Thermal - Mixed Boundary Example
(Conduction/Convection/Insulated)

Introduction

This tutorial was created using ANSYS 7.0 to solve simple thermal examples. Analysis of a simple conduction as well a mixed conduction/convection/insulation problem will be demonstrated.

The Mixed Convection/Conduction/Insulated Boundary Conditions Example is constrained as shown in the following figure (Note that the section is assumed to be infinitely long):

ANSYS Command Listing

/title, Simple Convection Example
/PREP7

! define geometry

length=1.0
height=1.0
blc4,0,0,length, height ! area - one corner, then width and height

! mesh 2D areas

ET,1, PLANE55 ! Thermal element only
MP,KXX,1,10 ! 10 W/mC
MAT,1
TYPE,1
ESIZE,length/20 ! number of element sub-divisions/side

Copyright 2003 - University of Alberta
AMESH,ALL

FINISH
/SOLU

ANTYPE,0 ! STEADY-STATE THERMAL ANALYSIS

! fixed temp BC's

NSEL,S,LOC,Y,height ! select nodes on top with y=height
D,ALL,TEMP,500 ! apply fixed temp of 500C
NSEL,ALL
NSEL,S,LOC,X,0 ! select nodes on three sides
D,ALL,TEMP,100 ! apply fixed temp of 100C
NSEL,ALL

! convection BC's

NSEL,S,LOC,X,length ! right edge
SF,ALL,CONV,10,100 ! apply fixed temp of 100C
NSEL,ALL

! Insulated BC's

NSEL,S,LOC,Y,0 ! bottom edge
SF,ALL,CONV,0 ! insulate edge
NSEL,ALL

SOLVE
FINISH

/POST1
PLNSOL,TEMP,,0, ! contour plot of temperatures