SANGANAGOUDA BIRADAR

Mobile no: 91-9663040445
Email ID : sangu.biradar10@gmail.com

Objective

Seeking an outstanding and intellectual career as a CAE Engineer, as a valuable team member contributing quality ideas and work for an organization where there is ample scope for individuals as well as organizational growth in design and development.

Professional Summary

6 years experience in Finite element Analysis i.e. Durability and Stress analysis, NVH analysis. Meshing of aero structure components, sheet metals, plastic, BIW components and Hexa meshing.

Professional Skill Profile:

- **Technical skills:**
  - FEA: **Meshing (sheet metals, BIW, FE Modeling of closures, interior trims and solid (Hexa meshing) components)**.
  - Analysis: **Durability and Stress analysis, NVH analysis**.
  - Tool knowledge **HYPERMESH, NASTRAN, ABAQUS and ANSA, Optisrtuct.**
  - Finite element modeling of aerospace and automotive components with connections.
  - Responsible for completion of the assigned project from scratch to delivery as scheduled.
  - Planning and co-ordination of day to day activities involved in projects.
  - Coordination with cross functional terms for smooth execution of projects.
- **Computer Aided Engineering Tools: CAE**

  HyperMesh V9-14, Hyperview V9-14, MSC.Nastran, Patran V14, ANSA, ABAQUS Optisrtuct.
- **Professional Experience:**
- **Previous Company**

  April 2015 to april 2019 Project Engineer, CSM software Pvt. Ltd, Bangalore.
- **Present Company**

  From April 2019 to till date working at TATA Technologies Pune.

Projects Worked:

Project#1: **FE modeling and analysis of Automotive exhaust system.**

Tools used: Preprocessor: Hypermesh Solver: Nastran Post processor: Hyperview Type of analysis carried out:

- Static analysis(Sol 101)
- Modal analysis(Sol 103)
- Frequency Response analysis (SOL 111)
- Nonlinear Static Analysis (Sol 106)

Project Description:

This project involves meshing the exhaust assembly which includes sub-assemblies like catalytic converter, rear muffler, front muffler, resonator, manifold turbo. Meshing of most sub-assemblies is Carried out using 2D (quad

Element). Welds are represented by hexa and penta elements. Flex is modeled using beam element, isolators with bush elements.

Project #2: **Brake and clutch pedal**

The requirement is to protect the BIW (mounting region and adjacent area) and Pedal when subjected to loading on brake and clutch pedal.

Simulation is carried out with the Brake and Clutch Pedal positioned such that the Pedals are in the car line (Vehicle Position).Pedal lever and pedal boss joined by welding (Welding on full periphery of boss). Brake and clutch mounting arrangement on bolt, Bolt-sleeve-Bush-Pedal Boss-Pedal lever. Loading should be applied as per below,

BCs: Cut BIW is constrained in all DOF

Loadings: Clutch Pedal Load applied on clutch pedal

Brake Pedal Load is applied on brake pedal

Nonlinear analysis is carried with non-linear material properties assigned, and proper contacts in the model

Project #3: **Model buildup/Integration for Durability analysis**

Creation of Full Vehicle in Include structure Format.

Creation and realization of Connections such as Spot-welds, Bolts, Adhesives etc...

Creation of Intermodular connection using CONN3D2 and KINCOUP 1D, DCOUP3D elements.

Creation of Sets and Contacts.

Creation of Boundary conditions.

Intersection and Penetration removal in the full vehicle.

Sub-assemblies contacts.

Model check and debugging.

Project #4: Side step strength analysis.

Assess the structural performance of step for strength requirements. The Analysis has been conducted in order to validate the performance of the tread board in regards to whether the specified number of fixings and bolted surfaces are sufficient to support the fully loaded step, this includes sheet metal performance, SPR performance, bracket mountings and the aux step itself.

.

Project Description:

The model is to be cut from the unglazed LHD/RHD full body durability model. The model is to include the main floor & body side assembly that together make up the sill section and BIW interfaces. The model is to include the tread board and any supporting bracketry. The model is to be fully constrained. Where the model is a 2-door derivative, apply a vertical load to the aux step or to the solid block rearward of the A-pillar intersection with the roof and in the centre of the largest span between brackets. However, if the length of the stepping area of the running board exceeds then use 4-door requirement. For a 4-door vehicle, apply two simultaneous vertical static loads at the "worst positions" and" extreme locations".

Nonlinear analysis is carried with non-linear material properties assigned, and proper contacts in the model.

.

Project #5: Nonlinear combined gravity and C-load static Analysis of exhaust system components using abaqus as solver.

Project Descriptions: In this local hanger analysis is carried out using hypermesh as modeling tool and solver as abaqus. This involves creation of sets and applying loads as per requirements. The boundary conditions are applied and all DOF in the Hanger location are constrained.

