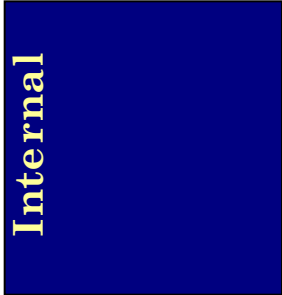
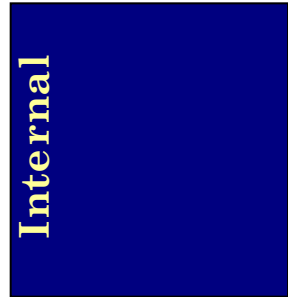


## **Contents**

<b>Introduction</b>	<b>3</b>
Input File Options	3
Inflow Types	3
Tasks	3
Required Data	3
May Be Required	4
Automatic Classification	4
Classification Criteria	5
Length Task	6
Turbulent Annulus	6
Annulus	6
<b>Boundary Condition Options</b>	<b>8</b>
Laminar, Entry	8
Laminar, Developed, Pipe	8
Laminar, Developed, Annulus	8
Turbulent, Developed, Low Prandtl No.	8
Other Turbulent Cases	8
Geometry Options for Laminar, Developed Flow	8
Output Results	9
Moody Chart	9
Required Data	9
Friction Factor Determination	10
INSTED's Definition of Friction Factor	11
<b>Required Geometric Data</b>	<b>12</b>
More on Annulus	14
Online Utilities	17



<b>Description of the Dialog Boxes</b>	<b>18</b>
The Project Administration Dialog Box	19
Problem Description Dialog Box	20
Fluid Properties Dialog Box	20
Pipe Attributes Dialog Box	21
Wall conditions Dialog Box	21
<b>Running a Sample Problem</b>	<b>22</b>



## INTRODUCTION

In this manual we discuss the capabilities that INSTED provides for internal flow analysis and the way to access these capabilities. The possible tasks with this module are discussed, as are the required data and overall details of the internal flow environment in INSTED. After you have provided the required data, the flow is locked into one of eight types. These flow types are discussed, as are the criteria for classification. Flow geometries are sketched, and the results from the internal flow analysis are listed. Finally, the friction factor (Moody chart) computation module is an integral part of internal flow analysis, and is therefore presented in this section. The three methods used for friction factor calculation are discussed.

### Tasks

- heat flow rate
- outlet temperature
- length of tube/duct

### Required Data

- inlet mean (bulk) temperature of fluid
- appropriate diameter of pipe/duct (available in INSTED/Database)
- wetted perimeter (available in INSTED/Database)
- inlet mass flow rate or velocity
- density of fluid (available in INSTED/Database)
- absolute viscosity (available in INSTED/Database)
- thermal conductivity (available in INSTED/Database)
- Prandtl number (available in INSTED/Database)

## May be Required

outlet temperature

- tube length (if task is not to find tube length)
- wall temperature
- absolute viscosity at the wall (for laminar, developed, moderate/high Prandtl number cases). Could be obtained from INSTED/Database

## Automatic Classification

After you have provided the required data (by direct input or from a database), INSTED locks the flow into one of the following eight types:

1. laminar, entry length, moderate/high Prandtl number
2. laminar, entry length, low Prandtl number (liquid metals)
3. laminar, developed, moderate/high Prandtl number
4. laminar, developed, liquid metals
5. turbulent, entry length, moderate/high Prandtl number
6. turbulent, entry length, low Prandtl number (liquid metals)
7. turbulent, developed, moderate/high Prandtl number
8. turbulent, developed, liquid metals

Options 2, 4, and 6 are not supported in the current version of INSTED

**Warning**

It is important to note that the above classifications and the method of solution, are due completely to the input data, and the only way to change the flow type is through the input data. Thus, if you have not made an estimate of your own, you will not know ahead of time the type of classification that you have.

