STEVENS
Institute of Technology

ME345 Modeling and Simulation - Professor Frank Fisher
3D Thermal Analysis, Finned Pipe
Last Updated October 2008 – Allen Umali

Introduction:
In this example you will learn to assess 3D geometries in heat transfer by modeling an object subjected to convection and temperature boundary conditions. ANSYS will allow you to generate a plot of the nodal temperature and heat flux distributions along the geometry.

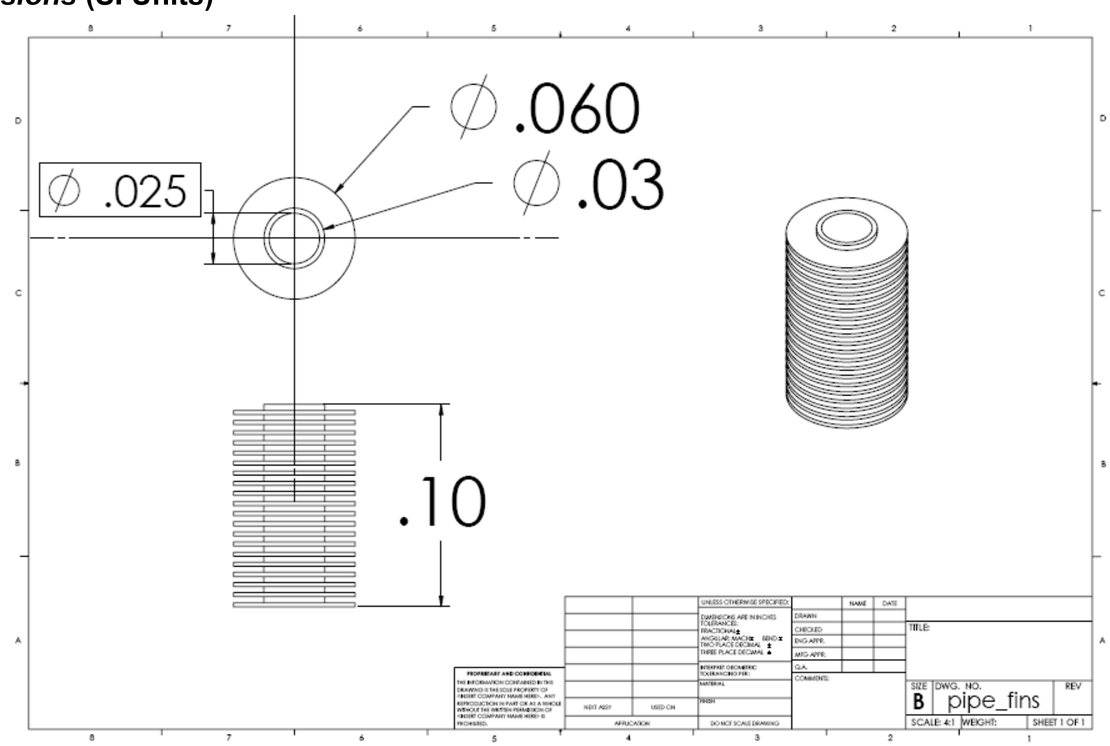
Problem Statement
Example 3-12 from *Heat and Mass Transfer – A Practical Approach 3rd Edition*
Steam in a heating system flows through tubes whose outer diameter is D_1 = 3 cm and whose walls are maintained at a temperature of 120 ^oC. Circular aluminum alloy fins (k=180 W/m-^oC) of outer diameter D_2 = 6cm and constant thickness t = 2mm are attached to the tube.. The space between the fins is 3mm, and thus there are 200 fins per meter length of the tube. Heat is transferred to the surrounding air at T_∞ = 25 ^oC, with a combined heat transfer coefficient of h = 60 W/m^2-^oC. Determine the heat transfer from the tube per meter of length.

Objective Circular aluminum alloy fins line the length of the pipe. For simplicity, the heat transfer from the tubes per 10 cm of pipe is to be determined. A contour plot of the nodal heat flux distribution will be generated.

Assumptions **1** Steady-state analysis **2** The heat transfer coefficient is uniform over the entire fin surface **3** Thermal conductivity is constant **4** Heat transfer by radiation is negligible

Boundary Conditions **1** All areas of the pipe/fin combination have convective boundary layers **2** The temperature at the base of the pipe is maintained at 120 ^oC.

Dimensions **(SI Units)**

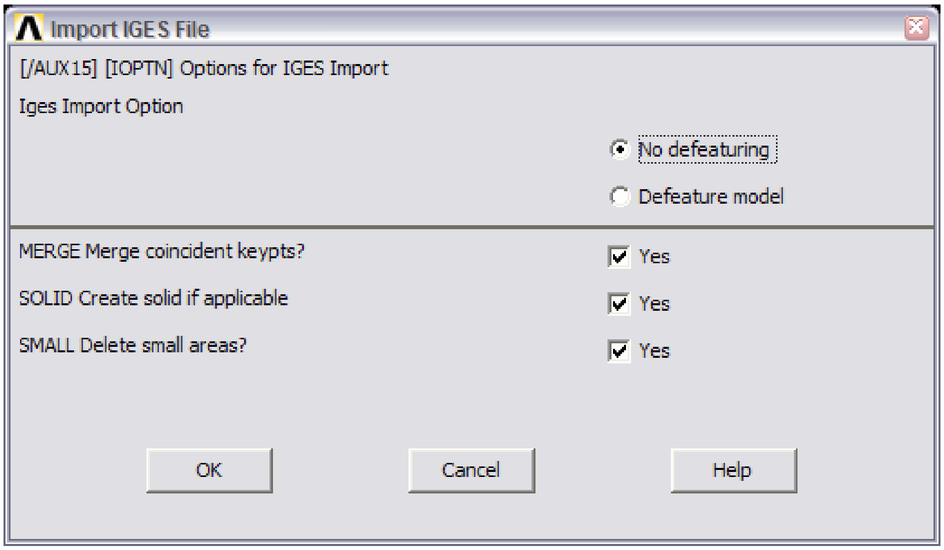


STEVENS
Institute of Technology

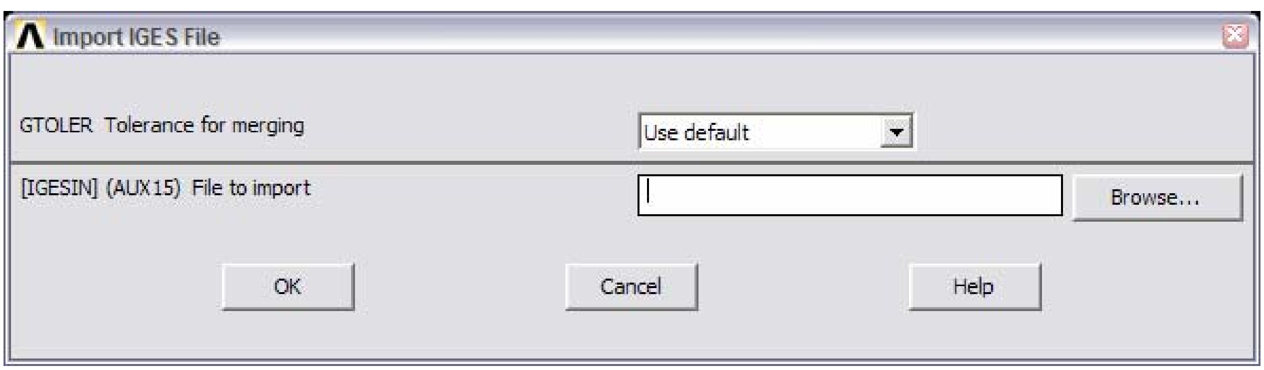
Starting ANSYS - Click on ANSYS 11.0 in the Programs menu.

