‘p’-type or ‘h’-type Elements?

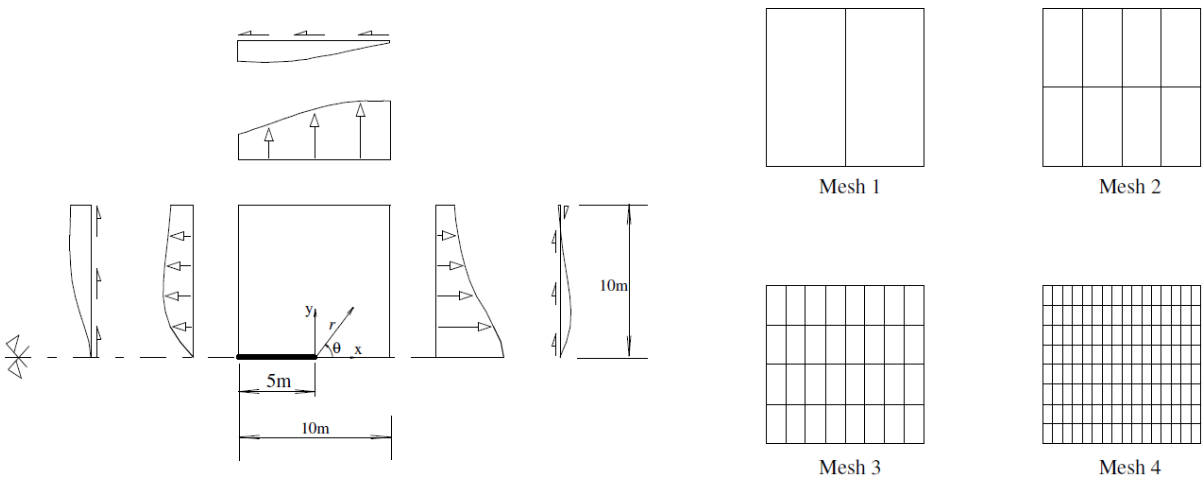
Introduction

The question that the title of this Technical Note poses is one that comes up fairly regularly in the finite element (FE) analyst community [1]. For the practicing engineer it is a question that can only really be answered by exploring the virtues of ‘p’-type elements in a finite element programme that includes these elements. There are commercial codes available that are based on ‘p’-type elements such as **Pro/Mechanica** and there are others, such as **ANSYS**, that have included them in their essentially ‘h’-type element software but now consider them to be obsolete. Finite element researchers often develop ‘p’-type elements and as RMA has experience in this field it was thought to be useful, particularly to those without access to one of these programmes, to present some results comparing ‘p’-type and ‘h’-type refinement for a practical engineering problem.

A Convergence Study with ‘p’-type Elements

The results presented use a hybrid-equilibrium plate-membrane element. These elements differ from the standard conforming displacement element in the manner in which the solution is approximated; they use equilibrating stress fields as opposed to conforming displacement fields. The details of this element type may be found in references 2 and 3 but for the purposes of this note they may simply be considered as a ‘p’-type membrane element.

The problem to be considered is that of a membrane with a crack of infinitesimal width and is shown in figure 1.



(a) The geometry and boundary conditions

(b) The meshes

Figure 1: Geometry, Static Boundary Conditions and Meshes for the Crack Problem

The extent of the crack is illustrated by the thick line and the boundary tractions are evaluated from the following stress field which is both statically and kinematically admissible i.e. it is a closed-form solution for the problem and has been plotted in figure 2.

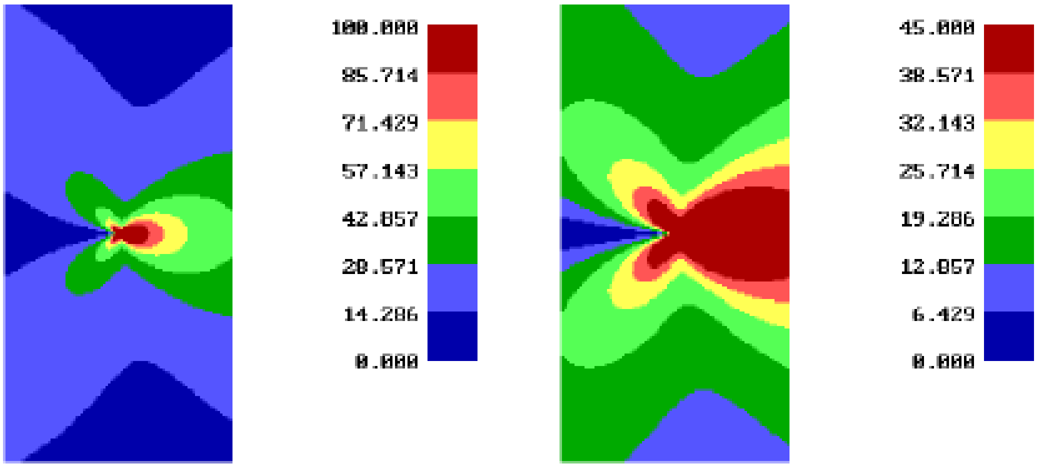
$$\sigma_x = \frac{100}{\sqrt{r}} \cos\frac{\theta}{2} \{1 - \sin\frac{\theta}{2} \sin\frac{3\theta}{2}\}$$
$$\sigma_y = \frac{100}{\sqrt{r}} \cos\frac{\theta}{2} \{1 + \sin\frac{\theta}{2} \sin\frac{3\theta}{2}\}$$
$$\tau_{xy} = \frac{100}{\sqrt{r}} \sin\frac{\theta}{2} \cos\frac{\theta}{2} \cos\frac{3\theta}{2}$$

