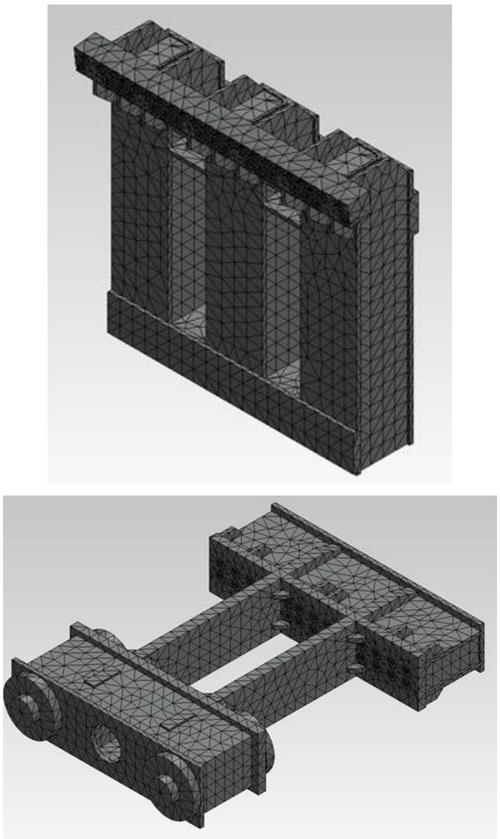

Fig. 2. The meshed models.

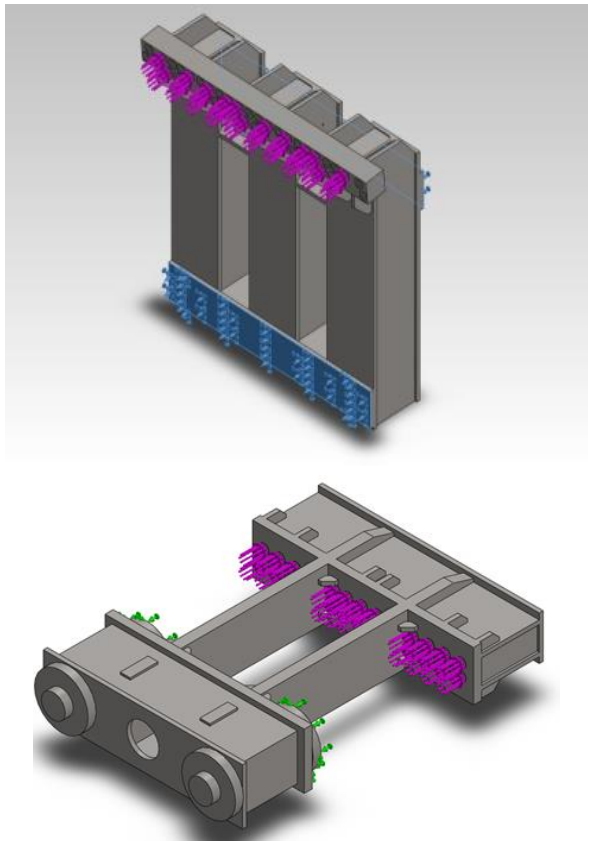

Fig. 3. The loads and the constraints.

