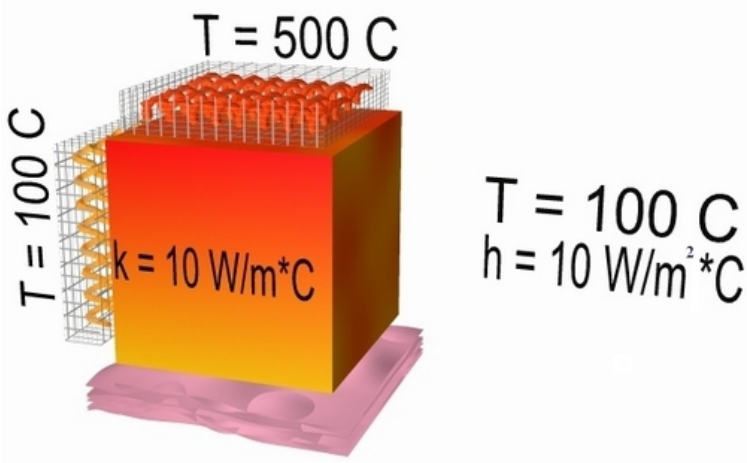
Thermal - Mixed Boundary Example (Conduction/Convection/Insulated)

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):



Preprocessing: Defining the Problem

1. **Give example a Title**
2. **Open preprocessor menu**
   ANSYS Main Menu > Preprocessor
   `/PREP7`
3. **Create geometry**
   Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners > X=0, Y=0, Width=1, Height=1
   `BLC4,0,0,1,1`
4. **Define the Type of Element**
   Preprocessor > Element Type > Add/Edit/Delete... > click 'Add' > Select Thermal Mass Solid, Quad 4Node 55
   `ET,1,PLANE55`

   As in the conduction example, we will use PLANE55 (Thermal Solid, Quad 4node 55). This element has 4 nodes and a single DOF (temperature) at each node. PLANE55 can only be used for 2 dimensional steady-state or transient thermal analysis.
5. **Element Material Properties**
   Preprocessor > Material Props > Material Models > Thermal > Conductivity > Isotropic > KXX = 10
   `MP,KXX,1,10`
   This will specify a thermal conductivity of 10 W/m*C.
6. **Mesh Size**
   Preprocessor > Meshing > Size Cntrls > ManualSize > Areas > All Areas > 0.05
   `AESIZE,ALL,0.05`
7. **Mesh**
   Preprocessor > Meshing > Mesh > Areas > Free > Pick All
   `AMESH,ALL`

