

How to draw structures in ANSYS

Problem description

Draw two beams in ANSYS, one is 100mm*10mm*10mm start from origin and the other is also 100mm*10mm*10mm connected to the first one.

ANSYS procedure

Main Menu → Preprocessor → Modeling → Create → Volumes → Block → By 2 Corners & Z → In the pop up window, input WPX “0”, WPY “0”, Width “100”, Height “10”, Depth “10” → Click “Apply” → Input WPX “100”, WPY “-90”, Width “10”, Height “100”, Depth “10” → Click “ok”

* You can also use “By Dimensions” to draw the beams by giving the coordinate of two opposite corners.

Thermal Analysis in ANSYS

Problem description

Calculate the temperature distribution on a 100mm*10mm*10mm beam with 25°C at one end and 1000W/m² heat flux on the other end

Method

Static analysis in ANSYS with thermal element types

ANSYS

1. Import the beam file from SolidWorks as described in last lecture
2. Define the analysis as a thermal analysis
Main Menu → Preference → check “Thermal” → ok
2. Select the element type as Solid 87
Main Menu → Preprocessor → Element Type → Add/Edit/Delete →
Click “Add” in the element dialog box →
Choose “Solid” under “Thermal” in the left scroll box of the library of element types →
Choose “Tet 10node 87” in the right scroll box →
Click “ok” to close the library →
Click “close” to close the element dialog box
3. Input the material properties
Main Menu → Preprocessor → Material Props → Material Models →
In the “Define Material Model Behavior” dialog box, double click “Thermal” on the right box →
double click “Conductivity” → double click “Isotropic” → Input “16” in the box for KXX → Click
“ok” to add the properties → double click “Specific Heat” → Input “500” in the box for C → Click
“ok” to add the property → double click “Density” → Input “8000” in the box for DENS → Click
“ok” to add the property → Close the “Define Material Model Behavior” dialog box
Main Menu → Preprocessor → Material Props → Temperature Units → Change the unit to Celsius
in the pop up window → ok
4. Mesh the structure
Main Menu → Preprocessor → Meshing → Mesh Tool →
In the “Mesh Tool” dialog box, check “Smart Size” and move the size down to fine “1” → Click
on “Mesh” → In the dialog box of pick, click “Pick All” → After meshing, click on “Refine” in the

"Mesh Tool" dialog box → Click "Pick All" → Change the level of refinement to "2" → Click ok
→ Click "Close" to close the Mesh Tool dialog box

5. Apply the load

Main Menu → Solution → Define Loads → Apply → Thermal → Temperature → On Areas →
Pick the left end area by mouse click (You can go to File Menu → PlotCtrls → Pan Zoom Rotate,
to find the area, check "Dynamic Mode" to rotate the model by mouse) → Click "ok" → Choose
"Temp" and put "25" as the value → Click "ok"

Main Menu → Solution → Define Loads → Apply → Thermal → Heat Flux → On Areas → Pick
the right end area by mouse click → Click "ok" → Input 1000 into the value → Click "ok"

6. Solve the problem

Main Menu → Solution → Current LS → Click "ok" → Click "close" after the solution is done and
close the window of commands

7. Review the results

Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solu → Choose the
results you want to review, for example, DOF Solution → Nodal Temperature → Click "ok"

8. Save the file

File Menu → Save as ... → Give a name to the file

Frequency Analysis in ANSYS

Problem description

Calculate the resonance frequency of a 100mm*10mm*10mm beam with one end fixed

Method

Modal analysis in ANSYS with Solid element types.

ANSYS procedure

1. Import the beam file from SolidWorks as described in last lecture

2. Define the analysis as a structural analysis

Main Menu → Preference → check "Structural" → ok

3. Select the element type as Solid 187

Main Menu → Preprocessor → Element Type → Add/Edit/Delete →

Click "Add" in the element dialog box →

Choose "Solid" under "Structural" in the left scroll box of the library of element types →

Choose "Tet 10node 187" in the right scroll box →

Click "ok" to close the library →

Click "close" to close the element dialog box

4. Input the material properties

Main Menu → Preprocessor → Material Props → Material Models →

In the "Define Material Model Behavior" dialog box, double click "Structural" on the right box →

double click "Linear" → double click "Elastic" → double click "Isotropic" → Input "1.9e11" in

the box for EX (Elastic Modulus) → Input "0.29" in the box for PRXY (Poisson's Ratio) → Click

"ok" to add the properties → double click "Density" → Input "8000" in the box for DENS → Click

"ok" to add the property → Close the "Define Material Model Behavior" dialog box

5. Mesh the structure

Main Menu → Preprocessor → Meshing → Mesh Tool →

In the "Mesh Tool" dialog box, check "Smart Size" and move the size down to fine "1" → Click on "Mesh" → In the dialog box of pick, click "Pick All" → After meshing, click on "Refine" in the "Mesh Tool" dialog box → Click "Pick All" → Change the level of refinement to "2" → Click ok → Click "Close" to close the Mesh Tool dialog box

9. Change the analysis type

Main Menu → Solution → Analysis Type → New Analysis → Check "Modal" in the pop up window → Click "ok"

Main Menu → Solution → Analysis Type → Analysis Options → Input 10 into the No. of modes to extract → Click "ok" → Change the End Frequency to 100000 → Click "ok"

10. Apply the load

Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Areas → Pick the fixed end area by mouse click (You can go to File Menu → PlotCtrls → Pan Zoom Rotate, to find the area, check "Dynamic Mode" to rotate the model by mouse) → Click "ok" → Choose "All DOF" and put "0" as the value → Click "ok"

11. Solve the problem

Main Menu → Solution → Current LS → Click "ok" → Click "close" after the solution is done and close the window of commands

12. Review the results

Main Menu → General Postproc → Results Viewer → Change "Choose a result item" into "Displacement vector sum" under "DOF solution" in the scroll down menu → move the bar to the right and the viewer will show different mode of resonance frequency while the frequency is showed in the blank at the bottom right corner → Click the button "Animate Results" → Change the animate type to "Mode shape" in the pop up window → Click "ok" → Click "ok" in the next pop up window → You can change the speed of animation by changing the delay in the animation controller window → Click "Close" to stop the animation → Close the result viewer toolbar to exit the result viewer.

13. Save the file

File Menu → Save as... → Give a name to the file