Example of selecting an optimal finite element mesh for calculating machinebuilding structures

Ramil Nezhmetdinov[1*], *Ramilya* Nezhmetdinova[1], *Damir* Bautdinov[2], and *Vladimir* Bogdanov[2]

[1]MSUT «STANKIN», 1, Vadkovsky per., 127055, Moscow, Russia
[2]The State University of Management, 99, Ryazansky Prospekt, 109542, Moscow, Russia

Abstract. When solving various problems in the field of mechanical engineering, an engineer has to face problems that cannot be solved by exact methods due to significant mathematical difficulties. Equations or systems of equations, as a rule differential equation, which describe the problem have no exact solution. When solving such problems in deformable solid mechanics, approximate methods of calculation are widely used, one of which is the finite element method (FEM). When solving the same problem of deformable solid mechanics by the finite element method (FEM), using different finite element meshes (FEM) and types of finite elements, the solution results may differ sharply from each other, and in some cases these differences may reach 50%. The question arises - what finite element size should be adopted for further investigations so that the calculation results correspond to reality? The answer to this question is given by the solution of a test verification problem, the results of which should be used to select the dimensions and determine the type of finite element suitable for further calculation.

1 Introduction

Analysis of the stress state of machine-building structures and structural elements (sometimes of rather complicated geometry) is usually carried out with the help of FEM. As studies show, solving the same problem, using different finite element meshes and types of finite elements, can give results different from each other, and in some cases these differences can reach 50%.

The question arises: what size of FEs should be used for research to ensure that the results of calculations correspond to reality?

This paper presents an example of selecting an optimal finite element mesh to study the stress state of a symmetric plate with a circular hole, which is subject to compression in one of the directions. The optimal mesh, i.e., the type and size of finite elements and their number, was determined based on the solution of a test verification problem.

* Corresponding author: neramil@gmail.com

2 Materials and methods

As a test verification problem, consider the problem of determining the stress state of a plate with a circular hole. A plate with a circular hole is an elastic isotropic medium in a plane-stressed state, weakened by a circular hole of radius a and subjected to compression by a distributed vertical load of intensity q. Such a problem has an exact analytical solution presented by Kirsch (Kirsch problem) [5]. Fig.1 shows the computational scheme of the verification problem.

The verification problem is solved in the polar coordinate system. If necessary, one can always switch to the Cartesian coordinate system using the transition formulas.

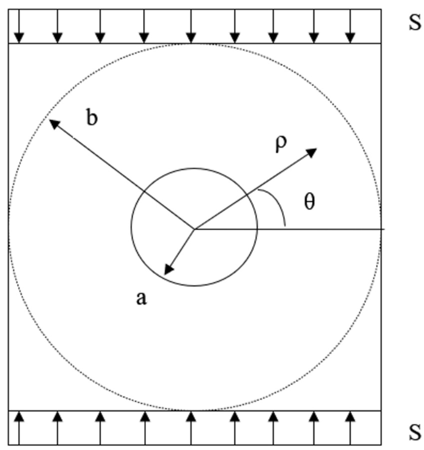

Fig. 1. Calculation scheme of the verification problem.

The biharmonic equation of the plane problem in the polar coordinate system is written in the form:

$$\left(\frac{\partial^2}{\partial r^2}+\frac{1}{r}\frac{\partial}{\partial r}+\frac{1}{r^2}\frac{\partial^2}{\partial \theta^2}\right)\cdot\left(\frac{\partial^2 \varphi}{\partial r^2}+\frac{1}{r}\frac{\partial \varphi}{\partial r}+\frac{1}{r^2}\frac{\partial^2 \varphi}{\partial \theta^2}\right)=0 \quad (1)$$

Let us take the stress function in the form:

$$\varphi = u(r)\cos 2\theta \quad (2)$$

where u(r) is the function to be defined.

