Using P-Elements

Introduction

This tutorial was completed using ANSYS 7.0. This tutorial outlines the steps necessary for solving a model meshed with p-elements. The p-method manipulates the polynomial level (p-level) of the finite element shape functions which are used to approximate the real solution. Thus, rather than increasing mesh density, the p-level can be increased to give a similar result. By keeping mesh density rather coarse, computational time can be kept to a minimum. This is the greatest advantage of using p-elements over h-elements.

A uniform load will be applied to the right hand side of the geometry shown below. The specimen was modeled as steel with a modulus of elasticity of 200 GPa.

ANSYS Command Listing

```
finish
/clear
/title, P-Method Meshing
/pmeth,on ! Initialize p-method in ANSYS
```
/prep7 ! Enter preprocessor

k,1,0,0 ! Keypoints defining geometry
k,2,0,100
k,3,20,100
k,4,45,52
k,5,55,52
k,6,80,100
k,7,100,100
k,8,100,0
k,9,80,0
k,10,55,48
k,11,45,48
k,12,20,0

a,1,2,3,4,5,6,7,8,9,10,11,12 ! Create area from keypoints

et,1,plane145 ! Element type
keyopt,1,3,3 ! Plane stress with thickness option
r,1,10 ! Real constant - thickness
mp,ex,1,200000 ! Young's modulus
mp,prxy,1,0.3 ! Poisson's ratio

esize,5 ! Element size
amesh,all ! Mesh area

finish
/solu ! Enter solution phase

antype,0 ! Static analysis
nsubst,20,100,20 ! Number of substeps
outres,all,all ! Output data for all substeps
time,1 ! Time at end = 1

lsel,s,loc,x,0 ! Line select at x=0
dl,all,,all ! Constrain the line, all DOF's
lsel,all ! Re-select all lines

lsel,s,loc,x,100 ! Line select at x=100
sfl,all,pres,-100 ! Apply a pressure
lsel,all ! Re-select all lines

solve
finish

/post1 ! Enter postprocessor

set,all ! Select last set of data
piesol,s,equiv ! Plot the equivalent stress