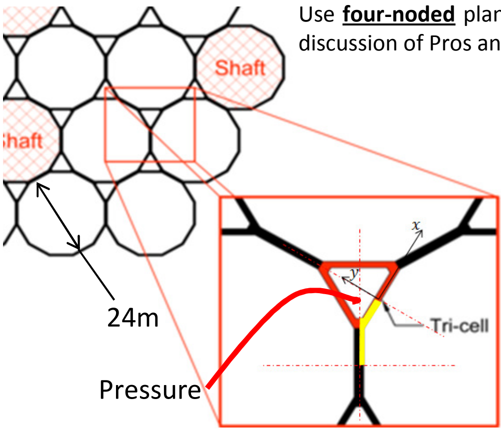
NBC Number 06 (April 2016)

The NAFEMS Benchmark Challenge

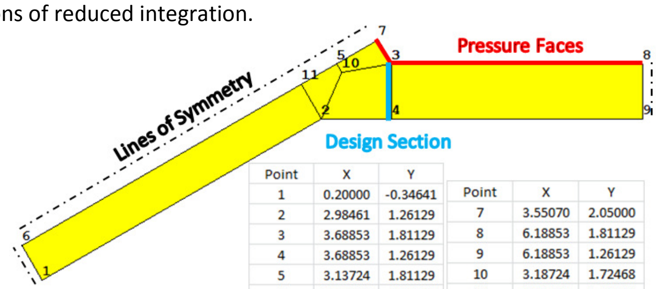
The Sleipner Incident - a Computer-Aided Catastrophe Revisted

In a recent article ('NAFEMS: the Early Days', January 2016), Peter Bartholomew describes how John Robinson, one of the founders of NAFEMS, noted in the early 1970s "… that both coding and modelling errors were commonplace and only time separated the community from computer-aided catastrophe (CAC)". In the early 1990s just such an incident of CAC occurred when the reinforced concrete Sleipner Platform A sank in a Norwegian Fijord. No one was injured, but the incident cost some $700M (US). The subsequent inquiry found that the FE modelling local to the failure had been inadequate, under-predicting the shear forces by some 45%, and that the reinforcement detailing in the region was not adequate to support the loading. This incident, now some 25 years ago, is a significant reminder of the importance of good simulation governance and, as there are useful lessons to learn from it, this challenge revisits the Sleipner Incident.

The platform comprised a honeycomb of shafts and tri-cells. The tri-cells were subject to hydrostatic pressure from the fijord causing a pressure differential across the walls of the tri-cells. A basic FE model, which approximates the actual geometry, comprising a symmetric sixth of a tri-cell is provided, figure 1, and uniform mesh refinement should be used to improve the solution.



Use **four-noded** plane stress elements **with reduced integration** - see NAFEMS FE Primer for discussion of Pros and Cons of reduced integration.



Point	X	Y
1	0.20000	-0.34641
2	2.98461	1.26129
3	3.68853	1.81129
4	3.68853	1.26129
5	3.13724	1.81129
6	0.00000	0.00000

Point	X	Y
7	3.55070	2.05000
8	6.18853	1.81129
9	6.18853	1.26129
10	3.18724	1.72468
11	2.78461	1.60770

Consider unit Pressure, E=30GPa and ν=0.2

Figure 1: Shaft and tri-cell honeycomb and a basic finite element model

The Challenge

Often the first step in the design of reinforced concrete structures is to use a linear-elastic model to establish the internal stresses due to the applied load. These are then used to size the reinforcement required to support the load and this is generally done by looking at stress resultants (shear force, axial force and moment in this case) across critical sections deemed to be of structural interest. Commercial FE systems generally use conforming finite elements which are based on assumed continuous displacement fields which ensure strain/displacement compatibility. Stresses are related to the strains through the appropriate constitutive relations - Hooke's law in the case of linear elasticity. The remaining condition, that of equilibrium of the stresses with the applied load, is generally only weakly satisfied with conforming elements. This can be a problem for the engineering designer but provided the mesh is adequately refined then the lack of equilibrium should be small.

In this challenge you are asked to consider how best to achieve a set of equilibrating stress resultants on the design section identified in figure 1 and the following techniques should be used:

- Stress Linearisation – see http://www.ramsay-maunder.co.uk/benchmark-challenge/stress-linearisation/
- Nodal Forces – these are in equilibrium with the applied load.
- Hand Calculations - based on static and kinematic requirements.
- Gauss point stresses – see the following references:

https://failures.wikispaces.com/Sleipner+A+-+North+Sea+Oil+Platform+Collapse
http://www.e-periodica.ch/digbib/view?var=true&pid=bse-cr-002:1996:15::127#1083

The reader is asked to provide a brief report detailing the studies they undertook and the results they achieved and it should conclude with a list of practical conclusions for the practising engineer.

Raison d'être for the Challenge

This challenge is an interesting one since it looks again at some of the mistakes that occurred in the original analysis of the Sleipner Platform and thus provides a reminder, should it be needed, that considerable care is required when dealing with finite element results.

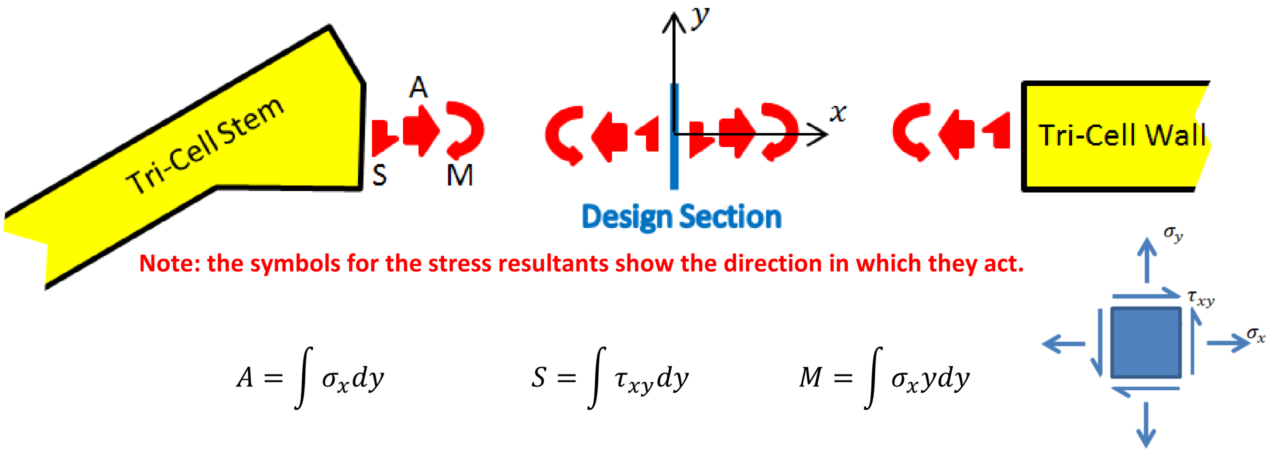
As the engineer charged with specifying the reinforcement for the platform you are interested in being able to establish the critical sections where the stress resultants are a maximum. If adequate reinforcement is not placed at these sections to resist the loads, then the design will fail. The engineer can often predict, from experience, where these sections *might be* and, as in the shear resultant for the tri-cell, the magnitude of a statically determinate resultant. Statically indeterminate stress resultants cannot be so easily determined particularly for complex structures or components and the engineer is then required to undertake appropriate structural analysis.

