**QUEEN'S
UNIVERSITY
BELFAST**

Sequential finite element modelling of lightning arc plasma and composite specimen thermal-electric damage

Millen, S. L. J., Murphy, A., Abdelal, G., & Catalanotti, G. (2019). Sequential finite element modelling of lightning arc plasma and composite specimen thermal-electric damage. *Computers and Structures*, 222, 48-62.
<https://doi.org/10.1016/j.compstruc.2019.06.005>

Published in:
Computers and Structures

Document Version:
Peer reviewed version

Queen's University Belfast - Research Portal:
[Link to publication record in Queen's University Belfast Research Portal](#)

Publisher rights

© 2019 Elsevier Ltd.

This manuscript is distributed under a Creative Commons Attribution-NonCommercial-NoDerivs License

(<https://creativecommons.org/licenses/by-nc-nd/4.0/>), which permits distribution and reproduction for non-commercial purposes, provided the author and source are cited.

General rights

Copyright for the publications made accessible via the Queen's University Belfast Research Portal is retained by the author(s) and / or other copyright owners and it is a condition of accessing these publications that users recognise and abide by the legal requirements associated with these rights.

Take down policy

The Research Portal is Queen's institutional repository that provides access to Queen's research output. Every effort has been made to ensure that content in the Research Portal does not infringe any person's rights, or applicable UK laws. If you discover content in the Research Portal that you believe breaches copyright or violates any law, please contact openaccess@qub.ac.uk.

Sequential Finite Element modelling of lightning arc plasma and composite specimen thermal-electric damage

S.L.J. Millen^a, A. Murphy^{a+}, G. Abdelal^a, G. Catalanotti^a,

^a School of Mechanical and Aerospace Engineering, Queen's University Belfast, Ashby Building, Belfast. Northern Ireland, U.K. BT9 5AH

⁺ Corresponding author: Tel.: +44 28 9097 4095; E-mail: a.murphy@qub.ac.uk

Abstract

Highly complex phenomena such as lightning strikes require simulation methods capable of capturing many different physics. However, completing this in one simulation is not always desired or possible. In such instances there can be a need for a methodology to transfer loading boundary conditions from one simulation to the next while accounting for the characteristic form of the loading and the dissimilar domain and mesh geometries. Herein, the objective is to combine two models to enable the automatic sequential simulation of a lightning arc and a composite test specimen. The approach is developed using Finite Element models, with a Magnetohydrodynamics model representing the lightning plasma and a thermal-electric model representing the specimen. The specimen mesh and loading boundary conditions are automatically generated based on the predicted output of the preceding plasma model. The precision, run-time and flexibility of the proposed approach is demonstrated, with thermal damage predictions generated in approximately 33 hours. Resulting from the integrated modelling capability is the first time prediction of damage representing the test electric boundary conditions rather than assumed specimen boundary conditions (herein using test 'Waveform B').

Keywords

Finite element modelling; Simulation coupling; Lightning strike; Thermal-electric modelling; Mesh generation.

1.0 Introduction

Lightning strike attachment on aircraft may cause damage. The use of composites in wing and fuselage construction has increased progressively over the last three decades due to demands on weight reduction and fuel efficiency [1]. Composite materials present a challenge to lightning strike protection due to their comparatively poor electrical and thermal conductivity. The insulating nature of the polymer resin surrounding the fibres can lead to an extreme temperature around the local attachment point caused by Joule heating due to lightning current, direct heat impact from hot plasma channel, and thermal radiation from plasma [2]. Historically external surface metallic meshes have been used to protect composite laminates [3], [4].

A lightning strike is a highly complex phenomenon that involves many interactive physics such as electrical, magnetic, thermal and mechanical effects, all occurring within a timeframe in the region of micro to milliseconds [5]. Experimental studies regarding composite behaviour under lightning strike have been conducted within literature [6]–[9]. However, due to the highly specialised facilities and the cost and very high energy involved in a strike, these experimental campaigns need to be supplemented by computational modelling. The most recent experimental works have focussed on multiple or sequential lightning strikes [10] or the investigation of novel protection methods [11]. Computational modelling has dominated lightning strike research over recent years with authors typically modelling electrical, thermal and mechanical loads separately [2], [12]–[17]. However, progress has also been made in terms of modelling the thermal plasma associated with a simulated strike, taking into account combined multiphysics effects [5], [18]–[20]. Modelling of the plasma can encompass all the key physics mentioned previously, however requires a highly complex procedure within a multiphysics simulation and results in high computational expense with recent works suggesting upwards of seven days despite only being modelled in 2D axisymmetric form [5]. Additionally, to best knowledge, only one author in this field has directly considered the influence of mesh size on simulation results and presented a clear convergence study [2]. However, potentially relevant work has been done in other fields to produce meshes by using algorithms to control the mesh size and shape [21]–[23]. There is a clear gap in the literature with respect to coupling such

models (plasma-mechanical/thermal-electrical) for lightning test simulation and the prediction of composite material damage. Moreover, there is no demonstrated understanding on the influence and sensitivity of mesh selection and coupling methods on simulation predictions.

Therefore, the focus of this paper is to propose a robust approach to transfer the key load boundary conditions between dissimilar Finite Element (FE) models and understand simulation sensitivity with respect to coupling and meshing. This will be done considering a FE plasma model in COMSOL multiphysics and a thermal-electric FE model in ABAQUS [2], [5]. These models will be sequentially completed with the plasma model running first and providing outputs to the thermal-electric model. The paper is divided into clear sections firstly describing the background to the lightning arc plasma and specimen thermal-electric models used herein as well as other key models in the field. Different meshing strategies within literature will then be discussed before moving to the methodology. In the methodology the baseline simulations are considered before each proposed coupling method in this work is discussed. The methodology ends with a description of the solution procedure, the material models and the mesh convergence study employed in this work. Results for each of the test cases are then presented comparing their relative accuracy and damage predictions before conclusions are drawn.

2.0 Background

2.1 Lightning arc plasma modelling

Lightning strike simulations have been predominantly focussed on modelling the damage caused by a strike using FE software such as ABAQUS [24]. Recently, progress has been made to accurately model the thermal plasma associated with simulated lightning strike testing [5], [18]–[20]. Typically lightning strike plasma simulations are based on multiphysics Magnetohydrodynamic (MHD) representations and have developed from the research conducted on arc welding [25]–[29]. These

simulations consider the standardised lightning waveforms as inputs and can predict plasma properties and specimen surface loads. These waveforms have been specified in SAEARP5412B [4] and are described as follows. Waveform A is a 200kA, high intensity first return stroke. Waveform B is the intermediate current which follows a first return stroke and lasts for up to 5ms. Waveform C, continuing current, has the longest duration lasting up to 1s while D is a subsequent stroke of 100kA.

Multiphysics models allow for the coupling of many physics which is convenient for the modelling of lightning since thermal, mechanical and electromagnetic effects all occur together. These models typically incorporate Navier-Stokes equations for fluid motion, Maxwell equations of electromagnetism and thermal conduction equations for heat transfer.

