

## Contents

<b>Introduction</b>	<b>2</b>
Features of the Code	2
<b>Running INSTED®/Heat Conduction Program</b>	<b>4</b>
Starting the Program	4
The Restart File	4
<b>Program Organization</b>	<b>5</b>
Tasks	6
Required Data	6
Optional Data	6
Results	6
Sketch of a Radial Wall	7
Sketch of a Slab Wall	7
Starting a New Project	8
Loading an Old Project	8
Saving a New Project	9
<b>Description of the Dialog Boxes</b>	<b>10</b>
The Main Dialog Box	10
The Project Administration Dialog Box	12
The Problem Description Dialog Box	13
The Layer Data Dialog Box	13
Boundary Conditions Dialog Box	14
Geometry Dialog Box	14
<b>Running a Sample Problem</b>	<b>15</b>
Procedure for Performing a Slab Analysis	15
Procedure for Performing a Radial Analysis	16
Sample Analysis	17

## **INTRODUCTION**

The procedures for executing INSTED®/Heat Conduction program are presented in this manual. The primary tasks, required data, and the output results are discussed, as well as the online utilities that are available within the program environment.

The heat conduction program analyzes heat flow rate across composite slabs or radial walls. The walls may contain fins at the surface to enhance heat transfer. The program also supports radiation on the surface.

The procedure is divided into two main activities:

- (1) Define the materials that compose the wall to be analyzed. The materials are defined by their thermal conductivity.
- (2) Describe the wall in terms of the layers that make up the wall.

A detailed description of the procedures to perform a sample calculation with this program is presented later in this manual.

### **Features of the Code**

- Supports boundary conditions of the specified temperature, film coefficient, and radiation types
- Supports fins at the surfaces
- Integrates with the Fin Analysis program in order to determine the effective film coefficient on surfaces with fins
- Allows the use of up to 50 material types and 50 layers.

### **Material Types**

Material types have been introduced in INSTED/Heat Conduction as a way of grouping together layers and/or segments in a layer that have the same thermal

conductivity. This obviates the need to define thermal conductivity separately for these layers or segments, thereby saving the time for data specification.

**Definition of a Layer**

A layer refers to the number of different materials encountered along the direction of heat flow.

**Definition of a Segment**

A segment refers to one of the different materials that are 'stacked' together to form a layer. Segments will sometimes be referred to as materials in this manual, as in the sketch of the composite slabs.

**Definition of a Surface**

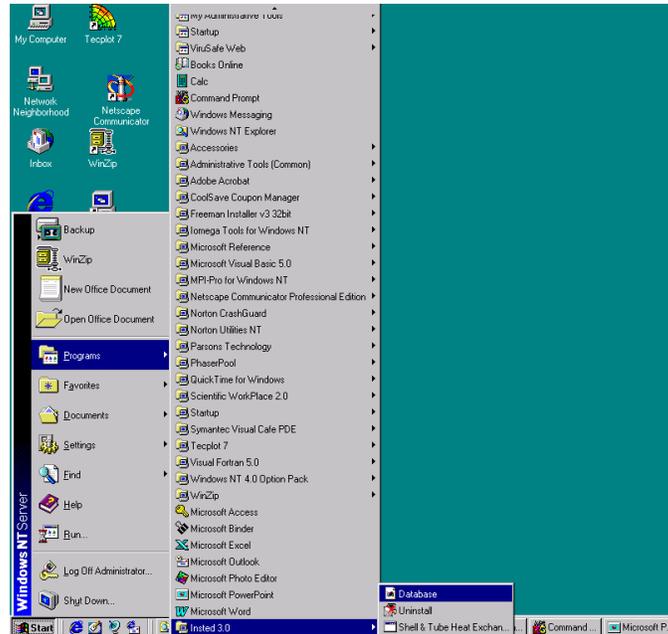
Heat flows from surface 1, through a wall, to surface 2. The wall could be made up of different layers.

