



技术邻 &gt; Technical Neighbor &gt; CAE仿真 &gt; CAE Simulation &gt; Abaqus

## abaqus UEL三维怎样解决边界散热问题，依据帮助文档中的二维改为三维，发现结果不对？

### How to solve the boundary heat dissipation problem in ABAQUS UEL 3D, and why the results are incorrect when changing from 2D in the help document to 3D?

浏览: 1707 Views: 1707

[DC3D8\\_FILM.zip](#) [UEL\\_DC3D8\\_FILM.zip](#) UEL\_DC3D8\_FILM.zip

DC3D8是abaqus自带单元。UEL\_DC3D8是UEL自编程序，结果跟自带单元不一样，不知道UEL程序错在哪！！求求有没有人帮帮忙，帮忙看看~~~要延毕了。。。

DC3D8 is an ABAQUS built-in element. UEL\_DC3D8 is a UEL self-programmed element, and the results are different from the built-in element. I don't know where the error in the UEL program is! Please help me, someone! I'm about to graduate...

ABAQUS

请问钻削仿真对工件需要... 如何在... 钻削的部分进行了局部网格SDYNA 细化分后，需要删除网格窗中用\*sen 口并将网格窗口系数设为0 sor关键 吗，还是保留网格窗口，网字记录 格窗口系数保持为1，然后 单元受 到的压 力？

When performing drilling simulation in LSDYNA, is it necessary to delete the mesh window and set the mesh window coefficient to 0 after locally refining the mesh for the part to be drilled, or should the mesh window be retained and the mesh window coefficient kept at 1, and then perform the simulation? How can the \*sensor keyword be used in LSDYNA to record the pressure on the element?

关注问题 Pay

attention to the

issue

邀请回答 Invite to answer

我来回答 I will answer



当前暂无回答 Currently no answers available

回答可获赠 200金币 Answering will earn 200 coins

相似问题 Similar Questions

查看全部 View All

ABAQUS模态分析UEL

暂无回答

ABAQUS在模态分析时，需要先设置step1frequency，才可以在step2中设置基于直接法...

ABAQUS modal analysis UEL has no answer. In ABAQUS modal analysis, it is necessary to set step1frequency first before you can set the steady-state dynamic analysis based on direct method, modal method, or subspace method in step2. The position of the two arrows in the help document. One says that the steady-state dynamic analysis based on modal is only called UEL in the extraction of natural frequencies in front, and considers the contributions of stiffness matrix and mass matrix, but not the damping matrix. The other says that both direct method and steady-state dynamic analysis based on modal call UEL, but only consider mass matrix and stiffness matrix, ignoring damping. I checked the information, and it is said that modal superposition method is not

ABAQUS UEL材料局部坐标系如何定义？

3个回答

复合材料纤维为各向异性材料，材料方向随单元位置而变化，自编的uel中计算纤维的...

How to define the local coordinate system of ABAQUS UEL material? 3 answers The composite material fiber is anisotropic, and the material direction varies with the position of the element. When calculating the global flexibility matrix of the fiber in the self-written UEL, the rotation axis formula is required. Since there are no global coordinates and material coordinates related to the parameters passed into UEL by ABAQUS, how can the rotation matrix be obtained in UEL?

abaqus子程序uel做模态分析？

3个回答

1、在abaqus中建立了由三个节点组成的两个桁架单元，各单元长度为1000mm，弹性模...

没解决？试试专家一对一服务 Not solved? Try one-on-one service 换一批 Refresh batch

ANSYS

FLUENT

1次答疑 One session

FLUENT专家 FLUENT Expert

17年 17 years

工程技术模拟仿真服务 Engineering Technology Simulation and Service

1000元/次 1000 yuan per time

预约答疑 Schedule consultation

2次答疑 Two sessions

孙一凡仿真 Simulation

19年 2019

专业从事有限元仿真15年，目前就职于排名第一的风电上市企业，主要从事结构部件强度计算，疲劳强...

268元/次 268 yuan per time

预约答疑 Schedule consultation

2次答疑 Two sessions

一杯敬明月 A toast to the bright moon

本科/10年/材料成型及控制工程 Bachelor's degree/10 years/Material Forming and Control Engineering

砂铸、熔模、金属型、低压、消失模等铸造工艺CAE分析有关问题

200元/次 200 yuan per time

预约答疑 Schedule consultation

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

沉淤 Sedimentation

¥350 \$350

ANSYS必修课\_workbench基础操作应用...

大龙猫 Big Cat

¥188 \$188

基于primer和hypermesh汽车CAE仿真系统分析... Automotive CAE simulation

渔船、油轮船、潜艇救援母舰、泰坦尼克号大型客船、拖船、救援艇、集装箱船、渡船等...

02 获取方式

扫码回复【整船】

How to perform modal analysis in ABAQUS subprogram UEL? 3 answers

1. In ABAQUS, two truss elements composed of three nodes were established, with each element having a length of 1000mm, an elastic modulus of 10000Mpa, a cross-sectional area of 100mm<sup>2</sup>, and a density of 7.85e-09ton/mm. The displacement in all directions of the nodes at both ends was fully constrained, while the middle node was free. Since the initial prestress is not considered in the UEL of ABAQUS,

example, and the correct result calculation is simply observed

https://www.jishulink.com/answer/1898417

2/2