APPENDIX : THE REFINING PROCESS

Jim's Example, consisting of a dual-thickness plate loaded in-plane, has been used to demonstrate the more important aspects of setting up a two-dimensional mesh, and to illustrate how a typical data file is prepared for the linear elastic program FEM1. The demonstration mesh was far too coarse to be considered a realistic approximation to the prototype; the present intention is to progressively subdivide - ie 'refine' - that mesh in a bid for greater realism, and at the same time to highlight some typical results which might be expected of such a refining process.

The undeformed meshes are illustrated opposite with a few node and element indices, the latter being italicised. Some overall particulars of the meshes are :

mesh subdivision step	0	1	2	3
number of nodes	11	31	101	361
number of elements	6	22	85	328
semi-bandwidth of assemblage equations	12	18	30	54
program - maximum nodes*semi-bandwidth	250*72	250*72	250*72	361*54
solution time, compiled (sec) - Macintosh SE	20	82	427	3098
- Macintosh SE/30	4	17	88	621
number of pages of output	1/3	1	3	10

An ideal square mesh forms a useful basis for estimating, as refining proceeds, the increase in the number of nodes and in other parameters. For such a square mesh :

	node number	element number	semi-bandwidth
if, at a certain stage of subdivision	n	e	w
then, after a further subdivision	$(2\sqrt{n}-1)^2 \rightarrow 4n$	$4e$	$2(w-3) \rightarrow 2w$

The 'area' required for the assemblage equations in memory is proportional to the product of node number*semi-bandwidth, and thus increases by a factor of 8 approximately over each subdivision of the square mesh. Jim's mesh is seen to be very similar to this. It will be noticed that the ratio of solution times also approaches this multiple.

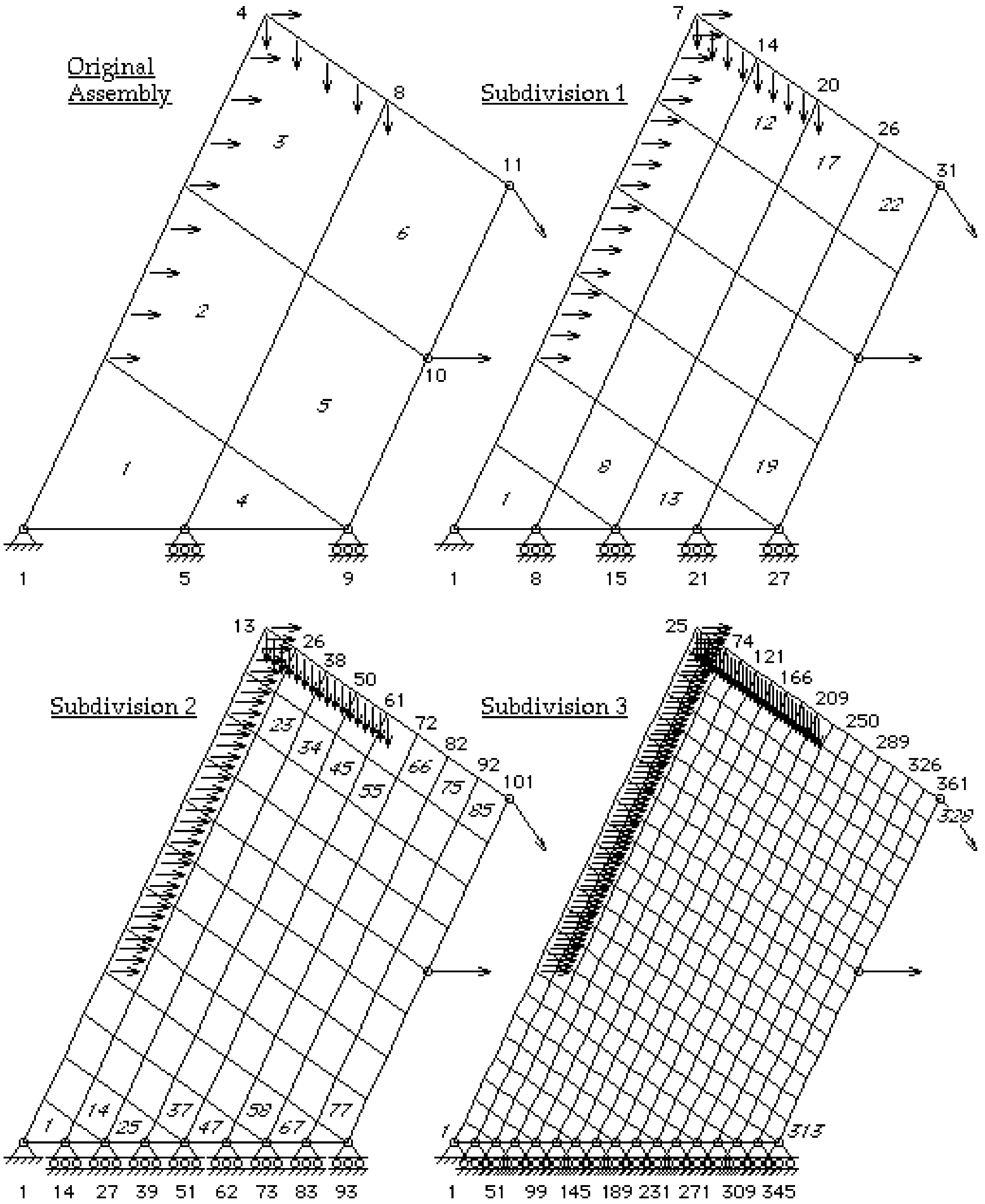
The standard program has been compiled to cater for maxima of 250 nodes and 72 semi-bandwidth - this was extended readily to cater for the last stage of refinement. Clearly, further subdivision is rendered impractical on the basic SE by time constraints as much as by memory limitations - though the speed of the SE/30 nullifies this argument. Solution time using interpreted code is about 5% longer than that above; solution on a Macintosh Plus takes only a few seconds longer than on an SE.

