

## 有限元理论基础及Abaqus内部实现方式研究系列17： 几何非线性的物理含义

### Theoretical Foundation of Finite Element Method and Research on Internal Implementation of Abaqus Series 17: Physical Meaning of Geometric Nonlinearity



SnowWave02



2020年3月9日 15:06 March 9, 2020 15:06

浏览: 3332 Views: 3332

评论: 7 Comments: 7

收藏: 11 Favorited: 11

(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17： 几何非线性的物理含义的图1

有限元理论基础及Abaqus内部实现方式研究系列17： 几何非线性的物理含义的图2

==概述== ==Overview==

有限元理论基础及Abaqus内部实现方式研究系列17： 几何非线性的物理含义的图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17篇：几何非线性的物理含义 == == Article 17: Physical Meaning of Geometric Nonlinearity ==

## 有限元理论基础及Abaqus内部实现方式研究系列17：几何非线性的物理含义的图6

几何非线性，即考虑大变形对于结构平衡位置的影响。爆炸冲击、冲压成型、大型结构件的弯曲等都含有几何非线性问题，几何非线性也是现代结构有限元商业软件的必备发展方向。在Abaqus中只要简单的在Step中勾选NL Geom这个开关就行。

Geometric nonlinearity, which considers the effect of large deformation on the structural equilibrium position. Explosive impact, stamping, and bending of large structural components all contain geometric nonlinearity issues, and geometric nonlinearity is also an essential development direction for modern structural finite element commercial software. In Abaqus, it is as simple as checking the NL Geom switch in the Step.

**Edit Step**

Name: Step-1

Type: Static, General

Basic Incrementation Other

Description:

Time period: 1

NLgeom: ☐ Off (This setting controls the inclusion of nonlinear effects of large displacements and affects subsequent steps.) ☒ On

Automatic stabilization: None

☐ Include adiabatic heating effects

OK Cancel

K.J.Bathe教授1979年提出的几何非线性理论也是目前应用于有限元分析最广泛的几何非线性力学。但要想在自主结构有限元程序中编程加入几何非线性的理论，远远不是Abaqus或者Ansys表面上的看起来只要加一个NLGeom=on/off这个开发那么容易。同时，要让自研程序的几何非线性做到和商软结果接近远比线性或者材料非线性难，我们在iSolver编写几何非线性的过程中也发现，除了刚度矩阵的修改，增量迭代法的自动步长选取，收敛准则等都有极大的影响。从本章开始，将介绍几何非线性的一些理论和Abaqus的实现方式，同时通过iSolver的程序验证Abaqus的实现方式。本章将介绍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。配合本章的视频解说和操作演示可看下

方:

The geometric nonlinearity theory proposed by Professor K.J.Bathe in 1979 is also the most widely used geometric nonlinearity mechanics in finite element analysis today. However, to program geometric nonlinearity theory into an independent structural finite element program is far from as simple as adding an NLGeom=on/off switch in Abaqus or Ansys. Moreover, making the geometric nonlinearity of the independently developed program approach the results of commercial software is much more difficult than linear or material nonlinearity. We also found during the process of writing geometric nonlinearity in iSolver that factors such as the modification of the stiffness matrix, automatic step size selection in the incremental iterative method, and convergence criteria have a significant impact. Starting from this chapter, we will introduce some theories of geometric nonlinearity and its implementation in Abaqus, and verify the implementation method of Abaqus through the program of iSolver. This chapter will introduce the simple physical meaning of geometric nonlinearity and understand the difference between geometric nonlinearity and linearity intuitively through the results of a cantilever beam in Abaqus and iSolver under small strain conditions. Accompanied by the video explanation and operation demonstration in this chapter, you can see below:

<http://www.jishulink.com/college/video/c12884> 20理论系列文章17-几何非线性的物理含义

<http://www.jishulink.com/college/video/c12884> 20 Theory Series Article 17 - Physical Meaning of Geometric Nonlinearity

## 有限元理论基础及Abaqus内部实现方式研究系列17：几何非线性的物理含义的图10

### 2.1 几何非线性的物理含义 2.1 Physical Meaning of Geometric Nonlinearity

#### 有限元理论基础及Abaqus内部实现方式研究系列17：几何非线性的物理含义的图11

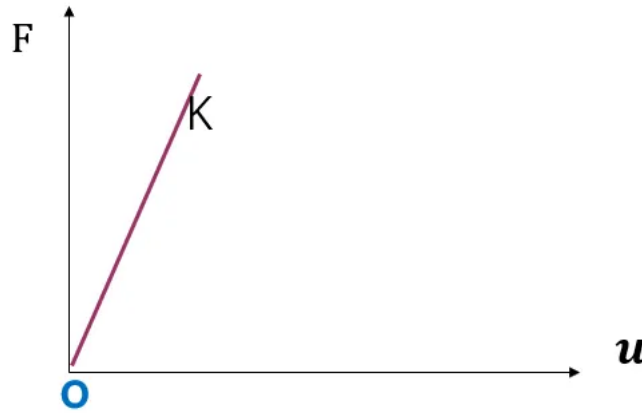
##### 2.1.1 从线性到几何非线性 2.1.1 From Linear to Geometric Nonlinearity

一个物体从初始状态A由于受到外部载荷运动，如果现在已知了另一种状态B的位移，那么其它的任意状态C的位移怎么求？

If an object moves from its initial state A due to external loads, and if the displacement of another state B is known, how can the displacement of any other state C be calculated?