Chemartin et al. [19] were the first authors to model the lightning plasma when considering arc attachment behaviour during a Waveform C swept stroke. Abdelal and Murphy [5] followed Chemartin et al. [19] presenting a solution for a Waveform B lightning strike test condition. More recently two works have proposed models for Waveforms A and C respectively [18], [20].

Chemartin et al. [19] used a 3D approach but only briefly described the solution procedure, physical properties represented and the mesh design. Both Chen et al. [18] and Wang et al. [20] used a 3D approach within CFD plasma models using ANSYS Fluent while Abdelal and Murphy [5] used a FE approach and a 2D-axisymmetric modelling strategy.

Chen et al. [18] only modelled a portion of the full waveform while both these authors and Wang et al. [20] included significant assumed initial boundary conditions in order to generate potentially representative behaviour. Abdelal and Murphy [5] improved on this approach by representing the initial plasma formation using cold-field electron emissions. This approach enabled the prediction of transient thermal, mechanical and electromagnetic behaviour. A preliminary 1D model was used to calculate initial electrical conductivity to remove the need for initial assumptions. This removed the need for initial assumptions and enabled simulations to start at room temperature conditions and capture initial arc attachment. This was not achieved by other works [18], [20]. This modelling approach, and Waveform B, will therefore be the focus herein as predictions will be

possible which capture the initial arc attachment behaviour. The main drawback of the simulation proposed by Abdelal and Murphy [5] was runtime. However, in order to reduce this, similitude theory was used to increase the utility of the model as a design tool.

Abdelal and Murphy [5] used a 12.5mm radius plasma simulation domain, Wang et al. used a 75mm radius domain, while Chen et al. used a 30 x 30 x 50 mm plasma region. Abdelal and Murphy were the only authors to discuss convergence however the meshes used by both Chen et al. and Wang et al. for the plasma domain were still much more structured. In the work of Chen et al., seven interpolation methods were tested to link the plasma simulation with a thermal-electric model. However, no convergence of the specimen mesh was discussed.

In all but one of the above works, damage behaviour has not been represented. In the work of Chen et al. where damage was presented, significant assumptions have been required in the plasma model. Abdelal and Murphy [5] were able to predict surface temperature of the specimen but not the internal resistive heating or the resulting specimen damage. Therefore, it would be beneficial to apply the predicted surface current density to determine the damage due to resistive heating. Note, it is not possible to apply a temperature boundary condition and also calculate the temperature at the specimen surface. The vast majority of the literature suggests that specimen temperature rise due to resistive heating will be much larger than heating from the plasma. Thus the current density is applied as the surface loading boundary condition and not the plasma surface temperature. Post simulation this idealisation will be reviewed through comparison of the plasma and the surface predicted temperatures.

2.2 Specimen thermal-electric modelling

Lightning strike damage simulations have developed incrementally since 2010 with different authors proposing different methods to apply the relevant loads and different meshes [2], [12], [13], [16], [30].

The majority of authors have used a model that matches the dimensions of the test specimens used in experiments. For example Ogasawara, Dong and Foster et al. [2], [13], [16] used a specimen which was 150 x 100 x 4.704 mm with a ply thickness of 0.147 mm, matching the specimen of Hirano et al. [6]. Abdelal and Murphy [12] used a quarter model composite laminate with symmetrical boundary conditions. This model was questioned by Foster et al. [2] due to the assumptions around symmetric results.

The meshes used have varied slightly between works but Ogasawara, Dong and Foster et al. [2], [13], [16] have all used structured meshes refined around the loading area. The mesh used by Abdelal and Murphy [12] is highly irregular and unstructured due to the partitioning method used.

Ogasawara et al. [16] applied the current profile to a single node at the specimen centre however several authors have commented on the potential inaccuracy of this method as it can lead to unrealistic temperature predictions. Abdelal and Murphy [12] improved upon this method by using an assumed arc radius of 5mm to apply the loading. Moreover other authors have made similar assumptions about the radius of loading, e.g. 5mm - Foster et al., 5 mm - Lee et al. [2], [30] , 2.5 mm - Dong et al. [13]. All works have used a zero potential boundary condition on the side and bottom surfaces of the specimen to replicate experimental conditions and observations [6].

Applying a circular load on a square or rectangular specimen has also the potential to cause a highly unstructured mesh. Foster et al. avoided this behaviour and suggested an analytical field loading approach to maintain a structured mesh but still assume and apply the loads in a circular pattern [2]. Foster's analytical field equation is shown in Equation (1), where r is the radius of the arc over which the load is applied and $field$ is the resulting area over which the load is applied.

$$Field = \left\lceil \left\lfloor \frac{r}{\sqrt{x^2 + y^2}} \right\rfloor / \left(\left\lfloor \frac{r}{\sqrt{x^2 + y^2}} \right\rfloor + 1 \right) \right\rceil \quad (1)$$

Foster et al. [2] also developed, for the first time, a method to characterise thermal damage to the specimen and used this during mesh convergence and the description of their results. So called moderate damage described a wide but shallow region of damage which encloses the severe damage area and includes physical behaviours such as resin degradation and fractured fibres. The severe

damage area is a narrow and potentially deeper area of damage which includes resin thermal decomposition, fractured fibres and fibre blow-out. Foster et al. [2] using the TGA results of Ogasawara et al. [16] determined that the 300°C contour could signify moderate damage while the 500°C contour boundary could indicate the severe damage zone.

While these authors have focussed on applying current loads, other authors have applied mechanical loads due to the formation of the lightning thermal plasma.

2.3 Specimen mechanical loading

Mechanical loading simulations have also been considered by a number of authors [15], [17], [31]. In each case the loads were derived from the fundamental pressure equations for lightning strikes, given in Equation (2), however the assumed equation denominator coefficient (specified as 8 in Equation (2)) varies between studies [7], [15], [17], [19], [32], [33].

$$\Delta p = \frac{\mu_o i^2}{8\pi^2 r^2} \quad (2)$$

where Δp is the pressure, μ_o is the magnetic permeability of air, i is the electric current and r is the arc radius.

Muñoz et al. [15] assumed the acoustic pressure and electromagnetic pressure were separate with the acoustic pressure having a constant value of 10 MPa over a radius of 12.5 mm throughout the analysis. The electromagnetic pressure acting on the specimen was assumed constant up to the arc radius. Both Foster et al. [17] and Millen et al. [31] used VDLOAD FORTRAN subroutines to apply different pressure loads to the test specimen surface. Foster et al. conducted three studies with a constant load over a 5 mm radius, varied loads over a 5 mm radius and a constant load over 5, 3 and 1 mm radii. Millen et al. used a custom-built VDLOAD subroutine which took output data from Abdelal and Murphy's [5] COMSOL simulation and interpolated this based on time and radial position to apply the pressure within a 12.5mm radius. However, this method was quite computationally expensive due to a larger loading area, high number of interpolations required and the amount of input data to be handled (17,000 unique values). Moreover, extending this approach to

also include current and heat flux loads would require upwards of 50,000 values to process and further add to the already significant computational cost. Consideration of all loads including pressure is important when attempting to link the plasma and composite specimen simulations since pressure along with current density and heat flux all require potentially simultaneous application and therefore need to be considered together.

