

تعريف مدل

نوع مدل: صفحه ای

نوع المان: Shell 181

نوع مصالح: الاستيک خطی

ضخامت ورق: ۱ سانتی متر

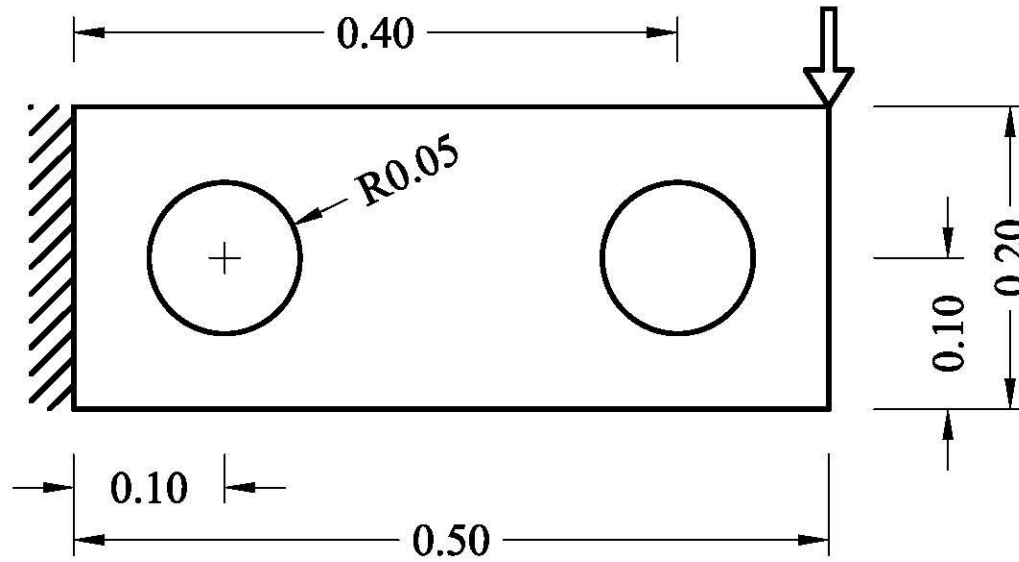
مقدار بار: ۱۸۰۰ نیوتن

واحدھا

مدول الاستیسیته نیوتن بر متر مربع،

چگالی: کیلوگرم بر متر مکعب،

طول: متر



Units:m

Start → All Programs → ANSYS 17 → ANSYS Product Launcher →



تعیین نوع المان

واحدھا: مدول الاستیستیة نیوتن بر متر مربع، چگالی: کیلوگرم بر متر مکعب، طول: متر

The image shows the ANSYS Multiphysics Utility Menu interface. The 'Main Menu' is visible on the left, with a pink arrow pointing to the 'Element Type' option under the 'Preprocessor' section, labeled 'شروع' (Start). The 'Element Types' dialog box is open in the center, showing 'NONE DEFINED' under 'Defined Element Types'. Below this dialog is the 'Library of Element Types' dialog box, which is also open. In this dialog, the 'Shell' category is selected in the left pane, and '3D 4node 181' is selected in the right pane. A pink arrow points to this selection. The 'Element type reference number' is set to '1'. The 'Library of Element Types' dialog also shows a list of other element types: Structural Mass, Link, Beam, Pipe, Solid, Shell, and Solid-Shell. The '3D 4node 181' element type is highlighted in blue. The 'Element type reference number' field contains the value '1'. The 'Library of Element Types' dialog has buttons for 'OK', 'Apply', 'Cancel', and 'Help'.

تعريف مشخصات مصالح

مدول الاستيسيته و ضريب پواسون

The screenshot illustrates the steps to define material properties in ANSYS:

- Main Menu:** The 'Material Models' option is highlighted, with a pink arrow labeled 'شروع' (Start) pointing to it.
- Define Material Model Behavior:** The 'Material Models Defined' list contains 'Material Model Number 1'. The 'Material Models Available' list shows 'Isotropic' selected under the 'Elastic' category.
- Linear Isotropic Properties for Material Number 1:** The dialog box shows the following properties:

Property	Value
Temperatures	T1
EX	2.1e11
PRXY	0.35

The screenshot displays the ANSYS Multiphysics Utility Menu interface. The 'Main Menu' on the left includes sections like Preferences, Preprocessor, and Solution. A pink arrow labeled 'شروع' (Start) points to the 'Material Models' option in the Preprocessor section. The 'Define Material Model Behavior' dialog box is open, showing 'Material Models Defined' (Density, Linear Isotropic) and 'Material Models Available' (Linear, Nonlinear, Thermal Expansion, Damping, Friction Coefficient). A pink arrow points to 'Density' under the 'Nonlinear' category. Below this, the 'Density for Material Number 1' dialog box is shown, with 'Temperatures' set to 0 and 'DENS' set to 7850. A pink arrow points to the '7850' value. The status bar at the bottom shows 'mat=1', 'type=1', 'real=1', 'csys=0', and 'secn=1'.

تعیین ضخامت المان

نام گذاری ضخامت و مقدار دهی بر حسب متر

The screenshot shows the ANSYS Multiphysics Utility Menu with the 'Create and Modify Shell Sections' dialog box open. The 'Lay-up' tab is selected, and the 'Add / Edit' option is highlighted in the 'Lay-up' menu. The dialog box shows a table with the following data:

	Thickness	Material ID	Orientation	Integration Pts	Pictorial View
1	0.01	1	0.0	3	

The 'Name' field is set to 'tp' and the 'ID' is set to '1'. The 'Section Offset' is set to 'Mid-Plane' and the 'Section Function' is set to 'None defined'. The 'Add Layer' and 'Delete Layer' buttons are visible below the table.

تعريف نقاط اصلي Keypoints

ANSYS Multiphysics Utility Menu

File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help

Toolbar

SAVE DB RESUM DB QUIT POWRGRPH

Main Menu

- Preferences
- Preprocessor
 - Element Type
 - Real Constants
 - Material Props
 - Sections
 - Modeling
 - Create
 - Keypoints
 - On Working Plane
 - In Active CS
 - On Line
 - On Line w/Ratio
 - On Node
 - KP between KPs
 - Fill between KPs
 - KP at center
 - Hard PT on line
 - Hard PT on area
 - Lines
 - Areas
 - Volumes
 - Nodes
 - Elements
 - Contact Pair
 - Circuit
 - Racetrack Coil
 - Transducers
 - Operate

ANSYS R17.0

Create Keypoints in Active Coordinate System

[K] Create Keypoints in Active Coordinate System

NPT Keypoint number: 1

X,Y,Z Location in active CS: 0 0 0

OK Apply Cancel Help

شروع

سایر KPها مطابق جدول زیر وارد شوند:

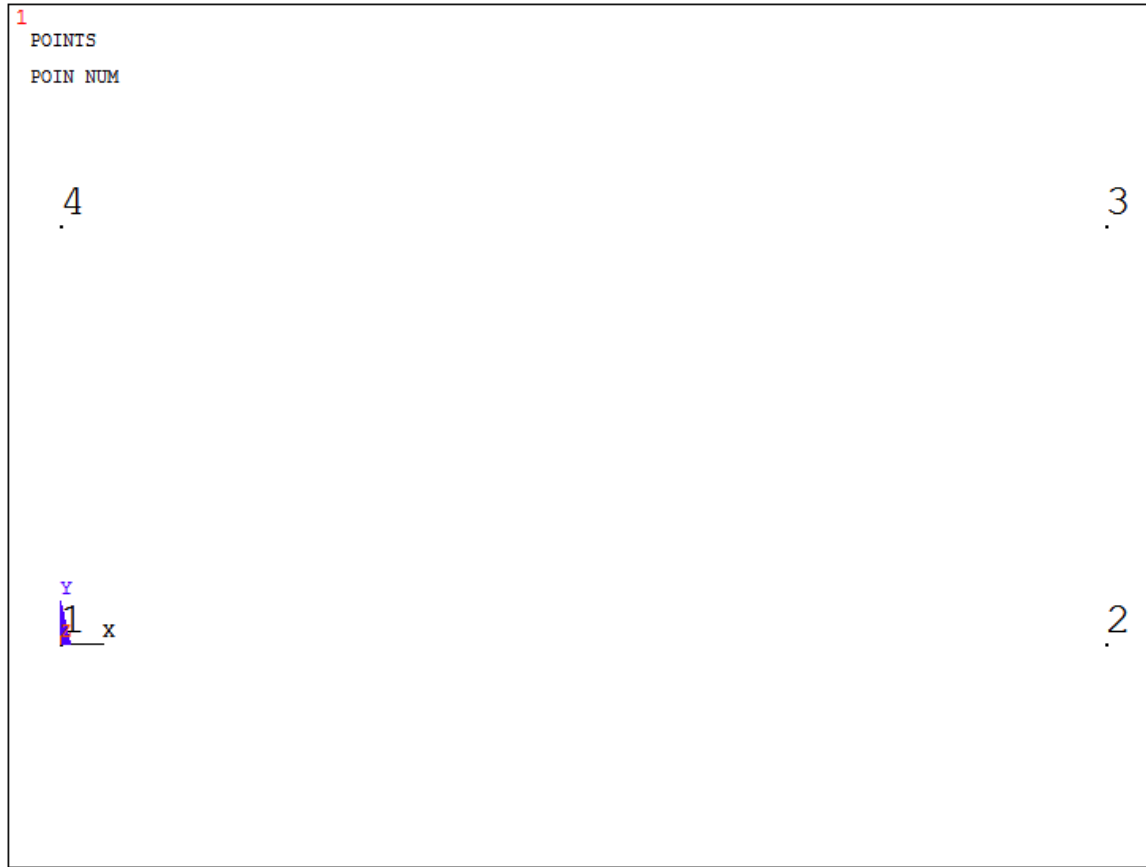
شماره KP	X	Y
1	0	0
2	0.5	0
3	0.5	0.2
4	0	0.2

Pick a menu item or enter a command (PREP7) mat=1 type=1 real=1 csys=0 secn=1

The screenshot shows the ANSYS Multiphysics Utility Menu interface. The 'Numbering' menu item is highlighted, and the 'Plot Numbering Controls' dialog box is open. The dialog box contains the following settings:

- [/PNUM] Plot Numbering Controls
- KP Keypoint numbers: On
- LINE Line numbers: Off
- AREA Area numbers: Off
- VOLU Volume numbers: Off
- NODE Node numbers: Off
- Elem / Attrib numbering: No numbering
- TABN Table Names: Off
- SVAL Numeric contour values: Off
- [/NUM] Numbering shown with: Colors & numbers
- [/REPLOT] Replot upon OK/Apply?: Replot

Buttons at the bottom of the dialog box are OK, Apply, Cancel, and Help. The status bar at the bottom of the window shows: mat=1 type=1 real=1 csys=0 secn=1.

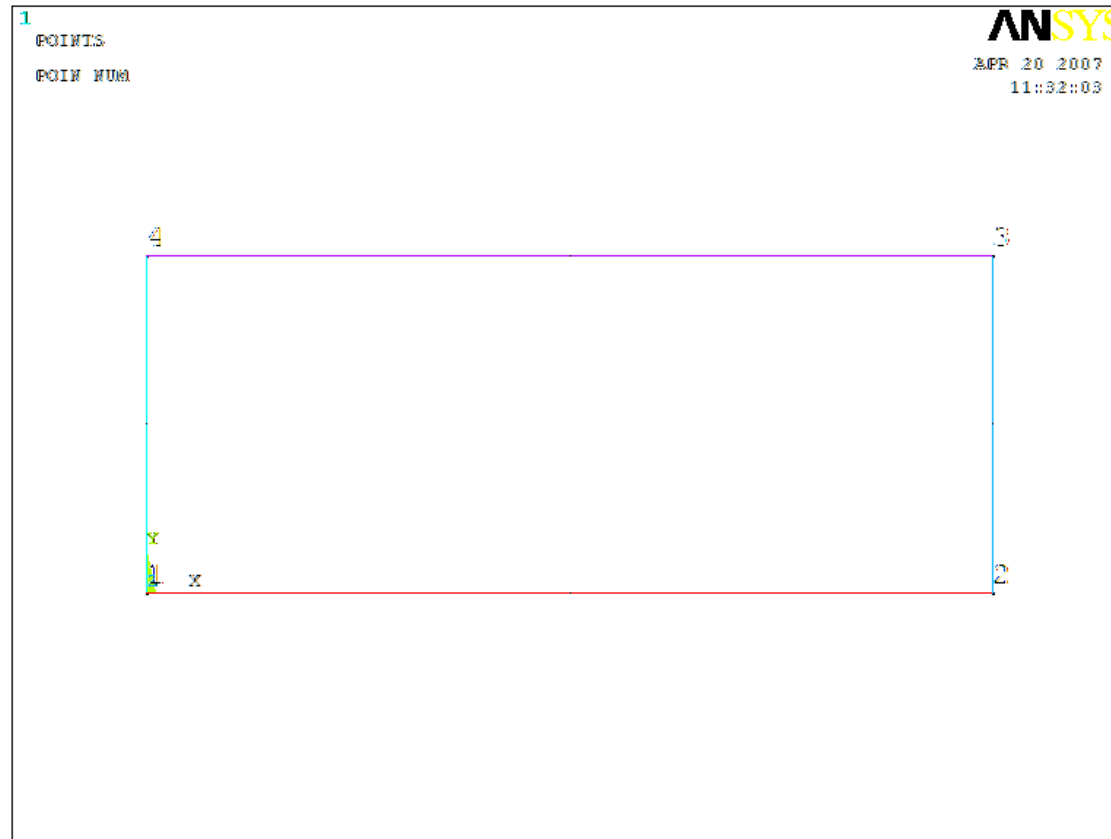


تعریف خطوط بین نقاط اصلی

The screenshot shows the ANSYS Multiphysics Utility Menu interface. The 'Main Menu' is expanded to 'Preprocessor' > 'Modeling' > 'Create' > 'Lines' > 'Straight Line'. A pink arrow labeled 'شروع' (Start) points to the 'Straight Line' option. The main window displays a coordinate system with points 1, 2, 3, and 4. A red line is drawn between points 1 and 2. A dashed box contains the following Persian text:

برای تعریف خطوط دو به دو بر نقاط اصلی کلیک نمایید. به این ترتیب چهار خط محیطی مستطیل ایجاد می شوند.

The status bar at the bottom shows: mat=1 type=1 real=1 csys=0 secn=1



تعریف سطح به کمک خطوط

ANSYS Multiphysics Utility Menu

File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help

Toolbar

SAVE_DB RESUM_DB QUIT POWRGRPH

Main Menu

- Preferences
- Preprocessor
 - Element Type
 - Real Constants
 - Material Props
 - Sections
 - Modeling
 - Create
 - Keypoints
 - Lines
 - Areas
 - Arbitrary
 - Through KPs
 - Overlaid on Area
 - By Lines**
 - By Skinning
 - By Offset
 - Rectangle
 - Circle
 - Polygon
 - Area Fillet
 - Volumes
 - Nodes
 - Elements
 - Contact Pair
 - Circuit
 - Racetrack Coil
 - Transducers
 - Operate

شروع

1
2
3
4

1 x

پس از زدن دکمه By Lines، بر روی چهار خط محیطی کلیک کرده و دکمه Ok را بفشارید

Pick a menu item or enter a command (PREP7) mat=1 type=1 real=1 csys=0 secn=1

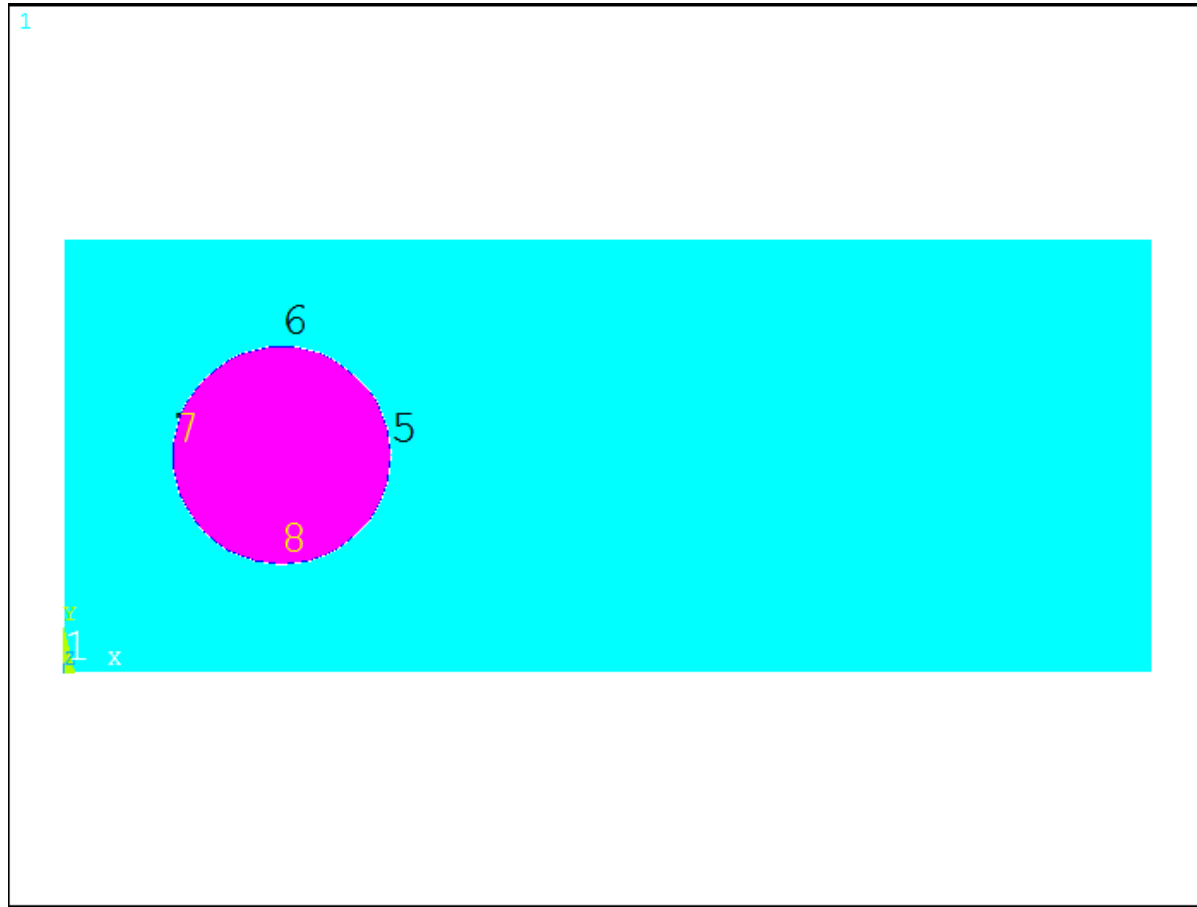
تغییر در سطح

ساخت یک دایره در محل سوراخ

The screenshot shows the ANSYS Multiphysics Utility Menu interface. The 'Main Menu' is expanded to 'Preprocessor' > 'Modeling' > 'Create' > 'Circle' > 'Solid Circle'. A pink arrow labeled 'شروع' (Start) points to the 'Solid Circle' option. The central workspace shows a cyan rectangular area with a coordinate system (x, y, z) and a '1' label. A 'Solid Circular Area' dialog box is open, with 'Pick' selected. The dialog box contains the following fields:

WP X	=	
Y	=	
Global X	=	
Y	=	
Z	=	
WP X		0.1
WP Y		0.1
Radius		0.05

The 'Apply' button is highlighted with a dashed circle and a pink arrow. The status bar at the bottom shows: [CYL4] Pick 2 WP locations -- center and radius | mat=1 | type=1 | real=1 | csys=0 | secn=1

