Wissen Baum Engineering Solutions

# Profile for Onsite Support

**Date: 17 November 2025**


**Wissen Baum Engineering Solutions**

**Germany | USA | India**


**Code: 8091**
**Domain: Crash**

**Resource CODE:** WB/CAE/8091/Aniket K

**Educational Qualification:**

- Bachelor of Engineering - Mechanical stream (4 years Program)

**Total Experience:**

- 4+ Years

**Software skills:**

- **Pre-process:** Hypermesh
- **Solution:** LSDyna
- **Post-process:** Hyperview, Hypergraph
- **Others:** MS tools

**Key Highlights:**

- Having 4+ years of experience in Automobile domain.
- Having good knowledge of Engineering Mechanics, strength of material and Machine design.
- Skilled communicator with leadership and quick learner with an analytical bent of mind coupled with zeal to utilize and enhance ideas, knowledge and skills.
- Exceptionally well organized with track records that demonstrate self-motivation, creativity and initiative to achieve both personal and corporate goals.

**Job Profile:**

- The mesh generation with given quality criteria followed by a quality assurance process using checklists for shell and solid meshing, variable thickness assignments, intersection penetration checks, and removal
- FE model buildup phase, which includes sub-assembly level model building and connection preparation for Door trims, Center console, Instrument panel etc. using Hypermesh
- The material definitions such as mat elastic, mat elastic-plastic, mat null, mat rigid, etc. property assignment such as (ELFORM, NIP, SHRF), contact definition, time history output blocks for nodes and elements, defining database cross-section, control cards, loading and boundary schemes, etc.
- Initial simulation study for intersections, penetrations, mass scaling, and connection scheme
- Post-processing (result interpretation) using Hyperview with key outputs such as energy balance, mass balance, force balance, force transfer path, deflections, stresses, failure modes etc.

- Executive reports preparation, with outputs such as deformation and plastic strain views at critical locations, regulation-specific observations, and other basic information related to the simulation model.
- Communicating with the team manager for effort evaluation, work-in-progress status, submission of final reports.

## Project – 1

**Client Name:** Leading European OEM
**Project Name:** Luggage Retention Test for Front Seat as per ECE R17 Compliance
**Solver:** LSDyna
**Software:** Hypermesh, Hyperview and Hypergraph
**Role:**

- Developed the finite element model (FEM) for the front seat luggage retention test, ensuring full compliance with ECE R17 regulations. This involved creating an accurate seat model, including all necessary components and material properties.
- Conducted meshing of seat components, as per quality criteria with thickness assignments. Performed quality checks, including intersection and penetration checks to ensure the integrity of the model.
- Applied material properties to seat components and set up connections like bolt, weld, joints, springs and ect. Assigned contacts between the seat and the luggage block for accurate simulation results. Creating time history evaluations.
- Applied acceleration loads, simulating the luggage retention using an unrestrained 20kg luggage block as per test specifications. Ensured that all conditions followed ECE R17 requirements for luggage retention during crash events.
- Interpreted simulation results using Hyperview, analyzing key outputs such as deflections, stresses, and plastic strain to evaluate seat performance under the test conditions.
- Compiled and delivered detailed executive reports, including animations of test scenarios, and stress/strain plots.
- Regularly communicated with the team manager to provide effort evaluations, update on work-in-progress, and ensure timely submission of final reports to team manager.

## Project - 2

**Client Name:** Leading European OEM
**Project Name:** Armrest Strength Test on Rear door assembly
**Solver:** LSDyna
**Software:** Hypermesh, Hyperview and Hypergraph
**Role:**

- Prepare the FE model for Armrest Strength Test on Rear door assembly using Hypermesh
- The main activities are meshing, variable thickness assignment, material assignment, quality checks and intersection, penetration checks
- Model building such as assigning the connections for snaps, assigning contacts, creating time history evaluations, positioning the spherical loading device having 60mm diameter perpendicular to the surface on the armrest, defining contact between them, applying a load of 70daN using loading device, applying boundary condition etc, and debug run is performed
- Post-processing is performed to analyze the animation of interaction between the loading device and armrest, check energy balance plot etc. using Hyperview and Hypergraph
- Generating executive report with animations, measuring deformations and its contours, stresses, and plastic strain plots.
- Regularly communicated with the team manager to provide effort evaluations, update on work-in-progress, and ensure timely submission of final reports to team manager.

## Project - 3

**Client:** Leading European OEM
**Project Name:** Misuse vertical load on Armrest of Center Console Assembly
**Solver:** LSDyna
**Software:** Hypermesh, Hyperview and Hypergraph
**Role:**

- Prepare the FE model for misuse vertical load on the armrest of the center console assembly using Hypermesh
- The main activities are meshing, variable thickness assignment, material assignment, quality checks and intersection, penetration checks
- Model building such as assigning the connections for snaps, assigning contacts, creating time history evaluations, applying a load of 400N in the vertical direction on the armrest, applying boundary condition etc, and debug run is performed
- Post-processing is performed to analyze the missing connections and if any other, the interaction of the armrest with adjacent parts, check energy balance plot etc. using Hyperview and Hypergraph

- Generating executive report with animations, measuring deformations and its contours, stresses, and plastic strain plots.
- Regularly communicated with the team manager to provide effort evaluations, update on work-in-progress, and ensure timely submission of final reports to team manager.

## Project - 4

**Client Name:** Leading European OEM
**Project Name:** Stiffness test of Glove Box of IP Assembly
**Solver:** LSDyna
**Software:** Hypermesh, Hyperview and Hypergraph
**Role:**

- Prepare the FE model for misuse load on Glove Box
- The main activities are meshing, variable thickness assignment, quality checks, intersection, penetration checks, material assignment, model building such as assigning the connections for snaps, assigning contacts, creating time history evaluations
- Applying a load of 100N in the normal to surface on the open position of the Glove Box, applying boundary condition and debug run is performed
- Post-processing (result interpretation) using Hyperview with key outputs such as deflections, stresses and plastic strain
- Generating executive report with animations, deformation, stresses and plastic strain plots.
- Regularly communicated with the team manager to provide effort evaluations, update on work-in-progress, and ensure timely submission of final reports to team manager.