If the structural analysis is performed using a standard conforming finite element (CFE) system then the engineer will have at his disposal a set of strongly equilibrating nodal forces and a set of weakly equilibrating finite element stress fields from which to determine the stress resultants. This response examines the ways in which these quantities may be used to determine the stress resultants and also how accurately they are captured. It will also look at the properties of Gauss point stresses and expel some of the myths that exist about their properties.

Henry Petroski, in his excellent book, 'Design Paradigms: Case Histories of Error and Judgement in Engineering', discusses an observation that, at least for bridges, major failures are spaced at thirty year intervals. The reason for this is postulated as being the result of a 'communication gap' between one generation of engineers and the next. It is therefore appropriate that one generation on from the Sleipner incident, practising engineers are reminded of why the failure occurred.

Stress Resultants

These are simply the forces and moments due to the stresses across a section. The critical Design Section in the Sleipner model is that between the stem and the wall (figure 1) and can be isolated, i.e., as a free-body, as shown in figure 2. The axial force, A, the shear force, S, and the moment, M, may then be drawn on either side as shown. The stresses acting on the section may then be integrated across the section to produce the three stress resultants.



$$A = \int \sigma_x dy \qquad S = \int \tau_{xy} dy \qquad M = \int \sigma_x y dy$$

Figure 2: Stress resultants on the Design Section (unit dimension into the page is assumed)

The shear resultant is determined exactly through an assumption of symmetry and consideration of statics. The axial force and moment resultants, however, are statically indeterminate with the exact values depending on the relative flexibility between the stem and the wall. This can be assessed through finite element analysis.

It is worth noting that the ideas of stress linearisation were developed in the pressure vessel industry where the codes of practice require engineers to estimate the stress resultants at critical sections called *stress classification lines* (SCL). In this challenge, we are interested in the SCL shown in figures 1 and 2.

Stress Resultants by Hand

It is always good practice, where possible, for the engineer to estimate the stress resultants by hand calculation. The loading on the tri-cell wall is shown in figure 3 together with diagrams representing the distributions in axial force, shear force and bending moment along the span of the beam. The figure shows the moment resultant being equal to the fixed end moment. This would be true if the tri-cell stem was rigid and, therefore, for the real flexible stem, the fixed end moment will provide a conservative (upper bound) of the actual moment.

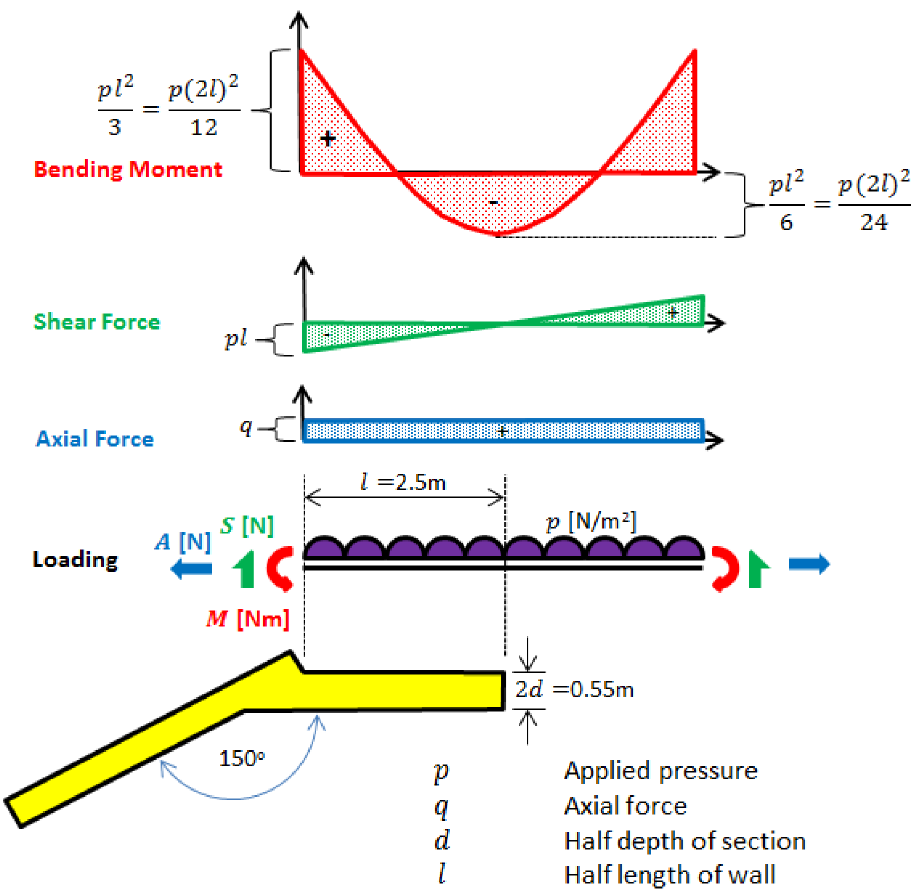


Figure 3: Hand calculation for shear and moment stress resultants

The axial stress resultant may be estimated through a consideration of the continuity of displacements (kinematics) at the design section – see figure 4.

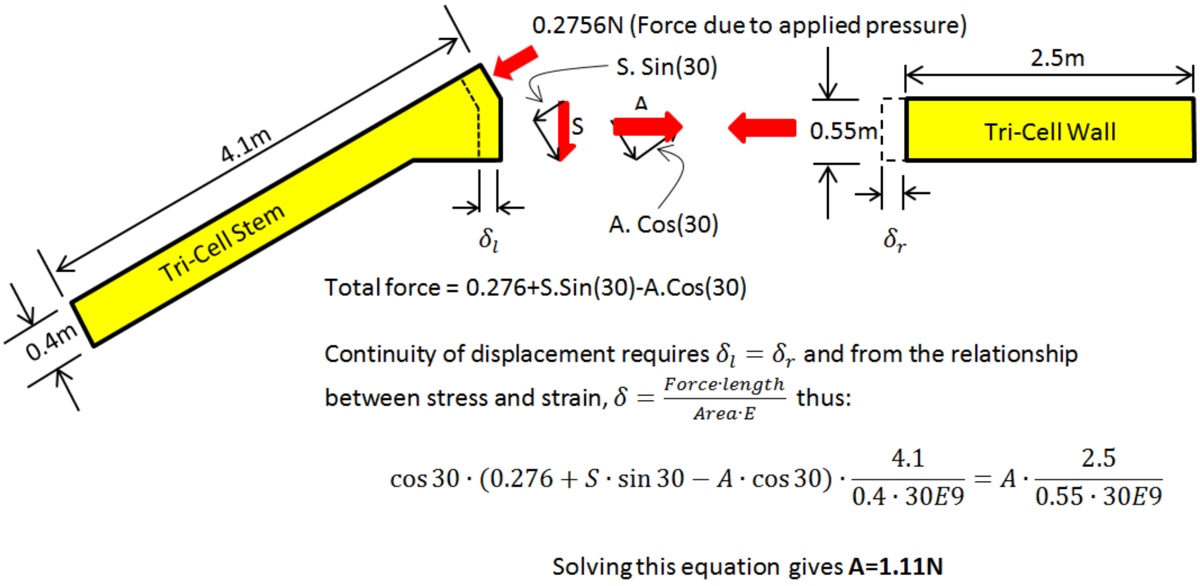


Figure 4: Hand calculation for axial stress resultant

The complete set of stress resultants as calculated by hand is shown in table 1. The shear resultant is exact, the moment resultant is a conservative upper-bound value and the axial resultant is a non-conservative lower-bound value as part of the tri-cell stem was assumed to be rigid.

	Beam
Axial	1.11N
Shear	2.5N
Moment	2.083Nm

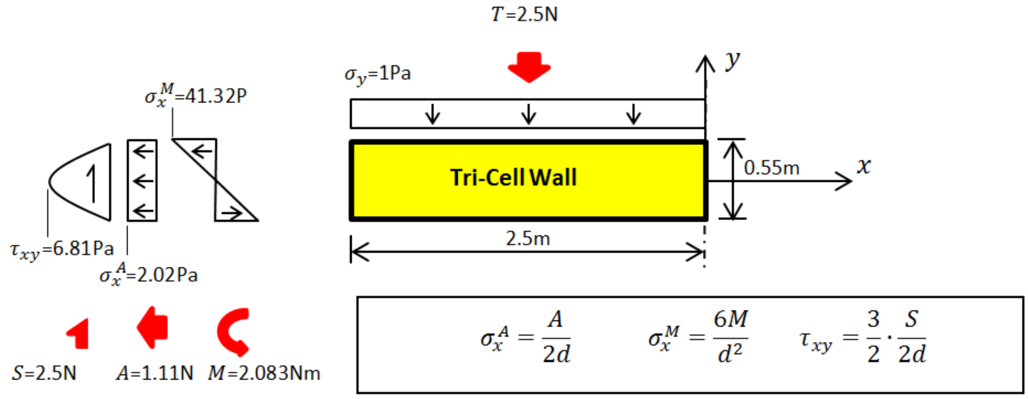
Table 1: Stress resultants from hand calculation

It should be noted here that the stress resultants calculated above are perfectly adequate for the design of the tri-cell. It would appear, however, that the Sleipner engineers did not originally do such a check although it is revealed in the references that the replacement for the failed platform was designed using the sort of simple hand calculation outlined above. Acknowledging that the engineering analysis could have stopped at this point, it is of interest to persue further the idea that accurate stress resultants, taking account of the flexibility of the stem, should also be available from a finite element analysis of the tri-cell.

Verification of the Finite Element System

Before launching into any finite element analysis it is wise, first, to ensure that the element type to be used in the selected FE system is functioning correctly. Most commercial software vendors supply benchmark problems with which they will have verified the software and which may be used by the engineer to confirm that all is correct. Many of these benchmarks have known theoretical solutions. In addition to checking the FE system is sound, such verification problems give the engineer the opportunity to understand how a particular element performs, for example, in the level of mesh refinement that might be needed to recover accurate stresses. If a benchmark problem can be found that is similar to the actual problem considered then this will also provide an initial understanding of how the structure works.

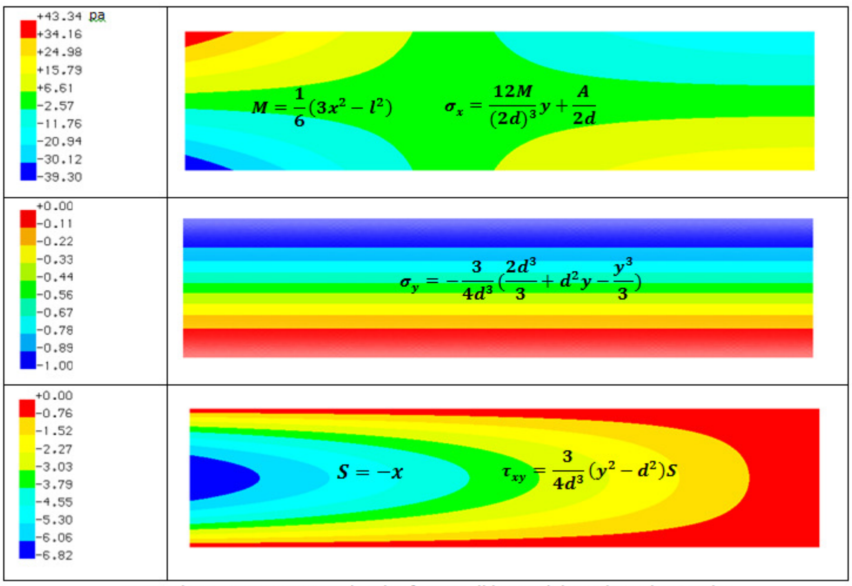
Given that the span/depth ratio of the tri-cell wall is relatively large, then engineer's bending theory may be considered valid; the influence of shear deformation will be small. If this is to be considered as a plane elasticity problem then the stress resultants will be distributed in the form of stresses in the usual manner for beams as shown in figure 5.



The shear stress distribution is assumed to be parabolic so that the peak value, at the centre of the section, is 1.5 times the average value.

Figure 5: Static boundary conditions for the tri-cell wall sub-model

The axial stress resultant does not change along the span of the beam whereas the shear resultant varies linearly with span and the moment resultant quadratically as shown in figure 3. At any position, x, along the span the shear stress varies quadratically across a section (x=constant) and the axial stress due to the moment varies linearly. The axial and shear stresses may then be written in terms of the stress resultants and these fields have been plotted in figure 6.



