Proceedings of the Third International Conference on
Parallel, Distributed, Grid and Cloud Computing for Engineering,
B.H.V. Topping and P. Iványi, (Editors),
Civil-Comp Press, Stirlingshire, Scotland

Paper 37

A Panel Element for Parallel Finite Element Analysis

T.Q. Li
Arup, London, United Kingdom

Abstract

This paper presents a panel element that can be large in size without compromising analysis results. The development of this element is to help overcome the conflict requirements in finite element analysis. Finite element analysis principle requires element size be sufficiently small, but engineers and analysts who use finite element analysis tools sometimes like to use large size elements to simplify the modelling process and reduce the unwanted analysis results. Traditionally shell elements do not have drilling stiffness (the rotational stiffness about the normal to the shell element), but it is sometimes required in engineering practice. To meet this requirement, an alternative method has been proposed in this paper to derive the drilling stiffness that can give more accurate results than previous proposed methods. As the panel element is large and meshed internally, each panel element can be treated as a substructure and processed in different processors if they are available, this can make it easier for writing parallel finite element analysis program.

Keywords: panel element, drilling stiffness, large element, panel, floor & wall.

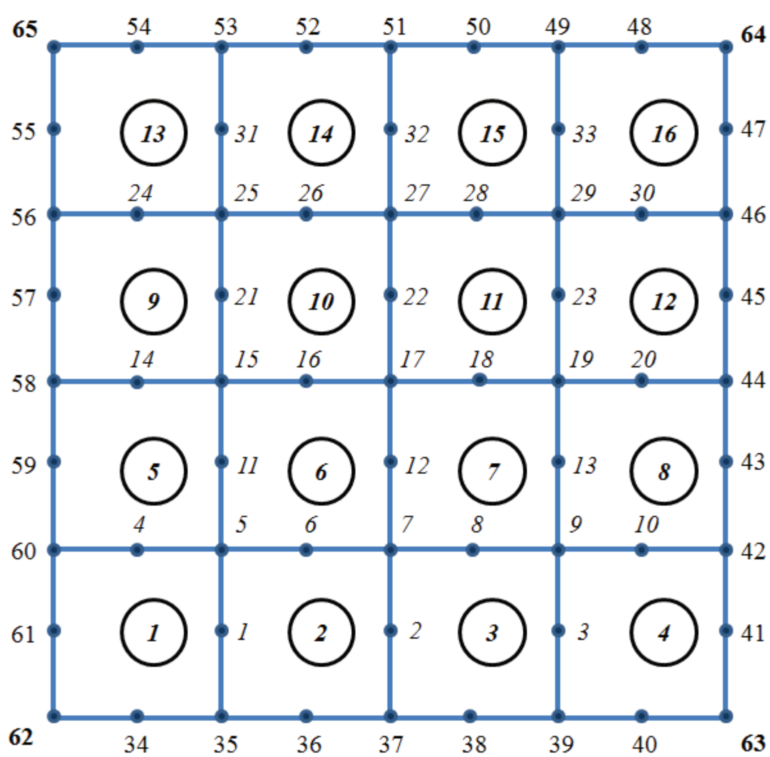
1 Introduction

Finite element analysis is well developed and widely used in all fields from engineering designs to academic researches, but there are still some improvements need to be done, e.g. the conflict requirements from engineering practice and finite element analysis principle. Finite element analysis principle requires element size to be sufficiently small to yield reliable analysis results, but engineers/analysts sometimes like to use large size elements to simplify the modelling process and reduce the amount of unwanted results. This illustrates the needs to develop large size elements that also give reliable analysis results. The other area that attracts lots of attentions in finite element analysis is the parallelisation of finite element analysis as finite element model become larger and larger nowadays and overrun the speed increase of single computer processors. To speed up the analysis, finite element analysis programs should be able to use all processors available in a local computer and/or from a network, i.e. parallelising finite element analysis. To help with the above two requirements, this paper develops a panel element which size can be large. The proposed panel element is developed based on substructure principle and the panel element is meshed internally by smaller shell elements. To the users, the panel element is just a 4 node large shell element capable of modelling large physical object such as a wall and a floor by using one element. Internally in the finite element analysis solver, the panel element is meshed by smaller shell elements. Since the panel element is meshed internally by small shell elements, there will be more nodes involved in the analysis than the 4 corner nodes used to define the element. The nodes are classified as internal nodes that are within the panel element and external nodes that are at the edges and corners of the panel element. The DOF of internal nodes will be condensed out and do not appear in global analysis. Therefore the total number of DOF of a finite element model using panel elements will be much less than that of the same model using small shell elements. Using panel elements is also very helpful in writing parallel finite element analysis program as each panel element can be easily processed independently in different threads/processors. There are also two other associated time saving benefits from using panel elements, one is the reduction of repetitive processes within one panel elements because only one of the small shell elements in a panel element needs to be processed, the other associated benefit is the reduction of repetitive processes of panel elements in a whole structural model because it is very likely that in a large structural model there are many identical panel elements and only one of them needs to be processed. The large size nature of the panel elements also makes it possible to predict drilling stiffness more accurately. Drilling stiffness of shell elements is an issue that has not been solved satisfactorily to meet engineering requirements, but it is highly demanded in engineering practice when modelling beam to shell element connections. The reason why drilling stiffness has not been predicted satisfactorily is due to the fact that drilling stiffness not only depends on the properties of shell element itself, but also depends on the area that is used to apply drilling moment to the shell element. Using small shell elements cannot solve this problem satisfactorily because it is impossible to take the moment application area into account when generating drilling stiffness. This paper recommends a method to calculate the drilling stiffness by taking the moment application area into account and the predicted drilling stiffness is more accurate than previously proposed methods [1-6] in which the drilling stiffness is unreasonably affected by element sizes/meshes. As drilling stiffness is not always required, e.g. if there are no beam elements connected to the panel element, the inclusion of drilling stiffness can be made optional in the proposed panel element by taking it as an extra user defined parameter.

