Crash Simulation of KTM “X-BOW” Car Front Impact Structure

Katharina Fischer[1], Phelippe Pereira[2], Madhukar Chatiri[3],
Matthias Hörmann[3], Andre Stühmeyer[3]

[1]*KTM Technologies GmbH, Salzburg/Anif, Austria, katharina.fischer@ktm-technologies.com*
[2]*ESSS, Sao Paulo, Brazil, phelippe.pereira@esss.com.br*
[3] *CADFEM GmbH, Grafing/Munich, Germany, mchatiri@cadfem.de*

Abstract

*The goal of this presentation is to study the structural behavior of the KTM “X-BOW” crash box front impact structure in a 0° impact test against a rigid wall. The energy absorbing crash box is made of laminated composite sandwich material. A “shell-solid-shell” numerical approach is used to model the sandwich composite structure. Shell elements are used for the face layers whereas solid elements are used for aluminum honeycomb core. Shell elements consider the composite layering using *ELEMENT_SHELL_OFFSET_COMPOSITE within LS-DYNA® and will be bonded to the solid elements without node sharing. The composite structure is modeled using *MAT_054 and honeycomb structure is modeled using *MAT_126 within LS-DYNA. For comparison reasons, numerical and experimental results for intrusion, deceleration, velocity and displacement over time are presented.*

Keywords: *KTM “X-BOW”, Frontal Impact, composite sandwich material*

1 Introduction

The present work is involved in examining the crash box structure behavior of KTM X-BOW (see Fig. 1.1) used to ensure driver´s safety in the event of a frontal impact. Crash box is a sacrificial impact structure which absorbs the car´s kinetic energy and limits the decelerations acting on human body. The main philosophy adopted by crashworthiness regulations is to assure that the driver is enclosed within a strong survival cell with absorbing structures in the front, back and sides

Fig. 1.1 – KTM X-BOW (right view)

Composite materials are used in the concept of the X-BOW because of strength/weight relation, best for energy absorption and relatively low cost. While metals absorb energy by plastic deformation, composites do so by braking and crushing into small fragments [1].

The aim of this study is to apply the finite element (FE) method by a commercial explicit FE code LS-DYNA to perform a dynamic nonlinear simulation of a 0° impact test against a rigid wall of KTM X-BOW front impact structure. This composite structure consists of the crash box, which is attached to the monocoque by bolts (see Fig. 1.2). The goal of this simulation is to capture the intrusion, general deformation and deceleration curves to know the crash behavior of the composite crash box unit. Details about geometry, mesh, boundary conditions, contact definitions, material modeling and numerical results will be presented in the next sections. Numerical results are compared to experimental crash test results.

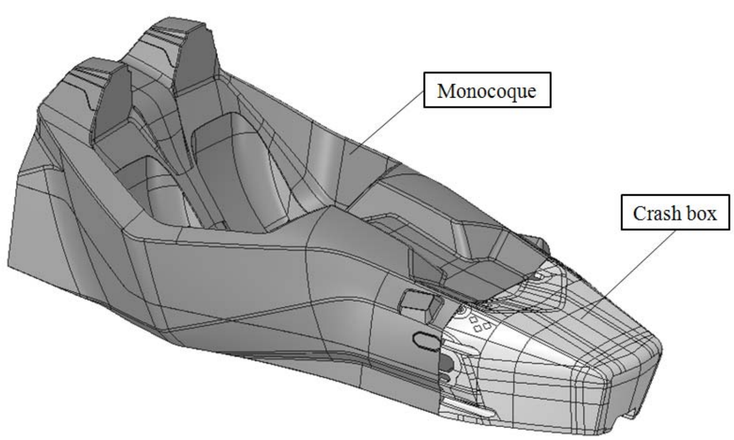

Fig. 1.2 – Illustration of X-BOW front structure components

2 Frontal Impact Crash Testing

This chapter describes the experimental frontal impact crash test of the KTM X-BOW. The crash test was performed by the CSI Certificazione e Testing according to FIA ART. 259 TECHNICAL REGULATIONS FOR PRODUCTION SPORT CARS (see Fig. 2.1). The frontal impact test specified by FIA forecast that the crash box is able to absorb the kinetic energy involved in the crash and the driver is protected from injuries provided by deceleration forces.

Fig. 2.1 – (left) Nose box before the test. (right) Nose box after the test.

The total mass considered is 925 kg and the front impact system (monocoque, sled, two dummies and crash box) will impact with an initial velocity of 12 m/s against a rigid wall. The total mass is distributed by the mass of monocoque (70 kg), the mass of two dummies (175 kg), the sled mass (675 kg) and finally the mass of the crash box (8 kg).

Experimental curves of displacement, velocity and deceleration over time were measured in the experimental frontal impact crash test and are compared to the numerical results from simulation.

