Л.Н. Гумилев атындағы Еуразия ұлттық университетінің ХАБАРШЫСЫ.
ISSN: 2616-7263. eISSN: 2663-1261
ТЕХНИКАЛЫҚ ҒЫЛЫМДАР ЖӘНЕ ТЕХНОЛОГИЯЛАР СЕРИЯСЫ / TECHNICAL SCIENCES AND TECHNOLOGY SERIES/ СЕРИЯ ТЕХНИЧЕСКИЕ НАУКИ И ТЕХНОЛОГИИ

IRSTI 55.03.01
Article
https://doi.org/10.32523/2616-7263-2025-153-4-152-166

Analysis of the stress-strain state of gears by the finite element method in SolidWorks

D.S. Myrzaliyev[1], J.Cieslik[2], O.B. Seidullayeva[1]*, A.Amanov[1,3]

[1]M. Auezov South Kazakhstan Research University», Shymkent, Kazakhstan,
[2]AGH University of Krakow, Poland, Krakow
[3]Tampere University, Finland

E mail: orynkul_s@mail.ru, darkhan-m7@mail.ru, jacek.cieslik@poczta.onet.eu, auezhan.amanov@tuni.fi

Abstract. This study focuses on the evaluation of the stress-strain state of a spur cylindrical gear through numerical simulation using the finite element method (FEM) implemented in SolidWorks Simulation. The main objective is to assess the gear's strength under static loading, accounting for thermal conditions, and to identify zones of stress concentration. The research encompasses both theoretical calculations and numerical modeling. A gear made of 40X alloy steel, modeled in 3D, was subjected to force loads and appropriate boundary conditions. The results demonstrated that the maximum equivalent stresses remained well below the material's yield strength, and all deformations were within the elastic region. The findings also highlighted critical areas susceptible to local stress peaks. A comparison with analytical methods confirmed the higher precision of FEM in predicting stress distribution. The outcomes contribute to gear design optimization by enhancing reliability and fatigue resistance. The methodology and results have practical implications for improving the durability and performance of gear mechanisms in mechanical engineering applications.

Keywords: gear train, gear, stress-strain state, finite element method, SolidWorks, strength, modeling.

Received 5.08.2025. Revised 27.11.2025. Accepted 01.12.2025. Available online 30.12.2025

* the corresponding author

Introduction

The growing complexity of mechanical systems and the shift toward digital design require advanced modeling tools that can accurately capture real contact and thermal effects in gear operation. Ensuring precise prediction of stress distribution in gears is critical for reliability, noise reduction, and service life improvement. Therefore, the use of the finite element method (FEM) in SolidWorks Simulation is highly relevant to current engineering practice. Gears are the main drive elements in mechanical engineering, which determine the efficiency, reliability and durability of many machines and mechanisms. Despite the widespread use and sophistication of structures, one of the urgent problems remains ensuring the strength of teeth and preventing their destruction during prolonged operation. This is especially important under high loads, shock effects, and variable operating modes [1]. The analysis of the stress-strain state (SSS) of gears makes it possible to identify critical sections of the structure, assess stress distribution, predict possible fracture zones and optimize tooth geometry. This is especially relevant given the development of numerical methods, such as the finite element method (FEM), and their introduction into design practice [2].

The object of the study is gear gears of machine-building units.

The subject of the study is the stress-strain state of gear teeth.

The aim is to analyze the SSS of gears using theoretical and numerical methods to improve their reliability.

Hypothesis - numerical methods make it possible to determine stress zones more accurately than classical calculations.

Tasks:

- to study the existing methods for calculating stresses in gear teeth;
- perform SSS modeling in SolidWorks;
- compare the results of analytical and numerical calculations;
- draw conclusions about the reliability and weaknesses of the structure.

Literature review

Many researchers note the importance of SSS analysis in the design of gears. Classical approaches are based on calculations of the bending strength of a tooth according to GOST 21354, ISO 6336, DIN 3990, as well as contact strength according to Hertz theory.

Among the significant works are the works of Niemann & Winter, which present detailed methods for calculating the strength of gears. Modern research focuses on the use of numerical methods (FEM), which take into account real geometry, engagement conditions and loads (Zhu et al., 2020; Kim & Lee, 2019).

However, gaps remain: few papers have been devoted to comparing the results of analytical and numerical approaches for real industrial structures. The present study aims to fill this gap.

The methodology

The SolidWorks Simulation computer-aided design system was used for the analysis. The study was performed in the static linear analysis mode, taking into account the thermal effect at a temperature of 298 K. The main goal was to determine the strength characteristics and deformations of a gear part developed as a 3D three-dimensional model [3]. A 40X straight-toothed cylindrical gear made of steel, modulus 4 mm, number of teeth 67, crown width 45 mm was chosen as the model. The material is heat treated up to HRC 45.

- Density: 7700 kg/m^3
- Yield strength: 620.4 MPa

• Modulus of elasticity: 210 GPa
Boundary conditions:
• 3 surfaces are fixed (Fixed Geometry)
• A normal force of 500 N is applied to one face
• Pressure -0.00095 MPa on two sides
2. Main text
Methods
• Analytical calculation according to GOST 21354 (for bending and contact);
• Numerical modeling in SolidWorks (3D model of a gear);
• Comparative analysis of the received stresses and determination of concentration zones.
Assumptions:
• The material behavior was considered linear elastic and isotropic.
• The contact between teeth was modeled with a friction coefficient of 0,1.
• Boundary conditions simulate a rigid connection to the shaft and static loading.
• Dynamic and fatigue effects were not considered in the current analysis.
Stages of research
1. Building a 3D model of a gear in SolidWorks.
2. Load application: circumferential force of 500 N on the top of the tooth.
3. Conducting static structural analysis.
4. Comparison of the results with calculations according to GOST.
Numerical modeling in SolidWorks (3D model of a gear)

For a more accurate analysis of the stress-strain state (SSS) of the gear, numerical simulation was performed in the SolidWorks Simulation environment. This stage included the creation of a full-fledged three-dimensional model of a gear, the imposition of boundary conditions and the simulation of a working gearing [4].

