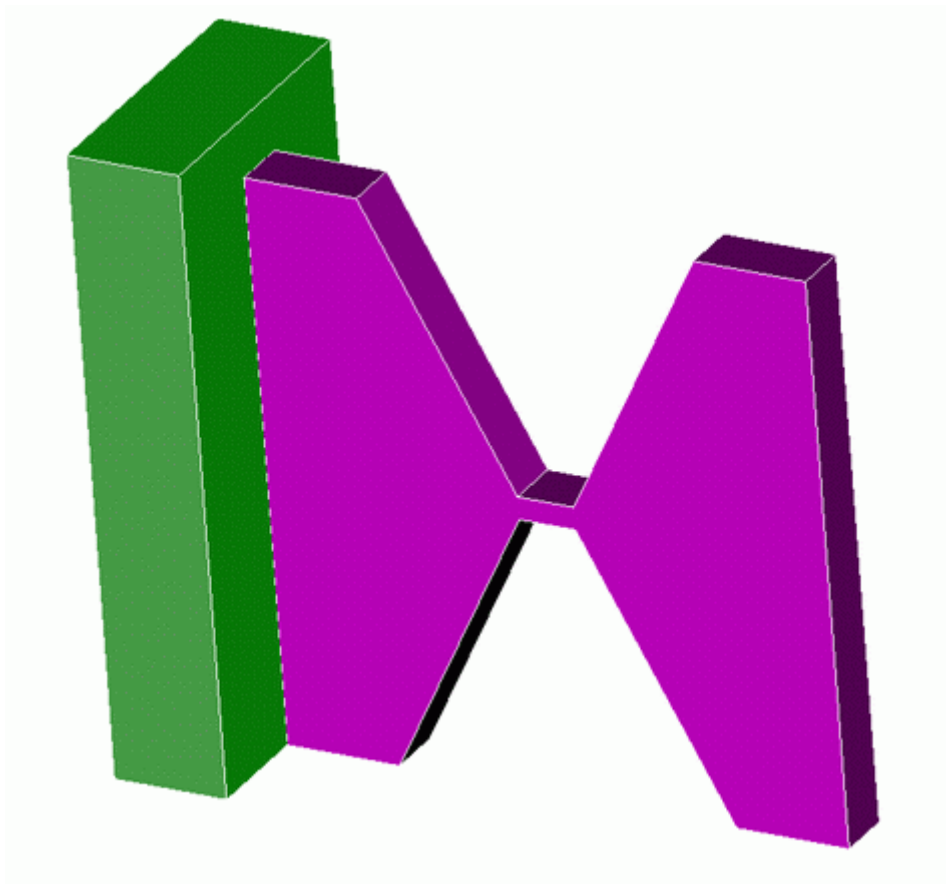


Using P-Elements

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

lselect,s,loc,x,0     ! Line select at x=0
d1,all,,all          ! Constrain the line, all DOF's
lselect,all           ! Re-select all lines

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

solve
finish

/post1               ! Enter postprocessor
set,last             ! Select last set of data
plesol,s,eqv         ! Plot the equivalent stress
```