有限元理论基础及Abaqus内部实现方式研究系列18： 几何非线性的应变

Theoretical Foundation of Finite Element Method and Internal Implementation of Abaqus Series 18: Geometric Nonlinear Strain

SnowWave02 关注 Focus 2020年4月1日 15:37 April 1, 2020 15:37 浏览：3413 Views: 3413 评论：2 Comments: 2 收藏：2 Favorites: 2

（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18： 几何非线性的应变的图1
有限元理论基础及Abaqus内部实现方式研究系列18： 几何非线性的应变的图2
==概述== ==Overview==

有限元理论基础及Abaqus内部实现方式研究系列18： 几何非线性的应变的图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。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.

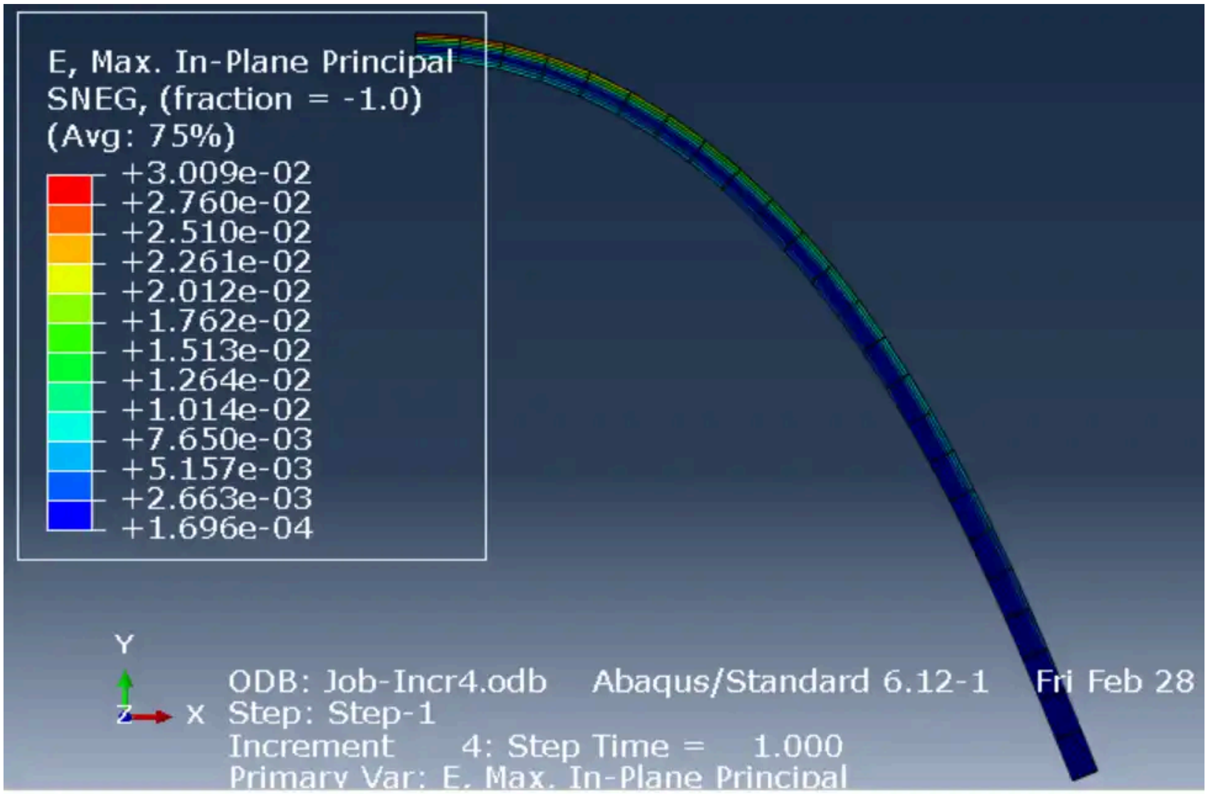
iSolver介绍视频： iSolver Introduction Video:

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

==第18篇：几何非线性的应变 == ==The 18th Article: Strain in Geometric Nonlinearity==

上一章我们介绍了几何非线性的物理含义，这章接着介绍几何非线性中重要的物理量：应变。物体在受到外力作用下会产生一定的变形，变形的程度称应变。理解应变的物理含义对有限元结果的判别和试验的对应有重要意义。线性情况下的应变非常容易计算，但如果存在几何非线性，就不是所有人都知道Abaqus等商软结果中的应变E代表的是什么了。本章首先从位移、变形和应变的区别说起，然后再具体介绍一下几何非线性下的应变是如何度量的。

In the previous chapter, we introduced the physical meaning of geometric nonlinearity. This chapter continues to introduce an important physical quantity in geometric nonlinearity: strain. When an object is subjected to external forces, it will undergo a certain deformation, and the degree of deformation is called strain. Understanding the physical meaning of strain is of great significance for the judgment of finite element results and the correspondence with experiments. In linear cases, strain is very easy to calculate, but if there is geometric nonlinearity, not everyone knows what the strain E represents in the results of commercial software like Abaqus. This chapter first discusses the differences between displacement, deformation, and strain, and then specifically introduces how strain is measured under geometric nonlinearity.

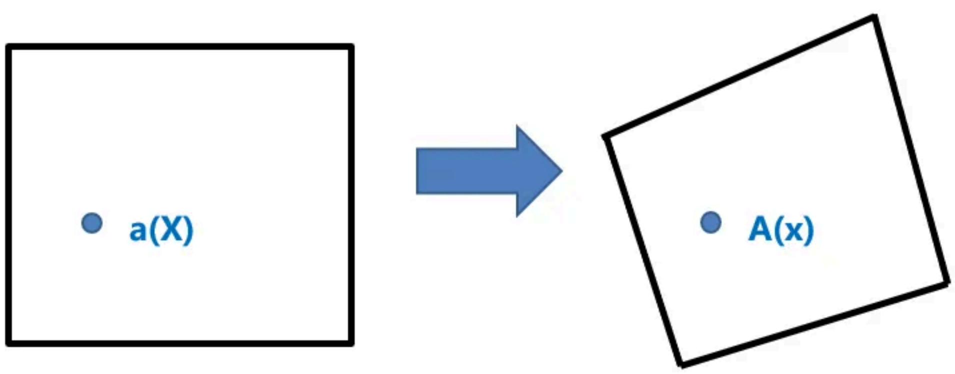


有限元理论基础及Abaqus内部实现方式研究系列18： 几何非线性的应变的图8

1.1 位移 Displacement 1.1 Displacement

一个物体从初始状态C0受到外部载荷运动到另一个状态C，由于材料不能凭空消失，那么C0下的任意的一个物质点a（设坐标为X）必然在C下有个对应点A（设坐标为x）。位移表示两个位置坐标相减，为u=x-X。

A body moves from its initial state C0 to another state C due to external loads, since the material cannot disappear arbitrarily, then any material point a (assumed to be at coordinates X) under C0 must have a corresponding point A (assumed to be at coordinates x) under C. Displacement represents the subtraction of two position coordinates, which is u = x - X.



有限元理论基础及Abaqus内部实现方式研究系列18： 几何非线性的应变的图11

1.2 变形 Deformation 1.2 Deformation

位移无法表示出变形情况，譬如大黄蜂变形为汽车时，身体360度转动，位移很大，但没有任何的变形。

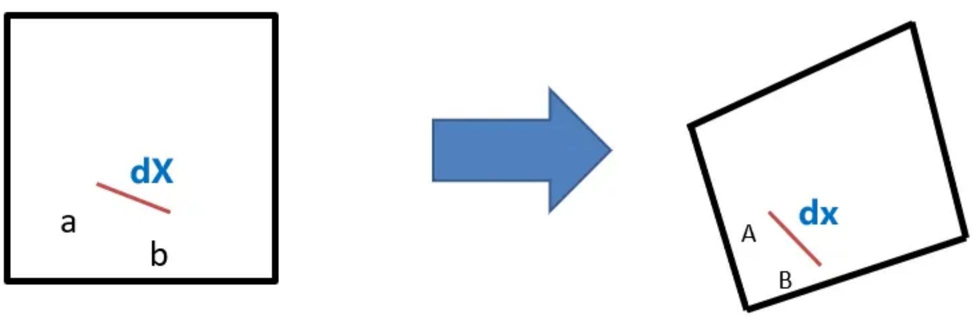
Displacement cannot represent the deformation situation, for example, when a bumblebee deforms into a car, its body rotates 360 degrees, and the displacement is large, but there is no deformation.

