STRUCTURE ANALYSIS OF THE FLANGING PRESS FRAME TYPE PIT 120 BY THE METHOD OF FINITE ELEMENTS

GABRIELA DINU, COSTICĂ ATANASIU
ŞTEFAN SOROHAN

The Strength of Materials Department, The Politechnic University in Bucharest

Abstract: This work contains a study of the induced stress and strain state, in static running, for the frame of a flanging press type PIT 120, using the modelling and analysis with finite elements.For estimating the strength of the frame, this one has been digitized in finite elements of a quadrangle plate kind (type SHELL) with four assemblage points.As results, there have been obtained the **shifts of the assemblage points** in which the plate finite elements interact and also, the **stress distributions** in these ones, for two loading cases. But, only one of these cases is brought up in this paper. The results are presented graphically, in a manner that facilitates their analysing and interpretation.

Keywords: press, frame, finite elements.

1.THE MODELLING OF THE FRAME STRUCTURE BY FINITE ELEMENTS

The strength structure of the frame is composed of steel walls of a plate kind, so, by modelling, the finite element type SHELL can be used.

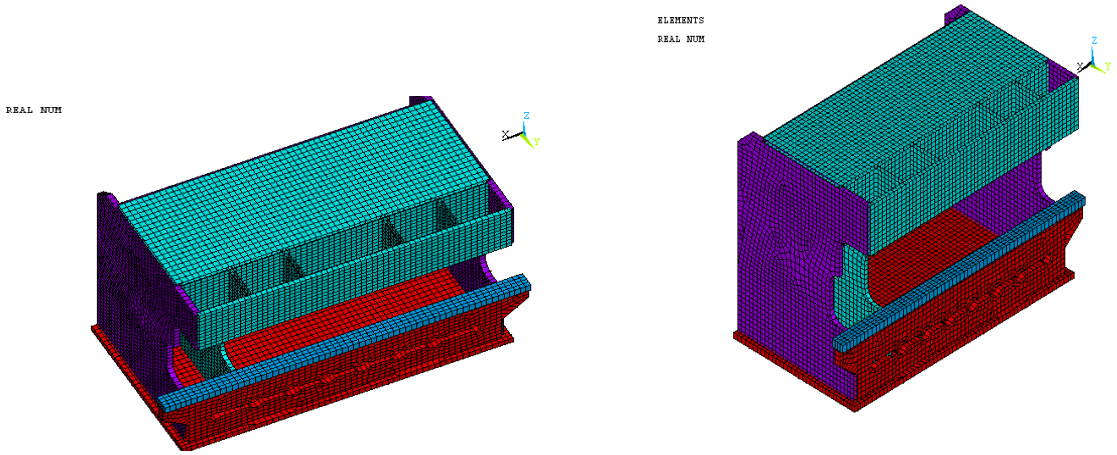
The physical - mechanical characteristics of this material are shown in the table no.1, using the following notation: E - longitudinal module of elasticity, ν - transverse contraction coefficient, $R_{p\ 0,2}$ – conventional ultimate strength, R_m – tensile strength.

Table 1

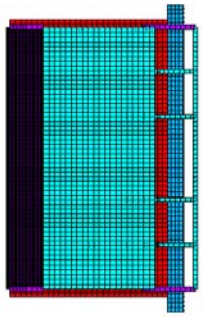
Material	*E* [MPa]	*v*	$R_{p0,2}$ [MPa]	R_m [MPa]
OL 37	210000	0,3	360...440	210...240

The determinations were performed by the general programme of analysis using finite elements, ANSYS.
The complete model of the structure shown in figure 1, a, b, c, d, e, f, presents the digitizing in finite elements of a quadrangle plate (type SHELL) with four assemblage points.

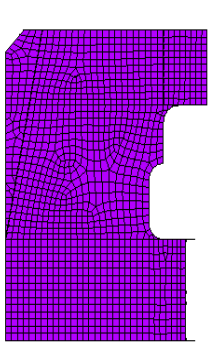
The structure was digitized with 14.595 elements and 14.444 assemblage points. For defining the geometry there were used 125 surfaces (Fig.2, a, b, c, d), 319 lines and 197 key points.



