Simulation of Stresses in A Drill Using the Ansys Program

Urinov Nasillo Fayzilloevich	Candidate Of Technical Sciences, Associated Professor
Saidova Mukhabbat Khamroevna	Senior Teacher
Ergashova Surayyo Raxmonovna	Undergraduate Bukhara Engineering-Technological Institute

ABSTRACT

The article presents a comparison of the maximum stresses that occur in the drill under the influence of the drilling torque, with the allowable stress, as well as a comparison of the maximum stress that occurs during the manufacture of the drill under the action of the cutting force, in the grinding operation.

Keywords: grinding operation, cutting force, finite element mesh, drilling, torque, simulation, stress, drill, ANSYS.

ANSYS is a finite element analysis program that has been dynamically developed and updated for over 35 years. This program is widely used among specialists involved in computer-aided calculations in the field of engineering (CAD), finite element calculations in various areas of theoretical mechanics, mechanics of deformation of solids, mechanics of various structures (sopromat) [1]. The program also solves a number of problems in the field of mechanics liquid and gas, heat transfer, heat transfer, electrodynamics, etc.

Also, the ANSYS program is applicable to quite promising areas of business, for example, 3D printing, as it allows you to create tools that make it possible to visualize 3D printing. The program allows for the design of 3D printing parts, consisting of a variety of materials, for example, SLM laser printing, which consists of a large number of fine metal powders. Now there is practically no engineering direction in which there would be no solution in the ANSYS program. It is used both for the defense industry and for aerospace facilities. And also in the field of microelectronics, medicine, and even in the field of simulators designed for software testing.

In many industries, pre-modeling and analysis helps companies avoid expensive and time-consuming investments by eliminating full development, which consists of design, manufacture and testing. The geometry of the system is based on the Parasolid core, and the finite element analysis system was developed directly by the American company ANSYS.

The ANSYS system is also good because the modeling and analysis tools used in it can be combined with other CAD software packages, for example, CATIA, SolidEdge, NX, SolidWorks and some others.

The program, which first appeared 35 years ago, has practically nothing to do with the modern version. It allowed solving only thermophysical problems and problems of linear strength. Such a program worked only in conjunction with electrical computers (computers).

In the early 1970s, completely new computing technologies were introduced into the system, and many changes were made at the request of users. The most basic addition is the appearance of non-linearities in various variations. It also became possible to use the subconstruction method. Finally, the developers have extended the finite element library. A lot of efforts were spent by developers on the invention of software developments intended for personal computers and vector graphic terminals that began to appear at that time.

In the late 1970s, the main achievement and update was the ability to work interactively in the ANSYS program. This approach made it possible to simplify the process of creating a model by the finite element method, and also simplified the evaluation of the results. It also became possible to apply an interactive graph in order to refine the geometry of the model, the properties of a given material, as well as the given boundary conditions before starting the calculation. This made it possible to display graphical information to control the results of the solution.

Simulation of stresses in a drill when a drilling torque is applied to it.

The goal is to compare the maximum stresses that occur in the drill under the influence of the drilling torque with the allowable torsional stress.

To do this, it is necessary to determine the maximum stress that occurs in the drill when the drilling torque is applied to it using the ANSYS program, and compare it with the allowable torsional stress.

1. Permissible torsion stress of the drill will take [τ]=250 MPa [3].

2. Import the model geometry from the Compass program (Fig. 1):

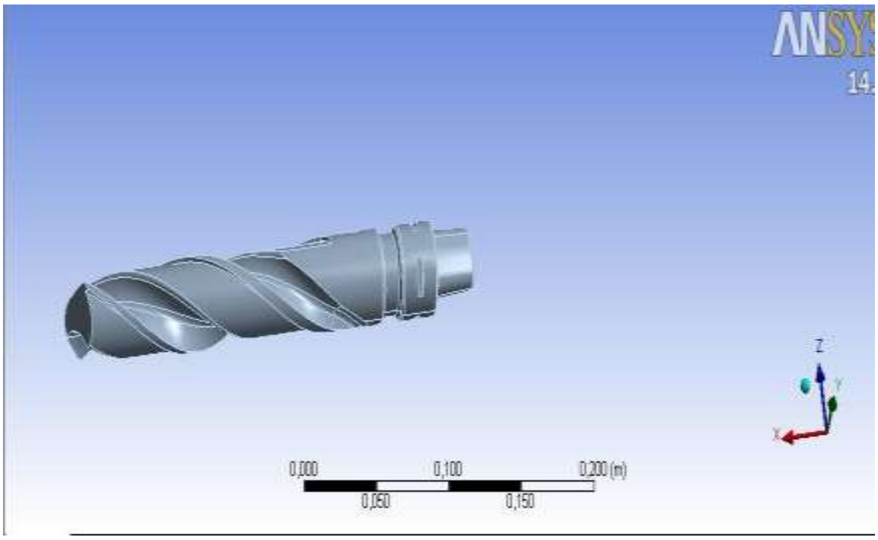


Figure 1 - Importing geometry from the Compass program.

3. Let's build a mesh of finite elements (Fig. 2):

Figure 2 - Finite element mesh.

4. Apply the force of fixing the drill in the machine spindle (Fig. 3):

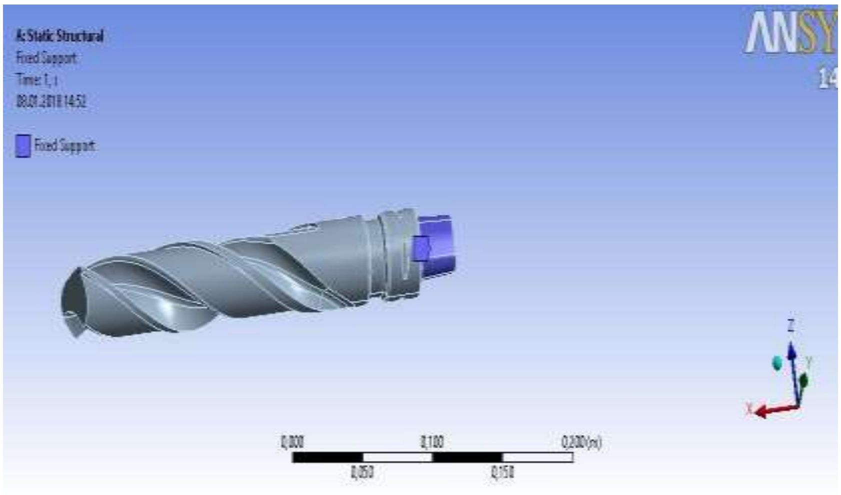


Figure 3 - Fixing force

5. Let's apply a bending moment (drilling moment, Fig. 4 Msv = 50N:

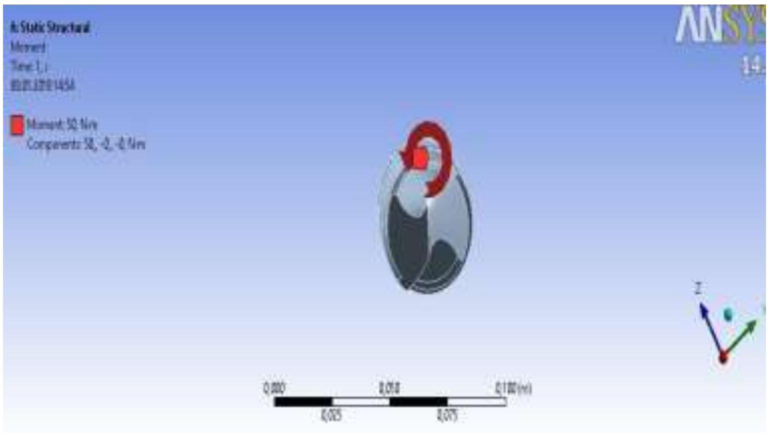
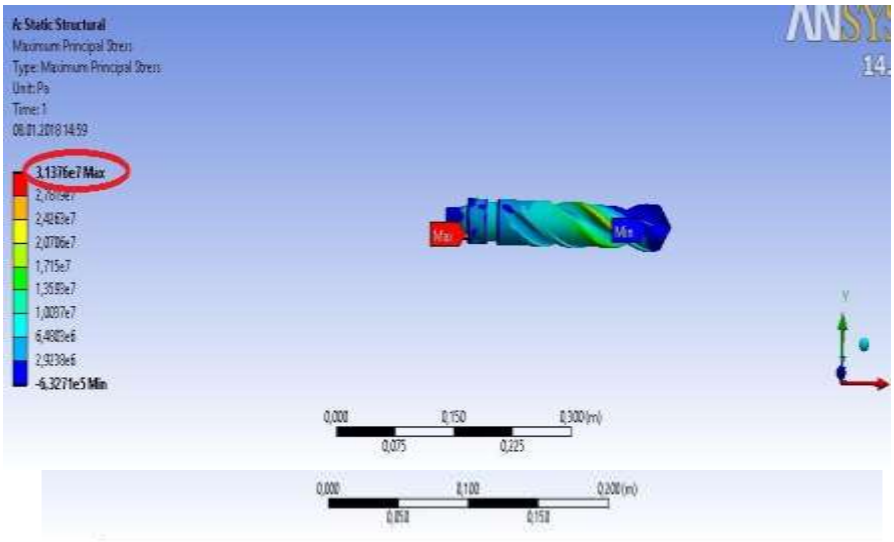


Figure 4 - Drilling moment.

6. Let's calculate the maximum stress that occurs due to the load



Torsion from the moment of drilling (Fig. 5):

Figure 5 - Calculation of the maximum stress.

As can be seen from Figure 4, the maximum voltage is:

τmax = 3.14· 107 Pa = 31.4 MPa. (one)

Let's compare the allowable and maximum voltage:

Tmax = 31.4 MPa < [τ]=250 MPa (2)

Conclusion: the maximum stress from the torque load of the drilling moment is less than the allowable stress.

Now let's compare the maximum stress that occurs in the drill under the influence of the cutting force with the allowable stress [2].

To do this, it is necessary to determine the maximum stress that occurs in the drill under the influence of the cutting force using the ANSYS program and compare it with the allowable stress. The maximum voltage must be less than the allowable one.

1. Let's take the allowable stress of the drill [τ]=250 MPa.

2. The geometry of the model is identical to the geometry (Fig. 6)

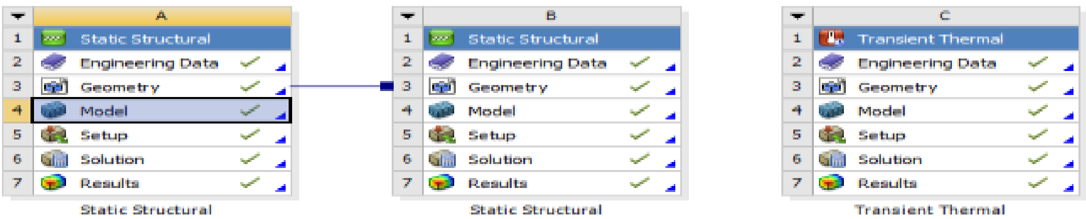


Figure 6 - Selecting the geometry of the model.

3. Let's build a mesh of finite elements (Fig. 7)

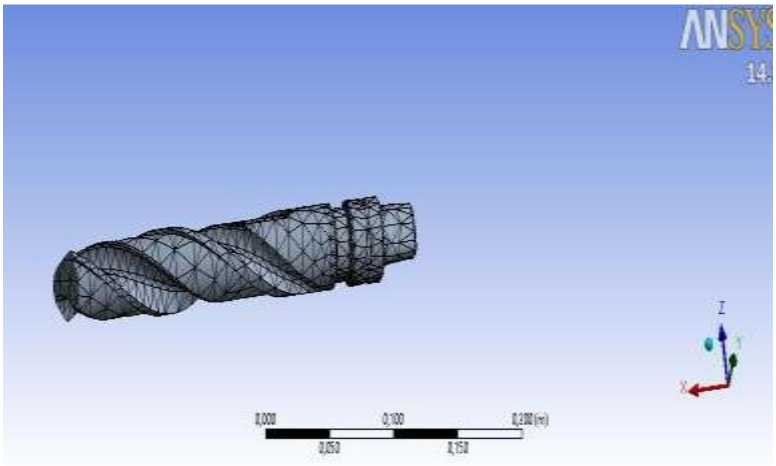


Figure 7 - Finite element mesh.

4. Apply the force of the cantilever fastening of the drill (Fig. 8):

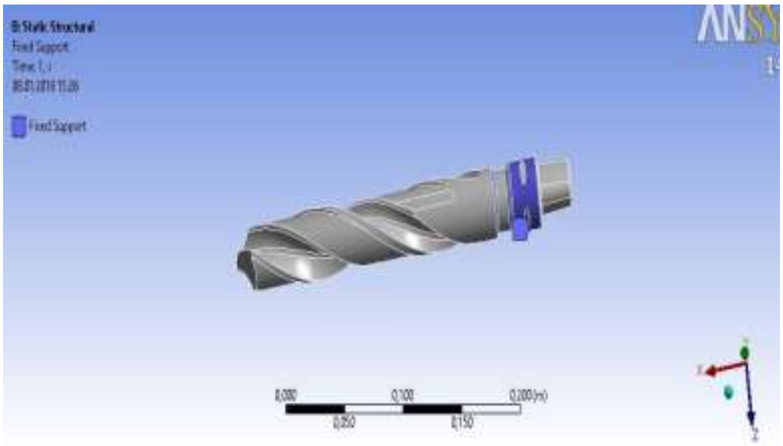
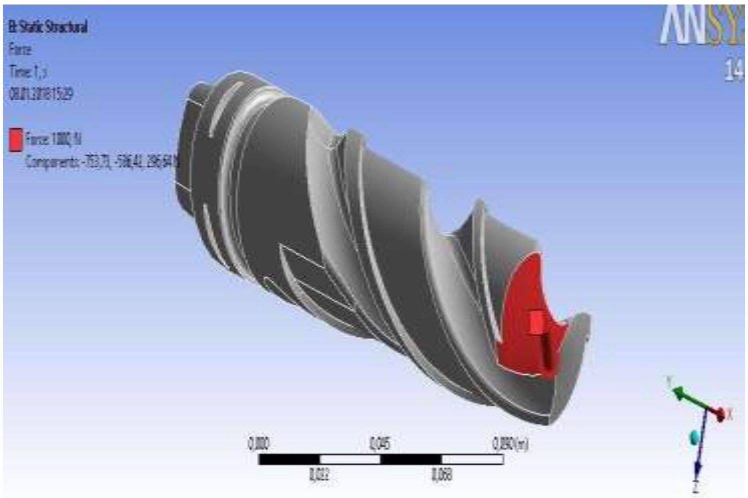


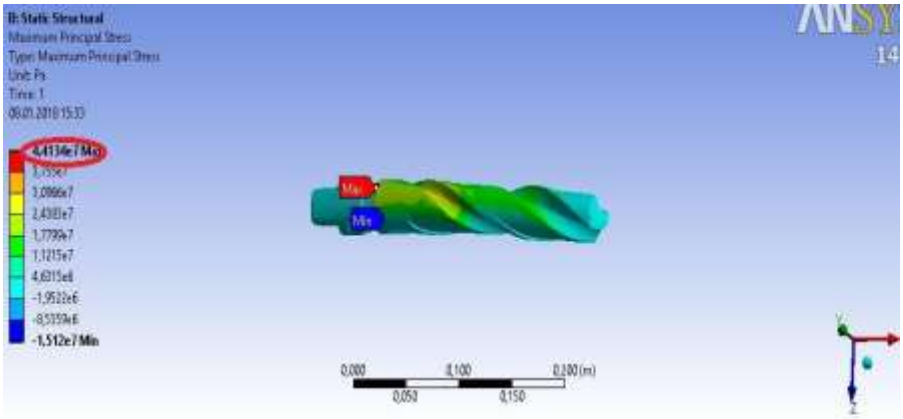
Figure 8 - Fixing force.

5. Apply the cutting force F = 1000N (Fig. 9):



Rice. 7 - Cutting force.

6. Let's calculate the maximum stress that occurs during the manufacture of a drill under The



action of a cutting force during a grinding operation (Fig. 8):

Figure 8 - Calculation of the maximum stress.

As can be seen from Figure 8, the maximum voltage is:

$\tau_{max} = 4.41 \cdot 107$ Pa = 44.1 MPa (3)

7. Compare the allowable and maximum voltage:

$\tau_{max} = 44.1$ MPa < $[\tau]=250$ MPa (4)

Conclusion: the maximum stress that occurs during the manufacture of a drill under the action of a cutting force is less than the allowable stress during a grinding operation.

Literature.

1. Fluent A. 12.0 Theory Guide // Ansys Inc. - 2009. - T. 5. - No. 5.

2. Gilovoy L. Ya., Molodtsov VV Investigation of the effect of centrifugal forces on the operational properties of the HSK connection by simulation methods // STIN. – 2011. – no. 12. - S. 2-7.

3. Progressive cutting tools and metal cutting modes: reference book / V.I. Baranchikov [i dr.]. - M. Mashinostroenie, 1990. - 400 p.