

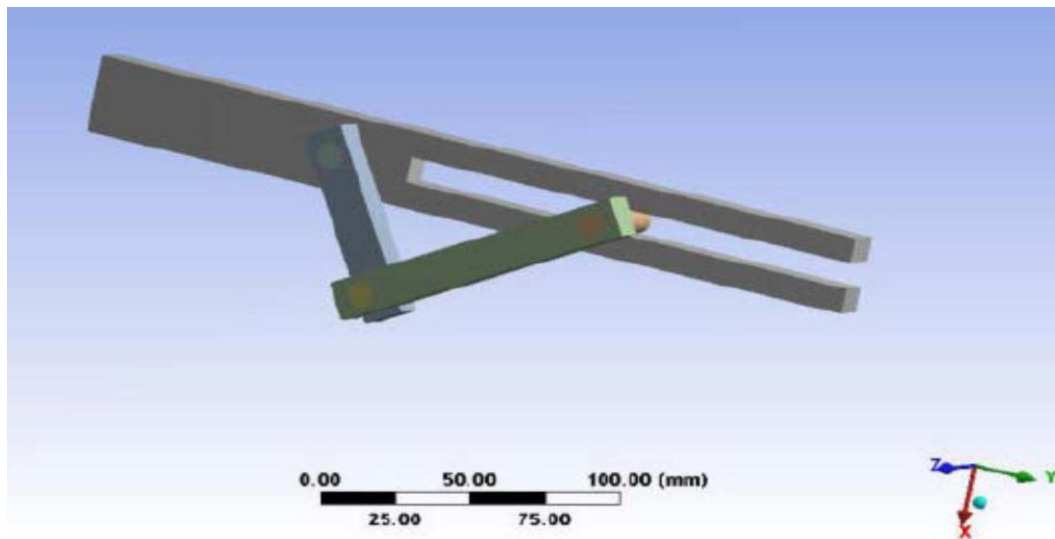
Dynamic substructuring

- Exercise 3 in MF2011 systems engineering 2011

Ulf Sellgren
KTH Machine Design
School of Industrial Engineering and Management



**KTH Industrial Engineering
and Management**



MF2011 Systems Engineering
School of Industrial Engineering and Management
March 2012

EXERCISE 3 – DYNAMIC SUBSTRUCTURING

The exercise

- The exercise requires Ansys 13 (or Ansys 12) APDL
- It is a group exercise (maximum three people per group)
- Set up and run two different multibody simulations - a flexible multibody, and a substructure (CMS) multibody simulation of a crank slot mechanism.
- Compare the results (stresses and solution times) from the two different simulations

The mechanism model

The crank slot model consists of several parts connected by joint elements. The task is to study the multibody dynamics motion of the crank mechanism when starting the mechanism at one of the joints from rest with a rotational acceleration of 25 rad/sec^2 . Furthermore, the transient stress in one of the slider rods (rod 2 In figure 2) has to be examined.

The geometry for the crank slot model consists of a base and two rods. The two rods are attached to each other and the base with three bolts. The material used for all components is structural steel, with Young's modulus (E) = 200 GPa, Poisson's ratio (ν) = 0.3, and the density (ρ) = 7850 kg/m^3 .

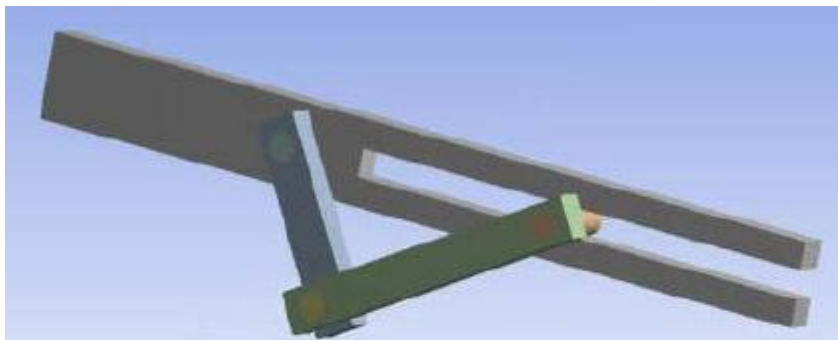


Figure 1. The crank-slot mechanism

The basic Ansys steps in this exercise

- Download the input library file (it contains the complete models)
- Performing the flexible body analysis and examine the motion, stresses & solution time – do this in ONE UNIQUE FOLDER
- Copy the input library file to another folder. Perform the component mode synthesis simulation in THIS UNIQUE FOLDER and study the results
- Compare the dynamic results and the processing times for the two simulations
- Mail the results to mf2011@md.kth.se

FOUR STEPS TO PERFORM

1. Create an empty folder and download the Ansys input file library file CrankSlot.lib (from the mf2011 home page).

The two ANSYS input files are used in the example analysis of the crank slot mechanism described in this section. The files were generated by the ANSYS Workbench product and packed in a single library file ([CrankSlot.lib](#)). The library file contains a number of macros, where each macro starts with the name of the macro, e.g. “model” and ends with “/eof”

The crank slot base model consists of one base component, two rods and three bolts (see figure 2). The base and rod 1 parts are FE-modeled as rigid links. Rod 2 is modeled as a flexible solid component with one pilot node in the center of each of the two bolt holes. The pilot nodes are constrained to the nodes on the hole surfaces. The three bolts are modeled as point mass elements (Ansys *mass21* elements). The bolt node is connected to the pilot node in the mating rod with the Ansys *mpc184* general joint element.

The input files used to run the crank slot analysis in the ANSYS program were generated by ANSYS Workbench.

The joints in the model

The joints that connect the parts of the supplied crank slot model. Revolute, slot, and cylindrical joints form the moving joints. The base of the model is fixed to the ground via a fixed joint.

The following figure shows the parts of the model, with the joints listed to the right:

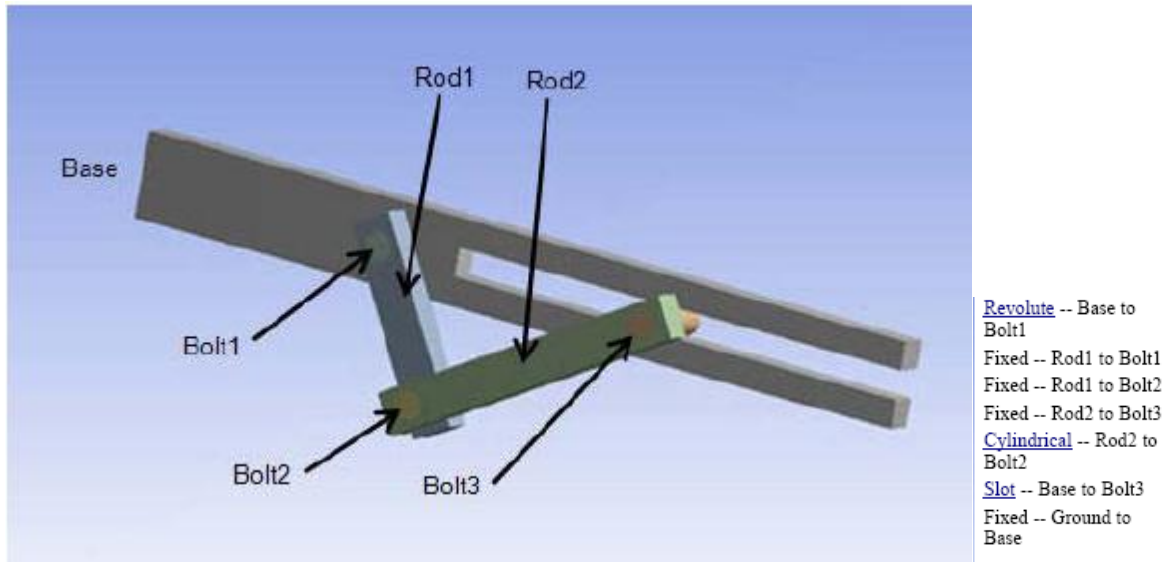


Figure 2 . The crank slot parts and joints

All joints are available via the [MPC184](#) element's KEYOPT(1) setting and, in some cases, the KEYOPT(4) setting.

2. Perform a flexible body simulation

Start Ansys from the Ansys Product Launcher (figure 3). Be sure to have directory where you downloaded the Crankslot.lib file as the *working directory*.

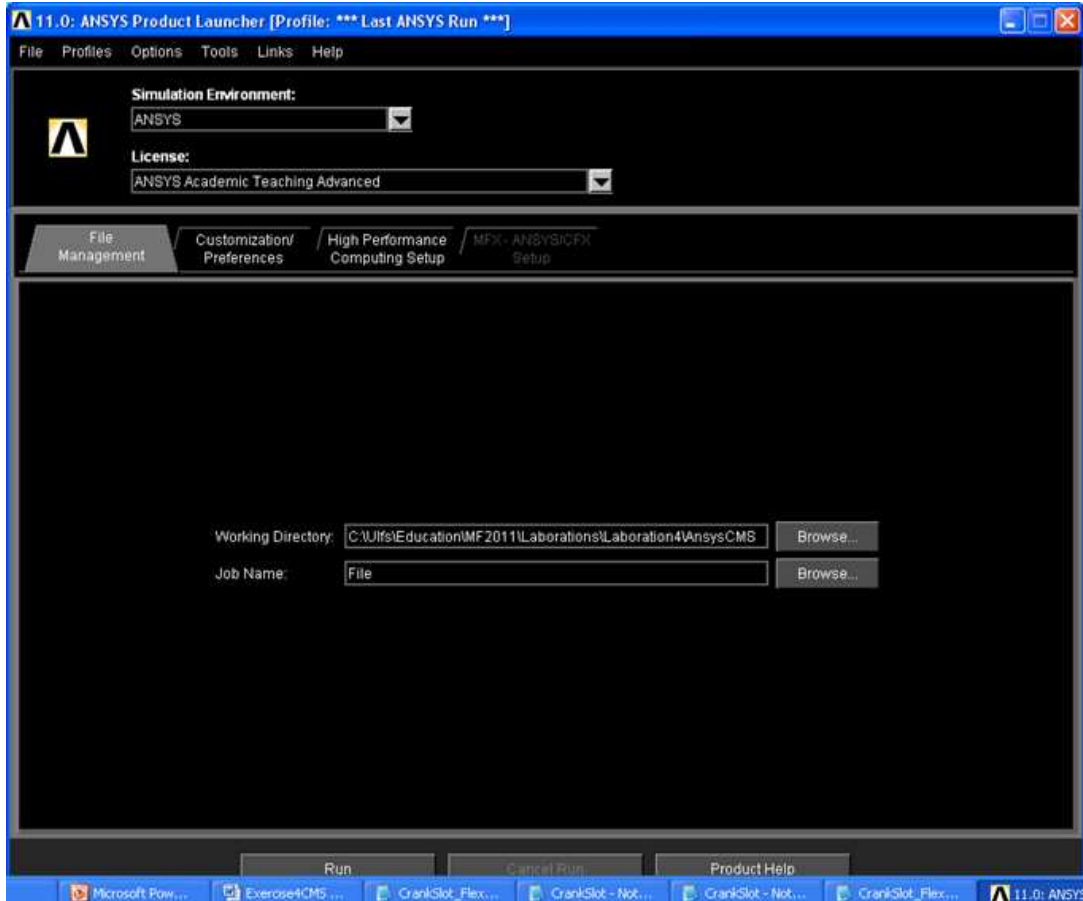


Figure 3. Start Ansys from the product launcher.

1 - Load the flexible crank slot model and perform the simulation by typing the commands:

- *ulib,crankslot,lib (point to the library file)
- *use,model (execute the *model* macro in the library)
- *use,solution (execute the *solution* macro in the library)

2 - Look for solution time (cp and wall clock) in the output window-

3 – Enter the general postprocessor and plot results:

- first deformed shape
- then contour plot of nodal results (stress – vonMises)

4 – Animate the results by pressing the PlotCtrl (see figure 5) and then select Animate, Over time and specify proper animation control data (see figure 6).

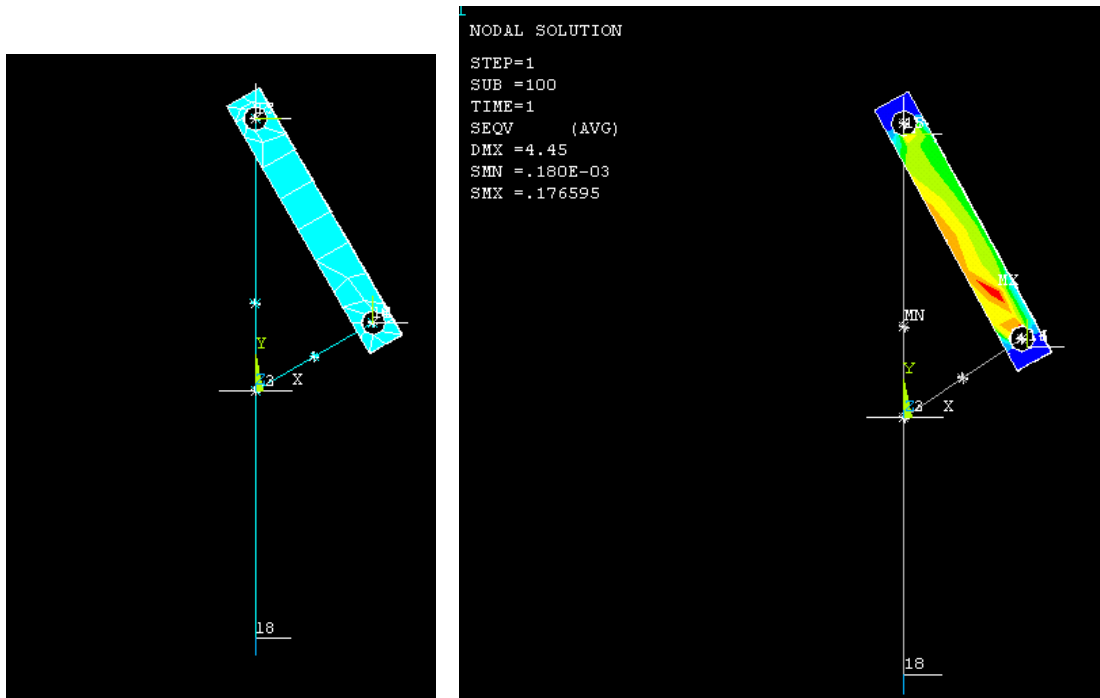


Figure 4. The model (left) and the vonMises stresses and one solution time (right)



Figure 5. Animation is available at the PlotCtrls menu.

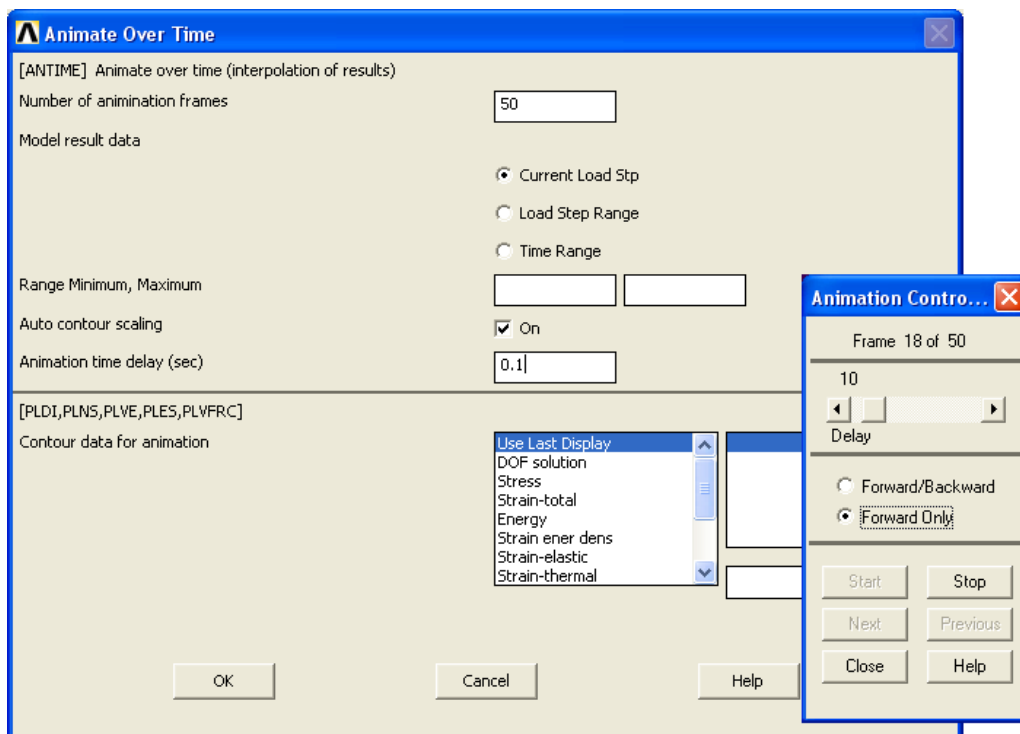


Figure 6. Select proper animation control data.

3. Create a new working folder that contains a copy of the library file, restart Ansys with the Product Launcher pointing to the new working directory. Use Component Mode Synthesis in the multibody simulation

The task is to use the flexible body created previously, to create a component mode synthesis (CMS) model with large rotation. Using CMS for the multibody analysis consists normally of three passes:

- 1) Creating a superelement of the flexible body (generation pass).
- 2) Using the superelement in the transient analysis (use pass).
- 3) Recovering stress and displacement results for the entire model (expansion pass).

These passes are available as macros in the crankslot.lib library file.

1 – First be sure to not work in the same folder as with the full model. Load the flexible crank slot model and perform the simulation by typing the commands:

(First enter the two Ansys commands *finish* and */clear* to clear the working memory)

*ulib,crankslot,lib (point to the library file)

*use,cmsmodel (execute the *model* macro in the library)

*use,cmsgenerate (condense the flexible rod to a CMS submodel)

*use,cmsuse (simulate the response with the condensed CMS model)

2 - Look for solution time (cp and wall clock) in the output window-

3 – Enter the general postprocessor and plot the displacement results (only).

4 – Animate the results (the motion) over time. What does it look like?

5 – Expand the results for the condensed CMS model:

*use,cmsexpand (expand the condensed rod 1 model for all time steps)

6 – Enter the general postprocessor and plot the displacement results and the nodal stresses.

7 – Animate the results over time. What does it look like?

8 – Append the condensed motion and the expanded rod 1 solution:

*use,cmsappend (append condensed and expanded results)

9 – Enter the general postprocessor and plot the displacement results and the nodal stresses.

10 – Animate the results over time. What does it look like?

4. Pack the generated animation files for the two models and mail it to mf2011@md.kth.se.

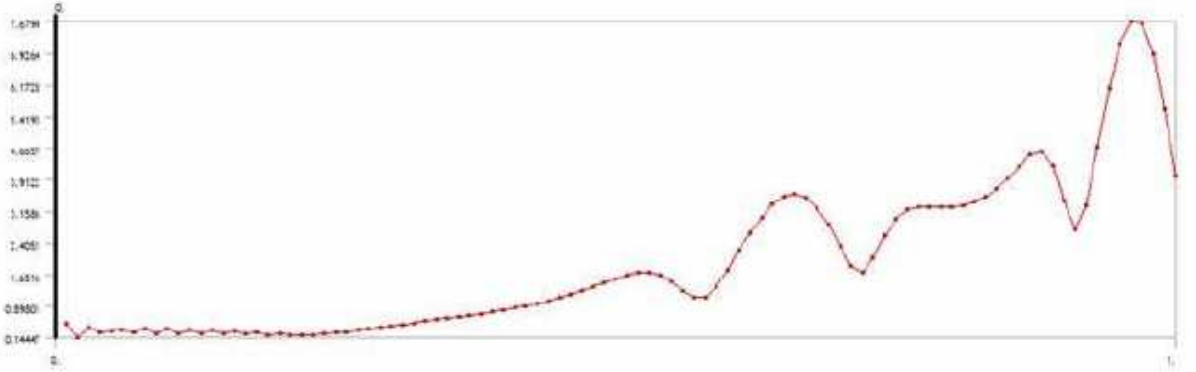
Zip the animation file (avi format files) and send them to mf2011@md.kth.se

The e-mail must also include:

1. A subject: *Exercise 3*
2. The simulation times (in CPU seconds) for the two cases (scrutinize the output window)
3. A short comment (your observation and reflection on the steps performed for the CMS model.
4. Your interpretation (by looking in the library file) of how a macro is defined (name, start, end and what is between) in the library. All commands are defined in the “Mechanical APDL Command Reference” in Ansys Help.
4. Don't forget to give your names in the e-mail.

Appendix

In addition to information about the displacement and stress in the structure, you can use the *joint probes* to obtain specific results information about the various joints in the model. Here the total force at a single joint is plotted as a function of time:



Comparison of the CPU times shows the advantage of using CMS even for a simple model such as the crank slot. The benefits of CMS for large-rotation and nonlinear analyses can multiply in cases involving larger and more complex models, especially those exhibiting more nonlinear behavior. The figure below shows a typical relation between the cpu times for a simulation performed with a flexible model and a simulation performed with a condensed model. The accuracy of the simulated results for the two models are comparable.

