To extract the reduced global matrix from Ansys mechanical, the following information might be helpful:
1) The file: How to extract structural matrices (mass,stiffness…) from Ansys and import them into Matlab. … by Alain Batailly is very helpful. (section 2 is most important)

The procedures are:
1) Creating the model in Ansys mechanical, then add these two commands before the solving step
EMATWRITE,YES
WRFULL
2) Exporting the input file for a batch mode run. Here only the direct solver are supported, so in step 1, you have to choose Direct solver to solve, or you can change it manually in the input file. The run has to be using 1 core in sequential mode, otherwise, the global matrix will mess up. It is claimed in Ansys documentation it support parallel running, but I failed to extract the matrix using parallel mode.
3) You will get *.FULL file after the run, and then you can run:
ansys -g
to open the GUI, then follow Alain's description to obtain the matrix and RHS vector in HB format. Then you can transform it to other storage format.