

# Concept Design of a Single-Aisle Aircraft Spoiler by Topology Optimization

P Purith<sup>1</sup>, C Kittikorn<sup>2</sup>, J Pakpong<sup>1</sup>, L Mezeix<sup>1</sup>

<sup>1</sup>Faculty of Engineering, Burapha University, 169 Long-Hard Bangsaen Road, Saensook, Muang, Chonburi 20131, Thailand

<sup>2</sup>Geo-Informatics and Space Technology Development Agency, Space Krenovation Park (SKP) 88 Moo 9, Thungsukhla, Sriracha, Chonburi 20230, Thailand

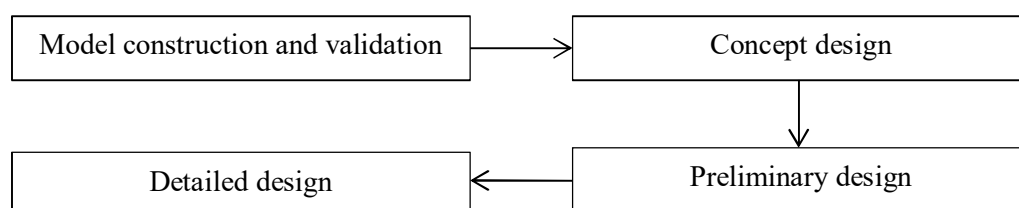
laurentm@eng.buu.ac.th

**Abstract.** Sandwich are widely used in the aerospace as flaps, spoiler, cabin floor or rotor where high flexural rigidity is required. Sandwich structures consist in a thick core material as honeycomb bonded with thin skins as carbon fibres. When the core material contains closed cells, water ingress must be considered. This water ingress may significantly increase the weight of the core and can induce damage. In this project, new spoiler design is proposed based on reinforced stiffeners in order to avoid the use of honeycomb. In a first step model is built and validated on current spoiler. Then, topology tool is used to locate the stiffeners. Finally, GFEM is performed to propose a first rough plies optimization under static analysis.

## 1. Introduction

Current aircraft spoiler is usually sandwich panels with Nomex honeycomb as core and carbon/epoxy as skins. Even if sandwich structures present a high ratio stiffness overweight, water ingress into honeycombs cells is one of the main concerns [1]. Redux film bonding between core and skins can be damaged and sandwich structure could completely fail. Therefore, water inside the honeycomb cells needs to be detected by Non-Destructive Tests (NDT). Different NDT methods are available for composited. For sandwich structures tap test is one of the most used even if it has several drawbacks [2]. Indeed, test quality depends of the inspector's skills and workplace noise can interfere it. In order to simplify the spoiler maintenance and control, honeycomb core could be avoided and replaced by composite stiffeners.

Design methodology can be summarized in 4 steps (figure 1). First step is the construction of the model and parameters validation, i.e. mesh, boundaries condition... Then, concept design can be proposed by using topology optimization [3].



**Figure 1.** Design process.



Content from this work may be used under the terms of the [Creative Commons Attribution 3.0 licence](https://creativecommons.org/licenses/by/3.0/). Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.

In order to reach the concept design, topology density-based approach is a powerful optimization tool [4]. General definition of the topology optimization is to find an optimal distribution of materials within a specify design domain. This approach is generally formulated and solved by considering the material distribution approach. In the design a density variable ( $\rho$ ) is assigned to all elements. Density variable is defined as a value that ranges from 0 to 1 during the computing. Most of the topology optimization problems based on maximum stiffness formulation or minimum global compliance. The topology optimization density-based approach optimization definitions can be written as:

$$\left\{ \begin{array}{l} \min_x \quad f(\rho) \\ \text{subject to} \quad \left\{ \begin{array}{l} 0 \leq \rho \leq 1 \\ \text{state function constraint} \\ \text{Manufacturing constraints} \end{array} \right. \end{array} \right.$$

**Figure 2.** Topology definition.

From the basic geometry given in the concept design step, preliminary design or Generalize Finite Element Method (GFEM) is performed. The aim is to obtain a first rough optimization by reducing the thickness of macro plies  $[0_n, 45_n, -45_n, 90_n]$ . Reaction Force (RF) between components of the structures can be also extracted. Finally, detailed design can be performed where ribs flanges, fillets... are designed. In this step ply by ply approach is used allowing lay-up optimization.

