

# The ANSYS Interface to ADAMS: new features and an example

Marc Brandenberger

CAD-FEM AG, Aadorf, Switzerland

## Summary

When analyzing multibody systems programs like ADAMS by MDI Inc. are used. If the flexibility of a component can not be neglected ADAMS relies on Finite Element programs like ANSYS to compute the data describing the behaviour of the flexible component. In ANSYS 6.1 a Graphical Interface will be introduced which allows the user to compute this modal neutral file easily. Further the new GUI will support the user when importing loads from ADAMS.

This paper shows for what systems the ANSYS-ADAMS interface can be used for, what features will be new in ANSYS 6.1 and demonstrates its purpose and how to use it with a simple example.

## Keywords

Multibody systems, flexible bodies, Ansys, Adams.

## 0. Introduction

The ADAMS software developed by Mechanical Dynamics Inc. is one of several special purpose programs used to simulate the dynamics of multibody systems. One drawback of these programs is that all components are assumed to be rigid. Tools to model component flexibility only exist for geometrically simple structures. To account for the flexibility of a geometrically complex component ADAMS relies on finite element codes such as ANSYS.

After analyzing a model in ADAMS the forces acting on a body can be stored in an ANSYS input file. The new interface supports the user when importing the forces. It should be used only for components that ADAMS assumes to be rigid or components undergoing small accelerations. For analyzing components undergoing large deformations the MSR toolkit from MDI should be used.

## 1. General remarks

### 1.1 When to use the Interface

You use the interface whenever you want to include flexibility of a body in ADAMS. Flexibility can be an important aspect in a multibody system, e.g. to recognize resonances, accurately simulate forces and movements of the components etc. Often flexibility of a system is not negligible. A typical example is the model of a piston moving in an engine (Figure 1). The movement of the piston significantly depends on the flexibility of the crankshaft and/or the conrod. Since the geometry of a conrod can be quite complex, the interface can be used to account for the conrod flexibility.

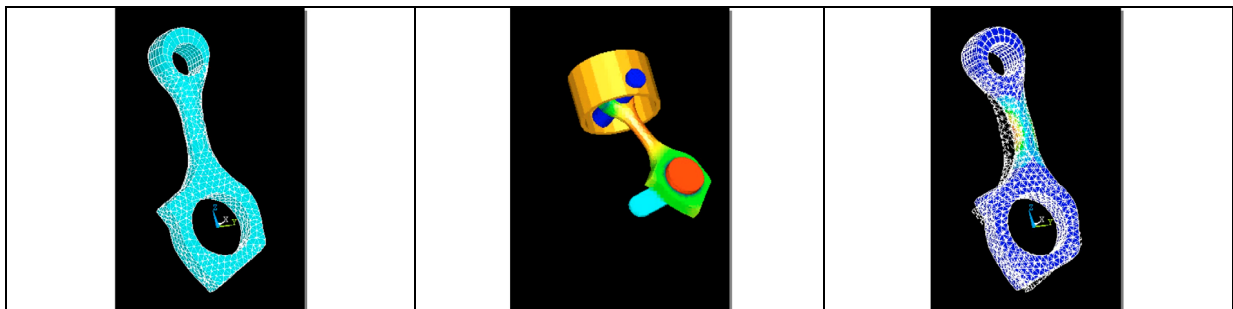


Fig. 1: Piston

The interface is relying on the component mode synthesis (also known as dynamic substructuring). This formulation is based on a modal analysis. Thus only linear flexibility can be considered in the ADAMS model. Both geometric and physical nonlinearities are ignored. If significant geometric nonlinear effects are present, the component has to be subdivided in several smaller components each of which has to be transferred separately. The flexible component can then be assembled in ADAMS from the subdivided components resulting in a flexible component with geometric nonlinearity.

When transferring forces from ADAMS back to ANSYS, you can utilize the export capabilities of ADAMS in order to write an ANSYS input file. This file contains acceleration and rotational velocity of the rigid part and forces acting in the joints of the component and is used to perform a stress and deformation analysis.

### 1.2 Algorithm used in ADAMS

The algorithm to compute the mnf file is based on the component mode synthesis. This method allows the user to apply variable boundary conditions at an Interface point and not to have to specify the boundary conditions in ANSYS. Different joints and forces can be applied to the Interface point in ADAMS.

The motion of a structural component with a set of interface points are spanned by the interface constraint modes and the fixed interface normal modes. Constraint modes are the static shape assumed by the component when one degree-of-freedom of one interface point is given a unit deflection while holding all other interface degrees-of-freedom fixed. Normal modes are the eigenmodes of the structure when all degrees-of-freedom of all interface points are held fixed. The

number of constraint modes is equal to the number of degrees-of-freedom of all interface points whereas the number of normal modes can be chosen by the user. For more information refer to [2],[3].

## 2. The interface in ANSYS 6.1

In release 6 ANSYS will provide a revised interface. Up to now only a macro existed that could be used to compute the mnf file.

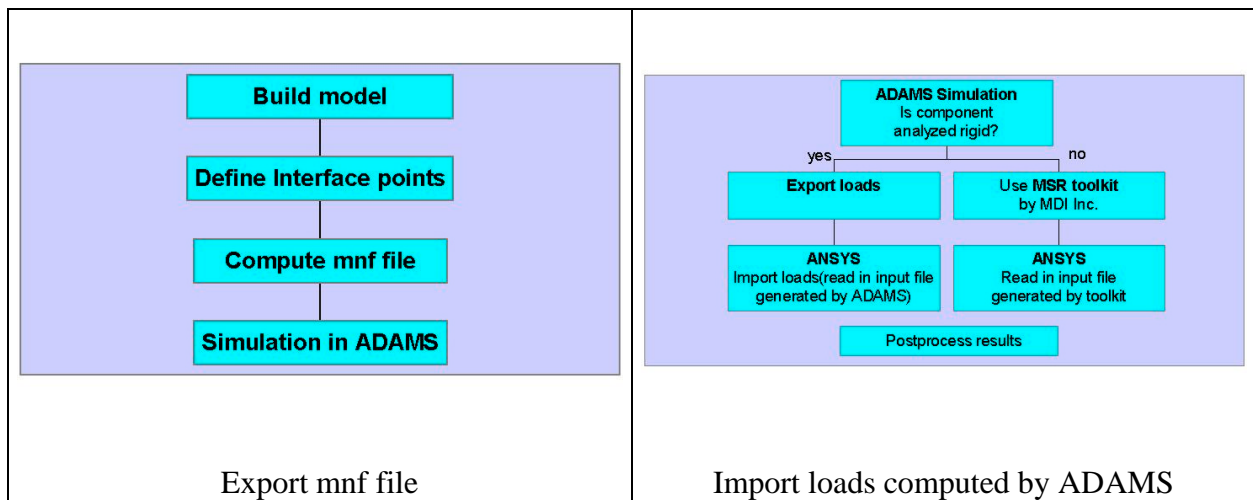
### 2.1 What's new

The new features in ANSYS 6.1 will be:

- Graphical User Interface that leads the user through the required steps when computing a mnf file.
- Graphical User Interface for importing the loads computed in ADAMS.
- Documentation: Online Help, Users Guide, Training Material.
- Stress and strain output for the computed modes.
- Marked interface nodes.
- Less computation time.
- Support of stress and strain output to mnf file.
- Use of mnf library 4.1.

### 2.2 Procedure

Figure 2 shows the steps the user has to go through when using the interface. The diagram on the left shows the usual steps when computing a mnf file. The diagram on the right shows the steps when transferring loads from ADAMS to ANSYS.



**Fig. 2:** Steps when using the interface

#### *Build model*

You build the model in ANSYS as you would do in any structural analysis. The Ansys Structural Analysis Guide will provide you with information about limitations of certain elements.

An important point are the interface points. These are the nodes where you are going to apply forces or joints in ADAMS. Since these are single nodes you have to be aware that the forces are applied to that single node. A reinforcement of the area is therefore recommended.

#### *Ansys: Export mnf file*

You then compute the mnf file using the new GUI.

### Adams: Import mnf file

In ADAMS you import the mnf file following the steps in the ADAMS/Flex guide. You then apply joints and forces to the flexible body on the interface nodes.

