

The Finite Element Simulations Analysis of Solid Heat Transfer

Xiong Jiang, Yu Zhang

Liaoning Institute of Science And Technology, Benxi 117004, China

Abstract: phenomena of heat conduction transfer was calculated with FEM software Ansys mechanical APDL, which concerns about a tube joint. The second class boundary is to be set as the boundary conditions. Several sizes of mesh grid have been processed. The calculated results of temperature fields and heat flux filed were plot and analyzed.

The mechanical APDL is a powerful language and tool to calculate and simulate the engineering problems prefer to the GUI method. The operate parameters such as mesh size and calculate cycle can be easily changed and manipulated.

Key words: heat conduction transfer; finite element method; mesh convergence; Ansys mechanical APDL

1. Introduction

Heat transfer is an area of thermal engineering the focuses on the transport, exchange, and redistribution of thermal energy. The three modes or ways that heat can be transferred have been termed conduction, convection, and radiation. In this paper, the conduction heat transfer will be studied.

Many kinds of methods were developed to solve this probleb. The most popular tools is called "Finite Element Method". The modern use of finite elements really started in the field of structural engineering. Probably the first attempts were by Hrennikoff (1941) and McHenry (1943), who developed analogies between actual discrete elements, for example bars and beams, and the corresponding portions of a continuous solid.

During the pre-processing phase, the finite element environment is developed from suitable input data. This includes mesh generation, calculation of the element matrices and assembly of the overall system of equations. By far the most complicated aspect of this phase is that of mesh generation. Several sizes of mesh are produced during this paper to determine the final scheme evn though the simlicity of calculated problem.

2. Calculated Methods

Ansys mechanal apdl 15.0 is the major tools applied in this paper. Two sizes of mesh are elabrated. Finish the preprocessor phase with GUI method to determine the units, simulation goemytry, element type, material properties. The boundary condition, results fethching and display are proceded by apdl.

2.1 Problem Described

A cylindrical tank is penetrated radially by a small pipe at a point on its axis remote from the ends of the tank. The inside of the tank is exposed to a fluid of 450°F. The pipe experiences a steady flow of 100°F fluid, and the two flow regimes are isolated from each other by a thin tube. The film coefficient in the tank is a steady 250 Btu/hr-ft²-°F. The film coefficient in the pipe varies with the metal temperature.

The purpose of the example is to determine the temperature distribution at the pipe-tank junction.

2.2 The Analysis Approach

The model in this example uses quarter-symmetry to represent the pipe-tank junction. The tank is assumed to be long enough for its remote end to be held at a constant temperature of 450°F. A similar assumption is made at the Y=0 plane of the tank.

Building the model involves defining two cylinder primitives and a Boolean overlap operation. A mapped (all-brick) mesh is used. The meshing operation produces warnings for a few distorted elements, but you can ignore the warnings because the cited elements are remote from the region of interest (the junction of the pipe and tank).

Because the analysis uses temperature-dependent material properties, the solution requires multiple substeps (50 in this case). Automatic time stepping also is used. After you solve the model, a temperature contour plot and a vector plot of thermal flux enables you to review the results.

3. GUI phase

3.1 Set Measurement Units

It's needed to specify units of measurement for the analysis. For this pipe junction example, measurements use the U. S. Customary system of units (based on inches). To specify this, type the command /UNITS,BIN in the ANSYS Input window and press ENTER.

3.2 Define The Element Type

The example analysis uses a thermal solid element. To define it, do the following:

Choose Main Menu> Preprocessor> Element Type> Add/Edit/Delete. The Element Types dialog box appears.

Click on Add. The Library of Element Types dialog box appears.

In the list on the left, scroll down and pick (highlight) "Thermal Solid." In the list on the right, pick "Brick20node 90."

Click on OK.

Click on Close to close the Element Types dialog box.

3.3 Define Material Properties

To define material properties for the analysis, perform these steps:

Choose Main Menu> Preprocessor> Material Props> Material Models. The Define Material Model Behavior dialog box appears.

In the Material Models Available window, double-click on the following options: Thermal, Density. A dialog box appears.

Enter .285 for DENS (Density), and click on OK. Material Model Number 1 appears in the Material Models Defined window on the left.

In the Material Models Available window, double-click on the following options: Conductivity, Isotropic. A dialog box appears.

Click on the Add Temperature button four times. Four columns are added.

