

Aditya Subhash Chougule

BE (Mechanical Engineering)

Email Id: adityachougule3387@gmail.com

Mobile No. 7709625249

CAREER OBJECTIVES

To serve an organization which provides me challenging carrier to bring out the best of my creative potential, which gives an opportunity to learn new things and to excel in my field of endeavor.

WORK EXPERIENCE

Work Experience: 4 years 10 Month (February 2017-Present)

<i>Organization</i>	DANA Technical Centre Pune(DITC)
<i>Designation</i>	Project Engineer FEA
<i>Experience</i>	4 Months
<i>Responsibilities</i>	<ul style="list-style-type: none">-To perform different types of analysis to check the Sealing capability of DANA cylinder head gaskets, various secondary gaskets.-Types of analysis: Structural analysis (Linear and Non-linear) Thermal analysis(steady state and transient), Sealing Analysis, modal analysis and Harmonic analysis.-Globally supported projects & actively involved in project discussions with design team.-Meshing of Different Engine parts in Hypermesh and Simlab with proper element quality checks.
<i>Organization</i>	Cummins Technical Centre India, Pune(Onsite engineer)
<i>Designation</i>	FEA Engineer
<i>Experience</i>	2 year 10 month (12th November,2018 - Present)
<i>Responsibilities</i>	<ul style="list-style-type: none">-To perform different types of analysis to check the structural integrity of engine exhaust parts & provide design recommendations based on results.-Types of analysis: Structural analysis (Linear and Non-linear) Thermal analysis(steady state and transient), Thermo-mechanical fatigue analysis, modal analysis and Harmonic analysis.-Globally supported projects & actively involved in technical discussions with specialists from different teams (design, material ,CFD etc.)-Preparing timely reports on progress of task & writing technical report at the end of project.-Supporting to team members & other teams in the department in problem solving and deciding analysis approach of different analysis.

<i>Project-I</i>	<p>Thermo Mechanical Fatigue Analysis of Exhaust Manifold</p> <p>Determine the crack initiation life & provide design recommendations to various types of exhaust manifold for thermal & structural loading by performing thermal mechanical fatigue analysis using Ansys Workbench & TFAT (Thermo mechanical Fatigue Analysis Tool).</p>
<i>Project-II</i>	<p>Modal Factor of Strength Analysis on Exhaust Manifold</p> <p>Perform thermal, static structural & modal analysis to calculate the resonant frequencies of exhaust manifold-turbocharger system & calculate fatigue life through a factor of strength from scaled mode shapes.</p>
<i>Project-III</i>	<p>Modal, Static and DBJ analysis of Intake Manifold</p> <p>Perform static-fatigue and modal analysis to check structural integrity of Intake manifold. Extracting bolt reaction forces to check strength & load carrying capacity of bolted joints.</p>
<i>Project-IV</i>	<p>Hot Bolted Joint Analysis on Exhaust System Joints</p> <p>Calculate margins to check sustainability of joints for structural & thermal loading & provide recommendations accordingly.</p> <ul style="list-style-type: none"> - First cycle yield margin, Gross slip margin, Bolt proof margin, Dynamic load margin, Thread strip margin & Clam Load Margin
<i>Project-V</i>	<p>Gasket Sealing Analysis at Exhaust Manifold & Cylinder Head Interface</p> <ul style="list-style-type: none"> - To determine the contact gap, sealing pressure, sliding distance to ensure the leak proof contact between exhaust manifold & cylinder head.
<i>Project-VI</i>	<p>Shrinkage and Warpage analysis of Exhaust manifold</p> <ul style="list-style-type: none"> -To determine the shrinkage phenomenon taking place in manifold ports after the 250-500 hrs. of thermal testing in Analysis itself. -To determine warping of manifold flange and check for the possible solutions if it exceeds the limit.
<i>Project-VII</i>	<p>Harmonic analysis to investigate the Bolt Joint Failure and Manifold Vibration issue</p> <ul style="list-style-type: none"> -Perform Harmonic analysis to predict the actual g levels observed on Manifold in testing and use those to get root cause of bolt loosening and manifold high vibrations. -Provide recommendations to avoid the failure based on results.
<i>Achievement</i>	<ul style="list-style-type: none"> -Recognized by ‘spot award-2020’ for special contribution in one of the Firg issue in Cummins technical Centre India, Columbus. -Recognized by ‘spot award-2021, for extra mile contribution to newly developed VPI program X14N. -Recognized by Cummins global team for implementing a new approach in manifold analysis process.