## RUNNING INSTED®/Heat Conduction Program

### Starting the Program

Follow this procedure to execute INSTED®/Heat Conduction Program.

1. Click on the Windows **Start** button
2. Select **Program**
3. Select **INSTED 3.0**
4. Select the **Conduction Analysis** icon

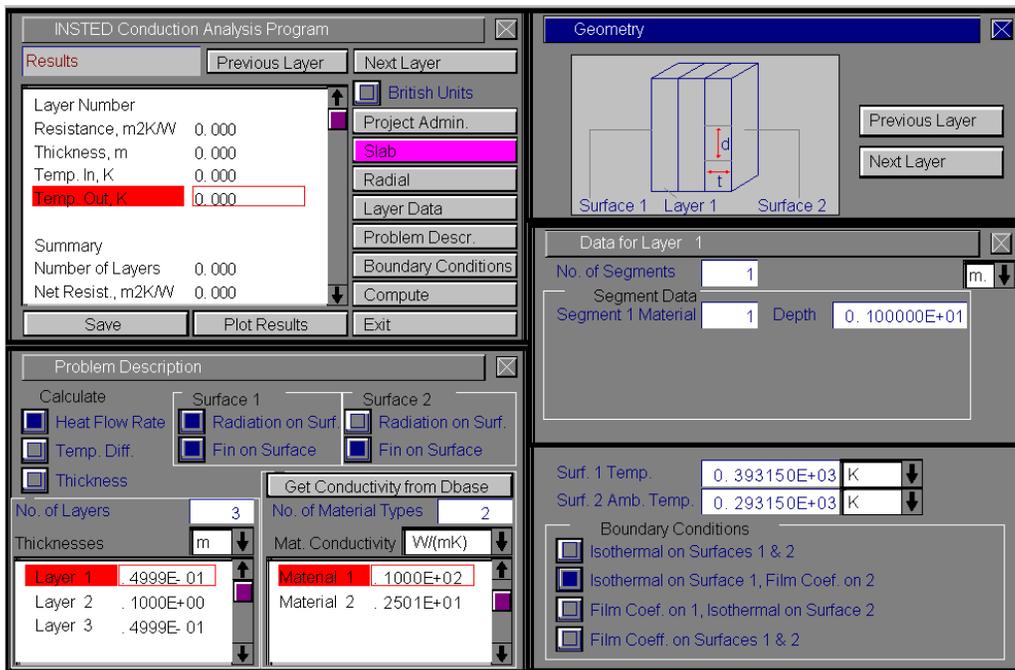


### The Restart File

The Heat Conduction program automatically loads the previous analysis if the program was terminated normally in the previous session. This is to allow previous sessions to continue naturally. However, to initialize all input fields, select the “New Project” option from the ‘Project Admin’ dialog box. This feature is explained in more details later in this manual.

**PROGRAM ORGANIZATION**

The INSTED®/Heat Conduction program is organized into six dialog boxes: the Main, Project Administration, Problem Description, Layer Data, Geometry, and the Boundary Conditions dialog boxes. The dialog boxes are launched from the Main dialog box. Result presentation is done also via the Main dialog box. A detailed description of each dialog box is presented later in this manual. The figure below shows the program interface.



**Tasks**

INSTED®/Heat Conduction program performs the following tasks:

- Calculation of heat flow rate
- Calculation of temperature difference
- Calculation of required thickness for a layer (for a target heat flow rate)

**Required Data**

The minimum data required for analysis are:

- Number of material types
- Material conductivity for every material type
- Number of layers
- Thickness and thermal conductivity for each layer
- Indicator for the presence or absence of fins at the surfaces
- Indicator for the presence or absence of radiation at the surfaces
- Surface boundary condition type

