



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.

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

3DPrintzkart, 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.

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.

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
- *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
<ul style="list-style-type: none">• 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
<ul style="list-style-type: none">• 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) <ul style="list-style-type: none">• 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) <ul style="list-style-type: none">• 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) <ul style="list-style-type: none">• Developed spring-damper contact model for net impact forces and dissipation.
Optimization Algorithms in Python — FAU (2024) <ul style="list-style-type: none">• Implemented GD, CG, Newton; adaptive line search; convergence diagnostics.
Design and Simulation of a 2-D Pantograph — IISc (2021) <ul style="list-style-type: none">• Ran Abaqus nonlinear FE to assess 2-D pantograph load response.
Nanoparticles-Aided Hyperthermia (2020) <ul style="list-style-type: none">• 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) <ul style="list-style-type: none">• 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.