【JY】 ABAQUS子程序UEL的有限元原理与应用

【JY】ABAQUS Subroutine UEL: Finite Element Principles and Applications

建源之光 - 减隔震 Jianyuan Light - Vibration Reduction and Isolation

2022年12月19日 06:08 December 19, 2022 06:08
浏览: 2667 Views: 2667
评论: 3 Comment: 3
收藏: 33 Favorited: 33

不等待 Do not wait **即关注 Immediate attention**

【简述ABAQUS中UEL子程序】 Brief introduction to the UEL subroutine in ABAQUS

ABAQUS作为成熟的商用有限元软件，可为高级用户提供特定的分析需求。ABAQUS常见的二次开发子程序包括：UMAT、VUMAT、UGENS、UEL和VUEL等。其中UEL/VUEL分别适用于ABAQUS的Standard/Explicit求解器。只有清楚有限元分析的基本原理，才能够较好地了解其分析的力学原理，才能对

特定的分析需求编写合适的分析单元。

ABAQUS, as a mature commercial finite element software, can provide specific analysis needs for advanced users. Common secondary development subroutines in ABAQUS include: UMAT, VUMAT, UGENS, UEL, and VUEL, among others. UEL/VUEL are respectively applicable to ABAQUS's Standard/Explicit solvers. Only by understanding the basic principles of finite element analysis can one better understand the mechanical principles of its analysis and be able to write suitable analysis elements for specific analysis needs.

本文对ABAQUS/Standard中UEL子程序的有限元原理及编写原则进行初步梳理，并编写了平面三角形单元静力分析的UEL子程序，以加深对UEL的理解与认识。

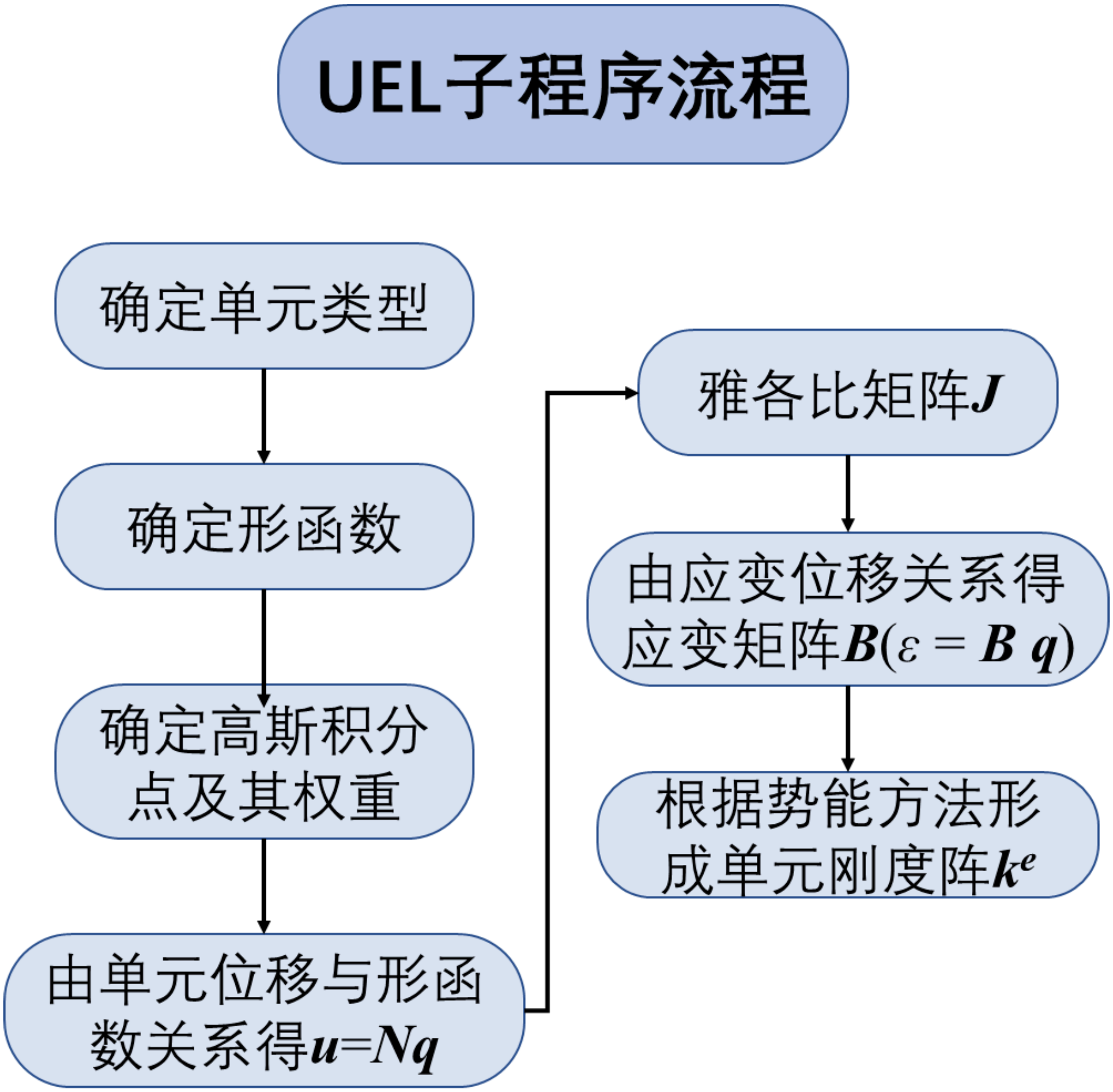
This article provides an initial overview of the finite element principles and writing principles of the UEL subroutine in ABAQUS/Standard, and also writes a UEL subroutine for the static analysis of plane triangular elements to deepen the understanding and recognition of UEL.

【ABAQUS的UEL单元之有限元基本原理简述】

Brief Introduction to the Finite Element Principles of ABAQUS UEL Elements

当用户需要用ABAQUS/Standard单元库中没有的单元进行分析时，可通过ABAQUS提供的UEL子程序接口进行二次开发，编写适用于特定分析的单元。下面以平面三角形单元为例，给出编写UEL单元的基本流程。

When users need to perform analysis with elements not available in the ABAQUS/Standard element library, they can carry out secondary development through the UEL subroutine interface provided by ABAQUS to write elements suitable for specific analysis. The following takes the plane triangular element as an example to give the basic process of writing UEL elements.



系统的势能Π由下式给出 The potential energy Π of the system is given by the following formula

$$\Pi=\sum_{e}\frac{1}{2}\int_{e}\varepsilon^{T}D\varepsilon t dA-\sum_{e}\int_{e}u^{T}ftdA-\int_{L}u^{T}Ttdl-\sum_{i}u_{i}^{T}P_{i}$$

其中

$$U_{e}=\frac{1}{2}\int_{e}\varepsilon^{T}\boldsymbol{D}\varepsilon t\mathrm{d}A$$

为单元的应变能。 The strain energy of the element is given here.

将应变-位移关系$\varepsilon=Bq$代入U_e中，由此对于势能方法形成的刚度矩阵如下：

Substitute the strain-displacement relationship ε = Bq into U_e to obtain the stiffness matrix formed by the potential energy method as follows:

$$U_{e}=\frac{1}{2}q^{T}B^{T}DB(\int_{e}Adx)q=\frac{1}{2}q^{T}k^{e}q$$

$$k^{e}=B^{T}DB(\int_{e}Adx)=t_{e}A_{e}B^{T}DB$$

其中 t_e 为单元厚度、 A_e 为单元面积。

Here, t_e represents the element thickness, and A_e represents the element area.