2 The panel element

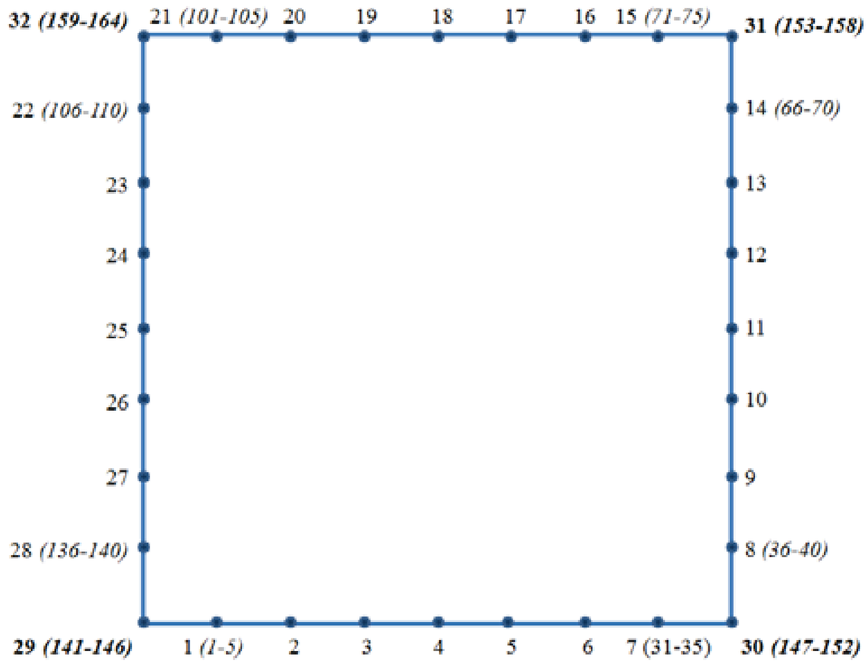
2.1 Meshing of the panel element

The panel element can be meshed by small shell elements in different ways, e.g. 4×4, 4×6 and 6×6 etc. The meshing scheme discussed in this paper is by 4×4 Quad8 shell elements as shown in Figure 1(a). Figure 1(b) shows the active (external) nodes of the panel element in global analysis and Figure 1(c) shows the panel element to be seen and used by the end users and it is a 4 node element. The derivations of small shell element stiffness matrix [K_e] are available from any finite element analysis books, e.g. from reference [7] and it is not discussed in this paper. This paper concentrates on the idea of the panel elements as well as the assembling of panel element stiffness matrix, load vector and how to generate drilling stiffness etc.

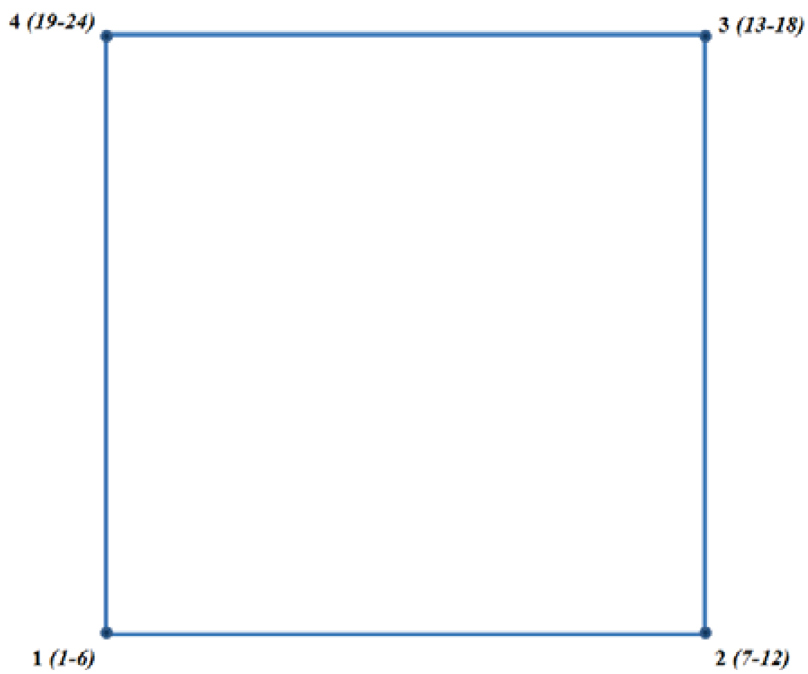


(a) Meshing of panel element

Figure 1 Panel element and the meshing scheme



(b) Panel element to the solver in global analysis



(c) Panel element to the users

Figure 1 (continued) Panel element and the meshing scheme

2.2 Stiffness matrix of the panel element

Assuming the stiffness matrix [K_e] of the small shell elements is known and the node and element numbering are as given in Figure 1(a), the stiffness matrix of the meshed up model shown in Figure 1(a) including all internal and external nodes can be assembled using the routine shown in Figure 2 in which 5 DOF at each node are used except the 4 corner nodes that have 6 DOF.

```
void AssembleK(std::vector(std::vector(double))& K)
{
   for(int iE = 0; iE < 16; ++iE)
   {
      for(int iR = 0; iR < 40; ++iR)
      {
         for(int iC = 0; iC < 40; ++iC)
         {
            const int iTR = vTopo[iE][iR/5];
            const int iTC = vTopo[iE][iC/5];
            int iRG = 5*(iTR-1) + iR%5;
            int iCG = 5*(iTC-1) + iC%5;
            if(iTR >= 62)
               iRG = 6*(iTR-1) + iR%5 - 61;
            if(iTC >= 62)
               iCG = 6*(iTC-1) + iC%5 - 61;
             K[iRG][iCG] +=  Ke[iR][iC];
         }
      }
   }
}
```

where:

vTopo:	Vector holding the topologies (node number) of each of the small shell elements as shown in Figure 1(a), e.g. vTopo[1][2] gives the second node number of element 1.
K_e:	Small shell element stiffness matrix and its size is 40×40
K:	Uncondensed stiffness matrix of the meshed up model