Note: The contour range used in this figure will be used throughout this article.

This solution for the axial and shear stress may be derived using an Airey Stress Function approach – see, for example, E.J. Hearn, 'Mechanics of Materials', Vol. 2, 2nd Edition, p699, Pergamon, (1985).

Figure 6: Axial and shear stress distributions for the tri-cell wall sub-model

The consistent nodal forces for a single element model of the tri-cell wall are shown in figure 7. The forces shown in blue, due to the uniformly distributed load and the shear, cancel out and thus do not appear in the final set of consistent forces. The four-noded element with reduced integration has a single integration or Gauss point at the centre of the element and the finite element stresses (axial and shear) are shown in the figure.

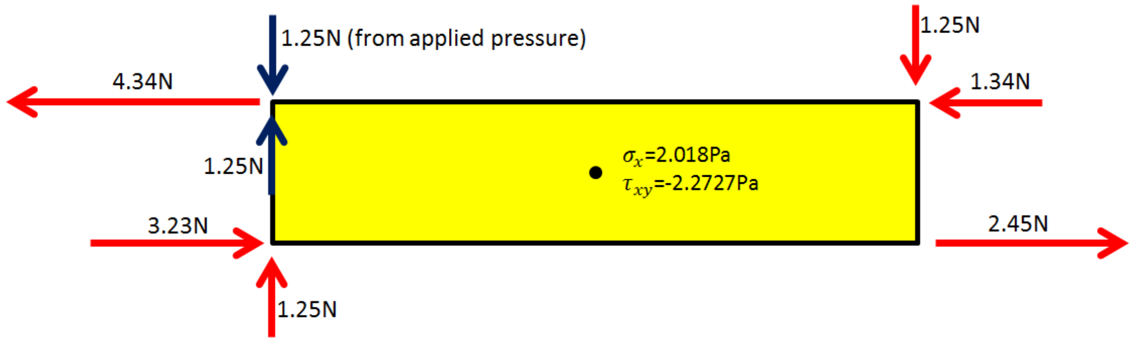


Figure 7: Consistent nodal forces for a single element model of the tri-cell wall sub-model

It is good to see that the total shear force on the left hand end agrees with the shear stress resultant as expected. It is, however, a little worrying that, if the finite element system only reports net nodal forces then the engineer may not easily check the shear resultant on the left hand end!

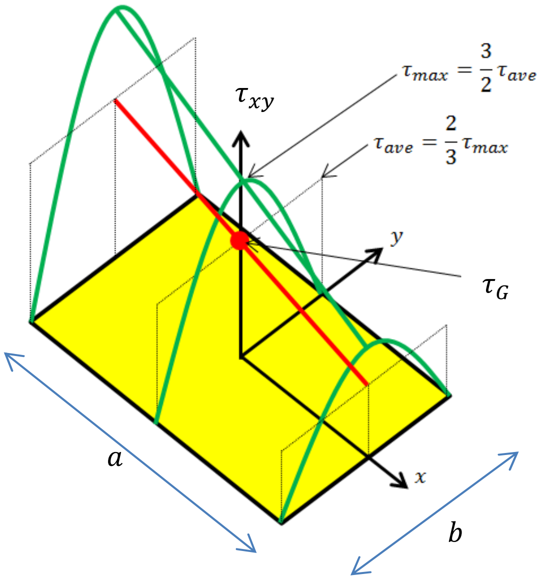
The stresses coming from the finite element model's single central Gauss point are compared with the exact values in table 2.

	FE	**Exact**
σ_x	2.018	2.018
τ_{xy}	-2.273	-3.409

Table 2: Comparison of stresses (N/m^2) at centre of tri-cell wall sub-model

The axial stress from the FE model is seen to be exact whereas the shear stress is not. It is interesting to note that the axial stress at the centre of the tri-cell wall is also equal to the average across the section, i.e., that due to the axial stress resultant. It might be the case that the same is true for the shear stress and, if the exact shear stress at the centre is scaled down by 2/3 to represent the average value across the section, it is seen that the finite element shear stress, like the axial stress, is also the average of the exact distribution across the central section.

Thus an interesting property of the Gauss point stresses is observed. However, the cautious engineer might question whether this is a general property or one that only occurs for the particular configuration currently being studied. Since, for this configuration, the shear stress varies linearly in the spanwise direction of the beam then a little thought will lead to another conclusion, this being, that the Gauss point stress is also the average value over a centrally positioned spanwise section. Further, the Gauss point stress is also the average value for the entire element! These ideas are illustrated in figure 8 where a shear stress plot for a spanwise section of the tri-cell wall is provided.



$$\tau_G = \frac{\int \tau_{xy} dy}{\int dy} \; ; x = 0$$

$$\tau_G = \frac{\int \tau_{xy} dx}{\int dx} \; ; y = 0$$

Length Weighted Average (LWA) Shear Stress

$$\tau_G = \frac{\int \tau_{xy} dA}{\int dA}$$

Area Weighted Average (AWA) Shear Stress

Where:

$$\int dx = a \quad ; \quad \int dy = b \quad ; \quad \int dA = a \cdot b$$

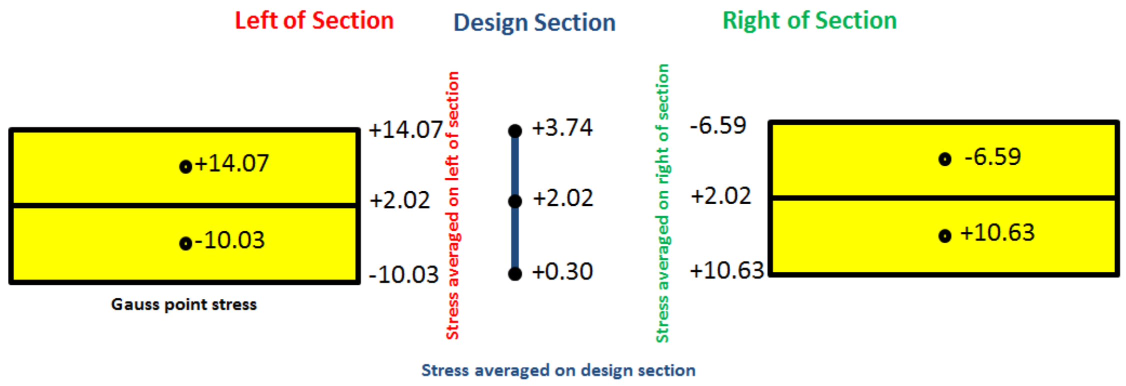
Figure 8: Shear stress distribution spanwise linear, depthwise quadratic

There is another method of finding stress resultants from a finite element model and this is to integrate the finite element stresses across the section. This method is known as stress linearisation.

Stress Linearisation

This is a method often used in finite element analysis to obtain stress resultants on sections of interest such as the Design Section. It performs the same integration as identified in figure 2, but on the finite element stress and using a numerical integration scheme. The method comes from the pressure vessel industry where the codes of practice require stress resultants to be checked at critical sections in an axisymmetric FE model. It is important, when using this approach, that the stresses are considered in a coordinate system normal and tangential to the section of interest. If this is not done, then the stress resultants cannot be correctly evaluated.

In the tri-cell wall example, the direct stress parallel to a section (σ_y) does not contribute to the stress resultants and the linearised value of this stress has no practical meaning. Similarly, the stress invariants (principal stresses, von Mises stress etc.) vary in direction or indeed have no direction along a section and thus it is difficult to see any physical meaning of linearised values of these quantities. It is worth understanding how stress linearisation is performed across a section and the axial stresses for a two element/edge mesh are shown in figure 9. Note that in this figure, the design section refers to a section half way along the tri-cell wall and not the design section defined in figure 1.

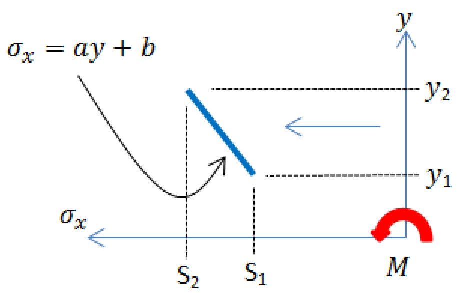


Note that the design section shown above is not in the same position as that of figure 1. The current design section is being used simply as an example of how stress linearisation can lead to different stress resultants depending on whether it is performed to the left or right of the section or using averaged values on the design section itself.

Figure 9: Nodal averaged axial stress on either side of, and at, a sample design section

The finite element stresses start life at the single central element Gauss points. These are then extrapolated, as a uniform field, to the nodes of the element and with only a single central stress this means that the nodes are given the same stress. The stresses are known as nodal unaveraged stresses. The unaveraged nodal stresses may then be averaged and this averaging takes place over the nodes on the selected elements. Thus, for a design section, it is possible to select just the elements on the left of the section, just the elements on the right of the section or the elements both sides of the section. The nodal averaged stresses for these three scenarios are shown in the figure. Stress linearisation can then be done along the left of section, the design section and the right of section and the stress variation on each will be piecewise linear.

As an example of how stress linearisation is performed, the equations for a linear stress variation are provided to evaluate the moment resultant in figure 10. The equations may be simply set up in a spreadsheet and the moment from each piece of the piecewise distribution determined. The total moment is then the sum of the moments from each piece. The values for calculating the moment resultant along the design section, taken from figure 9, are shown in the table at the bottom of the figure. Stress linearisation gives the moment as 0.087Nm. This may be compared to the exact value, which may be determined using the equation for moment as a function of axial position given in figure 5. This equation produces a moment of 0.260Nm and it is seen that there is a significant difference with the FE results (from linearisation) being only one third of the correct value! This is simply a manifestation of the fact that the finite element stresses from conforming displacement elements do not generally balance the applied loads.



$$M = \int \sigma_x\, y dy = \left[\frac{ay^3}{3} + \frac{by^2}{2}\right]_{y_1}^{y_2} = I_{y_2} - I_{y_1}$$

where:

$$a = \frac{s_2 - s_1}{y_2 - y_1}$$

$$b = s_1 - ay_1$$

$$I_{y_1} = \frac{ay_1^3}{3} + \frac{by_1^2}{2}$$

$$I_{y_2} = \frac{ay_2^3}{3} + \frac{by_2^2}{2}$$

y_1	y_2	S_1	S_2	a	b	Iy_2	Iy_1	Iy_2-Iy_1
-0.275	0	0.297	2.018	6.258	2.018	0.000	0.033	-0.033
0	0.275	2.018	3.740	6.262	2.018	0.120	0.000	0.120
								0.087Nm

Figure 10: Stress linearisation for moment stress resultant from linearly varying stress

With the stress resultants for the centre section of the tri-cell wall known, the error in those determined from stress linearisation of the finite element stresses can be evaluated. The results from a convergence study on the central stress resultants are shown in figure 11, which plots the relative error (defined in the figure) against the number of elements/edge in log-log space.

The shear resultant for the design section is exact even for the coarsest mesh and so is not included in the figure. For the coarse meshes there appears to be no consistency between the stress resultants worked out on the three sections. However, as the mesh is refined, the resultants evaluated on either side of the design section converge. The resultants calculated at the design section are more accurate (shear resultant is exact) than those both sides of the section, and converge more rapidly as the mesh is refined. It is only because the exact stress resultants for this problem are known that an error plot such as figure 11 may be prepared. Where the exact resultants are unknown, the practising engineer is faced with three sets of resultants, all generally different and all potentially in error to some unknown degree!

In this example reasonable engineering accuracy (taken here as <5%) is only achieved with the most refined mesh and the number of elements required (32 per model edge) might be considered as quite excessive by many practising engineers. Note though that if instead of the central design section, the moment resultants have been used from one of other side of the section, then even with the most refined mesh there would remain a 20% error. This level of accuracy is clearly unacceptable for practical engineering purposes. It is seen, in the figure, that the convergence does not settle down to a linear form until the four element/edge mesh; the moment calculated on the right hand side of the design section has greater error for this mesh than for the two elements/edge mesh. This sort of behaviour is not unusual for very coarse models where such behaviour is described as pre-asymptotic convergence.