**Classification Criteria**

1. flow is laminar if Reynolds number  $Re$  is less than 2300
2. hydrodynamic entry length, if distance  $x$  along the tube satisfies
  - $x < 10D$  (turbulent)
  - $x < 0.05ReD$  (laminar)
3. thermal entry length, if distance  $x$  along tube satisfies
  - $x < 10D$  (turbulent)
  - $x < 0.05ReD Pr$  (laminar), where  $D$ ,  $Re$ ,  $Pr$  are the diameter, Reynolds number, and Prandtl number, respectively
4. moderate/high Prandtl number implies  $Pr > 0.05$ . Liquid metal implies  $Pr \leq 0.05$
5. wall viscosity could change entry length/developed classification. This is relevant only to the laminar, non-annulus cases in INSTED. Discriminant is

$$(RePrD/L)^{1/3} (\mu/\mu_s)^{.14},$$

which is less than 2 for developed, laminar flow, but equal or greater than 2 for laminar, entry length cases. You will recognize  $(RePD/L)$  as the Graetz number and  $\mu$  and  $\mu_s$  as the value of the absolute viscosity of the fluid at bulk (mean) temperature and wall temperature, respectively. This

discriminant is used only for laminar (non-annulus) problems. For all other cases, the hydrodynamic entry length is used. This is not a serious limitation for turbulence, since both the hydrodynamic and thermal entry lengths are approximately equal. Concerning laminar, annulus cases, only fully-developed types are supported.

### **Length Task**

If the task is to calculate the length of the tube, INSTED internally forces the flow to be fully-developed. The laminar/turbulence, low/moderate-high Prandtl number classifications are not affected, however.

### **Turbulent, Annulus**

For this case, assume the inner and outer film coefficients are equal, model as a duct, with the appropriate diameter (discussed in the next subsection).

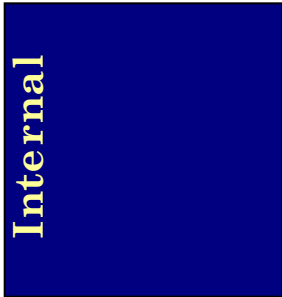
### **Annulus**

Use the hydraulic diameter,  $D_a - D_p$  for friction loss calculations, and the equivalent diameter

$$\frac{D_a^2 - D_p^2}{D_p}$$

for heat transfer calculations. Above,  $D_a$  and  $D_p$  are the inner diameter of outer tube and outer diameter of inner tube, respectively. For the laminar, developed case in the present Internal Flow module, the diameter ratio required (at some point during the analysis) pertains to the ratio of the actual diameters of the two pipes in the annulus. Annulus diameter in the laminar, developed case refers to the hydraulic diameter  $D_a - D_p$ . It is used for the heat transfer calculation. The measurement from which these data were obtained has accounted for the proper thermal-geometric data. However, this is not the case for turbulent flows, or in cases where an empirical equation is used to calculate Nusselt number (heat transfer coefficient) as a function of Reynolds number. Finally, you should note

that the wetted perimeter,  $P$ , of the annulus is  $\pi(D_a + D_p)$  for the friction calculation but  $\pi D_p$  for the heat transfer calculation. Flow area  $A$  in both cases is  $\pi/4[D_a^2 - D_p^2]$ . The expressions given previously for the hydraulic and equivalent diameters for the two cases may be obtained as  $4A/P$ .



## **BOUNDARY CONDITION OPTIONS**

### **Laminar, Entry**

- constant wall temperature

### **Laminar, Developed, Pipe**

- uniform surface heat flux, uniform peripheral temperature
- uniform surface heat flux
- uniform surface temperature

### **Laminar, Developed, Annulus**

- one surface insulated, other has constant heat flux
- uniform heat flux at both surfaces

### **Turbulent, Developed, Low Prandtl**

- constant surface heat flux
- constant temperature

### **Other Turbulent Cases**

- Analysis is fairly good with both constant temperature and constant heat flux

## **Geometry Options for laminar, developed flow**

- circular cylinder
- square or rectangles of arbitrary aspect ratios
- equilateral triangles
- hexagon
- horizontal ellipse
- cylindrical annulus



## Output Results

- Reynolds number
- flow area
- flow regime
- entry length - thermal and hydrodynamic
- tube length (if a task)
- friction factor
- pressure loss
- Nusselt number
- heat transfer coefficient
- bulk (mean) temperature,  $T_b$
- temperature difference  $\Delta T = T_w - T_b$
- log mean temperature difference,  $\Delta T_{lm}$
- outlet temperature
- heat flow rate based on  $\Delta T$  and  $\Delta T_{lm}$

## Moody Chart

The Moody environment is an important part of the Internal Flow analysis module. You will need to go through the friction factor calculation for most types of analysis. The exceptions are the laminar, fully-developed cases. Friction factor is needed for some of the model equations and for the pressure loss calculation.

