Nonlinear Finite Element Analysis of Airport Approach Lighting Structures under Impact Loading

M. Nejad Ensan[*1], D.G. Zimcik[1], S.T. Jenq[2] and F.B. Hsiao[2]

[1]*Institute for Aerospace Research, National Research Council Canada*
[2]*Institute of Aeronautics & Astronautics, National Cheng Kung University, Tainan, Taiwan*
[*] *Corresponding author. Tel: 1-613-998-0005; Fax: 1-613-990-3617; E-mail: Manouchehr.Nejad@nrc-cnrc.gc.ca*

Abstract

This paper describes computer simulation of the impact of airport approach lighting structures using the LS-DYNA nonlinear finite element analysis (FEA) software. Two tower designs were investigated in this simulation. First, a finite element model (FEM) was developed to simulate the impact of a representative tower typical of those used at Canadian airports. This was 6.6 m tall tower with a triangular cross section made of aluminum. The analysis simulated an aircraft wing striking the tower triangular cross section 1 m from the top in two different orientations of the tower, at the apex and on the side. Top mass of 2.72 kg or 5.44 kg, representative of lights and/or light fixtures was included in the model which was impacted at three different impact velocities: 50, 80 and 140 km/h.
Further simulation was carried out on a second model of an approach light masts with different geometry and made of a different material typical of those used at some European airports. These were 6 m tall masts, and contained a dummy top mass of 15 kg, simulating the effect of a cross-bar with three approach lights. The masts were configured as a composite lattice structure with a square cross section made of glass/epoxy. The impact of the mast at a height of 4.1 m above the ground due to an impact of the moving aircraft wing at velocity of 140 km/h was simulated.
The duration of the simulation in both cases was 100 milliseconds to captured the phase in which most of the damage to the tower took place. Simulation results were used to predict the deformation mode and magnitude, location and timing of failure, impact force and energy absorption curves as a function of time during the impact. These results were compared to experimental data from full-scale tests to validate the accuracy of the models. These data were necessary for the development of simplified requirements and test methods for the design of frangible structures that minimize the impact hazard to aircraft.

Keywords: Airport approach lighting structures, Frangibility, Nonlinear finite element analysis, Full-scale impact tests.

Introduction

Approach lighting towers are used at airports to give approaching aircraft correct information of direction and distance to the runway. The close proximity of these towers to the runways requires that the towers be designed such that a collision between an aircraft and the tower will not compromise flight capability. In order to minimize aircraft damage, approach lighting towers must be made “frangible”. *Frangibility* was defined as the property which allows an object to break, distort or yield at a certain impact load while absorbing minimal energy, so as to present the minimum hazard to aircraft.

The Frangible Aids Study Group (FASG) of the International Civil Aviation Organization (ICAO) is presently in the process to develop design specifications for a frangible approach lighting structure [1]. In the proposed frangibility specification for airport approach lighting towers, the FASG recommended that the structure should break, distort or yield when subjected to the sudden collision force of a 3000 kg aircraft, airborne and traveling at a velocity of 140 km/h or taxiing on the ground at 50 km/h.

A number of ICAO member countries have conducted impact testing on airport approach lighting towers [2-5]. In Canada, Transport Canada initiated a full-scale, test program in conjunction with the National Research Council, using truck mounted wing-shaped impactors that were impacted against the tower. A total of forty-five tests were completed [2,3], using a rigid impactor as well as two “soft’ impactors designed to replicate wing sections of typical aircraft. However, since the results of the tests indicated that the rigid impactor yielded conservative estimates of the maximum force and equivalent values of energy over the contact period, only those tests using the rigid impactor are used to compare to the simulation results. Also, impact tests on Exel airport approach light masts have been performed in Finland [4]. A wing section attached to a high speed truck was used as an impactor.