3 Mechanical model

The following sections describe the mathematical model that represents the mechanical behavior of a crash impact test. Firstly, a workflow is presented to demonstrate the steps used to develop this analysis. Starting from the CAD model, a Finite Element (FE) model has been generated and a dynamic nonlinear analysis has been performed using nonlinear contact definitions and nonlinear material definitions. Assumptions about the material modeling as well as material properties are described in this chapter.

3.1 Workflow

This numerical simulation follows the workflow showed in Fig. 3.1. The pre-processors Workbench (WB) and ANSYS SpaceClaim Direct Modeler (SCDM) are used to clean up the geometry. Workbench LS-DYNA (WB LS-DYNA) is used to assign materials, to mesh the model, to define contact interactions and to define boundary conditions. After this process the LS-DYNA input file (*.k) is generated and composite layers can be defined inside ANSYS

Composite PrepPost (ACP). LS-DYNA solver is used to solve the analysis and the LS-PrePost® (LSPP) is used to do the post-processing. Simulation results can be compared with experimental results by graphical comparison of the resultant displacement, velocity and deceleration over time.

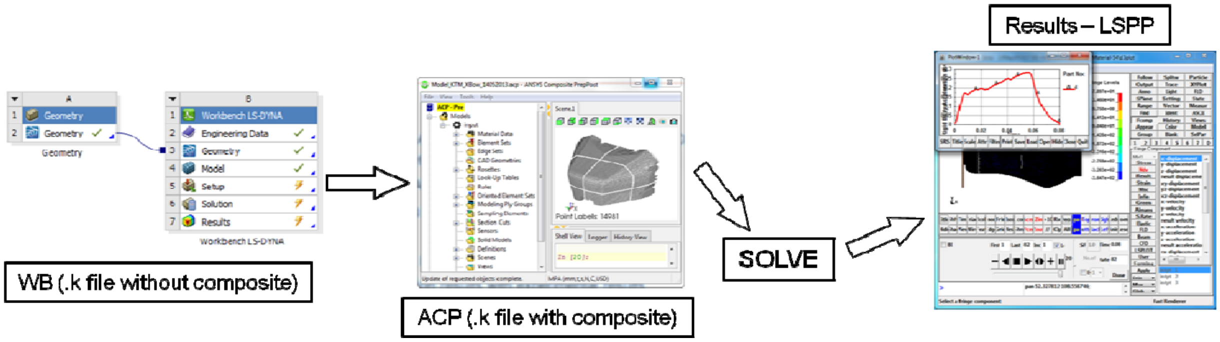

Fig. 3.1 – Workflow of the analysis

3.2 Geometric model

The CAD geometry of the KTM X-BOW is composed of one monocoque that is modeled as a rigid body, one crash box that is represented by a composite material and an impactor (rigid wall) that is fixed in space (see Fig. 3.2). Here, the sled and two dummies are not modeled but their masses are accounted in monocoque rigid surface body mass. Geometries received from the customer have been cleaned up using SCDM to remove some details that will not be considered in this study and to improve the mesh quality.

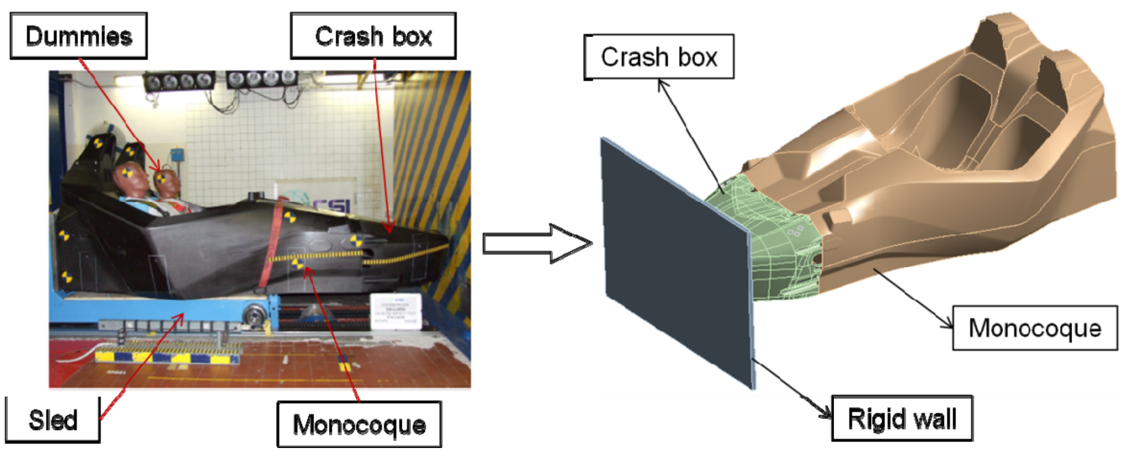

Fig. 3.2 – CAD structure with all components

A shell-solid-shell numerical model is used to model the crash box (sandwich composite structure). The crash box unit is composed by one solid body, two surfaces and 16 aluminum inserts. The CAD geometry and their parts are showed in Fig. 3.3.

