Elastic Waves Simulation on Aircraft Subcomponent for test correlation using piezoelectric sensors
Fernando SÁNCHEZ IGLESIAS¹, Ruben TEJERINA², Jaime GARCÍA ALONSO³
Pablo CAFFYN⁴, Manuel IGLESIAS VALLEJO⁵
¹, ², ³, ⁴, ⁵ Airbus Defence and Space, Paseo John Lennon, s/n, 28906 Getafe (Madrid), SPAIN
¹ fernando.i.sanchez@airbus.com, ² ruben.tejerina@airbus.com, ³ jaime.garcia@airbus.com,
⁴ pablo.caffyn@airbus.com, ⁵ manuel.iglesias@airbus.com

Abstract
The interest in Structural Health Monitoring (SHM) Systems that are able to detect and characterize events such as impacts for in-service aircraft is increasing. One of the main challenges for these systems is the detection of low energy accidental Barely Visible Impact Damage (BVID) on thin-walled structures such as fuselages or wing’s skin. These structures may be metallic or composite but the emphasis of this research has been placed on composite structures. The Airbus DS Stress Methods Team has designed a set of Numerical Simulations that has been validated with physical tests, to increase the knowledge of the structural response in terms of elastic waves to support the development of SHM Systems using these phenomena. The target of this study is to develop a simulation methodology that enables research on structural trend behavior in support of the development of a reliable SHM System for real aircraft structures while only carrying out a limited number of physical tests.

Keywords: Aerospace, Structures, Modelling, Verification, Validation, Guided Waves.

1 INTRODUCTION
In this article a set of simulations reproducing the characteristics of some impacts produced over an aircraft composite subcomponent are presented; the subcomponent impacted is a reinforced composite skin panel representative of a characteristic architecture (stiffeners & frames) of a section of composite fuselage. Configuration has been manufactured flat to help tuning and read-across between test results and simulation. And the impacts are produced at low energy using a gravity impactor.

The objective of the set of impacts tested is to capture the response of the structure. For this purpose, a network of piezoelectric sensors (in this case PZT discs) attached to the surface of the structure [1], and a numerical simulation approach are studied on this article.

The impact response data is acquired using a standard National Instruments® acquisition card. This card runs in-house developed acquisition software based on LabVIEW® environment. Then the data is stored in a computer and processed using in-house developed Matlab® software.
In order to perform the numerical simulation of the events studied, the authors have selected the Finite Element Method due to its capabilities to represent arbitrary geometries with a high level of complexity. Abaqus/Explicit® has been considered to currently be the best suited tool for this job.

Finally the time series measured using simulated PZT discs is correlated with the experimental acquisitions produced by the real PZT sensors in order to evaluate and increase the confidence on this kind of simulations and validate the methodology.

2 EXPERIMENTAL SET UP

The experimental set up consists of a composite stiffened flat panel with longitudinal omega stringers and transversal metallic frames. This panel is installed in a framework. It can be checked in Figure 1 (left).

![Figure 1 – Panel general view with PZT sensors installed, stiffeners side view (left). Panel with gravity impactor, flat side view (right).](image)

The panel has been sensorized with a network of piezoelectric disc (PZT) by the stiffeners side. The impacts are produced from the flat side using a gravity impactor as indicated in Figure 1 (right).
The sensor network is connected to data acquisition equipment (and it to the acquisition computer) placed directly by the panel following the schema indicated in Figure 2.

![Data acquisition schema for PZT sensors.](image)

The acquisition of data is launched using a manual triggering synchronized with the impact. All data is acquired using National Instruments LabVIEW based software and later processed using Matlab.

The acquisition configuration is as follows:
- 8 input channels acquired at a sampling rate of 250 ksps per channel.
- Input range +/- 10 volts.
- Manual acquisition triggering.

The impactor characteristics are the following:
- Mass 3.670 kg.
- Spherical tip with 15.8 mm of diameter.

Several impacts are produced in order to correlate them with numerical simulations. Two of the impacts positions and sensors used for acquisitions are shown in Figure 3 and 0.
Several impacts are produced in I1 and I2 locations and we select the impacts at 5 Jules energy level to correlate the with Abaqus simulations

3 NUMERICAL SIMULATION

Newton’s Second Law can be expressed in matrix format for finite element purposes as:

\[ M\ddot{u} + C\dot{u} + Ku = F_\text{ex} \]

(1)

Where \( M \) is the mass matrix, \( C \) is the damping matrix, \( K \) is the stiffness matrix, \( F_\text{ex} \) is the external loads vector and \( u \), \( \dot{u} \) and \( \ddot{u} \) is the displacement vector and its time derivatives. The equation can be rearranged to the form:

\[ M\ddot{u} = F_\text{ex} - C\dot{u} - Ku = F_\text{ex} - F_\text{in} \]

(2)

Having a diagonal (lumped) mass matrix the equation becomes trivial to solve, and thus, by using an explicit time domain integration scheme the response of the structure can be simulated.
Given that the space needs to be discretized in such a way that the shortest wavelength of interest is captured [2]; a large number of increments might be required to solve the model, however, since the Stiffness and mass matrix do not need to be formed nor inverted, each time increment becomes computationally inexpensive. This makes the explicit scheme particularly attractive for wave propagation problems.