The goal of this work was to develop and demonstrate the capability to accurately model a typical airport structure by comparing predicted results to full-scale impact tests data. Once verified, these models can then be used to investigate other configurations and parameters of impact to assess the performance of the structure. The computer simulation of two tower designs was investigated using the LS-DYNA [6] nonlinear finite element analysis (FEA) software. First, a finite element model was developed to simulate the impact of a representative tower typical of those used at Canadian airports. Further simulation was carried out on a second model, with different geometry and made of a different material, typical of those used at some European airports. These results were compared to experimental results from full-scale tests to validate the accuracy of the models. In addition, the tower dynamic behavior was compared to the high-speed video imagery to confirm failure mode predictions.

Case Study 1:

1.1 Full-scale Testing Conducted in Canada

The general configuration of an impact test is shown in Fig. 1. The tower was mounted in a pit on the side of a test track and the tower base-plate fixture was bolted to a concrete foundation. The impactor which was mounted on a truck such that the impact point was one meter down from the top of the tower, struck the tower at high, medium and low impact speeds (140, 80 and 50 km/h). Impacts on both the apex and on the side of the triangular cross section were conducted.

The tower consisted of two 3.3 m (10 ft) sections as shown in Fig. 1. The top section, referred to as the 7T10 section, was attached through a triangular aluminum plate to the bottom section referred to as the 9T10 section. Both sections had an equilateral triangular cross-section with the 7T10 section having a side length of 17.8 cm (7 in) and the 9T10 section having a side length of 22.9 cm (9 in). The vertical rods were 1.43 cm in diameter and were made of Aluminum-Alloy 6061-T6. The cross members were also made from 0.79 cm diameter rods of the same alloy. The tower supported a dummy top mass of 2.72 kg or 5.44 kg, representative of light fixtures and lights. These top masses were made from aluminum in the shape of cylindrical disks which were bolted to the top of the tower.
A rigid impactor was constructed from a semi-cylindrical steel tube 79 cm long, 30.5 cm in diameter, and 2.2 cm thick. The rigid impactor, also shown in Fig. 1, was mounted to the support structure and attached to the test vehicle. Impact speed, time history of impact force, period of contact between the tower and the impactor, time at which the different failures occurred, and impact force transferred to the tower over time were recorded.

1.2 Finite Element Model Development

The three-dimensional model shown in Fig. 2, was developed to simulate the impact of the tower. The tower was modeled using beam elements. The top mass and the plate connecting the top and the bottom sections of the tower were modeled using the Belytschko-Tsay shell elements. The impactor model also included Belytschko-Tsay shell elements with a uniform thickness.
The detailed model formulation for the LS-DYNA explicit analysis code included non-linear elastic-plastic material behavior. The material used in the tower was defined using the MAT_PLASTIC_KINEMATIC keyword to model isotropic and kinematic hardening plasticity. The effective plastic strain at failure was set as 15% (SF=0.15) in the material input data. The material properties used for the tower were those for Aluminum-Alloy 6061-T6.
The impactor was modeled using a rigid material model MAT_RIGID. The material properties used for the rigid impactor were those for Steel.

Fig. 1: General impact test configuration.

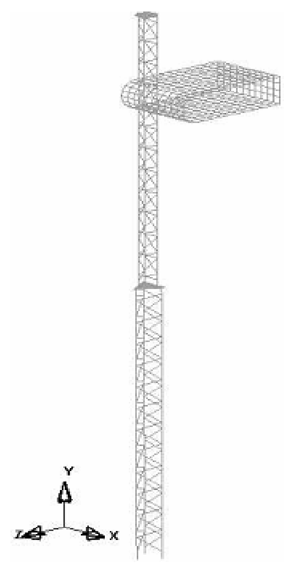


Fig. 2: Finite element model of the tower impact.

A CONTACT_AUTOMATIC_SINGLE_SURFACE definition was used to ensure contact between the various components.
A CONTACT_FORCE_TRANSDUCER_PENALTY was defined to monitor the contact forces at the interface. This transducer element allowed the total contact forces applied by the impactor to the tower to be registered but did not apply any force to the model itself.

