CHAPTER 5

5.0 FINITE ELEMENT ANALYSIS

➢ For a more sophisticated analysis of a structure, considering the composite panel to be subjected to a combination of forces, a technique such as Finite Element Analysis (FEA) might be used.

➢ In general terms, the shear forces normal to the panel will be carried by the each layers. Bending moments and in-plane forces on the panel will be carried as membrane forces in the facing skins.

➢ For many practical cases, where the span of the panel is large compared to its thickness, the shear deflection will be negligible. In these cases, it may be possible to obtain reasonable results by modeling the structure using composite shell elements.

➢ It should be noted that the in-plane stiffness of the composite panels is negligible compared to that of the facing skins.

➢ Where a more detailed model is required it is possible to model the composite panels using solid 3D elements. Attempts to model the individual layers of the composite panels should be avoided for normal engineering analyses.

➢ After successful modeling of the composite panel in CATIA V5, the model is imported into ALTAIR HYPERMESH V8.0 where a meshed model is prepared.

➢ The constraints required as per experimental norms are affixed in order to simulate the experimental conditions as close as possible.

➢ The virtual simulation & analysis is carried out in LS-DYNA using LS-PREPOST as the prime post processor.
5.1 ALTAIR HYPERMESH

- Altair Hypermesh one of the prime tools of Altair Hyperworks is built on a foundation of design optimization, performance data management & process automation.

Fig 5.1.1

- HyperWorks is an enterprise simulation solution for rapid design exploration and decision-making. As one of the most comprehensive CAE solutions in the industry, Hyperworks provides a tightly integrated suite of best-in-class tools for modeling, analysis, optimization, visualization, reporting, and performance data management.

- Features of Altair Hypermesh;

1. Expanded Leadership in Modeling and Visualization.
2. Rapid HEX solid meshing.
3. Shrink-wrap meshing to accelerate creation of large Solid models.
5. New automotive crash pre-processor HyperCrash.
6. Enhanced plotting and results manipulation functionality.
7. Technical Superiority in Optimization and Data Management.
10. Free sizing of computers.
Experimental Studies on Laminated Composites subjected to Impact loading

Fig 5.1.2 - Meshed Composite panel model having mesh size of 1mm.

Fig 5.1.3 - A Meshed model of the impactor in use with a mesh size of 1mm

Fig 5.1.4 - Assembled mesh model of Composite Panel & Impactor
Experimental Studies on Laminated Composites subjected to Impact loading

5.2 LS-DYNA.

- LS-DYNA is a general purpose finite element code for analyzing the large deformation static and dynamic response of structures including structures coupled to fluids.
- The main solution methodology is based on explicit time integration. An implicit solver is currently available with somewhat limited capabilities including structural analysis and heat transfer.
- A contact-impact algorithm allows difficult contact problems to be easily treated with heat transfer included across the contact interfaces. By a specialization of this algorithm, such interfaces can be rigidly tied to admit variable zoning without the need of mesh transition regions.
- Other specializations, allow draw beads in metal stamping applications to be easily modeled simply by defining a line of nodes along the draw bead.
- Spatial discretisation is achieved by the use of four node tetrahedron and eight node solid elements, two node beam elements, three and four node shell elements, eight node solid shell elements, truss elements, membrane elements, discrete elements, and rigid bodies. A variety of element formulations are available for each element type.
- Specialized capabilities for airbags, sensors, and seatbelts have tailored LS-DYNA for applications in the automotive industry. Adaptive remeshing is available for shell
Experimental Studies on Laminated Composites subjected to Impact loading

- LS-DYNA has a large material library (>150 material models), much of which is based on constitutive laws developed by researchers, and a robust set of contact laws to handle complex interactions between two objects in the model.
- It is capable of computing traditional structural impact problems in Lagrangian space, in which the finite element mesh is fixed to and deforms with the structures, but LS-DYNA also has features which allows it to solve fluid-structure interaction problems (i.e. a structure landing in water) using a combination Lagrangian-Eulerian meshing technique.

**APPLICATIONS OF LS-DYNA**

1. Automotive Crashworthiness & Occupant Safety.

2. Sheet Metal Forming With LS-DYNA

3. Metal forming applications for LS-DYNA include:
   
   I. Metal stamping  
   II. Hydroforming  
   III. Forging  
   IV. Deep drawing  
   V. Multi-stage processes

4. Aerospace Industry Applications:
   
   I. Blade containment  
   II. Bird strike  
   III. Windshield & engine blade  
   IV. Failure analysis

5. Other LS-DYNA applications include:
   
   I. Drop Testing  
   II. Can and shipping container design  
   III. Electronic component design  
   IV. Glass forming  
   V. Plastics, mould, and blow forming
VI. Biomedical.
VII. Metal cutting.
VIII. Earthquake engineering.
IX. Sports equipment (golf clubs, golf balls, baseball bats, helmets)
X. Civil engineering (offshore platforms, pavement design)

**Fig 5.2.3** - A snapshot of Low-Velocity Impact on Composite Panel during simulation trial using LS-Dyna

### 5.3 Correlation & Optimization in LS-DYNA

#### 5.3.1 Simulation & Analysis

**Fig 5.3.1** - Simulation & Graphs for specimen in LS-DYNA
Experimental Studies on Laminated Composites subjected to Impact loading

Graph 5.1 - Resultant Force Vs Time (Sec)

Graph 5.2 - Matsum Data Vs Time (Sec)

Table 5.1 – Experimental Results of Glass

<table>
<thead>
<tr>
<th>SPECIMEN</th>
<th>MASS (Kg)</th>
<th>HEIGHT (mm)</th>
<th>EXPERIMENTAL RESULTS</th>
<th>LS-DYNA RESULTS</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>LOAD(kN)</td>
<td>ENERGY(J)</td>
</tr>
<tr>
<td>Glass2mm</td>
<td>5.116</td>
<td>500</td>
<td>4.52</td>
<td>6.81</td>
</tr>
</tbody>
</table>
The above snapshots from the simulation trial in LS-DYNA clearly indicate that the impactor does not traverse through the specimen.

