Ramsay Maunder ASSOCIATES *Finite Element Specialists and Engineering Consultants*

Stress Linearisation for Practising Engineers

Angus Ramsay

Abstract

Background of Stress Linearisation

The process of stress linearisation was originally developed to assist practising engineers working in the design and analysis of pressure retaining equipment (pressure vessels, pipes, pumps, etc.) and, using general finite element models, to predict the stresses in these structures. In a mechanics of materials approach, structural forms such as pressure vessels are considered as shells and the codified assessment procedures, such as ASME, require the stresses to be cast in the form of stress resultants found in a shell member, i.e., membrane, bending shear resultants etc. When a pressure vessel, or similar, is analysed using continuum finite elements, then these stress resultants are not part of the standard output. These stress resultants may, however, be recovered by operating on the finite element stress field by the process of stress linearisation. The stress resultants may then further be operated on to obtain stress measures suitable for comparison with allowable limits prescribed in the codes of practice.

Overview of this Technical Note

This technical note was written with the aim of providing some background to stress linearisation for readers of the NAFEMS Benchmark Magazine tackling the sixth NAFEMS Benchmark Challenge. It attempts to describe the method in the context of the lower bound theorem of plasticity. This theorem offers the engineer a safe approach to structural design provided a stress field can be found that is everywhere in equilibrium with the applied loads and which does not violate the yield criterion. Whilst stress linearisation leads to a set of stress resultants on a design section, which is in equilibrium with the applied load, the process filters out the self-balancing part of the distribution and therefore pointwise equilibrium along the design section is lost. As such, strict appeal to the lower bound theorem of plasticity is no longer possible unless the material can be assumed to be sufficiently ductile to redistribute the stresses from the simplified linearised distribution to the true non-linear distribution.

This note is concerned with the method of stress linearisation rather than with any specific application. The author has therefore chosen to explain the approach through the example of uniform thickness plane elasticity problems rather than the axisymmetric form more applicable to pressure vessel analysis. As conforming finite element (CFE) models do not generally guarantee strong equilibrium, the convergence of stress resultants with mesh refinement is examined. Whereas mesh refinement leads to convergence stress resultants for ‘smooth’ stress fields, convergence appears less well behaved when the design section goes through a stress singularity.

Introduction

The engineering designer is concerned with ensuring that his/her design has sufficient ***strength*** to withstand any loads likely to be seen be the structure, and sufficient ***stiffness*** not to deflect excessively in service. In Limit State Design, also called Load and Resistance Factor Design, the strength constraint is covered as an Ultimate Limit State (ULS) condition whereas the stiffness constraint is covered by a Serviceability Limit State (SLS) condition. Both conditions need to be satisfied for a design to be considered serviceable and safe. Of course, there are other conditions that may need to be considered such as structural *stability*, e.g., buckling of members in compression, and fatigue failure where a structure undergoes cyclic loading. Of all these conditions, however, satisfaction of the ULS condition of collapse is essential for a safe design. Depending on the structural member, and how it is loaded, one or other of the limit states will govern the design. For example, the cross section of a beam used in a bookshelf might well be governed by the SLS condition of maximum deflection with the load to cause collapse being significantly greater than the load to cause an unacceptable deflection.

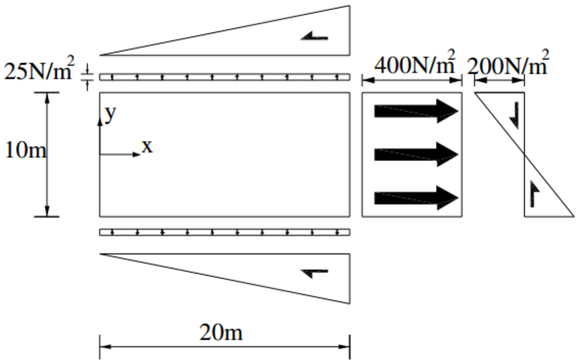
Whilst much engineering design is based purely on *elastic analysis*, there is often significantly more strength to be realised from a ductile structural member if it is allowed to be loaded up to plastic collapse. A uniformly loaded and simply supported beam with a rectangular cross section, for example, can take 1.5 times the load to cause first surface yield before a plastic hinge is developed at the centre of the beam and collapse occurs. In limit state design, the ULS condition of collapse may be tackled through *plastic analysis* in order to realise this additional strength. In a situation where ULS governs the design, a more efficient design, in terms of material utilisation, can thus be achieved through consideration of how the structural member actually collapses.

The ULS condition requires the engineering designer first to find a stress field that is in equilibrium with the applied loads. Having obtained this then he/she must ensure that the design has adequate strength to resist these stresses. There are, for a continuum, an infinite number of equilibrium stress fields, which balance the applied loads for a given structural problem. This is the case because the engineer may always add a *self-balancing stress field* to obtain a different overall stress field that remains in balance with the loads. The stress field obtained from an elastic analysis is just one particular equilibrium stress field (having the property that the corresponding strains are compatible with a unique displacement field) that might be chosen for this purpose. If the elastic analysis was conducted on virgin, unstressed, material then the elastic stress field is unlikely to be the one in the actual loaded structure since there are likely to be residual, self-balancing stress fields due to manufacture, assembly and thermal gradients. Nonetheless, provided it is an equilibrium stress field, a safe design may be developed by ensuring that sufficient material is placed to resist these stresses.

The ideas just outlined are embodied in the safe, *lower bound theorem of plasticity*, which states that provided a stress field can be found which is in equilibrium with the applied load and which does not anywhere violate the appropriate yield criterion, then the design is safe from plastic collapse. Implicit in this theorem is that the chosen material is sufficiently ductile that the equilibrium stress field can be realised without the material failing. Engineering materials that do provide sufficient ductility for this approach to be used include many structural steels and certain forms of under-reinforced concrete.

Equilibrating Stress Fields for a Continuum

Some of the points mentioned above are illustrated in the problem of figure 1, where a planar structural member is loaded with boundary tractions derived from the stress field provided.



$$\sigma_x = x^2$$

$$\sigma_y = y^2$$

$$\tau_{xy} = -2xy$$

Figure 1: Planar problem loaded with boundary tractions

The stress field is statically admissible (SA), i.e., it satisfies the relevant equations of equilibrium, but it is *not* kinematically admissible (KA) as the corresponding strain field does not satisfy the strain/displacement compatibility conditions.

Figure 2 shows three stress fields. The first is the SA stress field from figure 1. The second is a KA stress field obtained from a coarse conforming finite element model. The third stress field was obtained using a highly refined FE model. It is very close to the theoretically exact solution, which is both statically and kinematically admissible (SAKA).

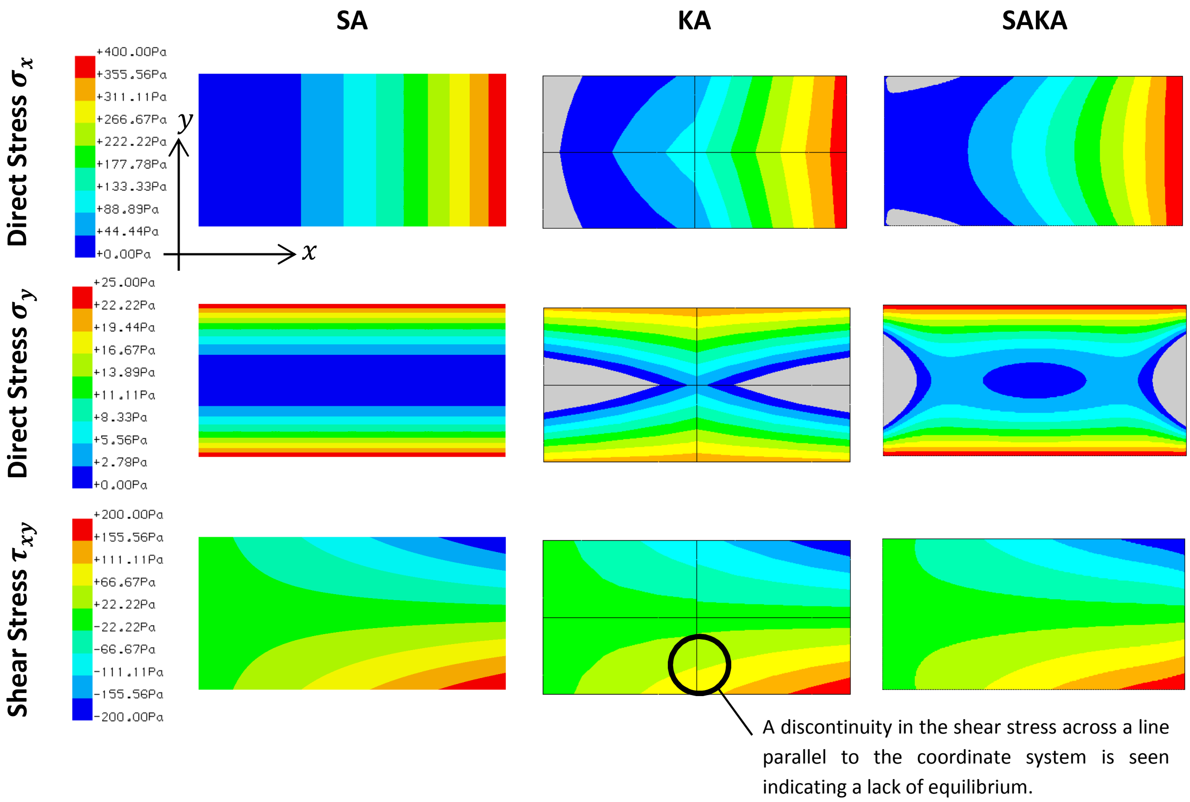


Figure 2: Stress fields for the planar problem of figure 1

If the SA field is subtracted from the SAKA field then the SA stress field shown in figure 3 is obtained. This stress field, which is not kinematically admissible, is self-balancing so that the boundary tractions are zero and for any design section, the stress resultants are zero.

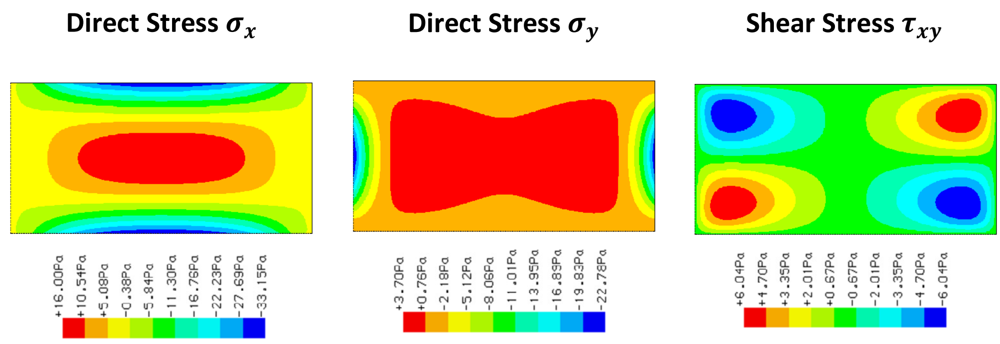


Figure 3: Self-balancing stress field (SAKA-SA)

Two equilibrium stress fields have been shown for this problem (SA and SAKA) with the difference being a self-balancing stress field. As such either of these are suitable candidates for a safe design to be developed using an appropriate yield criterion. In the context of stress classification, the SA stress field is a *primary* stress field whereas the self-balancing field is a *secondary* stress field. The distinction between primary and secondary stress fields is that it is the primary stress field that drives plastic collapse with the secondary stress field playing no part whatsoever in plastic collapse.

The yield criterion needs to be satisfied at all points in the structural member, and this may be achieved by ensuring that it is satisfied where the relevant stress measure is a maximum. The relevant stress measure for a ductile steel member would be the von Mises stress and in an *allowable stress design*, the maximum value of this stress taken from a linear-elastic analysis would be limited to a fraction of the yield strength of the material.

Equilibrating Stress Resultants on a Design Section

For structural members with sections subject to membrane, bending and shearing actions, e.g., beam, plate and shell type members, designs are often based on ensuring that stress resultants (forces and moments) on cross sections of the member do not exceed allowable limits. As with the assessment of the continuum discussed in the previous section, the engineer needs to establish the particular cross section(s) where the stress resultants are a maximum and therefore govern the strength of the design.

Whereas a '*strength of materials'* approach might deal directly with stress resultants, e.g., the bending moments and shear forces in a beam member, finite element analysis of a continuum representation of such a member, e.g., an axisymmetric analysis of a pressure vessel, would provide stress fields rather than stress resultants. If these stresses are to be assessed against the various codes of practice then they need to be converted into stress resultants.

The process of decomposing a stress distribution into a sum of more meaningful distributions and then converting these into stress resultants is shown in figure 4, for a stress distribution normal to a design section.

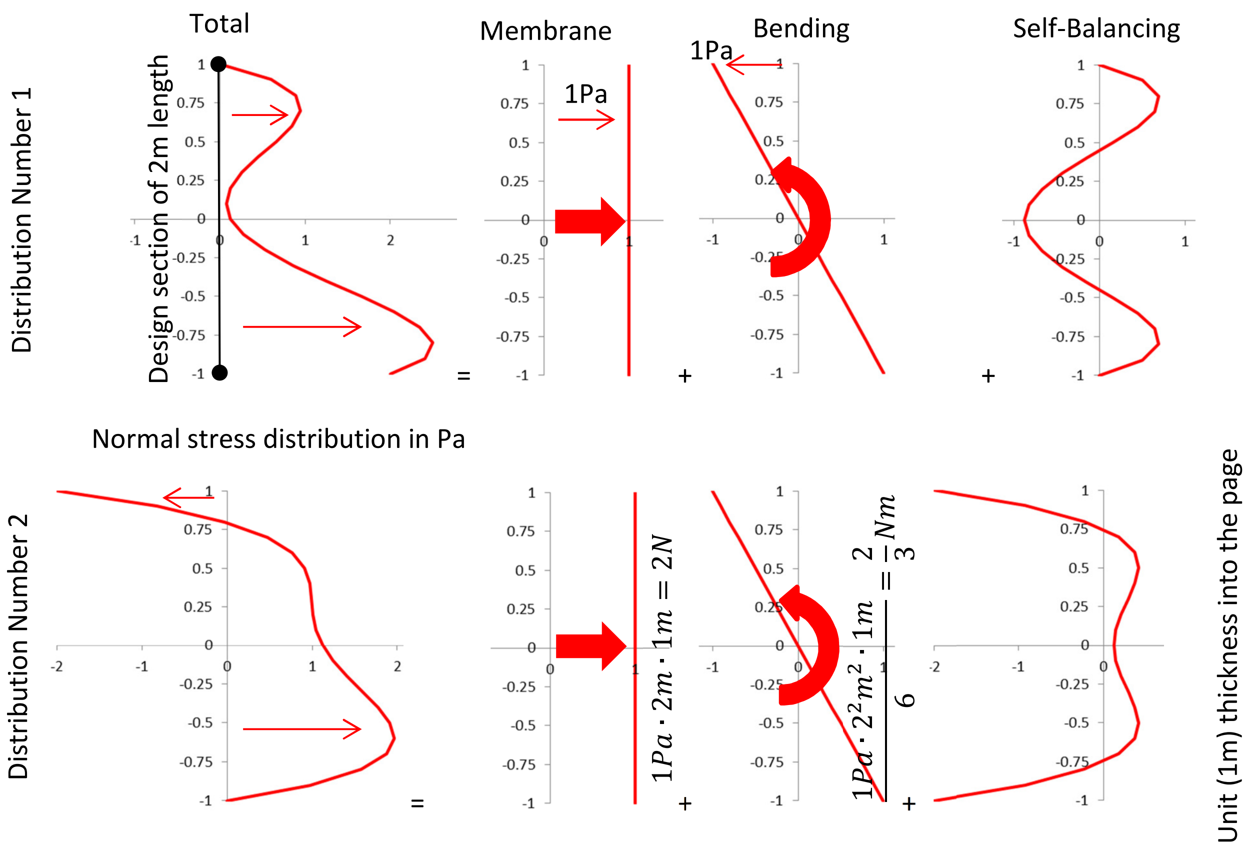


Figure 4: Normal stress distributions mapping into identical normal and moment stress resultants

The decomposition of the normal stress distribution leads to a membrane distribution with a resultant force, a bending distribution with a resultant moment and a residual distribution that is self-balancing on the section. By replacing the total distribution by the simpler membrane and bending distributions, overall equilibrium of the section is maintained at the expense of pointwise equilibrium, which is no longer satisfied. Two different total normal stress distributions are shown in the figure, which have different self-balancing parts but identical stress resultants.

Whilst the stress resultants maintain the overall equilibrium of the design section, by eliminating the self-balancing part of the distribution, pointwise equilibrium along the design section is no longer assured. The implication of this is that, in a strict sense, the lower bound theorem of plasticity no longer holds and, therefore, a safe design is not assured.