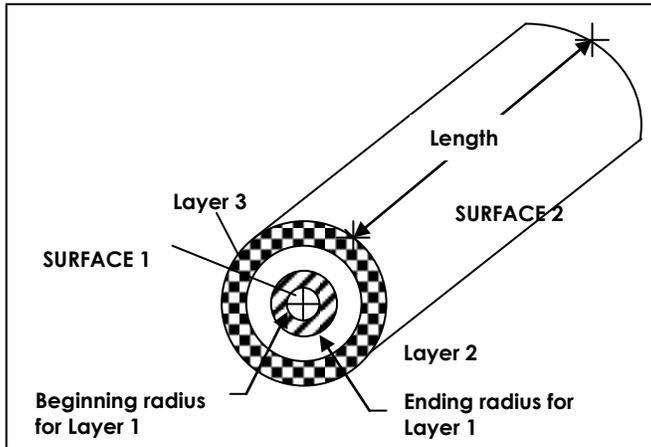
**Optional Data**

- Surface temperature or heat flow rate, depending on the task and surface conditions

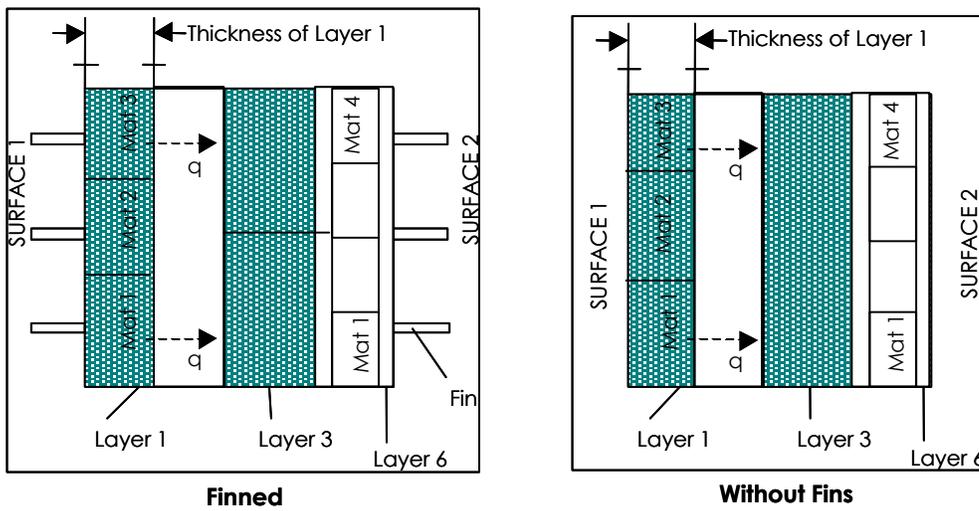
**Results**

- Net resistance
- Net temperature difference
- Total heat transfer rate
- Resistance and inlet and outlet temperature for each layer
- Thickness of a layer, if this is the specified task.

**Sketch of a Radial Wall**



**Sketch of a Slab Wall**



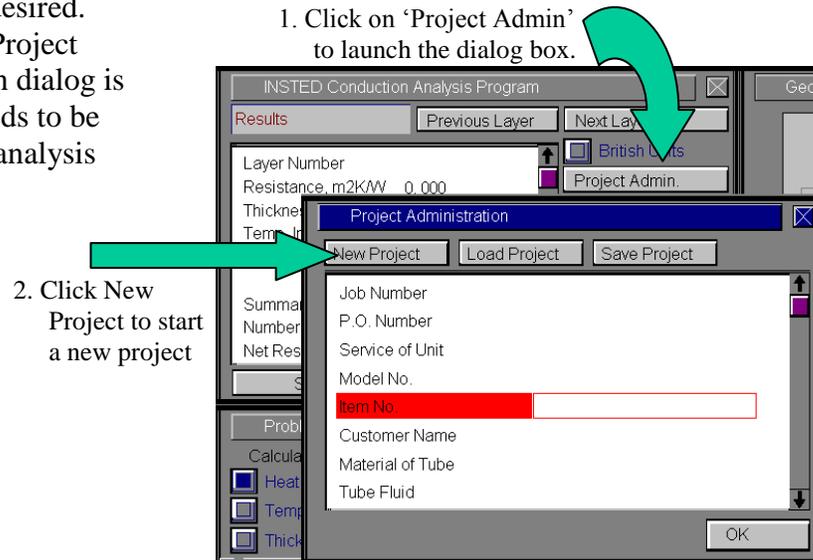
Note that for slabs, a layer could be composed of many different materials, or segments. For a radial wall, it is assumed that each layer has only one segment.