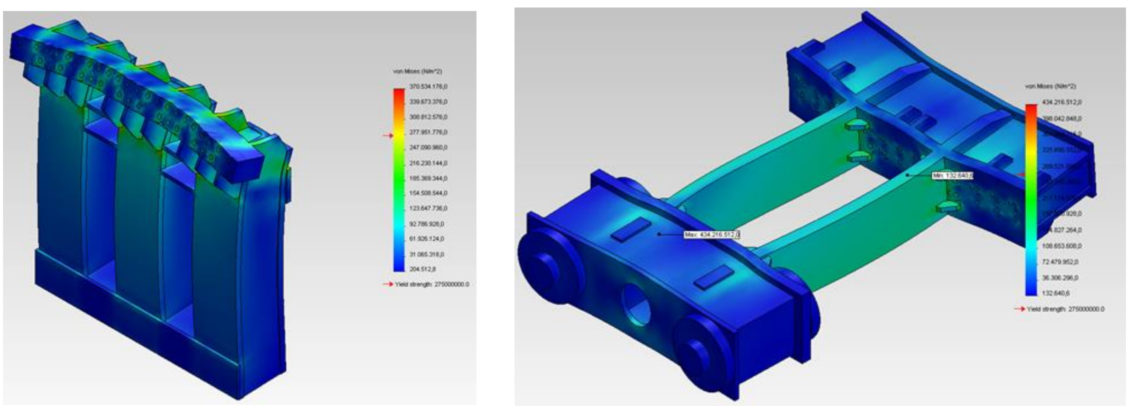

Fig. 4. The von Mises stresses distibution

These values are higher than the yield strength of the steel S355JO, $R_{p0,2}$ = 355 MPa from which these parts are made. In conclusion, it can be assumed that in the areas where it develops these tensions will produce plastic deformations.

Numerical Analysis with ANSYS

The same devices were modeled and analyzed with ANSYS program. Thus, on the basis of the overall design and execution of parts were chosen for their modeling, finite element as a basic element of type shell, SHELL43. For the blocking gear (fig. 5) was used and the finished item SOLID187.

Characteristic curve of conventional modeling of the material was based on the hypothesis that it deforms the plastic with nonlinear-hardening.

In Figure 5 are presented the models performed in ANSYS, and in Figure 6 meshed model.

