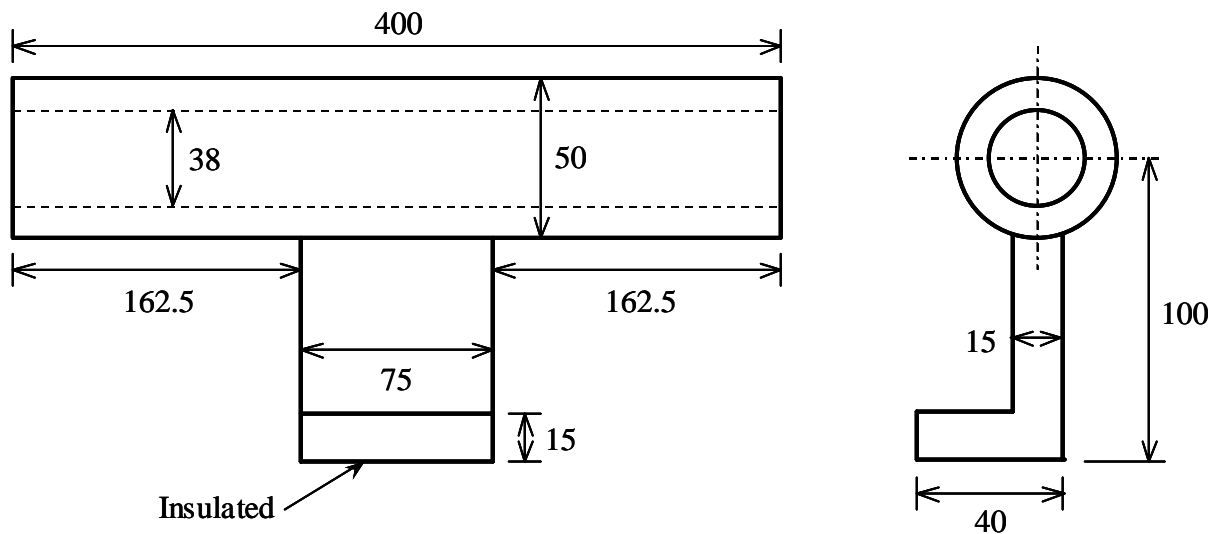


ANSYS Example: Transient Thermal Analysis of a Pipe Support Bracket

The section of pipe shown below is a representative section of a longer pipe carrying a hot fluid under pressure. The pipe is supported every 400 mm by a bracket that is welded to the pipe and subsequently attached to the wall. The pipe and bracket are made of 2024-T6 aluminum. The pipe has an outer and inner diameter of 50 mm and 38 mm, respectively. The bracket is attached to the wall with an insulating pad along the base, so there is no heat transfer between the wall and bracket. All dimensions in the figure below are given in mm.

The air temperature surrounding the pipe and bracket is 300 K, and the heat transfer (film) coefficient between the air and pipe/bracket is $h = 200 \text{ W}/(\text{m}^2\text{-K})$. The pipe and bracket also have a uniform initial temperature of 300 K. Fluid at a temperature of 450 K then begins flowing through the pipe, and the heat transfer (film) coefficient between the pipe and fluid is $h = 1200 \text{ W}/(\text{m}^2\text{-K})$.

In this example, ANSYS will be used to perform a transient thermal analysis on the pipe and bracket. The temperature in the pipe and bracket will be examined over a period of 20 seconds after the fluid begins flowing through the pipe. A contour plot of the temperature distribution can be generated at any point in time, and temperature vs. time plots can be generated for any node. Animation can also be used to display the temperature distribution in the entire part as a function of time. The analysis will be performed using 10 node, 3D thermal elements (SOLID 87).



The following thermal properties of 2024-T6 aluminum are required for the analysis.

Temperature (K)	100	200	300	400	600
Thermal Conductivity, k (W/m-K)	65	163	177	186	186
Specific Heat, c (J/kg-K)	473	787	875	925	1042

The density of the aluminum alloy is $2770 \text{ kg}/\text{m}^3$, which is constant within the temperature range considered here.

