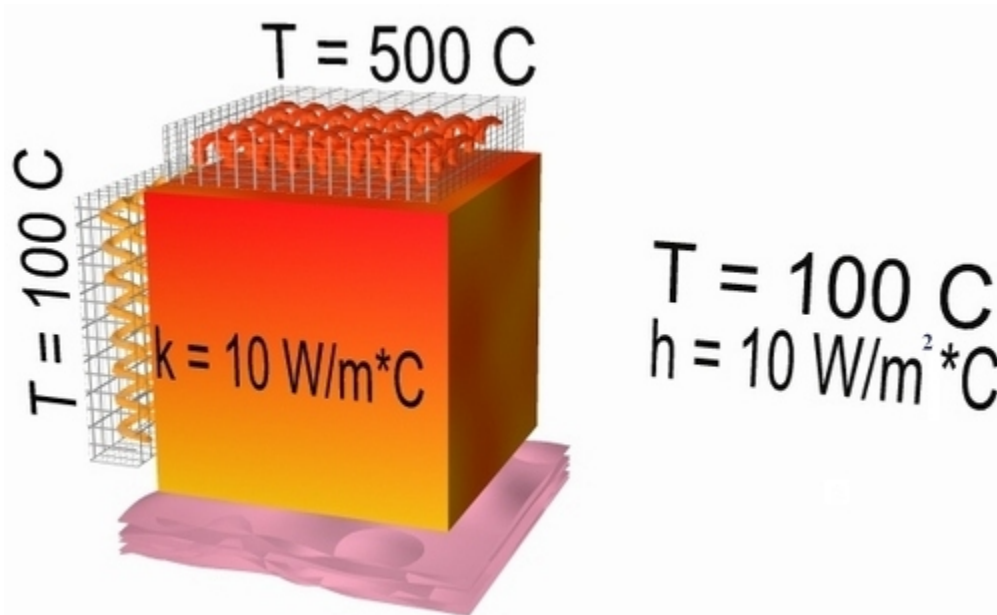


Thermal - Mixed Boundary Example (Conduction/Convection/Insulated)

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. Create geometry

Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners > X=0, Y=0,
Width=1, Height=1
BLC4, 0, 0, 1, 1

3. Define the Type of Element

Preprocessor > Element Type > Add/Edit/Delete... > click 'Add' > Select Thermal 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.

4. 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.

5. Mesh Size

Preprocessor > Meshing > Size Cntrl > ManualSize > Areas > All Areas > 0.05
AESIZE, ALL, 0.05

6. 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

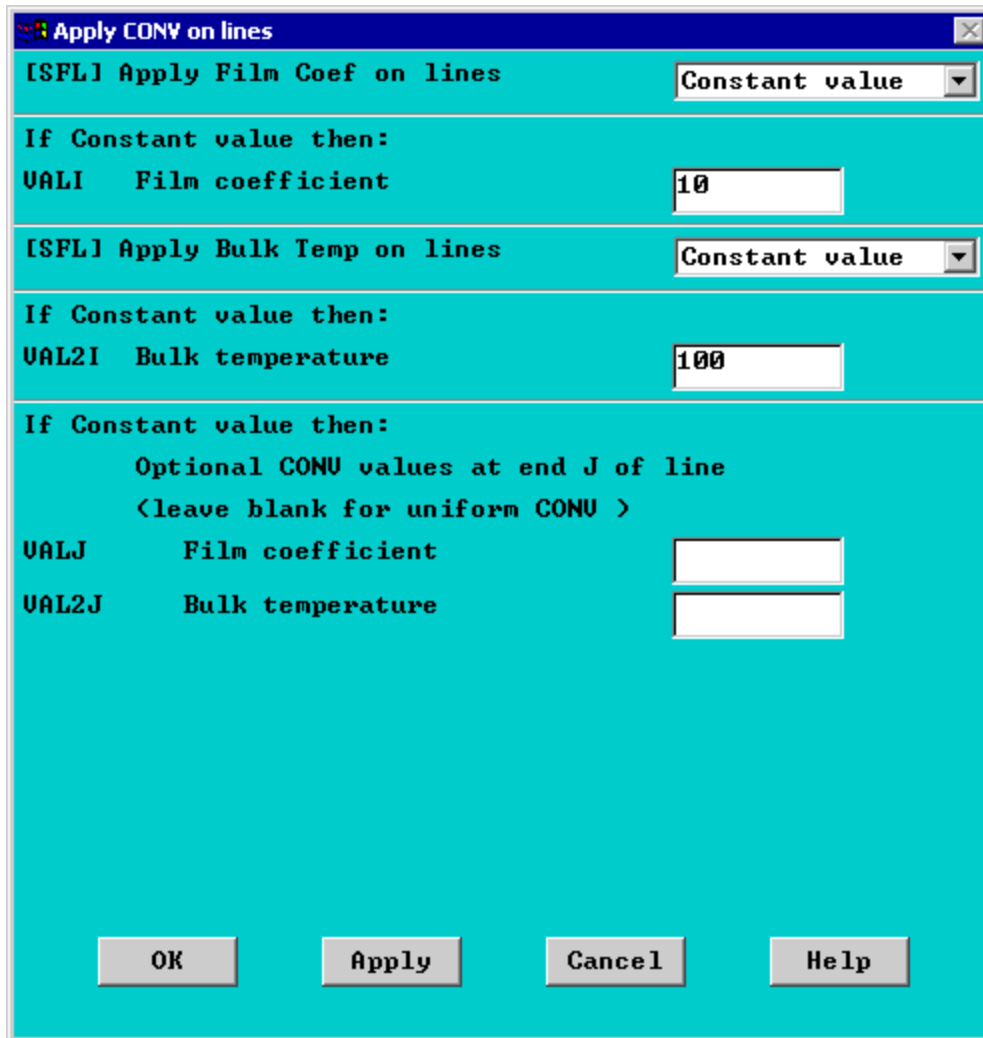
In this example, all 2 sides of the block have fixed temperatures, while convection occurs on the other 2 sides.