Knowing the stress function, it is possible to determine the stresses at any point in the computational domain using the following relationships:

$$\sigma_r = -\left(\frac{1}{r}\frac{\partial \varphi}{\partial r}+\frac{1}{r^2}\frac{\partial^2 \varphi}{\partial \theta^2}\right) \quad (3)$$

$$\sigma_\theta = -\frac{\partial^2 \varphi}{\partial r^2} \quad (4)$$

where:

σr - stresses in the radial direction;

σθ - stresses in the circumferential direction (circumferential or tangential stresses).

Substituting the stress function (2) into the strain coexistence equation (1) and performing all necessary transformations, we obtain the following differential equation in ordinary derivatives to determine the function u(r):

$$u^{IV} + \frac{2}{r}u^{III} - \frac{9}{r^2}u^{II} + \frac{9}{r^3}u^{I} = 0\,. \tag{5}$$

This is an Euler equation with variable coefficients. Let r=ex (Euler substitution), respectively $dr = e^x dx$. After substitution, the Euler equation with variable coefficients is reduced to an equation with constant coefficients:

$$u^{IV} - 4u^{III} - 4u^{II} + 16u^{I} = 0 \tag{6}$$

The solution of the equation with constant coefficients (6) has the form:

$$u = C_1e^{0x} + C_2e^{2x} + C_3e^{4x} + C_4e^{-2x} \tag{7}$$

Making the inverse substitution r=ex, x=ln r, we find the general solution of equation (5) to determine u(r):

$$u(r) = C_1 + C_2r^2 + C_3r^4 + \frac{C_4}{r^2} \tag{8}$$

The stress function (2) will be written in the form:

$$\varphi = \left(C_1 + C_2r^2 + C_3r^4 + \frac{C_4}{r^2}\right)\cos 2\theta \tag{9}$$

The constants C1, C2, C3, C4 are determined from the condition that the stresses on the circle of radius b will be the same as in the plate without a hole, and from the condition that the edge of the hole is free from external forces, since we consider the plate to be unlimitedly large a/b=0:

$$C_1 = \frac{a^2}{2}S,\; C_2 = -\frac{s}{4},\; C_3 = 0, C_4 = -\frac{a^4}{4} \tag{10}$$

After determining C1, C2, C3, C4 , the expressions for the normal radial and tangential stresses are written as:

$$\sigma_r = -\frac{S}{2}\left(1 - \frac{a^2}{r^2}\right) - \frac{S}{2}(1 + \frac{3a^4}{r^4} - \frac{4a^2}{r^2})\cos 2\theta \tag{11}$$

$$\sigma_\theta = -\frac{S}{2}\left(1 + \frac{a^2}{r^2}\right) - \frac{S}{2}\left(1 + \frac{3a^4}{r^4}\right)\cos 2\theta \tag{12}$$

The stresses on the hole contour, at r = a, will be equal:

$$\sigma_r = 0$$

$$\sigma_\theta = -S - 2S\cos 2\theta \tag{13}$$

Maximum tangential stresses occur in the upper and lateral sections:
- at θ=π/2, 3π/2 (upper and lower cross-sections) - σθ=S;

- at θ= 0, π (side sections) - σθ= -3S.

For a unit compressive uniformly distributed load

S=1 the tangential stresses are:

-when θ = π/2, 3 π/2 - σmax+=1 (highest tensile stresses);

-when θ = 0, π - σmax- = -3. (greatest compressive stresses).

Based on the obtained results, the optimal finite element mesh was selected for the parametric analysis of the stress state of the plate with a circular hole in the ANSYS software package.

To select the optimal mesh, it is sufficient to consider not the entire computational domain, but the fourth part, due to symmetry in the horizontal and vertical directions, which will reduce the number of finite elements and, consequently, reduce the calculation time. The calculation scheme of the test verification problem is presented in Fig. 2.

