CAE Workflow Coupling Stamping and Impact Simulations

Henry Shibayama, Rohit Ramanna, Sri Rama Murty Arepalli, Arthur Camanho

ESI South America *ESI North America* *ESI Software India* *ESI South America*

Abstract

Visual-Environment is an open and integrated user environment enabling simulation and analysis across multiple domains for efficient product engineering. A CAE workflow has been developed chaining stamping and impact simulations. The workflow originates from stamping matrix design (performing stamping simulations) to impact simulations considering the residual stress and thickness variation due to stamping process.

The objective to be achieved is the creation of a fast end-to-end workflow, aiming at accurate impact simulations while taking into account the results from manufacturing processes. So far, impact simulations are performed considering the stamping simulation results. The next step of the project is to perform welding simulations and consider its residual stresses and distortion, aiming at more accurate impact simulations, through chaining and considering the process effects coming from stamping and welding analysis as well.

Visual-Diemaker is a software tool focusing on the design of the stamping matrix, with some feasibility tools, such as tipping evaluations. Visual-Diemaker also integrated some tools for the set-up of stamping simulation. For development and evaluation of the methodology all simulations were performed using PAM-STAMP. Generated output results were M01 files (one per component), containing the residual stress and thickness variation.

In a first step Visual-Process, a mass-customization and automation tool to support the automation of CAE tasks, converts automatically M01 (PAM-STAMP) file structure to LS-DYNA® key-words respectively syntax (using ELEMENT_SHELL_THICKNESS and INITIAL_STRESS_SHELL). As a second step, the same tools allows the CAE engineer to open the LS-DYNA impact model in Visual-Environment, then setting the components to be chained with the stamping results, define the reference nodes for the construction of a reorientation matrix, and finally define a different number of integration points through the thickness. This process basically reads and converts the M01 syntax and also adds the INCLUDE_STAMPED_PART syntax into the global impact model which references the converted M01 file.

The purpose of the project is the ability of automated chaining between manufacturing and performance structural simulation in one and same environment. After achieving this development goal, further implementation and industrialization of this kind of analysis methodology into a CAE industrial department is expected as a reliable and fast way to proceed with chained analysis using different CAE solver.

Motivation

CAE Engineers are continuously looking for solutions to integrate all their product engineering projects in a single user environment to manage them most efficiently. And every product engineering team deals with questions in the virtual prototyping phase, many of these covering multiple engineering domains. The main objective of this project is to use Visual-Environment and adapt its features to allow a fast and robust (no errors) way to integrate the stamping simulations to the crash and safety simulations.

The development of this workflow, considering some process automation, was motivated by the necessity of integrate the stamping and crash simulation methodologies in the industrial world, where development cycle is getting short and numerical simulations needs to be done faster and the most accurate as possible. The workflow proposed in this paper uses some features of Visual-Environment, a numerical tool for pre and post processing of numerical simulations, developed by ESI Group [1].

Introduction

The workflow described in this paper was tested using a model developed by National Crash Analysis Center (NCAC) – Finite Element model of C1500 Pickup Truck [2]. The main objective of the project is to develop a fast and automatic way to run crash simulations considering the stamp effects done previously.

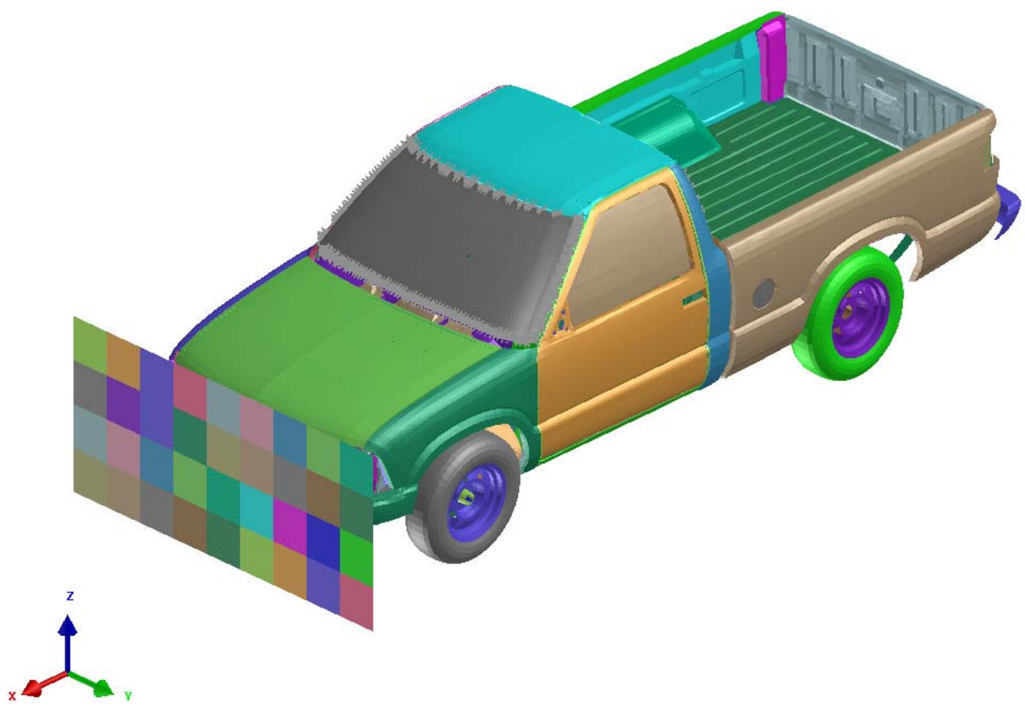

Figure 1: Finite Element Model of NCAC - C1500 Pickup Truck

The workflow developed considers the use of Visual-Environment, a pre and post processor numerical tool developed by ESI Group [1]. The workflow considers the M01 files generated by "One Step Solution" from PAM-CRASH solver, also developed by ESI Group [1]. The principle of "One Step Solution" is the "Virtual Work Energy", a function of the final and the initial shape of the component. The objective of the "One Step Solution" (also called "Inverse Method") is to find the "initial shape" considering the minimum value for "Virtual Work Energy"

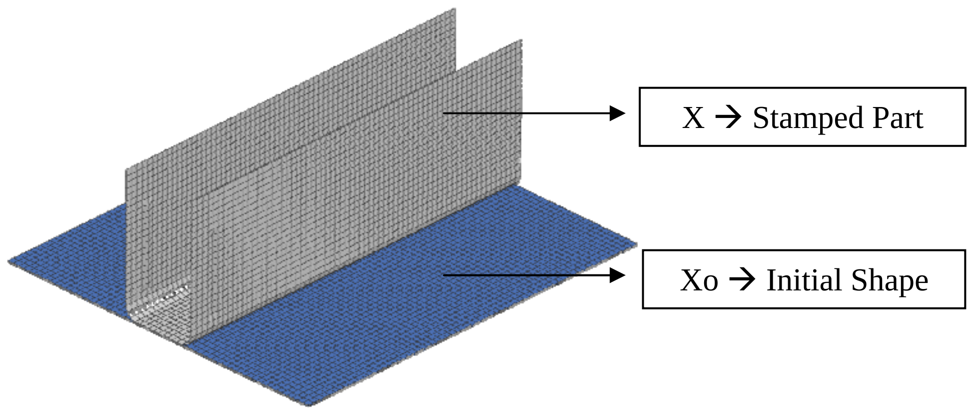

Figure 2: Initial (Blank) and Final (Stamped) Shape

Virtual Work Energy = W = Function of (Xo,X)

One Step Solution → find Xo to minimize W(Xo,X)

The basic idea of the workflow is described in the picture hereunder, where Visual-Environment is used to convert the results from PAM-CRASH solver syntax to LS-DYNA syntax. As mentioned before, the actual methodology involves the chaining of stamping results obtained from an implicit methodology, just to test the workflow. For future works, some real stamping simulations will be performed to get more accurate results

Figure 3: Stamping to Crash Workflow

CAE Workflow

Inside the Graphical User Interface (GUI) of Visual-Environment there are some tools to develop Python scripts in Visual-Process to automate any CAE workflow. The "PAM-STAMP Results to LS-DYNA" workflow was generated to automate the chaining of stamping simulation results to the impact & safety simulations.

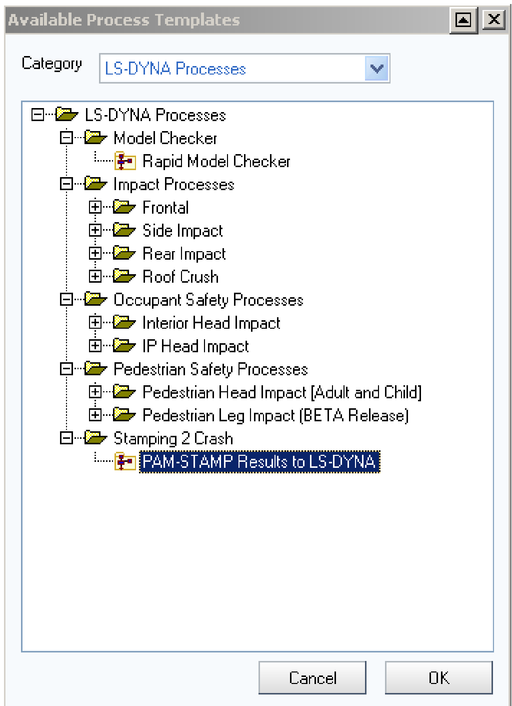

Figure 4: Stamping to Crash Workflow

If engineer has to bring stamping effects to many of the components, the automation tool provides a way to read a flat ASCII file in CSV (Comma Separated Values) format, which contains the path of the Stamp results, LS-DYNA part ID and optionally the re-orientation information.

The workflow automatically use the information in the "csv" file and develops syntaxes necessary to take into account the include files with the stamping results. The "csv" file contains the following information, organized in a sheet format (column and lines), as the table hereunder.

Table 1: Structure of "csv" file

M01 File and Directory	Part ID	N1S	N2S	N3S	N1C	N2C	N3C
C:\stamping\Part_2000001_M01	2000001	2220530	2220548	2222232	2000963	2000689	2064260
C:\stamping\Part_2000026_M01	2000026	2222507	2222739	2223544	2007785	2007729	2010437
C:\stamping\Part_2000028_M01	2000028	2228043	2227955	2227605	2011040	2010998	2013938
C:\stamping\Part_2000050_M01	2000050	2228632	2228994	2229738	2022386	2022391	2024969
C:\stamping\Part_2000052_M01	2000052	2233163	2233229	2233307	2025560	2025534	2028355

The “csv” file contains basically the directory of the M01 file to be mapped, the part ID of the component, the reference nodes of the stamping simulation (N1S, N2S and N3S) and the reference nodes of the crash simulation (N1C, N2C and N3C). These nodes are necessary to construct the reorientation matrix necessary to the mapping process into LS-DYNA solver.

The “csv” file can be created manually, using any text editor or similar, just respecting the tabulation and organization of the file. And the “csv” file can also be created by Visual-Environment, following the steps described in this paper. At the last step of the workflow proposed, there is the possibility to export and create a “csv” file with the information set along the process.

Without the “csv” file, it is possible to execute the workflow selecting all the necessary parts and reference parts using a graphical user interface developed for this task (this was the main reason for the creation of this workflow, to prevent any human error during it caused by a human error).

The first step, in the absence of “csv”, is to select the “original” crash model. The LS-DYNA model needs to be complete and ready to run, with all necessary information to be read by the solver. The process developed in Visual-Environment does not modify the main structure of the “original” crash model. It just adds the information necessary for the mapping process, basically the *INCLUDE_STAMPED_PART syntax. The flow-chart diagram below represents the workflow, step-by-step, using the features developed in Visual-Environment.

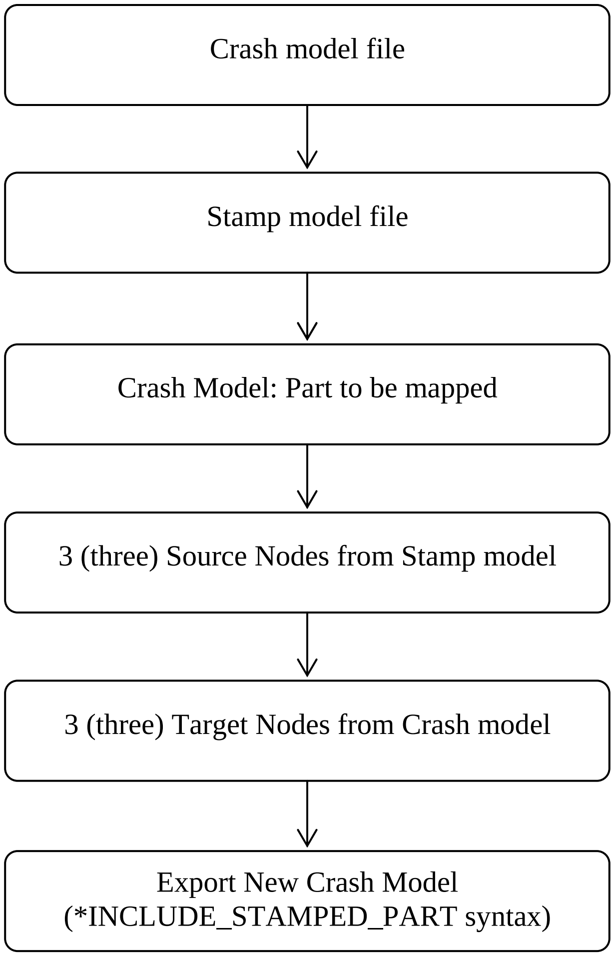

Figure 5: CAE Workflow

The next step is to define a directory to save all the converted files at the end of the process, working as a repository of all the export files created by the process.

As an option (not mandatory to set the process), it is possible to define a value for the maximum strain value and it is also possible to change the number of integration points along the thickness (as some stamping simulations considers 7 integration points).

The next step is to select a stamping model (the M01 file) and select the reference nodes (from the stamping and crash model) using the Graphical User Interface. Usually, the mesh discretization of the stamping model is different from the crash model. For a correct mapping process is necessary to select 3 nodes from the crash model as near as possible from the location of the nodes selected from the stamping model.