1.3 Numerical Simulation

Using this FEA model, numerical simulation was performed assuming that the impactor struck the tower which support a top mass of 2.72 kg or 5.44 kg, representative of light fixtures and lights, at an initial given velocity. The impacts were simulated in both the apex and side directions of the tower at three different impact velocities: 50, 80 and 140 km/h along the z-direction of the tower which simulated the impact conditions in the full-scale test program.
The impact events were analyzed for a time period of 0.1 s, which was chosen to be sufficiently long to simulate the initial events of the impact to compare with full-scale test results. Dynamic results were recorded at one hundred equal time steps (every 0.001 s).

Deformation Mode: The sequential views of the deformation mode obtained from the full-scale test and FEA simulation for impact at 140 km/h in the side direction with a 2.72 kg top mass present are shown in Fig. 3. On contact, the impactor flattened the 7T10 section (see section 1.1) of the tower at the point of contact and the top of the section began to bend around the impactor. The apex leg buckled at the first bay of the 7T10 section above the mid-plate at about 8 ms after impact. The buckling of the apex leg of the 9T10 section at the bottom of the tower occurred at about 26 ms after impact. The tower was clear of the impactor at about 80 ms after impact. It can be seen from these figures that the FEA model correlated very well with the experimental results for the deformation modes, timing of the buckling and magnitude of deformation.

Impact Force: The impact force-time curves showed that both FEA simulation and full-scale testing follow a very similar trend as shown in Fig. 4 for the example of impact at 140 km/h in the side direction with a 2.72 kg top mass present. It is believed that the oscillations in the full-scale testing impact force-time curve were a result of vibration in the full-scale testing impactor and support structure.
In general, the impact force-time curves showed two peaks that were very significant both in terms of their magnitude and the time at which they occurred. The first of these peaks occurred during the first 4 ms after impact and the second peak occurred during the later stage of the impact, between 22 ms to 40 ms, when the tower was bent around the impactor. For the tests at 50 km/h impact speed, with a top mass present, the first peak was always the global maximum. For the test at 140 km/h impact speed with a top mass present the second peak formed the global maximum.

Energy Absorption: The energy absorption versus time predicted by the FEA simulation was compared to the results of the full-scale tests for all available cases. Energy absorption curves also showed good agreement between the simulation and the full-scale test as shown in Fig. 5 for the representative example of the side direction at 140 km/h with a 2.72 kg top mass present. The curves for all cases followed similar trends and the final energy levels between test and simulation were very close. It would appear that the calculated values for energy approached a maximum value asymptotically for any impact case. The impact speed affected the time the tower was in contact with the impactor and the energy transferred to the tower during this contact period.