For the purpose of the Moody chart, the actual diameters of circular pipes or hydraulic diameters of non-circular ducts should be used. Equivalent diameter or equivalent radius should not be used.

### Required data

- absolute roughness - may be obtained from INSTED/Database
- (hydraulic) diameter - passed directly by calling module
- Reynolds number - passed directly by calling module

For the Moody chart environment, you will only need to specify absolute roughness (in meters) or obtain it from INSTED/Database. The diameter and Reynolds number are based on data that you should have specified for the Internal Flow analysis.

### Friction factor determination

For laminar flows, the Darcy-Weisbach equation is used to compute the friction factor  $f$ . For turbulent flows, the friction factor is calculated using the Colebrook's (nonlinear) equation and Churchill's explicit formula. Colebrook's formula is supposed to give more accurate  $f$  values. But its nonlinearity could cause the solution to diverge or behave unpredictably. INSTED uses Newton - Raphson linearization. Churchill's formula is explicit, and has been known to give accurate predictions of  $f$  for a wide range of Reynolds numbers.

When possible, the three approaches mentioned above are used to compute  $f$ . You have four options: choose which one you consider the best. The fourth option is the friction factor that INSTED suggests to you, which you can accept or ignore. You are given an option because of the nonlinear dependence of  $f$  on Reynolds number and diameter.

For laminar flows, the Darcy-Weisbach formula is usually the best choice. In moderate to high Reynolds number flows, Churchill's formula often gives correct solutions. On Colebrook's approach, results are probably not reliable if the number of Newton-Raphson iterations is few (one) or many (over one hundred). Finally, the  $f$  value should not exceed 0.14 for turbulent flows or go below 0.005 for both turbulent and laminar flows, except for smooth pipes where the friction factor may be several orders of magnitude less than 1.

**INSTED Definition of Friction Factor**

INSTED uses the Darcy-Weisbach friction factor,  $f_{DW}$ , whereas other people use Fanning's definition of friction factor,  $f_F$ . It is well known that  $f_{DW}=4f_F$ . Note that the Darcy-Weisbach definition results essentially from dimensional analysis in the hydraulic literature and is the usual friction factor associated with piping system design in engineering. Fanning's definition is more commonly used in gas dynamic applications.

## REQUIRED GEOMETRIC DATA

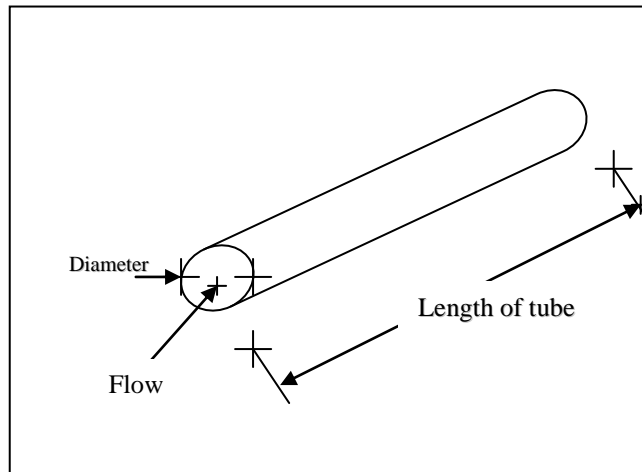
The following geometric data are usually required:

**Diameter:** This refers to the actual diameter, hydraulic diameter, or the equivalent diameter of the normal surface to fluid flow. For general non-circular geometries, use the hydraulic diameter for friction loss (pressure) and heat transfer calculations.