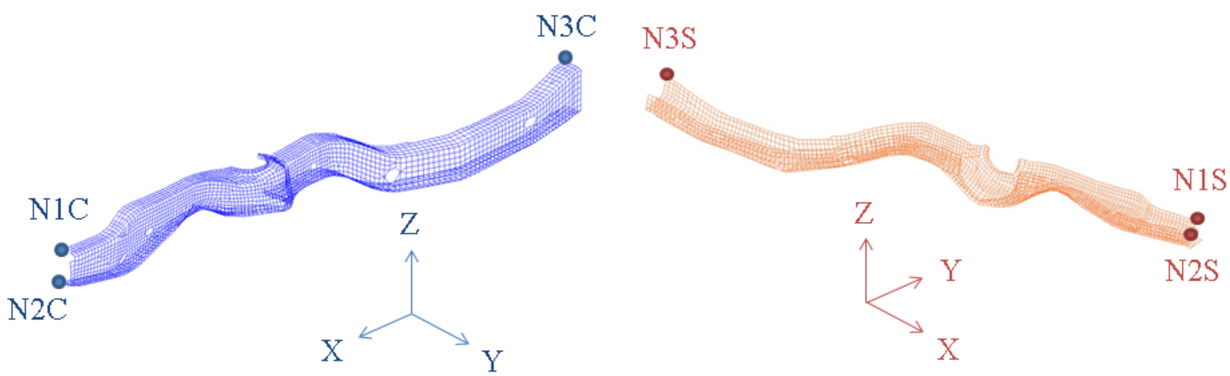

Figure 6: Reference Nodes from crash model (N1C, N2C, N3C) and stamp model (N1S, N2S, N3S)

The last step is to define the way that the files would be exported. There are 3 (three) options to define the export (1) export an input model based on the "original" crash model, just adding the "INCLUDE_STAMPED_PART" syntax; (2) add just a "INCLUDE" syntax to the "original" model referencing a new "inc" file with the "INCLUDE_STAMPED_PART" syntax; (3) export an new input model referencing the "original" crash model and containing the "INCLUDE_STAMPED_PART" syntax.

*INCLUDE_STAMPED_PART								
StampPart_2000001.inc								
$# PID	THICK	PSTRN	STRAIN	STRESS	INCOUT		RMAX	
2000001	0	0	0	0	1		0.	
$# N1S	N2S	N3S	N1C	N2C	N3C	TENSOR	THKSCL	
2220530	2220548	2222232	2000963	2000689	2064260			

Figure 7: INCLUDE_STAMPE_PART syntax

The workflow contains 3 phases: (1) Collect Inputs; (2) Map Stamping Results; (3) Export Mapping Results.

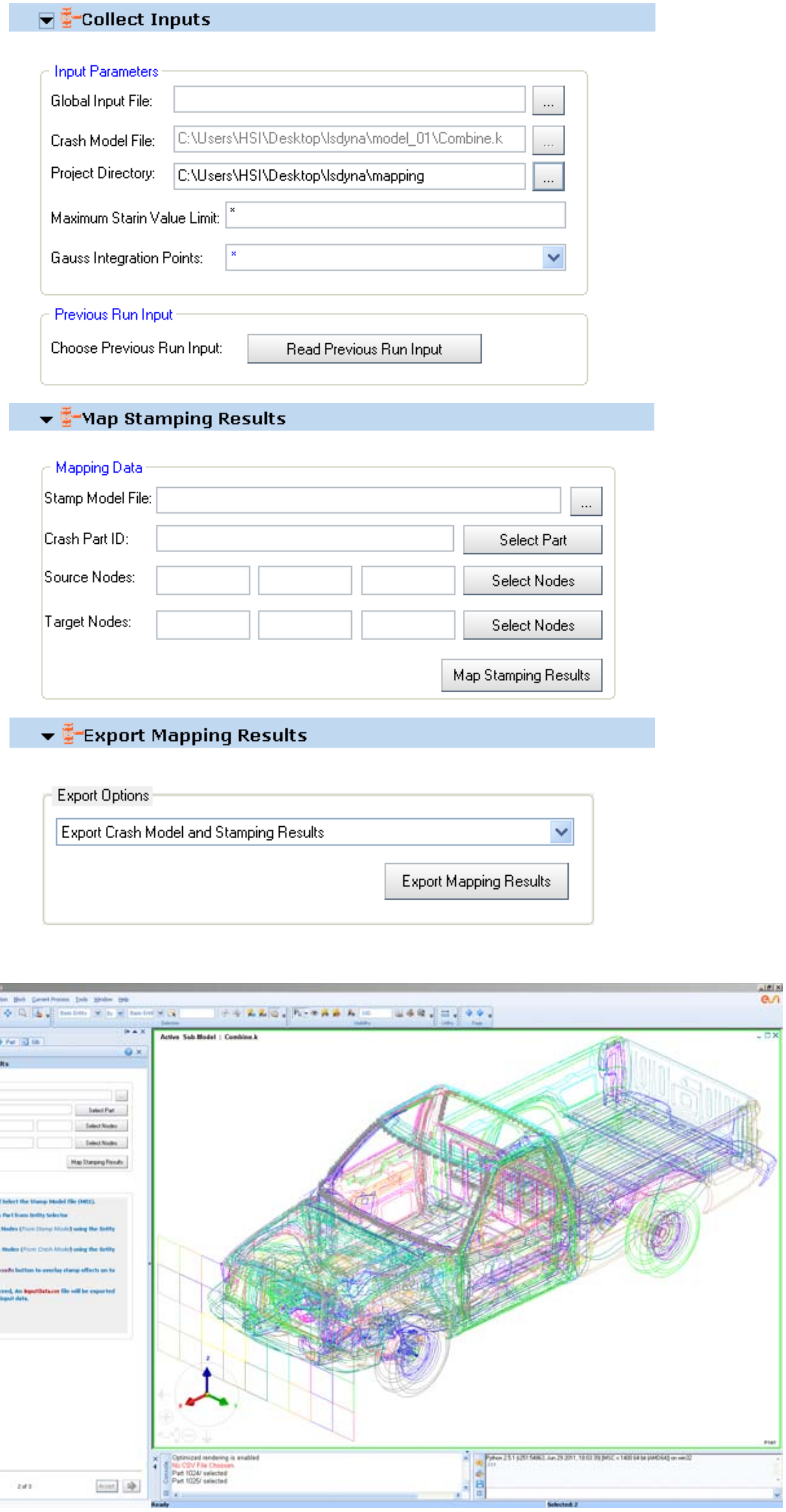

Figure 8: Graphical User Interface

Simulation Results

The project contains 2 (two) simulations. The first simulation called "original" is the original numerical model developed by NCAC. The second simulation called "mapped" is the simulation considering the stamping residual effects of some parts. In this test, just some parts are evaluated considering the stamping results. To choose which part to consider the stamping results, the "original" model was used to evaluate the internal energies of the components.

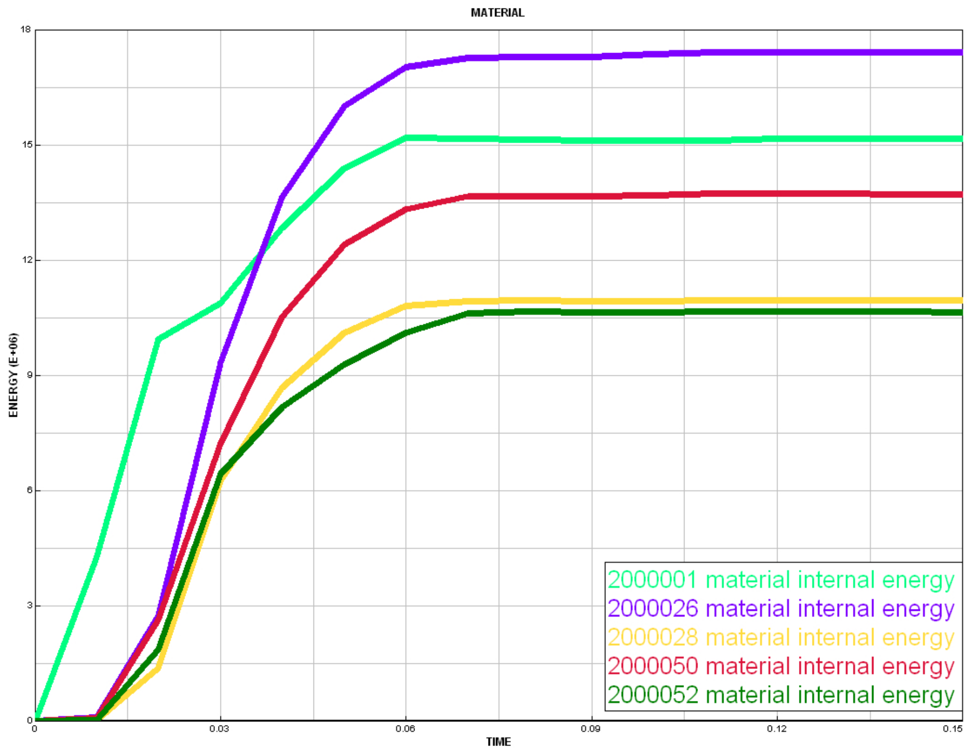

Figure 9: Stamping results (plastic strain) mapped to the crash model

6 (six) components were selected to consider its stamping results. These 6 (six) components showed the greatest value of "Internal Energy", showing a high level of deformation. This was the reason to choose these parts for the mapping process.

Table 2: Internal Energies (Original model)

Part ID	Internal Energy @ 150ms [mJ]
2000052	1,06E+07
2000028	1,09E+07
2000050	1,37E+07
2000001	1,51E+07
2000026	1,74E+07

These 6 (six) components are located at the front end of the vehicle structure. As the simulation from NCAC is a frontal impact, these are the parts with the highest level of internal energy at the end of the simulation.

