有限元理论基础及Abaqus内部实现方式研究系列15： 壳的剪切应力

Theoretical Foundation of Finite Element Method and Internal Implementation of Abaqus Series 15: Shear Stress of Shell

SnowWave02 关注 Focus
2019年12月16日 11:16 December 16, 2019 11:16 浏览：4234 Views: 4234 评论：1 Comments: 1 收藏：4 Favorites: 4

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

==概述== ==Overview==

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15篇：壳的剪切应力== ==15th Article: Shear Stress of Shells==

自编有限元应力的校核除了Mises等合力外，也应该校核各个应力分量。材料力学中六个应力分量如下：

In addition to the Mises equivalent stress, the verification of individual stress components should also be performed. The six stress components in material mechanics are as follows:

$$\begin{pmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{pmatrix}$$

其中Tau11，Tau22，Tau33为正应力，Tau12,13,23为三个剪切应力，对壳来说，Tau33=0，Tau12为面内剪应力，Tau13,23即为本文所说的横向剪切应力。

Among them, Tau11, Tau22, and Tau33 are normal stresses, while Tau12, 13, and 23 are the three shear stresses. For shells, Tau33=0, Tau12 is the in-plane shear stress, and Tau13 and 23 are the transverse shear stresses referred to in this article.

最近在做iSolver壳的应力分量和Abaqus比对时，发现Abaqus的横向剪切应力和预想的不一致。iSolver按照常用的壳的理论得到的剪切应力是个与厚度无关的常量，但Abaqus的横向剪切应力分量TSHR13，TSHR23，在各个截面方向积分点section point不一样。

Recently, while comparing the stress components of the iSolver shell with Abaqus, it was found that Abaqus's transverse shear stress does not match the expected results. According to the commonly used shell theory, the shear stress obtained by iSolver is a constant independent of thickness, but Abaqus's transverse shear stress components TSHR13 and TSHR23 vary at different section points along the section direction.

花了点时间细致的研究了一下，猜测Abaqus中剪切应力TSHR13、23是真实应力，但有限元理论和iSolver中计算的是板壳近似理论中平均剪切应力。本章将介绍壳单元中实际的和板壳近似理论中的剪切应力，也猜测了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，并通过iSolver来计算板壳理论的平均剪切应力。

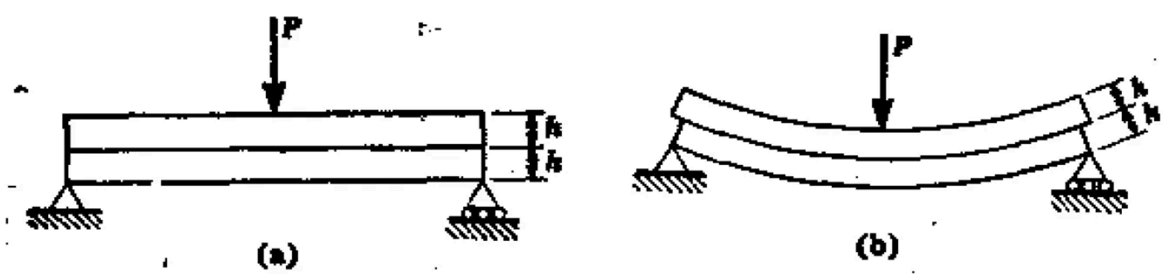
After spending some time studying in detail, I guess that the shear stresses TSHR13 and 23 in Abaqus are true stresses, but the finite element theory and iSolver calculate the average shear stress in the plate shell approximation theory. This chapter will introduce the actual and plate shell approximation theories of shear stress in shell elements, also guessing the internal implementation process of Abaqus, and finally verifying the true shear stress in Abaqus through an example, and calculating the average shear stress in the plate shell theory through iSolver.

有限元理论基础及Abaqus内部实现方式研究系列15： 壳的剪切应力的图4

1.1 壳的真实的剪切应力 1.1 Real Shear Stress of Shell

剪应力是材料由于抗拒面之间的滑动而产生的沿表面方向的应力。壳的中间层存在剪切应力，这个可以通过下面简单的例子验证。两块板叠加在一起，简支，中点加力，板间假定无摩擦，那么将会得到下面的形状，中间层表面上梁的伸长和下梁的缩短完全由x方向应力决定，此时中间层无抗拒滑动的力，也就不存在剪应力，。

Shear stress is the stress along the surface direction that occurs in materials due to resistance to sliding between surfaces. There is shear stress in the middle layer of the shell, which can be verified by the following simple example. Two plates are stacked together, simply supported, and loaded at the midpoint, assuming no friction between the plates, then the following shape will be obtained. The elongation of the beam on the middle layer surface and the shortening of the lower beam are completely determined by the x-direction stress. At this time, there is no resistance to sliding force in the middle layer, so there is no shear stress.



但如果两块板中面部分用胶水粘住，胶水将会阻碍中间层上下两个面的相对滑动，上边面的纤维长度会变少，下边面的纤维长度增加，使得中间层上下两个纤维长度相等，也就在中间层将产生剪应力。

But if the middle surface part of the two plates is glued together, the glue will hinder the relative sliding of the two surfaces of the middle layer. The fiber length on the upper surface will decrease, and the fiber length on the lower surface will increase, making the fiber lengths on the upper and lower surfaces of the middle layer equal, thus generating shear stress in the middle layer.

同时，很显然，上图最上层面任意一点没有任何切向的外力，所以不会有阻碍滑动的剪切应力了。而从中间层到最上层，可以猜测剪应力将逐步减小。根据材料力学的理论，实际的截面上的剪应力分布如下：

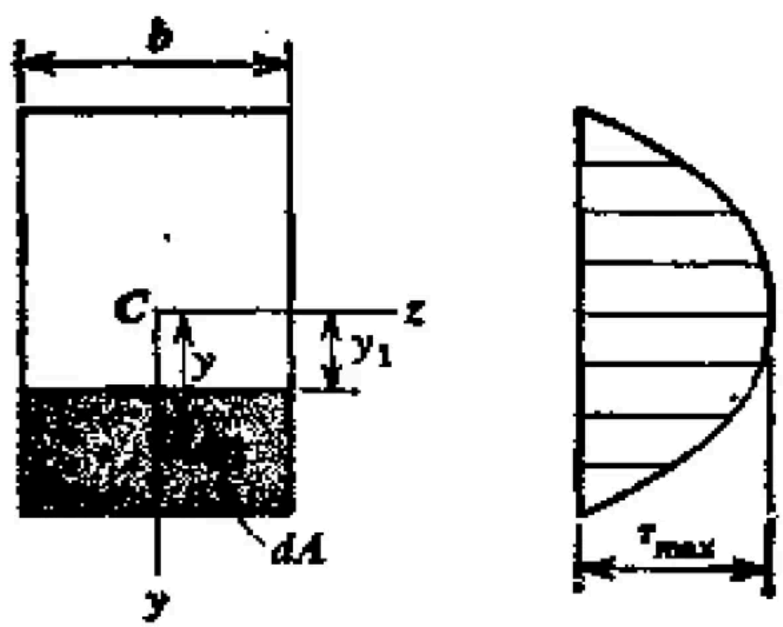
Simultaneously, it is obvious that at any point on the uppermost surface of the figure, there is no tangential external force, so there will be no shear stress to hinder sliding. From the middle layer to the uppermost layer, it can be guessed that the shear stress will gradually decrease. According to the theory of material mechanics, the actual distribution of shear stress on the cross-section is as follows:

$$\tau=\frac{V}{2I}\left(\frac{h^2}{4}-y_1^2\right)$$

其中V为剪力。 Where V represents the shear force.

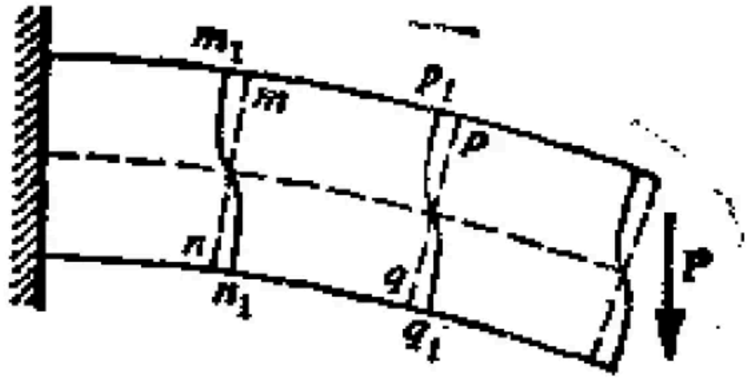
显然与材料点所在截面的y方向坐标y1是二次关系。用图形化表示为下图右侧，随截面厚度方向是抛物线，中面最大，上下表面为0：

It is obviously a quadratic relationship with the y-direction coordinate y1 of the material point on the cross-section. It is graphically represented on the right side of the figure, forming a parabola in the direction of the cross-section thickness, with the middle surface being the maximum and the upper and lower surfaces being 0.



在各向同性材料中，剪切应力和剪切应变也就是剪切角成正比，所以，如果一个橡皮条做成的悬臂梁，那么原来在铅直线上画的直线受力后将会变成如下图所示的m1n1的曲线。

In isotropic materials, shear stress and shear strain, that is, the shear angle, are proportional to each other. Therefore, if a cantilever beam is made of a rubber band, the straight line originally drawn on the vertical line will become the curved line shown in the figure below, m1n1.

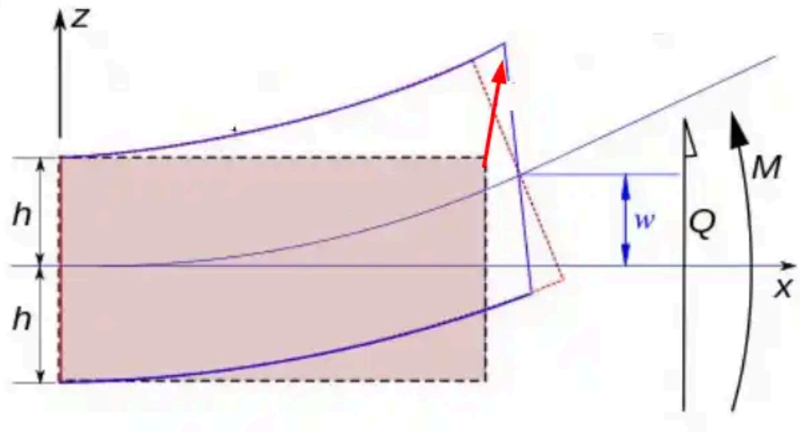


有限元理论基础及Abaqus内部实现方式研究系列15： 壳的剪切应力的图13

1.2 板壳近似理论的平均剪切应力 1.2 Average Shear Stress in Shell Approximation Theory

上面的剪切应力表达式中，需要预先知道剪力V，但实际有限元流程中在计算刚度矩阵K时并不知道V，K只由两者决定，一个是应力应变关系，也就是本构关系矩阵C，另一个是应变和位移关系，也就是常说的B矩阵。因此有限元中对壳做了直线法的近似，认为变形前垂直于中面的截面的所有材料点变性后依然位于一个平面内，譬如下图的红色箭头表示原先面上的所有材料点受力后组成新的材料点平面。根据这个假设，那么可以发现所有的剪切角也就是剪切应变是个恒定值，乘以各向同性的剪切模量G，那么得到的剪切应力也是恒定值，相当于一个平均效应的应力，但后面的例子通过iSolver的计算可以看到，这个平均剪切应力并不是简单的是剪力V在截面上的平均。

In the above expression for shear stress, it is necessary to know the shear force V in advance, but in the actual finite element process, when calculating the stiffness matrix K, V is not known; K is determined by both of them, one being the stress-strain relationship, also known as the constitutive relationship matrix C, and the other being the strain-displacement relationship, commonly referred to as the B matrix. Therefore, in the finite element method, a linear approximation of the shell is made, assuming that all material points on the cross-section perpendicular to the middle surface remain in a plane after deformation. For example, the red arrow in the figure below indicates that all material points on the original surface, after being subjected to forces, form a new material point plane. Based on this assumption, it can be found that all the shear angles, i.e., the shear strain, are constant values. Multiplying them by the isotropic shear modulus G, the resulting shear stress is also a constant value, which is equivalent to an average stress effect. However, as can be seen from the examples calculated by iSolver, this average shear stress is not simply the average of the shear force V over the section.