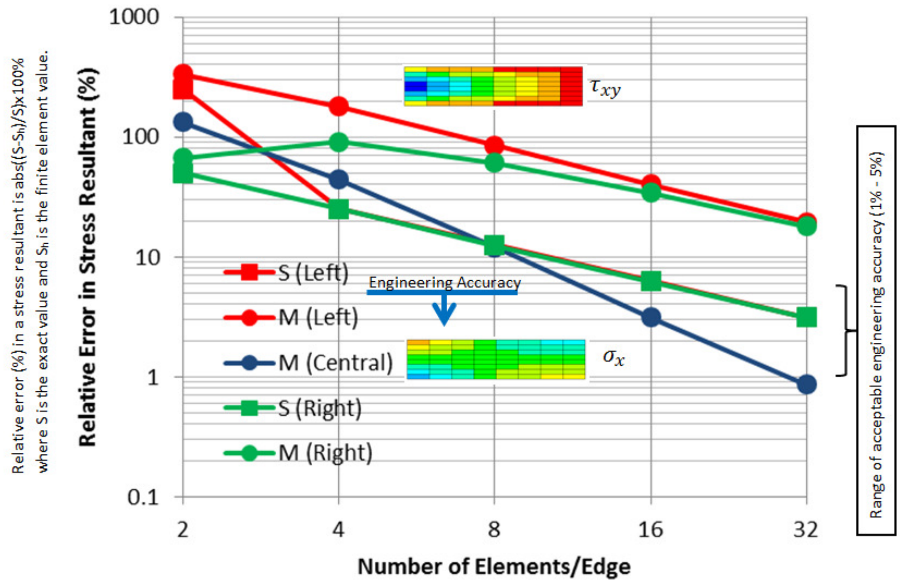


Figure 11: Convergence of error in shear and moment stress resultants (from linearization)

Having investigated the various methods of recovering stress resultants from a finite element model on the tri-cell wall sub-model, the full model of the tri-cell (figure 1) will now be investigated.

Analysis of the Sleipner Tri-Cell

An initial analysis was conducted on the basic mesh of figure 1. The nodal forces at the end of the tri-cell wall are shown in figure 12 and the stress resultants have been worked out using these nodal forces and may be compared with those from hand calculation.

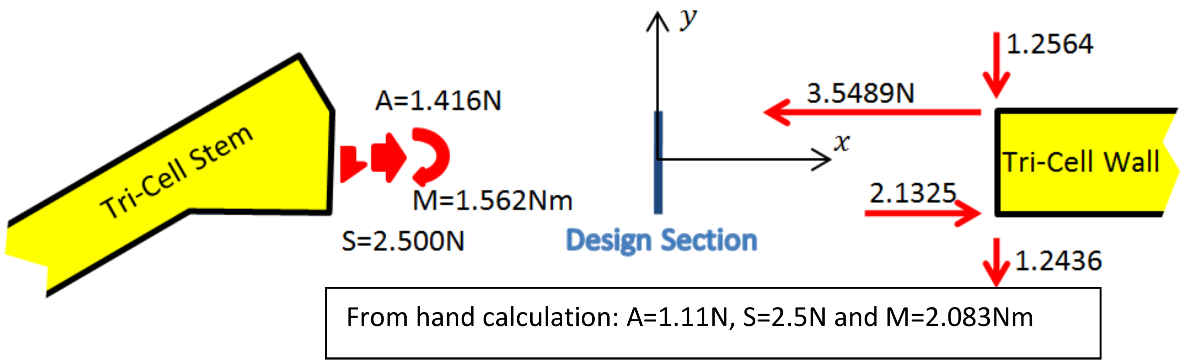


Figure 12: Stress resultants for full tri-cell model

As the axial and moment stress resultants are statically indeterminate, then they will converge with mesh refinement and this is shown in figure 13. There is a change of >10% in these resultants as they converge!

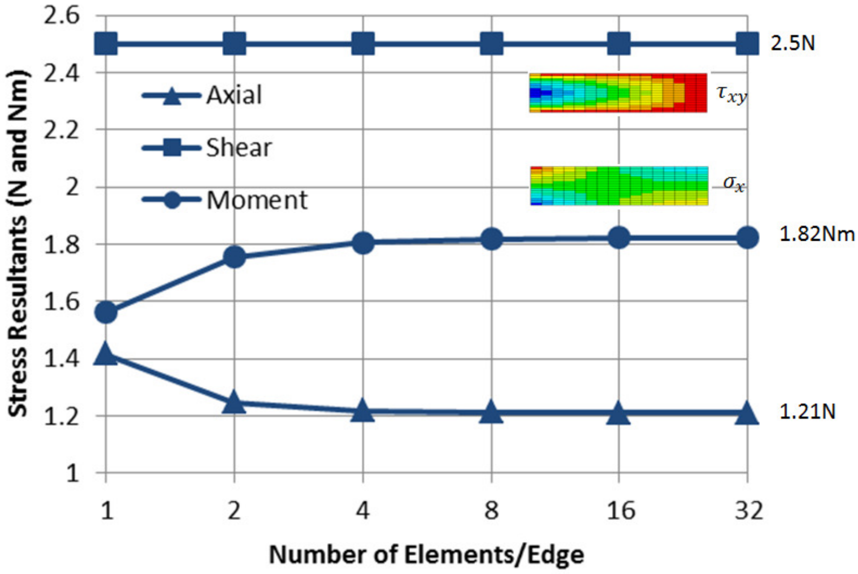


Figure 13: Convergence of stress resultants calculated from nodal forces

Comparing the converged stress resultants for the full model with those for the sub-model one sees, as expected, that the shear resultants match exactly. The axial stress resultant is 10% greater for the full model and this might be accounted for by the rigid portion of the stem that was not considered in the hand calculation of this quantity. The moment resultant for the full model is about 0.875 of that for the sub-model and this is the sort of value expected, as the tri-cell stem will rotate to a small degree. This can be clearly seen in figure 14, which shows the magnified displaced shape.

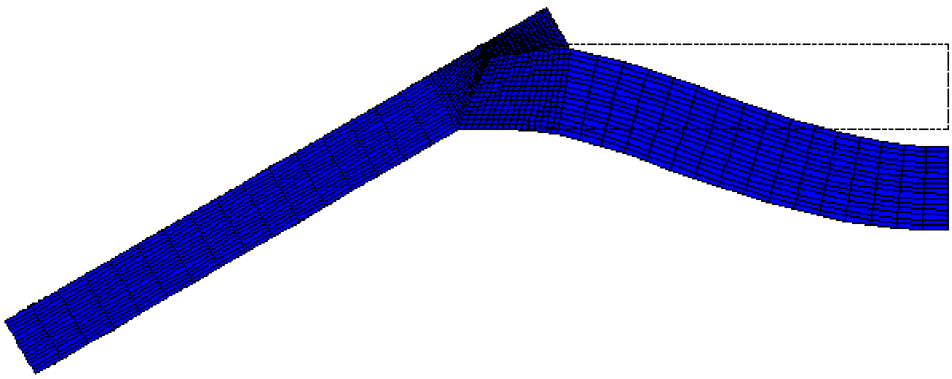


Figure 14: Magnified displaced shape (16 elements/edge mesh)

In the same way as stress linearisation was used to calculate the stress resultants at, and to the left and right of the design section for the sub model, this has also been carried out for the full model and the convergence of the resultants is shown in figure 15.

