有限元理论基础及Abaqus内部实现方式研究系列1：S4壳单元刚度矩阵研究

Theoretical Foundation of Finite Element Method and Research on Internal Implementation of Abaqus Series 1: Study on S4 Shell Element Stiffness Matrix

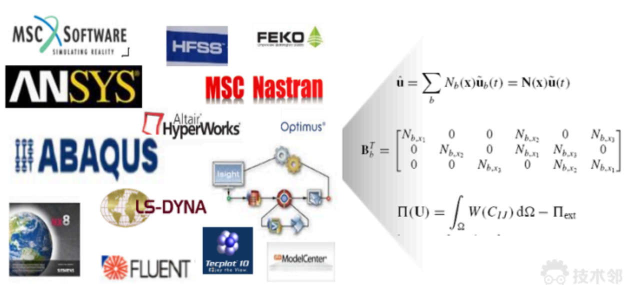
SnowWave02

更新于2019年11月27日 02:20 Updated on November 27, 2019, 02:20

==概述== ==Overview==

本系列文章研究成熟的有限元理论基础及在商用有限元软件的实现方式。有限元的理论发展了几十年已经相当成熟，商用有限元软件同样也是采用这些成熟的有限元理论，只是在实际应用过程中，商用CAE软件在传统的理论基础上会做相应的修正以解决工程中遇到的不同问题，且各家软件的修正方法都不一样，每个主流商用软件手册中都会注明各个单元的理论采用了哪种理论公式，但都只是提一下用什么方法修正，很多没有具体的实现公式。商用软件对外就是一个黑盒子，除了开发人员，使用人员只能在黑盒子外猜测内部实现方式。

This series of articles studies the mature finite element theory and its implementation methods in commercial finite element software. The development of finite element theory has matured over decades, and commercial finite element software also adopts these mature finite element theories. However, in the actual application process, commercial CAE software will make corresponding corrections on the basis of traditional theories to solve different problems encountered in engineering, and the correction methods of each software are different. Each mainstream commercial software manual will specify which theoretical formula each element uses, but only mention the correction method, and many do not provide specific implementation formulas. Commercial software is essentially a black box, and users can only guess its internal implementation methods from outside, except for developers.



一方面我们查阅各个主流商用软件的理论手册并通过进行大量的资料查阅猜测内部修正方法，另一方面我们自己编程实现结构有限元求解器，通过自研求解器和商软A的结果比较来验证我们的猜测，如同管中窥豹一般来研究的修正方法，从而猜测商用有限元软件的内部计算方法。我们关注CAE中的结构有限元，所以主要选择了商用结构有限元软件中文档相对较完备的Abaqus来研究内部实现方式，同时对某些问题也会涉及其它的Nastran/Ansys等商软。为了理解方便有很多问题在数学上其实并不严谨，同时由于水平有限可能有许多的理论错误，欢迎交流讨论，

也期待有更多的合作机会。

On the one hand, we consult the theoretical manuals of various mainstream commercial software and conduct a large amount of literature review to guess the internal correction methods, on the other hand, we ourselves program the structural finite element solver, and verify our guesses by comparing the results of our self-developed solver and commercial software Abaqus, studying the correction methods as if peering through a bamboo tube to understand the big picture, thus guessing the internal calculation methods of commercial finite element software. Since we focus on structural finite elements in CAE, we mainly choose Abaqus, which has relatively complete documentation for commercial structural finite element software, to study its internal implementation methods, and will also involve other commercial software such as Nastran/Ansys for some issues. Many problems are not mathematically rigorous for the sake of convenience in understanding, and due to our limited level, there may be many theoretical errors. We welcome communication and discussion and look forward to more cooperation opportunities.

iSolver介绍视频： iSolver Introduction Video:

http://www.jishulink.com/college/video/c12884

===第一篇：S4壳单元刚度矩阵研究。 === First Article: Study on S4 Shell Element Stiffness Matrix.

Abaqus的壳单元刚度矩阵的理论基础都是Kirchihoff（薄壳）和Mindlin（厚壳）理论，本章重点研究S4壳单元，该单元基于Mindlin理论，在自编程序中根据Mindlin理论编写后和Abaqus结果对比，可以发现Mindlin和Abaqus差异很大，然后结合帮助文档猜测Abaqus的S4单元的内部修正方法。

The theoretical foundation of Abaqus' shell element stiffness matrix is based on Kirchhoff (thin shell) and Mindlin (thick shell) theories. This chapter focuses on the S4 shell element, which is based on the Mindlin theory. By writing the program according to the Mindlin theory and comparing it with the results of Abaqus, it can be found that there is a significant difference between Mindlin and Abaqus. Then, combined with the help documentation, we guess the internal correction method of Abaqus' S4 element.

===S4单元修正方法总结 ===Summary of S4 Element Correction Methods

Abaqus的S4单元的薄膜效应刚度和面外弯曲刚度矩阵是完全积分，面外横向剪切刚度是减缩积分。

The membrane effect stiffness and out-of-plane bending stiffness matrices of Abaqus' S4 element are completely integrated, while the out-of-plane shear stiffness is reduced integration.

具体的刚度矩阵在Mindlin理论基础上的修正如下表：

The specific stiffness matrix is corrected based on the Mindlin theory as shown in the table below:

项次 Item number	刚度 Stiffness	修正情况 Correction		说明 Description
		修正 Correction	不修正 Not Correction	
1	薄膜效应刚度 Thin film effect stiffness	√		Abaqus为了消除壳单元出现的剪切锁死，计算薄膜效应时把壳作为体来处理 Abaqus treats the shell as a solid to calculate the thin film effect in order to eliminate the shear locking that occurs in shell elements
2	面外弯曲刚度 Out-of-plane bending stiffness		√	
3	面外横向剪切刚度 Out-of-plane shear stiffness	√		（1）Abaqus为了处理薄板情况，增加了一个几何因子 (1) Abaqus adds a geometric factor to handle thin plate situations （2）为了消除沙漏问题增加相关刚度 (2) Related stiffness is added to eliminate the hourglass problem
4	第6自由度刚度 6th degree of freedom stiffness	√		Abaqus取了和对角刚度相关的小量 Abaqus takes a small amount related to the diagonal stiffness
5	其它非对角元素 Other non-diagonal elements	√		（1）during analysis选项导致的小量 (1) Small quantities caused by the "During Analysis" option

项次 Item number	刚度 Stiffness	修正情况 Correction		说明 Description
		修正 Correction	不修正 Not Correction	
				（2）第6自由度刚度相关小量 (2) Small quantities related to the 6th degree of freedom stiffness

详细研究方法，见附件： Detailed research methods, see attachment:

[有限元理论基础及Abaqus内部实现方式研究系列1：S4壳单元刚度矩阵研究(SnowWave02 20170708).pdf]

Finite Element Theory and Abaqus Internal Implementation Research Series 1: S4 Shell Element Stiffness Matrix Research (SnowWave02 20170708).pdf