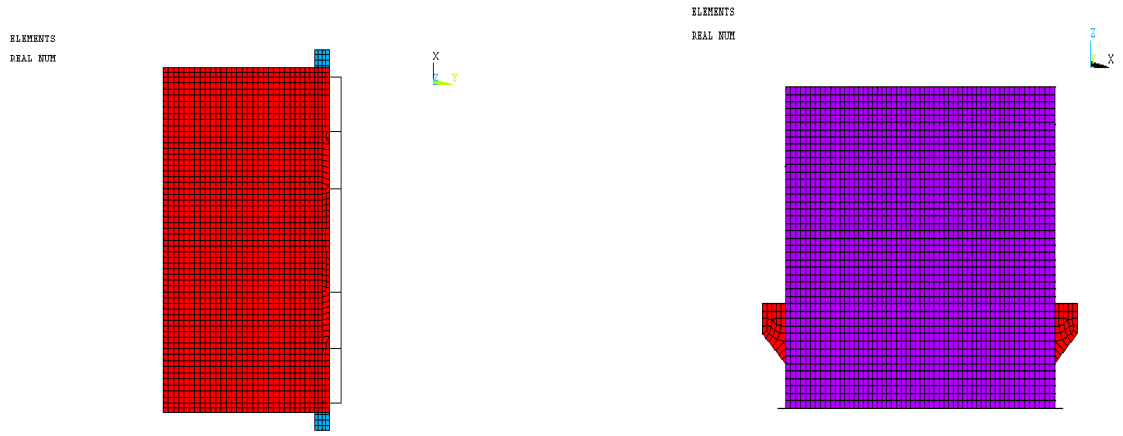
a. general view



b. top view

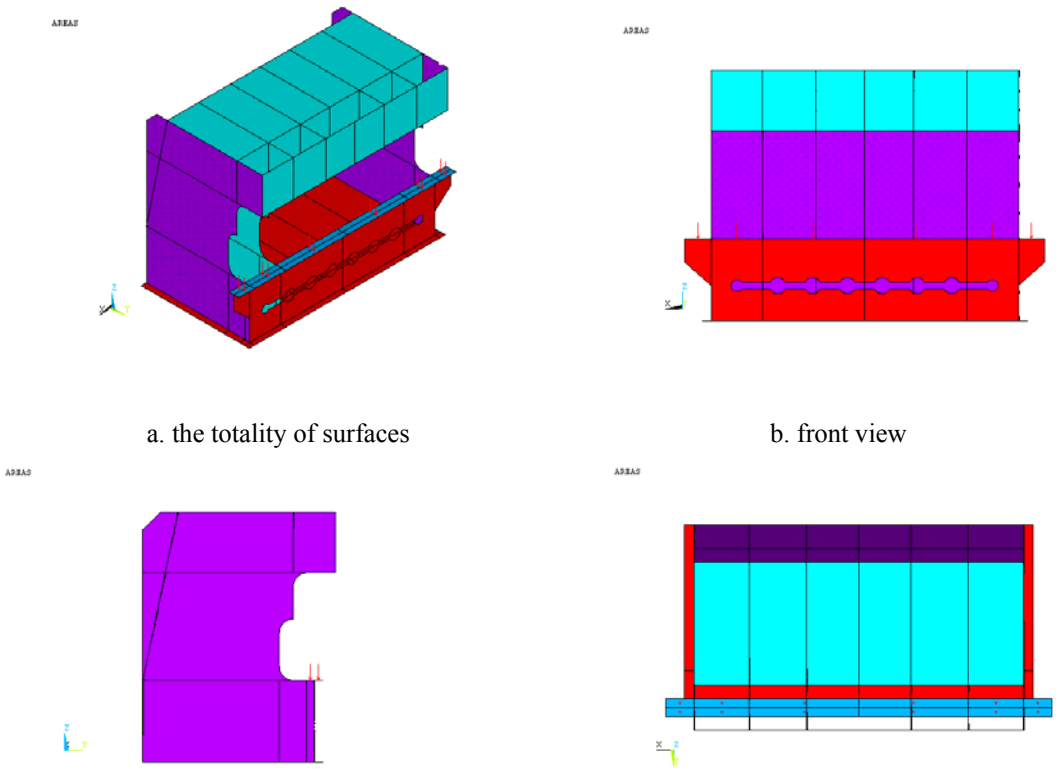


c. side view



d. base view e. back view

Fig.1. The digitizing of the frame



a. the totality of surfaces b. front view

c. side view d. top view

Fig. 2. The digitizing by surfaces

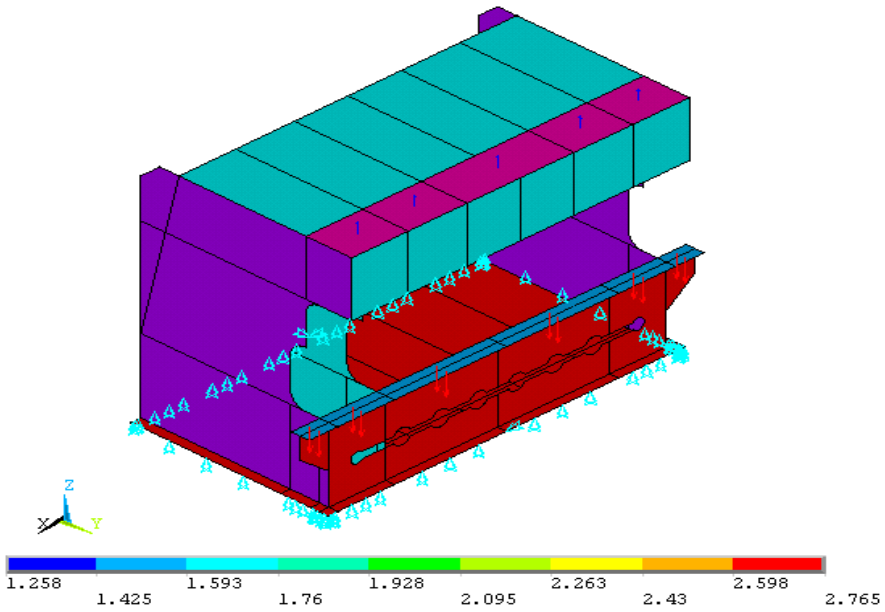


Fig. 3. The stress and suspension of the frame

Further on, there is presented the modelling of strains and suspensions for another case of stress which was also studied. The charging is made with a pressure of p = 2,765 N/mm^2 distributed uniformly on the platen surface (on a rectangular surface 3100x140mm^2). In the upper area, corresponding to the rabbet of the 6 hydraulic cylinders, there were inserted surfaces of a diaphragm kind (very slim), on which there was applied a pressure of p =1,258 N/mm^2, according to figure no.3. The resultant of these pressures corresponds to a force of 1200 kN. It is to be mentioned that, after applying the stresses on the surface of a giaphragm kind, this force was removed.

For a fast examination of the analyses results, these ones are presented graphically. There are figured the full shifts (USUM) and the equivalent stresses von Mises (SEQV) within the structure digitized with quadrangle finite elements type SHELL. The values of the shifts are given in mm,and the ones of the stresses in MPa .

2.THE RESULTS OF THE STATIC ANALYSES

The range of full shifts (USUM) is shown in figure no.4, these motions being chosen for comparison. There are shifts on the direction of Ox axix (Fig.4.1), shifts on the direction of Oy axix (Fig.4.2) and shifts on the direction of Oz axix (Fig.4.3).

The variation of the equivalent stresses is observed in figure no. 5, with represesntative values and data on the medial surface of the plate's element. The equivalent stresses on the upper surface of the plate's element are shown in Fig. 5.1, and the equivqlent stresses on the lower surface of the plate's element are to be seen in Fig.5.2. The maximum principal stresses at the center of the element are shown in Fig.5.3.