## Starting a New Project

To start a new project:

1. Click the “Project Admin.” button on the Main dialog box.
2. Click “New Project” on the ‘Project Administration’ dialog box.
3. Fill in the administrative data for the editbox on the Project Administration dialog, if desired.

Note that the Project Administration dialog is modal and needs to be closed before analysis can continue.

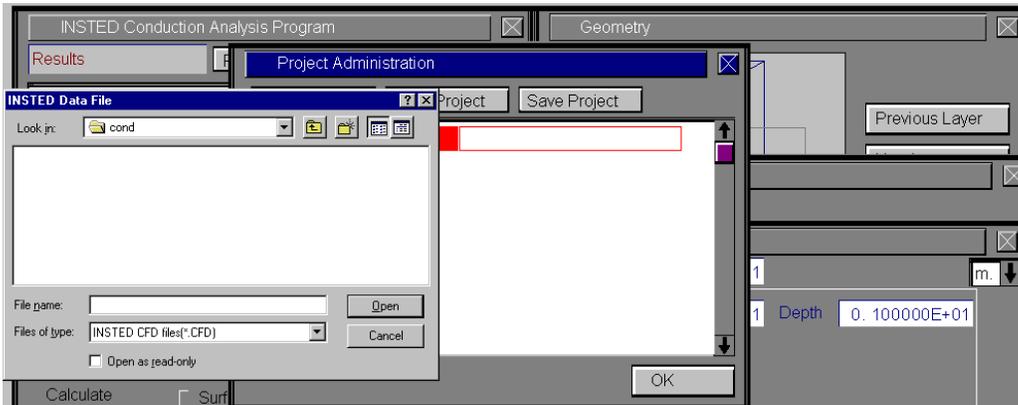


## Loading an Old Project

To load an old project:

1. Click the “Project Admin.” button on the Main dialog box
2. Click on the “Load Project” button
3. Type \*.\* in the file name text box on the Windows file menu which comes up
4. Press the Enter key

5. Select the file you wish to open by locating it on the Windows file open dialog. This dialog box is shown below



## Saving a New Project

To save a new project:

1. Click the “Project Admin.” button on the Main dialog box
  2. Click on the “Save Project” button
- ⇒ A Windows ‘File Save’ dialog box will appear
3. Select the path where you wish to save the file
  4. Type a name for the new project file

**DESCRIPTION OF THE DIALOG BOXES**

The following section describes the functions of the dialog boxes in INSTED®/Heat Conduction program.

**The Main Dialog Box**

This dialog box contains twelve buttons and an option button. The option button, if selected, indicates that all input data and results will use the British unit by default. Leaving this button unselected implies SI units for all calculations, input and results. The functions of the buttons are described below:

**Project Admin**

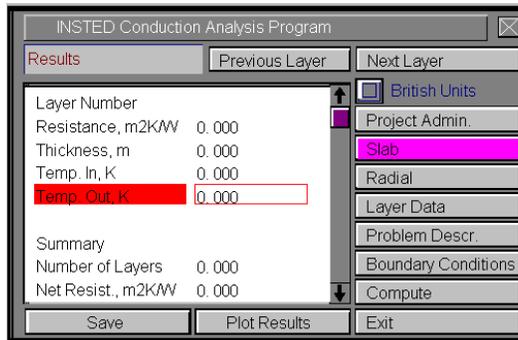
This button opens the ‘Project Administration’ dialog box, which performs administrative functions including loading and saving the session data, and initializing all data fields for starting a new session.

