Advances in Engineering Software 122 (2018) 93–105

Contents lists available at ScienceDirect

Advances in Engineering Software

journal homepage: www.elsevier.com/locate/advengsoft

In-plane torsional stiffness in a macro-panel element for practical finite element modelling

T.Q. Li[a,*], T. Ward[a], W.J. Lewis[b]

[a] Arup, London, UK
[b] University of Warwick, UK

ARTICLE INFO

Keywords:
Panel element
In-plane torsional stiffness
Large element
Sub-structuring
Floor & wall

ABSTRACT

Finite element (FE) analysis produces results, which, in most cases, gain in accuracy, as the size of the FE mesh is reduced. However, this is not necessarily the case when beam and shell element connections induce in-plane torsional effects in the shell. In such situations, shell elements either do not allow for an in-plane torsional stiffness, or, when present, the in-plane torsional stiffness is incorrectly affected by the sizes of the elements. To overcome this problem, we propose a macro- panel element that has fewer degrees of freedom, includes a new model for in-plane torsional stiffness, and produces results with sufficient accuracy to meet engineering requirements. The panel element is based on the principle of sub-structuring, i.e., the panel is meshed internally by smaller shell elements. As shown in the paper, the proposed panel element can be quite large, yet, it can give accurate analysis results. This work helps to overcome a common dilemma in practical use of finite element analysis, where finite element theory requires element sizes to be sufficiently small, but practical considerations suggest the use of large-size elements that simplify the modelling process and reduce excesses in generated results. A model built using macro-panel elements is equivalent to the model built using smaller shell elements, with the normal and shear stresses in the former being the same as the stresses in the finely meshed shell element model, We identify a number of performance benefits that become available as a consequence of modelling the shell elements at a higher level of abstraction.

1. Introduction

A common conflict in practice of finite element analysis is the trade-off between the practical benefits of using larger elements against the numerical benefits of finer meshing. We have found one solution to this problem by introducing an intermediate modelling abstraction, which resulted in a 'macro-panel' element.

The proposed panel element is designed using the principle of sub-structuring: the panel is divided internally into a set of smaller shell elements. From the point of view of a user, the macro-panel element is a four node "shell" element capable of modelling large physical objects, such as a shear wall or a floor slab, with a single element. From the point of view of the finite element analysis, the panel element is a 'condensed' mesh of smaller shell elements. The macro-panel element handles the intermediate detail between the two interfaces. The proposed transformation is based upon the work in reference [1], but we have carried out further verification and validation of results to demonstrate the practical and time saving benefits.

Obtaining in-plane torsional stiffness of a shell element can become problematic, as the stiffness is inevitably influenced by the size of the elements. We commonly observe in practice that, when a moment about the normal to a shell element is applied, the results are either not available or have poor accuracy. One typical example of this is a moment applied to shell elements via the twist of a beam perpendicularly connected to them. The larger size of the proposed macro-panel element makes it possible to conveniently accommodate the in-plane torsional moments over a wider area and predict the in-plane torsional stiffness more accurately. In this paper, we recommend a new method for calculating the in-plane torsional stiffness that makes use of the larger area offered by the transformation into a macro-panel element. We have found that this method produces greater accuracy than previously proposed methods (refs. [2–7]), in which the in-plane torsional stiffness is more sensitive to individual element sizes. To account for the situation where an in-plane torsional stiffness is not required, we make the inclusion of in-plane torsional stiffness optional by means of a user-defined parameter.

We demonstrate the effectiveness and applicability of the proposed macro-panel element model using two key examples. In Example 1, we

* Corresponding author.
E-mail address: thomas.li@arup.com (T.Q. Li).

https://doi.org/10.1016/j.advengsoft.2018.04.008
Received 14 August 2017; Received in revised form 13 March 2018; Accepted 22 April 2018
Available online 30 May 2018

model a 40-storey structure with linked shear walls and compare the results with three alternative finite element models that use shell elements to represent the walls and beam elements to represent the links. The alternative models given in this example include Allman/Cook formulation with in-plane torsional stiffness, as well as shell elements without that stiffness. In Example 2, we model a shear wall core of a 30-storey building subjected to horizontal uniform face loads. The core is modelled using the proposed macro-panel element model and the results are compared with the shell element model.

The results show that our proposed macro-panel element offers a number of benefits. Other than facilitating a more accurate method for predicting the in-plane torsional stiffness, while retaining the quality of a finely meshed model, it offers real practical advantages. The data associated with each panel element is independent and so the processing of any panel can be executed independently in a parallel way. We also find a reduction in the repetitive calculations of any individual panel, since it is only necessary to process a single element. Finally, we commonly observe a reduction possible at the structural level where we have found it very likely that in a large structural model, many identical panels exist, but where only one of them needs to be processed.

