Plate Element Usage Guidelines

Engineer's Studio
From Ver 6.0.0

Engineer's Studio supports two basic types of plate bending element. The 6 & 8 node elements available from the first version are members of the serendipity family of elements. Professor Maekawa used the serendipity element in his research during the 1980's in what is now known as the Maekawa Concrete model. Hence this element was adopted in Engineer's Studio initially. However, the serendipity plate element has some weaknesses. To overcome these weaknesses new 3 & 4 node elements have been introduced in V6. These contain the plate bending model proposed by Soh. The high order serendipity elements remain in V6 to support user models created previously. However, it is recommended that the low order 3 & 4 node elements are used in preference to the serendipity elements. The new low order elements have been tested against the original experimental results used in the verification of the Maekawa concrete model and found to give good results. From V6 the 4 node Soh element replaces the 4 node serendipity element present since V1. The 4 node serendipity element is no longer available in the product.

All element types are susceptible to certain common FEM issues. In addition the serendipity elements introduce a few of their own unique problems. This document gives guidance on issues encountered in the Engineer's Studio implementations. It includes hints for how to detect when problems have occurred and how to remedy them.

1 Shear Locking & Reduced Integration

The main problem with the serendipity plate element is a phenomenon known as shear locking. A solution to this is reduced integration and this was promoted by some authors. The following integration schemes are implemented in Engineer's Studio.

Element	Flexural Stiffness Integration Level	Out of Plane Shear Stiffness Integration Level	Mass Integration Level
Triangular 3 Node	2	2	2
Triangular 6 Node	2	2	2
Quadrilateral 4 Node	3	3	2
Quadrilateral 8 Node	2	2	2

The 6 node and 8 node elements have their integration level reduced from 3 to 2 for both flexural and out of plane shear stiffness. This is known as uniform reduced integration.

The above integration schemes for the 6 & 8 node elements resolve the main problem of shear locking and give reliable results for moments and displacements. However reduced integration introduces new issues. Namely, zero energy modes and poor out of plane shear force reporting. These are discussed further in the examples section below.

Integration schemes chosen for the 3 & 4 node elements are such that exact integrals result. i.e. Increasing the integration level does not change the result. These elements are free of shear locking.

2 General FEM Modelling & Result Verification

The overall approach to FEM modelling is to make a fairly rough initial mesh and calculate some results. The results are verified by inspection and problems noted. The mesh is improved and the model recalculated until the modeller is satisfied the original problem has been satisfactorily represented. It is an iterative process.

Meshes are often made finer to improve results. Making the mesh finer to improve the result is commonly known as ‘h’ convergence where h is the common symbol for element size. (A common related term is ‘p’ convergence which involves using higher order elements to achieve faster convergence). The basic concept of FEM analysis is that as h becomes smaller the approximation converges to the exact solution of the underlying differential equations. The ability to converge is a fundamental requirement of an FEM element. The plate elements in Engineer’s Studio do converge as h decreases. However as h becomes smaller the model size increases proportionally to the square of the reduction factor for 2D elements and hence the processing time increases. It is this balance between accuracy and speed the user must consider when creating models.

The shape of the individual elements is also important for obtaining good results. In general the 6 node element should be as close to an equilateral triangle as possible. The 4 and 8 node elements should be as close to square as possible. The auto mesh libraries in Engineer’s Studio attempt to achieve this. However in some geometrical cases achieving the ideal element shapes may be difficult for an auto mesh function. In such cases the user should consider breaking complex regions into smaller simpler regions and applying the auto mesh function to each such smaller region. This offers more control and can lead to better meshing.

Detecting modelling problems during the iterative modelling process is very important and this is where contour plots and the related post processing options are very important. When inspecting contour plots two types of issues in particular should be checked for, result steps at element boundaries and stress concentrations

2.1 Result steps at element boundaries

The plate element formulation in Engineer’s Studio is such that the shape functions for displacements and rotations are continuous over element boundaries, but moments and out of plane shears may not be. Steps should never occur in displacement and rotation contour plots, however, even in good models they will occur in bending moment and out of plane shear force plots.

Engineer’s Studio V5 contains a new post processing feature that enables these steps to be viewed. Contour plots can be prepared with no smoothing or any other post processing applied. This is called the “None” option for contour plot post processing and is referred to here as the “raw” results. Each element is plotted independently from all other elements. (Refer to the document “Contour Plots” for further details).