**Slab and Radial**

Press the “Slab” button to perform heat conduction calculations in a slab. Press the “Radial” button to perform calculations of through a radial wall.

**Layer Data**

Click on the “Layer Data” button to open the ‘Layer Data’ dialog box, which contains input field for the selected layer, including the thickness and thermal conductivity.



**Problem Description**

This button opens the ‘Problem Description’ dialog box, which contains a description of the problem such as the material types and layer thicknesses, the desired task, as well as the presence or absence of fins on the surfaces.

**Boundary Conditions**

This button opens the boundary conditions dialog box, where the boundary conditions data are specified.

**Compute**

The Compute button tells INSTED to compute the results. This assumes that all required input data for the task have been specified. If this is not the case, the computation may be unsuccessful and the missing input is indicated, via a (warning) message box.

**To Save**

Click on the “Save” button, which is conveniently situated on the Main dialog box. To safeguard work done in the session, it is advisable to intermittently perform a save operation. Note that an error occurs if no project has been loaded previously and saved using the “Save Project” button in the ‘Project Administration’ dialog box. The procedure for saving a new project has been described in a previous section of this manual.

**Plot Results**

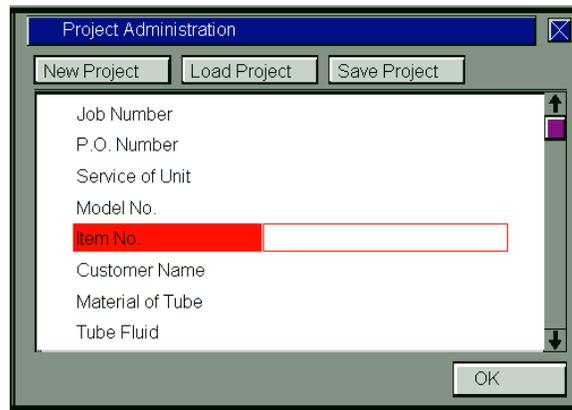
Press the “Plot Results” button to plot the temperature along the slab or radial wall. This plot may be printed from the File/Print menu at the top of the page. Clicking on any point on the page dismisses the plot and returns you to the main program.

## Exit

Press this button to close the program and end the current session. All session data are stored and are automatically retrieved on restarting the program.

## The Project Administration Dialog Box

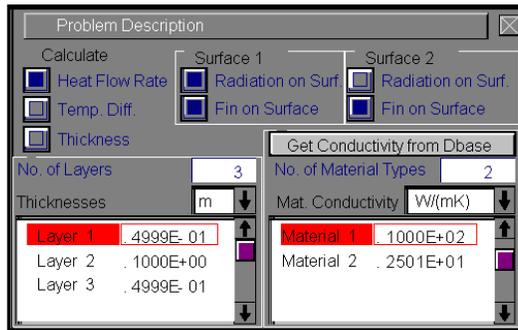
In the previous section, we covered much of the functions of the 'Project Administration' dialog box, including loading an existing project, saving a new project, or starting a new project.



The Project Administration dialog box also contains a description of the project including job number, customer names, etc.

**The Problem Description Dialog Box**

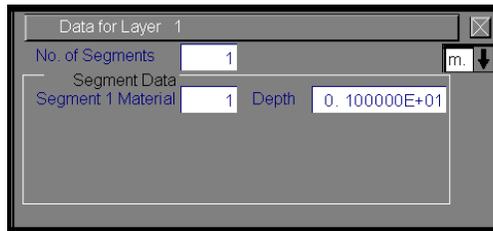
This dialog box contains a description of the problem. Select one of the options under ‘Calculate’ for the task you need to perform.



Other data to be specified in this dialog box include an indicator for the presence or absence of fins and radiation on either surface of the wall. The thermal conductivities of the wall are also defined here. (Thermal conductivity may also be obtained from the database by pressing the “Get Conductivity from Dbase” button.) A material type may not be used unless its thermal conductivity has been defined in this dialog box.