2.4 Specimen and plasma meshing

A key challenge demonstrated by Millen et al. [31] is the significant difference in simulation mesh densities used for the plasma analysis and the specimen analysis. The anode in the MHD simulations and the composite specimen in the thermal damage simulations represent the interface between the plasma and the specimen and thus it is these meshes which must initially be compared. Typical mesh sizes and partitioning methods can be seen in Figure 1.

Figure 1 - Overview of modelling regions and meshes for plasma (top right) and thermal-electric specimen simulations (bottom centre).

Starting with the plasma simulation by Abdelal and Murphy [5] the 2D-axisymmetric model used a 0.05 mm mesh seed along its width. If this was converted to 3D and applied to a typical 150 x 100 mm specimen the number of elements would exceed 5×10^6 , assuming a single element size. This is totally unreasonable for any damage simulation to produce results in a time period which could be used for design iteration. Thermal damage simulations have used different methods to control the mesh size. Ogasawara et al. [16] and Dong et al. [13] used 600 elements per ply. Millen et al. [31] used 1200 elements per ply for their initial simulation linking study. Abdelal and Murphy [12] used a varying mesh in their quarter model with 2 elements through the thickness of the first ply and one for the remaining plies. 4074 elements were used in ply one while 441 were used in further plies resulting in the least structured mesh of all the thermal damage simulation meshes used.

Structured meshes have also been a feature of other damage modelling studies. Several authors have used structured, refined meshes for their low and high velocity impact simulations on rectangular specimens [34]–[37]. All these authors typically use a refined mesh around the area of impact and a courser mesh at specimen extremities. Other workers have used circular geometry to model impact with a structured square mesh at the centre of the specimen [38]–[40]. The transition of square to circular geometry has been handled with slight variation between works but generally the same rules are used with a swept mesh and hex elements [38]–[42]. Adaptive meshing techniques can also be used in this instance [43]–[45]. However, most of these works are focussed on mechanical loading and herein the focus is initially electric current loading. Foster et al. [2] were the only group of co-workers to conduct and present a comprehensive mesh convergence study leading to 4200 elements per ply with an in-plane mesh seed of 1.5 mm and 2 elements through the thickness of each of the top 8 plies.

2.5 Test arrangement modelling

Preceding research has assumed and applied the loading conditions on the test specimen surface. Few works have recognised or modelled the exact test loading conditions in which typically a conical tip discharge probe (the cathode) a distance above the specimen surface and a copper plate or ring (the anode) below the specimen are used to create the lightning arc which strikes the specimen. The prescribed test conditions, in the form of a desired current waveform [4] is thus not directly applied to the specimen. Thus to remove the need for many assumptions on the specimen surface loading conditions it is necessary to model the lightning arc between the discharge probe and the specimen surface. However, from the literature it can be clearly observed the disparate modelling fidelity that is currently achievable given the significant differences in computational burden of plasma and specimen modelling. The literature confirms the need for a robust method to transfer the loading boundary results between the already demonstrated plasma and specimen FE models, and the need to understand the impact of model mesh selection, development and convergence.

3.0 Methodology

Based on the preceding literature summarised in Section 2 the objective herein is to create a sequential coupling between the model representing the simulated lightning plasma and the model of the composite material test specimen. Based on the literature the approach must be computationally efficient and must account for the significant differences typically seen between the plasma and specimen meshes. Also based on the literature a structured mesh within the thermal damage model is desirable [2].

3.1 Baseline simulations

The method described in this paper relies on a two stage procedure. The first of these is the MHD lightning plasma model of Abdelal and Murphy [5] (in COMSOL multiphysics). This model represents initial arc attachment and predicts the necessary specimen surface loads for the second stage. The second stage predicts specimen thermal-electric behaviour and its results are used to infer specimen damage. The plasma model predicted behaviour is used to load the specimen model – herein the specimen current density is used to load the specimen thermal-electric model.

The plasma is represented within a 2D-axisymmetric model with a conical electrode, a 12.5 mm radius simulation domain and copper anode. Navier-Stokes equations, Maxwell equations and thermal conduction equations govern fluid motion, electromagnetism and heat transfer. The primary model input is the standard waveform equation (Equation 3):

$$I'(t) = \frac{11300}{d\beta} (e^{-700 \cdot t' \cdot \gamma} - e^{-2000 \cdot t' \cdot \gamma}) = 113(e^{-70 \cdot t'} - e^{-200 \cdot t'}) \quad (3)$$

where $\gamma=1e-4$, $\beta=1e8$ and $d=2$. The specimen material is copper with air used for the fluid domain. Ground boundary conditions are applied to the bottom and side surfaces of the copper specimen to replicate experimental conditions [6]. Further details of the model and the specific simulation parameters can be found in the relevant reference [5].

At the specimen surface the plasma model predicts current density, pressure and heat flux. These three time and radially varying outputs are shown in Figure 2 with the x-axis representing radius, the y-axis representing the relevant load magnitude, and each line representing a different time increment for the first 2 ms of the simulation.

The second stage is the thermal-electric FE damage model in ABAQUS proposed by Foster et al. [2]. This model predicts the thermal damage as a result of resistive heating due to current loading. The composite specimen modelled is 150×100 mm, containing 32 plies each with a thickness of 0.147 mm, with IM600/133 material and the ply layup of $[45/0/-45/90]_{48}$.

Both of these models have been selected having been benchmarked against experimental work within recent literature and offering approaches which attempt to minimise modelling assumptions (particularly with respect to plasma initial boundary conditions) [2], [5].

Figure 2 - Output plots from plasma model (x-axis represents radius and y-axis is load magnitude while each line represents a different time increment, 0 to 2 ms)

3.2 Proposed coupling methods

Figure 1 shows a comparison between the baseline plasma simulation domain and mesh used by Abdelal and Murphy [5] and the baseline specimen simulation domain and mesh used by Foster et al. [2]. In order to accurately transfer the loads from one simulation to the next they must be resolved from 2D to 3D to account for the orthotropic material properties in the specimen thermal-electric simulation. The current density predicted at the anode surface in the plasma model is to be applied at the top surface of the specimen model. Other constraints on the approach are that pressure may only be applied to surfaces, an advantage as nodal loading was demonstrated to be inaccurate for thermal-electric simulations [2], [12], and no subroutine script is available in ABAQUS to apply current

density unlike the DLOAD and DFLUX routines used for continuous variation of pressure and heat flux in other works [46], [47]. Initially three coupling cases are considered:

Case 1

This is the baseline case and in this method the same model, mesh and loading approach developed by Foster et al [2] is used. Analytical fields are used to apply the radial expanding current loads to the top surface of the specimen model. In order to apply loads, surface rings are first defined via analytical fields. The equation used by Foster et al. is adjusted to capture the radial variance of the loads rather than applying them to one uniform circle and is given in Equation 4, where r_2 is the outer radius and r_1 is the inner radius and x and y are points within the radii. The radial progression of the current is represented through temporal variation of the loads applied to each surface ring using time versus load amplitude tables as shown in Figure 3. In this way the combined amplitude functions and surface loads allow the axisymmetric representation of the current density to be applied as the loading boundary conditions on the specimen surface. Exemplar load boundary conditions are shown in Figure 4 for two load instances, where the value of ‘Load existence’ equal to one represents loading and a value of zero represents no loading.

$$Field = \left[\left| \frac{r_2}{\sqrt{x^2 + y^2}} \right| / \left(\left| \frac{r_2}{\sqrt{x^2 + y^2}} \right| + 1 \right) \right] - \left[\left| \frac{r_1}{\sqrt{x^2 + y^2}} \right| / \left(\left| \frac{r_1}{\sqrt{x^2 + y^2}} \right| + 1 \right) \right] \quad (4)$$

Figure 3 – Illustration of a single surface, load and amplitude arrangement used to apply the plasma current density load on the test specimen surface.

Figure 4 – 3D illustration of loading method using modified analytical field (Equation 3).

Case 2

In Case 2 the mesh used by Foster et al [2] and recreated in Case 1 was refined to increase the number of partitions and the representation of the radial variance of the load. The main drawback of the analytical field method is that the minimum mesh size controls the minimum dimension of the loading rings. Thus in Case 1 the minimum partition was limited to 1.5 mm. Seed bias was therefore used to reduce the mesh size at the centre of the loading area and hence the number of loading rings. The resulting mesh had a minimum element size of 0.05 mm rising to 3 mm at the edge of the plasma model boundary (compared with the constant 1.5 mm size in the equivalent region in Case 1). The same loading method was applied to the refined mesh as described in Case 1.

Case 3

Visual inspection of the loading properties (pressure, current density, surface heat flux) shown in Figure 2 illustrates the same general trend for the majority of the data. Therefore, a turning point or point of inflection analysis may be used to subdivide the data into radial zones. Given the axisymmetric nature of the plasma simulation load discretisation is simplified to be circular and thus the points of inflection may be used to guide the specimen surface discretisation for the application of the load.

The graph of pressure was used as a test case. Figure 5 shows that for the first 0.3 mm the fluctuation and variation in pressure is very high but follows a general trend for >90% of the data. Between 0.3 mm and 2 mm features two linearly varying zones, between 2 and 3 mm represents a levelling of the data before the most complex region between 3 and 6 mm. The final zone (6 to 12.5 mm) features exponential and linear decay relationships.

Four criteria were used to find key points in the data:

- Point A < Point B > Point C – representing a peak
- Point A > Point B < Point C – representing a trough

- $(\text{Point C} - \text{Point B})/(\text{Point B} - \text{Point A})$ and $\text{Point C} > \text{Point B}$ – representing a doubling of gradient
- $(\text{Point C} - \text{Point B})/(\text{Point B} - \text{Point A})$ and $\text{Point C} < \text{Point B}$ – representing a halving of gradient

A test was then applied to check the criteria; the script was applied to the trigonometric function $\tan(\text{radius})$ from 0.0 to 12.5 mm as this function represents many of the characteristics of the plots seen in Figure 2. The script without user intervention successfully identified the transitions between zones with distinct characteristics, based on a post-test visual inspection.

Figure 5 - Zoomed graphs of pressure plots.

Figure 6 - Graph of pressure loads with partition points at $t=1.5\text{ms}$ and current loads/partitions at $t=0.05\text{ms}$.

The finalised script was then applied to all plots and the resulting points were plotted over the original data as shown in Figure 6 for both pressure and current at different time points. Each marker represented the potential boundary of a partition on the top face of the specimen. A small rounding script was added to the procedure which only added points if they occurred more than twice over all the plots and rounded those closer than 0.05 mm to avoid extremely small divisions resulting in an extremely high mesh density such as that in Case 2. The final Python script is shown in Algorithm 1.

The same partitioning procedure was then applied to the graphs of current density and heat flux and the divisions were compared. Similar divisions were consolidated while any missing divisions were added to the list. The full list of divisions was then used to generate the profile on the surface of the specimen as shown in Figure 7. This algorithm produced 24 surfaces each with their

own load and amplitude taken from the input data. As with Case 1 and 2 the partitioned surfaces captured the radial variance in the loads while the load versus amplitude data captures the time variance.

Figure 7 – Graphical representation of surface partitions produced by point of inflection analysis.

```

input : Data for pressure, flux and current density are imported
Number of rows and columns are counted for each input data set
for  $i$  in  $\text{range}(1, \text{numcols}_p)$  do
    for  $N$  in  $\text{range}(1, \text{numrows}_p - 1)$  do
        if  $\text{Current}[N, j] < \text{Current}[N-1, j]$  and  $\text{Current}[N+1, j] > \text{Current}[N, j]$ :
            then
                 $\text{turns.append}(\text{Current}[N, 0])$  /* min turning points */
        else if  $\text{Current}[N, j] > \text{Current}[N-1, j]$  and  $\text{Current}[N+1, j] < \text{Current}[N, j]$ :
            then
                 $\text{turns.append}(\text{Current}[N, 0])$  /* max turning points */
        if  $((\text{Current}[N+1, j] - \text{Current}[N, j]) / (\text{Current}[N, j] - \text{Current}[N-1, j])) > 2$ 
            and  $\text{Current}[N+1, j] > \text{Current}[N, j]$ : then
                 $\text{turns.append}(\text{Current}[N, 0])$  /* +2x change in gradient */
        if  $((\text{Current}[N+1, j] - \text{Current}[N, j]) / (\text{Current}[N, j] - \text{Current}[N-1, j])) > 2$ 
            and  $\text{Current}[N+1, j] < \text{Current}[N, j]$ : then
                 $\text{turns.append}(\text{Current}[N, 0])$  /* -2x change in gradient */
     $\text{turns} = \text{np.asarray}(\text{sorted}(\text{turns}))$  /* create a sorted array of points */
    for  $i$  in  $\text{range}(1, \text{len}(\text{turns}))$ : do
        /* round to nearest 0.5 if >1 */
        if  $\text{turns}[i] \geq 1.0$ : then
             $\text{rounded.append}(\text{round}(\text{turns}[i] * 2.0) / 2.0)$ 
        else
             $\text{rounded.append}(\text{round}(\text{turns}[i], 2))$  /* otherwise round to 2 d.p. */
     $\text{rounded} = \text{np.asarray}(\text{sorted}(\text{rounded}))$ 
     $\text{mesh} = \text{sorted}([z \text{ for } z, v \text{ in } \text{Counter}(\text{rounded}).\text{iteritems}() \text{ if } v \geq 2])$  /* keep
        only those points with an x axis value occuring more than once */

```

Algorithm 1 - Python script to find key points of inflection in input data.