有限元分析的第一步是将实体离散化为多个单元，此后构造出各个单元的单元刚度矩阵，在将单元刚度矩阵集成整体的刚度矩阵，最后通过整体刚度矩阵建立平衡方程从而求解各个节点的位移、应力、应变等响应。因此，根据有限元的分析原理，编写UEL的最终目的即是形成目标单元的单元刚度矩阵。以下对平面常应变三角形单元（CST）进行有限元分析，依据有限元分析思路编写UEL子程序，以初步了解有限单元的分析原理及UEL编写流程。

The first step in finite element analysis is to discretize the solid into multiple elements. Subsequently, the element stiffness matrices of each element are constructed, and then integrated into the overall stiffness matrix. Finally, the equilibrium equations are established through the overall stiffness matrix to solve the displacements, stresses, and strains of each node. Therefore, according to the analysis principles of the finite element method, the ultimate goal of writing the UEL is to form the element stiffness matrix of the target element. The following is a finite element analysis of the plane constant strain triangular element (CST), based on the finite element analysis approach to write the UEL subroutine, in order to initially understand the analysis principles and UEL writing process of the finite element method.

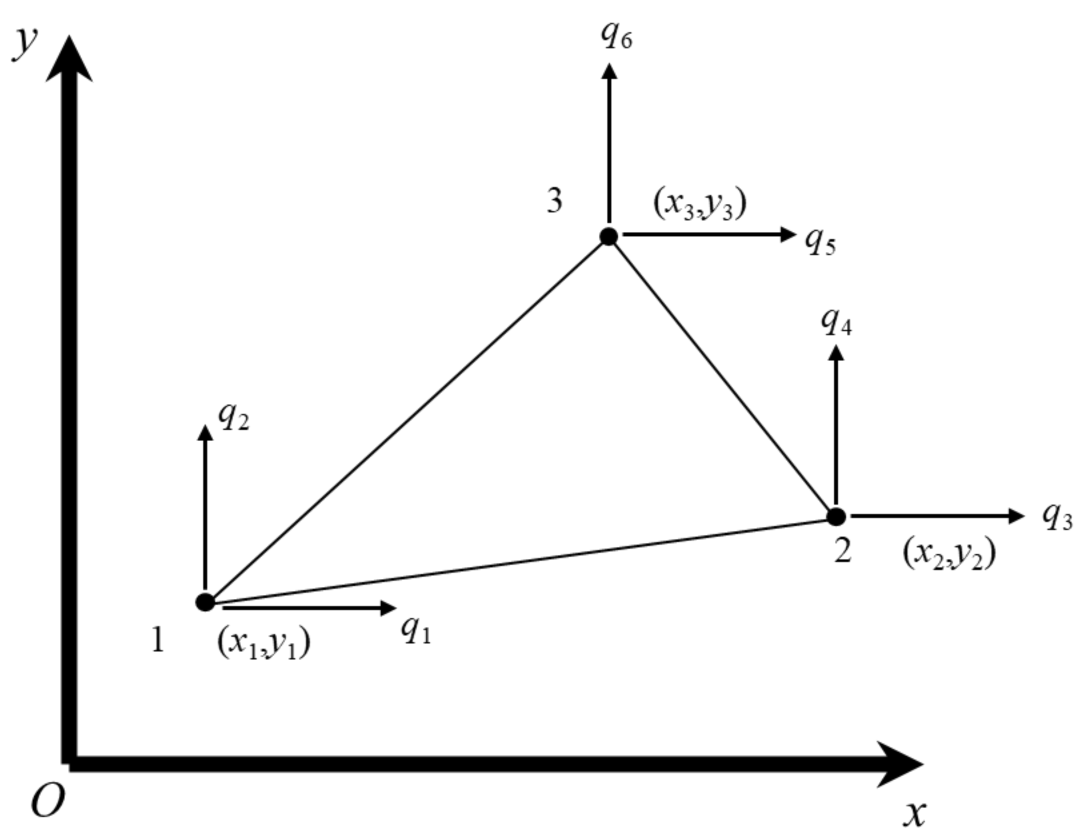


图2 平面三角形单元示意图 Figure 2 Schematic diagram of a plane triangular element

上图所示为平面三角形单元，由图示信息可得其节点坐标列阵 $\boldsymbol{u}$ 与位移列阵 $\boldsymbol{q}$ 分别为

The figure above shows a planar triangular element, from the information shown, the node coordinate array u and the displacement array q are as follows

$$u=(x_1,y_1,x_2,y_2,x_3,y_3)^T,\quad q=(q_1,q_2,q_3,q_4,q_5,q_6)^T$$

1.形函数 1. Shape functions

设三角形单元节点1、2、3对应的形函数分别为*N1*、*N2*、*N3*，形函数满足条件：N1+N2+N3=1（在点i处，形函数Ni值为1，其余形函数值为0），形函数采用自然坐标 ξ 和 η 描述，有

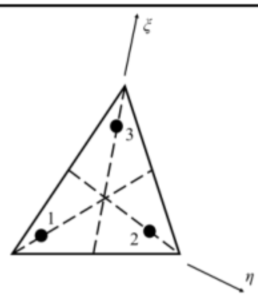
Let the shape functions corresponding to the triangular element nodes 1, 2, and 3 be N1, N2, and N3, respectively. The shape functions satisfy the condition: N1 + N2 + N3 = 1 (at point i, the value of the shape function Ni is 1, and the values of the other shape functions are 0). The shape functions are described using natural coordinates ξ and η, and have

$$N_1 = \xi, N_2 = \eta, N_3 = 1 - \xi - \eta$$

2.高斯积分点及权重 2. Gaussian integration points and weights

此平面三角形单元所选取的高斯积分点及权重如下： The Gaussian integration points and weights selected for this plane triangular element are as follows:

表1高斯积分点及权重 Table 1 Gaussian integration points and weights

积分点	ξ 坐标	η 坐标	积分点权重
	$\xi_1=1/6$ $\xi_2=2/3$ $\xi_3=1/6$	$\eta_1=\xi_1$ $\eta_2=\xi_1$ $\eta_3=\xi_2$	$\omega_1=1/3$ $\omega_2=1/3$ $\omega_3=1/3$

3.形函数与单元位移关系 3. Shape functions and element displacement relationship

得到形函数与单元节点位移后，根据等参元表示方法，单元内任意一点的位移都可用形状函数和未知结点位移场进行表示，有

After obtaining the shape functions and nodal displacements, according to the isoparametric element representation method, the displacement of any point within the element can be expressed using the shape functions and the unknown nodal displacement field.

$$\begin{cases} u = N_1 q_1 + N_2 q_2 + N_3 q_3 \\ v = N_4 q_4 + N_5 q_5 + N_6 q_6 \end{cases}$$

可表示为$\boldsymbol{u} = N\boldsymbol{q}$，其中*N*为 It can be expressed as u = Nq, where N is

$$N = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix}$$

对于三角形单元，可将x、y坐标表示为节点坐标的形式，可由此得到单元坐标的插值关系为：

For triangular elements, the x, y coordinates can be expressed in terms of nodal coordinates, from which the interpolation relationship of the element coordinates can be obtained:

$$\begin{cases} x = N_1 x_1 + N_2 x_2 + N_3 x_3 = x_{13}\xi + x_{23}\eta + x_3 \\ y = N_4 y_4 + N_5 y_5 + N_6 y_6 = y_{13}\xi + y_{23}\eta + y_3 \end{cases}$$

