Design for Manufacturing and Reliability

Mohammad Khalikujjaman
Md Gias Kamal
Avi Joshef Gomes
Md Abidur Rahman

Group 4

January 24th, 2026

Aarhus University

Reliability-Oriented Structural Analysis Using the Modern pyMAPDL Interface

Contents

Abstract

This project presents a reliability-oriented structural analysis workflow implemented using the modern Python-based interface pyMAPDL. A parameterized finite-element model of a hanger bracket subjected to static loading is developed, allowing systematic investigation of the influence of geometric variability on the structural response. The maximum von Mises stress is used as the governing performance metric.

To efficiently explore the design space, a Design of Experiments strategy based on a Central Composite Design is employed, and a second-order polynomial surrogate model is constructed from the finite-element results. The surrogate model is subsequently used for Monte-Carlo simulation to estimate the probability of failure under geometric uncertainty. The results indicate that the hanger bracket design remains robust within the investigated parameter range, with stress levels remaining well below the yield limit.

The study demonstrates how classical finite-element analysis and reliability assessment can be combined with modern Python-based automation. The use of pyMAPDL enables efficient parametric modelling, transparent post-processing, and seamless integration with numerical and statistical tools, making it a suitable platform for contemporary reliability-aware mechanical design.

1 Introduction

This group project focuses on implementing a reliability-oriented structural analysis workflow using the modern pyMAPDL interface. pyMAPDL provides a Python-based connection to ANSYS Mechanical APDL, enabling direct control of finite-element models through Python and allowing simulation, parameter variation, and post-processing to be handled within a single automated environment.

In the project, a hanger bracket subjected to static loading is analyzed using a fully parameterized finite-element model developed in pyMAPDL. The maximum von Mises stress is selected as the governing response quantity, as it directly reflects the structural integrity of the component under load. Key geometric dimensions of the bracket are treated as uncertain parameters and systematically varied to study their influence on the stress response.

To efficiently evaluate the effect of geometric variability, a Design of Experiments approach based on a Central Composite Design is employed. The resulting finite-element data are used to construct a quadratic surrogate model, which approximates the relationship between the geometric parameters and the maximum stress. This surrogate model is then used for Monte-Carlo simulation, enabling probabilistic reliability assessment at negligible computational cost compared to direct finite-element sampling.

2 Description and Geometry Parameterization

The hanger bracket under analysis is designed for mountaineering applications, where structural dependability is crucial. The model does not include the part of the bracket that is embedded in the rock. To enable effective variation of important dimensions within pyMAPDL, the geometry is parameterized.

Geometry and Parameters

The hanger bracket is defined by the following geometric parameters:

- L_1 – horizontal arm length
- L_2 – vertical arm length
- R_1 – inner fillet radius
- R_2 – outer fillet radius
- d_1 – hole diameter
- t – bracket thickness

Baseline dimensions are taken directly from project description and implemented as Python variables within the pyMAPDL script.

The nominal values are:

Parameter	Value (mm)
L_1	40
L_2	40
R_1	30
R_2	30
d_1	25
t	14

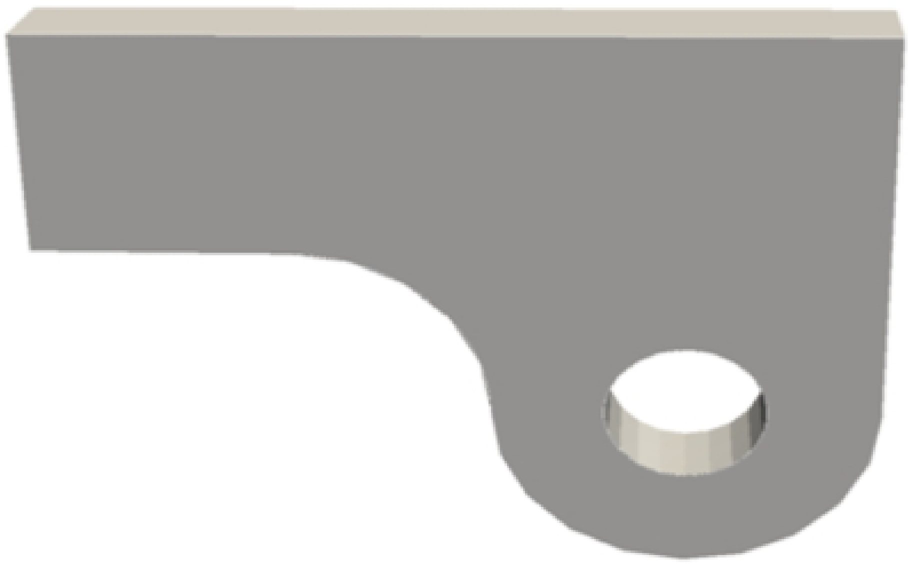

Figure 1: Parametrized Geometry for Hanger bracket

The geometry is constructed parametrically in pyMAPDL using keypoints, lines, and areas, followed by extrusion into a three-dimensional solid. This ensures that all subsequent analyses automatically reflect changes in the design variables.

3 Finite Element Model

3.1 Element Type and Material Properties

Three-dimensional SOLID285 elements, which are ideal for structural analysis of solid components under static loading circumstances, are used to discretize the hanger bracket. The element formulation supports linear elastic material behaviour and incorporates three translational degrees of freedom per node. The bracket material is structural steel (SIS 2140), which is modelled as linear elastic and isotropic using a Poisson's ratio of $\nu = 0.30$ and a Young's modulus of $E = 200$ GPa. The material properties are specified in the problem description. These characteristics are used consistently throughout the model, and the structural response of the bracket is described by evaluating the von Mises equivalent stress.

3.2 Boundary Conditions and Loading

The left side of the hanger bracket, which represents the part fastened into the rock and restricts all translational degrees of freedom, is entirely clamped when the bracket is modelled under static loading circumstances. An equivalent vertical force acting on the chosen nodes at the hole border is modelled as a downward load applied at the bottom of the hole on the right side of the bracket, with a magnitude equivalent to a climber mass of 100 kg. The von Mises equivalent stress criterion is used to assess the resulting stress condition.

3.3 Output Quantity

The quantity of interest in this analysis is the maximum von Mises equivalent stress in the hanger bracket. This value is extracted using the ETABLE functionality in ANSYS Mechanical APDL, ensuring full consistency with the original MATLAB–APDL coupled implementation. The von Mises stress distribution is used to evaluate the structural response of the bracket under the applied loading and boundary conditions.

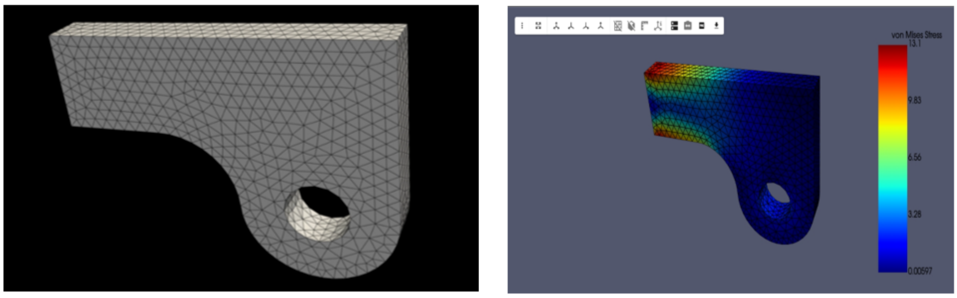

Figure 2: Meshed Model and Von Misses Stress Contours

4 Design of Experiments (DoE)

To efficiently explore the influence of geometric uncertainty on the structural response of the hanger bracket, a Design of Experiments (DoE) approach was adopted. A Central Composite Design (CCD) was selected due to its suitability for constructing second-order surrogate models with a limited number of simulations.

Three geometric design variables were considered:

- L_1 – vertical arm length
- L_2 – horizontal arm length
- R_1 – inner fillet radius

Each parameter was assumed to follow a normal distribution with a coefficient of variation (CoV) of 15%, consistent with typical manufacturing tolerances and identical to the assumptions.