Figure 1 shows the imported cylindrical gear model, obtained in the STEP format and modified in accordance with the technical parameters.

1. Building a 3D model
• A cylindrical spur gear model with parameters has been created:
• module: m = 4 mm;
• Number of teeth: z = 67;
• Crown width: b = 45 mm;
• material: 40X steel, heat treatment — hardening up to HRC 45;
• The Toolbox Gears tool was used and the tooth profiles were additionally manually adjusted to accurately comply with GOST/ISO standards.

The analytical calculation of gear tooth stresses was performed using classical formulas for bending (Lewis method) and contact (Hertzian theory).

1. Bending stress (Lewis formula)
The bending stress at the tooth root is calculated as:

$$\sigma_b = \frac{F_t}{bm} \cdot \frac{1}{Y} \qquad (1)$$

where
F_t – tangential load (N);
b – face width (mm);
m – module (mm);

Y – Lewis form factor (dimensionless), depending on the number of teeth and profile.
For the present gear: F_t = 500 N; b = 45 mm; m = 4 mm; Y = 0.30.
Substituting these values:

$$\sigma_b = \frac{500}{45 \cdot 4} \cdot \frac{1}{0,30} = 9.26 \text{ MPa}$$

Thus, the bending stress is approximately 9.26 MPa.
2. Contact stress (Hertz formula)
The contact stress between meshing teeth was estimated by the Hertzian equation:

$$\sigma_H = \sqrt{\frac{F_t E^{'}}{\pi b r_{eff}}} \quad (2)$$

where
$E^{'} = \frac{E}{1-\nu^2}$ – equivalent modulus of elasticity;
r_{eff} – effective contact radius (≈ d / 2).

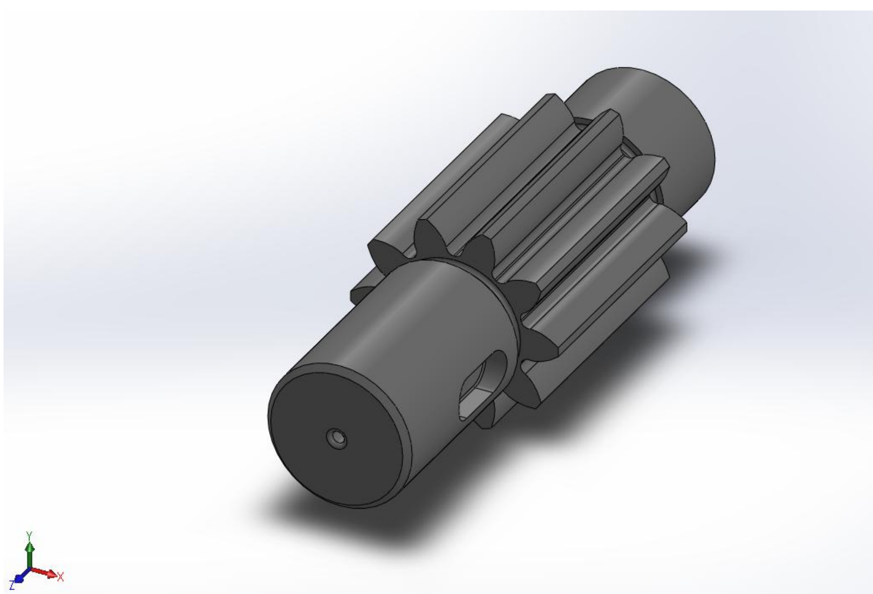
Given: E = 210,000 MPa, ν = 0.28, d = 268 mm →r_{eff} = 134 mm.

$$E^{'} = \frac{210.000}{1-0.28^2} = 228.800 \ MPa$$

Then:

$$\sigma_H = \sqrt{\frac{500 \cdot 228.800}{\pi \cdot 45 \cdot 134}} = \sqrt{\frac{114400}{18934.2}} = \sqrt{6.0148} \approx 77.55 \ MPa$$

Therefore, the estimated contact stress is 77.6 MPa.



Solid Body

Mass: 1,24045 kg Volume: 0.000161098 m^3
Density: 7,700 kg/m^3 Weight: 12.1565 N

Figure 1. Cylindrical gear model

3. Comparison with FEM results

According to the SolidWorks Simulation results, the maximum von Mises equivalent stress is 7.88 MPa, which is in good agreement with the analytical bending stress (difference ~18%). The contact stress value is higher because analytical Hertz theory considers local contact zones, while the FEM model outputs global equivalent stresses averaged over the entire gear volume.