Figure 2 Assemble of stiffness matrix for the meshed up model

According to the numbering system shown in Figure 1(a), the internal and external degrees of freedom are separated automatically in the stiffness matrix K, the internal DOF are in front followed by external DOF. The total number of DOF for the meshed up model N_t = 329 and the number of internal DOF N_i = 165. The uncondensed stiffness matrix K and the corresponding load vectors (discussed below) can be condensed using the routine in Figure 3.

```
void CondenseMatrix(std::vector<std::vector<double>>& K,
                    std::vector<std::vector<double>>& P,
                    int Nt, int Np, int Ni)
{
    for(int k = 0; k < Ni ; ++k)
    {
      for(int i = k + 1; i < Nt; ++i)
      {
        const double dFactor = K[i][k]/K[k][k];
        for(int j = k; j < Nt; ++j)
          K[i][j] = K[i][j] - dFactor*K[k][j];
        for(int j = 0; j < Np; ++j)
          P[i][j] =  P[i][j] - dFactor*P[k][j];
      }
    }
    for(int k = Ni-1; k > 0 ; --k)
    {
      for(int i = k-1; i >= 0; --i)
      {
        double dFactor = K[i][k]/K[k][k];
        for(int j = 0; j < Nt; ++j)
          K[i][j] = K[i][j] - dFactor*K[k][j];
        for(int j = 0; j < Np; j++)
          P[i][j] = P[i][j] - dFactor*P[k][j];
      }
    }
    // normalise
    for(int i = 0; i < Ni; ++i)
    {
      const double dFactor = K[i][i];
      for(int j = 0; j < Nt; j++)
        K[i][j] = K[i][j]/dFactor;
      for(int j = 0; j < Np; j++)
        P[i][j] = P[i][j]/dFactor;
    }

}
```

where:

- Nt: Total number of DOF of the meshed up model (329 in this example)
- Ni: Number of internal DOF (165 in this example)
- Np: Number of load cases
- K: Stiffness matrix, on beginning, K holds the stiffness matrix of the meshed up model assembled using routine in Figure 2, at the end, it contains:

$$\begin{bmatrix} [1] & [K]'_{ib} \\ [0] & [K]'_{bb} \end{bmatrix} = \begin{bmatrix} [1] & [K]_{ii}^{-1}[K]_{ib} \\ [0] & [K]_{bb} - [K]_{bi}[K]_{ii}^{-1}[K]_{ib} \end{bmatrix} \quad (1)$$

- P: Load matrix, each column represents a load case. On beginning, it holds the load matrix for the meshed up model, at the end, it contains:

$$\begin{bmatrix} [u]_i^i \\ [P]_b' \end{bmatrix} = \begin{bmatrix} [K]_{ii}^{-1}[P]_i \\ [P]_b - [K]_{bi}[K]_{ii}^{-1}[P]_i \end{bmatrix} \quad (2)$$

$[K]_{ib}'$: External node displacement to internal node displacement conversion matrix

$[K]_{bb}'$: The condensed stiffness matrix of the panel element with external DOF only

$[u]_i^i$: Internal node displacement generated by internal node loads with external nodes being fixed

$[P]_b'$: Condensed load matrix corresponding to the condensed stiffness matrix $[K]_{bb}'$ for global analysis

Figure 3 Condense of panel element stiffness matrix

From above operations, all useful matrices for both global and panel element level analyses are obtained. For the example discussed in this paper, the number of external DOF $N_e = N_t - N_i = 164$ and it is about half of the total number of DOF of the meshed up model, therefore the total number of DOF of the whole model using panel elements will also be much less than that of the same model using small shell elements, consequently, the computing time for solving the global stiffness matrix can also be reduced.

2.3 Load vector for the panel element

As panel element is large, it is not sufficient to convert panel element face loads to equivalent nodal loads only at the 4 corner nodes in the analysis. To have correct analysis results within the panel elements, the face loads need to be represented by equivalent nodal loads at all nodes (internal and external) of the meshed up model as shown in Figure 1(a). In global analysis, the equivalent nodal loads at all nodes need be condensed to external nodes that are compatible to the condensed stiffness matrix. Assuming the equivalent nodal load vector for the meshed up model is known and represented by $\{P\}$, it is partitioned into internal block $\{P\}_i$ and external block $\{P\}_b$ where $\{P\}_i$ is in front and followed by $\{P\}_b$, then the condensed nodal load vector $\{P\}_b'$ at external nodes for global analysis is:

$$\{P\}_b' = \{P\}_b - [K]_{bi}[K]_{ii}^{-1}\{P\}_i \quad (3)$$

Where:

$\{P\}_b$ External block of the equivalent nodal load vector to the face load, the length is 164 for the example discussed

$\{P\}_i$ Internal block of the equivalent nodal load vector to the face load, the length is 165 for the example discussed

$\{P\}_b'$ Condensed load vector at external nodes for global analysis, its length is 164 for the example discussed

$\{P\}'_b$ is not derived directly from Equation (3) in practice, it should be obtained in the same operation of condensing the stiffness matrix of the meshed up model that has been given in Figure 3. If there are multiple load cases, the load vectors can be extended to load matrix with each column representing a load case, multiple load cases have been considered in code routine given in Figure 3.

