525 Lowell Avenue, Apt. 4
Cincinnati, Ohio, 45220

Onkar Salunkhe

osalunkh@mtu.edu

+1 (906) 370 7294
LinkedIn | GitHub

EDUCATION

Michigan Technological University — Houghton, Michigan
Master of Science in Mechanical Engineering: Design | CAE | FEA | Machine Learning | ***GPA: 3.82/4.0*** — *August 2019 - December 2021*

Vishwakarma Institute of Technology (VIT), Pune — Pune, India
Bachelor of Technology in Mechanical Engineering| ***CGPA: 9.09/10*** — *July 2014 - May 2018*
Honors (Equivalent to Minors): Automobile Engineering| ***CGPA: 9.40/10*** — *July 2016 - May 2018*

SKILLS

- **Software**: ABAQUS, ANSYS, ADAMS, Arduino, AutoCAD, Autodesk FUSION 360, CATIA, COMSOL, Creo/ProE, Cura, HYPERMESH, Inventor, LS-DYNA, MS Office products, Optistruct, Radioss, Siemens NX, SolidWorks, TOSCA, 3DEXPERIENCE.
- **Programming Languages**: C++ (STL), FORTRAN 77, FlexPDE, Linux, MATLAB, Mathematica, Python.
- **Libraries and Enviornments**: Numpy, Keras, Pandas, Scikit-Learn, Multi-threading, Microsoft Visual Studio, Linux, Windows

EXPERIENCE

ABAQUS User Success Engineer Intern | Dassault Systèmes | Ohio, USA — June 2021 - **Present**
- Performed Standard, Implicit and Explicit/Dynamic simulations with Buckling, Impact and Frictional contact on ABAQUS.
- Aimed to support and assist both Front-end and Back-end SIMULIA Structural Mechanics team in Beta tests, Python automation environment, modelling complex FEA models and improve Technical communication and demonstration skills.
- Simulated Explicit/Dynamic study of Thief Knot focusing the effect of non-linear geometry, Elastic-Plastic material model, frictional contact coefficient, Mass-scaling and domain-level parallelization (Scaling study) on 3DEXPERIENCE.
- Simulated a Impact of a spherical ball on a thick plate with both Finite Element Method (FEM) and Discrete Element Method (DEM) with Smoothed Particle Hydrodynamics (SPH) modelling for large deformations and Non-linearity.

Software Development Meshing Intern | ANSYS | Cannonsburg, PA, USA — January 2021 - April 2021
- Developed an automation script in Python which will convert all input '.aedt' files to '.aedtz' files using Ansys EDT.
- Enhanced existing Python Test-suite runner for converting '.aedtz' files to Mesh input files. Improved the code for exception handling, custom user inputs and User Interface for real-time commands and progress tracking.
- Introduced a function to run the process on user defined Maximum Allowed Memory with Multi-threading.
- Developed Mesh Visualisation GUI using Object-Oriented Programming (OOPs) and Data Structures in EBU Prime app for Electromagnetic Finite Element Analysis (FEA) with STL libraries in C++ 11 and QT.
- Recursive functions and Defensive programming techniques are used to develop the code for more robustness and reusability.

Graduate Student Researcher | Michigan Technological University | Michigan, USA — September 2019 - **Present**
- Design and FEA of Carbon Nanotubes actuators for bending the Guide-wires for Biomedical applications.
- Designed and Optimised a bending actuator using geometrical combination of Electromechanical linear actuators with Mathematical Modeling and FEA. Bio-compatible nanomaterials are used wchich are useful in Cardiovascular treatments.
- Using High Performance Computing (HPC) and Nano-material modeling for simulating multi-element FEA model
- Solved the Analytical equation and got the optimal values for the equation using MATLAB solver

R&D Development Associate Engineer | Dassault Systèmes | Pune, India — June 2018 - August 2019
- Assisted senior developers in debugging and developing code and migrating CAD from CATIA V4 to V5 and above levels.
- Developed a Finite Element Solver in FORTRAN77 using Numerical Linear Algebra and Matrix Algebra.
- Promoted the source code changes in C++ and Automated Test cases and Regression Testing for customers like Boeing on CATIA V4 and 3DEXPERIENCE cloud platform for Software quality assurance and Documented the findings.

Research Intern | Indian Institute of Technology (IIT) Bombay | Mumbai, India — January 2018 - June 2018
- Finite Element Analysis (FEA) of Bulk Metallic Glass (BMG) composites.| Poster Presentation.
- Numerically investigated mechanical behavior of BMG through FEA using ABAQUS with parallel computing.
- User material library (UMAT) is used for Mathematical Modeling of Plasticity of BMG Material and developed MATLAB codes to perturb the values of Material parameters through the elements of 3D FE Mesh.
- 2 % strain plasticity enhancement is achieved with 5 % thickness of the copper coating on the monolithic BMG matrix.

PROJECTS

- **Development of algorithm for removing duplicate points in the point cloud** — *September 2020 - December 2020*
  - The objective of the algorithm is to remove duplicate points in the given point cloud with a specified tolerance of 10^{-6} units.
  - A Computational Geometry algorithm for points in 2D is developed with $O(n^2)$ time complexity and extended it to 3D.
  - A new algorithm is developed with lower time complexity of $O(nlog(n))$
- **2D Finite Difference Modeling of Linear Elastic material for bending of Plate** — *January 2020 - April 2020*
  - Modeled a 2D plate on Abaqus and Numerically solved the plate bending equations using matrix algebra.
  - Compared Solver time for computations with Finite Elements in Abaqus and Finite Difference in Mathematica.