Umesh Kodag

Sr. CAE Engineer (M. Tech. CAD/CAM/CAE)

CAE Professional with 4 years and 3 months of experience & expertise in meshing, vehicle integration, structural and modal analysis using various tools like ABAQUS, Nastran, HypeView, Metapost and ANSA.

Personal Info

1/9, Gauri Shankar Society,
Konkan Nagar, Bhandup (W),
Mumbai – 400078.
(+91) 9028735053
Umesh.kodag31@gmail.com

Computer & Software proficiency

ABAQUS – Proficient

Nastran – Basic

ANSA, Hyperview, Metapost – Proficient

Hypermesh, Pro-E, UG NX – Basic

MS Office - Expert

Employment Experience

Sr. CAE Engineer (2021 - Till Present)

Equilibrium Solutions Pvt. Ltd., Pune, MH.

- Static analysis for different parts in full vehicle like panels, knuckle, UCA, LCA etc., for various loading conditions.
- Hands on experience in structural and modal analysis using solvers - ABAQUS and Nastran.

CAE Engineer (2018 - 2021)

Equilibrium Solutions Pvt. Ltd., Pune, MH.

- Perform Full Vehicle Integration using different connection methods and separate includes for subcomponents.
- Design and analysis for validation of rear suspension by inertia relief analysis with iterations for defined loading conditions.

Intern - CAE Engineer (2017-2018)

Extencore Solutions Pvt. Ltd., Pune, MH.

- Finite element modelling of BIW and sheet metal parts according to model and customer requirement in pre-processors Ansa/hypermesh.
- Import and cleanup CAD model into Hypermesh Environment and understand meshing requirement.
- Follow cleanup procedure to achieve high quality mesh to match criteria given.

Education

2017 **M. Tech. CAD/CAM/CAE (8.65 CPI)**
Rajarambapu Inst. Of Technology – Islampur, Sangli.

2014 **B.E. Mechanical (71.65%)**
T.K.I.E.T, Warnanagar, Kolhapur.

2010 **H.S.C. Science (68.83%)**
Navjeevan College, Mumbai.

2008 **S.S.C. (78.46%)**
Y.C.S. School, Mumbai.

Industrial Projects

- **Project Title: Inertia relief analysis of Knuckle, UCA and LCA.**
  Environment: ANSA, ABAQUS, Nastran, Hyperview
  Synopsis:
    1. Geometry studied for hard points of different loading conditions and all possible load cases to be analyzed were figured out.
    2. Parts was modelled by using 3D solid elements and create rigids at hard points to applied load.
    3. Linear static analysis is performed for all loading conditions and find out critical locations where stresses are maximum.
    4. If the value of stresses exceeds yield value, performed nonlinear static analysis for that particular load case to get value of stresses in real conditions.

- **Project Title: Static Analysis for oil canning, panel stiffness of outer panel in vehicle (passenger door, driver door, Roof panel and outer panel).**
  Environment: ANSA, ABAQUS, Hyperview.
  Synopsis:
    1. Geometry (local and near parts are considered) studied for finding the locations where panel is so flexible which shows possible maximum deflection.
    2. Parts are modelled by using 2D shells and assign connection through connection manager in ANSA and check with modal analysis.
    3. Impactor position to different location and apply load at mid node of rigids to find deflection in panel.
    4. If permanent set value exceeds the target value then create different iterations by changing glue thickness, panel thickness and teacup contact region.

- **Project Title: Roof panel denting analysis in vehicle.**
  Environment: ANSA, ABAQUS, Hyperview.
  Synopsis:
    1. Study of roof panel stiffness to determine effect of denting. Body cut section are to be constrained in translation and rotational movements.
    2. Identify testing points for deflection based on modal analysis for initial study. Testing point selected based on design criteria (like most unsupported region, lean areas, weak points etc).
    3. Displacement measurement is taken from the nominal panel position, at each of the chosen indenter location.

- **Project Title: Connections of Full vehicle assembly through connection manager and perform modal analysis.**
  Environment: ANSA, Nastran and Hyperview.
  Synopsis:
    1. Geometry studied to check any intersections in assembly and parts are modelled by using 2D and 3D elements as per thickness of the parts.
    2. Do all possible checks to meet standard criteria.
    3. Assign connections through connections manager to meet design criteria and check all parts are connected.
    4. Do modal analysis to check missing connections in the assembly and create a runnable file for further study (load cases).

Academic Projects

M. Tech. Project: "Implementation of Virtual Internal Bond Model to investigate fracture behavior in Quasi-Brittle Material".
Environment: ABAQUS, Matlab.

• Virtual Internal Bond (VIB) Model is implemented by using VU-MAT subroutine, a mathematical model written in FORTRAN language. The code is applied in ABAQUS Software to identify crack behavior in Quasi-Static Brittle material.

B. E. Project: "Design and fabrication of hydraulic cut-off system for an overloaded vehicle"
Environment: Ansys, Catia

• Design and fabricate the hydraulic system mounted on chassis of vehicle (TATA Ace). If the vehicle carries load higher than the specified capacity load, the hydraulic system gets activated and cut-offs the current supply to engine.

Publications

• Umesh R. Kodag, Prof. Sudhindra N Jalwadi, "Implementation of Virtual Internal Bond Model to investigate fracture behavior in Quasi-Static Brittle material – Review, Proceedings of the Modern Era Research in Mechanical Engineering - 2016"

• Umesh R. Kodag, "Investigating fracture behavior in brittle material by using Virtual Internal Bond Model, Proceedings of 3rd RIT Post Graduate Conference".

GATE Qualified

GATE 2015 - 35.57 (Score 380)

Extra-Curricular & Achievements

• 2nd Rank holder in M. Tech. CAD/CAM/CAE 2017 Batch.
• 7th Rank holder in B.E. 2014 Batch.
• Presented seminar on "Thermal stress analysis of pressure vessel using ABAQUS".
• Attended workshop on "Research opportunities in NVH", at R.I.T.

Skills

• Adaptability
• Problem Solving skills
• Hardworking
• Teamwork

Declaration

I hereby declare that all the information furnished above is true to the best of my knowledge and belief.

Place: Pune
Date: 15/11/2021

- Umesh Ramchandra Kodag