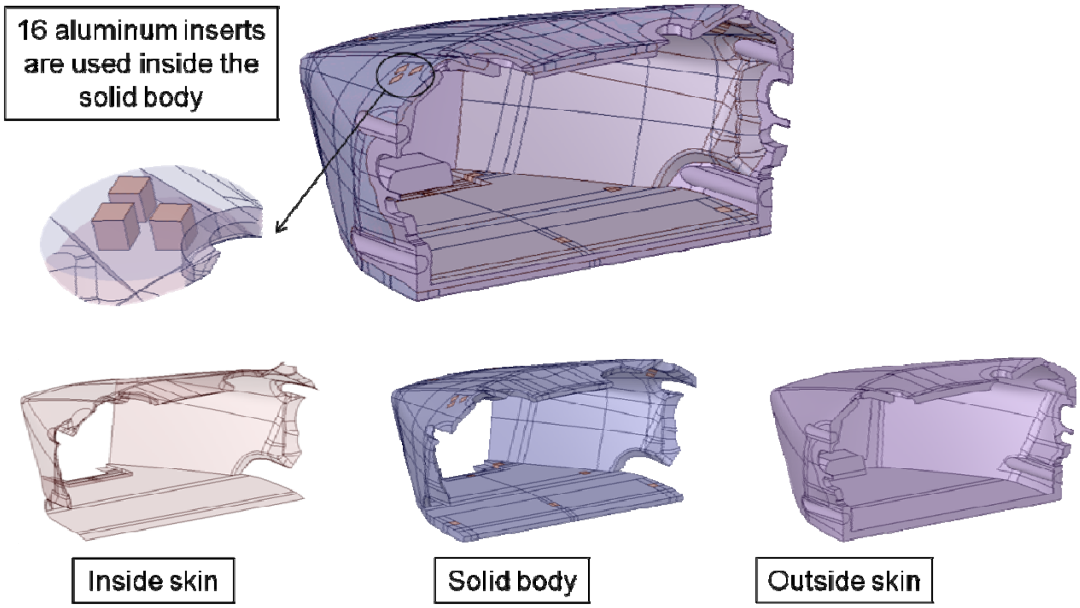

Fig. 3.3 – Crash box CAD geometry and their components

3.3 Meshing

Based on CAD data the mesh was generated in WB LS-DYNA. Each part has been separately meshed and the interface between the different domains (solid body and surface body) has been modeled using contact definitions. The inside skin and outside skin have been meshed by quadrilateral elements and the core has been meshed using hexahedral elements, predominantly. The model in Fig. 3.4 consists of 338034 nodes and 332078 elements.

