有限元理论基础及Abaqus内部实现方式研究系列29： 几何非线性的T.L.和U.L.转换关系

Theoretical Foundation of Finite Element Method and Internal Implementation of Abaqus Series 29: Transformation Relationship between T.L. and U.L. under Geometric Nonlinearity

SnowWave02

2020年10月28日 14:28 October 28, 2020 14:28

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

有限元理论基础及Abaqus内部实现方式研究系列29： 几何非线性的T.L.和U.L.转换关系的图1

有限元理论基础及Abaqus内部实现方式研究系列29： 几何非线性的T.L.和U.L.转换关系的图2

1 概述 1 Overview

有限元理论基础及Abaqus内部实现方式研究系列29： 几何非线性的T.L.和U.L.转换关系的图3

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，通过

This series of articles studies the mature finite element theory foundation and its implementation in commercial finite element software, through

（1） 基础理论 (1) Basic Theory

（2） 商软操作 (2) Commercial Software Operation

（3） 自编程序 (3) Self-written program

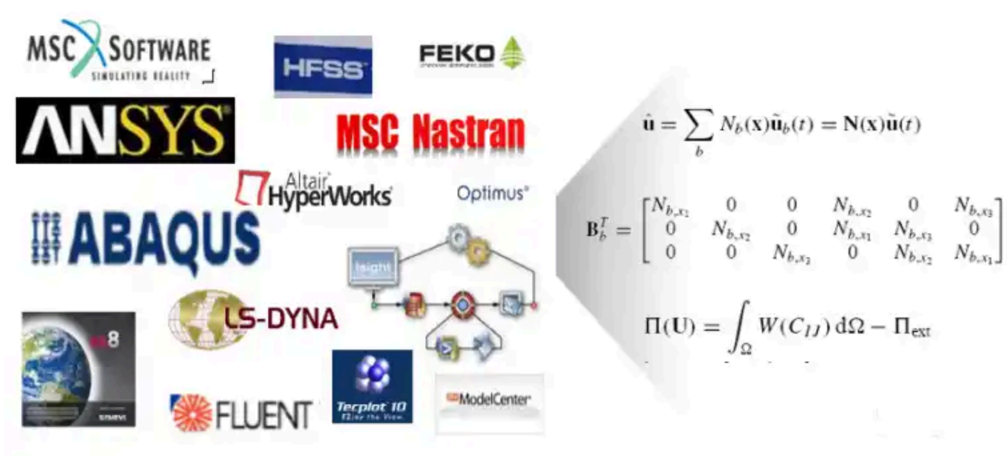
三者结合的方式将复杂繁琐的结构有限元理论通过简单直观的方式展现出来，同时深层次的学习有限元理论和商业软件的内部实现原理。

The combination of the three methods presents the complex and cumbersome structural finite element theory in a simple and intuitive way, while also deeply studying the internal implementation principles of finite element theory and commercial software.

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。

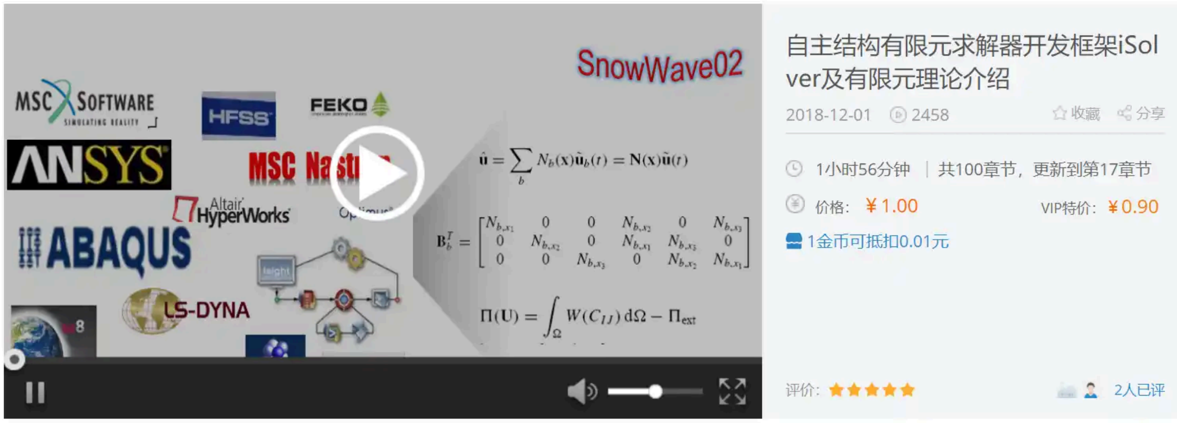
The theoretical development of finite elements has matured over decades, and commercial finite element software also adopts these mature finite element theories. However, in the actual application process, commercial CAE software will make corresponding corrections on the basis of traditional theories 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 a black box to the outside, and users can only guess the 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.

通用结构有限元求解器iSolver介绍视频： Introduction to the General Structural Finite Element Solver iSolver Video:



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

==第29篇：几何非线性的T.L.和U.L.转换关系==

==Article 29: Transformation Relationship between T.L. and U.L. under Geometric Nonlinearity==

最近很多人问我们T.L.和U.L是否都可以用于几何大变形问题，我们理解应该都可以，因为虽然公式不同，但他们所用的物理量是可以相互转换的，最终还是表达的同一套物理含义。本章将就最基本的应变介绍一下他们的转换关系的理论，同时采用Abaqus的一个简单的算例来验证他们的转换关系。

Many people have recently asked us whether T.L. and U.L. can both be used for geometrically nonlinear problems. We understand that they should both be applicable, as although the formulas are different, the physical quantities they use can be interconverted, ultimately expressing the same set of physical meanings. This chapter will introduce the theoretical conversion relationship of the basic strain between them, and also use a simple example from Abaqus to verify their conversion relationship.

无论是T.L.还是U.L.，同一个物理量只是表达式不同，但在同一时刻下的数值肯定是一样的，应变能也不例外，所以：

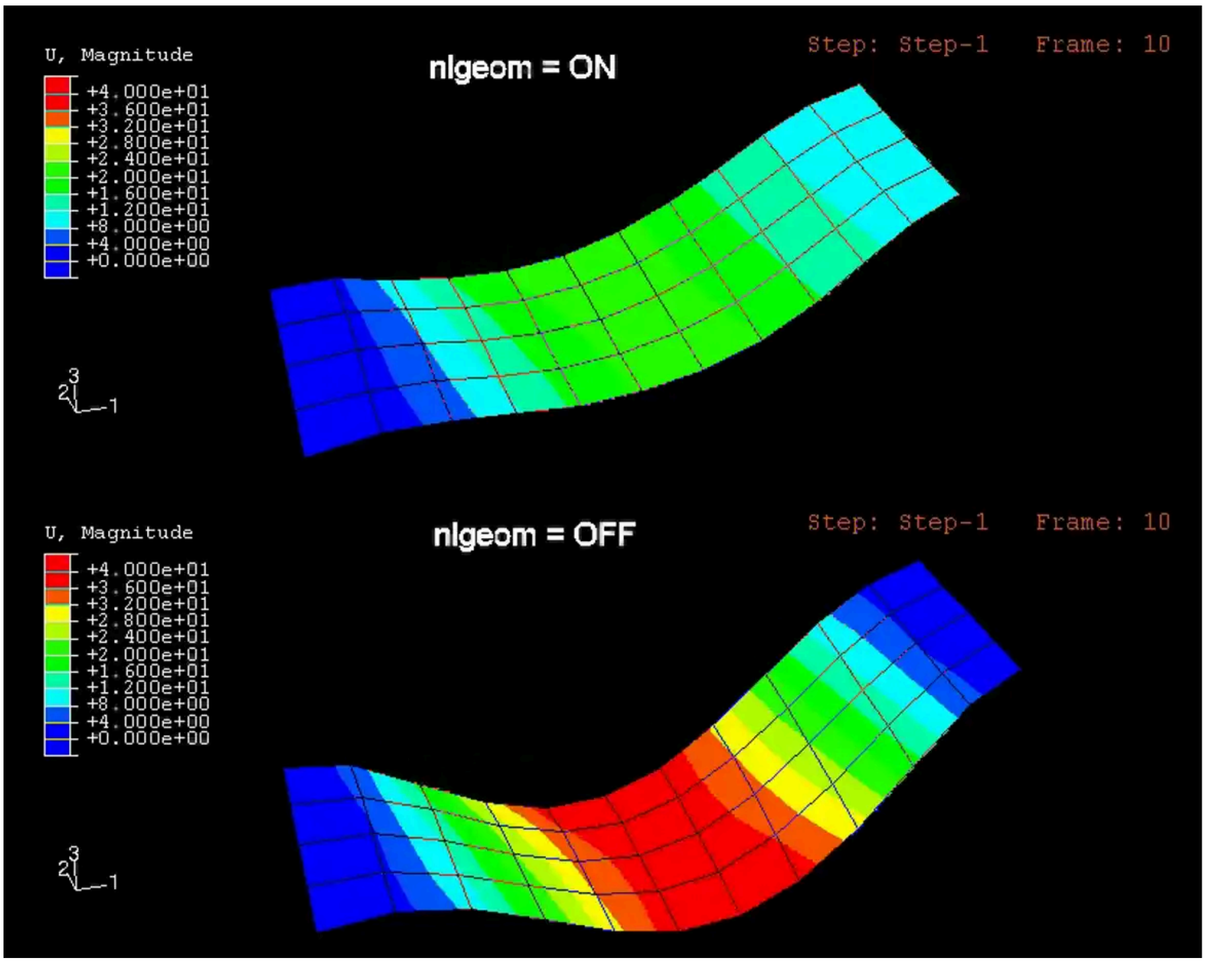
Whether it is T.L. or U.L., the same physical quantity is just expressed differently, and the values at the same moment must be the same, and this applies to strain energy as well. Therefore:

$$\int_{{}^{ndt}_{ndt}V} {}^{(n+1)dt}_{ndt}\sigma * {}^{(n+1)dt}_{ndt}\delta\varepsilon * {}^{(n+1)dt}_{ndt}J \, d{}^{ndt}_{ndt}V = \int_{{}^{0}_{0}V} {}^{(n+1)dt}_{0}S * {}^{(n+1)dt}_{0}\delta\varepsilon^{G} \, d{}^{0}_{0}V$$

这个等式说明T.L.和U.L.是可以通过各自应力和应变量的转换而相互等价，最终都能得到正确解。区别在于选取的应变和应力度量和试验怎么方便的对应，有限元中应变度量本身就是人为选取的，由位移总能得到各种应变的度量结果，具体可看前面**系列文章18：几何非线性的应变**。但材料的各种试验参数是真实的，必须通过试验得到，而试验中可测的量是有限的，有限元计算的应变等人为度量量最终还是要和试验对比，从这方面考虑也是Abaqus内

部主要采用U.L.的原因。

This equation shows that T.L. and U.L. can be made equivalent through the conversion of their respective stress and strain quantities, and both can ultimately yield the correct solution. The difference lies in how the strain and strain intensity measures are chosen to correspond conveniently with the experiment. In finite element analysis, the strain measure itself is arbitrarily selected, and various strain measures can be obtained from displacements. For details, see the previous series article 18: Strain in Geometric Nonlinearity. However, various material test parameters are real and must be obtained through experiments. The quantities measurable in the experiment are limited, and the strain measures calculated by finite element analysis, as arbitrary measures, ultimately need to be compared with experiments. From this perspective, it is also the main reason why Abaqus internally mainly adopts U.L.



1.1 T.L.和U.L.的物理量的转换关系理论

1.1 Theoretical relationship between the physical quantities of T.L. and U.L.

T.L.和U.L.的转换关系主要包括三个物理量的转换：

The conversion relationship between T.L. and U.L. mainly includes the conversion of three physical quantities:

1.1.1 应变的转换 1.1.1 Conversion of Strain

（1）T.L.采用的应变是Green应变。 (1) The strain adopted by T.L. is Green strain.

（2）U.L.中，在本**系列文章27：Abaqus内部计算和显示的应变**中，我们提到了Abaqus大变形下后处理显示的都是真实应变，但实际计算时采用了对数应变和变形率积分两种应变度量方式，而对数应变和T.L.中的Green应变没有直接关系，只有变形率积分度量方式和Green应变有直接关系。变形率积分应变和Green应变的关系如下：

(2) In the 27th article of this series: "Abaqus Internal Calculation and Display of Strain," we mentioned that under Abaqus large deformation, the post-processing display shows true strain, but the actual calculation uses two strain measures: logarithmic strain and deformation rate integral. There is no direct relationship between logarithmic strain and Green strain in T.L., but there is a direct relationship between the deformation rate integral measure and Green strain. The relationship between deformation rate integral strain and Green strain is as follows:

$$D = F' \dot{\varepsilon}^G F$$

1.1.2 应力的转换 1.1.2 Stress Transformation

（1） 在T.L.中，与Green应变形成功共轭关系的是2nd PK力S。

(1) In the Total Lagrange (T.L.) formulation, the 2nd Piola-Kirchhoff (PK) stress S is conjugate to the Green strain in terms of work formation.

（2） U.L.中，与变形率积分的应变度量形成功共轭关系的是真实应力，也就是Cauchy力。它们的关系如下：

(2) In the Updated Lagrange (U.L.) formulation, the true stress, which is also known as the Cauchy stress, is conjugate to the strain measure integrated over deformation rate. Their relationship is as follows:

$$\sigma = \frac{1}{J} F S F'$$

1.1.3 本构关系的转换 1.1.3 Transformation of Constitutive Relations

（1）T.L.中，虚功原理需要对应变能求导。 (1) In T.L., the principle of virtual work requires differentiation of strain energy.

$$K = dW = \int_{v_0} \left(dS * \delta\varepsilon^G + S * d\delta\varepsilon^G \right) dV_0$$

前面部分可以展开成应力应变关系矩阵和应变增量的积，对次弹性材料：

The preceding part can be expanded into the product of the stress-strain relationship matrix and the strain increment, for hyperelastic materials:

$$dS * \delta\varepsilon^G = d\varepsilon^G * C1 * \delta\varepsilon^G$$

（2）U.L.中，同样，对应变能求导后，前面部分可以展开成应力应变关系矩阵和应变增量的积，且相对初始构型，对次弹性材料：

(2) In the U.L., similarly, after taking the derivative of the strain energy, the preceding part can be expanded into the product of the stress-strain relationship matrix and the strain increment, and for hyperelastic materials relative to the initial configuration:

$$d\sigma * \delta D = dD * C2 * \delta D$$

在初始构型下，上述两者都表示初始构型下的应变能增量，那么得到

Under the initial configuration, both of these represent the strain energy increment under the initial configuration, so we obtain:

$$d\varepsilon^G * C1 * \delta\varepsilon^G = J * dD * C2 * \delta D$$

将应变的关系表达式代入，同时约去应变的导数，得到本构关系的转换如下，显然与变形梯度F有关：

Substituting the strain relationship expression and canceling out the derivatives of strain, the transformation of the constitutive relationship is obtained as follows, which is obviously related to the deformation gradient F:

$$C2_{ijkl} = \frac{1}{J} F_{ir} F_{js} F_{km} F_{ln} C1_{rsmn}$$

1.2 验证模型选取原则 1.2 Principle of Model Selection for Verification

为了验证T.L.和U.L.的等价转换，我们需要在软件或者编程中分别按两种描述实现同一个物理模型，理论上按照上面的三个物理量的转换，就能将T.L.描述完全转换为U.L.描述，但在Abaqus中，并不是能直接找到同一物理对象的两种描述方法，我们的选取原则如下：

In order to verify the equivalence of the T.L. and U.L. conversion, we need to implement the same physical model in software or programming in two different descriptions. Theoretically, according to the conversion of the above three physical quantities, the T.L. description can be completely converted to the U.L. description. However, in Abaqus, it is not possible to directly find two descriptions of the same physical object. Our selection principle is as follows:

（1）对T.L.描述，在本**系列文章19：Abaqus几何非线性的设置和后台**中，我们提到了Abaqus中只有梁壳模型由小应变单元，此时虚功原理为T.L.描述方式。所以我们采用壳单元研究T.L.的结果。

(1) For the T.L. description, in our series article 19: Abaqus Geometric Nonlinearity Settings and Background, we mentioned that in Abaqus, only beam-shell models use small strain elements, and the virtual work principle is the T.L. description method. Therefore, we adopt shell elements to study the results of T.L.

（2）对U.L.描述，Abaqus中体和壳采用的应变分别是变形率积分和真实应变，由前所述，我们仅研究体单元的U.L.结果。

(2) For the U.L. description, Abaqus uses deformation rate integration and true strain for the strain of solids and shells, respectively. As mentioned before, we only study the U.L. results of solid elements.

