CURRICULUM VITAE

Name: Ayyappa Jammula

Ph. No.: +919989989602

Current Location: Hyderabad

Email: [ayyappajammula03@gmail.com](mailto:ayyappajammula03@gmail.com)

|  |  |
| --- | --- |
| Degree: | B.E Mechanical |
| Career Level: | Fully Competent |
| Occupation: | CAE ANALYST |
| Career Title: | Mechanical Engineer, CAE Analyst |
| Target Title: | CAE Durability |
| Target Locations: | Hyderabad ,Bangalore, Pune, Chennai |
| Skills: | Hyper Mesh, ABAQUS, Optistruct, Hyper view, hyper graph |
| Goal: | Technology specialist in durability. Working with energy, passion & learning |
| Highlight: | Total years of experience: 4.6  Present Designation: CAE Analyst.   * Working as FEA engineer in Automotive Design & Engineering Solution from June 2017 till date * Highly motivated and skilled FE modeler and analyst with 4yers of experience in Hyper Mesh and Abaqus * Expertise in Abaqus to deal with linear and non-linear analysis for automobile components. * Best in class knowledge in durability analysis at various levels of product life cycle. * Extensively worked for component level which includes sub models preparation, data check, setting up boundary condition, solving, debugging and post processing. * Providing necessary suggestions based upon observations for geometric changes. * Well-developed Interpersonal and Communication skills with excellent Documentation and Presentation skills. Good team player with ability to grasp new concepts quickly and implement them in a productive manner   Software skills:  Pre-processors: Hypermesh  CAE Solvers: ABAQUS, Optistruct  Post-processors: Hyper-view & Hyper Graph. |
| Summary: | * Hand on experience in nonlinear analysis , linear, Optistruct * Mechanical engineer with great knowledge of finite element analysis with 4 years of experience in FEA model building, post processing, linear and nonlinear static analysis. |

# LIVE PROJECT AND ANALYSIS WORK EXPERIENCE:

**Job Responsibilities (Automotive Design & Engineering Solution from June 2017 to till date**

* Creating the finite element model of various types of parts based on the type of analysis.
* Performing linear and nonlinear static analysis and observing the results.
* Coordinating with team for developing suitable changes on design to get optimum results.
* Preparing reports of the project for summarizing the analysis results for ensuring the quality and accuracy of project.
* providing necessary suggestions on geometric changes based upon keen observations

**Professional Projects**

Worked for automobile steering components from base model to production model and performed various linear and nonlinear static analysis. To achieve the targets at various levels, product design is optimized with necessary geometric solutions.

Following are some of accomplished projects.

**Project 1:**

|  |  |  |
| --- | --- | --- |
| **Client / Project Name** | **BMW -Sub frame/ Non-Linear static analysis** | |
| **Role** | Team member | |
| **Environment (with skill versions)** | Software | Hypermesh17.0, Optistruct, Abaqus 2017 (Implicit) |

**Project description:**

Fea model built for Subframe which is under steering assembly using Rtria, Tetra, hexa elements and various rigid elements. Constraining with boundary conditions with respect to assembly links and applying various loading conditions.

Major analysis performed is Frequency response analysis and static stiffness.

**Project 2:**

|  |  |  |
| --- | --- | --- |
| **Client / Project Name** | **BMW-Rear Knuckle/ Non-Linear static analysis** | |
| **Role** | Team member | |
| **Environment (with skill versions)** | Software | Hypermesh13.0, Abaqus 6.12 (Implicit) |

**Project description:**

Fea model built for rear knuckle which is under steering assembly using Rtria, Tetra, hexa elements and various rigid elements. Constraining with boundary conditions with respect to assembly links and applying various loading conditions.

Major analysis performed is fracture loading analysis, pretension analysis.

**Fracture loading analysis:** Loads are applied in link directions with respect to the assembly links and unloading the loads to find permanent deformation.

**Pretention analysis:** assembly components are attached to links using contacts, contact properties and applying pretension force to calculate contact force and deformation.

**Project 2:**

|  |  |  |
| --- | --- | --- |
| **Client / Project Name** | **AIWAYS-Front Knuckle / Non-Linear static analysis** | |
| **Role** | Team member | |
| **Environment (with skill versions)** | Software | Hypermesh13.0, Abaqus 6.12 (Implicit) |

**Project description:**

FEA model built for front knuckle which is under steering assembly using Rtria, Tetra, hexa elements and various rigid elements. Constraining with boundary conditions with respect to assembly links and applying various loading conditions.