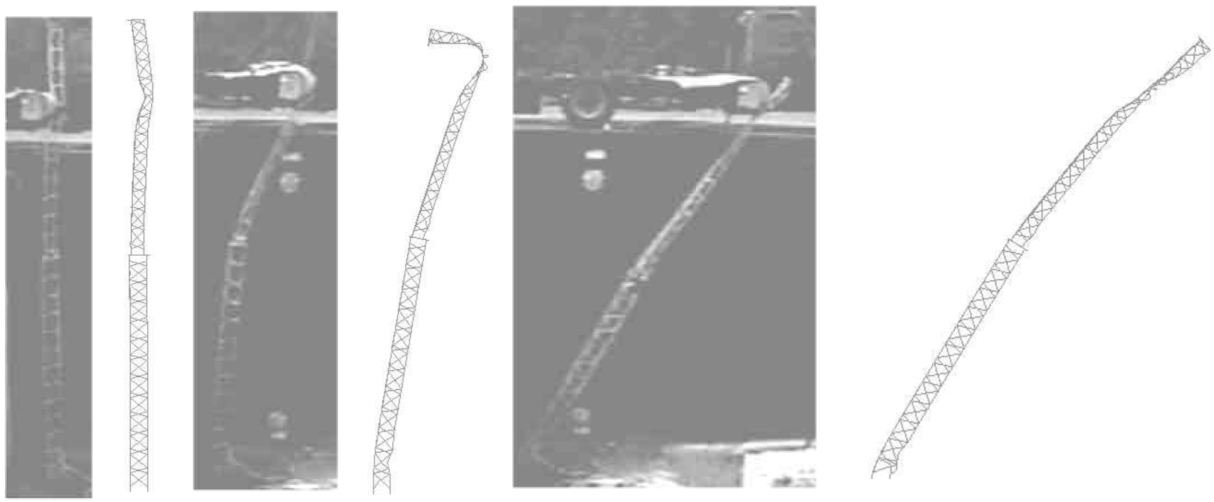
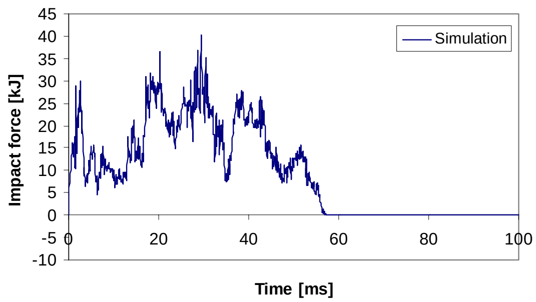
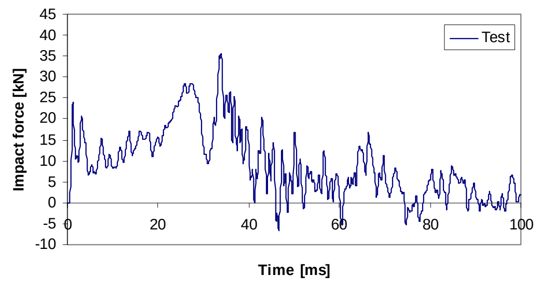


Fig. 3: Impact events from FEA simulation and full-scale test.
Direction: Side, Top mass: 5.44 kg, Velocity: 140 km/h
(Case Study 1)



(a) FEA Simulation



(b) Full-scale Test

Fig. 4a, b: Impact force over time from FEA Simulation and Full-scale Test. Direction: Side, Velocity: 140 km/h, Top mass: 5.44 kg.

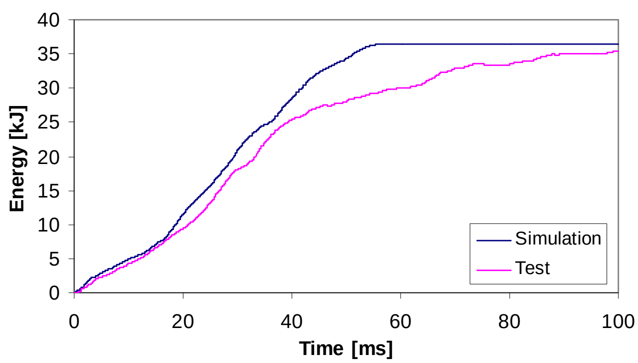


Fig. 5: Energy absorption over time from FEA simulation and full-scale test. Direction: Side, Velocity: 140 km/h, Top mass: 5.44 kg.

Case Study 2:

2.1 Numerical Simulation of an Approach Light Mast

As a second part of this work, another FEA model of a different approach light mast design with a different geometry and made of a different material was developed.
Experimental data from impact of six specimens of this design at 140 km/h by a wing section, representative of a light aircraft, mounted on a truck was available in Ref. [4]. The finite elements model developed to represent this physical design is shown in Fig. 6.
The masts were approximately 6 m tall, and contained a dummy top mass of 15 kg, simulating the effect of a cross-bar with three approach lights. The masts were configured as a fiber-glass/polyester lattic structure with a square cross-section of 400*400 mm, consisting of four vertical tubes, mounted on a steel bracket, and connected with diagonal tubes made of fiber-glass/polyester. The vertical tubes had a 30 mm diameter with 2 mm wall thickness and diagonal tubes had a 20 mm diameter with 2 mm wall thickness. The mast were impacted at the height of 4.1 m.
The mast and the top mass were modeled using beam elements. The impactor model included Belytschko-Tsay shell elements with a uniform thickness. The impactor was modeled using MAT_PLASTIC_KINEMATIC keyword. The material properties used for the impactor were those for Aluminum-Alloy 6061-T6.
A CONTACT_AUTOMATIC_SINGLE_SURFACE definition was used to ensure contact between the various components.
A CONTACT_FORCE_TRANSDUCER_PENALTY was defined to monitor the contact forces at the interface. This transducer element allowed the total contact forces applied by the impactor to the tower to be registered but did not apply any force to the model itself.
Using this FEA model, numerical simulation was performed assuming the impactor struck the tower at an initial velocity of 140 km/h along the z-direction of the mast.