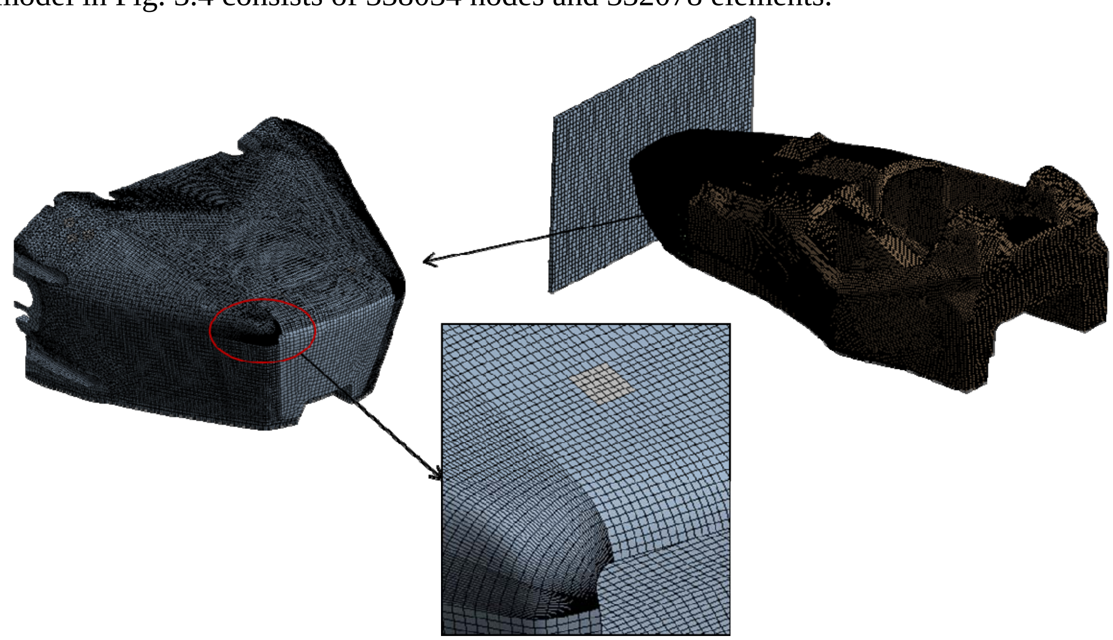

Fig. 3. 4 – Workbench LS-DYNA´s meshed model

3.4 Material model

The crash box is made of a lightweight sandwich composite structure consisting of carbon fiber skin laminates and aluminum honeycomb core. To represent this sandwich composite structure a shell-solid-shell approach is used that consists in using shell elements for the face layers and solid elements for the aluminum honeycomb core. This formulation has advantages like full 3D stress state in core, includes skin/core debonding and a representation for core shear deformations and core indentation in thickness direction. Shell elements will account for the composite layering and will be bonded to the solid elements.

For the carbon fiber skins (outside and inside skins) the composite material model *MAT_054 (*MAT_ENHANCED_COMPOSITE_DAMAGE) is used [2, 3]. This orthotropic, linear elastic material law is based on the Chang/Chang failure criteria, which controls failure in longitudinal and transverse direction under compressive, tensile and shear loads. This material model is based on stiffness reduction where the stress based criteria is applied. Parameters are set to zero in case of violation of the failure specific failure mode to model the material degradation. The used criteria are also shown in [4]

For the aluminum honeycomb core (solid body) the material model *MAT_126 (*MAT_MODIFIED_HONEYCOMB) is used [2, 3]. The major use of the Material Type 126 is for aluminum honeycomb crushable foam material where three yield surfaces are available (Type A, B and C). The type A will be considered in this study, where stress over strain material data are required to define the resistance in each direction for normal and shear stresses. All these stress vs. strain material properties data can be considered separately and fully uncoupled. The load curves LCA, LCB, LCC give the stress over strain for each component direction axis. The load curve LCS gives the shear stress versus shear strain for all directions. LCAB, LCBC and LCCA give stress over shear strain for each combined direction.

3.5 Contact definitions

Contacts pairs are used to represent the body interaction between flexible bodies or the body interaction between flexible and rigid bodies. The body interaction between aluminum honeycomb core and composite skins are modeled by a bonded contact (*CONTACT_TIED_NODES_TO_SURFACE_OFFSET) that has a permanent connection.

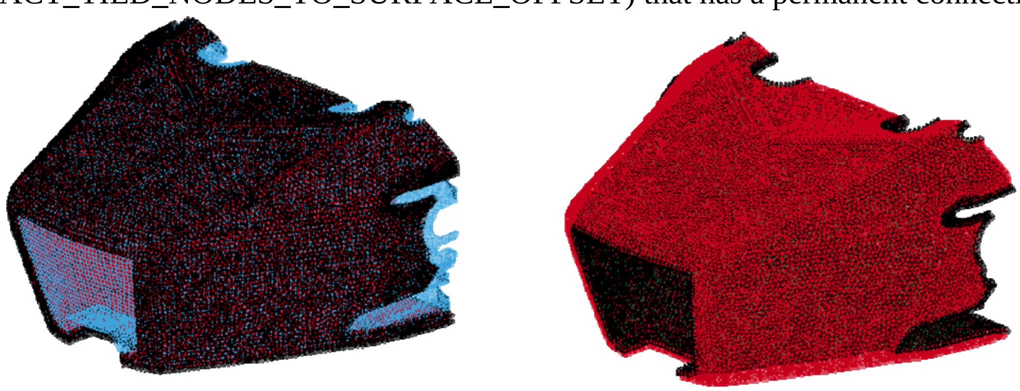

Fig. 3. 5 – Bonded contact between skin´s nodes with core body´s surface

Fig. 3. 5 shows the application of a bonded contact between the nodes of outside/inside composite skins and the aluminum honeycomb core body´s surface to model the sandwich structure of crash box.

To represent the interaction between crash box and the rigid wall a contact with friction is assumed. The coefficient of friction used is zero, allowing the sliding and separation movement however, the penetration between bodies is not allowed. This frictional contact (*CONTACT_AUTOMATIC_NODES_TO_SURFACE, see Fig. 3. 6) illustrates the frictional contact between inside composite skin and rigid wall (left), frictional contact between surfaces of aluminum honeycomb core and rigid wall (middle) and frictional contact between outside composite skin and rigid wall (right). The frictional contact between surfaces of aluminum honeycomb core and rigid wall is used to consider the interaction of these two bodies in a case of failure and erosion of the outside composite skin´s elements. The same assumption is used to consider the frictional contact between inside composite skin and rigid wall.

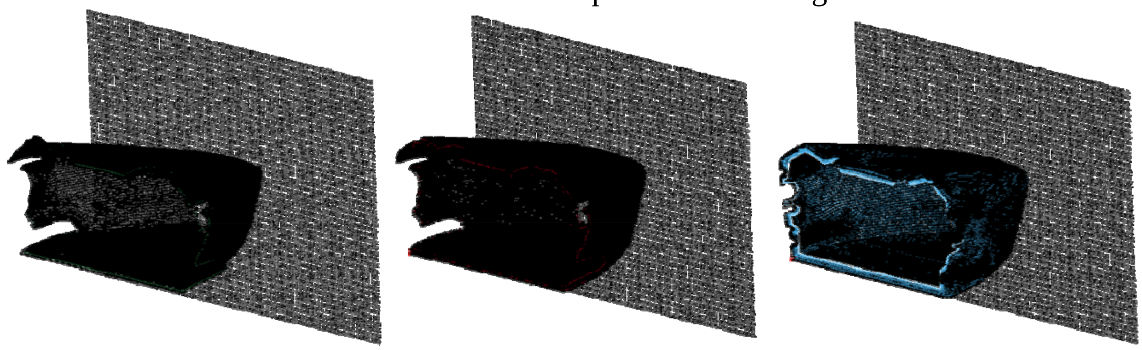

Fig. 3. 6 – Illustration of frictional contact

The interaction of outside composite skin and monocoque (see Fig. 3. 7) is modeled by a frictional contact with a frictional coefficient of zero that allows sliding and separation but constrains the penetration.

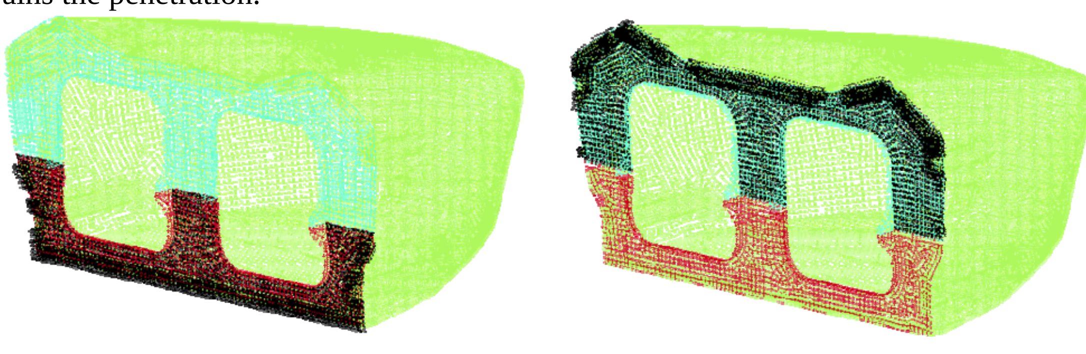

Fig. 3. 7 – Frictional contact of outside composite skin and monocoque

The interaction between outside and inside composite skins is modeled by a frictionless body interaction (*CONTACT_AUTOMATIC_SINGLE_SURFACE) that permits sliding but avoids any penetration. This frictionless body interaction is used to get a survival space between these two surfaces in case of a very high deformation of the aluminum honeycomb core material. The illustration of this body interaction is showed in Fig. 3. 8.

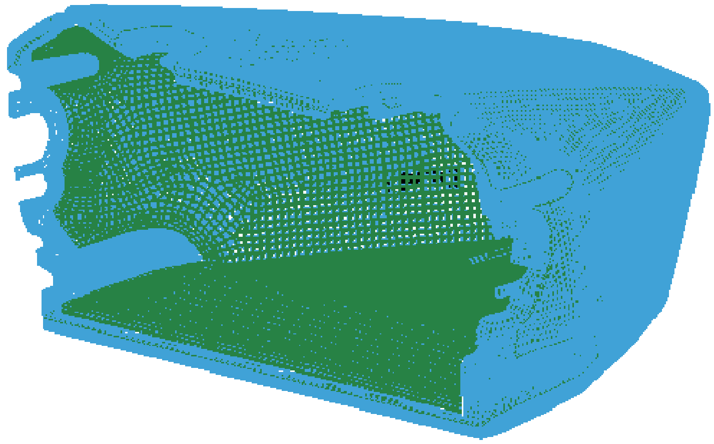

Fig. 3. 8 – Frictionless body interaction of inside and outside composite skins

3.6 Boundary conditions

The initial velocity of 12 m/s used in the experimental test is applied in this study. In the experiment an initial velocity is applied to the sled structure, but here this structure is not modeled, consequently this initial velocity is applied in the monocoque and crash box by an initial velocity command (see Fig. 3. 9).

