有限元理论基础及Abaqus内部实现方式研究系列6：General梁单元刚度矩阵
Theoretical Foundation of Finite Element Method and Research on Internal Implementation of Abaqus Series 6: General Beam Element Stiffness Matrix

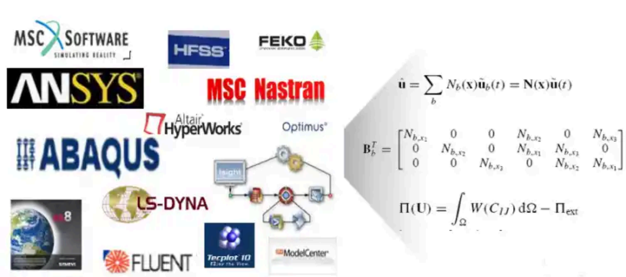
SnowWave02 关注 Focus

更新于2021年3月3日 11:29 Updated on March 3, 2021, 11:29 AM
浏览：2916 Views: 2916
收藏：4 Favorites: 4

==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 analysis, but rather investigate the implementation methods of commercial finite element software. The theoretical development of finite element analysis has matured over several 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 briefly mention the correction methods, with many lacking 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 methods 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 those of Abaqus, and studying Abaqus' correction methods as if through a tube, we can guess the internal calculation methods of commercial finite element software. While conducting research, 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.

==第六篇：General梁单元的刚度矩阵== == Sixth Article: General Beam Element Stiffness Matrix ==

相对壳来说，在实际应用过程中，商业软件对梁的修正方式相对较少，如果自己编程序，采用这些修正方式可以得到和商业软件完全一致的刚度矩阵，如果刚度矩阵完全一致，那么对任何的算例都可以得到和商业软件完全一致的结果了。

Compared to shells, in the actual application process, commercial software has relatively fewer correction methods for beams. If you write your own program and adopt these correction methods, you can obtain stiffness matrices that are completely consistent with those of commercial software. If the stiffness matrix is completely consistent, then for any example, you can obtain results that are completely consistent with those of commercial software.

实际的梁都是有截面形状的，商用软件分析时都采用两步走的形式：

Actual beams all have cross-sectional shapes, and commercial software analysis usually takes a two-step approach:

（1） 第一步：通过这些截面形状类型和参数得到构建梁单元所需的基本截面属性参数，譬如矩形面积=长*宽等。

(1) The first step: Obtain the basic cross-sectional property parameters required for the construction of beam elements through these types and parameters of cross-sectional shapes, such as rectangular area = length * width.

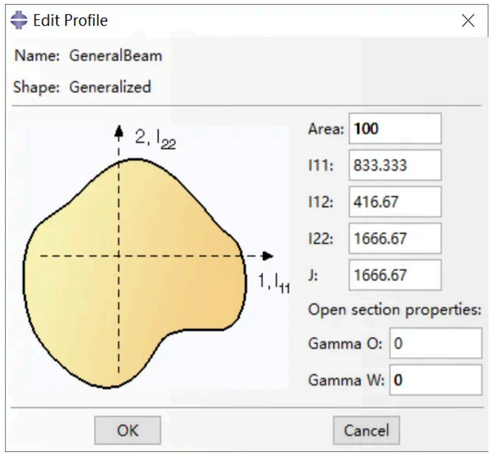
（2） 第二步：利用上面得到的截面属性参数组成梁单元的刚度矩阵。

(2) The second step: Utilize the cross-sectional property parameters obtained above to form the stiffness matrix of the beam element.

相对应的，一般商用软件的梁都有两类： Correspondingly, commercial software beams generally have two types:

（1） 一类是已知截面属性参数的General梁，在Abaqus中创建梁时选择General就是General梁。

(1) One type is the General beam with known section property parameters, in Abaqus, when creating a beam, selecting General is the General beam.



（2） 另一类是已知截面形状类型和几何尺寸的Geometry梁，在Abaqus创建梁截面时选择除General外的其它选项都是Geometry梁。

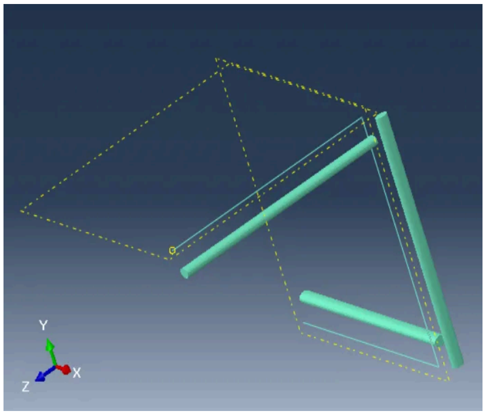
(2) The other type is the Geometry beam with known section shape type and geometric dimensions, in Abaqus, any option other than General when creating beam sections is the Geometry beam.