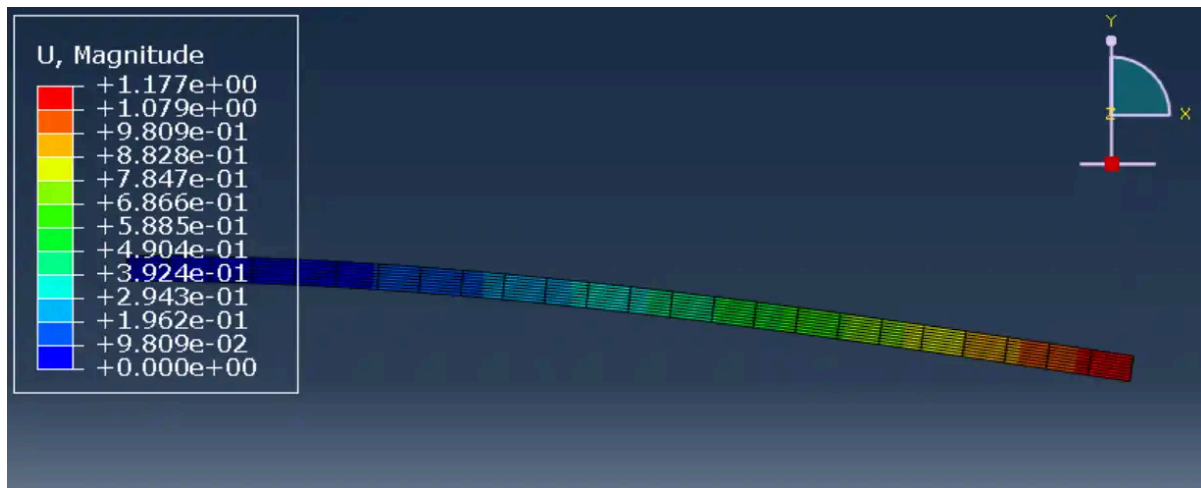
如果能直接从B的位移乘以一个常量就得到C，那么这个系统就是线性系统。譬如下面的800mm的悬臂梁问题，在Abaqus中用线性计算，载荷F和位移u是直线关系。

If the displacement of B can be directly obtained by multiplying it by a constant to get C, then this system is linear. For example, in the 800mm cantilever beam problem, linear calculations are used in Abaqus, and there is a linear relationship between the load F and displacement u.



载荷1N的时候Abaqus计算得到最大位移时1.177mm。

When the load is 1N, Abaqus calculates the maximum displacement to be 1.177mm.



那么载荷是1000N时是多少？显然，不用计算也知道就是1177mm。但1177这个值明显有问题，已经超过了梁的长度，按生活经验判断这个梁估计都断了或者极端扭曲了，所以这种情况需要用几何非线性来计算。

What is the value when the load is 1000N? Clearly, without calculation, it is 1177mm. But the value of 1177 is obviously problematic, as it exceeds the length of the beam. According to common sense, it is estimated that this beam has broken or is extremely distorted. Therefore, geometric nonlinearity needs to be used for this calculation.

#### 有限元理论基础及Abaqus内部实现方式研究系列17：几何非线性的物理含义的图16

##### 2.1.2 几何非线性简单物理含义 2.1.2 Simple Physical Meaning of Geometric Nonlinearity

虚功原理如下： The principle of virtual work is as follows:

$$W = \int_v \sigma * \delta D dV_v$$

在物理上可解释能量守恒原理，即在某一个时刻点，假定在外力作用下有个虚拟的位移，那么外力在虚拟位移下做的虚功=内部应变能的变化相同。

The physically interpretable principle of energy conservation, that is, at a certain moment, assuming there is a virtual displacement under the action of external forces, the virtual work done by the external force under the virtual displacement is equal to the change in internal strain energy.

