

COMPARATIVE ANALYSIS OF DIFFERENT SOFTWARES IN TERMS OF PARAMETERS OPTIMIZED BY TOPOLOGICAL OPTIMIZATION

JIRI STRUZ, LUKAS HRUZIK, LUKAS KLAPETEK, MIROSLAV TROCHTA

VSB – Technical University of Ostrava, Faculty of Mechanical Engineering, Department of Machine Parts and Mechanism, Ostrava, Czech Republic

DOI: 10.17973/MMSJ.2023_03_2022107

jiri.struz@vsb.cz

In recent years, topological optimization has become a powerful tool for designers and engineers in the technical world. The principle of this method is based on optimizing a given optimization region by minimizing or maximizing one or more objective functions, e.g. stiffness, mass, or natural frequency of the structure. For this reason, software companies developing CAD or FEM software have included topological optimization in their portfolio, either as a modular part of the software or as a dedicated program. In general, users of these programs do not have a mathematical definition of the software background, and it is not clear what variations in calculations may occur when using different software. For this reason, a comparative analysis of the selected software was performed to determine the differences between the results assuming the same boundary conditions and machine parts such as computing time, weight and volume.

KEYWORDS

topology optimization, design region, computing time, commercial software, weight reduction, 3D printing

1 INTRODUCTION

With the relatively rapid development and spread of 3D printing into various technical and non-technical fields, there is significant use of optimization approaches that could not be used in the recent past due to the manufacturing constraints that 3D printing eliminates. One of the optimization methods is the so-called Topological Optimization (TO), based on spatial optimization of material distribution or maximization of target parameters such as stiffness, natural frequency or weight. [Bendsøe, 2013] However, these parameters can be investigated by different approaches. In [Liu, 2015], TO focusing on harmonic frequencies in terms of deformation (MDM), acceleration (MAM), or combinations of them (FM) is investigated. A similar investigation is also addressed in [Bankoti, 2015]. A comparison of the different calculation methods is also presented in [Tyflopoulos, 2020], where the results of the mass and the differences between the maximum stresses are discussed in detail in the conclusions. Comparisons between the results of generative design (GD) and topological optimization (TO) are discussed in [Barbieri, 2022], where a relatively simple comparison of masses, deformations and stresses is made in a similar way to [Tyflopoulos, 2020]. Stresses are addressed in [Yang, 2020], who compares three types of lattice structures (FCC, ECC and VC) that have been proposed based on topology optimization and crystal inspiration. Emphasis is placed on compression testing to achieve comparison of deformation

modes, mechanical properties, and energy absorption capability. This research shows that the VC lattice is stronger than the FCC and ECC lattices in terms of failure stress. The differences between the different methods of calculating TO are discussed quite extensively in [Yago, 2022].

However, the criterion for calculating TO is not necessarily low mass but also an increase in stiffness, e.g. [Jeong, 2021] or, as already mentioned, the natural frequencies [Guo, 2021], [Bhalshankar, 2021]. Other optimized parameters can be shape, area, and parameters in terms of aerodynamics [Munk, 2015], [Ikeya, 2015], [Czarnecki, 2015], hydrodynamics and flow [Ghasemi, 2020].

Another important topic is the optimization of cracked structures, which aims to eliminate manufacturing defects at the design stage. For example, [Lahe Motlagh, 2021] shows that the peridynamic approach gives better support for structures with notches compared to the conventional FEM approach so that the total deformation energy is much lower than when using the FEM approach. The difference between the FEM approach and PD is up to 5% for simple geometries and up to 23% for complex geometries. If we talk about components without notches, the differences are negligible. Topological optimization in dynamic loading is addressed in [Zhu, 2015]. Another equally interesting application is the TO of piezoelectric plate shape in the MATLAB software environment [Homayouni-Amlashi, 2021].

The TO process can be broadly divided into two main parts – pre-processing and postprocessing. Pre-processing involves processing the design in available CAD software. These data are then used in FEM software, defining boundary conditions. This is followed by discretization of the design space and evaluation of the analysis results. As part of the evaluation, the designer has to decide whether or not to use TO and which software should be used for the problem. After the successful TO calculation, the model has to be modified for production, which can be realized by standard methods or, more often nowadays, by more modern methods such as additive manufacturing (AM). The TO process can be schematically outlined according to Figure 1. A similar process scheme is also considered by [Tyflopoulos, 2022].

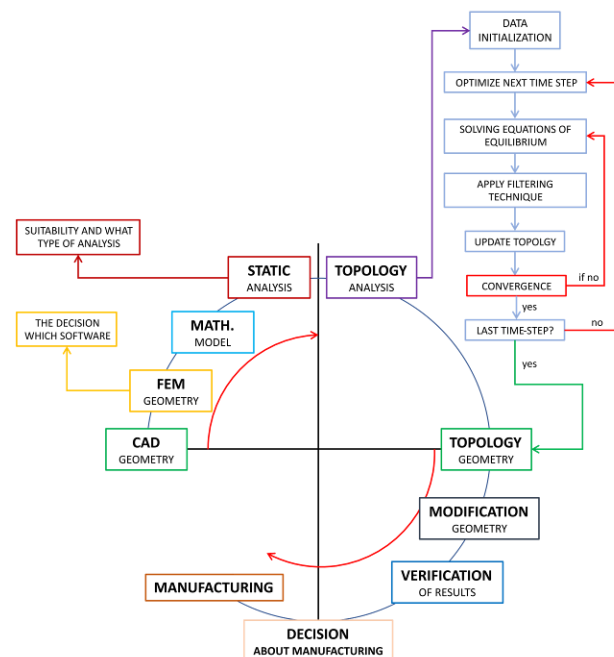


Figure 1. Process diagram

The above shows that TO is a very sophisticated tool that leads to more interesting designs in terms of design and material saving. Besides, it leads to design solutions that are lighter, more

robust, and also save development time. Related to this is the emphasis on TO during university teaching, as TO is proving to be an issue adaptable to teaching either as a stand-alone course or as a whole field of study. Hence the need to educate senior practising engineers in the subject matter. [Tyflopoulos, 2021]

