Civil Structures, Co-Simulation Architecture of ANSYS and MSC.ADAMS in Bridge Construction

Kotharu Srinivasa Rao,

Civil Engineering,
Sir c r reddy polytechnic,Eluru,India

Abstract: The blend of ANSYS and ADAMS can improve the accuracy of framework reproduction. In this paper, initial, a link stayed space support, comprising of stay links, segments, and the space bracket, is worked in ANSYS. Second, the modular impartial record (mnf) required in ADAMS is produced by executing the large scale order "ADAMS" and afterward brought into ADAMS programming. Third, the adaptable body in the mnf document is associated with the inflexible body worked in ADAMS through the outside hubs determined in ANSYS, and afterward the dynamic investigation can be done. Accordingly, the heap record required by ANSYS could be acquired by tapping on the menu "Document Export". At long last, the heap record is brought into ANSYS programming by the APDL language and the static examinations relating to various burden steps are completed. The reproduction examination shows a decent presentation and a few outcomes we are keen on are acquired.

Index Terms - *simulation; flexible body; truss; ANSYS; ADAMS*

I. INTRODUCTION

Considering the effect of flexible bodies can improve the simulation precision greatly, and ANSYS can create the flexible body conveniently. Meanwhile, the analysis results of ADAMS can also provide complicated boundary conditions for ANSYS. Therefore, through the interface between ANSYS and ADAMS, the effect of flexible bodies can be taken into account easily and results of stress analyses base on the accurate dynamic analysis will be obtained.

Meo and Zumpano [1] calculated the vibration properties by performing a modal analysis using the finite element analysis code and pre/post-processor system ANSYS. Kouroussis et al. [2] used ADAMS software to simulate the dynamic behavior of a competition kart, whose chassis frame was considered as a flexible body. Kmiec and McConville [3] developed an interface between ANSYS and ADAMS to transfer the modal information between the two tools. Jang et al. [4] studied the von Misses stresses on the boundary of the fillet using ANSYS.

II. DATA EXCHANGE BETWEEN ANSYS AND ADAMS

A. Modal neutral file (mnf) generated by ANSYS

The modal neutral file needed by ADAMS can be obtained by executing the macro command "ADAMS, _NMODES" in ANSYS, in which all necessary information of flexible bodies, such as mass, moment of inertia, frequency, mode shape, is included. First of all, a flexible body needs to be built in ANSYS and specified appropriate element types. Secondly, several nodes must be selected as external joints at the place where the flexible body and rigid body are joined together. Thirdly, the jobname.mnf file can be generated by the macro command in ANSYS. During the file-generating process, a few remarks must be made as follows:

a. The command "/UNITS" should be carried out first to specify the unit system used in analysis before the macro command "ADAMS" is executed. Because the MSC.ADAMS software can deal with different unit systems, the jobname.mnf file must contain the unit information used in ANSYS.

b. The external node is used to connect the flexible body with rigid body in the kinematics analysis in ADAMS. Generally speaking, only one external node is needed at the joint position. If there is not any node at the connecting part, we should create one at the place. The external node and its surrounding nodes of the flexible body are often defined as rigid regions.

c. Only nodes that will be used as external joints can be selected before the macro command "ADAMS" is executed, because all selected nodes will be regarded as external joints. In this case, the process is absolutely necessary.

B. Load file generated by ADAMS

First, the rigid body should be built in ADAMS software and the flexible body needs to be imported into ADAMS in the form of modal neutral file. Second, after the connection way is specified, the dynamic analysis can be carried out by applying the real load. Third, when the analysis is over, the load file needed in ANSYS can be gained, in which motion states and load information of the flexible body corresponding to different time during the movement are included.

Next, open the model file (jobname.db) of the flexible body in ANSYS and determine the reasonable boundary conditions. Then the load file can be called by executing the command "/INPUT, filename, lod" for analysis. After the solution is done, some results we are interested in, such as stress distribution, deformation of the flexible body, can be obtained.

III.   NUMERICAL EXAMPLE

A cable-stayed space truss, consisting of stay cables, columns and space truss, is built in ANSYS. The initial geometry of the structure is shown in Fig.1, and the material properties are given in Table **1** .

Fig.1 Cable-stayed space truss model built in ANSYS

--

As shown in Fig.1, the flexible body has been built in ANSYS. Next, the modal neutral file will be obtained by executing the macro command "ADAMS, _NMODES". Also you can create the file by the ANSYS Graphical User Interface (GUI) mode, that is "Main Menu>Solution>ADAMS Connection>Export to ADAMS". Then the modal neutral file is introduced into ADAMS through the ADAMS/Flex module and a modal analysis could be performed to identify the natural frequencies of the cable-stayed space truss. The 10 mode shapes gained by ADAMS are shown from Fig.2 to Fig.3.

TABLE I.   MATERIAL PROPERTIES FOR CABLE-STAYED SPACE TRUSS

Title	Element type	E ($\times 10^5 MPa$)	DENS (Kg/m^3)	Area ($\times 10^{-6} m^2$)
Upper chord	Link8	2.0	7800	11799.82
Bottom chord	Link8	2.0	7800	11799.82
Web member	Link8	2.0	7800	3795.04
Stay cable	Link10	1.7	7850	1300
Column	Solid65	0.335	3000	--

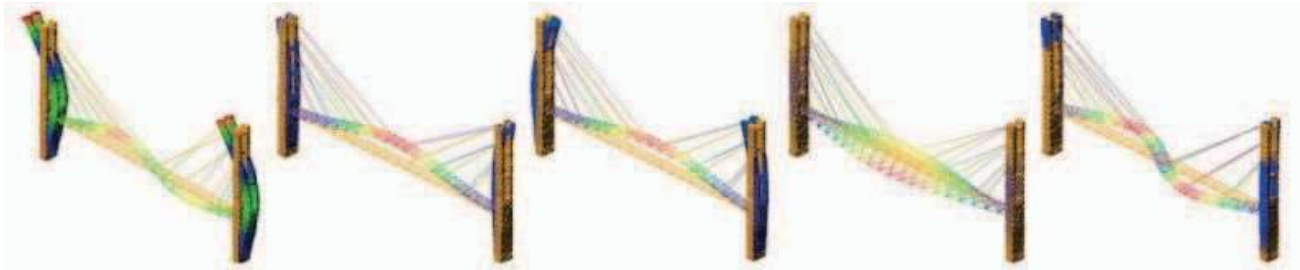Fig.2 Mode shapes from 1st order to 5th order in ADAMS.

Fig.3 Mode shapes from 6th order to 10th order in ADAMS.

The load is applied on the nodes of upper chords of the cable-stayed space truss in the form of Fourier sine series, whose format in ADAMS is "ABS(FORSIN(TIME,-0.25,PI,1000,30,40,50))". The dynamic load curve is shown in Fig.4.
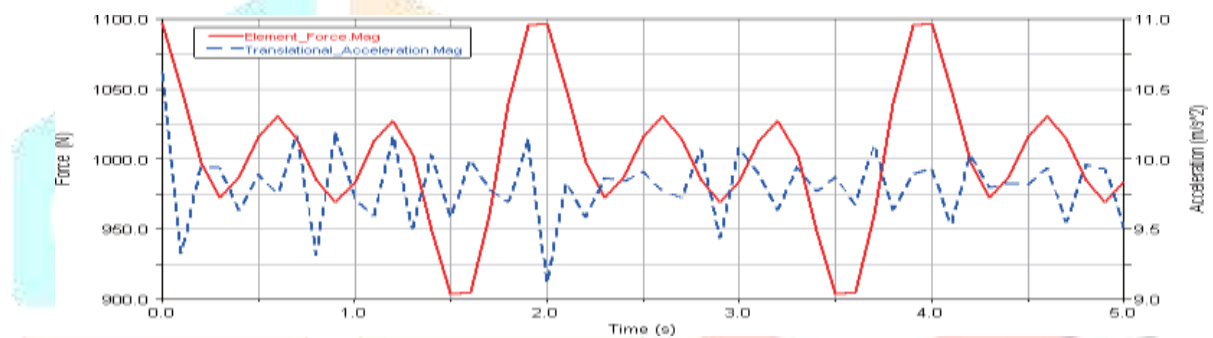

Fig.4 Dynamic load curve (red) and acceleration curve (blue)

Since the load has been applied to the structure, the dynamic analysis can be carried out and the load file will be gained subsequently by clicking on the menu "File>Export" in MSC.ADAMS. Then the load file can be introduced into ANSYS by executing the command "/INPUT, adamsload, lod", in which adamsload is the name of load file and lod is the file extension. Finally, the static analysis corresponding to different time can be carried out and some information we are interested in could be obtained by the General Postprocessor (POST1). Many results are displayed graphically from Fig.5 to Fig.7.
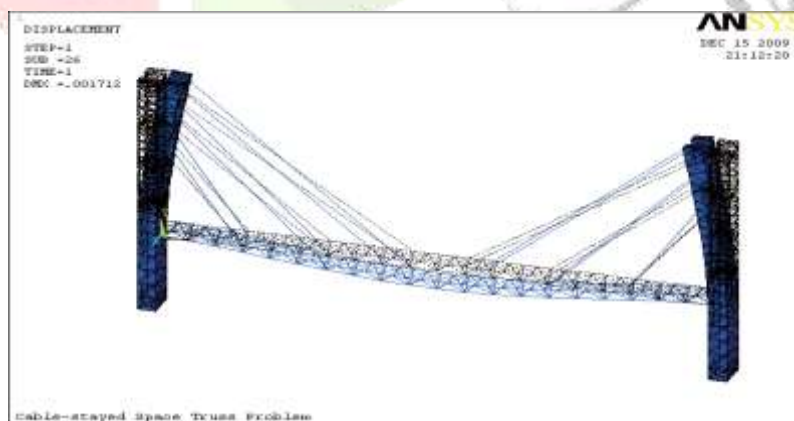

Fig.5 Deformed shape of the cable-stayed space truss
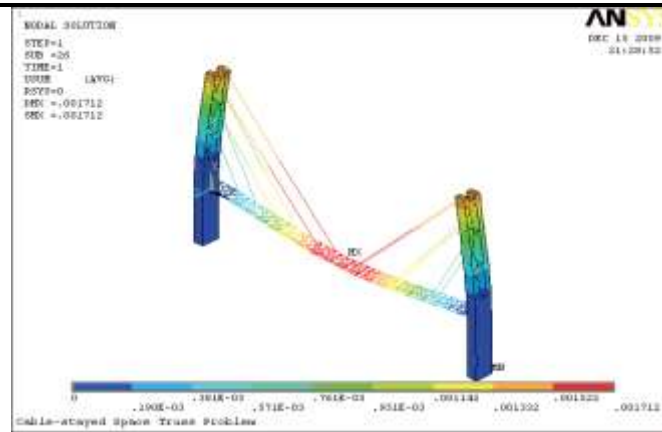
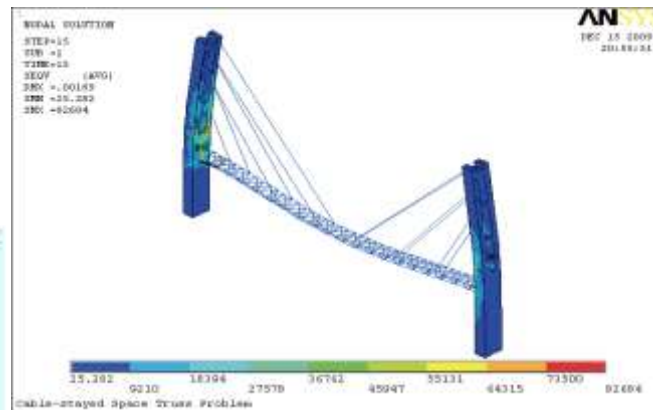Fig.6 Contour display of the total displacement

Fig.7 Contour display of the equivalent stress

In this paper, 15 load steps are specified for the static analysis of the cable-stayed space truss. The load spectrum is shown in Fig.4. The solving results of each step can be obtained through the General Postprocessor (POST1). Results of vertical displacements of nodes and axial stress of stayed cables are given in Table Ċ and Table ċ respectively. It can be seen clearly from Fig.8 and Fig.9 that the changing trends of node displacements and axial stresses show a good agreement with expected results.

TABLE II.   VERTICAL DISPLACEMENTS OF NODES UNDER EACH LOAD STEP

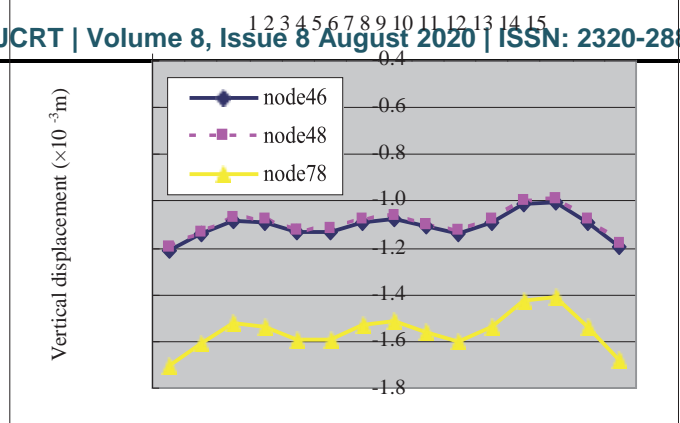| Load step		Node48	Node78
	$(h10^{-3}m)$	$(h10^{-3}m)$	$(h10^{-3}m)$
1	-1.21	-1.20	-1.71
2	-1.14	-1.13	-1.61
3	-1.08	-1.07	-1.52
4	-1.09	-1.08	-1.54
5	-1.13	-1.12	-1.59
6	-1.13	-1.12	-1.59
7	-1.09	-1.08	-1.53
8	-1.08	-1.06	-1.51
9	-1.11	-1.10	-1.56
10	-1.14	-1.12	-1.60
11	-1.09	-1.08	-1.54
12	-1.01	-1.00	-1.42
13	-1.00	-0.99	-1.41
14	-1.09	-1.08	-1.54
15	-1.20	-1.18	-1.68

Fig.8 Comparison of vertical displacements of nodes

TABLE III.   AXIAL STRESS OF STAYED CABLES UNDER EACH LOAD STEP

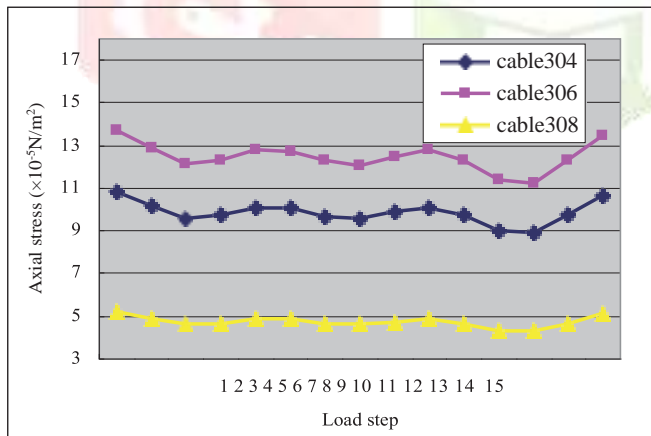| Load step	Cable304 ($\hbar 10^5 N/m^2$)	Cable306 ($\hbar 10^5 N/m^2$)	Cable308 ($\hbar 10^5 N/m^2$)
1	10.811	13.676	5.2089
2	10.175	12.871	4.9025
3	9.6264	12.177	4.6380
4	9.7265	12.303	4.6863
5	10.099	12.774	4.8656
6	10.087	12.760	4.8600
7	9.7110	12.284	4.6788
8	9.5715	12.107	4.6115
9	9.8943	12.516	4.7671
10	10.128	12.811	4.8797
11	9.7249	12.301	4.6854
12	9.0198	11.409	4.3457
13	8.9189	11.282	4.2972
14	9.7254	12.302	4.6857
15	10.668	13.495	5.1400

Fig.9 Comparison of axial stress of stayed cables

IV. CONCLUSIONS

The co-simulation of ANSYS and ADAMS is executed for the cable-stayed space truss problem in this paper. The modal neutral file and the load file, generated in ANSYS and ADAMS respectively, play a great role during the whole simulation process. The modal analysis is carried out in ANSYS and 10 mode shapes are extracted. At the same time, the modal analysis is executed in ADAMS after the mnf file is imported into it. The comparison of results shows a good performance in displaying natural frequencies and vibration shapes. The forces are applied to the cable-stayed space truss in the form of load file. The static analyses corresponding to different load steps show that the maximum displacement occurs in the mid-span and there exists stress concentration phenomenon at the space truss-column connections. At last, it can be seen from the charts mentioned in the previous section that the changing trends of displacements and stresses show a good agreement with expected results.

REFERENCE

[1] M. Meo, G. Zumpano, "On the optimal sensor placement techniques for a bridge structure," Engineering Structures, 2005, Vol.27, pp.1488-1497.

[2] G. Kouroussis, O. Verlinden, O. Champenois, C. Conti, "Multi-body model of a competition kart accounting for the chassis flexibility," Proceeding of the 7th National Congress on Theoretical and Applied Mechanics, Mons (Belgium), May 2006, National Committee for Theoretical and Applied Mechanics.

[3] D. Kmiec, J. McConville, "Structural flexibility influence using ADAMS/FEA," Proceedings of the 1996 International ADAMS User Conference, Michigan (USA), 1996.

[4] G. W. Jang, Y. Kim, K. Choi, "Remesh-free shape optimization using the wavelet-Galerkin method," International Journal of Solids and Structures, 2004, Vol.41, pp.22-23.

[5] S. Reh, J. Beley, S. Mukherjee, E. Khor, "Probabilistic finite element analysis using ANSYS," Structural Safety, 2006, Vol.28, pp.17-43.

[6] P. Yih, K. Satyan, J. Gerdes, "Safety Performance and Robustness of Heavy Vehicle AVCS," California Partners for Advanced Transit and Highways (PATH), 2005.

[7] D. Negrut, R. Rampalli, G. Ottarsson, A. Sajdak, "On the Use of the HHT Method in the Context of 3 Differential Algebraic Equations of Multi-body Dynamics," 2005 ASME International Design Engineering Technical Conferences & Computers and Information in Engineering Conference, Long Beach, California USA, January 2005.