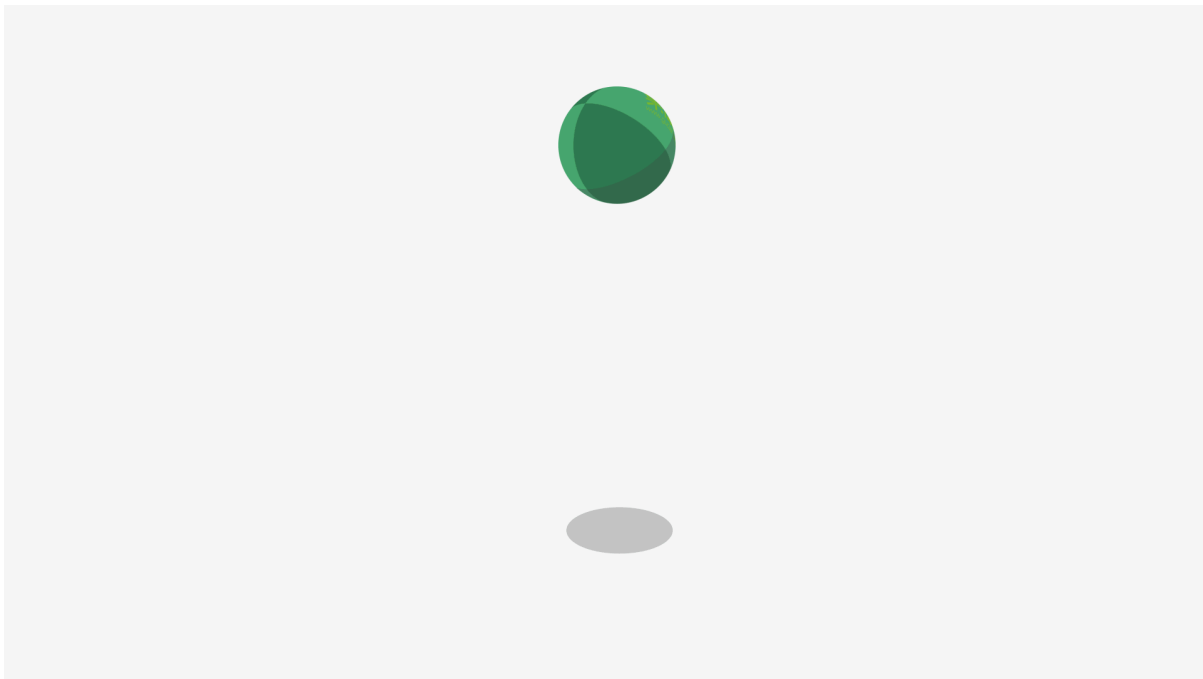
为了更好的理解上面的物理解释，如果我们把当成真实的位移，那么外层加上对时间的积分，可以理解为外力在虚拟位移下做的虚功=内部应变能的一段小时间内对应变能的积分：

To better understand the above physical explanation, if we treat the displacement as a real one, then by integrating over time, it can be understood that the virtual work done by the external force under the virtual displacement is equal to the integral of internal strain energy over a small time interval corresponding to the strain energy.

$$W = \int_t \int_v S(t) * d\varepsilon(t) dV(t) dt$$

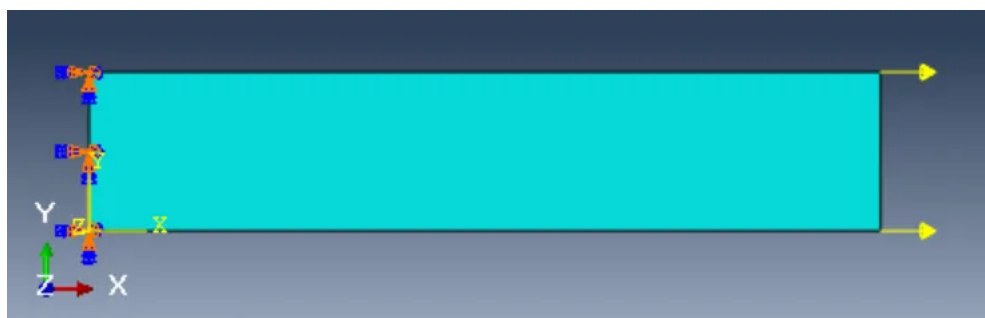
那么就和我们高中时学过的小球受重力作用后势能、动能相互转换是一样的原理。

So it is the same principle as the conversion between potential energy and kinetic energy of a small ball under the action of gravity that we learned in high school.



举个简单的有限元例子，譬如下方一个减缩积分S4R单元的右端受X方向两个力F：

Give a simple finite element example, such as the right end of the following reduced integration S4R element subjected to two forces F in the X direction:



得到的位移为 $U$ （相当于虚位移），那么力 $F$ 做的功是 $W=2F*U$ 。

The displacement obtained is  $U$  (equivalent to the virtual displacement), so the work done by the force  $F$  is  $W = 2F * U$ .

增加的应变能为 $S*E*V$ 在对所有时刻点 $t$ 的积分， $S$ 、 $E$ 、 $V$ 都是当前时刻的值。

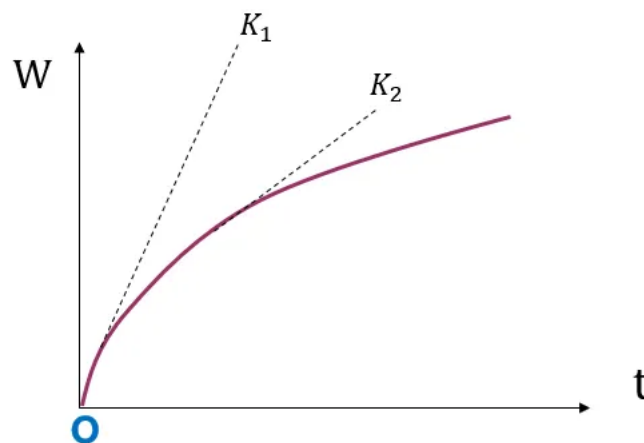
The increased strain energy is the integral of  $S*E*V$  over all time points  $t$ , where  $S$ ,  $E$ , and  $V$  are the values at the current moment.

应变 $E$ 的取法有很多种，采用真实应变，那么 $E$ 取为位移 $U(t)$ 和长度 $L(t)$ 的比值，按虚位移的定义，虚位移必然相对原始长度比较小，也就是 $L(t)=L_0+U(t)$ 可以用 $L_0$ 代替， $E=U(t)/L_0$ ，如果是线性系统， $U(t)=U^*t/\text{总时间}$ ，积分很容易计算出来，得到应变能 $V=S*U*b*h$ ，因为内力和外力平衡，减缩积分S4R面积内的所有点的应力和中心点一样，所以 $S=F/\text{截面积}=2F/(b*h)$ ，此时 $V=2F*U=W$ 。

There are many ways to determine the strain  $E$ . If the true strain is used, then  $E$  is taken as the ratio of displacement  $U(t)$  to length  $L(t)$ . According to the definition of virtual displacement, the virtual displacement must be relatively smaller than the original length, that is,  $L(t) = L_0 + U(t)$  can be replaced by  $L_0$ ,  $E = U(t)/L_0$ . If it is a linear system,  $U(t) = U^*t/\text{total time}$ , the integration can be easily calculated, resulting in the strain energy  $V = S*U*b*h$ . Since the internal force and external force are balanced, the reduced integral S4R area contains all points with the same stress as the center point, so  $S = F/\text{area} = 2F/(b*h)$ , at this time  $V = 2F*U = W$ .

如果是非线性系统，那么应变就没法简单的用 $E=U(t)/L_0$ ， $W$ 随 $t$ 的变化就是个非线性过程。每个时刻点可以求出一个斜率，这个斜率最终会形成当前时刻点的刚度矩阵。

If it is a nonlinear system, then the strain cannot be simply expressed as  $E = U(t)/L_0$ , and the change of  $W$  with  $t$  is a nonlinear process. A slope can be calculated at each moment, and this slope eventually forms the stiffness matrix at the current moment.



如果是对当前时刻的体积积分，那么对 $W$ 求导就很困难，因为 $V$ 也是与时间有关的，可以选择一个不变的初始构型 $V_0$ ，此时应力和应变也需要做相应的变化，我们假定分别变为了 $S$ 和 $E$ 。

If it is an integral of volume at the current moment, it is difficult to differentiate  $W$  because  $V$  is also related to time. One can choose an initial configuration  $V_0$  that does not change, and at this time, the stress and strain also need to change accordingly. We assume they have become  $S$  and  $E$ , respectively.



$$W = \int_{V_0} S * \delta \varepsilon dV_0$$

求导：

$$dW = \int_{V_0} (dS * \delta \varepsilon + S ** d\delta \varepsilon) dV_0$$

也就是刚度矩阵将分为两块，上式的前面一部分依然是以前的BDB形式，只不过B换成了当前时刻的应变位移矩阵，而后面新增项一般称为几何刚度阵，在Abaqus中称为初始应力矩阵（initial stress stiffness）。

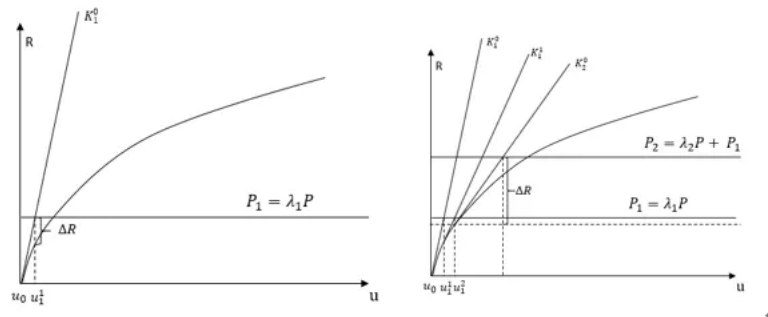
The stiffness matrix is divided into two parts; the front part of the above formula is still in the previous BDB form, but B is replaced with the strain-displacement matrix at the current moment, while the additional term is generally referred to as the geometric stiffness matrix, and in Abaqus, it is called the initial stress stiffness matrix.

有限元理论基础及Abaqus内部实现方式研究系列17：几何非线性的物理含义的图27

### 2.1.3 几何非线性的计算机求解方式 2.1.3 Computer solution methods for geometric nonlinearity

理论上受力曲线是一条光滑曲线，计算机没法求解曲线上每个时刻点的结果，只能求解部分有限间隔点的结果。非线性问题不是一条直线，所以需要多次迭代才能实现，而不再考虑 $u=F/K$ 这种一次性就能计算的简单问题。非线性问题求解有多种方法主要分为以下几类：增量法、迭代法、增量迭代法和弧长法。

Theoretically, the force curve is a smooth curve, and the computer cannot solve the results at each moment on the curve, but can only solve the results at some finite intervals. Nonlinear problems are not a straight line, so multiple iterations are needed to achieve the result, and simple problems like  $u=F/K$ , which can be calculated in one go, are no longer considered. There are many methods for solving nonlinear problems, mainly divided into the following categories: incremental method, iterative method, incremental iterative method, and arc length method.



具体的理论和Abaqus实现过程可参考我们以前系列文章：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。

The specific theories and Abaqus implementation process can be referred to in our previous series articles: the fourth article: Solution of Nonlinear Problems. It introduces the numerical solution methods adopted by Abaqus in nonlinear analysis.

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

有限元理论基础及Abaqus内部实现方式研究系列17：几何非线性的物理含义的图29



## 2.2 悬臂梁的几何非线性算例 2.2 Example of cantilever beam geometric nonlinearity

此次验证，依然使用自研求解器iSolver与Abaqus计算结果对比的方式。

In this verification, the self-developed solver iSolver is still used for comparison with Abaqus results.