2 MATHEMATICAL BACKGROUND AND METHODOLOGY

In this research, we decided to perform topological optimization of the considered component in the four most commonly used software types with modules for TO – Ansys, [Altair Inspire](#), Inventor and Solidworks. The last-mentioned software uses the SIMP (Solid Isotropic Model with Penalty) calculation method. In the case of Solidworks, it is traceable [SOLIDWORKS Help, 2022] that the mathematical basis of topological optimization is the energy equation:

$$E_{(\rho_e)} = \rho_e^p \cdot E_0 \quad (1)$$

In order to determine the global stiffness of the system, the following algorithm is considered:

$$K_{SIMP(\rho)} = \sum_{e=1}^n [\rho_{min} + (1 - \rho_{min}) \cdot \rho_e^p] K_e \quad (2)$$

The most commonly used equation is the equation for maximizing the stiffness or minimizing the compliance of the system:

$$\min C(\{\rho\}) = \sum_{e=1}^N (\rho_e)^p [K_e] [u_e]^T [u_e] \quad (3)$$

For example, according to [Sotola, 2021], the mathematical subtext of Ansys software is identical to the Nastran algorithm. In this case, the basis of TO is again the energy equation (1), and the minimization of the compliance uses the procedure given in (3). In general, the mathematical background of TO problems can be considered very similar across the software. However, the comparison is meaningful in the sense of the differences in the downstream algorithms, e.g., finite element mesh generation, etc. For example, [Bernd, 2007] compares different software with an integrated FEM environment in terms of the number of elements and nodes as well as individual results such as deformation or stresses.

2.1 Methodology

The methodology chosen in this research is to select one component to be subjected to static analysis based on defined boundary conditions. The static analysis results will serve as input data to TO, within which the design end exclusion regions will be defined. After TO, the number of elements and nodes, the computation time, the resulting weights, and the obtained shapes will be compared.

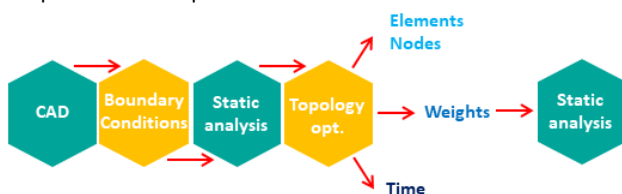


Figure 2. Schematic of the evaluation process

3 CASE STUDY AND BOUNDARY CONDITIONS

The research subject is a welded bracket used to reinforce the frame of a measuring device. This bracket was chosen for its simple loading and its simple construction. The original construction can be seen in Figure 3.

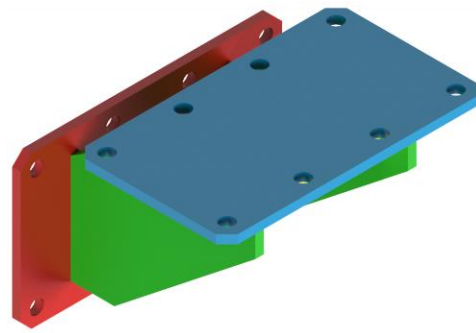


Figure 3. Original console design

Before defining the boundary conditions, a CAD model is first created to enable appropriate material removal. In other words, it is necessary to modify the design region so that the computation algorithm is minimally constrained by human intervention. The solution depicted in Figure 3 does not satisfy this condition, and the original model needs to be modified before calculation.

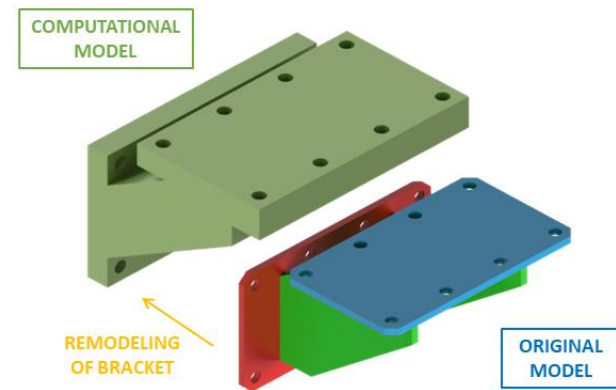


Figure 4. Creating the computational model of the bracket

Figure 4 shows the computational model in green with the relevant additions in the design region and the original model for easy comparison. On the CAD model defined in this way, the definition of boundary conditions, including the definition of forces and supports, as well as the regions that are included in the calculation and that need to be preserved, can be performed. The definition of the boundary conditions is shown in Figure 5. First, the material removal regions were decided. Next, the regions where material removal is prohibited were defined. In this case, these regions replicate the boundary conditions regarding loads and supports. The “fixed support” (in blue) was used, removing all degrees of freedom. A load (red) of 5,000 N was subsequently applied to the individual bearing surfaces. The other bracket parts (grey) were considered a “design region”.