2. The macro-panel element

2.1. Meshing of the panel element

The macro-panel element represents a substructure with internal meshing of shell elements at an arbitrary degree of refinement. Example refinements may represent a single panel by 4 × 4, 4 × 6 or 6 × 6 etc. shell elements. We restrict the discussion in this paper to a meshing scheme of 4 × 4 Quad8 shell elements, whose 'internal' assembly is shown in Fig. 1(a). The rectangular shape of the panel element is used here, but the method does not limit the shape to be rectangular and any quadrilateral shape can be used. We refer to all nodes within the panel element as 'internal' nodes, and reserve the name 'external' for all remaining nodes that are at the panel edges and corners. The node numbering is from internal nodes to edge nodes, and then the four corner nodes. Fig. 1(b) shows the 'external' nodes of the panel element as given to the global assembly where only these nodes will be active (the nodes have been re-numbered, for convenience). Fig. 1(c) shows the macro-panel element as seen by the end user; it is a typical four-node 2D element, and the four nodes are also re-numbered from one. The numbers shown in brackets indicate the degrees of freedom number.

All degrees of freedom (d.o.f.) associated with internal nodes are 'condensed' during the macro-panel element construction and do not appear in the global analysis. As a consequence, the total number of d.o.f. is significantly reduced (compared to the equivalent model that is using smaller shell elements).

The derivation of an individual shell element stiffness matrix $[K_e]$ can be found from a variety of sources e.g. reference [8] and is not discussed here. In this paper we concentrate on the idea of the panel element as a larger entity: the assembly of the macro-panel element stiffness matrix, the assembly of a corresponding load vector, and a method of constructing the in-plane torsional stiffness from the internal arrangement.

2.2. Stiffness matrix of the macro-panel element

Given a known stiffness matrix $[K_e]$ of a shell element (refer to Section 13.2 on pages 428 & 429 of reference [8]) for detailed derivation) and the element numbering system shown in Fig. 1(a), the stiffness matrix of the meshed panel in Fig. 1(a) can be assembled and partitioned by internal and external d.o.f. and take the following form:

$$[K] = \begin{bmatrix} K_{ii} & K_{ib} \\ K_{bi} & K_{bb} \end{bmatrix} \tag{1}$$

For the meshing scheme shown in Fig. 1(a), i is from 1 to 165 for the 33 internal nodes and each has 5 d.o.f., b is from 166 to 357 for the 32 external nodes and each has 6 d.o.f.

Using Guyan condensation method [9] to eliminate the internal d.o.f., the final stiffness matrix of the panel element with only external nodes can be obtained from:

$$[K^e] = [K]_{bb} - [K]_{bi}[K]_{ii}^{-1}[K]_{ib} \tag{2}$$

where the total number of d.o.f. of the panel element is 192- corresponding to the 32 external nodes (the node and d.o.f. numbering system for the panel element is shown in Fig. 1(b); Fig. 1(c) shows the node and d.o.f. numbering for the panel element as seen by the end user).

We can see from this example that the number of d.o.f. of the panel element is 192, while the total number of d.o.f. of the meshed model is 390. This shows that the total number of d.o.f. has been reduced by a half in this example. Consequently, for a structure modelled using panel elements, the size of the global stiffness matrix can be significantly reduced, producing time saving benefit in solving the matrix equation problem. The code routines for assembling the stiffness matrix of the meshed panel and then condensing the stiffness matrix to panel element stiffness matrix with only external d.o.f. active is based on reference [1] and can be found in the Appendix.

2.3. Load vector of the panel element

If there are distributed loads on the panel elements, the loads have to be represented by nodal loads in the analysis. The nodal loads act on both internal and external nodes. As internal nodes are not active in global analysis, the nodal loads also need to be condensed to external nodes in the similar way as condensing the stiffness matrix. Given a known nodal load vector $\{P\}$ for the meshed model, we can partition the load vector into an internal block $\{P\}_i$ followed by an external block $\{P\}_b$. The condensed nodal load vector $\{P\}_b^{'}$ at only external nodes for global analysis can be obtained from:

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

where:

$\{P\}_b$ The external block of the nodal load vector, the length is 192 for the example.
$\{P\}_i$ The internal block of the nodal load vector, the length is 165 for the example.
$\{P\}_b'$ The condensed load vector at external nodes for global analysis. The length is 192 for the example discussed.
$[K]_{bi}$ Defined by Eq. (1)
$[K]_{ii}$ Defined by Eq. (1)

In practice $\{P\}_b^{'}$ is not derived directly from Eq. (3), but gained in the same process of condensing stiffness matrix $[K]$, and the code routines can also be found from reference [1] and Appendix. When multiple load cases exist, the load vectors become a load matrix and each load vector is represented by a single column in the load matrix.

3. Analysis results of the panel element

3.1. Nodal displacements

Given the condensed stiffness matrix $[K]_{bb}^{'}$ and condensed load vector $\{P\}_b^{'}$, a global stiffness matrix and global load vectors for the whole structural model can be assembled and the matrix equilibrium equation can be solved in the usual way [8]. The displacements from