 有限元理论基础及Abaqus内部实现方式研究系列17：几何非线性的物理含义的图30

### 2.2.1 算例说明 Example Description

我们采用上面提到的悬臂梁的例子，我们采用壳单元来模拟整个梁。

We use the example of the cantilever beam mentioned above, and we use shell elements to simulate the entire beam.

参数如下： Parameters as follows:

尺寸：X方向长度 $L=800$ ，Y方向宽度 $b=20$ ，Z方向厚度 $h=20$ 。

Dimensions: Length in X direction  $L=800$ , width in Y direction  $b=20$ , thickness in Z direction  $h=20$ .

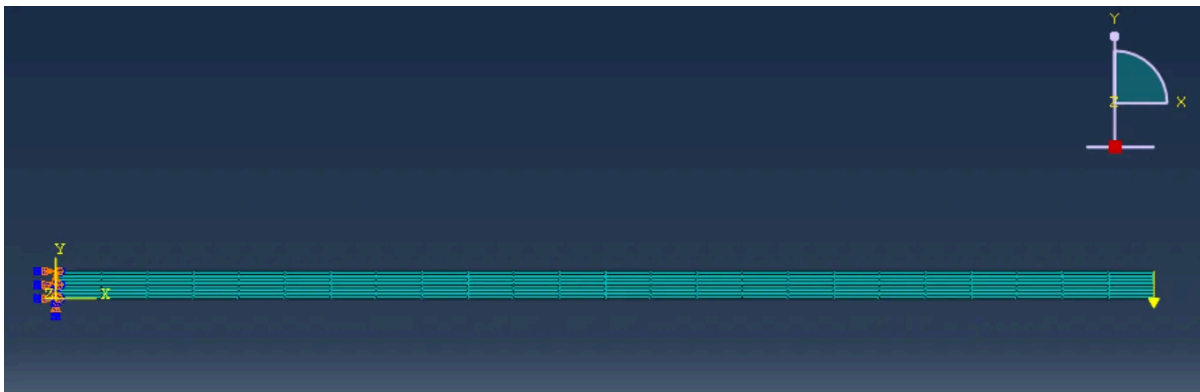
材料：Young's Modulus  $E=11000$ 。

Material: Young's Modulus  $E=11000$ .

点载荷： $F=1000$ 。 Point Load:  $F=1000$ .

网格：在X方向划分20份，Y方向划分8份（为了避免Abaqus和iSolver中都会出现的沙漏现象，厚度方向尽量多划分网格）。

Mesh: Divided into 20 parts in the X direction and 8 parts in the Y direction (to avoid the hourglass phenomenon that may occur in both Abaqus and iSolver, the thickness direction should be meshed as finely as possible).



Step中设置NLGeom打开，并且步长=0.25。

In Step, set NLGeom to open and the step length to 0.25.

- 我们发现就算不设置这个步长，采用自动步长，Abaqus也依然会自动调整为0.25的步长，为了和iSolver对比，我们都设置0.25，避免因收敛判据的不同造成的结果差异。

We found that even if this step size is not set, Abaqus will still automatically adjust to a step size of 0.25 when using automatic step size. In order to compare with iSolver, we all set it to 0.25 to avoid differences in results caused by different convergence criteria.

如果线性情况下，上面已经得到了就是1177，现在几何非线性情况，我们采用小应变的几何非线性。

If it is in a linear case, as has been obtained above, it is 1177. Now in the case of geometric nonlinearity, we adopt small strain geometric nonlinearity.