3 Analysis results of panel element

3.1 Nodal displacements

Once the condensed stiffness matrix $[K]'_{bb}$ and the condensed load vector $\{P\}'_b$ are obtained, global stiffness matrix and global load vectors for the whole model can be assembled in the same way as for any other elements [7]. From global analysis, the displacements $\{u\}_b$ at the external nodes of the panel elements are known directly and they are also the final displacements of the external nodes.

The final displacements at internal nodes contain two parts and they can be obtained from:

$$\{u\}'_i = [K]_{ii}^{-1}[P]_i + [K]_{ii}^{-1}[K]_{ib}\{u\}_b \tag{4}$$

Where:

- $\{u\}'_i$ Final nodal displacement vector of internal nodes
- $\{u\}_b$ Final nodal displacement vector of external nodes
- Other parameters are the same as above

As shown in Equation (4), the first part of internal node displacements is the displacements generated by the loads at internal nodes while external nodes are assumed to be fully fixed, the second part of internal node displacements is generated by external node displacements while no other loads are on the model.

3.2 Nodal forces/moments & total forces/moments at edges

Once external node displacements are known from global analysis, the forces/moments at external nodes can be calculated from:

$$\{F\}_b = [K]'_{bb}\{u\}_b \tag{5}$$

Where:

- $\{F\}_b$ Force vector representing the forces/moments at external node
- $\{u\}_b$ External node displacement vector

$\{F\}_b$ can be used to calculate the total forces and total moments at each of the 4 edges of the panel element.

In engineering design, it is often the case that the total forces and total moments at a section of a wall or a slab are more useful than the detailed stress distribution within the elements, e.g. the total forces/moments at a horizontal section of a wall is more useful than the detailed stress distribution when calculating the required reinforcements for the wall. Panel element size is large and the total forces and moments at an edge can represent the forces and moments at a section of a wall or a slab. The total forces/moments at an edge can be calculated from the known forces/moments at external nodes given by Equation (5). Figure 4 shows one option of the sign conventions for the total forces/moments at the edges of a panel element.

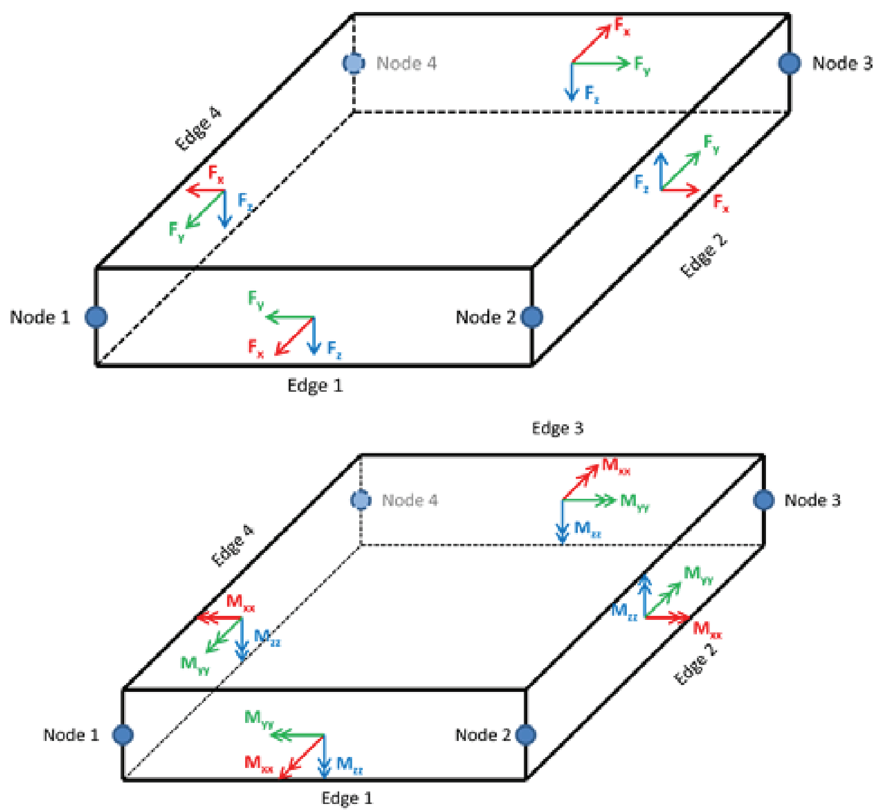


Figure 4 Sign convention of total forces and moments at panel element edges

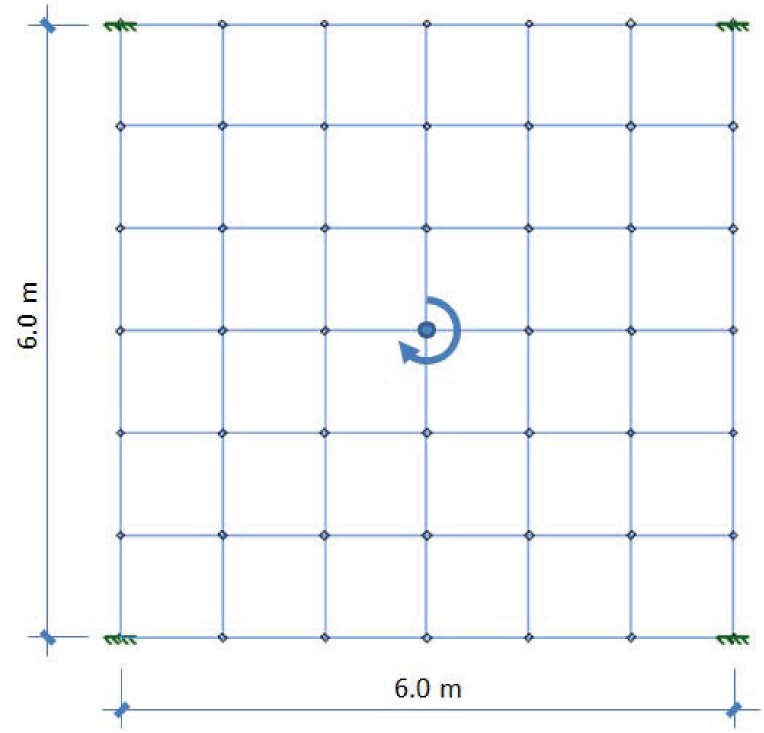
3.3 Forces, moments, stresses & strains within the element

