PRAVEEN KUMAR N

3rd floor, SST, PG, Bhadrappa Layout,
Devi Nagar Maruthi Nagar,
Bengaluru
PIN- 560094

+91 9738407481

Praveenkumar.npc@gmail.com

S/O Nandagopal S
Vasappa layout, Near Aralimara,
TARIKERE
PIN-577 228.

Profile Summary

- Having the experience of **4.1+ years** in the CAE and Structural Analysis field.
- **Currently serving notice period.**
- Good knowledge of **Finite element Method** and **Strength of materials, Fatigue and Damage Tolerance (F&DT), Composites laminates, Aero structures**.
- Knowledge of **Linear Static** Analysis, **Modal** analysis and **Fatigue** analysis using the **Stress Life (SN) approach** and **Damage Tolerance approach**.
- Familiar with Basic **Hand Calculation** like Lug-Pin analysis, Bolt analysis and Critical Section checks.
- Experience in preparation of **Spread sheet** and **certification reports**.
- Experience on **FE model** re-numbering, include file and master deck creation.
- Exposure to **Stress analysis** using **classical and FEA** of **aero structures**.
- Exposure of customer interaction, communication and co-ordination

Experience

❖ **09th MAY 2018 to till date**
Analysis Engineer
AXISCADES Engineering Technologies Ltd., BANGLORE

❖ **04th JAN 2016 to 8th MAY 2018**
CAE Trainee engineer
COPES TECH PVT. LTD, BANGLORE

Education

B.E in Mechanical Engineering

- MVJCE, VTU Belagavi passed in June 2014 with Aggregate of 65.36%.

Diploma in Metallurgical Engineering

- VISSJ.GPT, DTE Bengaluru passed in 2011 with Aggregete-76.23%.

Software skills

❖ **CAE Tools:** Hypermesh MSC Nastran Abaqus Hyperview Patran ANSA
❖ **Airbus Tools:** AirnavX, Zamiz, ISAMI, SRM, CCD
❖ **Programing Tools:** MATLAB, Python
❖ **Others:** MS Office

❖ Experience:

AXISCADES Engineering Technologies Ltd., BANGLORE

Period: 09th MAY 2018 to till date

Role: Analysis engineer

Job Description:

- **Fatigue justification** of the **concession** for Extended Service Goal (**ESG**) Intermediate Service Goal (**ISG**) using Airbus M2841, Concession user Guideline and Practical Guideline.
- **Hand Calculations** for Airbus Fatigue Quality Index (IQF) and Fatigue Life Estimation for Concessions Excel **Spread sheet** and **report** generation.
- Airbus **Structural Repair Manual (SRM)** Looping and SRM activity for the SRM Revisions and Database Preparation for Repair Automation.

COPES TECH PVT. LTD, BANGLORE

Period: 04th JAN 2016 to 8th MAY 2018.

Role: CAE Trainee engineer

Job Description:

- FE modeling of BIW, plastic components of automobile and 3D Hexa Penta mesh.
- Linear Static Strength analysis and Modal analysis using Nastran.
- Plotting results for stress, displacements and mode shapes using Hyperview as post-processor
- Hand Calculation like Lug-Pin analysis, Bolt analysis and Critical Section checks.

❖ Software:

- Hypermesh (Pre-processor) : 2D sheet metal, 3D (Tetra and Hexa &Penta) and 1D Spring & rigids.
- MSC Nastran (Solver) : Linear static and Modal analysis.
- Abaqus : Linear static and Contact Nonlinear analysis.
- Hyperview (Post-processor) : Plotting results for stress, displacements and mode Shapes
- Patran (Post-processor) : Plotting results for stress, displacements and mode Shapes
- MATLAB : analytical calculation, FEA coding and also plotting graphs
- ANSA : BIW component mesh and linear static Deck preparations for MSC Nastran
- Ms office : Ms word, Excel and Power Point
- Airbus tools : AirnavX, Zamiz, ISAMI, SRM, CCD.

❖ Brief List of Project Handled

Project #1: Extended / Intermediate Service Goal (ESG/ ISG) Concessions:

Description: **Fatigue Justification** of Extended Service Goal (**ESG**) Intermediate Service Goal (**ISG**) for Airbus Single Aisle (SA) family (A318, A319, A320 and A321) Intermediate Service Goal for Long Range (LR) family (A330 and A340) will be provided using Airbus Fatigue manual **M2841**, Concession user Guideline and Practical Guideline.

Analyzing and identifying the damage / NC and its locations on Aircrafts which are occur in Aircraft fuselage primary structures like Frames, stringers, skin and couplings etc, using **AIRBUS mechanical drawings**, **ZAMIZ 2D** drawing & **BOM** and to calculate fatigue life, inspection threshold and inspection intervals by determination of fatigue influence factors, collecting geometrical and GFEM Stress data. And supporting for preparation of SB embodiment documents. **Hand calculations** (According to **M2841** Fatigue Manual) will be performed to justify the design solution given for the Non-conformity by recalculating the new IQF and new fatigue life of the affected parts for the type of loading from the Stress Reports and Airbus Documents.

The F&DT analysis was carried out using AIRBUS methods, S-N curves and **ISAMI** (Joints) to ensure that the repair confirms as per ESG requirements by using

- **Damage Hole Analysis, 1D joint initiation analysis, Spot facing initiation analysis, Open hole initiation analysis, Lug &Pin initiation analysis** and **Dent Initiation Analysis** using the Airbus Tool **ISAMI Analyst**.
- **Hand Calculations** for Airbus Fatigue Quality Index (IQF) and Fatigue Life Estimation for Concessions Excel **Spread sheet** and **report** generation.