When large steps across element boundaries are observed in the raw results it may be a sign of poor meshing and improvements to the mesh should be considered. The size of the step across the element boundary compared to the magnitude of the moments in the adjacent elements gives an indication of the significance of the error. The user should refine the input until the errors are acceptable. Out of plane raw shear forces however often show steps which are difficult to reduce by mesh refinement. This is addressed in the examples section below.

2.2 Stress Concentrations

The term “stress concentration” here is adopted from general FEM theory. In the case of the plate element the stresses are the section forces N, M & S. Stress concentrations can occur at sharp corners and where concentrated loads or supports are applied. Stresses can become very large in these locations and tend to infinity. This is a feature of the combination of the geometry and the underlying differential equations when elastic models are used.

Mesh refinement using h convergence will improve the overall model results. The results will converge to the exact solution of the underlying differential equations as h reduces. However if the geometry and underlying differential equations have a stress concentration then this too will be accentuated as the mesh is refined. This does not indicate a failing of the element. All element types are susceptible to this issue. However real materials used in structures of course respond differently to a simple elastic model. Typical RC or steel structures will exhibit nonlinear properties and redistribute the stresses away from these regions. Thus the designer should consider using an average value taken over a wider area for obtaining realistic design values where stress concentrations are detected.

To assist with this, from V5, Engineer’s Studio includes a new contour plot post processing option that shows the average section forces for each element. With this option selected the average of the Gauss point values is applied to the whole element. Often these average values are suitable for use in design. However, there can be cases where the mesh is so fine that a whole element is within the stress concentration region. In this case even the average values may be too high for direct use in design. The size of the region the average is obtained from may need to be extended. Inspection of the contour plots will provide guidance in this case.

Engineer’s Studio automatically prepares various summary data containing maximum and minimum values for the contour plots. Peak values occurring in stress concentrations are included in these results and will hence distort the reported values. Care should be taken that these values are not used in design.

3 Examples

The following section presents samples of issues that can arise in the use of the plate element in Engineer’s Studio. Each example includes a brief description of the issue, the cause, how to detect it and how to mitigate it.

3.1 Zero Energy Modes

Description : The element deforms in an unusual shape that causes no energy to be expended within the element.

Cause : Reduced integration in 6 & 8 node elements causes the stiffness matrix to suffer from rank deficiency. This means that the element can deform in some particular patterns that require no strain energy. The 3 & 4 node elements do not suffer from this issue.

Detection : Inspection of displacement diagrams, including animations if dynamic analysis is done, reveals the zero energy mode shapes of the elements. The vibrations may occur in plane or out of the plane of the element. In dynamic analyses they are recognizable as a cyclic vibration that does not move the overall position of each vibrating element. In all cases detected so far, this phenomenon has been easy to detect when it occurs.

Comment : Figure 1 illustrates an occurrence in an 8 node element in static analysis however such simple models are not used typically. The issue in Figure 1 is easily resolved by meshing down to a 2×2 mesh.

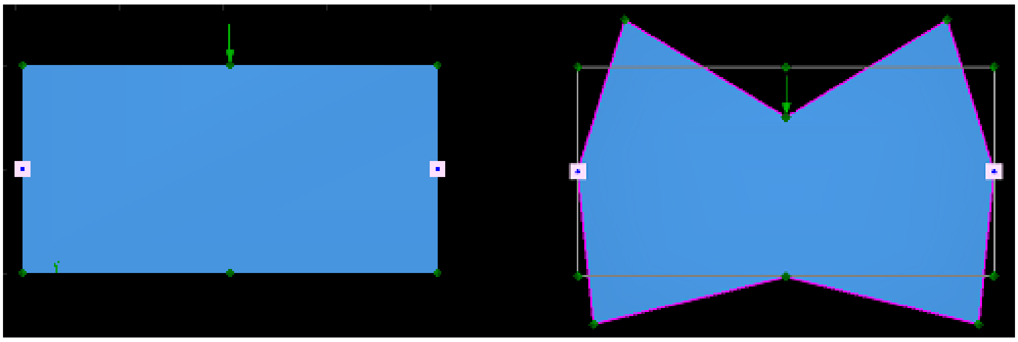

Figure 1: Zero energy mode in static analysis

In full structures this phenomenon has only ever been observed in a dynamic analysis as shown in Figure 2. In this case a model with non rectangular 8 node elements and zero damping showed excessively large zero energy mode out of plane vibrations that invalidated the overall analysis.