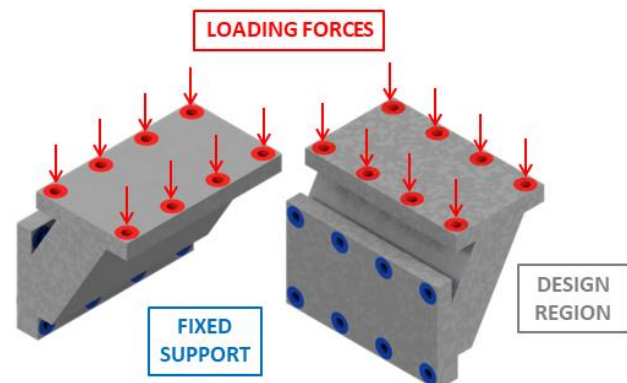


Figure 5. Definition of boundary conditions

As mentioned above, the boundary conditions were considered to make them as simple as possible to define. In order to perform the analysis correctly, it is also necessary to define the material. Standard available steel was considered in all analyses. The mechanical properties required for the linear analysis are given in Table 1.

Density [kg·m ³]	Poisson ratio [-]	Modulus of elasticity [GPa]
7850	0,3	210

Table 1. Mechanical properties of structural steel

Another possible parameter that can affect the results is the finite element mesh. In order to get a reasonably accurate view of how each software works, it was decided to set the same element size of 7 mm when generating the finite element mesh. At the same time, the default method of mesh generation will be set. The authors of the paper hope that this setting will lead to a better understanding of the behaviour of the different software types. The only difference is the approach of Autodesk Inventor software, where the element size is not defined, but its average and minimum size. For this reason, the software-defined default mesh settings have been retained. The finite element mesh statistics are summarized in Table 2.

	Solidworks	Ansys Workbench 2020	Autodesk Inventor	Altair Inspire
Elements	74471	95142	*7379	35675
Nodes	109083	137501	*12593	57924

Note: Refers to the default mesh settings specified by the software provider.

Table 2. Number of elements and nodes

4 ANALYSES RESULTS

4.1 Static analysis

Figure 6 to Figure 13 show the results of static analyses, which are a necessary precursor to topological optimization. The available results indicate that the maximum von Mises stress in the Solidworks case is approximately 3.99 MPa in the shape transition, while the deformation in the Y-axis direction is 0.0050 mm. In the case of Ansys Workbench software, the situation is similar. The maximum stress is approximately 4.11 MPa at the shape transition, similar to Solidworks. The Y-axis deformation reaches a maximum value of 0.0052 mm according to this calculation. The results of the analysis in Inventor and Altair Inspire show similar results to the previous cases. The maximum stress is again in the area of the shape transition of approximately 4.22 MPa and 4.17 MPa, respectively. The maximum deformation in the Y-axis is 0.0049 mm. In the case of Altair Inspire, the Y-axis deformation is 0.0051 mm.