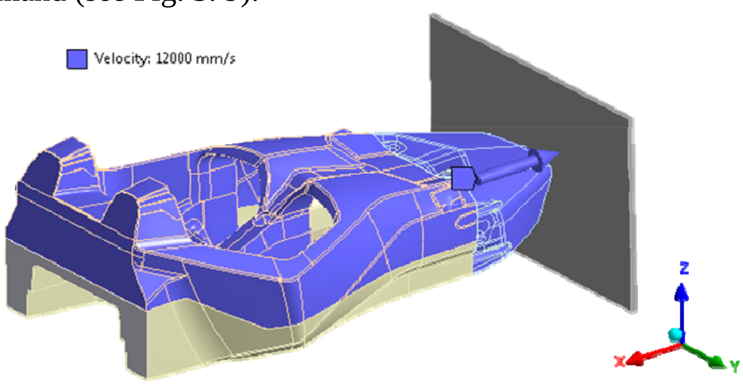

Fig. 3. 9 – Illustration of the Initial Velocity command

The total mass of this structure is 925 kg, where the mass of two dummies and the mass of sled structure is applied to monocoque total mass (917 kg) by changing the density. In the experimental test, a strip is used to constrain the monocoque structure, which is free to translate in x-direction only, keeping other degrees of freedom fixed. In this study, the tying strip used is not needed and not modeled because this constrain is applied directly on the monocoque structure that is modeled as a rigid body. The monocoque structure is divided in five rigid surface bodies that are constrained to one master rigid body that will receive all boundary conditions as showed in Fig. 3. 9, where the initial velocity is applied in the monocoque master rigid body only. A rigid body constraint command is applied to monocoque allowing the movement in x-direction only and for the impactor that is fixed in space (see Fig. 3. 10).

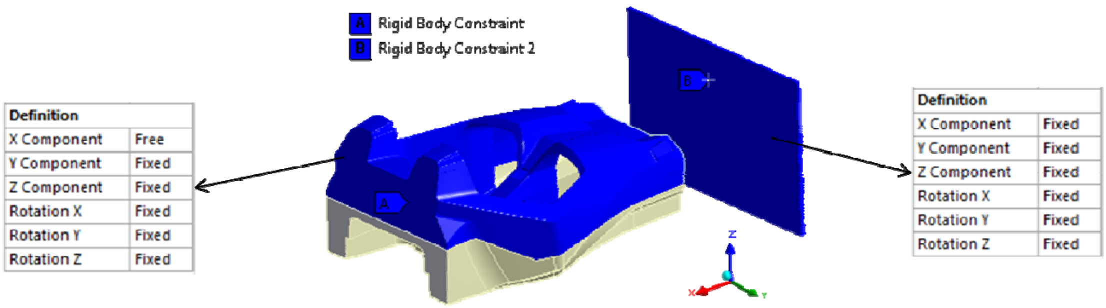

Fig. 3. 10 – Illustration of the Rigid Body Constraint command

3.7 Composite modeling using ANSYS Composite PrepPost

ANSYS Composite PrepPost (ACP) is used to model composite laminates for outside and inside skins of the crash box structure. Before we go to ACP, it´s necessary to create some named selections inside Workbench (WB) LS-DYNA that will be imported as Element Sets inside ACP. These Element Sets are used to identify elements that will receive different types of layers. The meshed input file (*.k) generated by WB LS-DYNA is imported inside ACP and only surfaces are read. Inside ACP, composites materials are created and assigned to each ply.

KTM Technologies provided the material data that also include the order of each layer that will be applied to generate the outside and inside composite skins.

The illustration of all plies assigned in ACP is showed in Fig. 3. 11, where it is possible to indentify the usage of 10 plies in the back part of the crash box outside skin, where only skins are used.

