A Study on the Automatic Generation of FE Mesh for Prestressing Tendon of PSC Containment Building

Hong-Pyo Lee,a Young-Sun Choun ,b Young-Chul Song,a
a Korea Electric Power Research Institute, Integrity Assessment Group, 103-16 Munji-Dong Yuseong, Daejeon, Korea
b Korea Atomic Energy Research Institute, Integrated Safety Assessment Division, P.O. Box 105,Yuseong, Daejeon, Korea
hplee@kepri.re.kr

1. Introduction

Numerical Formulation technique of automatic finite element (FE) mesh generation for prestressing tendon in the NUCAS-solid, which has been developed for ultimate pressure capacity and nonlinear analysis for prestressed concrete (PSC) containment building, is described in this paper. Sometimes, embedded prestressing tendon, when applied to problems with curved or draped prestressing cables, impose significant constraints on the selection of the overall mesh. Therefore, a need exists for a curved embedded representation of tendon that allows the choice of mesh to be somewhat independent of the tendon geometry and location.

2. Formulations of the embedded element

In a 3D nonlinear FE analysis of PSC structures, three methods available for the simulation of a tendon are the smeared, discrete and embedded method. The smeared and discrete formulations are dependent on the concrete element mesh. In 3D applications, this can lead to prohibitive computational costs due to the use of many unnecessarily small elements or inaccuracies caused by elements with undesirable aspect ratios. To remedy these problems, an embedded formulation is preferable

2.1 Inverse mapping

The inverse mapping procedure, which is a 3D extension of the method proposed by Elwi and Hrudey[3], is used to find the intersection point. A point such as P_1 is contained in a given concrete element if its coordinates, ${\xi_1}_{p_1}$, ${\xi_2}_{p_1}$ and ${\xi_3}_{p_1}$, in the element local axes satisfy

$$-1 \le {\xi_1}_{p_1},\ {\xi_2}_{p_1},\ {\xi_3}_{p_1} \le 1 \tag{1}$$

In the isoparametric formulation the global coordinates (x,y,z) of a generic point within a solid element are expressed as

$$\mathbf{x} = \sum_{a=1}^{8} N_a\left(\xi_i\right)\mathbf{x}_i^a,\ \left(i = 1,2,3\right) \tag{2}$$

where $\mathbf{x}_i^a$ is vectors of the element nodal coordinates and N_a represents the displacement-shape functions at node *a*. It follows that

$$\begin{Bmatrix} dx \\ dy \\ dz \end{Bmatrix} = \left[\mathbf{J}\right]^T \begin{Bmatrix} d\xi_1 \\ d\xi_2 \\ d\xi_3 \end{Bmatrix} \text{ or } \begin{Bmatrix} d\xi_1 \\ d\xi_2 \\ d\xi_3 \end{Bmatrix} = \left[\mathbf{J}\right]^{T^{-1}} \begin{Bmatrix} dx \\ dy \\ dz \end{Bmatrix} \tag{3}$$

where, $\left[\mathbf{J}\right]$ is the Jacobian matrix.

From the equation (2), the coordinates $\left(\xi_1, \xi_2, \xi_3\right)_{P_1}$ are the roots of the following set of equations:

$$\begin{Bmatrix} x \\ y \\ z \end{Bmatrix}_{P_1} - \begin{bmatrix} N_a & 0 & 0 \\ 0 & N_a & 0 \\ 0 & 0 & N_a \end{bmatrix} \mathbf{x}_i^a = \left\{0\right\},\ \left(i = 1,2,3\right) \tag{4}$$

The Newton-Raphson iterative procedure has been used for a solution. With an initial estimate of $\left[\left\{\xi_1\ \xi_2\ \xi_3\right\}_{P_1}^0\right]^T = \left\{0\right\}$, the solution after *n+1* iterations is determined as

$$\begin{Bmatrix} \xi_1 \\ \xi_2 \\ \xi_3 \end{Bmatrix}_{P_1}^{n+1} = \begin{Bmatrix} \xi_1 \\ \xi_2 \\ \xi_3 \end{Bmatrix}_{P_1}^{n} + \begin{Bmatrix} \Delta\xi_1 \\ \Delta\xi_2 \\ \Delta\xi_3 \end{Bmatrix}_{P_1}^{n+1} \tag{5}$$

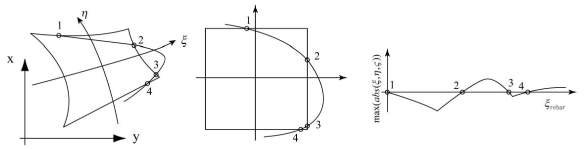
where
$$\begin{Bmatrix} \Delta\xi_1 \\ \Delta\xi_2 \\ \Delta\xi_3 \end{Bmatrix}_{P_1}^{n=1} = \left[J^n\right]^{T^{-1}} \left[\begin{Bmatrix} x \\ y \\ z \end{Bmatrix}_{P_1} - \begin{bmatrix} N_a & 0 & 0 \\ 0 & N_a & 0 \\ 0 & 0 & N_a \end{bmatrix} \begin{Bmatrix} x \\ y \\ z \end{Bmatrix} \right] \tag{6}$$

The preceding solution method has been found to converge rapidly. If the current element does not satisfy the equation (1), the procedure is repeated using the nodal coordinates of the next coordinates of the next element until the element containing $\mathbf{P}_1$ is identified.