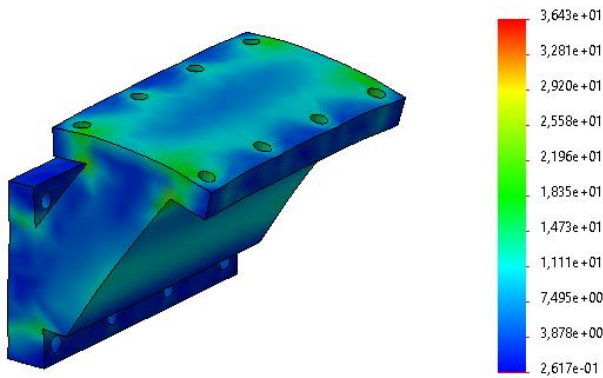


Figure 6. Results from static analysis in Solidworks – von Mises stress

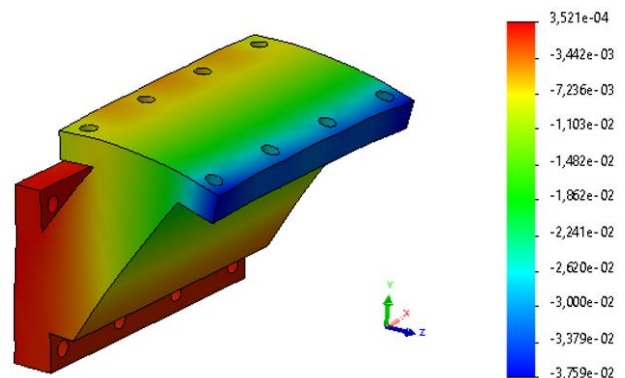


Figure 7. Results from static analysis in Solidworks – Y-axis deformation

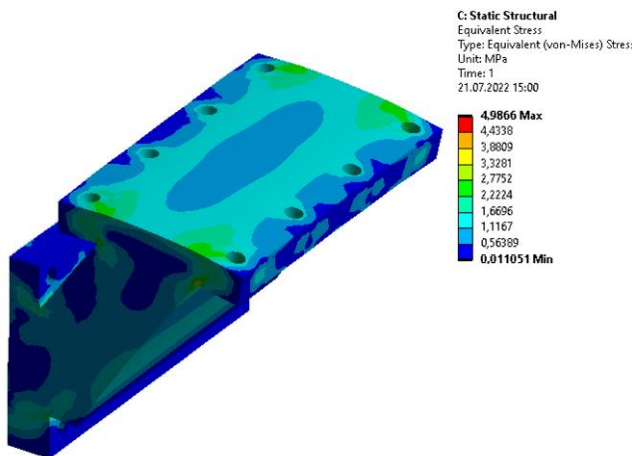


Figure 8. Results from static analysis in Ansys – von Mises stress

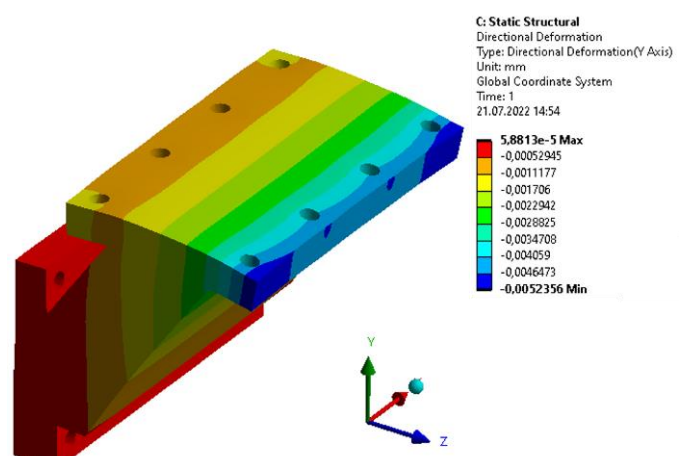


Figure 9. Results from static analysis in Ansys – Y-axis deformation

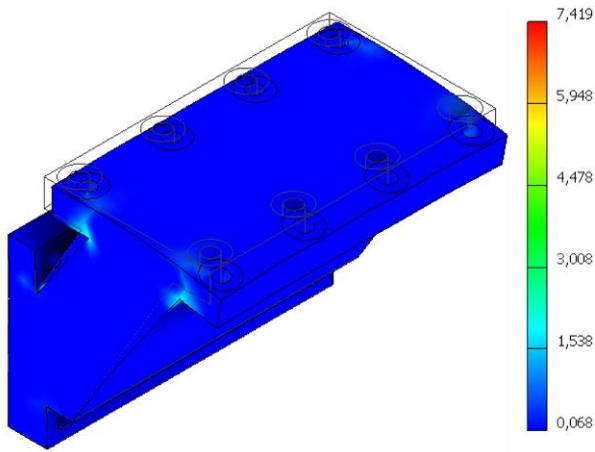


Figure 10. Results from static analysis in Inventor – von Mises stress

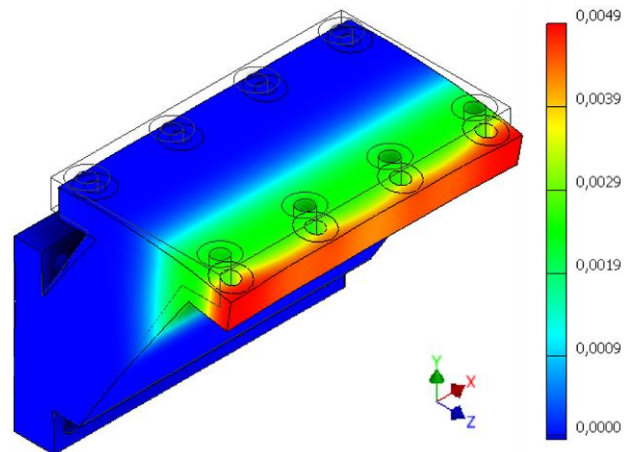


Figure 11. Results from static analysis in Inventor – Y-axis deformation

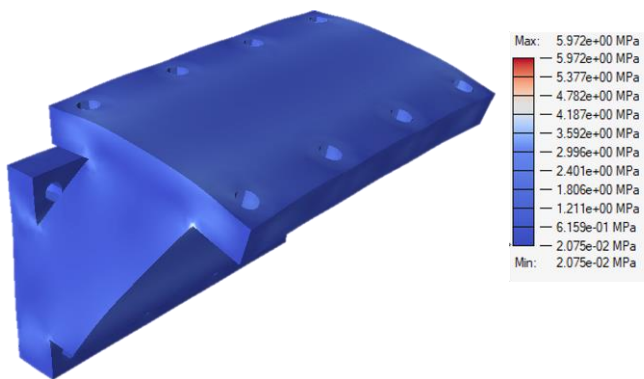


Figure 12. Results from static analysis in Inspire – von Mises stress

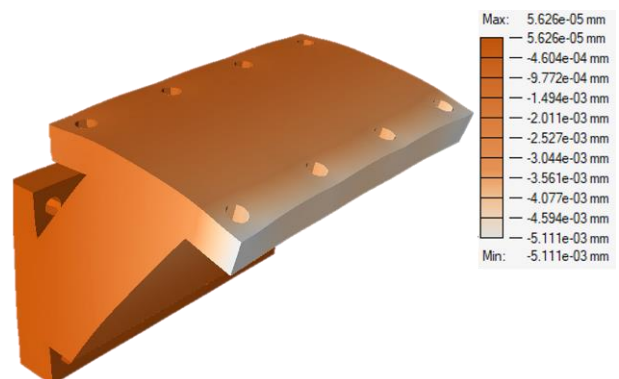


Figure 13. Results from static analysis in Inspire – Y-axis deformation

	Solidworks	Ansys Workbench 2020	Autodesk Inventor	Altair Inspire
Von Mises stress [MPa]	3.99	4.11	4.22	4.17
Y-axis deformation [mm]	0.0050	0.0052	0.0049	0.0051

Table 3. Comparison of stress and deformation results

From the above results, it can be concluded that the differences between them are minimal. However, care must be taken in setting up the analyses. As mentioned above, Autodesk Inventor allows setting the average and minimum element size in the case of a mesh. Solidworks, on the other hand, distinguishes the definition of forces, either individually or in total. It is the first option that has been set as the default definition method. It was, therefore, necessary to redefine this setting. Another interesting fact is that in the case of Autodesk Inventor and Solidworks software, static analysis takes place in the background when solving topological optimization. Ansys, on the other hand, requires a static analysis to be performed first, which must then be fed into the topological optimization module.

