Finite Element Analysis of the Asiana Airlines Flight 214 Crash

Ivan Pontes, Jonatas Silva, Lorena Neves, Lucas Rigobello,

Otavio Rosado, Paulo Victor Cerqueira, Raphael Magalhaes, Dr. Roberto Cammino, Dr. Aiman Shibli Illinois Institute of Technology

Abstract *- This report synthesizes the results of a Finite Element Analysis (FEA) modeling the Asiana Airlines Flight 214 crash, at San Francisco International Airport (SFO). The goal was to reproduce the movement of the aircraft and to validate how truthfully the method of FEA could represent the real crash dynamically. At first, only rigid elements were used to represent the model with the purpose of using the maximum number of rigid elements in order to save processing time. Secondly, a combination of rigid and deformable elements were used, in order to find the weak spots during impact. Thirdly, cohesive elements were introduced at the weak spots, located at the turbine, tail and wheels, in order to model part separation during impact. The model used 54,254 nodes, with some rigid, cohesive and deformable elements, proving that home computers are capable of running even a complex structure like the one studied here. In order to achieve a higher level of accuracy, all data and parameters used were loyal to the accident and to the Boeing model, including technical information of the impact, structure and material specifications.*

1. Introduction

On July 6th, 2013, an Asiana Airlines Boeing 777-200 flying from Seoul, South Korea to San Francisco, US with 307 people on board, crashed on final approach to runway 28L at the International Airport. This aircraft, which is world known for its outstanding safety standards, hit the seawall 115 meters (375 feet) before the beginning of the runway and the impact force separated the body from tail, gear and engines, coming to a rest left of the runway. The tail burst into flames and burned out, after the aircraft turned around by nearly 360 degrees as the tail fell on the ground. [Aviation Herald, 2013]

The accident was caused by human error, after an abnormally low altitude approach at low speed, resulting from a misinterpretation of the autopilot information.

Therefore, after understanding the accident, the main objective of this research is to discover whether or not is possible to use a finite element software (Abaqus and Hypermesh) to reproduce the impact forces and motion at the moment of crash and obtain similar dynamic results as the real accident in San Francisco. Furthermore, we describe the methodology to simulate the Asiana Airlines Flight Crash with a home computer.

In order to do what was intended, a model based on the accident aircraft was thoroughly analyzed and the elements were chosen as rigid, deformable or cohesive, influenced by how the crash affected them.

2. Materials and Methods

Analyzing the video of the airplane crash [youtube] it was possible to have a rough idea about which components would be relevant and main aspects to be considered during the simulation. Considering the complexity of the situation and the computer's limitations, it was impossible to run the simulation including all variables. Therefore, each single point should be thoroughly evaluated to keep the situation as close as possible to the real situation.

To start the analysis, the team looked for a CAD model of the Boeing 777-200. Using the website www.3dcadbrowser.com, it was possible to find a reasonable model with no major geometry problems, as shown in Fig. 1.

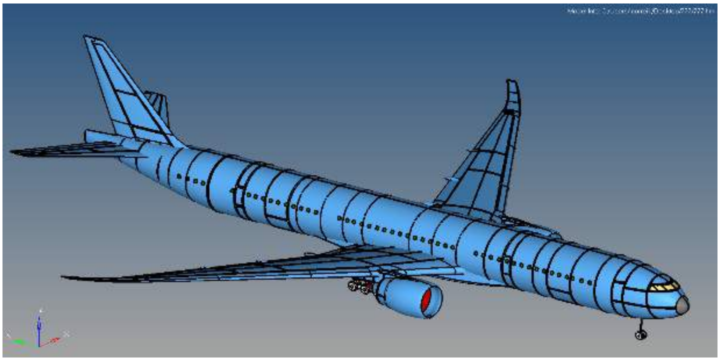

Fig. 1. CAD model for the Boeing 777-200

Considering that the model did not include seats, luggage, passengers, and all miscellaneous components we still maintained the whole weight of the aircraft, as it would affect the aircraft's trajectory after the impact. This lost weight was added to the body of the aircraft. The aircraft's weight was determined, to be 208 tons considering the maximum landing weigh according to the bibliography. [Swane, Yourkows] The entire mass of the aircraft was distributed through the density of each material over the components used in the Hypermesh model. In fact, the material used for the fuselage was Aluminum alloy of 2000 series (Al 2XXX); for the fin, stabilizer and aft bulkhead it was Al 6013; Al 7055-T77 was the material used for the wings; the turbine weight was based on the manufacturer specifications; the wheels weight were also based on the manufacturer specification. Fig. 2 and Table 1 represent the components and its respective weights.