When panel elements are used to model shear walls, the total forces/moments at the edges are normally sufficient for engineering design. However, when panel elements are used to model floors etc. that are subjected to out-of-plane bending, the forces, moments, stresses and strains within the panel elements will be required. Since panel elements are meshed internally, the forces and moments etc. within the panel elements are just the forces and moments etc. of the small shell elements in the panel element. They can be calculated from the known internal and external node displacements and the small shell element properties, so it is omitted here.

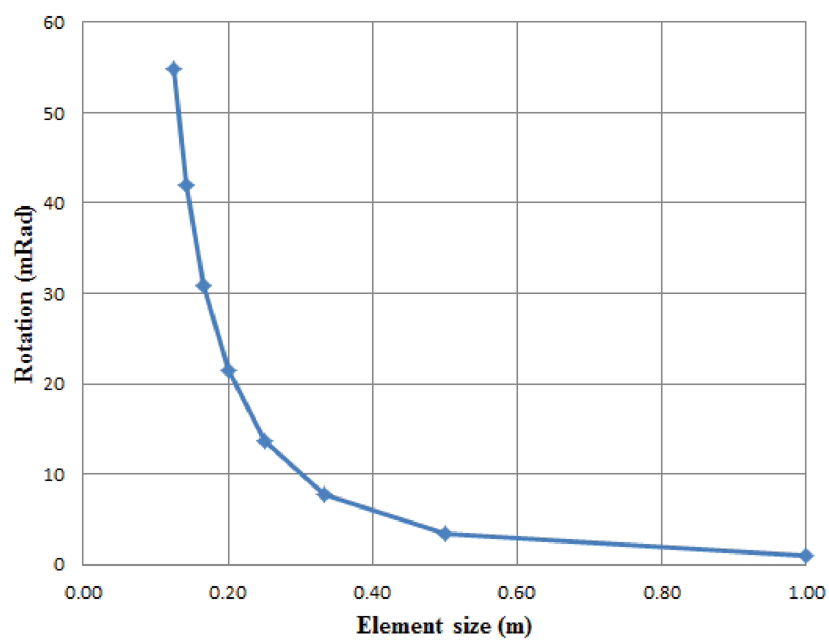
4 Drilling stiffness

4.1 Introduction

Traditionally shell elements only have 5 degrees of freedom at each node and the rotational stiffness (drilling stiffness) about the normal to the shell element plane is zero. 5 DOF shell element works fine if all shell elements connected to a node are exactly coplanar or completely non-coplanar. If the elements are exactly coplanar, the drilling degree of freedom can be removed in global analysis and only 5 DOF are used at each node, alternatively an arbitrary stiffness can be assumed for the drilling stiffness and still keep 6 DOF at each node in global analysis. If all elements at a node are completely non-coplanar, e.g. the elements are perpendicular each other at the node, the node will have perfect 6 DOF and there will be no problem in global analysis even though shell element only has 5 DOF. However, when the shell elements at a node are not exactly coplanar, but very close to coplanar, it is like to have problem. If keeping 6 DOF at the node, the drilling stiffness will be very small compared with other stiffness and it is likely to cause numerical problem. If ignoring the non-zero drilling stiffness and using 5 DOF in global analysis, it would generate some levels of analysis errors. The ideal solution is to work out the real drilling stiffness and include it in the analysis. The widely adopted methods proposed by Allman and Cook etc. [1-6] introduces a mid-side node at each edge of the elements, the drilling stiffness at the corner nodes are derived by the relative translational displacements between the corner node and the mid-side node within the element. The method is sound and works fine if certain conditions are met, e.g. if the element size is not too small and it is compatible with the beam size that is used to apply the drilling moment. However, it does not always work accurately enough in engineering practice and very often it underestimates the drilling stiffness when the element sizes are small. Figure 5 shows a numerical example that demonstrates the drilling stiffness (rotation) from Allman-Cook method [1-5] is significantly affected by the element sizes. Furthermore, when the element size approaches to zero, the drilling stiffness tends to be zero and this is incorrect. In reality, the drilling stiffness not only depends on the properties of the shell element itself, but also depends on the area that is used to apply drilling moment, e.g. the beam element section size connected to the shell elements. The beam section sizes cannot be considered in traditional shell element formulation in deriving drilling stiffness, because (1) the beam section size is unknown in advance; (2) the shell element size is normally not large enough to incorporate beam section size. The method proposed in this paper will be able to incorporate the beam section size as the panel element is much larger than the beam section. To consider the actual beam section size rather than estimate the size in deriving drilling stiffness, the beam section size is taken as an extra parameter for the proposed panel element, therefore the drilling stiffness can be predicted more accurately.



(a) The model (thickness = 0.5 m, E = 2.8×10^{10} Pa, Moment = 1×10^{6} N.m)

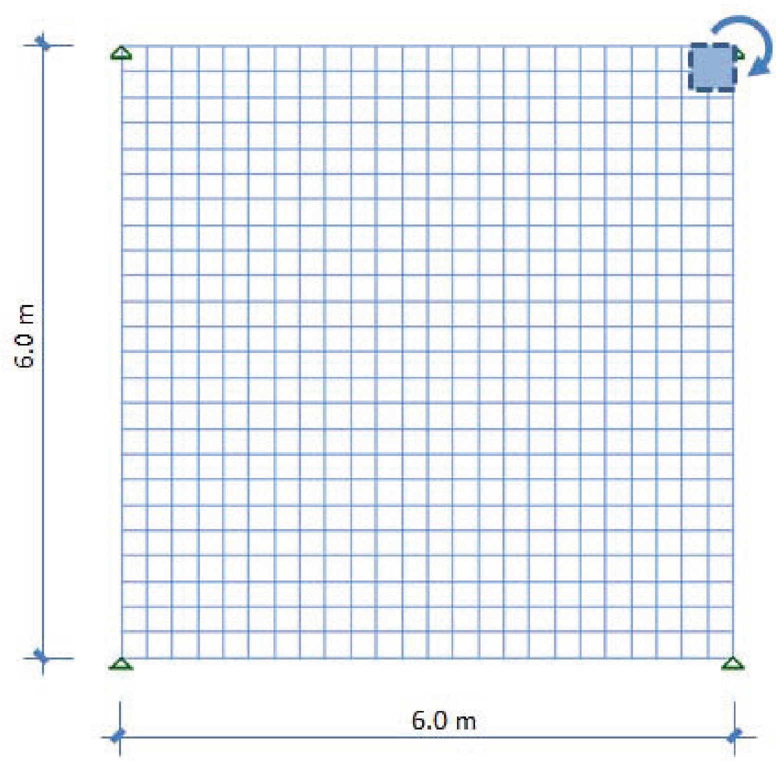


(b) Rotation versus element size

Figure 5 Effect of element size on drilling stiffness using Allman-cook method

4.2 Derivation of drilling stiffness

As mentioned above, the drilling moment has to be applied to shell elements through a finite moment application area, e.g. the section area of a beam connected to the shell elements. The moment application area has significant effect on the drilling stiffness and this is demonstrated by the finite element example shown in Figure 6. In this example, a 6×6 meter square panel is modelled by Quad8 shell elements with finer meshes. The drilling moment is applied to the top right corner through different square areas. The relationship between the rotational stiffness and the dimension of the square moment application area is drawn in Figure 6 (b). It shows that the effect of the area to the drilling stiffness is so significant that the area should be considered when deriving the drilling stiffness in order to get practically usable drilling stiffness.



(a) The model (thickness = 0.5 m, E = 2.8×10^{10} Pa)

Figure 6 Effect of drilling moment application area to the drilling stiffness

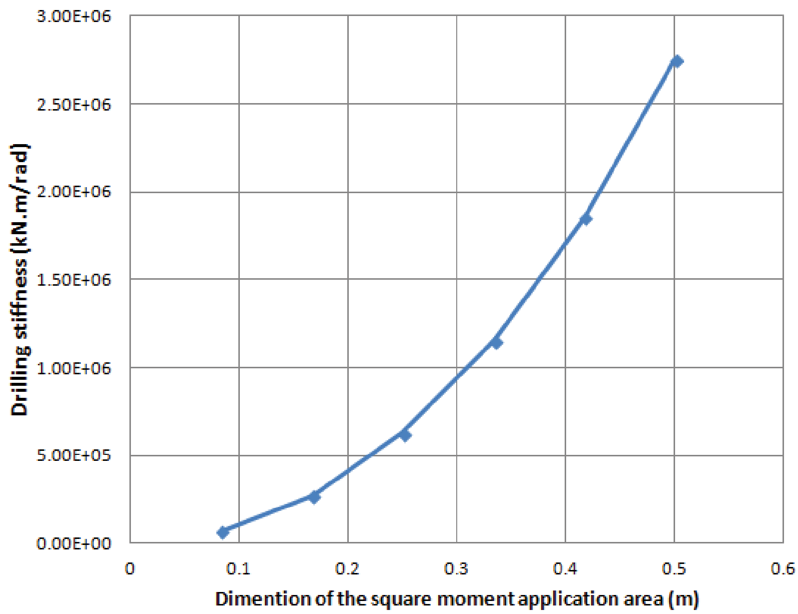


Figure 6 (continued)
Effect of drilling moment application area to the drilling stiffness

It is impossible to consider this moment application area by traditional shell elements as the element size is normally too small to cover the area. The proposed panel element is large and is meshed internally, so the moment application area can be considered correctly. Figure 7 shows how the drilling stiffness is derived by the panel element. A dummy node with only two translational DOF within the element plane is added near each of the 4 corner nodes. The distances of the dummy node to the two edges are determined by the moment application area, e.g. the section size of the beam element connected to the panel element. If the beam section is not rectangular, the equivalent rectangular area should be used. The displacements of the dummy nodes within the element plane are constrained by the adjacent edge displacements that are obtained through linear interpolation of the two adjacent nodes, e.g. for dummy node 3 as shown in Figure 7, its displacement can be calculated from (constrained by):

