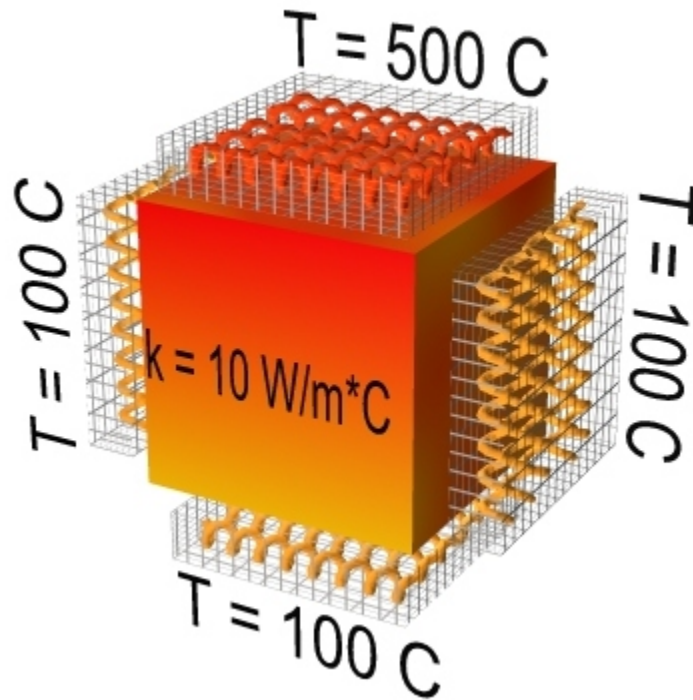


Simple Conduction Example

Introduction

This tutorial was created using ANSYS 7.0 to solve a simple conduction problem.

The Simple Conduction Example is constrained as shown in the following figure. Thermal conductivity (k) of the material is $10 \text{ W/m}^{\circ}\text{C}$ and the block is assumed to be infinitely long.



ANSYS Command Listing

```

/title, Simple Conduction 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
ESIZE,length/20                   ! number of element sub-divisions/side
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
NSEL,A,LOC,X,length
NSEL,A,LOC,Y,0
D,ALL,TEMP,100                          ! apply fixed temp of 100C
NSEL,ALL

SOLVE
FINISH

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