4.2 Topology optimization

In the previous sections, the boundary conditions, i.e. loads and supports, are described. Information on the design of the region is also given; see Figure 5. Another criterion to be defined in topological optimization is the objectives and constraints of topological optimization. Table 4 gives an overview of the criteria that can be defined in topological optimization in each

program. Solidworks makes it possible to define a criterion based on maintaining maximum stiffness while reducing weight. It is also possible to set the minimization of displacement or mass. Autodesk Inventor, on the other hand, in the shape generator module, automatically sets a criterion for maintaining maximum stiffness while reducing mass. The opposite is the case with Ansys Workbench, which allows for the definition of many criteria. It is mainly because Ansys software is focused on FEM analysis, whereas Autodesk Inventor or Solidworks are CAD programs mainly focused on structural design and engineering documentation. It should also be noted here that different calculation methods can be defined within Ansys topological optimization:

- Topology optimization - Density Based
- Lattice optimization
- Topology optimization - Level set based
- Topology optimization - Morphing

	Solidworks	Ansys Workbench 2020	Autodesk Inventor	Altair Inspire
Maximum stiffness while reducing weight	+	+	+	+
Minimizing displacement	+	+		
Minimization of mass	+	+		+
Minimization of volume		+		
Minimization of reaction force		+		
Centre of gravity		+		
Moment of inertia		+		
Global von Mises stress		+		
Local von Mises stress		+		
Criterion		+		
Maximum frequency				+

Table 4. Overview of criteria definition options

By default, the method is usually called Density based. This method was used as the default method in the development of the topology optimizations in this paper. So far, the most commonly used criterion is to maximize the stiffness while reducing the weight, the principle of which lies in defining a target weight. Altair Inspire makes it possible to set the criteria listed in Table 4. However, it is worth noting that in addition to standard criteria such as maximizing stiffness while reducing weight, the following frequency constraints can also be defined:

- Maximum frequency
- Defining the minimum frequency

Similarly, it is also possible to define a frequency or stress limitation or safety factor in Solidworks.

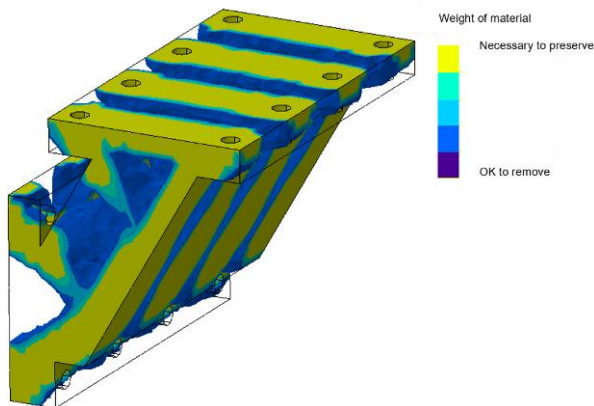


Figure 14. Topology optimization results from Solidworks

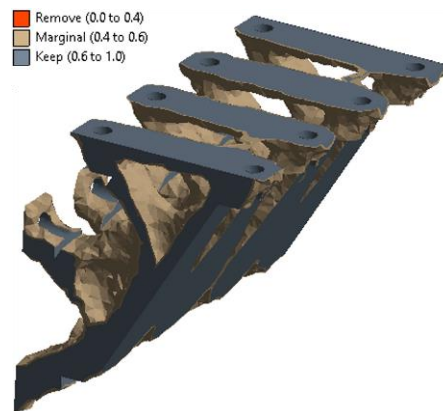


Figure 15. Topology optimization results from Ansys Workbench

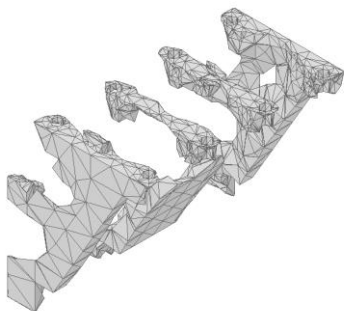


Figure 16. Topology optimization results from Autodesk Inventor corresponding static analysis mesh settings

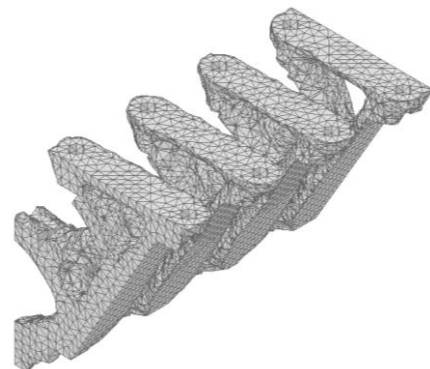


Figure 17. Topology optimization results from Autodesk Inventor corresponding to the default shape generator mesh settings



Figure 18. Topology optimization results from Altair Inspire

It is important to note that static analysis in Inventor and Solidworks is not linked to topological optimization and appears to be done in the background. This fact is evident in the fact that when setting up the shape generator in Inventor, it was again necessary to generate a mesh. The default mesh setting for the shape generator is many times finer than that of the static analysis, as can be seen in Figure 16 and Figure 17.

5 DISCUSSION

In the above chapters, the results are given based on the decision process and topological optimization process presented in Figure 1 and Figure 2 across different software. These results indicate that the differences in the results obtained during the static analysis are quite small, in the order of ten thousandths. In all these cases, the maximum von Mises stress was evident in the shape transition indicated in Figure 19 and also at the locations of the defining supports. On the other hand, the maximum deformations were evident at the edges of the bracket.

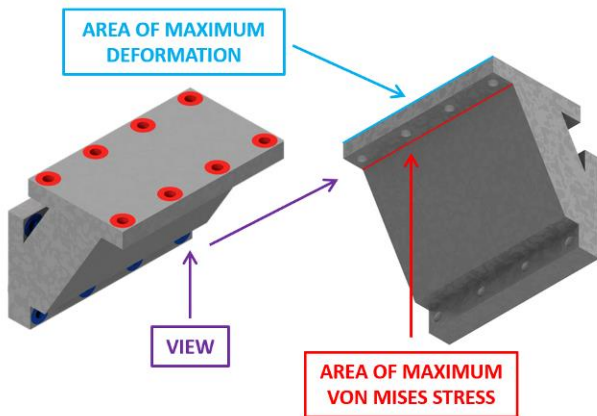


Figure 19. Representation of areas of maximum deformation and von Mises stress

The finite element mesh, where the emphasis was on defining the element size of 7mm, also came under scrutiny. The exception is Autodesk Inventor's shape generator, where the average element size and minimum element size are defined within the mesh settings. Table 2 indicates that Solidworks software shows a smaller number of elements and nodes than Ansys FEM software when defining the same element size. Similarly, Altair Inspire shows a much lower number of elements and nodes than the previous two cases. It should be noted here that the results presented in Table 3 are with relatively small deviations, and also, the computation times were not significantly different. The finite element mesh was also checked after the topological optimization. The data suggest that there is

a background adjustment in the number of elements or nodes. This can be seen in Figure 20, which shows that in the case of topological optimization in Solidworks, the number of nodes is reduced by approximately 85% even though the same network parameters are defined. A similar situation can also be seen in Altair Inspire, where there is a dramatic 91% reduction in the number of nodes and a 32% reduction in the number of elements. The opposite situation is true for Ansys, as the number of nodes remains the same and only the number of elements changes by about 11%. This phenomenon is also noticeable when using the "Level set based" calculation method. In Autodesk Inventor, on the other hand, the number of nodes increased by a factor of 7.9 and the number of elements by a factor of 9.

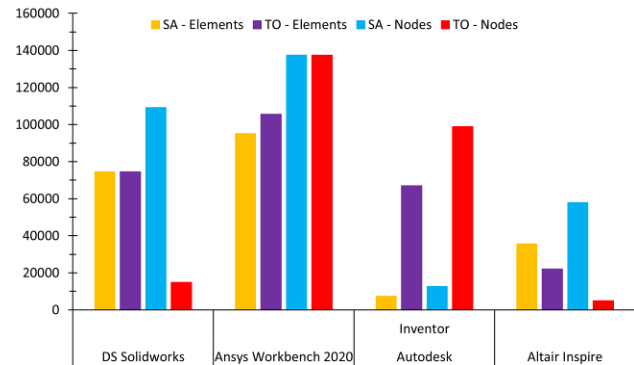


Figure 20. Comparison of the number of elements and nodes

Another observation is that many software programs perform a static background analysis before starting topological optimization. In our listing, these are Autodesk Inventor and Solidworks software. Ansys software, on the other hand, requires the static analysis to be manually fed into the topological optimization module. In contrast, Altair Inspire allows topological optimization to be run regardless of whether static analysis has been performed. However, the generally accepted practice is to run the static analysis and define the desired element size before starting the topological optimization. Figure 20 shows that despite this, the number of elements and nodes is reduced.

The next evaluated parameter was the computation time. The calculation of the topological optimization took the longest time in Ansys Workbench and Autodesk Inventor (assuming the use of the network generated for topological optimization). The calculation was relatively fast using Altair Inspire and Solidworks, see Table 5.

Solidworks	Ansys Workbench 2020	Autodesk Inventor	Altair Inspire
133	509	72/386	104

Table 5. Computing time [s]

Figure 14 to Figure 18 show the results of the topological optimizations. The graph below indicates that the greatest material removal occurs in the case of Altair Inspire, with a resulting weight of 8.616 kg. Solidworks has reached mass 8,863kg. A weight of 9.228 kg was then achieved by Ansys FEA software using a calculation method called "Density-based". The mass achieved using the "Level set based" method is 8.808 kg. Autodesk Inventor achieved a weight of 8.720 kg.

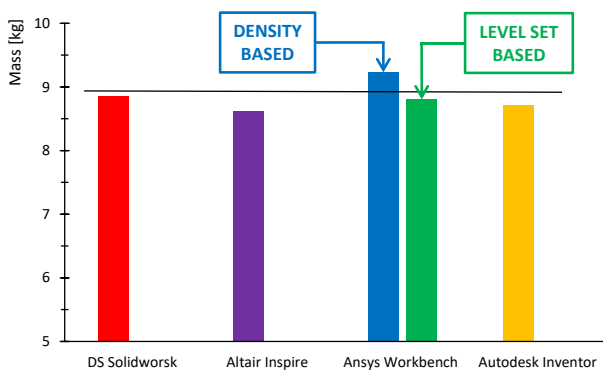


Figure 21. Mass results

Figure 14 to Figure 18 note that in some cases (Altair Inspire and Autodesk Inventor), the bracket has been completely split into four parts. In Ansys, a small bridge between the different support parts is maintained, even when using the “Level set based” method. The same region is marked as “suitable for removal” in Solidworks. The percentage difference from the predicted value is shown in the following table.

Solidworks	Ansys Workbench 2020	Autodesk Inventor	Altair Inspire
0,5%	4% / 1.1%	2%	3%

Table 6. Percentage comparison of results against the expected value

6 CONCLUSIONS

The research presents a collection of information that can assist designers, researchers and educators working on topological optimization problems and can help them select the appropriate software for a specific task. This paper discusses topological optimizations in terms of computation time, numbers of elements and nodes, their adjustment before starting the topological optimization, and the actual results in terms of both shape and final weight. The results of the static analyses are compared, and it can be said that the differences in terms of deformation results are in the order of thousandths and in the order of tens in the case of von Mises stresses. The above investigation briefly shows:

- The lowest masses were achieved by using **Altair Inspire**, which furthermore allows the results of the topological optimization to be modified, especially in terms of design, by using, for example, the PolyNurb function, which can be used to create very interesting design solutions using bionic structures;
- **Autodesk Inventor** achieved the lowest weight in second place, with the requirement to use a significantly different finite element mesh;
- **Ansys Workbench** achieved a weight of 9.228 kg using the “density-based” method. In contrast, it achieved a weight of 8.808 kg using the “Level set based” method. The broader range of methods by which topological optimization can be performed is also positive;
- **Solidworks** program reached a weight of 8,863kg.

Another significant finding is that many software types will adjust the mesh before starting the topological optimization on their background despite defining the same finite element mesh parameters. In particular, the smallest difference between the number of elements and nodes can be seen in the case of Ansys Workbench software.

ACKNOWLEDGEMENTS

This article was developed within the project SP2022/10-“Experimental and Computational Methods of Dimensioning Machinery Components 2022”.

REFERENCES

- [Bendsøe, 2013] BENDSØE, Martin P. a Ole SIGMUND. Topology Optimization Theory, Methods, and Applications. 2nd ed. Germany: Springer Berlin Heidelberg, April 2013. ISBN 3-540-429992-1.
- [Liu, 2015] LIU, Hu, Weihong ZHANG a Tong GAO. A comparative study of dynamic analysis methods for structural topology optimization under harmonic force excitations. Structural and Multidisciplinary Optimization. 2015, 51(6), 1321-1333. ISSN 1615-1488. Available from: doi:10.1007/s00158-014-1218-4
- [Bankoti, 2015] BANKOTI, Saurabh, Akash DHIMAN a Anadi MISRA. Comparative Analysis of Different Topological Optimization Methods with ANSYS. INTERNATIONAL JOURNAL FOR RESEARCH IN EMERGING SCIENCE AND TECHNOLOGY. 2015, 2(4), 84-89. ISSN 2349-7610.
- [Tyflopoulos, 2020] TYFLOPOULOS, Evangelos a Martin STEINERT. Topology and Parametric Optimization-Based Design Processes for Lightweight Structures. 10. 2020. ISSN 2076-3417. Available from: doi:10.3390/app10134496
- [Barbieri, 2022] BARBIERI, Loris a Maurizio MUZZUPAPPA. Performance-Driven Engineering Design Approaches Based on Generative Design and Topology Optimization Tools: A Comparative Study. 12. 2022. ISSN 2076-3417. Available from: doi:10.3390/app12042106
- [Yago, 2022] YAGO, Daniel, Juan CANTE, Oriol LLOBERAS-VALLS a Javier OLIVER. Topology Optimization Methods for 3D Structural Problems: A Comparative Study: A Comparative Study. Archives of Computational Methods in Engineering. 2022, 29(3), 1525-1567. ISSN 1886-1784. Available from: doi:10.1007/s11831-021-09626-2
- [Jeong, 2021] JEONG, Gil-Eon. Topology Optimization of Deformable Bodies with Linear Dynamic Impact and Frictionless Contact Condition. 11. 2021. ISSN 2076-3417. Available from: doi:10.3390/app112210518
- [Guo, 2021] GUO, Jincheng a Huaping TANG. Stiffness-Oriented Structure Topology Optimization for Hinge-Free Compliant Mechanisms Design. 11. 2021. ISSN 2076-3417. Available from: doi:10.3390/app112210831
- [Bhalshankar, 2021] BHALSHANKAR, Sudarshan. Dynamic Analysis of an Lower Control Arm Using Harmonic Excitation for Investigation Dynamic Behaviour. In: EasyChair [online]. 2021, 27 June 2021, s. 1-19 [cit. 2022-06-30]. Available from: <https://easychair.org/publications/preprint/8G3j>
- [He, 2018] HE, Z. C., Y. WU a Eric LI. Topology optimization of structure for dynamic properties considering hybrid uncertain parameters. Structural and Multidisciplinary Optimization. 2018, 57(2), 625-638. ISSN 1615-1488. Available from: doi:10.1007/s00158-017-1769-2
- [Munk, 2015] MUNK, David J., Gareth A. VIO a Grant P. STEVEN. Aerothermoelastic Structural Topology Optimisation for a Hypersonic TransportAircraft Wing. In: LI, Qing, Grant P. STEVEN a Zhongpu (Leo) ZHANG. Aerothermoelastic Structural Topology Optimisation for a Hypersonic TransportAircraft Wing: Proceedings of the 11th World Congress of Structural and Multidisciplinary Optimization (WCSMO-11) [online]. Sydney, Australia, 2015, June 2015, s. 31-36 [cit. 2022-06-30]. ISBN 978-0-646-94394-7. Available from:

<https://www.aeromech.usyd.edu.au/WCSMO2015/documents/Advances%20in%20Structural%20and%20Multidisciplinary%20Optimization.pdf>

[Ikeya, 2015] IKEYA, Kenichi a Masatoshi SHIMODA. Multi-objective Free-form Optimization for Shape and Thickness of Shell Structures with Composite Materials. In: LI, Qing, Grant P STEVEN a Zhongpu (Leo) ZHANG. Advances in Structural and Multidisciplinary Optimization: Proceedings of the 11th World Congress of Structural and Multidisciplinary Optimisation (WCSMO-11), June 7-12, 2015, Sydney, Australia [online]. Sydney, Australia, 2015, June 2015, s. 231-236 [cit. 2022-06-30]. ISBN 978-0-646-94394-7. Available from: <https://www.aeromech.usyd.edu.au/WCSMO2015/documents/Advances%20in%20Structural%20and%20Multidisciplinary%20Optimization.pdf>

[Czarnecki, 2015] CZARNECKI, Sławomir, Radosław CZUBACKI, Tomasz LEWINSKI a Paweł WAWRUCH. Topology optimization of continuum structures made of non-homogeneous materials of isotropic or cubic symmetry. In: LI, Qing, Grant P STEVEN a Zhongpu (Leo) ZHANG. Advances in Structural and Multidisciplinary Optimization: Proceedings of the 11th World Congress of Structural and Multidisciplinary Optimization (WCSMO-11) [online]. Sydney, Australia, 2015, June 2015, s. 83-88 [cit. 2022-06-30]. Available from: <https://www.aeromech.usyd.edu.au/WCSMO2015/documents/Advances%20in%20Structural%20and%20Multidisciplinary%20Optimization.pdf>

[Ghasemi, 2020] GHASEMI, Ali a Ali ELHAM. A Novel Topology Optimization Approach for Flow Power Loss Minimization Across Fin Arrays. 13. 2020. ISSN 1996-1073. Available from: doi:10.3390/en13081987

[Lahe Motlagh, 2021] LAHE MOTLAGH, Peyman a Adnan KEFAL. Comparative Study of Peridynamics and Finite Element Method for Practical Modeling of Cracks in Topology Optimization. 13. 2021. ISSN 2073-8994. Available from: doi:10.3390/sym13081407

[Zhu, 2015] ZHU, Mu. TOPOLOGY OPTIMIZATION OF FRAME STRUCTURES: DESIGN FOR CONSTRUCTABILITY AND

STOCHASTIC DYNAMIC LOADS. Baltimore, Maryland, 2015. Dissertation thesis. John Hopkins University. Supervisor graduate work Dr. James K. Guest.

[Homayouni-Amlashi, 2021] HOMAYOUNI-AMLASHI, Abbas, Thomas SCHLINQUER, Abdenbi MOHAND-OUSSAID a Micky RAKOTONDRABE. 2D topology optimization MATLAB codes for piezoelectric actuators and energy harvesters. Structural and Multidisciplinary Optimization. 2021, 63(2), 983-1014. ISSN 1615-1488. Available from: doi:10.1007/s00158-020-02726-w

[Tyflopoulos, 2022] TYFLOPOULOS, Evangelos a Martin STEINERT. A Comparative Study of the Application of Different Commercial Software for Topology Optimization. 12. 2022. ISSN 2076-3417. Available from: doi:10.3390/app12020611

[Tyflopoulos, 2021] TYFLOPOULOS, Evangelos, Cecilia HASKINS a Martin STEINERT. Topology-Optimization-Based Learning: A Powerful Teaching and Learning Framework under the Prism of the CDIO Approach. 11. 2021. ISSN 2227-7102. Available from: doi:10.3390/educsci11070348

[SOLIDWORKS Help, 2022] SOLIDWORKS Help: SIMP Method for Topology Optimization. SOLIDWORKS Help [online]. Dassault Systèmes, 2022 [cit. 2022-06-30]. Available from: https://help.solidworks.com/2021/english/SolidWorks/cworks/c_simp_method_topology.htm

[Sotola, 2021] SOTOLA, Martin, Pavel MARSÁLEK, David RYBANSKY, Martin FUSEK a Dusan GABRIEL. Sensitivity Analysis of Key Formulations of Topology Optimization on an Example of Cantilever Bending Beam. 13. 2021. ISSN 2073-8994. Available from: doi:10.3390/sym13040712

[Bernd, 2007] BERND, Roith, Alexander TROLL a Frank RIEG. Integrated Finite Element Analysis (FEA) in Three-Dimensional Computer Aided Design Programs (CAD) - Overview and Comparison. In: BOCQUET, J-C. Conference on Engineering Design, Paris, France [online]. Paris, France, 2007, s. 99-100 [cit. 2022-06-30]. ISBN 1-904670-02-4. Available from: <https://www.designsociety.org/publication/25371/Integrated+Finite+Element+Analysis+%28FEA%29+in+Three-Dimensional+Computer+Aided+Design+Programs+%28Cad%29+-+Overview+and+Comparison>

CONTACTS:

Ing. Jiri Struz

VSB-Technical University of Ostrava, Department of Machine Parts and Mechanism
17. listopadu 2172/15, 708 00, Ostrava-Poruba
jiri.struz@vsb.cz