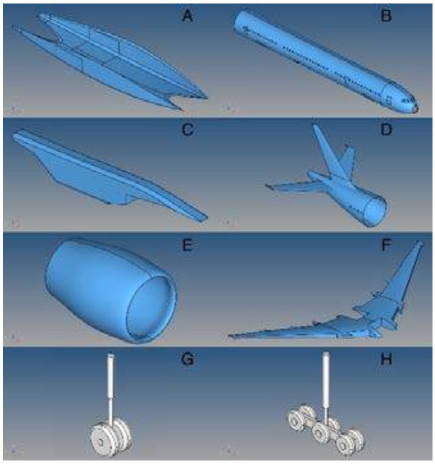

Fig. 2. Components in the final model Table 1. List of adopted weight

Figure	Component	Mass (Tonne)
A	Fairing	12.8
B	Main Fuselage and Flight Deck	16.6
C	Pylon Assembly	8.6
D	Aft Bulkhead, Fins and Stabilizers	3.6
E	Turbine Assembly (Each)	19.5
F	Wings	6.0
G	Front Gear Assembly	1.6
H	Left or Right Gear Assembly	2.0

The model was positioned according to the aircraft accident official report emitted by National Transportation Safety Board, which states that the angle of attack (the angle between horizon and aircraft) was 12 degrees, and the roll angle (angle in vertical axis) was 2.6 degrees. The report also states the aircraft's velocity in the moment of the crash as being 105.5 knots (54273.89 millimeters/second). [NTSB, 2014]

With the objective to simulate the exact conditions from the crash, the runway and the ground beside it were modeled exactly with the real specifications. A friction coefficient of 0.65 for the runway and 0.85 for the ground was used. Also, the runway and ground were kept as rigid bodies for the simulation.

Using the software Hypermesh, the model was simplified and just the main parts - such as the main fuselage, flight deck, wing, turbines, gears, fairing, aft Bulkhead, fins, and stabilizers - were kept to proceed with the mesh. The mesh was composed of 2D and 3D elements. Fig. 3 shows the first meshed model.

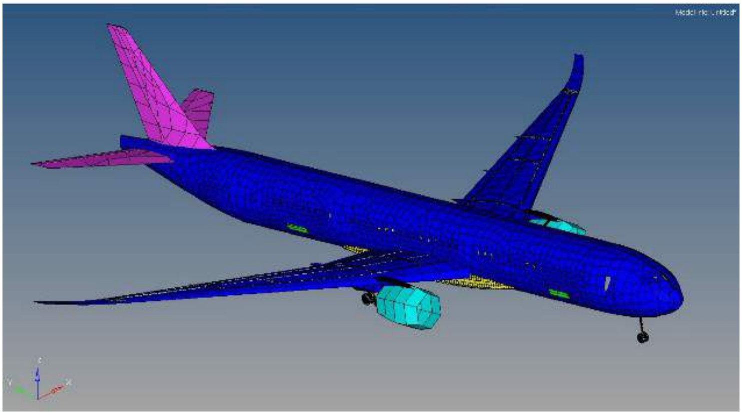

Fig. 3. First Meshed Model

We realized it is necessary to use some special techniques, since Abaqus has difficulty calculating contacts between two rigid bodies. If the contacts are not properly defined, the two bodies will pass through each other. The only way that was found to fix this problem, was using the contact penalty method, which must be used in a **DYNAMIC, EXPLICIT* simulation inside the

**STEP*, as shown below.

```
*STEP
*DYNAMIC, EXPLICIT
*CONTACT PAIR, MECHANICAL CONSTRAINT=PENALTY
     Component1,Component2
Component3,Component4.
*END STEP
```

An important detail is that, at least one of the two components involved in the contact pair must contain 3D elements. This kind of contact cannot be defined between two rigid surfaces.