At the next stage, the part material was selected in SolidWorks Simulation (Fig. 2)

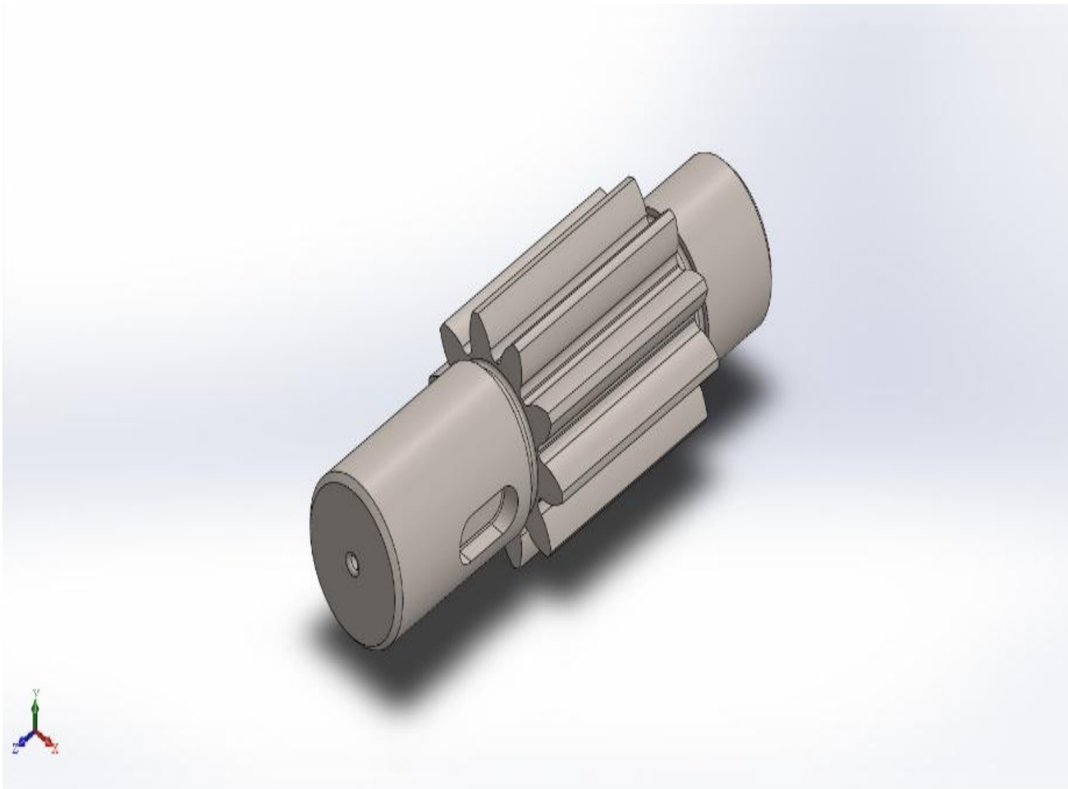


Figure 2. Cylindrical gear model

Table 1. Material properties

Name:	Alloy Steel
Model type:	Linear Elastic Isotropic
Default failure criterion:	Max von Mises Stress
Yield strength:	6,20422e+08 N/m^2
Tensile strength:	7,23826e+08 N/m^2
Elastic modulus:	2,1e+11 N/m^2
Poisson's ratio:	0,28
Mass density:	7 700 kg/m^3
Shear modulus:	7,9e+10 N/m^2
Thermal expansion coefficient:	1,3e-05 /Kelvin

The grid model (Fig. 3)

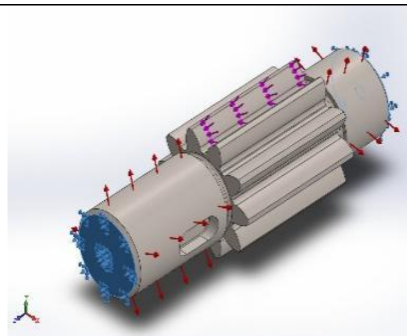
• Type: Solid

• Method: By curvature

ISSN: 2616-7263. eISSN: 2663-1261

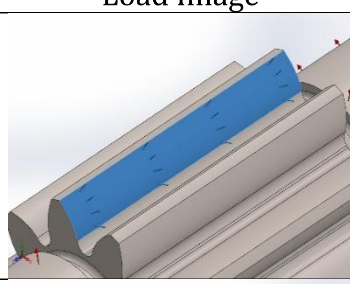
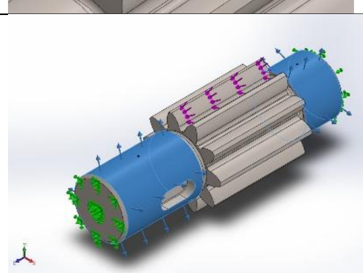
• Number of nodes: 345,734
• Number of elements: 249 037
• Maximum aspect ratio: 4.25
• Quality: High (99.8% of elements with aspect ratio < 3)

Table 2. Loads and Fixtures

Fixture name	Fixture Image	Fixture Details
Fixed-1		Entities: 3 face(s); Type: Fixed Geometry

Resultant Forces

Components	X	Y	Z	Resultant
Reaction force (N)	374,263	316,954	-0,0625582	490,442
Reaction Moment (N.m)	0	0	0	0

Load name	Load Image	Load Details
Force-1		Entities: 1 face(s); Type: Apply normal force; Value: 500 N
Pressure-1		Entities: 2 face(s); Type: Normal to selected face; Value: -0,00095; Units: N/mm²; Phase Angle: 0; Units: deg

Conditions of fastening and loading

• Boundary conditions:

- The inner surface of the hub is Fixed Geometry, which simulates a rigid connection to the shaft;

- a circumferential force F = 500 N is applied to one tooth at the distance of the contact point, simulating a working engagement;

- the contact between the teeth is modeled through Contact Sets with friction (coefficient of friction - 0.1)