The outputs from these refinements have not been included because the vast majority is of absolutely no interest whatsoever - a suitable post-processor is really a necessity - however the following table compares some facets of the solutions :

mesh subdivision step	0	1	2	3
error in reconstituted (6,-9)kN force, N	0, 0	0, 0	0.01, 0.02	0.01, 0
deflection at (6,-9) kN force, mm	(0.33,-0.48)	(0.41,-0.62)	(0.46,-0.67)	(0.52,-0.77)
force at node with specified 0.01mm deflection, kN	17.4	15.7	14.6	13.7
peak equivalent stress, MPa	126	237	342	503

The round-off errors, as reflected in the reconstituted concentrated (6,-9)kN force, are in all cases negligible. The arithmetic procedures of FEM1 would thus seem adequate for the scope of problems allowed by other program limitations.

Neither the deflection at the node where the concentrated force is applied, nor the force at the node where the specified horizontal deflection is 0.01mm, demonstrate the asymptotic tendency which one would expect to follow from the refining process. The reason for this unusual behaviour lies in the example, not in the method or program. This may be appreciated from the deformed meshes, shown overleaf, the most refined mesh in particular demonstrates large local deformations at the concentrated effects - the sharp change in slope where the concentrated force



JIM'S EXAMPLE - UNDEFORMED MESH

limits the nodal deflection to 0.01mm, and the deformation of the element where the (6,-9)kN concentrated force is applied. In practice such concentrated effects would not occur; for example the force might be applied through a two-force member welded to the plate, or via a pin bearing upon the interior of a hole in the plate, and so on - ie effects are distributed over the plate. Jim's example is therefore not realistic. (It was never meant to be realistic; it started life purely as an example for data file layout.) Despite the impractical quirks of Jim's plate, it is quite clear that increased refinement enables the finite element model to better approach realism. In other words - the more degrees of freedom, the more realism.

The unusual behaviour of Jim's plate is evident also in the stresses which occur in the finite element model. The table above includes values of the peak equivalent stress, which increase as refinement proceeds without displaying the asymptotic tendency expected from a 'practical' model. Further aspects of stressing may be appreciated from the stress patterns. The pattern option of FEM1 allows the user to specify a maximum stress contour, and each element is patterned according to the ratio which its equivalent stress bears to this maximum :

ratio equivalent stress / maximum stress contour	$< {}^{1}/_{3}$	$\geq {}^{1}/_{3}$	$\geq {}^{2}/_{3}$	≥ 1
element pattern	light gray	gray	dark gray	black

The four mesh patterns are shown with a common maximum stress contour of 275MPa, a value which was chosen somewhat arbitrarily to show desired features - it happens to be about the geometric mean of the four peak equivalent stresses.