Symmetry (kinematic) boundary conditions are applied to the line of symmetry and for a Young's Modulus of $210N/m^2$, Poisson's Ratio of 0.3 and material thickness of 0.1m and using a plane-stress constitutive relation the strain energy U for the symmetric half shown is 62.442963Nm.

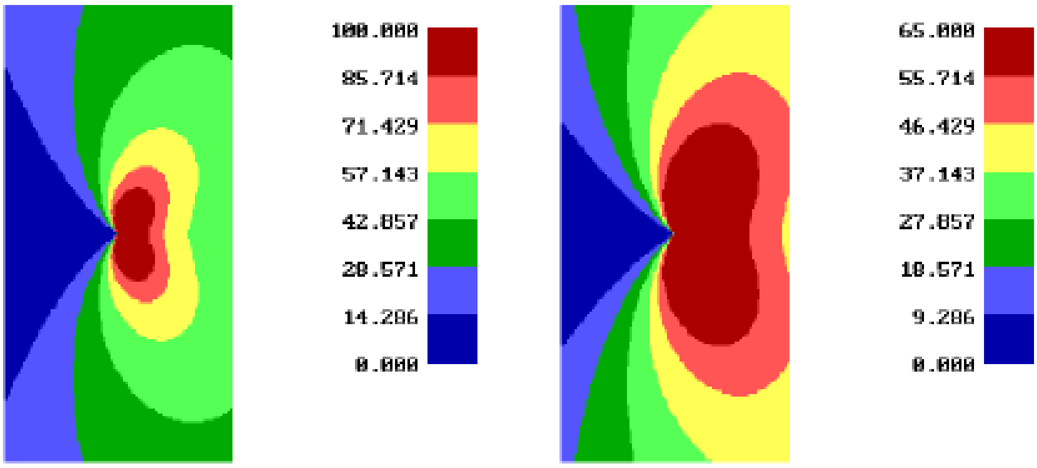
The convergence study looks at four meshes (figure 1) and for elements with degree of (statically admissible) stress field from 1 (constant) to 10. The finite element strain energies are presented in table 1. It is interesting to note that the finite element strain energy values are all greater than the exact value. This is a characteristic of the element type being used (equilibrium) and the manner in which it is loaded (force-driven). In contrast to this a force-driven displacement element will produce strain energy values less than the exact value; for Mesh 4 the standard eight-noded displacement element produced a strain energy of 61.056022Nm.

	Mesh 1		Mesh 2		Mesh 3		Mesh 4	
p	U_h^E	dof	U_h^E	dof	U_h^E	dof	U_h^E	dof
1	73.313361	28	67.107610	88	64.577097	304	63.470253	1120
2	66.731729	42	64.333771	132	63.356702	456	62.892746	1680
3	64.713918	56	63.525367	176	62.974219	608	62.706200	2240
4	63.909238	70	63.153881	220	62.794140	760	62.617507	2800
5	63.471788	84	62.946635	264	62.692650	912	62.567278	3360
6	63.207607	98	62.819312	308	62.629937	1064	62.536154	3920
7	63.034270	112	62.735024	352	62.588271	1216	62.515438	4480
8	62.914096	126	62.676244	396	62.559142	1368	62.500936	5040
9	62.827305	140	62.633612	440	62.537979	1520	62.490399	5600
10	62.762547	154	62.601702	484	62.522117	1672	62.482472	6160

