

# Topology Optimization of a Steering Knuckle

**Mailapalli Kumar Raja<sup>1</sup>, J Naveen Kumar<sup>2</sup>, B Poorna Chandu<sup>3</sup>,  
A Uday Kiran<sup>4</sup>, L Prem Kumar<sup>5</sup>, J Gowri Naidu<sup>6</sup>, B Koushik<sup>7</sup>**

<sup>1</sup> Assistant Professor, Dept of Mechanical Engineering, Sanketika Vidya Parishad Engineering College, Visakhapatnam, India.

<sup>2, 3, 4, 5, 6 & 7</sup> BTech 4th Year, Dept of Mechanical Engineering, Sanketika Vidya Parishad Engineering College, Visakhapatnam, India

## ABSTRACT

Steering knuckle is an important component from safety point of view since they are subjected to large amount of stresses. Hence to study the stress pattern in steering knuckle. ANSYS static structural workbench model is used. For analysis purpose basic model is prepared by picking data from available literature. For numerical analysis steering knuckle is prepared by using the software CATIA V5 and the stress analysis is done by FEM in ANSYS. Topology optimization have been performed for optimization of the volume and reduce the materials from the steering knuckle by using topology optimization module in ANSYS. Comparison between the results from post and pre-optimization will be done and discussed.

**Keywords:** Catia V5, Ansys V19.0, Solid works.

## 1. INTRODUCTION

Structural optimization is concerned with maximizing the utility of a fixed quantity of resources to fulfil a given objective. There are three categories of structural optimizations are exist: Shape, Size and Topology. The structural topology optimization is the most general of the three categories yielding information on the number, location, size and shape of openings within a continuum. Topology optimization was used to derive the optimal number, location and shape of holes. The walls were modelled as simple rectangular plates and optimized for vertical and horizontal loading combinations.

## 2. MODELING OF STEERING KNUCKLE

### ➤ Solid modelling

Solid modelling can be created and modified very quickly when compared to other types of modelling. Three approaches that are generally used for creating a solid model are:

- Constructive Solid Geometry
- Boundary representation
- Hybrid scheme



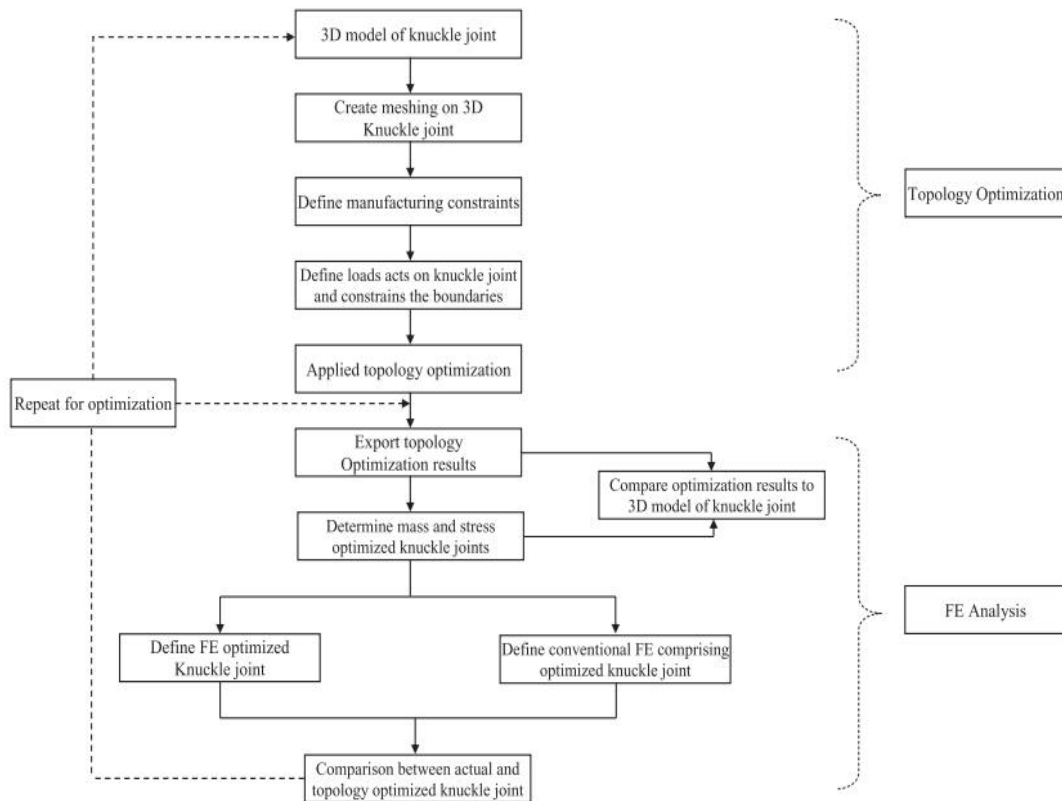
Fig1. Isometric view of a steering knuckle