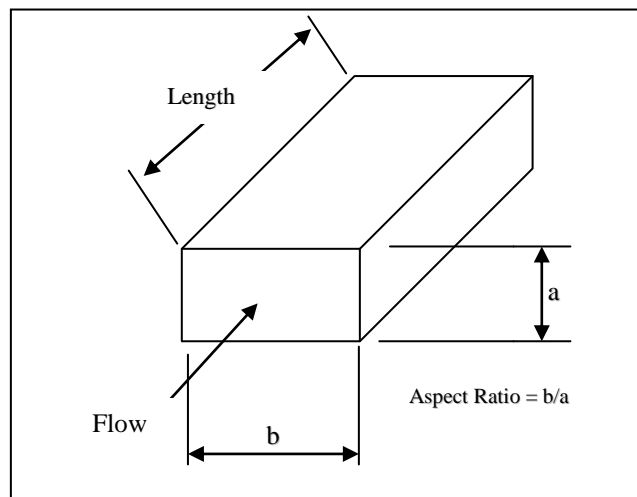
For an annulus, the hydraulic diameter should be used for the friction loss calculations and the equivalent diameter for the heat transfer calculations. Approximate diameter should be defined as  $4A/P$ , where  $A$  is flow area,  $P$  is wetted perimeter or heat transfer perimeter, for the friction loss and heat transfer calculations, respectively.

**Perimeter:** To accommodate a variety of inflow cross sections, you should provide the perimeter of the surface normal to the flow. Friction loss calculation uses the surface area per unit tube/duct length through which friction loss occurs. The perimeter for the heat transfer calculation is the surface area per unit length of tube/duct through which heat transfer takes place between the two regions of interest. As an example, for flow in an annulus, friction loss takes place on both the outer surface of inner tube and the inner surface of outer tube, whereas heat transfer between the fluid in the tube and the one in the annulus takes place only across the surface between the two fluids. The perimeter for friction loss and heat transfer will be quite different for an annulus and will depend on the ratio of diameters.

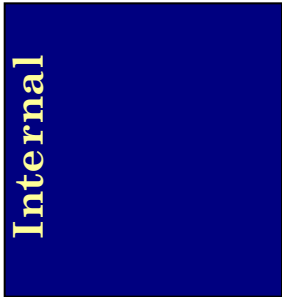
The figures below show some of the geometries supported by INSTED/ Internal Flow and Heat Transfer program and the required geometric data.