其中 Among

$$x_{ij} = x_i - x_j, y_{ij} = y_i - y_j$$

4.雅各比矩阵J 4. Jacobian Matrix J

由节点坐标x、y与ξ、η关系，在根据链式求偏导法则，可得到平面三角形单元的雅可比矩阵

The relationship between node coordinates x, y and ξ, η can be used, based on the chain rule of differentiation, to obtain the Jacobian matrix of a plane triangular element

$$J=\begin{pmatrix}\frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi}\\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta}\end{pmatrix}=\begin{pmatrix}x_{13} & y_{13}\\ x_{23} & y_{23}\end{pmatrix}$$

5.应变矩阵B 5. Strain Matrix B

根据应变ε与节点坐标u、节点位移q的关系可得应变计算公式为

The strain calculation formula can be obtained according to the relationship between strain ε and node coordinates u, and node displacement q

$$\varepsilon=\frac{1}{\det J}\begin{pmatrix}y_{23}q_1+y_{31}q_3+y_{12}q_5\\ x_{23}q_2+x_{31}q_4+x_{12}q_6\\ x_{32}q_1+x_{23}q_2+x_{13}q_3+y_{31}q_4+y_{21}q_5+y_{12}q_6\end{pmatrix}$$

其中xij与yij计算公式同上。写成矩阵形式为

The calculation formula for xij and yij is the same as above. It can be written in matrix form as

$$\varepsilon=Bq$$

由此可导出单元应变-位移矩阵B，其表达式为 From this, the element strain-displacement matrix B can be derived, whose expression is

$$B=\frac{1}{\det J}\begin{pmatrix}y_{23} & 0 & y_{31} & 0 & y_{12} & 0\\ 0 & x_{32} & 0 & x_{13} & 0 & x_{21}\\ x_{32} & y_{23} & x_{13} & y_{31} & x_{21} & y_{12}\end{pmatrix}$$

由此可看出B矩阵中所有元素均为常数，且每个常数均是通过节点坐标表示的。

It can be seen from this that all elements in the B matrix are constants, and each constant is represented by the node coordinates.

6.弹性矩阵D 6. Elastic matrix D

弹性矩阵**D**根据所研究问题为平面应力或平面应变问题有不同形式，其取值需根据所用材料确定，具体如下：

The elastic matrix D takes different forms depending on whether it is a plane stress or plane strain problem, and its value needs to be determined according to the material used, as follows:

$$\text{平面应力：}D=\frac{E}{1-v}\begin{pmatrix}1 & v & 0\\ v & 1 & 0\\ 0 & 0 & \frac{1-v}{2}\end{pmatrix}$$

$$\text{平面应变：}D=\frac{E}{(1+v)(1-2v)}\begin{pmatrix}1-v & v & 0\\ v & 1-v & 0\\ 0 & 0 & \frac{1}{2}-v\end{pmatrix}$$

7.应变-位移矩阵 7. Strain-Displacement Matrix

由已有的应变矩阵**B**与应力应变关系，得应变位移矩阵 $\sigma = \boldsymbol{D}\varepsilon = \boldsymbol{D}\ \boldsymbol{B}\boldsymbol{q}$ 。

From the existing strain matrix B and the stress-strain relationship, the strain-displacement matrix σ = Dε = D B q is obtained.

8.单元刚度矩阵与残余力向量 8. Element Stiffness Matrix and Residual Force Vector

根据势能方法导出得到三角形单元的单元刚度矩阵 k^e ，有

The element stiffness matrix k^e of the triangular element is derived from the potential energy method.

$$k^e = t^e A^e B^T D B$$

其中 t^e 为单元厚度 。

Among them, t^e is the element thickness.

最后，通过单元的边界条件可得单元的残余力向量。至此，UEL单元的编写完成，可进行初步调试直至结果无误。

Finally, the residual force vector of the element can be obtained through the element boundary conditions. Up to this point, the writing of the UEL element is completed, and it can be debugged preliminarily until the results are correct.

【 UEL的编写基本原则】 Basic Principles of UEL Writing

ABAQUS的子程序可通过FORTRAN 77/95 进行编写，本文采用FORTRAN77进行编写。

The ABAQUS subroutine can be written in FORTRAN 77/95, and this article uses FORTRAN 77 for writing.

在FORTRAN中编写UEL，需在以下框架中进行，才可被ABAQUS识别。以下框架也即是ABAQUS的UEL接口（见ABAQUS6.14 User Subroutines Reference Guide 1.1.28 UEL）。

In FORTRAN, UEL needs to be written within the following framework to be recognized by ABAQUS. This framework is also known as the ABAQUS UEL interface (see ABAQUS 6.14 User Subroutines Reference Guide 1.1.28 UEL).

```
SUBROUTINE UEL(RHS,AMATRX,SVARS,ENERGY,NDOFEL,NRHS,NSVARS,
     1 PROPS,NPROPS,COORDS,MCRD,NNODE,U,DU,V,A,JTYPE,TIME,DTIME,
     2 KSTEP,KINC,JELEM,PARAMS,NDLOAD,JDLTYP,ADLMAG,PREDEF,NPREDF,
     3 LFLAGS,MLVARX,DDLMAG,MDLOAD,PNEWDT,JPROPS,NJPROP,PERIOD)
C
      INCLUDE 'ABA_PARAM.INC'
C
      DIMENSION RHS(MLVARX,*),AMATRX(NDOFEL,NDOFEL),PROPS(*),
     1 SVARS(*),ENERGY(8),COORDS(MCRD,NNODE),U(NDOFEL),
     2 DU(MLVARX,*),V(NDOFEL),A(NDOFEL),TIME(2),PARAMS(*),
     3 JDLTYP(MDLOAD,*),ADLMAG(MDLOAD,*),DDLMAG(MDLOAD,*),
```

```
 4 PREDEF(2,NPREDF,NNODE),LFLAGS(*),JPROPS(*)

  user coding to define RHS, AMATRX, SVARS, ENERGY, and PNEWDT
```
（此部分是我们所要编写的内容）

```
  RETURN
  END
```

UEL的编写可分为以下几个部分内容： The writing of UEL can be divided into the following parts:

1.定义变量 1. Define Variables

编写UEL子程序最重要的工作是形成单元的刚度矩阵AMATRX与残余力向量RHS。单元刚度矩阵AMATRX与残余力向量RHS的形成可按照上一小节给出的流程进行。UEL子程序框架中出现的如SVARS、ENERGY、COORDS等变量无需定义（UEL子程序框架中各变量的具体含义见帮助文档），可直接赋值。

The most important task in writing the UEL subroutine is to form the element stiffness matrix AMATRX and the residual force vector RHS. The formation of the element stiffness matrix AMATRX and the residual force vector RHS can be carried out according to the process given in the previous section. Variables such as SVARS, ENERGY, COORDS, etc., that appear in the UEL subroutine framework do not need to be defined (the specific meanings of each variable in the UEL subroutine framework can be found in the help document), and can be assigned directly.

