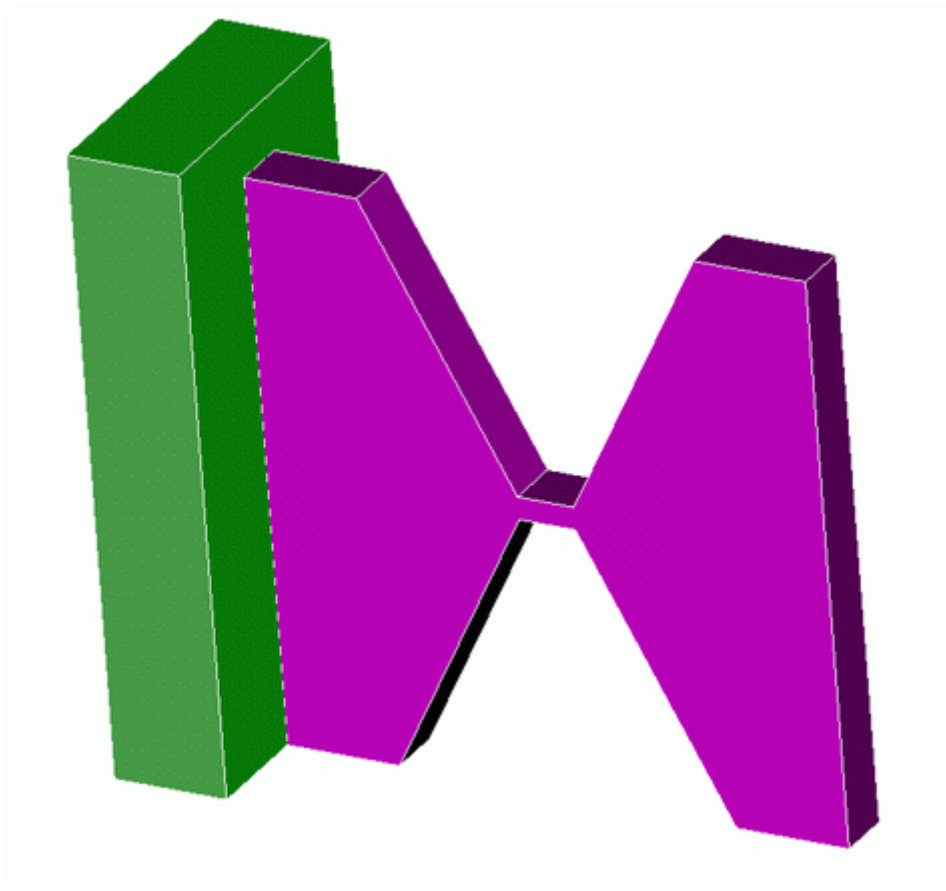


Using P-Elements

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.



Preprocessing: Defining the Problem

1. Give example a Title

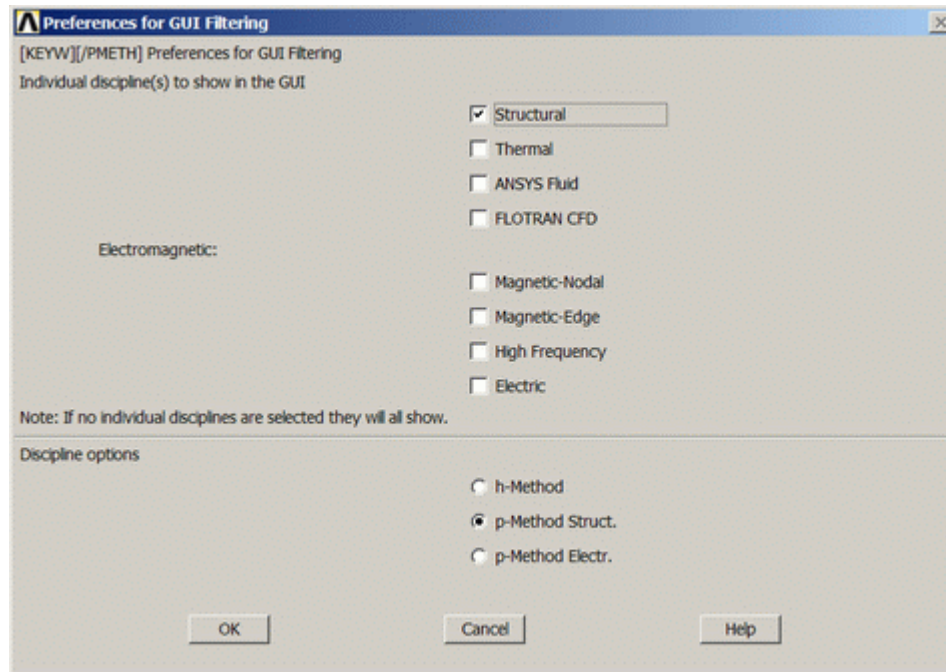
Utility Menu > File > Change Title ...
/title, P-Method Meshing