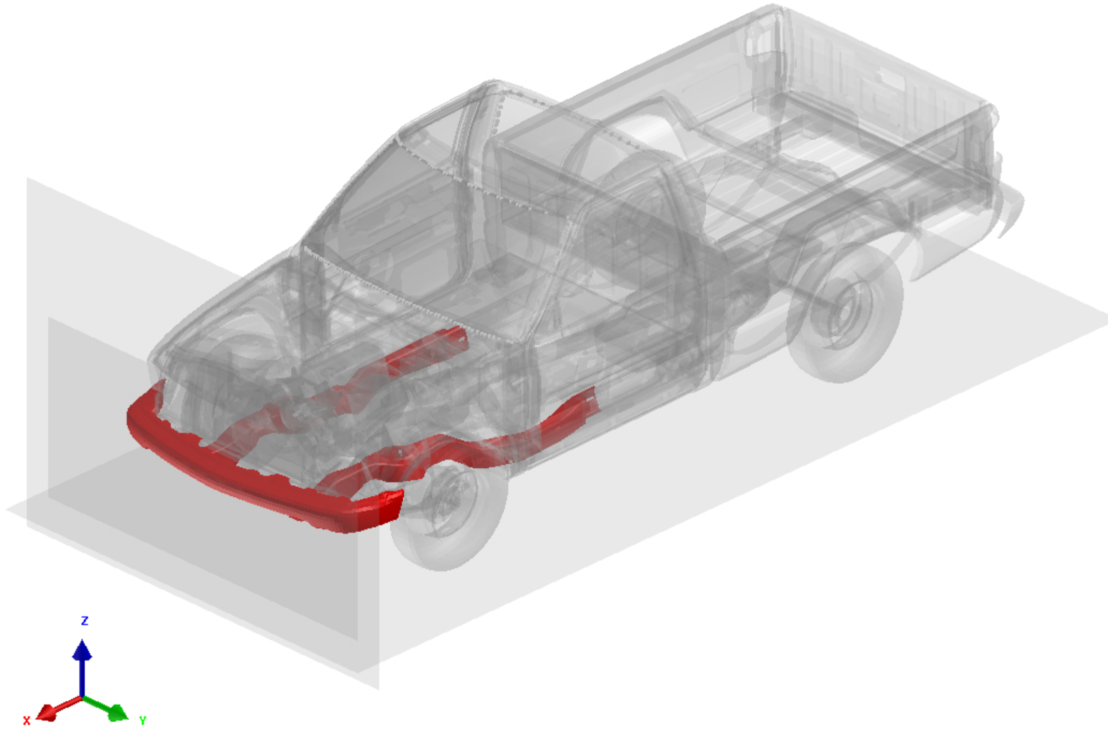

Figure 10: Parts to be mapped (Part ID: 2000001, 2000026, 2000028, 2000050, 2000052)

