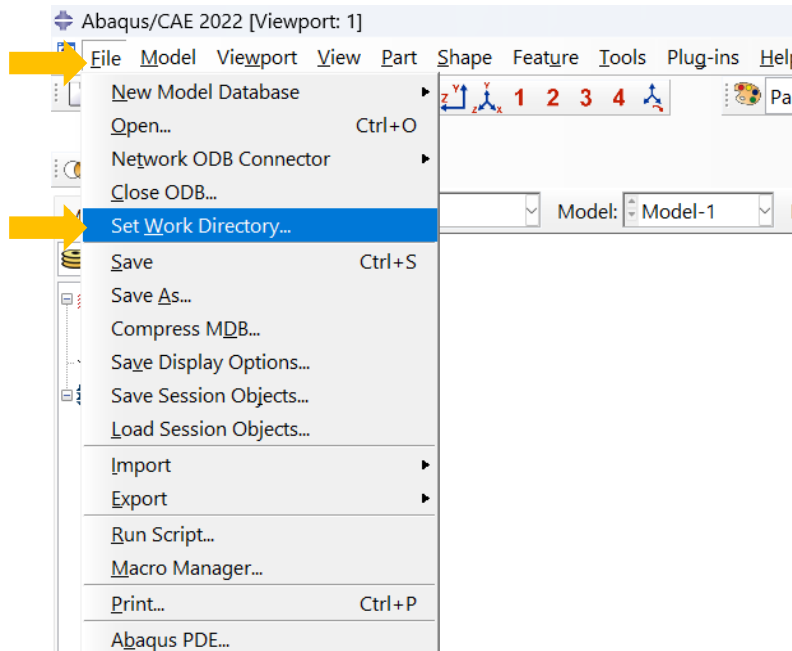
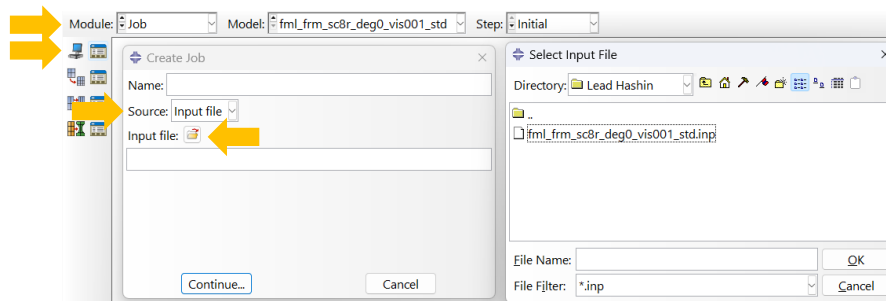


How to analyze the input file in Abaqus CAE

To simulate this problem in Abaqus, save the inp file in a folder and set the folder path as the **working directory**, as illustrated in the figure below.



Next, navigate to the **Job** module, create a new job, and directly open the .inp file with the specified title, as shown in the figure.



Then, submit the job and wait for the solution to complete.