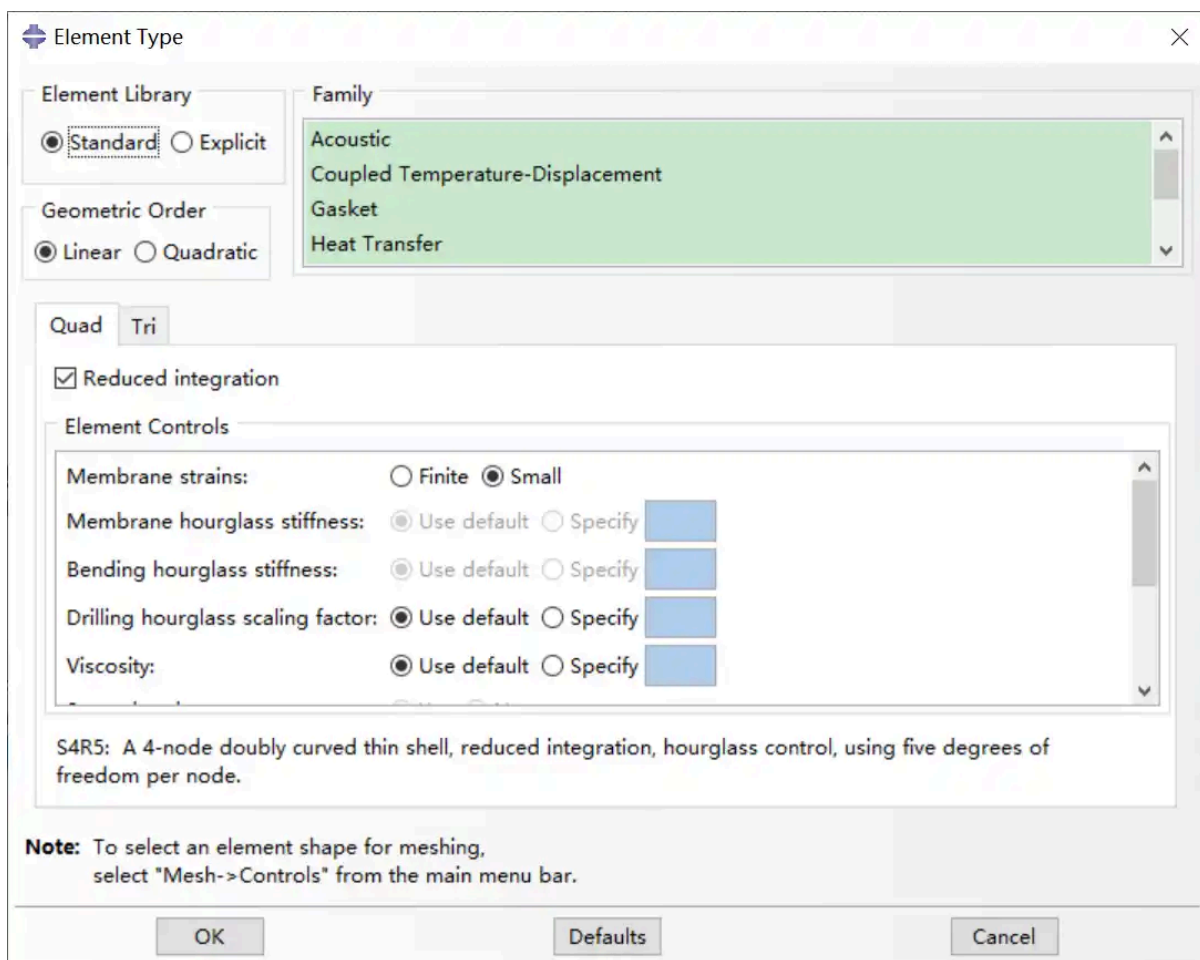
- 注：这个问题虽然位移很大，但依然还是个小应变问题，小应变单元的计算速度远远高于大应变，所以我们只用小应变单元来模拟。

Note: Although the displacement is large, it is still a small strain problem. The calculation speed of small strain elements is much faster than that of large strain, so we only use small strain elements to simulate.

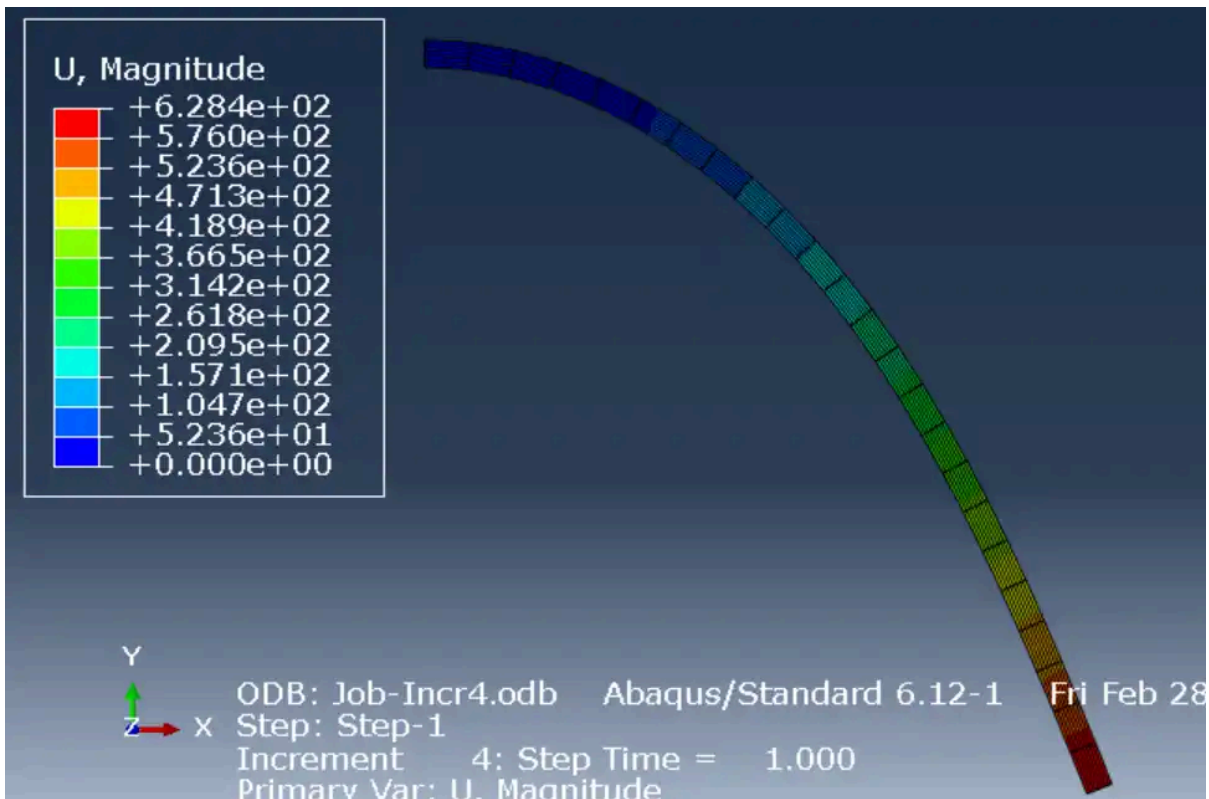
### 有限元理论基础及Abaqus内部实现方式研究系列17：几何非线性的物理含义的图33

#### 2.2.2 Abaqus结果 2.2.2 Abaqus Results

Abaqus中设置单元为S4R5。 Set the element type to S4R5 in Abaqus.