Good understanding of the recommended guidelines and standards for Aircraft manual **SRM**, Service Bulletin **SB**. Delivering and implementing the project as per scheduled deadlines and extending post-implementation support.

Project #2: FE modelling (Load model, detailed) and Static analysis of Boeing 787x fuselage structure lower lobe & upper lobe handling tools OHME (Overhead Mechanical Equipment) and XHF (Handling Fixture) given load condition as per Boeing D-doc and FE modelling and Static analysis of Boeing 787x PME272W3127-1 (Portable Mechanical Equipment) for the A Frame Install Tool for given load condition.

Description: Finite element modeling of OHME-1/2 & XHF-1/2 was modelled using combination of **1D & 2D (Shell) elements** and connection were done (**RBE2, Springs** etc.) and tool mass and C.G was matched with CAD model using NSM/density adjustment. Mass and Centre of gravity (C.G) was provided and applied using **CONM2** and connected using **RBE3**. Various boundary condition was applied and **linear Static analysis** of tool where performed under various given **gravitational load conditions**. All standard pre-processor and postprocessor checks where performed, and Loads were extracted and sorted for critical condition at various parts to performed hand calculation. Results summary and conclusions were reported.

Detailed **FE models and analysis** of different attachment components were performed. **Hand calculations** using classical approach like fastener group, lug and pin, bracket sections analysis were made and reported.

Project #3: FE modeling of BIW and plastic components of Automobile:

Description: Finite element modeling of **BIW components** like **Dash and cowl, Rear floor, Inner panel and plastic components** like **Seat back rest, Glove Box, Console, fascia, exterior trim, front bumper and lower trim** and necessary geometry clean ups were done. Quality parameters and setting up of **Material/Properties collector** according to requirement. After that **FE checks**, connectivity**, element quality, duplicates, Normals** ensured as per client requirement for various models.

Project #4: Static strength and Modal analysis of Motor bike pedal.

Description: Two wheeler motor bike brake pedal **Linear Static Analysis** was performed under normal load, **fatigue calculations** were done and also checked **for limit and ultimate loads**. Brake pedal was calculated for Fatigue life with scatter factor 3 using SN (stress life) approach. Component Modal Analysis were also computed. Analysis is carried out by FE modelling was done using **2D quad** elements, components are connected using **3D Penta** elements as weld. All standards pre- processor **FEM checks** are performed. **Fatigue life calculations** were done using **stress life approach** and the **Goodman**'s equation is considered.

Project #5: Strength and model analysis of Hanger Bracket and stiffness calculation at the attachments:

Description: FE modeling was done using **2D shell** elements in pipe and hanger and the **3D Hexa** elements in hanger rod and in welds.**Rbe2** elements are created at the ends of the hanger rod. All standards pre- processor **FEM checks** are performed and Static Analysis of the hanger bracket was performed under given multi loading condition and reported.

Project #6: FE modelling of composites structure and Optimization of Fuel Tank.:

Description: Finite element modeling of Composite fuel tank was modeled using combination of **2D Shell** elements and connection were done (**Rbe2, Springs (CBUSH) etc**.) and tool mass and C.G was matched with CAD model using density adjustment. Mass and Centre of gravity (C.G) was provided and applied using **CONM2** and connected using **RBE3**. All standard pre-processor and postprocessor FEM checks are performed. **Linear static analysis** to be performed under various inertia load cases. Results summary and conclusions were reported.

Project #7: Static strength and Modal analysis of automotive exhaust cold end sub assembly:

Description: Linear Static Analysis of the exhaust cold end assembly (Flanges Pipes, and Hanger) were carried under **1g** and **bad road loads**(**5g-Ver, 3g-Lat, 2g-Axial**). Modal Analysis of the assembly were computed. FE modeling of the pipe was carried out with 2D shell. Flange, hanger are modelled by using 3D HEX elements and isolator is modelled using CBUSH elements. Weld were represented using **Hexa/Penta** elements and **bolts using rigid RBE2** elements. All standard FEM checks are performed along with **Renumbering**. Carried analysis and Results were documented using MS PowerPoint.

❖ Academic Projects:

"Design and Fabrication of Footstep Power Generation System"

The Non-conventional method of electric power generation by simply walking or running on the footstep. The force acting on the foot step is converted into electrical energy using the electric Generator. Rack and pinion setup is used to translate linear to rotary motion and the chain drive and gear drive enhance the speed ratio. And the helical springs are used as elastic element to restore its original position.

"Finite Element formulation of four node quadrilateral plate element for Composite Lamina using third order shear deformation theory TOSDT and analysis of Composite lamina with various plate theory".

A C_0 continuous displacement finite element formulation of a TOSDT for thick arbitrary laminated composite plates under transverse loads. The assumed displacement model eliminates the use of shear correction coefficients. The discrete element chosen is a four nodded quadrilateral with five degrees-of-freedom per node. Results for plate deformations, internal stress-resultants and stresses for selected examples are compared with the finite element solutions with another higher-order elemental displacement with computer based FE-software (Nastran). A computer program using MATLAB coding which incorporates the realistic prediction of inter-laminar stresses from equilibrium equations.

❖ Declaration:

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

PRAVEEN KUMAR N