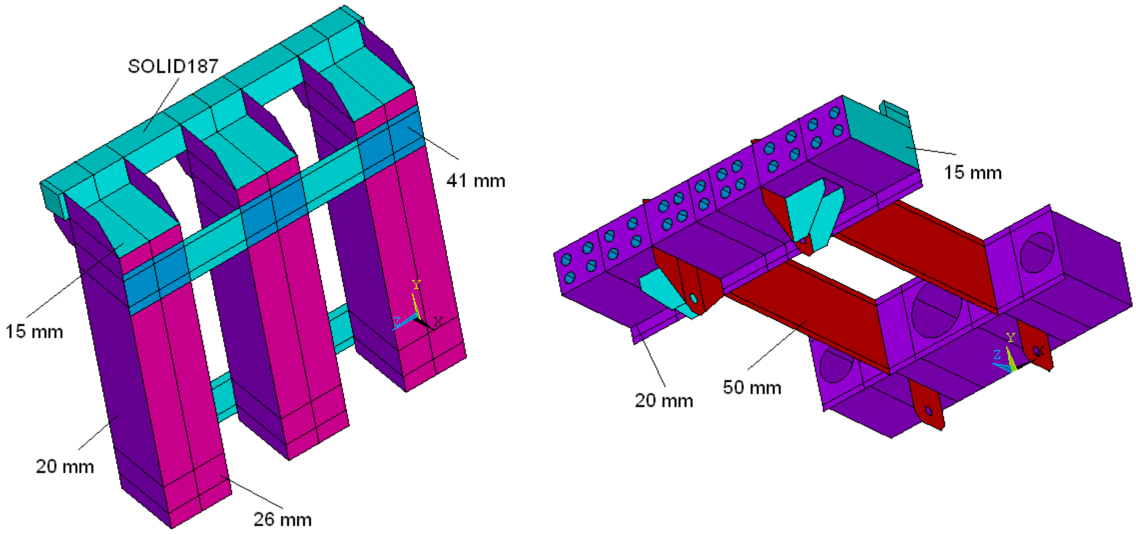

Fig. 5. ANSYS models

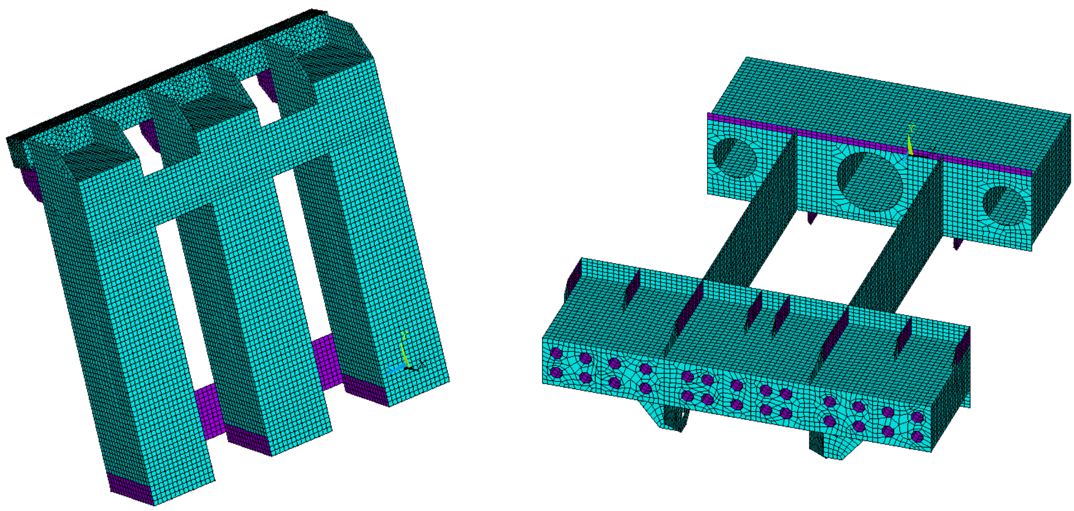

Fig. 6. The ANSYS meshed models

These models have been requested at loads and constrains as in the previous case. In the analysis were obtained distribution voltages, shown in Figure 8.

