有限元理论基础及Abaqus内部实现方式研究系列12： 几何梁单元的刚度矩阵

Theoretical Foundation of Finite Element Method and Research on the Internal Implementation of Abaqus Series 12: Stiffness Matrix of Geometric Beam Element

SnowWave02　关注 **Focus**

更新于2021年3月3日 11:30　Updated on March 3, 2021, 11:30 AM

浏览： **3837** Views: 3837　　评论： **9** Comments: 9　　收藏： **5** Favorites: 5
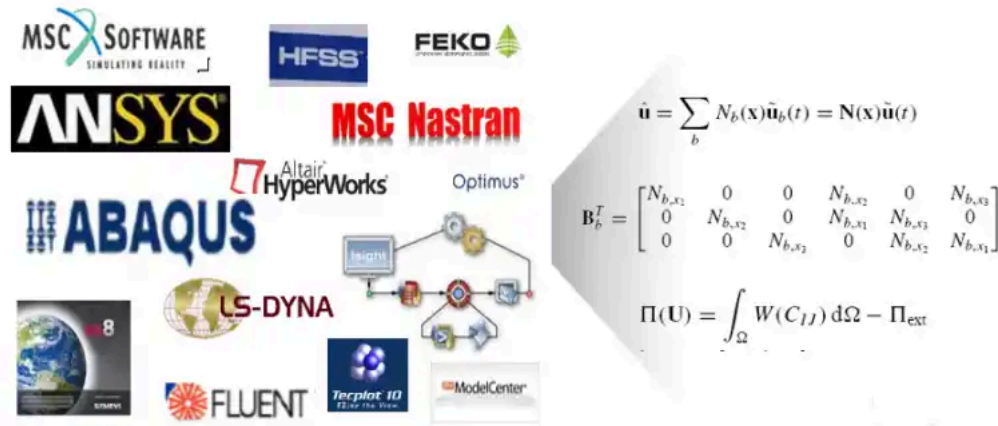
（原创，转载请注明出处）　(Original, please indicate the source for reproduction)

==**概述**==　==Overview==

有限元理论基础及Abaqus内部实现方式研究系列12： 几何梁单元的刚度矩阵的图1

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 theoretical foundation 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 specifies which theoretical formula each element uses, but only mentions 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.

$$\dot{\mathbf{u}} = \sum_b N_b(\mathbf{x})\tilde{\mathbf{u}}_b(t) = \mathbf{N}(\mathbf{x})\tilde{\mathbf{u}}(t)$$

$$\mathbf{B}_b^T = \begin{bmatrix} N_{b,x_1} & 0 & 0 & N_{b,x_2} & 0 & N_{b,x_3} \\ 0 & N_{b,x_2} & 0 & N_{b,x_1} & N_{b,x_3} & 0 \\ 0 & 0 & N_{b,x_3} & 0 & N_{b,x_2} & N_{b,x_1} \end{bmatrix}$$

$$\Pi(\mathbf{U}) = \int_\Omega W(C_{IJ})\,\mathrm{d}\Omega - \Pi_{ext}$$

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 one hand, we consult the theoretical manuals of various mainstream commercial software and guess the internal correction methods through extensive literature review. On the other hand, we program our own structural finite element solver and verify our guesses by comparing the results with those of commercial software. We study the correction methods like a glimpse through a tube, 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 among commercial structural finite element software, to study the internal implementation methods, and we will also involve other commercial software such as Nastran/Ansys for some issues. Many problems are not mathematically rigorous for the sake of understanding convenience, and due to our limited level, there may be many theoretical errors. We welcome discussions and look forward to more cooperation opportunities.

iSolver介绍： iSolver Introduction:

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

==第12篇：几何梁单元的刚度矩阵的研究== ==12th Article: Research on the Stiffness Matrix of Geometric Beam Elements==

一般商用软件的梁都有两类： General commercial software beams are divided into two categories:

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

(1) One type is the beam with known cross-sectional property parameters, known as General beam (also called parameter beam), when creating a beam in Abaqus, selecting General represents the General beam.
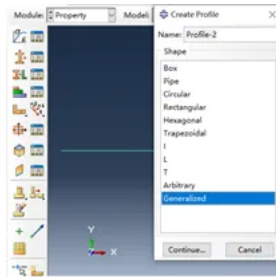
（2）　另一类是已知截面形状类型和几何尺寸的梁，即Geometry梁（也叫几何梁），在Abaqus创建梁截面时选择除General外的其它选项都是Geometry梁。

(2) The other type is the beam with known cross-sectional shape and geometric dimensions, known as Geometry beam (also called geometric beam), when creating beam sections in Abaqus, selecting options other than General are Geometry beams.