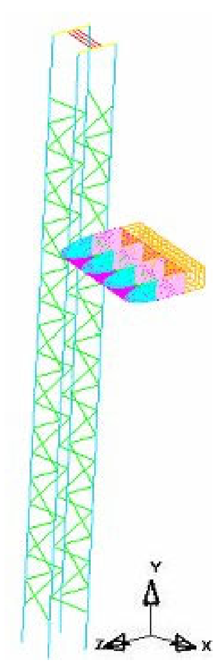


Fig. 6: Finite element model of the approach light mast impact.

Deformation Mode: The deformation mode obtained from the full-scale test and FEA simulation is shown in Fig. 7 at several time steps for the first 25.4 ms. The front vertical tubes cut through the wing nose, until they hit the wing spar. Subsequently, the mast moved with the wing, while the top and bottom of the mast remain in place. It can be seen from these figures that the FEA model correlated very well with the experimental results for the deformation modes, timing of the buckling and magnitude of deformation.

Energy Absorption: The energy absorption versus time predicted by the FEA simulation was compared to the results of the full-scale tests for all available cases. Energy absorption curves also showed good agreement between the simulation and the full-scale test as shown in Fig. 8 a,b. These curves followed similar trends and the final energy levels between test and simulation were very close. The predicted energy was overestimated by 10%, which is a conservative estimation.

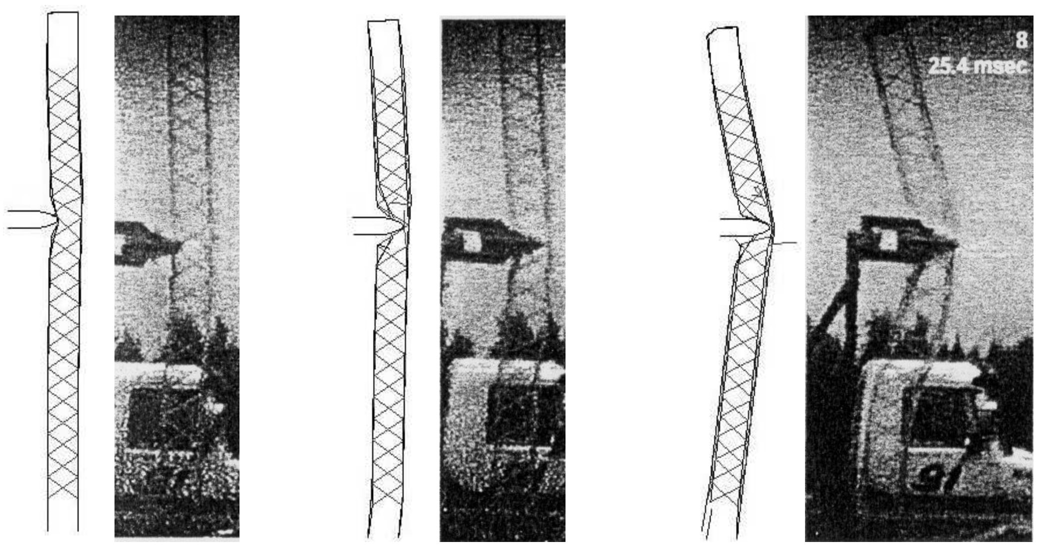


Fig. 7: Impact Events at 140 km/h from FEA Simulation and Full-scale Test*.
(Case Study 2)

