

*Chapter 3***INSTED®/CFD SOLVER**

This chapter presents user information for the solution of the flow and heat transfer equations using the INSTED/CFD (2D) program. The physical problems are governed by the two-dimensional incompressible time-dependent or steady-state Navier-Stokes equations, with or without the energy equation. The basic capabilities of Soft2D are listed in this chapter and three classifications of CFD are suggested in order to meet a variety of research and application needs. The governing equations for the three cases are also presented in this chapter.

**3.1 Target Users**

INSTED/CFD (2D) is primarily aimed at people who have an appreciation for the engineering design process and need CFD capabilities to supplement proven engineering relations and empirical approaches. The ideal client for whom INSTED is intended recognizes that, while there might be a need to front-end supercomputers to extract physics from engineering systems, the uncertainty level in engineering design rarely warrants CFD simulation with hundreds of thousands of nodal points. You must also appreciate that, in the current state of CFD technology, the data needed for the design of many thermal/flow systems of industrial interest cannot be provided directly by CFD simulations. The flows of engineering interest are always turbulent and are not well understood to the point where turbulence models of some generality can be developed. Moreover, for engineering applications, the turbulence cannot be simulated on the computer, via the direct numerical simulation (DNS) approach, because of inadequate computer speed and memory. The conclusion is that, while CFD is important, the engineer who actually does the design recognizes that CFD by itself can sometimes be very limited. Thus, CFD must be integrated with proved (usually empirical) engineering relations. This is the philosophy in INSTED: to integrate CFD, empiricals, and other energy design tools, in a single computing environment for energy systems.

### 3.1.1 Industries

Industrial mechanical systems and components that will benefit from INSTED include those associated with heat exchangers of all sorts, fin arrays, pumps, fans, compressors, delivery systems (conduits), turbines, engines, pipe networks, and HVAC & R systems. Some industries that could benefit from the use of INSTED include:

- Heat exchanger manufacturing (fin analysis, condenser tube design, general thermal, flow, and performance analysis).
- Heating, ventilating, and air conditioning (HVAC) (thermo/fluid analysis heating, ventilation, and air conditioning systems, condenser design, fire research, flow inside enclosed spaces and around buildings, insulation analysis, domestic and industrial piping systems, furnace design, etc.).
- Power generation (heat exchangers, cooling towers, furnace design)
- Electronics and computers (electronic cooling, piping systems, flow and thermal analysis in cabinets and chassis, circuit board thermal analysis, cooling system component analysis, etc.)
- Alternative energy efforts (thermal analysis, heating load and cooling load calculations).
- Aerospace, automotive (air flow around vehicles, flow and thermal simulation of passenger compartments, coolant flow in engine blocks, design of the cross flow heat exchanger that constitutes the radiator of a car or domestic refrigeration systems, flow and heat transfer in ducts, manifolds, flow in fuel lines and tanks, etc.).
- Chemical and food processing industries (pasteurization and sterilization of liquid foods, thermal processing, cooling process design, oven design, heat exchanger design, piping networks, etc.).
- Petroleum (piping systems for petroleum delivery, flow and thermal analysis, heat exchanger design).
- Nuclear engineering (research on cooling system design for high heat flux applications, piping analysis, heat exchangers and general thermal analysis).
- Instrumentation and control (simulation of flow and thermal fluids around sensing devices, vortex shedding studies, analysis of flow and pressure loss in valves, gates, and control devices).

### 3.2 Basic capabilities of INSTED/CFD (2D)

Some of the basic capabilities of INSTED/CFD (2D) include:

- Heat Conduction
- Natural Convection
- Forced Convection
- Mixed Convection
- Conjugate Heat Transfer
- Steady State Flows
- Time-Dependent Flows
- Any Complex Geometry (INSTED/CFD (2D) is based on the finite element method)
- Spatially-Dependent Thermophysical Properties
- Rotating Flows
- Dimensional Equations
- Non-Dimensional Equations Appropriate for Forced/Mixed and Natural Convection
- Automatic Time Step Size Selection
- Up to Seven Time Integration Schemes
- Time Integration Error Control
- Incompressible Flows
- Laminar and Turbulent Flows
- Display of the Time-History Data for Selected Points