2.变量赋值与信息传入 2. Variable Assignment and Information Transmission

在开始采用所编写的UEL单元时，需要从inp文件中传入单元所用材料的基本属性，如密度、弹性模量及泊松比等。同时，在inp文件中需要指定UEL单元的单元结点数NODES、单元类型命名TYPE、基本属性个数PROPERTIES、空间维数COORDINATES、产生的变量个数VARIABLES、单元节点编号*ELEMENT等信息。

When starting to use the UEL element that has been written, the basic properties of the material used by the element, such as density, elastic modulus, and Poisson's ratio, need to be transmitted from the inp file. At the same time, in the inp file, it is necessary to specify the number of element nodes NODES, the name of the element type TYPE, the number of basic properties PROPERTIES, the spatial dimension COORDINATES, the number of generated variables VARIABLES, and the element node numbering *ELEMENT for the UEL element.

参考Abaqus6.14帮助文档中调用UEL子程序时inp文件所用的命令流：

Refer to the Abaqus 6.14 help document for the command sequence used in the inp file when calling the UEL subroutine:

*USER ELEMENT, NODES=2, TYPE=U1, PROPERTIES=4, COORDINATES=3,

VARIABLES=12

1, 2, 3

*ELEMENT, TYPE=U1

101, 101, 102

*ELGEN, ELSET=UTRUSS

101, 5

*UEL PROPERTY, ELSET=UTRUSS

0.002, 2.1E11, 0.3, 7200.

上述命令表示所采用的UEL单元为两点的线性杆件单元，其中ELGEN表示逐步增加单元数量（详见帮助文档关键词解释*ELGEN）。要将inp文件中的单元特性传入UEL，可在UEL中写入以下命令：

The above command indicates that the UEL element used is a two-node linear bar element, where ELGEN represents the gradual increase in the number of elements (see the help document for keyword explanations *ELGEN). To pass the element properties from the inp file to UEL, the following command can be written in UEL:

AREA= PROPS(1) ! Cross-sectional area

AREA= PROPS(1) ! Cross-sectional area

E = PROPS(2) ! Young's modulus

E = PROPS(2) ! Young's modulus

MIU = PROPS(3) ! Poisson's ratio

MIU = PROPS(3) ! Poisson's ratio

RHO = PROPS(4) ! Density

RHO = PROPS(4) ! Density

3.变量计算 3. Variable Calculation

①根据前述有限元基本原理，计算各个变量，并形成刚度矩阵

① According to the aforementioned finite element basic principles, calculate each variable and form the stiffness matrix

②在编写UEL过程中，可通过输出每一步的计算变量，以方便后期debug判断错误所在的位置，每个输出变量可在.log文件中查看。采用FORTRAN77 输出变量的方式：

② During the process of writing the UEL, you can output the calculation variables at each step to facilitate later debugging and judgment of the location of errors. Each output variable can be viewed in the .log file. Use the FORTRAN77 method to output variables:

（假设变量为雅各比矩阵Jacobi）PRINT *, 'Jacobi=' ,Jacobi

(Assuming the variable is the Jacobian matrix) PRINT *, 'Jacobi=' ,Jacobi

4.编写完成，开始debug 4. The writing is completed, start debugging

编写完UEL子程序之后，先将UEL子程序应用于单个单元进行测试（从静力、线性摄动、动力时程等各个方面逐一测试），在保证单个单元的计算结果正确无误的前提下，再将UEL子程序应用于较多单元的分析案

例中，可保证分析结果的准确性。

After writing the UEL subroutine, first apply the UEL subroutine to a single element for testing (test from static, linear perturbation, dynamic time history, etc., one by one), and ensure that the calculation results of the single element are correct and accurate before applying the UEL subroutine to the analysis of more elements, which can ensure the accuracy of the analysis results.

【在ABAQUS中的UEL单元研究】 Research on UEL Elements in ABAQUS

根据以上平面三角形单元的有限元分析思路，编写对应的平面二维三角形单元UEL子程序，并通过两个不同的有限元分析算例验证该子程序的有效性。算例中采用的三角形单元为ABAQUS单元库中的二维三角形单元CPS3。

Based on the finite element analysis ideas of the above plane triangular elements, write the corresponding plane two-dimensional triangular element UEL subroutine, and verify the effectiveness of the subroutine through two different finite element analysis examples. The triangular elements used in the examples are the two-dimensional triangular elements CPS3 from the ABAQUS element library.

算例一：平面三角形单元 Example 1: Plane triangular element

一、三角形单元有限元计算参数 Section 1: Finite Element Calculation Parameters of Triangular Elements

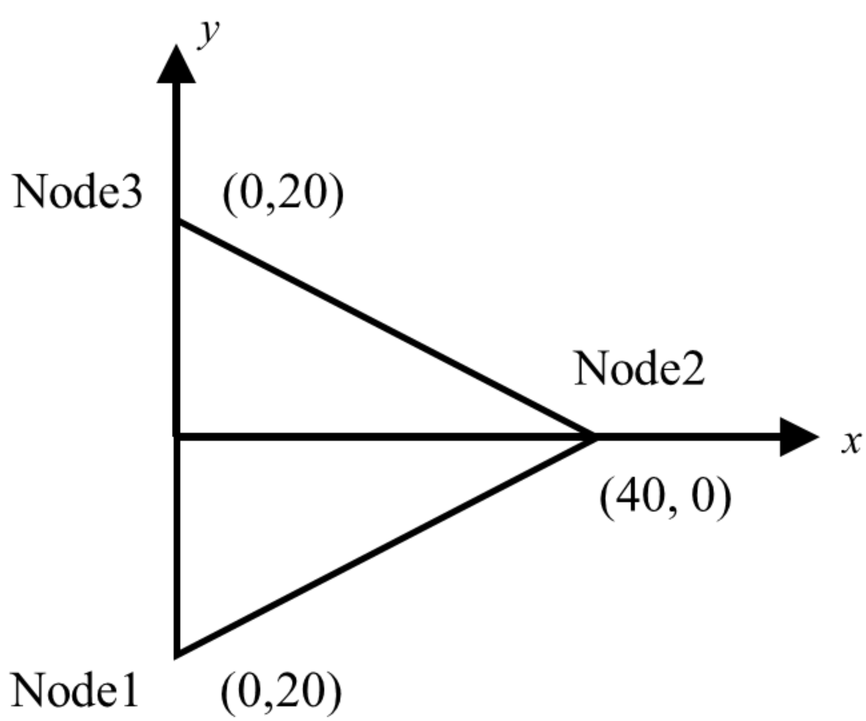


图3 平面三角形单元示意图 Figure 3 Schematic diagram of a planar triangular element

三角形单元的相关计算参数如表2所示： The relevant calculation parameters of the triangular element are shown in Table 2:

表2 三角形单元计算参数 Table 2: Calculation Parameters of Triangular Elements

弹性模量/Mpa	泊松比	三角形单元长度/mm	单元厚度/mm
210000	0.025	20	20

该三角形单元共有三节点（如图1所示），每个节点有2个自由度

This triangular element has three nodes (as shown in Figure 1), each node has two degrees of freedom

二、边界条件及载荷设置 Section 2: Boundary Conditions and Load Settings

将节点1与节点3的位移自由度施加约束，在节点2的水平x方向施加100N的节点力。

Apply displacement constraints to the displacement degrees of freedom of node 1 and node 3, and apply a 100N node force in the horizontal x direction at node 2.

