

# FINITE ELEMENT ANALYSIS OF BIRDSTRIKE ON AIRCRAFT STRUCTURES

By

ROLAND D FURTADO

Spring 2000

## ABSTRACT

The hazard of a bird impact on an aircraft could result in catastrophic damage to the affected aircraft. The Federal Aviation Authorities requires that the airplane be capable of successfully completing a flight during which likely structural damage occurs as a result of impact with a four-pound bird at  $V_c$  at sea level to 8,000 feet. This thesis reports the work carried out in presenting a finite element simulation of a four-pound bird impact on an aircraft component, viz., the wing of an aircraft. All the analyses were carried out using the LS-DYNA finite element software on the 'Hydra' supercomputing unit at N.I.A.R., Wichita State University. One of the most critical factors in the finite element simulation of a birdstrike is the material model selected to represent the bird. A Lagrangian material model, which allowed the definition of an equation of state characteristic of a bird impact, was selected for the analysis. This material model was validated by carrying out finite element analyses on aluminum panels (Al 7075 - T6) of varying thickness (0.25", 0.16" and 0.10"). The results obtained were compared to data obtained from actual tests carried out at Cessna Aircraft Company. The bird, idealized as a cylinder, was imparted with velocities ranging from 265 to 275 knots on the panel constrained along two of its parallel edges. The permanent deformation obtained in the analysis was 0.8 inches for the 0.25" thick and 1.3" for the 0.16" thick aluminum panels. This compared favorably with the deformation obtained in the tests i.e., 1" and 1.5" for the 0.25" thick and 0.16" thick aluminum panels respectively. Material failure was observed in the 0.10" thick aluminum panel in both the finite element simulation as well as in the tests carried out by Cessna. The Von Mises stresses and the plastic strain values reached in the plate were also determined in the finite element simulations. However this data could not be checked, as no corresponding data was available from the tests. The validated Lagrangian bird material model was then used to simulate a bird impact on a wing of an aircraft. The finite element model of the wing was obtained from Cessna. The bird was impacted on the leading edge of the wing with an initial velocity of 275 knots. As failure of the wing was observed due to the birdstrike, a failure model for the shell elements of the wing was incorporated in the analysis. The shell elements reaching the defined limit of 11 % were deleted from the model during the analysis. On primary impact with the wing, over 240 shell elements were deleted from the leading edge. The Von Mises stress levels reached in each of these elements were found to be significantly higher than the ultimate strength of the material. Secondary impact with the wing i.e., impact with the spar, resulted in five elements being deleted

from the spar. Thus the finite element analysis of the birdstrike on the wing showed a complete failure of the leading edge and partial failure of the spar. This thesis demonstrated that explicit finite element simulation presents a realistic and accurate picture of an impact problem. The finite element code LS-DYNA provided an accurate analysis as compared with test data for a bird impact on aircraft structures.