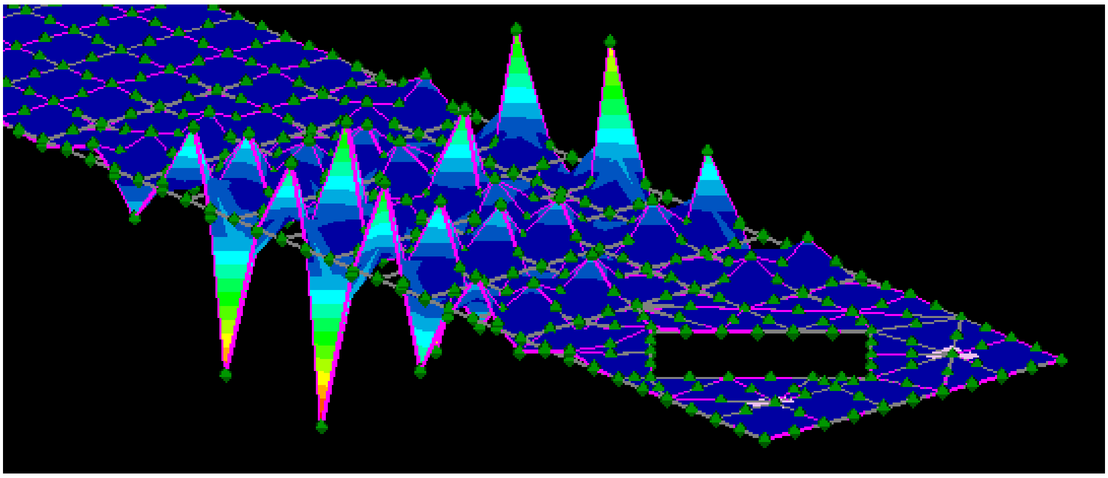

Figure 2: Zero energy mode in dynamic analysis

Remedy : In the dynamic model shown in Figure 2 a solution was obtained as follows. A large rhombus shaped region was divided into a large rectangular region and a smaller triangular region and a slightly finer mesh applied. Also, there was no damping in this model. The addition of a small amount of damping also resolved the problem independently to the re-meshing. Very few real structures have zero damping.

3.2 Stress Concentrations

Description : Very small regions contain very high stress (N, M or S) compared to stresses in adjacent regions.

Cause : As described above this issue can occur as element sizes are reduced and the results converge to the exact solution of the geometry and underlying differential equations.

Detection : The contour plots and the table data results for the max/min values per group should be inspected. The maximum and minimum values on the contour legends should be inspected. Figure 3 illustrates a stress concentration in out of plane shear forces. The model is a square plate simply supported on its left and right edges as viewed with a uniformly distributed out of plane load applied. Figure 4 is the same model and result but with the average post processing option selected. The legend colours are different to Figure 3 because in Figure 3 the peak values at the edge distort the overall spread of colouring. Figure 3 has a peak value of 10.4kN/m and Figure 4 has a peak value of 2.67kN/m. User defined contour legends could also be used to remove the distortion of the colouring in these cases. Note also that the meshing in this model is much finer than is typically required. This was done to illustrate stress concentrations.

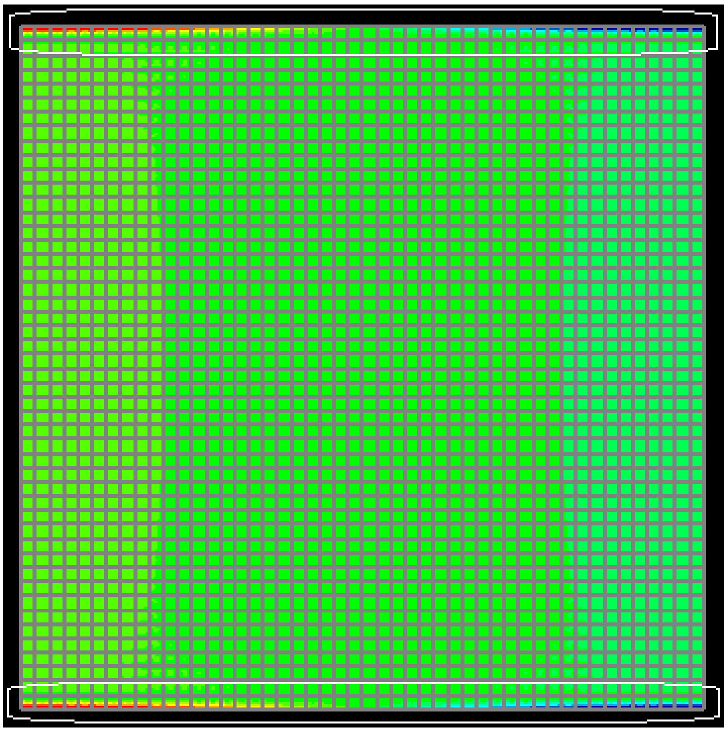

Figure 3: Stress concentrations in a simply supported plate (smoothing enabled)