Modeling

- From the ANSYS Utility Menu, click on **File > Import > IGES.** The following window appears:



- Click **OK.** The following window appears:

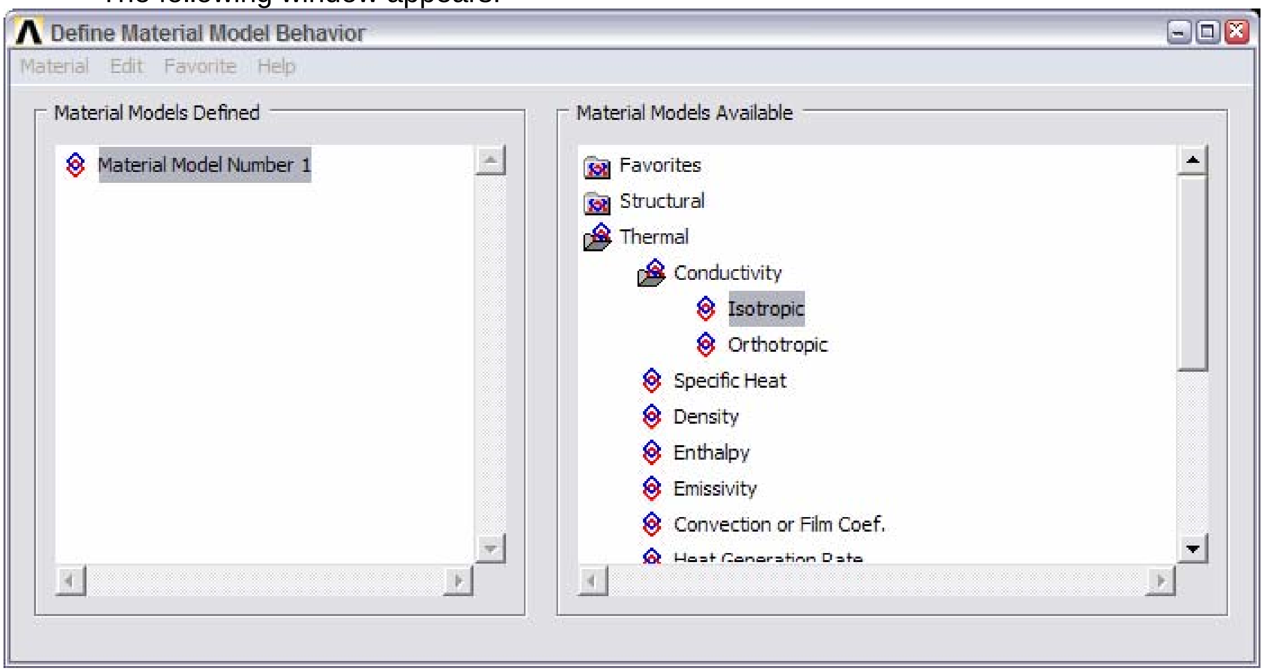


- Click on **Browse** and choose the location of the downloaded **pipe_fin.igs.** An isometric view of the pipe/fin part should come up. The modeling portion is now complete.

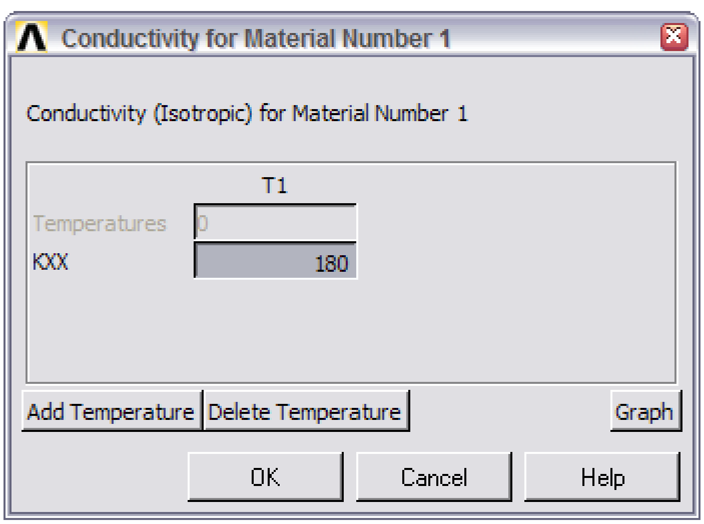
STEVENS
Institute of Technology

Material Properties - With the modeling completed, the material model behavior must be defined. ANSYS can then understand how heat travels through the composite solid (i.e. thermal conductivity of material).

- In the ANSYS Main Menu, select **Preprocessor > Material Props > Material Models.** The following window appears:



- For Material Model Number 1, click on **Thermal > Conductivity > Isotropic**. The following window will prompt you for the thermal conductivity.

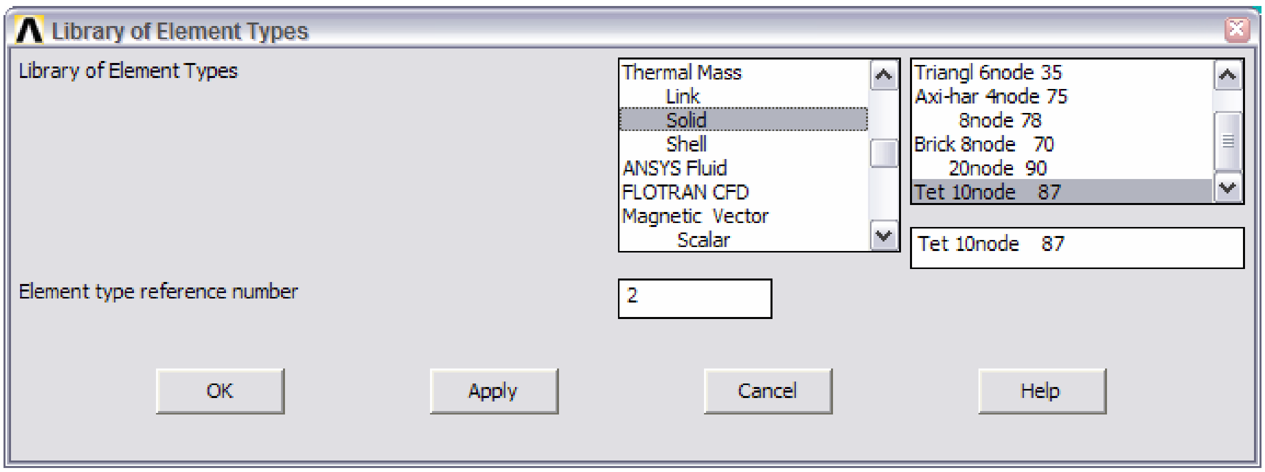


- Enter the thermal conductivity for aluminum alloy, **k = 180 W/m-^{o}C.** Material Model Number 1 now has an isotropic thermal conductivity of aluminum alloy.
- Exit the Material Model window.

STEVENS
Institute of Technology

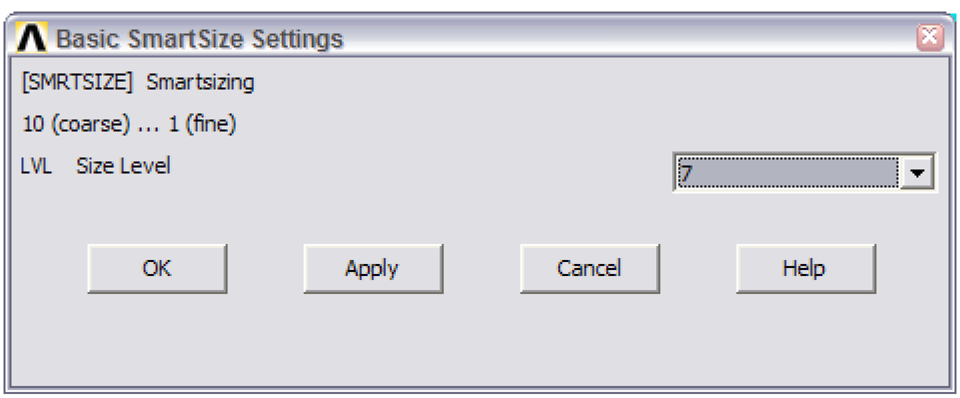
Element Properties

- The next step is to define the element type. From the ANSYS Utility Menu click on **Preprocessor > Element Type > Add/Edit/Delete.** In the Element Types window that appears click on **Add**.
- In the Library of Element Types window choose **Thermal Mass > Solid** in the left window and select **Tet 10node 87** in the right window. Click **OK**. Close the Element Types window.