In order to discover where the weak spots are, we created the second model, which is constituted by a mix of rigid and deformable elements. The regions that flew apart were probably the weak spots. This hypothesis was tested performing some analyses to find out where the high stresses would occur, by placing deformable elements in certain section of the aircraft and observing the stresses at those locations. Figure 5, shows the high stresses on the tail area during impact, once deformable elements are placed there. This approach will guide use to strategically locate cohesive elements in these areas, so that they can break off during impact

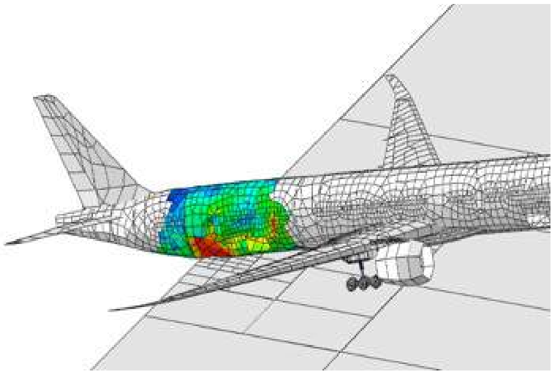

Figure 4. High stresses during impact at the tail area.

For the purpose of simulating the parts separating from the airplane in the crash, a special type of element was used. The cohesive elements were added between the turbine support and the wing; the tail and the main fuselage; the landing gear and the bottom fuselage, as seen in Fig.4. The cohesive element has the property to break up and delete itself when a specified stress is reached, resulting in the parts flying apart.

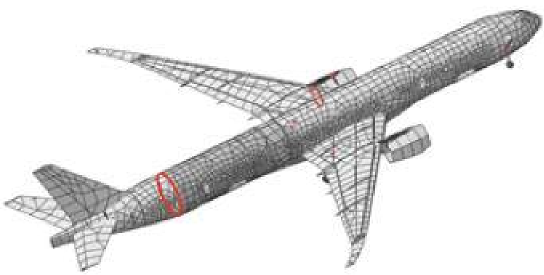

Fig. 5. Cohesive elements highlighted in red

The cohesive elements were declared according to the following procedure:

```
*COHESIVE SECTION, ELSET=Component_name, CONTROLS=Control
_name, MATERIAL=Material_name, RESPONSE=TRACTION SEPARATION
thickness=SPEC
IFIED 1.0,
```

The *TRACTION* SEPARATION method was used, in which a zero thickness is assumed, and section control is specified according to the following command:

```
*SECTION Control, name=Control _name, ELEMENT DELETION=YES
1.,1.,1.
```

The element deletion option must be activated by using two commands. First, at the template above by using the option ELEMENT DELETION=YES, and second, inside **STEP* by setting the *STATUS* in the **ELEMENT OUTPUT*.

The components that will be separating, must be released by the time the maximum stresses in the material are reached. The command that controls the release criteria is called damage criteria. To accomplish that, the damage initiation criteria was set as maximum stress, and since the damage evolution was not desirable, a very small number had been used.

```
*MATERIAL, name=Adhesive
*Damage Initiation, criterion=MAXS
Ultimate_Tensile,Ultimate_Shear,Ultimate_Shear
*Damage Evolution,
type=DISPLACEMENT 1e-30,
*Density
0.0000000078,
*Elastic,
type=TRACTION 7e+05,
5e+05, 5e+05
```

Having performed the fist tests on a rather crude model, in order to reduce the total elements and convince ourselves that we could run this type of simulation, we concluded our model needed a remesh, in order to more accurately capture the necessary features during impact. We therefore remeshed the structure allowing for smaller elements, now that we feel confident that our run times are reasonable. Fig. 5 shows the final meshed model.