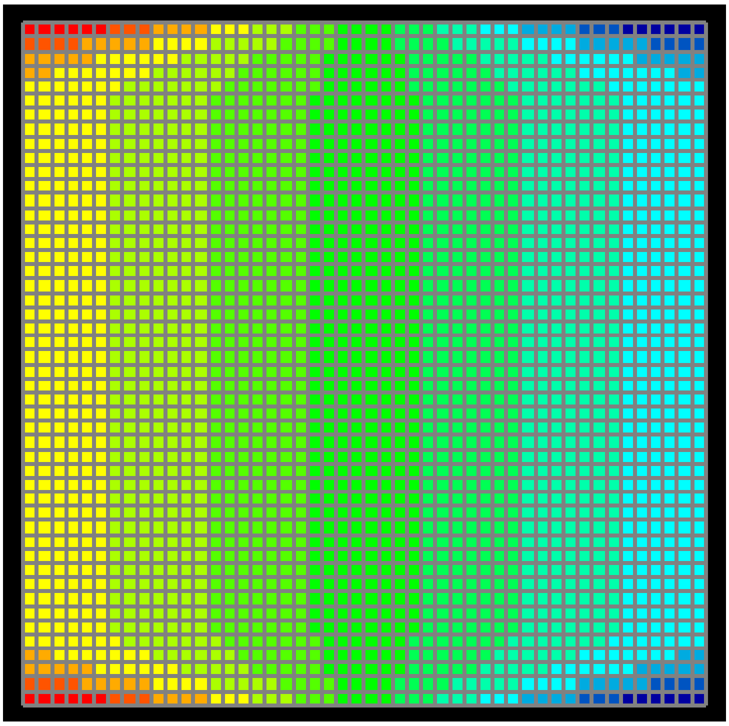

Figure 4: Stress concentration in a simply supported plate (averaging enabled)

Remedy : Avoid making elements too small in regions of stress concentration. Values to be used in design should be taken as the average values over some region. The “average” post processing option assists with this however, if element sizes are very small compared to the affected region the average may need to be taken over a region larger than the element size. The creation of material non-linear models can also be considered for RC structures.

3.3 Oscillation of Out of Plane Shears within an Element

Description : Out of plane shear forces within an element show excessive gradients in comparison to the overall trend visible over several adjacent elements. This has been observed in 6 & 8 node elements.

Cause : It is suspected that reduced integration is the cause of this. The 8 node element has a 2×2 Gauss point configuration and the values within the element seem to get distorted.

Detection : Inspect the raw out of plane shears and look for large gaps at element boundaries. Also create a cross section through the region using raw results. The overall trend can be seen, but individual element results are poor.

Comment : Figure 5 & Figure 6 show the same model and results based on the 8 node element. Figure 5 shows the raw results and here we can see the banding within each element. The values are very distorted. Figure 6 shows the results with the average post processing option applied. We can see much more acceptable values. Note that for this model the displacements and bending moments show high accuracy. Note also that this model contains a stress concentration at the corners. This may also be contributing to the banding. The trend of the stress concentration becomes clearer in Figure 6 with averaging. Similar to Figure 3 & Figure 4, we can see that the stress concentration and banding combine together to distort the colour distribution in the legend. The averaging used in Figure 6 obtains better results and resolves this.

Remedy : In cases encountered so far, applying the average post processing option produces an acceptable estimate of out of plane shears.