Case 4

A further study is required for specimens with copper lightning strike protection layers. Due to the heterogeneous structure of this layer, creating a suitable FE mesh is challenging, as demonstrated by Abdelal and Murphy [12], and partitioning such a structure with circular patterns produces a highly complex mesh which ABAQUS struggles to generate. For this reason a face selection algorithm was developed to select the faces within the limits from the point of inflection analysis and apply the relevant amplitude profiles. The code used for this procedure can be seen in Algorithm 2. This code finds the faces between upper and lower bounds from the turning point analysis. The centroid of each

face is then found and collated into an array of faces over which the loads are applied. Figure 8 shows the resulting loading surfaces generated using this method. Again, as with the preceding cases, each array of surfaces captured the radial variance in the loads while the load versus amplitude data captures the time variance.

```

input : ABAQUS model of copper mesh
output: Surfaces and loads for copper mesh
z=getByBoundingCylinder(2)-getByBoundingCylinder(1)/* finds faces
    within surface bounds */
for i in z: do
    y=faces.getCentroid()/* finds centroid of each face within surface
        bounds */
    q=y[0]
    m.p.Surface(name='Surf-n', side1Faces=m.p.faces.findAt(((q),)))

```

Algorithm 2 - Algorithm to find relevant surfaces for copper mesh loading.

Figure 8 - Loading surfaces generated on a standard copper mesh topology.

A summary of the key differences between the four models described herein are given in Table 1.

Table 1 - Summary of four proposed coupling methods

Case	Mesh Used	Total Element Count	Loading Method
1	Foster et al. [2]	33,670	Analytical Fields
2	Refined Foster et al. [2]	57,414	Refined Analytical Fields
3	COMSOL Output Dependent	26,950	Python script and tabular amplitudes
4	COMSOL Output Dependent	29,835	Python face selection and tabular amplitudes

3.3 Specimen model solution procedure and material modelling

In each analysis a fully coupled thermal-electric step is used. A transient solution procedure is used with DC3D8E elements. The same temperature dependent material properties as Foster et al. [2] are used in the analysis shown in Table 2. The energy released during resin decomposition is assumed to be 4.8×10^6 J, between 500°C and 800°C and the energy released during fibre ablation was assumed to be 43×10^6 J, between 3316°C and 3334°C. A zero electrical potential boundary condition is applied to the side and bottom surface of the specimen to replicate experimental conditions [6]. The material properties used for the copper LSP are shown in Table 3. These properties are consistent with those used in the plasma simulation and previous publications [5], [12], [30]. The results of these four thermal-electric simulations, three unprotected and one protected case, are presented and discussed in the next section.

Table 2 - Temperature dependent CFRP material properties.

Temperature dependent material properties				
		Thermal Conductivity		
Temperature (°C)	Specific Heat (J/kg°C)	Fibre (W/mm.K)	Transverse (W/mm.K)	Through- Thickness (W/mm.K)
25	1065	0.008	0.00067	0.00067
500	2100	0.004390	0.000342	0.000342
800	2100	0.002608	0.00018	0.00018
1000	2171	0.001736	0.0001	0.0001
3316	2500	0.001736	0.0001	0.0001
3334*	5875	0.001736	0.0001	0.0001
3335*	5875	0.0005	0.0005	0.0005
7000*	5875	0.001015	0.001015	0.001015
Temperature dependent material properties				
		Electrical Conductivity		
Temperature (°C)	Density (kg/mm ³)	Fibre (1/ Ω.mm)	Transverse (1/ Ω.mm)	Through- Thickness (1/ Ω.mm)
25	1.52x10 ⁻⁶	35.97	0.001145	1.79x10 ⁻⁶
500	1.52x10 ⁻⁶	35.97	0.001145	1.79x10 ⁻⁶
800	1.10x10 ⁻⁶	35.97	0.001145	1.79x10 ⁻⁶
3316	1.10x10 ⁻⁶	35.97	0.001145	1.79x10 ⁻⁶
3334*	1.11x10 ⁻⁹	35.97	2	1x10 ⁶
3335*	1.11x10 ⁻⁹	0.2	0.2	1x10 ⁶
7000*	1.11x10 ⁻⁹	1.5	1.5	1x10 ⁶
* - Gas				

Table 3 - Temperature dependent copper material properties.

Temperature (°C)	Density (kg/mm ³)	Specific Heat (J/kg°C)	Thermal Conductivity (W/mm.K)	Electrical Conductivity (1/ Ω.mm)
25	8.95x10 ⁻⁶	385	0.401	58140
500	8.95x10 ⁻⁶	431	0.37	20120
800	8.95x10 ⁻⁶	491	0.34	20120
1000	8.95x10 ⁻⁶	491	0.15	4651
2600	8.95x10 ⁻⁶	491	0.18	2404
7000	8.95x10 ⁻⁶	491	0.18	1500
8000	8.00x10 ⁻⁶	550	0.18	1400

3.4 Mesh convergence study

For Case 3 a mesh convergence was completed within a thermal-electric simulation. Convergence was assessed considering the size of the 300°C and 500°C contours on the top surface of the specimen (as these temperatures represent the moderate and severe damage areas and the same method undertaken by Foster et al. [2] when converging baseline case mesh i.e. Case 1). The mesh size was governed by three main criteria as shown in Figure 9, where ‘global size’ represents the size of elements at the specimen extremities (mm), ‘number of sectors’ represents the angular divisions at the specimen centre (within a 1/8th segment), and ‘number of partitions’ represents the radial division at the specimen centre (again within a 1/8th segment). Examining Table 4, the mesh convergence study indicates that varying the radial elements has limited effect on the predicted 300°C contour, only smoothing the edges, but had a noteworthy impact on the 500°C contour. Moreover, such mesh variation resulted in limited change to the simulation runtime. Varying the angular divisions of elements resulted in the greatest impact on element count and simulation runtime. For example, changing from 3 to 4 elements per 1/8th segment (simulation B to C) resulted in limited change to the monitored contours but a significant jump in simulation duration. Further increase in the number of

elements in the radial direction (simulations C to D) again results in limited change to the monitored contours but a significant jump in simulation runtime. To maintain a similar simulation duration reducing element size in the angular direction may be offset by increasing element size in the radial direction (D to E) but in this case the monitored contours were again not significantly impacted.

Figure 9 - Controls on mesh size

Table 4 - Comparison of mesh convergence and run-time for Case 3 thermal-electric simulation.

Sim. No.	No. of Sectors	No. of Partitions	Global Size (mm)	No. of Elements per ply	300/500°C contour area (mm ²)		Variance from previous (%)		Run Time (hrs)
A	3	40	5	3024	2648	735	-	-	24
B	3	46	5	3360	2735	918	3.3	24.9	33
C	4	44	5	4144	2635	930	-3.7	1.3	55
D	4	50	5	4592	2590	931	-1.7	0.1	65
E	5	37	4	4250	2681	921	3.5	-1.1	67

These results, Table 4, also demonstrate the variance of the simulations compared with the baseline mesh (Case 1, 4200 elements per ply). Figure 10 shows the convergence plots for the five mesh variants in the convergence study. The maximum variance between the predicted 300°C contour areas is 5% for the five meshes tested. The variance between the predicted 500°C contour areas are less than 1.5% for simulations B, C, D and E. Therefore, the mesh can be considered converged.