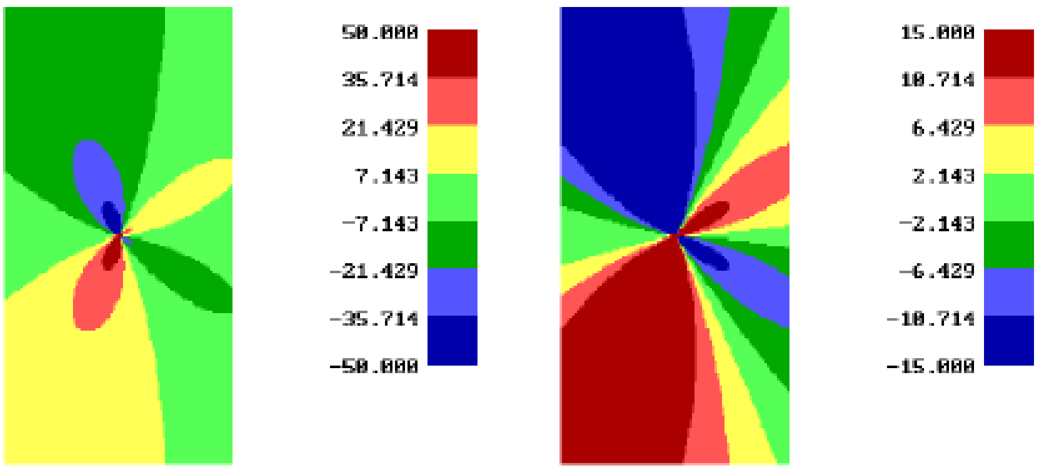
Table 1: Finite Element Strain Energies for the Crack Problem



(a) Stress component σ_x



(b) Stress component σ_y



(c) Stress component τ_{xy}

Figure 2: Contours for Closed-Form Solution (Two Contour Ranges Shown)