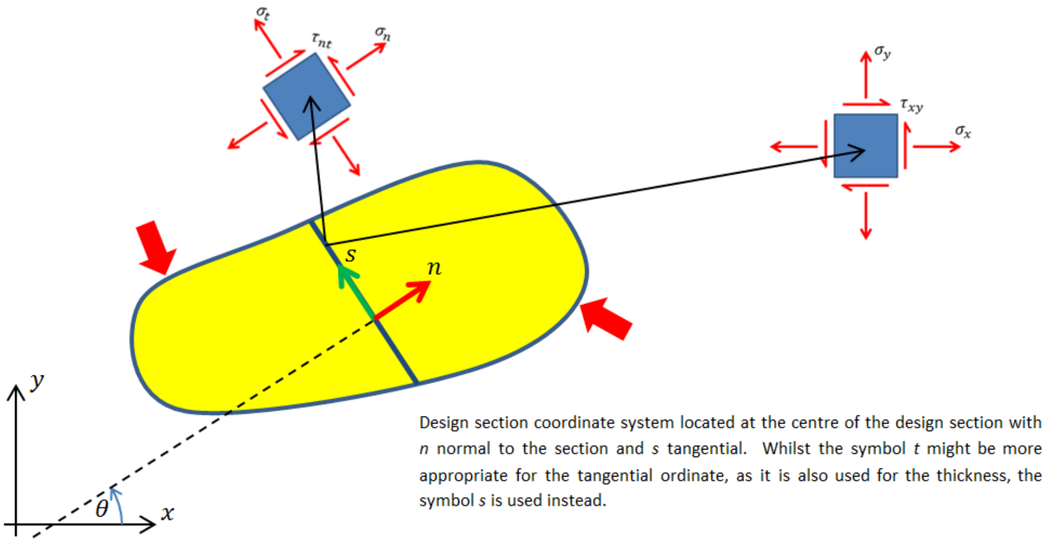
If one wishes to appeal to the lower bound theorem of plasticity for the safety of a design it is tempting to recast the lower bound theorem into a modified form that deals with stress resultants obtained by linearising the stresses along design sections. In doing so, the yield criterion would need also to be recast as an appropriate strength criterion. An example of this approach is shown in the appendix to this technical note. In the example, a beam representation of a plate is considered. For small plate widths, the elastic and plastic solutions for the beam provide good estimates of the plate strength but as the plate width increases this agreement deteriorates. Interestingly, thought, there is a limit to the difference in strength this being about 6% for the elastic solution and about 15% for the plastic solution. If the plate configuration considered is represented as a beam, prediction of failure by first yield is non-conservative. For plastic collapse, however, it is always conservative.

Thus, provided an appropriate strength criterion is used, then it is perfectly safe to assess the strength of a structural member based on stress resultants.

A procedure by which a practising engineer can obtain the stress resultant on a design section from a given stress distribution is that of ***stress linearisation***. It involves integrating stresses in an appropriate manner across the design section.

Stress Linearisation – the Method

An essential prerequisite to stress linearisation is that the stresses to be linearised must be in a coordinate system that is normal and tangential to the design section. Such a transformation is shown in figure 5.



A good reference for the transformation of stress as shown in this figure can be found at: http://users.ox.ac.uk/~kneabz/Stress5_ht08.pdf

Figure 5: Transformation of stress field into a system normal and tangential to the design section

The stress components expressed in the x, y coordinate system are transformed into the n, s design section coordinate system by rotating through the angle θ as shown in equation (1).

$$\begin{Bmatrix}\sigma_n\\ \sigma_s\\ \tau_{ns}\end{Bmatrix}=\begin{bmatrix}\frac{1}{2}(1+\cos 2\theta) & \frac{1}{2}(1-\cos 2\theta) & \sin 2\theta\\ \frac{1}{2}(1-\cos 2\theta) & \frac{1}{2}(1+\cos 2\theta) & -\sin 2\theta\\ -\frac{1}{2}\sin 2\theta & \frac{1}{2}\sin 2\theta & \cos 2\theta\end{bmatrix}\begin{Bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{Bmatrix} \tag{1}$$

One often sees in the literature, e.g., 'Knowledge Base – Don't Forget the Basics', NAFEMS Publication, 013, Inelastic Analysis, linearisation of principal stresses or even stress intensity and von Mises stresses. As the whole idea of linearisation is to obtain stress resultants such as normal force, tangential force and bending moment acting on the design section, this is simply incorrect; the direction of principal stresses is likely to vary along the design section and von Mises stresses have no associated direction at all. It is not helpful in this respect that in some finite element systems, e.g., ANSYS, linearisation is conducted on all available components of stress, in whatever coordinate system is being used and including the principal and von Mises stresses. This opens up the potential for *finite element malpractice* and software vendors would be doing a service to practising engineers if their software were designed in such a manner that an appropriate coordinate system was automatically selected and the principal and von Mises stresses are not linearised.

The loaded member of figure 5 is 'exploded' about the design section, of length l, exposing two new edges on which the stress resultants might be evaluated. A local coordinate system centred on the design section is shown in figure 6 together with a non-dimensional or normalised system for the tangential ordinate.

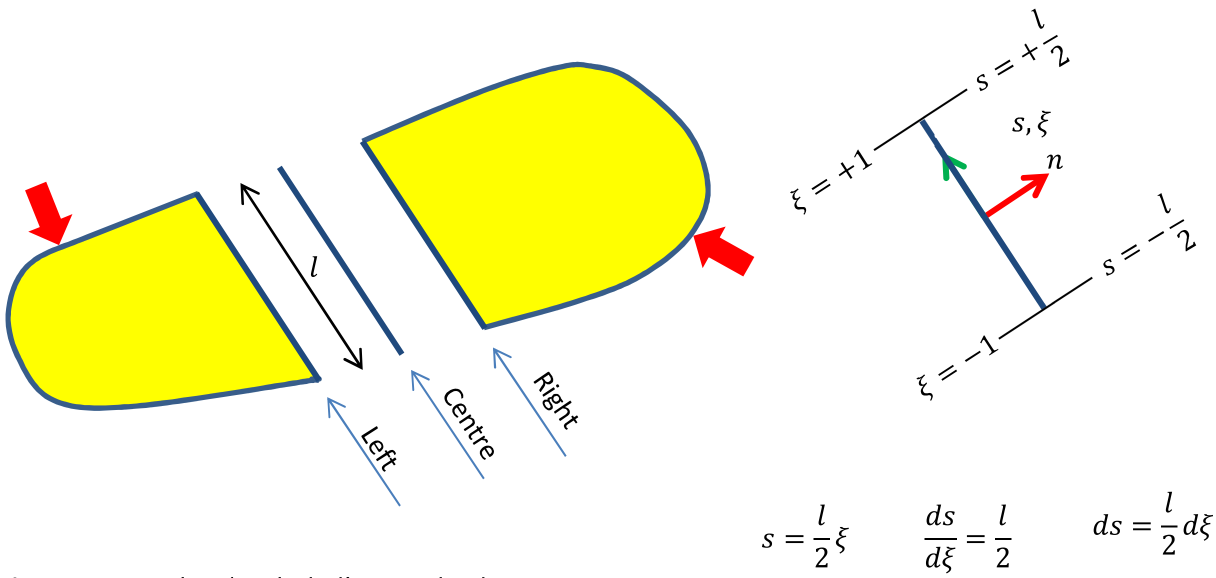


Figure 6: Member 'exploded' around a design section

Appropriate components of the stresses, transformed through equation (1) into the correct orientation, are multiplied, in equation (2), by the thickness t to produce tractions, which act normally and tangentially to the design section.

$$\begin{Bmatrix} t_n \\ t_t \end{Bmatrix} = t \begin{Bmatrix} \sigma_n \\ \tau_{ns} \end{Bmatrix} \qquad (2)$$

It will be noted, from equation (2), that the direct stress tangential to the section takes no part in the tractions; it is not required for equilibrium of the design section.

Consider now the distributions of normal and tangential tractions acting on a design section, as shown in figure 7.

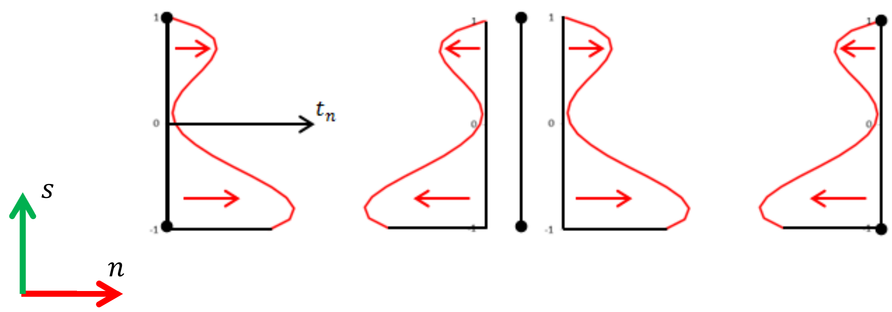


Figure 7: Illustrative normal traction distribution on a design section

The distributions of normal and tangential tractions may be integrated to obtain the three stress resultants acting at the centre of the design section and this process is shown in figure 8.

