



# Ashish Vayangankar

FEA Engineer

## About Me

Computational mechanics researcher with 4 years of research experience and strong theoretical foundations in mechanics, mathematics, and FEM. I'm a fast learner proficient in Abaqus and Python, building reliable simulation tools with careful verification and validation. Motivated to pursue doctoral research advancing numerical methods and high-fidelity modeling in collaborative, international teams.

## CONTACT

- **Phone:** (+49) 176 7627 2097
- **Email:** ashishsv99@gmail.com
- **Address:** 67551 Worms, Germany

## ONLINE

- **Website:** <https://ashishvayangankar.github.io/>

## SKILLS

- **Programming & Scripting:** Python, MATLAB, C++.
- **CAE / FEA Tools:** Abaqus, ANSYS, Altair HyperWorks/HyperMesh, HyperView, COMSOL Multiphysics, LS-DYNA, Siemens HEEDS.
- **CAD & Design:** SolidWorks, CATIA, Fusion 360.
- **Analysis Methods:** FEA/FEM, Nonlinear FEM, Static, Dynamic, Modal, Multibody Dynamics, Vibrations, NVH.
- **Materials & Structures:** Material Modeling, Composite Modeling, Fracture Mechanics.
- **PDM:** Teamcenter.
- **Testing & Validation:** Experimental Mechanics.
- **Productivity:** MS Office.

## Work Experience

### Praktikum / Abschlussarbeit (Master)

Hella GmbH & Co. KGaA, Lippstadt

Oct 2024 – Aug 2025

*Thesis: Optimization of Modal Parameters for Headlamp Components Under Vibration*

- Performed Random Vibration Analysis on a complex headlamp assembly.
- Calibrated FEA model to experiments using HEEDS; correlation from 50% to <10%.
- Pre/post-processed headlamp assembly in HyperMesh and HyperView.
- Solved optimization problems on modal parameters with HEEDS.
- Processed vibration data (PSD) from 3D camera setup using Python.
- Verified transferability on a multiple headlamp design.

### Hilfswissenschaftler (HIWI)

Lehrstuhl für Technische Mechanik — FAU Erlangen

Jun 2023 – Sep 2024

- Built AbqHomo: automated Abaqus RVE homogenization using Python.
- Applied periodic boundary conditions programmatically with Python.
- Computed macro-tangent matrices via Abaqus and Python.
- Modeled piezoelectric coupling and orphan-mesh workflows in Abaqus.
- Enabled batch parameter studies and cluster execution using Python.

### Project Assistant

Mechanics and Computations Lab — IISc Bangalore, India

Jul 2021 – Sep 2022

- Designed and simulated quasi-zero-stiffness isolator using Abaqus.
- Simulated linear static, frequency response and dynamic in Abaqus and ANSYS.
- Built base-excitation test rig with sensors and DAQ.
- Automated high-speed camera post-processing in C++; extracted vibration FRFs for validation.
- Validated transmissibility against experiments; isolation achieved near 2 Hz.

### Design Engineer

3DPrintkart, Bangalore, India

Jan 2021 – Jun 2021

- Designed CAD parts and assemblies in SolidWorks and Fusion 360.
- Prepared print-ready files using slicing tools.
- Optimized geometries for strength, weight, and manufacturability.

## PUBLICATIONS

- *Simulation Studies of Low-Velocity Impact Damage in FRPs — VETOMAC 2021*  
[https://link.springer.com/chapter/10.1007/978-981-99-4721-8\\_15](https://link.springer.com/chapter/10.1007/978-981-99-4721-8_15)
- *Nanoparticles use for the Effective Hyperthermia of Liver Tumor — IJISRT 2021*  
<https://ijisrt.com/assets/upload/files/IJISRT21DEC698.pdf>
- Optimization-Based Calibration and Validation of Connector Behavior in Linear Headlamp FEA Under Random Vibration with a Nonlinear Gap Study (Yet to be Published)

## LANGUAGE

- English: C2
- German: A2

## Education

### M.Sc. Computational Engineering

Friedrich-Alexander-Universität Erlangen-Nürnberg

Oct 2022 – Present

- **Address:** 91054, Erlangen, Germany
- **Field(s) of study:** Solid mechanics and dynamics
- **Final grade:** 2.2 (112.5 ECTS completed of 120 ECTS)
- **Thesis topic:** Optimization of Modal Parameters for Headlamp Components Under Vibration load case in Linear FEA simulation (Grade: 1.7)

### B.E. Mechanical Engineering

BMS College of Engineering

Aug 2016 – Sep 2020

- **Address:** 560019, Bangalore, India
- **Final grade:** 8.59 on 10
- **Type of credits:** CGPA | **Number of credits:** 200
- **Thesis topic:** Simulation study on low velocity impact on FRPs

## Projects

### LS-DYNA Crash & Explicit Simulation — Online Course (2025)

- Built and solved impact cases (crash box, 3-point bend) in LS-DYNA.
- Pre/Post in LS-PrePost: meshing, contacts, results plots and snapshots.

### Homogenization Analysis of a Piezoelectric Composite — FAU (2025)

- Built Abaqus FE-homogenization to capture PDMS-KNLN electromechanical response.
- Explored stiffness, inclusion shape, and volume effects on piezoelectric coupling.
- Mapped stress, strain, electric-field in RVEs to identify hotspots.

### Particle-Based Rockfall Protection-Net Simulation — FAU (2025)

- Developed spring-damper contact model for net impact forces and dissipation.

### Optimization Algorithms in Python — FAU (2024)

- Implemented GD, CG, Newton; adaptive line search; convergence diagnostics.

### Design and Simulation of a 2-D Pantograph — IISc (2021)

- Ran Abaqus nonlinear FE to assess 2-D pantograph load response.

### Nanoparticles-Aided Hyperthermia (2020)

- Demonstrated nanoparticles enhancing cancer-cell destruction during hyperthermia.
- Performed coupled thermal-bio simulations in COMSOL Multiphysics.

### Simulation Study on Low-Velocity Impact on FRPs — Bachelor Thesis (2020)

- Built AS4/8552 laminate models in Abaqus/Explicit with VUMAT.
- Modeled matrix and fiber failure; CZM delamination; Hashin/Puck criteria.
- Designed cohesive layers and ties to simulate crack initiation/propagation.
- Analyzed force/displacement histories and CAI curves across impact energies.
- Verified peanut-shaped delamination against published experimental benchmarks.