rocscience

RS3

Hybrid Mesh

Theory Manual

Table of Contents

Introduction

The accuracy of Finite Element modeling outcomes significantly relies on the chosen mesh type for discretizing the domain. Enhanced precision is achievable through finer meshing or by integrating higher-order elements. RS3 offers two distinct element types: the 4-noded tetrahedron and the 10-noded tetrahedron. While it is generally recommended to favor 10-noded elements, especially in SSR analysis or complex models to improve result's accuracy, this preference leads to time-intensive computations as it involves solving larger systems of equations. Meshing the entire model with 10-noded elements may not always be essential. In cases involving a domain section with low geometric or mechanical complexity, where linear analysis suffices, or when the area of interest is limited, using 10-noded elements across the entire domain may not be mandatory.

To address such scenarios, RS3 introduces a hybrid mesh with mixed orders, enabling users to selectively apply 10-noded elements to specific critical areas within the domain and using 4-noded elements for the rest of the model. The hybrid meshing approach optimizes computational efficiency and enhances result accuracy by strategically incorporating higher-order elements only where their benefits are most needed. This document outlines the theoretical approaches for implementing the hybrid mesh in finite element analysis and provides an explanation of the formulation used in RS3.

1. Overview

The fundamental prerequisites for integrating different discretization types within the spatial domain involve ensuring that the generated finite elements are both complete and compatible (Zienkiewicz, 1977; Bathe, 2016). Completeness, or the partition of unity in finite elements, dictates that the sum of all interpolation functions must equate to one at any point in the model. Compatibility, on the other hand, asserts that the field variables of interest (such as displacement, pore-fluid pressure, etc.) and the coordinates of nodes at the shared edges of two adjacent elements must coincide. Figure 1-1 illustrates an instance of an *incompatible* mesh employed to examine deformable solids.

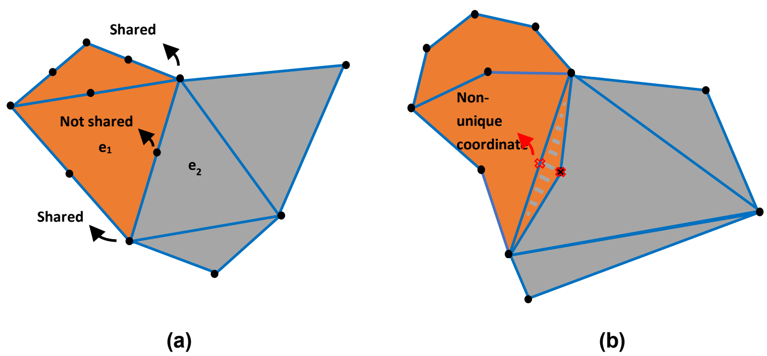

Figure 1-1: An incompatible finite element mesh of a deformable solid (a) before and (b) after deformation

In Figure 1-1b, the displayed deformed configuration is incorrect even after the simulation has converged. The discrepancy in the current deformed configuration arises from an issue of incompatibility in the generated mesh, leading to the shared node between elements 1 and 2 failing to converge to a unique coordinate. A correct deformed configuration of the same mesh is illustrated in Figure 1-2b, where the shared point between elements 1 and 2 occupies a unique coordinate after deformation takes place.

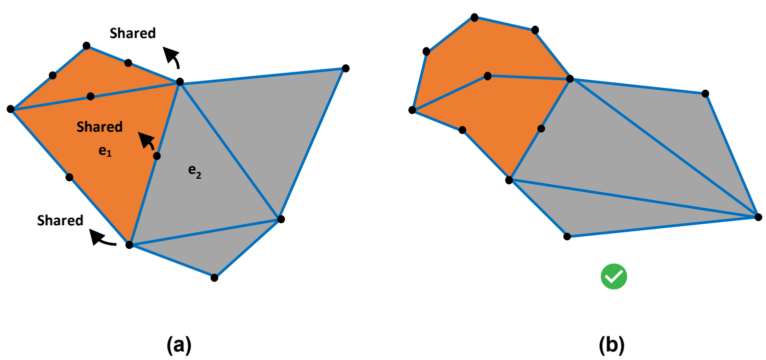

Figure 1-2: A compatible finite element mesh of a deformable solid (a) before and (b) after deformation

2. Classification of the approaches

Different approaches can be used to implement a hybrid mesh in finite element analysis, including using elements with arbitrarily augmented nodes, penalty augmentation, Lagrange multiplier adjunction, and master-slave elimination. All these approaches reduce the number of degrees of freedom compared to a domain entirely meshed using higher-order elements, resulting in faster solving. Although the results may vary slightly, these approaches ensure that the constructed assembled elements are compatible. RS3 employs the master-slave elimination approach due to its elegant formulation, high accuracy, and flexibility for use with different classes of elements (such as 3D solids, joints, liners, embedded structural elements, etc.). The following sections provide a summary of these approaches, with more details on the master-slave elimination approach. An extensively detailed explanation about augmenting arbitrary nodes to special elements can be found in Bathe (2016), and for penalty augmentation, Lagrange multiplier adjunction, and master-slave elimination, refer to Felippa (2004).

2.1.1. Elements with arbitrarily augmented nodes

Once additional node(s) are added to linear elements (e.g., 4-noded tetrahedra), their interpolation functions can be mathematically calculated, used in calculating the stiffness matrix, and assembled with other elements. By doing so, it ensures that the resultant mesh is compatible. For instance, in Figure 1-2a, element 2 is treated as a special triangle with 4 nodes (three nodes at the corners and one node at one of the edges). The interpolation functions for a tetrahedron with arbitrarily augmented node(s) in the natural coordinate system can be found in Bathe (2016, page 375).

2.1.2. Penalty augmentation

In the penalty augmentation approach, fictitious elastic elements, referred to as penalty elements (Zienkiewicz and Taylor, 1993), can be introduced along the boundary of adjacent higher and lower order elements, each with lower dimensions. The penalty elements include constraint equations aligned with the response of the regular higher and lower order elements at the boundary where penalty elements are located. The stiffness and force contributions of these elements are combined using the global assembler. These additional penalty elements incorporate a parametrized numerical weight, adjustable to ensure that the field variables along the boundary adhere to the requirements of both higher and lower order elements. Although this method involves a slightly larger number of elements compared to the approach using elements with arbitrarily augmented nodes, it maintains the same number of degrees of freedom.

The primary advantage of the penalty augmentation approach lies in its straightforward implementation. It also preserves positive definiteness, similar to the first approach. However, a significant drawback is the critical challenge associated with selecting weight values that strike a balance between solution accuracy and the violation of constraint conditions. While a square root rule often suffices for simple cases (Felippa, 2004), its effective application requires knowledge of the magnitudes of stiffness coefficients. In more complex scenarios, determining suitable weights may demand extensive numerical experimentation, leading to a time-consuming process of numerical trials that do not directly contribute to the primary goal—obtaining a solution (Felippa, 2004).

2.1.3. Lagrange multiplier adjunction

In contrast to penalty augmentation, the incorporation of Lagrange multipliers provides the advantage of exactness (Felippa, 2004). This method directly calculates constraint forces at nodes situated at edges shared between higher and lower order elements, eliminating the need for weight assumptions. However, it introduces additional unknowns compared to penalty augmentation and elements with arbitrarily augmented nodes, necessitating the expansion of the original stiffness matrix and more intricate storage allocation procedures. The use of Lagrange multipliers may result in positive indefiniteness in the stiffness matrix, posing challenges for linear equation solving methods relying on positive definiteness (Felippa, 1978). Its implementation is not straightforward, demanding careful attention to identifying singularities caused by constraint dependency and addressing the consequences of the loss of positive definiteness in the stiffness matrix (Felippa, 2004).

2.1.4. Master-slave elimination

The master-slave elimination approach, initially employed by the Boeing development team in the 1950s and detailed by Turner et al. (1964), had an early and general format. Felippa (2004) later refined it to handle versatile multi-freedom constraints in the finite element method. Despite its initial implementation in the NASTRAN code, challenges arose in identifying slave nodes. It is noteworthy that the application of master-slave elimination in hybrid meshing, as presented and employed in RS3, does not encounter the previously mentioned difficulties.

The primary objective is to eliminate the slave nodes from the system of equations by imposing constraints that define their response based on the behavior of the master nodes. The master-slave elimination approach is explored for various constraints, including both homogeneous and non-homogeneous ones, as discussed in Felippa (2004). However, this document specifically focuses on its application in implementing hybrid meshing, involving both higher and lower order elements (10-noded and 4-noded tetrahedra).

