



一种新思路用于实现ABAQUS用户自定义单元 A new approach for implementing ABAQUS user-defined elements



有限元先生 Finite
Element Master



更新于2024年9月19日 10:07 Updated on
September 19, 2024, 10:07

浏览:
1843 Views: 1843

收藏:
9 Favorited: 9

概述：帖子讲述了一种新思路用于实现ABAQUS自定义单元，该方法不需要在INP文件中采用“USER ELEMENT”等关键字定义单元，更重要的是，也**不需要在UEL接口中进行繁琐的FORTRAN编程**，只需要在INP文件中直接导入相应的矩阵即可。目前这种自定义单元适用的计算有：static, frequency extraction, modal dynamic, mode-based steady-state dynamics, complex eigenvalue extraction, and subspace-based steady-state dynamics。更多的功能还在探索之中。

Overview: The post discusses a new approach for implementing ABAQUS user-defined elements. This method does not require the use of keywords such as "USER ELEMENT" in the INP file to define elements, and more importantly, it also does not require cumbersome FORTRAN programming in the UEL interface. It is only necessary to directly import the corresponding matrices in the INP file. Currently, this user-defined element is applicable to calculations such as static, frequency extraction, modal dynamic, mode-based steady-state dynamics, complex eigenvalue extraction, and subspace-based steady-state dynamics. More features are still under exploration.

这种自定义单元方式为多种CAE软件协同二次开发提供了一种可能，众所周知，ABAQUS的非线性方程组求解能力是行业翘楚，这时候如果能得到描述方程组的关键矩阵，如固体力学的运动方程中的KK、MM和CC，直接导入ABAQUS便可以求解，相较于UEL二次开发，节省了向ABAQUS主程序输出关键矩阵的编程工作，至于KK、MM和CC，可以自己采用高级语言编程生成，也可以从其他软件中导出。再比如，COMSOL以其多场耦合计算功能著称，采用该方法就可以将COMSOL的多场耦合功能与ABAQUS结合，**本质是数学中的方程组求解**。

This customized element method provides a possibility for the secondary development of various CAE software. As everyone knows, ABAQUS is a leader in the industry for solving nonlinear equations. At this point, if the key matrices of the equation set, such as KK, MM, and CC in the motion equations of solid mechanics, can be obtained and directly imported into ABAQUS for solving, it saves the programming work of outputting the key matrices to the main program of ABAQUS compared to UEL secondary development. As for KK, MM, and CC, they can be generated by programming in advanced languages or exported from other software. For example, COMSOL is renowned for its multi-field coupling calculation capabilities. Using this method, the multi-field coupling function of COMSOL can be combined with ABAQUS, which is essentially equation set solving in mathematics.

这种自定义单元完美解决了ABAUS传统的用于自定义单元无法完成可视化的问题。搞过UEL二次开发的人都被自定义单元的可视化问题困扰，要么是将数据导出到第三方软件进行处理，要么是采用UMAT套一层单元进行

可视化，这两种方法都需要大量的编程工作。

This customized element perfectly solves the problem of visualization that traditional ABAQUS custom elements cannot achieve. Those who have done UEL secondary development have been troubled by the visualization problem of custom elements, either exporting data to third-party software for processing or using UMAT to add a layer of element for visualization, both of which require a lot of programming work.

这种自定义单元完美的解决了ABAQUS传统的用户自定义单元无法施加面力、体力等复杂力的短板。

This customized element perfectly solves the shortcoming of traditional ABAQUS user-defined elements in applying complex forces such as surface forces and body forces.

更多的功能还在探索之中... More functions are still under exploration...

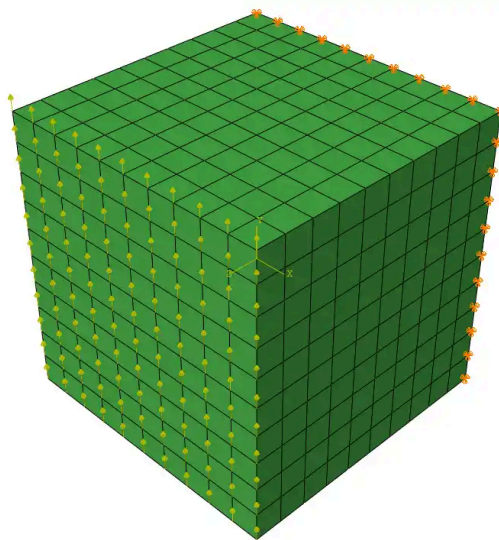
下面以一个悬臂端受切向力作用为例，讲解实现方法：

Taking a cantilever end subjected to a tangential force as an example, this paper explains the implementation method:

() 模型信息说明 Model information description

模型尺寸为10x10x10，弹性模量1e10，密度2400，泊松比0.24，一端完全固定，另一端受切向力作用，切向力以节点集合的形式添加，力幅为100，边界条件和荷载示意图为：

The model size is 10x10x10, with an elastic modulus of 1e10, density of 2400, Poisson's ratio of 0.24, one end completely fixed, and the other end subjected to a tangential force. The tangential force is added in the form of a node set, with an amplitude of 100. The boundary conditions and load diagram are as follows:



设置两种工况： Two working conditions are set:

- 1、ABAQUS的C3D8单元计算。 1. Calculation of ABAQUS C3D8 element.
- 2、采用新型自定义单元，新型自定义单元的刚度矩阵和质量矩阵采用MATLAB自编程序，然后输出**ABAQUS可以识别的外部文件形式**，并手动修改INP文件。
2. Using a new custom element, the stiffness matrix and mass matrix of the new custom element are calculated using a MATLAB program, and then output in an external file format that ABAQUS can recognize, and manually modify the INP file.

() INP文件的具体修改说明。 () Specific modification instructions for the INP file.

(1)、删除INP文件中的PART ASSEMBLY等关键词，即不采用PART建模方式。

(1) Delete keywords such as PART ASSEMBLY from the INP file, i.e., do not use PART modeling method.

(2)、添加如下关键字： (2) Add the following keywords:

*MATRIX INPUT, NAME=kk, INPUT=kk. txt, TYPE=UNSYMMETRIC

*MATRIX INPUT, NAME=mm, INPUT=mm. txt, TYPE=UNSYMMETRIC

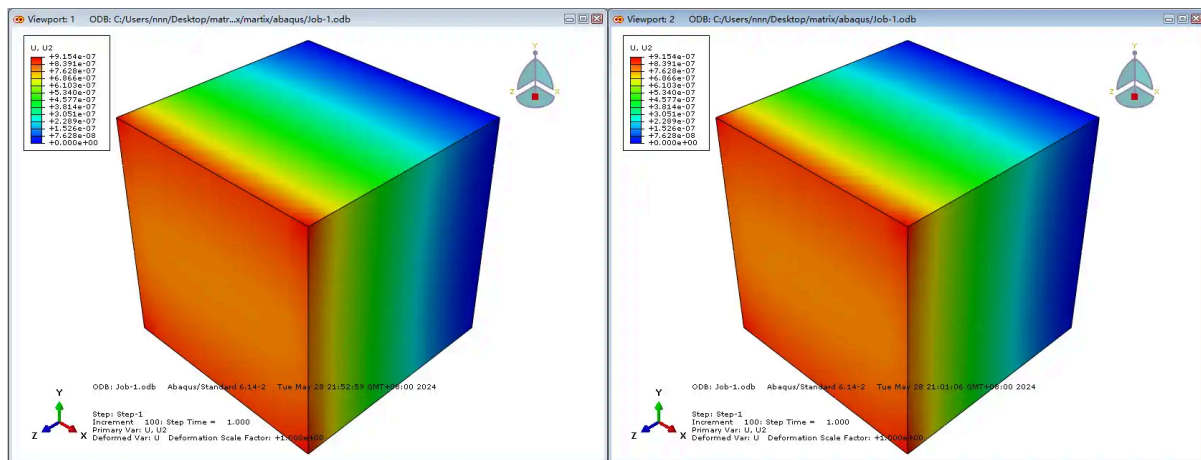
*MATRIX ASSEMBLE, STIFFNESS=kk

*MATRIX ASSEMBLE, mass=mm

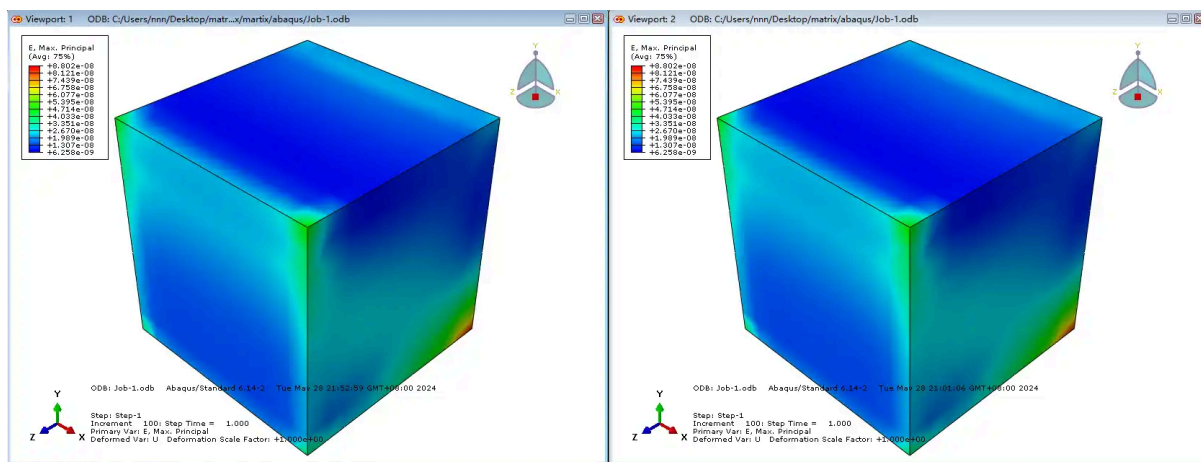
(3)、将属性定义部分的数值修改为极小值 (3) Modify the values in the property definition section to the minimum values

() 计算结果 () Calculation results

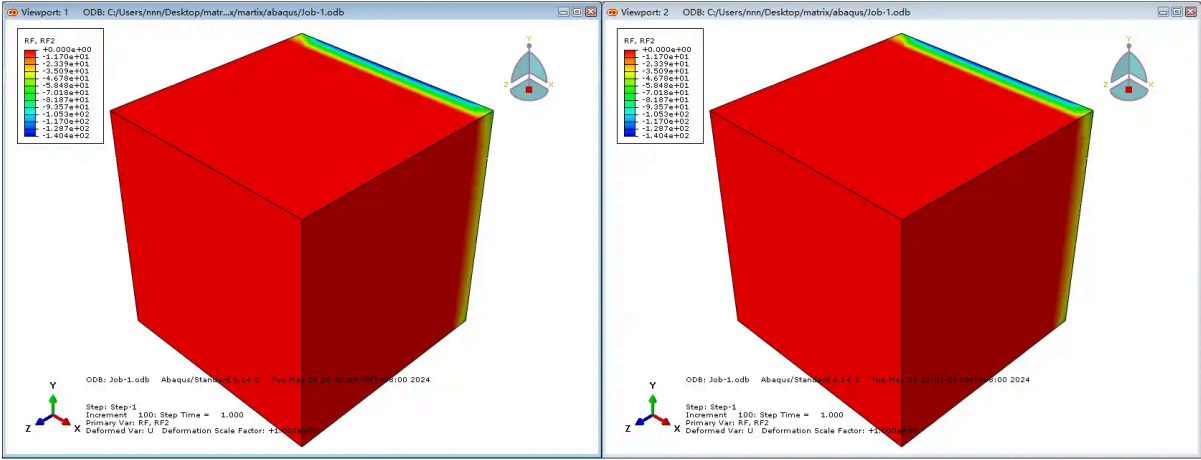
加载向位移云图对比如下： The comparison of displacement cloud diagrams is as follows:



第一主应变云图对比如下： The comparison of the first principal strain cloud diagrams is as follows:



加载向支反力云图对比如下： The comparison of the load reaction cloud diagrams is as follows:



点击链接获取更多内容： Click the link for more content:

一种新思路用于实现ABAQUS用户自定义单元 A new approach for implementing ABAQUS user-defined elements

推荐阅读 Recommended Reading

【专题课程】ANSA HEXABLOCK六面体网格划分专题(完结)...

Wonderful仿真 Wonderful simulation

¥399 \$399

非局部均值滤波和MATLAB程序详解视频算法及其保留图形细节应用...

正一算法程序 Zhengyi Algorithm Program

¥220 220 Yuan

车身设计系列视频之车身钣金顶盖横梁正向设计实例教程...

京迪轩 Jing Di Xuan

¥15 15 Yuan

hypermesh-cfd网格划分 HyperMesh CFD mesh

freshman

¥5