SolarPACES 2024, 30th International Conference on Concentrating Solar Power, Thermal, and Chemical Energy Systems

Solar Collector Systems

https://doi.org/10.52825/solarpaces.v3i.2421

Published: 26 Nov. 2025

Dynamic Wind Loading of Heliostats

Efficient Simulation of Resonance Effects for Heliostat Cost-Optimization

Justus Blum[1], Atli Tobiasson Helmer[2], and Andreas Pfahl[3,*]

[1]University of Applied Sciences Cologne, Institute of General Mechanical Engineering, Germany

[2]University of Applied Sciences Düsseldorf, Faculty of Mechanical and Process Engineering, Germany

[3]German Aerospace Center (DLR), Institute of Solar Research, Germany

*Correspondence: Andreas Pfahl, Andreas.Pfahl@dlr.de

Abstract. A method for the design of heliostats considering dynamic wind loads is presented. The transient FEM simulation is based on a CAD model of the heliostat and a pressure distribution time series measured in a wind tunnel. In order to minimise the calculation times, a simplified FEM model is used first to determine the period in which the maximum deformations occur. The stresses can then be determined for this period using a more precise model.

Keywords: Central Receiver, Concentrator, Heliostat, Wind Loads, Dynamic Amplification

1. Introduction

Due to climate change, it is essential to carry out research into renewable energies. Solar thermal plants are an important contribution to a climate-neutral energy supply. For high temperatures, especially solar tower plants with a central receiver are suitable. Heliostats concentrate the sunlight and are essential components of these plants. They must be dimensioned considering the fluctuating wind loads and the resulting dynamic loads.

DLR has developed a heliostat design of extraordinary low cost [1][2]. The main differences to previous heliostats are the small size combined with a favourable simple architecture that includes common azimuth and elevation rotation axes and some innovations in detail [2]. A first prototype with a 2 m² mirror surface is shown in Figure 1. The mirrors are supported by a steel structure. The pylon is also made from thin-walled sheet steel. Therefore, a relatively large diameter was chosen which also results in a more stable bearing of the rotating top.

Figure 1. *Prototype of the investigated heliostat design of 2 m²*

The dynamic wind loads for the stow position are to be determined for this heliostat. In the event of strong wind loads, the heliostat moves to the stow-position to reduce the forces on the mirror. The stow-position is defined by a horizontally aligned mirror surface. A heliostat in such a horizontal orientation was measured in a wind tunnel to gain a pressure distribution time series. This covered seven minutes and included both positive and negative pressure values. When vortices of air with vertical velocity component hit the side edge of the horizontal mirror, this flow detaches from the edge and turbulent eddies occur on both, the top and bottom of the mirror surface (Figure 2).