In the T1 through T5 fields, enter the following temperature values: 70, 200, 300, 400, and 500. Select the row of temperatures by dragging the cursor across the text fields. Then copy the temperatures by pressing Ctrl-c.

In the KXX (Thermal Conductivity) fields, enter the following values, in order, for each of the temperatures, then click on OK. Note that to keep the units consistent, each of the given values of KXX must be divided by 12. You can just input the fractions and have ANSYS perform the calculations.

8.35/12

8.90/12

9.35/12

9.80/12

10.23/12

In the Material Models Available window, double-click on Specific Heat. A dialog box appears.

Click on the Add Temperature button four times. Four columns are added.

With the cursor positioned in the T1 field, paste the five temperatures by pressing Ctrl-v.

In the C (Specific Heat) fields, enter the following values, in order, for each of the temperatures, then click on OK.

.113

.117

.119

.122

.125

Choose menu path Material> New Model, then enter 2 for the new Material ID. Click on OK. Material Model Number 2 appears in the Material Models Defined window on the left.

In the Material Models Available window, double-click on Convection or Film Coef. A dialog box appears.

Click on the Add Temperature button four times. Four columns are added.

With the cursor positioned in the T1 field, paste the five temperatures by pressing Ctrl-v.

In the HF (Film Coefficient) fields, enter the following values, in order, for each of the temperatures. To keep the units consistent, each value of HF must be divided by 144. As in step 7, you can input the data as fractions and let ANSYS perform the calculations.

426/144

405/144

352/144

275/144

221/144

Click on the Graph button to view a graph of Film Coefficients vs. temperature, then click on OK.

Choose menu path Material> Exit to remove the Define Material Model Behavior dialog box.

Click on SAVE_DB on the ANSYS Toolbar.

3.4 Define Parameters for Modeling

Choose Utility Menu> Parameters> Scalar Parameters. The Scalar Parameters window appears.

In the window's Selection field, enter the values shown below. (Do not enter the text in parentheses.) Press ENTER after typing in each value. If you make a mistake, simply retype the line containing the error.

RI1=1.3 (Inside radius of the cylindrical tank)

RO1=1.5 (Outside radius of the tank)

Z1=2 (Length of the tank)

RI2=.4 (Inside radius of the pipe)

RO2=.5 (Outside radius of the pipe)

Z2=2 (Length of the pipe)

Click on Close to close the window.

3.5 Create the Tank and Pipe Geometry

Choose Main Menu> Preprocessor> Modeling> Create> Volumes> Cylinder> By Dimensions. The Create Cylinder by Dimensions dialog box appears.

Set the “Outer radius” field to RO1, the “Optional inner radius” field to RI1, the “Z coordinates” fields to 0 and Z1 respectively, and the “Ending angle” field to 90.

Click on OK.

Choose Utility Menu> WorkPlane> Offset WP by Increments. The Offset WP dialog box appears.

Set the “XY, YZ, ZX Angles” field to 0,-90.

Click on OK.

Choose Main Menu> Preprocessor> Modeling> Create> Volumes> Cylinder> By Dimensions. The Create Cylinder by Dimensions dialog box appears.

Set the “Outer radius” field to RO2, the “Optional inner radius” field to RI2, the “Z coordinates” fields to 0 and Z2 respectively. Set the “Starting angle” field to -90 and the “Ending Angle” to 0.

Click on OK.

Choose Utility Menu> WorkPlane> Align WP with> Global Cartesian.

3.6 Overlap the Cylinders

Choose Main Menu> Preprocessor> Modeling> Operate> Booleans> Overlap> Volumes. The Overlap Volumes picking menu appears.

Click on Pick All.

3.7 Review the Resulting Model

Before you continue with the analysis, quickly review your model. To do so, follow these steps:

Choose Utility Menu> PlotCtrls> Numbering. The Plot Numbering Controls dialog box appears.

Click the Volume numbers radio button to On, then click on OK.

Choose Utility Menu> PlotCtrls> View Settings> Viewing Direction. A dialog box appears.

Set the “Coords of view point” fields to (-3,-1,1), then click on OK.

Review the resulting model.

Click on SAVE_DB on the ANSYS Toolbar.

3.8 Trim Off Excess Volumes

In this step, delete the overlapping edges of the tank and the lower portion of the pipe.

Choose Main Menu> Preprocessor> Modeling> Delete> Volume and Below. The Delete Volume and Below picking menu appears.

In the picking menu, type 3,4 and press the ENTER key. Then click on OK in the Delete Volume and Below picking menu.

3.9 Create Component AREMOTE

In this step, you select the areas at the remote Y and Z edges of the tank and save them as a component called AREMOTE. To do so, perform these tasks:

Choose Utility Menu> Select> Entities. The Select Entities dialog box appears.

In the top drop down menu, select Areas. In the second drop down menu, select By Location. Click on the Z Coordinates radio button.

Set the “Min,Max” field to Z1.

Click on Apply.

Click on the Y Coordinates and Also Sele radio buttons.

Set the “Min,Max” field to 0.

Click on OK.

Choose Utility Menu> Select> Comp/Assembly> Create Component. The Create Component dialog box appears.

Set the “Component name” field to AREMOTE. In the “Component is made of” menu, select Areas.

Click on OK.

3.10 Overlay Lines on Top of Areas

Do the following:

Choose Utility Menu> PlotCtrls> Numbering. The Plot Numbering Controls dialog box appears.

Click the Area and Line number radio boxes to On and click on OK.

Choose Utility Menu> Plot> Areas.

Choose Utility Menu> PlotCtrls> Erase Options.

Set “Erase between Plots” radio button to Off.

Choose Utility Menu> Plot> Lines.

Choose Utility Menu> PlotCtrls> Erase Options.

Set “Erase between Plots” radio button to On.

3.11 Concatenate Areas and Lines

In this step, you concatenate areas and lines at the remote edges of the tank for mapped meshing. To do so, follow these steps:

Choose Main Menu> Preprocessor> Meshing> Mesh> Volumes> Mapped> Concatenate> Areas. The Concatenate Areas picking menu appears.

Click on Pick All.

Choose Main Menu> Preprocessor> Meshing> Mesh> Volumes> Mapped> Concatenate> Lines. A picking menu appears.

Pick (click on) lines 12 and 7 (or enter in the picker).

Click on Apply.

Pick lines 10 and 5 (or enter in picker).

Click on OK.

3.12 Set Meshing Density Along Lines

Choose Main Menu> Preprocessor> Meshing> Size Cntrl> ManualSize>Lines> Picked Lines. The Element Size on Picked Lines picking menu appears.

Pick lines 6 and 20 (or enter in the picker) .

Click on OK. The Element Sizes on Picked Lines dialog box appears.

Set the “No. of element divisions” field to 4.

Click on OK.

Choose Main Menu> Preprocessor> Meshing> Size Cntrl> ManualSize> Lines> Picked Lines. A picking menu appears.

Pick line 40 (or enter in the picker).

Click on OK. The Element Sizes on Picked Lines dialog box appears.

Set the “No. of element divisions” field to 6.

Click on OK.

3.13 Mesh the Model

In this sequence of steps, you set the global element size, set mapped meshing, then mesh the volumes.

Choose Utility Menu> Select> Everything.

Choose Main Menu> Preprocessor> Meshing> Size Cntrl> ManualSize> Global> Size. The Global Element Sizes dialog box appears.

Set the “Element edge length” field to 0.4 and click on OK.

Choose Main Menu> Preprocessor> Meshing> Mesher Opts. The Mesher Options dialog box appears.

Set the Mesher Type radio button to Mapped and click on OK. The Set Element Shape dialog box appears.

In the 2-D shape key drop down menu, select Quad and click on OK.

Click on the SAVE_DB button on the Toolbar.

Choose Main Menu> Preprocessor> Meshing> Mesh> Volumes> Mapped> 4 to 6 sided. The Mesh Volumes picking menu appears. Click on Pick All. In the Graphics window, ANSYS builds the meshed model. If a shape testing warning message appears, review it and click Close.

3.14 Turn Off Numbering and Display Elements

Choose Utility Menu> PlotCtrls> Numbering. The Plot Numbering Controls dialog box appears.

Set the Line, Area, and Volume numbering radio buttons to Off.

Click on OK.

3.15 Define the Solution Type and Options

In this step, you tell ANSYS that you want a steady-state solution that uses a program-chosen Newton-Raphson option.

Choose Main Menu> Solution> Analysis Type> New Analysis. The New Analysis dialog box appears.

Click on OK to choose the default analysis type (Steady-state).

Choose Main Menu> Solution> Analysis Type> Analysis Options. The Static or Steady-State dialog box appears.