Post processing activities were carried out in HYPERVIEW.

Results and discussions were documented in ppt.

Project #6: Bolts analysis connected to two flanges by applying preload using Hyper mesh as modeling tool and Abaqus as solver.

Project Descriptions: Bolt analysis is carried out for clamp force (force calculated from bolt torque). Using Abaqus. This involves creation of element / component sets and creation of contact (contact nonlinearity) and interface between bolts and flange also creation of pre tension section for bolt to apply preload. The boundary conditions are applied as per requirements.

Post processing activities were carried out in HYPERVIEW.

Results and discussions were documented in ppt.

Project #7: Roof luggage carrier strength analysis

The roof mounted luggage carrier subsystem must be capable of withstanding a vertical load, longitudinal load, longitudinal load with 20 degree angle and lateral loading. There shall be no more than allowed deflection, no cracking and no perceptible permanent set of the luggage carrier components and roof panel

The model is to be cut from the glazed LHD/RHD full body durability model. Cut boundaries of the model are to be constrained in 1-6 DOF

Longitudinal loading: Tensile force of twice the permissible load of the luggage carrier applied horizontally in the driving direction

Longitudinal loading with 20 degree angle: Tensile force of twice the permissible load of the luggage carrier applied horizontally and 20°to the driving direction

Vertical loading: Vertical minimum load evenly distributed on the luggage carrier at a height of 50 mm +/- 10 mm above the crossbars. Apply 1g gravity load (75*9.81 = 736N) superimposed in all load cases in –ve Z direction

Nonlinear analysis is carried with non-linear material properties assigned, and proper contacts in the model.

Project #8: FE modeling of closures (Door, Hood, Tail gate). Tools used: ANSA.

Project Description:

FE modeling of closures and connections were done as per customer NVH standards. All the FE standard checks were performed as per given mesh criteria.

Project9#: Strength and rigidity analysis of IP & Centre Console CAE

Project Description: The objective of the analysis is to perform a strength and rigidity analysis, using Hypermesh and Abaqus. FE Modeling of IP (instrumental panel) and Centre console components using Hypermesh. Connection (at hinge and screw area used CONN3D2 and KINCOUP 1D elements), proper assemblies are made to simulate actual field condition. Different load condition are set up for model. Some load cases are to find the breakage and some load cases to meet the target displacement like 1mm, 2mm and 5mm performed a strength and rigidity analysis. Optimum design solution was provided to meet design expectation.

Software used: Hypermesh, Abaqus, hyperview.

Project10#: Customer Vehicle Jacking

The vehicle structure shall withstand repeated jacking at its designated jacking points, Front and Rear Sill Jacking under overload conditions

Project Description:

The Front sill jacking is analysed with a full model of Jacking plate and its neighbouring components side panel inner, side panel outer, reinforcements, wheel housing reinforcements, front floor and cross members. The cut model is constrained in x, y & z – translation and rotation directions. Load is applied on the jacking flange in the vertically upward (Inline),10deg rearward, 10deg forward, 10deg inboard & 10deg outboard directions to simulate the effect of jacking under even and uneven surfaces.Load has been calculated based on GVM(Gross vehicle Weight) of the vehicle i.e. load case is 0.5 * GVW * 9.81 * 1.25 = ----N (1.25 being factor of safety).

Nonlinear analysis is carried with non-linear material properties assigned, and proper contacts in the model.

Project11#: Modal analysis of seat assembly

Modal analysis of Seat assembly is carrying out to avoid the resonance condition and to ensure FE model connection for safe design.

Project Description

To achieve target natural frequency.

To provide feasible CAE proposal to designer to meet acceptance passing criteria of modal analysis for safe design.

Project12#: Modal analysis of steering mounting bracket.

As per Japanese Industrial Standards (JIS) excitation frequency is 33Hz and 67 Hz .In frequency response analysis Resonance condition of steering mounting bracket has been check to avoid the failure in system.

Project Description

To achieve natural frequency above Excitation Frequency. To provide feasible CAE Proposal to designer to meet acceptance passing criteria of Endurance limit in FRA for safe design.

Educational Qualifications:

M.tech-In machine design2014 pass out AHIT, Bangalore
B.E Mechanical - 2012 pass out MMEC, Belagavi.

Personal Profile

Name	:	Sanganagouda Biradar
Date of birth	:	14th jul 1990
Gender	:	Male
Marital Status	:	Married
Nationality	:	Indian
Languages known	:	English, Hindi, Kannada.
Hobbies	:	Playing Cricket, Social activities, reading.
Permanent Address	:	S/O M.S.Biradar At-Handral ,Post-Lingadalli TQ-MuddebihalDist-vijayapur, Karnataka 586116

DECLARATION:

I hereby declare that all the information mentioned above is true and I bear responsibility for the correctness of the above mentioned particulars.

Place: Pune

Sanganagouda. Biradar