ANSYS Analysis:

Start ANSYS Product Launcher, set the Working Directory to C:\temp, define Job Name as 'Pipe_Bracket', and click Run. Then define Title and Preferences.

Utility Menu → File → Change Title... → Enter 'Transient Thermal Analysis of a Pipe Bracket' → OK

ANSYS Main Menu → Preferences → Preferences for GUI Filtering → Select 'Thermal' and 'h-method' → OK

Enter the Preprocessor to define the model geometry:

Define Element Type and Material Properties. Since many of the Material Properties are temperature-dependent, we must specify the temperature units and define the properties at several temperatures.

ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete → Add... → Thermal Solid Tet 10 node 87 (define 'Element type reference number' as 1) → OK → Close

ANSYS Main Menu → Preprocessor → Material Props → Temperature Units → Kelvin or Rankin

ANSYS Main Menu → Preprocessor → Material Props → Material Models → Double Click Thermal → Conductivity → Isotropic → Add Temperature → Enter 100 for T1, 65 for KXX → Enter 200 for T2, 163 for KXX → Add Temperature → Enter 300 for T3, 177 for KXX → Add Temperature → Enter 400 for T4, 186 for KXX → Add Temperature → Enter 600 for T5, 186 for KXX → OK → Double Click Specific Heat → Repeat the process with values of 473, 787, 875, 925, and 1042 for C at each Temperature → OK → Double Click Density → Enter 2770 for DENS without specifying a temperature (density is constant) → OK → Click Exit (under 'Material')

Begin creating the geometry by defining a Hollow Cylinder (Volume) for the pipe.

ANSYS Main Menu → Preprocessor → Modeling → Create → Volumes → Cylinder → Hollow Cylinder → Enter 0 for WP X, 0 for WP Y, 0.019 for Rad-1, 0.025 for Rad-2 and -0.4 for Depth → OK

Change to an Isotropic View using the Plot Menu.

The support bracket will be created by defining two Rectangles (Areas), deleting the Area that overlaps the Cylinder, and then extruding the Areas into a Volume. First, the WorkPlane must be moved.

Utility Menu → WorkPlane → Offset WP to XYZ Locations + → Type '0, 0, -0.1625' in the Command Line of the 'Offset WP' window (Global Cartesian coordinates) → OK

ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Rectangle → By Dimensions → Enter 0.0075 and -0.0075 for X1 and X2, and -0.02 and -0.085 for Y1 and Y2, respectively → Apply → Enter 0.0075 and -0.0325 for X1 and X2, and -0.085 and -0.1 for Y1 and Y2, respectively → OK

ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Divide → Area by Area → Select (with the mouse) the rectangular Area to be divided → OK → Select the outer Area of the Cylinder → OK

ANSYS Main Menu → Preprocessor → Modeling → Delete → Area and Below → Select the remaining Area to be deleted → OK

Utility Menu → Plot → Replot

ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Add → Areas → Select (with the mouse) the two rectangular Areas → OK

ANSYS Main Menu → Preprocessor → Modeling → Operate → Extrude → Areas → By XYZ Offset → Select the Area that defines the bracket → OK → Enter -0.075 for DZ → OK

Utility Menu → Plot → Replot

ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Add → Volumes → Select 'Pick All'

Utility Menu → Plot → Replot

The component will now be Free Meshed with Tetrahedral Elements using a Global Size (Element edge length) of 6 mm. Then save the Database.

ANSYS Main Menu → Preprocessor → Meshing → MeshTool → Under 'Size Controls: Global' click Set → Enter 0.006 for 'Element edge length' → OK → Under 'Mesh:' select Volumes, Tet and Free → Click Mesh → Select (with the mouse) the Volume → OK

ANSYS Toolbar → SAVE_DB

Enter the Solution Menu to define boundary conditions and loads and run the analysis:

ANSYS Main Menu → Solution → Analysis Type → New Analysis → Select Transient → OK → Select Full for 'Solution method' → OK

The initial temperature of the pipe/bracket must first be defined.

ANSYS Main Menu → Solution → Define Loads → Apply → Initial Condition → Define → Select 'Pick All' → Select Temp for 'DOF to be specified' → Enter 300 for VALUE → OK

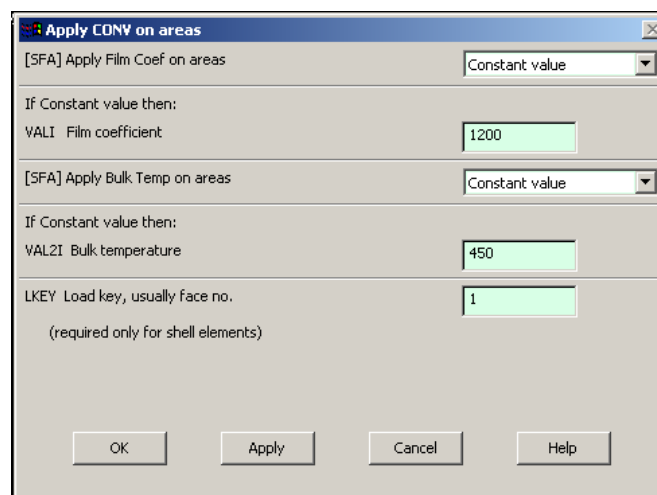
Now apply the Thermal Loads (Convections) to the pipe and bracket. The Film Coefficient on the inside of the pipe is 1200 (fluid temperature is 450), and the Film Coefficient on the outside of the pipe is 200 (air temperature is 300). Be careful to select the correct Areas when applying the Loads. It may be helpful to list the Areas to determine which Lines define each Area.

Utility Menu → Plot → Areas

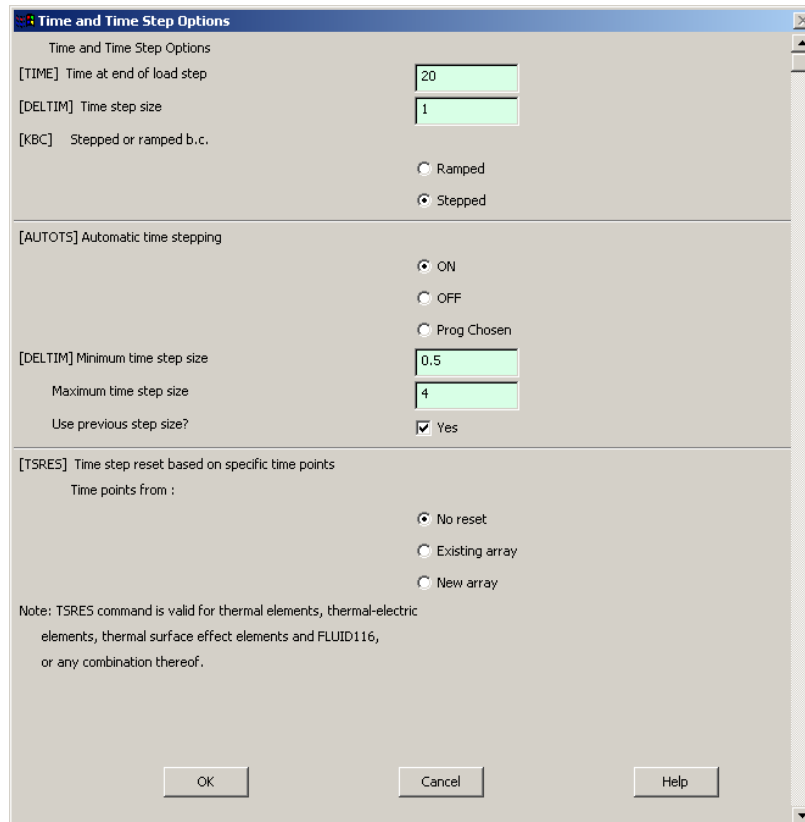
Utility Menu → PlotCtrls → Numbering... → Click Area numbers and Line numbers On → OK

