The stress analysis of the milling machine structure

K Lysek, A Gwiazda and K Herbuś
Silesian University of Technology, Faculty of Mechanical Engineering, Institute of Engineering Processes Automation and Integrated Manufacturing Systems, Konarskiego 18A, 44-100 Gliwice, Poland

E-mail: kamil.lysek@polsl.pl

Abstract. The article presents a ways of checking the stress of milling machine construction made of steel profiles. The aim of the article was designation of strength parameters. In order to designate basic parameters like deformation or stress, one of most popular mesh methods was used – finite elements method. In the work, load of the milling machine on the upper and the lower frame was analysed. The models were analyzed due to the quality of the mesh produced, based on the amount of occurrence of a given type of element. For this purpose, the statistics included in the program were used, which allows a scale of 0 to 1 to determine the quality of the element. Despite the use of most of the Tet10 type elements, and the minority of the type We15 and Hex20, the mesh analysis allowed to believe in the accuracy of computational solutions. Process of tension designation went through without errors, and designated properties didn’t get close to yield point for given material. Received deformations are also low therefore they shouldn’t affect the milling machine work precision in required range in any significant way.

1. Introduction
The stress analysis is an indispensable element of mechatronic machines being aggregation of three different subsystems [1-3]. It allows analysing the influence of control system on dynamic behaviour of analysed technical system [4-6].

Utilization of stress analysis allows checking in analytic way the stress and accompanying detail deformations made during the work [7-9]. For stress analysis, often there are CAD/CAE programs. CAD program (Computer Aided Design) is a software aiding modelling of some devices and other elements which can be shown using 2D or 3D model, also making documentation for them. While CAE (Computer Aided Engineering) is the software aiding any calculations associated with the model. Such software can use many calculation methods, such as boundary element method, finite difference method, finite volume method, meshless methods, but the most popular one is finite element method (FEM) [10-12].

This method involves discretization of calculation area using simple finite elements of known shape function. Because every element neighbours only with few other elements, resulting matrix is very rare. These results in lower amount of processed data, what allows for faster problems resolving. Unfortunately, this also results in usage of special procedures providing convergence of solutions [13,14].

2. Materials and methods
2.1. Used software
To do FEM calculations, there was used software called Ansys. In order to carry out an analysis, the key was getting to know the possibilities offered by this software. One of basic ones is the possibility
of controlling the mesh quality by functions like Method or Sizing. First of those allows for defining a method in which way the mesh will be made. While the second one allows inputting the minimal size of the mesh in reference to item type (face, body, edge etc.). Those and many more other useful functions make Ansys willingly used as a calculation tool. One of examples is its usage in automotive branch, what can be seen in figure 1. In this case, the program was used for dynamic analysis during a collision of a car with an obstacle. [15]

![Figure 1. Usage of Ansys in Automotive branch [15].](image)

2.2. Model preparation

Analysed elements consist mainly of steel profiles, of which basic material properties are Young’s module $E = 200$ GPa and density $\rho = 7850$ kg/m³ [16-18].

As a first one, upper frame of milling machine was analysed. In order to carry out an analysis, it was assumed that the base will be anchored, and the frame was loaded with following forces: 1400 N on Y axis and 900N or X axis. Forces have been applied to places where leading rails were attached. Model was divided into more than 2.1 million of finite elements, as shown in figure 3.

The prevailing elements are of type Tet10, while Wed15 are the least. Despite domination of tetragonal elements, quality of generated mesh looks very good. Basing on statistics analysis presented in the program, it can be seen that dominating quality for majority of elements fits in bounds 0.75-0.86. These data were presented in figure 2.

![Figure 2. The element quality for the upper frame mesh.](image)
Figure 3. The upper frame with applied mesh of finite elements

Lower frame was divided like in figure 4, into over 1.6 million of finite elements, what is a result of limiting minimal body element size to 5 mm. Created mesh entirely consists of Tet10 type finite elements.

Figure 4. The lower frame with finite elements mesh applied.
However, it has no significant influence on mesh quality, because as seen in figure 5, in dominating majority quality of elements is being graded between 0.75-0.88, what is a good result.

To generated mesh, a load of force of 2500 N was applied along -Z axis. Load was applied to places where there were rails, while restraint was applied at the bottom of shorter sections.

![Figure 5. The element quality for the bottom frame mesh](image)

3. Results and discussion

As can be seen in the figure 6, maximal deformation of upper frame is equal to $5.32 \times 10^{-2}$ mm. Therefore it can be specified that it is little and unable to have big impact during the machining of averagely precise elements. Of course, in CNC industrial milling machine such deformation would be unacceptable and it would allow rejection of many products produced by it.

![Figure 6. The analysis of deformation of upper frame of the milling machine.](image)
Maximal stress designated [7] during the analysis is approximately 31.3 MPa, what can be seen in figure 7. It is a value several times smaller than yield point for construction steel, which in case of this material is equal to 250 MPa.