The graphs generated are in close agreement with the experimental observation.

The spring back effect of the impactor on impact during simulation is attributed to the limitation in LS-DYNA to replicate & reproduce the experimental results.

The deviation in the Load v/s Time Graph is as a result of the liability of the material card in use.

The Energy v/s Time Graph is precise with the experimental Energy v/s Time Graph with a small deviation of about 2.75 %.

From the simulation it is also observed that a small portion of the region subjected to impact is undergoing cell-buckling.

Simulation & Analysis

![Simulation & Analysis](image1)

**Fig 5.3.2 Simulation & Analysis**

![Graph 5.3](image2)

**Graph 5.3**. Resultant Force Vs Time (Sec)
Graph 5.4 - Internal Energy Vs Time

- The above snapshots from the simulation trial in LS-DYNA clearly indicate that the impactor does not traverse through the specimen.
- The graphs generated are in close agreement with the experimental observation.
- The spring back effect of the impactor on impact during simulation is attributed to the limitation in LS-DYNA to replicate & reproduce the experimental results.
- The deviation in the Load v/s Time Graph is as a result of the liability of the material card in use.
- The Energy v/s Time Graph is precise with the experimental Energy v/s Time Graph with a small deviation of about 5%.
- From the simulation it is also observed that a small portion of the region subjected to impact is undergoing cell-buckling.

Fig 5.3.3 Simulation & Graphs for specimen C-7A in LS-DYNA
The above snapshots from the simulation trial in LS-DYNA clearly indicate that the impactor does not traverse through the specimen.

The graphs generated are in close agreement with the experimental observation.

The spring back effect of the impactor on impact during simulation is attributed to the limitation in LS-DYNA to replicate & reproduce the experimental results.

The deviation in the Load v/s Time Graph is as a result of the liability of the material.

The Energy v/s Time Graph is precise with the experimental Energy v/s Time Graph with a small deviation of about 6%.

From the simulation it is also observed that a small portion of the region subjected to impact is undergoing cell-buckling.

---

**Graph 5.5 - Resultant Force Vs Time (Sec)**

<table>
<thead>
<tr>
<th>SPECIMEN</th>
<th>MASS (Kg)</th>
<th>HEIGHT (mm)</th>
<th>EXPERIMENTAL RESULTS</th>
<th>LS-DYNA RESULTS</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>LOAD(kN)</td>
<td>ENERGY(J)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>LOAD(kN)</td>
<td>ENERGY(J)</td>
</tr>
<tr>
<td>Graphite</td>
<td>5.116</td>
<td>500</td>
<td>3.635</td>
<td>22.085</td>
</tr>
<tr>
<td>2mm</td>
<td></td>
<td></td>
<td>3.0472</td>
<td>26.02</td>
</tr>
<tr>
<td>SPECIMEN</td>
<td>EXPERIMENTAL RESULTS</td>
<td>LS-DYNA RESULTS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Graphite</td>
<td>LOAD(kN)</td>
<td>ENERGY(J)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2mm</td>
<td>LOAD(kN)</td>
<td>ENERGY(J)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Experimental Studies on Laminated Composites subjected to Impact loading

**Graph 5.6** - Internal Energy Vs Time

**Table 5.3** Comparative LS-DYNA Analysis for Specimens

<table>
<thead>
<tr>
<th>SPECIMEN</th>
<th>MASS (Kg)</th>
<th>HEIGHT (mm)</th>
<th>EXPERIMENTAL RESULTS</th>
<th>LS-DYNA RESULTS</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>LOAD (kN)</td>
<td>LOAD (kN)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>ENERGY (J)</td>
<td>ENERGY (J)</td>
</tr>
<tr>
<td>Glass 2mm</td>
<td>5.116</td>
<td>500</td>
<td>4.52</td>
<td>5.86</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>6.812</td>
<td>6.794</td>
</tr>
<tr>
<td>Graphite 2mm</td>
<td>5.116</td>
<td>500</td>
<td>3.635</td>
<td>3.047</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>22.085</td>
<td>26.02</td>
</tr>
<tr>
<td>Carbon 2mm</td>
<td>5.116</td>
<td>1000</td>
<td>4.980</td>
<td>4.374</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>41.624</td>
<td>39.004</td>
</tr>
</tbody>
</table>
5.3.2 Remarks

- Material card used for the purpose of analysis is “*MAT_054-055” (*MAT_ENHANCED_COMPOSITE_DAMAGE).

- The major properties required here are the Elastic Modulus in two perpendicular directions (of fibre orientation), Poisson’s ratio, Shear modulus, Effective Plastic Strain, mass density, failure strength and longitudinal tensile & compressive strengths.

- The deviation from the experimental is attributed to the liability in usage of this card along with the defects in manufacturing.

- Analysis carried out with the aid of ALTAIR HYPERMESH & LS-DYNA were very successful.

- The impact damage characteristics obtained by finite element method were in good agreement with that obtained by the impact test.

- A drop in both the absorbed load & absorbed energy of the sandwich panel was clearly observed with the increase in cell size & thickness of the different composite panel specimens under test.