УДК 622.276.34

GROUND OF APPLYING FINITE-ELEMENT METHOD FOR MULTIFUNCTION BLOWOUT PREVENTER BODY DEFLECTED MODE DETERMENATION

Pushchaev S.N.,
Scientific adviser Makushkin D.O.
Siberian Federal University

Engineering evaluation aiming to estimate the stress-strain state was restricted during a long time period to strain calculations using various formulae. State stress calculation is a sufficiently easy task till the object of calculations doesn't have a complicated structure. The more elaborate structure is the more differences a computed model and a real design. Ultimately, the solution can have a low accuracy[1].

That compelled engineers to overspend time creating the prototype models for carrying out further research.

Thus, the creation of the blowout preventer equipment had several stages. As a first step the structural form and the principle of action were developed. Then under the technical drawings the prototype model was created. The stress-strain state was estimated by means of installed strain-gage transducers.

This method has an essential fault. Considerable alteration in the construction of the prototype model makes additional creation of the prototype model necessary.

A number of mathematic simulation methods were developed for analytical solution of the stress-strain state estimation tasks. One of these methods was finite-element method (FEM), also known as finite element analysis (FEA) [2].

FEM is a mathematic simulation method for finding the solutions of the partial differential equations (PDE) and the simultaneous equations as well as (less frequently) integral equations.

It is considered that various phenomena regarded under science and engineering are commonly described in terms of differential equations by using their continuum mechanical models.

Solution of the differential equations under various conditions such as boundary and initial conditions leads to the understanding of the phenomenon and can predict its further development. Though, the incisive solutions of the differential equations are generally difficult to obtain.

These numerical methods are adopted to obtain approximate solutions of the differential equations. Those among these numerical methods which approximate continua with the number of degrees of freedom by a discrete body with the finite degree of freedom are called "discrete analysis".

The strain analyses procedure by the FEM is summarized as follows:

- Procedure 1 *Discretization*

- Procedure 2 *Selection of the interpolation function*
- Procedure 3 *Derivation of the element stiffness matrix*
- Procedure 4 *Assembly of stiffness matrices into the global stiffness matrix*
- Procedure 5 *Rearrangement of the global stiffness matrix*
- Procedure 6 *Designation of unknown forces and displacements*
- Procedure 7 *Calculation of strains and stresses*

One of the most difficult problems of any numerical method is verification problem. The Solid Works Simulation static analysis has a high accuracy demonstrated in various situations [3].

Let's make verification test of a cantilever beam. The cantilever beam (Fig.1) is subjected to a concentrated load (F = 1.2 kN) at the free end. Determine the displacement at the free end ($E = 0.71 \cdot 10^{11}\ Pa$). Dimensions of the cantilever are: L = 0.4m, b = 0.02m, t = 0.008m.

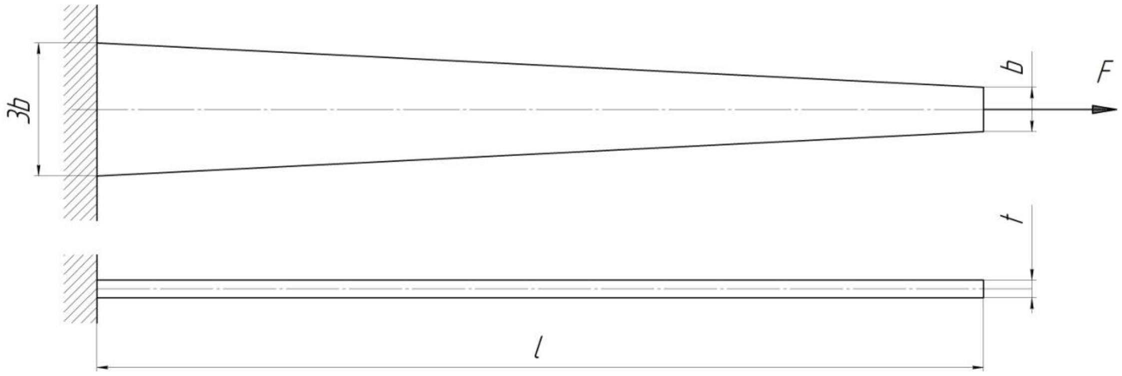

Fig.1 Cantilever beam

Theory:

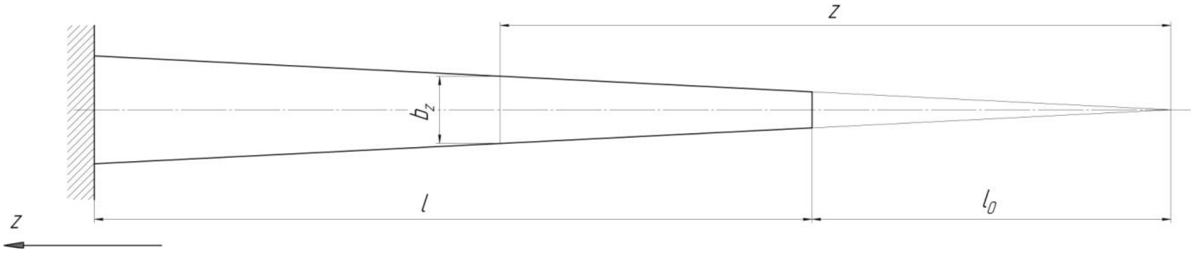

Fig.2 Cantilever beam model

$$\Delta l = \frac{F}{E}\int_l \frac{dz}{A}, \tag{1}$$

Where: A – cross section area.

$$\frac{3b}{l+l_0} = \frac{b}{l_0} \tag{2}$$

$$l_0 = 0.5l \tag{3}$$

From equation 2 and 3 summarized follows:

$$\frac{b_z}{z} = \frac{b}{l_0} \tag{4}$$

Thus area can be calculated by follows:

$$A = \frac{2 \cdot b_z \cdot t}{l} \tag{5}$$

The deflections at the free end can be calculated by follows:

$$\Delta l = \frac{F}{E}\int_{0.5 \cdot l}^{1.5 \cdot l} \frac{l \cdot dz}{2 \cdot b \cdot t \cdot z} = \frac{F \cdot l}{2 \cdot E \cdot b \cdot t} \cdot \ln 3 \tag{6}$$

$$\Delta l = 2.321 \cdot 10^{-5} m \tag{7}$$

SolidWorks Simulation:

1) Model creation

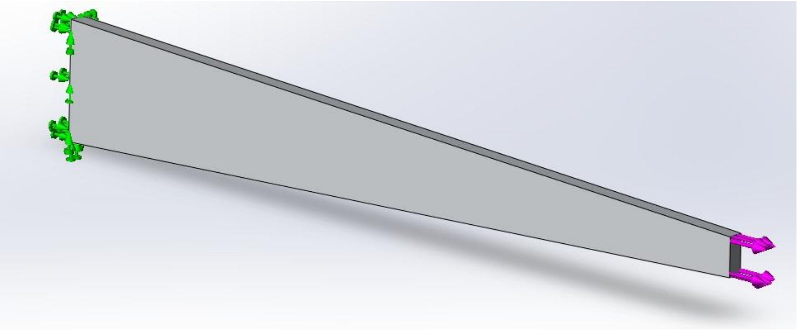

Fig.3 Cantilever beam model in SolidWorks Simulation

2) Mesh creation

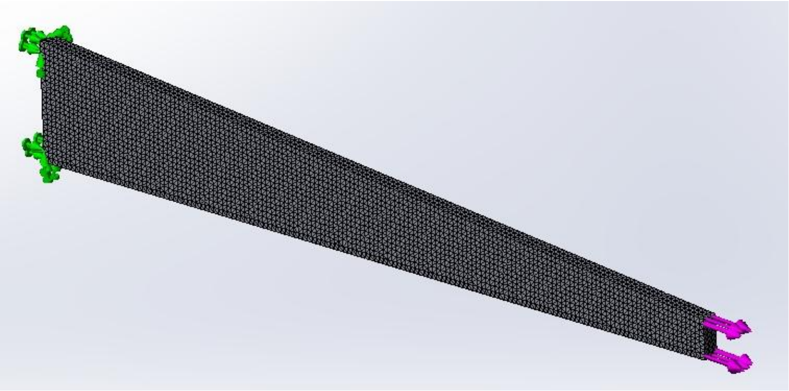

Fig. 4 Solid mesh for calculating model

Mesh information

Mesh type	Solid mesh
Element size	2.70964 mm
Number of elements	43 247

3) Result

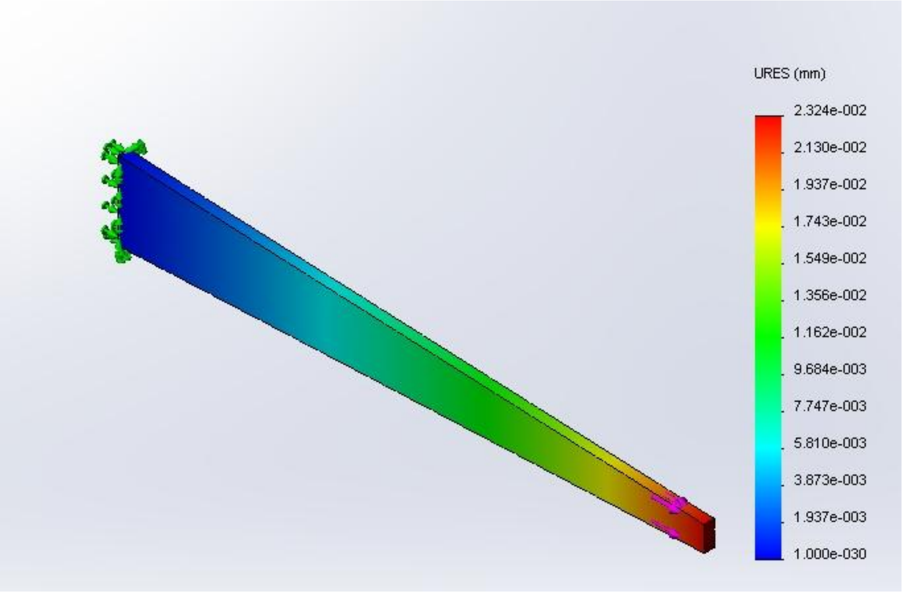

Fig. 5 Static displacement

As shown in Fig. 5 maximum displacement in model adopt a value $2.324 \cdot 10^{-5} m$. Results comparison demonstrate that Solid Works Simulation analysis have a high accuracy.

With reference to the real calculation model let's determine deflected mode for a multifunction blowout preventer (MFBOP) [4, 5] body (Fig.6). The MFBOP body is stressed under the pressure $70 \cdot 10^6 Pa$ in main value and $25 \cdot 10^6 Pa$ in head end.

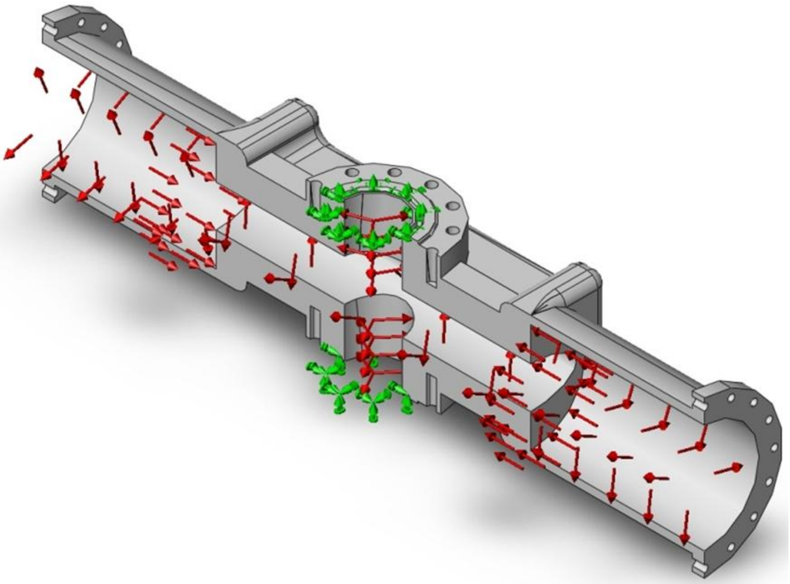

Fig.6 MFBOP body model

Mesh information

Mesh type	Solid Mesh
Maximum element size	131.274 mm
Minimum element size	26.2548 mm
Degrees of freedom	300 861
Number of elements	63 917