有限元理论基础及Abaqus内部实现方式研究系列15： 壳的剪切应力的图16

1.3 Abaqus内部实现流程猜测 1.3 Guess of the Abaqus Internal Implementation Process

Abaqus程序的内部流程由增量迭代法实现，猜测和普通的有限元理论是一致的，对静力分析步骤如下：

The internal process of the Abaqus program is realized by the incremental iterative method, and it is guessed that it is consistent with the ordinary finite element theory. The static analysis steps are as follows:

1. 根据本构关系和尺寸得到K。 1. Obtain K based on the constitutive relationship and size.

2. 由外力平衡得到位移d。 2. Obtain the displacement d by external force balance.

3. 由位移d计算内部剪切应力S13、S23，从而得到节点力。

3. Calculate the internal shear stresses S13 and S23 from the displacement d, thereby obtaining the nodal forces.

4. 求非平衡力，判断收敛，如果收敛，那么结束。

4. Seek the unbalanced force, judge convergence; if it converges, then terminate.

而S13、S23猜测应该是按板壳近似理论得到的与厚度无关的平均应力，真实的剪切应力并不参与迭代。

S13 and S23 are guessed to be the average stresses obtained from the plate-shell approximation theory, which are independent of the thickness, and the actual shear stresses do not participate in the iteration.

在迭代完毕后，如果用户需要输出截面真实的剪切应力，那么根据前面所说的剪力V来计算TSHR13/23。

After the iteration is complete, if the user needs to output the actual shear stress of the section, then calculate TSHR13/23 based on the shearing force V mentioned earlier.

有限元理论基础及Abaqus内部实现方式研究系列15： 壳的剪切应力的图17

1.4 算例 1.4 Example

我们只能验算Abaqus真实剪切应力TSHR13和23的结果，但没有找到方法来验证用于Abaqus流程的剪切应力是取的平均应力还是真实应力，因为Abaqus的S13，S23没有找到方法输出。但通过自编程序iSolver，我们还是计

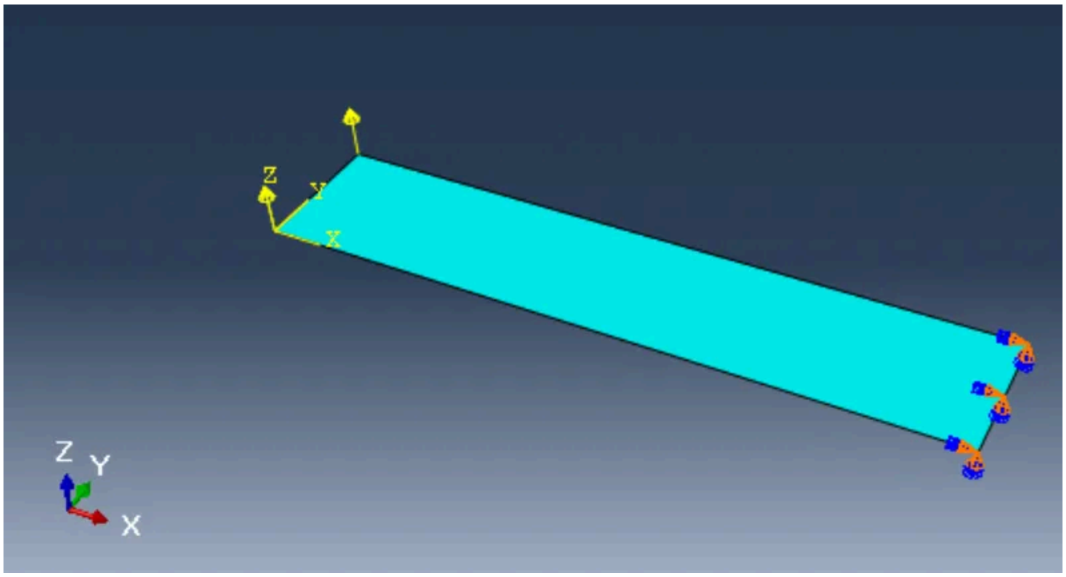
算了横向平均剪切应力的大小，如果有人也自己编程序，遇到类似问题时可以对比一下结果。