➤ **Part modeling**

It is the process of make a basic prototype by modeling each individual parts separately and Then assembled together to create model of the result

### 3.WORKING METHODOLOGY

First we designed a 3D model of the steering knuckle in the CATIA and design a meshed model of the component in ANSYS by FEM method. Converts the meshed model into static structure and then apply the existing component loads on the static structural boundary conditions with those loads, there will be some stresses are developed. With this stresses there are some deformation is occurred. Where there is less amount of the stresses are distributes, we will remove the material from that region and then the final design is produced. After this we will place the design in topology and ANSYS will automatically adopted this design to topology optimization. After that we will test the design for is this design tolerating the initial loads are not. Again applied the loads and there will be some deformation is occurred and finally make sure that the design is safe or not. The design is safe, our design is successful. For suppose the design is failed, again we will do the process again for safe design.



methodology flow chart

Working

#### 4.ANALYSIS OF STEERING KNUCKLE

##### ➤ 1.Build Geometry

Construct a 2D or 3D design in CATIA for the object to be modelled and tested using the work plane coordinate system in ANSYS.

##### ➤ 2. Define Material Properties

Now that part exists, define a study of the necessary material that compose the object being modelled. This includes thermal and mechanical properties

##### ➤ 3. Generate Mesh

At this point ANSYS understands the makeup of the part. Here we take 3D model. With this model generate the meshed model. Select 10mm as the size of the element and automatically the FEM meshed model will formed. Now define how the modelled system should be broken down into finite pieces.

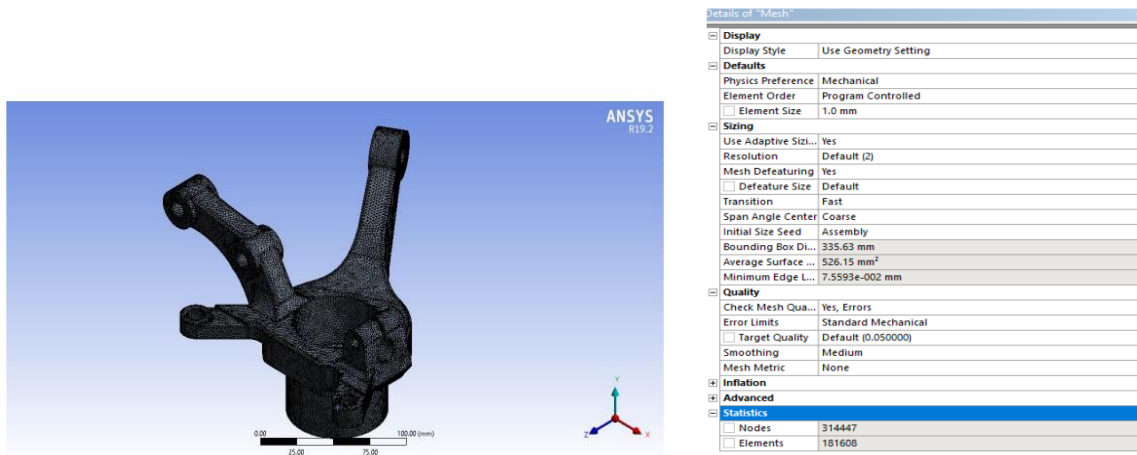


Fig 2. Meshed model

➤ **4. APPLY LOADS**

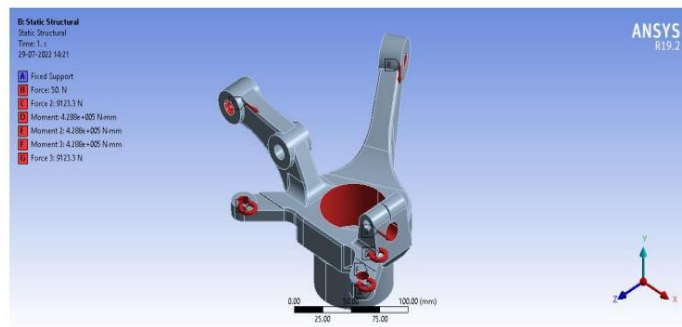


Fig 3. Boundary conditions

Object Name	Topology Optimization (C5)
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Topology Optimization
Solver Target	Mechanical APDL
Options	
Generate Only	Input No