To select the size and type of finite element, the computational domain was partitioned into meshes of uniform elements. Each finite element mesh was given a b-size finite element in fractions of the size of the computational domain H, which can also be considered of different sizes, and then densification of the mesh around the hole in the area bounded by r and r1, thus increasing the number of elements around the hole (n- mesh densification according to ANSYS) [1,2]. The finite element meshes with different side sizes and different types of elements are shown in Fig.3-6

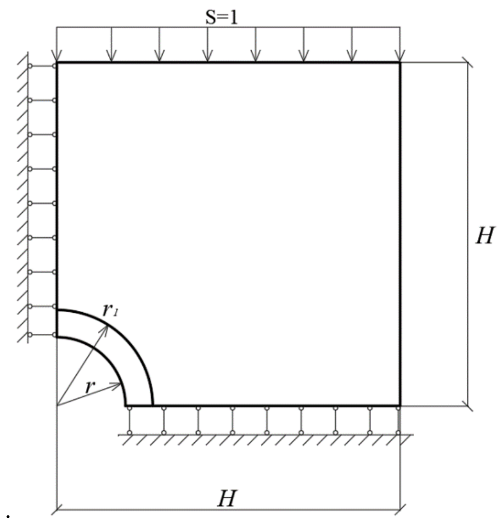

Fig. 2. Calculation scheme of the test verification problem.

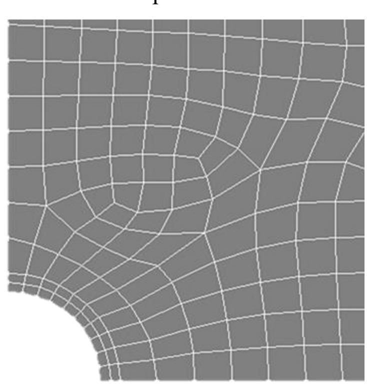

Fig. 3. Plane 42 finite element mesh, at n=1, b=0.4H.

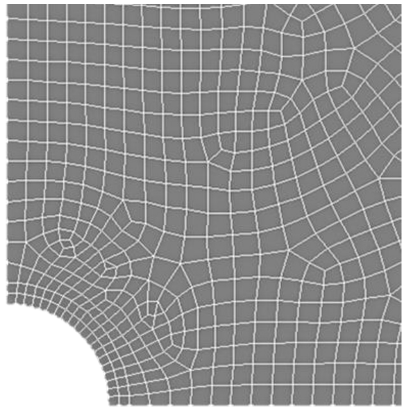

Fig. 4. Plane82 finite element mesh, at n=1, b=0,2H.

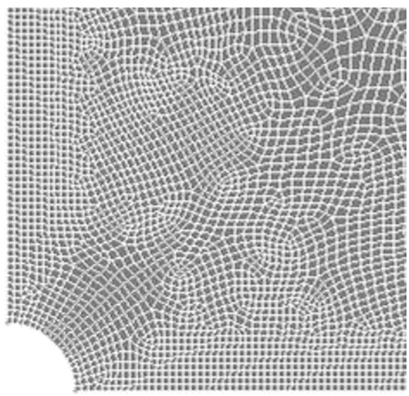

Fig. 5. Plane182 finite element mesh, at n=0, b=0,1H.

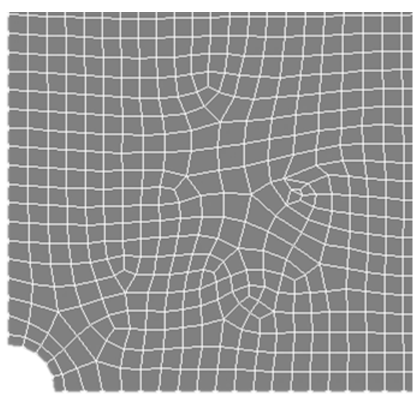

Fig. 6. Plane183 finite element mesh, at n=0, b=0.4H.

The results of solving the test problem in the program complex for different sizes and types of finite elements are shown in Table 1.

Table 1 shows that the values of maximum stresses closest to the analytical solution correspond to the mesh with elements of the following type and size:

-Element type: Plane 42;

-Size of the finite element: b=0,4H;

-Number of thickenings n according to ANSYS: 1;

-Dimension of the computational domain H=10, r=1, r1=1,2.