2. Activate the p-Method Solution Options

ANSYS Main Menu > Preferences

/PMETH, ON

Select **p-Method Struct.** as shown below



3. Open preprocessor menu

ANSYS Main Menu > Preprocessor

/PREP7

4. Define Keypoints

Preprocessor > Modeling > Create > Keypoints > In Active CS...

K, #, x, y, z

We are going to define 12 keypoints for this geometry as given in the following table:

Keypoint	Coordinates (x,y,z)
1	(0,0)
2	(0,100)
3	(20,100)
4	(45,52)
5	(55,52)
6	(80,100)
7	(100,100)
8	(100,0)
9	(80,0)

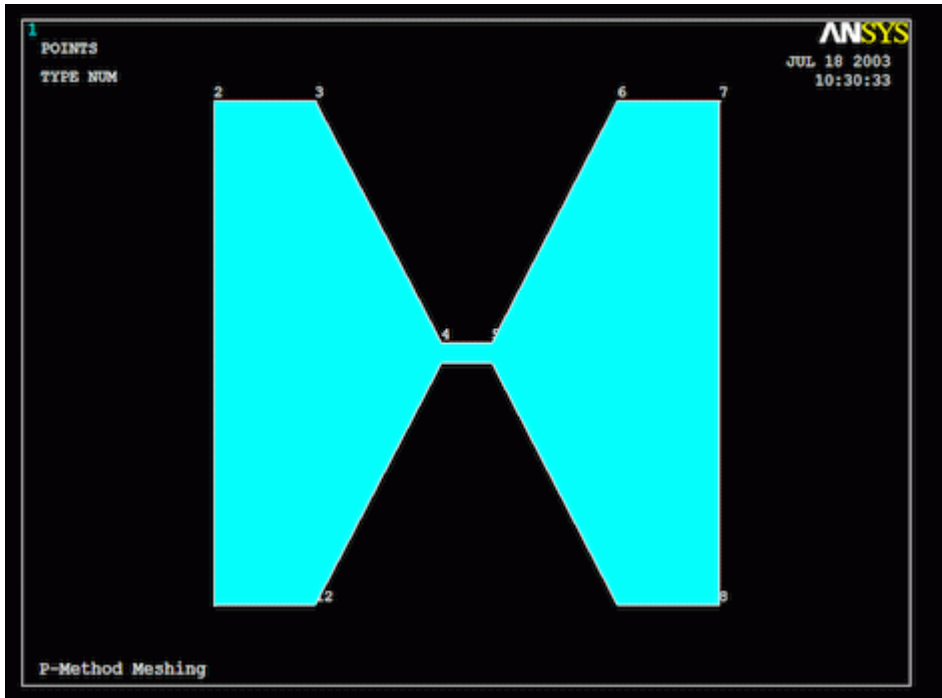
10	(55,48)
11	(45,48)
12	(20,0)

5. Create Area

Preprocessor > Modeling > Create > Areas > Arbitrary > Through KPs

A, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12

Click each of the keypoints in numerical order to create the area shown below.

