

## CFRP Panel FE Model subjected to Compression and Fire

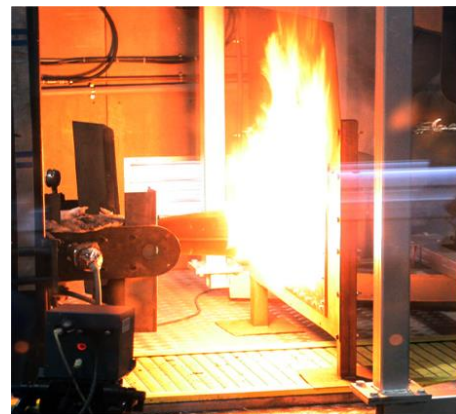
### INTRODUCTION

ICEMM has developed a material model to simulate CFRP panel degradation and loss of properties due to elevated temperatures when is subjected to fire on one face. This material model has been implemented in FEM code ABAQUS, and a coupled temperature-displacement analysis has been performed.

### COMPLETED PROJECT

Client: R&D ICEMM  
Date: 2013

- Development of an analytical material model that takes into account degradation due to elevated temperatures: thermal properties and mechanical properties variation with temperature. Implementation of the material model in FEM code ABAQUS and co-simulation of temperature and displacements.



Source: Infrared Lab Uc3m

Figure 1. CFRP Panel Fire test

### COMPLETED ACTIVITIES

- Analytical formulation to take into account material degradation when subjected to high temperatures: thermal and mechanical properties variation with temperature
- Implementation of material model in ABAQUS FEM code and coupled mechanical-thermal simulation.
- Comparison between FEM results and experimental tests
- Documentation.

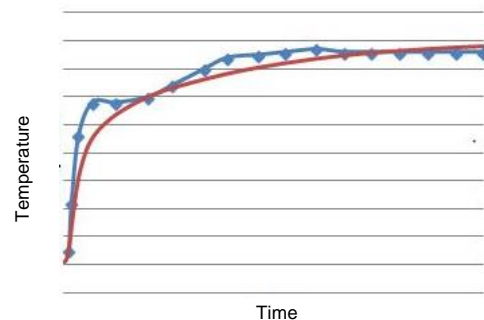


Figure 2. Temperature distribution with time: experimental results (blue) vs simulation results (red)

### TECHNOLOGY

All the project has been analyzed with the Finite Element Solver tool ABAQUS version 6.10.2 Windows Platform and the pre/post processing has been carried out with ABAQUS CAE.

- Thermal transient analysis: convection and conduction
- Coupled thermal-displacement analysis
- Linear Buckling analysis