有限元理论基础及Abaqus内部实现方式研究系列5：单元正确性验证

Theoretical Foundation of Finite Element Method and Internal Implementation of Abaqus Series 5: Element Correctness Verification

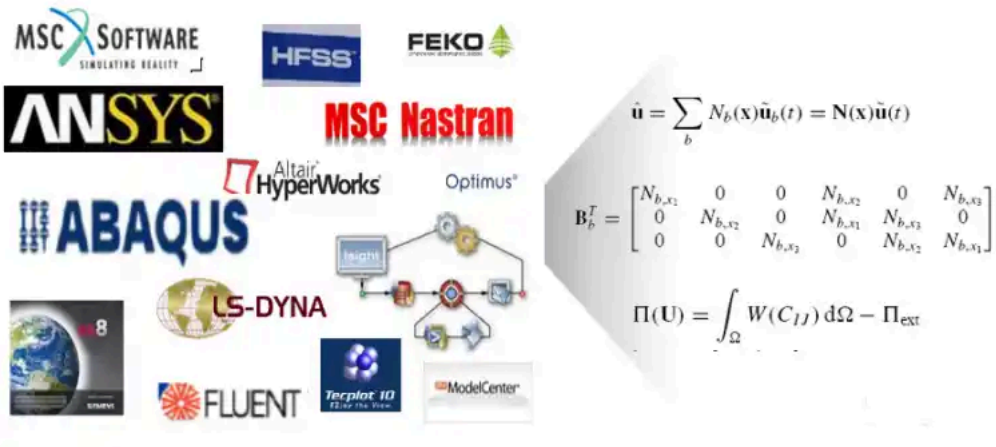
SnowWave02 关注 Focus

更新于2021年3月3日 11:27 Updated on March 3, 2021, 11:27 AM 浏览：2643 Views: 2643 收藏：5 Favorites: 5

==概述== ==Overview==

在CAE领域，从学校、实验室的自研算法到实现真正的商业化软件是一条无比漫长的道路。我们不研究有限元的新方法、新理论，只是研究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，商用软件在这些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。

In the field of CAE, the path from the independently developed algorithms in schools and laboratories to the realization of real commercial software is an incredibly long journey. We do not study new methods or theories of finite element, but only research the implementation methods of commercial finite element software. The theoretical development of finite element has matured for decades, and commercial finite element software also adopts these mature finite element theories. However, in the actual application process, commercial software will make corresponding corrections on the basis of these 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.



一方面我们查阅Abaqus软件手册得到修正方法的说明，另一方面我们自己编程实现简单的结构有限元求解器，通过自研求解器和Abaqus的结果比较结合理论手册如同管中窥豹一般来研究Abaqus的修正方法，从而猜测商用有

限元软件的内部计算方法。在研究的同时，准备将自己的研究成果记录下来写成一个系列文章，希望对那些不仅仅满足使用软件，而想了解软件内部实现方法甚至是做自己的软件的朋友有些帮助。由于水平有限，里面可能有许多错误，欢迎交流讨论。

On one hand, we obtain the description of the correction method from the Abaqus software manual, and on the other hand, we program a simple structural finite element solver ourselves. By comparing the results of our independently developed solver with Abaqus and studying the correction methods of Abaqus as if through a tube to see a dragon, we can guess the internal calculation methods of commercial finite element software. While studying, I am preparing to record my research findings in a series of articles, hoping to help those who are not only satisfied with using the software but also want to understand the internal implementation methods of the software or even develop their own software. Due to my limited abilities, there may be many errors, and I welcome discussions and exchanges.

==以往的系列文章== ==Previous Series Articles==

第一篇：S4壳单元刚度矩阵研究。介绍Abaqus的S4刚度矩阵在普通厚壳理论上的修正。

First article: Research on the Stiffness Matrix of S4 Shell Element. Introduces the correction of Abaqus' S4 stiffness matrix on the theory of thin shell.

http://www.jishulink.com/content/post/338859

第二篇：S4壳单元质量矩阵研究。介绍Abaqus的S4和Nastran的Quad4单元的质量矩阵。

Second article: Research on the Mass Matrix of S4 Shell Element. Introduces the mass matrices of Abaqus' S4 and Nastran's Quad4 elements.

http://www.jishulink.com/content/post/343905

第三篇：S4壳单元的剪切自锁和沙漏控制。介绍Abaqus的S4单元如何来消除剪切自锁以及S4R如何来抑制沙漏的。

Third article: Shear locking and hourglass control of S4 shell elements. This article introduces how Abaqus S4 elements eliminate shear locking and how S4R suppresses hourglassing.

http://www.jishulink.com/content/post/350865

第四篇：非线性问题的求解。介绍Abaqus在非线性分析中采用的数值计算的求解方法。

Fourth article: Solution of nonlinear problems. This article introduces the numerical computation methods adopted by Abaqus in nonlinear analysis.