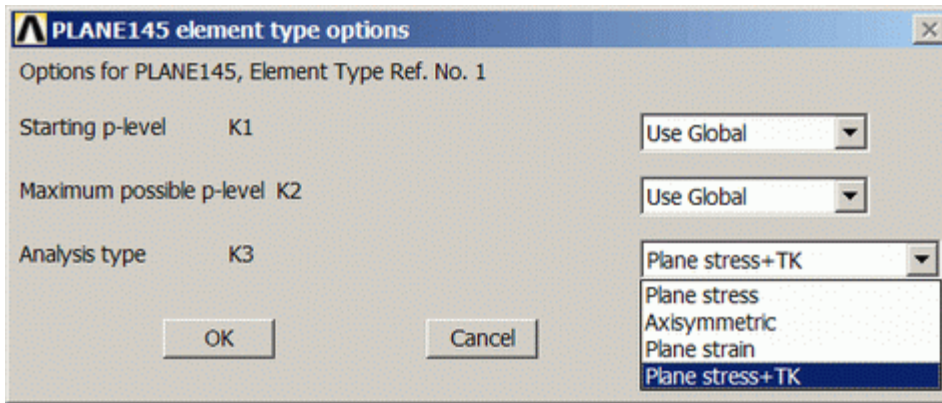


6. Define the Type of Element

Preprocessor > Element Type > Add/Edit/Delete...

For this problem we will use the PLANE145 (p-Elements 2D Quad) element. This element has eight nodes with 2 degrees of freedom each (translation along the X and Y axes). It can support a polynomial with maximum order of eight.

After clicking OK to select the element, click **Options...** to open the keyoptions window, shown below. Choose **Plane stress + TK** for Analysis Type.



Keyopts 1 and 2 can be used to set the starting and maximum p-level for this element type. For now we will leave them as default.

Other types of p-elements exist in the ANSYS library. These include Solid127 and Solid128 which have electrostatic DOF's, and Plane145, Plane146, Solid147, Solid148 and Shell150 which have structural DOF's. For more information on these elements, go to the Element Library in the help file.

7. Define Real Constants

Preprocessor > Real Constants... > Add...

In the 'Real Constants for PLANE145' window, enter the following geometric properties:

- i. Thickness THK: 10

This defines an element with a thickness of 10 mm.

8. Define Element Material Properties

Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic

In the window that appears, enter the following geometric properties for steel:

- i. Young's modulus EX: 200000
- ii. Poisson's Ratio PRXY: 0.3

9. Define Mesh Size

Preprocessor > Meshing > Size Cntrl > ManualSize > Areas > All Areas...

For this example we will use an element edge length of 5mm.

10. Mesh the frame

Preprocessor > Meshing > Mesh > Areas > Free > click 'Pick All'

Solution Phase: Assigning Loads and Solving

1. Define Analysis Type

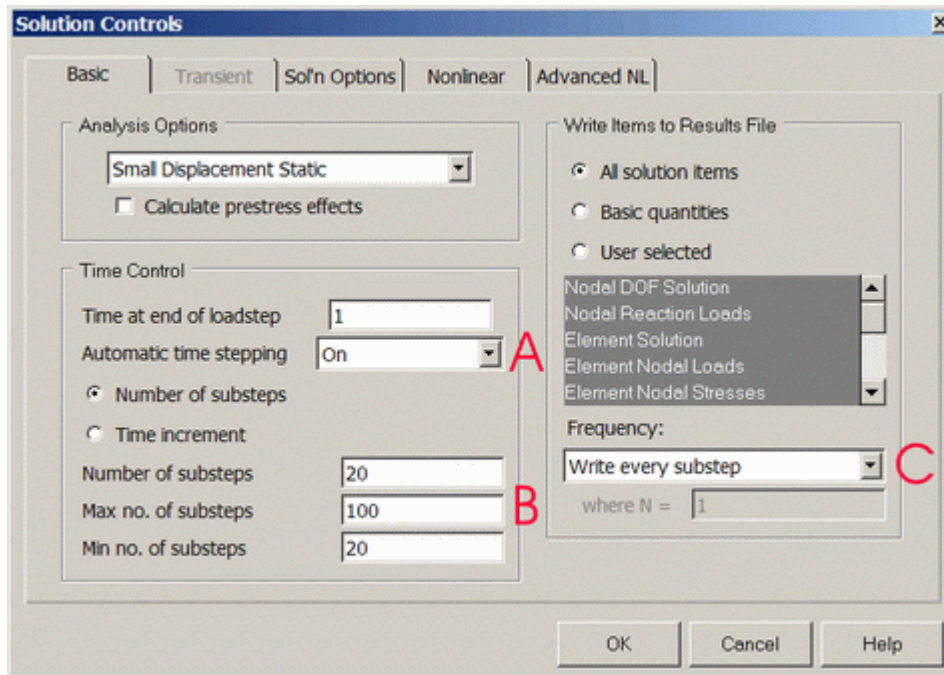
Solution > Analysis Type > New Analysis > Static