The loads applied on the existing knuckle like point load, twisting load, axial load etc.. will be applied on the structure manually to analysing the stress theory. After meshed model is completed, ANSYS will adopted the meshed model into structural model. Once the system is fully designed, the last task is to burden the system with constraints, such as physical loadings or boundary conditions. Static structural analysis theory will be performed and applied fixed loads 50N force at point [B], 9123.3N force at point

[C],  $4.288e+005$  N-mm twisting moments at point [D][E][F], and 9123.3N force at point [G]. Those points are connected with the bolts and shafts and absorbs the maximum amount of the stress. we should not able to remove the material from those points.

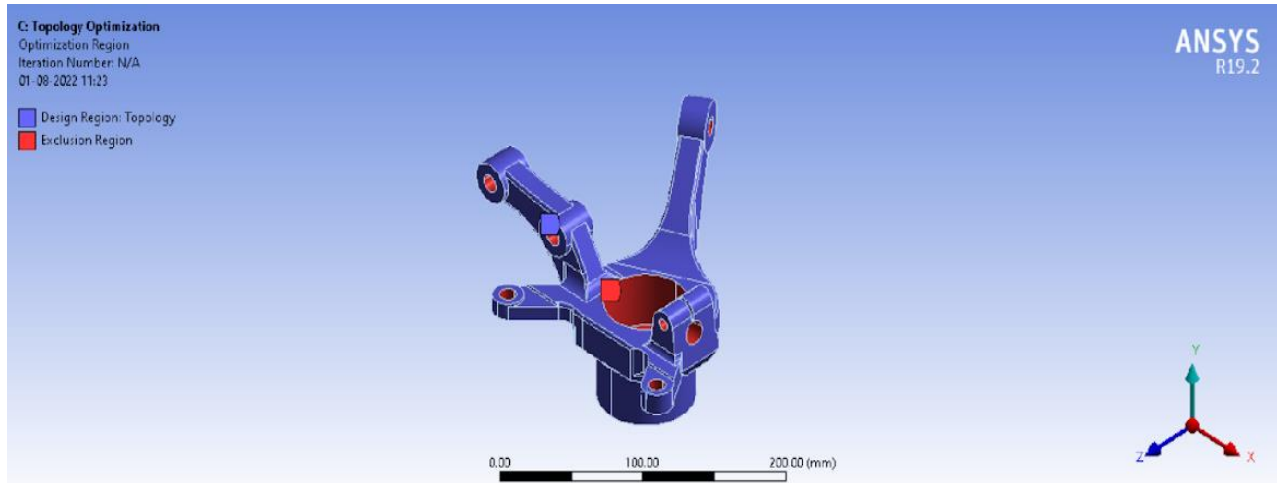


Fig 4. Optimization region

The blue coloured region is designed region for performing the topology and the red coloured region is exclusion region because that connects with the bolts and shafts that distributes the maximum amount of the stress.

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	2810	kg m <sup>-3</sup>
4	Isotropic Secant Coefficient of Thermal Expansion		
6	Isotropic Elasticity		
12	Tensile Yield Strength	5.03E+08	Pa
13	Tensile Ultimate Strength	5.72E+08	Pa
14	Isotropic Thermal Conductivity	204	W m <sup>-1</sup> C <sup>-1</sup>

Table 1. Aluminium alloy properties

We should take aluminium alloy properties for the topology. Once these all processes are done, the main step in this entire development will start that is topology optimization. For this process we should take the initial stresses applied all over the knuckle with different conditions in the first basic steering knuckle. And then apply topology optimization option in ANSYS and the stresses will apply on the entire structural model automatically.

