

有限元理论基础及Abaqus内部实现方式研究系列9： 编写线性UMAT Step By Step

Theoretical Foundation of Finite Element Method and Research on Internal Implementation of Abaqus Series 9: Writing Linear UMAT Step by Step



SnowWave02

[关](#)
[注](#) [Focus](#)
[s](#)2019年5月7日 04:59 May 7,
2019 04:59浏览: 3567 Views:
3567评论:
3 Comment: 3收藏:
8 Favorites: 8

有限元理论基础及Abaqus内部实现方式研究系列9： 编写线性UMAT Step By Step的图1

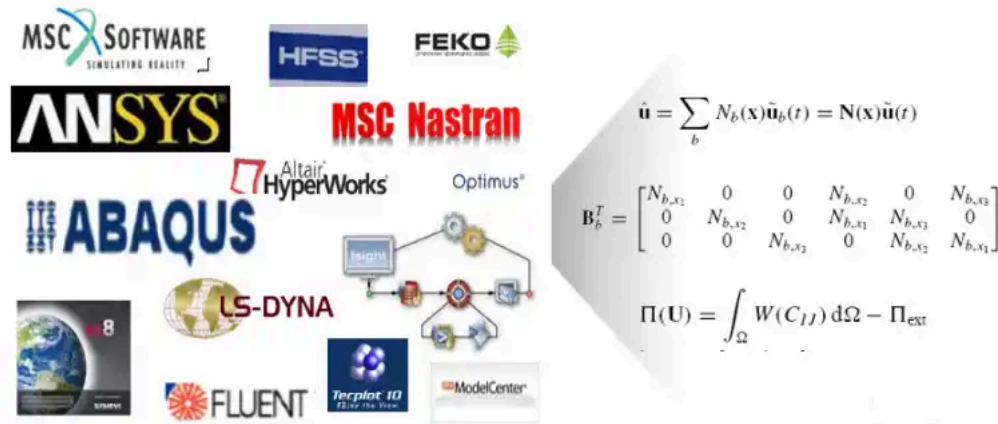
(原创，转载请注明出处) (Original, please indicate the source for reproduction)

==概述== ==Overview==

有限元理论基础及Abaqus内部实现方式研究系列9： 编写线性UMAT Step By Step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9篇：编写线性UMAT Step By Step==

== Article 9: Writing Linear UMAT Step by Step ==

有限元理论基础及Abaqus内部实现方式研究系列9： 编写线性UMAT Step By Step的图5

线性材料，即应力应变始终成正比关系的材料，常用于结构的线性静力分析，是各种常用分析的基础。而线性UMAT就是实现线性材料算法的接口，它的主要功能是计算单元的应力应变关系矩阵和单元应力，如下图所示。

Linear materials, where stress and strain are always proportional, are commonly used in structural linear static analysis and serve as the foundation for various common analyses. Linear UMAT is the interface for implementing linear material algorithms, mainly responsible for calculating the stress-strain relationship matrix and element stresses of the unit, as shown in the figure below.

Abaqus规定了UMAT的输入输出，下表就是一些关键变量的符号说明。

Abaqus specifies the input and output of UMAT, and the following table provides the symbol descriptions of some key variables.

变量类别 Variable Category	Variable	变量名称 Variable Name	变量说明 Variable Description
需要更新的变量 Variable to be updated		STRESS	输入为当前增量步开始之前的应力向量，在当前增量步内需要更新 The input is the stress vector before the current increment step, which needs to be updated within the current increment step
		DDSDDE	Jacobian矩阵，即应力应变关系矩阵 Jacobian matrix, i.e., the stress-strain relationship matrix
		STATEV	存储求解过程中的状态变量，用来传递状态变量，增量步结束时更新 Stores state variables during the solution process, used to pass state variables, and updated at the end of the increment step
传入变量 Input variable		STRAIN	当前增量步开始之前的总应变向量 Total strain vector before the start of the current increment step
		DSTRAIN	当前增量步内的应变增量 Strain increment within the current increment step
		TIME	当前分析步的时间步和总时间步 Time step and total time step of the current analysis step
		DTIME	增量步大小 Incremental step size
		NDI	直接应力分量维数 Dimension of direct stress components
		NSHR	剪切应力分量维数 Dimension of shear stress components
		NTENS	应力或应变分量的维数，等于NDI+NSHR Dimension of stress or strain components, equal to NDI + NSHR
		NSTATV	状态变量维数 Dimension of state variables
		PROPS	自定义材料常数 Custom material constants
		NPROPS	自定义材料常数的个数 Number of custom material constants
		NPT	积分点序号 Integral point number
		KSTEP	当前分析步序号 Current analysis step number
		KINC	当前增量步序号 Current increment step number

==演示视频== ==Demonstration Video==

Abaqus用户子程序UMat详解与开发工具：章节4

Abaqus User Subroutine UMat Detailed Explanation and Development Tools: Chapter 4

<https://www.jishulink.com/college/video/c13034?chapter=4>


==总结== ==Summary==

本文首先简单介绍了线性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，最后采用同一个算例对Abaqus自带材料和用户编写的线性UMAT两者分析结果进行对比，从而证明基于Matlab的线性UMAT的正确性。

This article briefly introduces the interface functions and meanings of key interface variables of linear UMAT, and then details the development steps of a linear UMAT based on Matlab through a simple cube static analysis example. Finally, using the same example, the analysis results of Abaqus' built-in materials and user-written linear UMAT are compared to prove the correctness of the linear UMAT based on Matlab.

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9: 编写线性UMAT Step By Step.pdf](#)

 Series 9: Theoretical Foundation of Finite Element Method and Internal Implementation of Abaqus: Writing Linear UMAT Step by Step.pdf

以往的系列文章： Previous series articles:



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



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在非线形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>

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
--	---	--	---