Click on OK to accept the default (“Program-chosen”) for “Newton-Raphson option.”

3.16 Set Uniform Starting Temperature

In a thermal analysis, set a starting temperature.

Choose Main Menu> Solution> Define Loads> Apply> Thermal> Temperature> Uniform Temp. A dialog box appears.

Enter 450 for “Uniform temperature.” Click on OK.

3.17 Apply Convection Loads

This step applies convection loads to the nodes on the inner surface of the tank.

Choose Utility Menu> WorkPlane> Change Active CS to> Global Cylindrical.

Choose Utility Menu> Select> Entities. The Select Entities dialog box appears.

Select Nodes and By Location, and click on the X Coordinates and From Full radio buttons.

Set the “Min,Max” field to R11 and click on OK.

Choose Main Menu> Solution> Define Loads> Apply> Thermal> Convection> On Nodes. The Apply CONV on Nodes picking menu appears.

Click on Pick All. The Apply CONV on Nodes dialog box appears.

Set the “Film coefficient” field to 250/144.

Set the “Bulk temperature” field to 450.

Click on OK.

3.18 Apply Temperature Constraints to AREMOTE Component

Choose Utility Menu> Select> Comp/Assembly> Select Comp/Assembly. A dialog box appears.

Click on OK to select component AREMOTE.

Choose Utility Menu> Select> Entities. The Select Entities dialog box appears.

Select Nodes and Attached To, and click on the Areas,All radio button. Click on OK.

Choose Main Menu> Solution> Define Loads> Apply> Thermal> Temperature> On Nodes. The Apply TEMP on Nodes picking menu appears.

Click on Pick All. A dialog box appears.

Set the “Load TEMP value” field to 450.

Click on OK.

Click on SAVE_DB on the ANSYS Toolbar.

3.19 Apply Temperature-Dependent Convection

In this step, apply a temperature-dependent convection load on the inner surface of the pipe.

Choose Utility Menu> WorkPlane> Offset WP by Increments. A dialog box appears.

Set the “XY,YZ,ZX Angles” field to 0,-90, then click on OK.

Choose Utility Menu> WorkPlane> Local Coordinate Systems> Create Local CS> At WP Origin. The Create Local CS at WP Origin dialog box appears.

On the “Type of coordinate system” menu, select “Cylindrical 1” and click on OK.

Choose Utility Menu> Select> Entities. The Select Entities dialog box appears.

Select Nodes, and By Location, and click on the X Coordinates radio button.

Set the “Min,Max” field to RI2.

Click on OK.

Choose Main Menu> Solution> Define Loads> Apply> Thermal> Convection> On Nodes. The Apply CONV on Nodes picking menu appears.

Click on Pick All. A dialog box appears.

Set the “Film coefficient” field to -2.

Set the “Bulk temperature” field to 100.

Click on OK.

Choose Utility Menu> Select> Everything.

Choose Utility Menu> PlotCtrls> Symbols. The Symbols dialog box appears.

On the “Show pres and convect as” menu, select Arrows, then click on OK.

Choose Utility Menu> Plot> Nodes. The display in the Graphics Window changes to show you a plot of nodes.

3.20 Reset the Working Plane and Coordinates

To reset the working plane and default Cartesian coordinate system, choose Utility Menu> WorkPlane> Change Active CS to> Global Cartesian.

Choose Utility Menu> WorkPlane> Align WP With> Global Cartesian.

4. Results

4.1 The Mesh

As shown in fig.2.

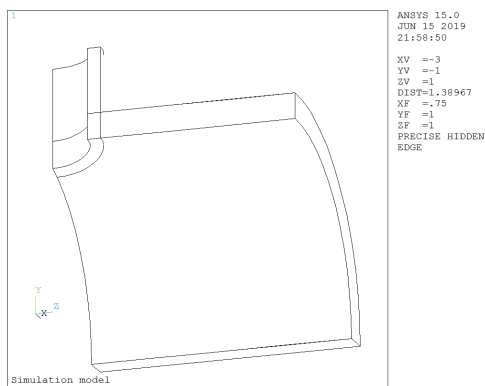


Fig.1 the geometry model

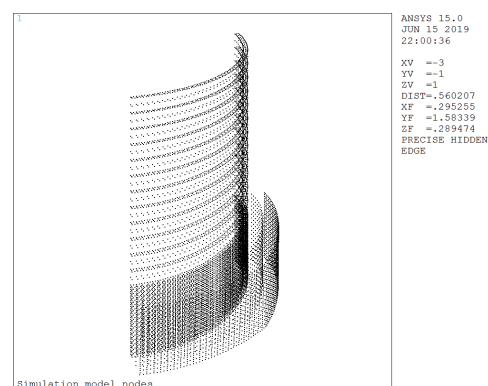


Fig.2 the final mesh

4.2 The Results of Two Different Mesh Sizes

Death is the end of the monster, when the death instinct turned to the outside world, the monster will project “death instinct” on the creator and the creator’ family. And when the monster project “death instinct” on himself, he will blame and resent himself, and finally, resulting to suicide.

