ARTICLE IN PRESS

Thin-Walled Structures (xxxx) xxxx–xxxx

Contents lists available at ScienceDirect

Thin-Walled Structures

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

Full length article

Constrained shell Finite Element Method, Part 2: application to linear buckling analysis of thin-walled members

Sándor Ádány*, Dávid Visy, Róbert Nagy

Budapest University of Technology and Economics, Department of Structural Mechanics, Műegyetem rkp. 3, 1111 Budapest, Hungary

A B S T R A C T

In this paper a novel method is employed for the buckling analysis of thin-walled members. The method is basically a shell finite element method, but constraints are applied which enforce the thin-walled member to deform in accordance with specific mechanical criteria, e.g., to force the member to buckle in flexural, or lateral-torsional or distortional mode. The method is essentially similar to the constrained finite strip method, but the trigonometric longitudinal shape functions of the finite strip method are replaced by polynomial longitudinal shape functions, and longitudinal discretization is used, which transform the finite strip into multiple finite elements, that is why the new method can readily be termed as constrained (shell) finite element method. In the companion to this paper a band of finite elements is discussed in detail, where 'band' is a segment of the member with one single finite element longitudinally. In this paper the constraining procedure is applied on thin-walled members discretized both in the transverse and longitudinal direction. The possible base systems for the various deformation spaces are demonstrated here, as well as numerous buckling examples are provided to illustrate the potential of the proposed method.

1. Introduction

Thin-walled members have complicated stability behaviour. Due to the high slenderness, stability is the governing phenomenon in many cases. If a thin-walled member is subjected to longitudinal compressive stresses, three characteristic buckling classes are usually distinguished: global, distortional, and local-plate buckling. When the effect of shear stresses is dominant, shear buckling may also occur. Transverse compressive stresses might lead to instability frequently referred to as web crippling. In practical situations these buckling modes rarely appear in isolation, but they are interacted with each other. Still, current design approaches first separate these phenomena, and determine a capacity to each of the potential buckling, and then check the possible interaction of them. This design approach appears also in current thin-walled design standards, e.g. [1,2].

Capacity prediction hence requires the critical loads associated with the various buckling modes. Nowadays, critical load calculation for thin-walled members can readily be accomplished by some numerical methods, the most widely used ones being the shell finite element method (FEM), the generalized beam theory (GBT), and the finite strip method (FSM).

FEM, by using shell finite elements, is general and can be used to analyse almost any thin-walled member. However, a general FEM is not able to separate the various buckling modes, which often makes it inefficient to directly use it in standard capacity calculations. GBT has shown that buckling deformations may be formally treated in a modal nature that mechanically separates global, distortional, local, and other deformations, see e.g. [3–5]. This formal separation is integral to GBT, and allows pure buckling mode calculations and measurements of modal participation in coupled modes. FSM is based on the work of Cheung [6], but popularized by Hancock [7] who provided the organizing thrust of today's member design, which later evolved into the Direct Strength Method (DSM) [8]. Hancock introduced the notion of the signature curve, from which quasi-pure buckling modes and associated loads could be determined, at least for typical design. The mechanical criteria embedded in GBT led to the development of a special version of FSM, the constrained Finite Strip Method (cFSM), see [9–13]. cFSM possesses the ability of modal decomposition as well as mode identification in a manner similar to GBT.

Both GBT and cFSM is easily available, since they are implemented into the free-to-use programs GBTUL [14] and CUFSM [15]. It is easy to understand, however, that either the method or its available implementation is not general enough. For example, the analysed member has to be prismatic, no holes are allowed in the members, and there are restrictions w.r.t boundary conditions as well as loading (at least in the available software implementations). Recent works aimed at partially

* Corresponding author.
E-mail address: sadany@epito.bme.hu (S. Ádány).

http://dx.doi.org/10.1016/j.tws.2017.01.022
Received 6 October 2016; Received in revised form 10 January 2017; Accepted 12 January 2017

removing these limitations, but these attempts have not led to a simple and easily applicable general method with modal features, as discussed in a companion paper [16].

In this paper a novel method is used. The proposed method follows the logic of cFSM, however, the longitudinal shape functions are changed and applied together with a longitudinal discretization. Thus, strips are transformed into multiple shell finite elements, therefore, the new method can readily be termed as constrained finite element method, abbreviated as cFEM.

The cFEM has first been reported in [17–19]. The cFEM method is using a special shell finite element, specifically designed for the method. The most basic feature of this special rectangular shell element is that it distinguishes the so-called longitudinal and transverse directions. Otherwise, the shell element uses classic polynomial shape functions, though in an unusual combination. The element, together with the derivation of the interpolation functions is presented in detail in [20]. The elementary stiffness matrices (i.e., elastic and geometric) can be determined analytically. As earlier studies highlighted the importance of some details of the derivations, the stiffness matrices are derived in various options, as discussed in [21]. Namely: the second-order effect of the various strain terms can be switched on or off, as well as it is optional whether the through-thickness variation of the stresses/strains is considered or disregarded.