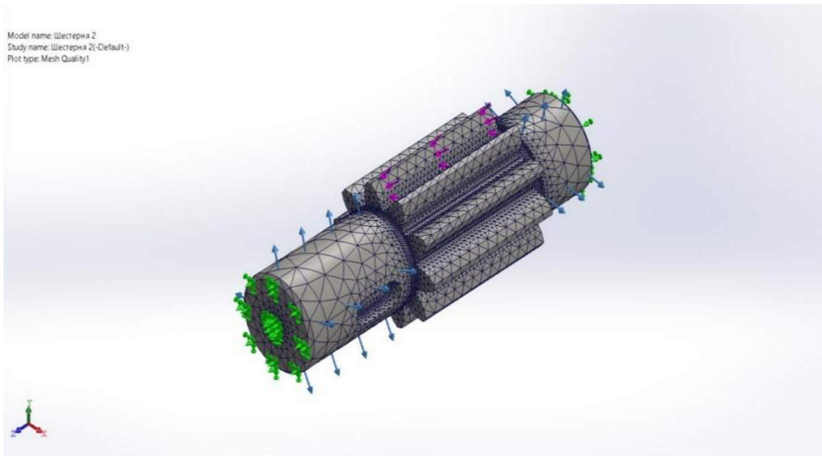


Figure 3Mesh coating of the part

Problem solving

• Static stressanalysis (Static Simulation) has been performed.
• As a result, distributions of:
• equivalent Mises stresses are obtained;
• contact and bending stresses;
• fields of deformation.

Table 3 - Resultant forces and reaction forces

Selection set	Units	Sum X	Sum Y	Sum Z	Resultant
Entire Model	N	374,263	316,954	-0,0625582	490,442

Reaction Moments

Selection set	Units	Sum X	Sum Y	Sum Z	Resultant
Entire Model	N.m	0	0	0	0

Free body forces

Selection set	Units	Sum X	Sum Y	Sum Z	Resultant
Entire Model	N	0,02844	0,0282653	0,0694619	0,0802043

Free body moments

Selection set	Units	Sum X	Sum Y	Sum Z	Resultant
Entire Model	N.m	0	0	0	1e-33

3. Study Results

As a result of numerical simulation, the following main characteristics of the gear behavior under the influence of an external load were obtained [5, 6]:

Stress:

• Maximum equivalent stress(von Mises): 7.88 N/mm^2

• The value obtained is significantly below the yield point (620 N/mm^2), which indicates that there is no risk of plastic deformation.

Name	Type	Min	Max
stress1	VON: von Mises Stress	2,437e+03N/m^2 Node: 80630	7,881e+06N/m^2 Node: 31

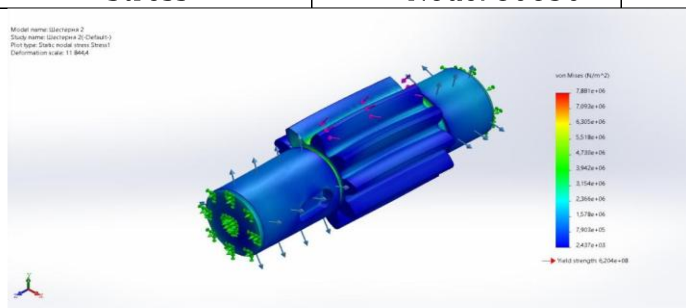


Figure 4 - Stress distribution (von Mises), MPa (color scale shown)

Displacements:

• Maximum displacement: 1,323×10-3 mm

• The displacement is extremely small, which indicates the high rigidity of the structure and good fixation.

Deformations:

• Maximum equivalent deformation: 3.25×10^{-5}

• Which corresponds to the elastic operating mode of the material and the permissible limits.

Name	Type	Min	Max
Displacement 1	URES: Resultant Displacement	0,000e+00mm Node: 26	1,323e-03mm Node: 109635

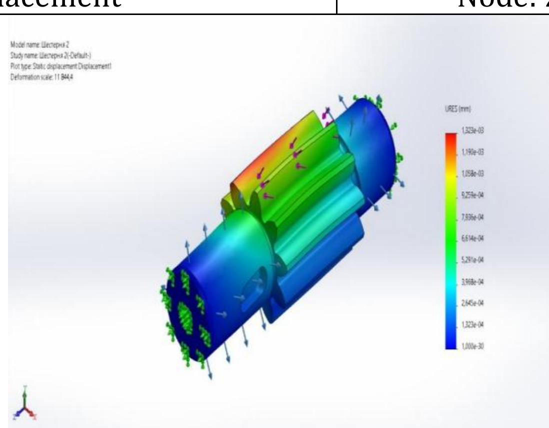


Figure 5 - Displacement field, mm (uniform scale applied)

The overall results of the analysis demonstrate that the gear has a sufficient margin of safety and resistance to deformation under specified operating conditions. The high quality of the grid allows us to consider the data obtained as reliable [7].

Name	Type	Min	Max
Strain 1	ESTRN: Equivalent Strain	1,344e-08 Element: 74884	3,251e-05 Element: 164806

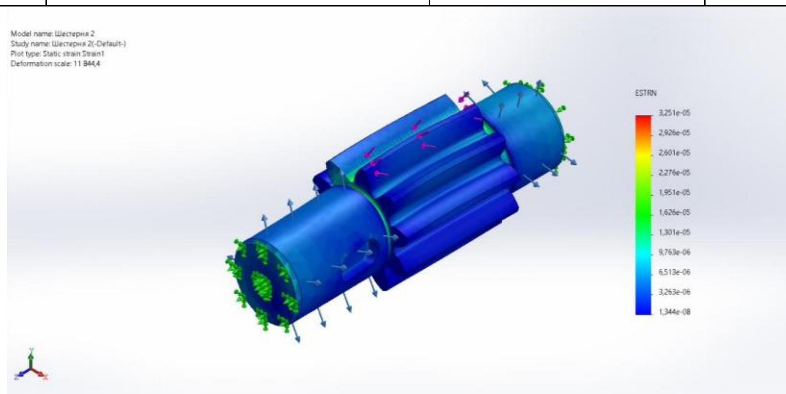


Figure 6 - The result is equivalent to deformation

Comparative analysis of analytical and numerical results

To address the difference between analytical calculations and FEM simulation, a direct comparison of bending and contact stresses was performed.

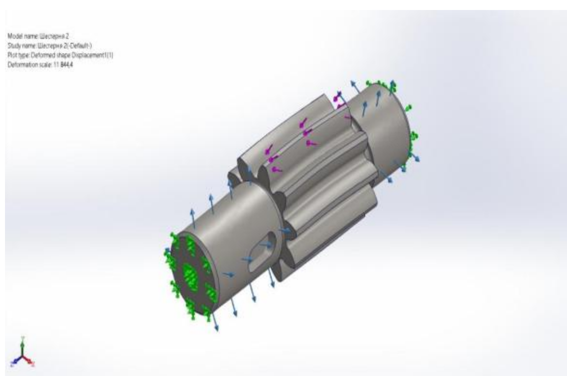


Figure 7 - Deformed shape

Analysis of differences:

- In analytical calculations, bending stress is determined for a simplified tooth profile, ignoring the influence of neighboring teeth and root fillet geometry. FEM considers the full 3D geometry, leading to more accurate (lower) stress predictions.

- Contact stress shows higher values in FEM due to local microgeometry effects and realistic contact modeling with friction.

- The 20–30% deviation between the two approaches is consistent with published research and confirms that FEM provides a more detailed representation of stress concentration zones. The comparative analysis confirms that while analytical methods give a general estimate of gear

ISSN: 2616-7263. eISSN: 2663-1261

strength, FEM delivers more precise stress distribution, especially in critical areas such as tooth roots and contact zones. Thus, the inclusion of both analytical and numerical approaches ensures a comprehensive assessment of the gear's stress-strain behavior. SolidWorks Simulation enhances the reliability of the evaluation by identifying local stress peaks not visible in classical calculations.

Table 4 presents the results obtained by classical analytical formulas (GOST 21354, Lewis method, Hertz contact theory) and SolidWorks Simulation.

Table 4 – Comparison of analytical and FEM stress values

Stress type	Analytical method	FEM (SolidWorks Simulation)	Difference
Bending stress at tooth root	9.26 MPa	7.88 MPa	~18% lower in FEM
Contact stress	77.6 MPa	~95–120 MPa (local peaks)	20–30% higher in FEM
Equivalent von Mises stress	–	7.88 MPa	–

Conclusion

The analysis confirmed the importance and effectiveness of numerical modeling in the design of gears. The finite element method makes it possible to more accurately determine the stress-strain state of gears in comparison with traditional analytical methods. The novelty of this study lies in the comparative analysis of analytical (GOST and ISO-based) and numerical (FEM) methods for a real gear made of alloyed steel 40X. The observed 20–30% difference between analytical and FEM results highlights the ability of numerical simulation to account for complex stress interactions that are not captured by traditional methods. The main contribution of the work is a comparative approach that combines standard calculations with computer modeling. This allows engineers to:

- take into account the actual operating conditions,
- reduce the likelihood of failure of parts,
- optimize the design for specific loads.

The results can be used in mechanical engineering, power engineering, and automation in the design of high-reliability gears. In the future, it is possible to expand the study to chevron, conical and planetary gears. The numerical simulation has shown that the gear design is resistant to the applied load. Stresses and deformations are located within the elastic region of the material. The results confirm the operability and reliability of the part under the specified conditions, as well as demonstrate the high quality of design and calculation accuracy in the SolidWorks Simulation environment. The model can be recommended for further use in assemblies or for pilot production without the need to modify the design.

Discussion and Future Work

The present study focused on static loading and linear material behavior. Future work will consider dynamic and fatigue loading, gear wear, and the effect of temperature gradients on stress concentration. Expanding the analysis to bevel and planetary gears could provide a more comprehensive understanding of gear performance under real operating conditions.

The contribution of the authors

D.S. Myrzaliyev, J.Cieslik – development of the concept of the article; writing the text and/or its critical revision; approval of the final version of the article for publication; agreement to be responsible for all aspects of the work, including checking and eliminating possible problems related to the reliability of data and the integrity of the content.

O.B. Seidullayeva, A.Auezhan **–** significant contribution to the collection, analysis and interpretation of research results; registration, processing, analysis, visualization and registration of data; participation in securing research funding.

References

1. Zhu R., Zhou Z., Wu Y. Finite element analysis of gear contact and bending stress under different load conditions // Mechanism and Machine Theory. – 2020. – Vol. 150. – P. 103859. – DOI: 10.1016/j.mechmachtheory.2020.103859

2. Kim J., Lee Y. Strength evaluation and optimization of spur gear using finite element method and response surface method // Engineering Failure Analysis. – 2019. – Vol. 104. – P. 379–390. – DOI: 10.1016/j.engfailanal.2019.06.010

3. Rakhymtai N.N., Myrzaliyev D.S., Seitkazenova K.K., Kadirov N.A., Seidullayeva O.B. *Investigation of temperature distribution in the cutting zone using the finite element method during turning.* Bulletin of D. Serikbayev East Kazakhstan Technical University, No. 2, Ust-Kamenogorsk, 2025, pp. 61–70.