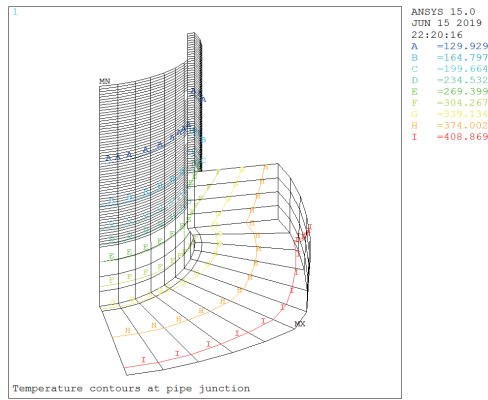


Fig.3 temperature contour of sparse mesh

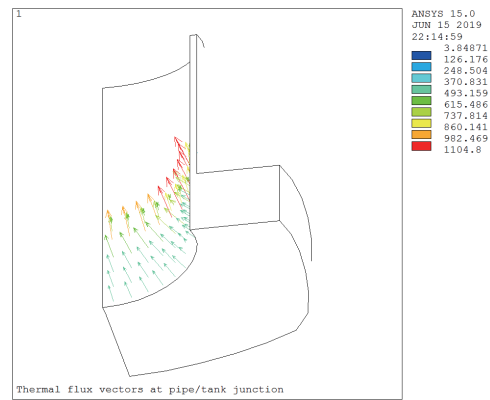


Fig.4 heat flux vector of sparse mesh

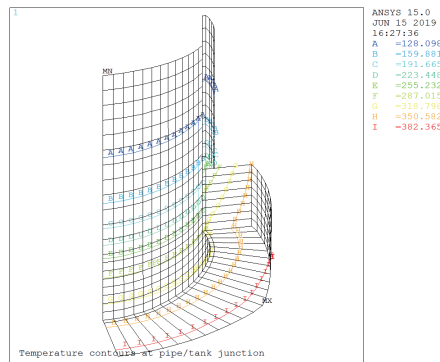


Fig.5 temperature contour of denser mesh

5. The Results

A solid conduction was simulated by Ansys 15.0 compound GUI and apdl methods. The results of Fig.5 and Fig.6 shows mesh size don't affect the analysis, though a little additional time is cost.

The results tell that both the temperature contour and heat flux reach a peak at the tube joint, which should be considered during the design of tube joint.

References

- [1] Brenner, S. C. and Scott, L. R. (1994). The Mathematical Theory of Finite Element Methods. Springer.
- [2] Zienkiewicz, O. C., Taylor, R. L. and Zhu, J. Z. (2005). The Finite Element Method: Its Basis and Fundamentals (6th edn). Elsevier.
- [3] Xiaolin Chen, Yijun Liu, (2015). Finite Element Simulations with Ansys Workbench, SDC.
- [4] Heui-Huang Lee, SDC(2015). Finite Element Simulations with Ansys Workbench 12 . SDC.
- [5] George R. Buchanan(1995). Schaum's Outline of Theory and Problems of Finite Element Analysis. McGraw Hill Companies, Inc.
- [6] Ansys inc.(2013). Ansys 15.0 Workbench Tutorial.

This article is a research fee project of the Department of Education of Liaoning Province (No. LJKZ1069) And Design the project of Liaoning Provincial Education Department based on the intelligent data failure analysis system of mining machinery LJKQZ20222339

Jiang Xiong, male (1968.09-) Manchu, associate professor of Liaoning Institute of Science and Technology. Research direction: Advanced Mathematics Teaching and mathematical modeling

Zhang yu, male (1985.07-) Han nationality, doctor degree, lecturer at Liaoning Institute of Science and Technology. Research direction: advanced numbers Teaching and Mathematical Modeling