**GANESH SUDHAKAR THORKAR**

Mobile: +91 90411 99799

E-Mail:[gthorkar@gmail.com](mailto:gthorkar@gmail.com) /[gthorkar@outlook.com](mailto:gthorkar@outlook.com)

***Seeking an excellent opportunity as a FEA Engineer to utilize my technical knowledge& experience that offers professional growth while being resourceful, innovative and flexible.***

**Academic Qualifications**

* Masters of Engineering (M.E.) - “Design Engineering” with CGPA of 9.08 from **Birla Institute of Technology and Science-Pilani** in 2010.
* Bachelorsof Engineering (B.E) –“Mechanical Engineering” with 75.06% from **Swami Ramanand Teerth Marathwada University, Nanded** in 2007.

**Software Skills–ABAQUS,ANSYS, Hyper-works, FE-Safe andPython**

**Work Exposure**

**Experience : Working since September 1st, 2016 to till date**

**Company : Senior Engineer-FEA, Oceaneering International India Ltd. Chandigarh, INDIA**

***Key Deliverables***

* Performing linear & non-linear (elastic-plastic) structural analysis of the Subsea Production Systems.
* Crank shaft bore distortion analysis of an engine assembly using ABAQUS.
* Elastomeric seals analysis using hyper-elastic material models to study the sealing behaviour.
* Crack growth analysis (fracture mechanics) in accordance with API 579-1/ASME FFS-1.

***Key Responsibilities***

* Handling the entire gamut of the activities pertaining to FE Analysis & calculations viz.
* Requirement & input gathering for FE Analysis as per load case defined for new product development.
* Performing linear, non-linear structural, vibration and fatigue analyses.
* Carrying out strength calculations and fatigue calculations.
* Analysisreport preparation, presenting FEA results and design recommendations to the internal/external customers.
* Pivotal role in analysis, optimization, loading calculations and lauded for active participation and presenting globally independent design reviews.
* Working with various global FEA teams across different geographic locations.
* Responsible for establishing FEA process flow & process checklist, creating simulation guidelines/methodology for various load cases.
* Responsible for technical review and participate actively in quality reviews of peers.
* Active participation in projectreview meeting and communicating the work plan for on time delivery.

**Experience : Working sinceNovember 1st, 2012 to August 30th 2016.**

**Company : Senior Engineer-FEA, Aker Solutions Pune, INDIA**

***Key Deliverables***

* Performing linear &non-linear (elastic-plastic) structuralanalysis of theSubsea Production Systems.
* Sealing analysis of the oil pan seals using ABAQUS/Explicit.
* Vibration (modal) analysis of the subsea structure using ABAQUS.
* Thermal structural analysis using ABAQUS.
* Hand calculations of structural components involving welded joints, threaded joints using MathCAD/Excel.
* Python scripting for pre and post processing.
* Preparation of the technical (calculation) report of the FE analyses performed.
* Furnishing design recommendations to the designers based on the FEA results for the optimization.
* Communicating with the customers/concerned partners regarding FEA approach and calculation reports.
* Working with various global FEA teams across different geographic locations.
* Responsible for technical review and participate actively in quality reviews of peers.

**Experience : Worked from October 04th, 2010 to October 24th, 2012**

**Company : Project Engineer, DANA India Technical Centre Pune, INDIA**

***Key Deliverables***

* Thermal structural analyses of the powertrain components viz. Cylinder Head Gaskets, Exhaust manifold Gaskets.
* Crank shaft bore distortion analysis of an engine assembly using ABAQUS.
* Non-linear structuralanalysis Engine oil coolers (EOC)& Transmission Oil Coolers (TOC) using ABAQUS.
* Elastomeric seals analysis using hyper-elastic material models to study the sealing behaviour using ABAQUS.
* Durability (fatigue) analysis of EOC and TOC using FE-Safe.
* Vibrational (Random response and steady state) analysis Engine Oil Coolers & Transmission Oil Coolers.
* Pre-processing (Solid hex, tetra and shell meshing)of components using Hypermesh.
* Preparation of the calculation report of the FE analyses performed.

**Professional Projects**

* ***Company* : *Dana India Technical Centre, Pune***

|  |  |  |
| --- | --- | --- |
| **Title** | **:** | **Structural analysis of Power Take off Unit (PTO) of the powertrain under different loading conditions.** |
| Tools Used | : | ABAQUS, Hyper-mesh, Ms-Excel and Pyhton |
| Scope | : | Evaluation of the structural integrity of the PTO Power unit housing of an harvester engine under inertial loads, bearing loads and pump weights. Performing bolted joint analysis of PTO housing joint to study contact pressure at the interface of joints and leakage behaviour between mating faces. This involved the modelling all PTO parts and engine in the Hyper-mesh, considering the geometry and contact nonlinearity. Non-linear materials were considered for the FE analysis. Deck preparation and analysis was done using ABAQUS. |

* ***Company* : *Dana India Technical Centre, Pune***

|  |  |  |
| --- | --- | --- |
| **Title** | **:** | **Coupled thermal structural FE analysis of the powertrain (engine assembly) under different operating conditions.** |
| Tools Used | : | ABAQUS, Hyper-mesh, Ms-Excel |
| Scope | : | The project involved pre-processing of Cylinder Head Gaskets (CHG), engine head, engine block, cylinder liners etc. in the Hyper-mesh. Deck preparation for the structural analysis in the ABAQUS, involving bolt loads, peak cylinder pressures and thermal map from the thermal analysis for the different operating conditions of hot fire and cool down. |

|  |  |  |
| --- | --- | --- |
| **Title** | **:** | **Vibration analysis and Durability analysis of Engine Oil Coolers (EOC).** |
| Tools Used | : | ABAQUS, FE-Safe, Hyper-mesh |
| Scope | : | This project involved structural & vibrational aspects; the structural analysis is simulated for maximum static pressure inside EOC considering geometric, material and contact non-linearity to study the stresses developed in the critical regions. Vibrational aspect included the modal analysis for extraction Eigen modes and natural frequencies, followed by harmonic analysis to estimate its durability performance by evaluating the life in the critical areas. The Pre-processing of cooler was carried out using hyper-mesh. Deck Preparation and post processing using ABAQUS. Fatigue analysis is done using FE-safe. |

**Academic Projects**

|  |  |  |
| --- | --- | --- |
| **Title** | **:** | **Investigation of effect of notch radius and thickness of CT specimen on fracture parameters.** |
| Duration | : | 6 Months |
| Scope | : | The project gives an idea about of how the crack initiates and propagates in the CT specimen. The detailed study involving experimentation and simulation in the ABAQUS were carried out on the CT specimen with varying notch radius and thickness. Their effect on the fracture parameters like J-integral and CTOD was found out |

**Strengths**

* An effective communicator with excellent interpersonal & relationship building skills.
* Analytical attitude and problem solving skills.
* Adapt in cementing healthy relationship with the clients for generating business.
* Possess a flexible, learning & detail oriented attitude.

**Personal Details**

* Language skills : English, Marathi, Hindi
* Date of Birth : 24th January 1984
* Residential Address : Second Floor, House No. 950, Sector – 10, Panchkula, Chandigarh, 134109