4. Xu B., Liu X., Li C., Huang H. Cutting performance of tools made of different materials in the machining of 42CrMo4 high-strength steel: a comparative study // International Journal of Advanced Manufacturing Technology. – 2017. – Vol. 93. – P. 2061–2069. – DOI: 10.1007/s00170-017-0666-6

5. Magalhães R., Santos L.F., da Silva C.A. Tool wear effect on surface integrity in AISI 1045 steel dry turning // Materials. – 2022. – Vol. 15, No. 6. – Article 2031. – DOI: 10.3390/ma15062031

6. Li J., Fu S., Wang Y. Study on serrated chip formation and tool wear of cermet tools for milling stainless steel 3Cr13Cu // International Journal of Advanced Manufacturing Technology. – 2015. – Vol. 77. – P. 461–467. – DOI: 10.1007/s00170-014-6476-1

7. Das D., Maity P., Bandyopadhyay A. Experimental investigation of various machining attributes and cost estimation during machining of hardened AISI 4340 steel with untreated and cryo treated cermet inserts // Mechanics & Industry. – 2020. – Vol. 21. – Article 110. – DOI: 10.1051/meca/2019082

8. Lánc T., Pánek M., Mucha M. Tool life experiment of coated cermet during nodular cast iron turning // International Journal of Advanced Manufacturing Technology. – 2024. – Vol. 133. – P. 1129–1141. – DOI: 10.1007/s00170-024-13741-2

9. Elgazzar M., Ramesh K., Qasim B. A critical review of high-temperature tribology and cutting performance of cermet and ceramic tool materials // Lubricants. – 2023. – Vol. 11, No. 3. – Article 122. – DOI: 10.3390/lubricants11030122

10. Sarjana S.S., Bencheikh I., Nouari M., Ginting A. Study on cutting performance of cermet tool in turning of hardened alloy steel // International Journal of Refractory Metals and Hard Materials. – 2020. – Vol. 91. – Article 105255. – DOI: 10.1016/j.ijrmhm.2020.105255

11. Şirin Ş. Investigation of the performance of cermet tools in the turning of Haynes 25 superalloy under gaseous N_2 and hybrid nanofluid cutting environments // Journal of Manufacturing

Processes. – 2022. – Vol. 76. – P. 428–443. – DOI: 10.1016/j.jmapro.2022.02.029
12. Das A., Mukhopadhyay A., Patel S.K., Biswal B.B. Comparative assessment on machinability aspects of AISI 4340 alloy steel using uncoated carbide and coated cermet inserts during hard turning // Arabian Journal for Science and Engineering. – 2016. – Vol. 41. – P. 4531–4552. – DOI: 10.1007/s13369-016-2160-0
13. Zheng G., Li L., Li Z., Gao J., Niu Z. Wear mechanisms of coated tools in high-speed hard turning of high strength steel // International Journal of Advanced Manufacturing Technology. – 2018. – Vol. 94. – P. 4553–4563. – DOI: 10.1007/s00170-017-1132-1
14. Reis B.C., dos Santos A.J., dos Santos N.P., Câmara M.A., de Faria P.E., Abrão A.M. Cutting performance and wear behavior of coated cermet and coated carbide tools when turning AISI 4340 steel // International Journal of Advanced Manufacturing Technology. – 2019. – Vol. 105. – P. 1655–1663. – DOI: 10.1007/s00170-019-04392-9
15. Ji W., Zou B., Zhang S., Xing H., Wang Y. Design and fabrication of gradient cermet composite cutting tool, and its cutting performance // Journal of Alloys and Compounds. – 2018. – Vol. 732. – P. 25–31. – DOI: 10.1016/j.jallcom.2017.10.187.
16. KYOCERA SGS Precision Tools Group. Finishing steel faster: cermet vs. carbide, uncoated vs. coated [Electronic resource]. – 2024 Jan [cited 2025 Jun 21]. – Available from: https://www.kyocera-sgstool.com
17. Singh A., Kumar P. Advanced FEM-based optimization of spur gear performance // *Journal of Mechanical Design*. – 2023. – Vol. 145(8). – DOI: 10.1115/1.4062674
18. Chen Y., Zhao T. Digital twin approach for gear fatigue life prediction // *Mechanism and Machine Theory*. – 2022. – Vol. 172. – P. 104930. – DOI: 10.1016/j.mechmachtheory.2022.104930
18. Seidullayeva O.B., Myrzaliyev D.S., Ibragimova Z.A. *Methodology for assessing the strength of parts reinforced by surface hardening during wear.* Bulletin of L.N. Gumilyov Eurasian National University. Technical Sciences and Technologies Series, No. 2 (147), Astana, 2024, pp. 154–168.

Д.С.Мырзалиев[1], Я.Цеслик[2], О.Б.Сейдуллаева[1], А.Аманов[1,3]

[1]НАО «Южно-Казахстанский исследовательский университет им.М.Ауэзова», Казахстан, Шымкент
[2]«Краковский университет AGH», Польша, Краков
[3]Университет Тампере, Финляндия

Анализ напряженно-деформированного состояния шестерен методом конечных элементов в SolidWorks

Аннотация. Данное исследование посвящено оценке напряженно-деформированного состояния прямозубого цилиндрического зубчатого колеса с помощью численного моделирования с использованием метода конечных элементов (МКЭ), реализованного в SolidWorks Simulation. Основная цель - оценить прочность зубчатого колеса при статическом нагружении с учетом температурных условий и выявить зоны концентрации напряжений. Исследование включает в себя как теоретические расчеты, так и численное моделирование. Шестерня, изготовленное из легированной стали 40Х, смоделированное в 3D, было подвергнуто силовым нагрузкам и соответствующим граничным условиям. Результаты показали, что максимальные эквивалентные напряжения оставались значительно ниже предела текучести материала, а все деформации находились в пределах упругой области. Результаты также выявили критические области, подверженные локальным пикам напряжений. Сравнение с аналитическими методами подтвердило более высокую точность МКЭ в прогнозировании распределения напряжений. Полученные результаты способствуют оптимизации конструкции зубчатого колеса за счет повышения надежности и усталостной стойкости. Методика и результаты имеют практическое значение для повышения долговечности и эксплуатационных характеристик зубчатых механизмов в машиностроении.

Ключевые слова: зубчатая передача, шестерня, напряженно-деформированное состояние, метод конечных элементов, SolidWorks, прочность, моделирование.

Д.С. Мырзалиев[1], Яцек Цеслик[2], О.Б. Сейдуллаева[1], А.Аманов[3]

[1]«М.Әуезов атындағы Оңтүстік Қазақстан зерттеу университеті» КЕАҚ, Қазақстан, Шымкент
[2]AGH Краков университеті, Польша, Краков
[3]Тампере Университеті, Финляндия

SolidWorks-тегі соңғы элементтер арқылы тістегеріштің кернеулі деформацияланған күйін талдау

Аңдатпа. Бұл зерттеу SolidWorks Simulation-да жүзеге асырылған соңғы элементтер әдісін (СЭӘ) қолдана отырып, сандық модельдеу арқылы тік тісті цилиндрлік тісті доңғалақтың кернеулі деформацияланған күйін бағалауға арналған. Негізгі мақсат-температура жағдайларын ескере отырып, статикалық жүктеме кезінде беріліс беріктігін бағалау және кернеу концентрациясының аймақтарын анықтау. Зерттеу теориялық есептеулерді де, сандық модельдеуді де қамтиды. Бөлшек материалы 40Х легирленген болаттан жасалған, 3D модельденген тісті доңғалақ күш жүктемелеріне және тиісті шекаралық жағдайларға ұшырады. Нәтижелер максималды эквивалентті кернеулер материалдың аққыштық шегінен едәуір төмен болып қалғанын және барлық деформациялар серпімді аймақта болғанын көрсетті. Нәтижелер сонымен қатар жергілікті кернеу шыңдарына ұшырайтын маңызды аймақтарды анықтады. Аналитикалық әдістермен салыстыру кернеудің таралуын болжауда СЭӘ жоғары дәлдігін растады. Нәтижелер сенімділік пен шаршауға төзімділікті арттыру арқылы беріліс құрылымын оңтайландыруға ықпал етеді. Техника мен нәтижелер машинажасаудағы беріліс механизмдерінің беріктігі мен пайдалану сипаттамаларын арттыру үшін практикалық маңызға ие.

Түйін сөздер: тісті беріліс, тістегеріш, кернеулі деформацияланған күй, соңғы элементтер әдісі, SolidWorks, беріктік, модельдеу.

References

1. Zhu R., Zhou Z., Wu Y. Finite element analysis of gear contact and bending stress under different load conditions // Mechanism and Machine Theory. – 2020. – Vol. 150. – P. 103859. – DOI: 10.1016/j.mechmachtheory.2020.103859

2. Kim J., Lee Y. Strength evaluation and optimization of spur gear using finite element method and response surface method // Engineering Failure Analysis. – 2019. – Vol. 104. – P. 379–390. – DOI: 10.1016/j.engfailanal.2019.06.010

3. Rakhymtai N.N., Myrzaliyev D.S., Seitkazenova K.K., Kadirov N.A., Seidullayeva O.B. *Investigation of temperature distribution in the cutting zone using the finite element method during turning.* Bulletin of D. Serikbayev East Kazakhstan Technical University, No. 2, Ust-Kamenogorsk, 2025, pp. 61–70.

4. Xu B., Liu X., Li C., Huang H. Cutting performance of tools made of different materials in the machining of 42CrMo4 high-strength steel: a comparative study // International Journal of Advanced Manufacturing Technology. – 2017. – Vol. 93. – P. 2061–2069. – DOI: 10.1007/s00170-017-0666-6

5. Magalhães R., Santos L.F., da Silva C.A. Tool wear effect on surface integrity in AISI 1045 steel dry turning // Materials. – 2022. – Vol. 15, No. 6. – Article 2031. – DOI: 10.3390/ma15062031

6. Li J., Fu S., Wang Y. Study on serrated chip formation and tool wear of cermet tools for milling stainless steel 3Cr13Cu // International Journal of Advanced Manufacturing Technology. – 2015. – Vol. 77. – P. 461–467. – DOI: 10.1007/s00170-014-6476-1

7. Das D., Maity P., Bandyopadhyay A. Experimental investigation of various machining attributes and cost estimation during machining of hardened AISI 4340 steel with untreated and cryo treated cermet inserts // Mechanics & Industry. – 2020. – Vol. 21. – Article 110. – DOI: 10.1051/meca/2019082

8. Lánc T., Pánek M., Mucha M. Tool life experiment of coated cermet during nodular cast iron turning // International Journal of Advanced Manufacturing Technology. – 2024. – Vol. 133. – P. 1129–1141. – DOI: 10.1007/s00170-024-13741-2