2.2 Curved embedded method [3]

As the tendon is described in terms of global coordinates by standard interpolation functions, the coordinates of the rebar within the intrinsic coordinate system of the parent element can be obtained by employing iterative procedure for the inverse mapping(global to local) which was already presented by Eq. (4)-(7). A point of the tendon is situated within the respective element or at the element boundary as long as Eq. (9) holds.

$$\max(abs(\xi_1, \xi_2, \xi_3)) \leq 1 \qquad (7)$$

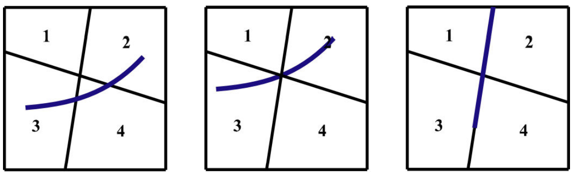


(a) Global configuration (b) Local configuration (c) Function given by $\max(abs(\xi,\eta,\varsigma))$

Figure 1. Intersection points of the tendon with the element faces

A generic plot of the function given by Eq. (7). It can be seen that this function is only piecewise continuous (C1 discontinuous). Finding all the roots analytically might become a mathematical challenge. Thus, the tendon is traced incrementally.

For the situation shown in Fig. 3, Eq. (7) will hold only for the old and for one new element at the interception point. The solution will not be unique for a situation similar to Fig. 3b, here Eq. (7) holds for more than one new element. A special treatment is needed as well for a case when Eq. (7) holds for two elements along a certain tendon portion (Fig. 3c).



(a) (b) (c)

Figure 2. Tendon within the parent elements

An efficient and simple remedy is to extend the limits for Eq. (7) by a small number, such that

$$\max(abs\left(\xi, \eta, \zeta\right) \leq 1 + \varepsilon, \quad \varepsilon = 1 \qquad (8)$$

3. Application

The automatic generation technique of FE mesh for prestressing tendon in the NUCAS-solid program which was developed to analysis ultimate capacity pressure and nonlinear behavior for PSC containment building on the Korea Standard Nuclear Power Plant is employed to be used curved embedded method. The input parameters to be simulated curved shape of a tendon element are starting point, ending point, total number of tendon, distance of tendon for the wall and total number of tendon, degree, starting point for the dome. The implemented automatic generation method is verified for a PSC containment build with hoop and meridional tendon (Fig. 3).

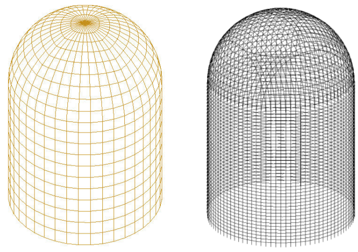


Figure 3. 3-dimensional FE mesh : concrete(left) and tendon(right)

4. Conclusion

In this paper, an automatic generation method using curved embedded element was implemented to generate the FE mesh for prestressing tendon for PSC containment building in the NUCAS-solid program. A detailed modeling of hoop and meridional tendon is not time-consuming in the FE analysis of PSC containment building any more. Finally, NUCAS-solid with the curved embedded tendon will be essential to develop the state of art FE analysis code for accurate safety assessment of containment building and our consistent research on safety analysis techniques.

Acknowledgement

The Research grant from the Ministry of Science and Technology, Korea, for the Nuclear Research & Development Program is gratefully acknowledged.

REFERENCES

[1] H.P. Lee, Y.S. Choun and J.M. Seo, A Study on the Standard 8-node Solid Finite Element for Nonlinear Analysis of Reinforced Concrete Containment, KAERI, KAERI/TR-2963/2005, 2005

[2] A.E. Elwi and T.M. Hrudey, Finite Element Model for Curved Embedded Reinforcement, Journal of Engineering Mechanics, Vol. 115, No. 4, 1989.

[3] Helmut Hartl, Development of a Continuum-Mechanics-Based Tool for 3D Finite Element Analysis of Reinforced Concrete Structures and Application to Problems of Soil-Structure Interaction, Doctoral Thesis, University of California in San Diego, 2002 .