本篇先讨论General梁单元的刚度矩阵的基本理论和Abaqus的修正方式，Geometry梁的计算方法只是比第一类梁多了一步怎么从截面几何参数得到截面属性参数，当然针对不同形状类型，Abaqus也做了许多的修正，将放到下一篇中讨论。

This article discusses the basic theory of the General beam element stiffness matrix and the correction methods in Abaqus first, and the calculation method of the Geometry beam is just one step more than the first type of beam on how to obtain section property parameters from section geometric parameters. Of course, Abaqus also makes many corrections for different shape types, which will be discussed in the next article.

本文首先简单介绍梁单元的基本理论，分析了每一部分刚度的来源，并研究了Abaqus中General梁的B31单元的刚度矩阵的修正方式，采用这些修正方式可以得到和Abaqus完全一致的刚度矩阵，自研软件的正确性证明是非常困难的，无论你测试多少标准算例或者和试验结果对比，都很难让用户踏实的相信你的结果和商业软件是一致的，但如果一个模型仅有General梁组成，那么无论这个模型多么的复杂，我们一般都可以得到和Abaqus完全一致的分析结果。最后我们使用了一个简单的算例综合验证，针对该算例我们对比采用了同样修正的自编程序iSolver和Abaqus的结果，可以发现结果如预期的一样，没有任何误差。

This article first briefly introduces the basic theory of the beam element, analyzes the source of each part of the stiffness, and studies the correction method of the B31 stiffness matrix of the General beam in Abaqus. By using these correction methods, we can obtain a stiffness matrix that is completely consistent with Abaqus. It is very difficult to prove the correctness of self-developed software, no matter how many standard calculation examples you test or compare with experimental results, it is difficult to make users believe that your results are consistent with commercial software. However, if a model is only composed of General beams, then no matter how complex the model is, we can generally obtain analysis results that are completely consistent with Abaqus. Finally, we use a simple example to comprehensively verify. For this example, we compare the results of the self-developed program iSolver with the same correction and Abaqus, and it can be found that the results are as expected with no errors.



==演示视频== ==Demonstration Video==

https://www.jishulink.com/college/video/c12884?chapter=1

梁的静力分析结果校核视频演示任意分段梁的静力分析，证明iSolver的分析结果和Abaqus没有误差，有两个算例：Generalized梁和L型梁。

Video demonstration of the verification of the static analysis results of beams, proving that the analysis results of iSolver have no error compared to Abaqus. There are two examples: Generalized beam and L-shaped beam.

==总结== ==Summary==

General梁的B31单元的刚度矩阵在Timoshenko梁理论基础上的修正如下表：

The stiffness matrix of the B31 element of the General beam is corrected based on the Timoshenko beam theory as shown in the table below:

项次 Item number	**刚度 Stiffness**	**修正 Correction**	**不修正 Not Correction**	**说明 Description**
1	轴向拉伸刚度 Axial tensile stiffness		√	
2	横向弯曲刚度 Lateral bending stiffness	√		采用减缩积分 Using reduced integration
3	轴向拉伸和横向弯曲耦合刚度 Coupled stiffness of axial tension and lateral bending	√		根据形心偏置进行修正 Correction based on centroid offset
4	轴向扭转刚度 Axial torsional stiffness	√		剪切中心的偏置会影响抗扭刚度系数J The offset of the shear center affects the torsional stiffness coefficient J
5	横向剪切刚度 Lateral shear stiffness	√		增加了一个几何因子，使得细长梁的时候该项趋于0。 An additional geometric factor has been added, making this term tend to 0 for slender beams.

项次 Item number	刚度 Stiffness	修正 Correction	不修正 Not Correction	说明 Description
6	轴向扭转和横向剪切耦合刚度 Axial torsion and lateral shear coupled stiffness	√		根据剪切中心的偏置进行修正。 Correction based on the offset of the shear center
7	其它元素 Other elements		√	都为0 All are 0

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6： General梁单元刚度矩阵(SnowWave02 20180830).pdf]

Finite Element Theory and Abaqus Internal Implementation Series 6: General Beam Element Stiffness Matrix (SnowWave02 20180830).pdf

==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。介绍有限元单元正确性的验证方法，通过多个实例比较自研结构求解器程序iSolver与Abaqus的分析结果，从而说明整个正确性验证的过程和iSolver结果的正确性。

Chapter 5: Element Accuracy Verification. Introduces the verification methods for finite element element accuracy, compares the analysis results of the self-developed structural solver program iSolver with Abaqus through multiple examples, thereby illustrating the entire accuracy verification process and the correctness of the iSolver results.

https://www.jishulink.com/content/post/373743