In [20] it is proved that the mechanical criteria, which are necessary for modal decomposition, can exactly be satisfied by the shape functions of the special shell element. Furthermore, in [16] the constraining procedure is discussed for members discretized into multiple finite elements. More precisely: in [16] multiple finite elements are assumed along the cross-section line, but only one single finite element is assumed longitudinally, which in other words means that one band of finite elements is discussed. Though one band of finite elements alone is not enough to solve practical problems, the discussion of a single band is important in order to understand the constraining procedure and the resulted base systems, and it helps to construct the modal base systems for a real member consisted of multiple bands. In fact, real members with multiple bands are in the main focus of this paper.

In this paper first the most important features of the new shell finite element are summarized, based on [20]. Also the constraining procedure in case of one single band of shell finite element is briefly summarized. Then two alternatives are presented for the construction of the modal base system of a member with multiple bands. Finally, numerous practical examples are shown for the constrained buckling analysis of thin-walled members.

As the numerical examples demonstrate, the here presented cFEM is much more general than any of the existing modal decomposition methods. Since it is based on a shell finite model, a variety of engineering problems can be solved. The element is rectangular, which means a certain limitation, but otherwise the method is general. Practically there are no limits for the loading and boundary conditions. Linear or non-linear static or even dynamic analyses can be performed, though in this paper only elastic linear buckling problems are shown. Holes can easily be handled, too, once they fit into the rectangular discretization. Though the problem of holes is not the topic of this actual paper, a few numerical examples with holes are also presented. Finally, though here the member to be constrained is assumed to be prismatic, piece-wise prismatic members can also be handled, by joining prismatic members together. Again, this question is not discussed here, only numerical examples demonstrate this ability of the proposed cFEM.

2. Basics of cFEM

2.1. Shell finite element for cFEM

A sample thin-walled member is shown on Fig. 1. It is discretized into shell finite elements. Local and global coordinate systems as well as some nodal displacements are also shown. The goal of the constrained finite element method is model the thin-walled member by shell finite elements, and to be able perform modal decomposition via enforcing the member to deform in accordance with pre-defined mechanical criteria. It is also essential to satisfy these criteria exactly, i.e., not only in specific locations (e.g., at some nodes), but all over the member: at any x and y locations within any finite element. As concluded in [16,20], this requires certain basic features from the interpolation functions, that is why the otherwise classic shape functions are used in a relatively unusual combination.

The derivation of the interpolation functions are discussed in [20], where the displacement functions are given, too. The nodal displacement degrees of freedom (DOF) are illustrated in Fig. 2. As can be seen, the proposed element has 30 DOF: each corner node has 7 DOF (1 for u, 2 for v, and 4 for w), while there are two additional nodes at $(x,y)=(a/2,0)$ and $(x,y)=(a/2,b)$ with one DOF per node for the u displacement.

2.2. cFEM in general

When a member is constrained, it is enforced to deform in accordance with some mechanical criteria. These criteria are identical to those applied in cFSM [e.g., 12–13], as summarized in the table of Fig. 3. As can be seen from Fig. 3, mostly displacement derivatives (i.e., strains) are used, and the important question is whether the given strain is zero or not (Y or N, respectively). In calculating the strains, all the u, v and w functions are interpreted at the middle surface of the plates, i.e., at $z=0$. In row 'transv. eq.' it is given whether the cross-section equilibrium (in the transverse direction) is satisfied or not (Y or N, respectively).

The deformation spaces defined in Fig. 3 are described in detail in [12,13]. It might be interesting to mention here that the mode spaces are separated into *primary* and *secondary* mode spaces. Primary modes are those deformations which are completely defined by the degrees of freedom (DOF) associated with the *main nodes* only, i.e., those nodes at the junction or end of the flat plates comprising the section. Secondary modes are defined by the DOF of the *sub-nodes*, i.e., those nodes within a flat plate discretized into multiple strips or elements.

It is to mention that though the table of Fig. 3 (mostly) clearly define the listed deformation spaces, these are only the 'displacement derivative is equal to zero' type criteria that can be transformed into equations. Therefore, only these so-called null criteria can and will directly be used in the constraining procedure, while the 'displacement derivative is not equal to zero' type criteria are used only indirectly in the construction of the deformation spaces.

As it is shown in [16], and will further be discussed here, the mechanical criteria can be transformed into constraint matrices. The application of the constraint matrix enforces to fulfil certain relationship between various nodal degrees of freedom. Another view of constraint matrix is that the column vectors of the matrix are the modal base vectors of the displacement field that is represented by the constraint matrix.

The $\mathbf{d}$ displacement vector may be constrained to any modal deformation space (i.e., to a $\mathbf{d_M}$ modal displacement vector) via:

$$\mathbf{d} = \mathbf{R_M}\mathbf{d_M} \tag{1}$$

where $\mathbf{R_M}$ is a so-called *constraint matrix*, the derivation of which can be found in [16] for a single band of finite elements, and will be presented in this paper for more general cases. 'M' might be G, D, L, S or T, or, in fact, 'M' might mean any combination of base vectors from any spaces.

By using the constraint matrix, solution in a reduced, specific deformation space is possible. For example, first or second-order static analysis can be done, the regular (i.e., unconstrained) problem of which takes the following form: