How to run input files from the command window with Abaqus
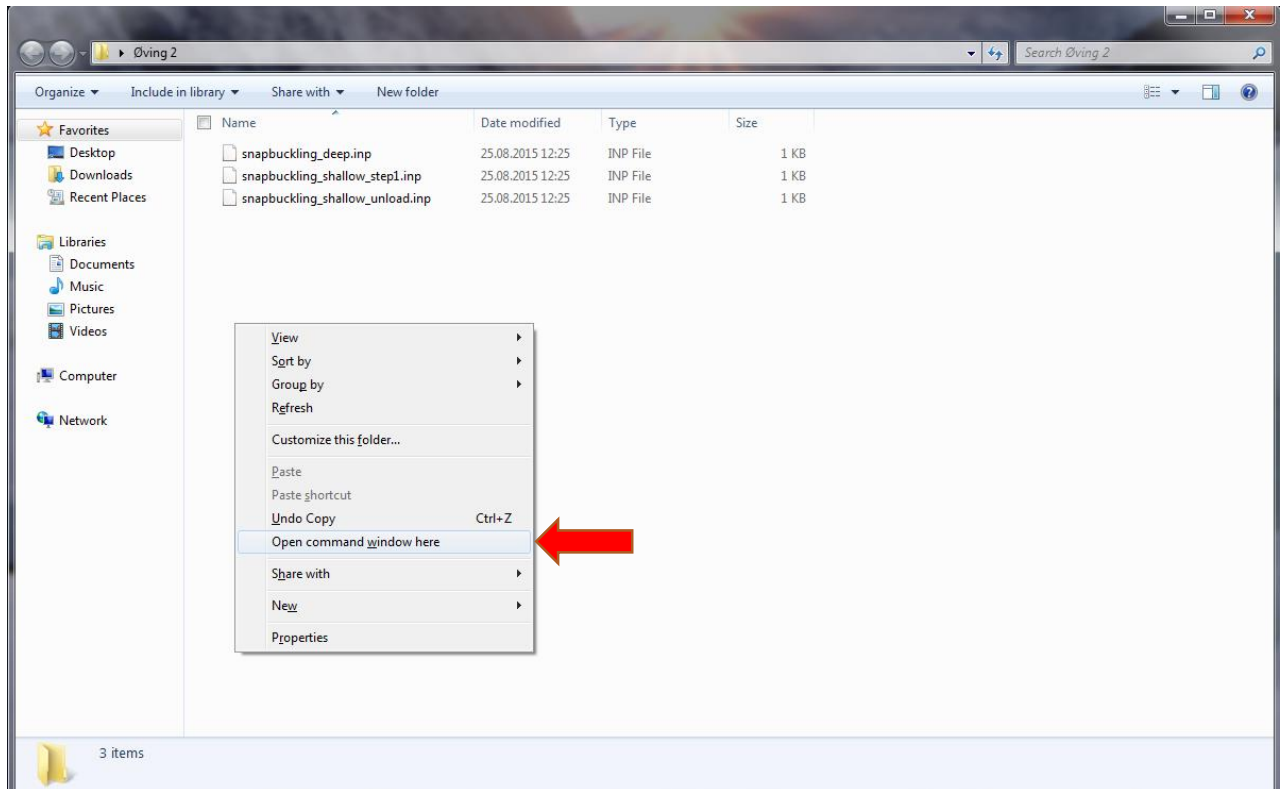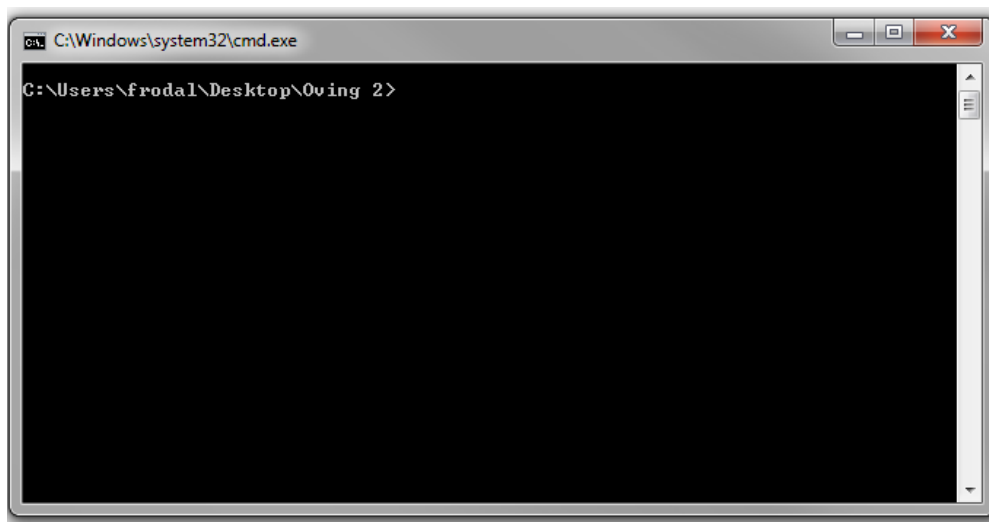
1) Open the folder containing the input files you want to run.
2) Press and hold down "shift" ⇧ on the keyboard and right click inside the folder.
3) Click the "Open command window here".

This should open a command window in the folder you are in.

4) In the command window you should write the commands, and hit "enter" after:
   abaqus job=snapbuckling_shallow_step1 int

This will run the input file named "snapbuckling_shallow_step1.inp" and create the result files for this job.

5) To do the restart analysis you should write:

abaqus job=snapbuckling_shallow_unload oldjob=snapbuckling_shallow_step1 int

This will run the input file "snapbuckling_shallow_unload.inp" and use some of the information from the previous job "snapbuckling_shallow_step1".

Now you have two .odb files with information.

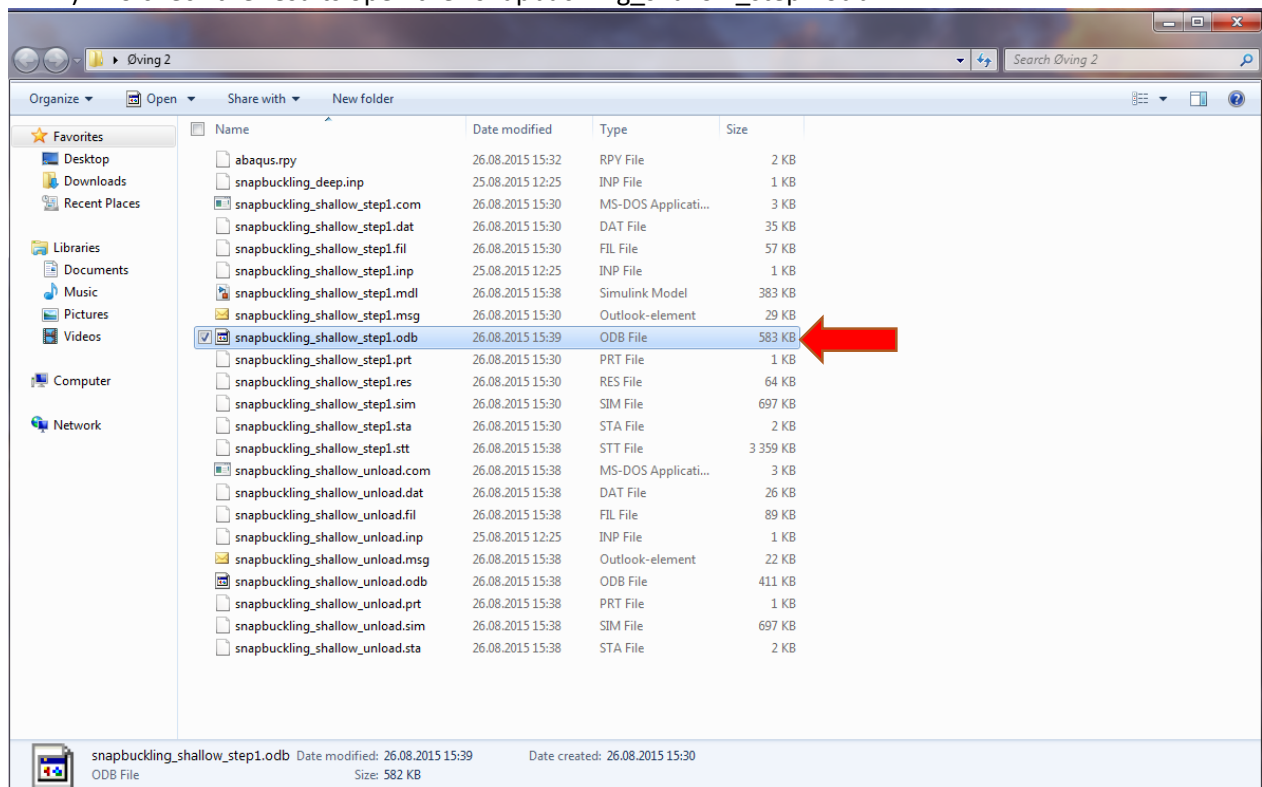6) To merge them write in the command window (in one line):

abaqus restartjoin originalodb=snapbuckling_shallow_step1 restartodb=snapbuckling_shallow_unload

This will merge the information from the two .odb files into the .odb file named "snapbuckling_shallow_step1.odb".

(In Chapter 3.2.2 in the Abaqus Analysis User's Manual you will find more detailed information regarding running analysis from the command window.)

(For more information about joining .odb files see Chapter 3.2.21 in Abaqus Analysis User's Manual ver. 6.14)

7) To check the results open the "snapbuckling_shallow_step1.odb".

8) Extract your data.