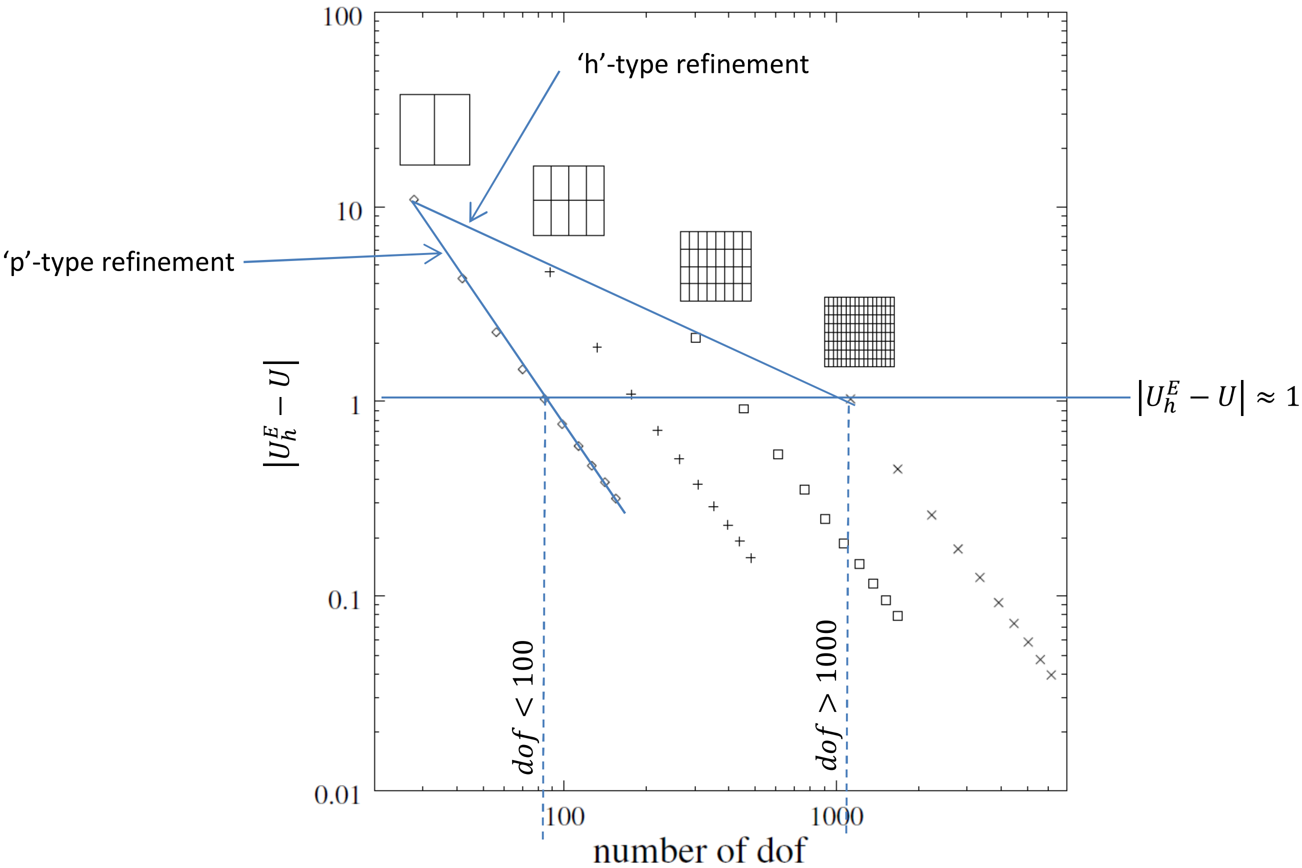


Figure 3: Convergence of the Error in Finite Element Strain Energy against Number of DOF

The error in the finite element strain energies is plotted in figure 3 against the number of degrees of freedom (dof); the graph uses logarithmic scales. In this figure lines have been added representing 'h'-type and 'p'-type refinement starting from Mesh 1 with p=1 (constant stress fields). The important point to note here, and the reason why 'p'-type elements are considered so attractive, is that the rate of convergence for 'p'-type refinement is significantly greater than for 'h'-type refinement. Another way of expressing the virtue of 'p'-type refinement is to take a horizontal line in the figure, which represents constant error, and compare the number of DOF required for the two methods of refinement. For the horizontal line in figure 3, which represents a unit error, the number of dof required for the two methods of refinement are less than 100 (Mesh 1, p=5) and greater than 1000 (Mesh 4, p=1), i.e. the number of dof for similar error is an order of magnitude more when using 'h'-type refinement!

To illustrate the convergence of the stresses for this problem the shear stress for a number of finite element models is shown in figure 4. Although 'p'-type refinement has a faster convergence rate, it is of interest to consider qualitatively the stress fields obtained in the four meshes for the same energy of the error. Figure 5 shows the shear stress for the four meshes with near unit error. It appears that the continuity and quality of the shear stress does improve with the number of dof achieved with 'h'-type refinement, i.e. a mixture of 'p'-type and 'h'-type refinement might be optimal.

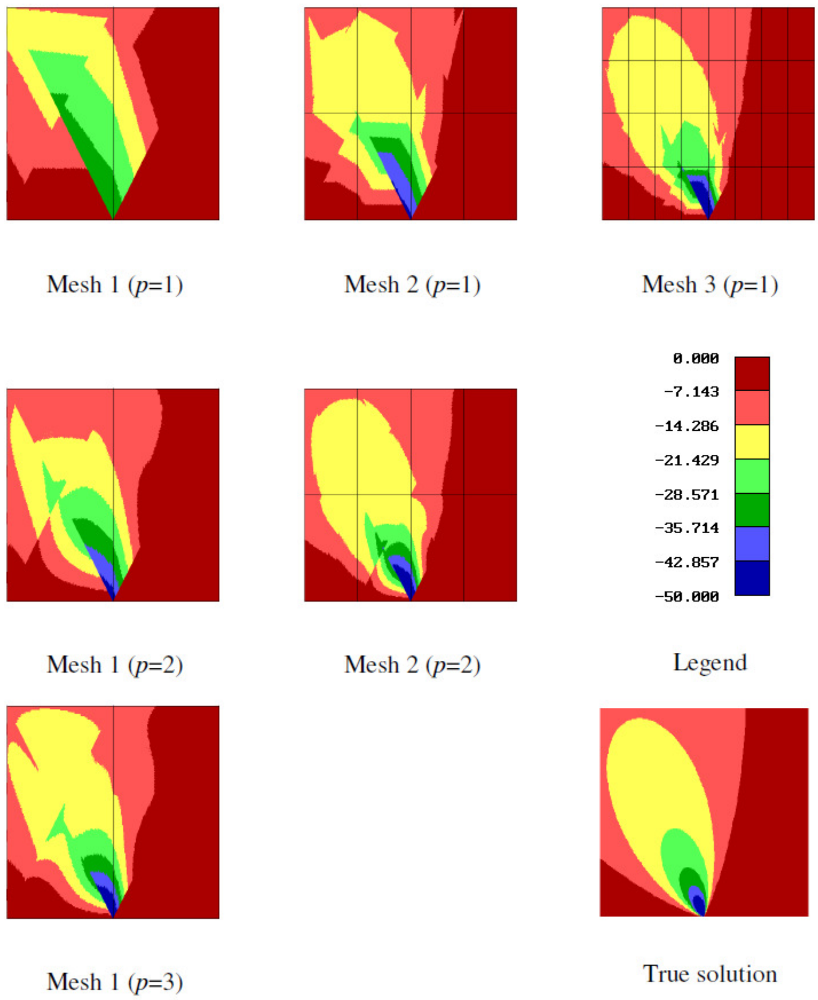


Figure 4: Contours of Shear Stress for Finite Element Models

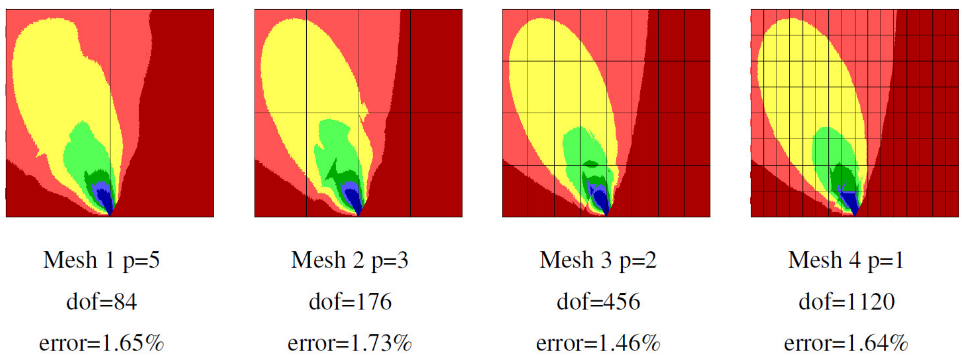


Figure 5: Contours of Shear Stress for Similar Errors in Strain Energy

The displaced shapes for the finite element models are compared in figure 6. Note that whilst hybrid-equilibrium elements satisfy equilibrium in a strong sense, the edge displacements are not continuous at the vertices. The discontinuities in vertex displacement are seen in the figure but converge (get smaller) as the mesh is refined.

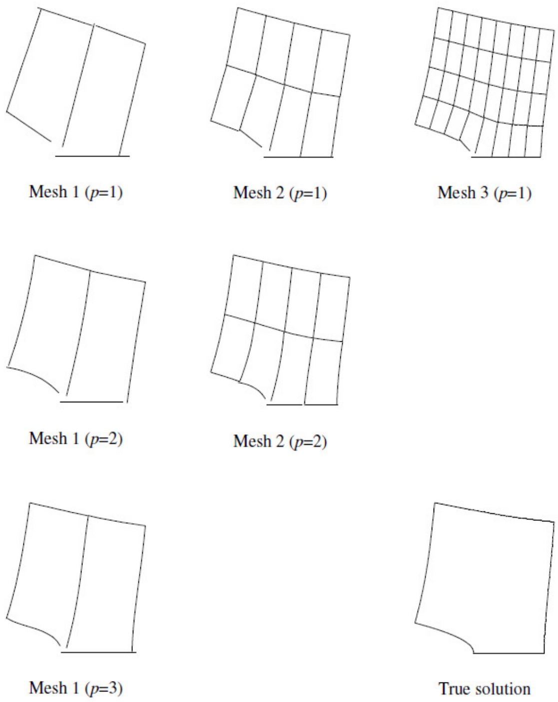


Figure 6: Displaced Shapes for Finite Element Models

Closure

This Technical Note has recycled some previously generated results to demonstrate the potential virtues of using a 'p'-type element. The original question that prompted the preparation of this note questioned the reasoning behind the two methods of refinement. Whilst the results for 'p'-type elements are shown to be impressive, it must be remembered that, in general, a mixture of 'p' and 'h'-type refinement will be optimal – consider the case of stress concentrations where more elements, 'h'-type refinement, and higher-degree elements, 'p'-type refinement, will be required around the concentration – e.g. as in figure 5.

References

[1] What is the reason behind a H-method and a P-method in FEA?

https://www.researchgate.net/post/What_is_the_reason_behind_a_H-method_and_a_P-method_in_FEA?

[2] E.A.W. Maunder, J.P.B. Moitinho de Almeida & A.C.A. Ramsay, ***A General Formulation of Equilibrium Macro-Elements with Control of Spurious Kinematic Modes: The Exorcism of an Old Curse***, International Journal for Numerical Methods in Engineering 12/1998; 39(18):3175 - 3194.

[3] A.C.A. Ramsay & E.A.W. Maunder, ***Sub-modelling and boundary conditions with p-type hybrid-equilibrium plate-membrane elements***, Finite Elements in Analysis and Design 01/2006; 43(2):155-167.

Note: references 2 and 3 are available at ResearchGate:

https://www.researchgate.net/profile/Angus_Ramsay