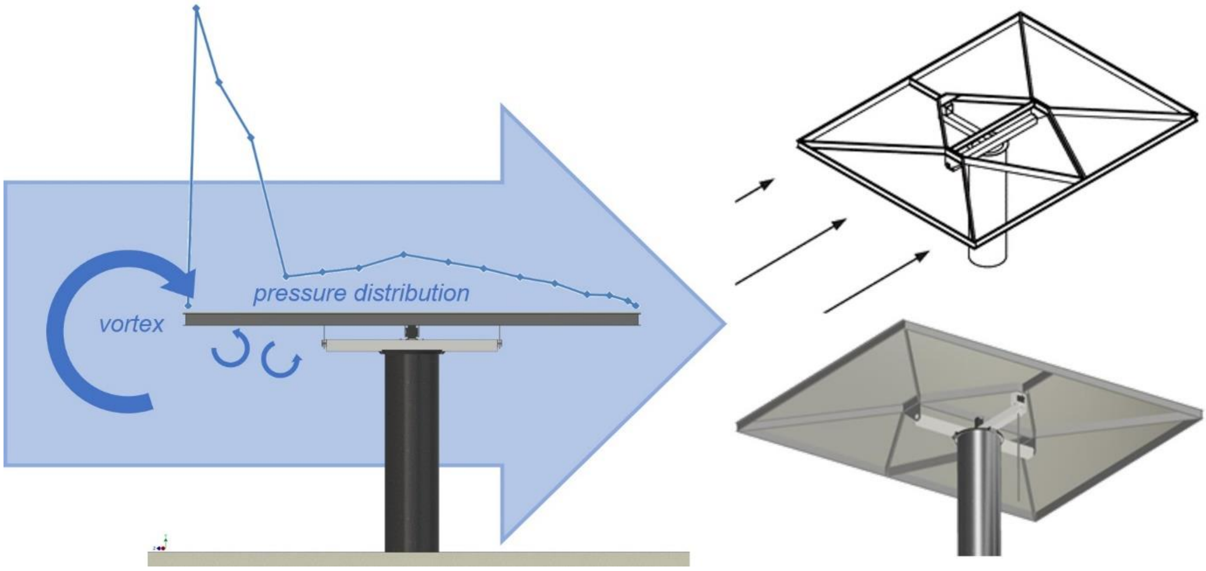

Figure 2. *Suction on the bottom part of the mirror surface in stow position caused by vortices*

2. Method

A transient simulation is carried out using the finite element method with ANSYS software. A CAD model of the heliostat and the pressure distribution time series, which was measured in a wind tunnel, serve as the basis for the calculation. The pressure distribution time series gives the pressure as a function of time on the mirror surface of the heliostat.

2.1 Defeaturing

The first step in setting up the calculation model is to create an idealised CAD model. The simplification essentially consists of neglecting details that have little influence on the structural mechanics.

2.2 Meshing

Two points are mainly relevant for the quality of the mesh. On the one hand, the type of finite elements, i.e. the form of the elements from which the mesh is created has a major impact. Finite elements can be divided into two main categories. There are surface elements called shells, which are used for shells, plates and generally for thin-walled parts, and volume elements, so-called solids, which are used for bodies whose dimensions are significant in all three spatial directions. The second important point regarding mesh quality is the number of elements. The more elements are included in the mesh, the more precise the calculation becomes. However, increasing the number of elements also leads to a significant increase in the computational time required. It is therefore common practice to look for a mesh density that provides a sufficiently precise solution with the lowest possible number of elements [3].

Shell elements are mainly used for thin-walled components with wall thicknesses of less than a tenth of the maximum component dimension. However, this leads to a higher modelling effort [4]. Especially for thin-walled profiles, shell elements are preferable, as flat solid elements such as hexahedrons with a high aspect ratio lead to poor calculation results [5] [6]. Creating hexahedrons with a low aspect ratio on a thin-walled profile results in a small element size and thereby for a high computational time required. In addition, shell elements can accurately predict deformation due to bending and natural frequencies of thin structures, which can be explained by the higher number of degrees of freedom per node with a value of five [6].

A mesh study is conducted to ensure the accuracy of the calculation. A convergence criterion of four percent deviation from the previous step of the calculation was chosen. The mesh study is carried out for the most important components.

2.3 Static analysis

At the beginning, the model is set up in a static-mechanical analysis, as a large number of calculations have to be carried out during modelling in order to be able to consider the effects of the defined boundary conditions and to be able to carry out plausibility checks. The static-mechanical analysis can be calculated much faster than the transient options.

2.4 Modal analysis

For structures with dynamic loads, it is recommended to carry out a modal analysis. Modal analyses do not calculate the structure's response to dynamic loads, but its natural frequencies. The modes reflect the vibration of the structure at certain frequencies. If an excitation occurs that is close to a natural frequency, resonance effects occur in the structure. These can lead to strong vibrations and thus can have undesirable effects [7].