变形必须要由两个点之间材料纤维的拉伸才能决定。对初始构想C0任意的两点a、b，变形后设为为AB，那么AB矢量和ab之间的关系可以由

The deformation must be determined by the stretching of material fibers between two points. For any two points a, b of the initial concept C0, after deformation set as AB, the relationship between the vector AB and ab can be

$$dx=F^*dX \quad dx = F^*dX$$

来表示。 to express.



其中F定义为deformation gradient matrix。

Where F is defined as the deformation gradient matrix.

有限元理论基础及Abaqus内部实现方式研究系列18： 几何非线性的应变的图16

1.3 应变 Strain 1.3 Strain

显然F与X和x的坐标系相关，dX和dx都是在特定三维空间笛卡尔坐标系下的矢量值。它是个张量，有9个量。F在理论上可以表示物体受力后的变形情况，但实际中试验没法测出这么多分量，因为拉伸试验等测试的几何量只能是位移，长度等有限的几个标量，这些基本量是无法得到F的全部分量的。所以只能寻找新的量来表示物体变形情

况。

It is obvious that F is related to the coordinate system of X and x. dX and dx are both vector values in a specific three-dimensional Cartesian coordinate system. It is a tensor with 9 components. In theory, F can represent the deformation of an object after it is subjected to force, but in practice, experiments cannot measure so many components because the geometric quantities in tests such as tensile tests can only be a limited number of scalars such as displacement and length. These basic quantities cannot obtain all the components of F. Therefore, new quantities need to be found to represent the deformation of the object.

一种很自然的想法就是取模，也就是取F的绝对值，称为拉伸率strech ratio：

A very natural idea is to take the modulus, that is, to take the absolute value of F, which is called the stretch ratio:

$$r=||dx||/||dX||$$

橡胶等材料，变形比较大，r可以达到2以上，那么在试验上很容易测量，但对金属等变形比较小的情况，一倍的力和两倍的力得到的r只是1.0001和1.0002，差异非常小，依然很难标定。

Rubber and other materials have a large deformation, with r reaching over 2, which is easy to measure in experiments. However, for metals and other materials with smaller deformation, the r values obtained with double the force are only 1.0001 and 1.0002, respectively, with a very small difference, making it still difficult to calibrate.

所以，为了测量和计算方便，人为假定了另一个表示物体变形情况的变量，就是应变，因为是人为设定的，所以应变的取法有很多种，主要考虑两个条件的便捷性：

Therefore, for the convenience of measurement and calculation, a variable representing the deformation of the object was artificially defined, which is called strain. Since it is artificially set, there are many ways to determine the strain, mainly considering the convenience of two conditions:

（条件a）在试验上可以简单的表示为位移，长度等基本量，且工况不同时，差异较大，能容易反应出物体加载情况。

(Condition a) Can be simply represented by basic quantities such as displacement and length, and the differences are significant under different working conditions, which can easily reflect the loading situation of the object.

（条件b）在理论上可以方便地和位移、应力一起形成最终正确的求解方程。

(Condition b) Can be conveniently combined with displacement and stress to form the final correct solution equation in theory.

有限元理论基础及Abaqus内部实现方式研究系列18： 几何非线性的应变的图21
1.4 应变的度量选择 1.4 Selection of Strain Measurement

为了简单起见，我们只针对一维情况来说明应变度量选择，应变的一般公式可以表示为r的一个函数：

For simplicity, we only discuss the strain measure selection for one-dimensional cases. The general formula for strain can be expressed as a function of r:

$$Strain=f(r)$$

由于应变是反应物体变形的量，那么当不变形时，也就是lambda=1时，显然Strain=0。

Since strain is a measure of the deformation of an object, when there is no deformation, that is, when lambda=1, it is obvious that Strain=0.

有限元理论基础及Abaqus内部实现方式研究系列18： 几何非线性的应变的图22
1.4.1 工程应变 1.4.1 Engineering Strain

当变形特别小时，也就是r~=1时，假定应变和位移是线性的，那么取伸长量和原始长度的比值。

When the deformation is particularly small, that is, when r≈1, it is assumed that strain and displacement are linear, so the ratio of elongation to original length is taken.

也就是： Namely:

Strain=u/X=(x-X)/X =r-1

Strain = u/X = (x - X) / X = r - 1

x和X分别为最终和最初的长度。一倍的力和两倍的力得到的应变变为了0.0001和0.0002，差两倍关系，很容易区分出来。

The lengths x and X represent the final and initial lengths, respectively. The strain obtained from a force of one times and two times is 0.0001 and 0.0002, respectively, showing a two-fold difference and easy to distinguish.

此时，显然条件a很容易满足，同时，在理论上将应变和位移看成是线性关系去求刚度阵，受力后得到的位移也容易计算，且结果和试验一致，因此，条件b也满足。

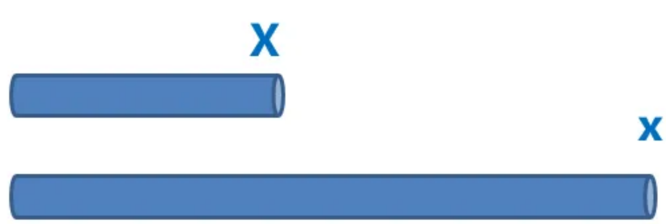
At this point, it is obvious that condition a is easily satisfied, and theoretically, if the strain and displacement are considered to be linearly related to obtain the stiffness matrix, the displacement after the force is applied is also easy to calculate, and the results are consistent with the experiment, thus, condition b is also satisfied.

有限元理论基础及Abaqus内部实现方式研究系列18：几何非线性的应变的图25

1.4.2 真实应变 1.4.2 True Strain

当存在几何大变形的情况，变形比较大时，此时x-X已经和X比拟了。

When there is a case of large geometric deformation, the deformation is relatively large, at this point, x-X has become comparable to X.



此时工程应变中的应变和位移是线性关系这种假定求出的位移已经和实际偏差较大了，因此，需要采用新的应变度量方式。一种选择是应变取从初始时刻到最终时刻点这段时间的累加，显然，这种选择就与材料点在整个时间段内

的变形路径有关。

At this point, the displacement in engineering strain is assumed to be linearly related to strain, and the displacement obtained from this assumption has already deviated significantly from the actual value. Therefore, a new strain measurement method needs to be adopted. One option is to accumulate the strain over the time period from the initial to the final moment, which is obviously related to the deformation path of the material point throughout the entire time period.

问题：那么从上图的X到最终的x，我们取哪个路径呢？

Question: Which path should we take from X to the final x in the figure above?

答：取真实的路径。 Answer: Take the actual path.

实际情况是什么路径，就应该取什么路径，此时在真实路径下得到的应变就是真实应变。当然真实路径不一定总是直接从X直线到x点的，譬如下方

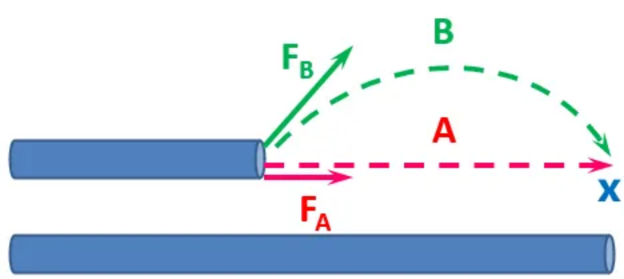
In reality, the path should be taken as it is, and the strain obtained under the actual path is the true strain. Of course, the actual path does not always have to be a straight line from X to point x, as shown below

(1) 如果受力是FA，那么的确是X直线到x点A路径；

(1) If the force is FA, then indeed the path from X to point A is a straight line.

(2) 但如果是受力FB，那么可能是路径B，只不过试验上总是采用FA来做拉伸实验罢了。

(2) But if it is subjected to FB, it may be path B, but in experiments, FA is always used for tensile tests.



在路径A下，可以得到 Under path A, one can obtain

$$Strain = \int_0^t dStrain = \int_0^t dx/X = ln(x) - ln(X) = ln(r)$$

也就是真实应变和拉伸率的对数相等，这也是为什么真实应变也称为对数或者自然应变的原因。

That is why the true strain is also called logarithmic or natural strain, because the logarithm of the true strain is equal to the stretch ratio.

有限元理论基础及Abaqus内部实现方式研究系列18： 几何非线性的应变的图32

1.4.3 Green应变 1.4.3 Green strain

真实应变在一维下很容易计算，但是在三维情况下，Strain和F相关，上面的积分就没法直接求了，只能得到一个类似

True strain is easy to calculate in one dimension, but in three dimensions, strain is related to F, and the above integral cannot be directly calculated, only a similar one can be obtained

$$Strain = \int_0^t dStrain$$

这种表达式求积分，时间的度量由增量来表示，真实应变由应变增量累加得到。因此，在计算机中每次求真实应变必须花费大量时间。

This expression calculates the integral, with time measured by increments, and the true strain is obtained by accumulating the strain increments. Therefore, each calculation of the true strain in the computer must take a significant amount of time.

- 注：实际中有限元的几何非线性求应变增量需要计算局部模态，而模态分析的求解时间成本很大。

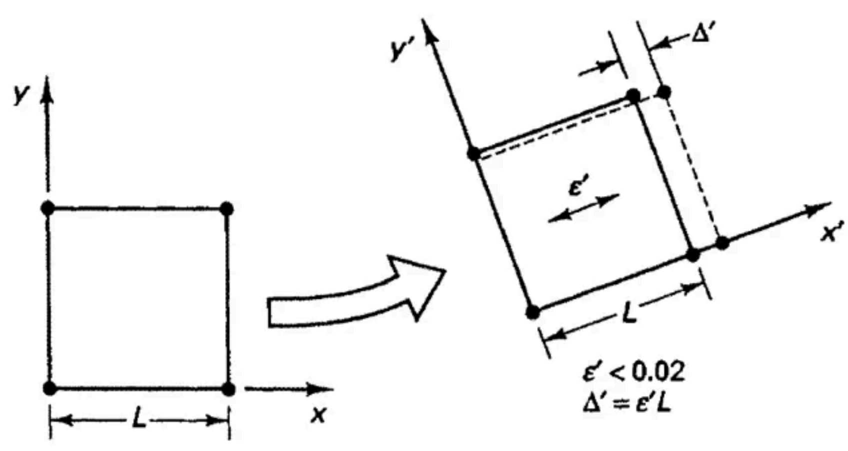
  Note: In practice, to calculate the strain increment of finite element geometric nonlinearity, it is necessary to compute the local modes, while the solution time cost of modal analysis is very high.

一般情况只能老老实实积分计算，但对大位移大转动小应变的特殊情况，发现应变如果取为Green应变，计算精度不受影响，同时，三维的应变也可以简单的表示出来。所以对小应变情况，可以采用Green应变，一维形式如下：

In general, one has to integrate and calculate honestly, but for the special cases of large displacements, large rotations, and small strains, it is found that if the strain is taken as Green strain, the calculation accuracy is not affected, and the three-dimensional strain can also be simply expressed. Therefore, for small strain cases, Green strain can be adopted, and the one-dimensional form is as follows:

Strain=0.5（r^2-1） Strain = 0.5(r^2 - 1)