### 3.3 Types of CFD Analysis Supported by INSTED

In the schemes described below,  $(x, y)$  are the distances (in meters, as an example) in the  $x$ - and  $y$ - coordinate directions.  $t$  is time,  $\mathbf{u} = (u, v)$  are the velocities (m/s) in the  $(x, y)$  directions,  $p$  is pressure (N/m<sup>2</sup>), and  $T$  is temperature (K).  $\rho_0$ ,  $C_p$ ,  $\mu$ ,  $\beta$ ,  $g_x$ ,  $g_y$ ,  $k$ ,  $\alpha$ , and  $q'''$  are typically the reference density (kg/m<sup>3</sup>), specific heat at constant pressure (J/kgK), absolute viscosity (Ns/m<sup>2</sup>), thermal expansion coefficient (1/K), the  $x$ - and  $y$ -components of the gravity vector (m/s<sup>2</sup>), thermal conductivity (W/mK), thermal diffusivity (m<sup>2</sup>/s), and volumetric heat flow rate (W/m<sup>3</sup>). Typically, the units provided above are examples and the equations themselves assume no units except those consistently defined by you.

#### Type 1: Physical CFD, Free Convection

This class is intended for flows dominated by natural convection. **The variables are non-dimensionalized with thermal diffusion scales.** Equations can be solved in either the inertial or rotating frames and buoyancy can be due to the normal gravity field and/or centrifugal force field. Note that all the variables (such as velocities, temperature, and heat flow rate) and thermophysical properties (thermal conductivity and viscosity) must

be non-dimensional. Furthermore, initial and boundary conditions must be given in terms of the non-dimensional forms.

The following non-dimensionalization scheme is used:

$$x = x' L; \quad y = y' L; \quad t = t' \frac{L^2}{\alpha}; \quad u = u' \frac{\alpha}{L}; \quad v = v' \frac{\alpha}{L}$$

$$p = p' \frac{\rho_0 \alpha^2}{L^2}; \quad T = T_0 + \overline{\Delta T T'}; \quad \Delta T = T_w - T_0$$

In the above equations, the primed quantities are non-dimensional quantities;  $T_w$  and  $T_0$  usually denote a hot wall and cold wall temperature, respectively. When the primes are suppressed, the non-dimensional equations can be written as:

$$\nabla \cdot \mathbf{u} = 0 \tag{1}$$

$$\begin{aligned} \frac{\partial u}{\partial t} + \mathbf{u} \cdot \nabla u - v \text{Re}_r \text{Pr} + Ra_r \text{Pr} T_x = \\ - \frac{\partial p}{\partial x} + \text{Pr} \nabla \cdot \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^*)_x + Ra \text{Pr} \sin\left(\frac{t \text{Re}_r \text{Pr}}{2}\right) T \end{aligned} \tag{2}$$

$$\begin{aligned} \frac{\partial v}{\partial t} + \mathbf{u} \cdot \nabla v - u \text{Re}_r \text{Pr} + Ra_r \text{Pr} T_y = \\ - \frac{\partial p}{\partial y} + \text{Pr} \nabla \cdot \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^*)_y + Ra \text{Pr} \cos\left(\frac{t \text{Re}_r \text{Pr}}{2}\right) T \end{aligned} \tag{3}$$

$$\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T = \nabla \cdot k \nabla T + H \tag{4}$$

In the above equations, an asterisk denotes a transpose. The non-dimensional parameters in the equations are defined as follows:

$$Re_r = 2\Omega L^2/\nu \quad - \text{Rotational Reynolds number}$$

$$Ra_r = L\Omega^2 \beta \overline{\Delta T} L^3 / (\nu \alpha) \quad - \text{Rotational Rayleigh number}$$

$$Pr = \nu/\alpha \quad - \text{Prandtl number}$$

$$\eta = \mu_{\text{dim}}/\mu_0 \quad - \text{non-dimensional viscosity}$$

$$Ra_g = g\beta \overline{\Delta T} L^3 / (\nu \alpha) \quad - \text{Gravitational Rayleigh number}$$

$$k = k_{\text{dim}}/k_0 \quad - \text{non-dimensional thermal conductivity}$$

$$H = q'' L^2 / k \overline{\Delta T} \quad - \text{non-dimensional heat generation}$$