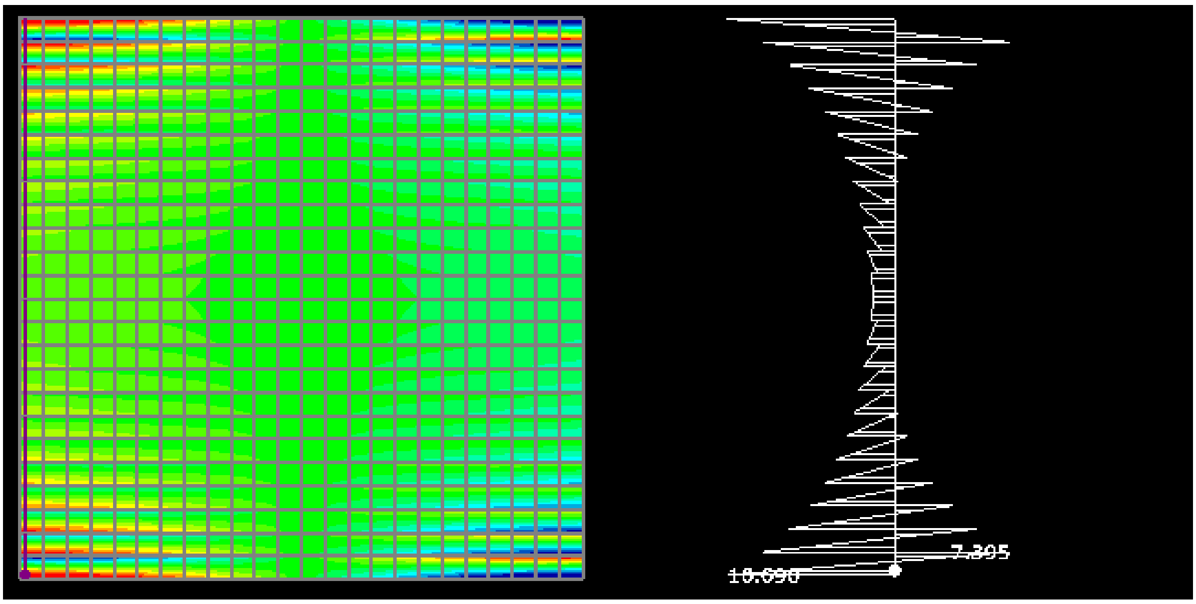

Figure 5: Intra element out of plane shear oscillations (raw)

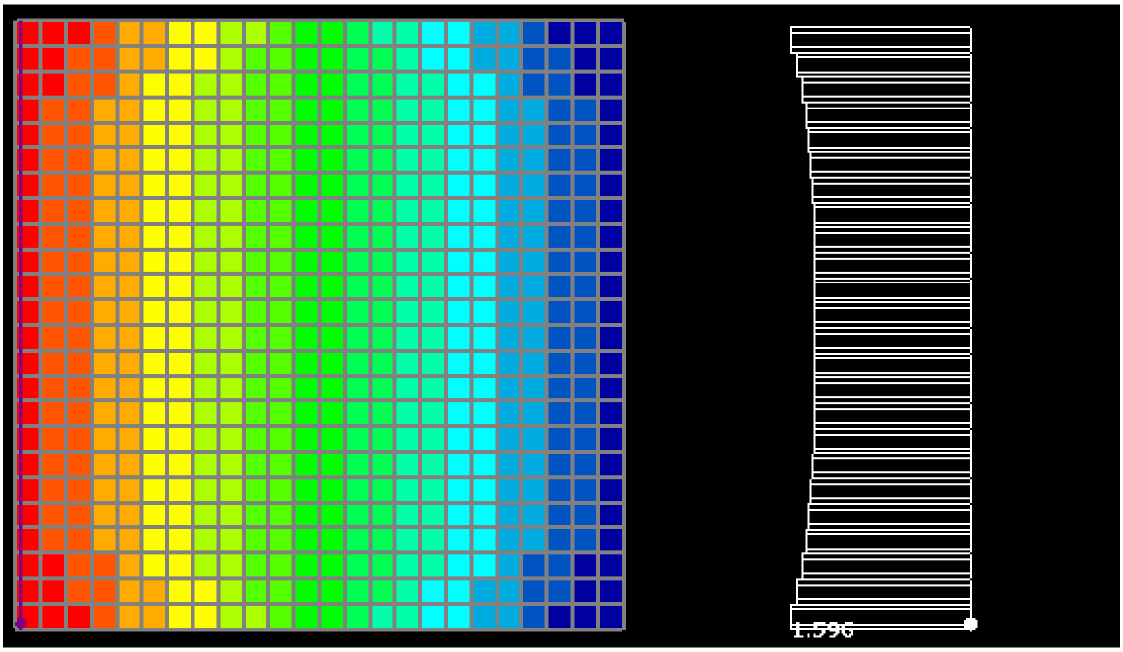

Figure 6: Intra element out of plane shear oscillations (averaged)

List of Figures