三、计算结果对比 Section 3: Comparison of Calculation Results

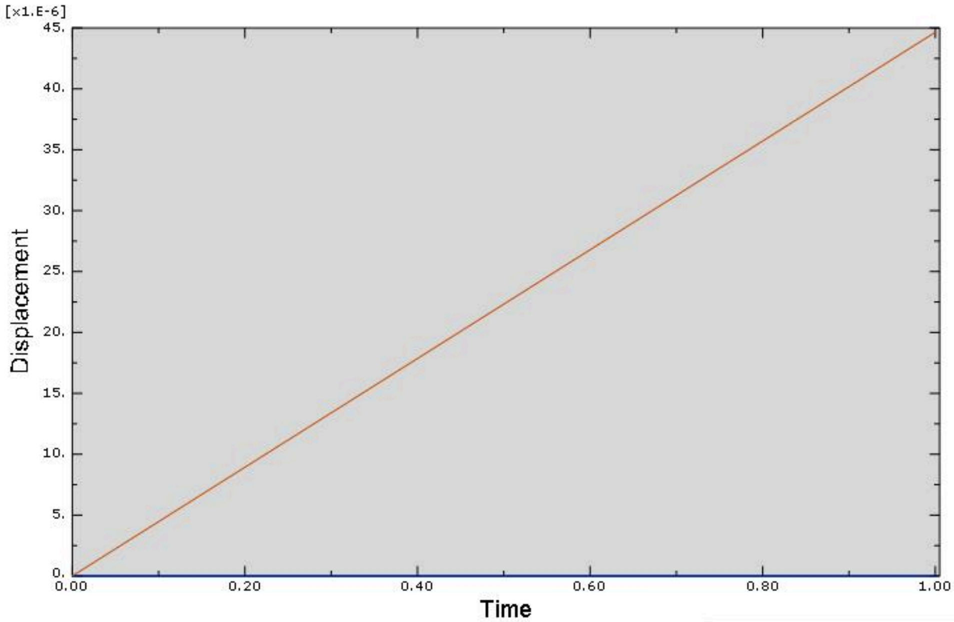
在相同边界条件、载荷条件下，二维三角形单元的UEL子程序计算结果与有限元ABAQUS计算结果如下表3及图4-5所示，从图中可看出，子程序的位移计算结果与abaqus中CPS3单元位移计算结果一致。

Under the same boundary conditions and loading conditions, the calculation results of the UEL subroutine for two-dimensional triangular elements are shown in Table 3 and Figures 4-5. From the figures, it can be seen that the displacement calculation results of the subroutine are consistent with the displacement calculation results of the CPS3 element in ABAQUS.

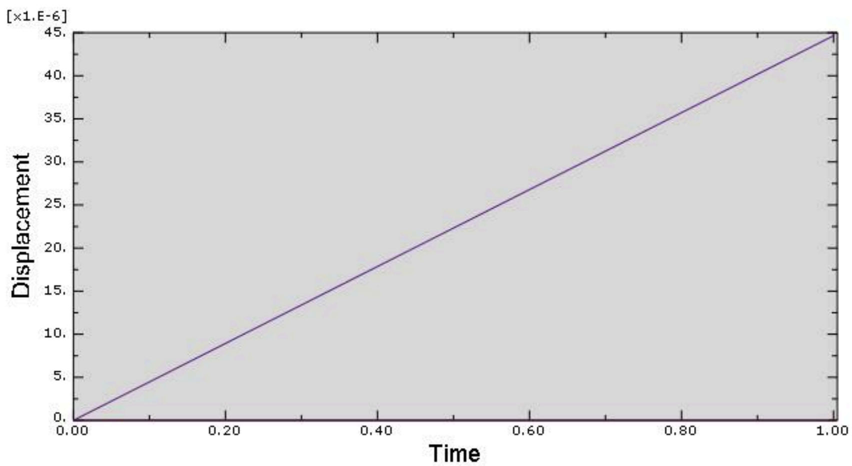
表3 UEL子程序与abaqus有限元最大位移对比

Table 3 Comparison of Maximum Displacement between UEL Subroutine and ABAQUS Finite Element

	节点编号	最大位移(U1)/mm
CPS3 单元	1	0
	3	4.4643E-05
UEL 子程序	1	0
	3	4.4643E-05



(a)CPS3加载时程 (a) CPS3 loading process



(b)UEL加载时程 (b) UEL loading process

图4两种不同单元的节点位移时程曲线 Figure 4: Displacement time history curves of two different elements

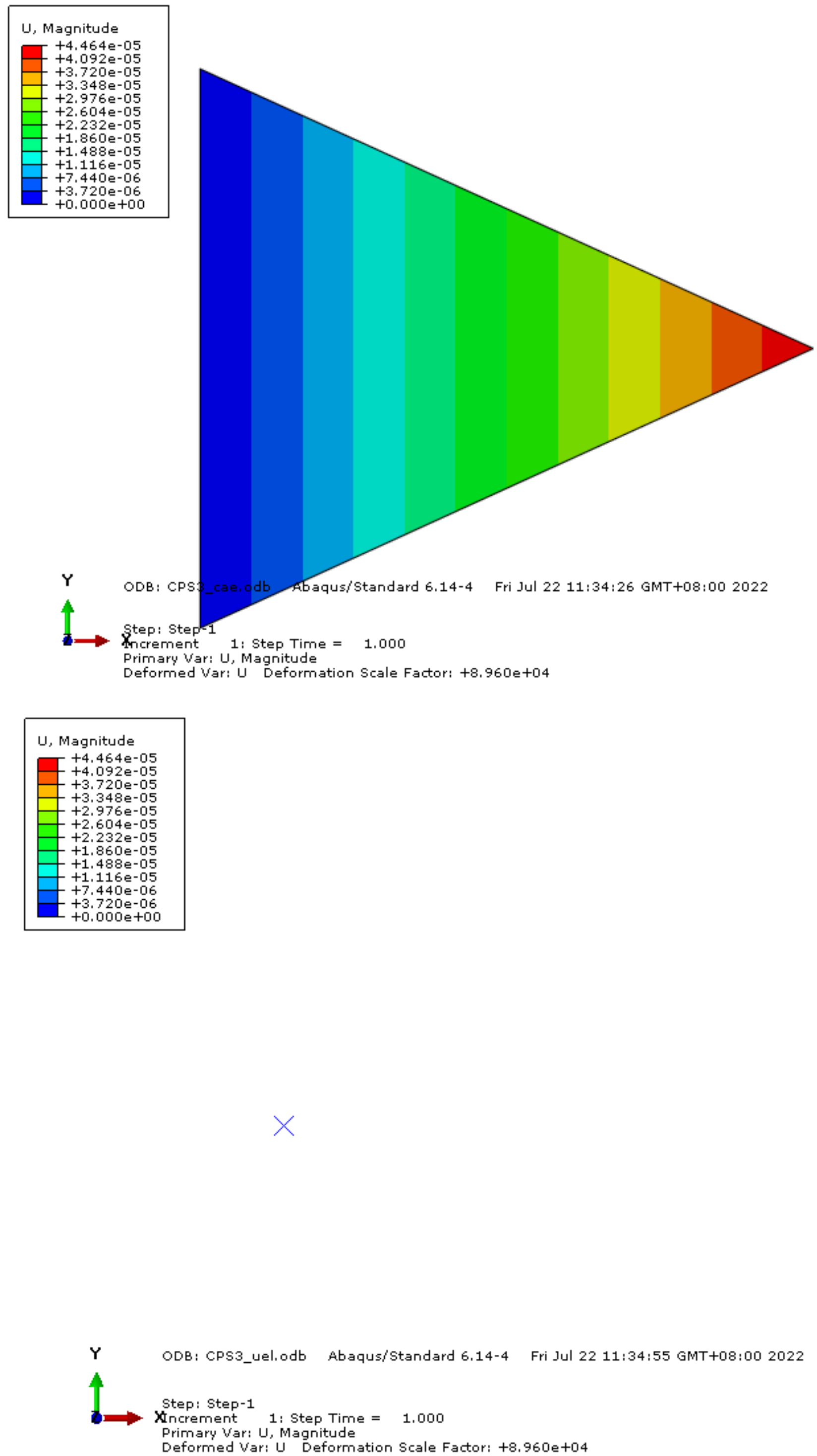


图5 ABAQUS杆单元计算结果与UEL单元计算结果对比

Figure 5 Comparison of ABAQUS bar element calculation results and UEL element calculation results