In the present paper, concept and preliminary design of spoiler will be proposed by replacing honeycomb core material by composite stiffeners. In a first step case study will be detailed. Then, model will be built and validated on current spoiler. Topology optimization will be used to determine the suitable stiffeners location and concept design will be proposed. Finally, GFEM will be performed in order to propose a first rough optimization.

## 2. Numerical simulation

### 2.1. Case study

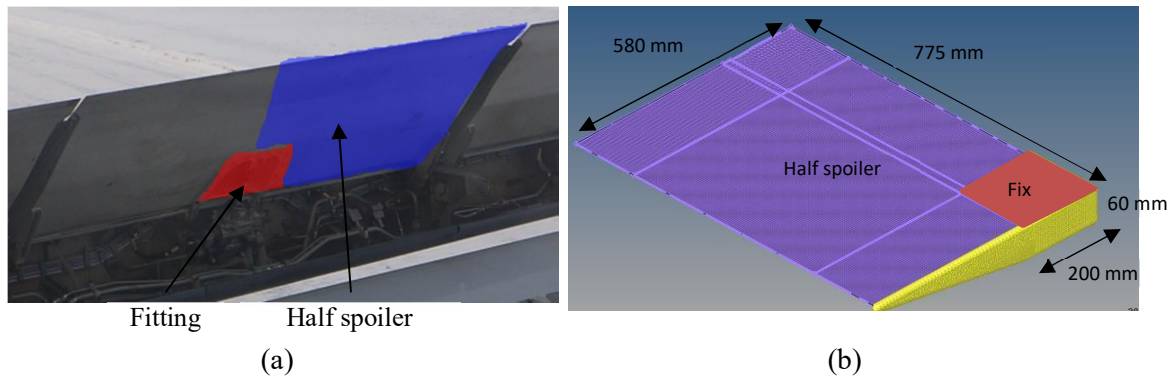
General dimension of studied spoiler is  $1.55 \times 0.6 \times 0.06 \text{ m}^3$ . Spoiler is mainly used during landing and therefore drag force is the main applied load. Design load,  $F$ , is applied perpendicularly to the upper surface of the spoiler and it is given by:

$$F = \frac{1}{2} \cdot S_f \cdot L_f (\rho \cdot v^2 \cdot A \cdot C_d) \quad (1)$$

Where  $C_d$  is Drag coefficient (1.3),  $A$  the spoiler area ( $1 \text{ m}^2$ ),  $v$  is the landing velocity of single aisle aircraft (86.64 m/s) and  $\rho$  density of air ( $1.225 \text{ kg/m}^3$ ). Load and safety factor are respectively 3 and 1.5 given finally a load,  $F$ , equal to 30,790 N

### 2.2. Model construction and validation

Numerical simulation has been firstly validated on current type of spoiler (figure 3a). Model consists in 3D elements for the Nomex honeycomb core ( $48 \text{ kg/m}^3$ ) and 2D shell for unidirectional carbon (T700)/epoxy upper and lower skins. Due to the symmetry only half of the spoiler has been simulated to reduce computing time. Metallic fitting allowing spoiler rotation has been considered as rigid and has been modelled by fix boundary condition (figure 3b). Finally, load was applied on upper surface of spoiler from equation (1).

**Figure 3.** Half spoiler with rotation features (3a)

Dimension of the spoiler (3b).

As this spoiler has been already designed and optimized to reduce the mass under aerospace design rules, the maximal compressive strain should be closed to the damage tolerance allowable, i.e.  $-4,000 \mu\text{def}$  in this case study. Numerical simulation results show that the maximal compressive strain is  $-3,700 \mu\text{def}$  that is closed to the allowable. Therefore, the numerical simulation model has been validated and can be used for topology optimization.