实际的梁都是有截面形状的，也就是几何Geometry梁，商用软件分析时都采用两步走的形式：

Actual beams all have cross-sectional shapes, that is, geometric Geometry beams, and commercial software analysis usually adopts 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.

梁相对壳来说，商业软件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梁单元刚度矩阵，如果刚度矩阵完全一致，那么对任何的梁的算例都可以得到和商业软件完全一致的结果了。在本系列第六篇我们讨论了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计算截面属性参数时也做了许多的修正，本篇中将进行讨论。然后在自编有限元程序iSolver实现同样的修正方式，最后验证iSolver的结果和Abaqus完全一致，从而证明Abaqus对几何梁的内部修正和我们设想的一致。同时，通过L梁的算例将会证明Abaqus用户手册的一点小错误（难得发现Abaqus文档的错误^.^），即General梁的惯性矩后台计算

时是相对于形心的（而不是文档所说的相对于1-2轴的）。具体验证过程也可以参考我们的演示录像。

Compared to shells, the correction methods for beams in commercial software are relatively few. If you write your own program, using these correction methods can obtain a beam element stiffness matrix that is completely consistent with commercial software. If the stiffness matrix is completely consistent, then for any beam example, you can obtain results that are completely consistent with commercial software. In the sixth article of this series, we discussed the basic theory of the General beam element stiffness matrix and the correction methods of Abaqus. The calculation method of Geometry beam is just one step more than the first kind of beam, that is, how to obtain the cross-sectional property parameters from the cross-sectional geometric parameters. Of course, Abaqus also makes many corrections when calculating the cross-sectional property parameters for different shape types. This article will discuss this. Then, the same correction method will be implemented in the self-written finite element program iSolver, and the results of iSolver will be verified to be completely consistent with Abaqus, thus proving that the internal correction of Abaqus for geometric beams is consistent with our expectations. At the same time, through the example of the L beam, it will be proven that there is a minor error in the Abaqus user manual (it's rare to find errors in Abaqus documents ^.^), that is, the inertia moment of the General beam is calculated relative to the centroid (not relative to the 1-2 axis as stated in the document). The specific verification process can also be referred to in our demonstration video.

==演示视频== ==Demonstration Video==

https://www.jishulink.com/college/video/c12884 1.1 单元篇-梁的静力分析结果校核

https://www.jishulink.com/college/video/c12884 1.1 Unit Article - Static Analysis Result Verification of Beams

 ==总结== ==Summary==

本文研究了Abaqus中几何梁的B31单元的刚度矩阵的求解方式，以L梁为例，介绍General梁用到的面积、惯性矩、扭转常数等参数在几何梁中是如何通过几何形状求得的，根据这些参数，可以得到和Abaqus完全一致的刚度矩阵，从而对只有几何梁组成的任意模型一般都能得到Abaqus完全一致的分析结果，并用一个简单的算例验证了该想法。

This article studies the solution method of the B31 element stiffness matrix of geometric beams in Abaqus, taking the L beam as an example, and introduces how the parameters such as area, inertia moment, and torsion constant used in General beams are obtained through geometric shapes in geometric beams. Based on these parameters, a stiffness matrix consistent with Abaqus can be obtained, so that for any model consisting only of geometric beams, Abaqus consistent analysis results can generally be obtained. This idea has been verified by a simple example.
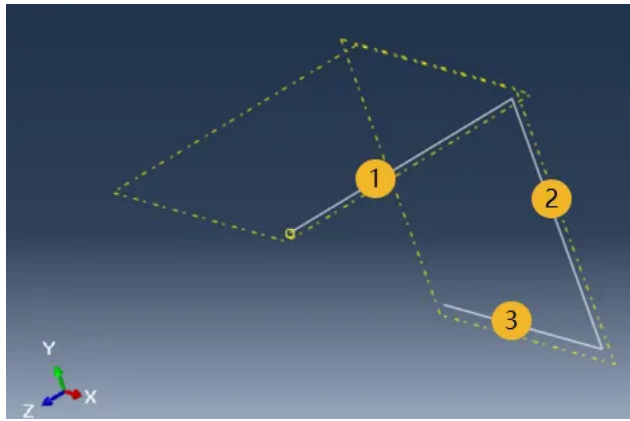
从一个几何梁到General梁的转化，需要计算的量和Abaqus的修正情况如下：

