# Mentor Expectations Guide: A Companion to the Mentor Guide
# T72S03: Perform Simple Finite Element Analysis

Compiled by Rick Bradford, Last Update: 29th September 2008

These Expectations Guides are a record of what the author of the Mentor Guide had in mind in posing the questions. They are not a definitive statement of the 'right' answer.

1. **Explain the mathematical formulation of the implicit finite element method for stress analysis.**

1.1 Describe in very broad terms how a finite element program goes about solving a stress analysis problem

The intention of this question is merely that the candidate should appreciate…

1) That the basic technique is to divide the continuum into a finite number of smaller regions (elements);

2) That these elements are connected together only at certain nodes (e.g. corners, mid-sides);

3) And hence there are a finite, discrete, number of degrees of freedom in the modified problem which can thus be formulated in matrix form and thus 'understood' by a computer.

4) As an immediate corollary, the candidate might be expected to see that gaps between the elements, or overlaps of adjacent elements, will in general occur between the node positions. Hence, the method is an approximation! Obvious, but rather important.

1.2 Define the term 'shape function' with examples

Candidates should be able to explain that the shape function specifies the displacement vector at any point within an element in terms of the nodal displacement vectors. They should be able to express this in algebraic form. A key requirement is that candidates appreciate that displacements are defined at all points within the continuum of the element interior.

Candidates should be able to give examples in isoparametric coordinates (in other words, for a square element). A key requirement is to appreciate that, in the isoparametric approach, the contribution is determined by functions like $(1-\eta)(1-\xi)$ [2D linear elements with nodes at $\eta, \xi \in \{0,1\}$] or like $\eta\xi(1-\eta)(1-\xi)/4$ [2D quadratic elements with nodes at $\eta, \xi \in \{-1,0,1\}$] which are non-zero at only one node, and zero at all other nodes.

1.3 State how the stresses and strains within an element are found from the shape functions and nodal displacements

This sub-role/question is concerned with linear elasticity only. Candidates should know that, since we have the displacement everywhere within the element, from 1.2, the strains follow simply by differentiation. [The required accreditation in T72S01 implies this knowledge]. Candidates should have enough mathematical nous to understand that it is the shape functions which are being differentiated, the nodal displacements being constants for a given element. They should be able to write this down algebraically, at least approximately (absolute precision not essential).

Candidates should know (one hopes!) that the stresses then follow from the strains by Hooke's Law. Again, knowledge of the 3D version of Hooke's Law can be assumed by virtue of accreditation in T72S01.

A good brain teaser at this point is to ask "why should the stresses be continuous across an element boundary"? The answer (I think) is that this depends purely on a sufficiently small and smooth variation of nodal displacements between the two elements – and hence on the elements being sufficiently small. In a real FE analysis, the stresses are *not* continuous, of course.

1.4 Explain the terms ‘linear element’ and ‘quadratic element’ and state how the displacements, stresses and strains vary across such elements

This follows from 1.2 and 1.3. The question is really to check that people have realised that ‘linear’ and ‘quadratic’ apply to the displacements, and hence that the stresses and strains are constant and linear respectively.

1.5 Write down the key equations defining the algorithm used to formulate the implicit finite element method for linear elastic stress analysis

Precise mathematics is not expected for this one. Candidates might be expected to explain the virtual work approach, though any valid derivation is acceptable. They should appreciate that the total work function can be written as a volume integral over the inner product of stress and strain (increment). This is clearly quadratic in the nodal displacements (confining attention to linear elasticity). Hence, the derivative of this work function with respect to a nodal displacement gives the corresponding generalised force which must be linear in the nodal displacements i.e. $F = Ku$ in matrix notation.

Mentees should understand that the (matrix) factor of proportionality is the stiffness matrix. They should be able to derive something that looks roughly like $K = \int B^T DB.dV$ in matrix notation, where B is a matrix of shape function derivatives and D is a matrix of elastic moduli. It is important to get them to write down something involving a volume integral in order to ask what a Gauss point is - and to follow that up with “why is integration required?”

For interest only (and well beyond expectation) there is a note on this web site which derives the form of the stiffness matrix without using virtual work, instead showing K to be a matrix representation in discrete space of a certain differential operator.

1.6 State how the problem, once expressed in discrete form, is solved

Simply stating “invert the stiffness matrix” is almost enough.