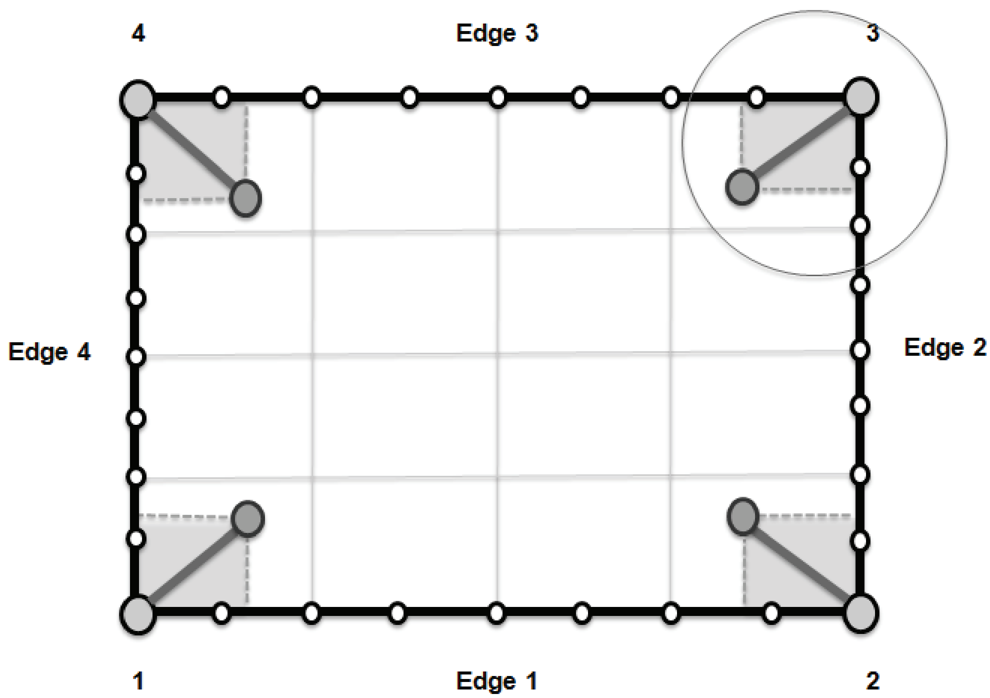
$$u_x = u_{1.x} + \frac{(u_{2.x} - u_{1.x})}{b}(d - b)$$
$$u_y = u_{3.x} + \frac{(u_{4.x} - u_{3.x})}{a}(c - a) \quad (6)$$

Where:

a, b	Nodal spacing at the horizontal and vertical edges respectively
c, d	The width and height of the rectangular moment application area
u_x, u_y	Horizontal and vertical displacements of the dummy node (constrained by the edge displacements and they are not independent)

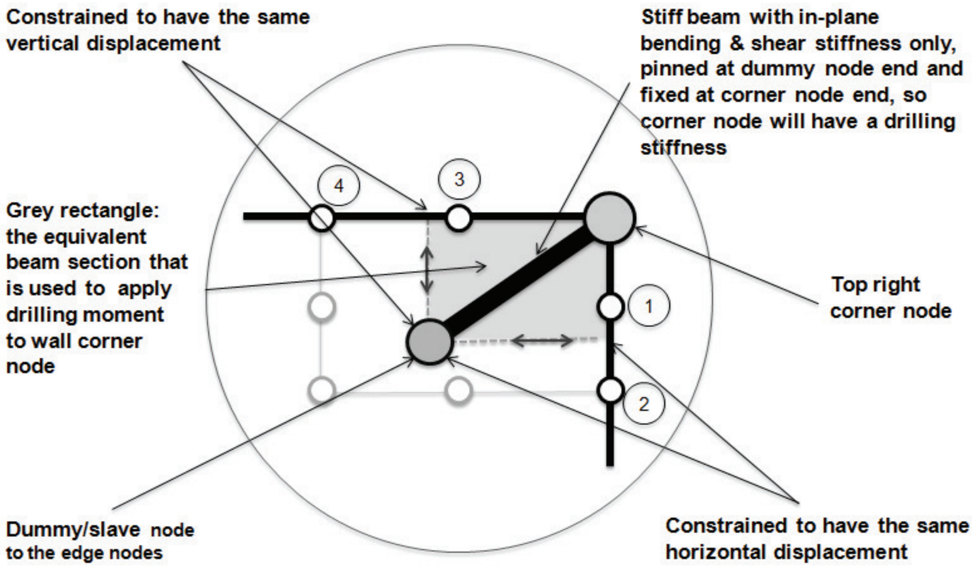
$u_{1.x}$, $u_{2.x}$	Horizontal displacements at nodes 1 & 2 as shown in Figure 7
$u_{3.y}$, $u_{4.y}$	Vertical displacements at nodes 3 & 4 as shown in Figure 7

The dummy nodes are then linked with the corner nodes by a rigid bar, so the corner node will have a drilling stiffness equivalent to that the moment is applied through the area defined by c×d as shown in Figure 7. In practice, a high stiffness beam without axial and out-of-plane bending stiffness is used to link the corner nodes and the dummy nodes in order to keep the corner nodes as independent nodes rather than a slave node. Since dummy nodes are slave nodes and not independent, their degree of freedom will not appear in global analysis, hence the introduction of dummy nodes does not change the total number of DOF of the panel element.



(a) The panel element with dummy nodes and rigid bars

Figure 7 Generation of drilling stiffness



(b) Generation of drilling stiffness

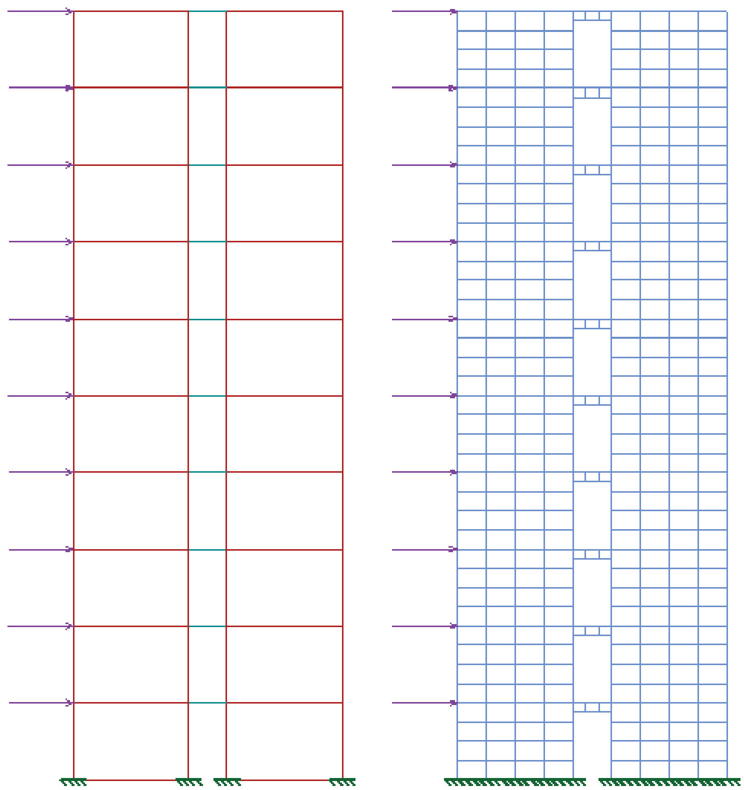
Figure 7 (continued) Generation of drilling stiffness