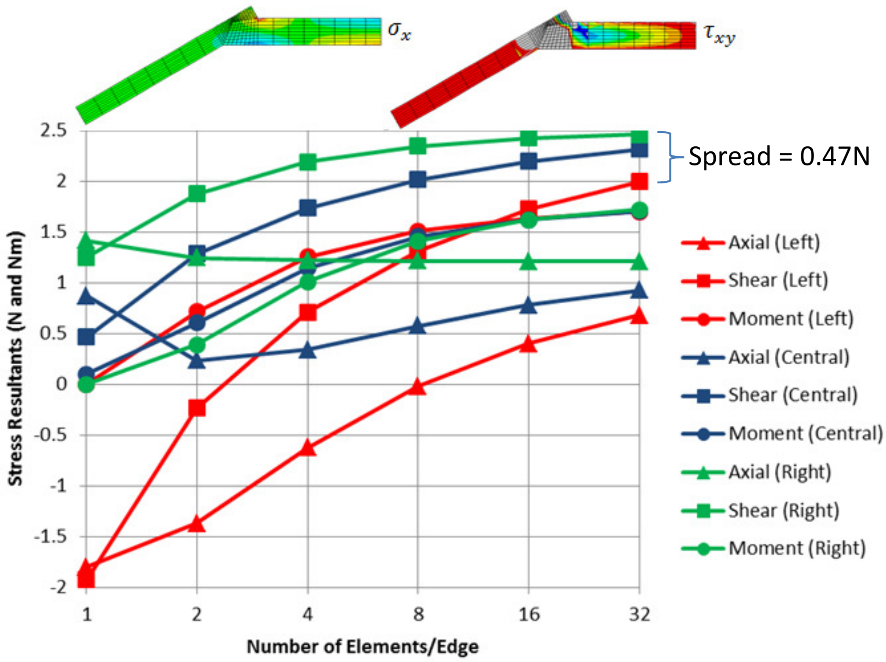


Figure 15: Convergence of stress resultants calculated from stress linearisation

For the coarse meshes the sign of the axial and shear resultants is incorrect on the left of the design section and even for the most refined mesh there remains significant spread depending where the linearisation was conducted – 0.47N in the shear which, as we know the true value to be 2.5N, represents a difference of some 20%.

As the exact stress resultants are known for each mesh (figure 13) the error in the values from stress linearisation may be determined and these have been plotted in figure 16. The error in the stress resultants from linearisation is significant and for the most refined mesh only the axial and shear resultants to the right of the design section are acceptable. If the engineer had not calculated the exact resultants from nodal forces then he/she would have been left with having to use figure 15, which would, to say the least, be rather unsatisfactory.

Thus, stress linearisation requires significant mesh refinement to recover accurate resultants even for a stress field found in a beam under a uniformly distributed load – see figure 11. When the design section goes through a point of stress singularity, then the results are polluted by the presence of the singularity and it becomes almost impossible to make a sound engineering decision on the data that is obtained.

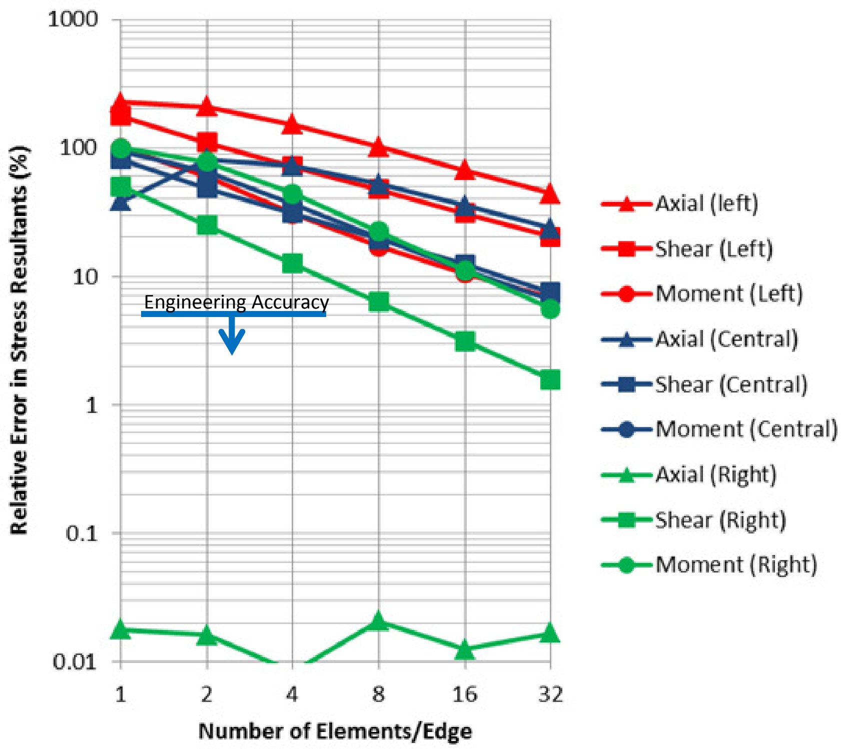


Figure 16: Convergence of error in stress resultants calculated from stress linearisation

By using the idea that for this problem the Gauss point stress, for an undistorted element, gives the exact average shear across a section one can modify the mesh such that a thin element is placed adjacent to the design section. As can be seen in figure 17, the shear stress resultant formed from the Gauss point stress is within 0.5% of the true value.

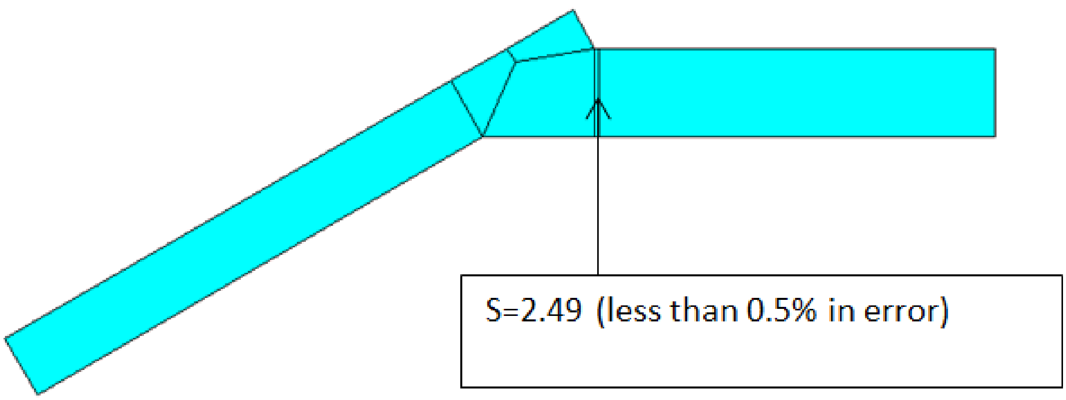


Figure 17: Convergence of Shear and Moment stress resultants

The method just described assumes no knowledge of the shear distribution along the length of the tri-cell wall. Having found a theoretical solution with similar characteristics to the tri-cell wall it is now known that the shear stress varies linearly along the wall and with two exact values, one can extrapolate to the design section. These could be obtained from the basic mesh since the value at the centre of the (full) tri-cell wall is zero and at the quarter point (the Gauss point location for the basic mesh) the value is 2.2727Nm^2. Extrapolating to the design section gives an average shear of 4.5454 Nm^2, which integrated across the thickness gives 2.5N!