Modal superposition (MSUP) is a simplified transient calculation method. This method can calculate non-constant variables over time. The basis of the calculation is modal analysis meaning it is based on the eigenmodes of linear systems. The calculation is conducted by factorising the eigenmodes from which the temporal response is composed [7]. This can be carried out if the behaviour of the structure is linear-elastic. The only contact type that can be defined in this form of analysis is the bonded contact. This is due to the fact that the other contact types exhibit non-linear behaviour [6,7]. The great advantage of this calculation method is the speed and the associated lower costs compared to the complete transient calculation [8].

The decisive factor for the accuracy of the modal superposition, which is used for the transient simulation, is the number of analysed modes in the preceded modal analysis. In order to define a criterion for the modes to be analysed, the participating mass is examined. The ratio of this effective mass, i.e. the mass that is in motion in the respective mode to the total mass of the structure is calculated. This calculation is carried out for each natural frequency. In total, an effective mass of 90 % should then be considered in the modal analysis in order to be able to precisely represent the time response of the structure [9].

The time step is selected to be an order of magnitude smaller than the oscillation period of the highest frequency considered. The degree of damping is also of significant impact on the MSUP method. The damping is extremely difficult to estimate in the design phase of a structure. But, as the influence of the degree of damping is high, it is recommended to carry out dynamic tests after the structure has been manufactured in order to verify the assumptions made. However, many practical applications have made it possible to determine a typical damping ratio of two percent for welded steel structures [10].

2.5 Transient analysis

The von Mises hypothesis is used to design the components. If the stress in the components is below the permissible stress, this hypothesis assumes that the components can withstand the loads. If the stresses are above this critical limit, it is assumed that the components will fail. Furthermore, the stresses should be as close as possible to the permissible values in order to optimise the use of materials and avoid over dimensioning. A simple structural steel, such as structural steel S235JR, is used for most of the components of the heliostat. This has a yield strength of Re = 235 MPa [11]. In addition, a safety factor of 1.35 is aimed for.

2.6 Determining the time period of largest deformations

At first, the time period with the highest deformations is determined (Figure 3). For this purpose, the previously created model is simplified so that a significantly more efficient calculation is possible. This makes it possible to calculate the entire pressure distribution time series of 340 seconds within a moderate calculation time. Using the deformation at two corner points of the mirror, the time of maximum deformation can then be determined by applying the complete pressure distribution time series on the simplified model. As the deformation is related to the stresses in the components, it is assumed that this point in time will also cause the maximum stresses.

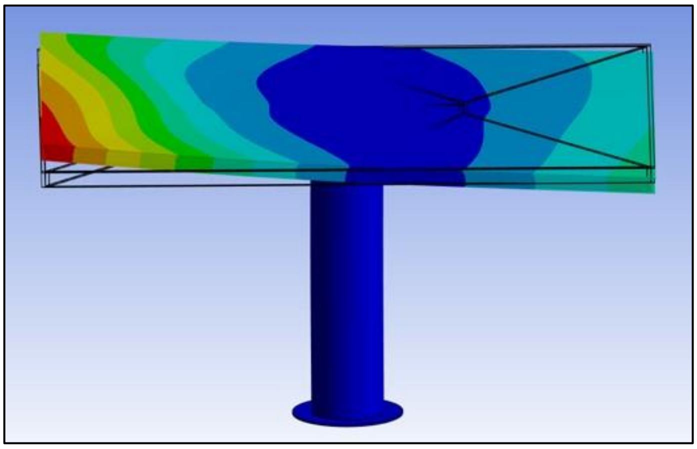

Figure 3. *Magnified deformation of heliostat under maximum dynamic wind loads*

The first important factor that impacts the calculation time is the number of nodes or the element size of the mesh respectively (Figure 4). The element size is gradually coarsened by 25 % for each component until a result is obtained that does still not deviate too much from the reference model and enables an efficient calculation.