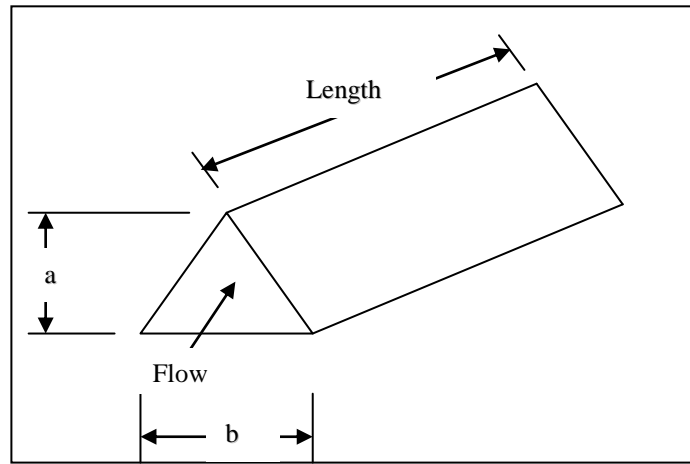


Circular Cylinder

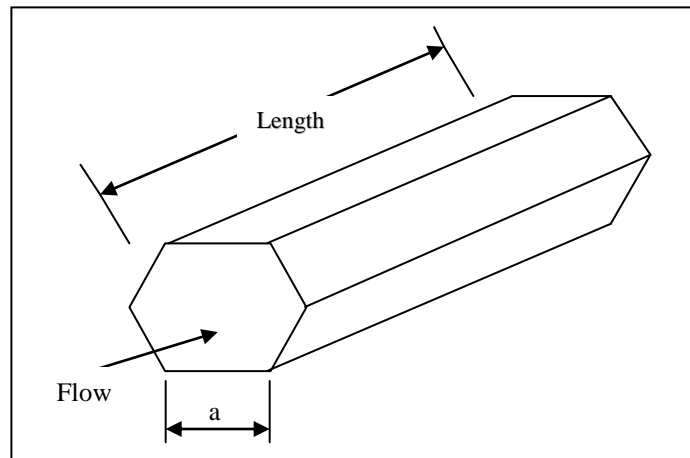


Rectangular Duct

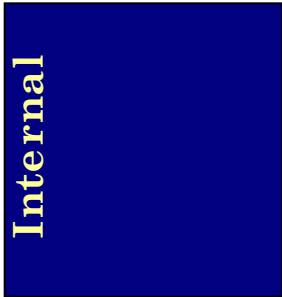


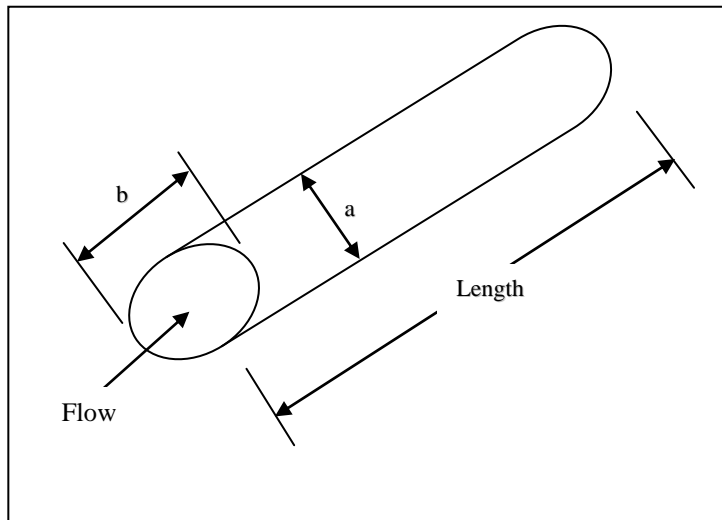


Equilateral Triangular Duct

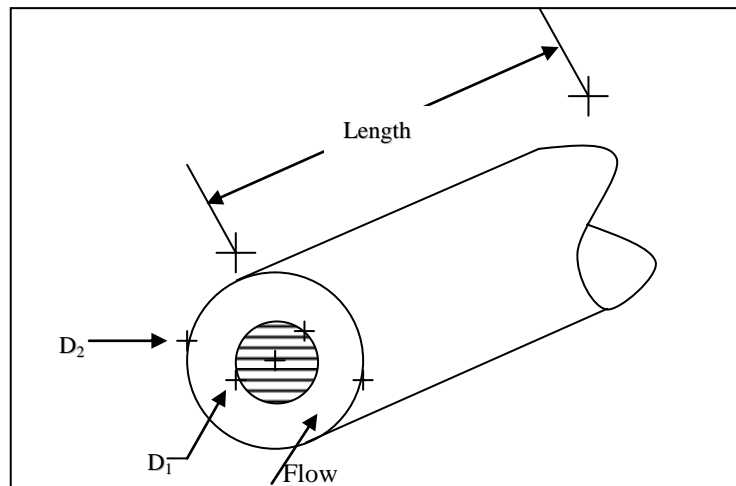


Hexagonal Duct

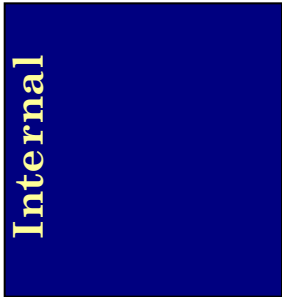




Elliptic Duct



Annulus



**More on Annulus**

You should be aware of the following situations in INSTED concerning annulus geometries.

1. For turbulent flows use the hydraulic diameter for the friction loss calculations and the equivalent diameter for the heat transfer calculations.
2. For laminar flows, only the fully-developed case is supported. Because of the way data is manipulated in INSTED, the hydraulic diameter should be specified for both the friction loss and the heat transfer calculations. Also, the Nusselt number data requires the specification of the ratio of the actual diameters of the tubes.
3. You do not know ahead of time whether the flow is laminar or turbulent, unless you have made some estimates. Therefore, you do not know what diameter to enter. This pertains only to Nusselt number (heat transfer coefficient) calculations, and not to friction loss calculations. If you assume a laminar flow and provide diameter information as in (2) above, and the flow turns out to be turbulent, that means you have provided the wrong data (diameter that may be too small since  $D_e > D_p$ ). If, on the other hand, you assume turbulence, and the flow turns out to be laminar, you might have used a diameter that is too large.
4. **Suggestion:** For annulus problems, make Reynolds number estimates, using both  $D_e$  and  $D_p$ . If the flow is laminar with  $D_e$ , it will be for  $D_p$ , and you should specify  $D_p$  for the diameter. If the flow is turbulent with  $D_e$ , you should accept  $D_e$  as the pipe diameter. The flow could be assumed to be turbulent for the 'gray region' in which flow is laminar for  $D_p$  but turbulent for  $D_e$ . If you do not perform the suggested estimation, then start out with  $D_e$  as the input diameter into the internal flow module, and change to  $D_p$  (if necessary), after INSTED has classified the flow as turbulent. To change, you merely recompute the solutions. You should be aware that the considerations above are



relevant only to annulus problems, and when the interest is in heat transfer.

