WRIGHT STATE UNIVERSITY
DEPARTMENT OF MECHANICAL AND MATERIALS ENGINEERING

ME7060 - Structural Reliability
Project Proposal

Koorosh Gobal

February 26, 2014

FLUID-STRUCTURE INTERACTION BASED RELIABILITY ANALYSIS

The wings are one of the important parts of a fixed-wing aircraft. When the aircraft travels forwards, air flows over the wings which have airfoil shape to create lift. Whether flexible or rigid, most wings, have a frame to give them their shape and to transfer lift from the wing surface to the fuselage. The basic structural elements are one or more spars running from root to tip, and many ribs running from the leading (front) to the trailing (rear) edge as shown in Figure 1.

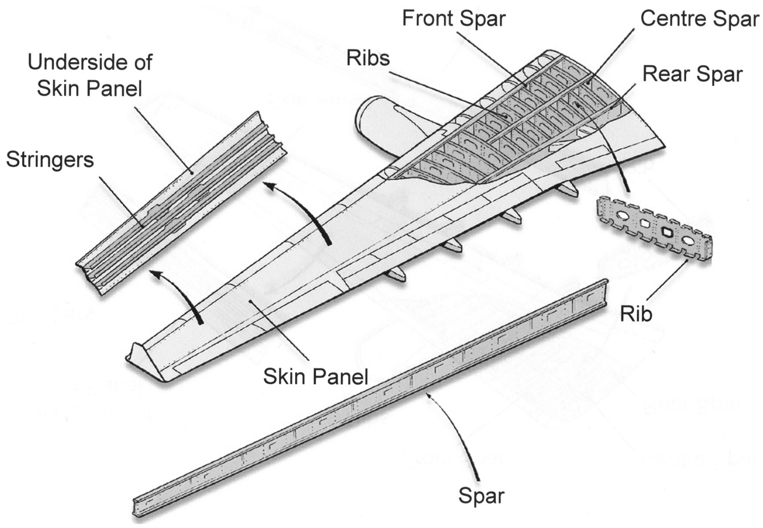

Figure 1: Wing-box structure.

Wings deform due to the aerodynamic forces which causes the change in the angle of attack and hence change the aerodynamic loading. Wing structure can be modeled using different fidelities. The highest fidelity models are generated using composite and brick elements. The next stage is using spars and ribs followed by modeling the entire wing as a single cantilever beam. In this research we chose the lowest fidelity model (cantilever beam) to represent the wing structure. Therefore, we can investigate different methods of reliability analysis instead of spending time on finite element simulation. The methods can be extended to a wing-box structure modeled with spars and ribs (ICW model). In this research, we will use *Nastran* finite element package to get the structural response. The structure is modeled using `CBAR` elements. It should me pointed out that the flow and structure are modeled in 2D domain.

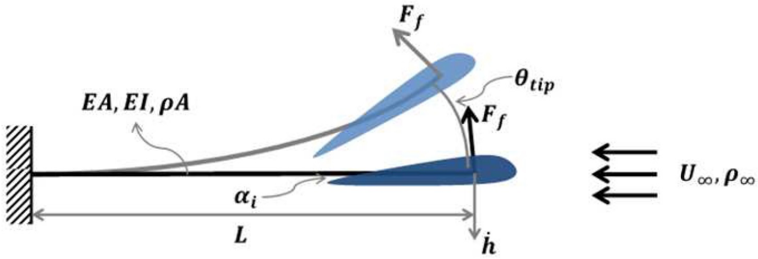

Figure 2: Simplified wing model.

The loading on the structure comes from the aerodynamic forces acting on the airfoil mounted at the end of the beam. The lift force can be calculated by solving the flow field over the airfoil and calculating the pressure profile. These will provide the force and moment applied on the wing structure. These forces can be used to calculate the stress in the wing which is chosen as our limit-state function for the reliability analysis as shown in Equation (1).

$$g\left(\bar{X}\right) = \sigma\left(E, I, L, \alpha_\infty, U_\infty, \bar{u}_\infty, \rho_\infty\right) - \sigma_{allowable} \tag{1}$$

As shown in Equation (1), the stress in wing depends on seven variables as specified below

1. E : Modulus of elasticity of wing structure
2. I : Area moment of inertia of wing structure
3. L : Length of wing
4. α_∞ : Angle of flow coming to the wing
5. U_∞ : Velocity of flow coming to the wing
6. $\bar{u}_\infty$: Sudden fluctuation in speed
7. ρ_∞ : density of fluid around the wing

There are different theories that can be used to calculate the aerodynamic response of the system. *Potential flow* describes the velocity field as the gradient of a scalar function: the velocity potential. As a result, a potential flow is characterized by an irrotational velocity field, which is a valid approximation for several applications such as aerofoils, water waves, electroosmotic flow, and groundwater flow. Nastran has the capability of using potential flow theory, *panel method*, to solve the current aeroelastic problem. Therefore, the β value can be calculated using reliability algorithms by calling Nastran to calculate the values for limit-state function and its gradients.

Another approach is to use *Euler equations* to solve the flow field. Euler equations are a set of equations governing inviscid flow. The equations represent conservation of mass (continuity), momentum, and energy, corresponding to the Navier-Stokes equations with zero viscosity and heat conduction terms.

$$\frac{\partial \rho}{\partial t} + \sum_{i=3}^{3} \frac{\partial \left(\rho u_i\right)}{\partial x_i} = 0 \tag{2a}$$

$$\frac{\partial \left(\rho u_j\right)}{\partial t} + \sum_{i=3}^{3} \frac{\partial \left(\rho u_i u_j\right)}{\partial x_i} + \frac{\partial p}{\partial x_j} = 0 \tag{2b}$$

Euler equations is a special/limiting case of the more general non-linear Navier-Stokes equation. Therefore, using it will give us more insight about the problem compared to potential flow. To solve the Euler equations around the airfoil a separate CFD solver is needed to calculate the pressure profile around the airfoil. In this research we are planing to connect an open source CFD solver (OpenFOAM) to solve the flow.