Figure 7. The stress of upper frame of milling machine analysis.

As can be seen in figure 8, maximal deformation of lower frame is equal to 4.03 mm*10^-3, this is an even smaller deformation than in case of upper frame. Therefore, also in this case it can be defined as low and unable to have big impact during the machining of averagely precise elements.

Figure 8. The deformation of lower frame of the milling machine analysis.
The stress which were created during the load can be seen in figure 9. They are low and their maximal value is approximately 2.73 MPa. One should stated that the results are in same case related with the material parameters that were designated for the analysed design. It is hover possibility to utilize also different materials [19,20].

Figure 9. The stress of lower frame of the milling machine analysis.

4. Conclusions
Mesh quality analysis showed that mostly the mesh is of good quality (0.75-0.85) and hence the received results can be taken into account as true.

Process of tension designation went through without errors, and designated properties didn’t get close to yield point for given material, therefore it can be assumed that after relief of the tensions, detail will come back to its original shape. Received deformations are also low therefore they shouldn’t affect the milling machine work precision in required range in any significant way. Calculations for lead screw weren’t done due to weight support on entire length on the fences.

The future works are intended to compare the results of virtual stress analysis with the results of actual systems obtained by typical manufacturing technologies. It is also analysed the application of temperature modules of virtual stress investigation to even more precisely match the results of simulation with actual tests.

5. References
[1] Nalepa B, Gwiazda A and Banaś W 2018 IOP Conf. Ser.: Mat. Sci. Eng. 400 1 doi: 10.1088/1757-899X/400/5/052006
[2] Gwiazda A 2014 App. Mech. Mat. 474 147 (doi: 10.4028/www.scientific.net/AMM.474.147)
[3] Hryniewicz P, Banaś W, Gwiazda A, Foit K, Sękala A and Kost G 2015 IOP Conf. Ser.: Mat. Sci. Eng. 95 1 doi: 10.1088/1757-899X/95/1/012086
[4] Nalepa B, Gwiazda A and Banaś W 2018 IOP Conf. Ser.: Mat. Sci. Eng. 400 1 doi: 10.1088/1757-899X/400/5/052000
[5] Hryniewicz P, Banaś W, Foit K, Gwiazda A and Sękala A 2017 IOP Conf. Ser.: Mat. Sci. Eng. 227 1 doi: 10.1088/1757-899X/227/1/012061
[6] Banaś W, Ćwikła G, Foit K, Gwiazda A, Monica Z and Sękala A 2017 IOP Conf. Ser.: Mat. Sci. Eng. 227 1 doi: 10.1088/1757-899X/227/1/012012
[7] Łodygowski T and Kąkol W 2003 Finite Element Method in selected issues of mechanics of engineering structures in Polish: Metoda Elementów Skończonych w wybranych zagadnieniach mechaniki konstrukcji inżynierskich (Kraków: Alma Mater)
[8] Gwiazda A, Sękala A, Banaś W, Topolska S, Foit K and Monica Z 2017 IOP Conf. Ser.: Mat. Sci. Eng. 227 1 doi: 10.1088/1757-899X/227/1/012054
[9] Gwiazda A, Banaś W, Sękala A, Foit K, Hryniewicz P and Kost G 2015 IOP Conf. Ser.: Mat. Sci. Eng. 95 1 doi: 10.1088/1757-899X/95/1/012103
[10] https://www.techopedia.com/definition/2063/computer-aided-design-cad accessed 20.03.2020.
[11] https://en.wikipedia.org/wiki/Computer-aided_engineering accessed 12.03.2020.
[12] Glodová I, Lipták T and Bocko J 2016 Procedia Eng. 96 131 doi: 10.1016/j.proeng.2014.12.131
[13] Bąkowski H and Posmyk A 2011 Finite Elements Method aide analysis of the wear of some selected sub-assemblies in technical means of transport (Gliwice: Wydawnictwo Politechniki Śląskiej)
[14] Osiński J 1997 Strength calculations of machine elements using the finite element method in Polish: Obliczenia wytrzymałościowe elementów maszyn z zastosowaniem metody elementów skończonych (Warszawa: Oficyna Wydawnicza Politechniki Warszawskiej)
[15] https://productdesign-development.com/impact-analysis accessed 12.03.2020.
[16] Ansys material library
[17] Topolska S and Łabanowski J 2017 Arch. Metall. Mater. 62(4) 2107 doi: 10.1515/amm-2017-0312
[18] Topolska S and Łabanowski J 2017 IOP Conf. Ser.: Mat. Sci. Eng. 227 1 doi: 10.1088/1757-899X/227/1/012131
[19] Grajcar A, Kożlowska A, Topolska S and Morawiec M 2018 Adv. Mater. Sci. Eng. 2018 1 doi: 10.1155/2018/7369827
[20] Adamowicz A 2016 J. Theor. App. Mech. 54(1) 205 doi: 10.15632/jtam-pl.54.1.205