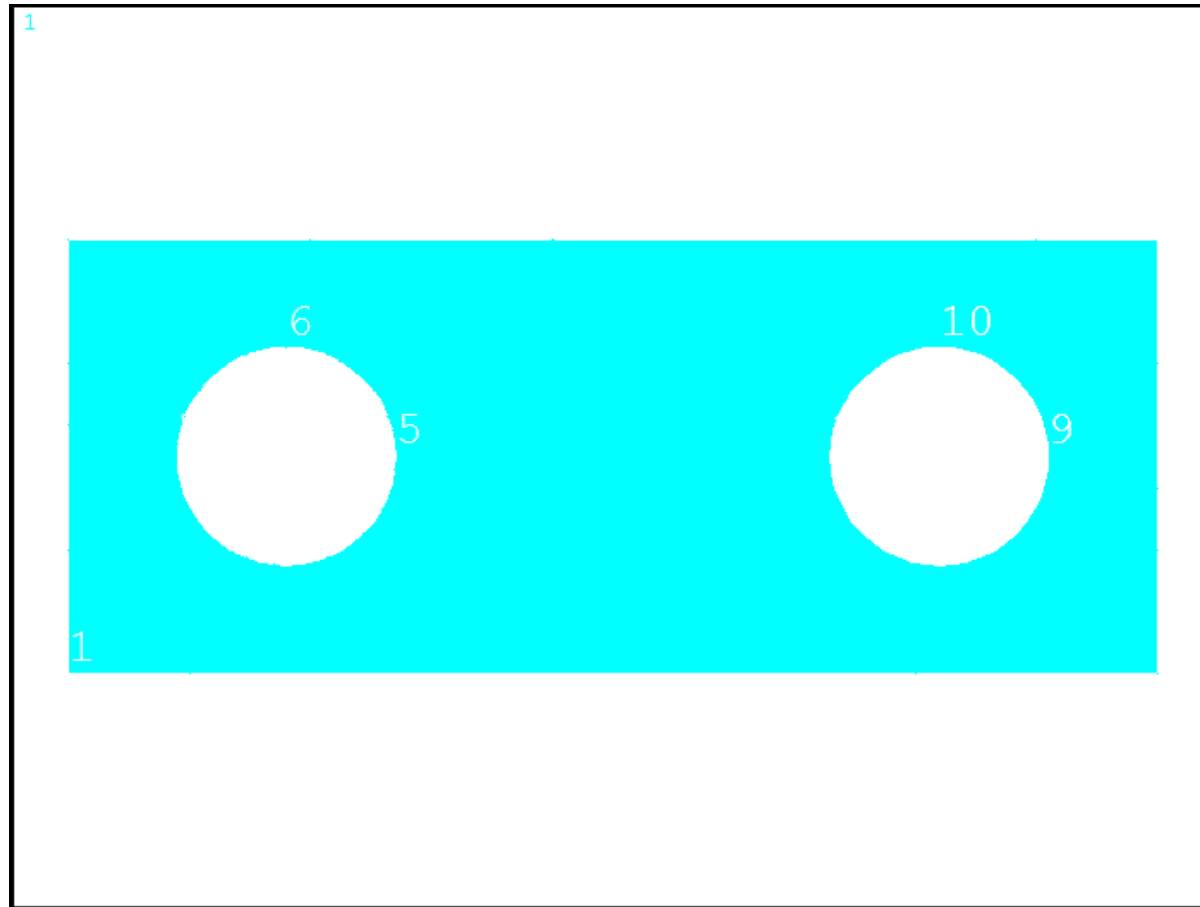


کم کردن سطح دایره‌ای از مستطیل

The screenshot shows the ANSYS Multiphysics Utility Menu interface. The 'Main Menu' is expanded to 'Booleans', and the 'Subtract' option is highlighted. A pink arrow with the Persian word 'شروع' (Start) points to this option. The main window displays a 2D model of a cyan rectangle with a white circular hole. The hole is labeled '6' at the top and '5' on the right side. The rectangle is labeled '1' at the top-left and bottom-left corners. The status bar at the bottom shows 'Pick a menu item or enter a command (PREP7)' and various parameters: mat=1, type=1, real=1, csys=0, secn=1.

برای subtract کردن سطح دایره‌ای از سطح مستطیلی، ابتدا روی مستطیل کلیک کرده، پس از زدن دکمه apply روی دایره کلیک کرده و دکمه ok را فشار دهید.

به همین ترتیب دایره دوم تعریف شده و از مستطیل کم می شود.



نسبت دادن مشخصات به سطح

تعیین نوع المان، مشخصات المان و نوع مصالح

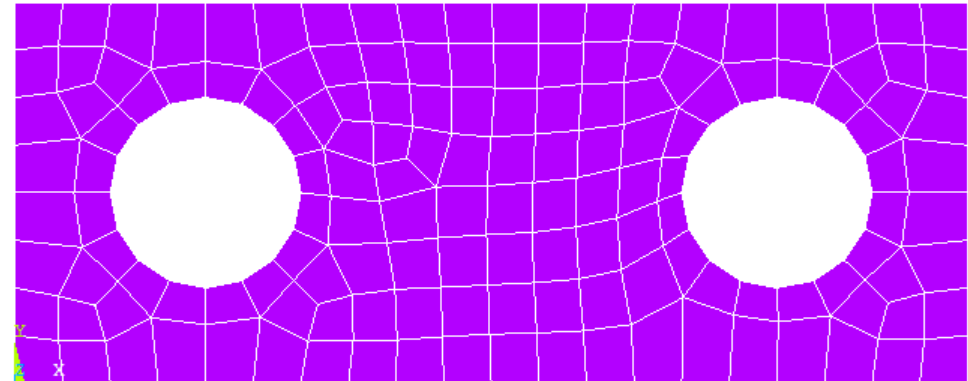
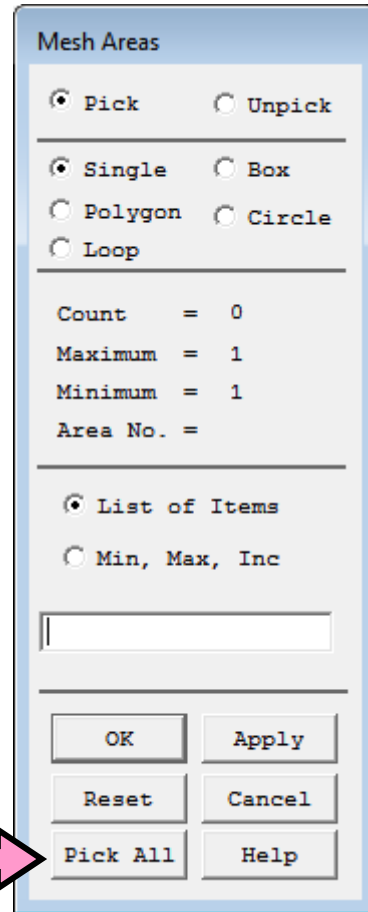
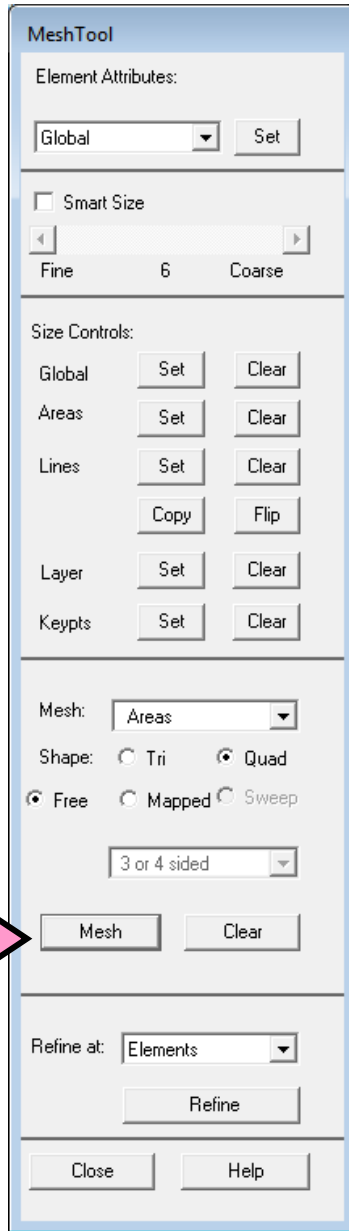
The screenshot shows the ANSYS Multiphysics Utility Menu interface. The 'Main Menu' is open, and the 'Picked Areas' option under the 'Meshing' section is highlighted. A pink arrow points to this option with the word 'شروع' (Start) written next to it. The 'Area Attributes' dialog box is open, displaying the following settings:

Field	Value
MAT Material number	1
REAL Real constant set number	None defined
TYPE Element type number	1 SHELL181
ESYS Element coordinate sys	0
SECT Element section	1 tp

Buttons at the bottom of the dialog include OK, Apply, Cancel, and Help. The background window shows a 3D model of a hole in a plate with a cyan selection box around it. A text box in the background reads: 'با ورود به Picked Areas، سطح را انتخاب کرده و دگمه Enter را بفشارید.' (By entering Picked Areas, select the surface and press the Enter key.)

تعريف اندازہ المانها روی کلیه خطوط

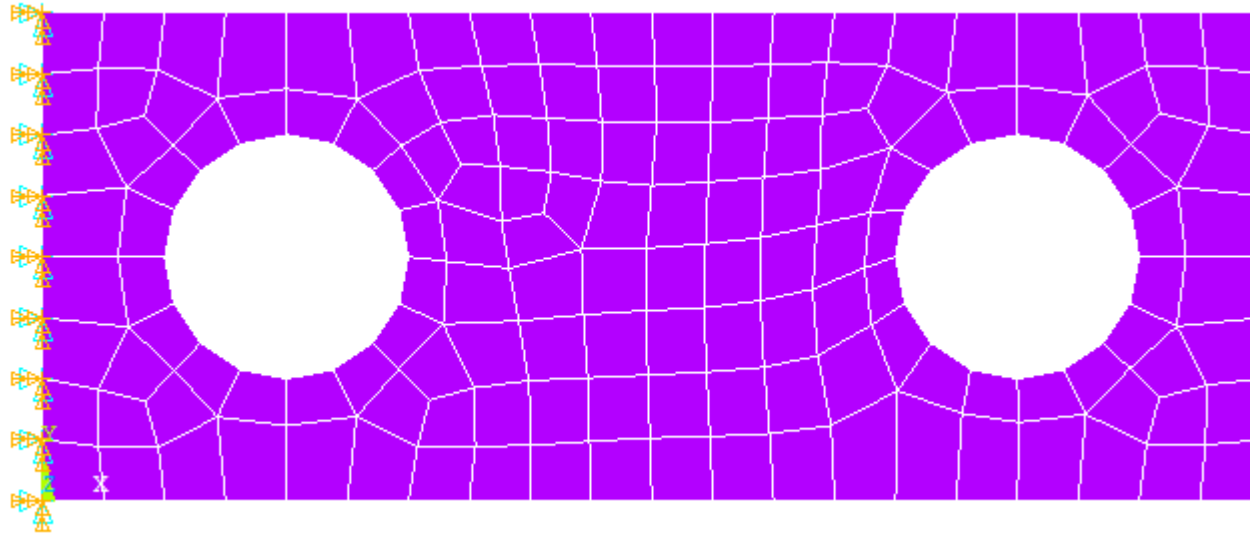
The image shows the ANSYS Multiphysics Utility Menu interface. The **MeshTool** dialog is open, with the **Meshing** section selected in the Main Menu. A pink arrow labeled "شروع" (Start) points to the **Meshing** option. The **Element Size on Picked Lines** dialog is also open, showing options for **Pick**, **Unpick**, **Single**, **Box**, **Polygon**, **Circle**, and **Loop**. A pink arrow points to the **Mesh** button in the MeshTool dialog. Another pink arrow points to the **OK** button in the Element Size dialog. A third pink arrow points to the **SIZE** input field in the Element Size dialog, which contains the value 0.025. A fourth pink arrow points to the **KYNDIV** checkbox, which is unchecked. The background shows a 3D model of a rectangular plate with a central hole, with a cyan rectangular area highlighted on the top edge.



The screenshot displays the ANSYS Multiphysics Utility Menu interface. The 'Main Menu' on the left lists various options, with 'New Analysis' highlighted under the 'Analysis Type' section. A pink arrow points to this menu item. The 'New Analysis' dialog box is open in the center, showing the 'Type of analysis' section with 'Static' selected. A second pink arrow points to the 'Static' radio button. The background shows a 2D meshed part with a hole. The status bar at the bottom shows the command 'mat=1 type=1 real=1 csys=0 secn=1'.

تعیین محل تکیه گاهها و نوع آنها

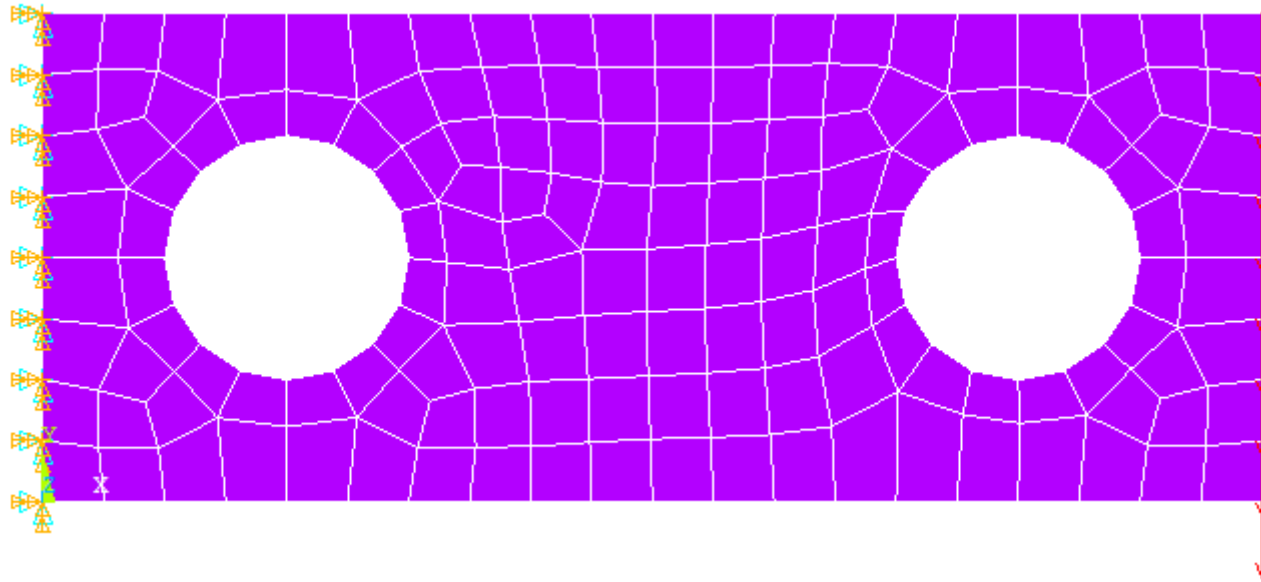
The screenshot shows the ANSYS Multiphysics Utility Menu interface. The main menu on the left is expanded to 'Solution' > 'Define Loads' > 'Apply' > 'Structural' > 'Displacement' > 'On Nodes'. A pink arrow labeled 'شروع' (Start) points to the 'On Nodes' option. The central window displays a meshed arch structure with a vertical line of nodes highlighted in yellow. A dialog box titled 'Apply U,ROT on Nodes' is open, showing options for 'Pick' and 'Unpick', and 'Single' and 'Box' selection methods. The 'Count' is 9, 'Maximum' is 192, 'Minimum' is 1, and 'Node No.' is 30. Below the dialog, another 'Apply U,ROT on Nodes' dialog is shown, with 'All DOF' selected in the 'DOFs to be constrained' list. A pink arrow points to this list. The status bar at the bottom indicates '[D] Pick or enter nodes for displacement constraints' and shows parameters: mat=1, type=1, real=1, csys=0, secn=1.



تعیین محل وارد کردن بارهای متمرکز

The screenshot shows the ANSYS Multiphysics Utility Menu interface. The main menu on the left is expanded to 'Apply' > 'Structural' > 'Force/Moment' > 'On Nodes'. A pink arrow labeled 'شروع' (Start) points to this menu path. The central window displays a meshed plate with a circular hole. A blue box highlights a vertical edge of the mesh. A dialog box titled 'Apply F/M on Nodes' is open, showing options for 'Pick' (selected), 'Unpick', 'Single', 'Box' (selected), 'Polygon', and 'Circle'. Below these are statistics: Count = 9, Maximum = 192, Minimum = 1, and Node No. = 22. The 'List of Items' option is selected. A second dialog box, also titled 'Apply F/M on Nodes', is open in the foreground, showing the configuration for the force application: Lab Direction of force/mom is set to 'FY', Apply as is set to 'Constant value', and the Force/moment value is '-200'. Pink arrows point to the 'FY' dropdown, the 'Constant value' dropdown, the '-200' input field, and the 'OK' button. A white arrow points to the highlighted edge in the mesh. The status bar at the bottom shows '[F] Pick or enter nodes for force/moment loading' and 'secn=1'.

مدل نهایی پس از المان بندی و بار گذاری



ذخیره مدل

در این مرحله مدل را ذخیره کنید.

File → Save as Jobname

ANSYS Multiphysics Utility Menu

File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help

Toolbar

SAVE_DB RESUM_DB QUIT P

Main Menu

- Preferences
- Preprocessor
- Solution
 - Analysis Type
 - Define Loads
 - Load Step Opts
 - SE Management (CMS)
 - Results Tracking
 - Solve
 - Current LS**
 - From LS Files
 - Manual Rezoning
 - Multi-field Set Up
 - ADAMS Connection
 - Diagnostics
 - Unabridged Menu
 - General Postproc
 - TimeHist Postpro
 - ROM Tool
 - Radiation Opt
 - Session Editor
 - Finish

/STATUS Command

```

SOLUTION OPTIONS
PROBLEM DIMENSIONALITY. . . . . 3-D
DEGREES OF FREEDOM. . . . . UX  UY  UZ  ROTX  ROTY  ROTZ
ANALYSIS TYPE . . . . . .STATIC (STEADY-STATE)
GLOBALLY ASSEMBLED MATRIX . . . . . .SYMMETRIC

LOAD STEP OPTIONS
LOAD STEP NUMBER. . . . . 1
TIME AT END OF THE LOAD STEP. . . . . 1.0000
NUMBER OF SUBSTEPS. . . . . 1
STEP CHANGE BOUNDARY CONDITIONS . . . . . .DEFAULT
PRINT OUTPUT CONTROLS . . . . . .NO PRINTOUT
DATABASE OUTPUT CONTROLS. . . . . .ALL DATA WRITTEN
FOR THE LAST SUBSTEP
    
```

Solve Current Load Step

[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.

OK Cancel Help

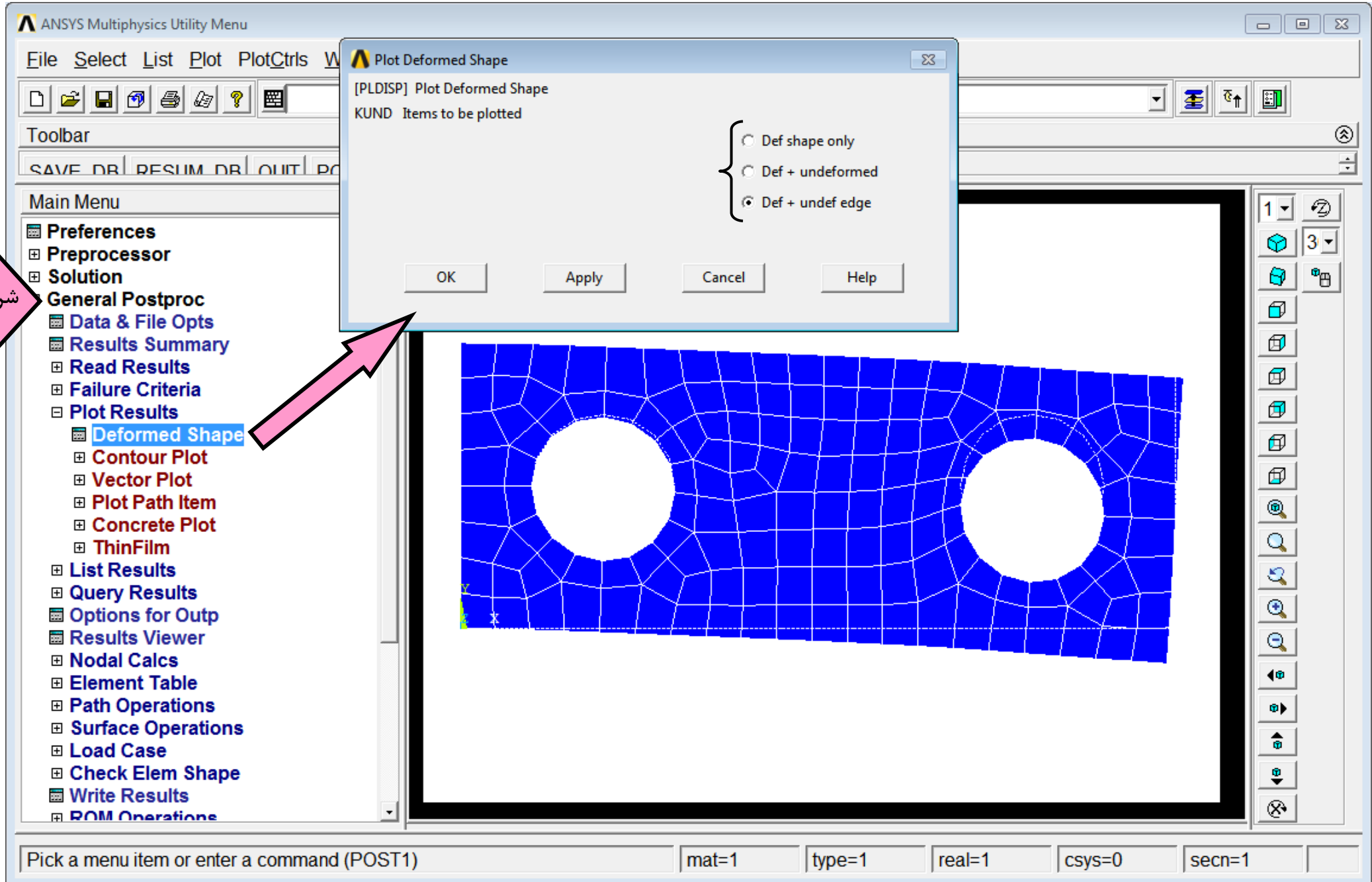
Note

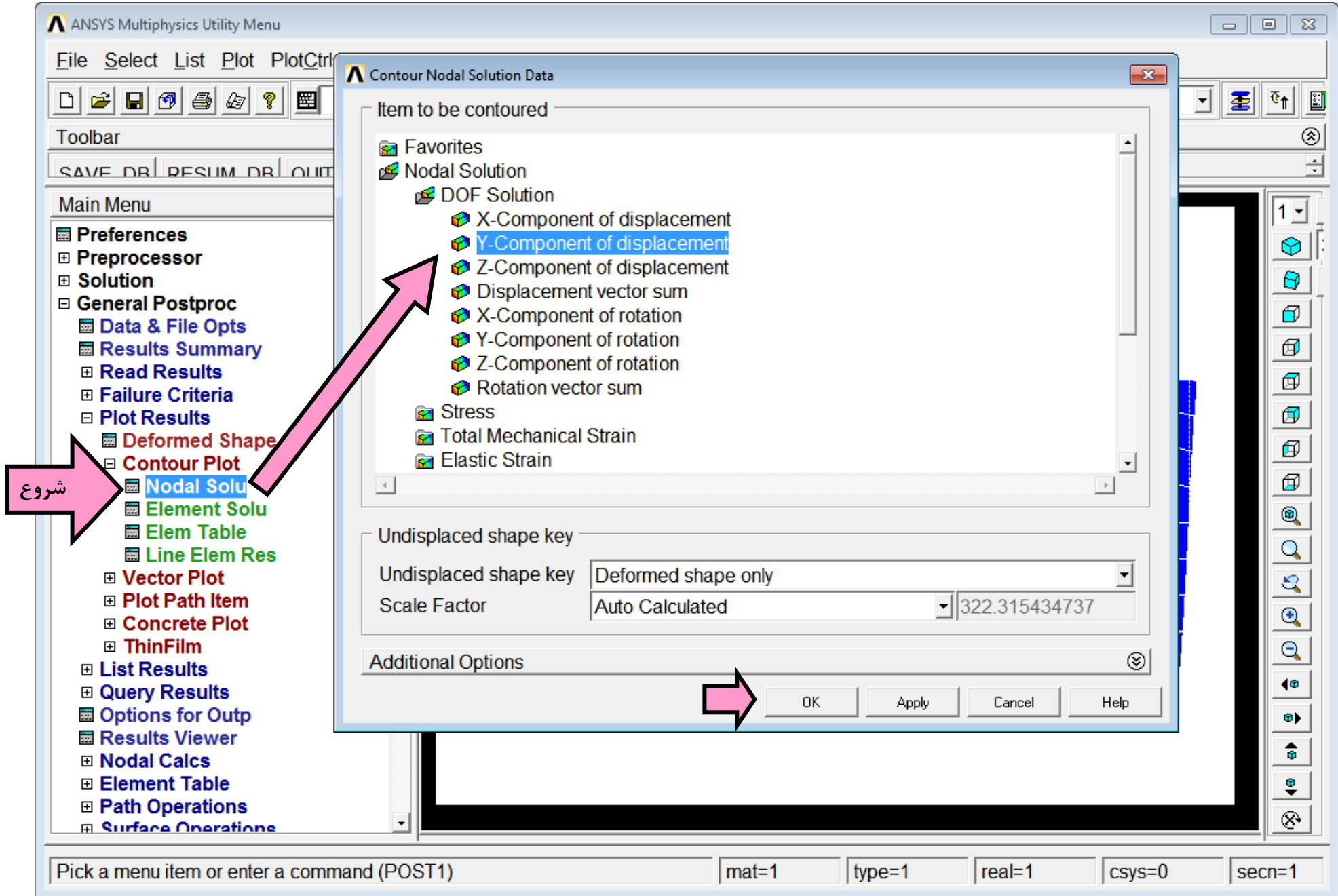
Solution is done!

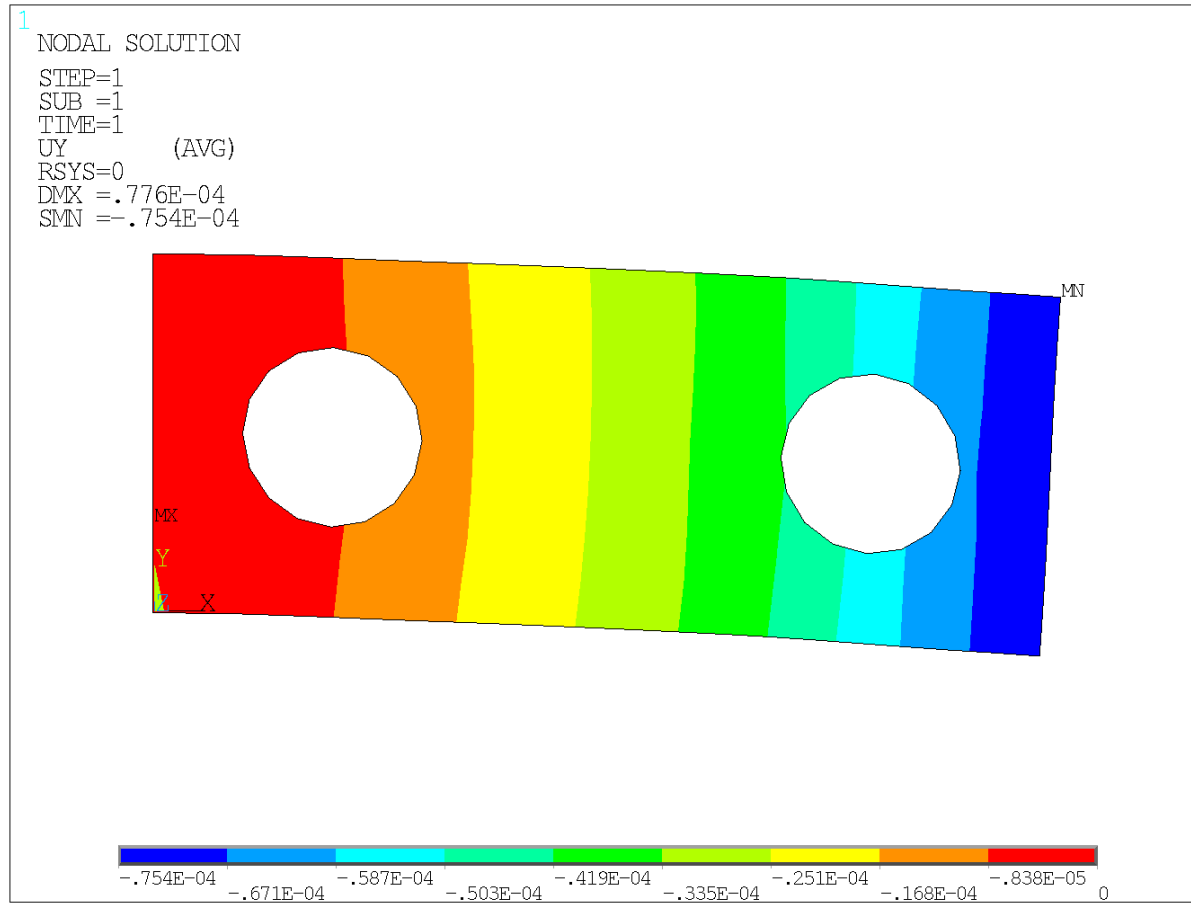
Close

Pick a menu item or enter a command (SOLUTION)

شروع

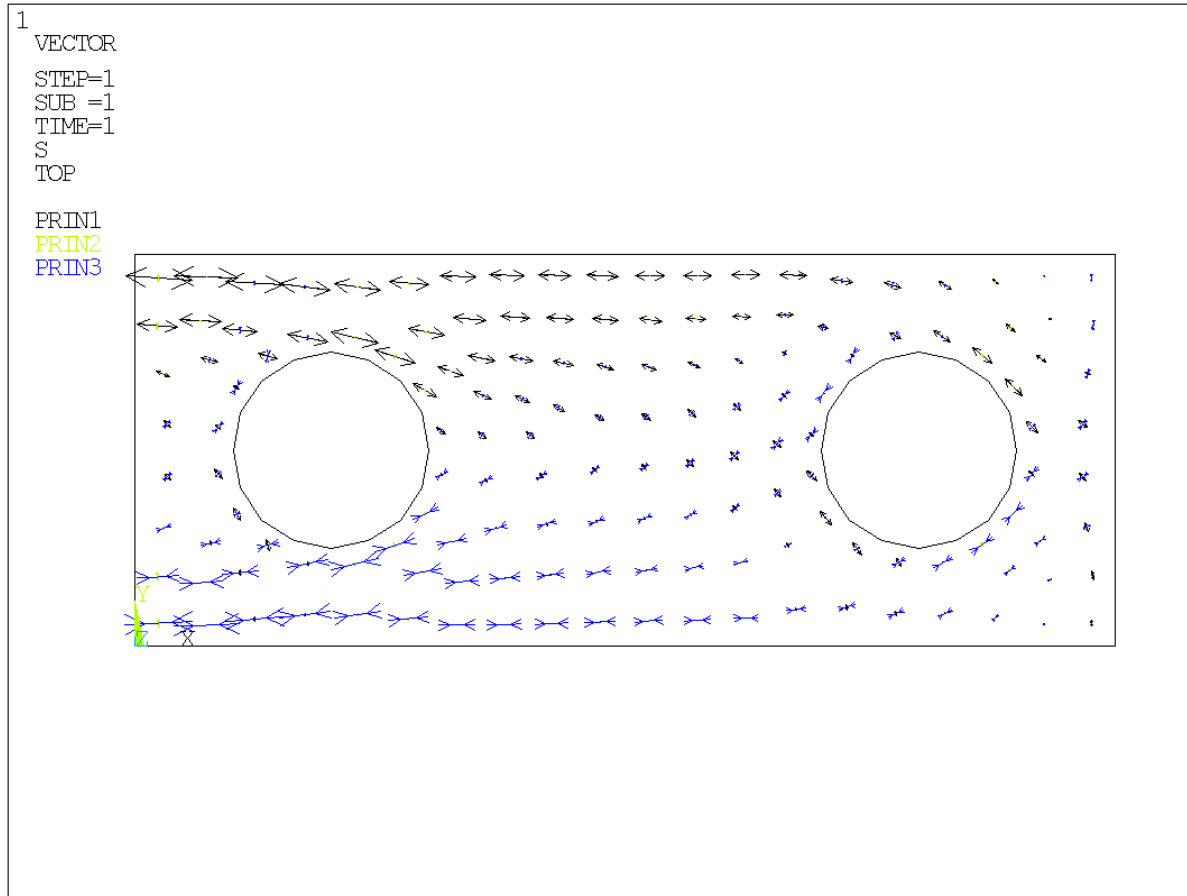




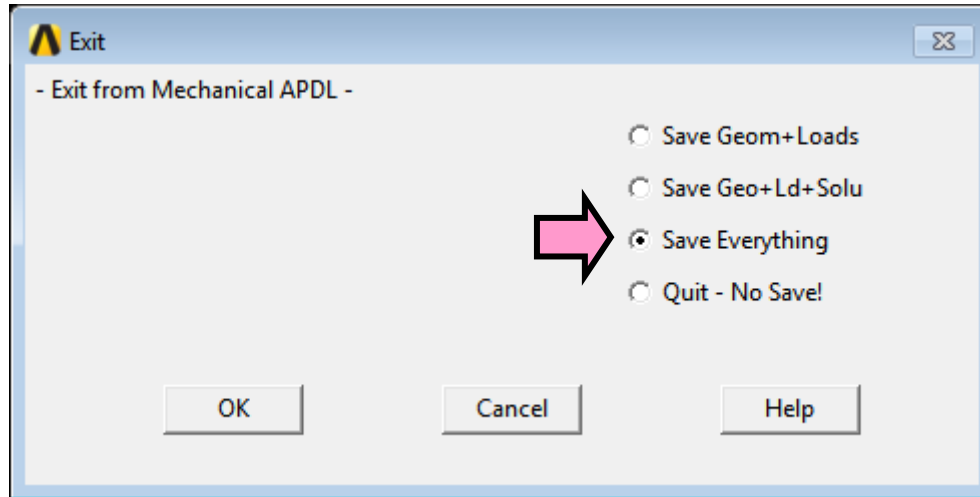


مشاهده تنشهای اصلی بصورت برداری

The screenshot displays the ANSYS Multiphysics Utility Menu interface. On the left, the 'Main Menu' is visible, with 'Plot Results' expanded and 'Predefined' selected. A pink arrow points to this menu item with the Persian word 'شروع' (Start) written next to it. The central dialog box, titled 'Vector Plot of Predefined Vectors', is open. It features a list of 'Item' options: 'DOF solution', 'Stress', 'Strain-total', 'Strain-mech+thrm', 'Strain-elastic', 'Strain-thermal', and 'Strain-plastic'. 'Stress' is highlighted, and a pink arrow points to it. To the right of this list, 'Principal' and 'S' are displayed. Below the list, there are radio buttons for 'Vector Mode' (selected) and 'Raster Mode'. Other settings include 'Loc' set to 'Elem Centroid', 'Edge' set to 'Element edges', and 'OPTION' set to 'Undeformed Mesh'. At the bottom of the dialog are 'OK', 'Apply', 'Cancel', and 'Help' buttons. The status bar at the bottom shows 'Pick a menu item or enter a command (POST1)' and various parameters like 'mat=1', 'type=1', 'real=1', 'csys=0', and 'secn=1'.



ذخیره کردن اطلاعات و خروج از برنامه



File→Exit→

Start → All Programs → ANSYS 17 → ANSYS Product Launcher →



