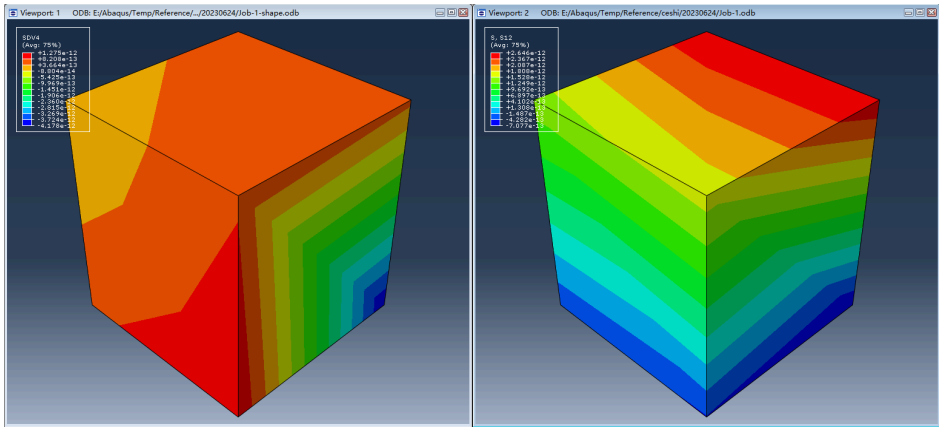
C3D8单元结果不同? Why are the results of the C3D8 element different?

浏览: 1875 回答: 3 Views: 1875 Answers: 3

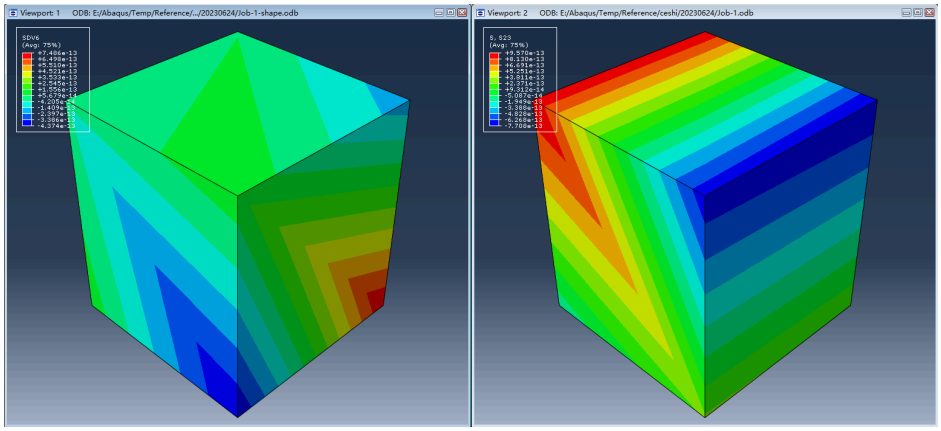
用UEL编写的C3D8 单元进行程序验证时，结果不同是什么原因造成?

Why do the results differ when verifying the program with a C3D8 element written in UEL?

S12方向的应力 Stress in the S12 direction



S23方向的应力 Stress in the S23 direction



.inp文件 .inp file

相似问题 Similar Questions 查看全部 View All

什么是压缩边? ? ?
1个回答
干啥用的?
What is the compression edge??? 1 answer What is it used for?

TimeHist Postpro 中minimum 和 maxmum...
1个回答
是什么原因?

Why is the minimum and maximum displayed as invalid in the TimeHist Postpro? 1 reply, what is the reason?

如何设置才可让ADAMS信息窗口的字体显...
暂无回答
如何设置才可让ADAMS信息窗口的字体显示更大?

How to set the font size to be larger in the ADAMS information window? Currently no answer, how to set it?

```
** PARTS
**
*Part, name=Part-1
*Node
      1,          1.,          1.,          1.
      2,          1.,          0.,          1.
      3,          1.,          1.,          0.
      4,          1.,          0.,          0.
      5,          0.,          1.,          1.
      6,          0.,          0.,          1.
      7,          0.,          1.,          0.
      8,          0.,          0.,          0.
**********************************************
*User element, nodes=8, type=U1, properties=2, coordinates=3, VARIABLES=96
1, 2, 3
*Element, type=U1, Elset=solid
1, 5, 6, 8, 7, 1, 2, 4, 3
**********************************************************************
*Uel property, elset=solid
200e3, 0.3
*Element, type=C3D8, elset=Visualization
11,5, 6, 8, 7, 1, 2, 4, 3
*Nset, nset=Set-1, generate
 1,  8,  1
*Elset, elset=Set-1
 1,
** Section: Section-1
*Solid Section, elset=Visualization, material=Material-1
,
*End Part
**
```

UEL.for文件 UEL.for file

```
 do kintk=1,ninpt

  call kshapefcn(kintk,ninpt,nnode,ndim,dN,dNdz)
  call kjacobian(jelem,ndim,nnode,coords,dNdz,djac,dNdx,mcrd)
  dvol=wght(kintk)*djac


  b=0.d0
  do inod=1,nnode
     b(1,3*inod-2) = dNdx(1,inod)
     b(2,3*inod-1) = dNdx(2,inod)
     b(3,3*inod)   = dNdx(3,inod)
     b(4,3*inod-2) = dNdx(2,inod)
     b(4,3*inod-1) = dNdx(1,inod)
     b(5,3*inod-1) = dNdx(3,inod)
     b(5,3*inod)   = dNdx(2,inod)
     b(6,3*inod-2) = dNdx(3,inod)
     b(6,3*inod)   = dNdx(1,inod)

  end do

  dstran=matmul(b,du(1:ndim*nnode,1))

  call kstatevar(kintk,nsvint,svars,statevLocal,1)

  stress=statevLocal(1:ntens)
  stran(1:ntens)=statevLocal((ntens+1):(2*ntens))


**************************************************************


  call kumat(props,ddsdde,stress,dstran,ntens,statevLocal)

  stran=stran+dstran


  statevLocal(1:ntens)=stress(1:ntens)
  statevLocal((ntens+1):(2*ntens))=stran(1:ntens)


  call kstatevar(kintk,nsvint,svars,statevLocal,0)

  amatrx(1:24,1:24)=amatrx(1:24,1:24)+
1 dvol*(matmul(matmul(transpose(b),ddsdde),b))

  rhs(1:24,1)=rhs(1:24,1)-
1 dvol*(matmul(transpose(b),stress))


put
```

形函数： Shape function:

```
subroutine kshapefcn(kintk,ninpt,nnode,ndim,dN,dNdz)

include 'aba_param.inc'

parameter (gaussCoord=(1.d0/3.d0)**0.5d0)
dimension dN(nnode,1),dNdz(ndim,*),coord38(3,8)

data  coord38 /-1.d0, -1.d0, -1.d0,
2               1.d0, -1.d0, -1.d0,
3               1.d0,  1.d0, -1.d0,
4              -1.d0,  1.d0, -1.d0,
5              -1.d0, -1.d0,  1.d0,
6               1.d0, -1.d0,  1.d0,
7               1.d0,  1.d0,  1.d0,
8              -1.d0,  1.d0,  1.d0/

3D 8-nodes

determine (g,h,r)
g=coord38(1,kintk)*gaussCoord
h=coord38(2,kintk)*gaussCoord
r=coord38(3,kintk)*gaussCoord

shape functions
dN(1,1)=(1.d0-g)*(1.d0-h)*(1.d0-r)/8.d0
dN(2,1)=(1.d0+g)*(1.d0-h)*(1.d0-r)/8.d0
dN(3,1)=(1.d0+g)*(1.d0+h)*(1.d0-r)/8.d0
dN(4,1)=(1.d0-g)*(1.d0+h)*(1.d0-r)/8.d0
dN(5,1)=(1.d0-g)*(1.d0-h)*(1.d0+r)/8.d0
dN(6,1)=(1.d0+g)*(1.d0-h)*(1.d0+r)/8.d0
dN(7,1)=(1.d0+g)*(1.d0+h)*(1.d0+r)/8.d0
dN(8,1)=(1.d0-g)*(1.d0+h)*(1.d0+r)/8.d0

derivative d(Ni)/d(g)
dNdz(1,1)=-(1.d0-h)*(1.d0-r)/8.d0
dNdz(1,2)= (1.d0-h)*(1.d0-r)/8.d0
dNdz(1,3)= (1.d0+h)*(1.d0-r)/8.d0
dNdz(1,4)=-(1.d0+h)*(1.d0-r)/8.d0
dNdz(1,5)=-(1.d0-h)*(1.d0+r)/8.d0
dNdz(1,6)= (1.d0-h)*(1.d0+r)/8.d0
dNdz(1,7)= (1.d0+h)*(1.d0+r)/8.d0
dNdz(1,8)=-(1.d0+h)*(1.d0+r)/8.d0


derivative d(Ni)/d(h)
dNdz(2,1)=-(1.d0-g)*(1.d0-r)/8.d0
dNdz(2,2)=-(1.d0+g)*(1.d0-r)/8.d0
dNdz(2,3)= (1.d0+g)*(1.d0-r)/8.d0
dNdz(2,4)= (1.d0-g)*(1.d0-r)/8.d0
dNdz(2,5)=-(1.d0-g)*(1.d0+r)/8.d0
dNdz(2,6)=-(1.d0+g)*(1.d0+r)/8.d0
dNdz(2,7)= (1.d0+g)*(1.d0+r)/8.d0
dNdz(2,8)= (1.d0-g)*(1.d0+r)/8.d0

derivative d(Ni)/d(r)
dNdz(3,1)=-(1.d0-g)*(1.d0-h)/8.d0
dNdz(3,2)=-(1.d0+g)*(1.d0-h)/8.d0
dNdz(3,3)=-(1.d0+g)*(1.d0+h)/8.d0
```

```
      dNdz(3,4)=-(1.d0-g)*(1.d0+h)/8.d0
      dNdz(3,5)= (1.d0-g)*(1.d0-h)/8.d0
      dNdz(3,6)= (1.d0+g)*(1.d0-h)/8.d0
      dNdz(3,7)= (1.d0+g)*(1.d0+h)/8.d0
      dNdz(3,8)= (1.d0-g)*(1.d0+h)/8.d0

      return
      end
C**********************************************************
```

出现误差的原因是否是由于形函数的编号和Abaqus中不同造成呢?

Is the cause of the error due to the difference in the numbering of the shape functions and Abaqus?

abaqus软件UEL abaqus software UEL | ABAQUS

关于abaqus模型不收敛的问? Question about the non-convergence of the abaqus model?

Abaqus幂律模如何设置? How to set up the ABAQUS power-law model?

关注问题 Focus on the issue

邀请回答 Invite to answer 我来回答 I will answer

全部回答 (1) All answers (1)

最默认 Default 新 Latest

yunching 关注 Focus

ABAQUS C3D8采用的是选择缩减积分，并不完全是全积分。

ABAQUS C3D8 uses selective reduced integration, not fully integrated.

2023年6月26日 June 26, 2023 评论 2 点赞 Comment 2 Likes

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

换一批 Refresh batch

1次答疑 One Q&A 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

4次答疑 4 Q&A sessions

冷月 Cold Moon

本科/18年/工程力学 Bachelor/18 years/Engineering Mechanics

ansys，abaqus土木结构分析，弹性动力时程分析，弹塑性分析，基坑开挖分析，隧道开挖分析，复杂项...

125元/次 125 yuan per time 预约答疑 Schedule consultation

兵荒马乱 Chaos and disorder

硕士/9年/机械工程及自动化 Master/9 years/Mechanical Engineering and Automation

软件：FLUENT、WORKBENCH、MATLAB等；主要领域：FLUENT中空化（cavitation）仿真、热分析、...

100元/次 100 yuan per time 预约答疑 Schedule consultation

推荐阅读 Recommended Reading

ansys结构动力学仿真 ANSYS structural dynamics simulation

技术邻小李 Technical Neighbor Xiao Li ¥150 150 yuan

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

冷月 Cold Moon ¥15 15 Yuan

Fluent传热——随波逐流（更新） Fluent Heat Transfer -...

随波逐流 Drifting with the Waves ¥200

Adams Car 扭力梁悬架模型频...

仿真分析 Simulation Analysis