Solution Phase: Assigning Loads and Solving

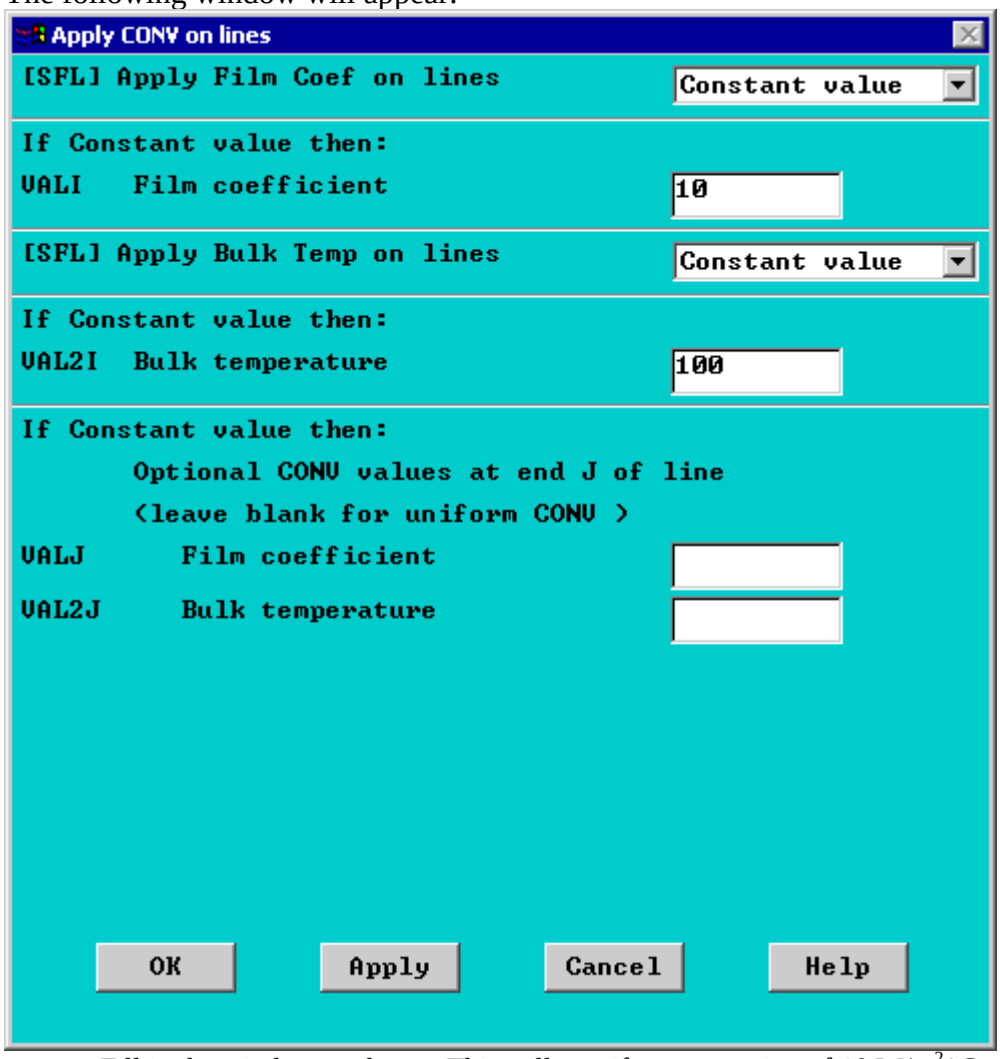
1. **Define Analysis Type**
   Solution > Analysis Type > New Analysis > Steady-State
   `ANTYPE,0`
2. **Apply Conduction Constraints**

In this example, all 2 sides of the block have fixed temperatures, while convection occurs on the other 2 sides.

- **Solution > Define Loads > Apply > Thermal > Temperature > On Lines**
- Select the top line of the block and constrain it to a constant value of 500 C
- Using the same method, constrain the left side of the block to a constant value of 100 C

3. **Apply Convection Boundary Conditions**
   - **Solution > Define Loads > Apply > Thermal > Convection > On Lines**
   - Select the right side of the block.

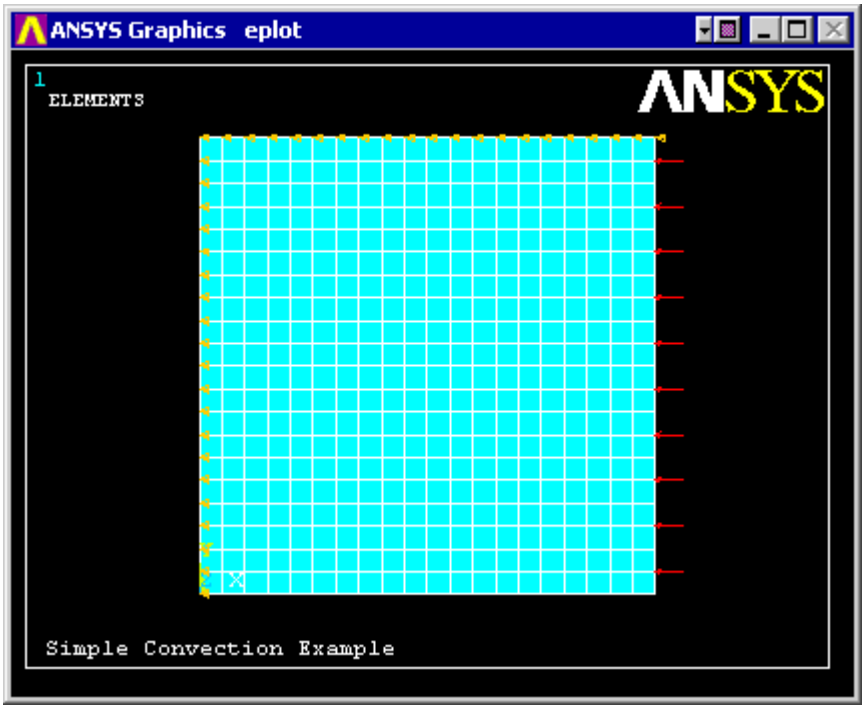
   The following window will appear:



   - Fill in the window as shown. This will specify a convection of 10 W/m^2*C and an ambient temperature of 100 degrees Celcius. Note that VALJ and VAL2J have been left blank. This is because we have uniform convection across the line.

4. **Apply Insulated Boundary Conditions**
   - **Solution > Define Loads > Apply > Thermal > Convection > On Lines**
   - Select the bottom of the block.
   - Enter a constant Film coefficient (VALI) of 0. This will eliminate convection through the side, thereby modeling an insulated wall. Note: you do not need to enter a Bulk (or ambient) temperature

You should obtain the following:



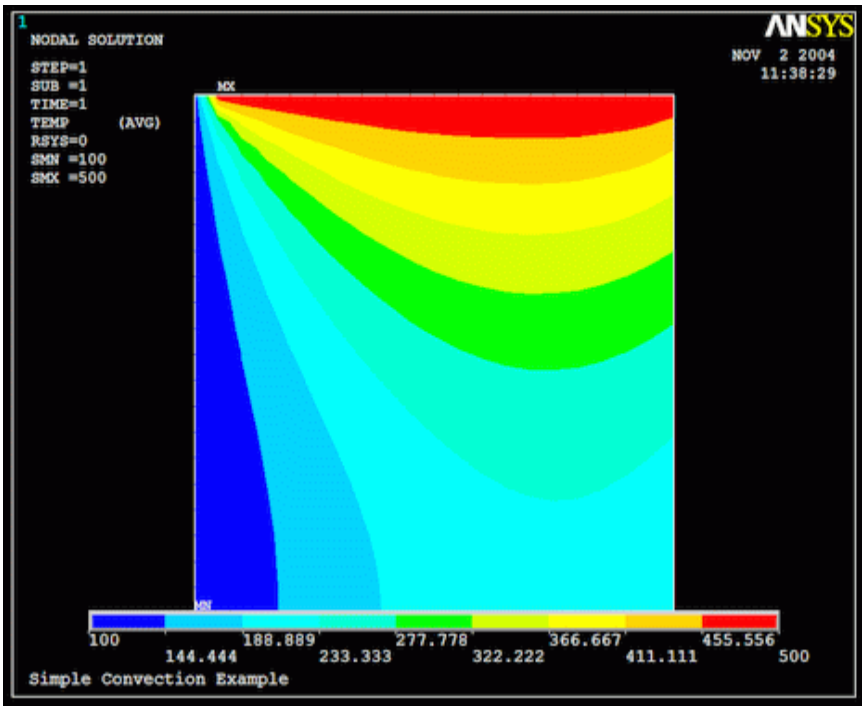
5. **Solve the System**
   Solution > Solve > Current LS
   SOLVE

Postprocessing: Viewing the Results

1. **Results Using ANSYS**

Plot Temperature

General Postproc > Plot Results > Contour Plot > Nodal Solu ... > DOF solution, Temperature TEMP



Command File Mode of Solution

The above example was solved using a mixture of the Graphical User Interface (or GUI) and the command language interface of ANSYS. This problem has also been solved using the ANSYS command language interface

that you may want to browse. Copy and paste following code into Notepad or a similar text editor and save it to your computer. Now go to '**File > Read input from...**' and select the file.

```
/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
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
```

Reference

http://www.mece.ualberta.ca/tutorials/ansys