ANTYPE, 0

2. Set Solution Controls

Solution > Analysis Type > Sol'n Controls

The following window will pop up.



A) Set Time at end of loadstep to 1 and Automatic time stepping to ON

B) Set Number of substeps to 20, Max no. of substeps to 100, Min no. of substeps to 20.

C) Set the Frequency to Write every substep

3. Apply Constraints

Solution > Define Loads > Apply > Structural > Displacement > On Lines

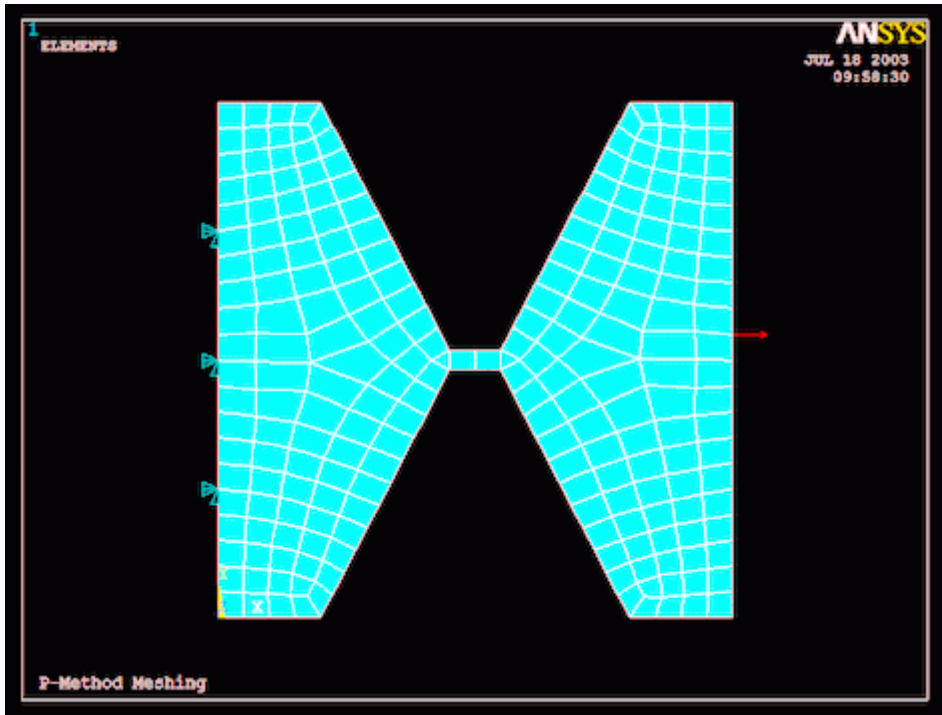
Fix the left side of the area (ie all DOF constrained)

4. Apply Loads

Solution > Define Loads > Apply > Pressure > On Lines

Apply a pressure of -100 N/mm²

The applied loads and constraints should now appear as shown in the figure below.