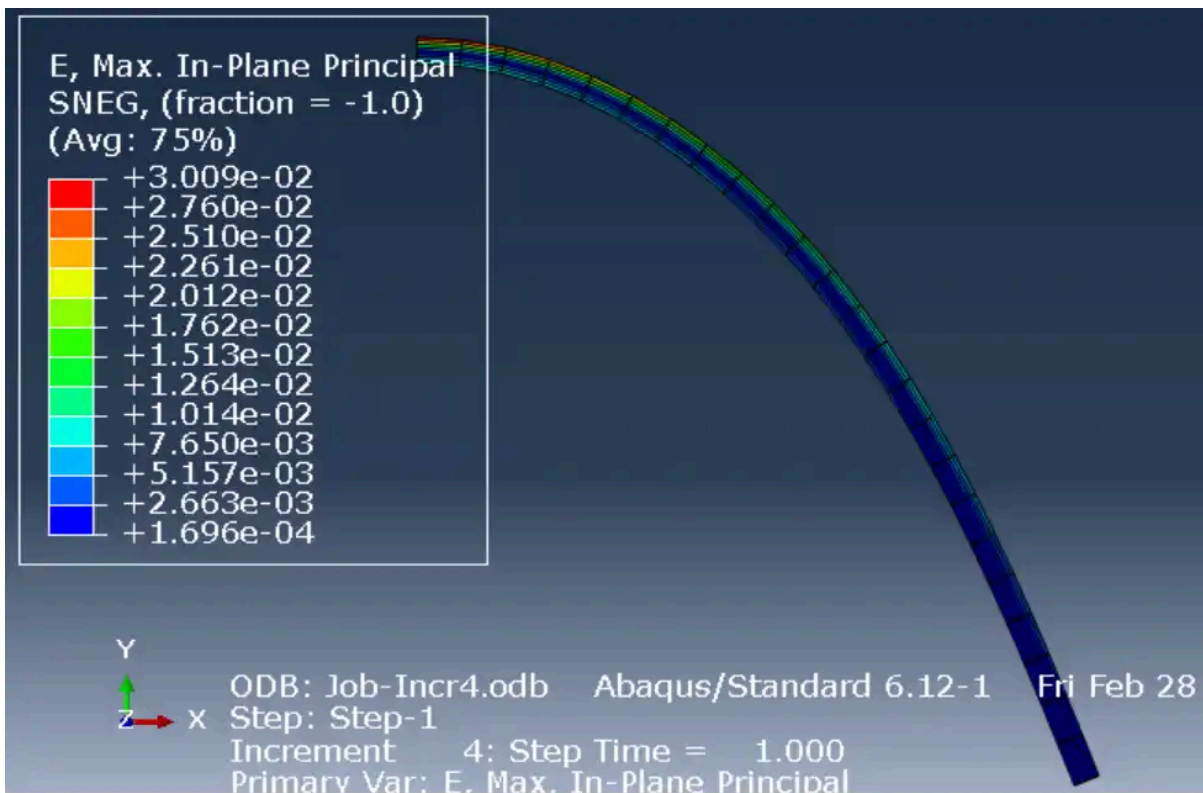


得到最后一个增量步的位移为628.4： The displacement of the last increment step is 628.4:



同时可以发现应变为0.03，和1相比是小量，所以可以用小应变来模拟。

It can also be found that the strain is 0.03, which is a small amount compared to 1, so it can be simulated with small strain.