We can only verify the results of the true shear stress TSHR13 and 23 in Abaqus, but we have not found a method to verify whether the shear stress used in the Abaqus process is the average stress or the true stress, as there is no method to output S13 and S23 in Abaqus. However, through the self-written program iSolver, we still calculated the size of the lateral average shear stress, and if someone else writes their own program and encounters similar problems, they can compare the results.

有限元理论基础及Abaqus内部实现方式研究系列15： 壳的剪切应力的图18

1.4.1 算例说明 1.4.1 Example Description

只取一个简单的长方形。 Only a simple rectangular shape is taken.



参数如下： Parameters as follows:

尺寸：5X1，厚度0.1。 Dimensions: 5X1, thickness 0.1.

材料：Young’s Modulus 1e8， Poisson Ratio 0.3。

Material: Young's Modulus 1e8, Poisson Ratio 0.3.

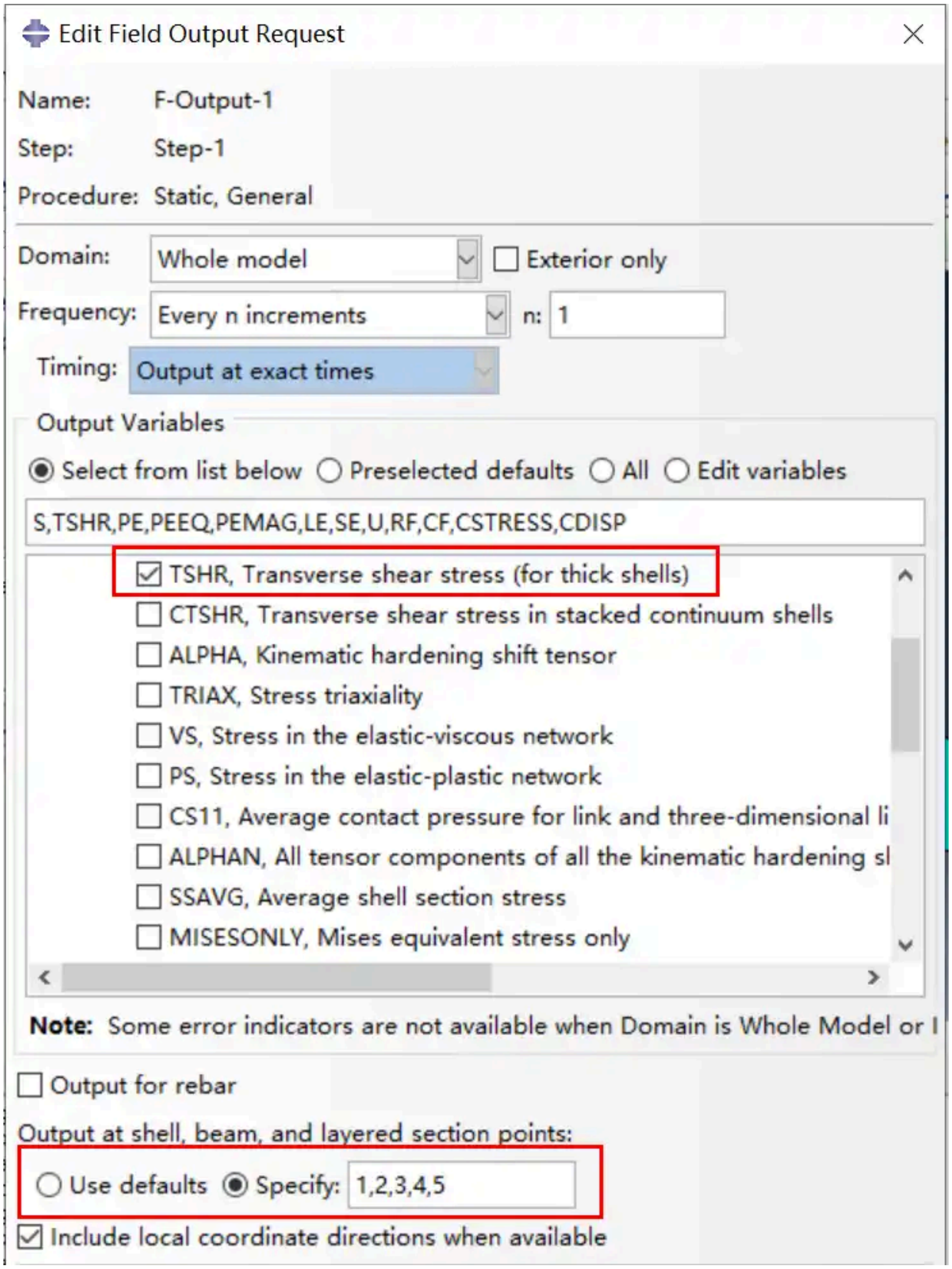
右侧两个节点固支。 The two nodes on the right are fixed.