Closure

This response has shown the virtue of holding back from the real analysis until one has gained a better understanding of the problem, and how it might be modelled using finite elements. Had the Sleipner engineers done this they would have realised, as they did later, that the solution does not require a finite element analysis at all; hand calculation would have been sufficient. They would also have realised that that Gauss point shear stresses are only exact in the sense of average sectional shear values when the elements are rectangular. To demonstrate this last point figure 18 shows the shear stress resultants calculated from the Gauss point stresses for a mesh of distorted elements similar to the one used by the Sleipner engineers. The error in the stress resultant at the section through the Gauss point is some 48% and if this value is used as the worst shear resultant then it is some 62% in error with the value at the design section.

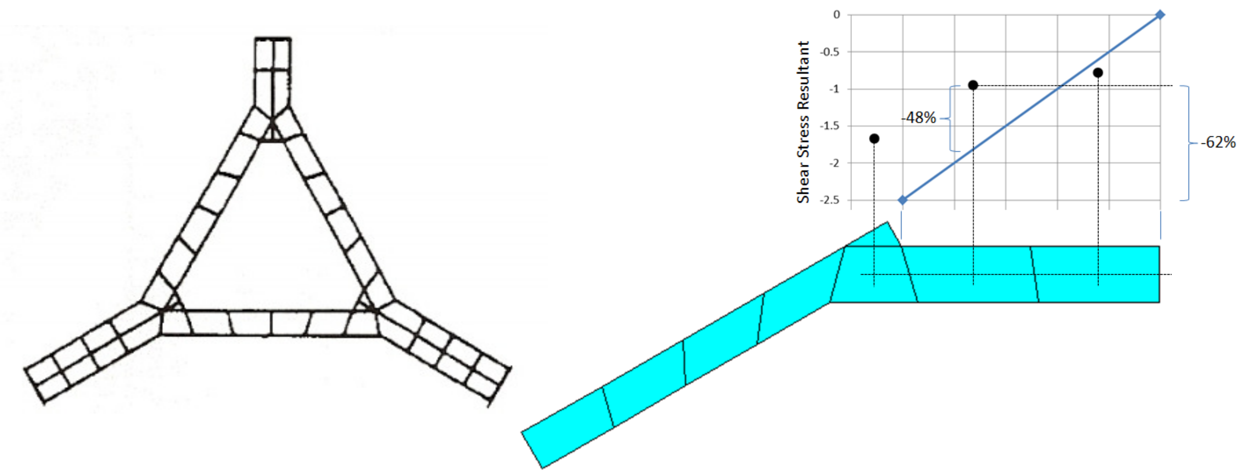


Figure 18: Convergence of Shear and Moment stress resultants

It is perhaps surprising that the Gauss point stress are so sensitive to element distortion but in reality it is nothing more than a happy coincidence that they are exact, in some sense, for the undistorted rectangular element and under the loading considered.

The reason that such a coarse mesh was used by the Sleipner engineers was that the complete structure was modelled in eight-noded hexahedral elements and with the computational resources available, which was the sort of mesh required. When such a coarse mesh is used, it is normal practice to consider sub-modelling regions of particular interest. However, if this had been done then, as demonstrated through the theoretical solution, a significant level of mesh refinement would be required to capture with reasonable engineering accuracy the peak shear stress of the stress resultants through stress linearization.

It is interesting to note that the error in the stress resultants calculated by linearization is the error in the equilibrium of the stress resultants since they are being compared with values that are in equilibrium with the applied load. This is the nature of the low-degree conforming finite element used in the majority of commercial finite element systems. The stresses are of low-fidelity and in order to obtain equilibrating stress resultants requires, often, a significant effort in mesh refinement. It is also noted that in this case the stresses and therefore the resultants appeared to converge from below the true value so that the results are both erroneous and non-conservative. It has also been demonstrated how strong equilibrium may be recovered, at least for stress resultants, when nodal forces are used and given the poor performance of stress linearisation with low-fidelity non-equilibrating stress fields, this approach should be favoured by the practising engineer.

In the Sleipner model, the design section had a stress singularity at one end. No standard finite element can cope with infinite stresses and, from an engineering perspective, these are often ignored as a manifestation of an idealised finite element model (a sharp corner in this case). If the stress singularity is ignored then the stresses across the design section are linear and quadratic, respectively, for the axial and shear stress. The four-noded element used in this study is not much better than a constant stress element and, therefore, it is not surprising that it performs so poorly. The higher-order (eight-noded) element would have performed better but, with little more than a linear stress capability, it would still have required significant mesh refinement to achieve sensible stress resultants. Since many practical engineering problems involve bending moments and shear forces of the type seen in the tri-cell wall, it seems somewhat inappropriate that higher-fidelity elements are not provided by software vendors. For example, in a displacement formulation, a cubic (p=3) displacement field element would have recovered exactly, with a single element, the stress field in the sub model. In the presence of the singularity in the full model, however, it would probably have struggled and would certainly not have provided a stress field at the design section in equilibrium with the applied loads.

Practical Conclusions

This study, based on the Sleipner failure, has provided some useful results for the practising engineer:

1) Statically determinate stress resultants may be found by hand but statically indeterminate ones require some form of analysis that includes the relative flexibility of the structure, e.g., finite element analysis.
2) Stress resultants may be determined from nodal forces if they are interpreted correctly. The statically indeterminate resultants will converge with mesh refinement whereas statically determinate ones are exact irrespective of the mesh chosen.
3) Gauss point stresses have certain properties that may be useful to the engineer but as these properties do not necessarily extend to distorted elements and general stress fields the engineer needs to be certain that they apply to his/her situation before making a potentially rash decision to use them.
4) Stress linearisation is a valid way of determining the stress resultants across a section. However, because CFEs provide finite element stress fields that are only weakly in equilibrium with the applied load, then considerable mesh refinement might be needed to obtain accurate results. If the design section runs through a point of stress singularity then the stress field adjacent to the singularity is polluted and it is unlikely that equilibrating stress resultants can be obtained through stress linearisation.