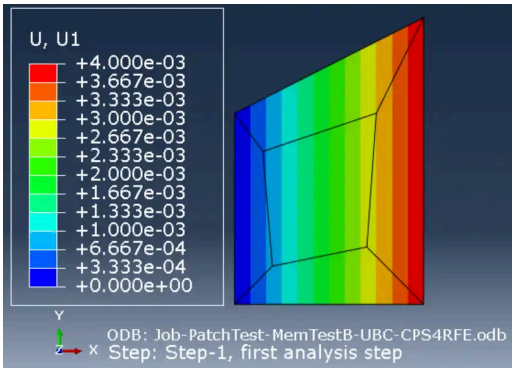
http://www.jishulink.com/content/post/360565

==第五篇：单元正确性验证== ==Fifth article: Element Correctness Verification==

本文介绍了单元正确性的验证方法并对Abaqus的一次梁和壳单元进行了部分算例验证，当然Abaqus正确性是肯定没问题的，我们只是将Abaqus的结果作为一个标准，后面主要是对自研程序iSolver梁壳单元进行分析并与Abaqus结果进行比较来详细说明整个的验证过程。水平有限，现在还没法做出一个完备到单元每个功能点都能测试到的算例集，只能尽力覆盖单元的功能点，算例有一部分是自己做的，还有一些是其它书或者论文中的，后者的算例也提供了来源说明方便你做更多的了解，同时，附上文章中提到的所有模型文件，你可以下载在Abaqus中运

行验证Abaqus的正确性，也可以用来测试你自己编的的有限元程序。

This article introduces the verification methods for element correctness and performs partial case verification on Abaqus's one-time beam and shell elements. Of course, Abaqus's correctness is definitely not an issue; we are just using Abaqus's results as a standard. The main part of the article is to analyze the self-developed iSolver beam-shell element and compare it with Abaqus results to detail the entire verification process. My level is limited, and I haven't been able to create a complete set of test cases that can test each functional point of the element yet. I can only try to cover the functional points of the element. Some of the test cases are self-made, and some are from other books or papers. The sources of the latter test cases are also provided for you to gain more understanding. At the same time, all the model files mentioned in the article are attached, which you can download and run in Abaqus to verify the correctness of Abaqus, or to test your own finite element program.



==单元正确性验证总结== ==Element Correctness Verification Summary==

本文从三个方面来对自研程序iSolver梁壳单元进行分析并与Abaqus结果进行比较来详细说明整个的验证过程。结果如下：

This article analyzes the self-developed iSolver beam-shell element from three aspects and compares it with the Abaqus results to detail the entire verification process. The results are as follows:

考核内容 Evaluation Content	考核结果 Evaluation Results	说明 Description
刚度矩阵 Stiffness Matrix	梁完全和理论一致，壳很大不同 The beam is completely consistent with the theory, but the shell is significantly different	Abaqus和iSolver内部B31完全按照Timoshenko梁理论实现，壳算法不同，结果不同 Abaqus and iSolver internally implement B31 completely according to Timoshenko's beam theory, but the shell algorithms are different, resulting in different results
分片试验 Segment Test	Abaqus和iSolver都通过 Abaqus and iSolver both pass	-

基本算例 Basic example	iSolver 结果和Abaqus部分结果只有计算机精度差异，部分存在差异 The results of iSolver and Abaqus only have differences in computer precision, with some differences	Abaqus和iSolver所采用的单元算法不同，在网格细化后结果可以更加接近 The element algorithms used by Abaqus and iSolver are different, and the results can be closer after mesh refinement

详细研究方法，见附件： A detailed research method, see the attachment:

有限元理论基础及Abaqus内部实现方式研究系列5：单元正确性验证(SnowWave02 20180613).pdf

Finite Element Theory and Abaqus Internal Implementation Series 5: Element Accuracy Verification (SnowWave02 20180613).pdf

操作视频: https://www.jishulink.com/college/video/c12884

Operation video: https://www.jishulink.com/college/video/c12884

文章包括的Abaqus模型如下： The Abaqus models included in the article are as follows:

推荐阅读 Recommended Reading

Abaqus、iSolver与Nastran梁单元差异...
SnowWave02 免费 Free

转子旋转的周期性模型-水冷电机散热仿真 Periodic Model of Rotor...
技术邻小李 Technical Neighbor Xiao Li ¥100 100 Yuan

网架设计之SFCAD第七节终极大招 Design of Grid Structure: The...
高工 Senior Engineer ¥138

ansys经典半刚性脚手架分析（2） Ansys Classic Semi
冷月 Cold Moon ¥1