算例二：平面三角形单元组合 Example 2: Plane triangular element assembly

一、三角形单元有限元计算参数 Section 1: Finite Element Calculation Parameters of Triangular Elements

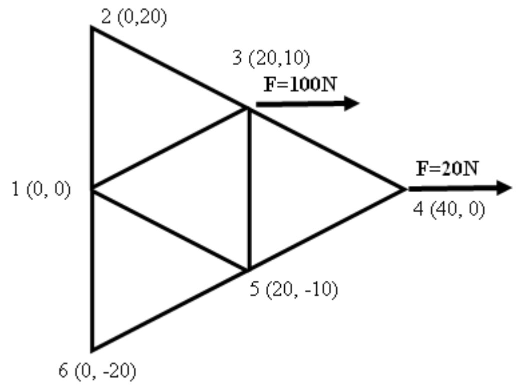


图6平面三角形结构 Figure 6 Plane triangular structure

由上图6可知该结构为四个相同的二维三角形单元组成。该杆件结构的相关参数如下表4所示：

As shown in Figure 6, the structure is composed of four identical two-dimensional triangular elements. The relevant parameters of the rod structure are shown in Table 4:

表4 三角形单元参数 Table 4: Triangular element parameters

弹性模量/Mpa	泊松比 v	三角形单元长度/mm	单元厚度/mm
230000	0.2	20	30

该平面三角形共有六个节点（如图4所示），共由三个平面二维三角形单元组成，每个节点有2个自由度。

The plane triangle has six nodes (as shown in Figure 4), composed of three two-dimensional plane triangular elements, with each node having two degrees of freedom.

二、边界条件设置 Second: Boundary condition settings

在三角形节点3施加100N水平力、节点4施加20N的水平力，并约束节点1、节点2的所有自由度，其余节点均为自由边界。

A 100N horizontal force is applied at node 3 of the triangle, and a 20N horizontal force is applied at node 4, with all degrees of freedom of nodes 1 and 2 constrained, and the remaining nodes are free boundaries.

三、计算结果对比 Section 3: Comparison of Calculation Results

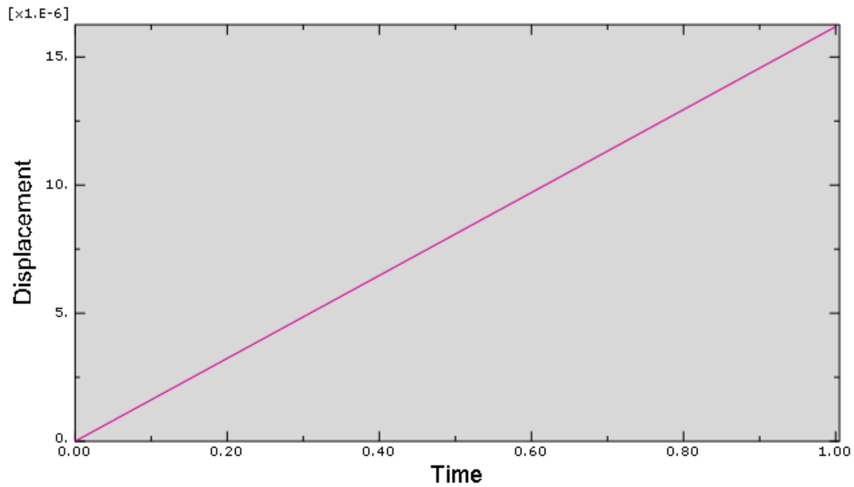
在相同边界条件、相同载荷条件下，所编写的子程序位移计算结果与abaqus中的CPS3单元位移计算结果如下表4及图5-6所示，可从下图表得知，二者计算结果一致。

Under the same boundary conditions and loading conditions, the displacement calculation results of the written subroutine are compared with the displacement calculation results of the CPS3 element in ABAQUS as shown in Table 4 and Figures 5-6. It can be seen from the following table that the two calculation results are consistent.

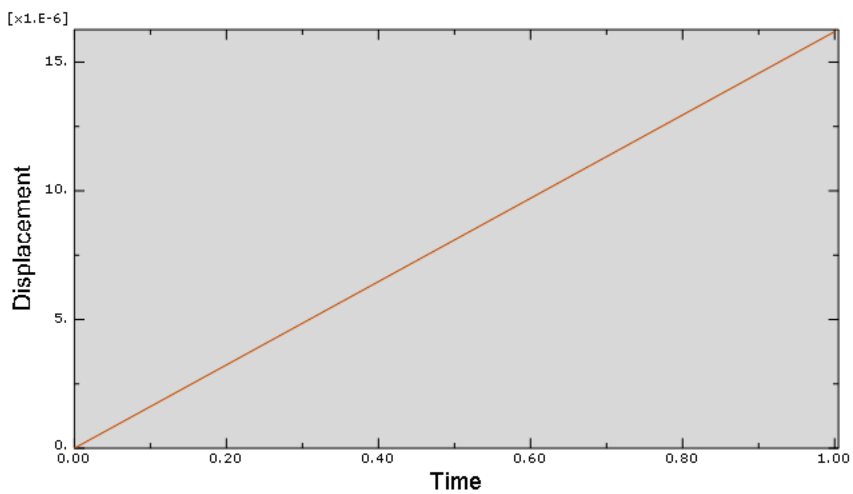
表5 UEL子程序与abaqus有限元最大位移对比

Table 5: Comparison of the maximum displacement between the UEL subroutine and the finite element analysis in ABAQUS

	节点编号	U1 最大位移/mm	U2 最大位移/mm
CPS3 单元	1	3.3550 E-05	-1.4791 E-05
	3	2.6353 E-05	-4.7042 E-05
	4	0	0
UEL 子程序	1	3.3550 E-05	-1.4791 E-05
	3	2.6353 E-05	-4.7042 E-05
	4	0	0

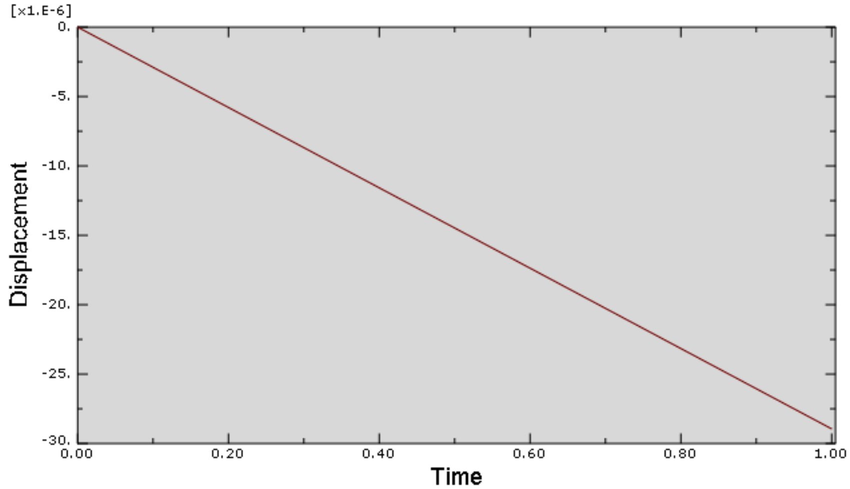


(a)CPS3加载时程-U1 (a) CPS3 loading process - U1

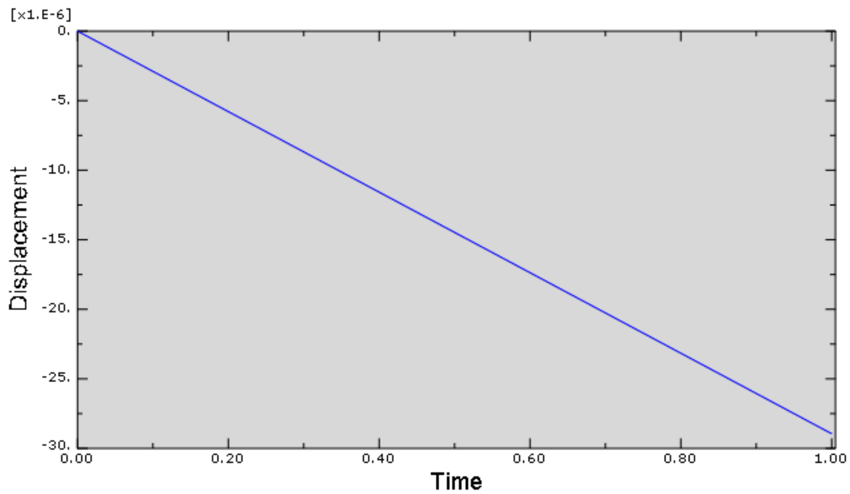


(b)UEL加载时程-U1 (b) UEL Loading Time History - U1

图7-1两类单元节点的U1位移时程曲线 Figure 7-1: Time History Curves of U1 Displacement for Two Types of Element Nodes

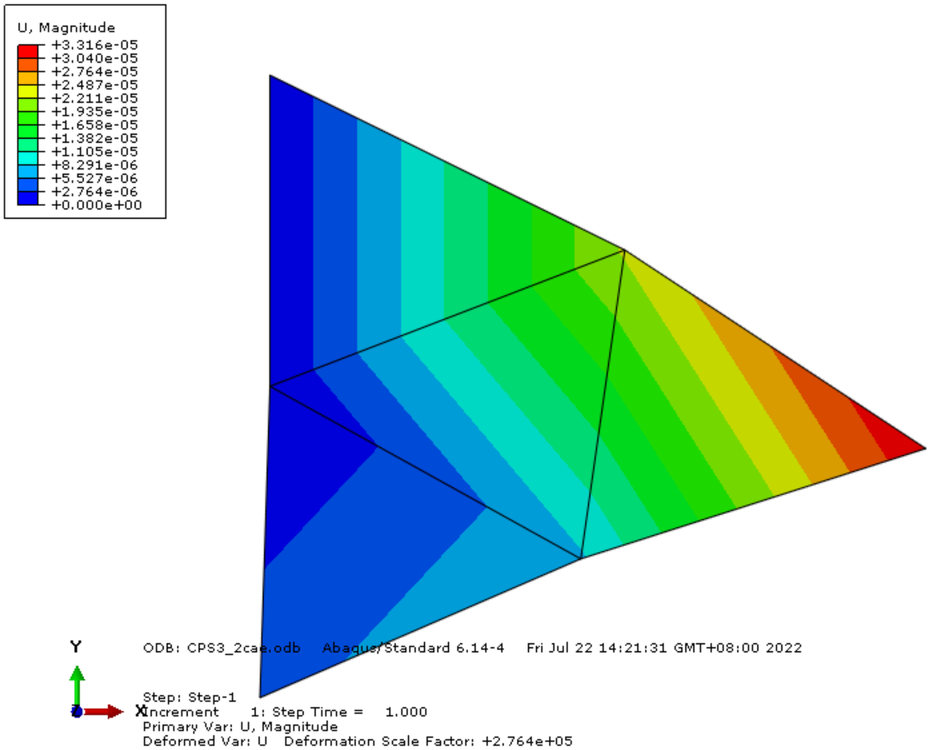


(a)CPS3加载时程-U2 (a) CPS3 loading process - U2



(b)UEL加载时程-U2 (b) UEL Loading Time History - U2

图7-2两类单元节点的U2位移时程曲线 Figure 7-2: Time History Curves of U2 Displacement for Two Types of Element Nodes



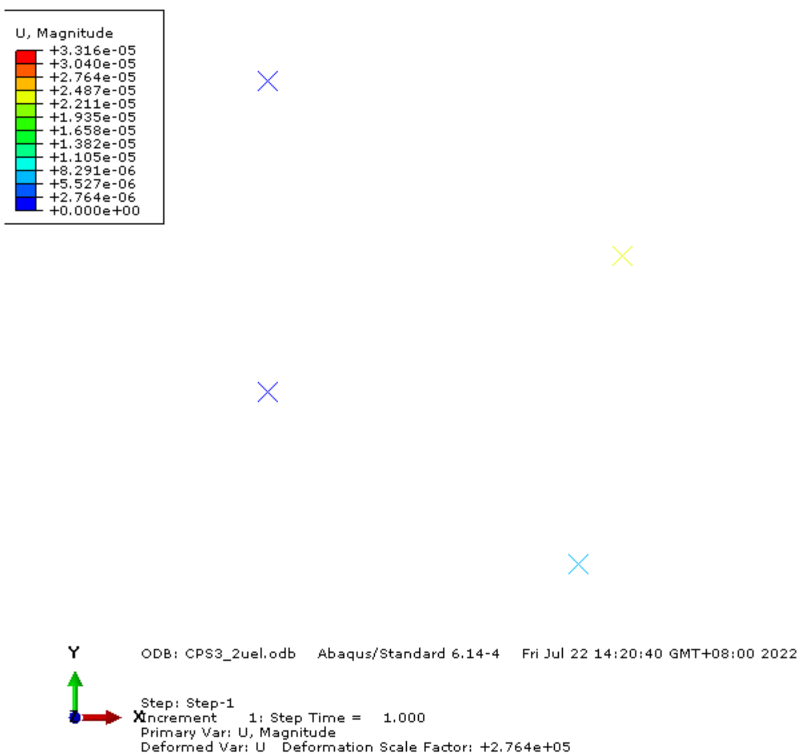


图8 CPS3单元计算结果与UEL单元计算结果对比

Figure 8 Comparison of CPS3 Element Calculation Results and UEL Element Calculation Results

通过以上两个有限元算例可知，采用所编写的UEL二维三角形单元子程序所计算得到的位移及时程结果与有限元软件ABAQUS中采用CPS3单元对应计算结果一致，说明该三角形单元子程序可靠且合理。

It can be seen from the above two finite element examples that the displacement and time history results obtained by using the UEL two-dimensional triangular element subroutine written are consistent with the corresponding calculation results using the CPS3 element in the finite element software ABAQUS, indicating that the triangular element subroutine is reliable and reasonable.

