

ANSYS MAPDL

ANSYS Multiphysics software offers a comprehensive product solution for both multiphysics and single-physics analysis. The product includes structural, thermal, fluid and both high- and low-frequency electromagnetic analysis. The product also contains solutions for both direct and sequentially coupled physics problems including direct coupled-field elements and the ANSYS multi-field solver.

To run ANSYS MAPDL in batch mode you can utilize/modify the default mapdl.pbs script and execute it via the qsub command.

```
' #!/bin/bash #PBS -l nodes=2:ppn=16 #PBS -q qprod #PBS -N $USER-ANSYS-Project #PBS -A XX-YY-ZZ
```

! Mail to user when job terminate or abort

PBS -m ae

!change the working directory (default is home directory)

cd (working directory must exists)

```
WORK_DIR="/scratch/$USER/work" cd $WORK_DIR
```

```
echo Running on host hostname echo Time is date echo Directory is pwd echo  
This jobs runs on the following processors: echo cat $PBS_NODEFILE
```

```
module load ansys
```

Set number of processors per host listing

(set to 1 as \$PBS_NODEFILE lists each node twice if :ppn=2)

```
procs_per_host=1 ##### Create host list hl="" for host in cat $PBS_NODEFILE  
do if [ "$hl" = "" ] then hl="host :procs_per_host" else hl="hl :host:$procs_per_host"  
fi done
```

```
echo Machines: $hl
```

-i input.dat includes the input of analysis in APDL format

-o file.out is output file from ansys where all text outputs will be redirected

-p the name of license feature (**aa_r=ANSYS Academic Research**, **ane3fl=Multiphysics(commercial)**, **aa_r_dy=Academic AUTODYN**)

```
/ansys_inc/v145/ansys/bin/ansys145 -b -dis -p aa_r -i input.dat -o file.out -  
machines $hl -dir $WORK_DIR '
```

Header of the pbs file (above) is common and description can be find on this site. SVS FEM recommends to utilize sources by keywords: nodes, ppn. These keywords allows to address directly the number of nodes (computers) and cores (ppn) which will be utilized in the job. Also the rest of code assumes such structure of allocated resources.

Working directory has to be created before sending pbs job into the queue. Input file should be in working directory or full path to input file has to be specified. Input file has to be defined by common APDL file which is attached to the ansys solver via parameter -i

License** should be selected by parameter -p. Licensed products are the following: **aa_r (ANSYS Academic Research)**, **ane3fl (ANSYS Multiphysics)-Commercial**, **aa_r_dy (ANSYS Academic AUTODYN)**> More about licensing here