**Layer Data Dialog Box**

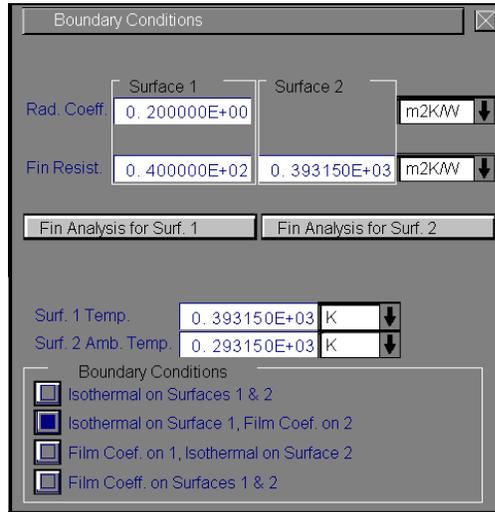
The thickness of a layer and material types of each segment in the layer should be specified in this dialog box.



A radial layer has only one segment, the thickness of the layer and type of material are entered in this dialog box. Slab layers are allowed to be composed of four segments. However, the total depth of the segments must be approximately equal to the depth of adjacent layers.

**Boundary Conditions Dialog Box**

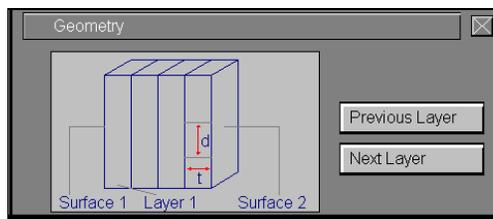
This dialog box contains the input fields for the boundary conditions. The boundary condition type is specified in the ‘Problem Description’ dialog box. For instance, if your input in the ‘Problem Description’ dialog box indicates fins on a surface, the fin resistance must be specified in the present dialog box.



The temperatures and/or film coefficients at the surfaces are also specified in this dialog box.

**Geometry Dialog Box**

This dialog box shows a schematic of the radial or slab wall being analyzed. Clicking on a label in the picture transfers you to the input field for the variable that corresponds to the label.



## RUNNING A SAMPLE PROBLEM

The steps involved in using the INSTED/Heat Conduction program are listed in this section.

### Procedure for performing a slab conduction analysis

1. Select the task of interest such as: Calculate 'Heat Flow Rate', 'Temp. Diff' or 'Thickness' from the 'Problem Description' dialog box
2. On the 'Problems Description' dialog box, enter the total count of the different materials involved in the system to be analyzed  
→ The thermal conductivity listboxes are automatically formatted to receive input for the number of material types entered
3. Enter the thermal conductivity of each material type in the 'Conductivity' listbox. Thermal conductivity may be obtained from INSTED/Database by clicking on the appropriate button
4. Enter the total number of layers also in the 'Problem Description' dialog box
5. Select each layer consecutively in the 'Thicknesses' listbox by clicking on it in the listbox. For each layer,
  - (a) Enter the thickness of the layer in this listbox
  - (b) Enter the number of segments in the current layer in the 'Data for Layer' dialog box usually located beside the 'Main' dialog box
  - (c) Enter the material type of each section consistent with your type declaration
  - (d) Enter the depth of each section. The total depth must be equal to the depth of a full layer
6. In the 'Problem Description' dialog box, describe the surface condition by clicking on "Fin on Surface" or "Radiation on Surface" buttons for the heat inflow and outflow surfaces, respectively
7. In the 'Layer Data' dialog box, select the boundary conditions for both surfaces
8. Enter the temperature or heat transfer coefficient values that may be required for the boundary condition type selected

9. Open the 'Surface Values' dialog box by clicking "Surface Values" button in the Main dialog box
10. Enter the values required in the dialog box
11. Click on the "Compute" button on the Main dialog box  
→ The results appear on the Main dialog box
12. Scroll through the results for each layers result by clicking "Previous Layer" or "Current Layer"
13. View the temperature profile by clicking on 'Plot Results' on the Main dialog box