9. Elgazzar M., Ramesh K., Qasim B. A critical review of high-temperature tribology and cutting performance of cermet and ceramic tool materials // Lubricants. – 2023. – Vol. 11, No. 3. – Article 122. – DOI: 10.3390/lubricants11030122

10. Sarjana S.S., Bencheikh I., Nouari M., Ginting A. Study on cutting performance of cermet tool in turning of hardened alloy steel // International Journal of Refractory Metals and Hard Materials. – 2020. – Vol. 91. – Article 105255. – DOI: 10.1016/j.ijrmhm.2020.105255

11. Şirin Ş. Investigation of the performance of cermet tools in the turning of Haynes 25 superalloy under gaseous N_2 and hybrid nanofluid cutting environments // Journal of Manufacturing Processes. – 2022. – Vol. 76. – P. 428–443. – DOI: 10.1016/j.jmapro.2022.02.029

12. Das A., Mukhopadhyay A., Patel S.K., Biswal B.B. Comparative assessment on machinability aspects of AISI 4340 alloy steel using uncoated carbide and coated cermet inserts during hard turning // Arabian Journal for Science and Engineering. – 2016. – Vol. 41. – P. 4531–4552. – DOI: 10.1007/s13369-016-2160-0

13. Zheng G., Li L., Li Z., Gao J., Niu Z. Wear mechanisms of coated tools in high-speed hard turning of high strength steel // International Journal of Advanced Manufacturing Technology. – 2018. – Vol. 94. – P. 4553–4563. – DOI: 10.1007/s00170-017-1132-1

14. Reis B.C., dos Santos A.J., dos Santos N.P., Câmara M.A., de Faria P.E., Abrão A.M. Cutting performance and wear behavior of coated cermet and coated carbide tools when turning AISI 4340 steel // International Journal of Advanced Manufacturing Technology. – 2019. – Vol. 105. – P. 1655–1663. – DOI: 10.1007/s00170-019-04392-9

15. Ji W., Zou B., Zhang S., Xing H., Wang Y. Design and fabrication of gradient cermet composite cutting tool, and its cutting performance // Journal of Alloys and Compounds. – 2018. – Vol. 732. – P. 25–31. – DOI: 10.1016/j.jallcom.2017.10.187.

16. KYOCERA SGS Precision Tools Group. Finishing steel faster: cermet vs. carbide, uncoated

vs. coated [Electronic resource]. – 2024 Jan [cited 2025 Jun 21]. – Available from: https://www.kyocera-sgstool.com
17. Singh A., Kumar P. Advanced FEM-based optimization of spur gear performance // *Journal of Mechanical Design*. – 2023. – Vol. 145(8). – DOI: 10.1115/1.4062674
18. Chen Y., Zhao T. Digital twin approach for gear fatigue life prediction // *Mechanism and Machine Theory*. – 2022. – Vol. 172. – P. 104930. – DOI: 10.1016/j.mechmachtheory.2022.104930
18. Seidullayeva O.B., Myrzaliyev D.S., Ibragimova Z.A. *Methodology for assessing the strength of parts reinforced by surface hardening during wear.* Bulletin of L.N. Gumilyov Eurasian National University. Technical Sciences and Technologies Series, No. 2 (147), Astana, 2024, pp. 154–168.

Information about the authors:

Myrzaliyev Darkhan Saparbaevich - candidate of Technical Sciences, Associate Professor, Faculty of Mechanics and Oil and Gas Engineering, NAO «M. Auezov South Kazakhstan Research University», Kazakhstan, Shymkent
Jacek Cieslik - Doctor of Technical Sciences, Professor, AGH University of Krakow, Poland, Krakow
Seidullayeva Oryngul Bakhytkyzy - corresponding author, Doctoral student in Mechanical Engineering, Faculty of Mechanics and Oil and Gas Engineering, NAO «M.Auezov South Kazakhstan Research University», Kazakhstan, Shymkent
Amanov Auezhan - Associate Professor, Tampere University, Finland

Мырзалиев Дархан Сапарбаевич - к.т.н., доцент, факультет «Механика и нефтегазовое дело» НАО «Южно-Казахстанский исследовательский университет им.М.Ауэзова», Казахстан, Шымкент
Яцек Цеслик - д.т.н., профессор «Краковский университет AGH», Польша, Краков
Сейдуллаева Орынгул Бахытовна - автор для корреспонденции, докторант по ОП «Машиностроение», факультет «Механика и нефтегазовое дело», НАО «Южно-Казахстанский исследовательский университет им.М.Ауэзова», Казахстан, Шымкент.
Аманов Ауезхан - доцент, Университет Тампере, Финляндия.

Мырзалиев Дархан Сапарбайұлы - т.ғ.к., доцент, «Механика және мұнайгаз ісі» факультеті, «М.Әуезов атындағы Оңтүстік Қазақстан зерттеу университеті» КЕАҚ, Қазақстан, Шымкент
Яцек Цеслик - т.ғ.д., профессор «AGH Краков университеті», Польша, Краков
Сейдуллаева Орынгүл Бахытқызы - хат-хабар авторы, «Машинажасау» БББ бойынша докторант, «Механика және мұнайгаз ісі» факультеті, «М.Әуезов атындағы Оңтүстік Қазақстан зерттеу университеті» КЕАҚ, Қазақстан, Шымкент
Аманов Ауезхан - доцент, Тампере Университеті, Финляндия