### Online Utilities

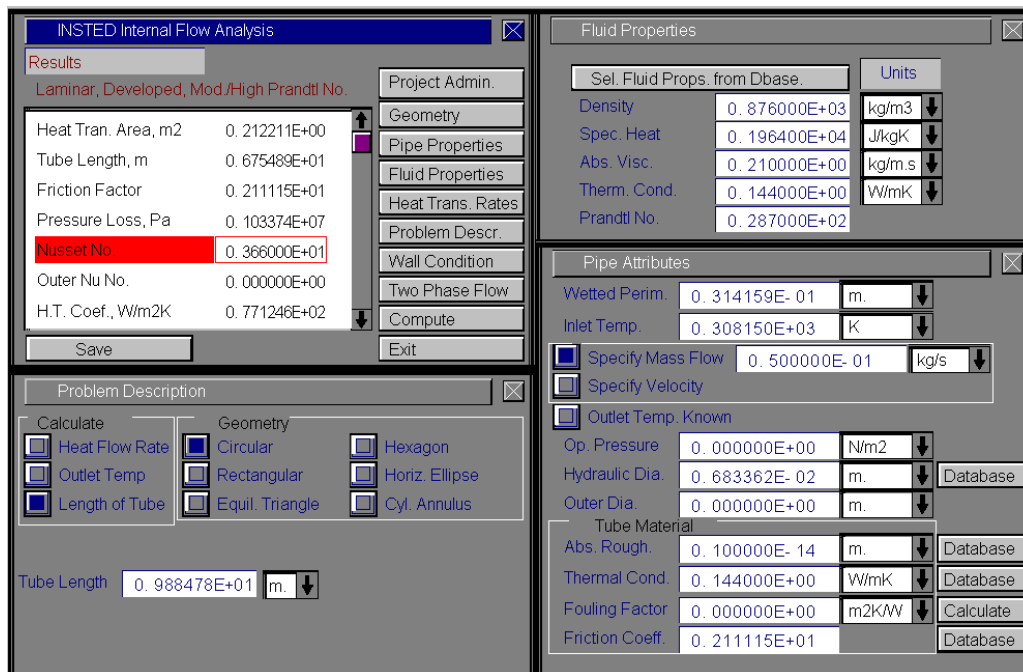
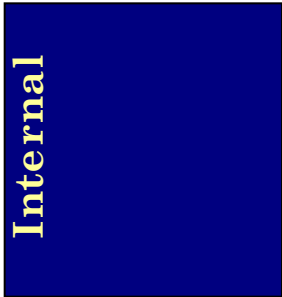
- unit conversion from British to SI
- database of thermophysical properties
- database of pipe schedules and dimensions
- database of film coefficient
- database of absolute roughness
- Moody chart (friction factor calculation)

Note also that the Internal Flow module is called by the INSTED heat exchanger programs.

Internal

## DESCRIPTION OF THE DIALOG BOXES

The program interface is presented below. The Main dialog box is the one on the top left hand of the screen at the start of the program. All the other dialog boxes are launched from the Main dialog box. The Main dialog box also includes the “Save” and the “Exit” buttons. The “Save” button is used to save a project which already has a file associated with it. Use the “Save Project” button from the ‘Project Admin’ dialog box to save a new project that does not yet have a file associated with it. This procedure is described in the next section. The “Exit” button is used to end the program.

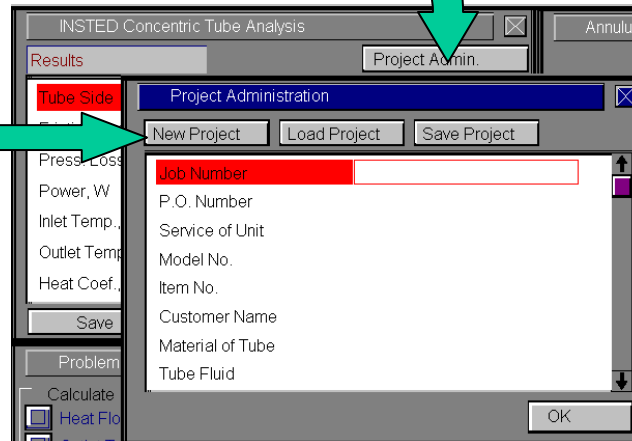


## The Project Administration Dialog Box

The functions of the 'Project Administration' dialog box includes loading an existing project, saving a new project, and starting a new project.

1. Click on "Project Admin" to launch the

2. Click "New Project" to start a new project



### Starting a New Project

Click the "New Project" button. All input fields are initialized and the memory is cleared in preparation for a new project.

### Loading an Old Project

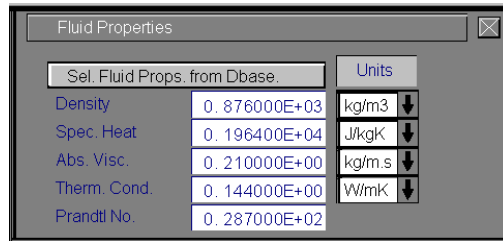
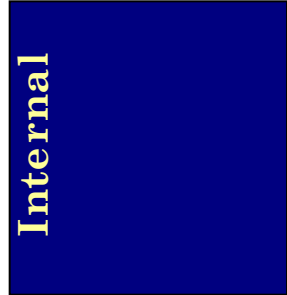
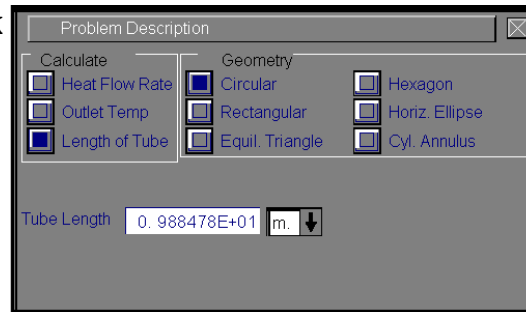
Click the "Load Project" button to load a project you previously saved or to load one of the sample problems supplied with the program. An operating system "Open File" dialog box appears. You may select the file you wish to load after navigating through the directories on your hard disk to the directory in which the file resides. In addition, you may need to type "\*.\*)" in the file name space and press the enter key to view all files in the directory.

### Saving a New Project

Click the "Save Project" button to save a new project. An operating system "Save As" dialog box appears. You will need to provide a name for the project file and select the directory in which you wish to save file.

## Problem Description Dialog Box

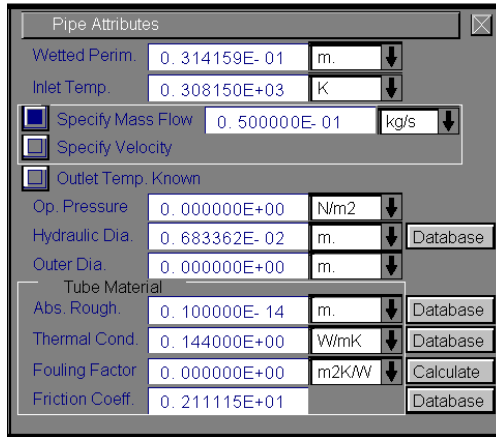
You specify the desired task in this dialog box. The geometry is also specified here as well as the tube length. A description of the geometries supported by the program and the input data required for each geometry has been presented in a previous section of this manual.



## Fluid Properties Dialog Box

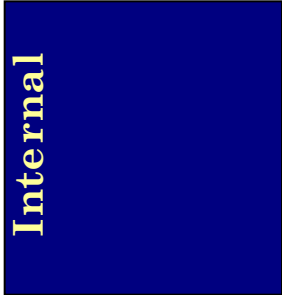
The thermophysical properties of the fluid are specified in this dialog box. This includes the density, specific heat, thermal conductivity and absolute viscosity.

The thermophysical properties may also be selected from INSTED/Database. The procedure for using the Database, including the addition of custom data is explained in the Database portion of this manual.