Figure 10 – Thermal-electric mesh convergence plots

Thus the mesh of simulation B was chosen for the study due to its favourable run time and its comparable element count and prediction performance to Foster et al. [2]. Indeed this simulation was able to complete a Waveform B simulation with the same runtime as the Foster et al. simulations which represented a much shorter but higher magnitude test waveform ('Waveform A'). For ease of comparison of results this mesh was also used for the underlying plies in Case 4. Figure 11 shows the final mesh and partitioning method used in Cases 3 and 4. A combination of o-grid and mid-point subdivision methods [40], [48]–[50] were used to generate a mesh suitable for this radial problem. This meshing procedure, to the author's best knowledge, is novel within this field and for this application and presents an alternative to the traditionally user-crafted meshes with square areas of refinement near the centre of the specimen.

Figure 11 – Meshing partitions and final mesh using o-grid and mid-point subdivision methods (right-hand figure shows detail at the centre of the specimen).

4.0 Results

For each of the four test cases outlined in Section 3 full thermal-electric simulations were tested. This involved using the same initial current density data from the plasma model prediction and processing this for application to each thermal-electric model. Results show how well each case represents the radial and temporal evolution of the loads for a single time point by comparing the input loads with the raw data profiles. Again surface temperature contours (300 and 500°C) are compared. The influence of each loading method and its required mesh on element count and run times are also discussed. Finally, the copper protected case is discussed separately as it has a significantly different design and behaviour.

4.1 Case 1, 2 & 3

To check the ability of each case to capture the radial and time variance of the input data, graphs of plasma model output data were compared with the applied load within each thermal-electric simulation. The comparison between the original and partitioned data is shown in Figure 12 with instantaneous integrals at $t = 0.05$ ms of 9096, 7352 and 7176 A^2mm for Cases 1, 2 and 3 respectively. These plots are compared at $t = 0.05$ ms where the greatest differences are seen with respect to the original data output from COMSOL. At this time point the original COMSOL output has an

instantaneous integral of $7047 \text{ A}^2\text{mm}$ resulting in errors of 29.1%, 4.3% and 1.8% respectively for Case 1, 2, and 3. Case 1 overestimates the current density to be applied in several sections of the graph. The comparison is improved for Case 2 and Case 3 with both having almost identical predictions except for between 3.5 to 7.5 mm where Case 2 overestimates the load. In order to achieve the improvement in accuracy between Case 1 and 2 the run-time and element count is increased significantly, Table 4. The Case 3 loading is the most acceptable given the average error between plasma model output and the applied load is less than 1% and the low model element count and run-time.

Figure 12 - Comparison of plasma model output with specimen loading applied in Case 1,2 & 3 ($t = 0.05\text{ms}$).

Table 5 – Mesh densities, run times and damage areas for the three unprotected test cases.

Test Case	Total Element Count	Run Time (hrs)	300/500°C contour area (mm ²)	Damage depth (number of plies from impacted surface)
1	33,670	62	2976/1267	4
2	57,414	220	3272/1327	2
3	30,150	33	2735/918	6

Cases 1 and 3 have broadly similar predictions for 300°C contours with the difference being 8%, Table 5. However, due to the overestimate of the load applied, as noted earlier, the 500°C contour is 30% larger, Figure 13 and Figure 15. This correlates well with the 30% over representation in the applied current load between 1 to 2.25 mm, as shown in Figure 12, due to the constraints on mesh and load as discussed in Section 3.2.

Figure 13 - Temperature contours for Case 1 thermal-electric simulation.

Case 2, despite a very dense mesh, and theoretically matching the raw data (Figure 12) struggles to capture the behaviour at the centre of the arc as shown in Figure 14. This leads to separate temperature contours at the centre of the specimen, behaviour not observed in the other simulations. The difficulty here is likely due to forcing extremely narrow circular loading patterns onto a square/rectangular mesh, hence the benefit of the structured circular mesh used in Case 3. Through-thickness contours are also highly irregular for Cases 1 and 2 compared with other works in the field

with damage being four plies deep for Case 1 and two plies deep for Case 2. Case 2 and 3 despite having very similar incident loads and the same through-thickness element count have different through-thickness damage profiles showing the effect of the mesh in all directions even if the input load is captured accurately. Figure 15 shows the full depth of thermal damage for Case 3 using the python method. Thermal damage extends to six plies deep due to the long time period of Waveform B and opportunity for thermal and electrical conduction between plies.

Figure 14 - Temperature contours for Case 2 thermal-electric simulation.

Figure 15 - Temperature contours for Case 3 thermal-electric simulation.

Comparing Case 3 with other published works it can be observed that Waveform B creates a different damage contour shape compared with Waveform A predictions, e.g. Foster et al. [2]. The Waveform B contours have a more uniform and symmetrical shape whereas the Waveform A contours have a large circular area of damage which extends to the extremities of the specimen with a uniform width. The present results are also more comparable with the diamond shaped damage profiles found in experimental works e.g. [6]. Damage depth predictions are also deeper for Waveform B than A, 6 plies versus 5. At this stage it is worth noting that in Abdelal and Murphy's [5] plasma model the anode is assumed to be copper and thus further work is required to represent the anode more accurately with composite properties. In addition the benefits of modelling resistive heating in the thermal-electric simulation have been proven given the different temperatures between the simulations. The peak plasma temperature is 33,708°C and the peak specimen surface temperature is 234°C in the plasma simulation. However, the peak specimen temperature due to resistive heating in ABAQUS is approximately 2,500°C, an order of magnitude larger, proving that this behaviour is not adequately captured with this plasma model alone. Given the short duration of the event (5ms) and the noted surface temperature, under Waveform B conditions resistive heating is likely to result in much greater specimen heating compared against heating from the arc plasma. However, due to fundamental modelling constraints the developed thermal-electric model will under estimate the specimen temperature as it does not capture the additional plasma heating effect.

4.2 Case 4

Figure 16 illustrates the thermal profile generated for the protected laminate. Thermal damage only extends one ply deep as observed by other authors using copper mesh layers [30]. The temperature rise in the specimen is negligible and the reasons for this result are the high electrical conductivity of copper and the relatively low incident current density predicted by the plasma model. Once again this will be checked in future work when modifications are made to the anode properties but this initial result would suggest that for test Waveform B a copper protection system is more than capable of dispersing the current effectively to minimise damage. This is in contrast to the studies for test

Waveform A and D where Abdelal and Murphy [12] showed damage extended three plies deep for Waveform D, and Lee et al. [30] predicted damage one ply deep for Waveform A but with a temperature rise in the order of hundreds of degrees. Nevertheless, the same general trend applies that the presence of a protection system reduces the amount of thermal damage on the specimen when comparing Cases 3 and 4.