The CCD consisted of 15 design points, comprising:

- 8 factorial points

- 6 axial (star) points
- 1 center point

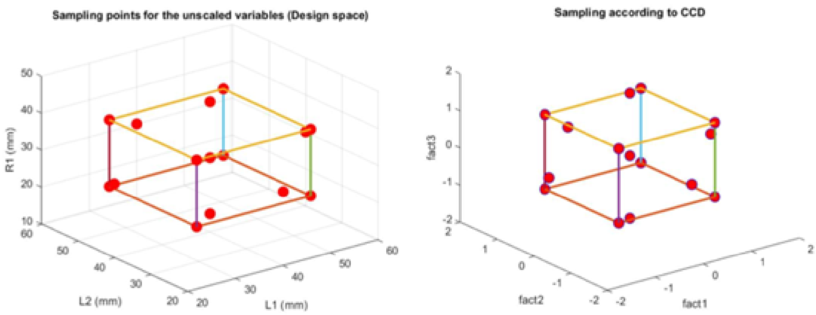

Figure 3: CCD sampling points in design space.

The design space was centered around the nominal geometry $(L_1, L_2, R_1) = (40 \text{ mm}, 40 \text{ mm}, 30 \text{ mm})$ with parameter intervals defined as:

$$\Delta x_i = 2 \cdot \text{CoV} \cdot x_{i,\text{mean}}$$

Each CCD point was mapped from the normalized design space into physical dimensions and evaluated using a fully automated pyMAPDL finite-element workflow. For every design configuration, the FE model was regenerated, meshed, solved, and post-processed to extract the maximum von Mises stress using the APDL ETABLE method.

5 Surrogate Model Construction

Based on the CCD dataset, a second-order polynomial surrogate model was constructed to approximate the relationship between the geometric parameters and the maximum von Mises stress.

$$\sigma_{vm} = a_0 + a_1 L_1 + a_2 L_2 + a_3 R_1 + a_4 L_1 L_2 + a_5 L_1 R_1 + a_6 L_2 R_1 + a_7 L_1^2 + a_8 L_2^2 + a_9 R_1^2$$

The regression coefficients $a_0 \ldots a_9$ were obtained using a least-squares solution of the linear system:

$$\mathbf{A}\mathbf{a} = \boldsymbol{\sigma}$$

where the design matrix $\mathbf{A}$ contained the linear, interaction, and quadratic terms evaluated at the CCD points, and $\boldsymbol{\sigma}$ represented the corresponding FE-computed stresses.

