1. Abstract

This Finite Element (FE) analysis of Hood Release Assembly (manufactured using released the hood under static load where nonlinear material used. A set of experiments are conducted on Plastic material . The experimental study together with FE modelling to capture the delamination failure is presented.

Keywords**: Nonlinear Analysis, Plastic Material, Hypermesh, Abaqus**

2. Introduction

The failure modes in structural plastic material, the initiation of a crack/fracture does not indicate ultimate failure. Generally, a stable crack propagation stage, associated with steady increase in external load, precedes a catastrophic failure. This is frequently observed in structural applications of plastic. In this case, analytical and experimental investigation of stiffeners is studied. Design and assessment of such structures require adequate consideration of strength and fracture toughness.

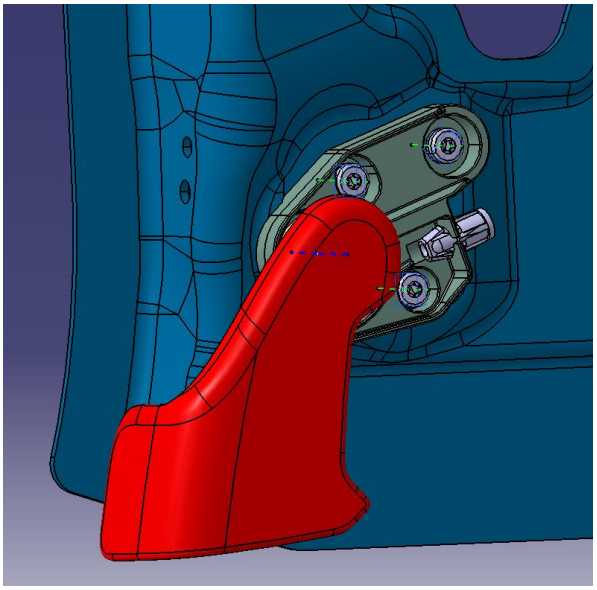


Figure 1: Hood Release Assembly

These bolts load the Bracket through washer plates, which high loadings (combination of tensile / compression, flexure and torsional loads), the design of supporting structure in this region is critical. However, because of complex loading, this region is often over-designed to prevent failure. Thus it is necessary to focus on the structural design of Handle structure to minimize human and material losses, which can be achieved by understanding the tensile and shear stress distribution and their effect on failures. This determines the load-carrying capacity and damage mechanism of hood release assembly.

Bolts usually connect the directly to the brackets between transverse floors and longitudinal, relying on high compressive strength of the Handle. The structural configuration of an attached to a handle pin and is shown in Fig 1. Stainless steel bolts are normally used to attach the bracket.

3. Discussion of FEA approaches

Hood Release Assembly are generally used in the fabrication of stiffened panels such as handle, bracket. One other important use of handle pin is to reinforce the joining of the handle to bracket. Key benefits of Handle pin to attain high bending stiffness and torsional resistance. The prime characteristic of out-of-plane joints using plastic is that, because of a lack of continuity of reinforcing fibers across the joint, they are susceptible to failure by peel or delamination well before the ultimate in-plane Plastic material stress is reached. The authors presented a parametric study on nodal thickness, gap size, back fill angle and fillet radius of top handle. Three loading configurations to analysis the different failure modes of top handle.

4. Numerical Simulation

Finite element simulation of a curved plastic panel is carried out using the commercially available finite element package ABCQUS 6.12. This solver is capable of performing linear and non-linear stress analysis in a static or dynamic framework. In order for it to be employed in this simulation procedure, the solver is equipped with additional isotropic/orthotropic material models for the description of plastic materials to perform the nonlinear static analysis of the structural model.

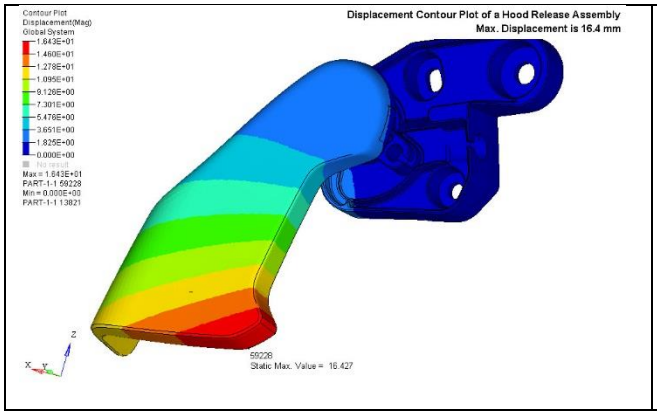
5. Structural Modelling

During experimental investigation, the load was applied at the end of the handle, using a 12 mm diameter bolt at the center of the specimen, at a rate of two mm/min (cross-head movement). FEA was performed on a 3D model. Local co-ordinate systems were defined for each element to match the local orientation.

Three boundary conditions were defined to represent the arrangement existing in the handle experimental analysis. Nodes at the bracket were secured in all directions and a displacement load was applied on handle nodes in the y-direction while the backing plate nodes were fixed in the all direction. Load was applied linearly with respect to time. Contact condition was modelled using the direct constraint method. A single load case was defined in the simulation. Generally, a small number of increments provide an inaccurate damage mechanism while a large number of increments increase the computer runtime; hence a reasonable number of increments needs to be provided. After a certain number of iterations, 200 fixed increments (load steps) were given to identify the damage initiation and progression with high accuracy and acceptable processing time.

6. FEA Results

The numerical analysis was conducted using small strain analysis and steel backing plates were modelled with elastic-plastic behavior. Displacement & Stress distribution at the handle, just before the first failure is shown in Fig.2



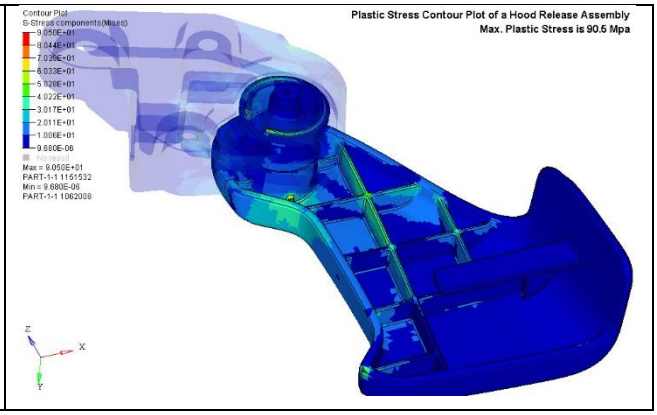


Figure 2: Displacement and Stress Contour Plots of a Hood Release Assembly

7. Conclusion

In this paper, an experimental and analytical investigation of hood release assembly that fail due to delamination is presented. It was observed that a better understanding of the plastic behavior and shear stress distribution makes the design more robust and minimizes material and human catastrophic.

FE analysis of stress distribution around the bends is analyzed. In general the handle is deforming and it was observed that the load is transferred to the bracket, as the bracket is overhang structure and it is deforming to an extent.

The handle failed at the curved section due to excessive high stresses. The strength is a resin dependent mechanical property in the through-thickness direction. The limiting factor for failure initiation is the opening of the handle rather than sliding of the handle pin. The predictions using FEA are in reasonable agreement with the experiments. Experimental results include scatter and uncertainty whereas the finite element model is free from manufacturing and testing inaccuracies.