- Buoyancy resulting from a centrifugal force field is included in the equations.
- Initial and boundary conditions must be specified in non-dimensional form.
- For temperature, the user must first determine a high temperature (dimensional) and a low temperature to use in defining  $\Delta T$ .
- The pressure that is calculated is the one relative to the hydrostatic pressure from the balance of the pressure gradient, the normal gravity field, and the centrifugal force field. That is, the calculated pressure results wholly from fluid motion. Using familiar notations, the hydrostatic pressure (in dimensional form) is obtained from

$$\nabla p = \rho \mathbf{g} - \rho \boldsymbol{\Omega} \times \boldsymbol{\Omega} \times \mathbf{r},$$

where  $\boldsymbol{\Omega}$  is the rotation vector

- $Re_r = Ra_r = 0$  for solution in the inertial frame.
- In the present code, the Coriolis terms  $Re_r Pr (\boldsymbol{\Omega} \times \mathbf{u})$  in (2) and (3) have no effects because the flow is two-dimensional. However, the parameter  $Re_r$  has significance through its appearance in the argument of the normal gravity term.
- It is assumed in Type 1 CFD problems that the gravity vector is parallel (antiparallel) to one of the two coordinate axes. Type 2 and Type 3 problems allow an arbitrary orientation of the gravity vector.

Type 2: Physical CFD, Forced/Mixed Convection

This class is intended for flows dominated by forced convection. **Forced convection scales are used for scaling the velocities and pressure**, meaning that free convection effects are assumed to be of secondary importance. Equations can be solved in either the inertial or rotating frames. Note that all the variables (such as velocities, temperature, heat flow rate) and thermophysical properties (thermal conductivity and viscosity) must be non-dimensional. Furthermore, initial and boundary conditions must be given in terms of the non-dimensional forms.

The following non-dimensionalization scheme is used:

$$x = x' L; \quad y = y' L; \quad t = t' \frac{L}{U_0}; \quad u = u' U_0; \quad v = v' U_0$$

$$p = p' \rho_0 U_0; \quad T = T_0 + \overline{\Delta T} T'; \quad \overline{\Delta T} = T_w - T_0$$

In the above equations, the primed quantities are the non-dimensional quantities while  $T_w$  and  $T_0$  usually denote a hot wall and cold wall temperature. When the primes are suppressed, the non-dimensional equations can be written as:

$$\nabla \cdot \mathbf{u} = 0 \tag{5}$$

$$\frac{\partial u}{\partial t} + \mathbf{u} \cdot \nabla u - \nu \text{Re}_{rf} + Ra_{rf} x = -\frac{\partial p}{\partial x} + \frac{1}{\text{Re}} \nabla \cdot \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^*)_x - Ra_f g_x T \tag{6}$$

$$\frac{\partial v}{\partial t} + \mathbf{u} \cdot \nabla v + u \text{Re}_{rf} - Ra_{rf} y = -\frac{\partial p}{\partial y} + \frac{1}{\text{Re}} \nabla \cdot \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^*)_y - Ra_f g_y T \tag{7}$$

$$\text{Re Pr} \left( \frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) = \nabla \cdot k \nabla T + H \tag{8}$$

In the above equations, an asterisk denotes a transpose. The non-dimensional parameters in the equations are as follows:

$$Re_{rf} = 2\Omega L/U_0$$

$$Ra_{rf} = (\Omega L/U_0)^2$$

$$Re = U_0 L/\nu \quad - \text{Reynolds number}$$

$$\eta = \mu_{dim}/\mu_0 \quad - \text{non-dimensional viscosity}$$

$$Ra_f = \beta \overline{\Delta T} L / U_0^2 \quad - \text{Gravitational Rayleigh number}$$

$$Pr = \nu/\alpha \quad - \text{Prandtl number}$$

$$k = k_{dim}/k_0 \quad - \text{non-dimensional thermal conductivity}$$

$$H = q''' L^2 / k \overline{\Delta T} \quad - \text{non-dimensional heat generation}$$

In addition, there are the parameters  $g_x$  and  $g_y$ , which are the components of the normal gravity in the  $x$ - and  $y$ - directions respectively, with units of acceleration. Note that  $Ra_f$  and  $g_x$  (or  $g_y$ ) are each dimensional but the product  $Ra_f g_x$  is non-dimensional. Note also that  $Ra_f g$  is the Richardson number. The product  $Re Pr$  is the Peclet number.

- Bouyancy resulting from a centrifugal force field is neglected.
- Initial and boundary conditions must be specified in non-dimensional form.
- For temperature, you must first determine a high temperature (dimensional) and a low temperature to use in defining  $\Delta T$ .
- The pressure that is calculated,  $p_{calc}$ , is the one relative to the hydrostatic pressure distribution assuming no rotation. Centrifugal pressure is included in the calculation. The hydrostatic pressure, assuming no rotation, is obtained from the dimensional equation:

$$\nabla p_h = \rho g$$

Thus, guage pressure,  $p_g$ , is given by  $p_g = p_{calc} + p_h$ , while absolute pressure is  $p_g + p_{atmos}$ , with  $p_{atmos}$  as the atmospheric pressure

- Note that, the Coriolis terms in (6) and (7) have no effect because the flow is two-dimensional. Also, the parameter  $Re_{rf}$  does not appear in the argument of the normal gravity term, because of the assumption that forced convection is dominant.