## 5.RESULTS AND DISCUSSIONS

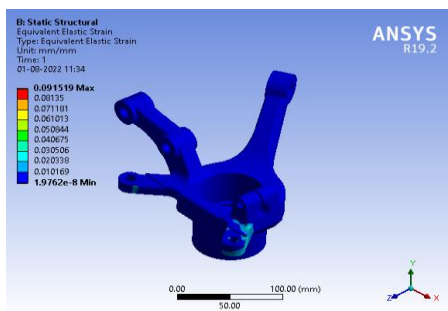


Fig 5. Deformation

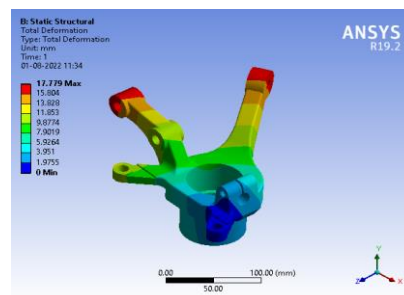


Fig 6. Equivalent elastic strain

There are developed some stresses with the total loads we applied on the static structure manually. Deformation will happen by the developed stresses from 0 to 17.799Max. The blue coloured region will be 0 and red coloured region produces maximum amount of stresses. The red coloured region is fixed points so it is safe. The equivalent elastic strain is located at blue region as minimum. No strains are located at red region so it is also safe.

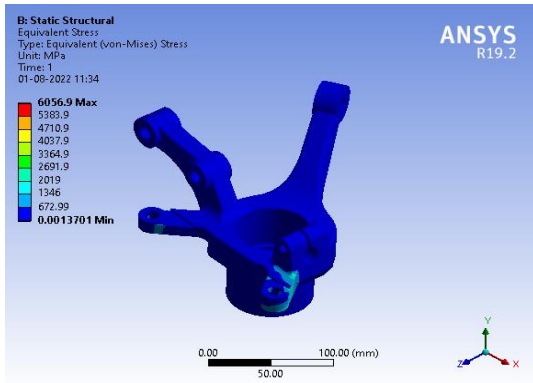


Fig 7. Equivalent stress

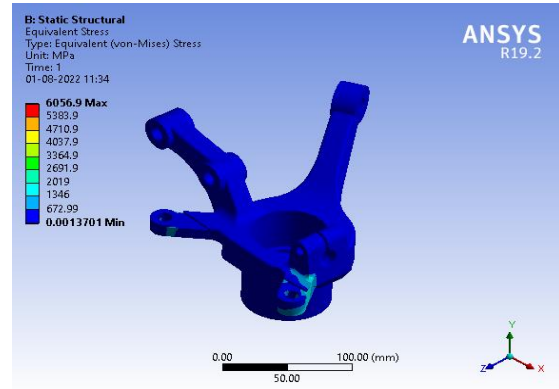


Fig 8. Equivalent stress

Take Von-Mises stress theory and performed equivalent stresses will be done and these two conditions have produced the equivalent stresses at blue region, no red region is shown in the static structure. All results are optimum with this and our design is safe by the loads applied on the static structure from existing component loads.

Place this design in Topology in ANSYS then the material will be removed automatically

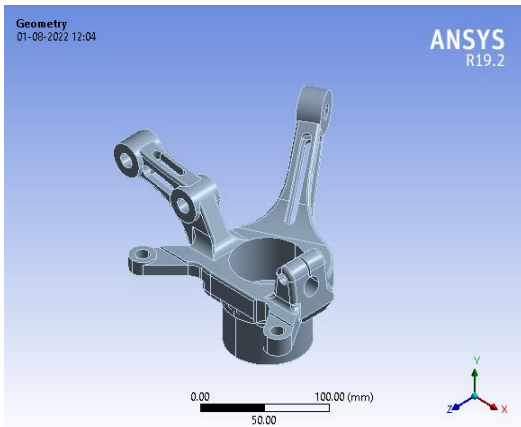


Fig 9. Topology density

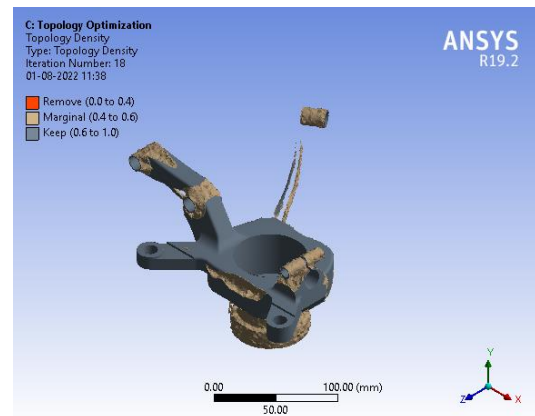


Fig 10. Geometry

After the removal of material again form a geometry with some holes in the body those holes are material removed regions. This design is tolerating the total amount of loads applied on the body before the material removed. For testing, we see total deformation, stress and strain in below

