Computational time reduction in Chassis FEA Structural simulations using FEA-Superelement

Selvamanikandan.M[1], Venkatesan.S[2]

[1]P.G. Scholar, Department of Mechanical Engineering,
Vinayaka Mission's Kirupananda Variyar Engineering College, Salem, Tamil Nadu, India
selvamanikandan@vmkvec.edu.in

[2]Professor, Department of Mechanical Engineering,
Vinayaka Mission's Kirupananda Variyar Engineering College, Salem, Tamil Nadu, India
thasenvenkat09@gmail.com

Abstract: ***Due to continuous improvements in computer performance, finite element models continue to grow. This means that finite element analysis (FEA) solutions can get in some hours of computation time.***

Chassis development FEA simulations have to map the entire vehicle environment in order to perform well in static and dynamic analyzes. This environmental component adds rigidity to the chassis members, especially for flexural and torsional loads.

The total number of elements has increased as environmental components have been added to the chassis system. However, these additional components are not included in the study component. Therefore, if the number of elements in the FE model increases the computation time, then the environment components in the simulation must be rendered without adding elements to the FE model.

FEA solvers, efficiently solves the large systems of Matrix equations using sparse matrix algorithms; but still FEA–Superelements offers greater efficiency in the computational time without much compromise in the FEA results

Superelement is a finite element method that defines a new type of finite element by grouping the Stiffness matrix, Mass matrix, Inertia matrix of set of finite elements and processing a group of finite elements.

In this research study, complete data of the vehicle upper body environment has converted into superelements, which shortens the computation time and examines the deviation of the result from the complete element represented by the Normal FE element.

Keywords: FEA, Chassis load case, superelements, ls-dyna superelement, Chassis Simulations, Implicit-Modes, Element-direct-Matrix-Input**.**

1. Introduction

Superelements are part of the problem and can be resolved locally before they are implemented into a global problem. Superelements, an add-on to FEA, play an important role in solving very large and complex finite element models by breaking larger structures into a series of smaller substructures called superelements.
FEA - Super Element can be used with all FEA analysis functions. It is particularly efficient for comprehensive system analysis of entire aircraft, vehicles and ships Perform incremental and partial assembly solutions. Using superelements to undermine problems can simplify the division of labor and eliminate computer memory limitations. Superelement accelerates innovation by improving the solution's efficiency and reduces the risk of product design by increasing the number of iterations. Enables enterprise-wide collaboration by combining models from multiple sources (internal and external) and masking unique data. Maximize the output of valuable computing resources. Reduce the amount of computer memory and space required.
We can transfer model components from one company to another (in the form of a superelement matrix) and hide the design details so that superelements can also be used in collaborative design (stiffness terminology) only).

2. Methodology

FEA simulation plays an important role to complete a product with short duration and without compromise in the quality of product.
LS Dyna is a Nonlinear FEA solver, which is supporting super elements to reduce the computational time.
The FEA solver, solves the numerical problems with implicit codes, in the simplest form of {F} = [K]. {U}
[K] is the stiffness matrix, which represents the structural stiffness of the assemblies.
In LS Dyna Implicit solving methodology has used to convert the Normal FE elements into the superelements.
In order to use this feature, an implicit analysis must be requested using IMFLAG = 1 on *CONTROL_IMPLICIT_GENERAL, and a non-zero termination time should be on *CONTROL_TERMINATION. To get best accuracy results, double precision version of LS-DYNA is preferable. To

eliminate static rigid body motion, need to apply a sufficient number of constraints to the model. Ls Dyna will write the Mass matrix, Stiffness matrix, Damping Matrix information for the assembly, Inertia Matrix information once superelement conversion done

.

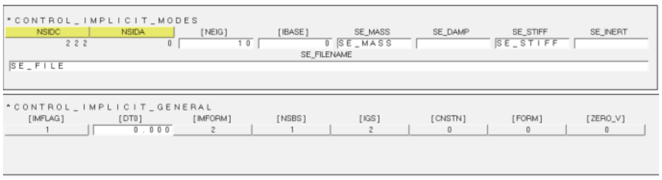

Figure 1.1 Input Control Cards for Superelement Conversion

The user can create the superelement representation of the reduced model by specifying the SE_MASS, SE_DAMP, SE_STIFF, SE_INERT and SE_FILENAME fields. The inertia matrix is necessary if body forces, e.g., gravity loads, are applied to the superelement.

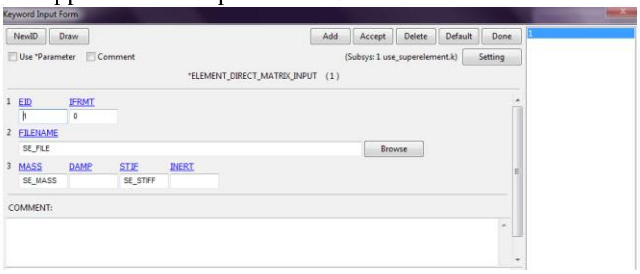

Figure 1.2 Control Card for represent the superelement results in the reduced FE mode

The BINARY keyword option can be used to create a binary representation for the superelement which can be used with *ELEMENT_DIRECT_MATRIX_INPUT_BINARY to reduce the file size.

3. FEA Chassis Analysis

3.1 Bending Load Case

The bending stiffness of a chassis extracted by the bending analysis, in this load case the chassis are constrained at wheel mounting locations.

Also, upper body will add some more resistance against bending of chassis rails.

Three different simulations has performed to find the performance of the super element.

1. Chassis Bending Analysis with Full meshed upper body
2. Chassis Bending with Superelement of upper body
3. Chassis Bending without upper body

These study FEA results are shown in fig 3.1.1 and 3.1.2

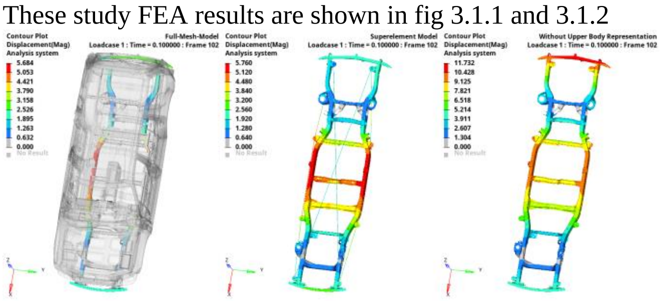

Fig. 3.1.1 bending analysis Displacement contours comparison (Full mesh vs superelement vs without upper body)

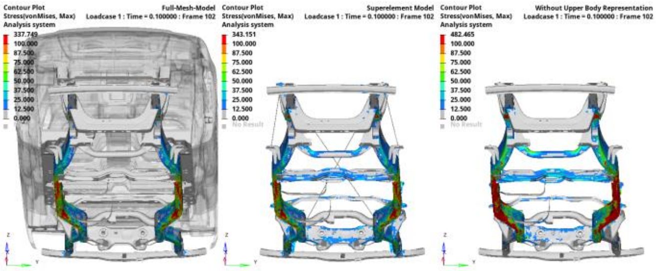

Fig. 3.1.2 bending analysis Von Mises Stress contours comparison (Full mesh vs superelement vs without upper body)

The relative displacement graph plotted between these three FEA models, It shown in the fig3.1.3,

From the graph Superelement method results are very closely matching with the Upper body meshed model.

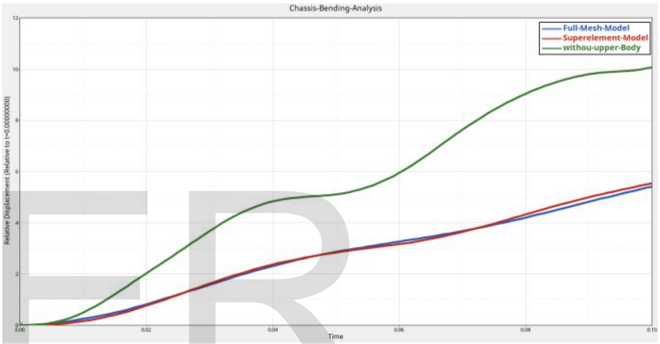

Fig. 3.1.3 bending analysis relative displacement curve comparison (Full mesh vs superelement vs without upper body)

3.2 Twisting Load Case

The twisting stiffness of a chassis extracted by the Chassis torsional analysis, in this load case the chassis are constrained at rear wheel mounting locations and center point of front axial member location in all directions except X Rotational degrees of freedom.

Also, upper body will add some more resistance against twisting of chassis rails.

Three different simulations has performed to find the performance of the super element.

1. Chassis Twisting Analysis with Full meshed upper body
2. Chassis Twisting with Superelement of upper body
3. Chassis twisting without upper body

These study FEA results are shown in the fig 3.2.1 and 3.2.1.

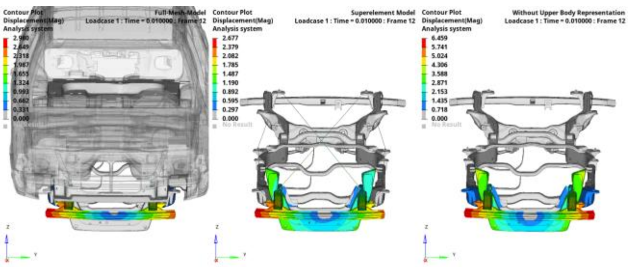

Fig 3.2.1 Twisting analysis Displacement contours comparison (Full mesh vs superelement vs without upper body)

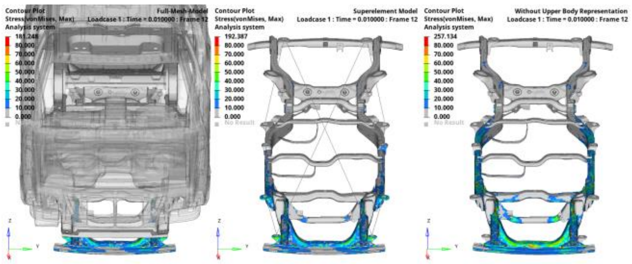

Fig 3.2.2 Twisting analysis Von Misses Stress contours comparison (Full mesh vs superelement vs without upper body)

The relative displacement graph plotted between these three FEA models, it shown in the fig3.2.3,
From the graph Superelement method results are closely matching with the Upper body meshed model.

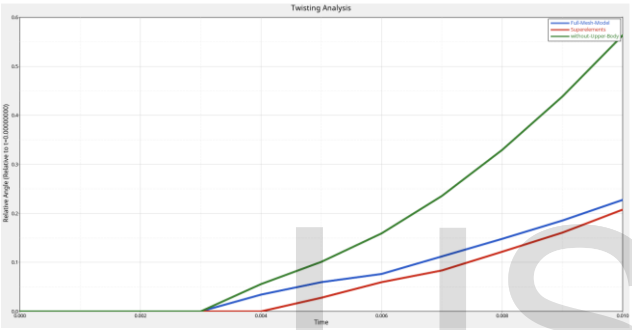

Fig 3.2.3 Twisting analysis relative displacement curve comparison (Full mesh vs superelement vs without upper body)

4. Conclusion

The research study of computational time reduction by superelements has concluded by results shown in the table 4.1.

Table 4.1 Results summary for Superelement study

	Bending Load case		
	Full Mesh Model	Superelement Model	Without Upper Body
Displacement (mm)	5.68	5.76	11.73
Stress (MPa)	337	343	482
Run Time (minutes)	195	15	13

	Twisting Load case		
	Full Mesh Model	Superelement Model	Without Upper Body
Displacement (mm)	2.9	2.6	6.4
Stress (MPa)	181	192	257
Run Time (minutes)	8.01	0.86	0.78

By using the superelements in the quasi static simulations, it gives almost closer results with the Full mesh model, and large time reduction in the computational time. These run has performed in the same no CPU.
The super element supports the linear behavior of the assembly, so if any large nonlinear deformation observed in the upper body during any of the chassis load cases, then there will be some considerable deviations in the superelement method results.

References

[1] Nitin S. Gokhale. (2008). "Practical Finite Element Analysis". Finite To Infinite. ISBN: 8190619500
[2] Julian Happian-Smith (2001). "An Introduction to Modern Vehicle Design" Elsevier ISBN: 0750650443
[3] Cicek Karaoglu, N. Sefa Kuralay (2001) 'Stress analysis of a truck chassis with riveted joints'. Elsevier: V.38 (2002) 1115–1130
[4] H.J. Beermann, (1989) "The Analysis of Commercial Vehicle Structures", Mechanical Engineering Publication Limited, Vol.5 PP. 28-46.
[5] Zu-Qing Qu, Panneer Selvam (2000) 'Dynamic Superelement Modeling Method for Compound Dynamic Systems' AIAA. Vol: 38 Issue: 06.
[6] Zu-Qing Qu (2004) 'Model Reduction on Component Level—Superelement Modeling Technique' Springer-Verlag London pp 255-294.
[7] Dupont Pillet S. Cogan (2008) 'Superelement verification in complex structural models' IOS Press and the authors. ISSN 1070-9622/08.
[8] W Kuntjoro J Mahmuda (2017) 'Wing Structure Static Analysis using Superelement' Elsevier PP: 1877-7058.

Author Profile

Selvamanikandan received the B.E, degree in Mechanical Engineering from Government College of Engineering, Salem in 2016. During 2016-2019, he worked as CAE Project Engineer in Hepatica Technologies Pvt Ltd., Bangalore. Currently pursuing his Masters in CAD from Vinayaka Mission's Kirupananda Variyar Engineering College Salem, Tamil Nadu. Also he is working as CAE Engineer in Hinduja Technologies Chennai.