

第十四章 有限元方法在起重机结构分析中的应用

第一节 有限元方法简介

第二节 有限元分析程序ANSYS简介

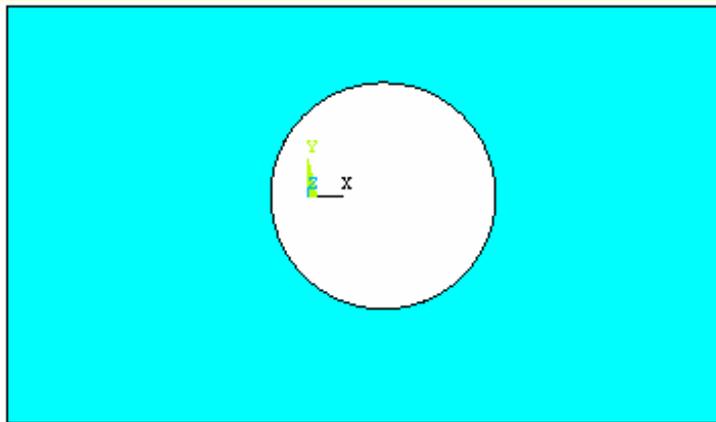
第三节 有限元分析实例——

龙门起重机金属结构有限元分析

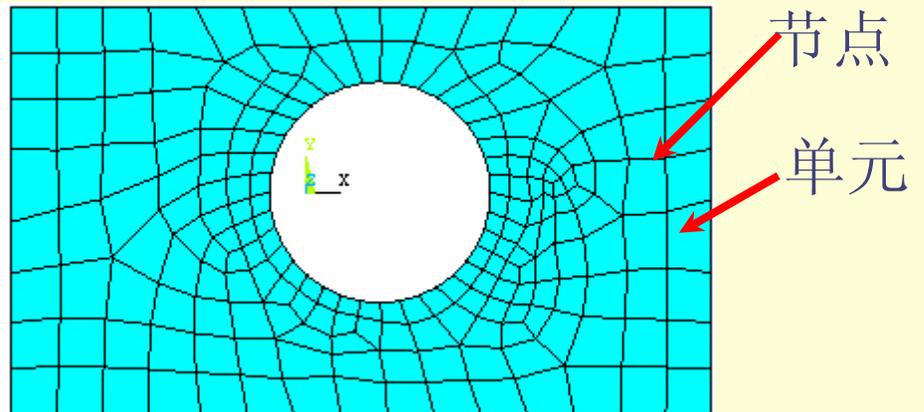
第一节 有限元方法简介

1. 有限元法的概念

有限元法 (FEM, Finite Element Method)就是把物理结构分割成有限个区域, 这些区域称为单元。每个单元中有有限个节点, 单元间通过节点相连。对每一个单元建立作用力方程, 组集成整个结构的系统方程, 求解该系统方程, 得到结构的近似解。



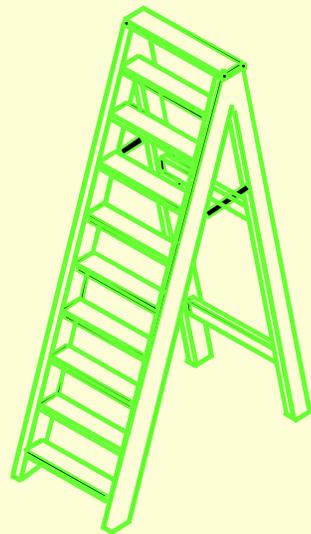
真实结构



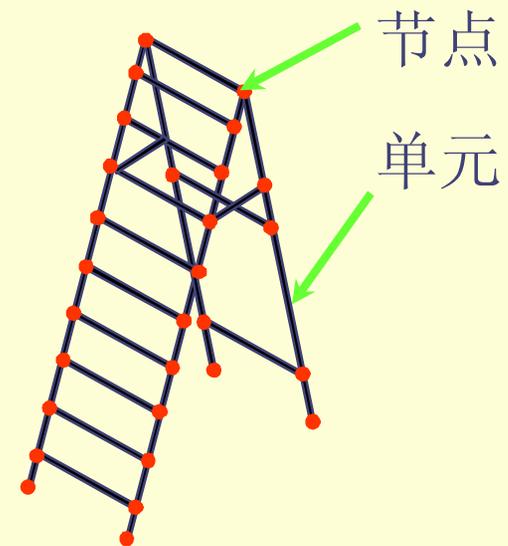
有限元模型

有限元方法用来解决实际工程需要解决而理论分析又无法解决的复杂问题。

有限元模型由一些简单形状的单元组成,单元之间通过节点连接,并通过结点传递内力。

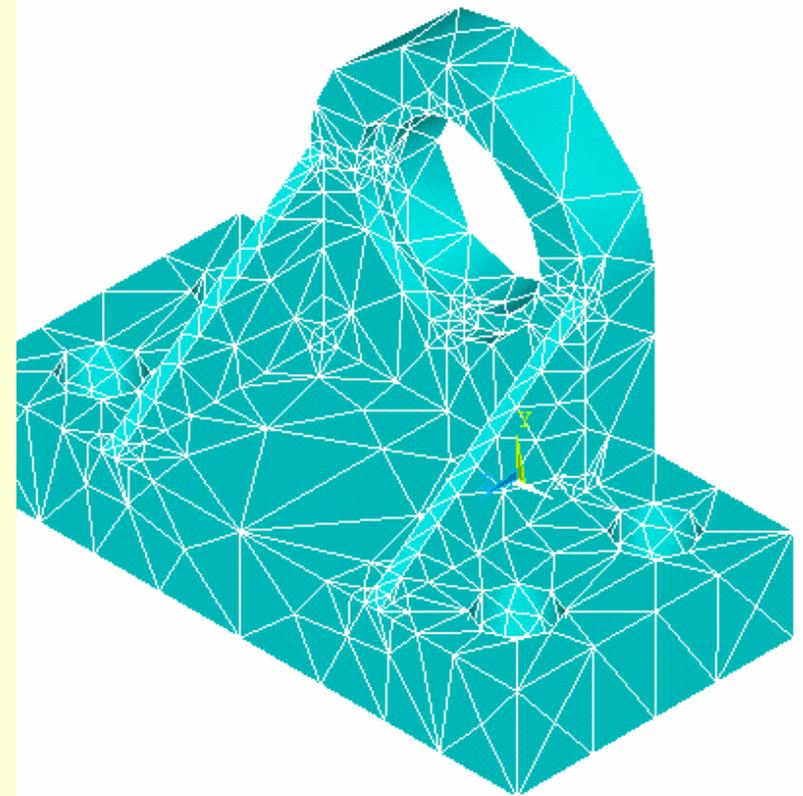
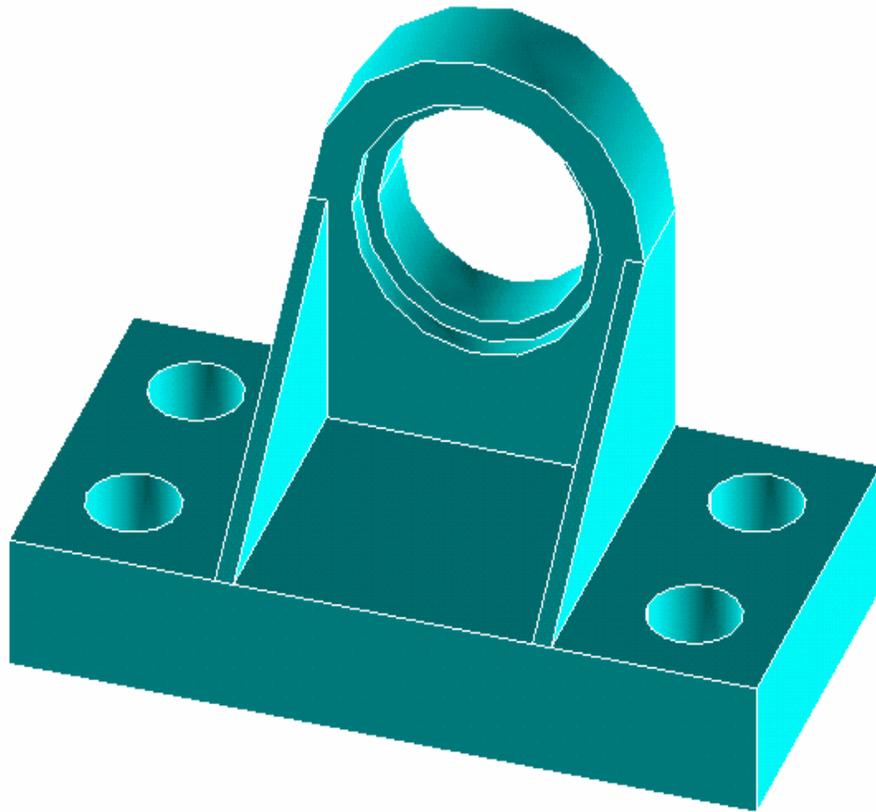


真实结构

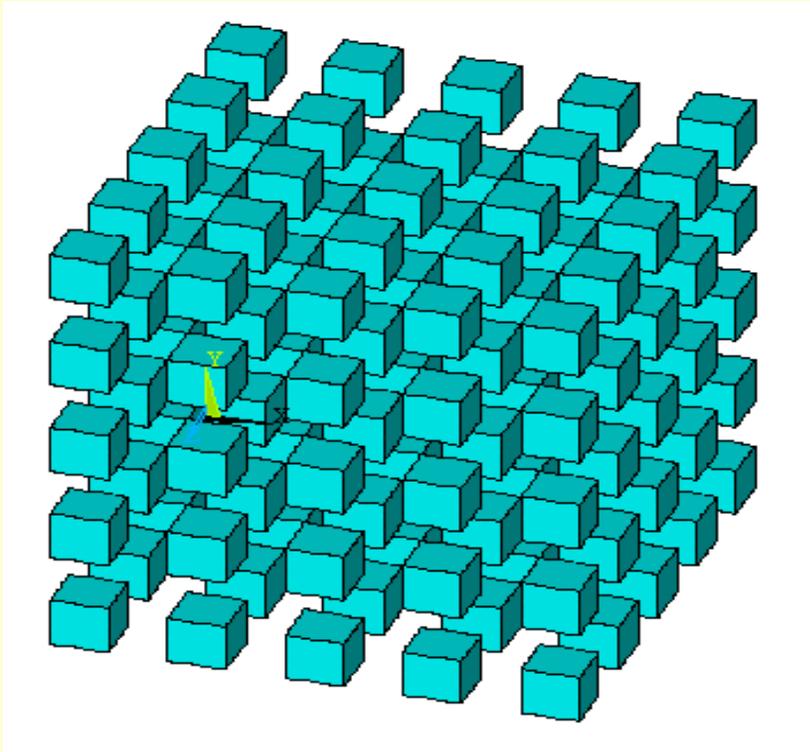
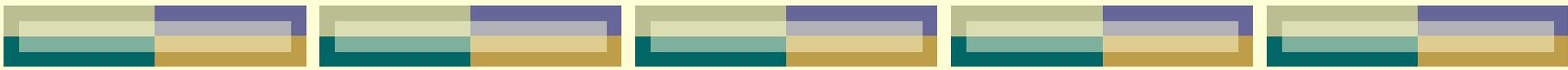


有限元模型

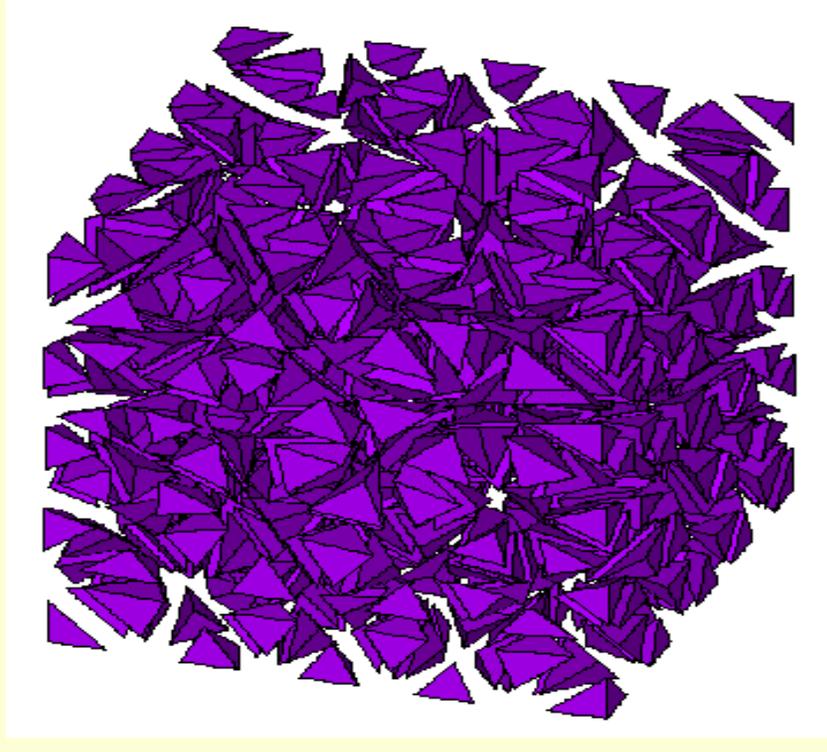
有限元方法涉及力学原理、数学方法和计算机程序设计等几方面。



轴承座实际结构与有限元模型

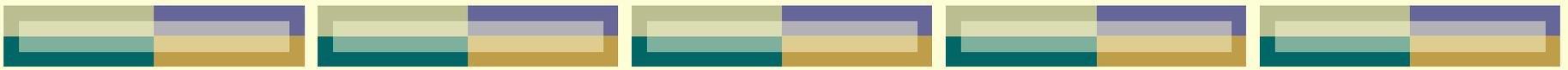


块单元网格



四面体单元网格





2. 有限元法的发展概况

- 有限元法的思想最早出现于20世纪40年代。
- 1960年，Clough教授在论文中首次使用“有限元法”一词。
- 20世纪70年代以后，随着计算机和软件技术的发展，有限元法也随之迅速地发展。
- 目前，有限元法已广泛应用于固体力学、流体力学、热传导、电磁学、声学、生物力学等各个领域。

常用有限元分析软件：

- 德国：ASKA
- 英国：PAFEC
- 法国：SYSTUS
- 美国：NASTRAN 、 ADINA、 ANSYS、 SAP、
ABQUS、 BERSAFE、 BOSOR、 COSMOS、
ELAS、 MARC和STARDYNE等。

有限元方法及软件发展趋势：

- 从单纯的结构力学计算发展到求解多物理场问题
- 由求解线性工程问题进展到分析非线性问题
- 增强可视化的前置建模和后置数据处理功能
- 与CAD软件的无缝集成
- 在Wintel 平台上的发展

3. 有限元法的计算步骤

(1) 连续体离散化

● 选择合适的单元。

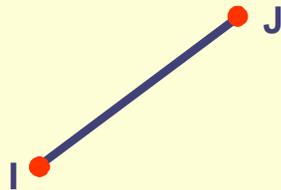
常用单元：杆单元，梁单元，三角形单元，矩形单元，四边形单元，曲边四边形单元，四面体单元，六面体单元以及曲面六面体单元等。

● 进行单元划分（即有限元网络划分）

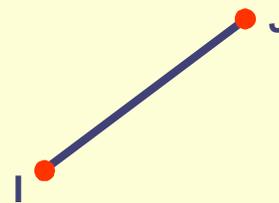
● 施加载荷及约束

常用单元

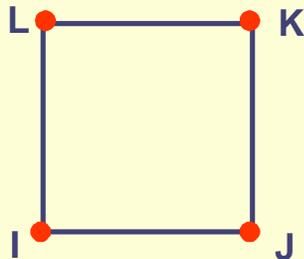
节点自由度随连接该节点的 单元类型 变化。



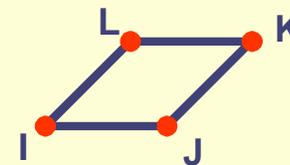
三维杆单元 (铰接)
 UX, UY, UZ



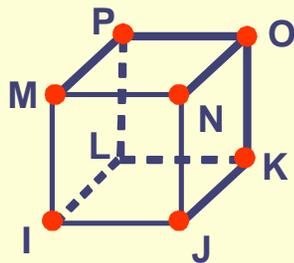
三维梁单元
 $UX, UY, UZ,$
 $ROTX, ROTY, ROTZ$



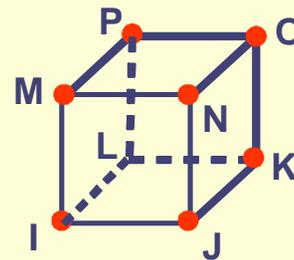
二维或轴对称实体单元
 UX, UY



三维四边形壳单元
 $UX, UY, UZ,$
 $ROTX, ROTY, ROTZ$



三维实体结构单元
 UX, UY, UZ



三维实体热单元
 $TEMP$

(2) 单元分析

建立各单元节点位移和节点力之间的关系式。

● 位移分量列阵

$$\{\delta\}^e = [u_i \quad v_i \quad u_j \quad v_j \quad u_m \quad v_m]^T$$

● 节点力分量列阵

$$\{F\}^e = [F_{ix} \quad F_{iy} \quad F_{jx} \quad F_{jy} \quad F_{mx} \quad F_{my}]^T$$

● 节点位移与节点力之间的关系

$$\{F\}^e = [k]^e \{\delta\}^e$$

式中： $[k]^e$ ——单元刚度矩阵。

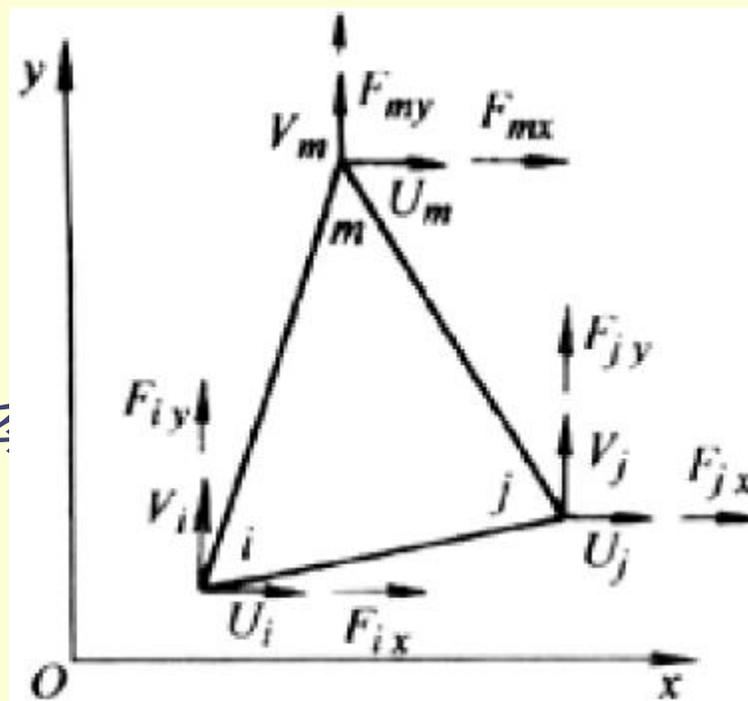


图 三角形平面单元

(3)整体分析

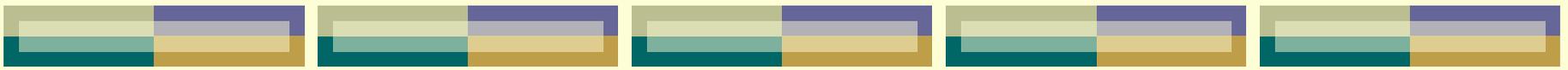
利用节点的力平衡和节点变形协调条件来建立整个连续体的节点力和节点位移的关系式。

$$[K]\{\delta\}=\{R\}$$

式中： $[K]$ ——整体刚度矩阵；
 $\{\delta\}$ ——全部结点位移组成的列阵；
 $\{R\}$ ——全部结点荷载组成的列阵。

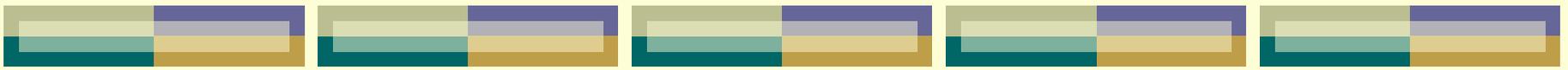
第二节 有限元分析程序ANSYS简介

- ANSYS软件是美国ANSYS公司研制的大型通用有限元分析软件。
- 能够进行包括结构、热、声、流体、电磁场等学科的研究。
- 在核工业、铁道、石油化工、航空航天、机械制造、能源、汽车交通、国防军工、电子、土木工程、造船、生物医学、轻工、地矿、水利、家用电器等领域有着广泛的应用。



1. ANSYS的功能

- 创建有限元模型（前处理）
- 施加载荷进行求解（有限元分析）
- 查看分析结果（后处理）
- 进行优化设计
- 做数值模拟实验



2. ANSYS的分析类型

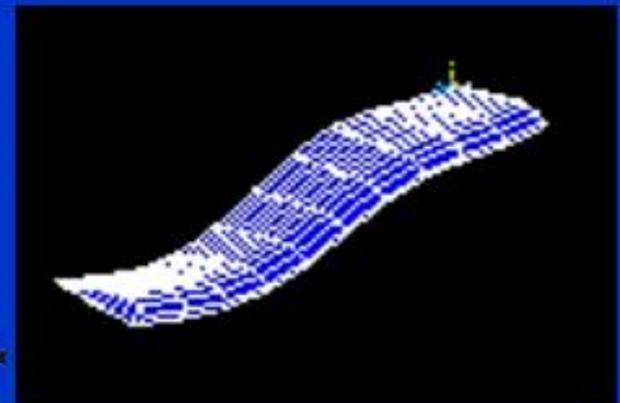
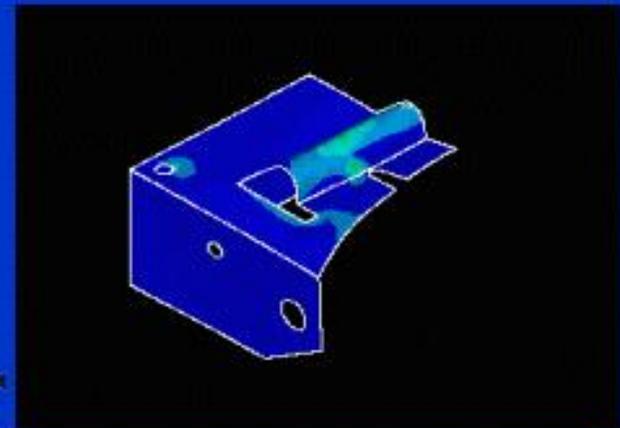
- 结构分析
- 热分析
- 电磁场分析
- 声学分析
- 压电分析
- 流体动力分析
- 多场耦合分析
- 设计灵敏度及优化分析

ANSYS 结构分析 概览

结构分析用于确定结构的变形、应变、应力及反作用力等。

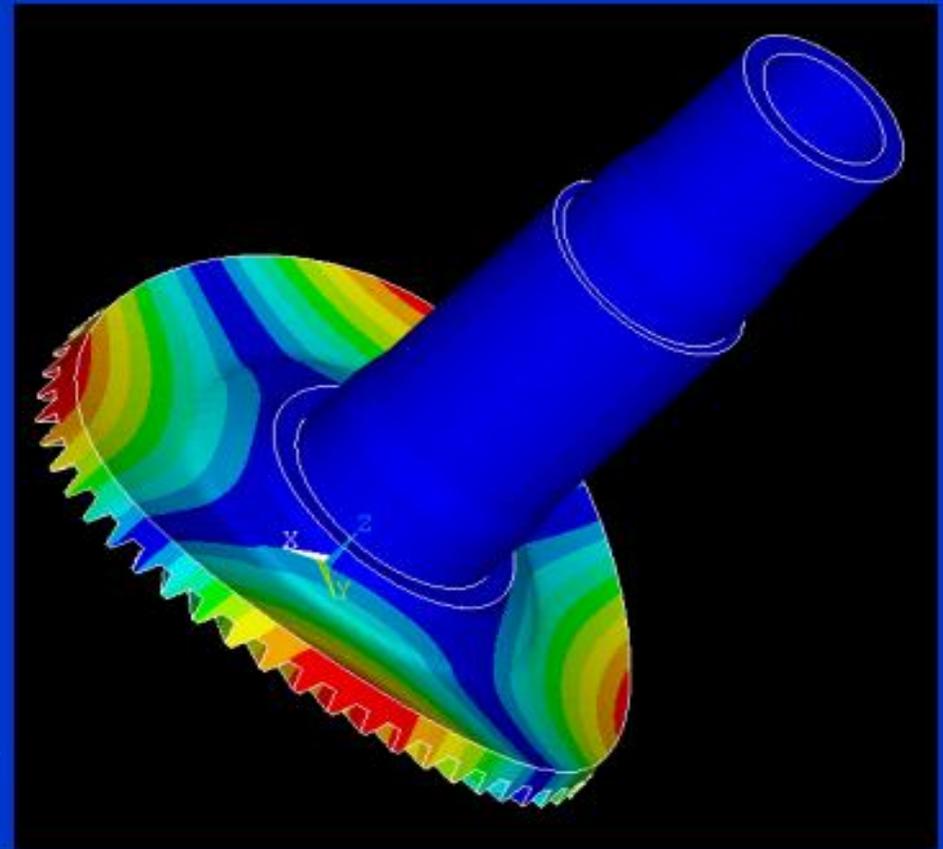
结构分析的类型:

- **静力分析** - 用于静态载荷, 可以考虑结构的线性及非线性行为, 例如: 大变形、大应变、应力刚化、接触、塑性、超弹及蠕变等。
- **模态分析** - 计算线性结构的自振频率及振形, **谱分析** 是模态分析的扩展, 用于计算由于随机振动引起的结构应力和应变 (也叫作 **响应谱或 PSD**).



ANSYS 结构分析 概览(续)

- **谐响应分析** - 确定线性结构对随时间按正弦曲线变化的载荷的响应.
- **瞬态动力学分析** - 确定结构对随时间任意变化的载荷的响应. 可以考虑与静力分析相同的结构非线性行为.
- **特征屈曲分析** - 用于计算线性屈曲载荷并确定屈曲模态形状. (结合瞬态动力学分析可以实现非线性屈曲分析.)
- **专项分析**: 断裂分析, 复合材料分析, 疲劳分析

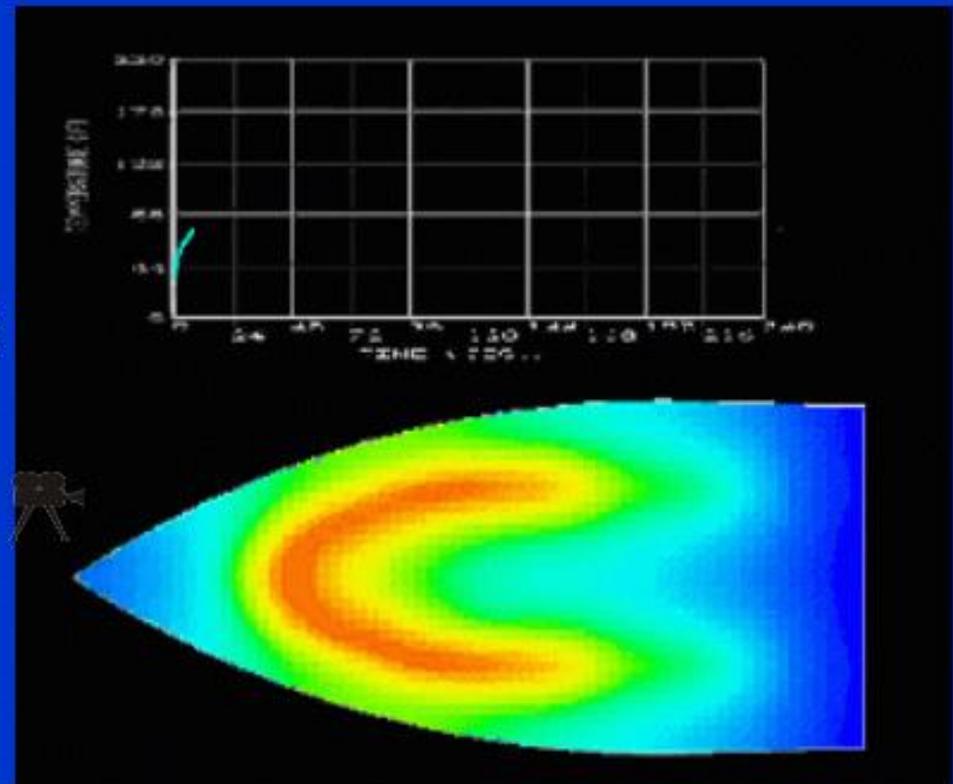


Courtesy: Sikorsky Aircraft

ANSYS热分析概览

ANSYS 热分析计算物体的稳态或瞬态温度分布，以及热量的获取或损失、热梯度、热通量等。

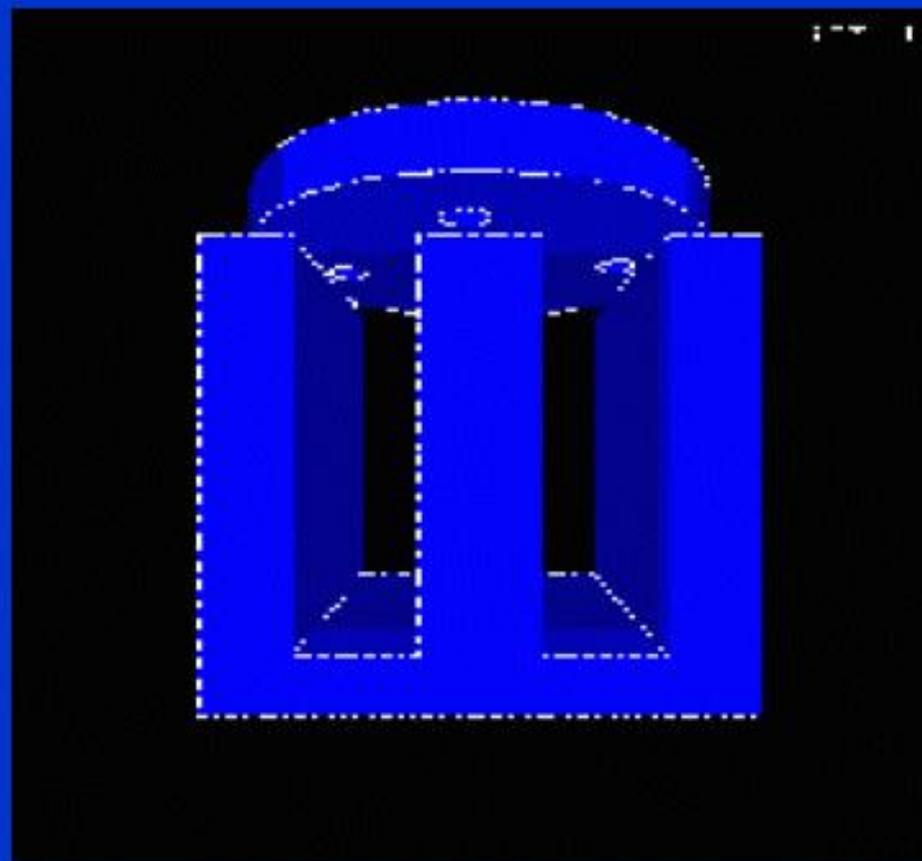
- 热分析之后往往进行结构分析,计算由于热膨胀或收缩不均匀引起的应力.
- **ANSYS**功能:
 - 相变 (熔化及凝固), 内热源 (例如电阻发热等)
 - 三种热传递方式 (热传导、热对流、热辐射)



ANSYS电磁分析概览

磁场分析 用于计算磁场。

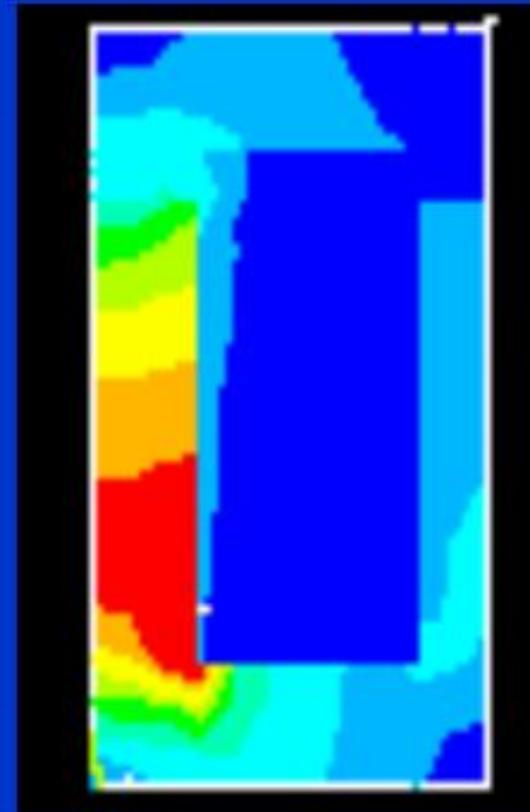
- 磁场分析中考虑的物理量是磁通量密度、磁场密度、磁力、磁力矩、阻抗、电感、涡流、能耗及磁通量泄漏等。
- 磁场可由电流、永磁体、外加磁场等产生。



ANSYS电磁分析概览(续)

磁场分析的类型:

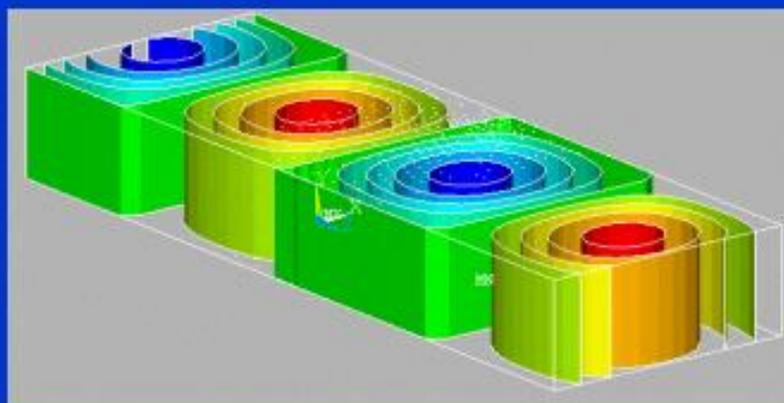
- **静磁场分析** - 计算直流电(DC)或永磁体产生的磁场.
- **交变磁场分析** - 计算由于交流电(AC)产生的磁场.
- **瞬态磁场分析** - 计算随时间随机变化的电流或外界引起的磁场.



ANSYS电磁分析概览(续)

电场分析 用于计算电阻或电容系统的电场。典型的物理量有电流密度、电荷密度、电场及电阻热等。

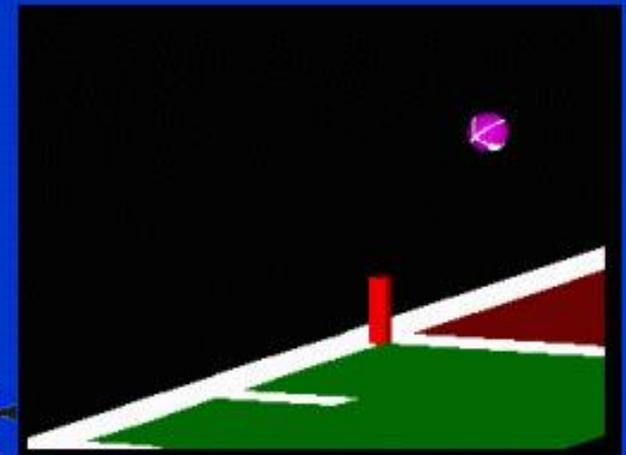
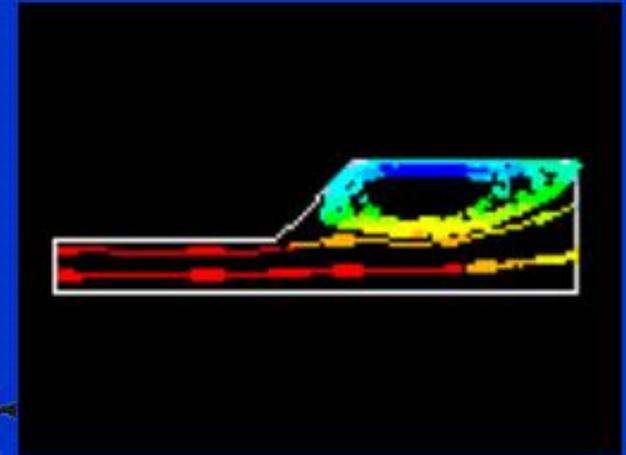
高频电磁场分析 用于微波及RF无源组件，波导、雷达系统、同轴连接器等分析。



ANSYS 流体分析 概览

流体分析 用于确定流体的流动及热行为。流体分析分以下几类：

- **CFD - ANSYS/FLOTRAN** 提供强大的计算流体动力学分析功能，包括不可压缩或可压缩流体、层流及湍流，以及多组份流等。
- **声学分析** - 考虑流体介质与周围固体的相互作用，进行声波传递或水下结构的动力学分析等。
- **容器内流体** 分析 - 考虑容器内的非流动流体的影响。可以确定由于晃动引起的静水压力。
- **流体动力学耦合分析** - 在考虑流体约束质量的动力响应基础上，在结构动力学分析中使用流体耦合单元。



ANSYS 耦合场分析概览

耦合场分析 考虑两个或多个物理场之间的相互作用。如果两个物理场之间相互影响，单独求解一个物理场是不可能得到正确结果的，因此你需要一个能够将两个物理场组合到一起求解的分析软件。

例如：在压电力分析中，需要同时求解电压分布（电场分析）和应变（结构分析）。

其他需要耦合场分析的典型情况有：

- 热—应力分析
- 流体—结构相互作用
- 感应加热（电磁—热），感应振荡

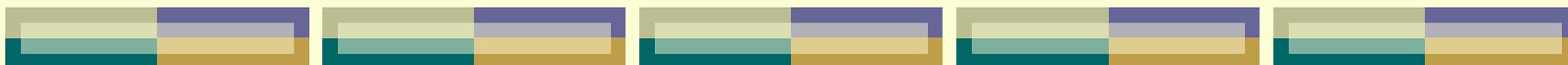


两根热膨胀系数不同的棒焊接在一起，图示为加热后的变形。

第三节 有限元分析实例

——龙门起重机金属结构有限元分析





计算模型分析

- **结构**: 按实际结构建模, 但对受力不大的次要部分可以进行简化。
- **载荷**: 根据《起重机设计规范》GB3811-83中有关规定, 确定如下计算工况。

分析类型		计算载荷
强度	小车在跨中	$\varphi_4 P_G, \varphi_4 G_{xc}, \varphi_2 P_B, P_H, P_{W,i}, P_S$
	小车在悬臂端	
刚度	小车在跨中	P_B, G_{xc}
	小车在悬臂端	

- **约束**: 将支座视为铰支座。

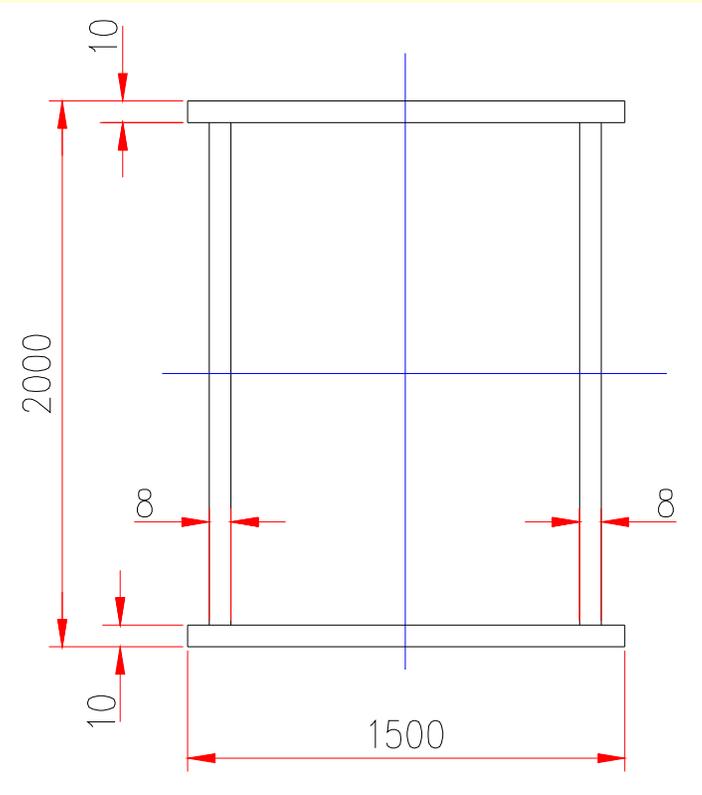
轨道式集装箱龙门起重机主要参数

起重量：50t

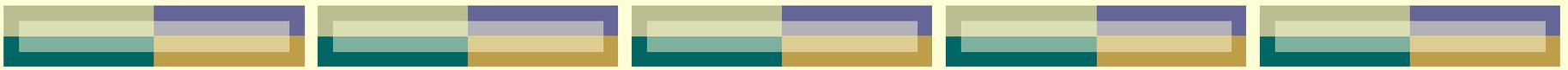
跨度：30m

有效悬臂长：7.5m

起升高度：12m



主梁截面尺寸



单元类型选择

ANSYS中有100多种单元。当板的长度与厚度之比大于20时应选用ANSYS中的壳单元（Shell）。

ANSYS中Shell单元的类型：

- Shell 41：采用薄膜理论。忽略弯曲和横向剪切，只包含薄膜效应。
- Shell 63：是“薄”壳单元。包含弯曲和薄膜效应但忽略横向剪切变形。
- Shell 43, 143, 181, 91, 93, 99：是“厚”壳单元。包含弯曲、薄膜和横向剪切效应。

对所分析结构，应选用Shell 63。

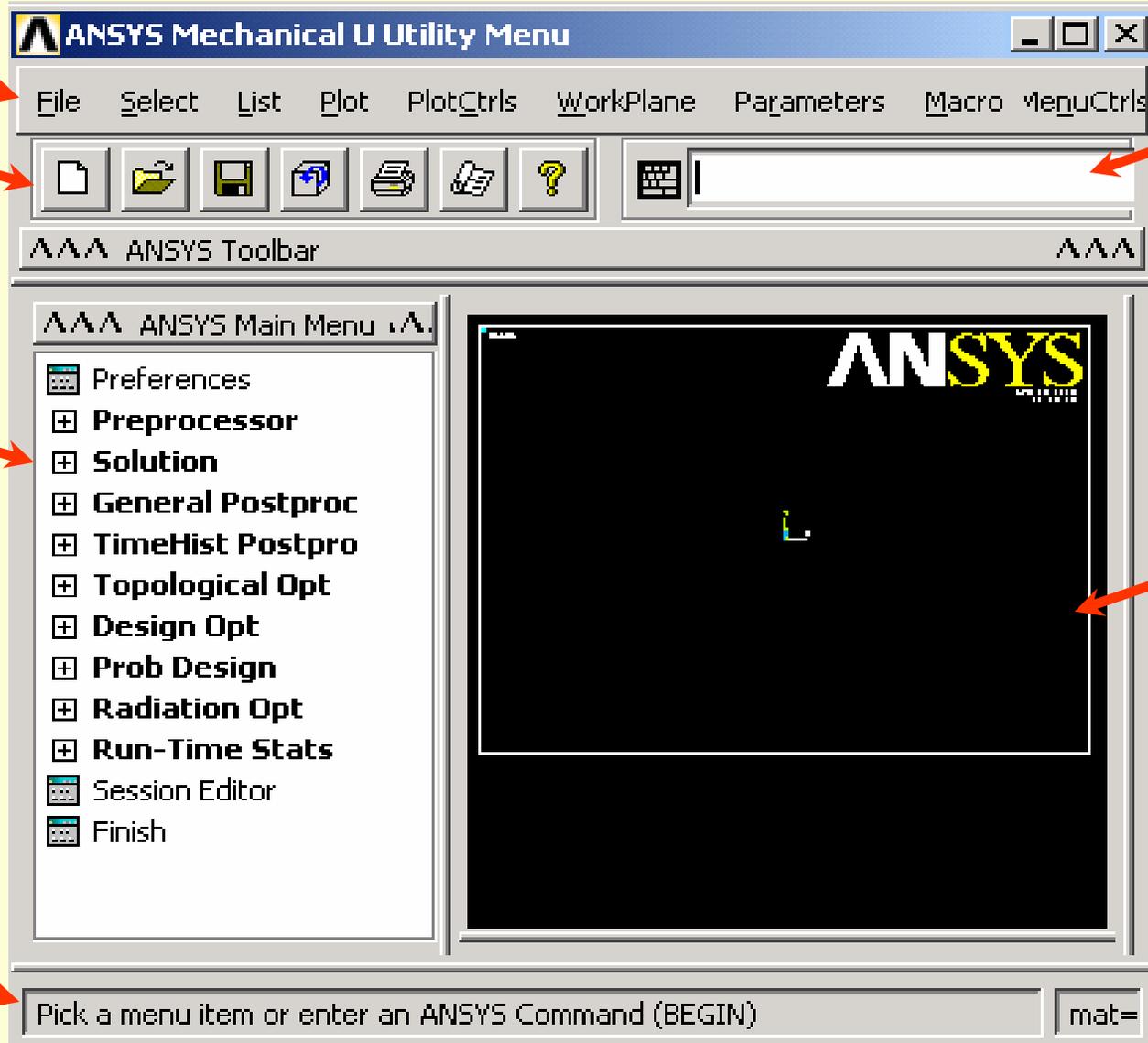
ANSYS的用户界面

应用菜单

工具条

主菜单

对话框



输入窗口

图形窗口

ANSYS Mechanical U

***** ANSYS COMMAND LINE ARGUMENTS *****

GRAPHICS DEVICE REQUESTED = win32
GRAPHICAL ENTRY = YES
LANGUAGE = en-us

00254413 VERSION=INTEL NT RELEASE= 7.0 UP20021010
CURRENT JOBNAME=file 17:36:09 APR 16, 2003 CP= 0.621

/SHOW SET WITH DRIVER NAME= WIN32 , RASTER MODE, GRAPHIC PLANES = 8

RUN SETUP PROCEDURE FROM FILE= C:\PROGRAM FILES\ANSYS INC\U70\ANSYS\apdl\start70
.ans

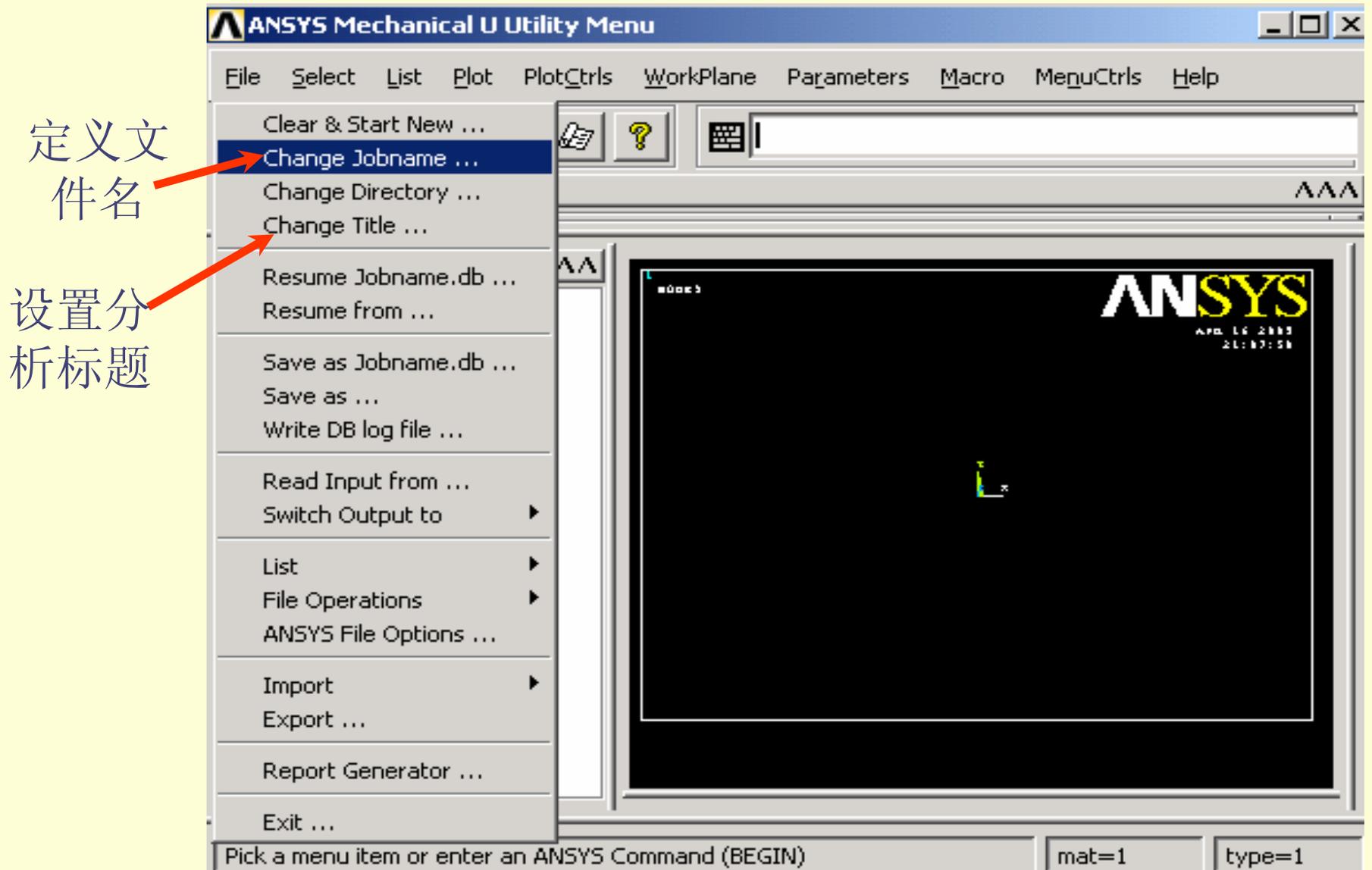
/INPUT FILE= menust.tmp LINE= 0

/INPUT FILE= C:\PROGRAM FILES\ANSYS INC\U70\ANSYS\apdl\start70.ans LINE=
0

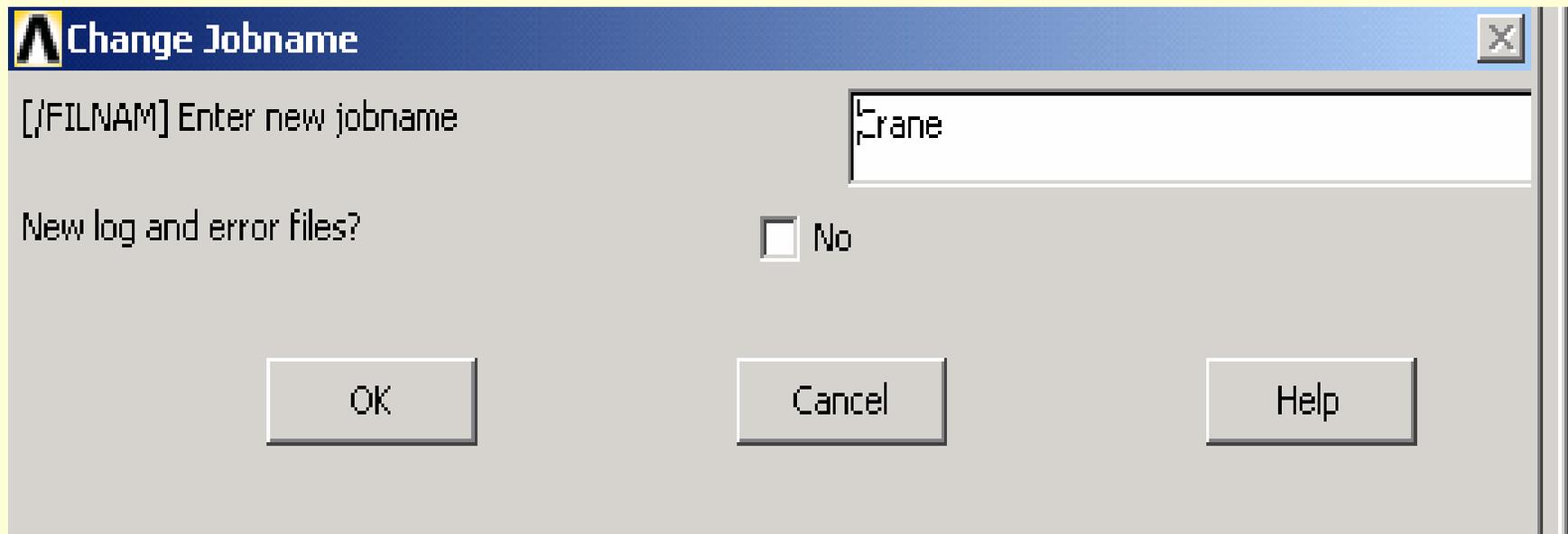
ACTIVATING THE GRAPHICAL USER INTERFACE <GUI>. PLEASE WAIT...

PRODUCE NODAL PLOT IN DSYS= 0

1. 定义文件名、设置分析标题



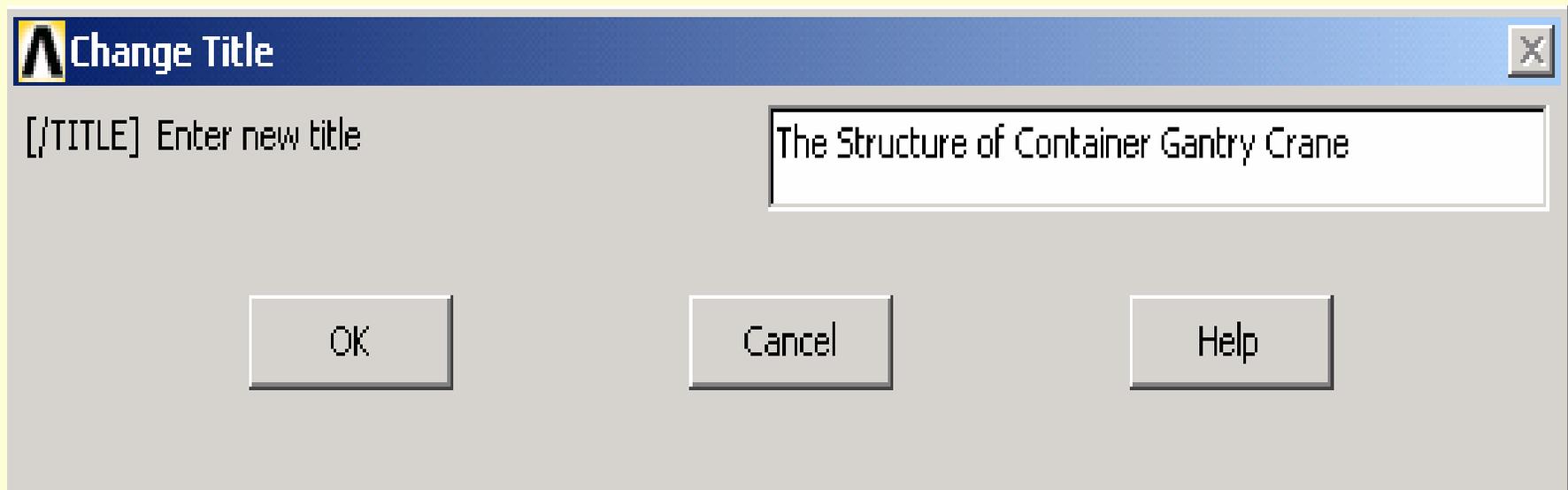
定义文件名

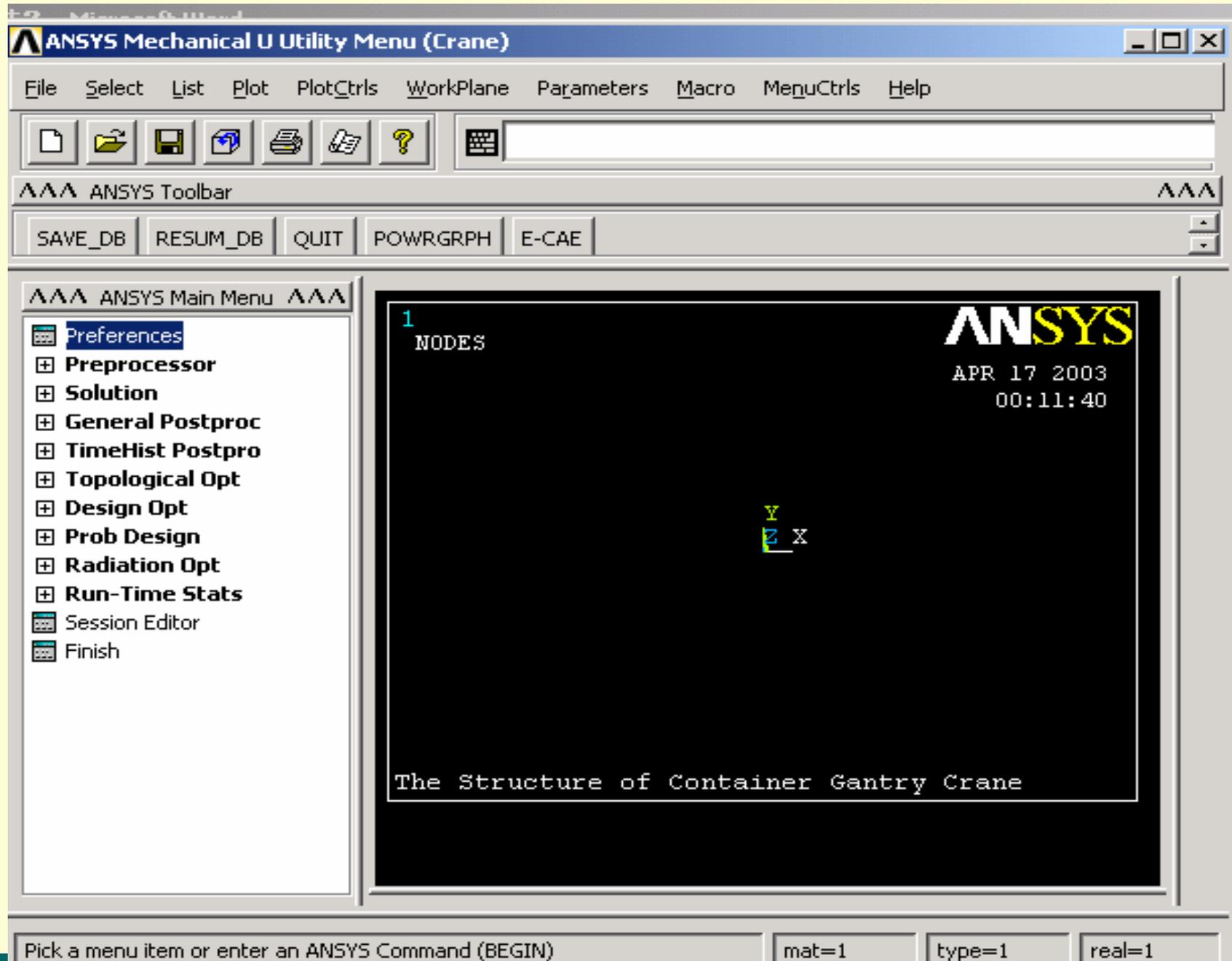


A screenshot of a Windows-style dialog box titled "Change Jobname". The dialog has a blue title bar with a close button (X) in the top right corner. The main area is light gray and contains the following elements:

- A label "[/FILNAM] Enter new jobname" on the left.
- A text input field on the right containing the text "crane".
- A label "New log and error files?" on the left.
- A checkbox labeled "No" on the right, which is currently unchecked.
- Three buttons at the bottom: "OK", "Cancel", and "Help".

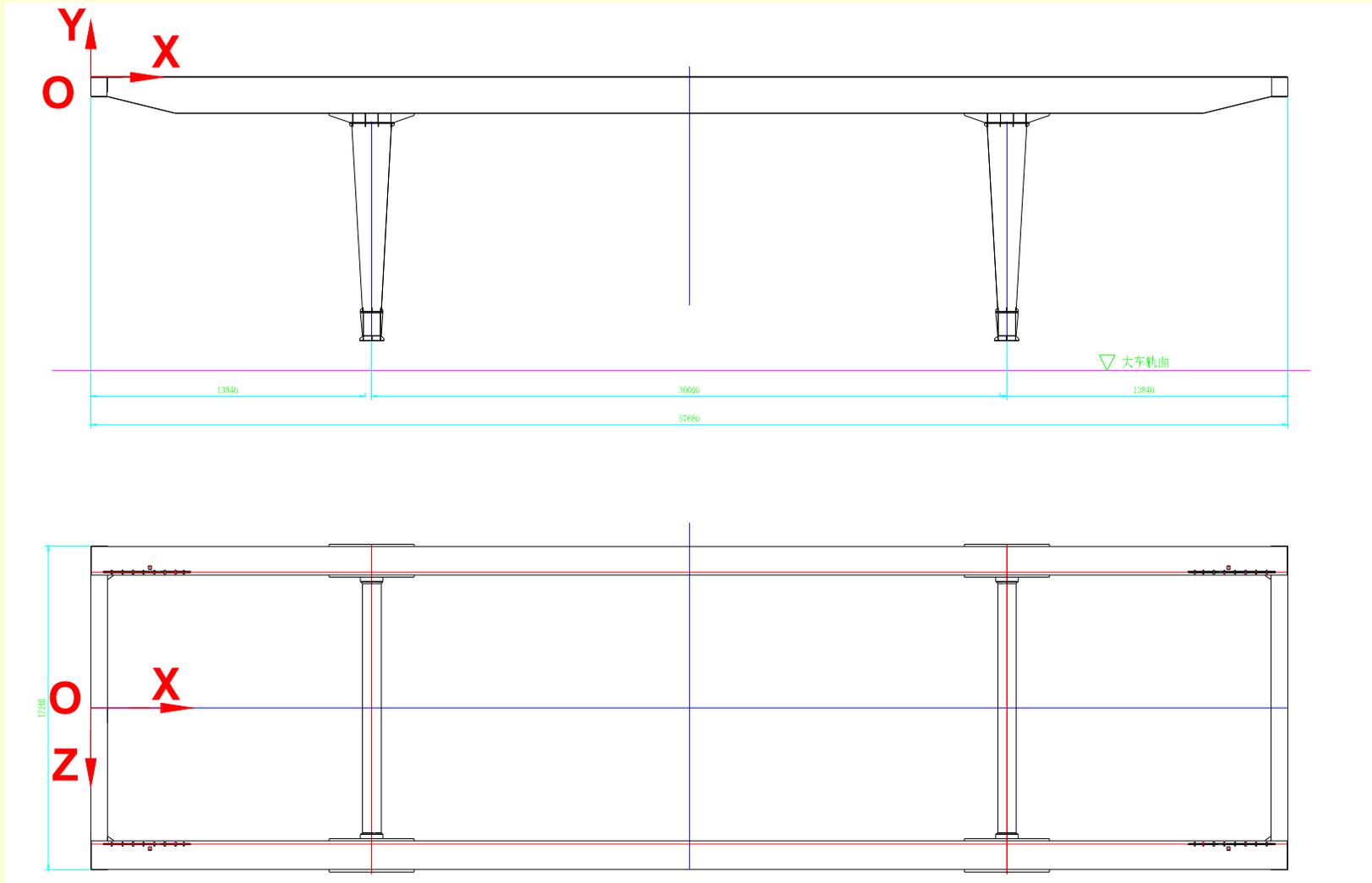
设置分析标题



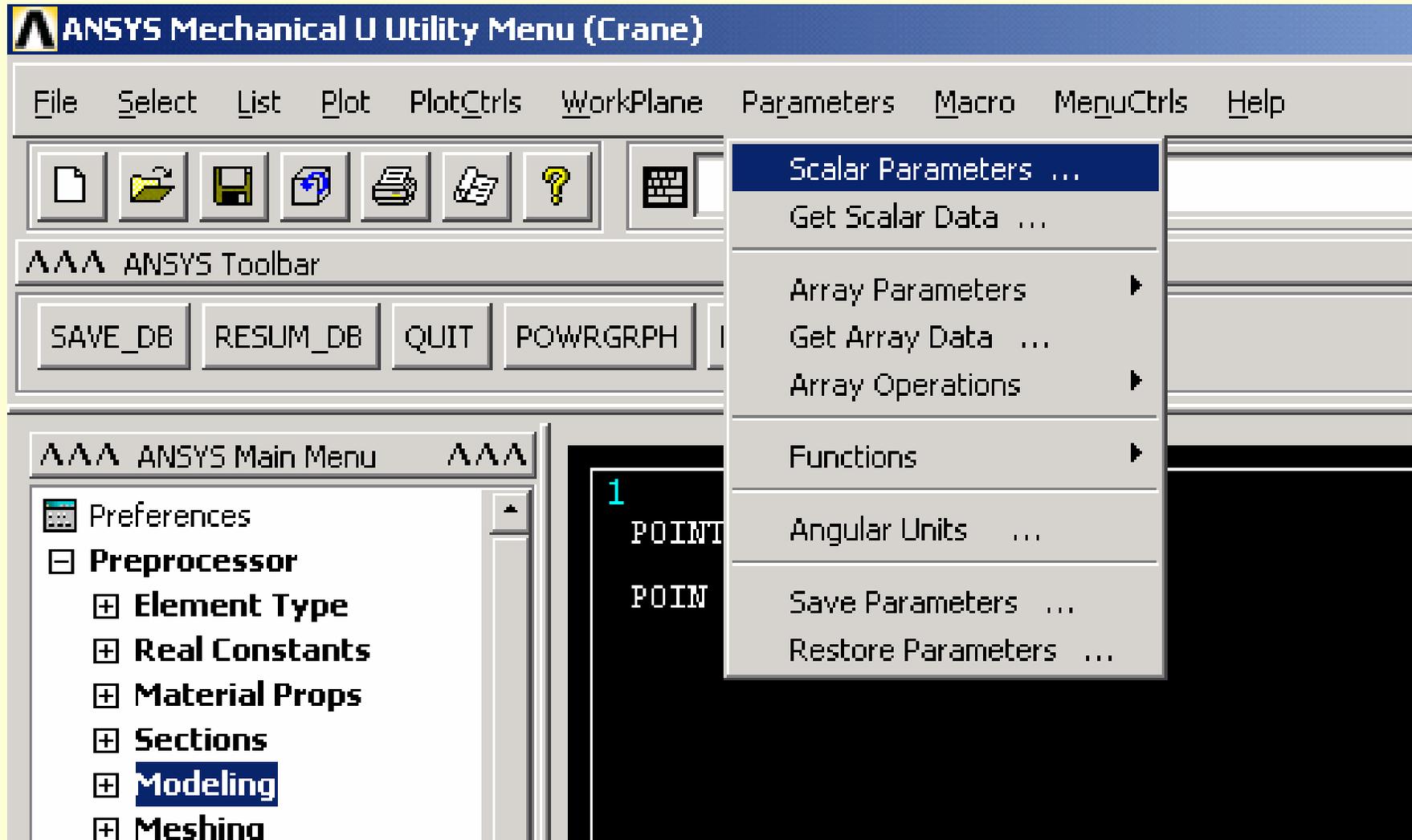


2. 定义参数表

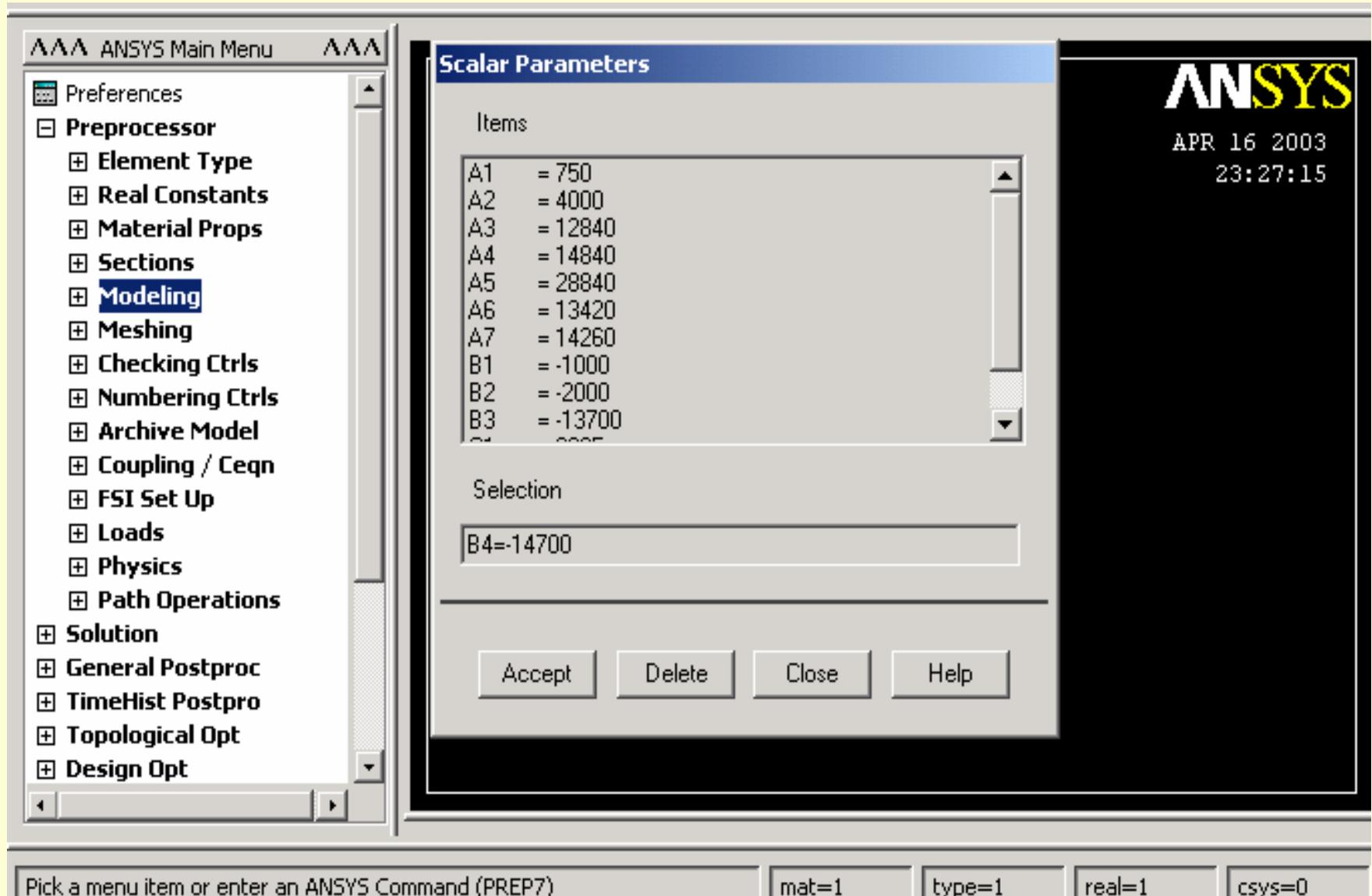
● 确定整体坐标位置



定义标量参数

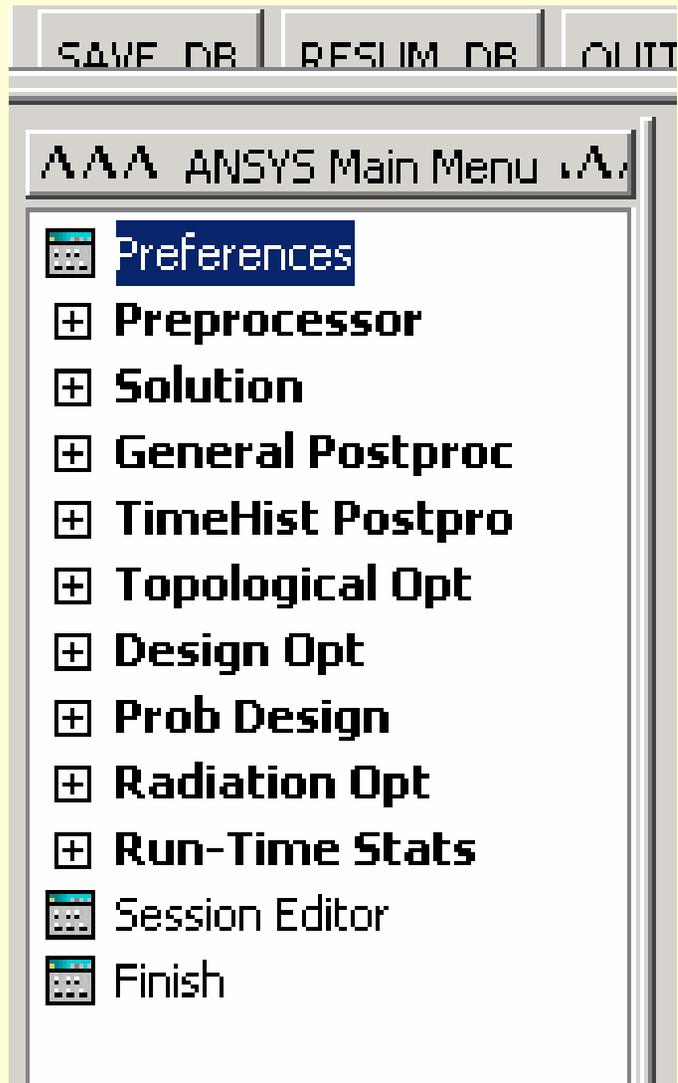


定义标量参数



3. 定义单元类型

● 设置分析类型



Preferences for GUI Filtering

[KEYW][,PMETH] Preferences for GUI Filtering

Individual discipline(s) to show in the GUI

- Structural
- Thermal
- ANSYS Fluid

Note: If no individual disciplines are selected they will all show.

Discipline options

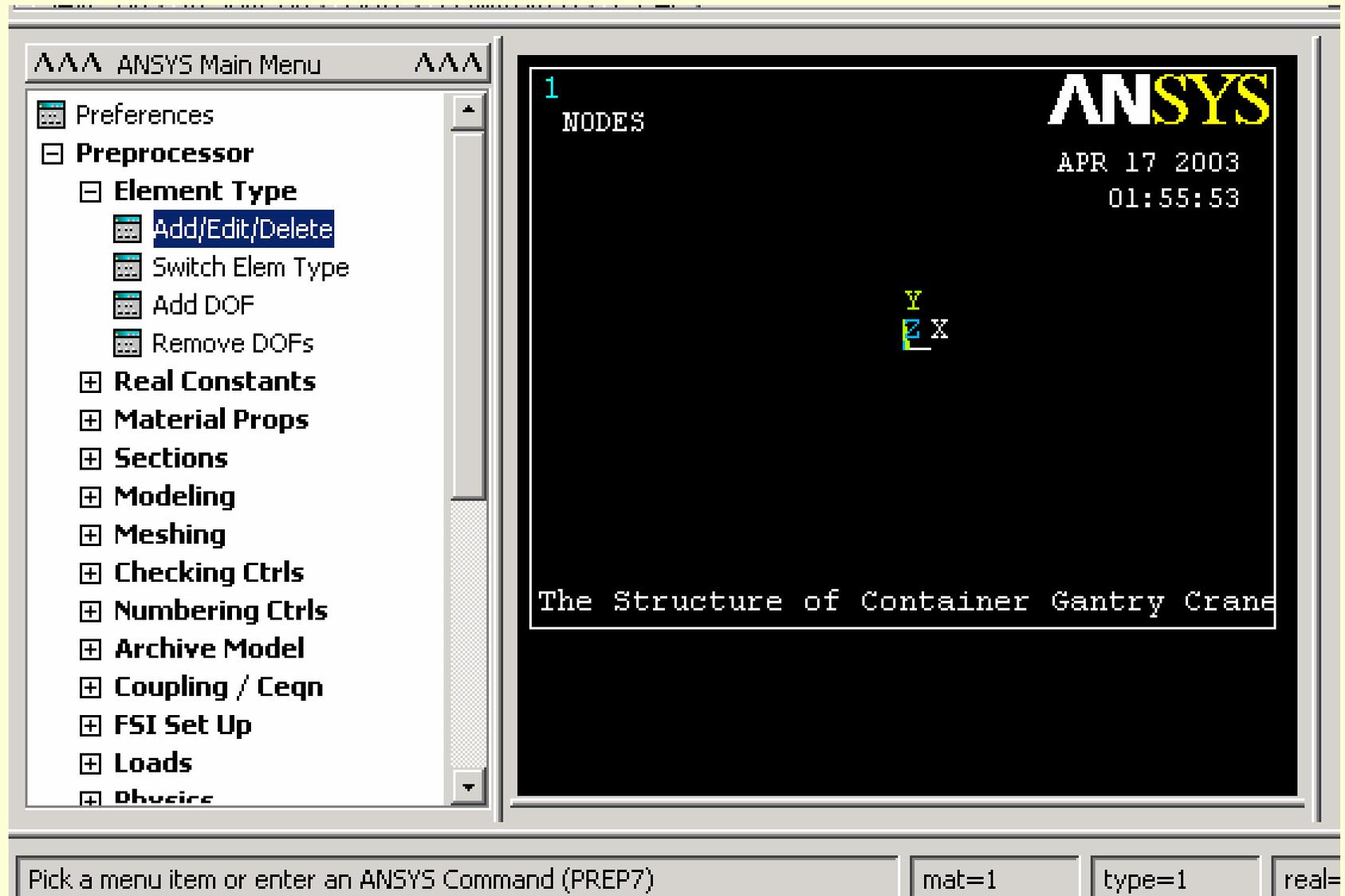
- h-Method
- p-Method Struct.

OK

Cancel

Help

定义单元类型



Element Types

Defined Element Types:

NONE DEFINED

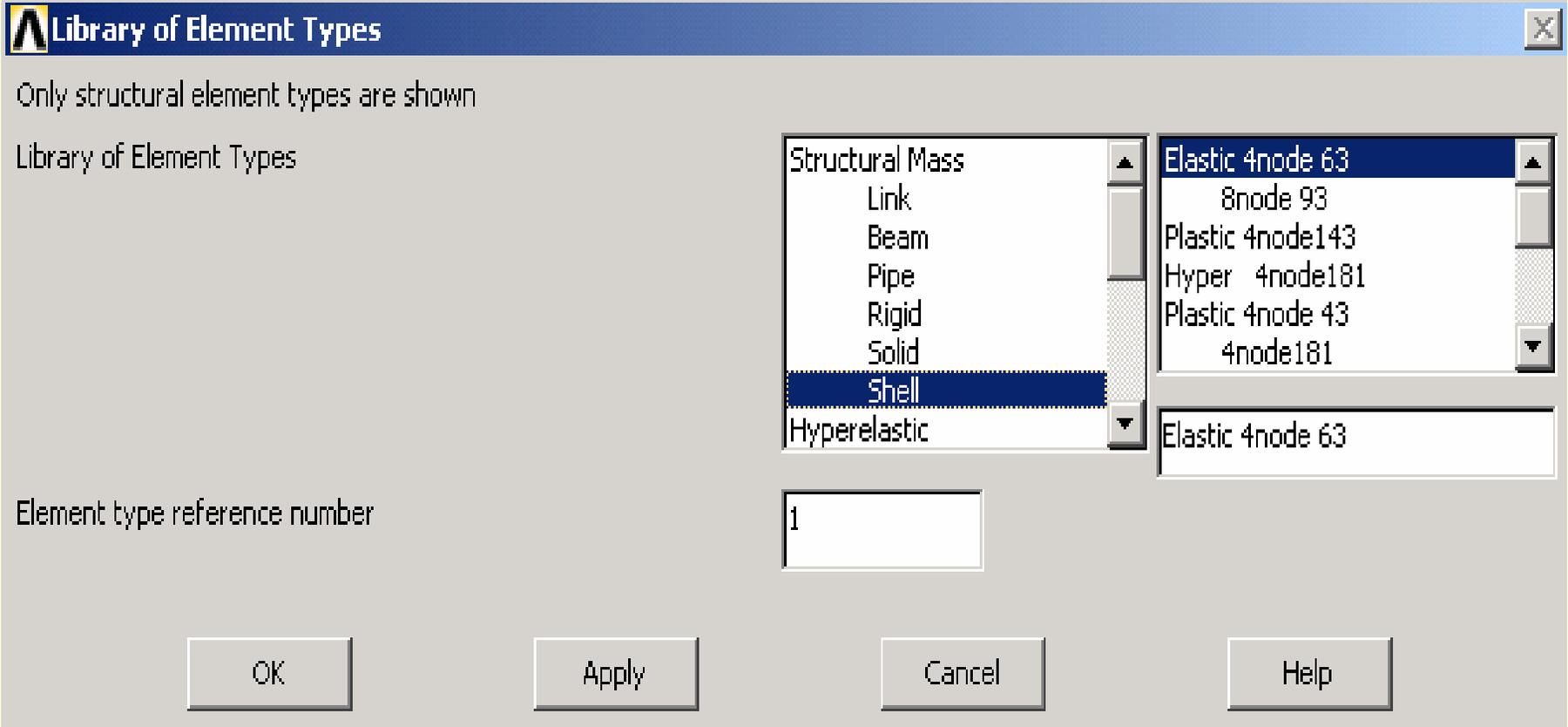
Add...

Options...

Delete

Close

Help



Element Types

Defined Element Types:

Type	1	SHELL63
------	---	---------

Add...

Options...

Delete

Close

Help

SHELL63 element type options

Options for SHELL63, Element Type Ref. No. 1

Element stiffness K1

Bnding and membr

Stress stiffening option K2

Bnding and membr

Membrane only

Bending only

Extra displacement shapes K3

Include

Extra stress output K5

No extra output

Pressure loading K6

Reduced loading

Mass matrix K7

Consistent

Stiffness matrix K8

Consistent

Element coord sys defined by K9

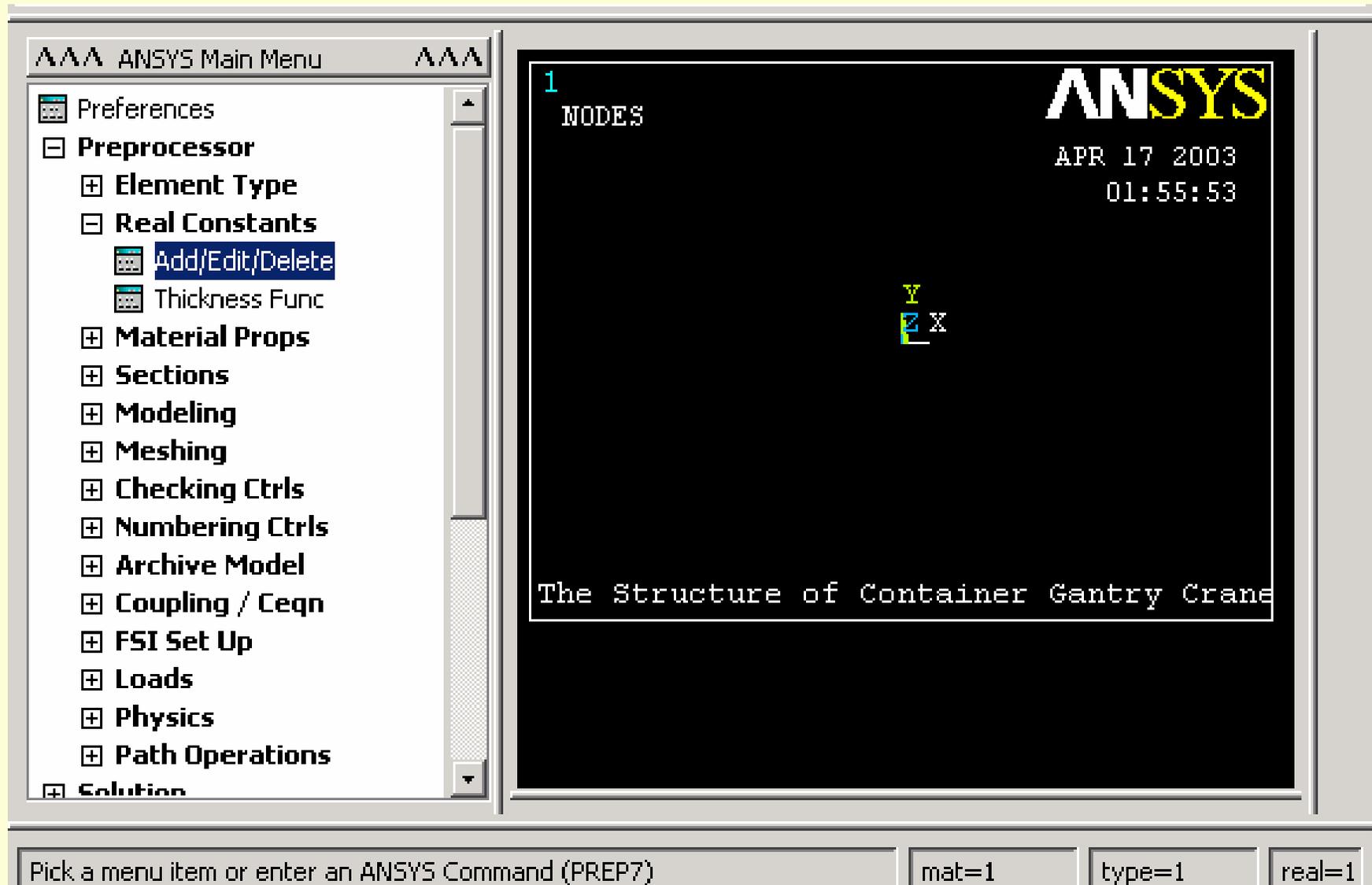
Elem orientation

OK

Cancel

Help

4. 定义实常数



Real Constants [X]

Defined Real Constant Sets

NONE DEFINED

Add...

Edit...

Delete

Close

Help



Real Constant Set Number 1, for SHELL63 [X]

Element Type Reference No. 1

Real Constant Set No.

Shell thickness at node I TK(I)

at node J TK(J)

at node K TK(K)

at node L TK(L)

Elastic foundation stiffness EFS

Element X-axis rotation THETA

Bending mom of inertia ratio RMI

Dist from mid surf to top CTOP

Dist from mid surf to bot CBOT

Added mass/unit area ADMSUA

OK

Apply

Cancel

Help

Real Constant Set Number 2, for SHELL63

Element Type Reference No. 1

Real Constant Set No.

Shell thickness at node I TK(I)

 at node J TK(J)

 at node K TK(K)

 at node L TK(L)

Elastic foundation stiffness EFS

Element X-axis rotation THETA

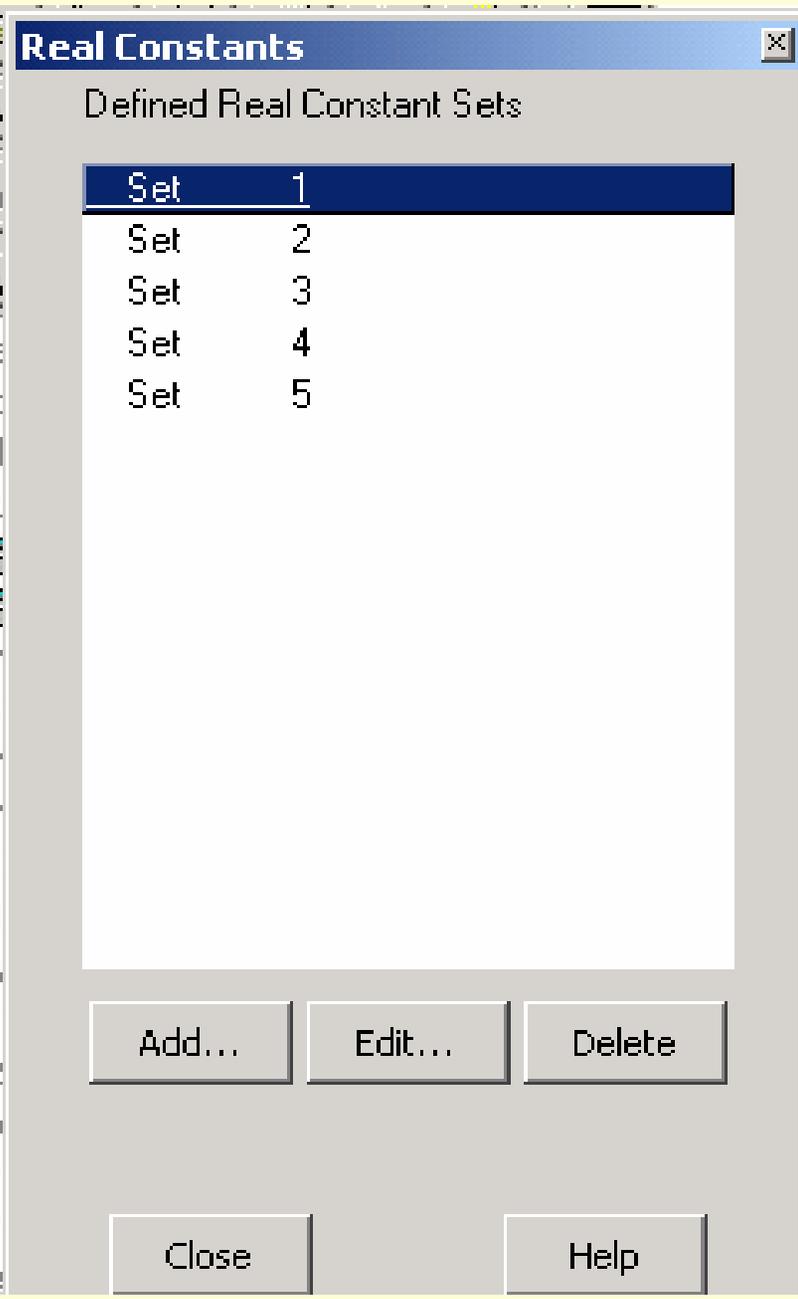
Bending mom of inertia ratio RMI

Dist from mid surf to top CTOP

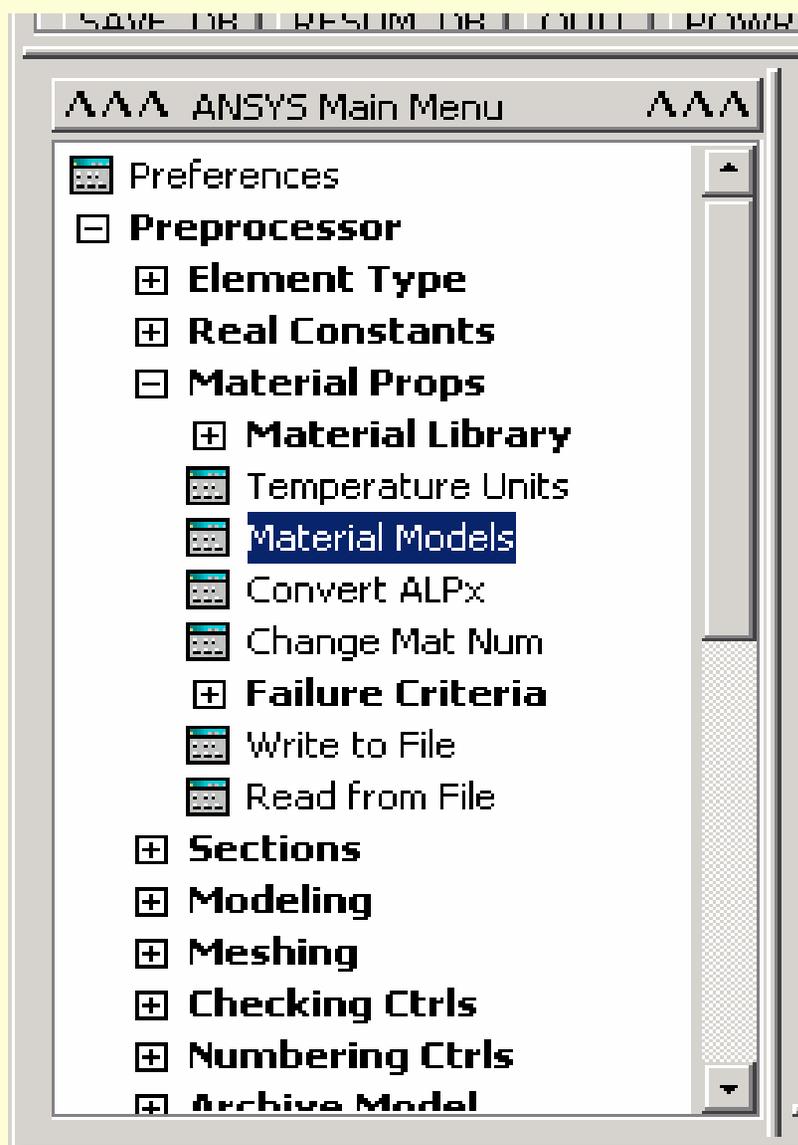
Dist from mid surf to bot CBOT

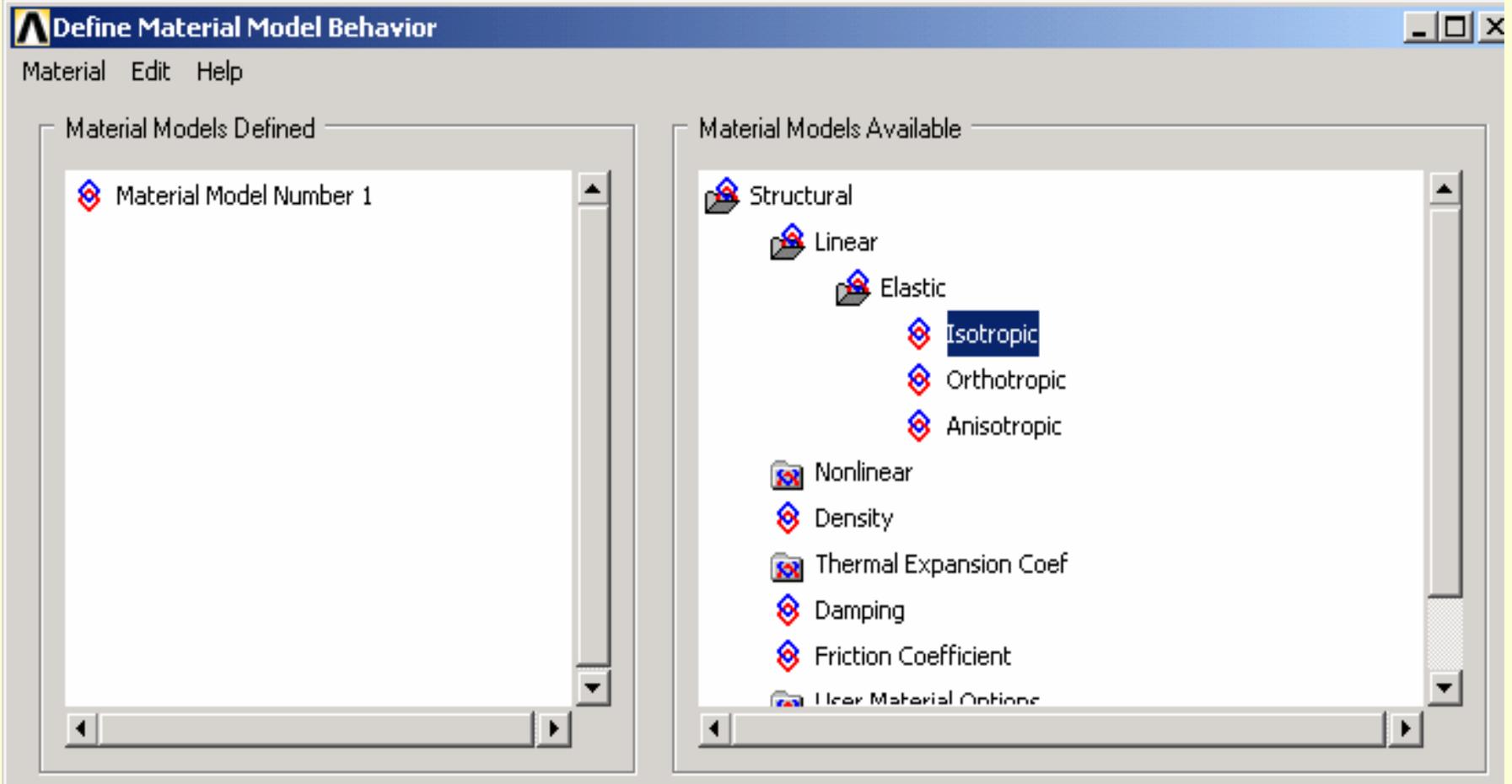
Added mass/unit area ADMSUA

OK Apply Cancel Help



5. 定义材料特性





Linear Isotropic Properties for Material Number 1

Linear Isotropic Material Properties for Material Number 1

T1

Temperatures

EX

2.1e5

PRXY

0.3

Add Temperature

Delete Temperature

Graph

OK

Cancel

Help

Define Material Model Behavior

Material Edit Help

Material Models Defined

- Material Model Number 1
 - Linear Isotropic

Material Models Available

- Structural
 - Linear
 - Elastic
 - Isotropic
 - Orthotropic
 - Anisotropic
 - Nonlinear
 - Density
 - Thermal Expansion Coef
 - Damping
 - Friction Coefficient
 - User Material Options

Density for Material Number 1

Density for Material Number 1

	T1
Temperatures	
DENS	7.85e-5

Add Temperature

Delete Temperature

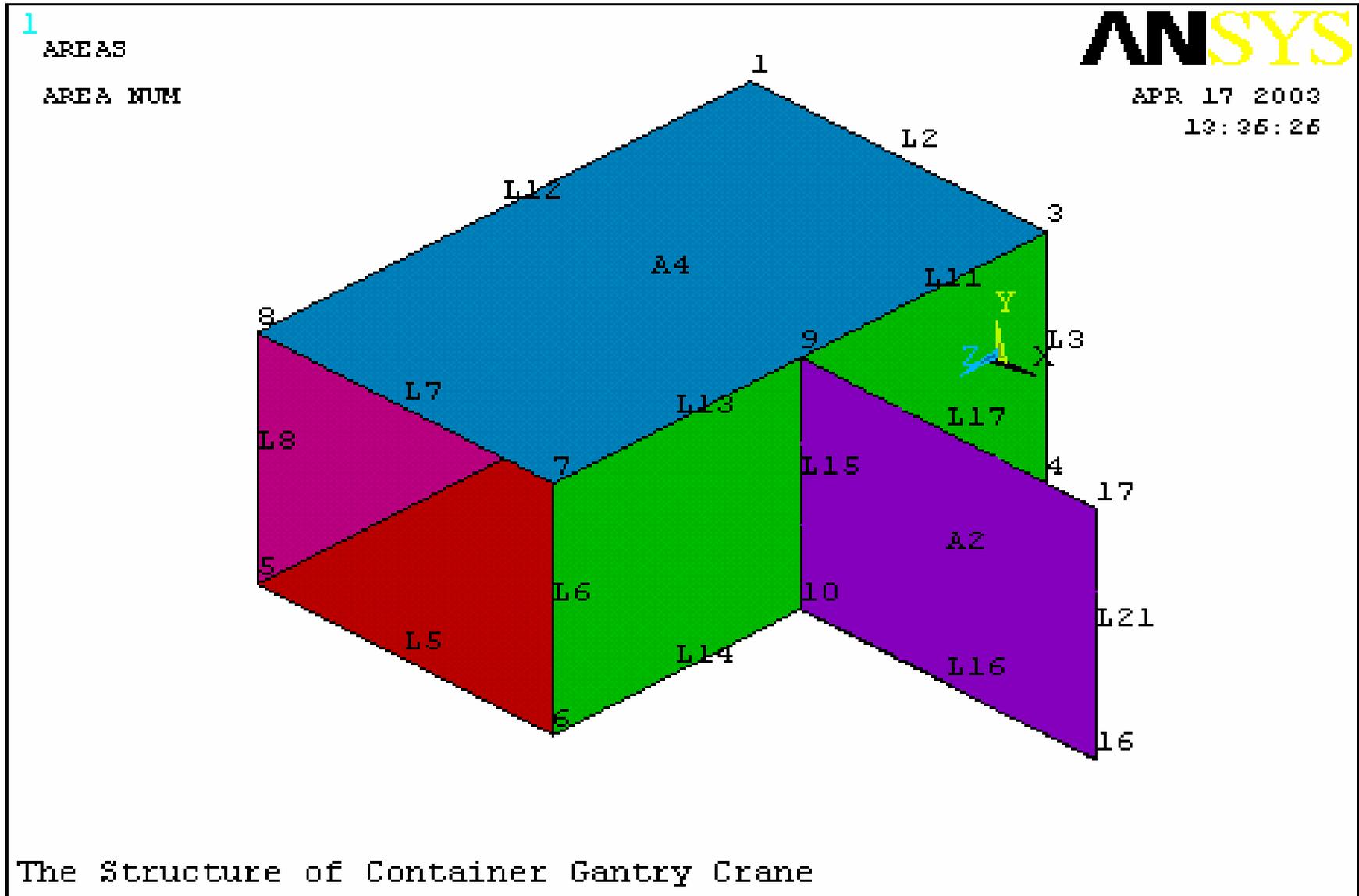
Graph

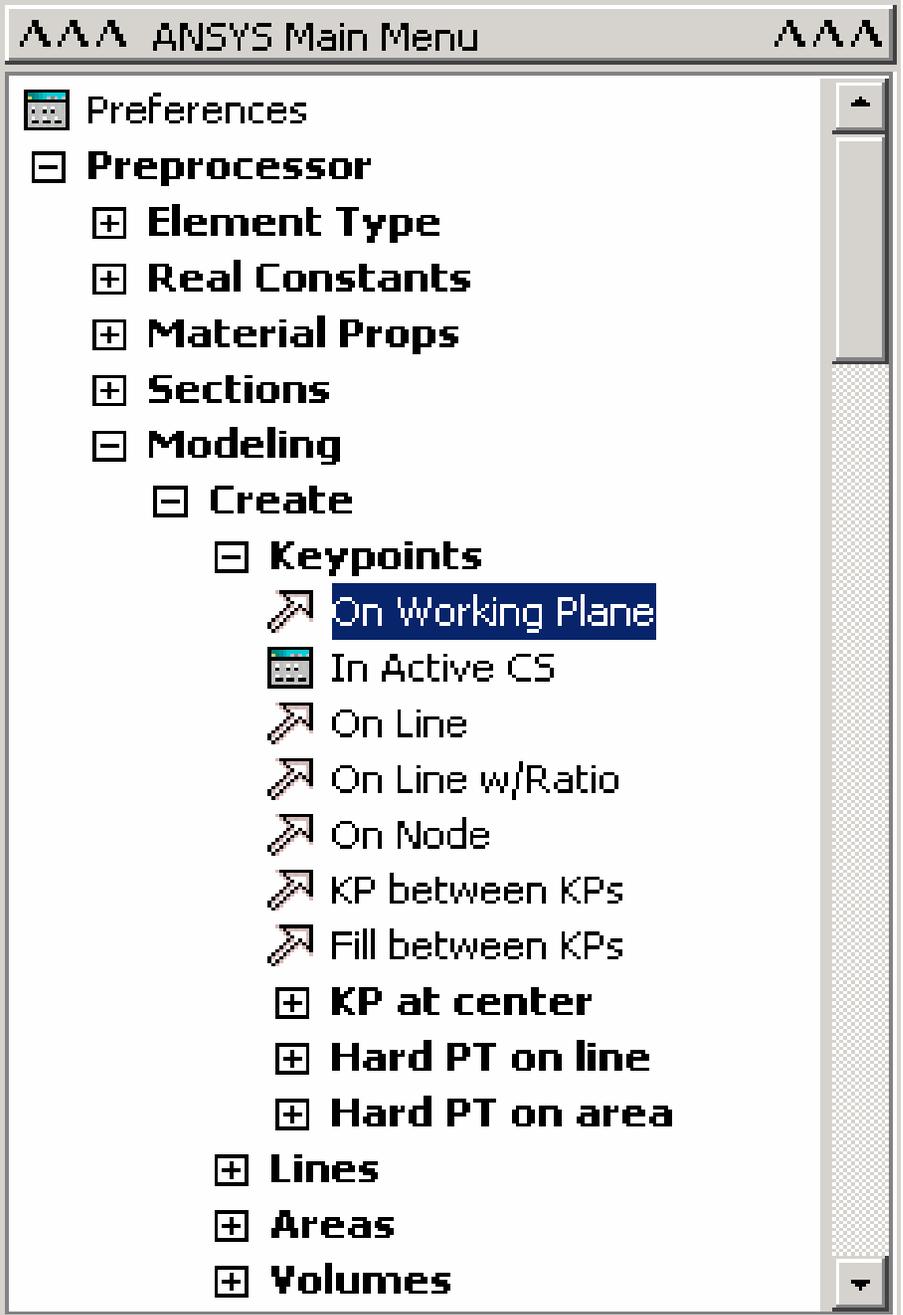
OK

Cancel

Help

6. 创建关键点





Create Keypoints in Active Coordinate System

[K] Create Keypoints in Active Coordinate System

NPT Keypoint number

X,Y,Z Location in active CS

 Create Keypoints in Active Coordinate System ✕

[K] Create Keypoints in Active Coordinate System

NPT Keypoint number

X,Y,Z Location in active CS

1
POINTS Y
POIN NUM 1 4X
2 3

ANSYS

APR 17 2003
14:25:30

5 8
9 13
10 14
11 16
15

17 21
18 22
20 24
19 23

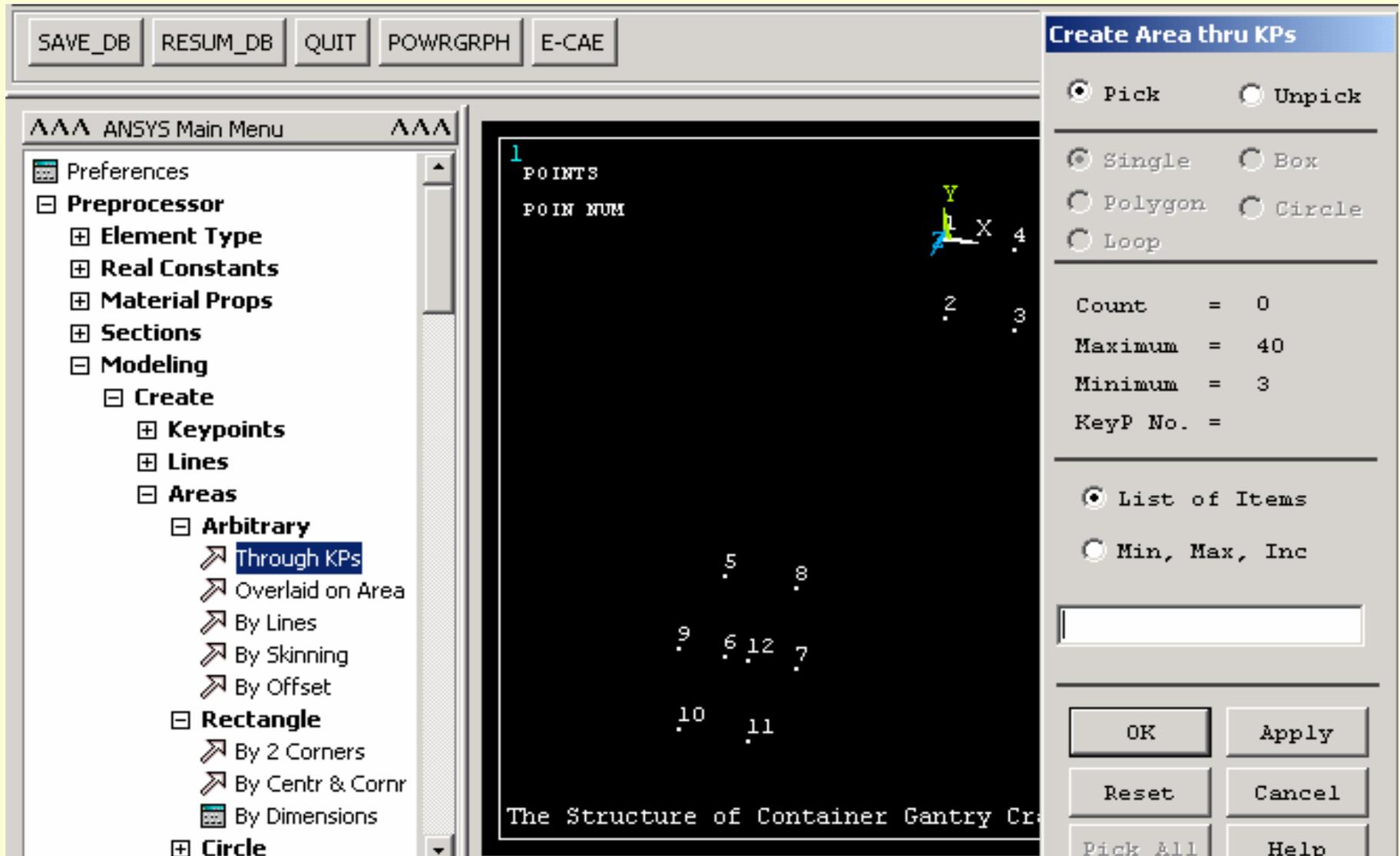
25
26
28
27

39 38
40 37

29 32
30 35
33 36

The Structure of Container Gantry Crane

7. 创建面



1

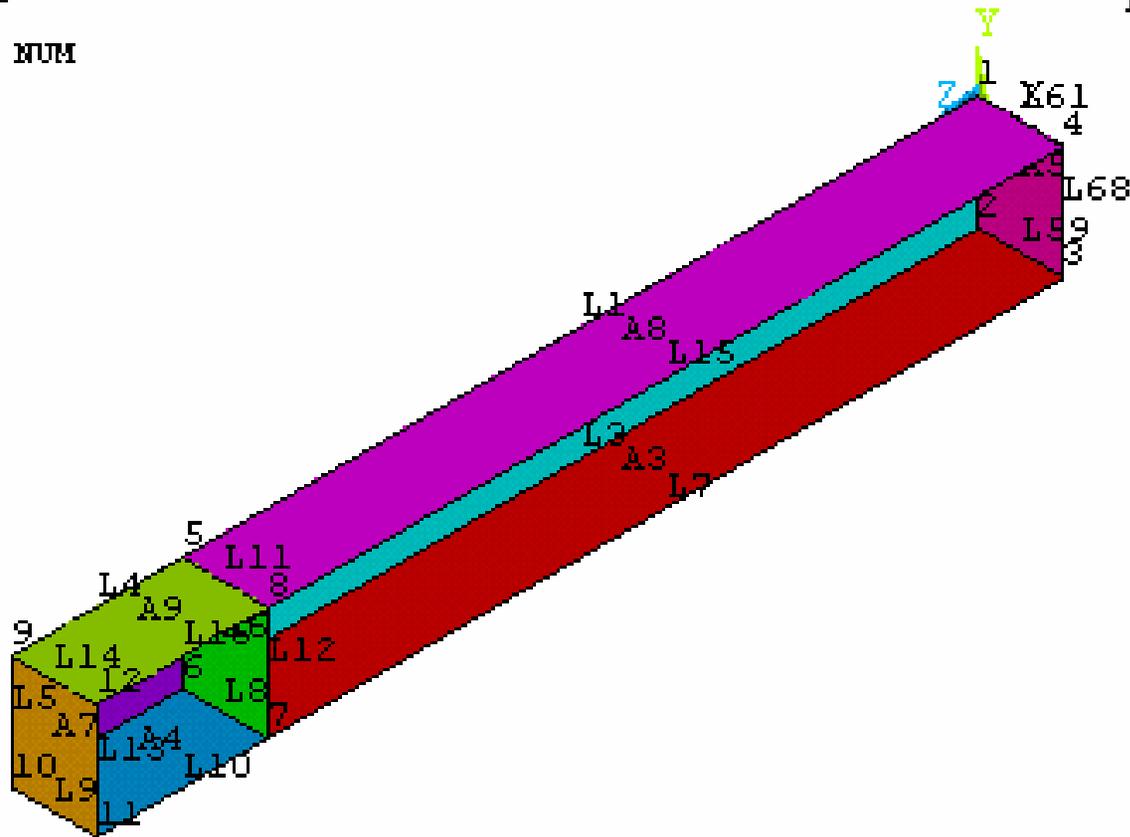
AREAS

AREA NUM

ANSYS

APR 17 2003

14:44:15



The Structure of Container Gantry Crane

1

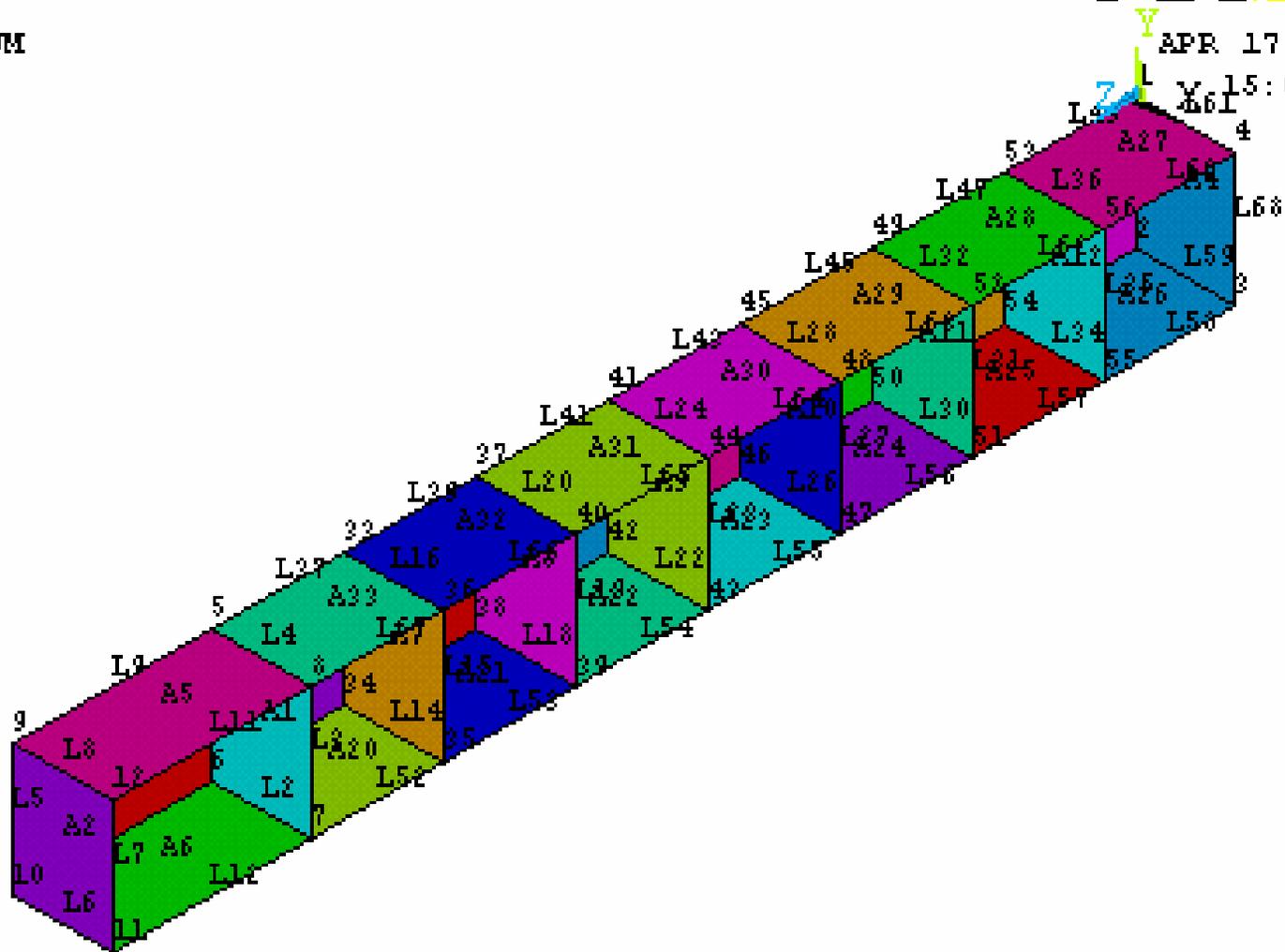
AREAS

AREA NUM

ANSYS

APR 17 2003

15:01:06



1

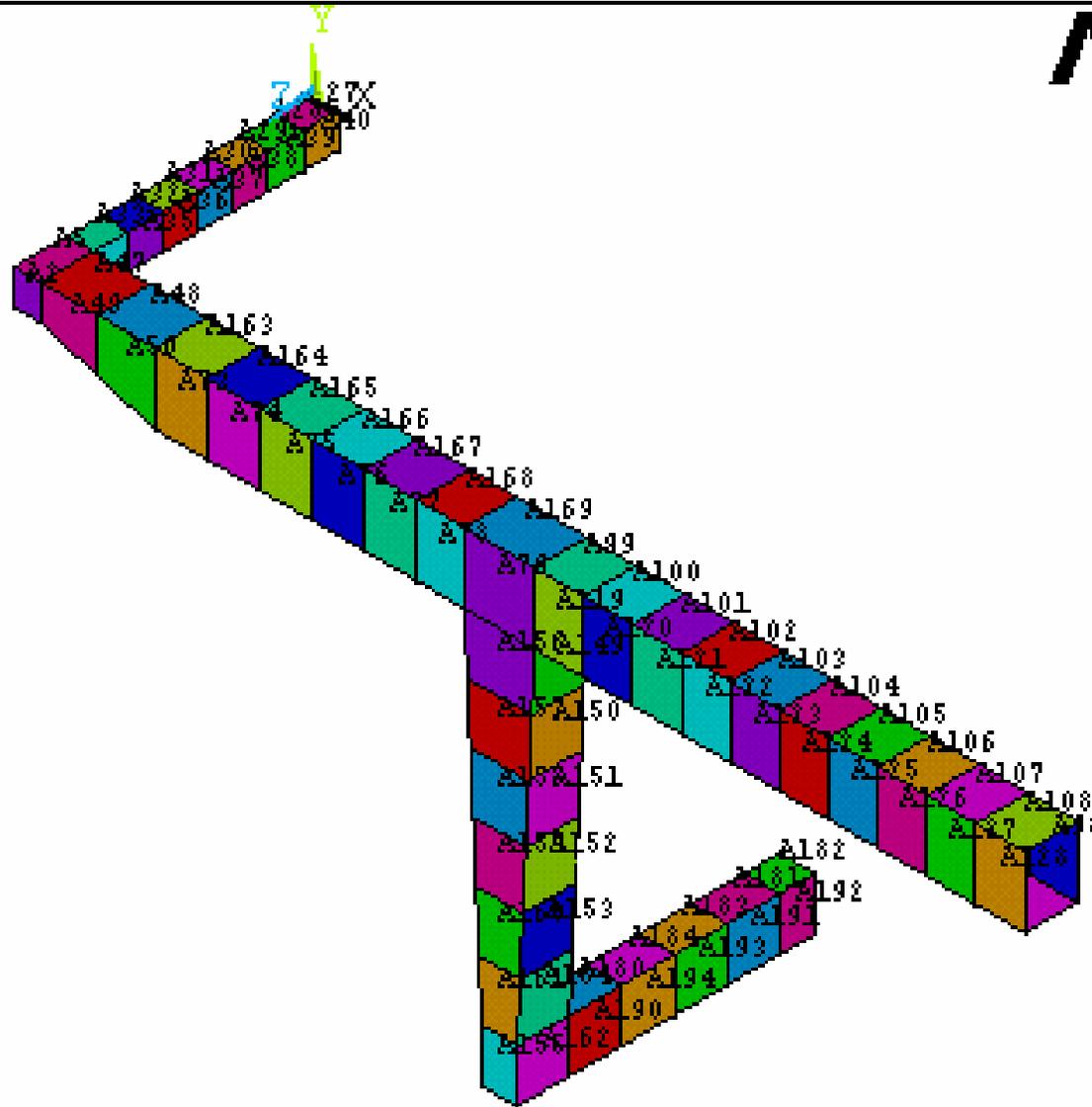
AREAS

AREA NUM

ANSYS

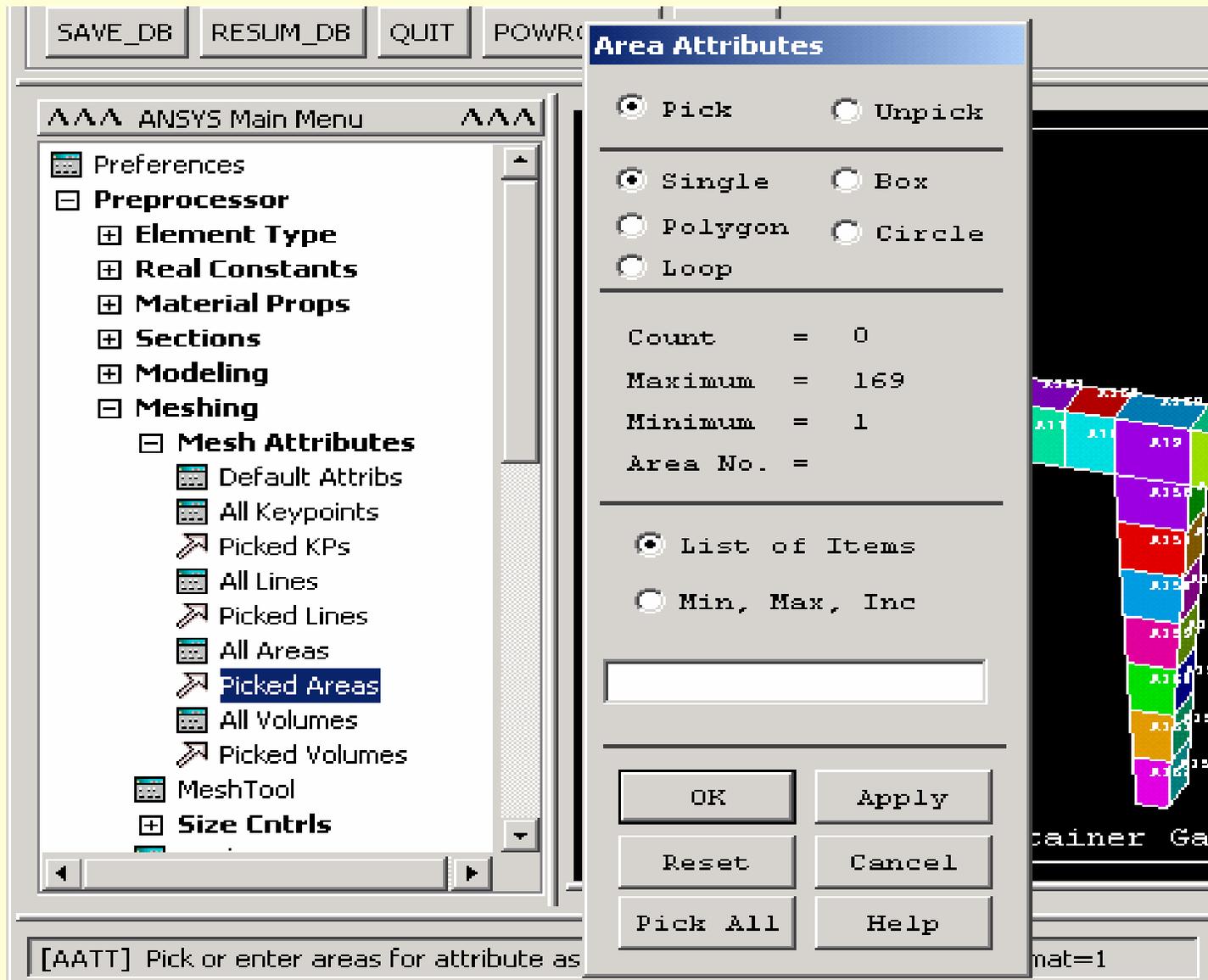
APR 17 2003

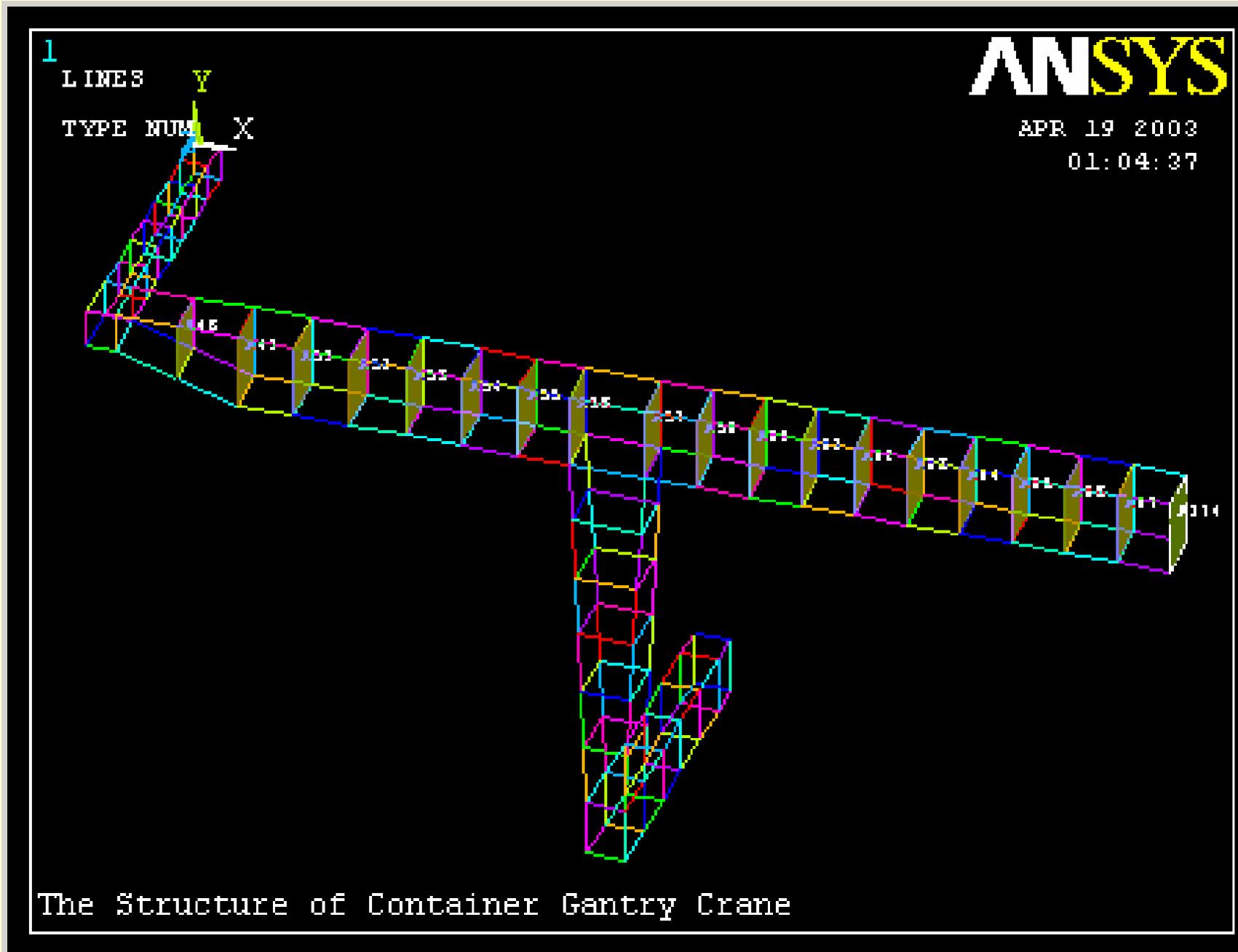
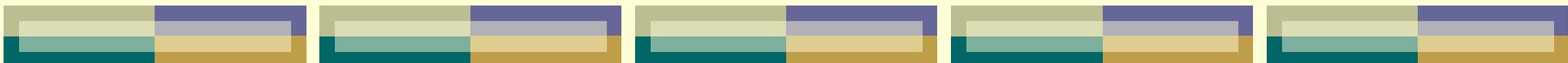
15:08:44



The Structure of Container Gantry Crane

8. 设置板厚





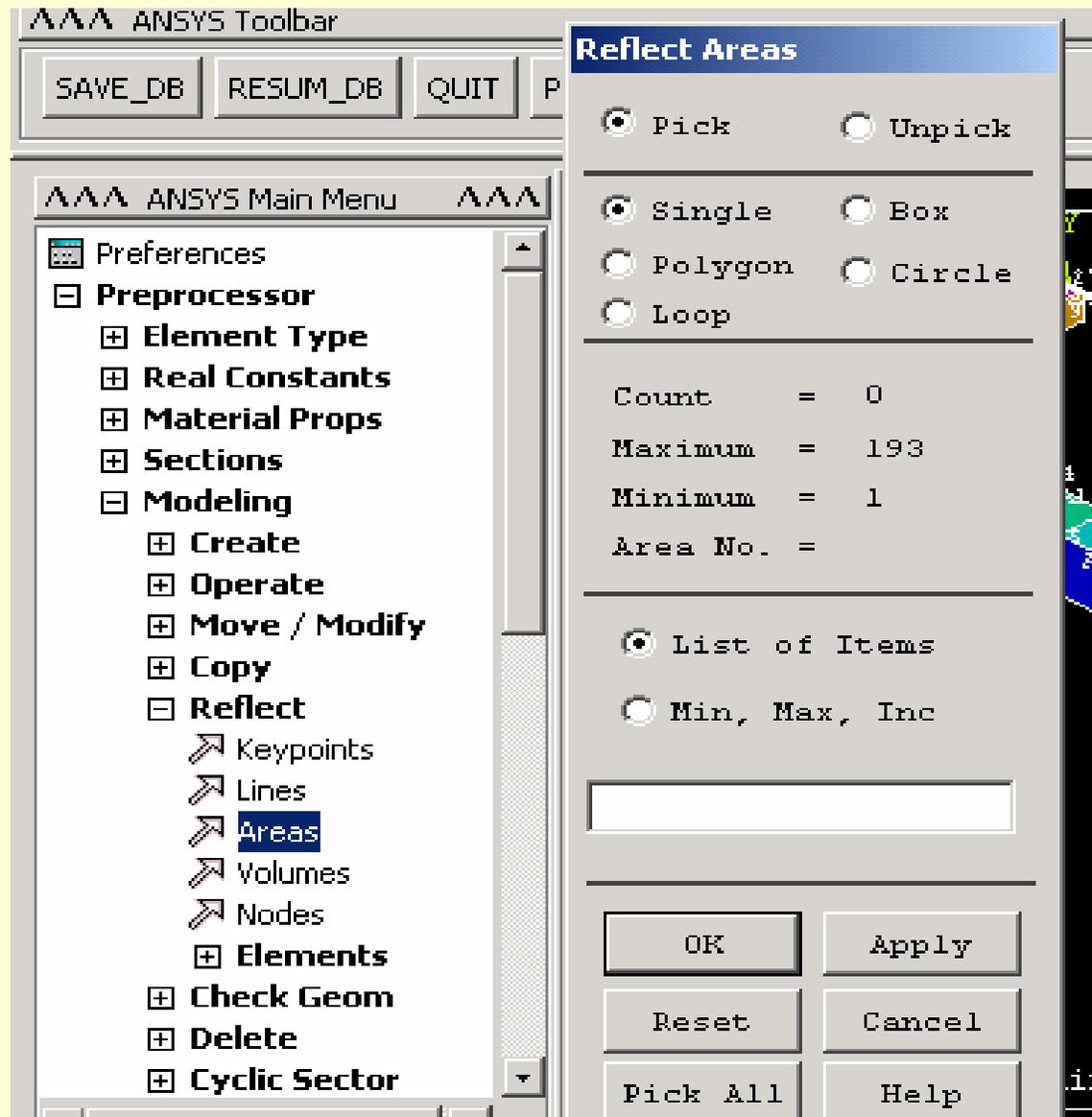
Area Attributes [X]

[AATT] Assign Attributes to Picked Areas

MAT	Material number	1
REAL	Real constant set number	1
TYPE	Element type number	1 2 3 4 5
ESYS	Element coordinate sys	
SECT	Element section	None defined

OK Apply Cancel Help

9. 创建整体结构



Reflect Areas

[ARSYM] Reflect Areas

Ncomp Plane of symmetry

Y-Z plane X

X-Z plane Y

X-Y plane Z

KINC Keypoint increment

NOELEM Items to be reflected

IMOVE Existing areas will be

OK

Apply

Cancel

Help

1

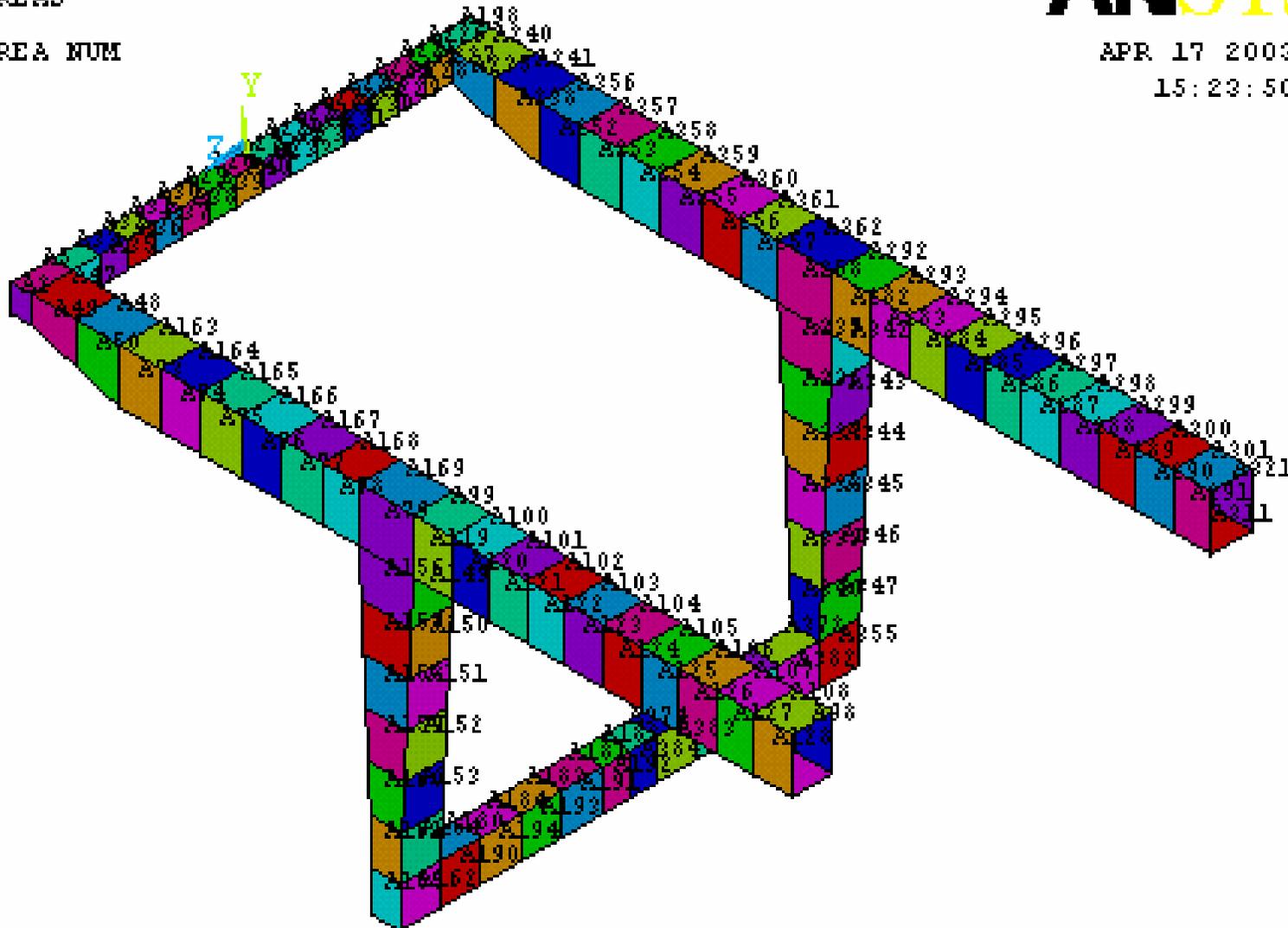
AREAS

AREA NUM

ANSYS

APR 17 2003

15:23:50



The Structure of Container Gantry Crane

1

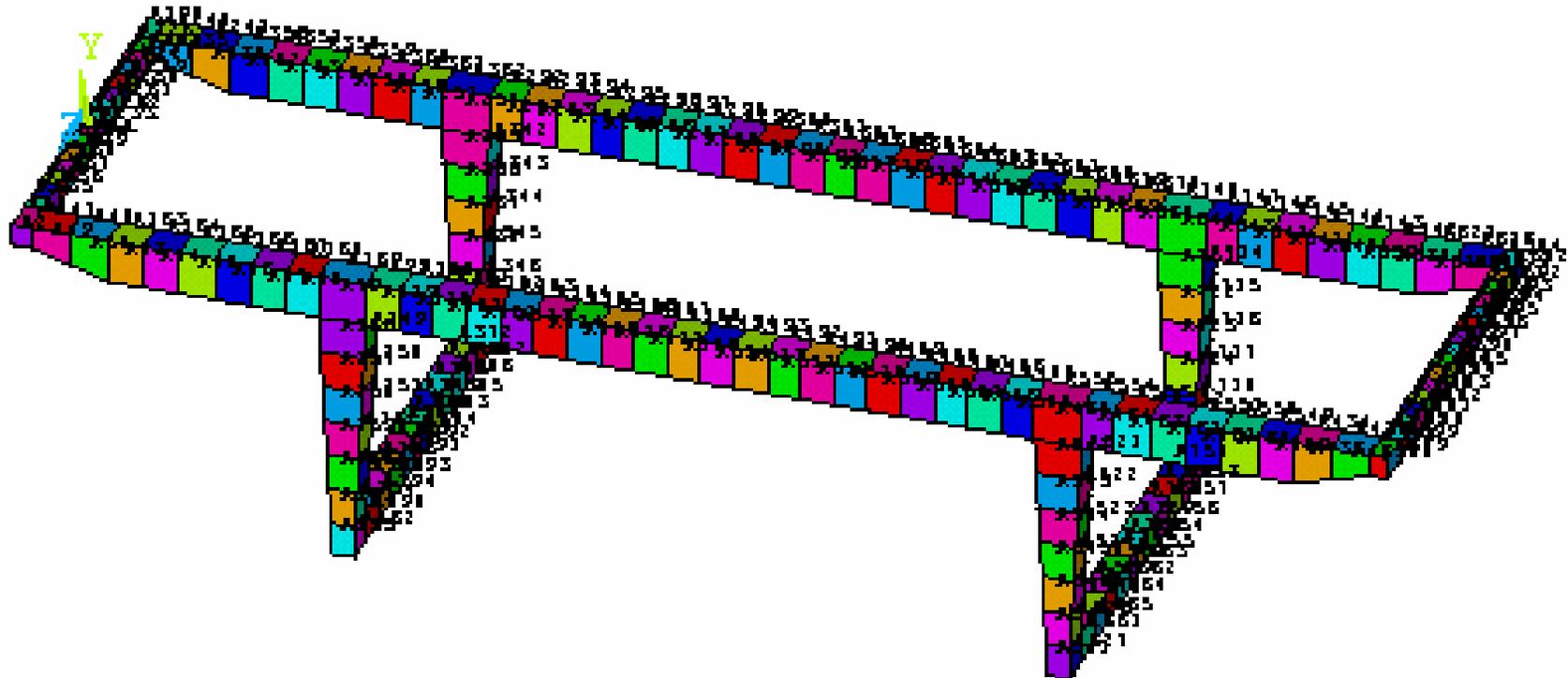
AREAS

AREA NUM

ANSYS

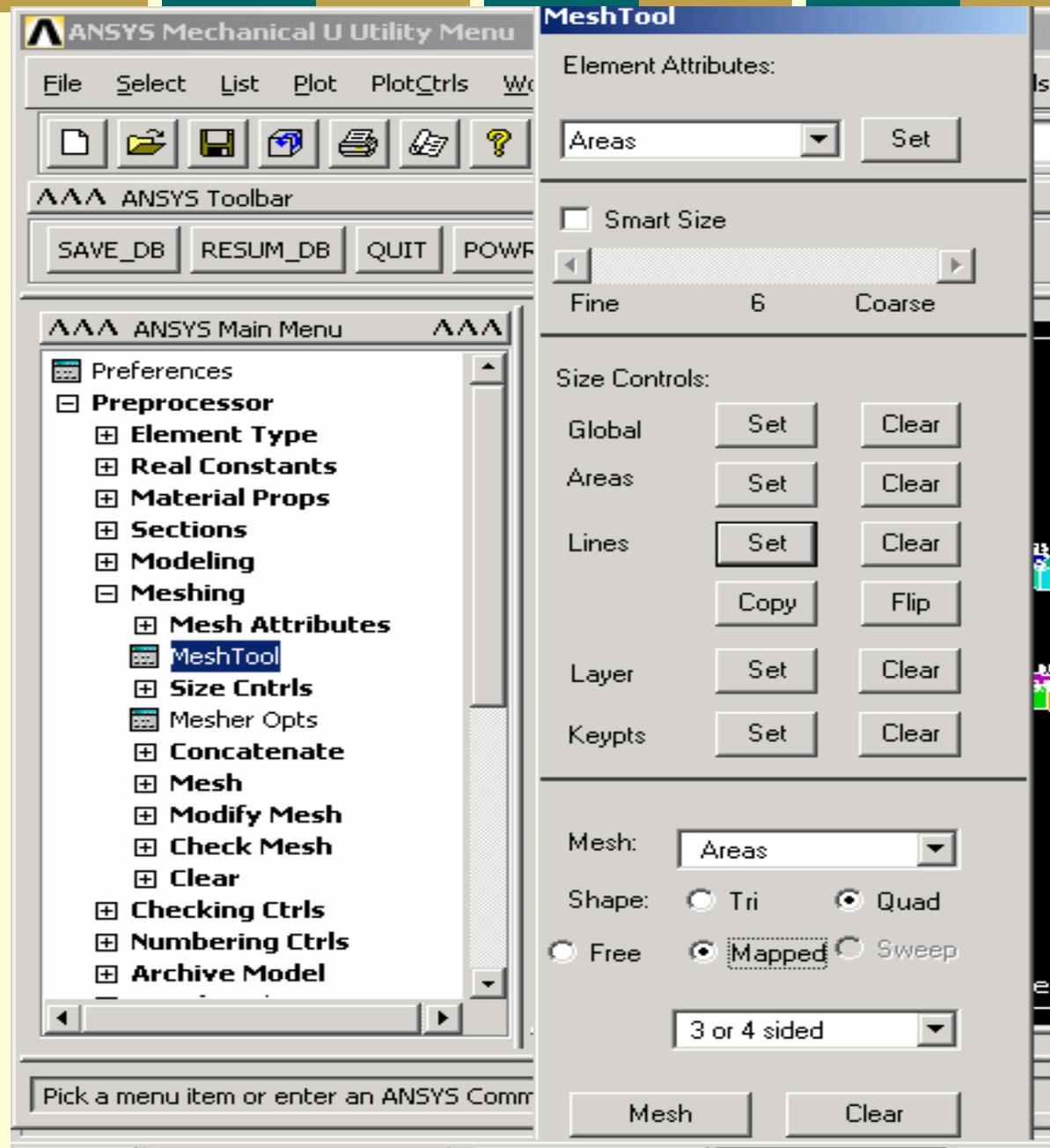
APR 17 2003

15:30:32



The Structure of Container Gantry Crane

10. 划分网络



Element Sizes on Picked Lines

[LESIZE] Element sizes on picked lines

SIZE Element edge length

NDIV No. of element divisions

(NDIV is used only if SIZE is blank or zero)

KYNDIV SIZE,NDIV can be changed

 Yes

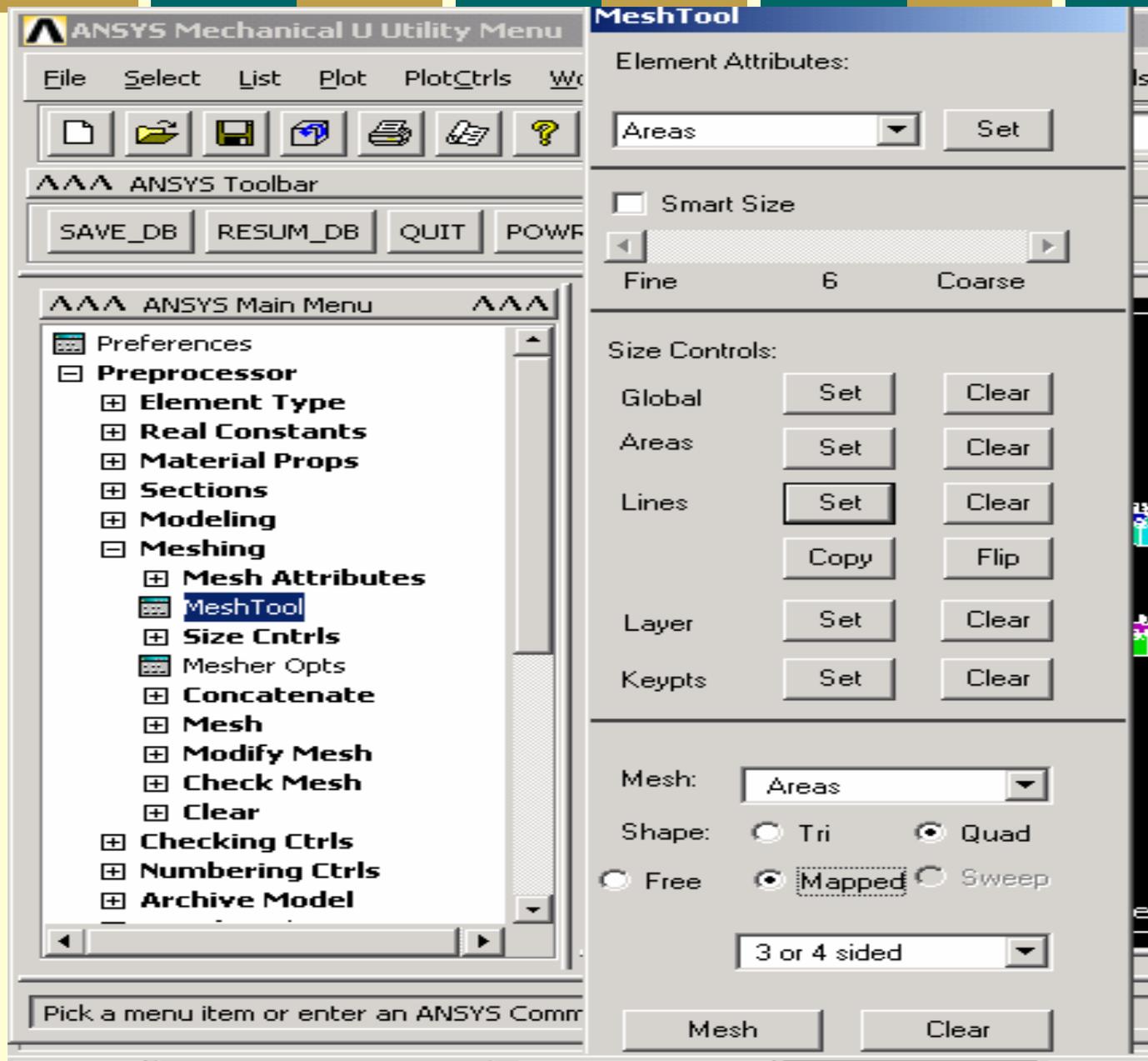
SPACE Spacing ratio

ANGSIZ Division arc (degrees)

(use ANGSIZ only if number of divisions (NDIV) and
element edge length (SIZE) are blank or zero)

Clear attached areas and volumes

 No



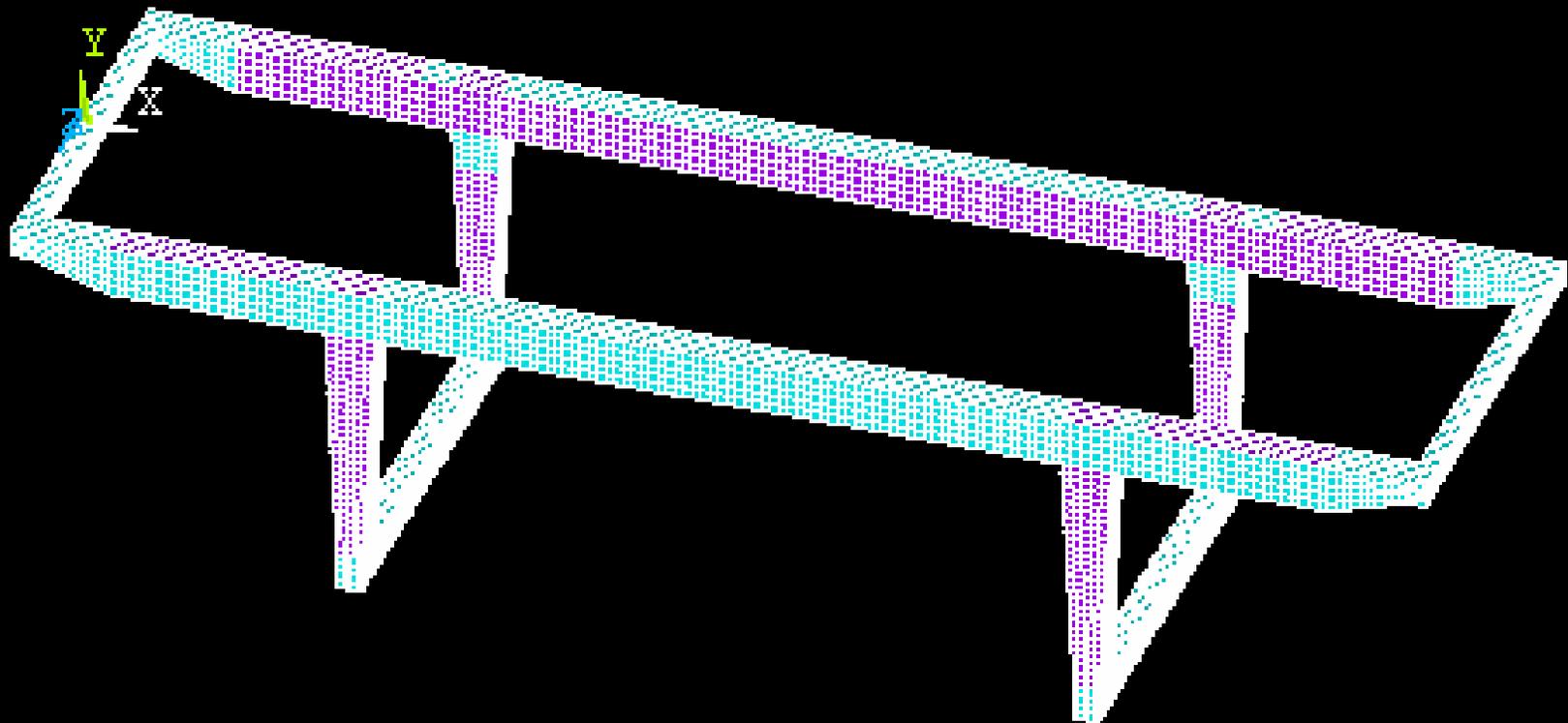
1

ELEMENTS

ANSYS

APR 18 2003

00:46:09



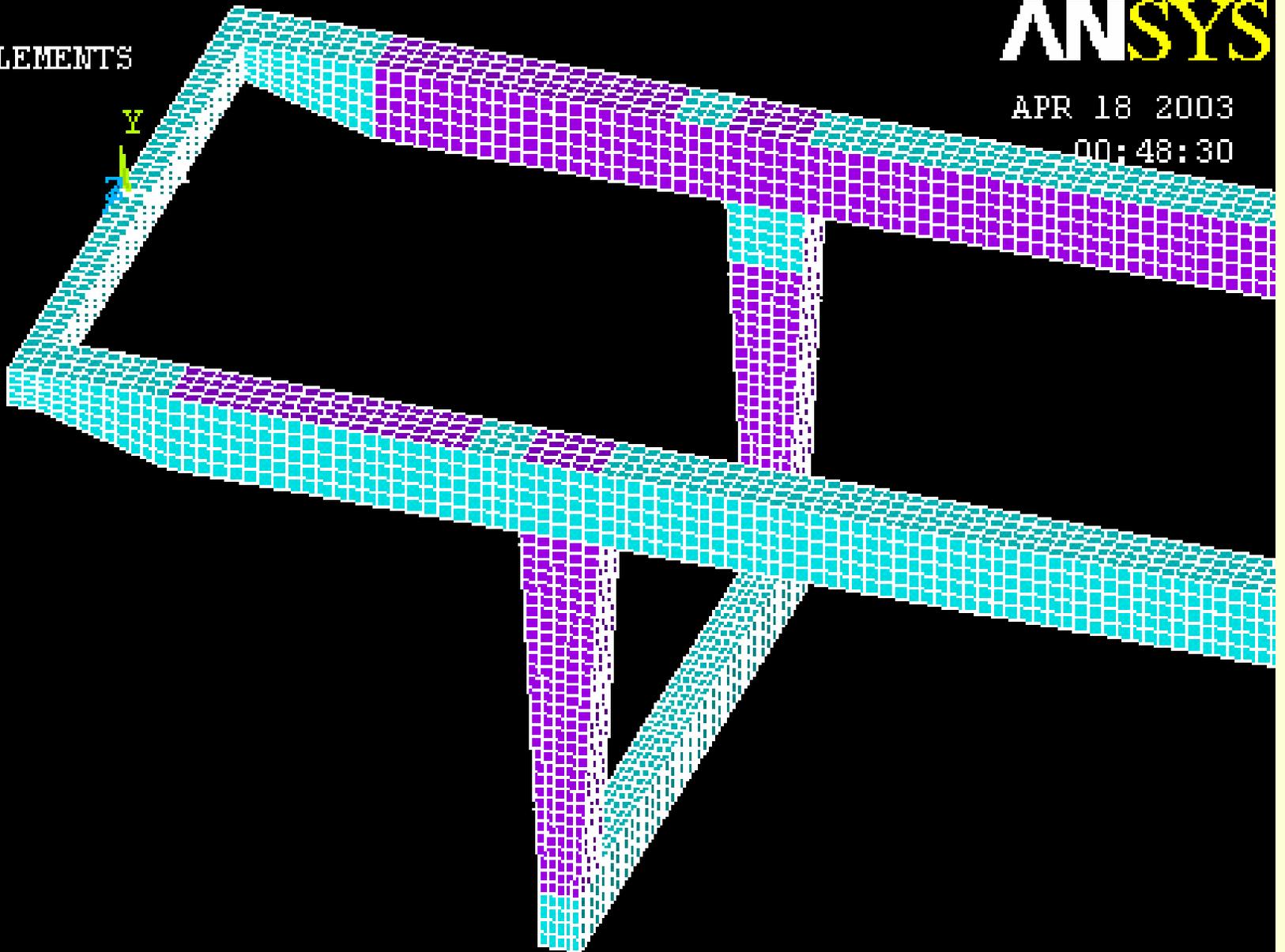
The Structure of Container Gantry Crane

1
ELEMENTS

ANSYS

APR 18 2003

00:48:30

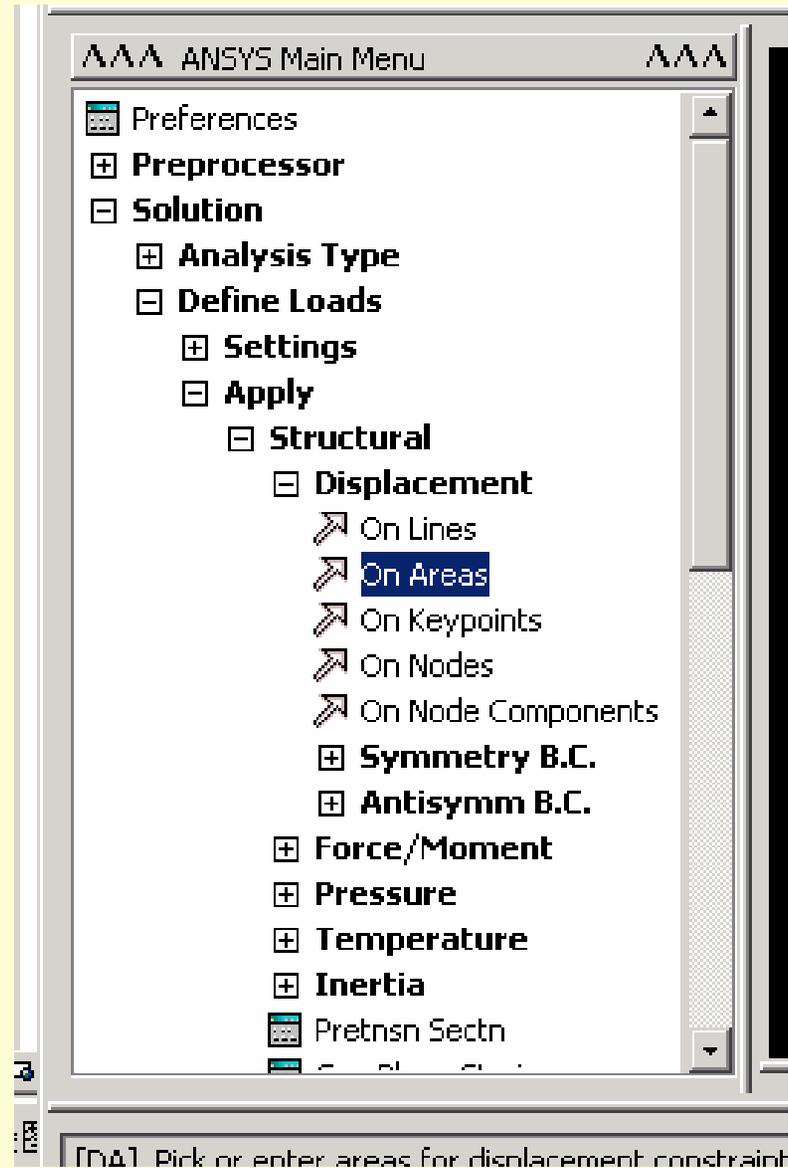


The Structure of Container Gantry Crane

ANSYS 7.0 Output Window

```
** Meshing of area 776 completed ** 15 elements.  
  
** Meshing of area 777 in progress **  
  ** AREA 777 MESHED WITH 12 QUADRILATERALS, 0 TRIAN  
** Meshing of area 777 completed ** 12 elements.  
  
** Meshing of area 778 in progress **  
  ** AREA 778 MESHED WITH 12 QUADRILATERALS, 0 TRIAN  
** Meshing of area 778 completed ** 12 elements.  
  
NUMBER OF AREAS MESHED = 778  
MAXIMUM NODE NUMBER = 12436  
MAXIMUM ELEMENT NUMBER = 13644  
  
PRODUCE ELEMENT PLOT IN DSYS = 0  
  
***** ROUTINE COMPLETED ***** CP = 178.246  
  
***** ANSYS SOLUTION ROUTINE *****
```

11. 施加位移边界条件



1

LINES

TYPE NUM

ANSYS

APR 17 2003

21:10:09

The Structure of Container Gantry Crane

Apply U,ROT on Areas

[DA] Apply Displacements (U,ROT) on Areas

Lab2 DOFs to be constrained

- All DOF
- UX**
- UY
- UZ
- ROTX
- ROTY
- ROTZ

UX

Constant value

Apply as

If Constant value then:

VALUE Displacement value

0

OK

Apply

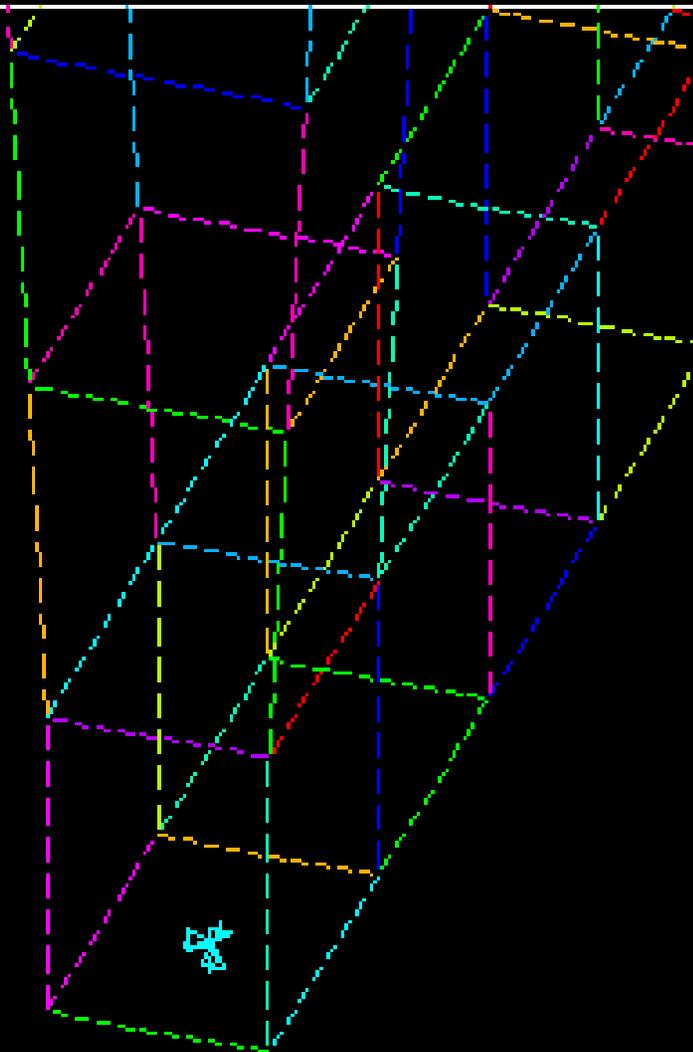
Cancel

Help

```
1
LINES
TYPE NUM
U
```

ANSYS

APR 17 2003
21:24:10



The Structure of Container Gantry Crane

1

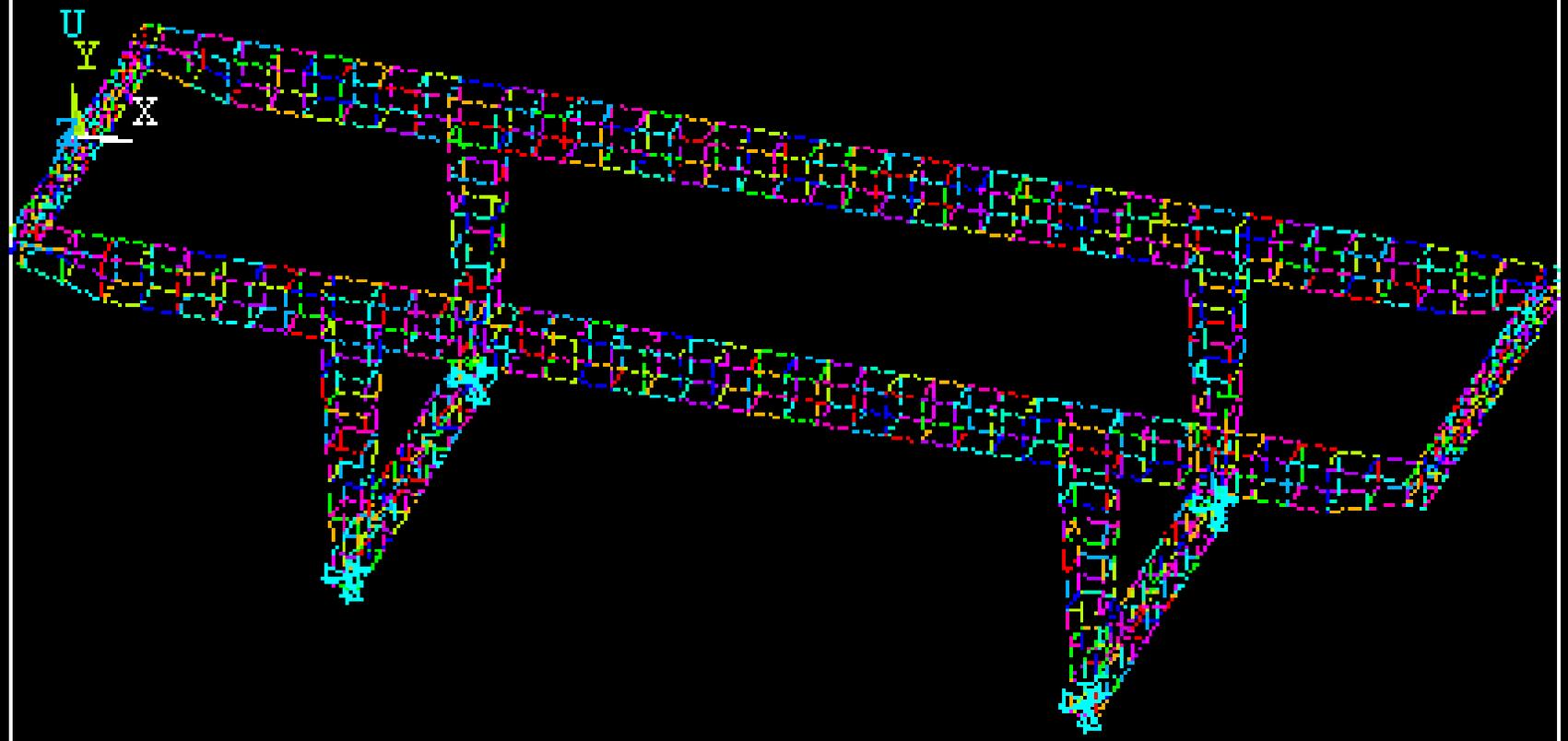
LINES

TYPE NUM

ANSYS

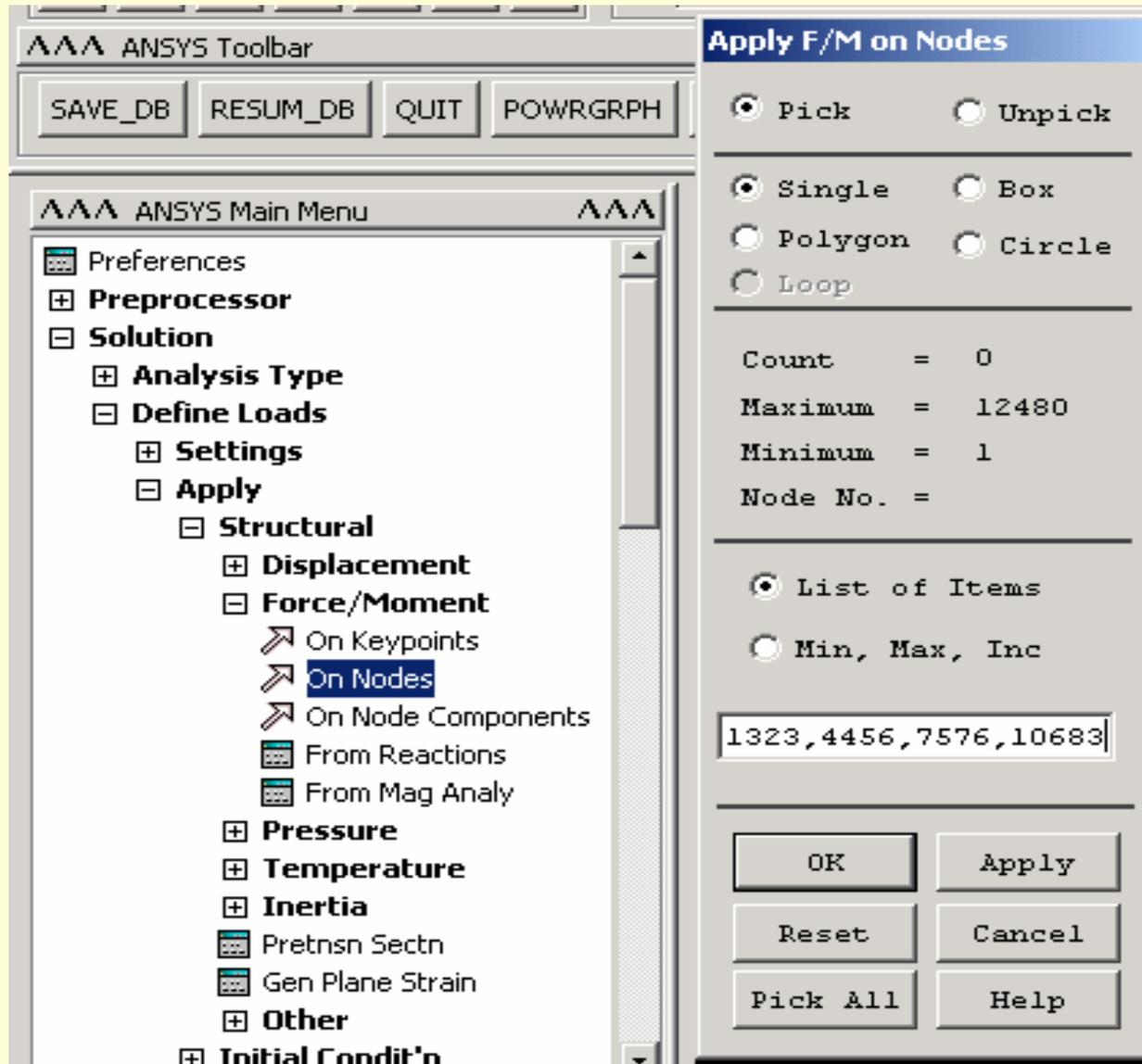
APR 17 2003

21:27:54



The Structure of Container Gantry Crane

12. 施加载荷



Apply F/M on Nodes

[F] Apply Force/Moment on Nodes

Lab Direction of force/mom

FY

Apply as

Constant value

If Constant value then:

VALUE Force/moment value

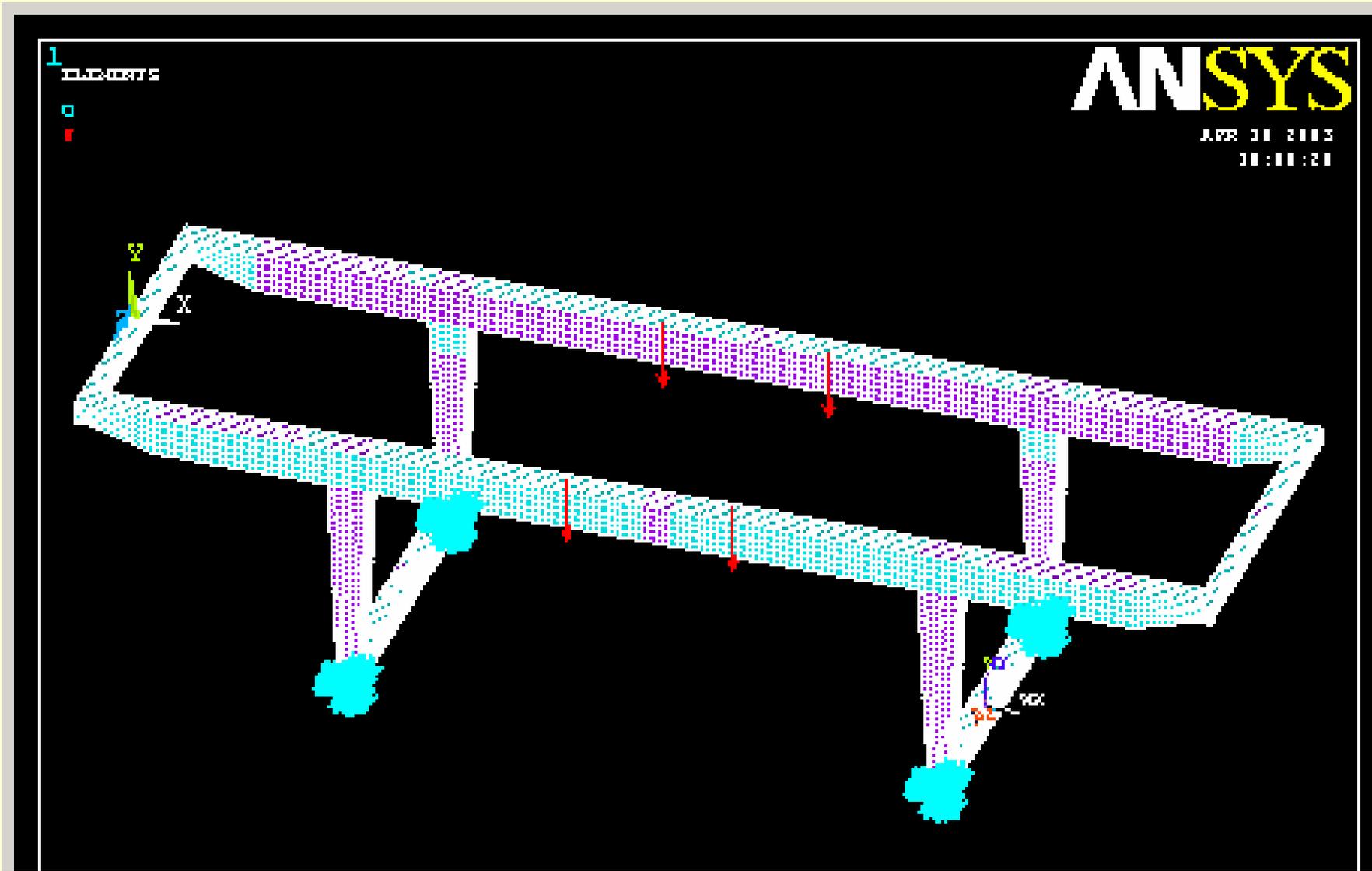
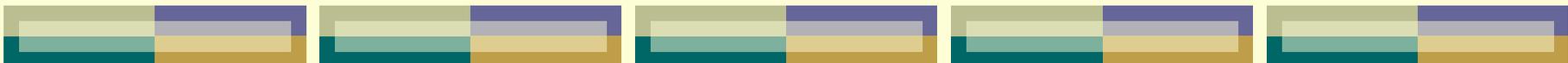
F1

OK

Apply

Cancel

Help



13. 求解

The image shows the ANSYS Main Menu on the left and the /STATUS Command dialog box on the right. The /STATUS Command dialog box displays the following text:

```
SOLUTION OPTIONS  
  
PROBLEM DIMENSIONALITY. . . . . 3-D  
DEGREES OF FREEDOM. . . . . UX  UY  UZ  ROTX ROTY ROTZ  
.....  
.STATIC (STEADY-STATE)  
  
I O N S  
  
.      1  
. 1.0000  
.      1  
.     NO  
  
PRINT OUTPUT CONTROLS . . . . . NO PRINTOUT  
DATABASE OUTPUT CONTROLS. . . . . ALL DATA WRITTEN  
FOR THE LAST SUBSTEP
```

The /STATUS Command dialog box also contains a sub-dialog box titled "Solve Current Load Step" with the following text:

```
[SOLVE] Begin Solution of Current Load Step  
  
Review the summary information in the lister window (entitled "/STATUS  
Command"), then press OK to start the solution.
```

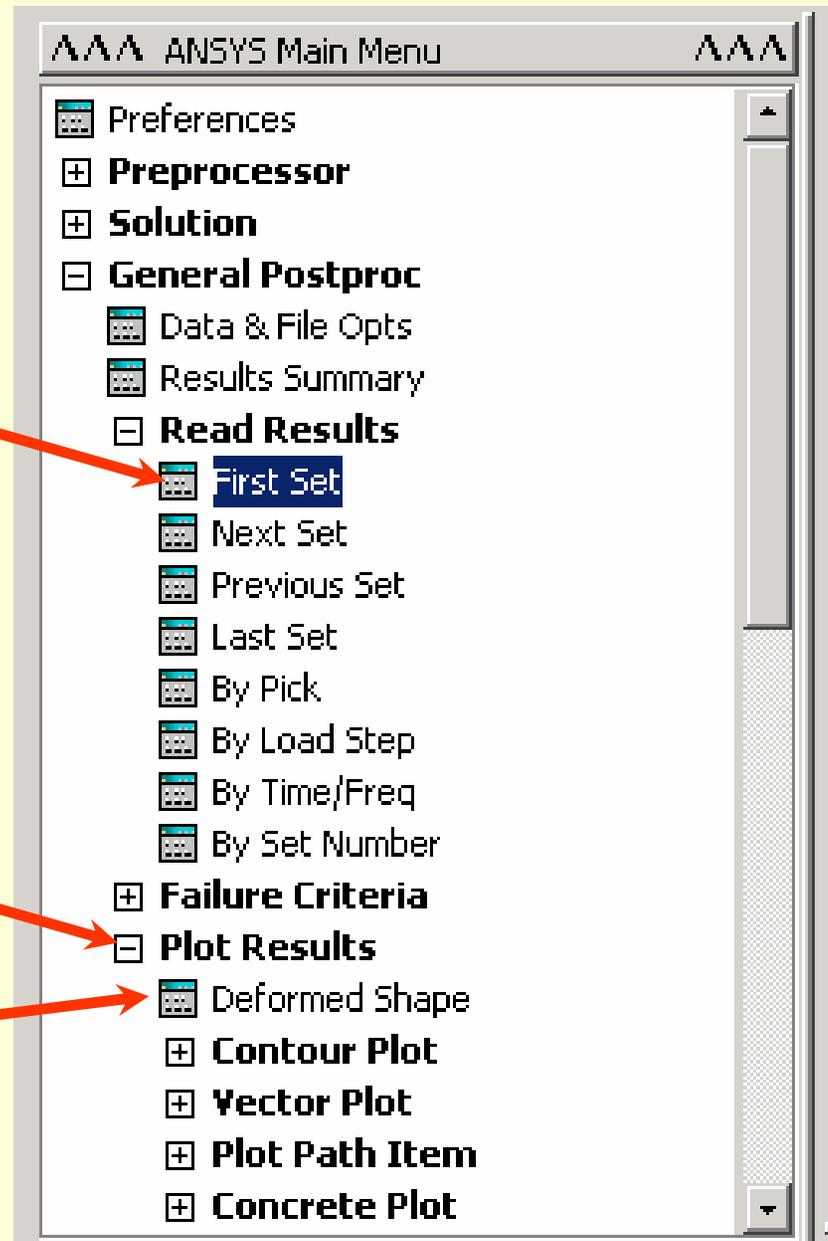
The sub-dialog box has three buttons: OK, Cancel, and Help.

14. 图形显示计算结果

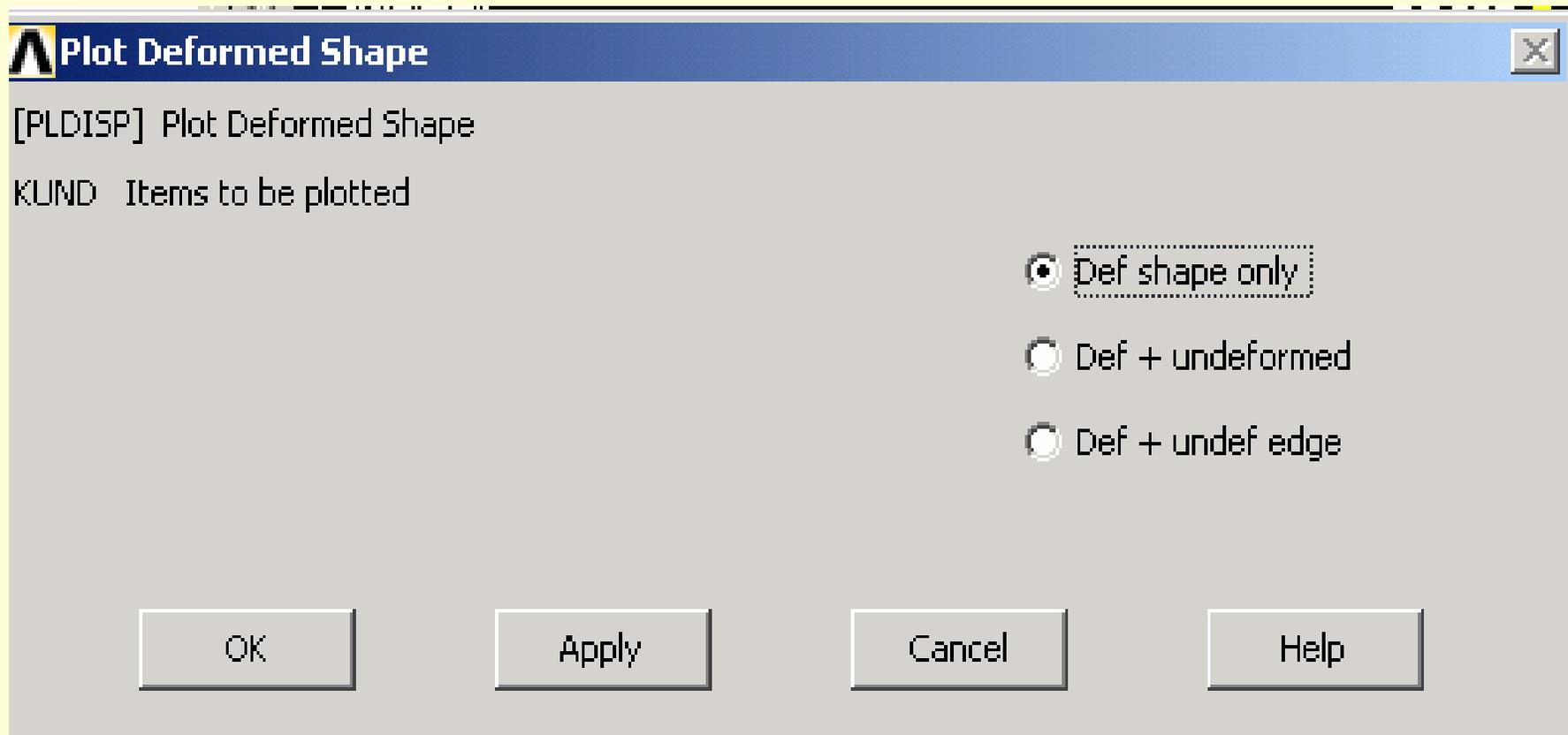
读入计算结果

图形显示计算结果

变形后的图形显示



(1) 变形后的图形显示



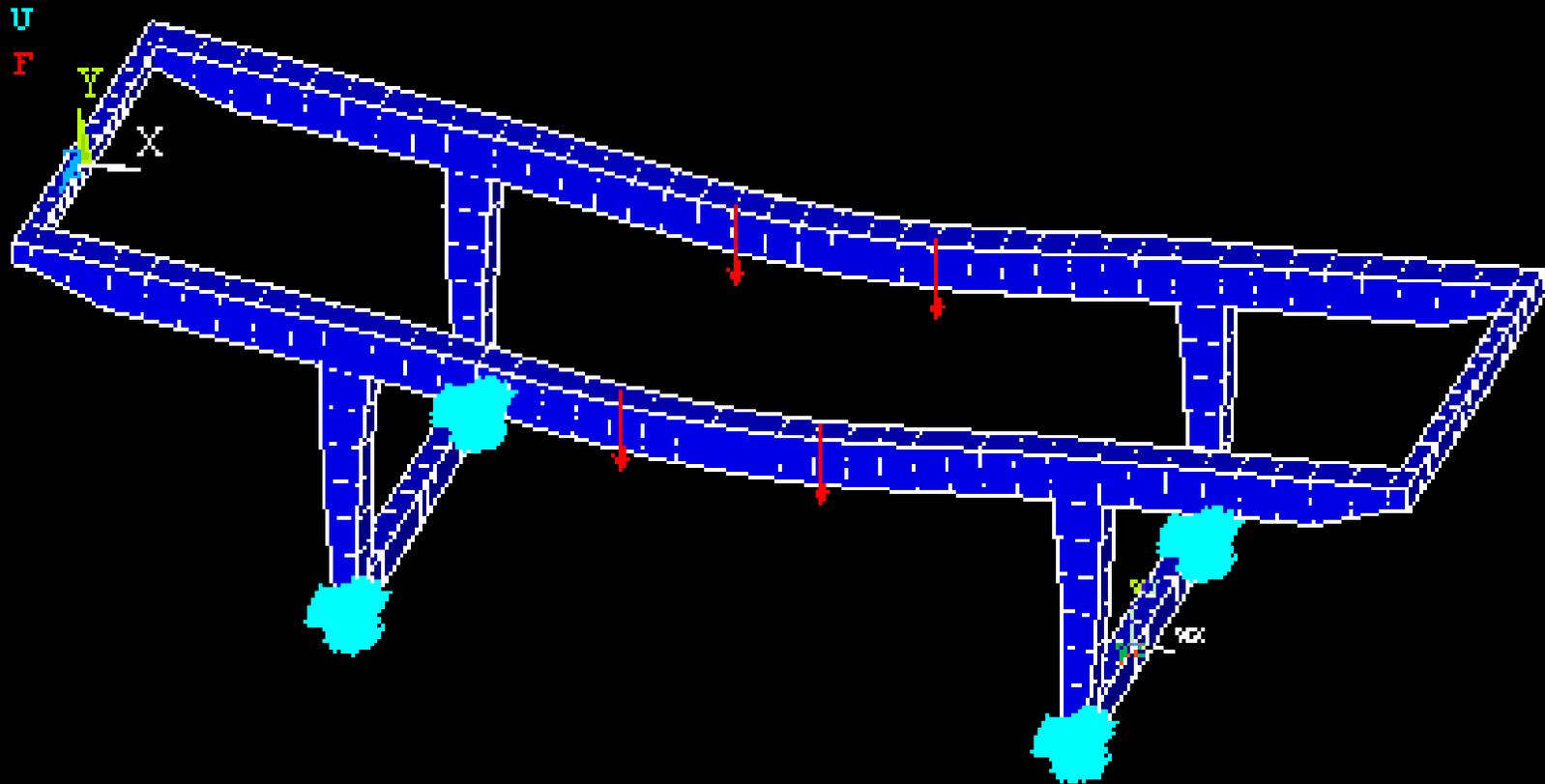
1
DISPLACEMENT

变形放大倍数: 100

ANSYS

STEP=1
SUB =1
TIME=1
DMX =16.205

APR 18 2003
10:48:40



The Structure of Container Gantry Crane

Plot Deformed Shape

[PLDISP] Plot Deformed Shape

KUND Items to be plotted

- Def shape only
- Def + undeformed
- Def + undef edge

OK

Apply

Cancel

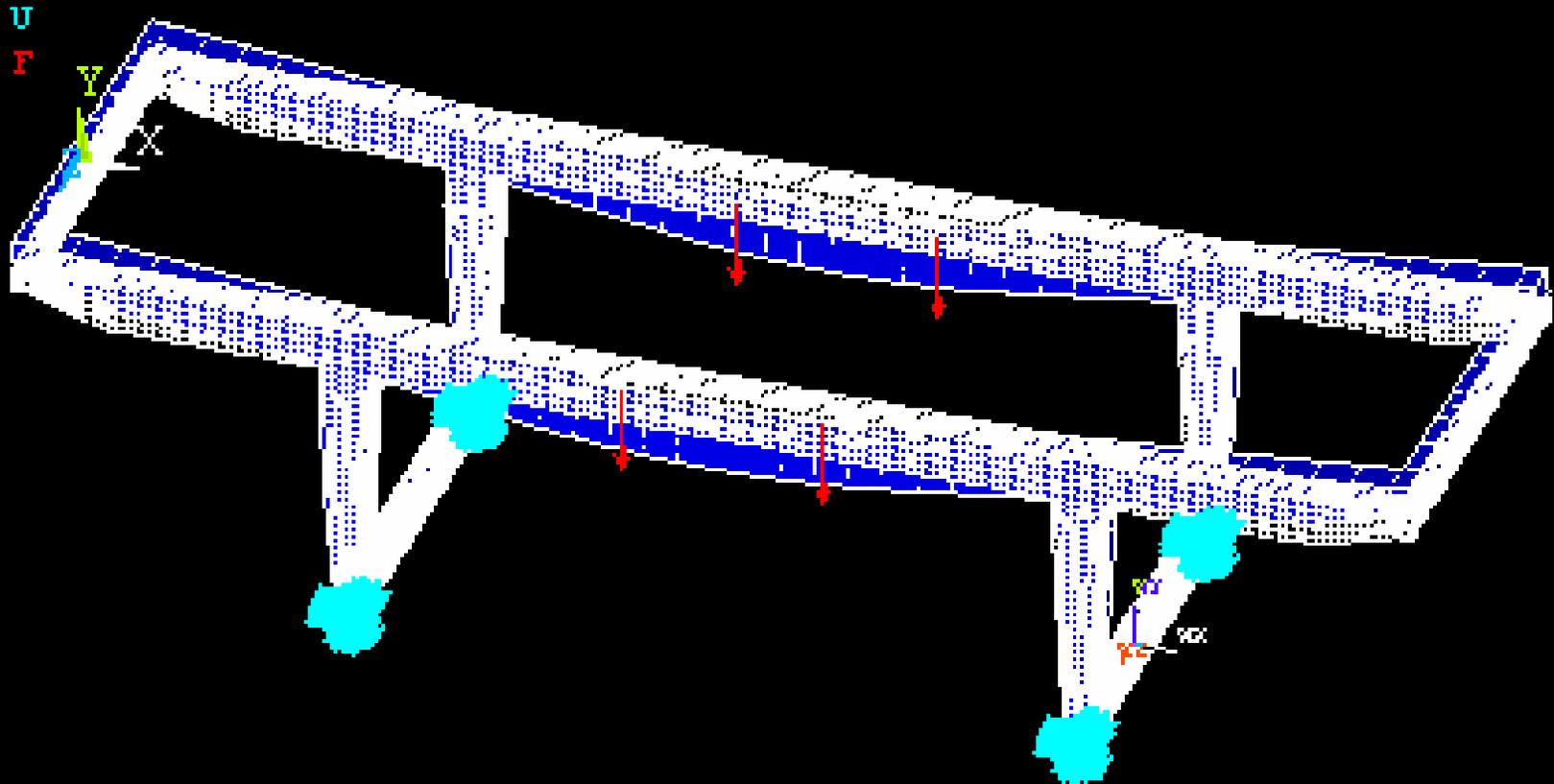
Help

1
DISPLACEMENT

STEP=1
SUB =1
TIME=1
DMX =16.205

ANSYS

APR 18 2003
11:01:53



The Structure of Container Gantry Crane



Plot Deformed Shape



[PLDISP] Plot Deformed Shape

KLUND Items to be plotted

- Def shape only
- Def + undeformed
- Def + undef edge

OK

Apply

Cancel

Help

1

DISPLACEMENT

STEP=1

SUB =1

TIME=1

DMX =16.205

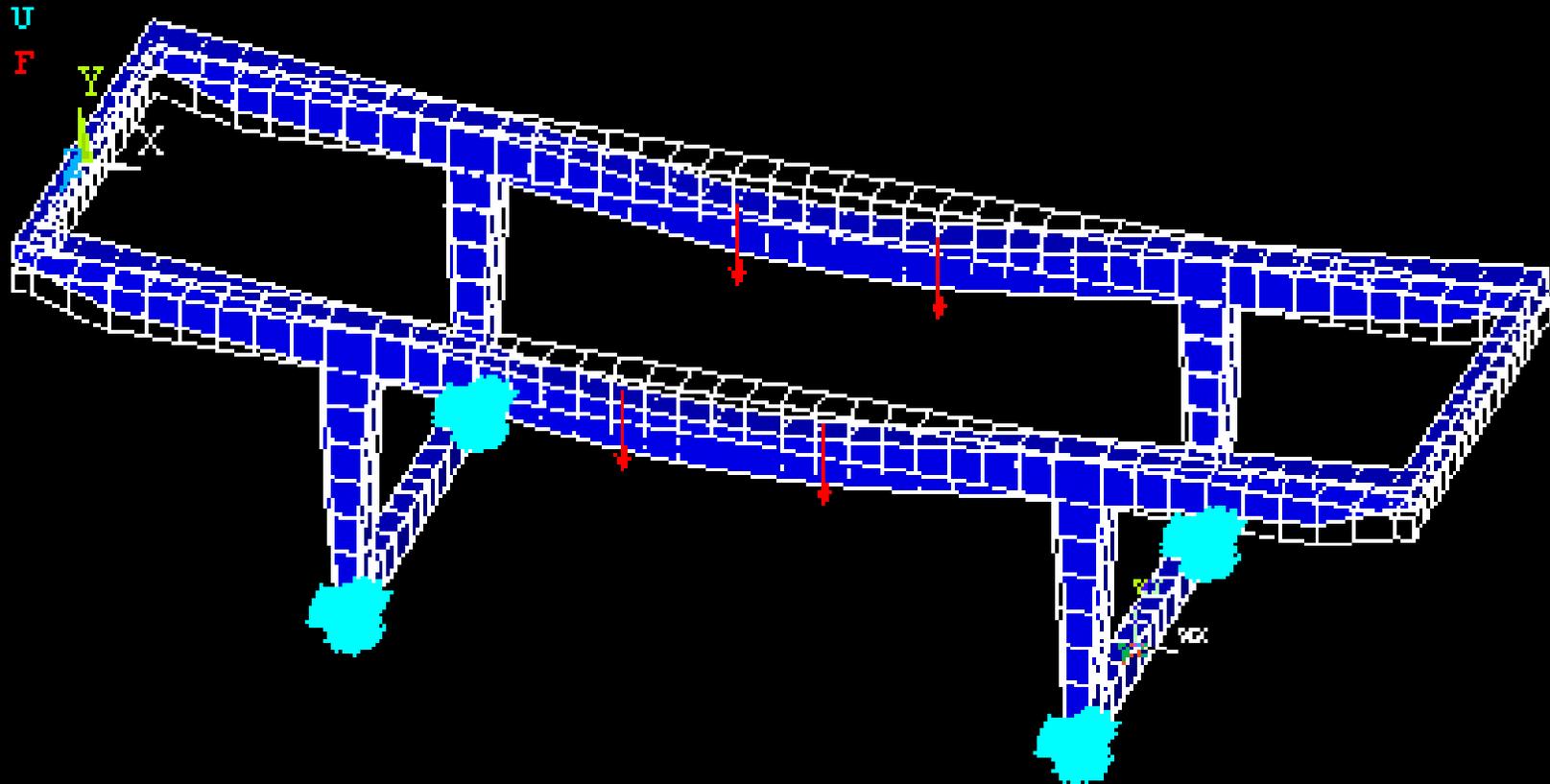
ANSYS

APR 18 2003

11:03:34

U

F



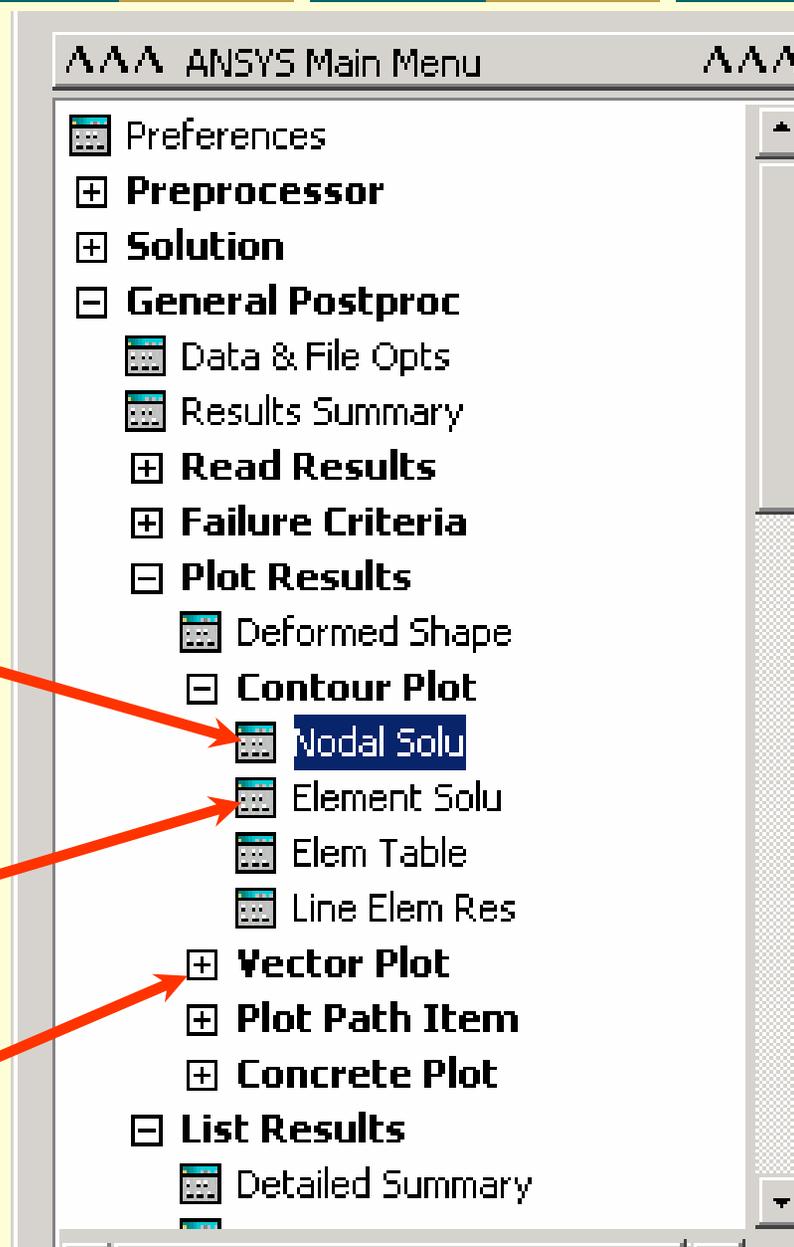
The Structure of Container Gantry Crane

(2) 等值线图显示结果

显示节点求解结果

显示单元求解结果

矢量图显示计算结果



Contour Nodal Solution Data

[PLNSOL] Contour Nodal Solution Data

Item,Comp Item to be contoured

DOF solution

Stress

Strain-total

Energy

Strain ener dens

Strain-elastic

Strain-thermal

Strain-plastic

Translation UX

UY

UZ

USUM

Rotation ROTX

ROTY

UY

KUND Items to be plotted

Def shape only

Def + undeformed

Def + undef edge

Fact Optional scale factor

1

[/EFACET] Interpolation Nodes

Corner only

Corner + midside

All applicable

[AVPRIN] Eff NU for EQV strain

0.3

OK

Apply

Cancel

Help

1
NODAL SOLUTION

节点求解结果：Y方向位移



APR 18 2003

20:13:35

SUB =1

TIME=1

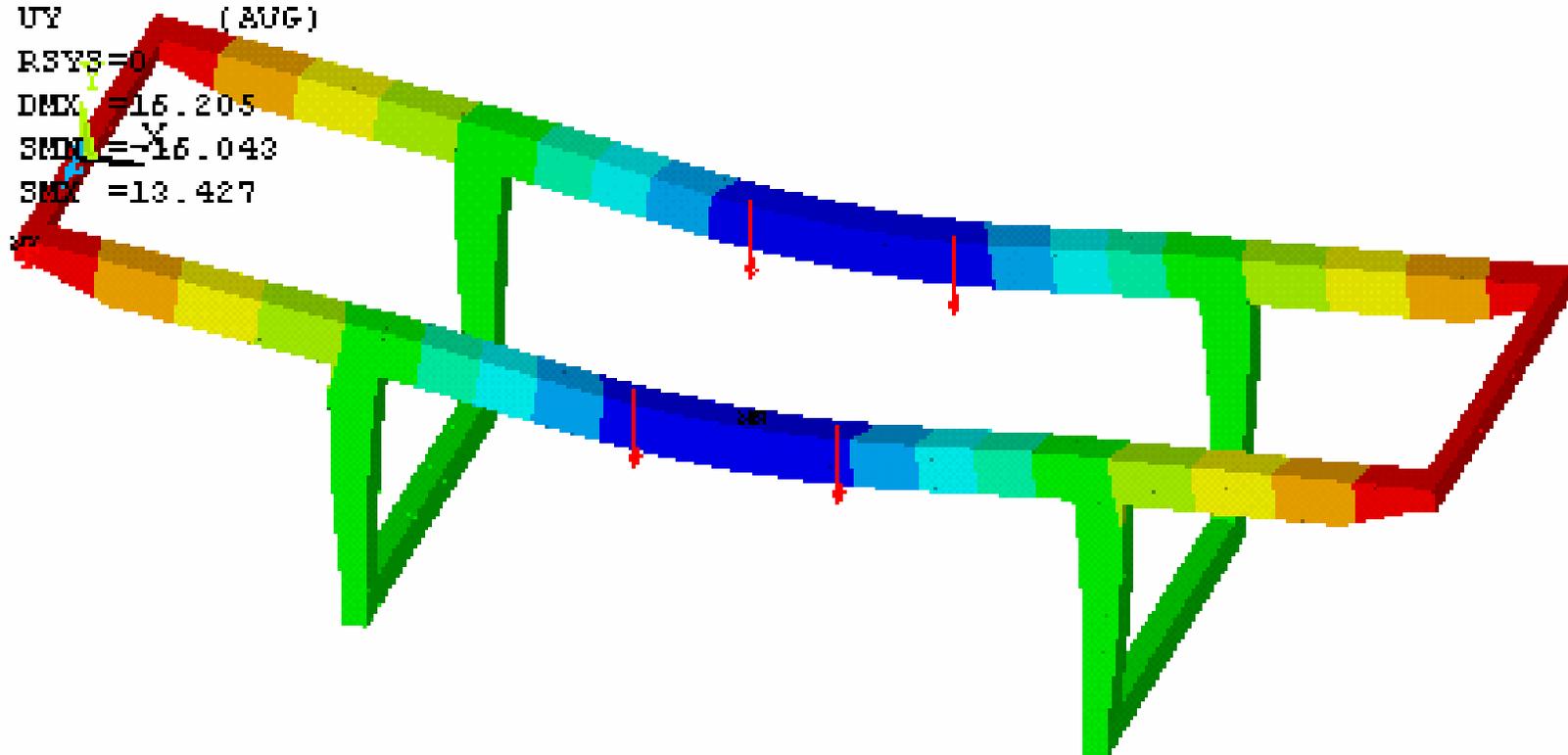
UY (AUG)

RSYS=0

DMX =15.205

SMDL =-15.043

SMD =13.427



The Structure of Container Gantry Crane

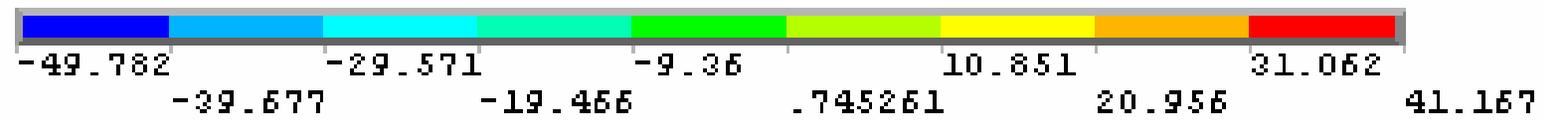
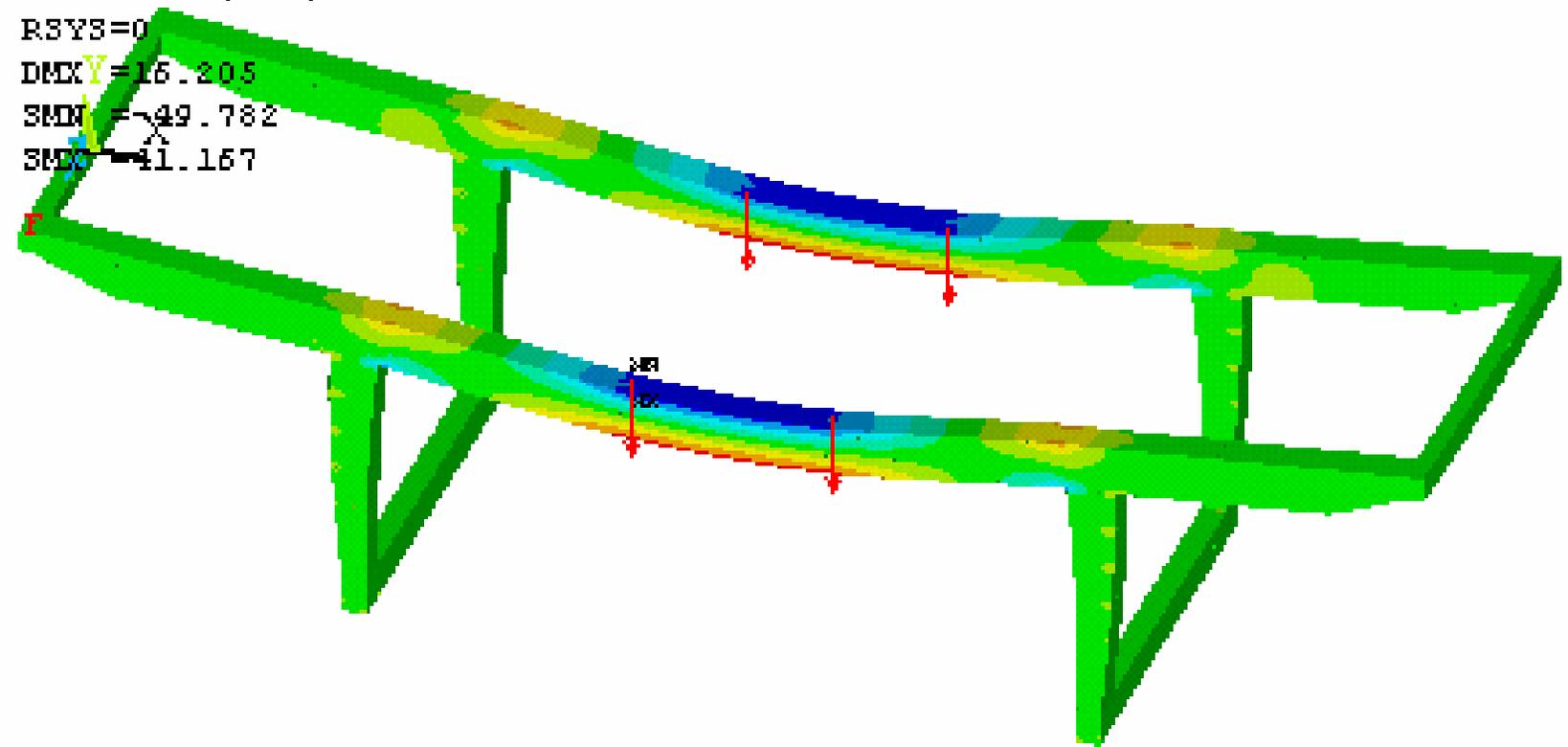
1

NODAL SOLUTION
SUB =1
TIME=1
SX (AVG)
RSYS=0
DMX=16.205
SMN=-49.782
SMX=41.167

节点求解结果：沿X轴的应力



APR 18 2003
20:21:46



The Structure of Container Gantry Crane

1

NODAL SOLUTION

节点求解结果：沿Y轴的应力

ANSYS

SUB =1

APR 18 2003

TIME=1

20:23:52

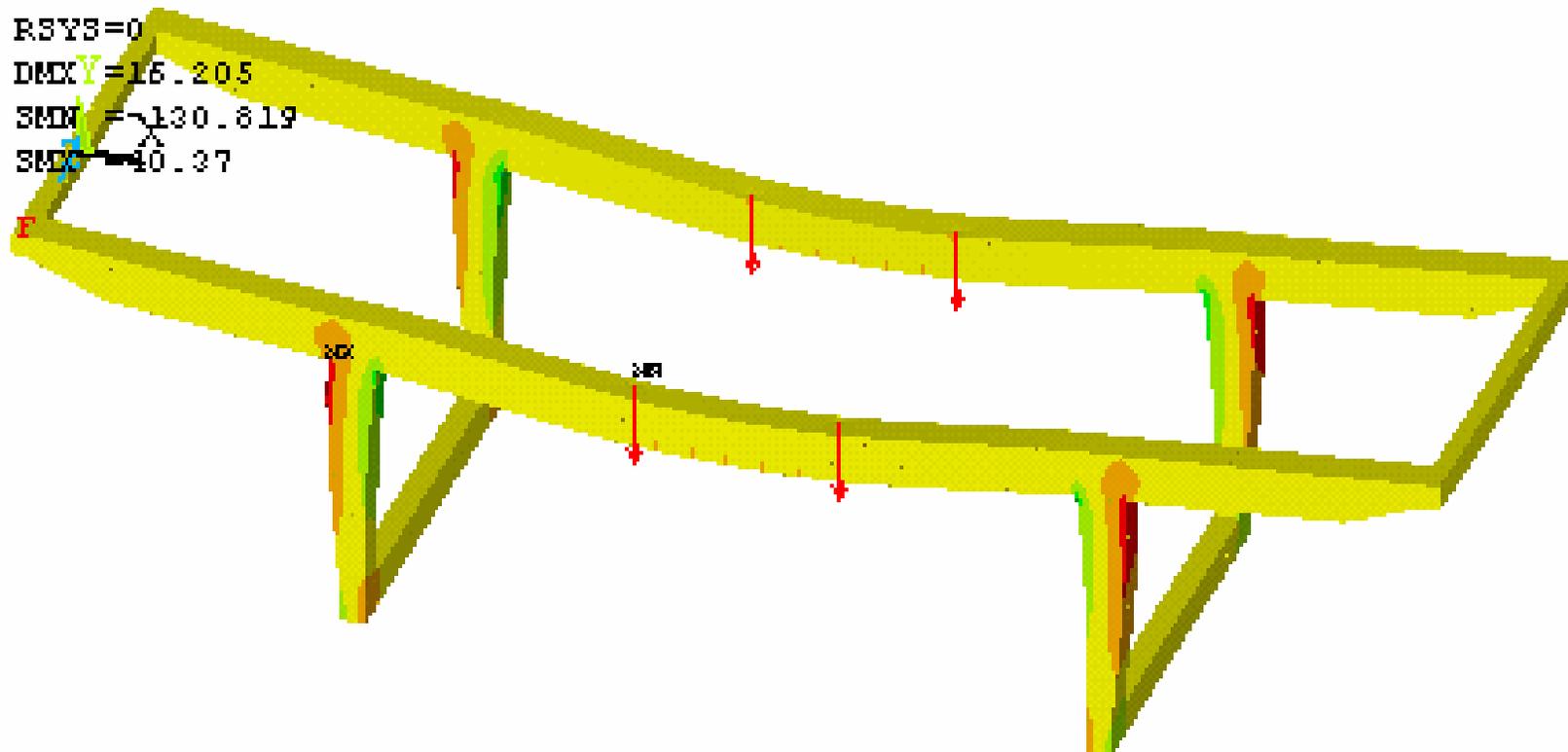
SY (AVG)

RSYS=0

DMX=16.205

SMD=130.819

SMS=40.37



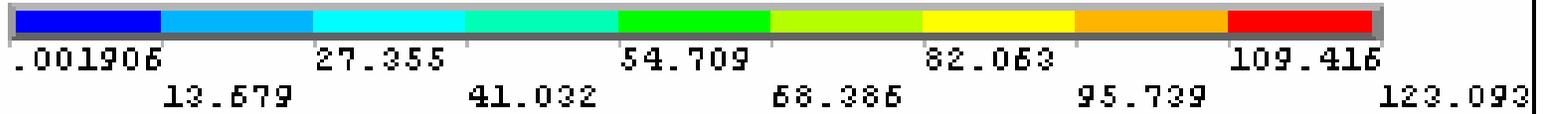
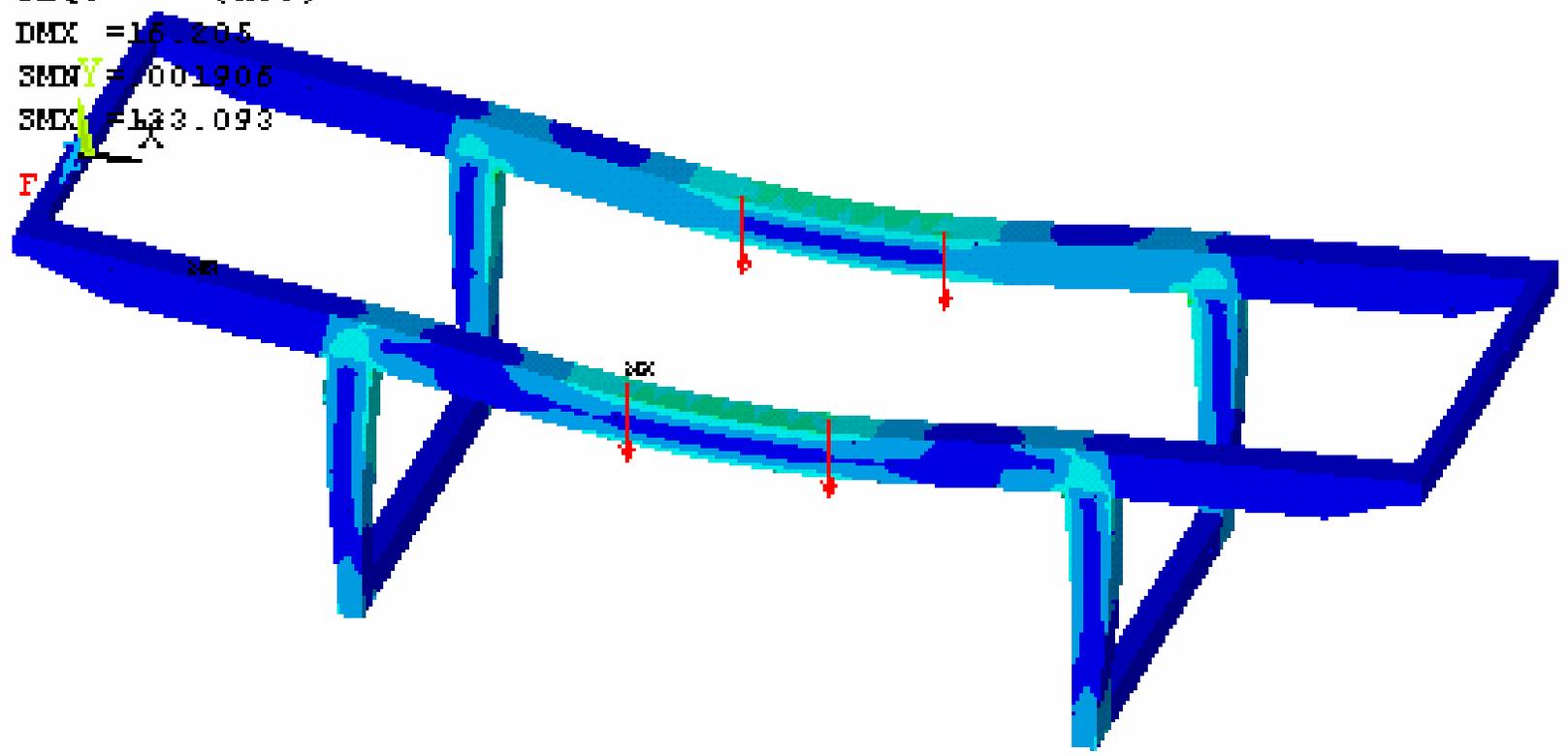
The Structure of Container Gantry Crane

1
NODAL SOLUTION
SUB =1
TIME=1
SEQV (AVG)
DMX =15.205
SMRY =001906
SMC1 =123.093

节点求解结果：第四强度理论应力



APR 18 2003
20:25:57



The Structure of Container Gantry Crane

1

ELEMENT SOLUTION

SUB =1

TIME=1

3X (NOMIG)

RSYS=0

DMX=16.205

SMM=-50.541

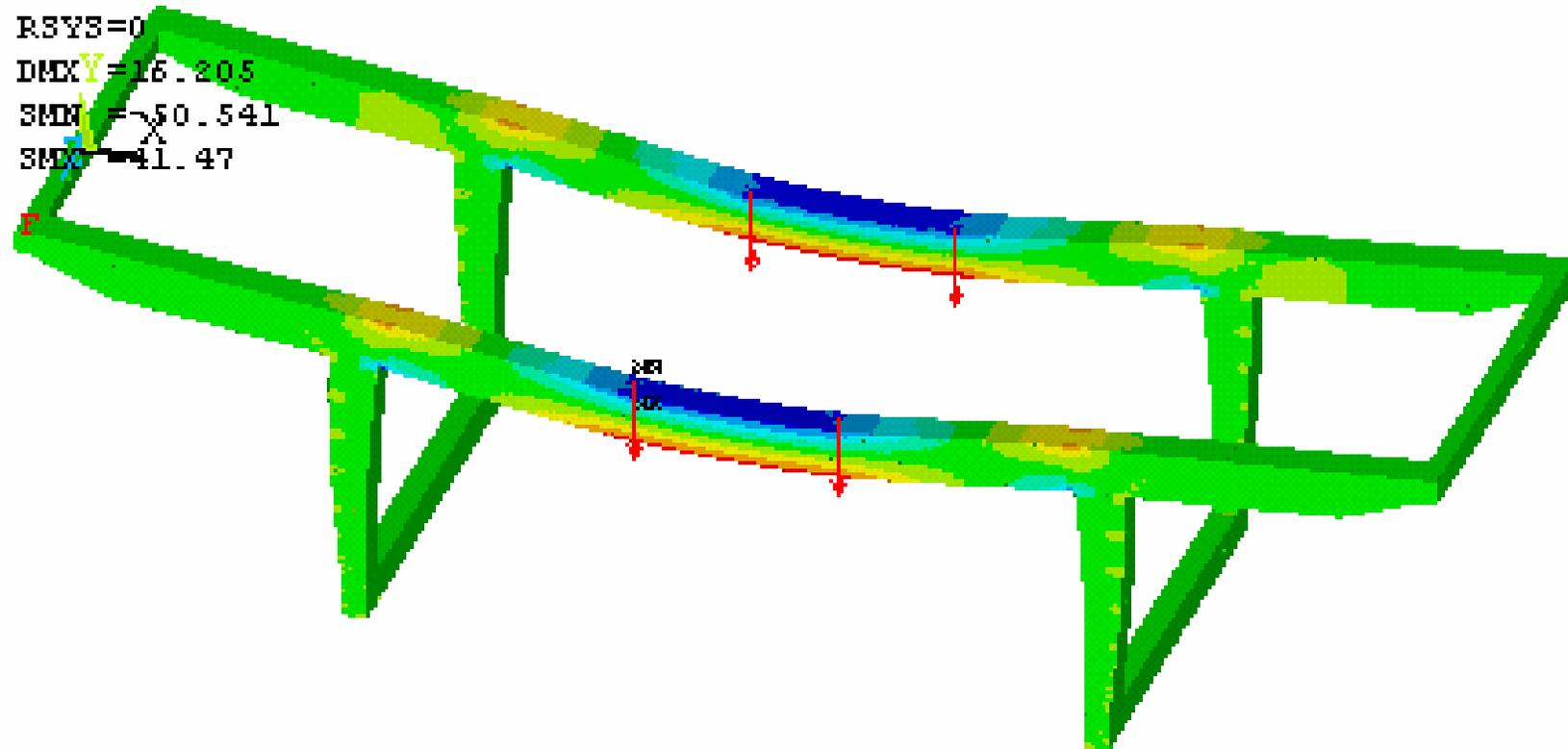
SME=41.47

单元求解结果：沿X轴的应力

ANSYS

APR 18 2003

20:27:46



The Structure of Container Gantry Crane

1

ELEMENT SOLUTION

单元求解结果：第四强度理论应力



SUB =1

APR 18 2003

TIME=1

20:29:41

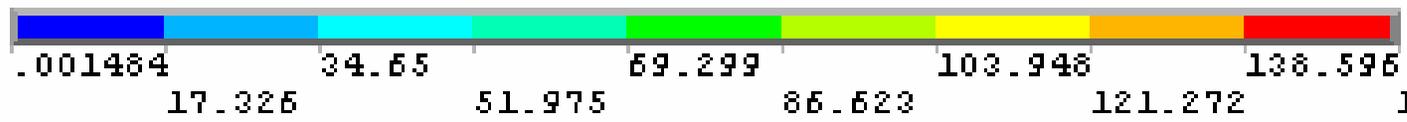
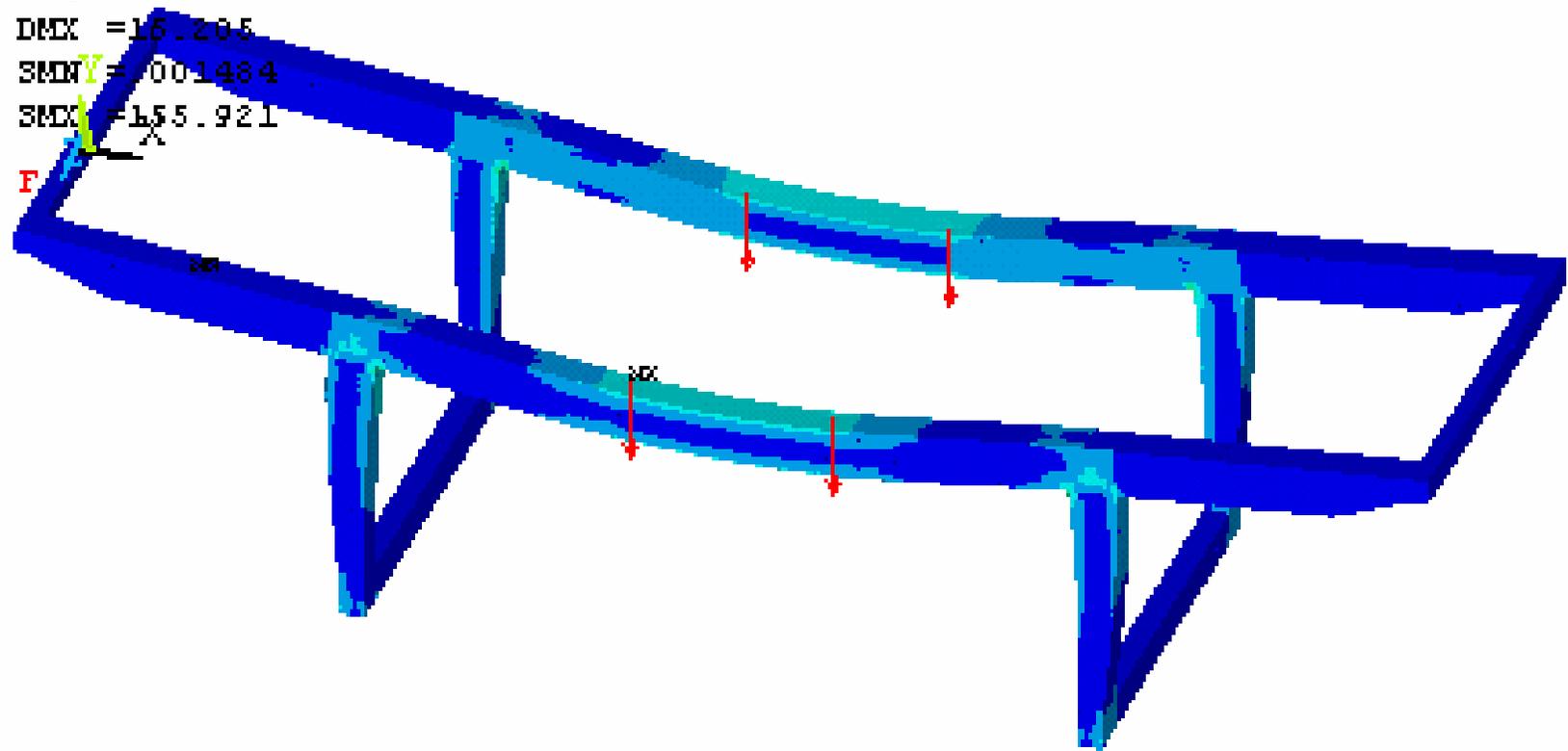
SEQV (NO AVG)

DMX =15.295

SMMY =.001484

SMDX =155.921

F



The Structure of Container Gantry Crane

1

ELEMENT SOLUTION

单元求解结果：沿Y轴的节点力

ANSYS

STEP=1

APR 18 2003

SUB =1

13:45:14

TIME=1

FYY

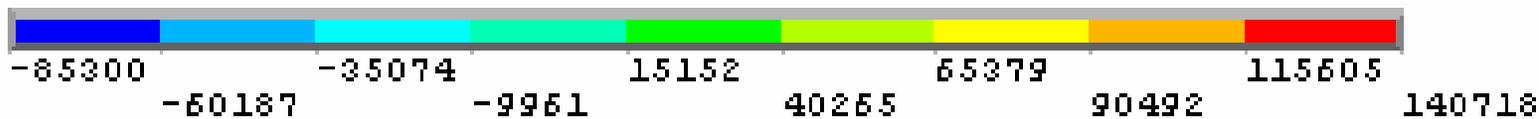
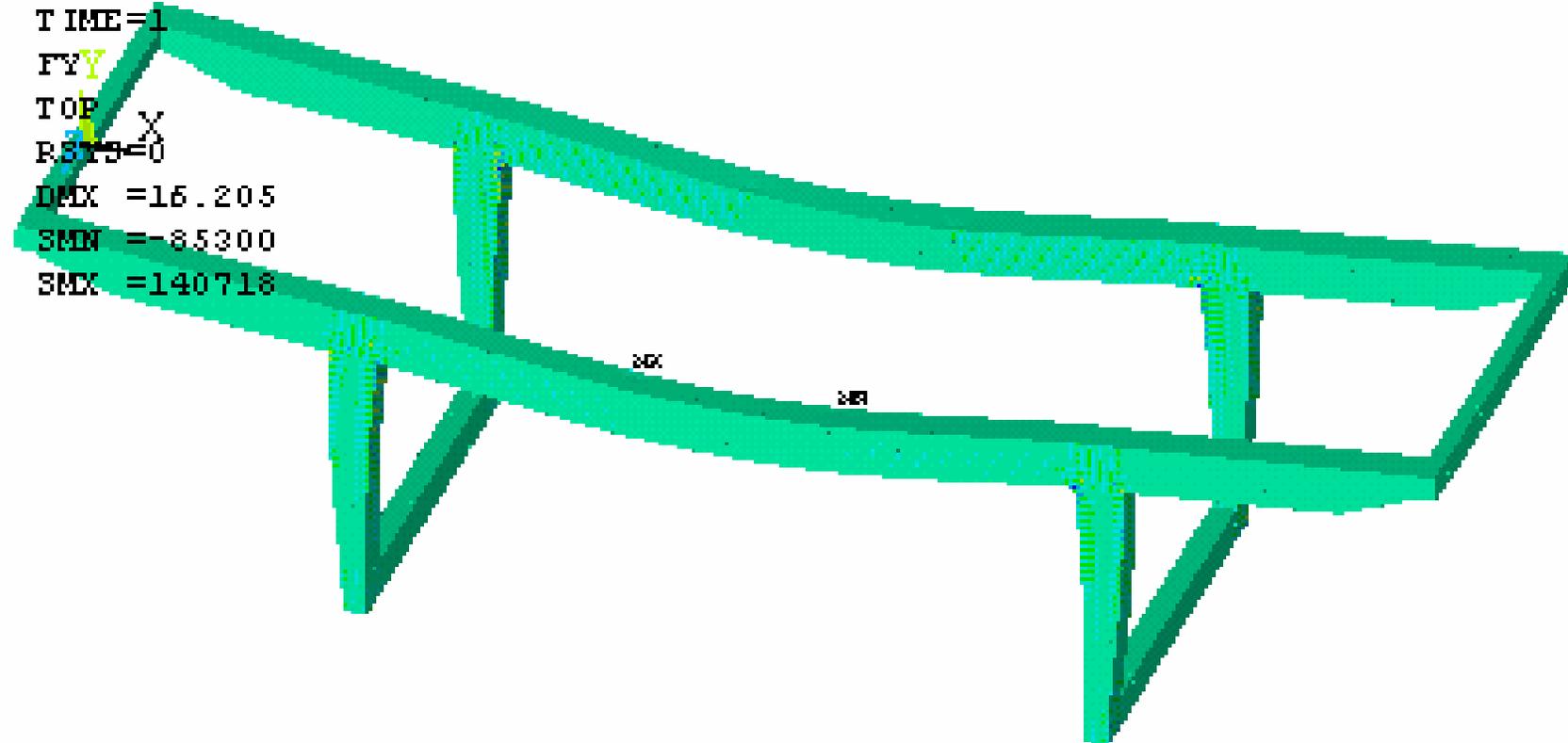
TOP

RSTN=0

DMX =16.205

SMN =-85300

SMX =140718



The Structure of Container Gantry Crane

1

ELEMENT SOLUTION

单元求解结果：沿X轴的节点力矩

APR 18 2003

13:53:12

STEP=1

SUB =1

TIME=1

MX Y

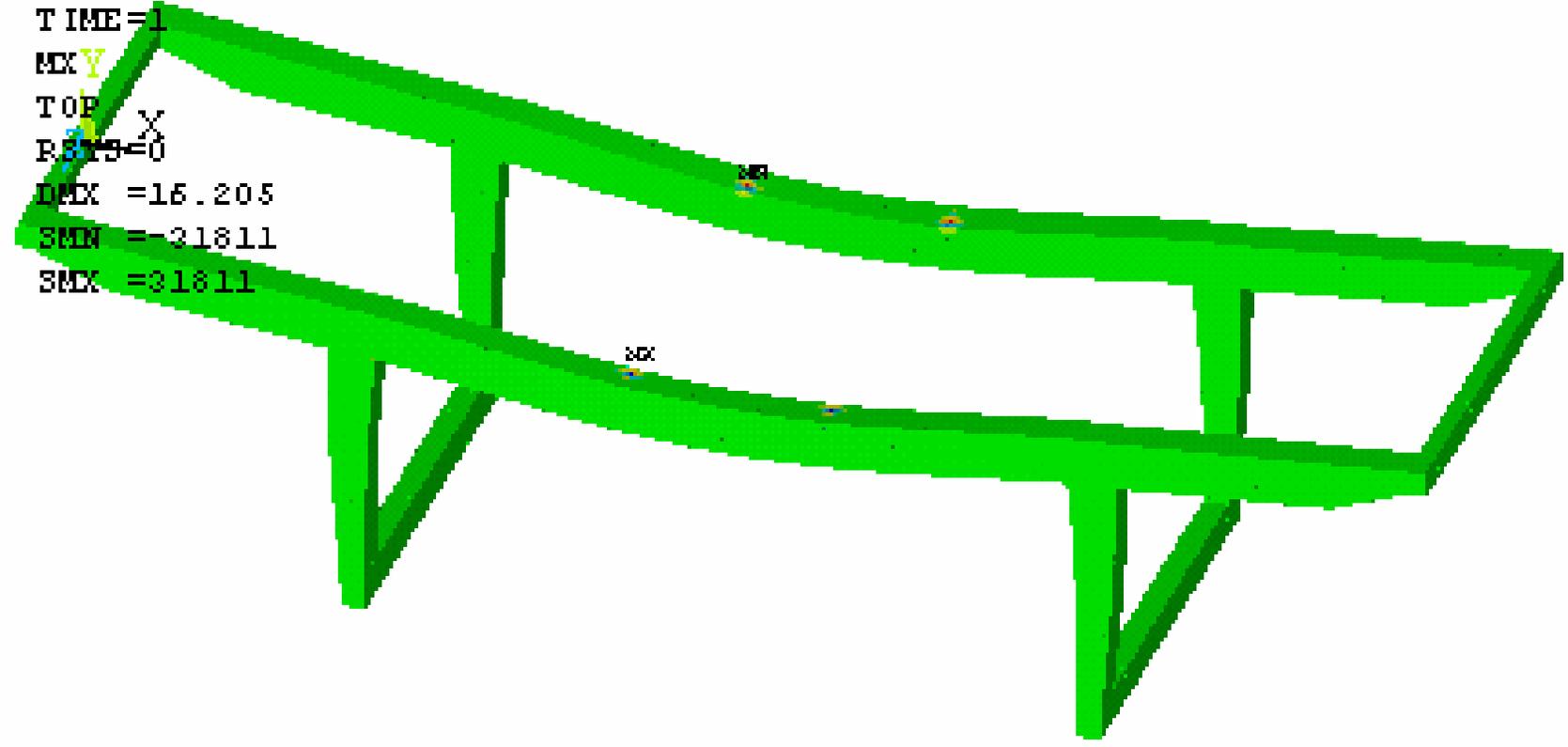
TOP X

RSYS=0

DMX =16.205

SMM =-31811

SMX =31811



The Structure of Container Gantry Crane

1

ELEMENT SOLUTION

单元求解结果：应变能

ANSYS

SUB =1

APR 18 2003

TIME=1

20:31:50

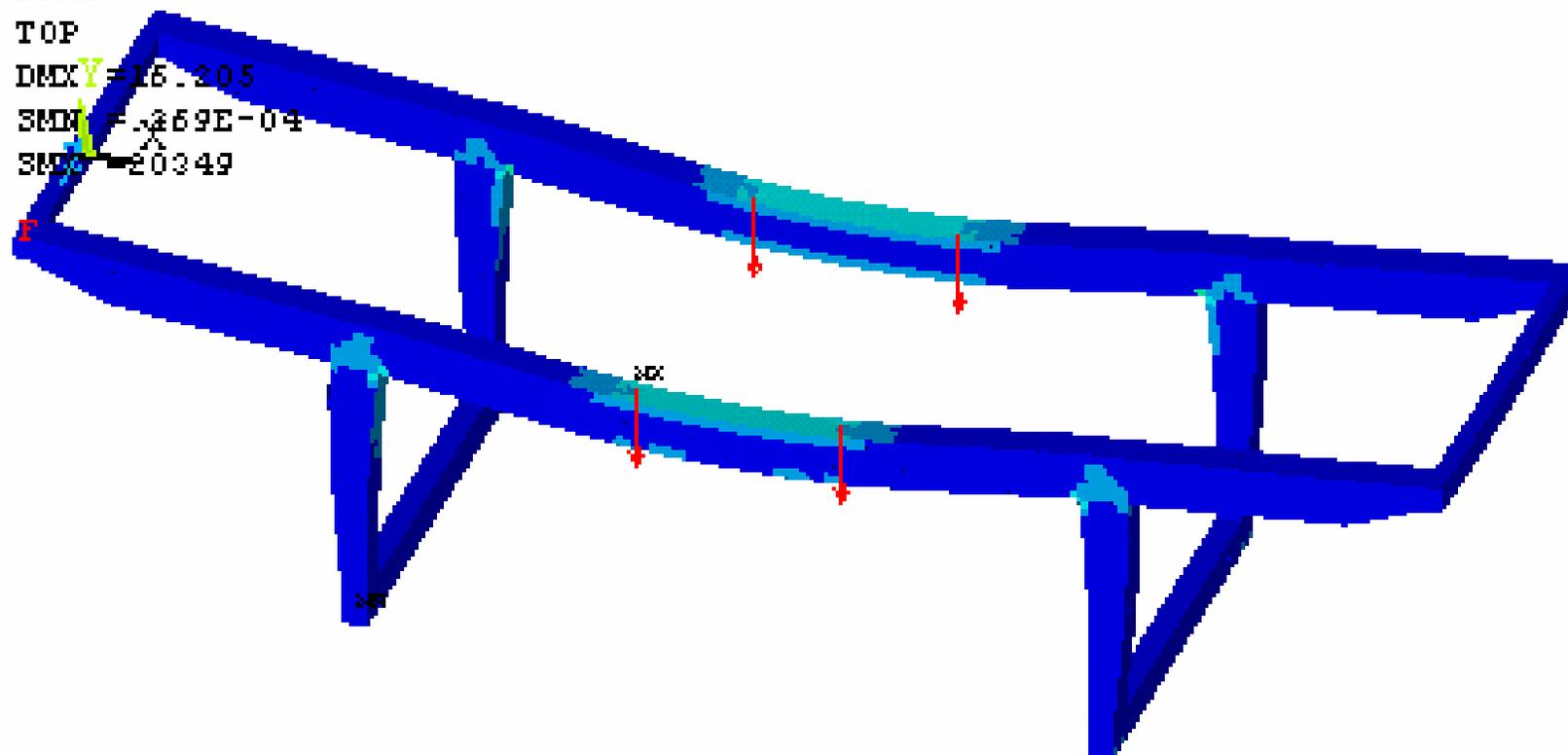
SENE

TOP

DMX(Y)=16.205

SMDV =.269E-04

SMDZ =20349



The Structure of Container Gantry Crane

1

ELEMENT SOLUTION

单元求解结果：应力偏差

ANSYS

SUB =1

APR 18 2003

TIME=1

20:33:59

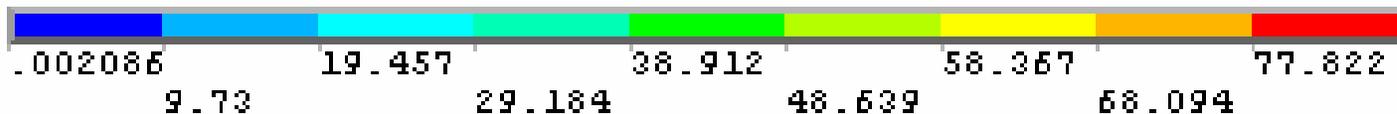
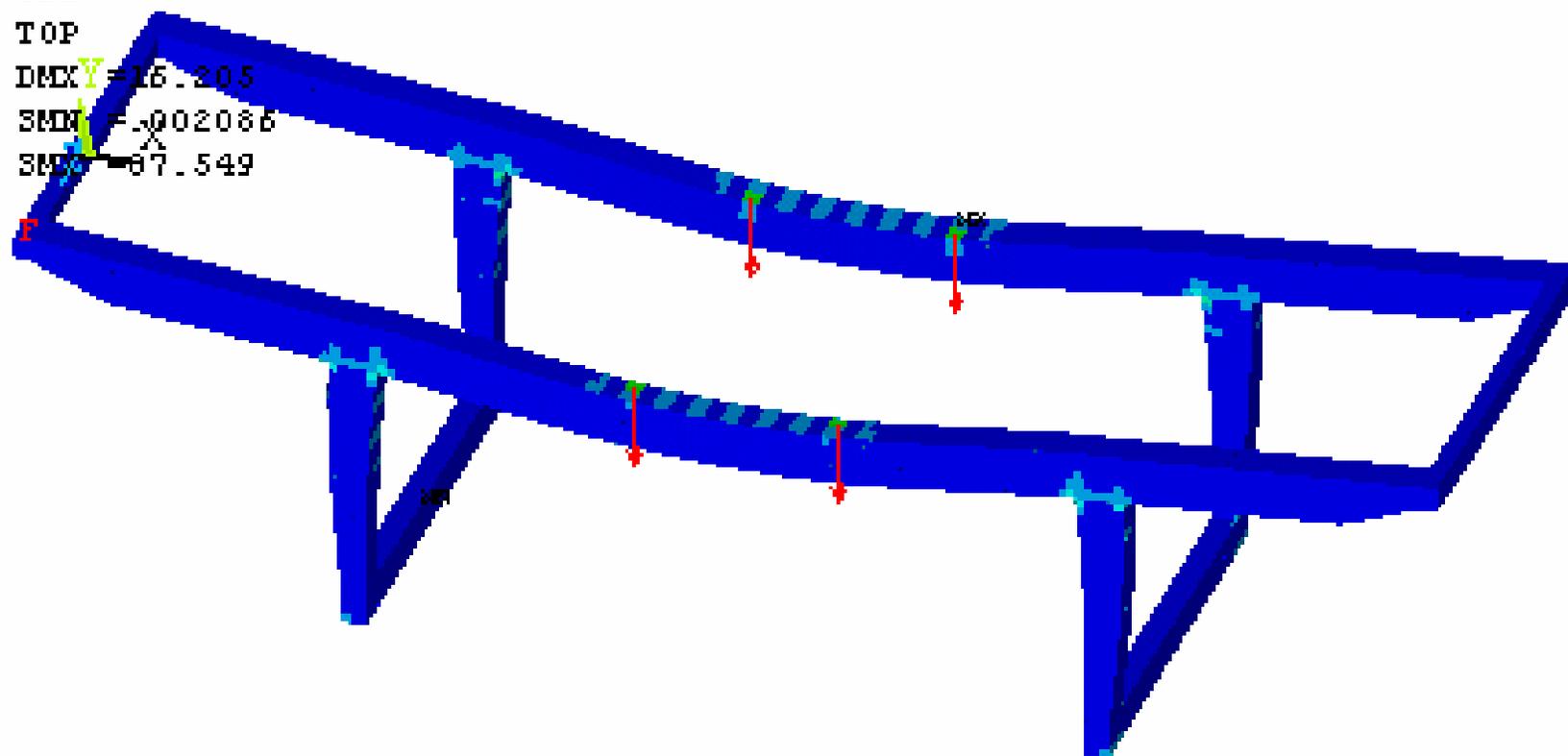
SDSG

TOP

DMX \bar{Y} = 16.205

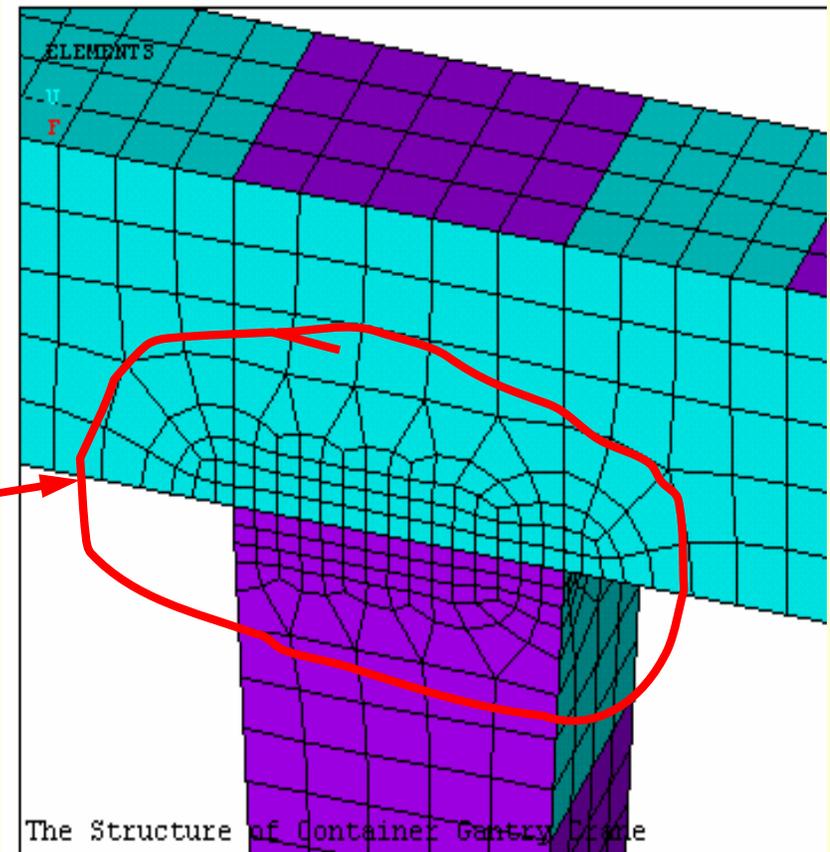
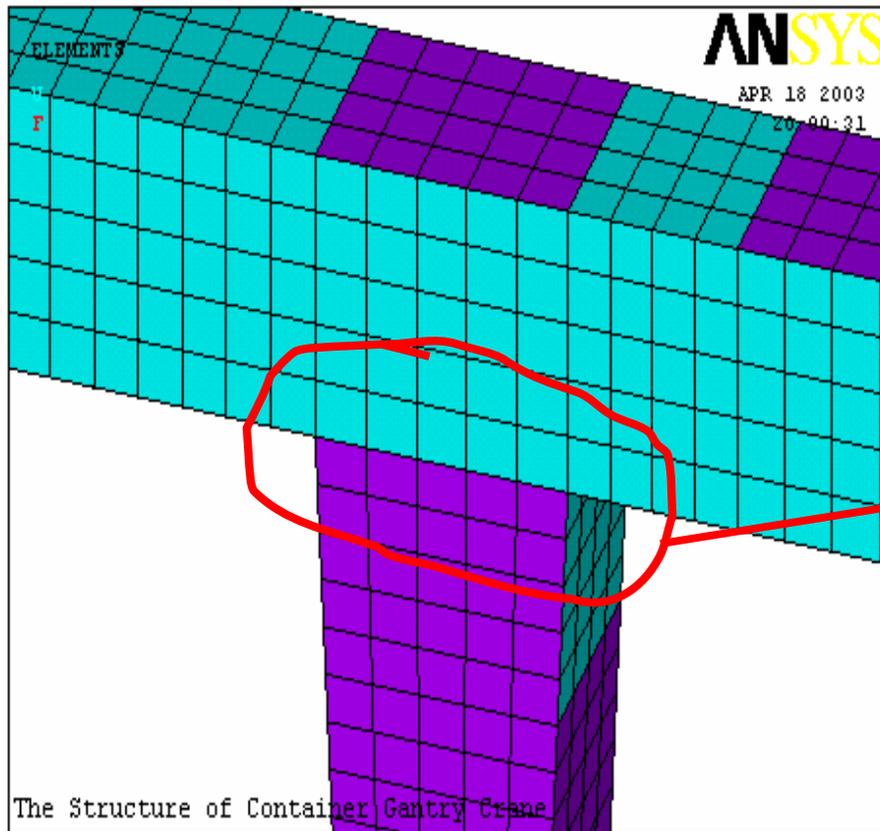
SMD \bar{X} = .002086

SME \bar{Z} = 87.549

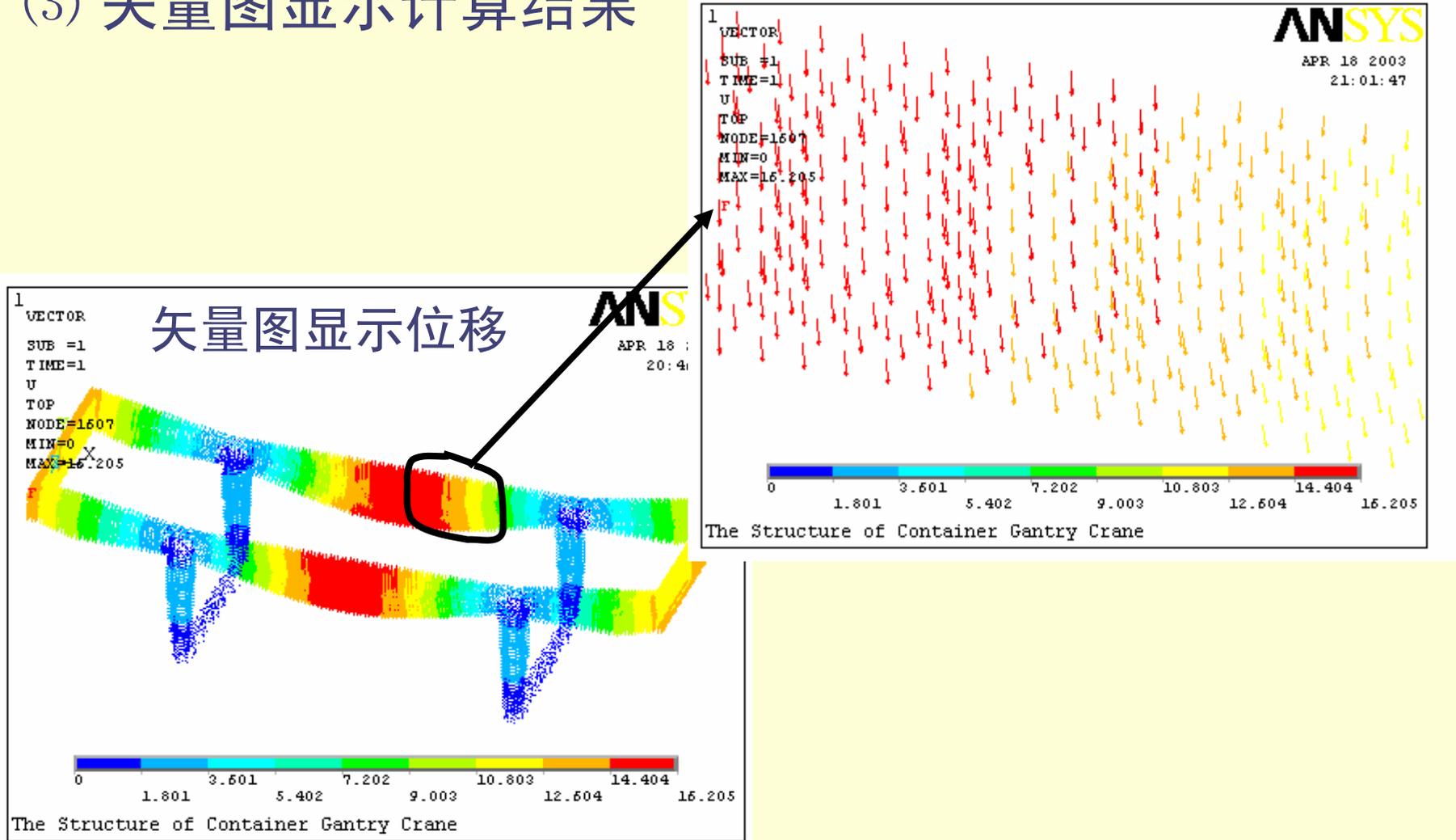


The Structure of Container Gantry Crane

对应力梯度较大区域进行网格细化



(3) 矢量图显示计算结果



(4) 动画显示计算结果

15. 列表显示结果数据

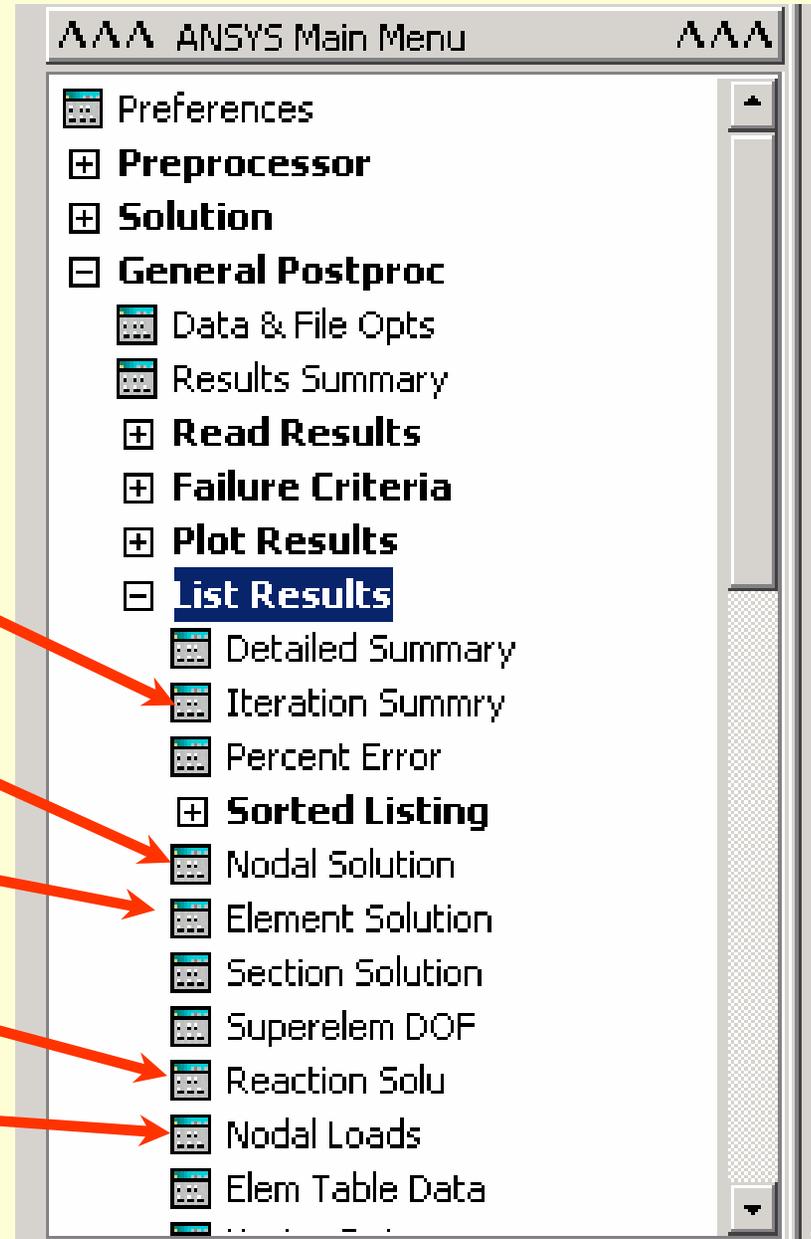
列表显示摘要信息

列表显示节点求解结果

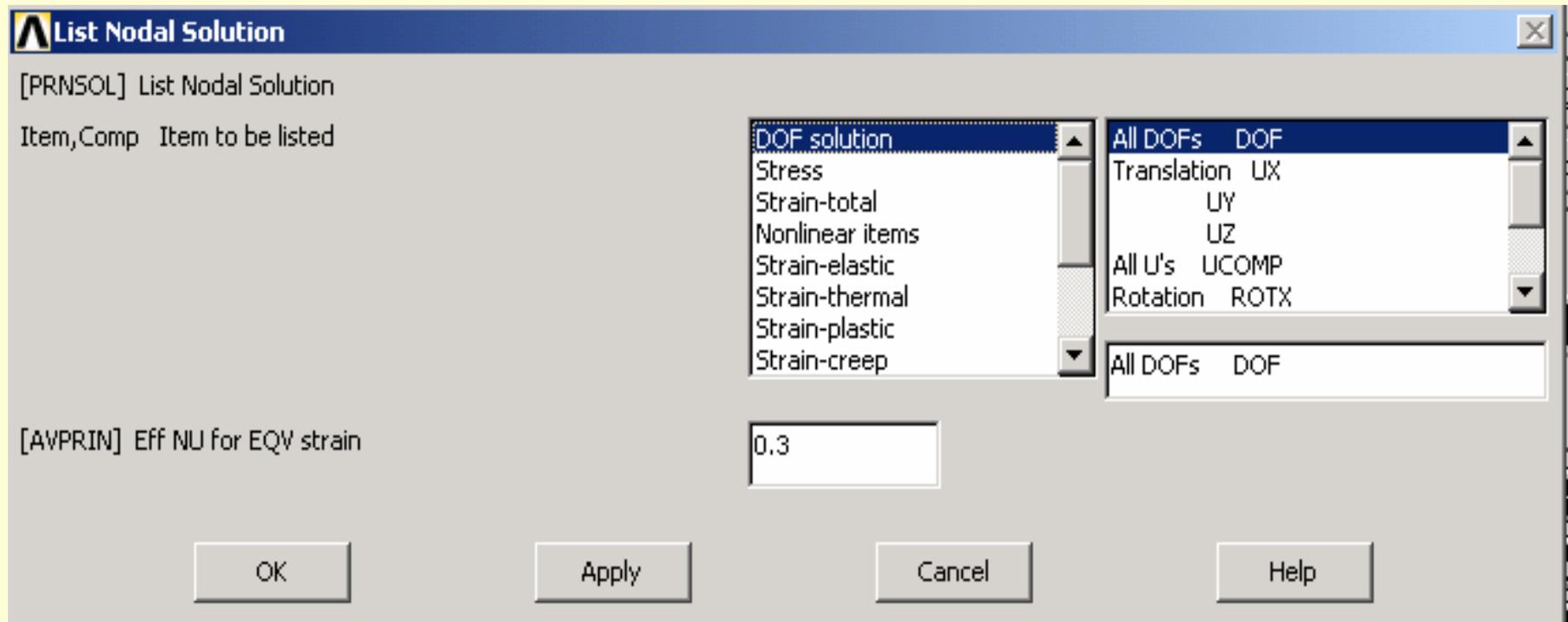
列表显示单元求解结果

列表显示约束点处支反力

列表显示节点力



(1) 列表显示节点求解结果



PRNSOL Command

File

PRINT DOF NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP= 0 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN GLOBAL COORDINATES

NODE	UX	UY	UZ	ROTX	ROTY	ROTZ
1	1.0452	13.202	-0.35611E-01	-0.14429E-03	0.11101E-03	-0.10003E-02
2	0.48539E-01	13.203	0.10435	-0.14616E-03	0.10250E-03	-0.10009E-02
3	0.84554	13.202	-0.79528E-02	-0.13705E-03	0.10527E-03	-0.99806E-03
4	0.64600	13.202	0.19517E-01	-0.13878E-03	0.10355E-03	-0.99745E-03
5	0.44656	13.202	0.47166E-01	-0.13859E-03	0.10261E-03	-0.99707E-03
6	0.24729	13.202	0.75112E-01	-0.14280E-03	0.10206E-03	-0.99413E-03
7	0.47285E-01	12.451	0.27327E-01	-0.14835E-03	0.10394E-03	-0.10021E-02
8	0.47870E-01	12.827	0.65944E-01	-0.14803E-03	0.10243E-03	-0.10019E-02
9	1.0451	12.452	-0.11588	-0.15161E-03	0.11593E-03	-0.10011E-02
10	0.24656	12.451	-0.80112E-03	-0.13816E-03	0.10640E-03	-0.99644E-03
11	0.44605	12.451	-0.28583E-01	-0.13990E-03	0.10852E-03	-0.99829E-03
12	0.64552	12.451	-0.56626E-01	-0.14007E-03	0.11129E-03	-0.99803E-03
13	0.84507	12.452	-0.85079E-01	-0.14835E-03	0.11453E-03	-0.99954E-03
14	1.0452	12.827	-0.75322E-01	-0.14829E-03	0.10111E-03	-0.10001E-02

```

PRNSOL Command
File
12431  0.18077  -0.15346  0.25896E-02-0.33078E-05-0.29264E-05-0.21628E-03
12432  0.17993  -0.15357  0.12691E-02 0.22111E-05-0.20890E-05-0.21629E-03

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP=      0  SUBSTEP=      1
TIME=      1.0000  LOAD CASE=      0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN GLOBAL COORDINATES

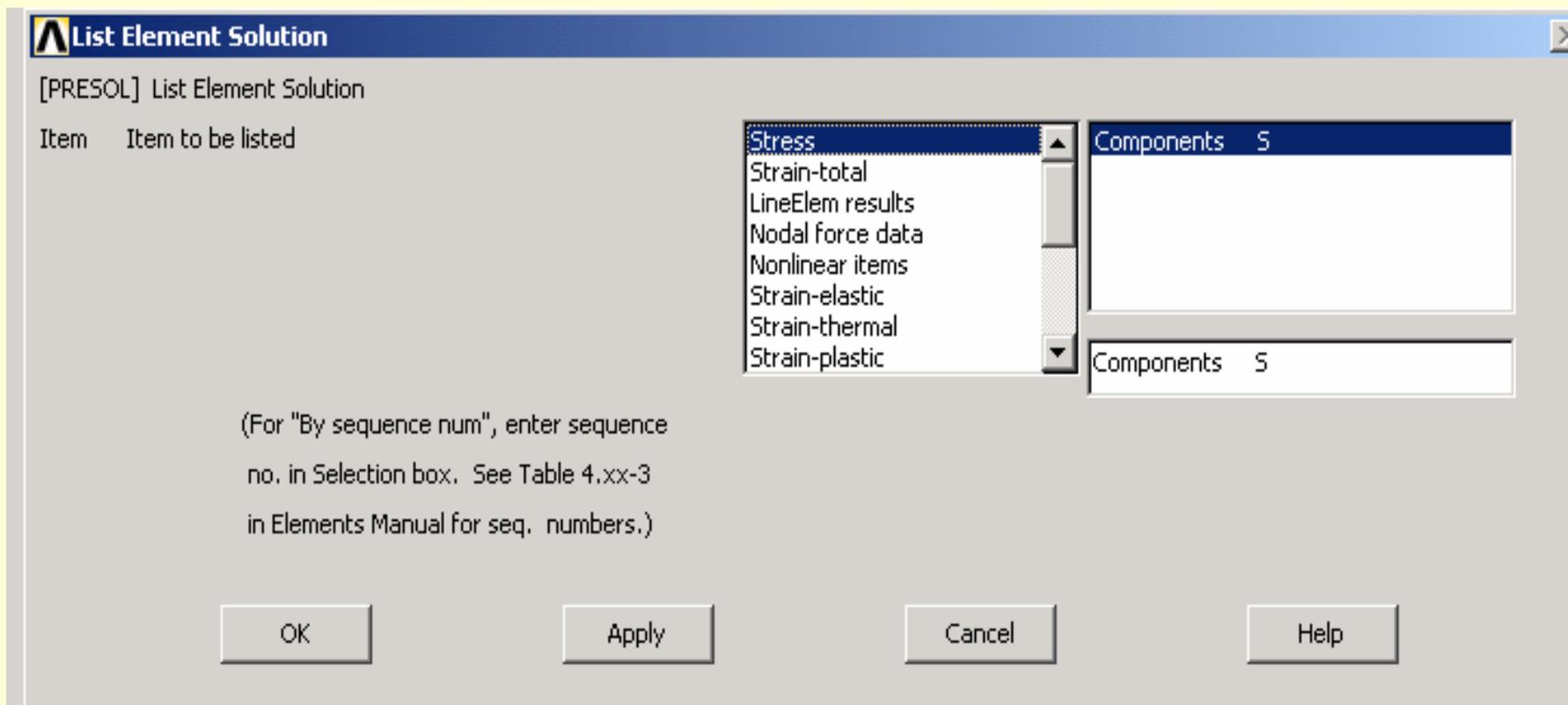
      NODE      UX      UY      UZ      ROTX      ROTY      ROTZ
12433  0.99729E-01-0.15383  0.24023E-02-0.14629E-05-0.32492E-05-0.21598E-03
12434  0.98890E-01-0.15395  0.11983E-02 0.11908E-05-0.17928E-05-0.21598E-03
12435  0.18759E-01-0.15418  0.22450E-02 0.33235E-05-0.30722E-05-0.21586E-03
12436  0.17919E-01-0.15430  0.11035E-02-0.13091E-05-0.19702E-05-0.21587E-03

MAXIMUM ABSOLUTE VALUES
      NODE      8562      4717      7940      10624      8651      1441
      VALUE      2.3097      -16.043      -2.4140      0.11479E-02-0.16369E-03-0.15770E-02

```

各方向最大位移所在节点号

(2) 列表显示单元求解结果



File

PRINT S ELEMENT SOLUTION PER ELEMENT

***** POST1 ELEMENT NODAL STRESS LISTING *****

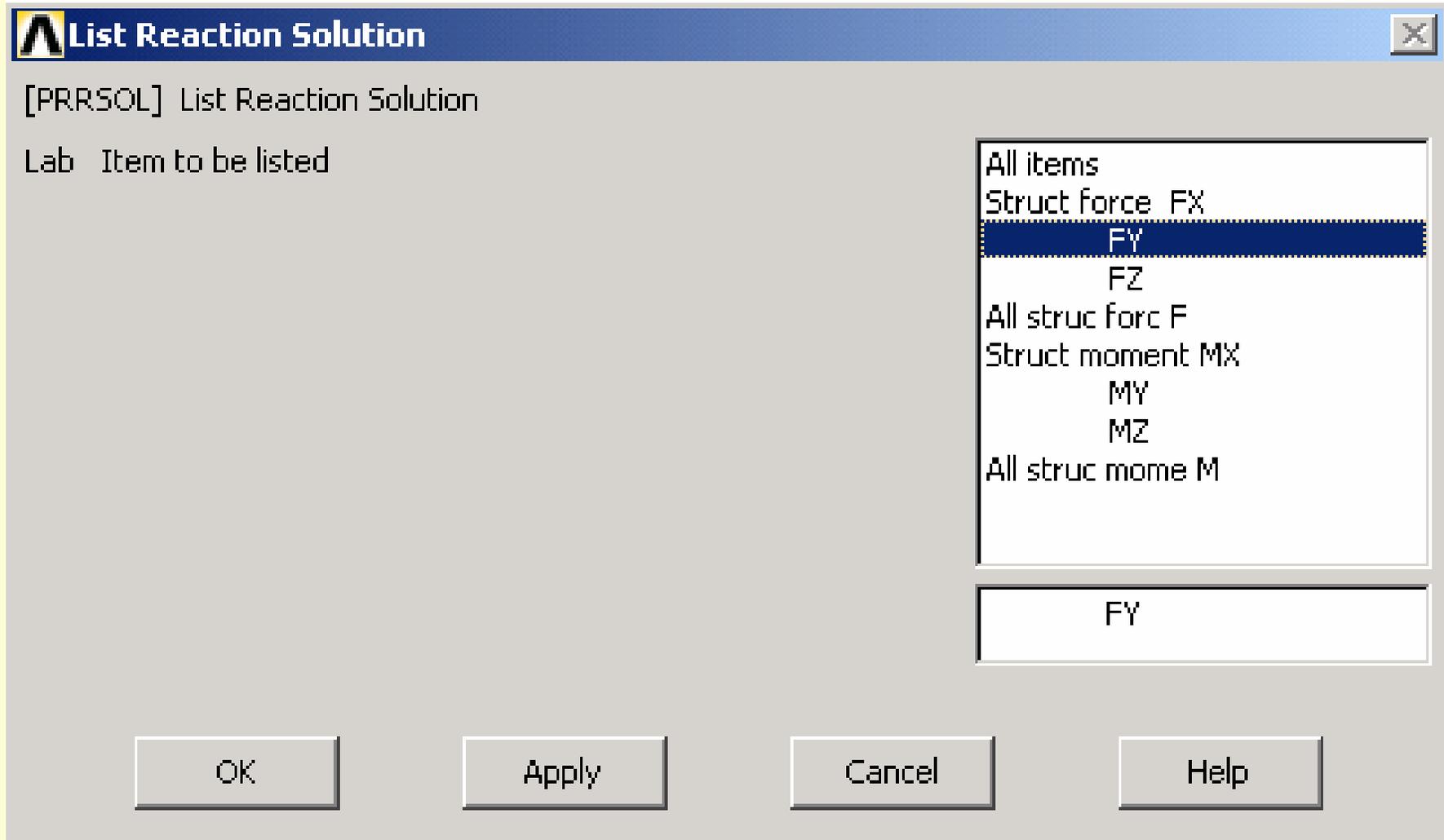
LOAD STEP= 0 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0
SHELL RESULTS ARE FOR TOP/BOTTOM

THE FOLLOWING X,Y,Z VALUES ARE IN GLOBAL COORDINATES

ELEMENT= 1 SHELL63

NODE	SX	SY	SZ	SXY	SYZ	SXZ
1	0.14605E-01	0.13363	0.0000	-0.17810	0.0000	0.0000
3	-0.33846E-01	0.54728E-01	0.0000	-0.17419	0.0000	0.0000
15	-0.83177E-02	-0.12810E-01	0.0000	-0.18392	0.0000	0.0000
14	-0.67023E-02	-0.58523E-01	0.0000	-0.18783	0.0000	0.0000
1	0.11624E-01	0.14759E-01	0.0000	-0.20404	0.0000	0.0000
3	0.65041E-02	0.93658E-01	0.0000	-0.20795	0.0000	0.0000
15	-0.19024E-01	-0.84859E-01	0.0000	-0.19822	0.0000	0.0000
14	0.32931E-01	-0.39146E-01	0.0000	-0.19431	0.0000	0.0000

(3)列表显示约束点处支反力



```

PRRSOL Command
File
PRINT FY REACTION SOLUTIONS PER NODE

***** POST1 TOTAL REACTION SOLUTION LISTING *****

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING X,Y,Z SOLUTIONS ARE IN GLOBAL COORDINATES

NODE      FY
1411  0.13405E+06
1412   85993.
1413   84371.
1414   73560.
1415   80649.
1416  -58008.
1417   34560.
1418   15169.
1419  -944.92
1420  -19323.
1421  -73673.
1422  -48781.

```

```

***** POST1 TOTAL REACTION SOLUTION
LISTING *****

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

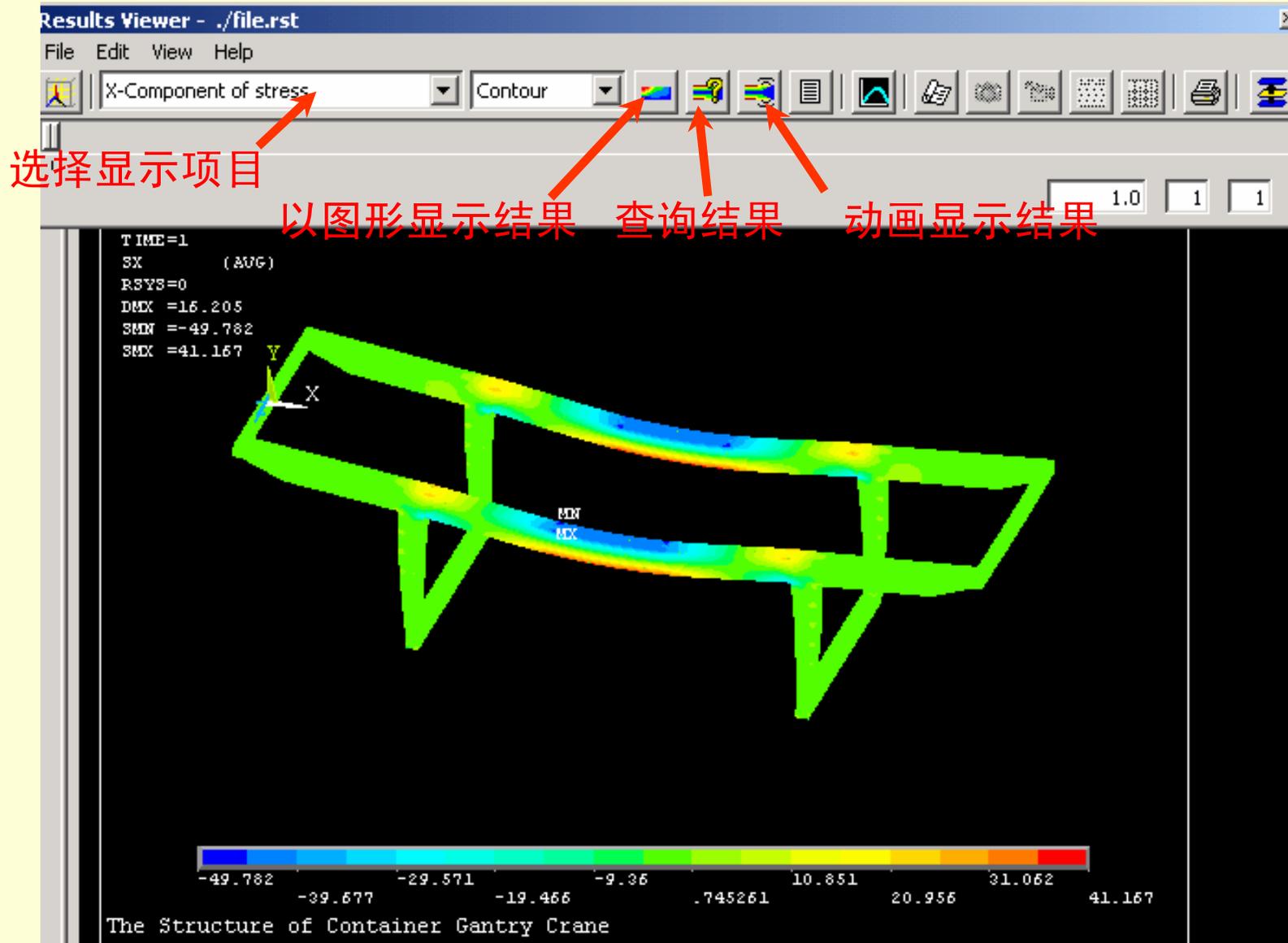
THE FOLLOWING X,Y,Z SOLUTIONS ARE IN
GLOBAL COORDINATES

NODE      FY
10732   26.431
10733  -16.443
10734    6.5282
10735   -3.7786
10736   17.650
10737  -28.417
10738    2.4348
10739   -4.6444
10740   23.640

TOTAL VALUES
VALUE  0.11000E+07

```

16. 通过结果浏览器显示计算结果



查询结果