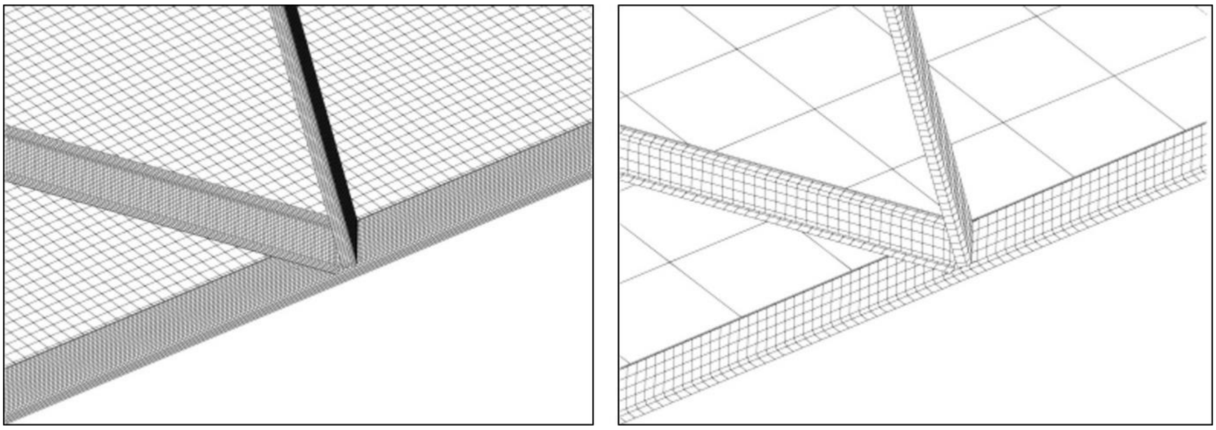

Figure 4. *Detail of the mirror with support structure with fine net (left) and coarsened net (right)*

Another strong impact factor on the calculation time is the number of modes included. This number is reduced to one third of the previously calculated modes. Consequently, ten modes are included in the modal analysis for the coarse model.

Fast transient calculations are also possible with rigid body simulations. This approach was therefore also attempted. The model is shown in Figure 5.

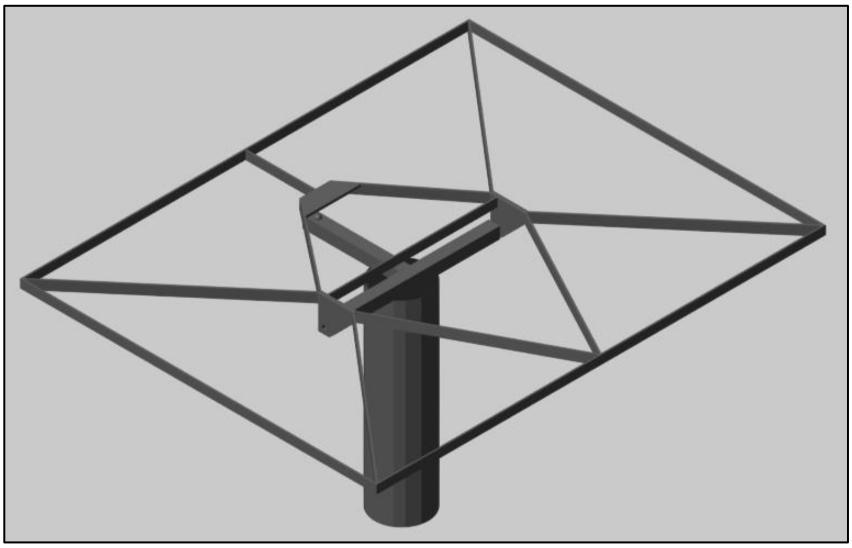

Figure 5. *Rigid body model*

2.7 Calculation of the maximum stresses in the structure

For the time period of maximum deformations, the maximum stresses in the structure can be calculated with the detailed FEM model. Since a time period of a few seconds is sufficient, this can be done with an affordable computational effort. The ratio of these stresses to the maximum stresses calculated with a static analysis gives the dynamic amplification factor. For the static analysis, the point in time of maximum relevant total force or moment has to be considered.