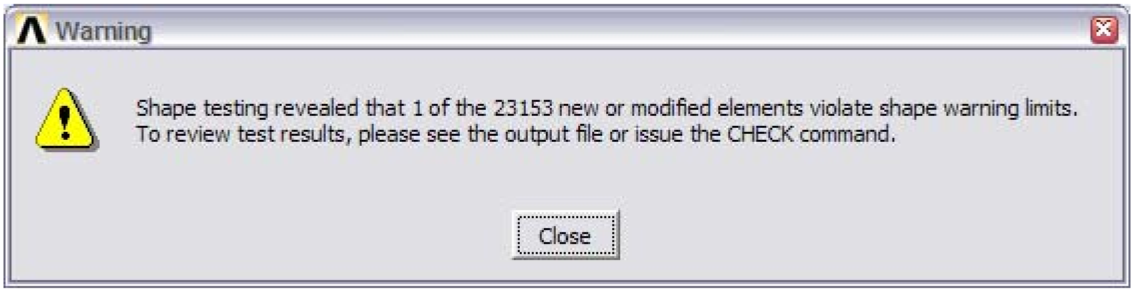
Meshing – This next section is responsible for defining the size of the elements, and ultimately the degree of accuracy of the solution.

- In the ANSYS Main Menu click on **Meshing > Mesh Attributes > Default Attributes**, and confirm that the **Element type number is 1 SOLID 87** and that the **Material number is 1**. Click **OK**.
- Next we define the element size. In the ANSYS Main Menu, click on **Meshing > Size Controls > Smart Size > Basic**. The following window appears:

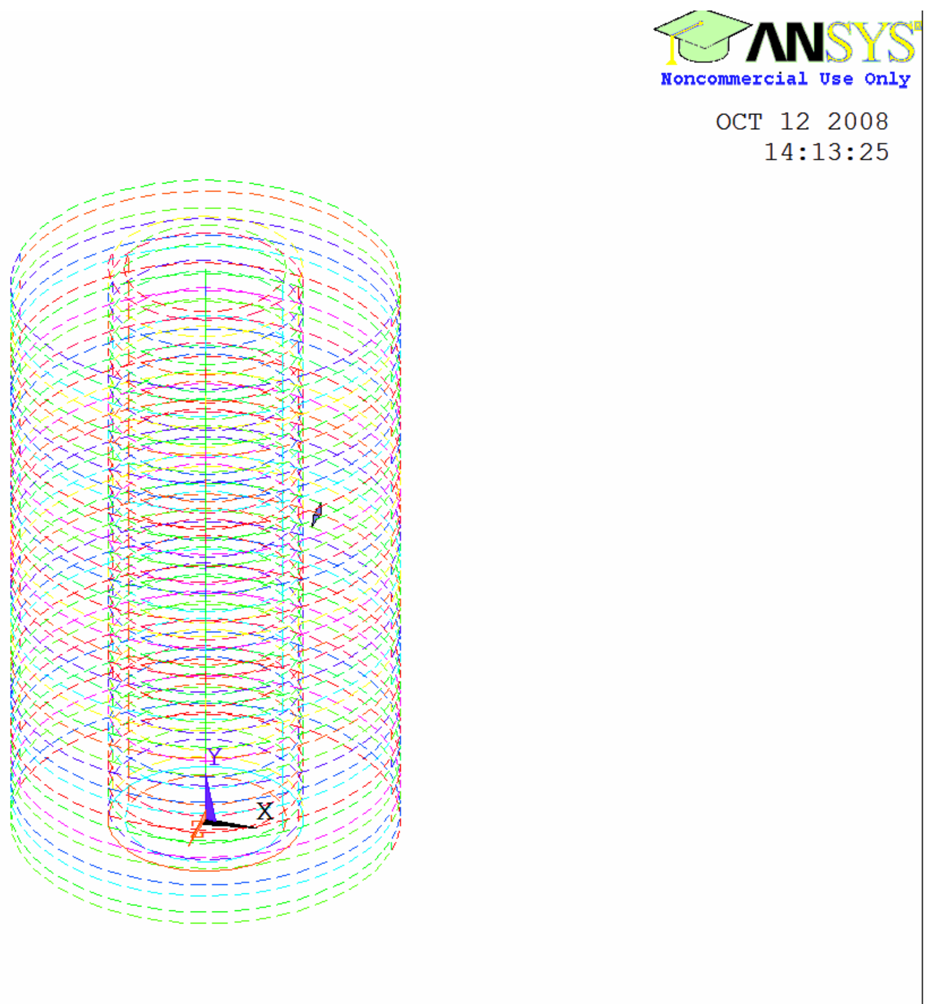


- Choose a **size level of 7** and click **OK**.

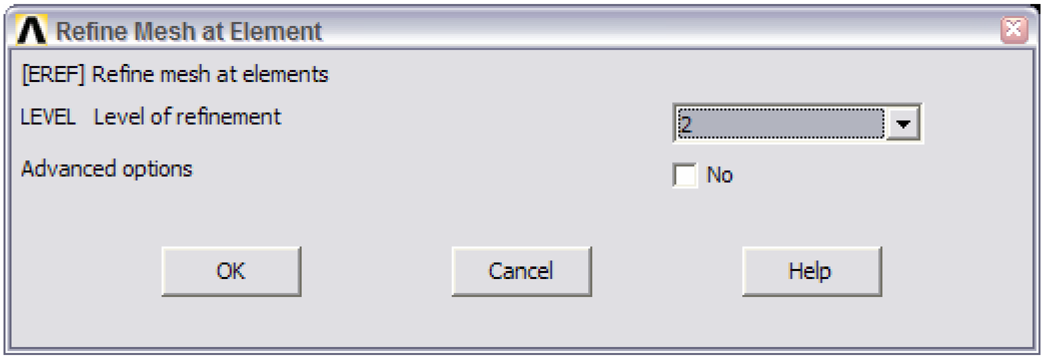
- Next click on **Meshing > Mesh > Volumes > Free** and click anywhere on the pipe/fin volume. Click **OK**. An error warning should occur.



  - If the warning does occur, we need to refine our mesh. In order to do this we must plot the elements that violate shape limits. Click on **Meshing > Check Mesh > Individual Elm > Plot/Error Elements**.
  - Click **OK** on the next window that comes up. **ANSYS should plot the volume using lines to reveal the error element.**



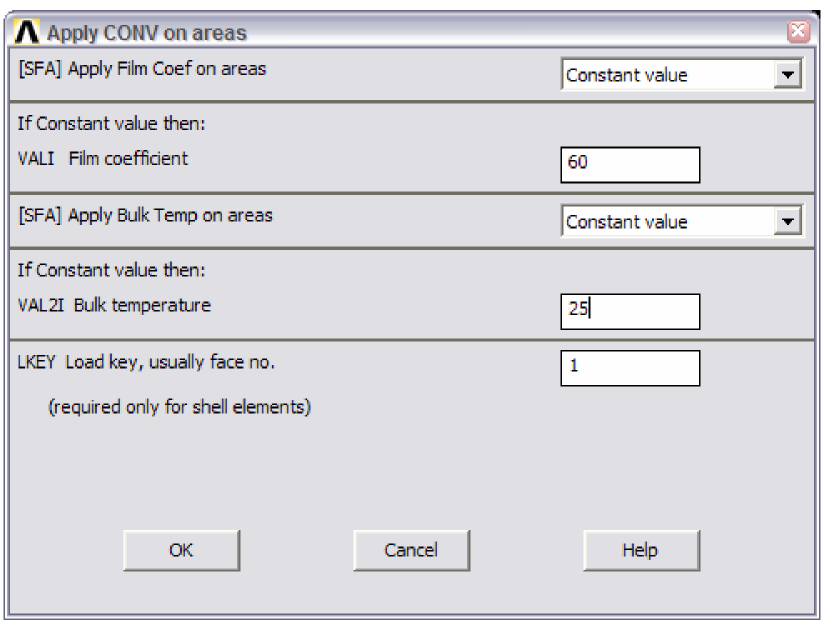
- To refine the error element, click on **Meshing > Mesh Tool**. On the Mesh Tool window that appears, click **Refine**. **Select the error element** by clicking on it and click **OK**. The Refine Mesh at Element window appears.



- **Select 2 for the level of refinement** and click **OK**. The volume should now be completely meshed with no error elements. Close the Mesh Tool.

Loading - The next step is to define the loads or boundary conditions acting on the volume.

- First we define the convection on the pipe area. Click on **Preprocessor > Loads > Define Loads > Apply > Thermal > Convection > On Areas**.
- On the **Apply CONV on Areas** window select **Pick All**.
- The film coefficient (convective heat transfer coefficient) was given as a constant **h = 60 W/m^2-^{o}C**. **Enter 60 for the Film coefficient**.
- The surrounding temperature is given earlier as **T_∞ = 25 ^{o}C**. **Enter 25 for the Bulk temperature**.
- Click **OK**.