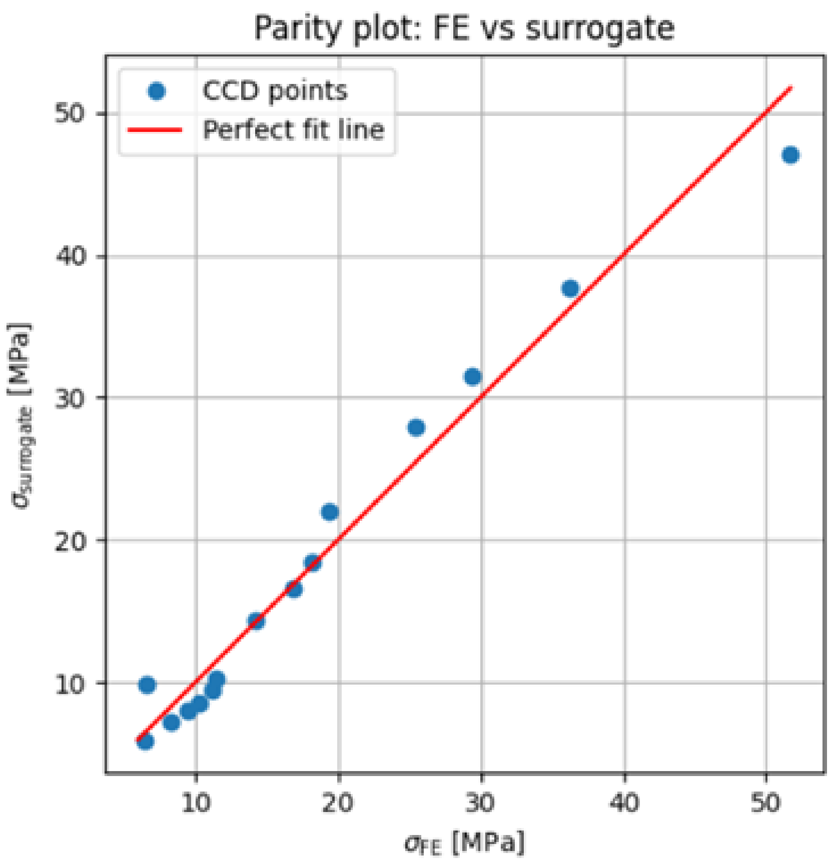

Figure 4: Comparison between stresses and surrogate predictions

The surrogate fitting was performed entirely in Python using NumPy, mirroring the mathematical formulation previously implemented in MATLAB. The resulting coefficients captured both the primary trends and interaction effects between the geometric variables. In particular, the surrogate correctly reflected the strong influence of L_1 and R_1 on stress levels, which is physically consistent with bending behavior near the fillet region.

The use of a quadratic surrogate strikes a balance between model accuracy and computational efficiency, making it well suited for subsequent reliability analysis.

6 Surrogate Model Validation

To assess the accuracy of the surrogate model, a parity plot was generated comparing the surrogate-predicted stresses against the corresponding finite-element results at the CCD points.

In the parity plot, the FE stress values are plotted on the horizontal axis, while the surrogate predictions are plotted on the vertical axis. A diagonal line representing a perfect fit ($\sigma_{\text{surrogate}} = \sigma_{\text{FE}}$) was included as a reference.

The results show that all CCD points lie close to the perfect-fit line, with no systematic bias or large deviations. This confirms that the quadratic surrogate accurately reproduces the finite-element response within the sampled design space. Minor scatter around the diagonal is expected due to numerical discretization effects and the limited number of design points.

Overall, the parity plot demonstrates that the surrogate model provides a reliable and computationally efficient approximation of the FE model.

7 Reliability Analysis

7.1 Monte-Carlo Simulation

A reliability analysis based on Monte-Carlo simulation is performed using the previously constructed surrogate model. Many solver evaluations in direct Monte-Carlo simulation using the finite element model makes the computation complicated. In that case, surrogate model measures efficient evaluation at negligible computational cost. The geometric parameters are treated as independent variables and the mean values correspond to the nominal design dimensions. Standard deviations are defined using a coefficient of variation (CoV) of 15%. Each sample geometry evaluates the surrogate model to compute max Von Mises stress and repeating the process at large number measures stress distribution which is the basis of reliability assessment.

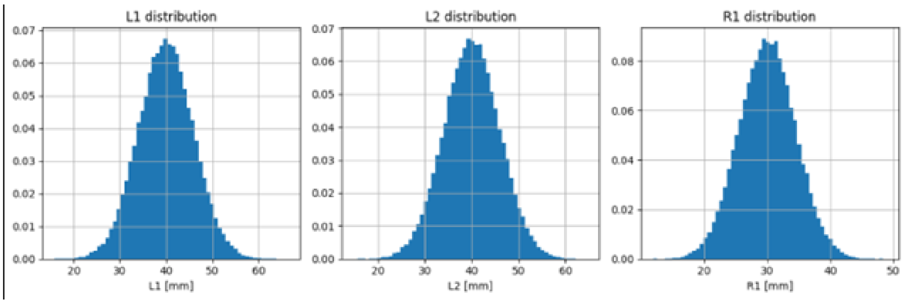

Figure 5: Input histograms L_1, L_2, R_1 distributions

7.2 Failure Criterion

A model is considered to have failed if the predicted Von Mises stress exceeds the material yield strength:

Yield stress 50 MPa is adopted. Here, failure is corresponding to the onset of plastic deformation rather than catastrophic structural failure. In reality-based design these criteria ensure safety margins under uncertainty.