* *"Impact test of EXEL approach light masts" by J. Hanka and M. Vahteri, October 1991*

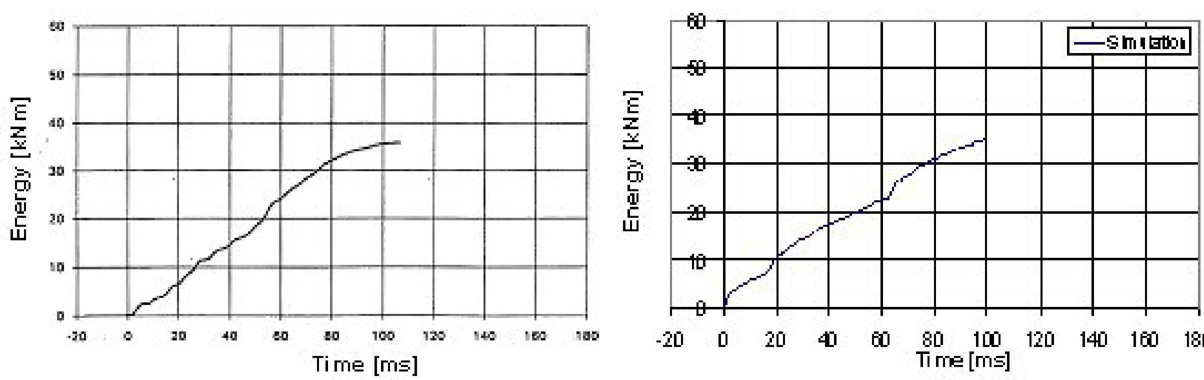


Fig. 8a, b: Energy absorption over time from full-scale and test FEA simulation. Velocity: 140 km/h, Top mass: 15 kg.

Summary

Two finite element impact models have been developed for approach lighting towers typical of those used at Canadian and some European airports. These models have been shown to accurately simulate impact of the tower by a light aircraft. Results from the impact model of the tower provided data on the overall event for several configurations and speeds that compared very well to experimental data. There was good correlation between deformation mode, location and timing of failure, impact force and energy absorption curves obtained from full-scale test results and simulation.

Recommendations from the results of experimental and simulation were presented to the FASG of ICAO. These recommendations have been accepted by the FASG to assist in the development of simplified requirements and test methods for the design of frangible structures to minimize the impact hazard to aircraft.

Acknowledgments

The work described in this paper was partially supported through a collaborative research project under the National Research Council Canada (NRC) and the National Science Council (NSC) of Taiwan Collaborative Research Program. Additional support for this work was provided by the Aerodrome Safety Branch of Transport Canada under contract number T8016-02-0035.

References

[1] International Civil Aviation Organization (ICAO) "Aerodrome Design Manual, Draft, Part 6, Frangibility" Revised Edition, 2003.

[2] Zimcik, D.G., Selmane, A. and Farha M.H. (1999) "Experimental Study on the Frangibility of the Canadian Airport Approach Lighting Towers," Canadian Aeronautics and Space Journal, Vol. 45, No. 1, pp. 32-38.

[3] Zimcik D.G., Nejad Ensan M. and Farha M.H. (2002) "Frangibility of Airport Approach Lighting Towers," Seventh International Conference on Structures Under Shock and Impact (SUSI 2002), Montreal, QC, May 27-29.

[4] Hanka, J. and Vahteri, M. (1991) "Impact Test of Exel Approach Light Masts", Report by Material Research and Testing Department, Neste Composite Technology, Finland.

[5] Robbersmyr, K.G. and Bakken, O.K. (1997) "Impact Test for Lattix Airport Approach Light Towers", Report No. 14/97, Mechanical Department Agder College and Juralco AS, Norway, July.

[6] Hallquist, J.O. (2001) "LS-DYNA Keyword User's Manual", Version 960, Livermore Software Technology Corporation, Livermore, CA.