Type 3: Engineering CFD, General Calculations

This class is intended for calculations in which the actual values of the solution variables are of interest. **Equations are in dimensional form and are solved only in the inertial frame.** The dimensional equations for this case are:

$$\nabla \cdot \mathbf{u} = 0 \tag{9}$$

$$\frac{\partial u}{\partial t} + \mathbf{u} \cdot \nabla u = -\frac{1}{\rho_0} \frac{\partial p}{\partial x} + \frac{1}{\rho_0} \nabla \cdot \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^*)_x - \beta g_x T \tag{10}$$

$$\frac{\partial v}{\partial t} + \mathbf{u} \cdot \nabla v = -\frac{1}{\rho_0} \frac{\partial p}{\partial y} + \frac{1}{\rho_0} \nabla \cdot \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^*)_y - \beta g_y T \tag{11}$$

$$\rho C_p \left( \frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) = \nabla \cdot k \nabla T + q''' \tag{12}$$

- Rotating flow problems can only be solved in the inertial frame. This prevents the need to transform data before use.
- Initial and boundary conditions must be specified in dimensional form.
- The pressure that is calculated,  $p_{calc}$ , is the one relative to the hydrostatic pressure distribution, from:

$$\nabla p_h = \rho g$$

Thus, gauge pressure,  $p_g$ , is given by  $p_g = p_{calc} + p_h$ , while the absolute pressure is  $p_g + p_{atmos}$ , with  $p_{atmos}$  as the atmospheric pressure

### 3.3.1 Species Transport Equations

For all CFD types, species equations are solved in the form:

$$P_1 \left( \frac{\partial(\rho\phi_i)}{\partial t} + \frac{\partial(\rho u_j \phi_i)}{\partial x_j} \right) = \frac{\partial(\rho P_2 \phi_i)}{\partial x_j} + P_3,$$

where the subscript "i" denotes a particular species. You should provide the proper values for the three parameters  $P_1$ ,  $P_2$ , and  $P_3$  for each species. Currently, INSTED®/CFD (2D) will solve for up to 20 species, depending on the type of license you have to use INSTED®. Only Dirichlet and zero flux boundary conditions are currently supported for species calculations.

### 3.3.2 Axisymmetric Models

INSTED®/CFD (2D) is also capable of calculating flow, heat transfer, and species distribution in cylindrical and axisymmetric problems. The axial (z-) coordinate direction should be interpreted as the x-coordinate and the radial coordinate direction is interpreted as the y-coordinate. **It is imperative that you use this convention.**

If you assume that the azimuthal velocity and the gradient of the azimuthal direction are zero, then the governing equations can be written as follows. Note that CFD Type 1 is assumed for this illustration.

#### Continuity

$$\frac{1}{r} \frac{\partial}{\partial r} (\rho r u_r) + \frac{\partial}{\partial z} (\rho u_z) = 0$$

**Momentum**

$$\rho \left( \frac{\partial u_z}{\partial t} + u_r \frac{\partial u_z}{\partial r} + u_z \frac{\partial u_z}{\partial z} \right) = \frac{1}{r} \frac{\partial}{\partial r} (r \sigma_{rz}) + \frac{\partial \sigma_{zz}}{\partial z}$$

$$\rho \left( \frac{\partial u_r}{\partial t} + u_r \frac{\partial u_r}{\partial r} + u_z \frac{\partial u_r}{\partial z} \right) = \frac{1}{r} \frac{\partial}{\partial r} (r \sigma_{rr}) - \frac{1}{r} \sigma_{\theta\theta} + \frac{\partial \sigma_{rz}}{\partial z}$$