Figure 16 - Temperature contours for Case 4 thermal-electric simulation (legend values shown in °C).

5.0 Conclusions

An automated approach to enable sequential Finite Element modelling of lightning arc plasma and composite specimen thermal-electric damage has been presented. The key objectives of this work have been met through the automatic linking of the two Finite Element models. The models represent the initial arc attachment process (via Magnetohydrodynamics) and the behaviour of the composite test specimen under thermal-electric loading. The predicted current loading, from the plasma on to the test specimen, is used to load the thermal-electric model but critically also used to define an appropriate mesh for the simulation. The method has been shown to be capable of overcoming the issues posed by different mesh densities between models while being able to generate a structured specimen mesh, suitable for modelling with and without copper mesh layers. Mesh density and design has been shown to be very important in producing damage predictions, even if the input loading is accurate. A mesh convergence study was used to eliminate the variability of results with mesh design. The new loading strategy has produced a result in approximately 33 hours and since this result needs to be passed to a further simulation, to apply pressure loads and consider thermal expansion, this is an advantage to a full plasma/damage simulation encompassing three stages. These loading techniques allowed for an initial quantification of the effect of plasma loads on specimen damage looking specifically at current density and comparing loading methods. Of particular significance is the fact that the presented results are the first predictions of damage for a test 'Waveform B' arrangement where the test electric input conditions are directly represented rather than assumed specimen surface loading conditions. These initial simulations use a static arc, expanding radially from a fixed point. Future work should consider arc movement [2], [19] in both the plasma and damage models and the adaptation of the automatic meshing approach to effectively enable this. In order to fully capture damage predictions, further simulations will be required which modify the anode material within the plasma simulation and represent the lightning arc pressure and temperature loading.

Acknowledgements

The authors would like to thank EPSRC for funding this work as part of the PhD research of S. Millen.

ACCEPTED MANUSCRIPT

References

- [1] M. Bossak and J. Kaczowski, "Global / local analysis of composite light aircraft crash landing," *Comput. Struct.*, vol. 81, pp. 503–514, 2003.
- [2] P. Foster, G. Abdelal, and A. Murphy, "Understanding how arc attachment behaviour influences the prediction of composite specimen thermal loading during an artificial lightning strike test," *Compos. Struct.*, vol. 192, pp. 671–683, 2018.
- [3] M. Gagné and D. Therriault, "Lightning strike protection of composites," *Prog. Aerosp. Sci.*, vol. 64, pp. 1–16, 2014.
- [4] SAE Aerospace, "Aerospace Recommended Practice ARP5412B," 1999.
- [5] G. F. Abdelal and A. Murphy, "A multiphysics simulation approach for efficient modeling of lightning strike tests on aircraft structures," *IEEE Trans. Plasma Sci.*, vol. 45, no. 4, pp. 725–735, 2017.
- [6] Y. Hirano, S. Katsumata, Y. Iwahori, and A. Todoroki, "Artificial lightning testing on graphite/epoxy composite laminate," *Compos. Part A Appl. Sci. Manuf.*, vol. 41, no. 10, pp. 1461–1470, 2010.
- [7] H. Kawakami, "Lightning Strike Induced Damage Mechanisms of Carbon Fiber Composites," Washington, 2011.
- [8] N. Hosokawa, T. Ooto, S. Kubo, M. Anzai, A. Yoshiya, and A. Nakagoshi, "Lightning Strike Protection for Composite Laminates by Pitch-Based Carbon Fibre Skin," in *International Conference on Composite Materials*, 2013, pp. 1–10.
- [9] P. Feraboli and M. Miller, "Damage resistance and tolerance of carbon/epoxy composite coupons subjected to simulated lightning strike," *Compos. Part A Appl. Sci. Manuf.*, vol. 40, no. 6–7, pp. 954–967, 2009.
- [10] J. Sun, X. Yao, X. Tian, J. Chen, and Y. Wu, "Damage Characteristics of CFRP Laminates

- Subjected to Multiple Lightning Current Strike,” *Appl. Compos. Mater.*, 2018.
- [11] B. Wang, Y. Duan, Z. Xin, X. Yao, D. Abliz, and G. Ziegmann, “Fabrication of an enriched graphene surface protection of carbon fiber/epoxy composites for lightning strike via a percolating-assisted resin film infusion method,” *Compos. Sci. Technol.*, vol. 158, pp. 51–60, 2018.
- [12] G. F. Abdelal and A. Murphy, “Nonlinear numerical modelling of lightning strike effect on composite panels with temperature dependent material properties,” *Compos. Struct.*, vol. 109, no. 1, pp. 268–278, 2014.
- [13] Q. Dong, Y. Guo, X. Sun, and Y. Jia, “Coupled electrical-thermal-pyrolytic analysis of carbon fiber/epoxy composites subjected to lightning strike,” *Polym. (United Kingdom)*, vol. 56, pp. 385–394, 2015.
- [14] Z. Q. Liu, Z. F. Yue, F. S. Wang, and Y. Y. Ji, “Combining Analysis of Coupled Electrical-Thermal and BLOW-OFF Impulse Effects on Composite Laminate Induced by Lightning Strike,” *Appl. Compos. Mater.*, vol. 22, no. 2, pp. 189–207, 2015.
- [15] R. Muñoz, S. Delgado, C. González, B. López-Romano, D. Y. Wang, and J. Llorca, “Modeling lightning impact thermo-mechanical damage on composite materials,” *Appl. Compos. Mater.*, vol. 21, no. 1, pp. 149–164, 2014.
- [16] T. Ogasawara, Y. Hirano, and A. Yoshimura, “Coupled thermal-electrical analysis for carbon fiber/epoxy composites exposed to simulated lightning current,” *Compos. Part A Appl. Sci. Manuf.*, vol. 41, no. 8, pp. 973–981, 2010.
- [17] P. Foster, G. Abdelal, and A. Murphy, “Quantifying the Influence of Lightning Strike Pressure Loading on Composite Specimen Damage,” *Appl. Compos. Mater.*, vol. 26, no. 1, pp. 115–137, 2018.
- [18] H. Chen, F. S. Wang, X. T. Ma, and Z. F. Yue, “The coupling mechanism and damage prediction of carbon fiber/epoxy composites exposed to lightning current,” *Compos. Struct.*,

vol. 203, pp. 436–445, 2018.

