Ashish Vayangankar

FEA Engineer

Contact

- Phone: (+49) 17676272097
- Email: ashishsv99@gmail.com
- Address: Warburger Strasse 07, 33098, Paderborn, Germany

Online

- Website: ashishvayangankar.github.io

Skills

- Abaqus
- Altair Hyperworks
- Altair Hypermesh
- Python
- ANSYS
- Siemens HEEDS
- COMSOL Multiphysics
- MATLAB
- C++
- Solidworks
- Fusion 360
- Catia
- CAE Engineering
- FEA / FEM
- nonlinear FEM
- Experimental Mechanics
- Vibrations Mechanics
- Static Analysis
- Dynamic Analysis
- Modal Analysis
- Material Modeling
- Composite Modeling
- Fracture Mechanics
- MS Office

About Me

I'm a results-driven engineer with hands-on experience in simulations and experimental validation, particularly in FEA, vibration, and modal analysis. I've worked with tools like Abaqus, ANSYS, Python and HyperMesh / Hyperview to solve real-world problems and deliver efficient solutions. I'm tenacious, resourceful, and quick to adapt when challenges arise. What I bring to the table is not just technical skill, but a mindset focused on going beyond expectations. If you're looking for someone who learns quickly, works hard, and delivers results, I'm ready to contribute from day one.

Work Experience

Oct, 2024 - Aug, 2025 (10mo)
Hella GmbH & Co. KGaA
Praktikum / Abschlussarbeit (Master)
Thesis Topic: Optimization of Modal Parameters for Headlamp Components Under Vibration Load Case in Linear FEA Simulation

- Aligned linear FEA simulations with experimental modal data by optimizing stiffness and damping parameters in HEEDS.
- Processed 3D camera measurements and computed displacement RMS/PSD using Welch transform for accurate experimental modal analysis.
- Introduced play/gap between headlamp components and applied equivalent dashpot models to evaluate linear approximation limits.
- Used coherence theory on accelerometer signals to identify regions of potential nonlinearity due to contact gaps or play.
- Optimized modal parameters of the connectors and the play using HEEDS with ABAQUS, and Hypermesh/Hyperview (Pre/Post-processing) workflow.

June, 2023 - Sep, 2024 (1yr, 3mo)
Lehrstuhl für Technische Mechanik - FAU Erlangen
Hilfswissenschaftler (HIWI)

- Developed Python-based AbaqHomo tool which automated the RVE homogenization using Abaqus.
- Applied periodic boundary conditions to accurately represent microscale behavior in the composite model.
- Computed macro-stress and macro-tangent from microscale FEA to predict material behavior of piezoelectric polymer composites.
- Streamlined material modeling workflow, improving accuracy and reducing manual effort for computing Homogenized results.

July, 2021 - Sep, 2022 (1yr, 2mo)
Mechanics and Computations Lab – IISC
Project Assistant

- Designed and analyzed a quasi-zero stiffness (QZS) vibration isolator combining a linear and a non-linear component.
- Built static and dynamic experimental setups with load cells, lead screw mechanisms, and high-speed tracking for transmissibility tests.
- Simulated static and frequency response using ABAQUS and ANSYS; performed multi-step 2D analyses with base excitation and modal validation.
- Validated frequency isolation bands by comparing simulation transmissibility curves with dynamic test results across lower frequency range of 2–12 Hz.

Jan, 2021 - June, 2021 (6mo)
3DPrintzkart
Design Engineer

- Designed and prepared print-ready 3D models for additive manufacturing using CAD and slicing tools, optimized for structural applications.

Publications

- Simulation Studies of Low-Velocity Impact Damage in FRPS – VETOMAC 2021
- Nanoparticles use for the Effective Hyperthermia of Liver Tumor – IJISRT 2021
- 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

Oct, 2022 – Present
M.Sc. Computational Engineering
Friedrich-Alexander-Universität Erlangen-Nürnberg

- **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

Aug, 2016 – Sep, 2020
B.E. Mechanical Engineering
BMS College of Engineering

- **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

Homogenization Analysis of a Piezoelectric Polymer–Ceramic Composite – FAU (2025)

- Built an Abaqus-driven finite-element homogenization model to capture the effective electromechanical response of a PDMS–KNLN composite.
- Explored how polymer stiffness and particle/fiber inclusion shapes and vol % modulate overall piezoelectric coupling for flexible sensing and energy harvesting.
- Mapped stress, strain and electric-field distributions in the RVE to uncover study hotspots and build durable composite design.

Particle-Based Rockfall Protection-Net Simulation – FAU (2025)

- Developed a spring-damper contact-dynamics model to predict impact forces and energy dissipation in wire-rope rockfall nets.

Optimization Algorithms in Python – FAU (2024)

- Implemented and benchmarked gradient-descent and Newton-type methods with adaptive line search and convergence diagnostics.

Design and Simulation of a 2-D Pantograph – IISC (2021)

- Carried out nonlinear finite-element analysis in Abaqus to assess the load-bearing response of a simplified two-dimensional pantograph mechanism.

Nanoparticles-Aided Hyperthermia (2020)

- Demonstrated the effect of nanoparticles in enhancing cancer-cell destruction during hyperthermia.
- Performed coupled thermal–biological simulations in COMSOL Multiphysics.

Simulation study on low velocity impact on FRPs – Bachelor Thesis (2020)

- Developed detailed finite element models of AS4/8552 laminates using ABAQUS/Explicit and VUMAT subroutine.
- Modelled matrix cracking and fibre failure using Hashin and Puck criteria; implemented interply delamination via CZM.
- Designed cohesive layers and tie constraints using fracture mechanics to simulate crack initiation and propagation between plies.
- Analyzed impact response through force–time, displacement–time, and CAI load–displacement curves for various energies.
- Verified the predicted damage patterns (peanut-shaped delamination) against published experimental benchmarks.