**Amul Kushwaha**

**201 bharaut nagar satna , MADHYA PRADESH pin 485001 , 9165138554 | amulkushwaha54321@gmail.com**

# Objective

Detail-oriented and results-driven CAE engineer with a strong background in mechanical engineering and expertise in ABAQUS, meshing, linear static analysis, and AI-driven simulations. Proficient in leveraging AI and ML techniques for mechanical engineering applications, including quantum computing for simulation and deep learning neural networks. Seeking to leverage my skills and experience to contribute to innovative engineering projects.

# Education

**Master certification program in HEV**  ( DEC 2022) **B.E. Mechanical Engineering,** RGPV University Bhopal , **CGPA 7.64/10**  (May 2021) **Diploma Engg. In Mechanical Engineering Vindhya institute of technology, CGPA 7.21/10** (June2018) **High. School (10th) , CGPA 6.2 /10 (**May2013)

# Experience

## Equilibrium Solution Private Limited, Pune (July 2023 to current ) Designation:- CAE Engineer Responsibility:-

* Preparing mesh in 2D Shell , Tetra, Hexa, and Hybrid meshing using ANSA.
* Work on Interior sheet metal and plastic components
* Work on the frame assembly
* Build a connection between two parts by using the connection manager
* Deliver the file on the given timeline
* Learned the ABAQUS for 1d static linear analysis, bucking analysis

## SACHA Engineering ( FORD MOTOR ) , Chennai (Aug 2022 to July 2023) Designation:- CAE Engineer Responsibility:-

* Preparing the mesh for Crash and NVH analysis using ANSA
* Generating the mesh for Sheet metal and plastic components
* Assign the Thickness for plastic and sheet metal components
* Clearing the intersection and penetration with the help of Cygwin software
* Deliver the file on the given timeline
* Clearing all Quality

**Skill lync :-** (FEB. 2022 to Aug 2022)

## Designation:- Technical Support Engineer Responsibility:-

* Explaining the fundamentals of FEA to Students
* Explaining the concept of CAE Engineering with the help of CAE tools such as ANSA, Hypermesh, Radios
* Preparing the presentation for ANSA and Hypermesh

# Projects

## ABAQUS CAE, Static and Dynamic Analysis Linear Static or Quasi-static Analysis By Using ABAQUS

* Conducted detailed Linear Static and Quasi-static analyses using ABAQUS to evaluate structural integrity and performance of mechanical components and systems.
* Developed and validated finite element models (FEM) to simulate real-world loading conditions and predict component behavior under various scenarios.
* Optimized meshing techniques to ensure accurate and efficient simulations, enhancing the reliability of analysis results
* Utilized ABAQUS scripting and automation tools to streamline analysis workflows, reducing simulation time and enhancing productivity

## ABAQUS CAE, Static and Dynamic Analysis Nonlinear analysis by using ABAQUS

* Conducted advanced nonlinear analyses using ABAQUS to evaluate complex mechanical systems and components under realistic operating conditions.
* Developed and validated nonlinear finite element models (FEM) incorporating material nonlinearity, large deformations, and contact interactions to simulate real-world scenarios accurately.
* Implemented sophisticated material models and constitutive equations to capture nonlinear behavior of materials, such as plasticity, hyperelasticity, and creep.
* Performed static, dynamic, and transient nonlinear analyses to assess structural performance, fatigue life, and failure mechanisms under varying load conditions.

## Course –MATALB for Mechanical Engineers Parsing NASA Thermodynamic Data

* given data is that clear total 1000+species are there each species are different cofficent and each is unique specific heat , entropy, and enthalpy so we are calculating all these parameters in our MatLab programming
* extract the value of coefficients which is use to calculate a specific heat , enthalpy, and entropy use fgetl common one we get the coefficients line of thermal data

# Internship

 Stress-reducing analysis and finished product quality check (Fitting shop) in Birla corp. satna  Quantum computing for solving the complex meth problems and Simulation in IBM  **Skills**

1. Simulation Software: ABAQUS, ANSA, Hypermesh, Hyperview, AutoCAD,
2. Meshing: Automated Meshing, AI-Generated Meshing
3. Analysis: Linear Static Analysis, Nonlinear Analysis
4. AI & ML: Machine Learning, Deep Learning, Neural Networks
5. Quantum Computing: Quantum Algorithms, Simulation Acceleration
6. Programming: Python, MATLAB
7. Project Management: Team Leadership, Cross-functional Collaboration

# Certificates

## Rocket modelling on Open rocket (OMSPACE , ISRO) Research accepted certificate (OrbitX india Aerospace pvt. Ltd ) AutoCAD master certificate

**Creating connection in Rear ROD in ANSA**   Rigid body elements are connected.

* Spot welding component connections are made.
* CBUSH elements are represented
* Mass of Handle is represented

#  Degree of freedom assigned for hinges Extra curricular / Leadership Activities

* Winning gold medal in National level Championship
* Winning science exhibition in the college level

# Software Packages

* Preprocessing: HyperMesh, ANSA, ANSYS Meshing , Pointwise
* Computational Analysis: MATLAB, ABAQUS
* Statistical Data Analysis: MS-Excel ChatGPT