Applications of Finite Element Analysis Using Examples from Small and Large Deformation Mechanics

Raghu Echempati, Ph.D., P.E.
Professor of Mechanical Engineering and Director of M. E. Graduate Programs
Department of Mechanical Engineering
Kettering University
Flint, MI 48504 (U.S.A.)

Abstract

In this lecture, two separate studies will be reported. Both are conducted from an industrial and educational perspective. Several educational CAE tutorials have been developed using the available software at Kettering University. These tutorials are believed to be unique in the sense that similar tutorials using the basic theory are not to be found in the literature.

The first part of this lecture is a work in progress that discusses the possibility of integrating kinematics, dynamics and finite element analysis of mechanical assemblies and mechanisms. The main emphasis is on integrating theory with advanced CAE tools. The so called multi-body analysis is usually taught at the graduate level at several schools. Also, many industrial establishments use multi-body dynamics on a regular basis to analyze complicated 2D and 3D mechanisms. MSC/ADAMS or ANSYS products are often used for this purpose. From a teaching perspective, one of the main reasons for attempting to integrate two course topics is to give undergraduate students experience in how to better design engineering products. Several industrial applications of this concept can be realized, which include a piston cylinder or an over-head valve linkage mechanism of a conventional automobile engine or wind shield wiper mechanism. Applications in bio-engineering and other high-speed machinery can also be mentioned.

Flexible body (mechanism) theory and analysis is difficult to teach at the undergraduate or even at the graduate levels. Numerous studies were reported in the past by analyzing the ‘instantaneous structure’ of the mechanism in different orientations to identify the most critical orientation(s) of the mechanism and the high stress or high deflection locations in the members of the flexible mechanisms. Several special purpose computer programs have also been written by different academic and industry researchers that address this issue to some extent, but there seems to be no evidence of an undergraduate course that combines these two approaches. Simplified theory that covers the important mechanism design and FEA concepts can be complemented by validation using simulation tools. This may be a good starting point before a fully-blown course may be attempted. Availability of a CAE lab with state of the art CAE software is one of the important ingredients for the success of implementing this idea. A four bar mechanism based on a conventional wind shield wiper is used while developing the tutorials.

For the second part of the paper, advanced CAE simulation tools can also for the simulation of sheet metal parts that are used in automotive and other industries. The lecture covers basics of sheet metal forming and forming simulation technology. Then, the basic process of modeling and analysis of one of the sheet metal components used in instrument panels of automotive applications is discussed. DOE studies to optimize its design are discussed. Solid modeling of the individual sheet metal components using different CAD programs like Unigraphics are imported to different CAE programs (like HyperForm) to be meshed and then subsequently exported to high-end solvers like LS-Dyna to study the formability characteristics of the component. Finally, modal analysis and gauge optimization of the entire instrument panel assembly have been conducted using CAE tools. The main purpose of doing these types of analyses is to choose the optimum design based on the set constraints.