3. Results and conclusions

In order to exclude undesirable resonance effects, a modal analysis is carried out. The maximum excitation frequency of the wind is around 5 Hz. The lowest natural frequency of the heliostat should therefore be well above this value to avoid strong dynamic resonance effects. This condition is fulfilled with a value of 12.6 Hz for the first mode of the heliostat. When calculating 30 modes, the frequency of the highest mode is 116.3 Hz (Figure 6).

1,	12,579	11,	64,23	21,	97,888
2,	14,027	12,	72,981	22,	102,67
3,	14,998	13,	78,178	23,	104,67
4,	19,937	14,	80,97	24,	105,44
5,	22,481	15,	83,409	25,	107,58
6,	27,579	16,	87,036	26,	108,25
7,	31,308	17,	89,914	27,	110,94
8,	36,99	18,	91,831	28,	112,18
9,	51,056	19,	92,975	29,	114,85
10,	55,783	20,	96,507	30,	116,34

Figure 6. *First 30 eigenmodes of the heliostat*

For the transient calculations regarding the deformation, only the first 10 modes were considered. Due to the lower maximum mode with an eigenfrequency of only 56 Hz, the time step could be increased to 0.0018 s. This also led to a significant reduction of the calculation time.

For validation, a time period of 0.2 s around the point in time of maximum deformation was calculated with the simplified model as well as with the detailed model. The curves of two corner points are used to compare the two transient calculations. These are the corner points of the mirror edge at which the wind attacks. The curves of the first corner point are shown in Figure 7. The maximum value of the simplified calculation differs by only 5 % from that of the precise calculation and also the shapes are similar.

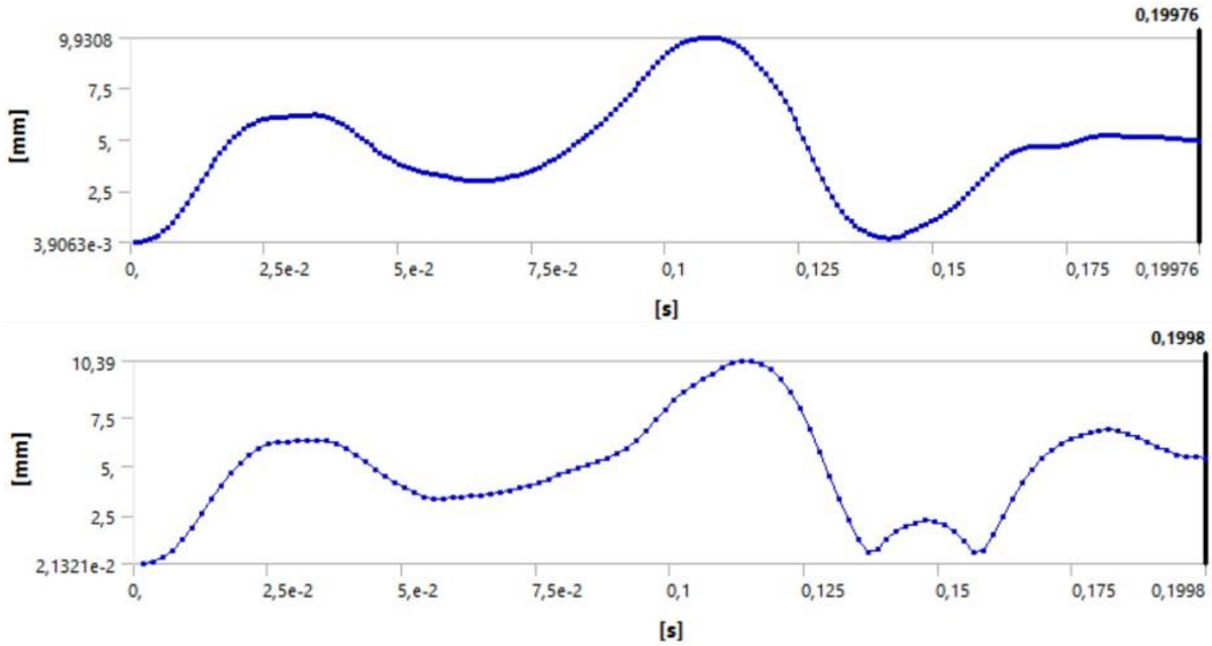

Figure 7. *Vertical displacement of 1st mirror corner point of fine (top) and coarsened model (bottom)*