5. Solve the System

Solution > Solve > Current LS
SOLVE

Postprocessing: Viewing the Results

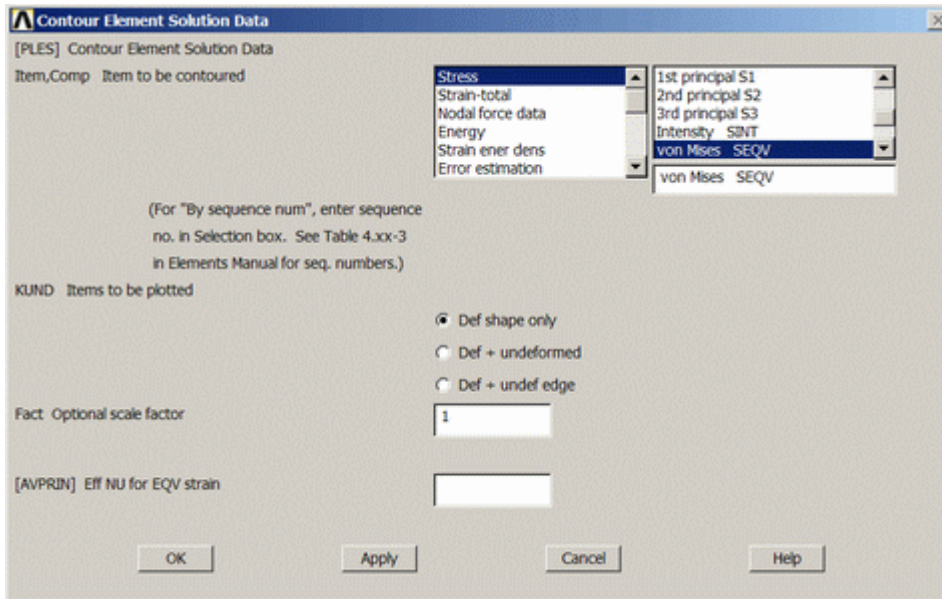
1. Read in the Last Data Set

General Postproc > Read Results > Last Set

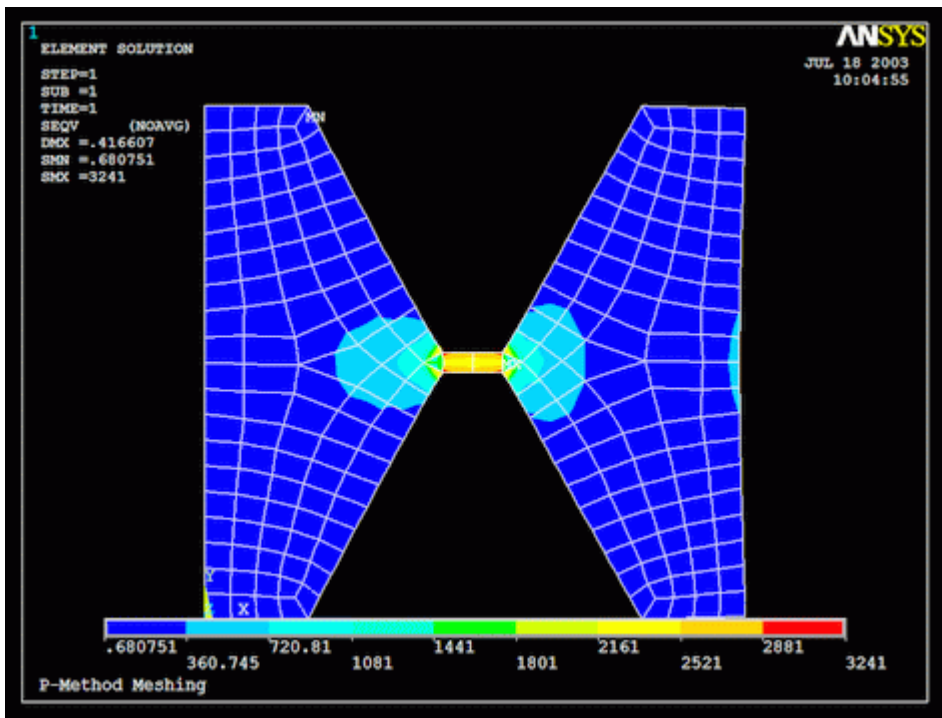