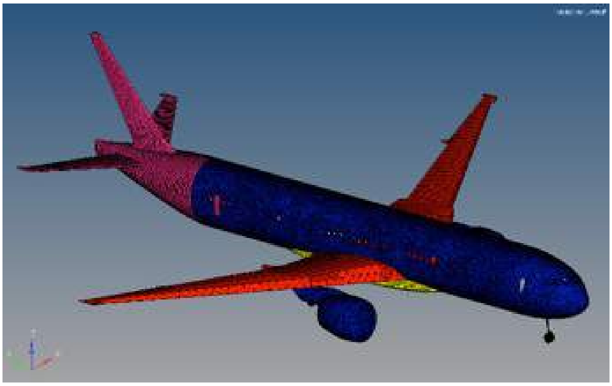

Fig. 5. Remeshed Model

3 Results

This research was able to reproduce great part of the accident using finite element method software such as Hypermesh and Abaqus. This can be useful for future analysis of crashes and impacts.

However, to perform this analysis some simplifications had to be made. Some structural details were suppressed in this simulation. Some simplifications were required, such as the removal of some structural parts, seats, and engines, allowing the model to be as simple as possible to run in a home computer.

The rigid elements were not enough to reproduce the motion, being a huge problem since this kind of element does not absorb the energy from the impact. In addition, the gravity force does not work for rigid elements. In addition, we had the issue of rigid to rigid contact, where the rigid elements were penetrating the floor as soon as the aircraft touched it, unless the aforementioned changes were made. These three issues contributed for an inaccurate simulation.

To solve the rigid elements problem, the model was turned into deformable. This way, the model stopped penetrating the floor and the impact's energy was correctly absolved, producing a satisfactory result.

4 Discussion

The first model was made just of rigid bodies. However, it was noticed that Abaqus presents some limitations using this kind of elements: at least some deformable elements are needed. The limitations that we found during the simulations were the contact between two rigid bodies, and the computational cost when using an EXPLICIT simulation with just rigid bodies in Abaqus.

To solve the first problem, we used the penalty contact function in Abaqus, as described in the methodology. This contact pair must be declared with at least one 3D component involved in the contact pair, it cannot be defined between two surfaces.

It is important to mention that because this command act as if it had "elastic springs" in the contact regions, no energy is dissipated in the process. The kinetic energy is stored as potential energy in these "springs" that is transformed in kinetic energy again. So the total energy in this system was almost conserved because only rigid elements were used to construct the model, so no energy dissipation by deformation was present in the system. The only energy dissipation present in this system was the one caused by the friction between the airplane and ground/grass and the airplane and runway. Friction, was assigned in the following manner. All keywords have been obtained from the Abaqus User's Manual online.

```
*SURFACE INTERACTION, NAME=ALFRICTIONrunway
*FRICTI
ON
0.65,
*SURFACE INTERACTION, NAME=ALFRICTIONgrass
*FRICTI
ON
0.85,
```

Another important thing to mention is that by using only rigid elements and setting the contact penalty, in a *DYNAMIC, EXPLICIT, may cause a severe computational cost for the computer to run this file because when Abaqus computes this kind of explicit simulation it use the deformable elements to calculate the increment time, but because there aren't any of them the software struggles in calculating this increment.

After facing this problems it was decided to combine the two kind of elements, deformable and rigid, because it was realized that only rigid elements were not able to describe the exact deformation of this problem.

In the second model it was decided to use some parts of the aircraft as deformable elements as well. At first, we used deformable elements to check the specific location of each weak spot, and the failure points, as it is shown in the figure 5.

At this point, it was still a concern for us that gravity was not captured on rigid elements. As shown in Fig. 6, the aircraft presented an almost correct motion but without gravity, it did not land on the floor, but rather bounced around.