### **Procedure for performing a radial conduction analysis**

1. Select the task of interest such as: Calculate 'Heat Flow Rate', 'Temp. Diff' or 'Thickness' from the 'Problem Description' dialog box
2. On the 'Problems Description' dialog box, enter the total count of the different materials involved in the system to be analyzed  
→ The thermal conductivity listboxes become automatically formatted to receive input for the number of material types entered
3. Enter the thermal conductivity of each material type under 'Conductivity'. Thermal conductivity may be obtained from INSTED/Database by clicking on the appropriate button
4. Enter the total number of layers also in the 'Problem Description' dialog box
5. Select each layer consecutively in the 'Thicknesses' listbox by clicking on it in the listbox. For each layer,
  - (e) Enter the beginning and ending radius of the layer
  - (f) Enter the material type of each layer on the 'Data for Layer' dialog box
  - (g) Enter the length of each layer. The length of all layers may not deviate in total by more than 20%
6. In the 'Problem Description' dialog box, describe the surface condition by clicking on "Fin on Surface" or "Radiation on Surface" buttons for the heat inflow and outflow surfaces, respectively
7. In the 'Data for Layer' dialog box, select the boundary conditions for both surfaces

8. Enter the temperature or heat transfer coefficient values that may be required for the boundary condition type selected
9. Open the 'Surface Values' dialog box by clicking "Surface Values" button in the Main dialog box
10. Enter the values required in the dialog box
11. Click on the "Compute" button on the Main dialog box  
→ The results appear on the Main dialog box
12. Scroll through the results for each layers result by clicking "Previous Layer" or "Current Layer"
13. View the temperature profile by clicking on 'Plot Results' on the Main dialog box

The sample problems are located in the 'Samples' subdirectory of the directories of each INSTED module.

## Sample Analysis

A sample problem best illustrates the procedures involved in running INSTED/Heat Conduction.

### Problem Statement

A masonry wall of a building consists of an outer layer of facing brick ( $k = 1.32$  W/m°C) 10 cm thick, followed by a 15 cm thick layer of common brick ( $k = 0.69$  W/m°C), followed by a 1.25 cm thick layer of gypsum plaster ( $k = 0.84$  W/m°C). An outside coefficient of  $30$  W/m<sup>2</sup>°C may be expected, and a coefficient of  $8$  W/m<sup>2</sup>°C is a reasonable value to use for the inner surface of a ventilated room. What will be the rate of heat gain, per unit area, when the outside air is at 35°C and the inside air is conditioned to 22°C? What will be the temperature of the plastic?

**Source**

Alan J. Chapman, 1987. "Fundamentals of Heat Transfer". Macmillan Publishing Co., NY, Page 46.

**Comments**

- Data in the source is used.
- Area is based on unity.
- Film coefficient is specified at the two surfaces; no fins, no radiation

**Name of Data**

The input data for the present sample problem are contained in the file  
\\insted30\cond\samples\Chapman.46.

**Instructions for running a sample file**

Sample files for INSTED/Conduction Program are located in of the installation drive. To run the sample file '*Chapman.46*', as an example, follow these procedures:

1. Click "Project Admin" on the Main dialog box
2. Click the "Load Project" button
3. Go to the subdirectory 'samples' by double-clicking on the 'samples' folder
4. Type " \*.\* " in the filename position and press Enter  
→ A list of the files present in the directory is displayed
5. Select the file '*Chapman.46*' and click on the "Open" button
6. Close the 'Project Admin' dialog by clicking its "Ok" button
7. Click "Compute" on the Main dialog box  
→ Results are presented on the Main dialog box
8. Scroll through the results for each layer by clicking "Previous Layer" or "Current Layer" above the 'Results' listbox
9. View the temperature profiles by clicking on "Plot Results" on the Main dialog box