ONKAR K

Objective:

To work efficiently in the CAE industry by blending my skill sets along with my industrial and researchexperience to develop solutions and cater to the company's growth in delivering accentuated results.

Summary:

I am a graduate Mechanical engineer with **3** years of experience in CAE field especially in Automotive Durability and partly NVH domain.

Sofware Skills:

CAE Pre-Processor: Hypermesh.

Solver: Abaqus, Nastran, Femfat.

Operating systems: Linux and Windows.

Other software: MS- office.

Experience:

East Engineering & Technology Ltd. **April '2019 – September '2020**

East Engineering & Technology Ltd **August '2023 – Present****

****Continued same company after graduation**

Academic Year: August '2020 – July '2023**

Successfully Completed BTech In Mechanical Engineering with **8.15** CGPa 1st class with distinction.

Major Projects Executed:

A) Analysis Projects:

1 . Torsional Stiffness calculation for BiW assembly.

Tool Used: Abaqus, HyperView, Hypermesh.

Components analyzed: BIW Full Vehicle.

Objective: To calculate Torsional and Bending Stiffness.

Description: FE model was received to find out Torsional and vertical bending stiffness of the BiW. Linear analysis was carried out to simulate the behavior of the BIW while it is subjected to twisting moment when car is moving through pot holes. Stiffness targets were set for all front, rear and vertical bending and analysis was carried out. This also gives initial idea about how would BiW react to service loadcases. Weaker stiffness, worst would be reaction to service loads. Hence stiffness performance is achieved as priority.

2. Door sagging analysis:
Tool Used: Abaqus, HyperView, Hypermesh.

Components analyzed: Front and rear door.

Objective: To calculate door sag at various opening angles.

Description: Sag Analysis is carried out to check feasibility of door closing under the event of sag loading. To evaluate same we received CAD data from Designer. Performed meshing as per the required quality criteria. All spot welding, bolting, hemming, arc welding and connections were given according to design data. Boundary Conditions was applied on cut-section of BIW and loads were applied according in following sequence

i. Gravity Load

ii. Sagging Load

iii. Unload

Deformation in results were observed and reported. Stress strain plots were given for additional information.

Recommendations were given in order to achieve the target.

3. Oil canning and denting assessment on skin panels:

Tool Used: Abaqus, HyperView, Hypermesh.

Components analyzed: Closures.

Objective: To find oil canning and denting performance of assemblies.

Description: Skin panels are exposed to ourt environment where it may subject to unexpected small loads e.g. lean on door by a customer which may lead to dent on outer skin. Hence oil canning and denting analysis is carried out in order to overcome such conditions. Oil canning gives idea about buckling of panel due to load and denting avoids any permanent set. This assessment is carried out all over skin panels at numerous locations such as body side outer, door, tailgate and bonnet.

4. Pre-loading clevis joints and evaluating load loss in clamping:

Tool Used: Abaqus, HyperView, Hypermesh.

Objective: To calculate Pre-loading clevis joints and evaluating load loss in clamping.

Description: In order to accommodate manufacturing tolerances and to ease the assembling of joints, there are intended gaps designed in clevis joints. However, these joints, if not optimized for appropriate gaps, may lead to high load loss for clamping and may generate high stresses/strains in model which accumulates damage in model. Hence it becomes necessary to assess the design intended gaps for pre-determined load loss and strength targets as decided by customers. Non-linear analysis was performed to calculate the load loss. Material, geometric and contact non-linear were included in model. A graph was plotted as Recommendations were given for improving the stress distribution and avoid stress

concentration while also reducing the load loss.

5. Strength and Durability Analysis of Engine Side Mount Bracket:

Tool Used: Abaqus, HyperView, Hypermesh and FemFat.

Components analyzed: Engine Mount.

Objective: Strength and Durability assessment.

Description: Engine Mounts are exposed to various excitations from IC Engines as well as different road conditions. The G level achieved in these can be as high as 3G/4G. Hence, it required to assess mounts for those loads. As per customer requirement, service loads are divided in two main categories i.e. Fatigue cases and extreme cases. Analyses are carried out considering pre-tension in engine mounts. For fatigue load cases, LCF analysis was carried out using abaqus and FEMFAT and corresponding life cycles for individual load case pairs were reported. For severe load cases, analysis was done based on acceptable limits of plastic strain target decided by customer and analysis was carried out as static non-linear using abaqus as solver. All failure locations were observed and reported along with recommendations.

B) Meshing Projects:

1. BIW Parts:
Domain: Automotive
Software Utilized: Hypermesh, Abaqus, Hyperview
Description: CAD model received and checked for any geometry errors appearing in model. Decision is taken on mesh to be used (2D/3D elements). Auto-Mid-surface or manual method of surface offset was used to get mid-surfaces. Meshing was carried out with mesh quality parameters of FE model. Materials were assigned along with measured thickness of components. RBE2-Beam-RBE2 connections were made to represent bolted connections and RBE3-Hex-RBE3 to represent spot and adhesive connections. Modal analysis was done in Abaqus to check for any missing connections and to assess natural frequency. Results were reviewed in hyperview & any changes required in model were done.

2. Plastic Parts:
Domain: Automotive
Software Utilized: Hypermesh, Abaqus, Hyperview
Description: Dashboard CAD model is reviewed for any unintended design surfaces appearing in model. Accordingly mesh to be used is decided (2D/3D elements). In this project, all components were plastic parts. Mid-surfaces were extracted manually as components had variable thicknesses. Materials properties were assigned. Rigid connections were made to represent bolted connections. Modal analysis was done in Abaqus to check for any missing connections and to assess natural frequency. Results were reviewed in hyperview & any changes required in model were done.

3. Solid Parts:
Domain: Automotive

Software Utilized: Hypermesh, Abaqus, Hyperview

Description: CAD model were checked for geometry errors present in model. Surface clean-up of the components was done. 2D mesh were generated on surface of the components and checked for quality parameters were satisfied and if edge were generated on the mesh. The 2D mesh was converted to tetra mesh(3D) for solid components. Finite element quality parameters were again checked for 3d mesh generated. Rigid and RBE3 connections were given to represent the bolted joints. Modal analysis was done in Abaqus to check for any missing connections and to assess natural frequency. Results were reviewed in hyperview & any changes required in model were done.

Education:

Examination	University	Academic Year	Percentage	Division
BTech (Mechanical)	D' Batu University	2020 - 2023	8.15	1st Class with Distinction
Diploma (Automobile)	(MSBTE)	2012 - 2015	62.20	1st Class
H.S.C	Pune	2009 - 2011	69.83	1st Class
S.S.C. (X)	Pune	2008 - 2009	50.91	2nd Class

Personal Information:

Name : Onkar Manmath Kshirsagar
Date of Birth : 31 March 1993.
Gender : Male.
Marital Status : Married.
Address : A/P Ule Dist Solapur
Languages Known : English, Hindi, and Marathi.

Declaration:

I hereby declare that the above-mentioned information is correct up to my knowledge and I bear theresponsibility for the correctness of the above-mentioned particulars.

Date:

Place: Solapur **(Onkar K)**

`