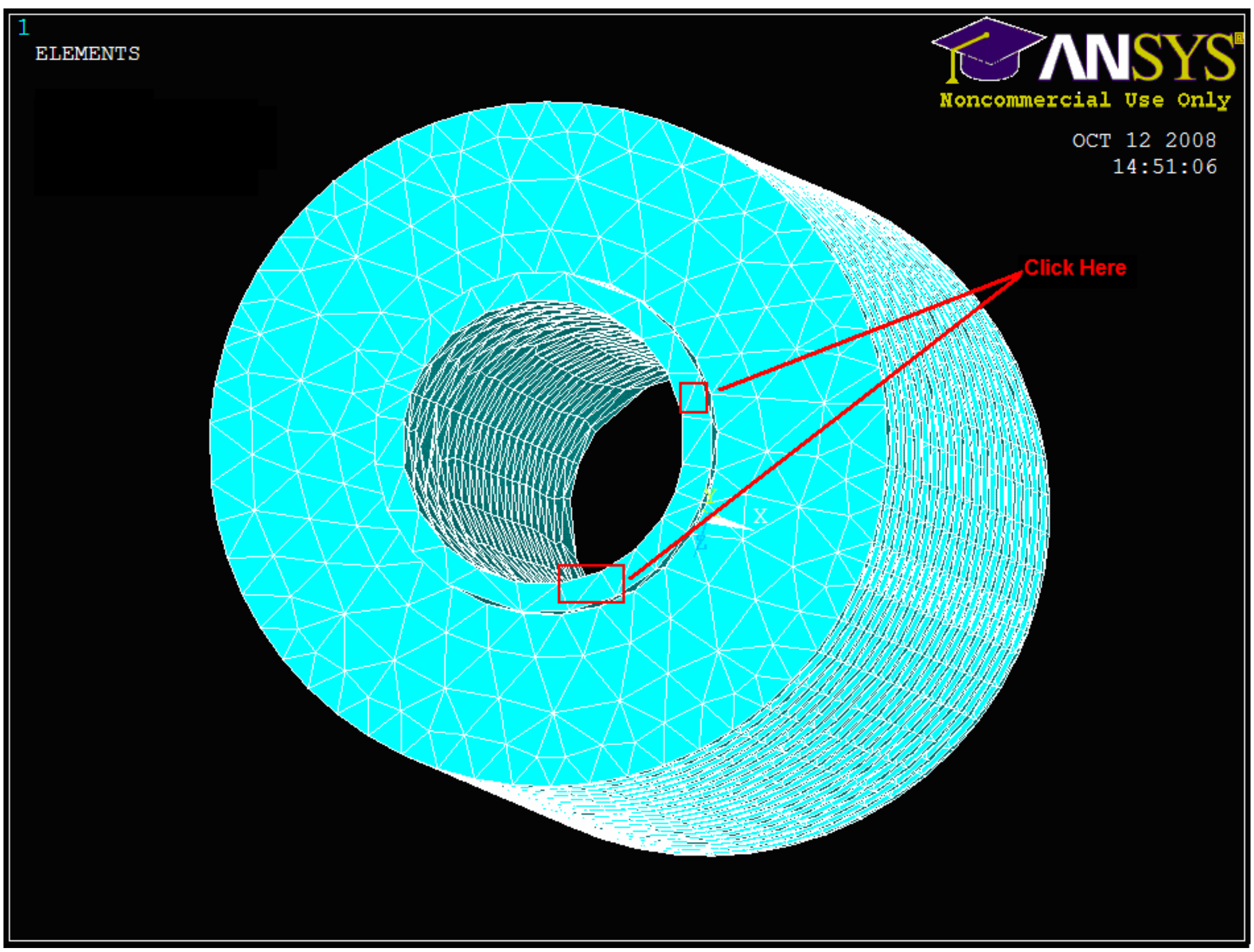
STEVENS
Institute of Technology

The next step is to apply the given base temperature to the insides of the pipe.

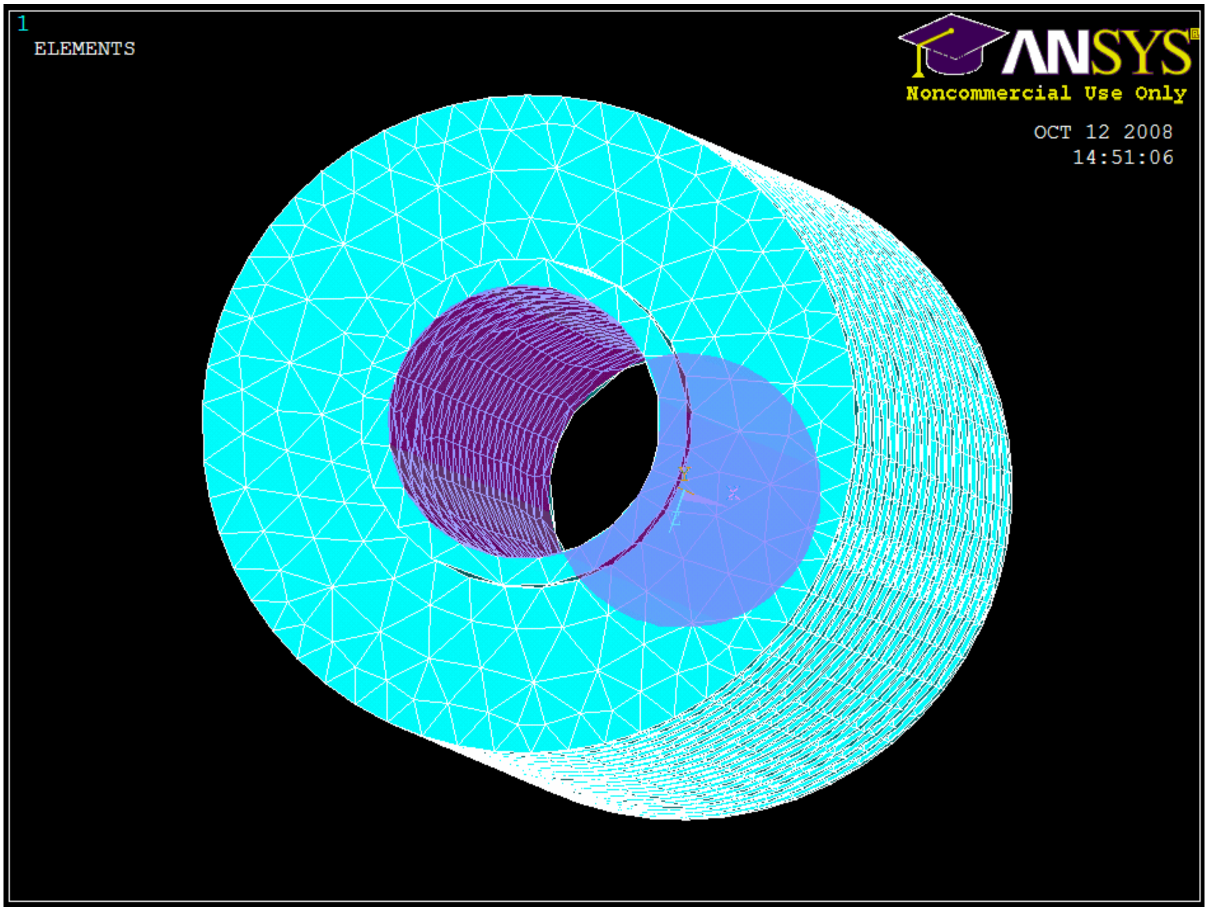
- **NOTE: Selecting the inside of the pipe may be a difficult task. It may be easier to change the view of the pipe as shown, and select in the vicinity of the red boxes shown**.

- First Click on the **Dynamic Model Mode** button  in the lower right hand corner of the ANSYS window**.**
- **Press and hold the right mouse button to rotate the pipe**. When finished **click on the Dynamic Model Mode button again to deactivate.**
- Click on **Preprocessor > Loads > Define Loads > Apply > Thermal > Temperature > On Areas**.