左侧两个节点每个加集中力1e5，z方向。这样将产生面外弯曲。

Each of the two nodes on the left is subjected to a concentrated force of 1e5 in the z-direction. This will generate out-of-plane bending.

在Step中勾选输出截面应力和输出截面积分点的变量。

Check the variables for output section stress and output section integration points in the Step.



有限元理论基础及Abaqus内部实现方式研究系列15：壳的剪切应力的图23

1.4.2 中面真实剪切应力理论结果 1.4.2 Theoretical Results of Real Shear Stress on Middle Surface

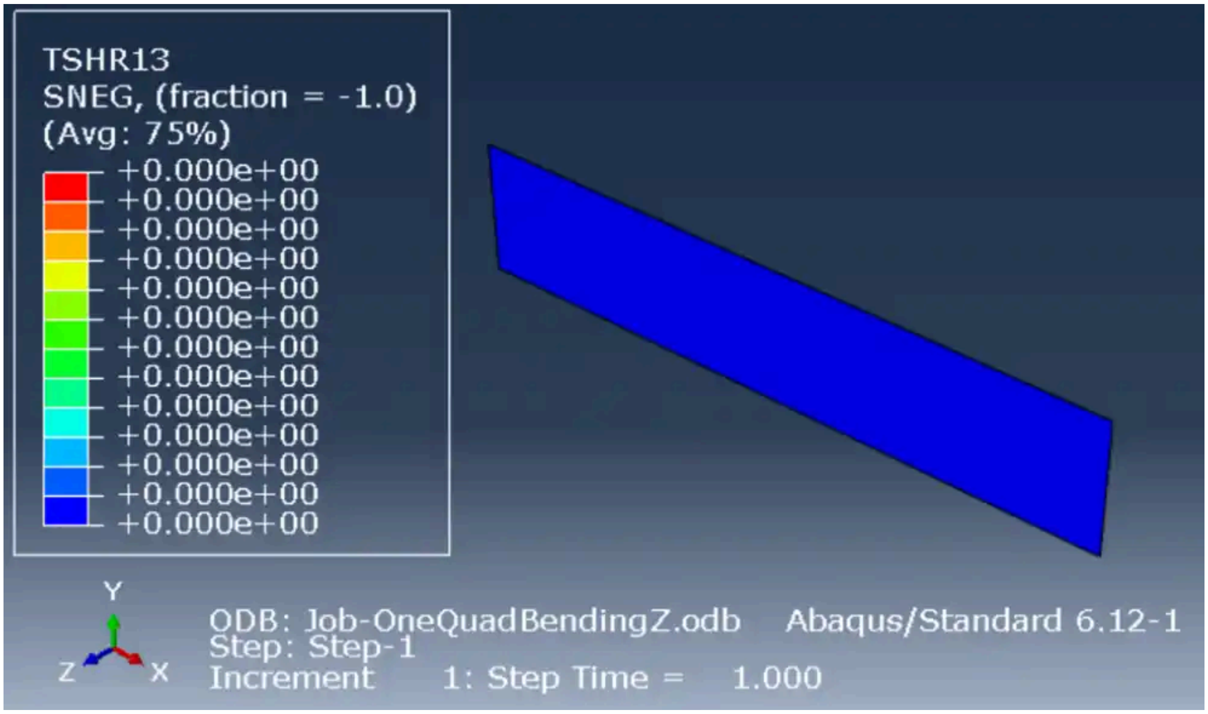
V=100000*2，b=1，h=0.1 V=100000*2, b=1, h=0.1

TauMax=3/2*V/(b*h)=3e6 Max Shear Stress = 3/2 * V / (b*h) = 3e6

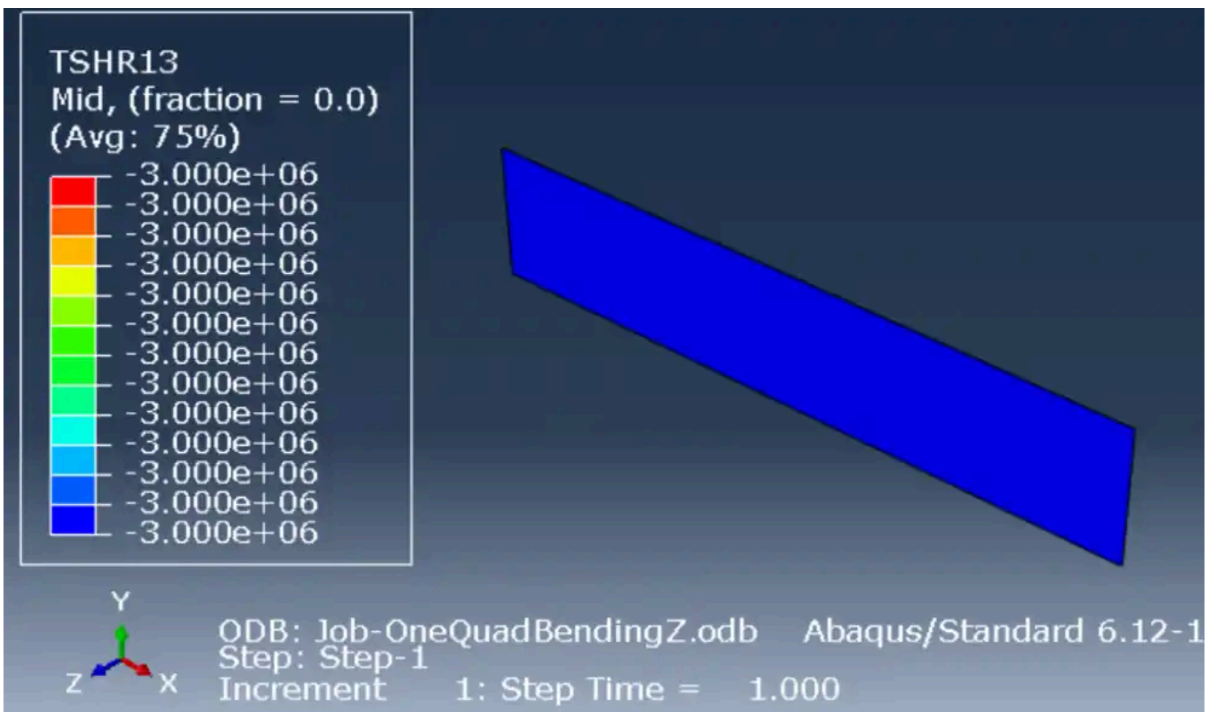
有限元理论基础及Abaqus内部实现方式研究系列15：壳的剪切应力的图24

1.4.3 Abaqus真实剪切应力结果 1.4.3 Abaqus Real Shear Stress Results

下表面TSHR13=0： Bottom Surface TSHR13=0:



中面为3e6，和理论完全一致。 The middle surface is 3e6, which is completely consistent with the theory.



有限元理论基础及Abaqus内部实现方式研究系列15： 壳的剪切应力的图29

1.4.4 平均剪切应力结果 1.4.4 Average Shear Stress Results

Abaqus中无法输出S13和S23，也就是平均剪切应力，为了便于大家理解S13这个值大概是什么量级，我们采用iSolver，在内部计算得到积分点上的剪切应力得到S13=8.1e5，而只有采用平均面积得到的V/(b*h)=2e6的2/5左右，就算加上剪切修正因子5/6也不是简单的按平均面积得到的结果，猜测是因为在这种情况下，壳只在两个节点处加载荷，而不是在一个自由边上均匀加载，而平均面积计算的剪切应力是假定均匀加载得到，所以不一致。当

然，有限元中采用这个积分点应力来求节点力，得到的节点力依然和外力是平衡的。

In Abaqus, it is not possible to output S13 and S23, which are the average shear stresses. To help everyone understand what the magnitude of the S13 value is, we use iSolver to internally calculate the shear stress at the integration points, obtaining S13=8.1e5. However, this is only about 2/5 of the value obtained using the average area, V/(b*h)=2e6. Even when adding the shear correction factor of 5/6, it is not a simple result obtained by averaging the area. It is speculated that this is because in this case, the shell is only loaded at two nodes, rather than uniformly loaded on a free edge. The average area calculated shear stress assumes uniform loading, so there is a discrepancy. Of course, in finite element analysis, using this integral point stress to calculate the nodal forces still results in a balance with the external forces.

有限元理论基础及Abaqus内部实现方式研究系列15： 壳的剪切应力的图30

==总结== ==Summary==

本章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。同时还有两个疑问也希望与大家讨论，得到大家的指点。

This chapter introduces the actual shear stress in shell elements and the shear stress in the plate-shell approximate theory, and also makes a simple guess about the internal implementation process of Abaqus. Finally, a calculation example is used to verify the actual shear stress in Abaqus. In addition, there are also two questions that I hope to discuss with everyone, and I hope to receive everyone's guidance.

（1）我们暂时没有找到方法来验证用于Abaqus流程的剪切应力是取的平均应力还是真实应力，因为Abaqus的S13，S23没有找到方法输出，不知道谁了解怎么输出壳的这两个应力？

(1) We have not found a method to verify whether the shear stress used in the Abaqus process is the average stress or the true stress, because there is no method to output Abaqus' S13 and S23. Does anyone know how to output the two stresses of the shell?

（2）按照下面的公式，Mises力应该包括正应力和剪切应力，但查看壳的Mises应力结果可以看出，壳的Mises力没有计入Tau13，Tau23，不知道为什么这样取Mises力？这样用来校核壳的Mises应力达到屈服是否会有问题？

(2) According to the following formula, the Mises force should include both normal stresses and shear stresses. However, when checking the Mises stress results of the shell, it can be seen that the Mises force does not include Tau13 and Tau23. I don't know why the Mises force is taken this way. Will there be a problem with checking whether the Mises stress of the shell reaches yield by this method?

$$\sigma_v = \sqrt{\frac{1}{2}[(\sigma_{11}-\sigma_{22})^2+(\sigma_{22}-\sigma_{33})^2+(\sigma_{33}-\sigma_{11})^2+6(\sigma_{12}^2+\sigma_{23}^2+\sigma_{31}^2)]}$$

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15： 壳的剪切应力的图33

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

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

第九篇：**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.

有限元理论基础及Abaqus内部实现方式研究系列15：壳的剪切应力的图34
有限元理论基础及Abaqus内部实现方式研究系列15：壳的剪切应力的图35
有限元理论基础及Abaqus内部实现方式研究系列15：壳的剪切应力的图36

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.

http://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