In the master-slave elimination approach for hybrid meshing, mid-nodes situated at the edge shared between higher and lower order elements are designated as slave nodes. The remaining nodes along the shared edge are termed master nodes, influencing the response of the slave nodes. It is worth noting that since the slave nodes are exclusive to higher order elements (and not present in lower order elements), these higher order elements are termed buffer elements in this document (e.g., element 1 in Figures 1-1 and 2-1).

The subsequent section presents the formulation of the master-slave elimination approach in hybrid meshing of deformable solids. It is crucial to note that problem formulations related to steady-state and transient flow, coupled hydromechanical analysis, dynamic analysis, joints, and structural elements can be derived based on the concepts presented. RS3 employs the master elimination approach to address all the aforementioned problems.

Consider an assembly of two neighboring elements with different orders, specifically a 10-noded and a 4-noded tetrahedron, sharing a face defined by three common edges, as depicted in Figure 2-1. The number of slave nodes in this mesh is $s = 3$. Denoting n and d as the total number of nodes and degrees of freedom of each node, this assembled mesh is associated with $n = 11$ and $d = 3$. In this figure, the 10-noded buffer and 4-noded elements are labeled as 1 and 2, respectively. Element 2 is composed of nodes $\{1,2,3,4\}$, while element 1 includes nodes $\{1,2,3,5,6,7,8,9,10,11\}$. Additionally, nodes coloured in orange, green, and black are designated as slave, master, and regular nodes, respectively.

The original system of equations that arises from the variational principle of the balance of linear momentum, combined with the spatial discretization, leads to the following format (Zienkiewicz, 1977):

$$[\boldsymbol{K}]\{\boldsymbol{u}\} = \{\boldsymbol{f}\} \qquad (2.1)$$

where $[\boldsymbol{K}]$ is the $nd \times nd$ stiffness matrix with, and $\{\boldsymbol{u}\}$ and $\{\boldsymbol{f}\}$ represent the displacement and force vectors, each with nd components.

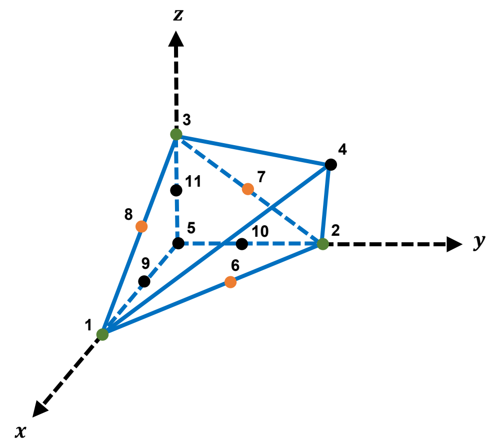

Figure 2-1: An assembly featuring a 10-noded tetrahedron (labeled 1) and a 4-noded tetrahedron (labeled 2), highlighting slave, master, and regular nodes in orange, green, and black colours, respectively

To ensure compatibility within this assembled mesh, the displacement of slave nodes must be determined by the master nodes. In the sample mesh represented in Figure 2-1, this is achieved by employing the interpolation functions (N_i) of element 2 (the 4-noded element), as follows:

$$\begin{cases} u_6 = N_1 u_1 + N_2 u_2 \\ v_6 = N_1 v_1 + N_2 v_2 \\ w_6 = N_1 w_1 + N_2 w_2 \end{cases} \quad \begin{cases} u_7 = N_2 u_2 + N_3 u_3 \\ v_7 = N_2 v_2 + N_3 v_3 \\ w_7 = N_2 w_2 + N_3 w_3 \end{cases} \quad \begin{cases} u_8 = N_1 u_1 + N_3 u_3 \\ v_8 = N_1 v_1 + N_3 v_3 \\ w_8 = N_1 w_1 + N_3 w_3 \end{cases} \qquad (2.2)$$

where u, v, and w represent the displacements along the x, y, and z axes, respectively. Throughout the subsequent derivations, it is assumed that the slave nodes are positioned at the exact midpoint coordinates of the shared edges, maintaining an equal distance from their associated master nodes ($N_i = \frac{1}{2}$; for $i = 1,2,3$). Equation (2.2) illustrates a set of constraints that are well-suited to be expressed in the following generic form:

$$\{\boldsymbol{u}\} = [\boldsymbol{T}]\{\hat{\boldsymbol{u}}\} \qquad (2.3)$$

where $\{\boldsymbol{u}\}$ and $\{\hat{\boldsymbol{u}}\}$ represent displacement vectors, each with nd and $(n-s)d$ components, respectively. It is important to note that $\{\hat{\boldsymbol{u}}\}$ is the same as $\{\boldsymbol{u}\}$ but excludes the displacements associated with slave nodes. Meanwhile, $[\boldsymbol{T}]$ is an $nd \times (n-s)d$ matrix containing the constraint equations that represent the response of slave nodes concerning that of master nodes, as defined in Equation 2.2.

To construct $[\boldsymbol{T}]$, begin by creating an identity matrix with dimensions $nd \times nd$. Then, eliminate the columns associated with degrees of freedom of slave nodes, resulting in a matrix of dimensions $nd \times (n-s)d$. Denoting $i \in \{1,2,3,\dots,nd\}$, and k and l as the degrees of freedom associated with slave and master nodes:

$$T_{ij} = \frac{1}{2}; \quad if\ i = k\ and\ j = l \tag{2.4}$$

Substituting Equation (2.3) into Equation (2.1) and multiplying $[\boldsymbol{T}]^T$ to the resulting expression yields:

$$[\widehat{\boldsymbol{K}}]\{\widehat{\boldsymbol{u}}\} = \{\widehat{\boldsymbol{f}}\}; \qquad [\widehat{\boldsymbol{K}}] = [\boldsymbol{T}]^T[\boldsymbol{K}][\boldsymbol{T}] \quad \text{and} \quad \{\widehat{\boldsymbol{f}}\} = [\boldsymbol{T}]^T\{\boldsymbol{f}\} \tag{2.5}$$

where $[\widehat{\boldsymbol{K}}]$ is a reduced $(n-s)d \times (n-s)d$ stiffness matrix that ensures positive definiteness if $[\boldsymbol{K}]$ is positive definite. Solving Equation (2.5) under the imposed boundary conditions yields the unknown components of $\{\widehat{\boldsymbol{u}}\}$ and $\{\widehat{\boldsymbol{f}}\}$. Then, Equation (2.3) can be utilized to find $\{\boldsymbol{u}\}$.

Alternatively, local constraint matrices, $[\boldsymbol{T}]_{loc}$, can be calculated solely within buffer elements and not on the globally assembled mesh. Subsequently, the reduced stiffness, $[\widehat{\boldsymbol{K}}]_{loc}$, and force vector, $\{\widehat{\boldsymbol{f}}\}_{loc}$, are constructed and then assembled with the regular elements existing in the meshed domain. This approach, which yields the same result, is flexible and can be rigorously implemented for other classes of elements (e.g., joints, structural elements, etc.) and is employed in RS3.

As demonstrated in the derivations, the implementation of the master-elimination approach in hybrid meshing is highly straightforward. This approach benefits not only from positive definiteness but also results in a reduced system of equations smaller than those of the first three approaches. Similar to the Lagrange multiplier adjunction, the constraints are precisely embedded in the solution. As mentioned earlier, the hybrid formulations used in RS3 for steady-state and transient flow, coupled hydromechanical analysis, dynamic analysis, and the utilization of features such as joints and structural elements in the models are all comparable to the formulation presented above for deformable solids, with slight differences in their governing equations and stiffness and vector calculations.

3. References

1. Bathe, K-J., *Finite Element Procedures*, 2nd ed., 4th print. Prentice Hall, Pearson Education, Inc., Waterdown, Massachusetts, 2016.
2. Felippa, C. A., *Introduction to Finite Element Methods*, University of Colorado, Boulder, Colorado, 2004.
3. Felippa, C. A., Iterative procedures for improving penalty function solutions of algebraic systems, International Journal for Numerical Methods in Engineering, 12, 821-836, 1978.
4. Zienkiewicz, O. C. and Taylor, R. E., *The Finite Element Method*, 4th ed., McGraw-Hill, London, Vol II: 1993.
5. Zienkiewicz, O. C., *The Finite Element Method*, 3rd ed., McGraw-Hill, London, 1977.