【参考文献】 References

[1]：Abaqus6.14 User Subroutines Reference Guide

[1] Abaqus 6.14 User Subroutines Reference Guide

[2]: 工程中的有限元方法（第四版） [2] Finite Element Method in Engineering (4th Edition)

概念为先，机理为本，期待下篇！ Concept first, mechanism as the foundation, looking forward to the next article!

往期推荐 • Previous recommendations •

#性能分析

【JY】基于性能的抗震设计浅析（一） A Brief Analysis of Seismic Design Based on Performance (Part 1)

【JY】基于性能的抗震设计浅析（二） A Brief Analysis of Seismic Design Based on Performance (Part 2)

【JY】浅析消能附加阻尼比 Analysis of Additional Damping Ratio for Energy Dissipation

【JY】近断层结构设计策略分析与讨论 Analysis and Discussion of Near-Fault Structural Design Strategies

【JY】浅析各动力求解算法及其算法数值阻尼(人工阻尼)

Analyzing Various Dynamic Solving Algorithms and Their Algorithmic Numerical Damping (Artificial Damping)

理念 Concept

【JY|体系】结构概念设计之(结构体系概念) 【JY|System】Conceptual Design of Structures (Structure System Concept)

【JY|理念】结构概念设计之(设计理念进展) 【JY|Concept】Conceptual Design of Structures (Progress of Design Concept)

【JY】有限单元分析的常见问题及单元选择 Common Issues in Finite Element Analysis and Element Selection

【JY】结构动力学之显隐式 Explicit and Implicit Methods in Structural Dynamics

【JY】浅谈结构设计 An Overview of Structural Design

【JY】浅谈混凝土损伤模型及Abaqus中CDP的应用

An Introduction to Concrete Damage Models and the Application of CDP in Abaqus

【JY】浅谈混凝土结构/构件性能试验指标概念（一）

An Overview of Concrete Structure/Component Performance Test Index Concepts (Part 1)

【JY】浅谈混凝土结构/构件性能试验指标概念（二）

An Overview of Concrete Structure/Component Performance Test Index Concepts (Part 2)

【JY】橡胶系支座/摩擦系支座全面解析 Comprehensive Analysis of Rubber-Type and Friction-Type Supports

#概念机理

【JY】基于Ramberg-Osgood本构模型的双线性计算分析

Bilinear Calculation Analysis Based on the Ramberg-Osgood Constitutive Model

【JY】结构动力学初步-单质点结构的瞬态动力学分析

【JY】An Introduction to Structural Dynamics - Transient Dynamics Analysis of Single-Point Structures

【JY】从一根悬臂梁说起 【JY】Starting from a Cantilever Beam

【JY】反应谱的详解与介绍 An Explanation and Introduction to Response Spectrum

【JY】结构瑞利阻尼与经济订货模型 Structural Rayleigh Damping and Economic Order Quantity Model

【JY】主成分分析与振型分解 Principal Component Analysis and Mode Decomposition

【JY】浅谈结构多点激励之概念机理（上） A Brief Discussion on the Concept and Mechanism of Multi-Point Excitation in Structures (Part 1)

【JY】浅谈结构多点激励之分析方法（下） An Overview of the Analytical Methods for Multi-point Excitation of Structures (Part 2)

【JY】板壳单元的分析详解 Detailed Analysis of Shell Elements

【JY】橡胶支座的简述和其力学性能计算 A Brief Description of Rubber Supports and Calculation of Their Mechanical Properties

【JY】振型求解之子空间迭代 Solving Mode Shapes Using Subspace Iteration

【JY】橡胶支座精细化模拟与有限元分析注意要点 Key Points for the 精细化 Simulation and Finite Element Analysis of Rubber Supports

【JY】推开土木工程振型求解之兰索斯法(Lanczos法)的大门

Opening the Door to the Lanczos Method in Civil Engineering Vibration Mode Solution

【JY】基于OpenSees和Sap2000静力动力计算案例分析

Case Analysis of Static and Dynamic Calculations Based on OpenSees and Sap2000

【JY】建筑结构施加地震波的方法与理论机理 【JY】Methods and Theoretical Mechanisms for Applying Seismic Waves to Building Structures

【JY】力荐佳作《结构地震分析编程与应用》 Highly Recommended Masterpiece: Programming and Application of Structural Seismic Analysis

#软件讨论

【JY】复合材料分析利器—内聚力单元 Tool for Composite Material Analysis - Cohesive Zone Element

【JY】SDOF计算教学软件开发应用分享 Sharing of SDOF Calculation Teaching Software Development and Application

【JY】Abaqus案例—天然橡胶隔震支座竖(轴)向力学性能

Abaqus Case - Vertical (Axial) Mechanical Properties of Natural Rubber Isolation Support

【JY】Abaqus6.14-4如何关联fortran?

【JY】How to associate Fortran with Abaqus6.14-4?

【JY】如何利用python来编写GUI? 【JY】How to write a GUI using Python?

【JY】如何解决MATLAB GUI编程软件移植运行问题?

【JY】How to solve the porting and running issues of MATLAB GUI programming software?

【JY】浅谈结构分析与设计软件 【JY】A brief discussion on structural analysis and design software

【JY|STR】求解器之三维结构振型分析 Analysis of Three-Dimensional Structural Vibration Modes of the Solver

【JY】SignalData软件开发应用分享 Sharing of SignalData Software Development and Application

【JY】基于Matlab的双线性滞回代码编写教程

Tutorial on Writing Bilinear Hysteresis Code Based on Matlab

【JY】动力学利器 —— JYdyn函数包分享与体验

Dynamics Tool - Sharing and Experience of JYdyn Function Package

【JY】混凝土分析工具箱：CDP模型插件与滞回曲线数据

ABAQUS Subroutine UEL: Finite Element Principles and Applications - Technical Neighbors

【JY】结构工程分析软件讨论（上） Discussion on Structural Engineering Analysis Software (Part 1)

【JY】结构工程分析软件讨论（下） ABAQUS Subroutine UEL: Finite Element Principles and Applications - Technical Neighbors

#YJK前处理参数详解

【JY】YJK前处理参数详解及常见问题分析（一）

【JY】An Overview of YJK Preprocessing Parameters and Common Problem Analysis (Part 1)

【JY】YJK前处理参数详解及常见问题分析：控制信息（二）

【JY】An Overview of YJK Preprocessing Parameters and Common Problem Analysis: Control Information (Part 2)

【JY】YJK前处理参数详解及常见问题分析：刚度系数（三）

【JY】An Overview of YJK Preprocessing Parameters and Common Problem Analysis: Stiffness Coefficient (Part 3)

【JY】YJK前处理参数详解及常见问题分析：二阶效应和分析求解（四）

【JY】An Overview of YJK Preprocessing Parameters and Common Problem Analysis: Second-Order Effects and Analysis Solution (Part 4)

【JY】YJK前处理参数详解及常见问题分析（五）：风荷载信息

Detailed Explanation of YJK Preprocessing Parameters and Common Problem Analysis (Part Five): Wind Load Information

【JY】YJK前处理参数详解及常见问题分析（六）：地震信息

Detailed Explanation of YJK Preprocessing Parameters and Common Problem Analysis (Part Six): Seismic Information

#其他

【JY】位移角还是有害位移角？ Displacement Angle or Harmful Displacement Angle?

【JY】如何利用python来编写GUI？ How to Use Python to Write a GUI?

【JY】今日科普之BIM Today's Popular Science: BIM

~关注未来更精彩~ ~Focus on the more exciting future~