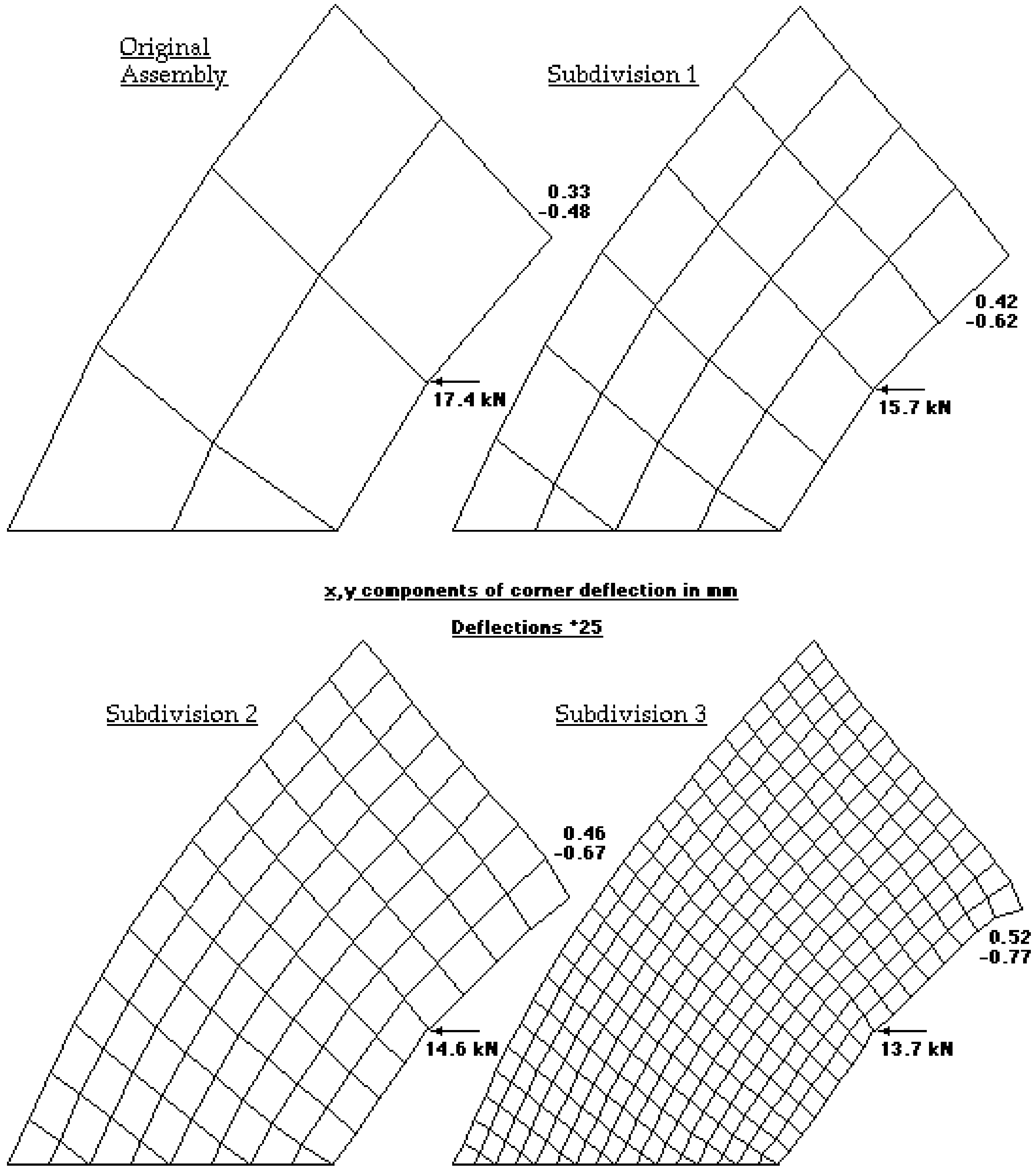
The plate's behaviour is sensibly that of a cantilever, with high stresses on the LH and RH faces. It is interesting that the maximum stress occurs at the base of the original mesh and of the first two refinements, but the peak stress of the most refined mesh occurs at the point of application of the concentrated (6,-9)kN force. This is another quirk of Jim's unrealistic plate - though a geometric singularity like a crack front would lead to similar results.

There is not much more of any immediate interest in the above.

Subdivision 4 has been devised to duplicate the fine mesh of subdivision 3 in way of the stress concentrations and large deflections along the RHS of the plate, without the attendant disadvantages of the more refined meshes regarding memory usage and execution time. Subdivision 4 was achieved by adopting a fine mesh in the areas of interest, and a coarse mesh in the areas of little interest. The mesh is shown together with the deformed shape of the plate and the stress contour patterns.