4 Worked examples

The proposed panel element has been implemented in Oasys General Structural Analysis package GSA and two models have been built using the proposed panel elements in GSA, one is a linked shear wall model and one is the core model of a high rise building, the same models are also built using Quad8 shell elements in GSA and the models are shown in Figure 8 and Figure 9 respectively. The displacements at each floor level are summarised in Table 1 and it shows that the panel and shell element models give very similar results. It is obvious that the models built using panel elements are much simpler than the model using shell elements. As less number of elements are used in panel element model, the amount of results are also much less and easy to interpret.

5 Conclusion

The conflict requirements from finite element analysis principle and modelling process can be resolved by the proposed large panel elements. The use of the large panel elements makes the modelling of shear wall and floor plate etc. much simpler. The use of panel elements also makes it easier/possible to produce total forces and moments at a section/edge that are more useful in engineering designs. The proposed method of predicting drilling stiffness can give more accurate results and meet engineering requirements as this method takes the actual moment application area into account. The use of the panel elements can also reduce the repetitive processes at both element level and global analysis level to increase analysis speed. Using panel elements also makes it easier to write parallel finite element analysis program as each panel element can be taken as a substructure and is processed independently in separate processors.

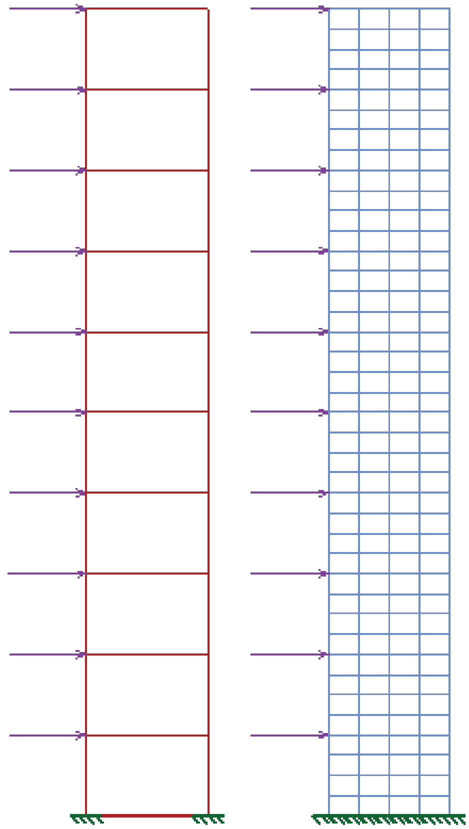


(a) Panel element model (b) shell element model

total height = 40 m; total width = 14 m; link beam length = 2 m

thickness = 200 mm; $E = 2.8x10^{10}$ N/m^2 point load = 30 kN

Figure 8 Linked shear wall modelled by panel and shell elements



(a) Panel element model (b) shell element model

total height = 40 m; 6x6 meter square in plan; wall thickness = 200 mm
$E = 2.8 \times 10^{10} N/m^2$ point load (2 at each level) = 2 × 30 kN

Figure 9 Elevation of square core modelled by panel and shell elements

Level	Linked Shear Wall			Square Core		
	panel	shell	%	panel	shell	%
1	0.192	0.197	-2.49%	0.230	0.236	-2.65%
2	0.549	0.550	-0.32%	0.654	0.661	-1.14%
3	1.025	1.020	0.43%	1.240	1.249	-0.71%
4	1.573	1.560	0.82%	1.951	1.961	-0.52%
5	2.156	2.133	1.07%	2.753	2.764	-0.42%
6	2.750	2.716	1.24%	3.617	3.630	-0.35%
7	3.335	3.290	1.36%	4.520	4.534	-0.31%
8	3.902	3.846	1.47%	5.443	5.458	-0.28%
9	4.448	4.380	1.56%	6.371	6.388	-0.26%
10	5.008	4.923	1.71%	7.314	7.329	-0.21%

Note: see Figures 8 & 9 for the dimensions and material properties etc. of the models

Table 1 Comparison of horizontal displacements of panel and shell element models

References

[1] D.J. Allman. A compatible triangular element including vertex rotation for plane elasticity analysis. Computers and Structures, 19:1-8, 1984

[2] D.J. Allman. A Quadrilateral finite element including vertex rotation for plane elasticity analysis. International Journal for Numerical Methods in Engineering. 26:717-730, 1988

[3] D.J. Allman. Evaluation of the constant triangle with drilling rotations. International Journal for Numerical Methods in Engineering. 26:2645-2655, 1988

[4] R.D. Cook. On the Allman triangle and a related quadrilateral element. Computers and Structures, 2:1065-1067, 1986

[5] R.D. Cook. A plane hybrid element with rotational d.o.f. and adjustable stiffness. International Journal for Numerical methods in Engineering, 24:1499-1508, 1987

[6] H Richard et al A refined four-noded membrane element with rotational degrees of freedom. Computers and Structures, 1:75-84, 1986

[7] O.C. Zienkiewicz & R.L. Taylor. The Finite Element Method for Solid and Structural mechanics. Sixth edition, 2005