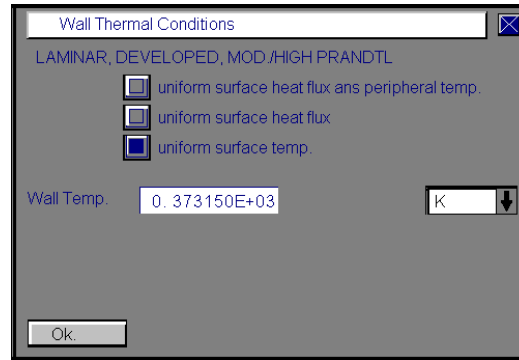
## Pipe Attributes Dialog Box

This dialog box contains the input fields for temperature, flow rate, fouling, roughness, tube dimensions and the thermal conductivity of the tube material. The value of the friction factor may be specified directly or obtained from the Database.



## Wall Conditions Dialog Box

The wall boundary conditions are entered in this dialog box. The values of some thermophysical properties at the wall such as viscosity or Prandtl number, may be required depending on the flow type.

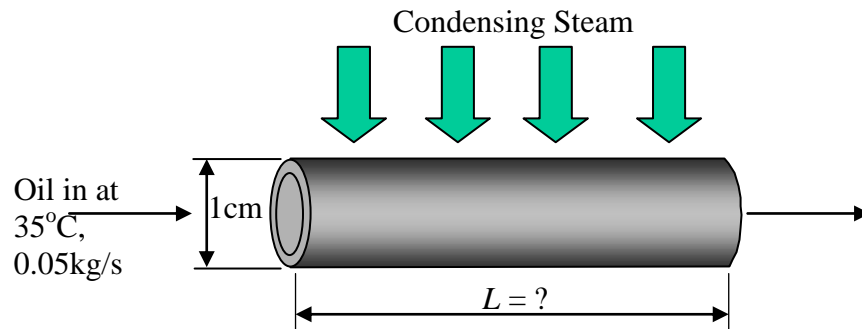


The need for the specification of wall conditions is usually only after the flow has been initially classified. As a result, this dialog box, if relevant, is usually presented after the computation has been initiated.

## RUNNING A SAMPLE PROBLEM

This section illustrates the use of INSTED/Internal Flow and Heat Transfer program. A sample problem best illustrates the procedures.

### Problem Statement:



Used engine oil can be recycled by a patented reprocessing system. Suppose that such a system includes a process during which engine oil flows through a 1-cm-ID, 0.02-cm-wall copper tube at the rate of 0.05 kg/s. The oil enters at 35°C and is to be heated to 45°C by atmospheric-pressure steam condensing on the outside. Calculate the length of tube required.

### Source:

Frank Kreith & Mark Bohn. 1993. Fifth Edition. Principles of heat transfer. West Educational Publishing, Boston. Page 393.

## Comments:

- In INSTED, choose the 'Tube Length' task and the option to enter mass flow rate (not velocity)
- From the source:  $C_p=1964$  J/kg K,  $\rho=876$  kg/m<sup>3</sup>,  $k=0.144$  W/m K,  $\mu=0.210$  N s /m<sup>2</sup>,  $Pr=28.7$
- Based on the input data, INSTED will classify this problem as Laminar, Developed, Moderate/High Prandtl number
- The wall has a fixed temperature. The source assumes this to be 373.15 K
- INSTED requires a friction factor. Absolute roughness is obtained from INSTED/database. The smooth pipe option is selected from this database. No additional data is required to obtain a friction factor. The friction factor value suggested at the Moody chart environment is used, which is 2.11

## Name of Data File

The input data for this sample problem are contained in the file  
\insted30\internal\samples\Krbohn.393.

## Instructions for running a sample file

Sample files for INSTED/Internal Flow and Heat Transfer program are located in the installation drive. To run the sample file "krbohn.393", follow these procedures:

1. From the 'Project Admin' dialog box, click on the "Load Project" button.
2. Locate the directory \insted30\internal\samples\ on the installation drive.
3. Type " \*.\* " in the filename position and press the Enter key.  
→ A list of the files present in the directory is displayed.
4. Select the file "krbohn.393" and click on the "Ok" button.
5. Click "Compute" on the Main dialog box to obtain the results.