### Transfer Forces / Compute deformation and stresses in ANSYS

After the ADAMS simulation you may transfer loads back to ANSYS. There are two ways to do this:

- If the body is modeled rigid in ADAMS (you can switch a flexible body to rigid if necessary) you transfer back constraint reaction forces and external forces at specified time steps. The ANSYS GUI supports you when importing the loads.
- If the body is modeled flexible in ADAMS you transfer back the deformation at specified times. This can be done using the MSR toolkit provided by MDI in the near future. No ANSYS GUI exists yet to support the user when importing the deformation.

## 3. A small example

A spatial joint mechanism (figure 3) with a kinematic loop and one rigid degree of freedom is considered. Such a system is typically used as a transmission element for fast motion in shaking or positioning processes. Because of the high operational velocity, the main link (figure 4) is designed as a slender beam from lightweight material. Therefore the structural flexibility may result in vibrations of the main link, which should be taken into account in a reasonable simulation. The idea of such a joint mechanism is taken from [1].

The simulation is carried out with a constant imposed angular velocity of the crank. This input motion is then transmitted to the output joint at the base. For the flexible and the rigid system the angular motion of the output are compared to each other for a sufficiently high input velocity. The rigid reference solution is generated by FwdKinematics.m. The acceleration to the constant angular velocity is done via the STEP5 function.

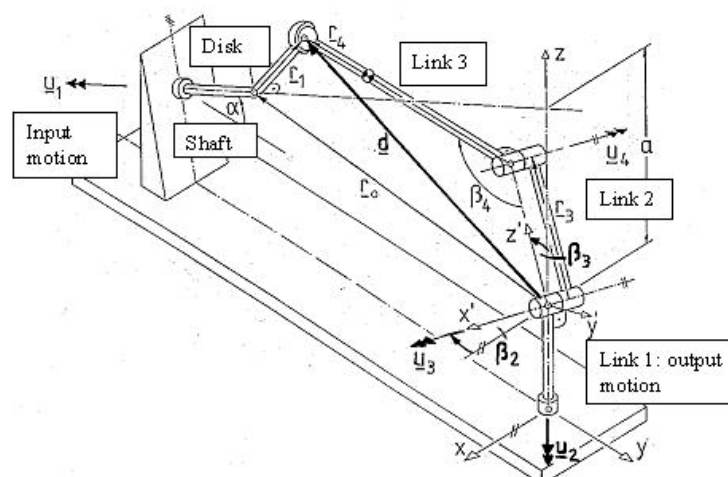
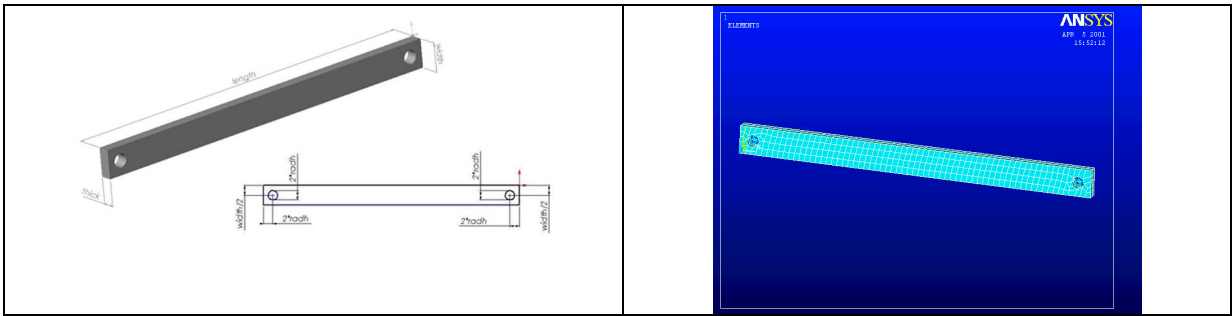


Fig. 3: Spatial joint mechanism



**Fig. 4:** Link 3: Geometry and ANSYS model

### Interface Points

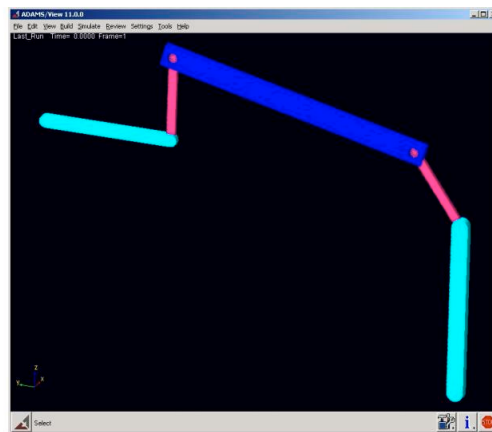
The interface points are the nodes where we want to apply constraints in the ADAMS model. These points are the middle points of the cylindrical bearing surfaces. These nodes are connected to the bearing surfaces using BEAM4 elements ('spider web'). See figure 4.

### Export mnf file

The mnf file is then computed using the new ANSYS GUI.

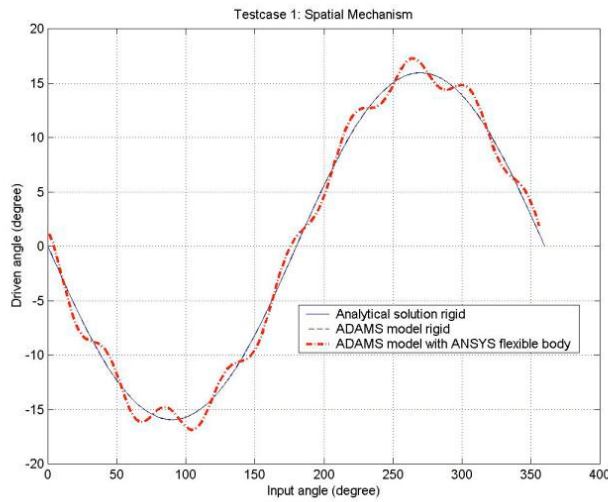
### Import mnf into ADAMS / Simulation

Import the mnf file in ADAMS using the appropriate commands or the GUI.



**Fig. 5:** ADAMS model

Start the simulation and look at the motion of link1. In figure 6 we compare the motion for a flexible system and a rigid system. Clearly the resonance frequency of the system can be seen in the response of the flexible system.

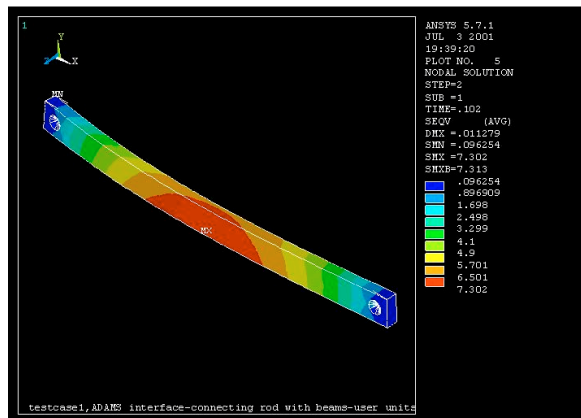


03-Jul-2001 / CAD-FEM AG MAB / ANSYS-ADAMS connection

**Fig. 6:** Motion of link1

### Transferring forces back to ADAMS

Using the results of the simulation of the rigid system we export the constraint reaction forces at a critical time step and import that file into ANSYS. A static deformation and stress analysis is started. The deformation of link3 is shown in figure 7.



**Fig. 7:** Structural analysis using forces from ADAMS

### References

- [1] M. Hiller: „Kinematik und Dynamik für Mechanismen, Fahrzeuge und Roboter“, Studienskript, Universität Duisburg, 1997.
- [2] Craig R.R. and Bampton, M.C.C., “Coupling of Substructures for Dynamics Analyses”, AIAA Journal, vol.6,no.7,1968,pp.1313-1319
- [3] Ottarsson Gisli, “Modal Flexibility Method in ADAMS/Flex”, March 31, 1998; provided with the ADAMS/Flex documentation or available from MDI Inc. in pdf format.
- [4] Marc Brandenberger, „Simulation von Mehrkörpersystemen mit elastischen Komponenten in ANSYS, LS-DYNA und ADAMS: ein Vergleich“, 17. CADFEM Users Meeting, 6.-8.Oktober 1999, Sonthofen