

有限元理论基础及Abaqus内部实现方式研究系列10： 耦合约束的研究

Theoretical Foundation of Finite Element Method and Research on Internal Implementation of Abaqus Series 10: Study on Coupling Constraints



SnowWave02

[关](#)
[注](#) [Focus](#)
[s](#)更新于2021年3月3日 11:28 Updated
on March 3, 2021, 11:28 AM浏览:
5093 Views:
5093评论:
6 Comments: 6收藏:
12 Favorites: 12

有限元理论基础及Abaqus内部实现方式研究系列10： 耦合约束的研究的图1

(原创，转载请注明出处) (Original, please indicate the source for reproduction)

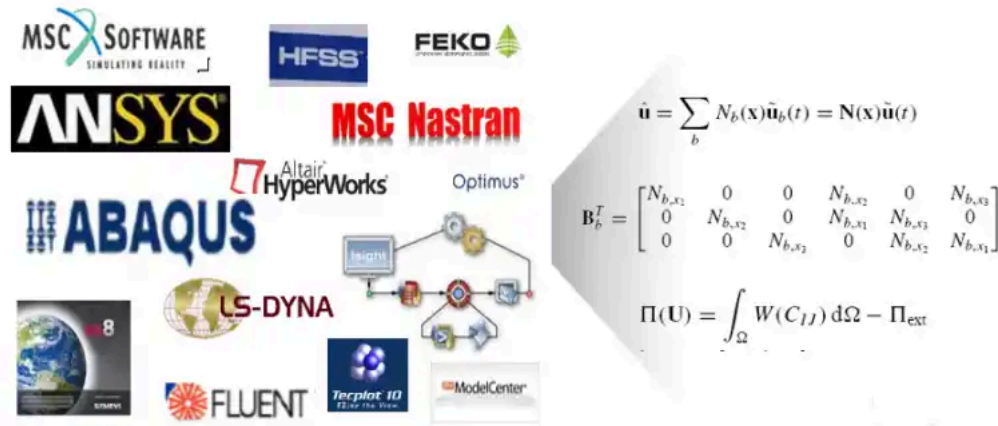
==概述== ==Overview==



有限元理论基础及Abaqus内部实现方式研究系列10： 耦合约束的研究的图2

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.



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。

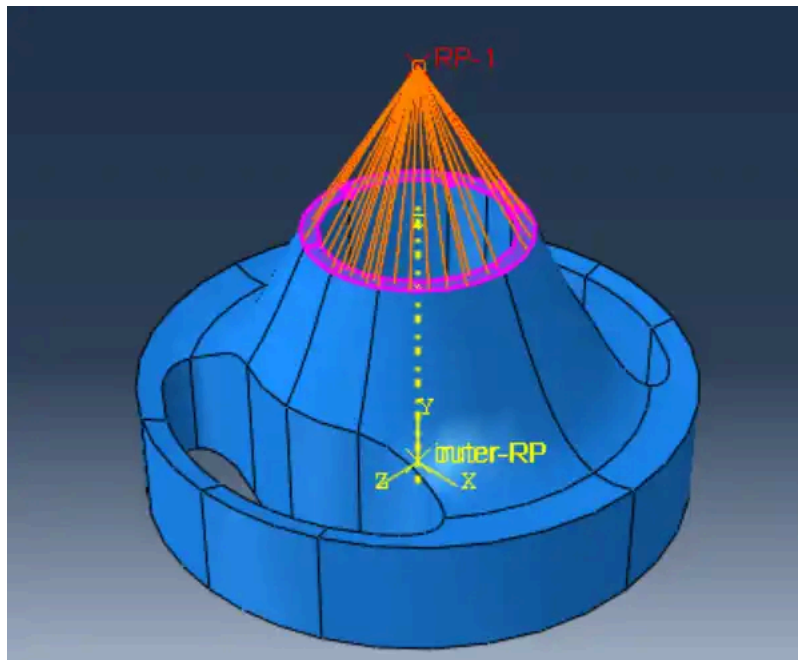
On one hand, we consult the theoretical manuals of various mainstream commercial software and guess the internal correction methods through extensive literature review. On the other hand, we program our own structural finite element solver and verify our guesses by comparing the results with those of 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10篇：耦合约束 (Coupling constraints) 的研究==

==Chapter 10: Study on Coupling Constraints (Coupling constraints)==



有限元理论基础及Abaqus内部实现方式研究系列10：耦合约束的研究的图6

耦合约束对应Nastran的MPC，是最常用的约束方式之一，用于定义一个表面集（Surface Set）内节点与控制节点位移自由度之间的相互关系，可以模拟节点的刚性连接或指定节点位移间的组合约束。

Coupling constraints correspond to MPC in Nastran and are one of the most commonly used constraint methods, used to define the relationship between the displacement degrees of freedom of nodes within a surface set and control nodes, which can simulate rigid connections between nodes or specify combined constraints on node displacements.