Figure 8 shows the temporal progression of the displacement of the second point. The deviation here is 14 %. The curves of the calculations for this point are also very similar. The simplified model could thus be validated for the purpose of determining the point in time of maximum deformation.

The original simulation model requires 190 minutes of calculation time for one second of the pressure distribution time series. The simplified model, on the other hand, requires only 42.5 minutes. These values result from a calculation with a processor that has 48 physical cores. The most critical point in time of the pressure distribution time series can therefore be determined within approximately ten days.

The created rigid body model probably has too many simplifications, so that it was not possible to achieve sufficient agreement with the detailed FEM model. Furthermore, the calculation times were not significantly shorter than with the simplified FEM model. Since an FEM model is required for the stress analysis anyway, calculating the most critical point in time using a simplified FEM model seems to be the most favourable approach overall.

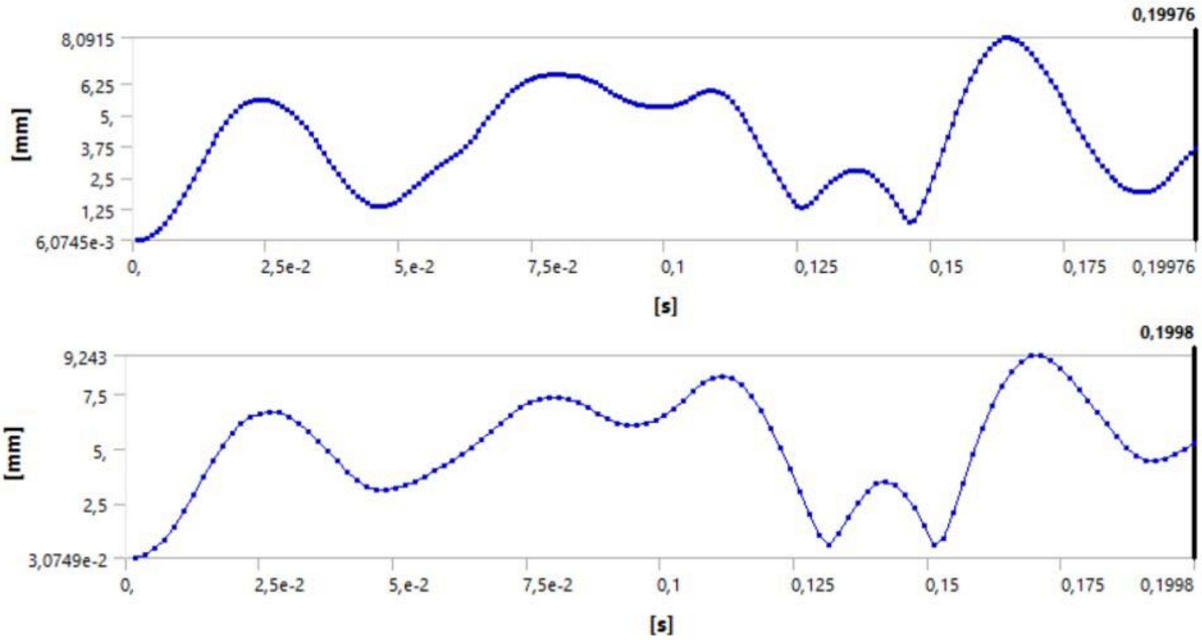

Figure 8. *Vertical displacement of 2nd mirror corner point of fine (top) and coarsened model (bottom)*