ANSYS Main Menu → Solution → Define Loads → Apply → Thermal → Convection → On Areas → Select (with the mouse) the Areas defining the inside of the pipe (A5 and A6 in this case) → OK → Enter 1200 for 'Film coefficient' and 450 for 'Bulk temperature' → OK

ANSYS Main Menu → Solution → Define Loads → Apply → Thermal → Convection → On Areas → Select the outside Areas, except the Area which is insulated (A3, A16, A1, A2, A7, A13, A14, A8, A9, A10, and A11) → OK → Enter 200 for 'Film coefficient' and 300 for 'Bulk temperature' → OK



Since the problem is nonlinear (in time and temperature), several options must be defined for the nonlinear solver. Specifically, Time Step Options must be defined. The transient solution will be performed for 20 seconds, using an initial Time Step Size of 1 second. Since the temperatures are applied instantly (rather than gradually), they will be applied as Stepped, rather than Ramped Boundary Conditions. Automatic Time Stepping will be activated, as this may reduce the Solution time. Minimum and Maximum Time Step Sizes of 0.5 and 4 seconds will be defined. **ANSYS Main Menu → Solution → Load Step Opts → Time/Frequency → Time – Time Step →** Enter 20 for TIME, 1 for DELTIM, select ‘Stepped’ for KBC, click ‘Automatic time stepping’ ON, enter 0.5 for ‘Minimum time step size’ and 4 for ‘Maximum time step size’ (if the temperature distribution for every second is needed then this number will be set to 1) → OK



Define the frequency with which results will be written to the Database and Results File. Then save the Database and initiate the Solution.

ANSYS Main Menu → Solution → Load Step Opts → Output Ctrl → DB/Results File → Select ‘All items’ to be controlled, and select ‘Every substep’ for FREQ (File write frequency) → OK

ANSYS Toolbar → SAVE_DB

ANSYS Main Menu → Solution → Solve → Current LS → OK → Close the information window when solution is done → Close the /STATUS Command window

Enter the General Postprocessor to examine the results:

A Contour Plot of the nodal temperatures at any substep (time increment) can be generated by reading the appropriate set of results from the Results File.

ANSYS Main Menu → General Postproc → Results Summary → Close

ANSYS Main Menu → General Postproc → Read Results → First Set

ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solution
→ Select 'DOF Solution' and 'Nodal Temperature' → OK

ANSYS Main Menu → General Postproc → Read Results → Next Set

ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solution
→ Select 'DOF Solution' and 'Nodal Temperature' → OK

*** *This procedure can be repeated until all desired substeps have been viewed.* ***

The temperature of a particular node can be viewed on the plot, or the nodal temperatures can be listed and saved to a file for further analysis.

ANSYS Main Menu → General Postproc → Query Results → Subgrid Solu → Select (with the mouse) various Nodes to see the temperatures

ANSYS Main Menu → General Postproc → List Results → Nodal Solution → Select 'DOF Solution' and 'Nodal Temperature' → OK

The transient temperature distribution can also be Animated over the 20 second time period.

Utility Menu → PlotCtrls → Animate → Over Time... → Set 'Number of animation frames' to the desired value (20 used here), select 'Current Load Stp' (the entire transient solution solved in this example comprises one Load Step), set the 'Animation time delay' to the desired value (0.5 s used here) with 'Auto contour scaling' On, and select 'DOF solution' and 'Temperature TEMP' → OK

A Temperature vs. Time Plot for any node can be generated within the Time History Postprocessor using the following procedure.

ANSYS Main Menu → TimeHist Postpro → Define Variables → Add → Select 'Nodal DOF result' → OK → Select (with the mouse) the desired node → OK → Click OK to close window → Close

ANSYS Main Menu → Time Hist Postpro → Graph Variables → Enter 2 (since the selected node is set 2) for NVAR1 (1st variable to graph) → OK

The analysis should be rerun with a finer mesh to check for convergence of the solution. The procedure will be the same, but a smaller Global Element Size will be defined. If the results (nodal temperatures) using the two meshes are in close agreement, the model can be considered to have small discretization error.