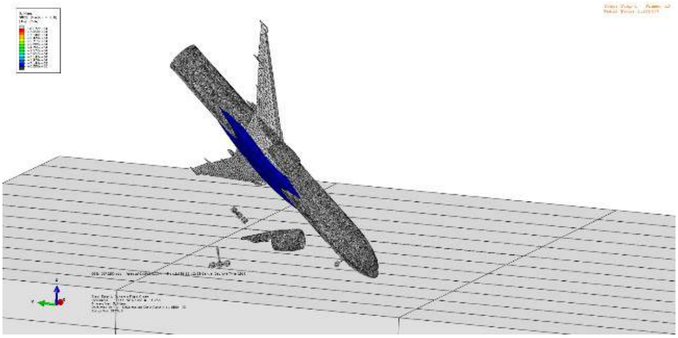

Fig. 6. Rigid model with Some Deformable Elements to allow the simulation to run in Abaqus

The immediate solution was to run the whole model as deformable, as represented by Fig.

7. We will also provide an alternative to this option of making the entire structure deformable in order to capture gravity effects.

One might use point masses elements with the mass value of the part that you want to use for the individual rigid portions and placed at the center of gravity of the body studied. These mass elements are susceptible to the gravity command in ABAQUS.

```
*STEP
*DYNAMIC, EXPLICIT
                                    .

*DLOAD
,GRAV,gravity_number_value,0,0,
-1 (Ex.: ,GRAV,9820,0,0,-1)
                                    .
*END STEP
```

The results for the last simulation are shown in Fig. 7, where we put the majority of the elements as deformable, and leaving just the parts that flew apart as rigid. The pros of doing this is that the gravity worked properly and we got more deformation that leads to more energy dissipation and more accurate results to the airplane motion.

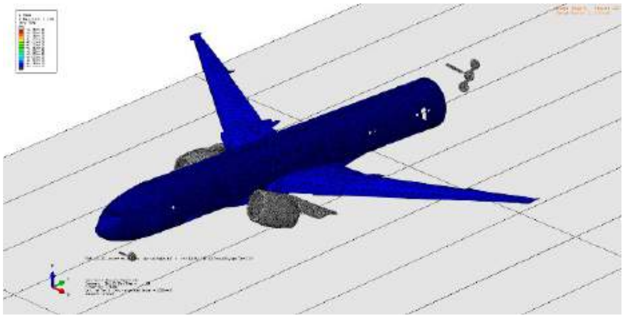

Fig. 7. Deformable model with gravity effects

On the other hand, this change makes the simulation more costly, thus spending more time to run the analysis. We were still able to complete this on desktop computer without using any special servers. Figure 7 shows the aircraft as it slides on the runway, when the tail has fallen off and the wheels and turbines are separating as they impact the ground.

5. Conclusion

In this research, we simulated close to exactly the Asiana Airlines Boeing 777-200 crash as it occurred on July 6th, 2013 at the San Francisco International Airport. In future works, it is suggested that an ideal mix between rigid and deformable elements to be used, thus saving a lot of computational power needed to perform this task. Doing the methodology described in this paper such as choosing wisely the spots where the elements need to be deformable or rigid, finding the correct spots to use the cohesive elements, and doing every technique used in this paper to declare the contact between the components, it is highly likely that by using Finite Elements Analysis Tool will be enough to recreate the exact movement of this crash leading to many other research possibilities such as how to avoid life losses during an aircraft crash, or maybe new runway parameters to avoid material and human losses.

6. Acknowledgements

Special thanks to the Armour College of Engineering

Special thanks to Capes (Coord. De Aperfeiçoamento de Pessoal de Nível Superior) for funding the Engineering Themes course ENGR498

7. References

1 "Descent Below Visual Glidepath and Impact With Seawall Asiana Airlines Flight 214." 14 June 2014. National Transportation Safety Board.

2 “Accident: Asiana B772 at San Francisco on July 6th 2013, touched down short of the runway, broke up and burst into flames”. The Aviation Herald, July 6th 2013.

3 "Boeing 777-200 3D Model." 3DCadBrowser, 4 Dec. 2008.

4 "Abaqus Analysis User's Manual." Web. 10 June 2015.

5 Yourkows, Swane. "777 Tour Fun Facts & Data." University of Washington, 11Mar. 2003..

6 "San Francisco Airport Crash: Asiana Flight214 Crash Reconstructed." Youtube. TheTelegraph, 15 July 2013