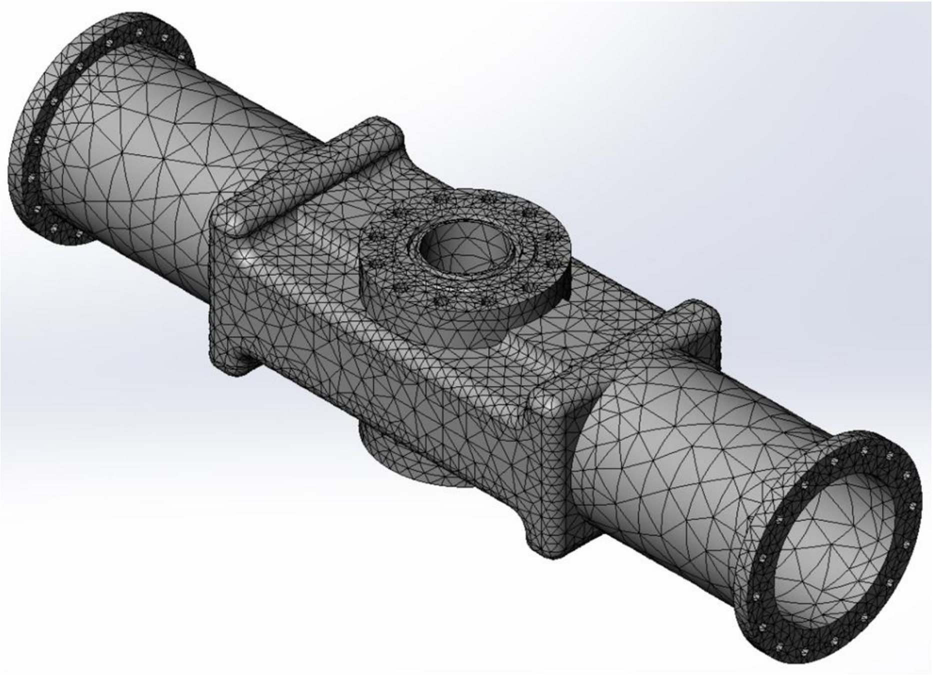

Fig.7 MFBOP body mesh

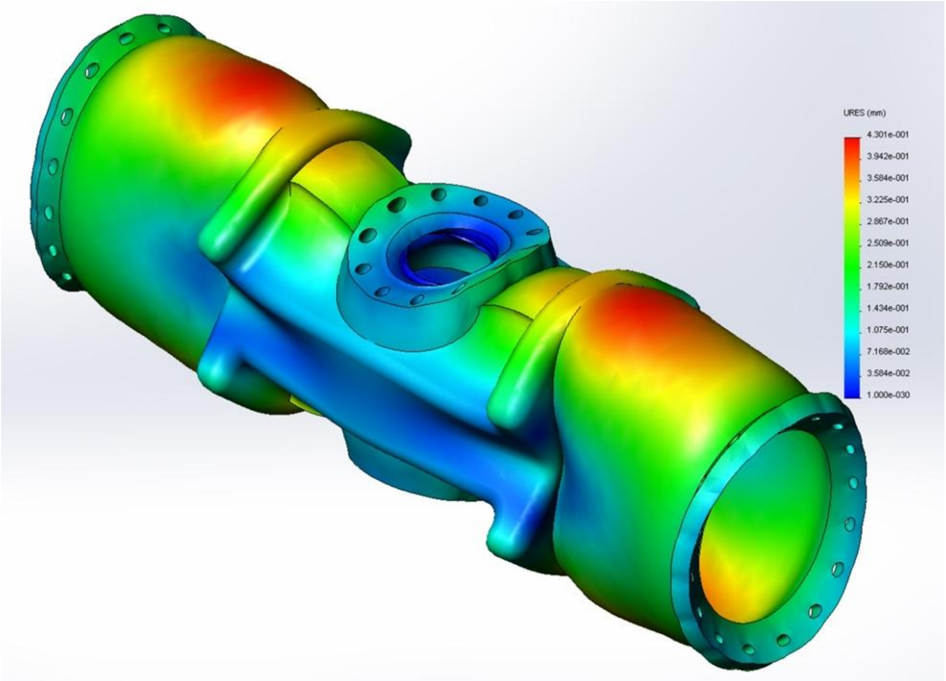

Fig.8 Result

The maximum displacement magnitude is 0.4 mm that is normal result for blowout preventer body. And we can assert that this design of MFBOP body have enough stiffness.

References

1) V. Kolcherin, I. Kolesnikov, I. Antonov. Blowout preventer equipment. Volgograd, 2009
2) Y. Nakasone, S Yoshimoto Engineering Analysis With ANSYS Software, Tokyo, Japan, 2006
3) SolidWorks Simulation Verification Problems and NAFEMS Benchmark Tutorials, Dassault Systems SolidWorks Corp., 2012
4) RF patent # 2274727 C1, MPK E 21 B 33/03, published in
5) S. Pushchaev, D. Makushkin. Multifunction Blowout Preventer. «Burenie & Neft» journal, Aug, 2010