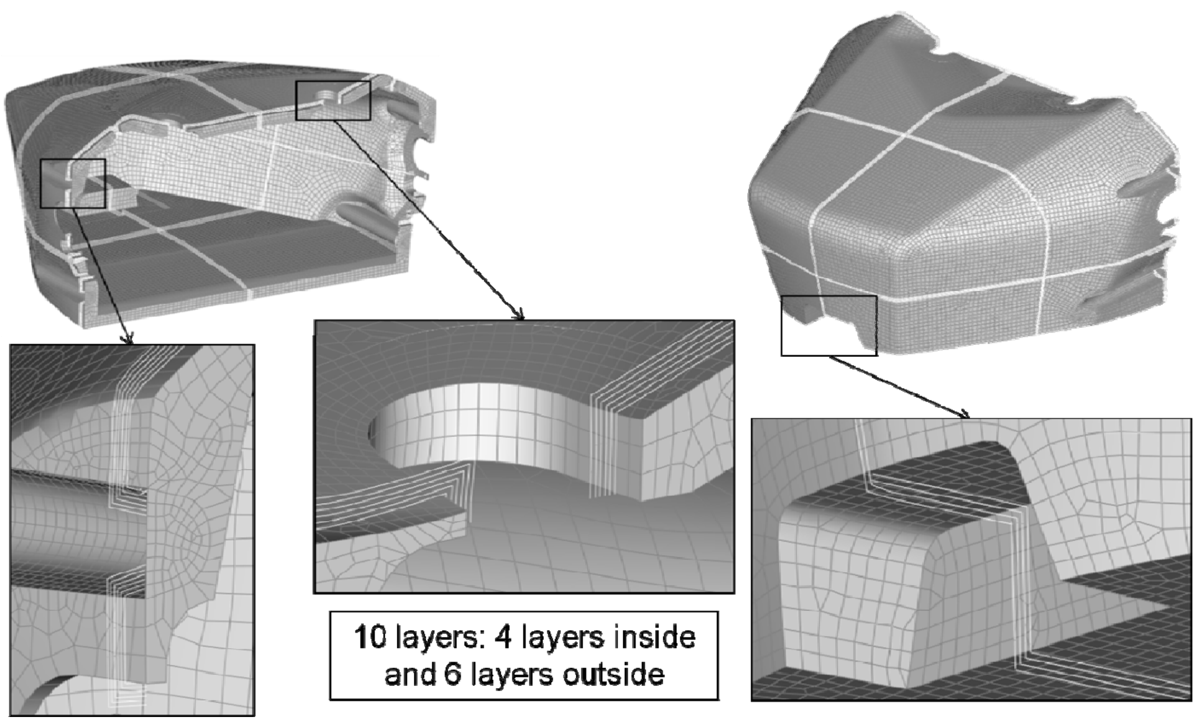

Fig. 3. 11 – Illustration of layers modeled in ACP

4 Results

The energy balance response curve for the model is illustrated in the Fig. 4. 1, where it´s possible to see at 70 ms the X-BOW starts to bounce back.

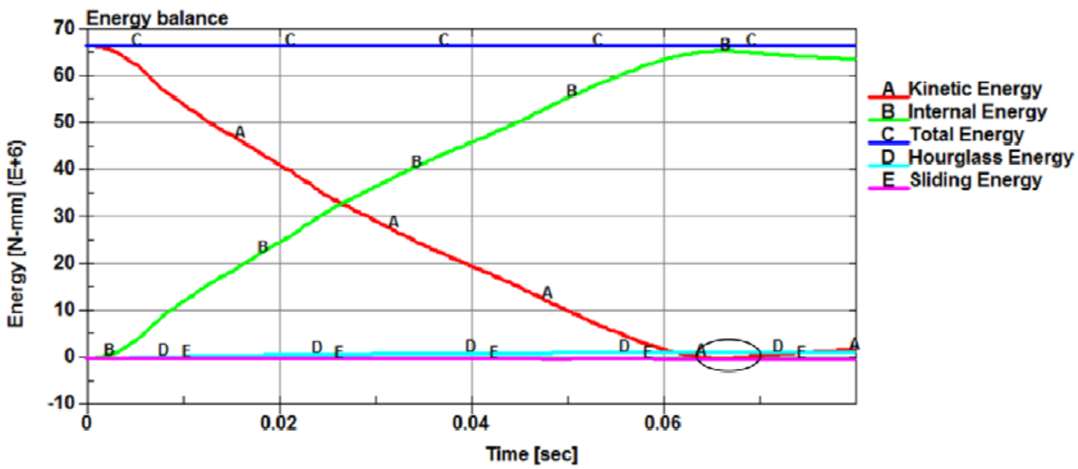

Fig. 4. 1 – Illustration of the Energy balance – model 4

For comparison reasons the numerical intrusion response are shown in Fig. 4. 2, at specific time steps (0.0, 25.0, 50.0 and 75.0 ms).