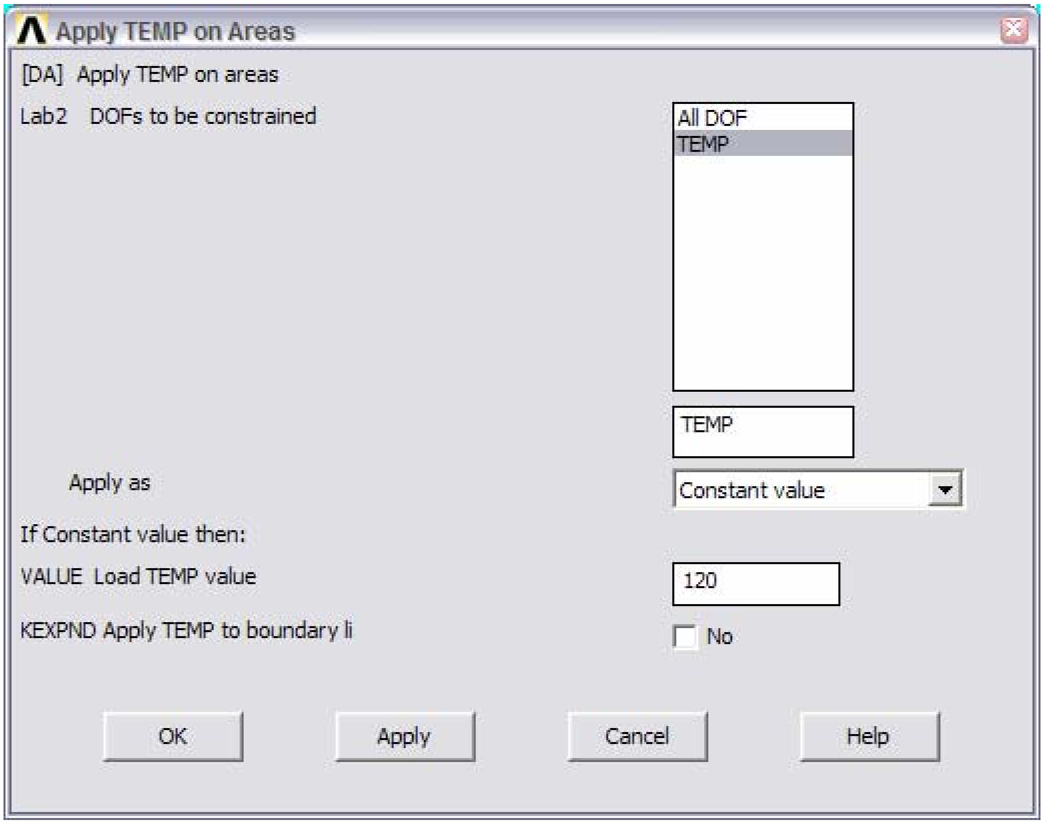
- NOTE: If an undesired area is selected, you can **toggle between "picking" and "unpicking" by clicking the right mouse button.**
- To select the inside of the pipe, **click in the vicinity of the red boxes shown:**



- When selected, the inside of the pipe should be highlighted purple as shown. Click **OK**.



- In the following window that appears, click on TEMP and insert a **Load Temp value of 120 °C**.



- Click **OK**.
- The loading of the pipe/fin is now complete.

Solution – The final step is to run the solution.

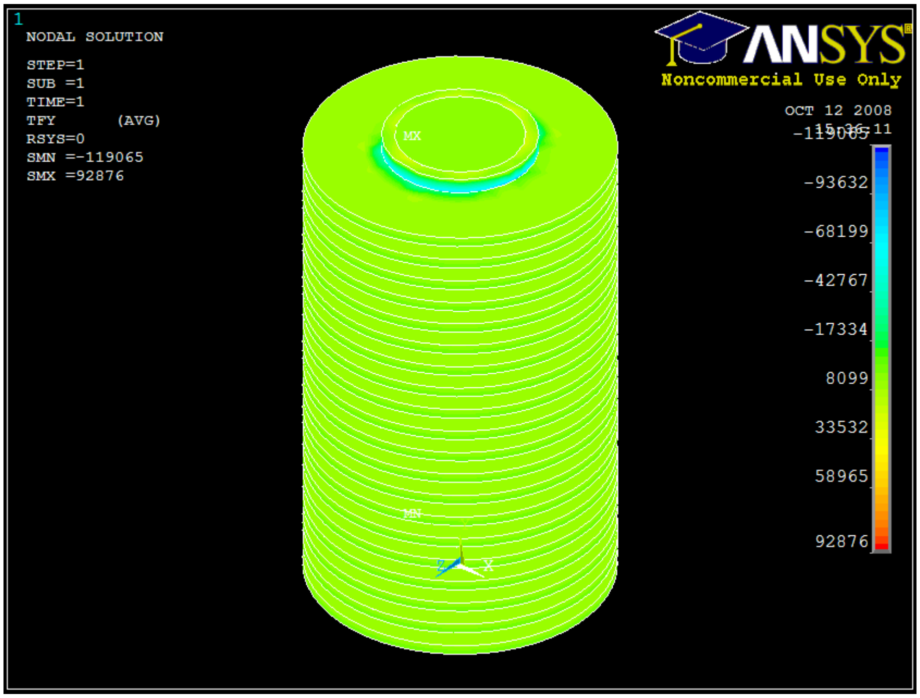
- Click on **Preprocessor > Solution > Analysis Type > New Analysis**. In the following window be sure that **Steady-State** is selected. Click **OK**.

- Click on **Preprocessor > Solution > Solve > Current LS** and click **OK**. Give ANSYS a few seconds to solve. The following window will appear.

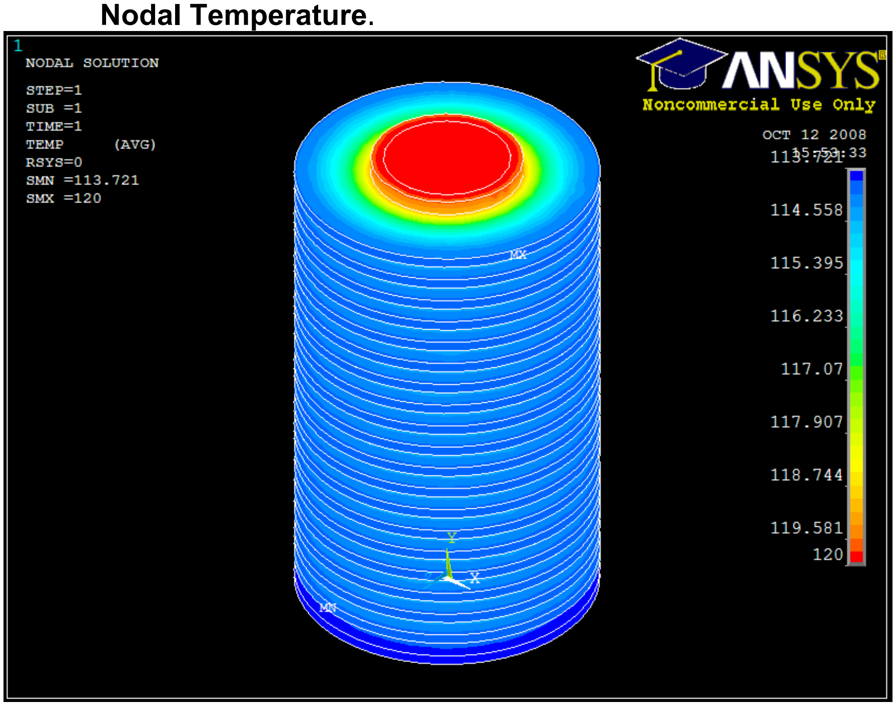
STEVENS
Institute of Technology

General Post-Processing

- To view the solution, we can generate a contour plot of the **heat flux**.
- Click on **General Postproc > Plot Results > Contour Plot > Nodal Solution > Thermal Flux > Y-Component of thermal flux.**



- If the **temperature distribution** was required, ANSYS can also perform this by clicking **General Postproc > Plot Results > Contour Plot > Nodal Solution > DOF Solution > Nodal Temperature.**



Stevens Institute of Technology
ME345 Modeling and Simulation
Professor Frank Fisher
ANSYS 3D Thermal Analysis
Written by Allen Umali