Strain=0.5（r^2-1）

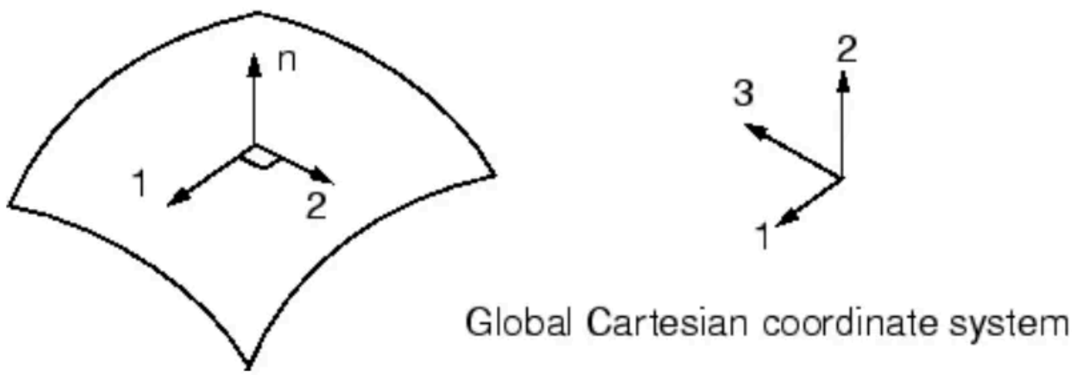


有限元理论基础及Abaqus内部实现方式研究系列18：几何非线性的应变的图37

1.5 壳的应变度量 1.5 Shell Strain Measurement

如果是壳单元，那么用的是局部坐标系，假设为L坐标系，初始构型下存在坐标系变换dX=F2*dL。类似的，几何大变形中任意一个构形也有类似的变换。此时变形量F或者真实应变就和L坐标系相关了。

If it is a shell element, it uses the local coordinate system, assumed to be the L coordinate system, where there exists a coordinate transformation dX = F2 * dL under the initial configuration. Similarly, in geometric large deformation, any configuration also has a similar transformation. At this point, the deformation amount F or the true strain is related to the L coordinate system.



具体的也可看一下第十四篇：**壳的应力方向**或者下面配合iSolver求解器的视频解说

Specifically, you can also take a look at the fourteenth article: the stress direction of shells or the video explanation below for the iSolver solver

https://www.jishulink.com/college/video/c12884 20.14 理论系列文章14：壳的应力方向

https://www.jishulink.com/college/video/c12884 20.14 Series of Theoretical Articles 14: Shell Stress Direction

有限元理论基础及Abaqus内部实现方式研究系列18：几何非线性的应变的图40

==总结== ==Summary==

本章首先从位移、变形和应变的区别说起，然后通过一维的简单例子具体介绍了几何非线性下的应变的度量方式。

This chapter first discusses the differences between displacement, deformation, and strain, and then specifically introduces the measurement methods of strain under geometric nonlinearity through a one-dimensional example.

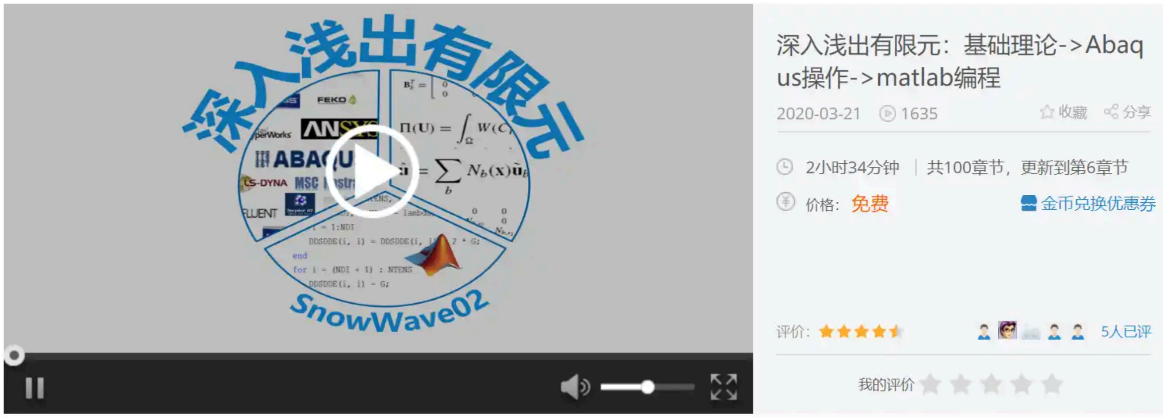
至于线性的应变在商软或者自主软件中在单元中实现，可以参考下面的视频：

As for the implementation of linear strain in commercial or independent software within the element, you can refer to the following video:

https://www.jishulink.com/college/video/c14948

深入浅出有限元：基础理论->Abaqus操作->matlab编程

A Deep Dive into Finite Element Method: Basic Theory -> Abaqus Operation -> Matlab Programming



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18： 几何非线性的应变的图43

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 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：

有限元理论基础及Abaqus内部实现方式研究系列18：几何非线性的应变的图44
有限元理论基础及Abaqus内部实现方式研究系列18：几何非线性的应变的图45
有限元理论基础及Abaqus内部实现方式研究系列18：几何非线性的应变的图46

编写线性UMAT Step By Step。介绍基于Matlab线性零基础，从零开始Step by Step的UMAT的编写和调试方法，帮助初学者UMAT入门。

Chapter 9: Writing Linear UMAT Step by Step. Introduces the writing and debugging methods of UMAT based on Matlab linear zero foundation, starting from scratch step by step to help beginners get started with UMAT.

