Finite Element Analyses of Adhesively Bonded Composite-Steel Joints for Lightweighting Applications

Yu-Ping Yang, George Ritter, Jerry Gould, Mike Foster, and Jon Jennings

Edison Welding Institute (EWI)

1250 Arthur E. Adams Drive, Columbus, Ohio 43221

Email: yyang@ewi.org; Phone: 614-688-5253

KEYWORDS

Finite element analysis, Adhesive bonding, composite

ABSTRACT

Manufacturers of planes, trains, automobiles, trucks and tractors are seeking for new materials that improve efficiency and reduce weight. Engineering plastics and carbon-fiber composites are popular choices, but they create challenges when they have to be joined to dissimilar materials, such as aluminum, steel or titanium. To assist the design of a bonded joint system for a composite-metal interface, a three-dimensional (3D) finite-element computational procedure was developed in an industrially-funded project. This analysis procedure was used to predict stress and strain distributions, joint strength and failure modes of an adhesive-bonded composite joint during loading.

The computational procedure was developed based on commercial finite element software, ABAQUS. 3D models were used in the procedure in which the metal and adhesive were meshed with solid elements and the composite was meshed with solid elements and cohesive elements. Isotropic elastic and plastic material properties were assumed for both the metal and the adhesive. Orthotropic elastic material properties were assumed for the composite. Progressive damage and failure were modelled by defining failure criteria (damage initiation and evolution) to the adhesive and composite. To validate the analysis method, double lap shear (DLS) tests were conducted and

modelled. The model predictions had good agreement with experimental tested results.

The computational procedure was applied to predict the strength and failure modes of various designed adhesive composite-steel joints at room temperature (23°C) and an elevated temperature (60°C). The failure at 23°C was interlaminar-shear fracture which occurs in the exterior composite skin, while the failure at 60°C was interfacial-shear fracture (cohesive failure) in the adhesive. In addition, finite element analysis was also used to investigate the effect of adhesive thickness on the joint strength and to monitor the change in failure mechanism with changes in joint dimensions.