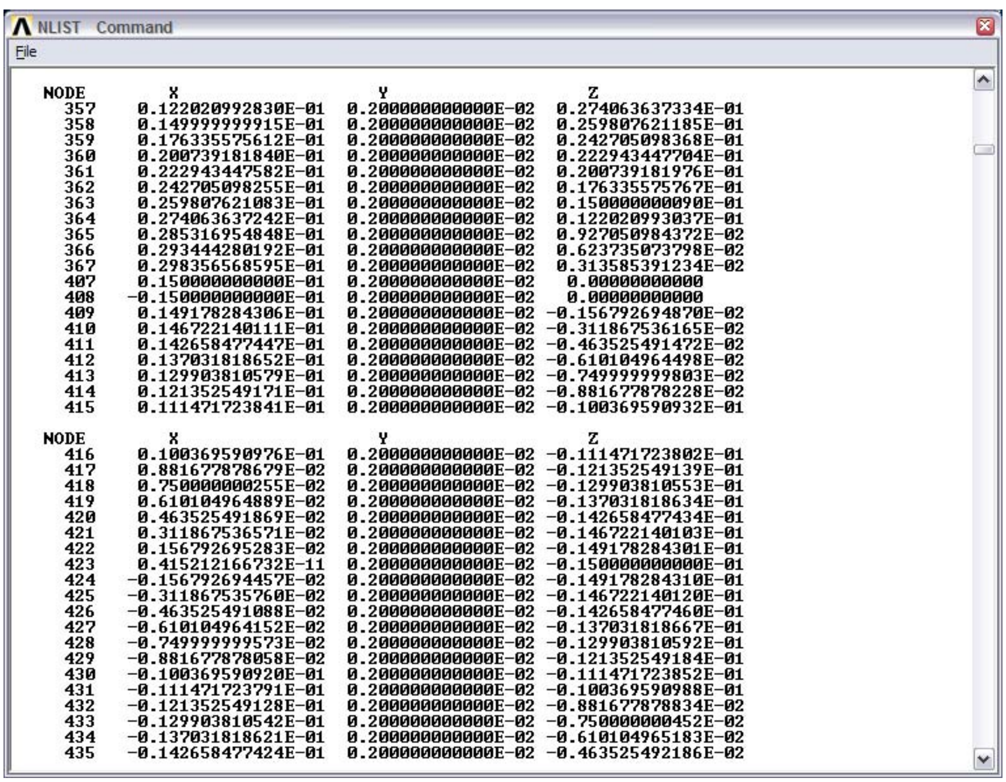
Example 3-12 from <u>Heat and Mass Transfer: A Practical Approach 3rd Ed.</u> *Yunus A. Cengel*

Does ANSYS generate a reasonable solution?

The second part of the exercise involves the comparison of ANSYS results to the hand calculations; our objective is to determine whether or not the ANSYS results are reasonable. The problem statement asks us to solve for the heat dissipated by the pipe fins. ANSYS can only solve for the *heat flux* along the fin, but not the actual heat loss. Still, we can arrive at a solution for the actual heat loss if we multiply the component fluxes by the corresponding normal fin areas. That is,

$$Q = qA$$

where Q is the heat loss, q is the component heat flux, and A is the normal area. Using the "NLIST" and the "PRNSOL" functions in ANSYS, we can output the flux solution for all of the nodes. In addition, we can output the coordinates of each node to determine which nodes are relevant to the solution (i.e. nodes at the base of the pipe and along a fin). We can filter the data in excel to find a sample of the nodes.



NODE	X	Y	Z
357	0.122020992830E-01	0.200000000000E-02	0.274063637334E-01
358	0.149999999915E-01	0.200000000000E-02	0.259807621185E-01
359	0.176335575612E-01	0.200000000000E-02	0.242705098368E-01
360	0.200739181840E-01	0.200000000000E-02	0.222943447704E-01
361	0.222943447582E-01	0.200000000000E-02	0.200739181976E-01
362	0.242705098255E-01	0.200000000000E-02	0.176335575767E-01
363	0.259807621083E-01	0.200000000000E-02	0.150000000090E-01
364	0.274063637242E-01	0.200000000000E-02	0.122020993037E-01
365	0.285316954848E-01	0.200000000000E-02	0.927050984372E-02
366	0.293444280192E-01	0.200000000000E-02	0.623735073798E-02
367	0.298356568595E-01	0.200000000000E-02	0.313585391234E-02
407	0.150000000000E-01	0.200000000000E-02	0.00000000000
408	-0.150000000000E-01	0.200000000000E-02	0.00000000000
409	0.149178284306E-01	0.200000000000E-02	-0.156792694870E-02
410	0.146722140111E-01	0.200000000000E-02	-0.311867536165E-02
411	0.142658477447E-01	0.200000000000E-02	-0.463525491472E-02
412	0.137031818652E-01	0.200000000000E-02	-0.610104964498E-02
413	0.129903810579E-01	0.200000000000E-02	-0.749999999803E-02
414	0.121352549171E-01	0.200000000000E-02	-0.881677878228E-02
415	0.111471723841E-01	0.200000000000E-02	-0.100369590932E-01

NODE	X	Y	Z
416	0.100369590976E-01	0.200000000000E-02	-0.111471723802E-01
417	0.881677878679E-02	0.200000000000E-02	-0.121352549139E-01
418	0.750000000255E-02	0.200000000000E-02	-0.129903810553E-01
419	0.610104964889E-02	0.200000000000E-02	-0.137031818634E-01
420	0.463525491869E-02	0.200000000000E-02	-0.142658477434E-01
421	0.311867536571E-02	0.200000000000E-02	-0.146722140103E-01
422	0.156792695283E-02	0.200000000000E-02	-0.149178284301E-01
423	0.415212166732E-11	0.200000000000E-02	-0.150000000000E-01
424	-0.156792694457E-02	0.200000000000E-02	-0.149178284310E-01
425	-0.311867535760E-02	0.200000000000E-02	-0.146722140120E-01
426	-0.463525491088E-02	0.200000000000E-02	-0.142658477460E-01
427	-0.610104964152E-02	0.200000000000E-02	-0.137031818667E-01
428	-0.749999999573E-02	0.200000000000E-02	-0.129903810592E-01
429	-0.881677878058E-02	0.200000000000E-02	-0.121352549184E-01
430	-0.100369590920E-01	0.200000000000E-02	-0.111471723852E-01
431	-0.111471723791E-01	0.200000000000E-02	-0.100369590988E-01
432	-0.121352549128E-01	0.200000000000E-02	-0.881677878834E-02
433	-0.129903810542E-01	0.200000000000E-02	-0.750000000452E-02
434	-0.137031818621E-01	0.200000000000E-02	-0.610104965183E-02
435	-0.142658477424E-01	0.200000000000E-02	-0.463525492186E-02

Figure 1. NLIST function is used to find the coordinates of relevant nodes.

PRNSOL Command

File

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0
NODAL RESULTS ARE FOR MATERIAL 1

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

NODE	TFX	TFY	TFZ	TFSUM
4990	-30747.	5408.9	17647.	35861.
4991	-33464.	5353.7	10700.	35539.
4992	-35255.	5509.3	3694.7	35874.
4993	-35482.	5269.2	-3730.2	36065.
4994	-34065.	5370.6	-10936.	36178.
4995	-30967.	5349.1	-17880.	36156.
4996	-26676.	5420.5	-23773.	36140.
4997	-21110.	5453.6	-28820.	36138.
4998	-14569.	5407.3	-32618.	36130.
4999	-7413.6	5391.6	-34870.	36055.
5000	-77.872	5393.4	-35882.	36286.
5001	7369.2	5401.7	-35114.	36283.
5002	14507.	5357.1	-32750.	36218.
5003	21039.	5344.2	-28787.	36054.
5004	26311.	5359.8	-23785.	35871.
5005	30730.	5389.7	-17734.	35887.
5006	33499.	5362.7	-10737.	35584.
5007	-5137.4	-5296.4	-7.0322	7378.7
5008	5077.2	-5290.8	-3.2398	7332.8
5010	-5035.5	-5313.7	1124.1	7406.5
5012	-4637.8	-5339.4	2054.4	7364.8
5014	-4120.7	-5354.0	2991.6	7388.9
5016	-3338.6	-5320.7	3717.5	7299.0
5018	-2426.2	-5303.6	4332.5	7265.4
5020	-1484.9	-5355.5	4785.3	7333.9
5022	-519.28	-5362.7	5040.6	7378.1
5024	615.90	-5350.2	5008.2	7354.3
5026	1518.1	-5254.9	4757.9	7249.6
5028	2516.4	-5323.8	4278.3	7278.6
5030	3347.2	-5350.7	3659.0	7295.4
5032	4085.6	-5316.4	2885.4	7299.4
5034	4638.9	-5354.4	2077.9	7382.9
5036	5067.8	-5460.9	1052.2	7524.0
5039	4967.4	-5310.0	-1108.4	7355.2
5041	4633.1	-5320.1	-2050.2	7346.5

***** POST1 NODAL THERMAL FLUX LISTING *****
PowerGraphics Is Currently Enabled

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0
NODAL RESULTS ARE FOR MATERIAL 1

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

NODE	TFX	TFY	TFZ	TFSUM
5043	4121.6	-5353.4	-2994.0	7389.9
5045	3329.2	-5340.9	-3725.1	7313.4
5047	2327.8	-5331.7	-4431.5	7313.2
5049	1542.8	-5318.8	-4806.0	7332.6
5051	530.26	-5358.4	-5044.3	7378.2
5053	-512.45	-5302.2	-5024.2	7322.4
5055	-1447.0	-5349.2	-4829.8	7350.8
5057	-2535.3	-5359.0	-4403.4	7384.9
5059	-3437.7	-5337.6	-3683.9	7340.3
5061	-4082.4	-5316.7	-2967.3	7330.6
5063	-4646.2	-5351.2	-2075.5	7384.5
5065	-4967.6	-5322.3	-1008.8	7350.0
5397	98184.	-5901.6	9229.0	98793.
5398	94736.	-5584.3	29767.	99459.

Figure 2. PRNSOL function to list the heat fluxes for all nodes.

Convection along the Fin: Y-Direction Heat Flux

It is important to first understand that the heat transfer to the surroundings by convection is not constant throughout the fin, because of the varying temperature distribution along the fin. The ANSYS results agree with this, as the Y-Flux steadily decreases as radius increases (with the exception of the three outliers). Because the flux is variable, we can either perform an integral of the Y-heat flux per differential area along the fin, or take the average flux and multiply by the normal fin area. That is

$$Q = \int_{r_1}^{r_2} q(r)dA = Q_{average} * A$$

In order to perform an integral, an equation for the flux as a function of radius must be obtained by fitting a best fit regression with the data. Instead, we use the average flux of a sample of relevant nodes for the sake of simplicity.

Node	x coordinate	Y-Flux	Absolute Value Y-Flux
5028	1.50E-02	-5324.1	5324.1
5047	1.50E-02	-5332	5332
5399	1.53E-02	-5718	5718
5424	1.53E-02	-3263.1	3263.1
5425	1.68E-02	-4374.8	4374.8
5430	1.70E-02	-5383.3	5383.3
5454	1.70E-02	-5377.1	5377.1
5397	1.76E-02	-5781.1	5781.1
5426	1.76E-02	-2654.8	2654.8
5461	1.86E-02	-5405.3	5405.3
5484	1.86E-02	-5320.1	5320.1
5429	1.91E-02	-5500.6	5500.6
5455	1.92E-02	-5444.8	5444.8
5030	2.01E-02	-5351.1	5351.1
5427	2.05E-02	-5450.7	5450.7
5456	2.06E-02	-5464.5	5464.5
5428	2.10E-02	-5390.9	5390.9
5460	2.17E-02	-5374.3	5374.3
5485	2.17E-02	-5373.1	5373.1
5457	2.38E-02	-5360.4	5360.4
5486	2.39E-02	-5391.5	5391.5
5458	2.49E-02	-5417.8	5417.8
5459	2.49E-02	-5178.8	5178.8
5041	2.74E-02	-5320.4	5320.4
5039	2.93E-02	-5310.3	5310.3
5008	3.00E-02	-5291.2	5291.2

Average Flux	**5175.157692 (W/m^2)**

Figure 3. Flux values and x-coordinate (radius) of relevant nodes after sorting in excel.

Referring to Figure 3, the average y-directional heat flux is calculated at

$$q_y = 5175.2\ W/m^2.$$

The corresponding normal area is the area along the diameter of the fin.

$$A_y = \pi(r_2^2 - r_1^2) * 2 = \pi(0.03^2 - 0.015^2) * 2 = 4.2412 \times 10^{-3}\ m^2$$

The heat transfer via Y-Flux is

$$Q_y = q_y A_y = 5175.2 * 4.2412 \times 10^{-3} = 21.949W$$

Convection at the base of pipe and at fin tips: X-Direction and Z-Direction Heat Flux

Similarly, the x-component fluxes along the base of the pipe and at the tip of the fin can be multiplied by the base area and fin tip area, respectively, to find the additional convection at these locations. It makes sense to take the x-component fluxes along the x-axis at the mentioned locations and distribute these values along the circumference of the geometry. Hence, the z-direction fluxes need not be read. Moreover, one will find that the x-fluxes along the x-axis are similar to the z-fluxes along the z-axis.

The x-component fluxes along the x-axis at the base of the pipe and tips of the fins were found with PRNSOL on average to be approximately

$$q_{xz\_base} = 10582.2\ W/m^2$$
$$q_{xz\_tip} = 5151.2\ W/m^2$$

One un-finned area of the pipe base is the base circumference multiplied by one base length.

$$A_{xz\_base} = C_{base}\, l_{base} = 2\pi r_{base} l = 2\pi(0.015)(0.003) = 2.8274 \times 10^{-4}\ m^2$$

One fin tip area is the fin circumference multiplied by the thickness of the fin

$$A_{xz\_tip} = C_{tip} t_{fin} = 2\pi r_{tip} t_{fin} = 2\pi(0.03)(0.002) = 3.7699 \times 10^{-4}\ m^2$$

The total additional heat transfer from the base and fin tips is

$$\begin{aligned} Q_{xz\_total} &= q_{xz\_base} A_{xz\_base} + q_{xz\_tip} A_{xz\_tip} \\ &= (10582.2)(2.8274 \times 10^{-4}) + (5151.1)(3.7699 \times 10^{-4}) = 4.9339\ W \end{aligned}$$

The total heat transfer by convection for 200 fin and un-finned areas is

$$Q_{total} = 200(Q_y + Q_{xz\_total}) = 200(21.949 + 4.9339) = 5376.6\ W$$

Comparing ANSYS with Hand Calculations

The answer using the method presented in the book gives an answer of 5380 W. ANSYS generated a heat loss of 5376.6 W. The **error is approximately -0.06% or -0.0006** and is sufficiently small enough to verify that the **ANSYS results are reasonable**.

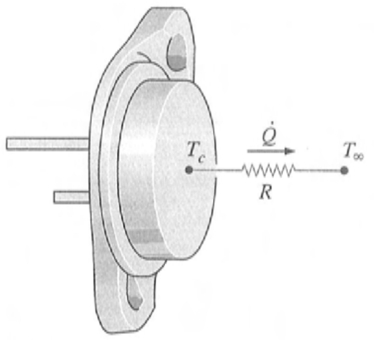


FIGURE 3–47
Schematic for Example 3–10.

Analysis The power transistor and the thermal resistance network associated with it are shown in Fig. 3–47. We notice from the thermal resistance network that there is a single resistance of 20°C/W between the case at T_c = 85°C and the ambient at T_∞ = 25°C, and thus the rate of heat transfer is

$$\dot{Q} = \left(\frac{\Delta T}{R}\right)_{\text{case-ambient}} = \frac{T_c - T_\infty}{R_{\text{case-ambient}}} = \frac{(85 - 25)^\circ\text{C}}{20^\circ\text{C/W}} = 3 \text{ W}$$

Therefore, this power transistor should not be operated at power levels above 3 W if its case temperature is not to exceed 85°C.

Discussion This transistor can be used at higher power levels by attaching it to a heat sink (which lowers the thermal resistance by increasing the heat transfer surface area, as discussed in the next example) or by using a fan (which lowers the thermal resistance by increasing the convection heat transfer coefficient).

EXAMPLE 3–11 Selecting a Heat Sink for a Transistor

A 60-W power transistor is to be cooled by attaching it to one of the commercially available heat sinks shown in Table 3–6. Select a heat sink that will allow the case temperature of the transistor not to exceed 90°C in the ambient air at 30°C.

SOLUTION A commercially available heat sink from Table 3–6 is to be selected to keep the case temperature of a transistor below 90°C.

Assumptions **1** Steady operating conditions exist. **2** The transistor case is isothermal at 90°C. **3** The contact resistance between the transistor and the heat sink is negligible.