- [19] L. Chemartin *et al.*, “Direct Effects of Lightning on Aircraft Structure : Analysis of the Thermal , Electrical and Mechanical Constraints,” *J. Aerosp. Lab*, no. 5, pp. 1–15, 2012.
- [20] F. Wang, X. Ma, H. Chen, and Y. Zhang, “Evolution simulation of lightning discharge based on a magnetohydrodynamics method,” *Plasma Sci. Technol.*, vol. 20, no. 7, 2018.
- [21] G. Subramanian, A. Prasanth, and V. V. S. Raveendra, “An algorithm for two-and three-dimensional automatic structured mesh generation,” *Comput. Struct.*, vol. 61, no. 3, pp. 471–477, 1996.
- [22] M. V. Pathan, S. Patsias, and V. L. Tagarielli, “A real-coded genetic algorithm for optimizing the damping response of composite laminates,” *Comput. Struct.*, vol. 198, pp. 51–60, 2018.
- [23] G. H. Kwon, S. W. Chae, and K. J. Lee, “Automatic generation of tetrahedral meshes from medical images,” *Comput. Struct.*, vol. 81, no. 8–11, pp. 765–775, 2003.
- [24] ABAQUS 2016 Documentation, *Abaqus User Subroutines Reference Guide*. 2017.
- [25] H. G. Fan and Y. W. Shi, “Numerical simulation of the arc pressure in gas tungsten arc welding,” *J. Mater. Process. Technol.*, vol. 61, pp. 302–308, 1996.
- [26] J. J. Lowke, R. Morrow, and J. Haidar, “A simplified unified theory of arcs and their electrodes,” *J. Phys. D. Appl. Phys.*, vol. 30, no. 14, pp. 2033–2042, 1997.
- [27] C. S. Wu and J. Q. Gao, “Analysis of the heat flux distribution at the anode of a TIG welding arc,” *Comput. Mater. Sci.*, vol. 24, pp. 323–327, 2002.
- [28] F. Lago, J. J. Gonzalez, P. Freton, F. Uhlig, N. Lucius, and G. P. Piau, “A numerical modelling of an electric arc and its interaction with the anode: part III. Application to the interaction of a lightning strike and an aircraft in flight,” *J. Phys. D. Appl. Phys.*, vol. 39, no. 10, pp. 2294–2310, 2006.
- [29] K. K. Kim and I. M. Karpova, “A magnetogasdynamic model of a welding arc,” *Russ. Electr.*

Eng., vol. 87, no. 1, pp. 46–50, 2016.

- [30] J. Lee, T. E. Lacy, C. U. Pittman, and M. S. Mazzola, “Thermal Response of Carbon Fiber Epoxy Laminates with Metallic and Nonmetallic Protection Layers to Simulated Lightning Currents,” *Polym. Compos.*, 2017.
- [31] S. L. J. Millen, A. Murphy, G. F. Abdelal, and G. Catalanotti, “Quantifying the Influence of Plasma - Solid Mechanics Coupling on the Prediction of Lightning Strike Composite Specimen Damage,” in *1st International Conference on Advances in Aerospace Structures, Systems & Technology (AASST)*, 2018, pp. 103–112.
- [32] P.-N. Gineste, R. Clerc, C. Castanié, H. Andreu, and E. Buzaud, “Assessment of Lightning Direct Effects Damages By Modelling Techniques,” in *Int. Aerospace and Ground Conf. on Lightning and Static Electricity*, 2009, pp. 1–11.
- [33] S. J. Haigh, “Impulse Effects during Simulated Lightning Attachments to Lightweight Composite Panels,” in *Int. Aerospace and Ground Conf. on Lightning and Static Electricity*, 2007, pp. 1–21.
- [34] S. Metoui, E. Pruliere, A. Ammar, and F. Dau, “A reduced model to simulate the damage in composite laminates under low velocity impact,” *Comput. Struct.*, vol. 199, pp. 34–45, 2018.
- [35] W. Tan, B. G. Falzon, L. N. S. Chiu, and M. Price, “Predicting low velocity impact damage and Compression-After-Impact (CAI) behaviour of composite laminates,” *Compos. Part A Appl. Sci. Manuf.*, vol. 71, pp. 212–226, 2015.
- [36] A. Yoshimura, T. Okabe, M. Yamada, T. Ogasawara, Y. Tanabe, and A. C. Group, “Damage Simulation of CFRP Laminates Under High Velocity Projectile Impact,” in *18th International Conference on Composites Materials (ICCM)*, 2011, pp. 1–6.
- [37] E. Panettieri, D. Fanteria, and F. Danzi, “A sensitivity study on cohesive elements parameters: Towards their effective use to predict delaminations in low-velocity impacts on composites,” *Compos. Struct.*, vol. 137, pp. 130–139, 2016.

- [38] S. Xu and P. H. Chen, "Prediction of low velocity impact damage in carbon/epoxy laminates," *Procedia Eng.*, vol. 67, pp. 489–496, 2013.
- [39] Y. Shi, T. Swait, and C. Soutis, "Modelling damage evolution in composite laminates subjected to low velocity impact," *Compos. Struct.*, vol. 94, no. 9, pp. 2902–2913, 2012.
- [40] M. F. S. F. de Moura and J. P. M. Gonçalves, "Modelling the interaction between matrix cracking and delamination in carbon-epoxy laminates under low velocity impact," *Compos. Sci. Technol.*, vol. 64, no. 7–8, pp. 1021–1027, 2004.
- [41] G. Catalanotti, P. P. Camanho, P. Ghys, and A. T. Marques, "Experimental and numerical study of fastener pull-through failure in GFRP laminates," *Compos. Struct.*, vol. 94, no. 1, pp. 239–245, 2011.
- [42] D. H. Li and F. Zhang, "Full extended layerwise method for the simulation of laminated composite plates and shells," *Comput. Struct.*, vol. 187, pp. 101–113, 2017.
- [43] B. H. V. Topping and B. Cheng, "Parallel and distributed adaptive quadrilateral mesh generation," *Comput. Struct.*, vol. 73, no. 1–5, pp. 519–536, 1999.
- [44] P. M. Mohite and C. S. Upadhyay, "Adaptive finite element based shape optimization in laminated composite plates," *Comput. Struct.*, vol. 153, pp. 19–35, 2015.
- [45] J. K. Wilson and B. H. V. Topping, "Parallel adaptive tetrahedral mesh generation by the advancing front technique," *Comput. Struct.*, vol. 68, no. 1–3, pp. 57–78, 1998.
- [46] Y. Wang and O. I. Zhupanska, "Lightning strike thermal damage model for glass fiber reinforced polymer matrix composites and its application to wind turbine blades," *Compos. Struct.*, vol. 132, pp. 1182–1191, 2015.
- [47] Y. Wang and O. I. Zhupanska, "Thermal Ablation in Fiber-Reinforced Composite Laminates Subjected to Continuing Lightning Current," *57th AIAA/ASCE/AHS/ASC Struct. Struct. Dyn. Mater. Conf.*, no. January, pp. 1–17, 2016.

- [48] C. G. Armstrong, H. J. Fogg, C. M. Tierney, and T. T. Robinson, "Common themes in multi-block structured quad/hex mesh generation," *Procedia Eng.*, vol. 124, pp. 70–82, 2015.
- [49] T. S. Li, R. M. McKeag, and C. G. Armstrong, "Hexahedral meshing using midpoint subdivision and integer programming," *Comput. Methods Appl. Mech. Eng.*, vol. 124, no. 1–2, pp. 171–193, 1995.
- [50] N. Weatherill and C. Forsey, "Grid generation and flow calculations for aircraft geometries," *J. Aircr.*, vol. 22, no. 10, p. 1985, 1985.