	<ul style="list-style-type: none"> -Implemented automatic scripting process for HBJ margin calculations. -Reduced thermo mechanical analysis run time in Ansys Workbench by 7-8 hrs.
<i>Organization</i>	Analyzer CAE Solutions Pvt.Ltd.,Pune
<i>Designation</i>	FEA Engineer
<i>Experience</i>	1 year 9 months (February 2017- November 2018)
<i>Job Role</i>	Structural, Modal Analysis, Report Preparation
<i>Project-I</i>	Static analysis of Winglet ULR-WingTip-Subjig-Frame FEA -Carried out structural analysis for optimization frame structure. Critical region in frame were fine meshed as per customer requirement. During analysis, parts which are not in area of interest were suppressed and point masses applied instead of it.
<i>Project-II</i>	Static and Modal analysis of a Platform -Hollow tubes with rectangular cross section for a platform that supports mechanical equipment. This structure is modeled using ANSYS workbench. A modal analysis was performed to check whether the natural frequencies of structure are in the range of excitation frequencies because of operation of the equipment mounted. Structural analysis was performed to find out stresses on the structure because of the weight of mounted equipment.
<i>Project-III</i>	Structural analysis of refrigerator Bin -Static structural analysis was carried out to verify the load carrying capacity of refrigerator Bin. Bilinear material was assigned for Bin. Frictional contacts were defined at appropriate locations.
<i>Project-IV</i>	Static analysis of pressure vessel -cylinder pressure vessel with covering plates bolted to its openings was analyzed using ANSYS workbench. Bolt pretention and internal pressure was applied on the vessel and static analysis was carried out to determine stresses in the vessel. The stresses were found to be within mate
<i>Organization</i>	KIRLOSKAR OIL ENGINES-
<i>Designation</i>	Project Intern
<i>Experience</i>	2 months (Vocational training)
<i>Project</i>	Standardization of Spares Under this project I worked with maintenance department and checked for Spares quantity optimization project.

EDUCATION

<i>June 2016</i>	Bachelor of Technology in Mechanical
<i>Institute</i>	D.Y.Patil college of engineering and technology
<i>Percentage</i>	71.94%

<i>May 2012</i>	HSC (with Mechanical Maintenance)
<i>Institute</i>	New College, Kolhapur
<i>Percentage</i>	75.50%
<i>June 2010</i>	SSC
<i>Percentage</i>	92.91 %

ACADAMIC PROJECT ACCOMPLISHED

<i>Academic Title</i>	B.E.
<i>Description</i>	Comparative assessment between standard and wiper insert on turning of Titanium Alloy Study the effect on surface roughness if Ti6A14V alloy with change in speed, feed and depth of cut during machining with standard and wiper insert.

TECHNICAL COMPETENCE

1. Ansys | Static structural linear analysis with basic calculations for stress-life based fatigue, Static structural non-linear analysis – Material, Contact and Geometric non-linearity and Dynamic Analysis –Modal, Harmonic response and random vibration
2. Creo Parametric
3. TFAT (Thermo mechanical Fatigue Analysis Tool)
4. Hyper mesh(Preprocessing)

PERSONAL DETAILS

<i>Date of Birth</i>	22 nd March 1994
<i>Address</i>	A4-1002, The grater ganga panama, Pimple Nilakh, PCMC, Pune.
<i>Gender</i>	Male
<i>Language</i>	English, Hindi and Marathi
<i>Hobbies</i>	Playing and following Football,Trekking
<i>Strength</i>	Hard working, Creativity and willingness to learn

I hereby affirm that the information provided is true to best of my knowledge.

Aditya Subhash Chougule