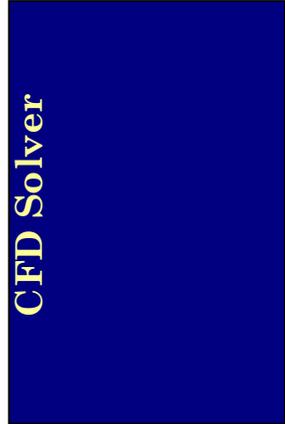
$$\sigma_{rr} = -p + 2\mu \frac{\partial u_r}{\partial r}, \sigma_{\theta\theta} = -p + 2\mu \frac{u_r}{r}, \sigma_{rz} = \mu \left( \frac{\partial u_r}{\partial z} + \frac{\partial u_z}{\partial r} \right), \sigma_{zz} = -p + 2\mu \frac{\partial u_z}{\partial z}$$

**Temperature**

$$\rho c_p \left( \frac{\partial T}{\partial t} + u_r \frac{\partial T}{\partial r} + u_z \frac{\partial T}{\partial z} \right) = \frac{1}{r} \frac{\partial}{\partial r} \left( kr \frac{\partial T}{\partial r} \right) + \frac{\partial}{\partial z} \left( k \frac{\partial T}{\partial z} \right)$$

**Species**

$$\frac{\partial}{\partial t} (\rho \phi_i) + \frac{\partial}{\partial r} (\rho u_r \phi_i) + \frac{\partial}{\partial z} (\rho u_z \phi_i) = \frac{1}{r} \frac{\partial}{\partial r} \left( \rho \alpha_i r \frac{\partial \phi_i}{\partial r} \right) + \frac{\partial}{\partial z} \left( \rho \alpha_i \left( \frac{\partial \phi_i}{\partial z} \right) \right)$$



**3.4 Turbulence Models in INSTED**

The turbulence models supported in INSTED include

1. Zero-equation model (Baldwin and Lomax)
2. Zero-equation model (Smagorinsky)
3. Low- and high-Reynolds number  $k-\varepsilon$  model
4. RNG  $k-\varepsilon$  model
5.  $k-\varepsilon$  model of Abe/Kondoh
6.  $k-\varepsilon$  model of Kolmogorov wall scales
7. A realizable  $k-\varepsilon$  model
8. Anisotropic, nonlinear algebraic stress model (ASM).

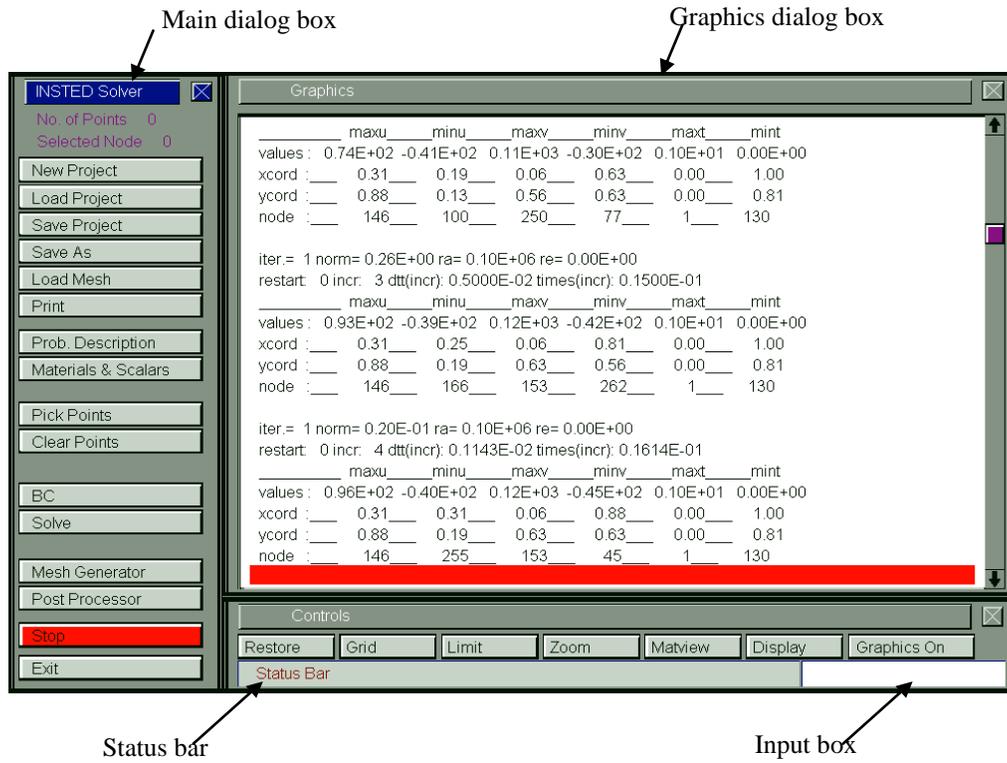
The details of the mathematical formulation of each turbulence model is discussed in the theoretical manual. Note however, that the current version of INSTED/CFD (2D) is based entirely on model 4 above: RNG  $k-\varepsilon$  model

Chapter 4

**User Instructions for CFD Analysis**

This chapter presents the instructions for running a CFD Analysis using INSTED/CFD (2D). Natural convection in a square cavity, introduced in 2.14 will be used to illustrate the CFD solution steps. In this chapter, the conditions ‘YES’ and ‘ON’ are used interchangeably, to mean the same thing. Similarly, ‘NO’ and ‘OFF’ are used synonymously.

