`

ABHIJEET.R.TAMBOLE
M.Tech (Machine Design)
Mob–7204414635
Email–*abhirajtambole1997@gmail.com*

PROFILE

An enthusiastic, dynamic and versatile engineering student with explicit machine and designing knowledge who functions at high levels in various settings. Highly adaptive and a quick learner in all fields.

CAREER OBJECTIVE

To be a successful professional in a globally respected company and to achieve the objectives of the company with honesty and fairness and to expertise in the practical field through commitment, perseverance and continuous improvement.

TECHNICAL SKILLS

- Pre-processor: - Hyper-mesh, Ansa (Basic).
- Solver: -Abaqus.
- Post-processor: -Hyper view, Hypergraph.

PROFESSIONAL SUMMARY:

ONWARD TECHNOLOGY

- Hyper mesh as tool expertise.
- Period – March 2023 to till date.
- Role – CAE Engineer.
- Support Customer to WABTEC.

SUYAN ENGINEERING SOLUTIONS

- Hyper mesh as tool expertise.
- Period – January 2022 to March 2023.
- Role – CAE Engineer.

VIN DESIGN SOLUTION.

- 1.5 years of experience in the field of CAE.
- Hyper mesh as tool expertise.
- Period – August 2020 to December 2021.
- Role – CAE Engineer.

Project 1: Modal analysis of Door assembly

Tools used: Abaqus, Hyper mesh, Hyper view.
Description: `
- Modelling of door assembly was done.

- Assign material properties and relevant connection are carried out.
- Dynamic behaviour of the door assembly by extracting Eigen values (frequencies) and Eigen vectors (mode shapes).
- To check the structural integrity of the door assembly.

Project 2: Wheel cap snap fit stiffness analysis

Tools used: Abaqus, Hyper mesh, Hyper view.

Description:
- Modelling of Wheel cap was done and relevant connections are carried out.
- Assign material properties and boundary conditions to FE model.
- Deck preparation, load cases are considered and results are post processed.

Project 3: Non-Linear Static Structural Strength Analysis for Knuckle.

Tools used: Hypermesh, Abaqus.

Description:
- This involved Finite Element Modeling, analysis setup (connections, materials, boundary conditions loading and deck preparation) and post processing of the results.
- Developed target countermeasures and determining the design feasibility of components for product changes and improvements to meet functional and performance specifications.
- Post processing of the result by reporting the displacement, Strain & compared the acceptance criteria.

Project 4: Non-Linear Static Structural Strength Analysis for Door Grab Handle.

Tools used: Hypermesh, Abaqus.

Description:
- Nonlinear static strength analysis for the Door Grab Handle as per the physical test set up for predicting the component failure & to improve the strength of Door Grab Handle.
- This involved Finite Element Modeling, analysis setup (connections, materials, boundary conditions loading and deck preparation) and post processing of the results.
- Load applied on Door Grab Handle using a rigid applicator (500-600N).
- Developed target countermeasures and determining the design feasibility of components for product changes and improvements to meet functional and performance specifications.
- Post processing of the result by reporting the displacement, Strain & compared the acceptance criteria.

Project 5: Non-Linear Static Structural Strength Analysis for Footrest.

Tools used: Hypermesh, Abaqus.

Description:
- Nonlinear static strength analysis for the Footrest as per the physical test set up for predicting the component failure & to improve the strength of the Footrest.
- This involved Finite Element Modeling, analysis setup (connections, materials, boundary conditions loading and deck preparation) and post processing of the results.
- Load applied on Footrest using a rigid applicator of 50mmdiameter (250N).
- Developed target countermeasures and determining the design feasibility of components for product changes and improvements to meet functional and performance specifications.

- Post processing of the result by reporting the displacement, Strain & compared the acceptance criteria.

Project 6: FE Modelling of sheet metal & Plastic Trim parts such as Floor, Door, Fender, Hood, Front Bumper & Rear Bumper, IP, Console, Head Lamp, Rear Lamp, A,B & C Pillar Trims.

Tools used: Hypermesh
Description:
- Project involves geometry clean up, cad visual checks, Symmetry checks.
- FE modelling using 2D elements.
- Clearing quality as per criteria provided.
- FE checks, intersection and penetration removal.

Project 7: FE Modelling of Solid Mesh parts (Tetra) such as Engine Parts, Foam Parts, Tail gate.

Tools used: Hypermesh
Description:

- Project involves geometry clean up, CAD visual checks, Symmetry checks.
- FE modelling using" equilateral Tria & Ortho Tria" elements And Tetra meshgeneration.
- Avoided dents at critical and hole regions.
- Clearing quality as per criteria provided.
- FE checks, intersection and penetration removal.

Project 8: Step Ladder

Tools used: Hypermesh, Ansys, Hyperview
Description:
- FEA Modeling was generated as per client requirement.
- Assign material properties and relevant connection are carried out.
- Carried out different load cases for person standing and 2G lifting.

EDUCATIONAL QUALIFICATION

- M.Tech. Machine design – Year 2020, University B.D.T Engineering college, Davangere. (83.71%)
- B.E. Mechanical - Year 2018, Atria Institute of Technology, Bangalore.(67.24%)
- Class XII-Higher Secondary School Year 2014, Diamond PU Science College -Bhalki. (73.33%)
- Class X- Higher Secondary School Year 2012, Govt. High School-Bhalki (75.84%)

DECLARATION

I hereby declare that the above written particulars are true to the best of my Knowledge.

Place:

Date: ABHIJEET