（3）在T.L.中我们采用最简单的线弹性材料来描述本构关系，正常来说，在U.L.中也应该采用上面2.1.3对应的U.L.的线弹性本构关系，可惜的很，这种线弹性关系在Abaqus中自带材料不存在。Abaqus在几何非线性的U.L.描述中，对次弹性材料，Abaqus认为塑性大变形，弹性小变形，此时Abaqus采用的本构关系矩阵和小应变的完全一样，也就是与变形梯度F无关，而不是2.1.3对应的本构关系。因此，我们只能采用UMAT自定义材料来设置U.L.的线弹性本构关系。我们将在UMAT中实现St.Venant-Kirchhoff材料本构模型，此模型下，U.L.的C2矩阵可以用B=F*F’简单的表示为：

(3) In T.L., we use the simplest linear elastic material to describe the constitutive relationship. Normally, in U.L., the linear elastic constitutive relationship corresponding to 2.1.3 should also be used. Unfortunately, this linear elastic relationship does not exist in the built-in materials of Abaqus. In the geometric nonlinear U.L. description of Abaqus, for hyperelastic materials, Abaqus considers large plastic deformation and small elastic deformation. At this time, the constitutive relationship matrix used by Abaqus is the same as that of small strain, which is independent of the deformation gradient F, rather than the constitutive relationship corresponding to 2.1.3. Therefore, we can only use UMAT to define a custom material to set the linear elastic constitutive relationship of U.L. We will implement the St. Venant-Kirchhoff material constitutive model in UMAT, under which the C2 matrix of U.L. can be simply expressed as B = F * F'.

$$C2_{ijkl} = \frac{1}{J}\left[\lambda B_{ij}B_{kl} + \mu(B_{ik}B_{jl} + B_{il}B_{jk})\right]$$

同时，由于Abaqus自带的Fortran编写复杂，我们在iSolver中采用Matlab的UMAT实现St.Venant-Kirchhoff材料。

Meanwhile, due to the complexity of Fortran programming in Abaqus, we adopt Matlab's UMAT implementation for the St. Venant-Kirchhoff material in iSolver.

1.3 Abaqus和iSolver的算例证明 1.3 Example Verification of Abaqus and iSolver

1.3.1 模型描述 1.3.1 Model Description

取一个长方体，参数如下： Take a rectangular prism with the following parameters:

尺寸：5X1X0.1 Size: 5X1X0.1

材料：Young’s Modulus 200， Poisson Ratio 0。

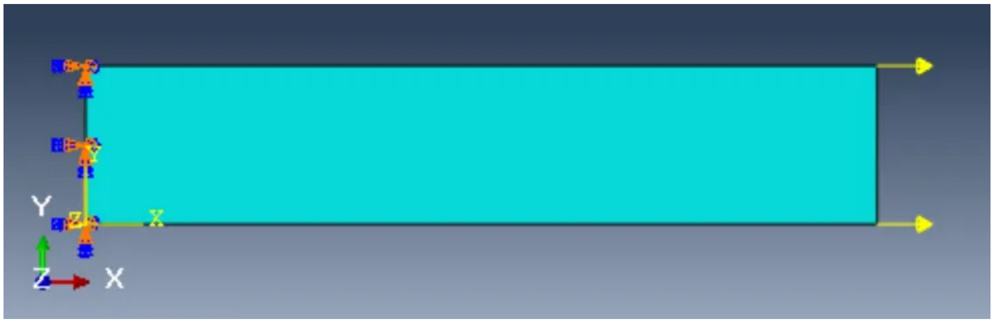
Material: Young's Modulus 200, Poisson Ratio 0.

所有节点的yz方向约束，同时左侧节点固支。 All nodes have constraints in the yz direction, and the nodes on the left are fixed.

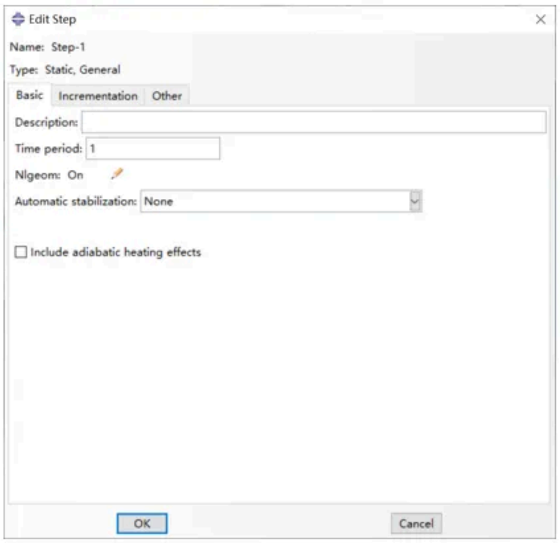
右侧节点只有x方向加力，力的合力为5。 The nodes on the right only have a force applied in the x-direction, with a resultant force of 5.

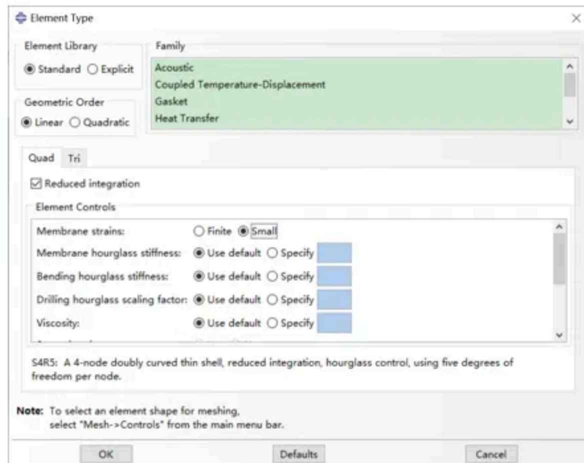
1.3.2 T.L.描述 1.3.2 T.L. Description

采用壳单元建模，划分为一个单元： Modeling using shell elements, divided into one element:

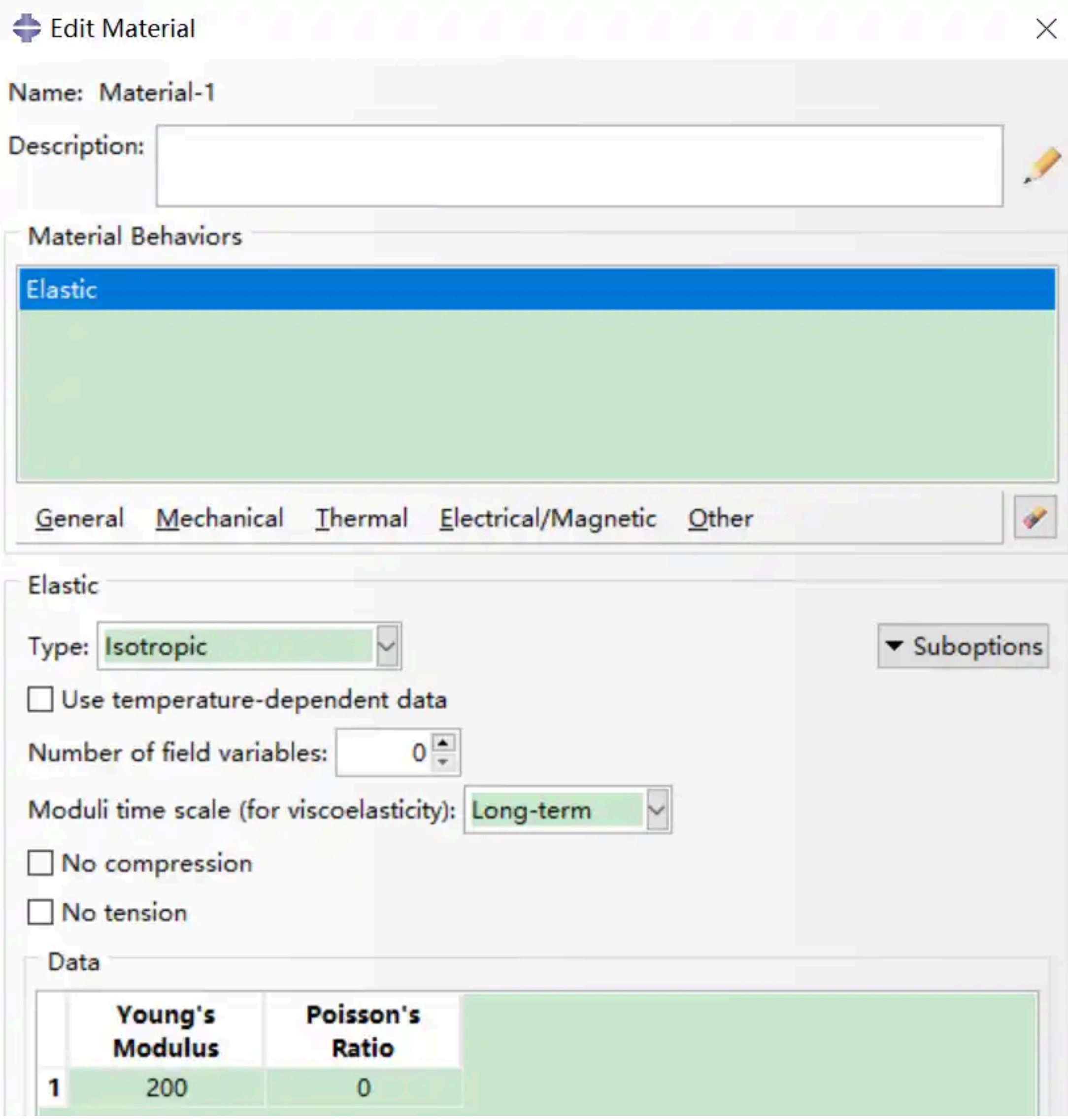


打开几何非线性，且用S4R5单元： Open geometric nonlinearity and use the S4R5 element:



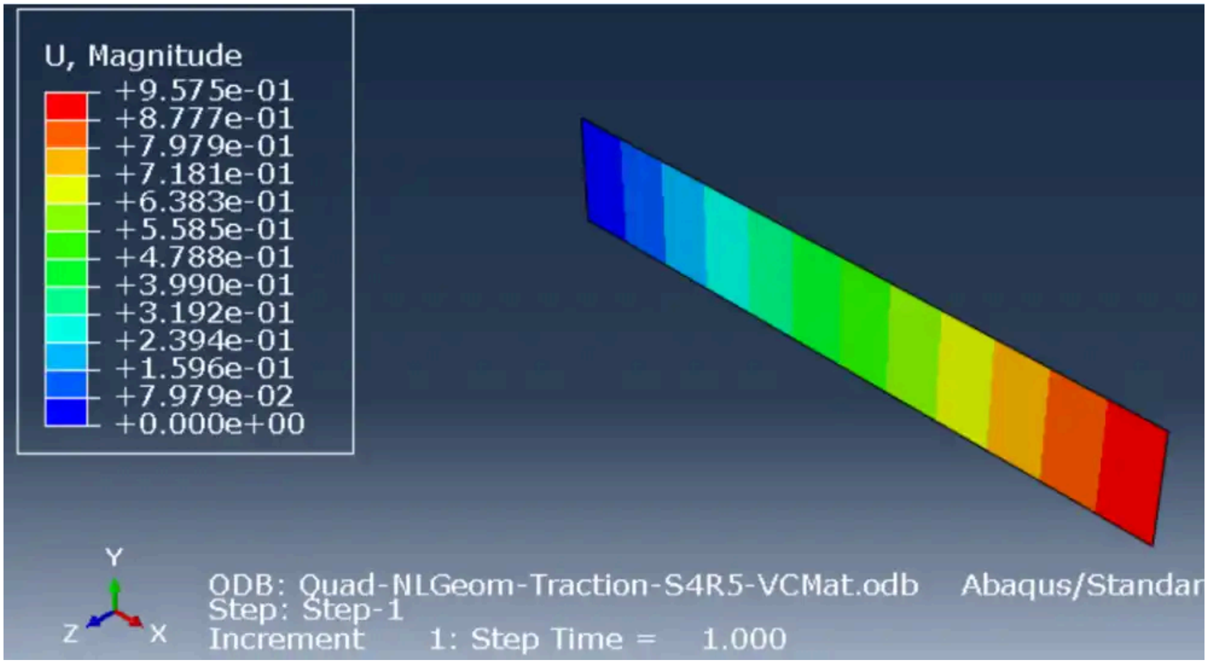


材料为Abaqus自带材料： Material is the Abaqus built-in material:



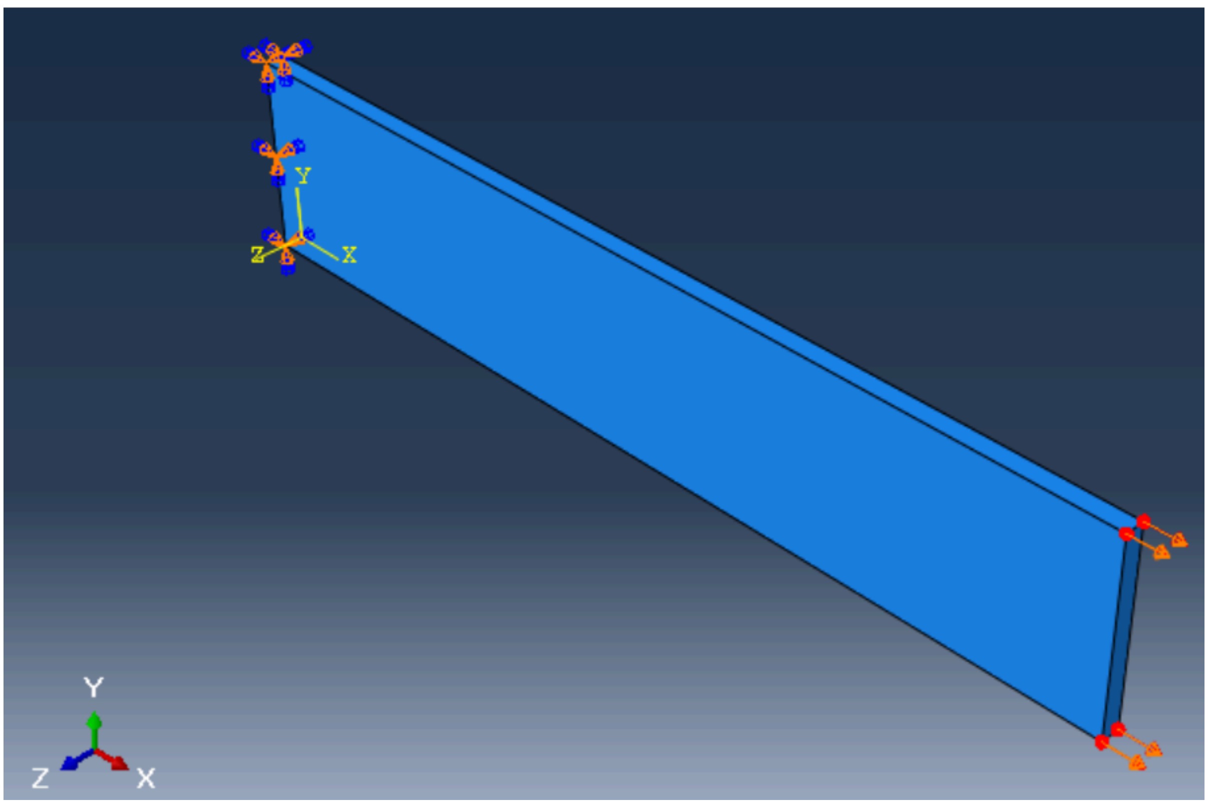
右端加集中载荷2.5，Abaqus计算得到最终的位移为9.575e-1：

A concentrated load of 2.5 is applied at the right end, and Abaqus calculates the final displacement to be 9.575e-1.



1.3.3 U.L.描述 1.3.3 U.L. description

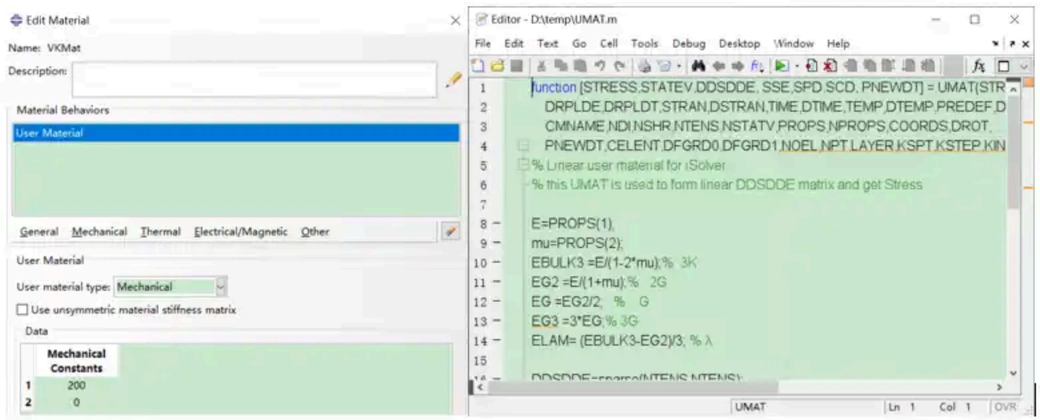
采用体单元建模，划分为一个单元： Modeling using a solid element, divided into one element:



打开几何非线性，且用C3D8R单元。 Open geometric nonlinearity and use the C3D8R element.