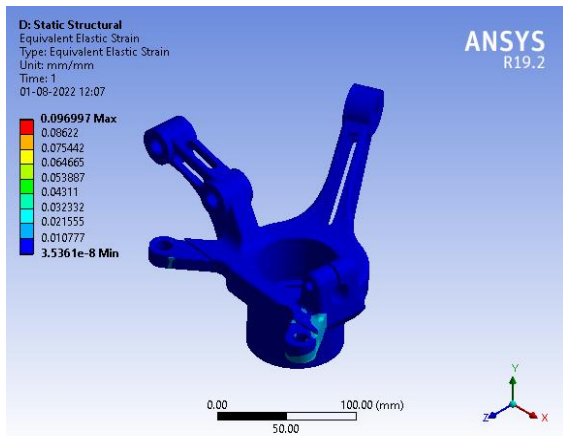


Fig 11. Total deformation

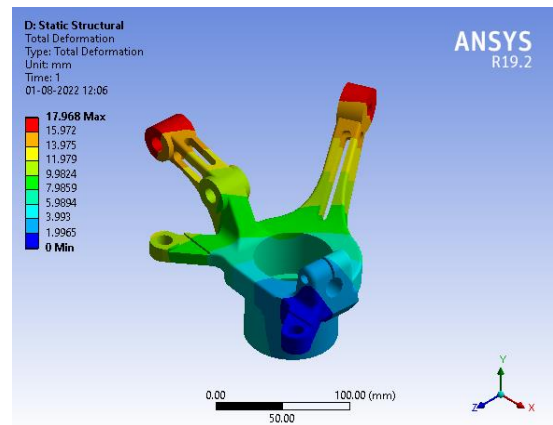


Fig 12. Equivalent elastic strain

In total deformation, it shows removed material is more and then the equivalent elastic strain shows that the design is failed to tolerating the total loads applied from the existing component loads. So we can add some amount of the material to the design and then the final design will be produced.

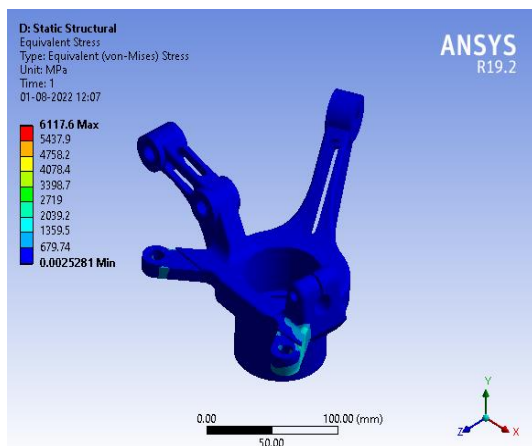


Fig 13. Equivalent stress

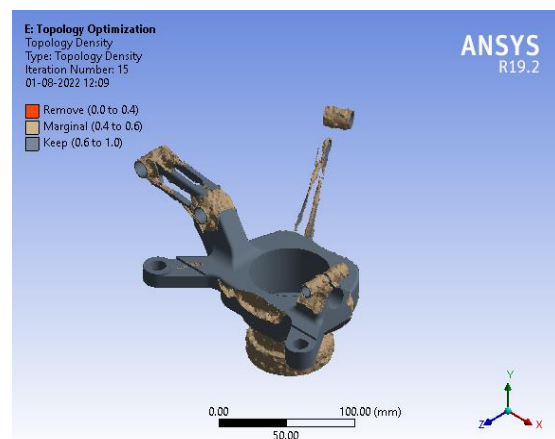


Fig 14. Topology density

Finally add some material to the design and performed topology and the final result will be done. The topology will formed like first one but this time some material will be added. The reason why those two designs are shown similarly is the component have some weak points and those points are shown as failed but there is no problem with is because those places are joints and connections so that is negligible. Finally our design is successes.

Object Name	Topology Density
State	Solved
Scope	
Scoping Method	Optimization Region
Optimization Region	Optimization Region
Definition	
Type	Topology Density
By	Iteration
Iteration	Last
Retained Threshold	0.5
Exclusions Participation	Yes
Calculate Time History	Yes
Suppressed	No
Results	
Minimum	1.e-003
Maximum	1.
Average	0.53023
Original Volume	3.286e+005 mm <sup>3</sup>
Final Volume	1.8321e+005 mm <sup>3</sup>
Percent Volume of Original	55.755
Original Mass	0.92337 kg
Final Mass	0.51482 kg
Percent Mass of Original	55.755
Visibility	
Show Optimized Region	Retained Region
Information	
Iteration Number	15

Object Name	Topology Density
State	Solved
Scope	
Scoping Method	Optimization Region
Optimization Region	Optimization Region
Definition	
Type	Topology Density
By	Iteration
Iteration	Last
Retained Threshold	0.5
Exclusions Participation	Yes
Calculate Time History	Yes
Suppressed	No
Results	
Minimum	1.e-003
Maximum	1.
Average	0.55342
Original Volume	3.6794e+005 mm <sup>3</sup>
Final Volume	2.0495e+005 mm <sup>3</sup>
Percent Volume of Original	55.701
Original Mass	1.0339 kg
Final Mass	0.5759 kg
Percent Mass of Original	55.701
Visibility	
Show Optimized Region	Retained Region
Information	
Iteration Number	18

Table 3. Topology density 1<sup>st</sup> time

Table 4. Topology density 2<sup>nd</sup> time

From these two tables, we reduce the material from original material 1.0229kg to topology material 0.5759kg in first time. The removal of material is high in this step so, it is failed. In second time we will remove the material from original material 0.92337kg to topology material 0.51482kg this is less than the first material so, the target is success we will make a design by removal of half of the material by the topology optimization and the design tolerating the loads from the existing material loads

## 6. CONCLUSION

- As these optimizations are targeted for material optimization the dead mass is optimized.
- Successive models are to be redesigned excluding the volumes removed during optimization.
- In first iteration the mass of the object is optimized to 55.701% of its original mass. (1.03 kg To 0.53kg)
- In the second iteration after redesigning the model, the mass of the object is optimized to 55.755% of its original mass. (0.923 kg To 0.514 kg)
- By optimizing these model we can reduce the material cost by 50% and additionally material handling cost, machining cost etc. are also reduced significantly

## 7. FUTURE SCOPE

As an AI language model, I cannot predict the future. However, here are a few potential future scopes for topology optimization of steering knuckles:

1. Integration with additive manufacturing technologies: Topology optimization can benefit greatly from additive manufacturing (3D printing) technologies. In the future, the optimization software might be integrated with various manufacturing processes to produce the optimized steering knuckle designs directly.



2. Multi-objective optimization: Currently, topology optimization of steering knuckles only considers the structural performance. Future optimization techniques might consider multiple objectives, such as weight reduction, manufacturability, and cost reduction.
3. Use of machine learning: Machine learning algorithms can enhance the optimization process by learning from previous optimization results and refining the design with each iteration. This could improve the efficiency and accuracy of the process.
4. Improved analytics and visualization tools: With the development of advanced data visualization technology, engineers might be able to analyze and evaluate complex optimization results easily. Innovative tools might enable them to manipulate and interact with optimized designs more intuitively.

## 8. REFERENCES

1. HSL(2011). A collection of Fortran codes for large scale scientific computation. <http://www.hsl.rl.ac.uk>.
2. E.H.L. Aarts and J. Korst. Simulated annealing and Boltzmann machines: a stochastic approach to combinatorial optimization and neural computing. WileyInterscience series in discrete mathematics and optimization. Wiley, 1989.
3. W. Aichtziger. On simultaneous optimization of truss geometry and topology. *Structural and Multidisciplinary Optimization*, 33(4):285–304, 2007.
4. W. Aichtziger and M. Kořvara. On the maximization of the fundamental eigenvalue in topology optimization. *Structural and Multidisciplinary Optimization*, 34(3):181–195, 2007.
5. W. Aichtziger and M. Kořvara. Structural topology optimization with eigenvalues. *SIAM Journal on Optimization*, 18(4):1129–1164, 2007.
6. Wolfgang Aichtziger and Mathias Stolpe. Truss topology optimization with discrete design variables - Guaranteed global optimality and benchmark examples. *Structural and Multidisciplinary Optimization*, 34(1):1–20, December 2007.