However, candidates should have some appreciation that this is adequate only in the (most unusual) case that all the nodal forces are known and all the nodal displacements are unknown, so that $u = K^{-1}F$ is the solution. More generally, some displacements are known and some forces are known, so that the knowns and unknowns appear on both sides of $F = Ku$. I believe different codes have different ways of dealing with this. The key step is always inversion of the stiffness matrix, but there may be rather more jiggery-pokery going on than Users often recall. The Mentor would be in a stronger position if he knows a little about said jiggery-pokery. Basically it involves shuffling all the knowns to one side of the equation and all the unknowns to the other side.

1.7 Outline an algorithm which may be used to invert the stiffness matrix

It is a good idea to start this one by asking what step usually takes the bulk of the computer's analysis time. It is because the answer to this is the stiffness matrix inversion that we have an interest in the algorithms employed. I would expect candidates to be able to describe Gaussian elimination. This is the most obvious method. It is equivalent to solving simultaneous equations by systematically eliminating a variable at a time, but expressed in matrix notation. The advantage of getting a candidate to flog through this algorithm is that it becomes immediately clear both what the bandwidth is, and why the bandwidth is important as regards the computation time. [Off the top of my head, I think you can see that the computation time must be proportional to $b^2 N$ , where N is the number of nodes and b is the bandwidth – but I could be wrong].

The Mentee should appreciate the importance of bandwidth. This is less common than it used to be, as computers have become faster and bandwidth is no longer the very severe constrain that it once was. However, we still run non-linear jobs which take the order of months – so it is still important to know what the computer is doing all that time. Also, for a given mesh, the bandwidth can be very different if the elements/nodes are ordered differently. Again appreciation of this is lacking these days, since FE codes tend to optimise the ordering for the User. But the Mentee should understand for, say, a rectangular mesh of N x M elements which ordering is optimal – and roughly what the resulting bandwidth is. (See also 2.4)

Candidates should be aware that there are more sophisticated algorithms out there. They may be expected to have heard of sparse solvers and for what sort of stiffness matrices they might be appropriate.

1.8 Describe the isoparametric method for dealing with curved elements

I need to check this myself, but if I remember correctly "isoparametric" means that the same mapping functions are used for the element shape as for the displacements. Thus, the boundaries of the element are defined by the same functions of the nodal coordinates as the displacements are in terms of the nodal displacements, i.e. via the shape functions. This means that quadratic elements have parabolic shaped sides in general. An important implication of this is that they are not circular! Hence, an accurate representation of a circular arc (e.g. a notch root radius, or simply a pipe) requires a number of elements. (What's the rule of thumb – at least 4 elements per $90^o$? I can't remember).

Candidates should be aware that the terms in the stiffness matrix are evaluated by first mapping to a unit square (or cube), and that a geometrical factor, the Jacobean, results. It is the conversion factor between the volume element of the mapped geometry and that of the true geometry. Thus, the deviation of the Jacobean from unity is a measure of the curvature or distortion of the element (see below).

## 2. **Describe how to design a good finite element mesh for linear elastic stress analysis**

2.1 Discuss the advantages and disadvantages of linear, quadratic and higher order elements for linear elastic finite element analyses.

This is the classic question: "Is it better to use a 20 x 20 mesh of linear elements or a 10 x 10 mesh of quadratic elements?" The number of degrees of freedom and bandwidth are very similar, so the analysis time is similar. The 10 x 10 quadratic mesh is better, though. This is ingrained in custom & practice, i.e. we usually use quadratic elements. Why? My argument draws an analogy with fitting an arbitrary curve using quadratic splines. This is much better than piecewise linear. So why are cubic elements not better still? I think the only answer is that experience has shown quadratics to be optimal. Ultimately you could just use one $20^{th}$ order element – but this is obviously poor because it amounts to imposing a $20^{th}$ order polynomial, and high order polynomials are usually seriously silly between the fitted data points.

2.2 Discuss the advantages and disadvantages of uniform element size versus varying element size across the mesh

This question is best introduced in the form of asking the Mentee to mesh an example geometry. My favourite is a compact tension specimen. Candidates must appreciate that it is desirable to have a refined mesh in the area of interest. For a CTS this means the crack tip and the ligament. However, elements near the upper and lower edges of the specimen can be less refined. In particular, there would be little point in expending effort in refining the mesh near the loading holes. In fact, it may be better to omit the holes altogether. It may be appropriate to ask why this is valid (St.Venant's Principle – local geometry affects the stresses only locally – candidates who explain the relevance of elliptic versus hyperbolic differential equations at this point get no extra points for showing off).