Major analysis performed is Strut push through analysis, pretension analysis followed by service loads.

**Strut push through analysis:** Displacement load (BC) applied in link directions with respect to the assembly links to find out the max force.

**Pretention analysis:** assembly components are attached to links using contacts, contact properties and applying pretension force to calculate contact force and deformation. After pretension service load is applied at that link to observe the deformation of the link.

**Project 3:**

|  |  |  |
| --- | --- | --- |
| **Client / Project Name** | **VW-Rear Knuckle / Non-Linear static analysis** | |
| **Role** | Team member | |
| **Environment (with skill versions)** | Software | Hypermesh17.0, Abaqus 6.14 (Implicit) |

**Project description:**

FEA model built for Rear knuckle using Rtrias elements with quad, tetra, and hexa elements along with various rigid elements and contact surfaces. Constraining with boundary conditions with respect to assembly links and applying various loading conditions.

Analysis performed is Stiffness, linear and non-linear load cases.

**Project 4:**

|  |  |  |
| --- | --- | --- |
| **Client / Project Name** | **GM-Control arms / Non-Linear static analysis** | |
| **Role** | Team member | |
| **Environment (with skill versions)** | Software | Hypermesh17.0, Abaqus 6.14 (Implicit) |

**Project description:**

Fea model built for Rear Upper control arm using Rtria, Tetra, hexa elements and various rigid elements. Constraining with boundary conditions with respect to assembly links and applying various loading conditions.

**Press fit:** bushes are attached to links using contact surfaces, applied clearances and press in forces to find the deformation.

**Press out:** bushes are attached to links using contact surfaces, applied clearances and pushed out to find the deformation and reaction forces.

**Ball Joint Stud Draw in:** assembly components are attached to links using contacts, contact properties and applying displacement at pretension node to calculate contact force and deformation.

**Project 5:**

|  |  |  |
| --- | --- | --- |
| **Client / Project Name** | **Ford –Front spindle/ Non-Linear static analysis** | |
| **Role** | Team member | |
| **Environment (with skill versions)** | Software | Hypermesh17.0, Abaqus 6.14 (Implicit) |

**Project description:**

Fea model built for Front Spindle which is under steering assembly using Rtria, Tetra, hexa elements and various rigid elements. Constraining with boundary conditions with respect to assembly links and applying various loading conditions.

Major analysis performed is Stem press fit followed by lateral load to observe the bending of stem.

**Project 6:**

|  |  |  |
| --- | --- | --- |
| **Client / Project Name** | **Geely-Front Knuckle / Non-Linear static analysis** | |
| **Role** | Team member | |
| **Environment (with skill versions)** | Software | Hypermesh17.0, Abaqus 6.14 (Implicit) |

**Project description:**

Fea model built for Front knuckle which is under steering assembly using Rtria, Tetra, hexa elements and various rigid elements. Constraining with boundary conditions with respect to assembly links and applying various loading conditions.

Linear Static loads analysis performed, when assembly experiences loads, each link deforms respectively. So, it’s important to know the stiffness of all links so ensure the structural integrity. So, unit load is applied on each link in the respective directions and deformation is noted. Then the stiffness is calculated by dividing the deformation per unit load. If the link stiffness is below the target limits, link is redesigned to strengthen.

## EDUCATIONAL DETAILS:

|  |  |  |  |
| --- | --- | --- | --- |
| University / Institute | Qualification | Year of Passing | Percentage |
| Gudlavalleru Engineering College | B.E Mechanical | 2017 | 70.93 |
| Sri chaitanya Jr college - Guntur | Intermediate | 2013 | 95.3 |
| Balaji High School-Andhra Pradesh | SSC | 2011 | 93.67 |

## EXTRACURRICULAR ACTIVITIES:

* Kho-Kho Champions in JNTUK C-Zone tournament.
* Organize the college annual day and other technical events.

**STRENGTHS**

* Ability to work hard for project or purpose. Excel myself at solving technical or abstract problems. Have strong willingness to walk extra miles to achieve excellence & flexibility.
* Creating high degree of standards in system or project. Prefer new ideas over old and offers them frequently.
* Manage to build good team relationships.
* The exceptional problem solving ability. Prefer to consider new and better ways to do things.

**PERSONAL DETAILS:**

Name : Ayyappa Jammula

Nationality : Indian

Languages Known : English, Telugu and Hindi.

Current Location : Hyderabad

Date of Birth : 26-03-1996

Marital Status : Single

Hobbies : Browsing, Watching movies, Playing Cricket.

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

Place: Hyderabad. Date: