Obtaining a Listing of the Stiffness Matrix / Load Vector from ANSYS, Version 8.1

To obtain a listing of the stiffness matrix and load vector from an ANSYS model, use the "Substructure" option in the Solution processor, and solve. The method is outlined in the following instructions. There are two sets of instructions. The first set is based on use of the Graphical User Interface (GUI). There are also command line entries that can be used to accomplish the same tasks. Instructions for the command line approach are shown after the instructions for the GUI approach.

GUI Option:

1. Create the model, including all required real constants, material properties, constraints, and applied loads.

2. Enter the solution processor. On the ANSYS Main Menu, select "Solution".

3. On the "Solution" menu, select "Analysis Type", then "New Analysis".

4. On the "New Analysis" menu, select "Substructuring/CMS", and click "OK".

5. On the "Solution" menu, under "Analysis Type", select "Analysis Options".

6. On the box that opens, verify that "Substructuring" is selected, then click "OK".

7. On the "Substructuring Analysis" menu, there is a "SEPR Items to be printed" option. For this option, select "LoadVect+Matrix", then click "OK".

8. Select master degrees of freedom. Assuming you wish to include all unconstrained degrees of freedom in your printed matrices, then select all dof in the model as master dof. To do this, at the ANSYS command prompt, type the command: *m,all,all*

9. Direct the ANSYS output to an output file. At the ANSYS command prompt, type the command: */output,filename* (where filename is any name the user chooses).

10. Solve. On the "Solutions" menu, under the "Solve" heading, select "Current LS", then click "OK". Upon solving, you may get warnings, such as: *Node 2 UY master is superseded by a specified constraint.* This is simply because some of the master dof selected were constrained to zero displacement. These warnings are not a problem.

11. Redirect the ANSYS output back to the output window. At the ANSYS command prompt, type: */output*

<u>Command Line Option</u>:

First, create the model, including all required real constants, material properties, constraints, and applied loads. Then, enter the commands below in italics. These are entered at the ANSYS command line.

/solu
antype,substr
seopt,file,1,1
m,all,all
/output,filename   (where *filename* is any name you desire)
solve
/output

After the above steps, whether the GUI option or command line option is used, an ASCII text file will be stored under the filename chosen (see Step 9 of the GUI Option instructions) in your ANSYS working directory. This file will contain the load vector and stiffness matrix listing. It can be viewed with any text editor, such as Notepad, or any word processing program, such as Microsoft Word. The listing may be preceded by other information, so you may have to scroll through the file some to find it. Using Notepad, if desired, this information can be cut and pasted into another file. There is an example listing on the following page.

An example listing is provided below for the two-spring system shown. The system uses ANSYS Combin14 spring elements. All dof are constrained to zero at node 1. Nodes 2 and 3 have UY and UZ dof constrained to zero displacement.
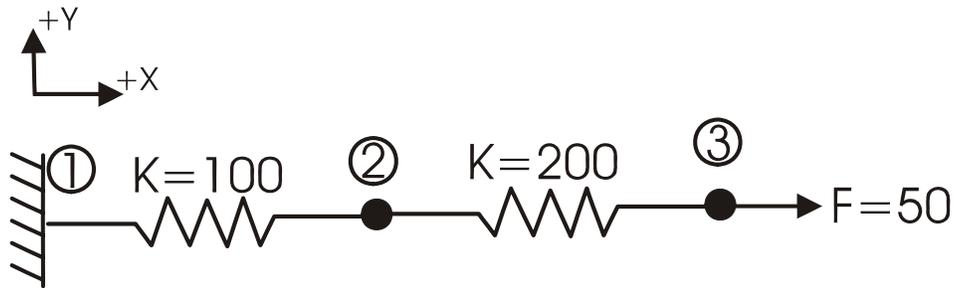
EXAMPLE LISTING OF LOAD VECTOR AND STIFFNESS MATRIX:

```
*********************************************************************
*                                *
*  ANSYS VERSION FOR DEGREE-RELATED, PUBLIC DOMAIN RESEARCH ONLY  *
*                                *
*********************************************************************

  ***** LOAD VECTOR NUMBER   1 *****

 ROW   NODE   DIR    VALUE


   1      2  UX   0.000000
   2      3  UX   50.00000


  ***** STIFFNESS MATRIX *****

 ROW  1  NODE   2   DEG. OF. FR. =  UX

  1 0.30000000E+03   2-0.20000000E+03

 ROW  2  NODE   3   DEG. OF. FR. =  UX

  1-0.20000000E+03   2 0.20000000E+03
```