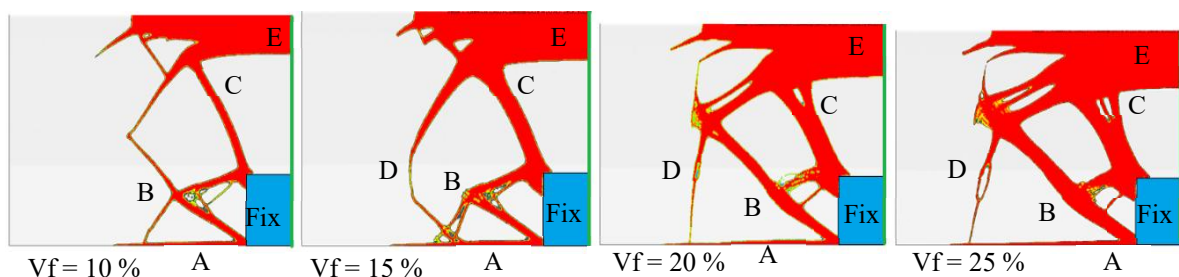
### 2.3. Concept design

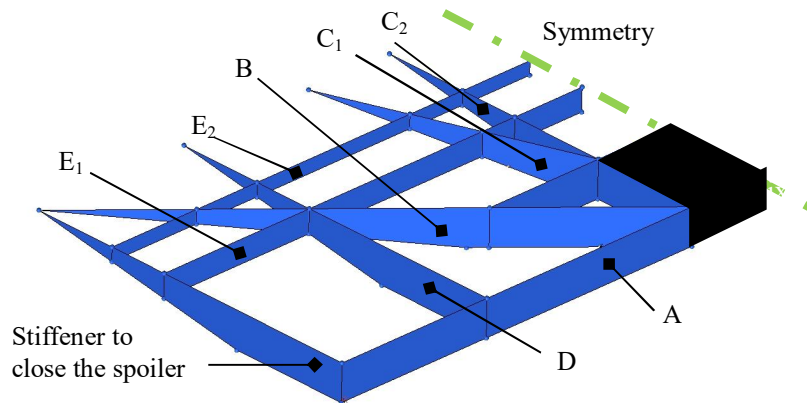
#### 2.3.1. Topology parameters

The objective is to replace the honeycomb core by composite stiffeners. To locate the suitable stiffeners position, topology tool was used. The current spoiler model has been used for the optimization. As upper and lower skins cannot be changed, it was considered as non-design area when the core volume have to be considered as design area. Minimizing the weight compliance has been inputted as topology objective with Volume Fraction,  $V_f$ , as constraint. Four  $V_f$  has been studied in order to identify the location of the stiffeners. Finally, extrusion function was used as manufacturing constrain to obtain only the stiffeners web.

#### 2.3.2. Topology results

The 4 different studied volume fractions are shown in the figure 4. By analysing the evolution of the material density in function of the volume fraction, stiffeners location can be obtained. 3 main stiffeners (A-C) starts from the fix boundary condition. Moreover, a secondary stiffener is necessary (D). Finally, reinforcement is required perpendicularly to the symmetry plane (E). From conceptual step, a design can be proposed by using the stiffeners location (A-E). Another stiffener has been used to close the spoiler (figure. 5).

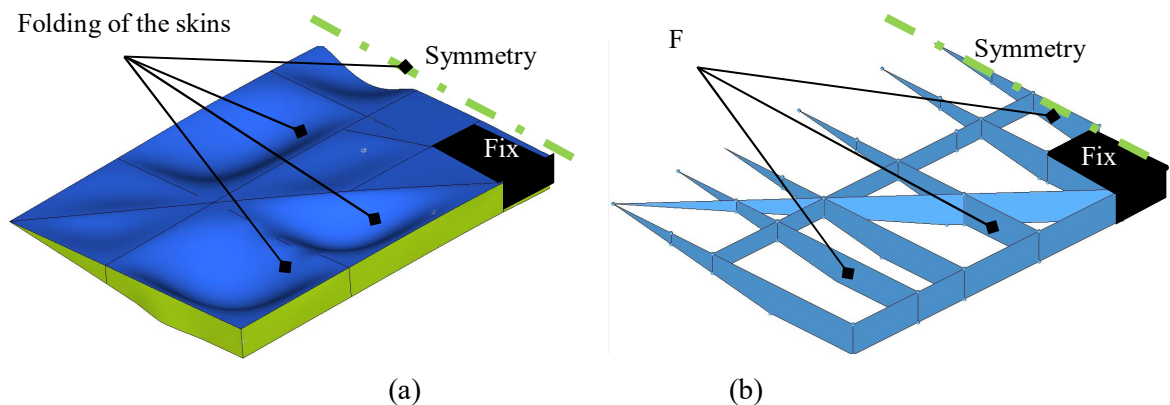
**Figure 4.** Topology optimization results in function of the volume fraction,  $V_f$  (Elements with density value one only).