2. Plot Equivalent Stress

General Postproc > Plot Results > Contour Plot > Element Solu

In the window that pops up, select Stress > von Mises SEQV



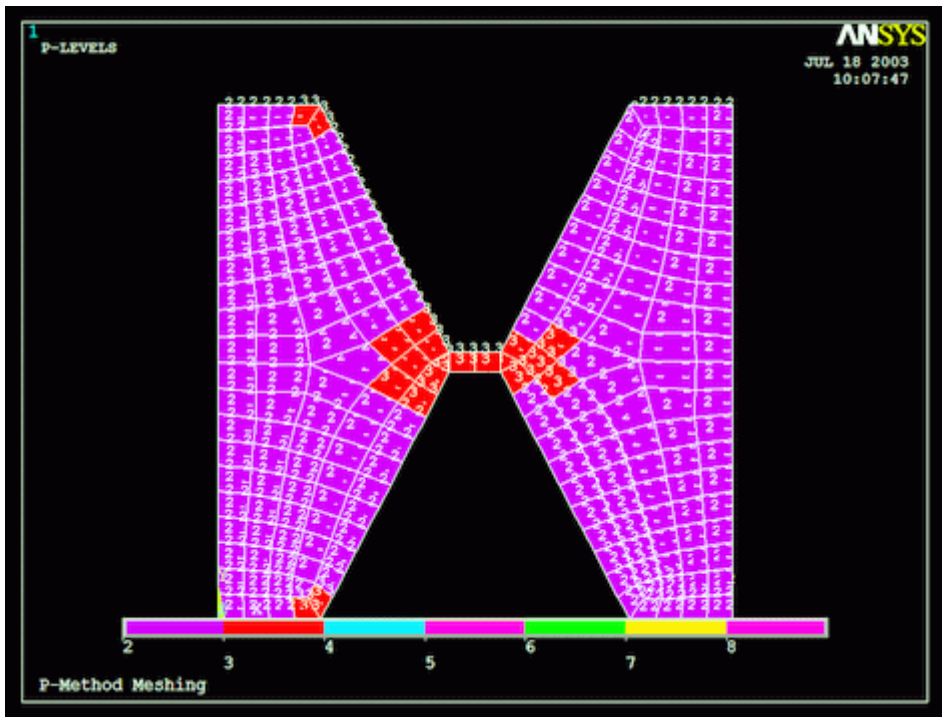
The following stress distribution should appear.