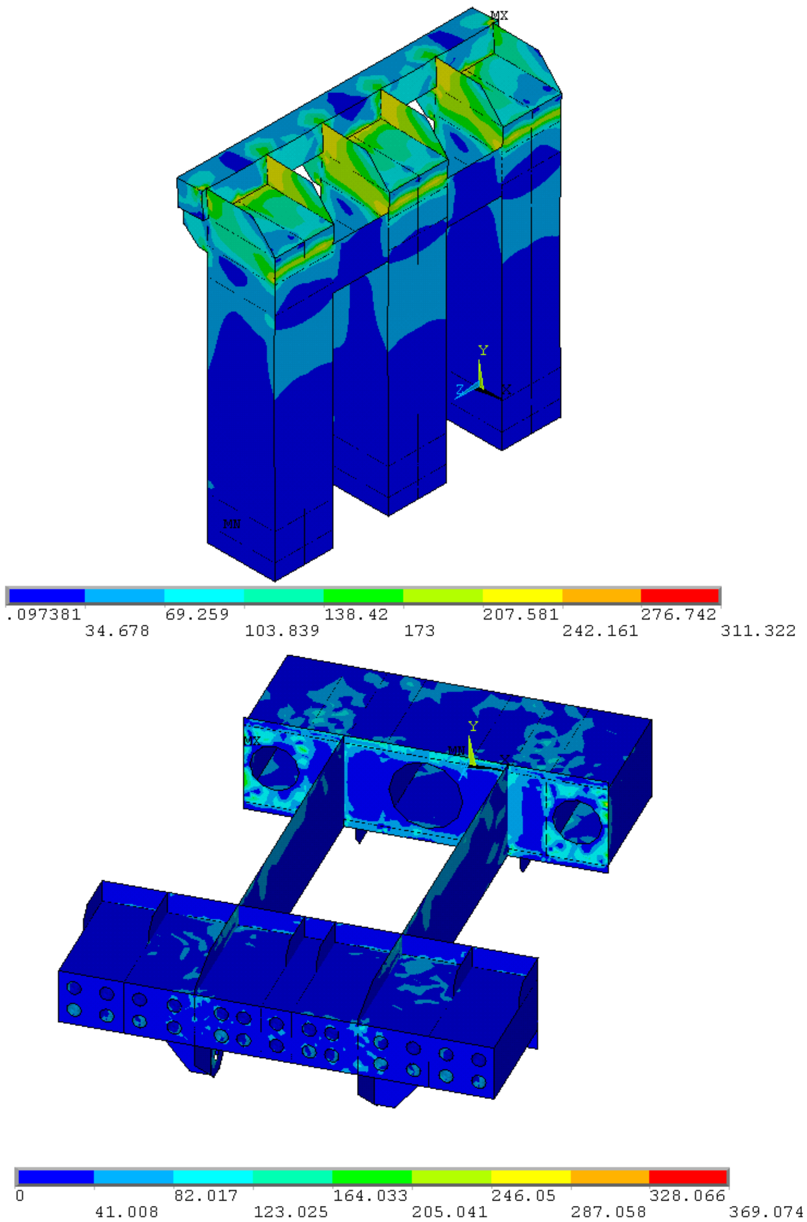

Fig. 8. Von Mises stresses distribution for ANSYS models

After the finite elements analysis has been established for (fig. 9):

- for blocking gear the maximum stress is σ_{max} = 311.3 MPa;
- for tensile anchor the maximum stress is σ_{max} = 369.1 MPa.

A first conclusion: the maximum stresses corresponding to the 2D model are significantly lower as maximum stresses corresponding to the 3D model.

Conclusions

It was on a numerical analysis of blocking gear and tensile anchor of the auxiliary device for pre-stressing strands of wire ropes used to bridges trough. We used two models: one 3D, the other 3D.

Both models have been loaded by a force F = 300 tf implemented equivalent pressures and have had the same constrains.

Calculation of stresses and deformations were made with a numerical analysis based FEM: the analysis of 3D model using Solid Works Simulation, the 2D model using ANSYS. The type of analysis performed was the nonlinear analysis in that nonlinear material behavior and the perfect structure geometry was.

After the finite elements analysis has been established for (fig. 9):
- for blocking gear the maximum stress was:
  - σ_{max} = 311.3 MPa for 3D model;
  - σ_{max} = 370.5 MPa for 2D model;
- for tensile anchor the maximum stress was:
  - σ_{max} = 369.1 MPa for 3D model;
  - σ_{max} = 404.6 MPa for 2D model.

A first conclusion: for 3D model the maximum stresses values are higher than the yield strength of the steel S355JO, $R_{p0,2}$ = 355 MPa from which these parts are made. In conclusion, it can be assumed that in the areas where it develops these tensions will produce plastic deformations.

The second conclusion: the maximum stresses corresponding to the 2D model are significantly lower as maximum stresses corresponding to the 3D model. In the case of tensile anchor the maximum stresses are little more than the yield strength.

In conclusion when making a numerical finite element analysis is very important to properly shape the structure according to the stresses that develop in it.

References

1. * * * – SR EN 10025-2:2004, *Produse laminate la cald din oţeluri de construcţii*. Partea 2: *Condiţii tehnice de livrare pentru oţeluri de construcţii nealiate*.
2. Pupăzescu, Al. – *Mecanică teoretică şi rezistenţa materialelor*, Vol. II, Editura Universităţii Petrol-Gaze din Ploieşti, 2007.
3. Olariu, V., Brătianu, C. – *Modelare numerică cu elemente finite*, Ed. Tehnică, Bucureşti, 1986.

Utilizarea modelelor 2D în raport cu modelele 3D în analiza numerică

Rezumat

Odată cu dezvoltarea puternică a procesoarelor numerice utilizarea metodei elementelor finite (MEF) a cunoscut din ce în ce mi puţine limitări. Acest fapt coroborat cu posibilitatea proiectării asistate CAD a condus la înglobarea de programe de analiză numerică în software de proiectare. În acest fel proiectantul are posibilitatea evaluării rapide a stării de tensiuni şi deformaţii în elementele proiectate cu consecinţe directe asupra proiectării acestora.

În lucrare este prezentată proiectarea pe bază de analiză numerică a principalelor elemente din cadrul dispozitivului de pretensionat toroane folosite la chesoanele din beton armat ale podurilor rutiere. Analiza efectuată atât pe modele 3D – la care s-au utilizat elemente finite tetraedrice – cât şi pe modele 2D – la care s-au folosit elemente 2D (SHELL), a scos în evidenţă atât concordanţele cât şi diferenţele dintre cele două modele.