Analysis The rate of heat transfer from a 60-W transistor at full power is $\dot{Q}$ = 60 W. The thermal resistance between the transistor attached to the heat sink and the ambient air for the specified temperature difference is determined to be

$$\dot{Q} = \frac{\Delta T}{R} \longrightarrow R = \frac{\Delta T}{\dot{Q}} = \frac{(90 - 30)^\circ\text{C}}{60 \text{ W}} = 1.0^\circ\text{C/W}$$

Therefore, the thermal resistance of the heat sink should be below 1.0°C/W. An examination of Table 3–6 reveals that the HS 5030, whose thermal resistance is 0.9°C/W in the vertical position, is the only heat sink that will meet this requirement.

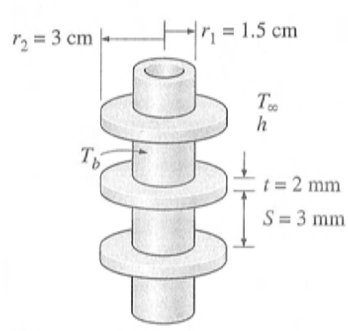


FIGURE 3–48
Schematic for Example 3–12.

EXAMPLE 3–12 Effect of Fins on Heat Transfer from Steam Pipes

Steam in a heating system flows through tubes whose outer diameter is D_1 = 3 cm and whose walls are maintained at a temperature of 120°C. Circular aluminum alloy fins (k = 180 W/m · °C) of outer diameter D_2 = 6 cm and constant thickness t = 2 mm are attached to the tube, as shown in Fig. 3–48. The space between the fins is 3 mm, and thus there are 200 fins per meter length of the tube. Heat is transferred to the surrounding air at T_∞ = 25°C, with a combined heat transfer coefficient of $h = 60\ \text{W/m}^2 \cdot {}^\circ\text{C}$. Determine the increase in heat transfer from the tube per meter of its length as a result of adding fins.

SOLUTION Circular aluminum alloy fins are to be attached to the tubes of a heating system. The increase in heat transfer from the tubes per unit length as a result of adding fins is to be determined.

Assumptions 1 Steady operating conditions exist. **2** The heat transfer coefficient is uniform over the entire fin surfaces. **3** Thermal conductivity is constant. **4** Heat transfer by radiation is negligible.

Properties The thermal conductivity of the fins is given to be $k = 180\ \text{W/m} \cdot {}^\circ\text{C}$.

Analysis In the case of no fins, heat transfer from the tube per meter of its length is determined from Newton's law of cooling to be

$$
\begin{aligned}
A_{\text{no fin}} &= \pi D_1 L = \pi(0.03\ \text{m})(1\ \text{m}) = 0.0942\ \text{m}^2 \\
\dot{Q}_{\text{no fin}} &= hA_{\text{no fin}}(T_b - T_\infty) \\
&= (60\ \text{W/m}^2 \cdot {}^\circ\text{C})(0.0942\ \text{m}^2)(120 - 25)^\circ\text{C} \\
&= 537\ \text{W}
\end{aligned}
$$

The efficiency of the circular fins attached to a circular tube is plotted in Fig. 3–43. Noting that $L = \frac{1}{2}(D_2 - D_1) = \frac{1}{2}(0.06 - 0.03) = 0.015$ m in this case, we have

$$
\begin{aligned}
r_{2c} &= r_2 + t/2 = 0.03 + 0.002/2 = 0.031\ \text{m} \\
L_c &= L + t/2 = 0.015 + 0.002/2 = 0.016\ \text{m} \\
A_p &= L_c t = (0.016\ \text{m})(0.002\ \text{m}) = 3.20 \times 10^{-5}\ \text{m}^2
\end{aligned}
$$

$$
\left.
\begin{aligned}
&\frac{r_{2c}}{r_1} = \frac{0.031\ \text{m}}{0.015\ \text{m}} = 2.07 \\
&L_c^{3/2}\sqrt{\frac{h}{kA_p}} = (0.016\ \text{m})^{3/2} \times \sqrt{\frac{60\ \text{W/m}^2 \cdot {}^\circ\text{C}}{(180\ \text{W/m} \cdot {}^\circ\text{C})(3.20 \times 10^{-5}\ \text{m}^2)}} = 0.207
\end{aligned}
\right\} \eta_{\text{fin}} = 0.96
$$

$$
\begin{aligned}
A_{\text{fin}} &= 2\pi(r_{2c}^2 - r_1^2) = 2\pi[(0.031\ \text{m})^2 - (0.015\ \text{m})^2] \\
&= 0.004624\ \text{m}^2
\end{aligned}
$$

$$
\begin{aligned}
\dot{Q}_{\text{fin}} &= \eta_{\text{fin}} \dot{Q}_{\text{fin, max}} = \eta_{\text{fin}} h A_{\text{fin}} (T_b - T_\infty) \\
&= 0.96(60\ \text{W/m}^2 \cdot {}^\circ\text{C})(0.004624\ \text{m}^2)(120 - 25)^\circ\text{C} \\
&= 25.3\ \text{W}
\end{aligned}
$$

Heat transfer from the unfinned portion of the tube is

$$
\begin{aligned}
A_{\text{unfin}} &= \pi D_1 S = \pi(0.03\ \text{m})(0.003\ \text{m}) = 0.000283\ \text{m}^2 \\
\dot{Q}_{\text{unfin}} &= hA_{\text{unfin}}(T_b - T_\infty) \\
&= (60\ \text{W/m}^2 \cdot {}^\circ\text{C})(0.000283\ \text{m}^2)(120 - 25)^\circ\text{C} \\
&= 1.6\ \text{W}
\end{aligned}
$$

Noting that there are 200 fins and thus 200 interfin spacings per meter length of the tube, the total heat transfer from the finned tube becomes

$$\dot{Q}_{\text{total, fin}} = n(\dot{Q}_{\text{fin}} + \dot{Q}_{\text{unfin}}) = 200(25.3 + 1.6)\text{ W} = 5380\text{ W}$$

Therefore, the increase in heat transfer from the tube per meter of its length as a result of the addition of fins is

$$\dot{Q}_{\text{increase}} = \dot{Q}_{\text{total, fin}} - \dot{Q}_{\text{no fin}} = 5380 - 537 = 4843\text{ W} \qquad \text{(per m tube length)}$$

Discussion The overall effectiveness of the finned tube is

$$\varepsilon_{\text{fin, overall}} = \frac{\dot{Q}_{\text{total, fin}}}{\dot{Q}_{\text{total, no fin}}} = \frac{5380\text{ W}}{537\text{ W}} = 10.0$$

That is, the rate of heat transfer from the steam tube increases by a factor of 10 as a result of adding fins. This explains the widespread use of finned surfaces.

3–7 ■ HEAT TRANSFER IN COMMON CONFIGURATIONS

So far, we have considered heat transfer in *simple* geometries such as large plane walls, long cylinders, and spheres. This is because heat transfer in such geometries can be approximated as *one-dimensional,* and simple analytical solutions can be obtained easily. But many problems encountered in practice are two- or three-dimensional and involve rather complicated geometries for which no simple solutions are available.

An important class of heat transfer problems for which simple solutions are obtained encompasses those involving two surfaces maintained at *constant* temperatures T_1 and T_2. The steady rate of heat transfer between these two surfaces is expressed as

$$Q = Sk(T_1 - T_2) \qquad \text{(3–79)}$$

where S is the **conduction shape factor**, which has the dimension of *length,* and k is the thermal conductivity of the medium between the surfaces. The conduction shape factor depends on the *geometry* of the system only.

Conduction shape factors have been determined for a number of configurations encountered in practice and are given in Table 3–7 for some common cases. More comprehensive tables are available in the literature. Once the value of the shape factor is known for a specific geometry, the total steady heat transfer rate can be determined from the equation above using the specified two constant temperatures of the two surfaces and the thermal conductivity of the medium between them. Note that conduction shape factors are applicable only when heat transfer between the two surfaces is by *conduction.* Therefore, they cannot be used when the medium between the surfaces is a liquid or gas, which involves natural or forced convection currents.

A comparison of Eqs. 3–4 and 3–79 reveals that the conduction shape factor S is related to the thermal resistance R by $R = 1/kS$ or $S = 1/kR$. Thus, these two quantities are the inverse of each other when the thermal conductivity of the medium is unity. The use of the conduction shape factors is illustrated with Examples 3–13 and 3–14.

TAB
Con
tran