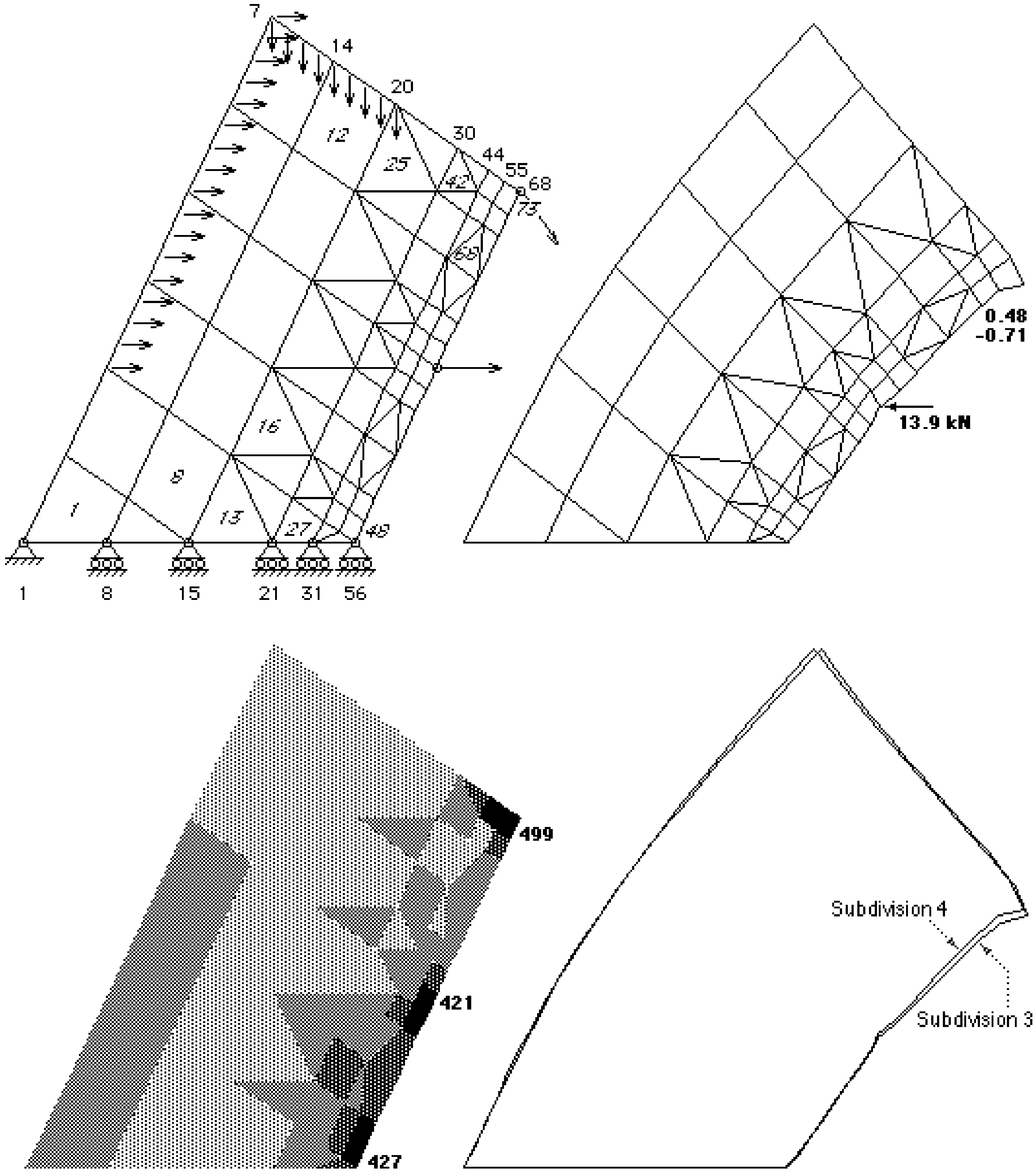
Comparing the output for Subdivision 4 with the above, there are 68 nodes, 73 elements, a semi-bandwidth of 52, and just over 2 pages of output; the solution took 466 s and so in many respects the mesh is similar to Subdivision 2. However the selective refining means that the performance is more in line with Subdivision 3, as the force at the 0.01mm deflection node is 13.9kN, the deflection at the (6,-9)kN load is (0.48,-0.71)mm, and the peak stress is 499MPa - the highly stresses areas are seen to be very similar to those of Subdivision 3. The similarity between Subdivisions 3 and 4 is further borne out by the superposition of their deflected shapes.

The conclusion from this last step is that it is possible to model realistically localised stress concentrations by means of meshes which have been selectively refined in way of the high stress/strain gradients, without incurring the memory demand and execution time penalties associated with globally refined meshes.



JIM'S EXAMPLE - DEFORMED MESH

JIM'S EXAMPLE - STRESS CONTOURS



JIM'S EXAMPLE - PREFERENTIAL REFINEMENT (Subdivision 4)