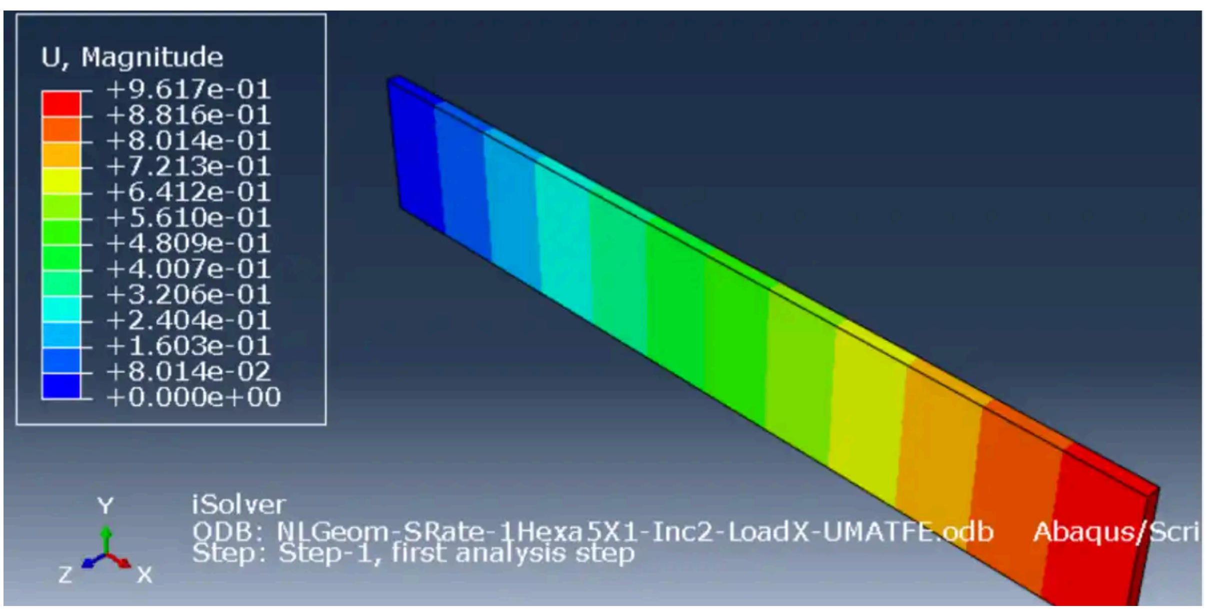
材料为UMAT自定义材料，在CAE中设置两个参数200和0，在iSolver的UMAT.m中实现St.Venant-Kirchhoff材料。

The material is a UMAT custom material, with two parameters set to 200 and 0 in the CAE, and the St. Venant-Kirchhoff material is implemented in the UMAT.m of iSolver.



右端加集中载荷1.25（合力还是1），调用iSolver计算得到最终的位移为9.617e-1，和T.L.的结果误差为0.44%，基本一致。

The concentrated load on the right end is 1.25 (whether it is the resultant force or 1), and the final displacement obtained by calling iSolver is 9.617e-1, with an error of 0.44% compared to the T.L. result, which is basically consistent.



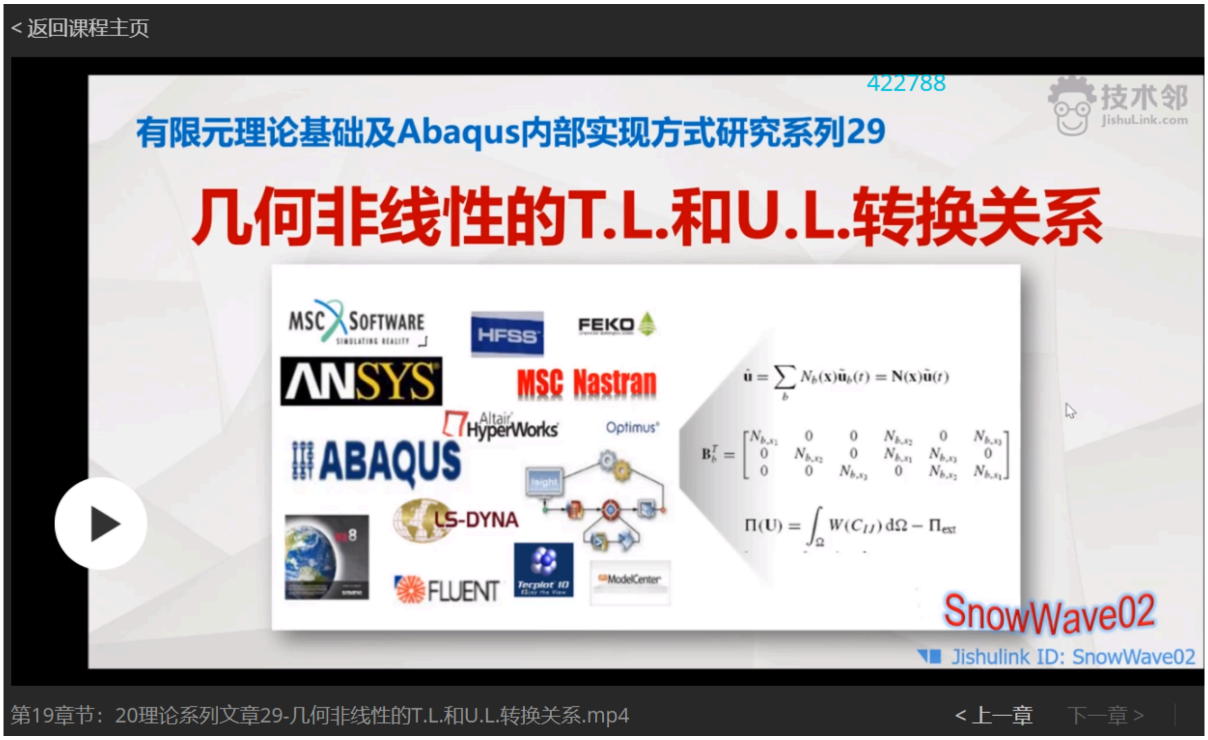
1.4 视频讲解和操作验证演示 1.4 Video Explanation and Operation Verification Demonstration

如果觉得上面的文字太复杂，也可以看一下视频的简要讲解，包括基于Abaqus和iSolver的操作验证，地址如下：

If you find the above text too complex, you can also watch the brief explanation in the video, including the operation verification based on Abaqus and iSolver. The address is as follows:

https://www.jishulink.com/college/video/c12884 20理论系列文章29-几何非线性的T.L.和U.L.转换关系

https://www.jishulink.com/college/video/c12884 20 Theory Series Article 29 - Transformation Relationship between T.L. and U.L. in Geometric Nonlinearity



==总结== ==Summary==

至此，我们可以说只要三个物理量实现转换，整个T.L.的结果就和U.L.的结果完全一致。而如果其中有一项没实现转换，那么结果可能完全不一致，譬如U.L.中没有将本构关系转换，而依然采用弹性的本构关系，也就是采用Abaqus自带的弹性材料，得到U=1.427，和T.L.的结果有很大差异：