**Figure 5.** Conceptual design of the stiffeners by using topology tools.

### 3. GFEM

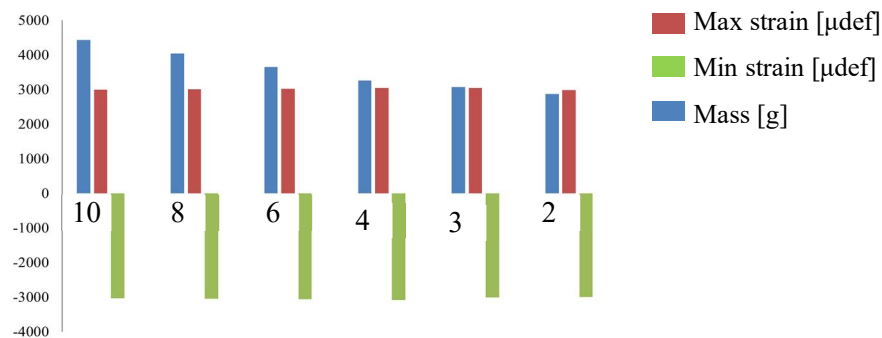
The aim of GFEM is to propose a preliminary design. As it is a new design, latest composite reference has been used. Unidirectional carbon/epoxy IMA/M21 was applied to the geometry obtained from the topology (figure 5). Same skin lay-up has been used as for the current spoiler. In a first step, optimization of macro plies  $[0_n, 45_n, -45_n, 90_n]$  has been performed in order to reduce the mass. First results show folding of the upper skin due to the load (figure 6a). Therefore, additional stiffeners (F) has been designed (figure 6b). Moreover, stiffeners  $C_1$  and  $E_2$  do not present any utility and has been therefore removed.



**Figure 6.** Folding of the skins (6a)

Additional stiffeners (6b)

GFEM from optimized geometry (figure 6b) has been simulated under static analysis. Skins lay-up are always the same as before. Different number of macro plies of stiffeners,  $n$ , has been investigated and strains have been reported (figure 7). As observed from  $n=10$  to 2, maximum and minimum strain do not present significant evolution and are lower than allowable. Critical strain value is located on the upper skin/ stiffener  $C_2$  edge due to the GFEM basic geometry (figure 5).



**Figure 7.** Summarize of the strain value each macro ply thickness.

#### 4. Conclusion

Single aisle spoiler was studied in this paper with objective to replace Nomex honeycomb by composite stiffeners. The conclusion can be summarized as follow:

- Numerical simulation has been successfully validated on current spoiler where the minimum compressive strain is closed to the allowable.
- Stiffeners location has been determined thanks to topology optimization applied to the core material and by investigated different volume fraction.
- From topology a first CAD has been proposed
- GFEM has been performed on the CAD to reduce plies number under static analysis.

Works are in progress to extract Reaction Force between stiffeners and skins in order to design the stiffeners flange. Detailed designed will be then performed.

#### Acknowledgement

The authors wish to thank the Geo-Informatics and Space Technology Development Agency (GISTDA) for the financial, operational and other related matters in ensuring the progress of research.

#### References

- [1] LaPlante G, Marble Andrew E, MacMillan B, Lee-Sullivan P, Colpitts Bruce G, Balcom Bruce J 2005 Detection of water ingress in composite sandwich structures: a magnetic resonance approach, *NDT & E Int.* **38**(6) 501-7.
- [2] Gary E G, Scott L, Jeff H 1996 Electronic tap hammer for composite damage assessment In Proceedings of SPIE, *Int. Soc. Optical Eng.* **2945**.
- [3] Grihon S, Krog L, Bassir D 2009 Numerical Optimization applied to structure sizing at AIRBUS: A multi-step process, *Int. J. Simulation Multidisciplinary Design Optimization*, **3** 432-42.
- [4] Verbart A, van Keulen F, Langelaar M 2015 *Topology Optimization with Stress Constraints* Ipskamp Drukkers TU Delft, 2-14.
- [5] Bouvet, C. and S. Rivallant 2016 Damage tolerance of composite structures under low-velocity impact. *Dynamic Deformation, Damage and Fracture in Composite Materials and Structures*: 7-33.