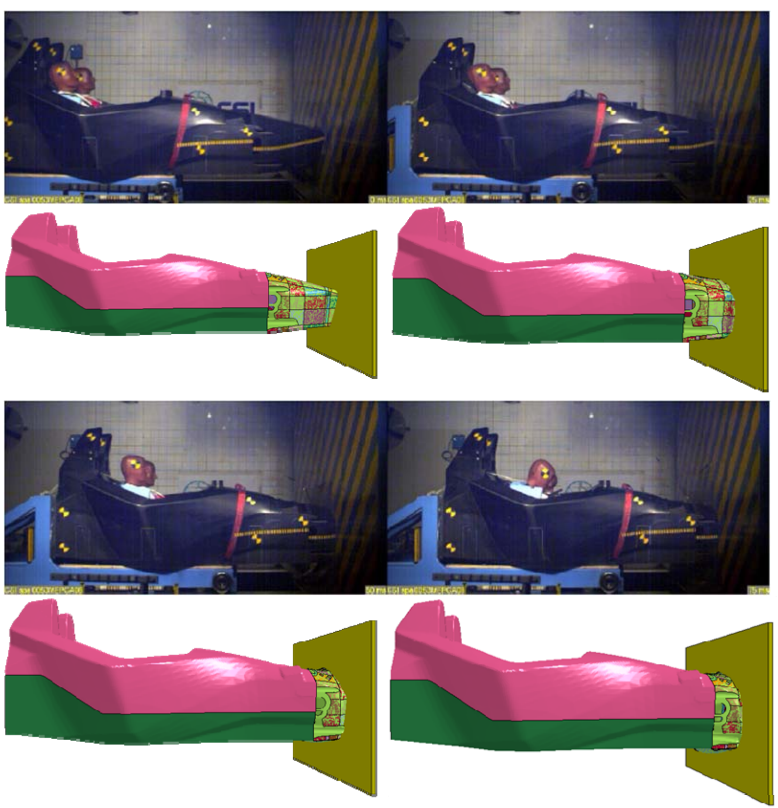

Fig. 4. 2 – Experimental and numerical intrusion results

The numerical and experimental deceleration response over time is shown in Fig. 4. 3.

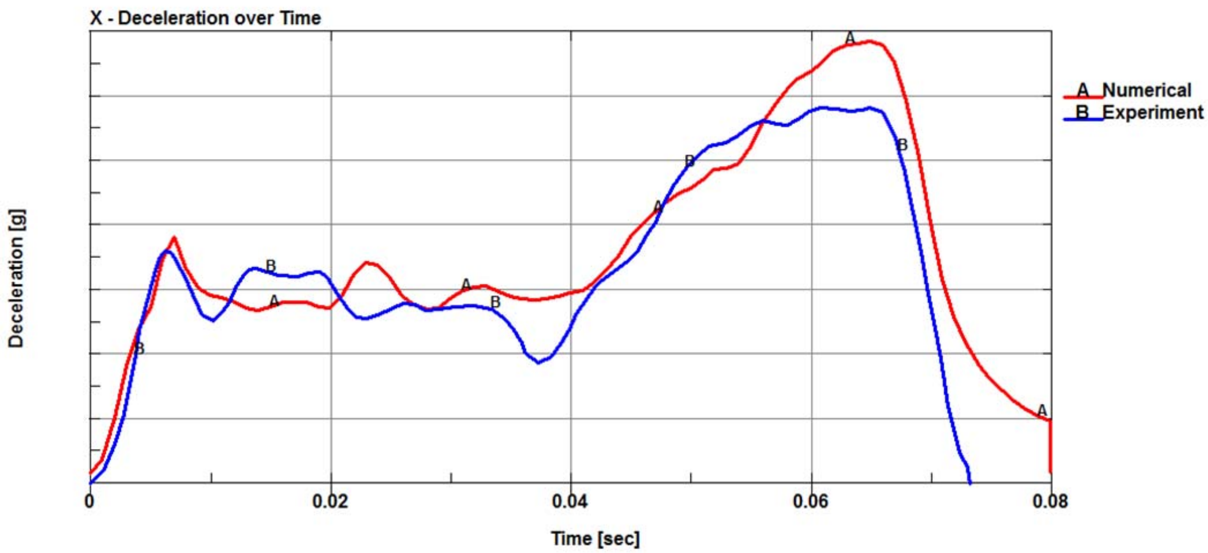

Fig. 4.3 – Comparison between experimental and numerical deceleration result

5 Summary and conclusions

Impact and crushing simulations KTM "X-BOW" Front Impact Structure striking a rigid wall have been conducted and validated against experimental crash test data. Crash front algorithm inside LS-DYNA is used to replicate the crushing of the composite layered shell model. Good correlation with respect to qualitative and quantitative results is obtained with this model.

6 References

[1] Heimbs, S., Strobl, F., Middendorf, P., Gardener, S., Eddington, B. and Key, J.: Crash Simulation of an F1 Racing Car Front Impact Structure, 7th European LS-DYNA Conference, 2009

[2] Hallquist, J.O.: LS-DYNA Theoretical Manual, Livermore Software Technology Corporation, Livermore, 2006

[3] Hallquist, J.O.: LS-DYNA Keyword Manual Version 971, Livermore Software Technology Corporation, Livermore, 2013

[4] Hörmann, M., and Wacker, M.: Simulation of the crash performance of crash boxes based on advanced thermoplastic composite. 5th European LS-DYNA Conference, 2007