Up to this point, we can say that as long as the conversion of three physical quantities is achieved, the entire T.L. result is completely consistent with the U.L. result. However, if one of them fails to be converted, the results may be completely different, for example, in U.L., the constitutive relationship is not converted and still uses the elastic constitutive relationship, that is, using the elastic material built-in Abaqus, obtaining U=1.427, which has a significant difference from the T.L. result:

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

有限元理论基础及Abaqus内部实现方式研究系列29： 几何非线性的T.L.和U.L.转换关系的图46

有限元理论基础及Abaqus内部实现方式研究系列29：几何非线性的T.L.和U.L.转换关系的图47

有限元理论基础及Abaqus内部实现方式研究系列29：几何非线性的T.L.和U.L.转换关系的图48

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29：几何非线性的T.L.和U.L.转换关系的图49

有限元理论基础及Abaqus内部实现方式研究系列29：几何非线性的T.L.和U.L.转换关系的图50

1.6.1 ========第一阶段========

1.6.1 ========First Phase========

第一篇：**S4壳单元刚度矩阵研究。** First article: Research on the stiffness matrix of S4 shell elements.

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

第二篇：**S4壳单元质量矩阵研究。** The Second Article: Research on the Mass Matrix of S4 Shell 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。** The Third Article: Shear Locking and Shear Band Control of S4 Shell Elements.

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

第四篇：**非线性问题的求解。** The Fourth Article: Solution of Nonlinear Problems.

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

第五篇：**单元正确性验证。** The Fifth Article: Element Accuracy Verification.

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

第六篇：**General梁单元的刚度矩阵。** Sixth Article: Stiffness Matrix of General Beam Elements.

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

第七篇：**C3D8六面体单元的刚度矩阵。** Seventh Article: Stiffness Matrix of C3D8 Hexahedral Elements.

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

第八篇：**UMAT用户子程序开发步骤。** Eighth Article: Steps for Developing UMAT User Subroutines.

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

第九篇：**编写线性UMAT Step By Step。**

Ninth Article: Writing Linear UMAT Step By Step.

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

第十篇：**耦合约束（Coupling constraints）的研究。**

The tenth article: Research on coupling constraints.

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

有限元理论基础及Abaqus内部实现方式研究系列29：几何非线性的T.L.和U.L.转换关系的图51

有限元理论基础及Abaqus内部实现方式研究系列29：几何非线性的T.L.和U.L.转换关系的图52

1.6.2 ========第二阶段========

1.6.2 ========Second Stage========

第十一篇：**自主CAE开发实战经验第一阶段总结。** The eleventh article: Summary of the first phase of independent CAE development experience.

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

第十二篇：**几何梁单元的刚度矩阵。** The twelfth article: Stiffness matrix of the geometric beam element.

http://www.jishulink.com/content/post/534362

第十三篇：**显式和隐式的区别。** The Thirteenth Article: The Difference Between Explicit and Implicit.

http://www.jishulink.com/content/post/537154

第十四篇：**壳的应力方向。** The Fourteenth Article: Stress Direction of Shells.

https://www.jishulink.com/content/post/1189260

第十五篇：**壳的剪切应力。** The Fifteenth Article: Shear Stress of Shells.

https://www.jishulink.com/content/post/1191641

第十六篇：**Part、Instance与Assembly。**

Chapter 16: Part, Instance, and Assembly.

https://www.jishulink.com/content/post/1195061

第十七篇：**几何非线性的物理含义。** The 17th article: The physical meaning of geometric nonlinearity.

https://www.jishulink.com/content/post/1198459

第十八篇：**几何非线性的应变。** The 18th article: Strain of geometric nonlinearity.

https://www.jishulink.com/content/post/1201375

第十九篇：**Abaqus几何非线性的设置和后台。** The 19th article: Settings and background of Abaqus geometric nonlinearity.

http://www.jishulink.com/content/post/1203064

第二十篇：**UEL用户子程序开发步骤。** The 20th article: Steps for developing UEL user subroutines.

https://www.jishulink.com/content/post/1204261

有限元理论基础及Abaqus内部实现方式研究系列29：几何非线性的T.L.和U.L.转换关系的图53
有限元理论基础及Abaqus内部实现方式研究系列29：几何非线性的T.L.和U.L.转换关系的图54

1.6.3 ========第三阶段========

1.6.3 ========Third Phase========

第二十一篇：**自主CAE开发实战经验第二阶段总结。** Chapter 21: Summary of the Second Stage of Autonomous CAE Development Practical Experience.

https://www.jishulink.com/content/post/1204970

第二十二篇：**几何非线性的刚度矩阵求解。** The 22nd article: Solution of the stiffness matrix for geometric nonlinearity.

http://www.jishulink.com/content/post/1254435

第二十三篇：

有限元理论基础及Abaqus内部实现方式研究系列29：几何非线性的T.L.和U.L.转换关系的图55

编写简单面内拉伸问题UEL Step By Step。

The 23rd article: Step By Step guide to writing a simple in-plane tensile problem UEL.

http://www.jishulink.com/content/post/1256835

第二十四篇：

有限元理论基础及Abaqus内部实现方式研究系列29：几何非线性的T.L.和U.L.转换关系的图56

显式求解Step By Step。 The 24th article: Step By Step explicit solution.

https://www.jishulink.com/content/post/1261165

第二十五篇：**显式分析的稳定时间增量。** The 25th article: Stability time increment for explicit analysis.

http://www.jishulink.com/content/post/1263601

第二十六篇：**编写线性VUMAT Step By Step。**

第二十六篇：Step by Step Guide to Writing Linear VUMAT.

https://www.jishulink.com/content/post/1266640

第二十七篇：**Abaqus内部计算和显示的应变。** 第二十七篇：Strain Calculation and Display in Abaqus Internal.

https://www.jishulink.com/content/post/1273788

第二十八篇：**几何非线性的T.L.和U.L.描述方法**

Chapter 28: Description Methods for Geometric Nonlinearity T.L. and U.L.

https://www.jishulink.com/content/post/1282956