All the simulations present on this article are performed with Abaqus/Explicit® v6.14-1.

Figure 4 shows a global view of the finite element model used. Mesh is not shown on this picture due to the high element density. Since the amplitude of the wave is sufficiently small, linear elasticity is assumed on the simulation [4] [5].

3.1 Mesh definition

Composite parts are represented with continuum shell elements (SC8R), as it increases computational efficiency and accuracy. Multiple elements are placed through the thickness to properly represent the transmission of the wave [3].

Due to its low thickness compared to the plate and to mitigate undesirable hourglassing modes, piezoelectric sensors are modelled as reduced integration shell elements (S4R), sharing nodes with the last layer of the plate elements (Figure 5).

Frames and test tooling are modelled with shell elements and with a far more coarse mesh, as the influence they have over the area of interest is limited.

Figure 4 – Global view of the model.
The number of elements per wavelength in the model is a critical parameter due to the aliasing introduced by the discretization. According to [6], the number of elements $M$ which must be used to cover a length $L$ with $P$ elements per wavelength is given by:

$$M = \frac{PL}{\lambda} = \frac{PLf}{c}$$  \hspace{1cm} (3)

Where, $c$ is the wave group velocity, $\lambda = c/f$ is the wavelength and $f$ is the frequency.

Due to the nature of the problem, the shortest wavelength of interest is the wavelength of the compressional waves. According to [6], 10 elements per wave length can give sufficient accuracy for the problem, so by applying the previous equation a maximum element length of 1 mm is guaranteed to be sufficient to satisfy this criterion.

### 3.2 Materials and properties

Metallic parts are modelled as an isotropic material. Design values are obtained from MMPDS-05 [8].

The composite parts are modelled as layers of orthotropic material. Composite material elastic properties are obtained from the Airbus Group design value material database; in the case of the embedded elastomeric material layer (for acoustic or other specific functional improvements), the properties are taken from its commercial brochure. Laminate properties are assigned on the element section.

PZT material is represented as an orthotropic material. Elastic material properties are obtained from literature [7] [9].
3.3 Contour conditions and constraints

Aluminum frames, test rig and omega stiffeners are bonded to the skin with tie constraints. The model is simply supported on the four corners of the test rig frame.

3.4 Impact definition

The impactor is modeled as an analytical rigid sphere with a radius of 7.9mm and a concentrated mass in its center of 3.67Kg. The initial velocity is adjusted to give the required energy for each impact.

General contact with no friction is established between the impactor surface and the panel.

3.5 Simulation run

Simulation is run with the double precision solver in the Airbus Defence and Space cluster. The approximate wall-clock time per increment is of 0.732s, which gives run times of 4h to 13h depending on the time period simulated. (On 64 cores Intel Xeon E5-2667 v3 – 3.2GHz, 128 GB Ram, 200 GB SSD)

3.6 Output

For verification purposes, displacement output is requested at all nodes of the model, while stress and strain output is limited to the elements representing the piezoelectric sensors.

The analysis simulates a total time of 0.003s with an output requested every 500 equidistant intervals of time. The following figures show the displacement plot for impact I1 at two instants of time.

![Displacement plot](image-url)
The strain recorded on the elements belonging to each of the piezoelectric sensors is compared with the data recorded on the test. The results are presented on the following section.

4 CORRELATION BETWEEN SIMULATION AND TEST

The target of the correlation is the matching of the simulated waveform and the acquired waveform using PZT sensors.

To perform this step the signals must be converted into common units, and, in this case, also the simulation output stresses at PZT sensor from MPa to volts using a simplified formula [9].

\[ V(t) = \frac{d_{13} \cdot A_{pzt}}{C_p} \cdot \sigma(t) \]  

(4)

Where, \( d_{13} \) is the piezoelectric constant for the PZT sensor, \( A_{pzt} \) is the PZT sensor area and \( C_p \) is the PZT sensor capacitance. The results are shown in Figure 8.
It can be observed on the previous image that the model is able to capture the tendency of most of the signals and predict the behavior of most of the sensors with an acceptable accuracy. The worst correlation appears on the PZT sensors located closer to the impact and better in the furthermost sensors, but the overall behavior and adjustment ranges are good.
5 CONCLUSIONS

A set of simulations have been run in order to correlate the experimental results in terms of the simulated wave form arriving to the sensor in the first milliseconds. A good correlation has been achieved for most of the sensors; nevertheless adjustments should be made in the structure model and in the PZT modeling in order to improve the results.

The final target should be to gain knowledge in this kind of simulations in order to allow the possibility of virtual testing, reducing the number of physical tests to the minimum to validate the functionality.

As shown in this article, the Explicit Finite Element Model has proven to be a valid tool to simulate the structural behavior of complex composite structures under low energy impacts and to capture the response at PZT sensors of a SHM system, although some improvements could still be made to enhance the correlation.

6 REFERENCES