The sub-text is, of course, that accurate results are obtained only for meshes in which the element size is not too great compared with the stress gradients. The alternative to refinement is therefore to use this minimum required refinement everywhere – the disadvantage being computer time. I think this is the only disadvantage to using a uniform mesh, but it is a prohibitive one in most practical cases.

The disadvantages of using a refined mesh are: (a)User effort to define the mesh; (b)There may be more doubt about whether the mesh is adequate in regions of low refinement; (c)Adjacent element sizes must not differ too much (see below), so this may introduce a source of inaccuracy.

2.3 State the routine checks for good mesh geometry

1) Refinement in the required places;

2) Ratio of adjacent element size is not too great (what's the guidance on the upper limit?);

3) Element aspect ratios are not too great (this differs considerably in 2D and 3D if I remember correctly – again what is the advised upper limit?);

4) Element included angles not too small (lower limit – is it >30$^{o}$? Do check, I can't remember);

5) Do not mix left handed and right handed elements. Preferably use only right-handed (indicates as having positive area or volume);

6) Jacobeans are within advised minima/maxima.

7) There must be more…add your own favourites. Most decent proprietary FE codes will produce a set of diagnostics on request.

2.4 Describe how node or element numbering or ordering can affect analysis time

Node or element numbering or ordering can affect the bandwidth (depending upon the software used). For a simple 3D block of material modelled as (say) 100 elements by 20 elements by 5 elements, candidates should be able to define the orderings which minimise the bandwidth and hence time. [In this case, getting it wrong could make a difference in analysis time of up to $(100\times 20/20\times 5)^2 = 400$, so a 5 minute job ends up as 33 hours.

People may be used to using an automatic bandwidth optimiser these days. Do they know how good these are? In the latter example, would the optimiser convert the 100 x 20 case automatically into the optimal 20 x 5 case?

2.5 Discuss the options for the numbers of integration (Gauss) points per element

Not a desperately big issue, but people should be aware that there *are* options, e.g. so-called reduced integration, and the relation to 'mechanisms' (not that I can remember what these are, amittedly). Has the Mentee tried comparing reduce integration and standard Gauss point configurations? What difference was found?

2.6 Discuss the advantages and disadvantages of different commercial mesh generation packages

This is open ended, with no particular expectation other than the candidate displays some knowledge of the issue. The obvious packages for BEG/SAG at the time of writing are ABAQUS-CAE and FEMGEN, but people should be encouraged to discuss other software which they have experience of. The issue here is awareness. People should display some knowledge of the relative strength and difficulties in meshing – as opposed to geometrical modelling. In my (extremely limited) experience, there can be problems with meshing when using 'top down' geometry modelling approaches like ABAQUS-CAE and PATRAN. With "bottom up" approaches like FEMGEN, which build the mesh upwards from the elements, the disadvantage is the absence of a nice, powerful, geometry modeller. But there's an infinite amount to go at here. I'm too ignorant to say much.

This is an important area as regards the cost of an FE analysis. These days the cost is dominated by mandays, not computer time. And, at least for 3D jobs, the bulk of the man-time is probably spent on developing the mesh.

## 3. **Discuss the interpretation of the output from finite element analyses and potential error traps**

3.1 State what quantities will be continuous between adjoining elements and which discontinuous.

Nodal data (degrees of freedom), such as displacements, rotations and temperatures, are the same for any element containing those nodes - because they are defined for the node rather than the element. An exception is rotational degrees of freedom at a junction between plate or shell elements and 'brick' elements. Rotational degrees of freedom do not exist for the latter. This is a potential error trap (see below).

Quantities which depend upon the shape functions (usually their derivatives) are element data and will generally be discontinuous between elements. Examples are stress, strain and heat flux. (Someone check I've got this right as regards heat flux – it depends on temperature gradient, so I think so).

3.2 Discuss the different types of numerical output available for a given variable, i.e. nodal, element and Gauss point. Discuss their advantages and disadvantages.

This is essentially an amplification of 3.1. It is generally believed that the 'continuum' variables (stress, strain, heat flux) are most accurately evaluated at Gauss points (is this really true? – Discuss! What do the text books say?).

When using post-processors, say for contour plots, there is often a choice between the different types of data. So, as a practical matter, if you ask for a Gauss point based plot but only wrote node/element data to your analysis output file, then you are going to be disappointed. Gauss point plots tend to look rather *pointilliste*, i.e. blocks of colour appear in quarter-element chunks. Node/element contour plots look smoother, but this doesn't mean that they are any more accurate.

The practical advantage of nodal stresses is that they are averages over all the attached elements. Does this mean that they might be more accurate than Gauss point stresses?

3.3 State how the forces of restraint would be expected to vary over a restrained section bearing uniform pressure or tension and composed of quadratic elements

For a single 3D brick element the corner:mid-side node reaction forces are in the ratio -1:4. These must be multiplied by the number of elements attached to the given node. This will be 4 elements per corner node and 2 elements per mid-side node for an element in the middle of a section. Hence the reaction loads, in this example, would be in the ratio -1:2.

For 2D elements the corner nodes have positive reactions and the corner:mid-side node reaction forces are in the ratio 1:4 for a single element.

A good exercise is to ask the Mentee to derive these ratios from first principles using the shape functions. I may put a derivation on this web site – but no peeking!

3.4 Discuss the pitfalls of using a mesh consisting of both shell or plate elements and also solid brick elements

I'm just after the classic 'joining' problem here – though you may have your own issues. Because bricks do not have rotational degrees of freedom, if a shell or plate is just attached to one line of nodes on a brick element then the connection is free to rotate (pin joint). Assuming this is not the intention, it is necessary to 'overlap' the element types – in which case some tweaking of element properties is needed to avoid over-stiffness of the connection.

3.5 Discuss how the stresses on the net section ahead of a crack may be in error in a course mesh

The first element on the crack face (the crack tip element) continues to carry load, though it should not. The reason is that the stresses will be non-zero (in fact large) at the crack tip. And hence the crack face on this element, though it should be traction free ideally, is in effect carrying some load. If the mesh is sufficiently refined the error will be small. For coarse meshes, the net section (ligament) stress will be smaller than it should be and will not balance against the applied load.

3.6 Discuss how elements with reversed topology order might cause errors

This means elements with negative volume. Most programs will warn of this error trap in the 'printed' output. Some errors which might result from this include:-

(a)Centrifugal loads acting on this element will be in the wrong direction (negative mass!);

(b)Energies evaluated from energy densities will have the wrong sign (because of the negative volume). Hence, meshes which consist of some right-handed and some left-handed elements will evaluate the magnitude of the total energy incorrectly.

(c)For the same reason, the calculation of fracture mechanics parameters like J may be in error.

4. **Describe the salient features and potential pitfalls of thermal analyses, thermal stress analyses and the explicit method**.

4.1 State the basic equation of thermal conduction in time-dependent, transient form and describe how it is solved in a finite element program

Something looking like $K\nabla^2 T = -C\frac{\partial T}{\partial t} + q$ will do, where q is the heat production per unit volume, K is the conductivity of the material and C its specific heat. Surface applied heat fluxes or temperatures are applied as boundary conditions. Formulations differ according to whether a steady state or a transient solution is required. The basic formulation of a transient problem involves time-stepping. I'd have to swot up on the spatial formulation myself, but I guess some matrix X is defined which represents the conductivity and depends upon the derivatives of the element shape functions, suitably integrated over volume. The static problem would then look something like $XT = Q$ where T is a vector of nodal temperatures, and Q is a vector of applied heat loads. So solving comes down to inverting X, in analogy to a stress problem. The transient problem could be solved (I guess) using simply $CT(i+1) = CT(i) + q(i) - X(i)T(i)$ repeatedly, where 'i' represents the i'th time point. However, a first order finite difference can be improved using higher order integration (e.g. fourth order Runge-Kutta). The matrix X needs re-evaluating at each time step in general (e.g. due to temperature dependent material properties). Don't quote me on any of this – look it up!

4.2 Discuss what temperature gradients may give rise to thermal stresses, and what the mechanism is which gives rise to thermal stresses

Possibly this one belongs in T72S01 rather than in this FEA Mentor Guide. What I have in mind is that people are likely to answer this question along the lines "thermal stresses arise when the structure has a temperature gradient". This is wrong in two ways. Firstly, thermal stresses can arise at uniform temperature – if the structure is suitably restrained and changes temperature. (Or, alternatively, if the structure has a non-uniform coefficient of thermal expansion). Secondly, temperature gradients do not necessarily give rise to thermal stresses. The exception is a uniform temperature gradient (i.e. a temperature which varies linearly with respect to a Cartesian coordinate system). If such a structure is also unrestrained, then there are no thermal stresses. (It is an interesting exercise to prove this – though probably more relevant to T72S01 than here. It is proved in a note on this web site).