- **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:

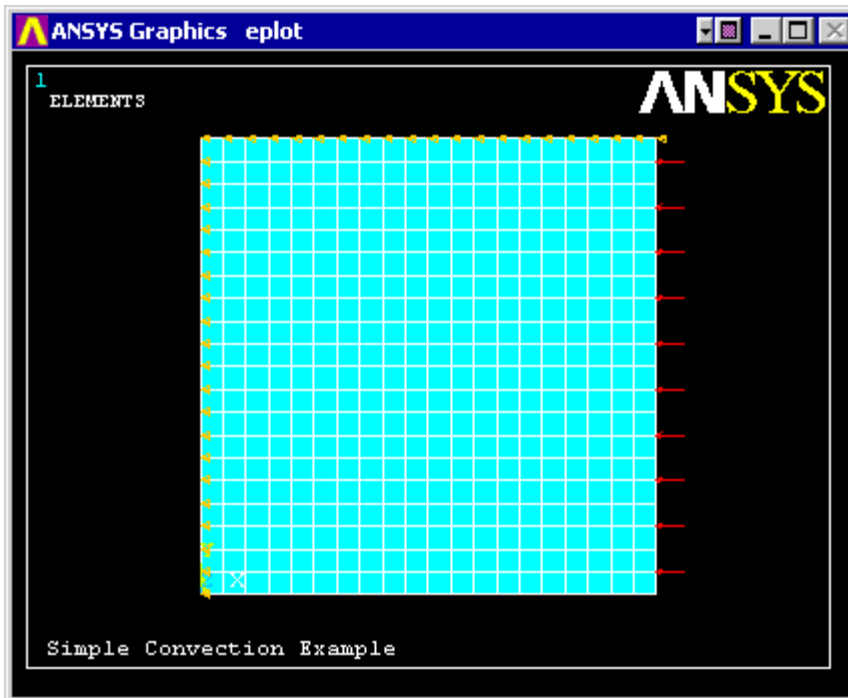


- Fill in the window as shown. This will specify a convection of $10 \text{ W/m}^2\text{C}$ and an ambient temperature of 100 degrees Celcius. Note that VALJ and VAL2J have been left blank. This is because we have uniform convection across the line.

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

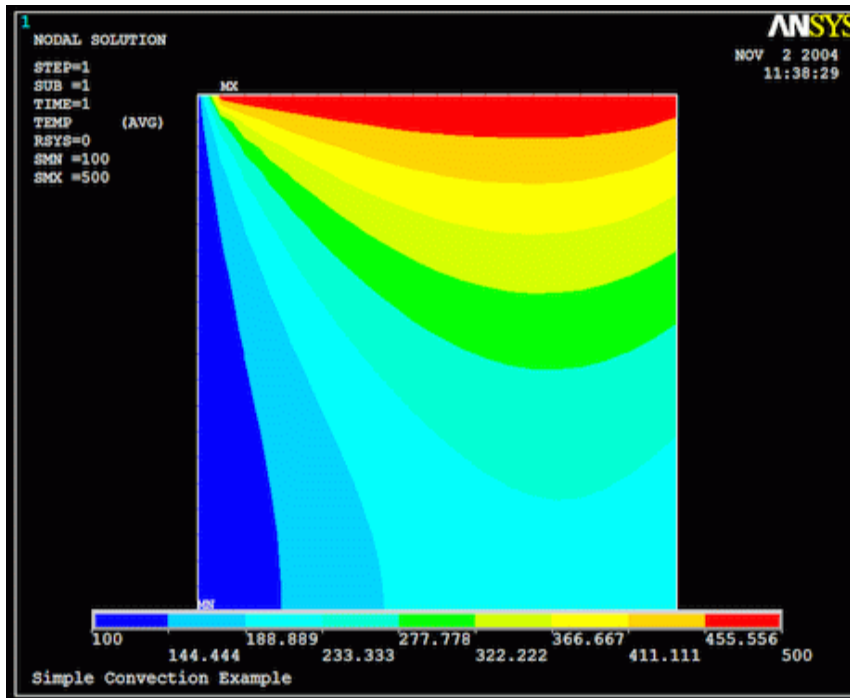
Solution > Solve > Current LS
SOLVE

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. Open the file and save it to your computer. Now go to '**File > Read input from...**' and select the file.