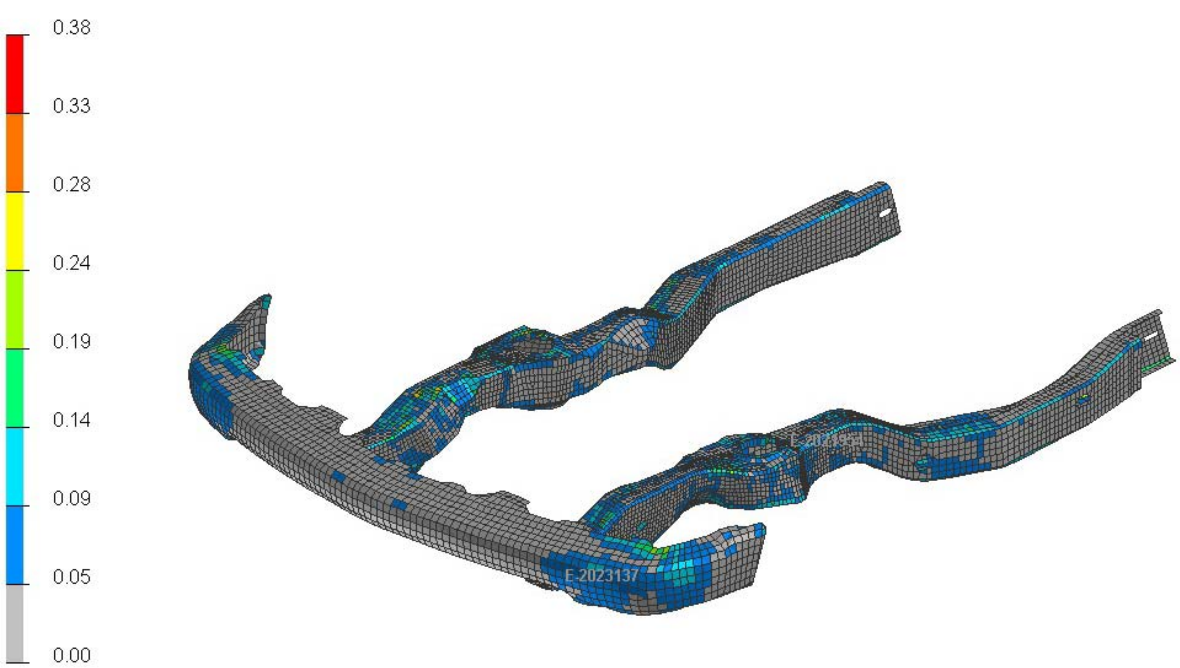

Figure 11: Stamping results (plastic strain) mapped to the “original” model

The diagram below represents the curves (internal energy) from the "original" crash model and "mapped" model with the stamping effects considered.

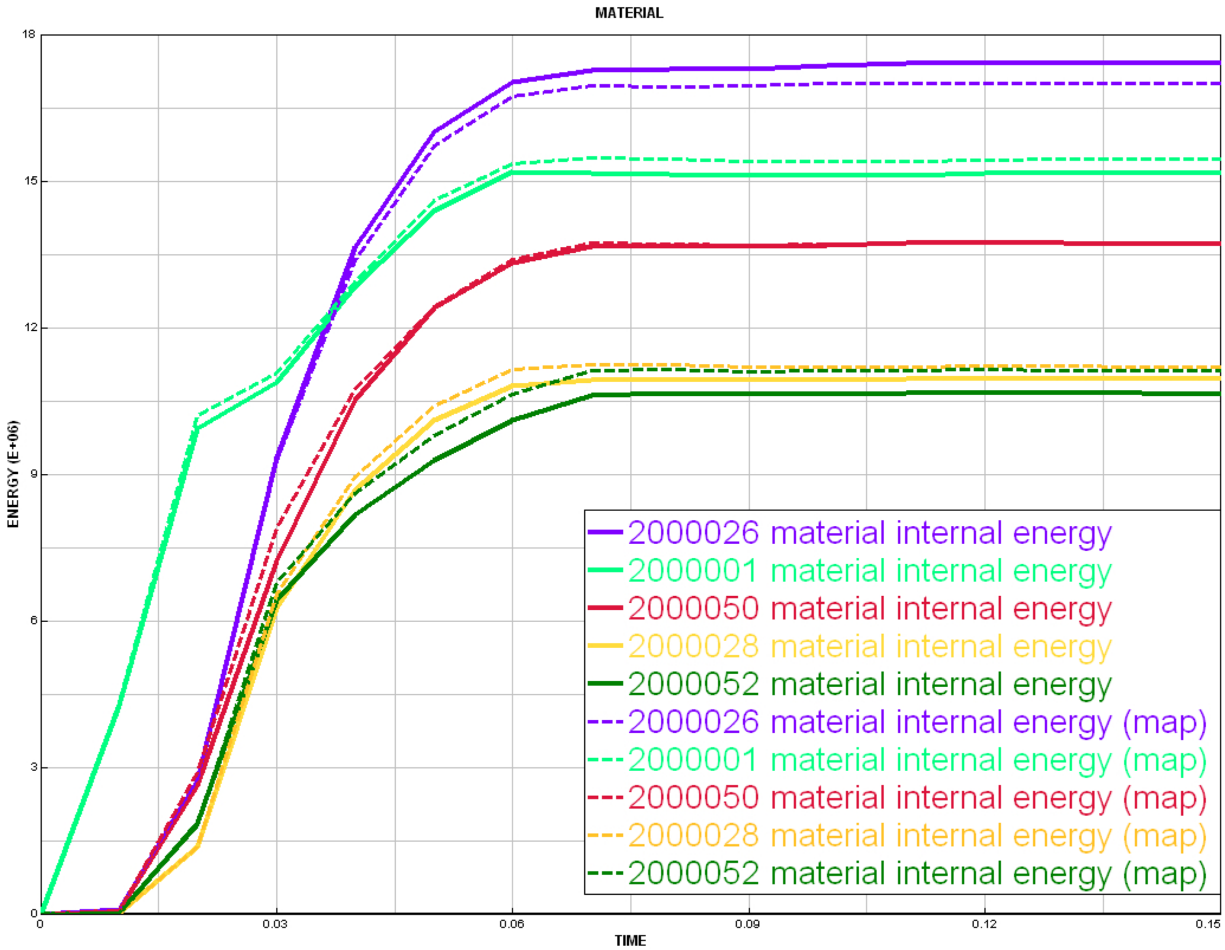

Figure 12: Stamping results (plastic strain) mapped to the crash model

Table 3: Internal Energies (Original X Mapped)

Part ID	Original Internal Energy @ 150ms [mJ]	Mapped Internal Energy @ 150ms [mJ]	Percentual Difference
2000052	1,06E+07	1,11E+07	4,48%
2000028	1,09E+07	1,12E+07	2,32%
2000050	1,37E+07	1,37E+07	0,58%
2000001	1,51E+07	1,54E+07	2,18%
2000026	1,74E+07	1,70E+07	-2,30%

The accelerations of some nodes were also evaluated to compare the results from the "original" and the "mapped" simulations. 7 (seven) points as shown in below figure and table were evaluated.

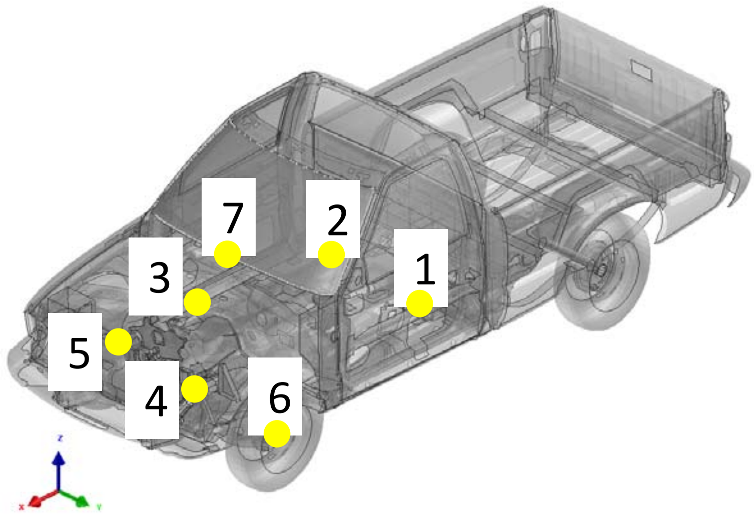

Figure 13: Position of Accelerometers

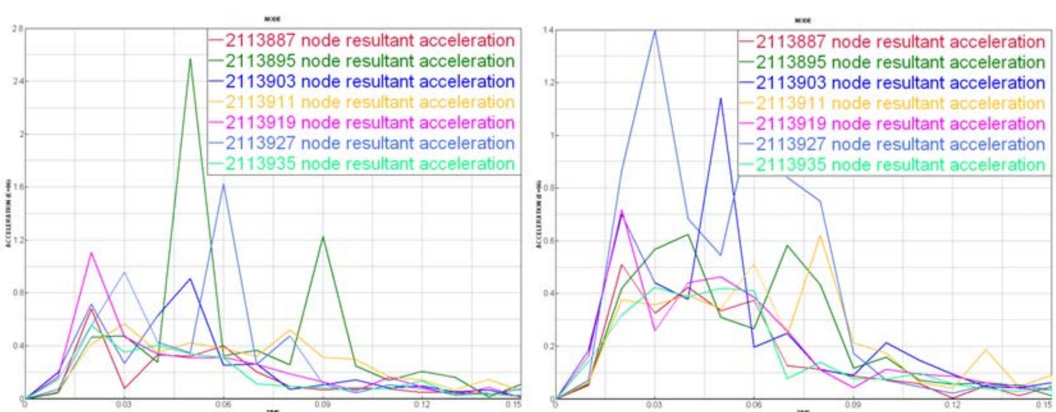

Figure 14: Acceleration Curves (Left = "original"; Right = "mapped")

Table 4: Maximum Acceleration for "original" and "mapped" models

	Location	Node ID	"original" (mm/s2 @ sec)	"mapped"(mm/s2 @ sec)
1	Left Seat	2113895	2,569,012 @ 0,05	623,977 @ 0,04
2	Right Seat	2113911	567,413 @ 0,03	621,196 @ 0,08
3	Engine Top	2113935	553,830 @ 0,02	423,241@ 0,03
4	Engine Bottom	2113887	679,049 @ 0,02	509,692 @ 0,02
5	R brake caliper	2113919	1,104,337 @ 0,02	717,697 @ 0,02
6	L brake caliper	2113903	905,887 @ 0,05	1,141,481 @ 0,05
7	IP top	2113927	1,621,748 @ 0,06	1,398,756 @ 0,03

Conclusions and Observations

The results demonstrated the importance in considering the stamping effects in the impact and safety simulations. And the CAE automated workflow presented a user-friendly workflow tool demonstrating the feasibility to use in the industrial world, minimizing the possibility of human input errors.

The objective to be achieved is the creation of a fast end-to-end workflow, aiming at accurate impact simulations while taking into account the results from manufacturing processes. So far, impact simulations are performed considering the stamping simulation results. The next step of the project is to perform welding simulations and consider its residual stresses and distortion, aiming at more accurate impact simulations, through chaining and considering the process effects coming from stamping and welding analysis as well.

References

[1] ESI Group Website <https://www.esi-group.com> Visited in March 18th 2014

[2] Finite Element Model Archive from National Crash Analysis Center (NCAC) – The George Washington University <http://www.ncac.gwu.edu/filmlibrary/index.html> Visited in March 20th 2014.