4. Summary and outlook

In summary, the stresses in the mechanical structure of a heliostat that occur under fluctuating wind loads were determined. For this purpose, a model was developed to determine the most critical point in time of the pressure distribution time series in a computationally efficient manner. For this, the existing model was simplified in terms of mesh size and the number of modes considered in order to enable a significantly faster calculation. The stresses in the structure could then be calculated with a fine model for a short time period that includes the maximum deformations. Overall, an efficient method was developed that can be used in future to dimension heliostats taking dynamic wind loads into account.

Data availability statement

All relevant data is given in the text or the referenced literature.

Underlying and related material

All relevant underlying and related material is given in the text or the referenced literature.

Author contributions

Justus Blum: Data curation, formal analysis, investigation, methodology, visualization, writing – original draft.

Atli Tobiasson Helmer: Data curation, formal analysis, investigation, methodology, visualization.

Andreas Pfahl: Conceptualization, methodology, supervision, writing – review & editing.

Competing interests

The authors declare that they have no competing interests.

References

[1] A. Pfahl and V. Dohmen, “Low-Cost Materials for Heliostats,” SolarPACES 2023 conference.

[2] A. Pfahl, “Heliostat,” International patent applications PCT/EP2024/085978 and PCT/EP2024/085999.

[3] T. A. Burkhart, D. M. Andrews, and C. E. Dunning, “Finite element modeling mesh quality, energy balance and validation methods: a review with recommendations associated with the modeling of bone tissue,” Journal of biomechanics, vol. 46, no. 9, pp. 1477–1488, 2013, doi: 10.1016/j.jbiomech.2013.03.022.

[4] M. Schabacker, Solid Edge 2023 für Fortgeschrittene – kurz und bündig, 3rd ed. Wiesbaden, Heidelberg: Springer Vieweg, 2023. Accessed: May 20 2024. [Online]. Available: https://link.springer.com/978-3-658-41085-8

[5] M. Peeters, G. Santo, J. Degroote, and Van Paepegem, “Comparison of Shell and Solid Finite Element Models for the Static Certification Tests of a 43 m Wind Turbine Blade,” Energies, pp. 2–19, 2018, doi: 10.3390/en11061346.

[6] J. D. Thomas, “Hex Element Aspect Ratio and Formulation,” Sandia National Laboratories, pp. 2–5, 2019, doi: 10.2172/1762928.

[7] C. Gebhardt, Praxisbuch FEM mit ANSYS Workbench: Einführung in die lineare und nichtlineare Mechanik., 3rd ed.: Carl Hander Verlag GmbH & Co. KG, 2018. Accessed: May 9 2024. [Online]. Available: https://www.hanser-elibrary.com/doi/book/10.3139/9783446457409

[8] J. Rubacek, “Time Effective Transient Analysis Using ANSYS Mechanical and Matlab Simulink,” Philips Oral HealthCare, 2004. [Online]. Available: https://www.researchgate.net/file.PostFileLoader.html?id=55960a3260614be2d28b4569&assetKey=AS%3A273670398775298%401442259516350.

[9] S. Elias, V. Matsagar, and T. K. Datta, “Effectiveness of distributed tuned mass dampers for multi-mode control of chimney under earthquakes,” Engineering Structures, vol. 124, pp. 1–16, 2016, doi: 10.1016/j.engstruct.2016.06.006.

[10] F. N. Kudu, Ş. Uçak, G. Osmancikli, T. Türker, and A. Bayraktar, “Estimation of damping ratios of steel structures by Operational Modal Analysis method,” Journal of Constructional Steel Research, vol. 112, pp. 61–68, 2015, doi: 10.1016/j.jcsr.2015.04.019.

[11] https://www.thyssenkrupp-materials-processing-europe.com/en/c-steel/hot-strip/s235jr-s355k2/s235jr.