People should be able to explain the formulation of a thermal stress problem, i.e. that it is essentially an applied strain: $\varepsilon_{ij}^{TOT} = \varepsilon_{ij}^{el} + \alpha \Delta T \delta_{ij}$ (isotropic). The modification of the usual continuum stress equations is completed by noting that it is the elastic strains, $\varepsilon_{ij}^{el}$ , which are related to stress via Hooke's Law, but that it is the total strains, $\varepsilon_{ij}^{TOT}$ , which are constrained to respect compatibility. See the notes on this web site for the basic formulation of the continuum stress problem, and how compatibility is formulated.

4.3 Discuss how the time of peak stress may be identified for a transient condition

Good question. Anyone got an answer? People may offer "look for the time of maximum temperature gradient". This may be a rough guide, but as discussed in 4.2, temperature gradient is not always a reliable indicator. In general the pragmatic approach is to analyses a sequence of times and find out which is the most onerous.

4.4 Discuss what time step sizes may be required to solve a transient thermal conduction problem, and how this relates to the required mesh refinement

This is one of my old favourites. It is not hard for people to see that if $t_0$ is a timescale typifying the transient, then the analysis time step size should be such that $t_{step} << t_0$. However, it is less obvious that this has implications for the required element size. The reason is that the solution procedure goes bonkers if the time step chosen is smaller than the time it takes for heat to traverse one element. This transit time is $t_{transit} \approx L^2 / 6\kappa$ where $\kappa$ is the thermal diffusivity given by $\kappa = K / \rho C$ and L is the element size. Hence, the requirements are that, $t_{transit} = L^2 / 6\kappa < t_{step} << t_0$, so this shows that the mesh refinement for a transient problem has to be such that, $L << \sqrt{6\kappa t_0}$ . This is not immediately obvious but extremely important. If you violate this requirement in ABAQUS (and it's easily done) then you get no warnings but the output temperatures are crazy.

4.5 Describe the thermal boundary conditions which typically apply for boundaries adjacent to fluids, insulation or vacuum

This should be easy, but is included for completeness.

(a)Insulation: At the crudest level, treat as adiabatic, but some problems may require a specific effective conductivity to be modelled – in which case a boundary condition is required at the outer surface of the insulation.

(b)Fluid: Candidates should be able to explain what a heat transfer coefficient is (heat flux per unit temperature difference between the surface and the adjacent fluid).

(c)Vacuum (and fluid): Radiation – Stefan's constant times absolute temperature to the power 4 (and also times some emissivity factor). But the surface both radiates and receives heat by absorbing radiation. So, the overall effect may involve a difference term like $T_s^4 - T_e^4$ where $T_s$ is the absolute surface temperature and $T_e$ is the absolute temperature of the environment. There may be lots of different components to the environment, at different temperatures – in which case this radiation issue ceases to be a boundary condition and instead requires the surrounding region to be adopted into the problem and the radiation modelled.

It may be appropriate to ask people why absolute temperatures (K) are often used in thermal problems – the answer being "radiation". There is an error trap here. There can be misunderstandings between thermal modellers, working in K, and structural analysts, who usual work in $^{o}$C.

4.6 Describe the basic formulation of the explicit method for finite element stress analysis

The implicit method is clearly not the way nature solves the problem of how to make a structure respond to the application of a load. Nature does not invert matrices. The explicit method is closer to nature in that it is based on treating the problem as a dynamic problem. Thus, if the static problem to be solved is $Ku = F$ then this is replaced by the dynamic problem $M\ddot{u} + C\dot{u} + Ku = F$ where u and F are not time dependent, but their large-t value is the static solution, for which the velocity and acceleration become zero. The implicit method inverts K to find the displacements u which exactly balance the applied loads F. The explicit method does not have to invert K. Instead it accepts that the forces are not balanced, and hence the structure is dynamic. The explicit method solves the time-dependent problem by time-stepping, finding the new displacements and velocities from the previous ones, e.g.

$\dot{u}(i+1) = \dot{u}(i) + M^{-1}\left[F - C\dot{u}(i) - Ku(i)\right].(t_{i+1} - t_i)$, although I expect a better algorithm like 4$^{th}$ order Runge-Kutta would be used by respectable codes. At this point we are beyond what would be expected for the "simple FEA" SQEP area.

I think it must be necessary to include some damping to ensure that velocities become zero, otherwise the structure would oscillate forever. Critical damping would be most efficient.

4.7 Discuss the benefits and disbenefits of the implicit and explicit methods

The part of the analysis which takes the longest time in the implicit method, the inversion of K, is not required in the explicit method. This potentially gives the explicit method an advantage in terms of computation time. A snag is that it appears from the above equation that inversion of the mass matrix is required – which would potentially take just as long. I *think* the answer to this is that, in the explicit method for a static problem, the mass matrix is approximated as diagonal so inversion is trivial. This is ok for a static problem since the use of the mass-acceleration term is just a ruse for getting to the static solution, the mass term ending up as zero anyway. Whether the explicit method is actually quicker depends upon how long the time-domain iteration takes to converge. Does anyone know from experience in the case of linear elastic problems?

Another advantage of the explicit method is that non-linear behaviour can be incorporated in a straightforward manner, e.g. by replacing Ku by $\int K_T \delta u$ . In contrast, the implicit method needs to introduce specific algorithms to deal with non-linearity, since simple inversion of a single K matrix is no longer enough.

However, the explicit method has really already paid this price – since it needs to employ some algorithm to get the time-stepping process to converge anyway – even if the material response is linear. The books talk about conditionally and unconditionally convergent algorithms. I have virtually no knowledge of these things, but I suspect the temptation to regard the unconditionally convergent algorithms as “a good thing” is probably wrong. I think they are probably just an error trap. They will converge even when the answer is wrong.

So when is the explicit method used? I stand to be corrected, but if a problem is both non-linear and dynamic then the explicit method is probably going to be more efficient. For simpler problems, implicit is generally used. Is custom & practice a fair reflection of the capabilities of the two techniques? I don’t know.

5. **State the Verification requirements for finite element analyses within the context of the BEG/ETB/SAG Quality Assurance arrangements**.

5.1 Identify the procedure which should be used to verify a finite element analysis within BEG's Structural Analysis Group's Quality Assurance arrangements.

At the time of writing (29/9/08) the verification procedure is E/PROC/ENG/BI/048, and the proforma check sheets to be completed are in ENG/FORM/029.

5.2 Specify the key aspects of verification of a finite element analysis to this procedure.

- The central philosophy of the Verification work instruction E/PROC/ENG/BI/048 is that there is no such thing as a verification computer programme or spreadsheet. Fully independent verification of an FE task would therefore require, in principle, a second analysis by an independent SQEP using a different FE code. Unfortunately this is impractical in terms of both time and cost. Hence, the work instruction sanctions "reasonableness checks" as a substitute for the second analysis.
- Self-Checking. There must be documented self-checks before handing the work over to the independent Verifier.
- Verification by an independent accredited SQEP under this sub-role.
- The Verifier must check the standard set of specified items, as listed in the proformas (see above Forms).
- Completion, and signature, of the required verification proformas (see above Forms).
- QA Grade 1: Checking must be by fully independent means, implying a second FE analysis by the Verifier using a different finite element program. This is very unusual. Most FE work will be done at QA Grades 2 or below.
- QA Grades 2a, 2b, 3: In lieu of a fully independent verification, i.e. checking by another person using a different FE code, the procedure E/PROC/ENG/BI/048 requires "reasonableness checks". Some suggestions as to what reasonableness checks might entail are included in the procedure proformas (Sheets 3c and 3d). However, the essence of the "reasonableness checks" is to be creative in thinking what can possibly be used to enhance confidence in the *results* of the analysis. Ideally the Originator should carry out his own reasonableness checks, which the Verifier either verifies or augments with his own checks.
- QA Grade 4: This does *not* mean "unverified". It means that the same checks should be carried out as for a QA Grade 3 task, but these checks are carried out by the Originator. In particular, the Originator is obliged to carry out and document reasonableness checks.

The acceptance of reasonableness checks in lieu of a fully independent verification has been sanctioned with some reluctance. It is really in recognition of the impracticality of repeat analyses in most cases. Verifiers should be aware of this, and therefore give appropriate – heavy – emphasis to the reasonableness checks.

It should also be noted that the use of reasonableness checks in lieu of an independent analysis with a different code is only sanctioned if the program used has sufficient validation and User experience in the application area in question. Programs without the requisite standing for the task will require independent checking against another code.

5.3 Gives examples of the specific checks that would be carried out for a stated example analysis.

The purpose of this question is to allow the Mentor to specify a problem, and have the candidate reel off a list of basic checks plus reasonableness checks. The Mentor should look for wide-ranging thinking as regards the reasonableness checks. The candidate should have the attitude, "there's a mistake in here somewhere, how can I find it?"

This document was created with Win2PDF available at http://www.win2pdf.com.
The unregistered version of Win2PDF is for evaluation or non-commercial use only.
This page will not be added after purchasing Win2PDF.