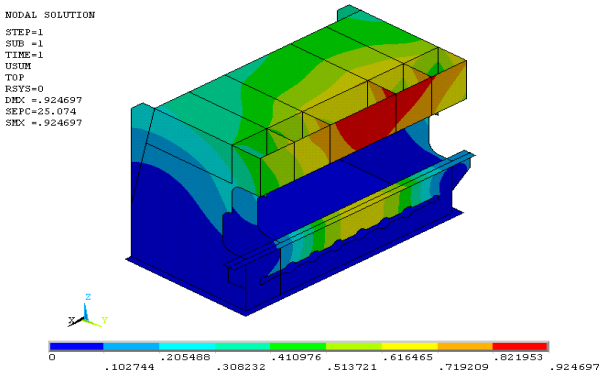


Fig. 4. The distribution of the ductile full shifs' field

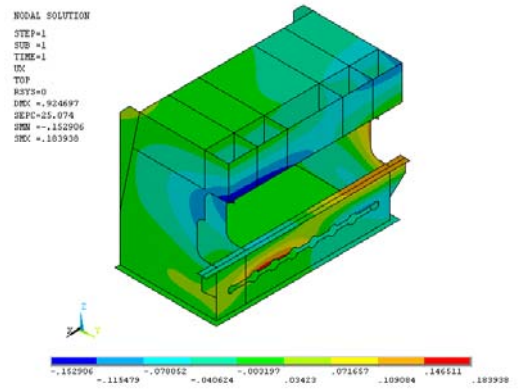


Fig. 4.1. The distribution of the ductile shifts' field on the direction of Ox axix

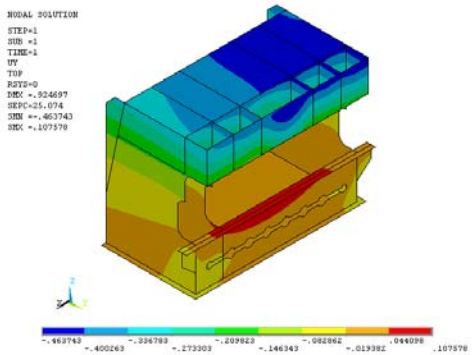


Fig.4.2. The distribution of the ductile shifts' field on the direction of Oy axix

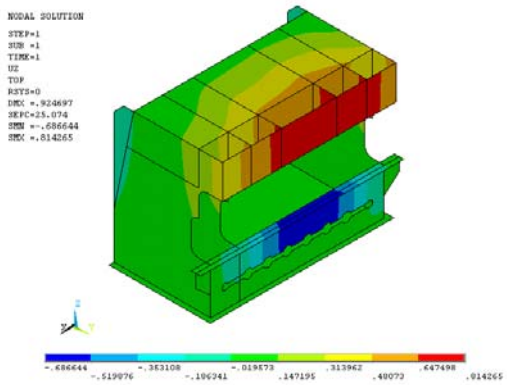


Fig.4.3. The distribution of the ductile shifts' field on the direction of Oz axix

3. CONCLUSIONS

Based on the results presented above there were drawn up the tables 2 and 3, which show that the values of the equivalent stresses within the plates are admissible, being below the value of the ultimate strengths of the material, which means that this frame can stand the stresses taken into account.

Table 2

Maximum Equivalent Stresses in the plates [MPa]			Figures
On the upper surface	On the midle surface	On the lower surface	
179,049	150,3	163,182	5, 5.1, 5.2

Table 3

Shifts [mm]				Figures
Maximum full shift	Dx	Dy	Dz	
0,924697	0,1839	0,1075	0,8142	4., 4.1, 4.2, 4.3

The rigidity of the structure has a high level, the maximum ductile full shift being of 0,924 mm, value which is accepted for this kind of machine. The values of the equivalent stresses were compared, being considered as reference values. The stresses are distributed, approximately uniformly, on the stage piece of the frame, leading to a rised stability of the machine. The results obtained by FEM (Finite Elements Method) allowed the producer of the press to take some measures in order to additionally strengthen the sensitive areas, by supplementary ribs and walls reinforcements. During the performance tests there were made measurements of the press frame's strains, results which were close to those obtained by FEM. There were fluctuations of the values of not more than 20%, but they appeared after the strengthening modifications proposed. In the same time, it was pursued to reduce the weight of the machine, so, an economy of the material. In this way, the areas less stressed were modified, reducing the depth of the walls or ribs.

REFERENCES

[1] LEVINA, Z.M., *Rigiditatea organelor de masini*, Moscova, 1971
[2] Manual de utilizare a programului ANSYS