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

第十篇：**耦合约束（Coupling constraints）的研究。**介绍Abaqus中耦合约束的原理，并使用两个简单算例加以验证。

Chapter 10: Research on Coupling Constraints. Introduce the principle of coupling constraints in Abaqus and verify it with two simple examples.

https://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。**介绍了iSolver开发以来的阶段性总结，从整体角度上介绍一下自主CAE的一些实战经验，包括开发时间预估、框架设计、编程语言选择、测试、未来发展方向等。

The eleventh article: Summary of the first phase of independent CAE development experience. It introduces the phase-by-phase summary of the development of iSolver, and gives an overall introduction to some practical experiences of independent CAE, including development time estimation, framework design, programming language selection, testing, and future development directions.

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

第十二篇：**几何梁单元的刚度矩阵。**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。

Twelfth article: Stiffness Matrix of Geometric Beam Element. This article studies the method of solving the stiffness matrix of the B31 element of geometric beam in Abaqus, taking the L beam as an example, and introduces how the parameters such as area, moment of inertia, and torsion constant used in General beam are obtained through geometric shape in geometric beam. Based on these parameters, a stiffness matrix consistent with Abaqus can be obtained, so that for any model composed only of geometric beams, Abaqus can generally obtain consistent analysis results. This idea is verified by a simple example.

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

第十三篇：**显式和隐式的区别。**介绍了显式和隐式的特点，并给出一个数学算例，分别利用前向欧拉和后向欧拉求解，以求直观表现显式和隐式在求解过程中的差异，以及增量步长对求解结果的影响。

Thirteenth article: The difference between explicit and implicit. It introduces the characteristics of explicit and implicit methods, and provides a mathematical example, using forward Euler and backward Euler methods respectively to solve, in order to intuitively demonstrate the differences between explicit and implicit methods in the solution process, as well as the influence of the increment step size on the solution results.

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

第十四篇：**壳的应力方向。**简单介绍了一下数学上张量和Abaqus中壳的应力方向，并说明Abaqus这么选取的意义，最后通过自编程序iSolver来验证壳的应力方向的正确性。

14th article: Stress direction of shells. A brief introduction to the tensor of stress direction in mathematics and in Abaqus, and an explanation of the significance of Abaqus's selection, and finally, the correctness of the stress direction of shells is verified through the self-written program iSolver.

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

第十五篇：**壳的剪切应力。**介绍了壳单元中实际的和板壳近似理论中的剪切应力，也简单猜测了一下Abaqus的内部实现流程，最后通过一个算例来验算Abaqus中的真实的剪切应力。

15th article: Shear Stress of Shell. Introduces the shear stress in actual shell elements and in the plate-shell approximate theory, also makes a simple guess about the internal implementation process of Abaqus, and finally verifies the actual shear stress in Abaqus through a calculation example.

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

第十六篇：**Part、Instance与Assembly。**介绍了Part、Instance与Assembly三者之间的关系，分析了Instance的网格形成原理，并猜测Abaqus的内部组装实现流程，随后针对某手机整机多part算例，通过自编程序iSolver的结果比对验证我们的猜想。

Chapter 16: Part, Instance, and Assembly. Introduces the relationship between Part, Instance, and Assembly, analyzes the principle of grid formation of Instance, and guesses the internal assembly implementation process of Abaqus. Subsequently, for a multi-part assembly example of a mobile phone, the results of the self-written program iSolver are compared and verified to confirm our conjecture.

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

第十七篇：**几何非线性的物理含义。**介绍了几何非线性的简单的物理含义，并通过几何非线性的悬臂梁Abaqus和iSolver的小应变情况的结果，从直观上理解几何非线性和线性的差异。

Chapter 17: Physical Meaning of Geometric Nonlinearity. Introduces the simple physical meaning of geometric nonlinearity and illustrates the difference between geometric nonlinearity and linearity through the results of small strain of the cantilever beam with geometric nonlinearity in Abaqus and iSolver.

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