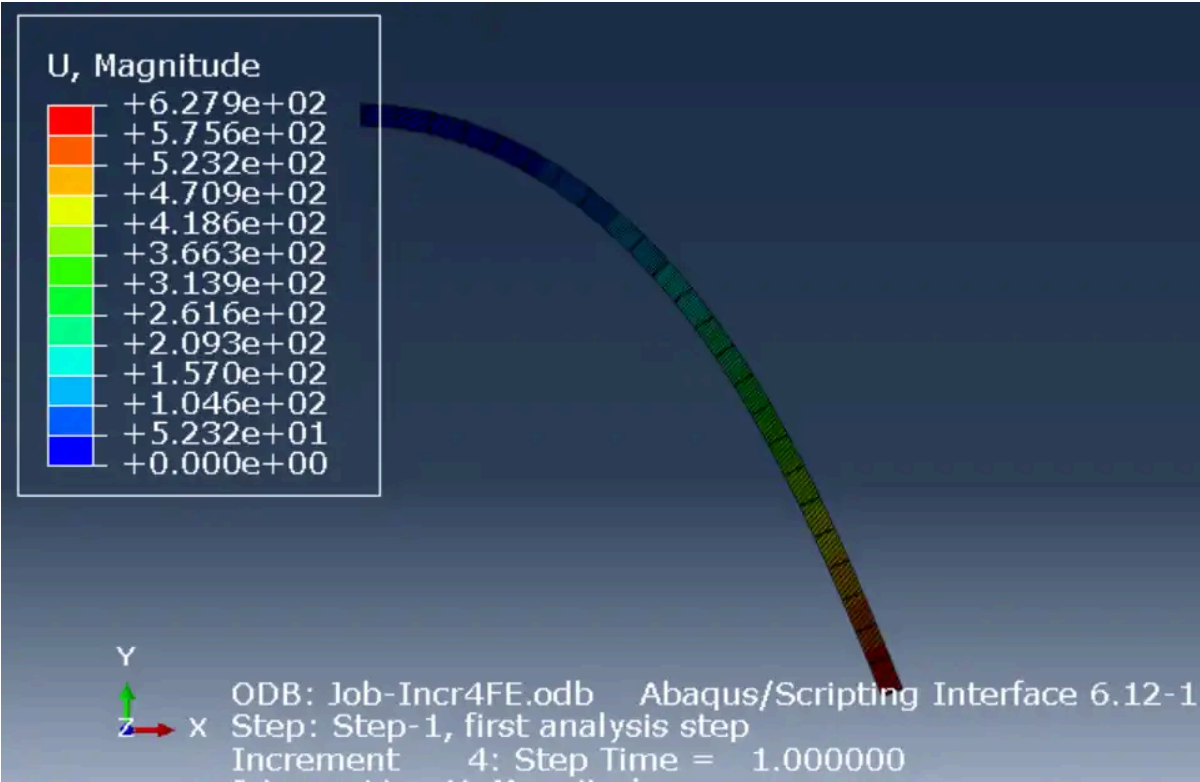


有限元理论基础及Abaqus内部实现方式研究系列17：几何非线性的物理含义的图39

### 2.2.3 iSolver结果 2.2.3 iSolver Results

在运行结束后，得到的位移为627.9，和Abaqus相差0.08%。

At the end of the run, the obtained displacement is 627.9, which is 0.08% different from Abaqus.



有限元理论基础及Abaqus内部实现方式研究系列17： 几何非线性的物理含义的图42

2.3 遗留问题 2.3 Residual Issues

Abaqus应力判据为： The Abaqus stress criterion is:

$$\gamma_{max}^{\alpha} \leq R_n^{\alpha} \tilde{q}^{\alpha}$$

其中， Among which,

$$\gamma_{max}^{\alpha}$$

为最大应力残差； is the maximum stress residual;

$$R_n^{\alpha}$$

为误差因子，默认为5e-3。 is the error factor, with a default value of 5e-3.

q为平均时间应力，即.msg中的Average Force值，在Abaqus的帮助文档中有具体的计算公式，但我们按照公式第一个增量步的结果和Abaqus完全一致，后面的后面的增量步差异很大，这是导致我们的迭代次数和Abaqus

没法完全一致的原因，有知道q第二增量步怎么计算的大神还请不吝赐教。

q is the average time stress, which is the Average Force value in the .msg file. The Abaqus help document provides a specific calculation formula. However, according to the formula, the result of the first increment step is completely consistent with Abaqus, but there is a significant difference in the subsequent increment steps, which is the reason why our number of iterations cannot be completely consistent with Abaqus. If there is any expert who knows how to calculate the second increment step of q, please kindly give me some guidance.

CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION					1
AVERAGE FORCE	1.503E-02	TIME AVG. FORCE	1.503E-02		
LARGEST RESIDUAL FORCE	-2.503E-11	AT NODE	124	DOF	2
INSTANCE: PART-1-1					
LARGEST INCREMENT OF DISP.	-1.028E-02	AT NODE	225	DOF	2
INSTANCE: PART-1-1					
LARGEST CORRECTION TO DISP.	-1.028E-02	AT NODE	225	DOF	2
INSTANCE: PART-1-1					

 有限元理论基础及Abaqus内部实现方式研究系列17：几何非线性的物理含义的图49

### ==总结== ==Summary==

本文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。

This article introduces the simple physical meaning of geometric nonlinearity and illustrates the difference between geometric nonlinearity and linearity through the results of a cantilever beam with small strain in Abaqus and iSolver, from an intuitive perspective.

视频解说及算例操作演示如下： The video explanation and example operation demonstration are as follows:

<http://www.jishulink.com/college/video/c12884> 20理论系列文章17-几何非线性的物理含义

<http://www.jishulink.com/college/video/c12884> 20 Theory Series Article 17 - Physical Meaning of Geometric Nonlinearity

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](http://www.jishulink.com)

email: [snowwave02@qq.com](mailto:snowwave02@qq.com)



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



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。

Chapter 4: Solution of Nonlinear Problems. Introduces the numerical solution 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17：几何非线性的物理含义的图51

有限元理论基础及Abaqus内部实现方式研究系列17：几何非线性的物理含义的图52

有限元理论基础及Abaqus内部实现方式研究系列17：几何非线性的物理含义的图53

**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>

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

**网架设计之SFCAD第七节终极大招 Design of Grid Structure: The...**

高工 Senior Engineer

¥138

**ansys经典半刚性脚手架分析 (2) Ansys Classic Semi**

冷月 Cold Moon

¥1