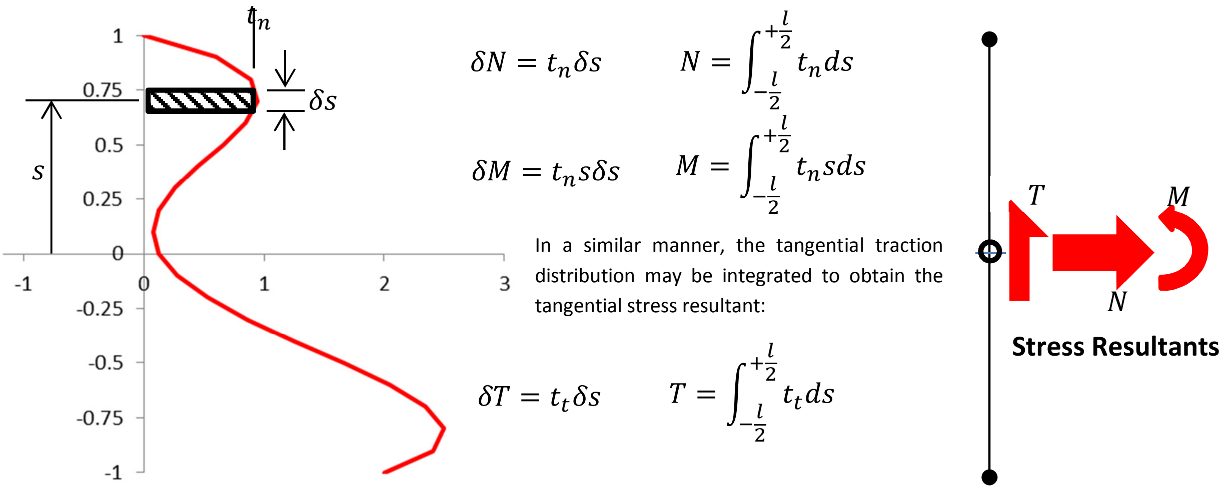


Figure 8: Integrating the traction distributions to produce the stress resultants

The stress resultants were obtained by integrating the traction distributions. The integrations shown in the figure were conducted over the tangential ordinate s but it is equally valid to conduct these integrations over the non-dimensional ordinate ξ, as shown in equations (3), if the appropriate relationship between the two ordinates (see figure 6) is adopted.

Normal Stress Resultant
$$N = \int_{-\frac{l}{2}}^{+\frac{l}{2}} t_n ds = \frac{l}{2} \int_{-1}^{+1} t_n d\xi$$

Tangential Stress Resultant
$$T = \frac{l}{2} \int_{-1}^{+1} t_t d\xi \tag{3}$$

Moment Stress Resultant
$$M = \frac{l^2}{4} \int_{-1}^{+1} t_n \xi d\xi$$

A polynomial may be written in many ways, and one form that is particularly useful in stress analysis is the Legendre form. The first five Legendre polynomials are presented in figure 9.

If the distributions of the Legendre polynomials are considered as normal tractions then it is seen that for degree 2 (quadratic) and higher, the distributions produce no normal or moment stress resultants, i.e., they are self-balancing along a design section. To demonstrate this point, consider a normal traction distribution defined as a cubic Legendre polynomial:

$$t_n = \frac{1}{2}(5\xi^3 - 3\xi)$$

	i	L_i
Constant	0	1
Linear	1	ξ
Quadratic	2	$\frac{1}{2}(3\xi^2 - 1)$
Cubic	3	$\frac{1}{2}(5\xi^3 - 3\xi)$
Quartic	4	$\frac{1}{8}(35\xi^4 - 30\xi^2 + 3)$

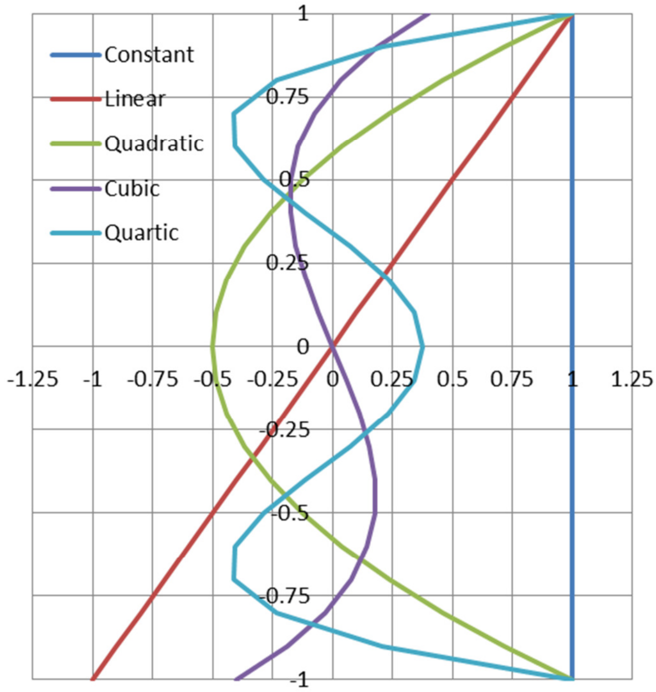


Figure 9: Legendre polynomials (see - https://en.wikipedia.org/wiki/Legendre_polynomials)

The normal and moment resultants for this cubic traction distribution are evaluated below.

$$N = \frac{l}{2}\int_{-1}^{+1} t_n d\xi = \frac{l}{4}\left[\frac{5}{4}\xi^4 - \frac{3}{2}\xi^2\right]_{-1}^{+1} = 0$$

$$M = \frac{l^2}{4}\int_{-1}^{+1} t_n \xi d\xi = \frac{l^2}{8}[\xi^5 - \xi^3]_{-1}^{+1} = 0$$

The normal stress resultant is given by the constant Legendre polynomial and the moment stress resultant by the linear polynomial. This means that provided we can write the stress distribution along a design section in the form of Legendre polynomials, then it is rather easy to work out the stress resultants. The following equation expresses the normal traction distribution as a sum of Legendre polynomials of increasing degree with ϕ_i representing the coefficients of the normal traction polynomial and ψ_i the coefficients for the tangential traction distribution.

$$t_n = \sum_{i=0}^{n} \phi_i L_i \qquad\qquad t_t = \sum_{i=0}^{n} \psi_i L_i \tag{4}$$

The three stress resultants can now be very simply expressed in terms of the coefficients of the Legendre polynomial traction distributions and the length of the design section:

Normal Stress Resultant $$N = \frac{l}{2}\int_{-1}^{+1} t_n d\xi = \frac{l}{2}\int_{-1}^{+1} \phi_0 d\xi = \frac{\phi_0 l}{2}[\xi]_{-1}^{+1} = \phi_0 l$$

Tangential Stress Resultant $$T = \frac{l}{2}\int_{-1}^{+1} t_t d\xi = \frac{l}{2}\int_{-1}^{+1} \varphi_0 d\xi = \frac{\varphi_0 l}{2}[\xi]_{-1}^{+1} = \psi_0 l \qquad (5)$$

Moment Stress Resultant $$M = \frac{l^2}{4}\int_{-1}^{+1} t_n \xi d\xi = \frac{l^2}{4}\int_{-1}^{+1} \phi_1 \xi^2 d\xi = \frac{\phi_1 l^2}{4}\left[\frac{\xi^3}{3}\right]_{-1}^{+1} = \frac{\phi_1 l^2}{6}$$

The traction distributions of figure 7 were formed from a sum of Legendre polynomials with different coefficients. The individual polynomials are shown in figure 10 - for both normal and tangential distributions the coefficient of the cubic polynomial was (arbitrarily) chosen to be zero.

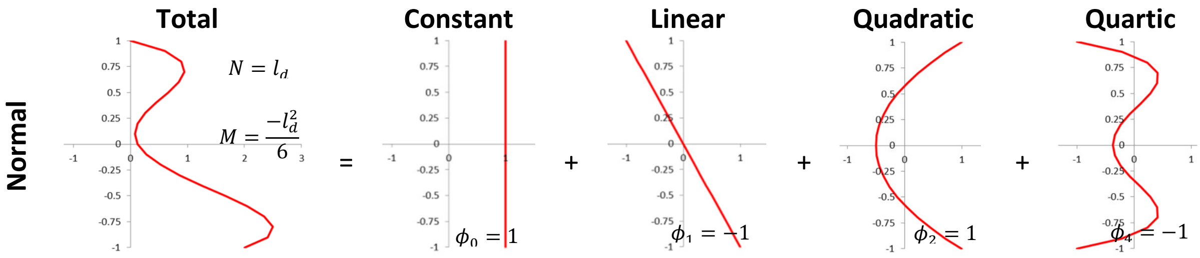


Figure 10: Decomposition of the normal traction distribution

With the coefficients of the Legendre polynomials from figure 10, and using equations (3), the stress resultants for the 2m design section shown in figure 11 are evaluated.

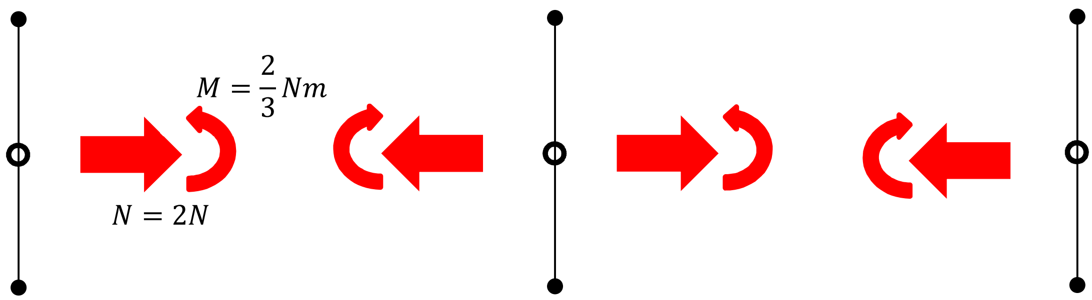
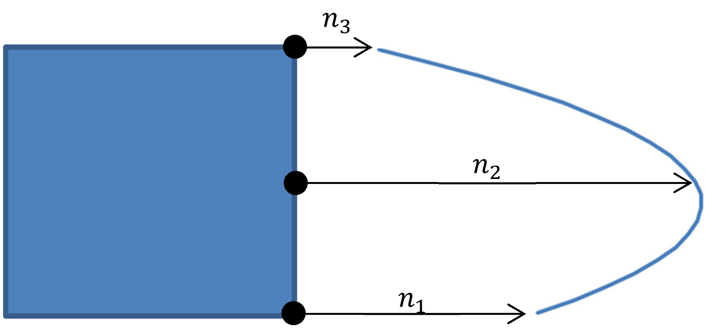


Figure 11: Stress resultants for the traction distributions of figure 7

Stress Linearisation Applied to Finite Element Results

When it comes to evaluating stress resultants for a conforming finite element (CFE) model then the stress distribution along an edge of an element will either be linear (lower-order elements) or quadratic (higher-order elements), defined either by two or three nodal values. These may be converted to nodal tractions by multiplying by the thickness. The coefficients of the constant and linear Legendre polynomials can be written directly in terms of the nodal traction values as shown for the normal nodal tractions on the edge of a higher-order element in figure 12.



$$\phi_0 = \frac{1}{6}(n_1 + 4n_2 + n_3)$$

$$\phi_1 = -\frac{1}{2}(n_1 - n_3)$$

$$N = \frac{l}{6}(n_1 + 4n_2 + n_3)$$

$$M = \frac{-l^2}{12}(n_1 - n_3)$$

Figure 12: Normal and moment resultants calculated from normal nodal tractions

If a lower-order element is being used then it is a simple matter to set the mid-side nodal traction to the average of the corner node values. By replacing the normal nodal tractions by the tangential nodal tractions in the expression for the normal stress resultant, the tangential stress resultant may be obtained. It is, thus, a simple matter to calculate the stress resultants acting at the centre of an element edge from the nodal traction values and without explicitly having to integrate. Of course, if there are a number of element edges along the chosen design section then these stress resultants will need to be transferred to the centre of the design section to have any physical meaning.

An example of stress linearisation for a multi-element model is shown in figure 13 where two lower-order elements are used across the design section and the stresses have been averaged between the elements at the centre of the design section. In this example, the expressions for the normal and moment stress resultants given in terms of the nodal tractions were used (see figure 12).

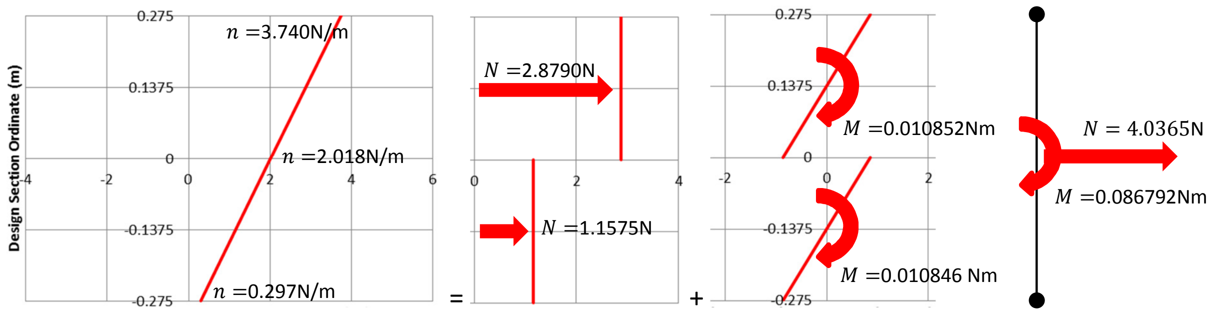


Figure 13: Normal and moment resultants from stress linearisation for a two-element model

The total normal traction distribution is shown at the left of the figure and is equal to the sum of the membrane and bending distributions as shown. The normal and moment stress resultants are calculated for each element edge and are then transferred to the centre of the design section at the right of the figure in the appropriate manner.

A potential problem facing the practising engineer when undertaking stress linearisation on a CFE model is that, because the finite element stress field is not in equilibrium, the stress resultants calculated at the left and right of the design section are not likely to agree unless the mesh is well refined. Indeed, if the design section goes through a point of stress singularity then however refined the mesh, there is likely to be significant differences in these stress resultants. There will also be a difference in the stress resultants depending on whether the nodal stresses are averaged or not. This characteristic of conforming finite elements is rather unsatisfactory and the best advice to the practising engineer, if he/she is to avoid finite element malpractice, is to make sure that the mesh is sufficiently refined to provide an accurate prediction of the stresses.

It is clear, then, that the practising engineer using CFE models needs to exercise caution when conducting stress linearisation to ensure that the stress resultants are actually in equilibrium with the applied load. *If they are not in equilibrium then the engineer may no longer appeal to the lower bound theorem of plasticity for the safety of his/her structural member*.

In order to establish whether the stress resultants are in equilibrium, the engineer can conduct a mesh refinement study and observe the convergence of the resultants.

Convergence of Stress Resultants for a Smooth Stress Field

A planar elasticity problem is chosen to examine how the stress resultants calculated by stress linearisation converge with mesh refinement. The problem is shown in figure 14 along with the stress field and the corresponding boundary tractions. This stress field is both statically and kinematically admissible and thus it is the true solution to the problem.

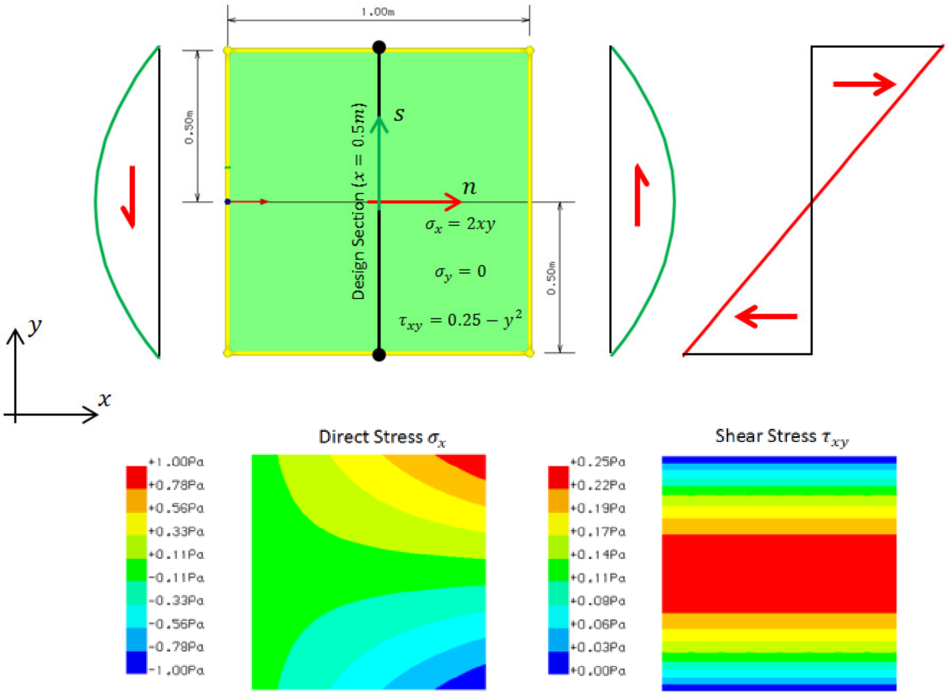
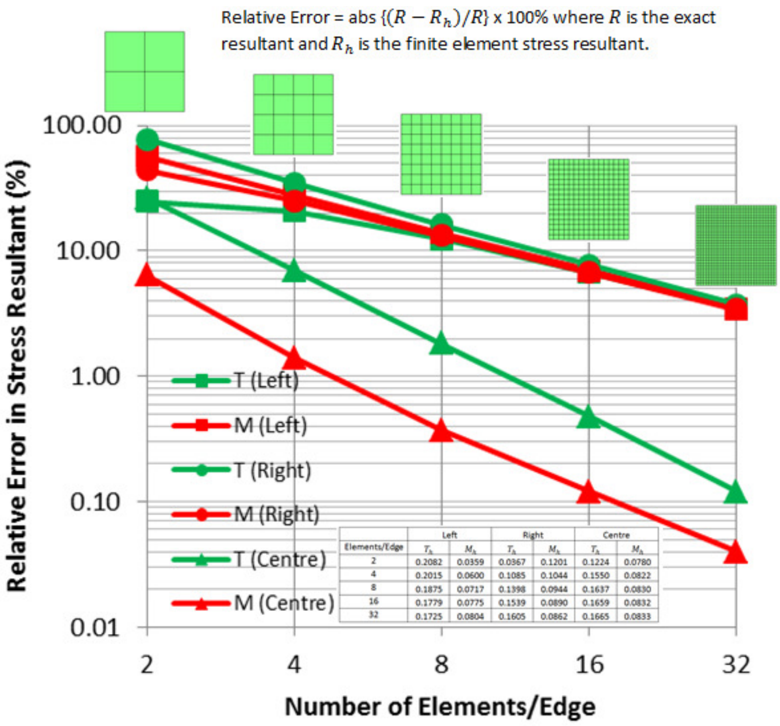


Figure 14: Boundary tractions and stress fields for a planar problem

As the design section coordinate system is parallel to the global coordinate system, no transformation of the stresses is required with $\sigma_n = \sigma_x$, and $\tau_{ns} = \tau_{xy}$. The problem has unit (1m) thickness and so the tractions on the design section are $t_n = y$, and $t_t = 0.25 - y^2$. The normal stress resultant is clearly zero. The tangential resultant is simply the average shear traction multiplied by the length of the design section. The average shear traction is 2/3 of the maximum value of 0.25, and as the length of the design section is 1m the tangential stress resultant is 1/6N. For the moment resultant, we can see that the coefficient of the linear Legendre polynomial is 0.5N/m, and, from equation (5), the moment resultant is 1/12Nm

Using meshes of fully integrated four-noded plane stress elements, with the number of elements/edge starting at two and then being doubled up to 32, the problem was analysed and stress linearisation used to determine the stress resultants on either side of the design section. The results are presented in figure 15, which shows a log-log plot of the convergence of the stress resultants. Since the true solution is known, the relative error in the stress resultants may be calculated and has been expressed as a percentage in the figure.

For the coarse meshes, the error in the stress resultants is considerable (about 80% for the tangential resultant calculated to the right of the design section) and a significant lack of co-diffusivity in the resultants is observed – those calculated to the left of the section differ significantly from those calculated to the right of the section. As the mesh is refined, both the error and the lack of co-diffusivity reduces, though even with the most refined mesh, there remains an error in the resultants of over 3%. The stress resultants at the centre of the design section, here taken as the average of the values to either side of the section, are much more accurate with both resultants exhibiting an error of less than 5% for the eight elements/edge model. Clearly, had the same study been conducted using higher-order elements, then the results would have exhibited a much reduced error.



	Left		Right		Centre	
Elements/Edge	T_h	M_h	T_h	M_h	T_h	M_h
2	0.2082	0.0359	0.0367	0.1201	0.1224	0.0780
4	0.2015	0.0600	0.1085	0.1044	0.1550	0.0822
8	0.1875	0.0717	0.1398	0.0944	0.1637	0.0830
16	0.1779	0.0775	0.1539	0.0890	0.1659	0.0832
32	0.1725	0.0804	0.1605	0.0862	0.1665	0.0833

Figure 15: Convergence of stress resultants for problem in figure 14

It should be noted that the convergence plots are, after the very coarse first result, essentially linear on the log-log plot and this is to be expected for a smooth stress field. As such sufficient mesh refinement should yield the stress resultants to the desired accuracy. It is of interest now to see how the method performs in the presence of a stress singularity.

Convergence of Stress Resultants for a Singular Stress Field

It is to be suspected that if a design section runs through a stress singularity then the stress resultants obtained through stress linearisation are likely to be polluted by the presence of the singularity. To explore this idea further the built-in beam under self-weight of figure 16 is examined.

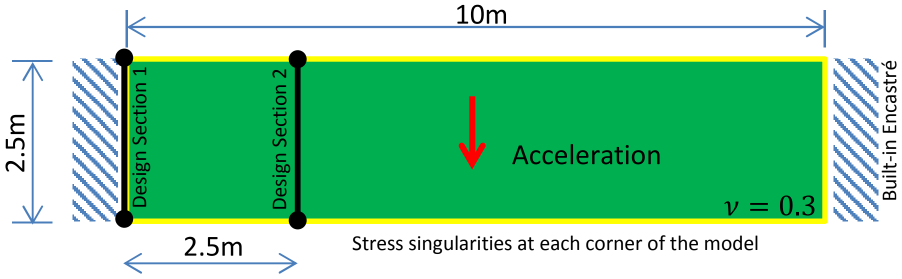


Figure 16: A planar problem with stress singularities

This problem exhibits stress singularities at each corner of the beam. Two design sections will be considered, *viz.*, Design Section 1 positioned at the end of the beam and Design Section 2 positioned at the quarter point. The beam is modelled with a single element mesh which is then uniformly refined and the convergence of the shear resultant on the design sections is observed. For the acceleration and density used the theoretical shear resultant are 39.5kN and 19.75kN respectively for the two sections. Two forms of convergence plot are offered in figure 17. This first is a semi-log plot showing convergence of the shear resultant and the second is a log-log plot showing convergence of the relative error in the resultant expressed as a percentage. The performance of the four-noded element is significantly influenced by whether reduced or full integration is adopted. The reduced integration element appears to converge monotonically from below the theroetical value whereas for the fully integrated element convergence, eventually, comes from above the true value. The eight-noded element (reduced integration) produces a result that is almost exact irrespective of the mesh.

The log-log plot shows, for the eight-noded element, that the error increases with mesh refinement. The convergence curve for this element is shown in two parts as, for the coarser meshes, the error was not able to be accurately assessed since the resultant was reported with insufficient significant figures by the software used.

From a practical stand point, the level of mesh refinement required to achieve reasponable engineering accuracy, say 5%, is rather significant. There is also a concern that with the eight-noded element, whilst achieving good accuracy with coarser meshes, there is a tendancy for the quality to decrease with mesh refinement.

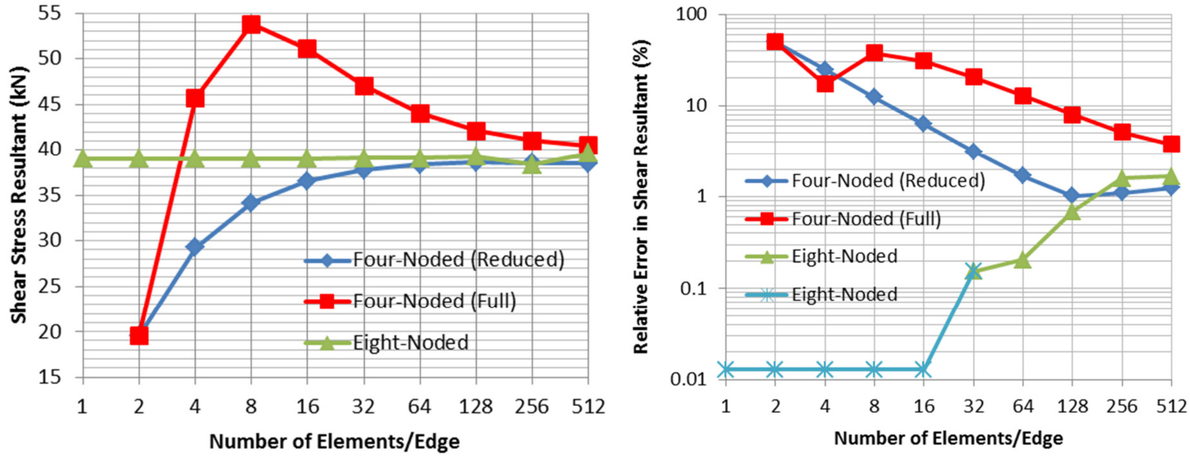


Figure 17: Convergence of shear resultant on Design Section 1

Similar convergence plots are shown in figure 18 for the second design section. In this case, where the design section is remote from the singularities, the four-noded reduced integration element and the eight-noded element give the exact result and this is independent of mesh refinement. The fully integrated four-noded element again shows poor performance requiring a 64 elements/edge mesh before 1% accuracy is achieved.

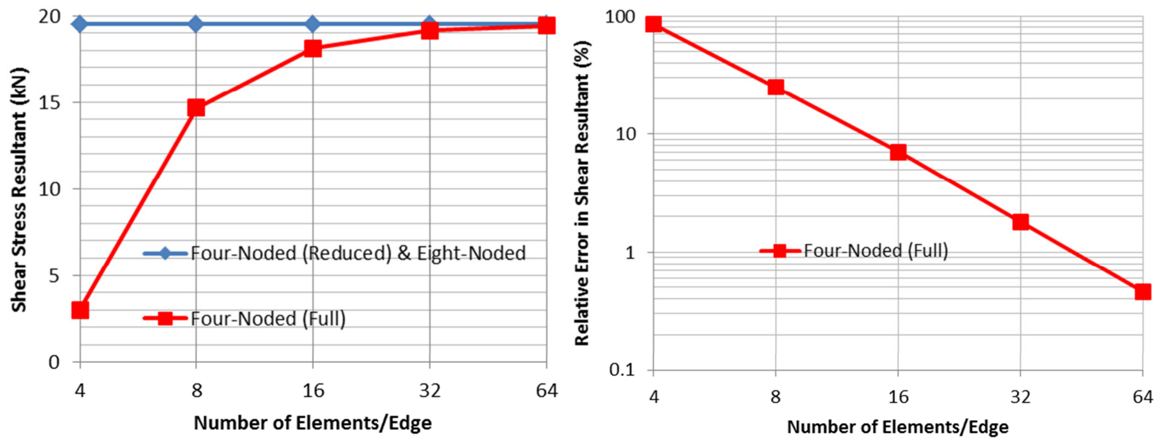
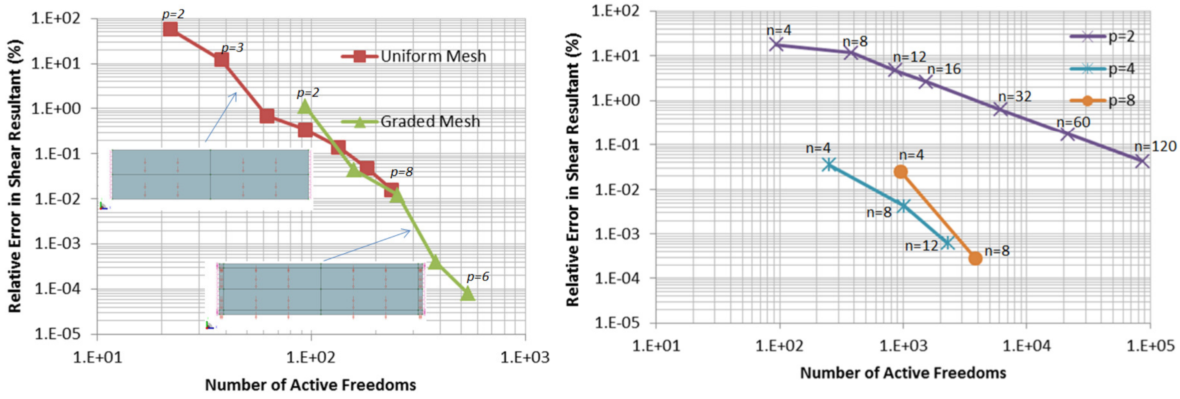


Figure 18: Convergence of shear resultant on Design Section 2

This problem could also be tackled using *p*-type CFEs, which allow the mesh to be refined both by *h*-type refinement (increasing the number of elements) and by *p*-type refinement (increasing the polynomial degree of the elements). The results from such a study are shown in figure 19.

The convergence of the error using *p*-type elements is monotonic and higher levels of accuracy are achievable than was obtained using the standard lower and higher-order displacement elements at least for the first design section. The results for the second design section were obtained using uniform mesh refinement with the number of elements/edge varying between 4 and 120.



(a) Design Section 1

(b) Design Section 2

Figure 19: Convergence of shear resultant on Design Sections (ESRD: StressCheck)

The software used for the *p*-type analysis preseted in figure 19 was ESRD:StressCheck. The *p*-type element available in ANSYS was also used and the results are shown in figure 20.

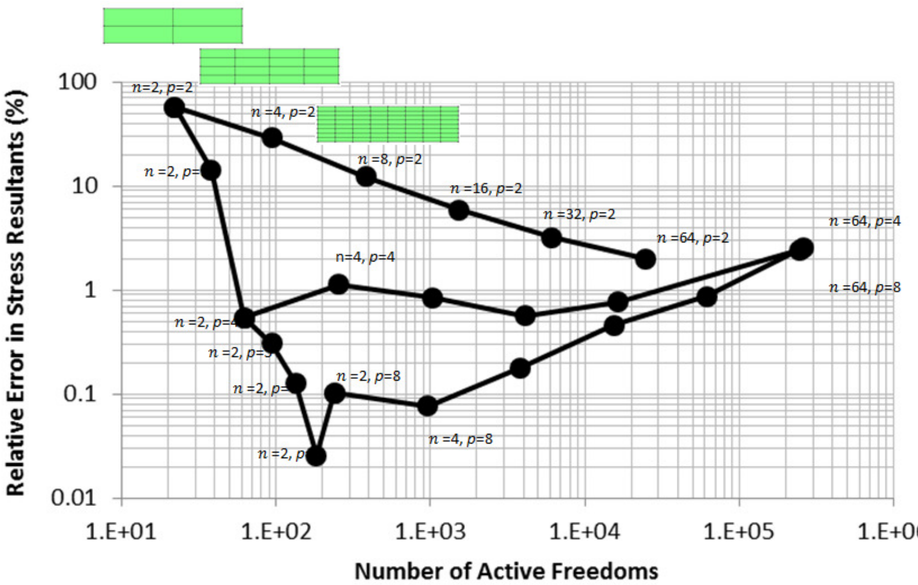


Figure 20: Convergence of shear resultant on design section 1 (ANSYS)

Whereas for StressCheck the observed convergence was monotonic, the convergence for ANSYS was rather erratic and for the higher degree elements, the error increases with *h*-refinement! The difference in the results between the two software packages was surprising since both adopt the same element formulation. It is known that StressCheck uses the raw finite element stresses along the design section to perform the stress linearisation and it is suspected that ANSYS only approximates the finite element stresses thus leading to the erratic convergence behaviour observed.

Discussion

This technical note has presented, hopefully in approachable manner for practising engineers, the procedure for determining stress resultants through stress linearisation. The ideas behind achieving a safe design, in terms of the ULS condition of collapse, were presented in terms of the lower bound theorem of plasticity. *The theorem is only valid if the stress fields used for the design are in equilibrium with the applied loads*. For particular structural forms, e.g., pressure vessels, practising engineers often base their assessment of the required strength on stress resultants and these are often obtained through stress linearisation of the finite element stresses from a CFE model.

Two example problems were examined using stress linearisation to determine stress resultants. For both problems, the theoretical stress resultants were known and it was, thus, possible to determine the error in the stress resultants predicted by the finite element model. In the first example, the theoretical stress field was smooth, and it was observed that with mesh refinement, the finite element stress resultants converged satisfactorily, albeit rather slowly, to the theoretical values. The second problem involved a stress singularity and the design section was deliberately placed such as to go through the singular points. In this example, convergence of the stress resultants was rather unsatisfactory.

The sort of mesh refinement studies conducted in this report are time consuming and, in the author's experience, it is unusual for such studies to be conducted in engineer design/analysis offices. The engineer not conducting such studies is risking the safety of his/her design, as without strong equilibrium, the lower bound theorem of plasticity may not be invoked.

As an alternative to obtaining stress resultants through stress linearisation, the engineer would be better advised to use the nodal forces, which are guaranteed to be in equilibrium with the applied load. If nodal forces are to be used then the engineer needs to design the finite element mesh such that element edges align with his/her chosen design section. This is a reasonable approach if the engineer happens to know where the critical design section lies. In general, however, the location of this section will not be known and, indeed, it might be that different design sections are critical for different stress resultants. Thus, it can be seen that capturing the critical stress resultants is, potentially, a rather toilsome activity and, as such, one that might not be done properly in a commercial environment where the engineer is likely to be under pressure to deliver results quickly.

Conclusions

If the author was asked whether a practising engineer should use stress linearisation to determine stress resultants from a CFE model then he would have to advise against it. Far better to make use of nodal forces which are guaranteed (all else being equal) to be in equilibrium with the applied loads. It is noted, though, that if the aim of stress linearisation is to classify the stresses according to, for example, the ASME pressure vessel codes, then the use of nodal forces, whilst giving the stress resultants, will not provide the self-balancing traction distribution. In such cases, one might recommend that the stress resultants obtained by stress linearisation are verified through those calculated from nodal forces.

The nature of CFEs, which are used by most if not all commercial finite element systems, is such that unless the mesh is sufficiently refined then the stresses are unlikely to be in equilibrium with the applied load. This is a rather unsatisfactory situation since without equilibrium the practising engineer cannot ensure his/her design is safe. With CFEs the only way to ensure safety is through mesh refinement and convergence studies.

Thus, the status quo in the finite element industry is, in the author's opinion, rather unsatisfactory. *Computer-Aided Catastropes* (CAC) do occur, with the Sleipner incident being a significant example. That more CAC do not occur is probably more due to the fact that many designs are manufactured in ductile materials with strengths based on elastic analyses. In this manner the design is often rather forgiving in terms of possessing significant residual strength. There is a significant move, driven by industry, towards the democratisation of simulation. This is understood to mean that sophisticated simulation software tools are being placed in the hands of engineers who are, potentially, inadequately experienced or trained. As this report has demonstrated, there is, with CFEs, significant scope for *finite element malpractice* and if this is significant enough to lead to a failed structure or component then death or injury may arise and companies may go out of business.

Acknowledgements:

The author would like to thank Matt Watkins at ESRD Inc., for organising the analysis of the stress singularity problem through his company's finite element software StressCheck.

The author was grateful for the review comments on this technical note offered by Dr Edward Maunder, Consultant at Ramsay Maunder Associates, Professor Nawal Prinja, Technical Director, Clean Energy, AMEC Foster Wheeler plc, and Professor Savvas Triantafyllou of the Faculty of Engineering at the University of Nottingham.

Appendix: Stress Linearisation in a Plate Problem

The purpose of this example is to demonstrate that stress resultants obtained by stress linearisation along a design section can lead to an incorrect prediction of the strength of a member. A uniformly loaded rectangular plate, simply supported on two opposite sides, is shown in figure 21 together with a beam representation of the plate.

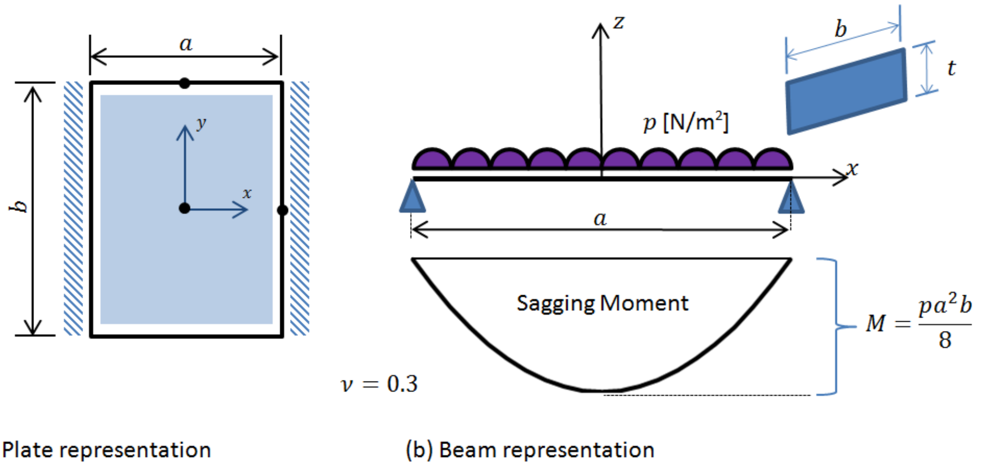


Figure 21: Plate configuration and beam representation

For the beam representation of the plate, the maximum moment occurs at the centre of the beam and has the value shown in the figure. The pressure to cause first yield at the centre of the beam may be written in terms of the yield stress S_y and the shape of the plate; note that the result is independent of the width b:

$$p_f = \frac{4S_y t^2}{3a^2}$$

For the beam, the pressure to cause plastic collapse is simply:

$$p_c = \frac{3}{2} p_f$$

Thus, for a plate with a =1m, b =2m, t =10mm and S_y=275MPa the pressure to cause first yield is p_f=36.67kPa.

In 1899, Levy provided a solution for the elastic moment to this particular plate configuration. It was published in Timoshenko's "Theory of Plates and Shells". The Levy solution provides the theoretically exact Cartesian moment fields, which may be converted into principal moments and von Mises moments. These moment fields are shown for the plate dimensions being considered in figure 22. The loading is 1kPa and the units of the moments are Nm/m.

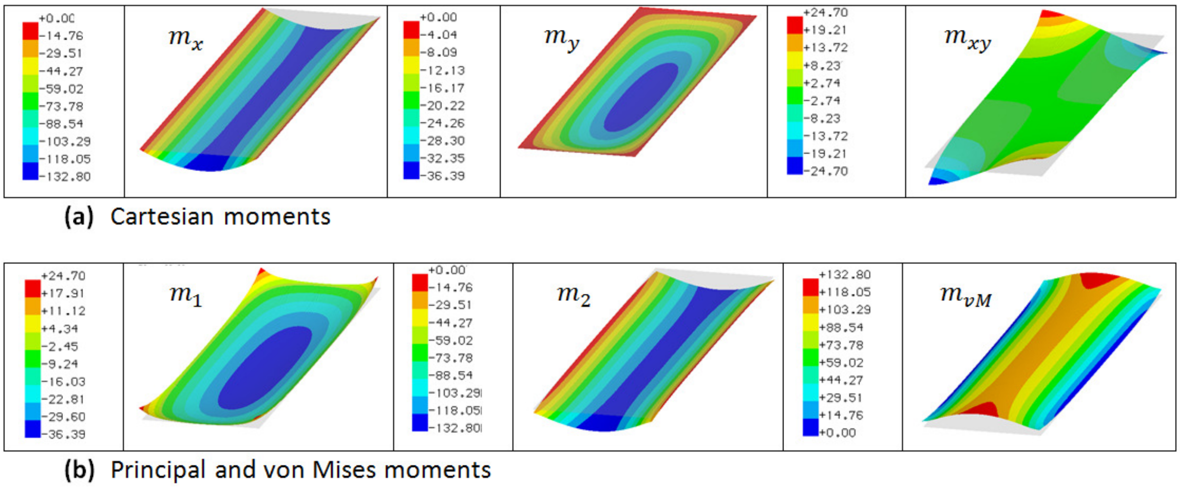


Figure 22: Elastic moments from the Levy solution

The m_x moment field varies in an essentially quadratic manner as the bending moment diagram for the beam representation of the plate. There is, however, a variation across the width of the beam. If stress linearisation were used to determine the moment stress resultants along lines parallel to the supported edges, then these moment resultants would correspond exactly to the bending moment diagram for the beam.

The von Mises moments show a maximum at the centre of the unsupported edges having a value of 132.8Nm/m for an applied load of 1kPa. The corresponding surface stress is 7.97MPa and, from this, one can scale to obtain the pressure to cause first yield for our 275MPa steel. The value is 34.51kPa which demonstrates that for this particular plate configuration, the value obtained using a beam representation, is nearly 6% non-conservative. The difference between the beam and plate solutions becomes small when the width of the plate is small and converges to a constant when the width is large – see figure 23.

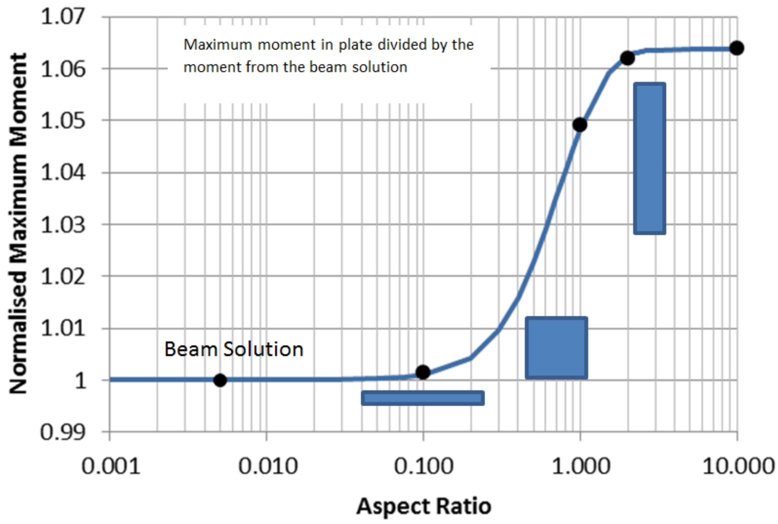


Figure 23: Maximum moment for plate as a function of aspect ratio (Elastic)

In a similar manner, the loads to cause plastic collapse for the plate and beam representations may be compared. In this case, the plastic solution for the plate comes from limit analysis using a rigid, perfectly plastic material model. The results are summarised in figure 24, which plots the collapse load of the plate, normalised with the collapse load for the beam, as a function of aspect ratio.

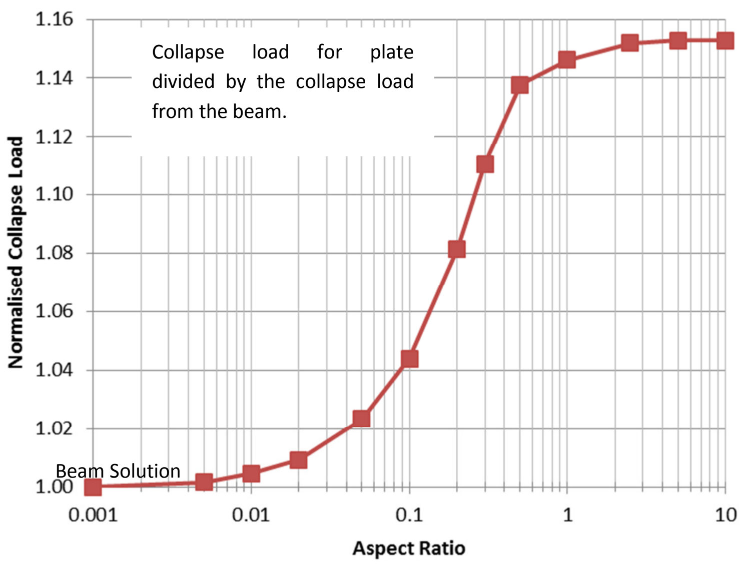


Figure 24: Collapse Load for plate as a function of aspect ratio (Plastic)

Whereas for the elastic case of first yield, the beam solution gave a non-conservative prediction of failure, for the plastic case the beam solution under-estimates the strength of the plate by a factor of up to $2/\sqrt{3}$.

The purpose of this example was to demonstrate that the use of stress resultants on design sections could lead to an inaccurate prediction of the strength of the structural member. This inaccuracy can, however, be corrected if the yield criterion is replaced by a ‘strength criterion’ which accounts for the difference between, in this case, the plate and the beam representation. For example, the stress resultants achieved by linearisation could be used to predict the collapse load for a beam and then this can be corrected using figure 24.