The transformation from a geometric beam to a General beam requires the following quantities to be calculated and the corresponding corrections in Abaqus:

项次	计算梁	修正情况		说明
		修正	不修正	
1	梁方向	√		由界面得到互相垂直的 1 方向 (1-direction)和 2 方向(2-direction)，注意，和 Patran 完全颠倒，这也是 abaqus 自带的转换 bdf 经常失败的原因之一。
2	面积		√	
3	惯性矩	√		采用沿积分点做积分的方式。
4	扭转常数	√		根据截面形状不同而不同，对于矩形、梯形截面，Abaqus 采用 Prandtl 应力积分法。
5	刚度矩阵其它元素	√		选择 during analysis 选项，将导致一个小量

通过L梁也可以得到，General梁的惯性矩是相对于形心的（而不是相对于1-2轴的），abaqus用户手册写错了。

The moment of inertia of the General beam is with respect to the centroid (not with respect to the 1-2 axis), which is incorrectly written in the Abaqus user manual.

有兴趣的可以自行下载iSolver进行验证，因为看不到Abaqus的源代码，上述B31的修正方式也仅是猜测，如果你在使用iSolver测试其它的由几何梁组成的模型结果时发现和Abaqus结果不一致，欢迎联系我们。

Those who are interested can download iSolver for verification, as the source code of Abaqus is not visible, and the correction method of B31 mentioned above is just a guess. If you find that the results of other models composed of geometric beam elements tested with iSolver are inconsistent with the results of Abaqus, please contact us.

如果有任何其它疑问或者项目合作意向，也欢迎联系我们：

If you have any other questions or intentions for project cooperation, feel free to contact us:

snowwave02 From www.jishulink.com

email: snowwave02@qq.com

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

技术邻   首页 Home   学院 College   直播 Live Streaming   问答 Q&A   悬赏 Bounty   🔥会议   🔥Conference

[有限元理论基础及Abaqus内部实现方式研究系列12：几何梁单元的刚度矩阵.pdf](#)

[Finite Element Theory and the Internal Implementation of Abaqus: Series 12 - Stiffness Matrix of Geometric Beam Elements.pdf](#)

以往的系列文章：   Previous series articles:

有限元理论基础及Abaqus内部实现方式研究系列12： 几何梁单元的刚度矩阵的图10

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 in the theory of ordinary thick 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. 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。

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

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

第六篇：**General梁单元的刚度矩阵**。介绍梁单元的基础理论和Abaqus中General梁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梁单元完全一致的刚度矩阵。

Sixth article: Stiffness matrix of General beam element. Introduces the basic theory of beam elements and the correction methods of the General beam element stiffness matrix in Abaqus. By using these correction methods, it is possible to obtain a stiffness matrix that is completely consistent with the Abaqus beam element.

https://www.jishulink.com/content/post/403932

第七篇：**C3D8六面体单元的刚度矩阵**。介绍六面体单元的基础理论和Abaqus中C3D8R六面体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六面体单元完全一致的刚度矩阵。

Seventh article: Stiffness matrix of C3D8 hexahedral element. Introduces the basic theory of hexahedral elements and the correction methods of the C3D8R hexahedral element stiffness matrix in Abaqus. By using these correction methods, it is possible to obtain a stiffness matrix that is completely consistent with the Abaqus hexahedral element.

https://www.jishulink.com/content/post/430177

第八篇：**UMAT用户子程序开发步骤**。介绍基于Fortran和Matlab两种方式的Abaqus的UMAT的开发步骤，对比发现开发步骤基本相同，同时采用Matlab更加高效和灵活。

Eighth article: Steps for UMAT user subroutine development. Introduces the development steps of Abaqus UMAT based on both Fortran and Matlab, and finds that the development steps are basically the same. At the same time, Matlab is found to be more efficient and flexible.