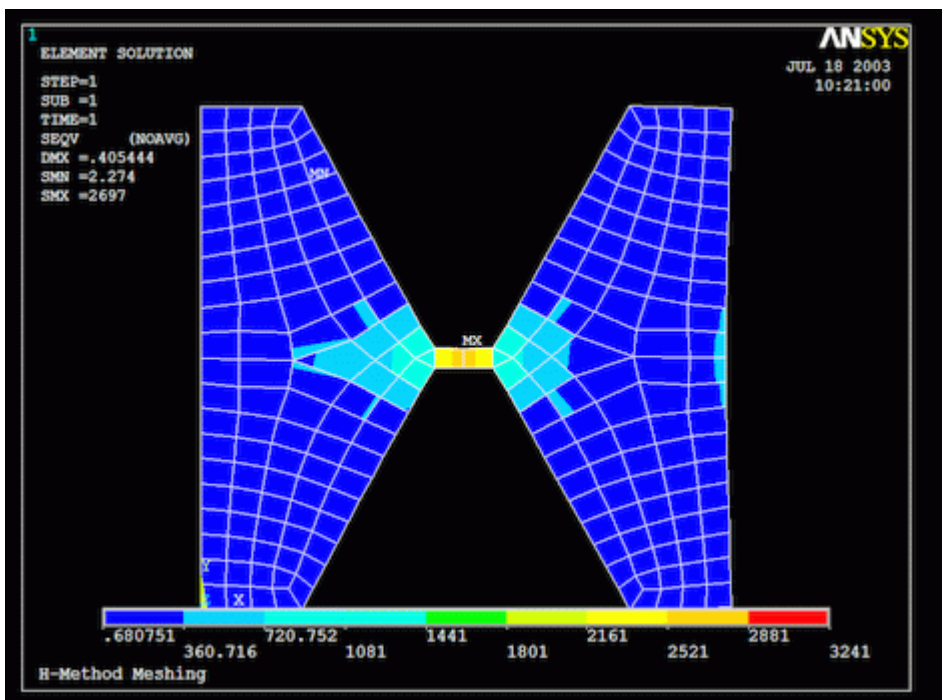
3. Plot p-Levels

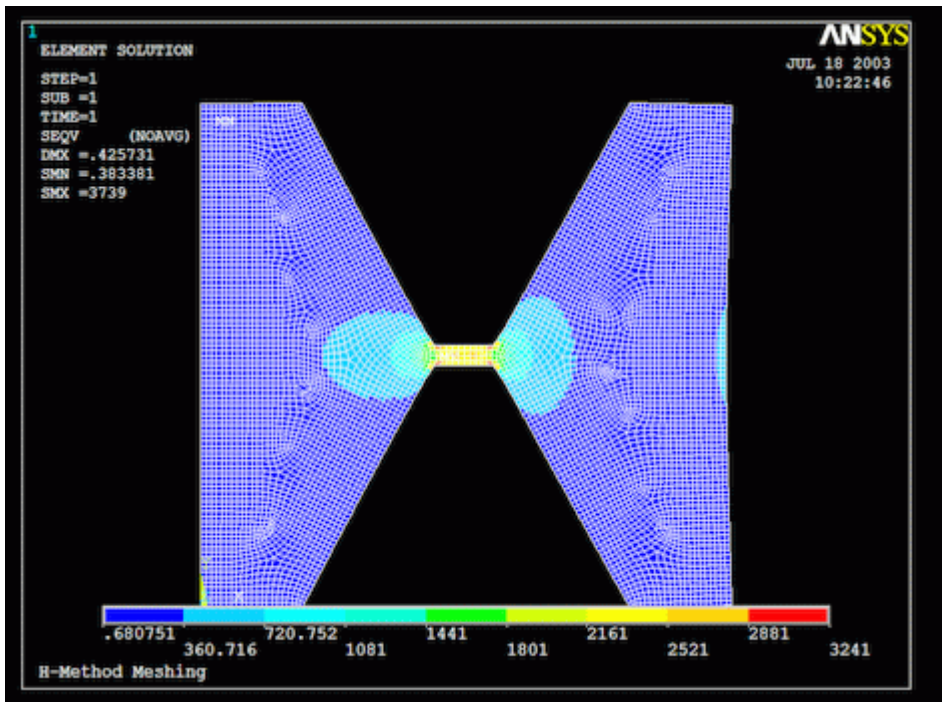
General Postproc > Plot Results > p-Method > p-Levels

The following distribution should appear.



Note how the order of the polynomial increased in the area with the greatest range in stress. This allowed the elements to more accurately model the stress distribution through that area. For more complex geometries, these orders may go as high as 8. As a comparison, a plot of the stress distribution for a normal h-element (PLANE2) model using the same mesh, and one with a mesh 5 times finer are shown below.





As one can see from the two plots, the mesh density had to be increased by 5 times to get the accuracy that the p-elements delivered. This is the benefit of using p-elements. You can use a mesh that is relatively coarse, thus computational time will be low, and still get reasonable results. However, care should be taken using p-elements as they can sometimes give poor results or take a long time to converge.

Command File Mode of Solution

The above example was solved using a mixture of the Graphical User Interface (or GUI) and the command language interface of ANSYS. This problem has also been solved using the ANSYS command language interface that you may want to browse. Open the .HTML version, copy and paste the code into Notepad or a similar text editor and save it to your computer. Now go to '**File > Read input from...**' and select the file. A .PDF version is also available for printing.