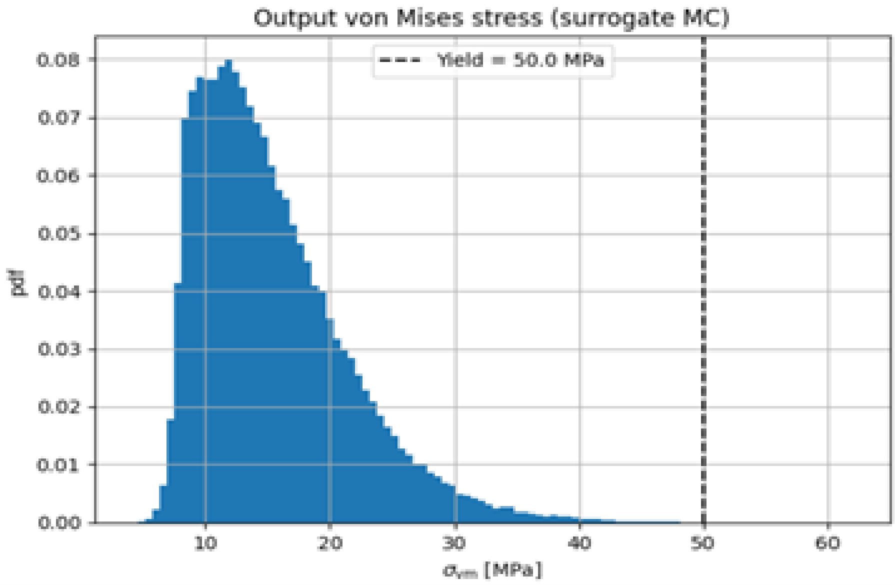

Figure 6: Output stress histogram with yield line (50 MPa)

7.3 Results

For all cases, the stress distribution remains well below the yield limit of 50 MPa. No Monte-Carlo samples exceed the failure threshold resulting in zero failures in the simulation. The histogram of the Von Mises stress also confirms that the probability mass is concentrated far from the yield boundary indicating a safety margin.

It is important to mention that no failure does not indicate absolute safety. Rather, it indicates that the considered geometry and loading conditions can extremely low the probability of yielding. Overall, the results show that the hanger bracket design is robust against geometric uncertainty within the investigate design space.

8 Discussion

The reliability analysis demonstrated how a surrogate-based design assesses structural performance in efficient and reliable means under various uncertainties. As per plot and histogram obtained from simulations, it has been seen that stress distribution remains below the yield limit for all cases which ensure the robustness of the nominal design.

In a relationship between the finite element results and the surrogate predictions, as per plot it indicates that the surrogate model accurately captures the dominant nonlinear effects the Central Composite Design domain. Due to numerical discretization minor deviations observed in the parity plot.

From a computational view, the surrogate model reduces the cost of reliability analysis compared to direct finite element Monte-Carlo simulation. This enables extensive probabilistic analysis which would be impractical in traditional finite element method.

9 Conclusions

Overall, this project demonstrates how a reliability-oriented structural analysis workflow can be efficiently implemented using the modern pyMAPDL interface within a Python environment. By developing a fully parameterized finite-element model, automating a Design of Experiments, constructing a quadratic surrogate model, and performing Monte-Carlo reliability analysis, the study shows that classical reliability concepts can be combined with contemporary computational tools in a transparent and reproducible manner. The use of pyMAPDL enables direct interaction with ANSYS Mechanical APDL without external system calls, simplifies the automation of parametric studies, and allows seamless integration with Python libraries for numerical analysis and visualization. The results indicate that the hanger bracket design remains robust under the considered geometric uncertainties, with stress levels remaining well below the yield limit throughout the investigated design space. From a computational perspective, the surrogate-based approach significantly reduces the cost of probabilistic analysis compared to direct finite-element sampling, making large-scale reliability assessments feasible. It should be noted, however, that the surrogate model is valid only within the defined Central Composite Design domain and should not be extrapolated beyond this range. Future extensions of this work could include higher-order or adaptive surrogate models, consideration of nonlinear material behavior, or analysis under transient and multi-load conditions. Overall, the project highlights the suitability of pyMAPDL as a powerful and flexible tool for modern reliability-aware mechanical design and analysis.

Appendices

Appendix A: Full pyMAPDL code for geometry creation, meshing, and stress.

Appendix B: Python implementation of CCD generation, surrogate fitting and Monte-Carlo reliability analysis.