耦合约束常用于某些有限元模型要求特定自由度连接关系的场合，包括：

Coupling constraints are commonly used in situations where certain finite element models require specific relationships between degrees of freedom, including:

- 1、描述非常刚硬的结构元件，使用约束方程代替大刚度弹性单元能够使有限元模型更为合理；
1. Describing very rigid structural elements, using constraint equations instead of large stiffness elastic elements can make the finite element model more reasonable;
- 2、在不同类型的单元间传递载荷，如将壳单元的力偶传递到实体单元中（实体单元没有转动自由度）；
2. Transferring loads between different types of elements, such as transferring the moment of force from shell elements to solid elements (solid elements do not have rotational degrees of freedom);
- 3、定义节点间的刚性连接。 3. Defining rigid connections between nodes.

Abaqus中耦合约束分为运动耦合（Kinematic Coupling）和分布式耦合（Distributing Coupling），分别对应Nastran中的RBE2单元和RBE3单元，详见《[Abaqus Analysis User's Manual](#) Table 3.2.25-1》。

In Abaqus, coupling constraints are divided into kinematic coupling (Kinematic Coupling) and distributed coupling (Distributing Coupling), which correspond to the RBE2 and RBE3 elements in Nastran, respectively. See "Abaqus Analysis User's Manual Table 3.2.25-1" for details.

==演示视频== ==Demonstration Video==

该视频演示了iSolver中实现KCoupling的功能，证明iSolver结果和Abaqus完全一致：

This video demonstrates the implementation of the KCoupling function in iSolver, proving that the iSolver results are completely consistent with Abaqus:

<https://www.jishulink.com/college/video/c12884> 第6章节：3.1 载荷和边界-K-Coupling耦合约束

<https://www.jishulink.com/college/video/c12884> Chapter 6: 3.1 Load and Boundary - K-Coupling Coupling Constraint

==总结== ==Summary==

本文简单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，并通过两个简单算例加以验证。在有限元分析中，耦合约束应用极广，研究其原理有助于我们选择合理的约束方式，从而保证建模的准确性。不同商软对耦合约束的定义也不同，Abaqus/Nastran/Ansys的定义分别如下：

This article briefly introduces the definition and application of coupling constraints, specifically elaborates on the principles of motion coupling constraints and distributed coupling constraints in Abaqus, and verifies them through two simple examples. Coupling constraints are widely used in finite element analysis, studying their principles helps us choose reasonable constraint methods to ensure the accuracy of modeling. Different commercial software has different definitions of coupling constraints, the definitions of Abaqus/Nastran/Ansys are as follows:

项 次 number	Item Question	运 动 耦 合 约 束 Motion coupling constraint	分 布 耦 合 Distributed Coupling
1	Abaqus	K-Coupling	D-Coupling
2	Nastran	RBE2	RBE3
3	Ansys	CERIG	RBE3

注：对于非线性分析，Ansys采用MPC184单元来创建耦合约束。

Note: For nonlinear analysis, Ansys uses the MPC184 element to create coupling constraints.

如果有任何其它疑问，欢迎联系我们： If you have any other questions, please feel free to contact us:

snowwave02Fromwww.jishulink.com

email: snowwave02@qq.com


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10：耦合约束（Coupling constraints）的研究.pdf](#)

[Finite Element Theory and Abaqus Internal Implementation Research Series 10: Research on Coupling Constraints \(Coupling constraints\).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10：耦合约束的研究的图8

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。



技术邻

首页 Home

学院 College

直播 Live Streaming

问答 Q&A

悬赏 Bounty

会议

Conference

Conference



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>

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

17人购买 17 people purchased

收费内容为空，如果觉得文章对你有帮助，也可以打赏一下，谢谢支持

The paid content is empty. If you find the article helpful, you can also make a donation, thank you for your support

¥1 立即购买 Buy Now

推荐阅读 Recommended Reading

<p>Abaqus、iSolver与Nastran梁单元差异...</p> <p>SnowWave02</p> <p>免费</p>	<p>转子旋转的周期性模型-水冷电机散热仿真</p> <p>技术邻小李</p> <p>¥100</p>	<p>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</p> <p>正一算法程序 First Algorithm Program</p> <p>¥220</p>	<p>车身设计系列视频之车身钣金正向设计实例教程...</p> <p>京迪轩 Jinding 轩</p>
--------------------------------------------------------------------------	------------------------------------------------------------	--------------------------------------------------------------------------------------------------	-------------------------------------------------------------