https://www.jishulink.com/content/post/432848

第九篇：**编写线性UMAT Step By Step**。介绍了线性UMAT的接口功能和关键接口变量的含义，并通过简单立方体静力分析的算例详细说明了基于Matlab线性UMAT的开发步骤。

Chapter 9: Writing Linear UMAT Step by Step. Introduces the interface functions of linear UMAT and the meanings of key interface variables, and illustrates the development steps of linear UMAT based on Matlab through a simple cube static analysis example.

http://www.jishulink.com/content/post/440874

第十篇：**耦合约束（Coupling constraints）的研究**。介绍了耦合约束的定义和用途，具体阐述了Abaqus中运动耦合约束和分布耦合约束的原理。

Chapter 10: Research on Coupling Constraints. Introduces the definition and application of coupling constraints, and specifically elaborates on the principles of motion coupling constraints and distributed coupling constraints in Abaqus.

http://www.jishulink.com/content/post/531029

第十一篇：**自主CAE开发实战经验第一阶段总结**。结合自研有限元求解器iSolver第一阶段开发的实战经验，从整体角度上介绍自主CAE的开发难度、时间预估、框架设计、编程语言选择、测试、未来发展方向等。

Chapter 11: Summary of the First Phase of Independent CAE Development Practice. Based on the practical experience of the first phase development of the independently developed finite element solver iSolver, this article introduces the difficulties, time estimation, framework design, programming language selection, testing, and future development directions of independent CAE development from an overall perspective.

http://www.jishulink.com/content/post/532475

****

***
线下培训 Offline Training
***

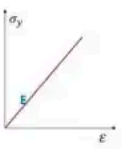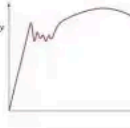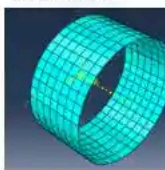**********************************

如想了解更多或者需要与我们当面交流，欢迎参加近期我们的线下培训。

If you want to learn more or need to communicate with us in person, please join our upcoming offline training.

【7月20-21日 上海】Abaqus UMAT用户子程序二次开发技术培训:

【July 20-21, Shanghai】Abaqus UMAT User Subroutine Secondary Development Technical Training

http://www.jishulink.com/content/post/531598

培训大纲如下：　Training outline as follows:

时间	主题	主要内容
第一天上午	总体介绍及实例展示	1、Abaqus二次开发总体介绍： 　a）用户子程序 　b）基于python的GUI开发 2、实例展示和练习： 　a）用户子程序 　b）基于python的GUI开发 　c）用户子程序实例练习 3、UMAT整体理论： 　a）UMAT接口原理和使用方法； 　b）整体理论练习。 4、学员实际项目问题解答（课间）。
第一天下午	线弹性UMAT	1、线弹性UMAT理论 2、基于Fortran线弹性UMAT 　a）基于Fortran的线弹性UMAT开发演示； 　b）基于Fortran的线弹性UMAT开发练习； 3、基于Matlab线弹性UMAT 　a）基于Matlab的线弹性UMAT开发演示； 　b）基于Matlab的线弹性UMAT开发练习；
第二天上午	塑性UMAT	1、塑性UMAT实例演示； 2、塑性UMAT基础理论； 3、径向返回算法； 4、基于Matlab的塑性UMAT编写演示； 5、基于Matlab的塑性UMAT编写练习
第二天下午	非线性硬化UMAT和壳的UMAT	1、非线性硬化： 　a）非线性硬化及牛顿迭代法理论； 　b）基于Matlab的非线性硬化塑性材料UMAT编写演示； 　c）基于Matlab的非线性硬化塑性材料UMAT编写练习； 2、壳单元UMAT： 　a）壳单元刚度理论； 　b）壳单元UMAT编写演示； 　c）壳单元UMAT编写练习。 3、学员实际项目问题解答（课间）。
培训完	长期技术支持	

以下内容为付费内容，请购买后观看 **This content is paid, please purchase to watch**

2人购买　2 people purchased

¥**1**　**立即购买** **Buy Now**

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

非局部均值滤波和MATLAB程序详解视频算法及其保留图形细节应用…

正一算法程序 Zhengyi Algorithm Program　　　¥220 220 Yuan

车身设计系列视频之车身钣金正向设计实例教程…

京迪轩 Jing Di Xuan　　　¥1