DYNAMIC ANALYSIS OF SUBSTRUCTURES WHEN TESTED IN ISOLATION USING FINITE ELEMENT METHOD

Ramin Amid, Kamran Behdinan and Hamid Ghaemi
Department of Aerospace Engineering, Ryerson University, Toronto, On, Canada, M5B2K3

Introduction

In large complex engineering systems, often only a sub- system or a small part of the system design needs to be modified to adapt or improve performance in some way. In most cases of design for crashworthiness, the sub-system behaviour is strongly coupled to that of the overall system in such a way that even small changes to the sub-system can strongly affect the behaviour of the entire structure. Since it is extremely expensive to physically fabricate and test all conceivable design alternatives, analysts use numerical techniques to predict the crash characteristics of a proposed design.

The strength of simulations lies in being able to rapidly perform important parametric studies that allow for quick elimination of unsuitable prototype designs [1] and [2]. It is a routine practice for engineers to simulate vehicle crashworthiness [3], [4] and [5]. Various modeling techniques were developed to drastically reduce the size of FE model, while preserving degree of accuracy. The reduced-order model is computationally efficient but no longer contains all the information due to the different boundary conditions [6].

This work aims at developing the technique that allows analysts to change the design of any one subsystem or a collection of subsystems of a structure in such way that structural behavior of the full system remains unchanged under impact test.

Finite Element simulation of full structure impact

The FE model of the structure was constructed from CAD data. Once the discretized model was constructed, the FE model was exported into a explicit solver (LS-DYNA) to perform impact simulation. The structure was made from mild steel AISI 1010 and AISI 1020. The appropriate elastic-plastic material properties were incorporated into the FE model, see Fig.1.

Once the impact simulation was performed, the nodal accelerations as a function of time for the nodes around the boundary of the substructure were extracted. The FFT of the acceleration data were taken and the dominant frequency for each node was calculated.

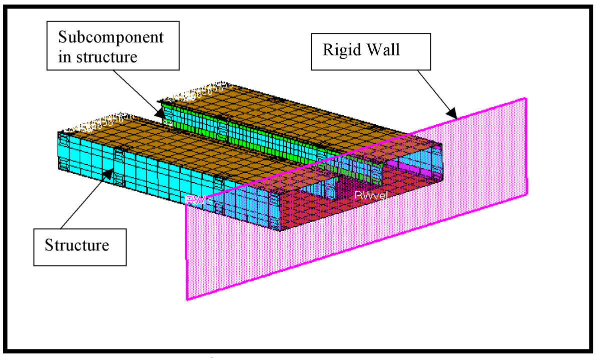

Figure 1: FE model of the model and the rigid impactor (wall)

Finite Element analysis substructure in isolation

The FE model of substructure was developed using identical mesh size as that of full structure. From the FE analysis of the full structure, the energy transferred to the sub-structure, the fundamental frequencies of the nodes around the boundary of substructure and participation of each frequency were calculated. The following parametric equation was proposed.

$$V_{Comp-impact}\sum_{i=1}^{n}\Gamma_i f_{comp-i} = V_{sub-comp}\sum_{1=1}^{n}\Gamma_i f_{sub-comp-i} \quad (1)$$

$$E = (1/2)mV^2 \quad (2)$$

Additionally, knowing the velocity obtained from equation (1), the mass of the impactor (rigid wall) for testing of substructure in isolation was determined using equation2. Table 1 tabulated the mass velocity and energy of impact of full and isolated models.

Table 1: Energy, mass and velocity for both the full model and isolated model

Simulation	Mass	Velocity	Energy
Full Model	72 kg	10.5 m/s	3970 J
Isolated Model	27.5 kg	7 m/s	696 J

Result and discussion

Here we have proposed that substructural testing in isolation can be performed with acceptable accuracy if the substructure in isolation at a selected location is excited at such frequencies that result in similar mode shapes of that of substructure in full model. The following Fig.2 depicts the effective plastic strain of the FEA results for the substructure in isolation.

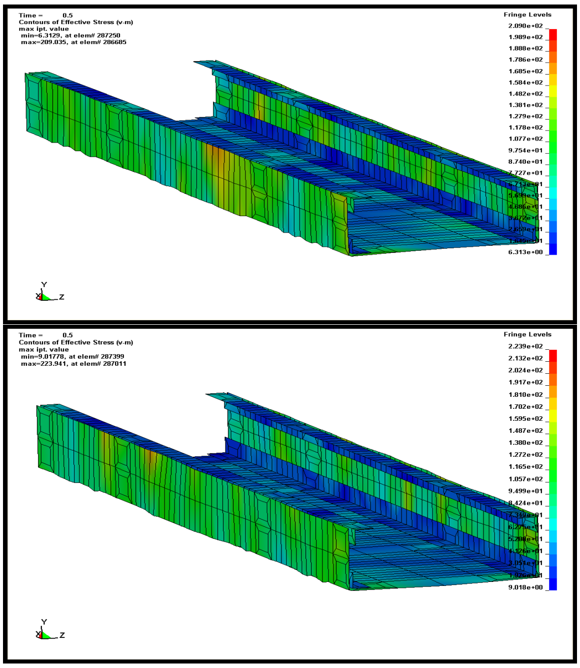

Figure 2: Contour plot of effective plastic strain a) subcomponent in isolation, b) subcomponent in full model

The full model FE impact simulation resulted in von Mises stresses of 168.5 MPa, 178.6 MPa and 148.7 MPa in front, middle and end section respectively. Equally, the FE analysis of substructure in isolation resulted in von Mises stresses of 159.5 MPa, 170.2 MPa and 140.7 MPa for front, middle and rear section respectively. Moreover, the plastic strains in the full model analysis for the front, middle and rear locations were determined to be 9050 $\mu\varepsilon$, 7820 $\mu\varepsilon$ and 8405 $\mu\varepsilon$ respectively. The corresponding plastic strains at the same locations on the sub-structure in isolation were determined to be 8859 $\mu\varepsilon$, 7710 $\mu\varepsilon$ and 8310 $\mu\varepsilon$ respectively.

The errors corresponding effective plastic strain and effective von Mises stress are determined to be between 1.4% to 2.1% and 4.7% to 5.4% respectively. Moreover, the computational time has been reduced by approximately 58% when performing FE analysis on the substructure in isolation.

Conclusions

The technique presented showed that the FE results of the full model and isolated model were in very good agreement signifying its applications and potentials. Additionally, since the method relies on impacting the substructure in isolation with appropriate mass and velocity, it can be applied to experimental setup. This in effect has a potential to reduce the time and cost of redesigning any one part of a vehicle, whether making a wheelchair accessible vehicle or reinforcing a weak part without changing the original crash characteristic of the vehicle.

References

[1] Thacker, J.G., Reagan, S.W., Pellettiere, J.A., Pilkey, W.D., Crandall, J.R., and Sieveka, E.M., 1998. *Experiences during development of a dynamic crash response automobile model*, *Finite Element in Analysis and Design*, Vol. 30, pp.279-295.

[2] Kan, C.D., Tamborra, N., and Fein, O., 1999. *Development of finite element model of a small-size vehicle for crashworthiness applications*, Proceedings of 32nd International Symposium on Automotive Technology and Automation, Vienna, Austria, pp.319-326.

[3] Pyttel, T., Floss, A., Thibaud, C., Goertz, C., 2007. *Realistic simulation models for airbags and humans-new possibilities and limits of FE simulation*, International Journal of Crashworthiness, Vol. 12, No. 5, pp. 481-492.

[4] Markiewicz, E., Marchand, M., Ducrocq, P., and Drazetic, P., 2001. *Evaluation of different simplified crash model: Application to the under-frame of a railway driver's cab*, International Journal of Vehicle Design, Vol. 26, pp. 187-203.

[5] Pipkorn, B., 2002. *A mathematical hybrid model for evaluating vehicle performance in car-to-car side impacts*, International Journal of Vehicle Design, Vol. 2, pp. 48-69.

[6] Mkrtchyan, L., Sahr, C.H., Berger, L., and Maier, M., 2008. *Interactions among structural components during complex impact events*, International Journal of Crashworthiness, Vol. 13, No. 1, pp. 9-24.