This optimal mesh can be adopted for further studies of the stress-strain state of plates with circular holes, which are subjected to tension-compression in their plane.

It should be noted that by increasing the number of finite elements and densifying the mesh around the hole, it is not always possible to improve the accuracy of the results, despite the fact that there is a widespread opinion that a more accurate result is obtained by dividing the computational domain into a larger number of finite elements [3,4].

Table 1. Results of solving the test problem in the program complex for different sizes and types of finite elements.

№	Dimensions: H. r. r_1	Type element	Finite element size	Number of densifications n	σ^+_{max}	σ^-_{max}
1	H=8 r=1 r_1=1.2	Plane 182	0.4	0	0.86	-2.74
2			0.4	1	0.974	-2.94
3			0.32	0	1.051	-3.038
4			0.25	0	0.88	-2.79
5			0.2	0	0.88	-2.805
6			0.15	0	0.9804	-2.952
7			0.1	0	0.9826	-2.958
8		8 node 183	0.4	0	1.119	-3.164
9			0.4	1	1.112	-3.145
10			0.25	0	1.079	-3.104
11			0.2	0	1.068	-3.08
12			0.1	0	1.095	-3.121
13		Plane42	0.32	0	0.989	-3.012
14			0.25	0	1.021	-3.043
15			0.25	1	1.074	-3.108
16			0.2	0	1.036	-3.06
17			0.1	0	1.089	-3.12
18	H=4.5 r=0.5 r_1=0.7	Plane182	0.3	0	0.896	-2.829
19			0.225	0	0.919	-2.874
20			0.225	1	1.026	-3.036
21			0.2	0	0.933	-2.889
22			0.2	1	1.029	-3.043
23			0.15	0	1.029	-3.036
24	H=10 r=1 r_1=1.2	Plane 82	0.25	0	1.04	-3.055
25			0.25	1	1.063	-3.081
26			0.2	0	1.029	-3.04
27			0.1	0	1.056	-3.071
28		Plane182	0.25	0	1.016	-3.017
29			0.2	0	1.022	-3.026
30		Plane42	0.4	0	0.9032	-2.93
31			0.4	1	0.999759	-3.02
32			0.25	0	0.981	-3
33			0.2	0	0.998784	-3.012
34			0.1	0	1.05	-3.072

3 Conclusions

As shown by the conducted investigations, the values of maximum stresses closest to the analytical solution of the verification problem correspond to the grid with elements of the following type and size - Plane 42, finite element size - b=0.4H, number of densifications n according to ANSYS - 1, dimensions of the computational domain H=10, r=1, r1=1,2. This optimal mesh can be adopted for further studies of the stress-strain state of plates with circular holes, which are subjected to tensile compression in their plane.

An increase in the number of finite elements and densification of the mesh around the hole does not always lead to an increase in the accuracy of the results, despite the fact that there is a widespread opinion that more accurate results are obtained when the computational domain is divided into a large number of elements [3,4]. The above studies show that the choice of the finite element type, its size and, as a consequence, the finite element mesh, is the most important and responsible moment in the application of software systems in engineering calculations, which determines the accuracy and correctness of both the obtained and subsequent results.

Acknowledgment

This work was supported by the Ministry of Science and Higher Education of the Russian Federation under project FZNW-2024-0013.

References

1. A.B. Kaplun, E.M. Morozov, M.A. Olferyeva, *ANSYS in the engineer's pukki: Practical Guide* (Unitorial Urss, Moscow, 2003)
2. A.V. Chigarev, A.S. Kravchuk, A.F. Smalyuk, *ANSYS for Engineers: Reference Manual* (Mashinostroenie-1, Moscow, 2004)
3. K. Bate, E. Wilson, *Numerical Methods of Analysis and Finite Element Method* (Stroyizdat, Moscow, 1982)
4. R. Gallagher, *Finite Element Method. Fundamentals* (Ir, M., 1984)
5. S.P. Timoshenko, J. Goodyear, *Theory of Elasticity* (Main Editorial Office of Physical and Mathematical Literature, Nauka Publishing House, 1975)