**4.1 The INSTED Soft2D Interface**



The INSTED solver consists of six dialog boxes - the Main dialog box located at the left side of the screen at the launch of the program, the Controls, Graphics, Problem Description, Materials and Scalar, and the Limit dialog box.

## 4.1.1 The Graphics dialog box

This is the dialog that displays the grid. This dialog box also displays the log file interactively while solution is in progress. The user may toggle between text (information) mode and mesh graphics display mode by clicking the **Graphics On** or **Graphics Off** button on the 'Controls' dialog box.

## 4.2 Performing a CFD Analysis in INSTED

The steps involved in running a CFD sample problem are as follows:

- Load a mesh file
- Specify solver and flow parameters
- Specify material properties
- Select sample points (if needed) for storing time-history data
- Solve the problem

## 4.3 Loading a Mesh File

Ensure the mesh for the problem has been generated; with boundary conditions properly specified at the various boundaries of the model. The procedures for doing this have been discussed in section 2.14.

A mesh created from INSTED Mesh Generator will be automatically loaded if INSTED/CFD (2D) Solver was invoked from the mesh generator. Otherwise, a mesh will need to be loaded by clicking the Load Mesh button on the Main dialog box. An operating system 'Open File' dialog box is presented. Follow operating system procedures for selecting the required file. The file must be called *mesh.dat* and must have an accompanying *bound.dat* and *xnuslt.dat* in the same directory. These three files are generated internally by INSTED/CFD (2D) Mesh Generator. You must provide the files if you do not use INSTED to generate your mesh.

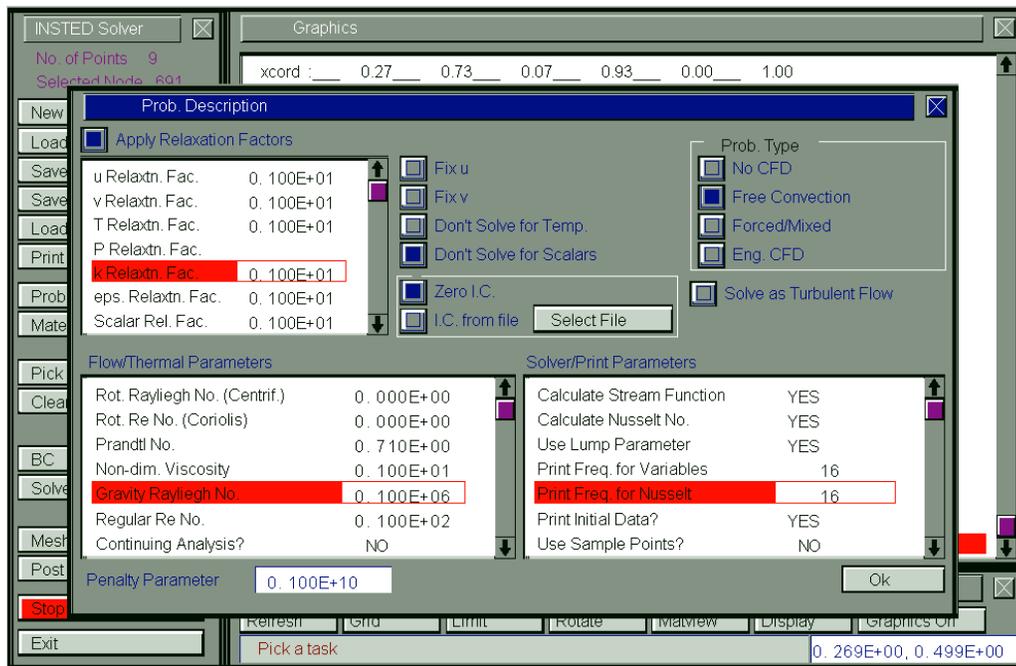
#### 4.4 Specifying Problem Type, Solver, and Flow Parameters

All solver, flow, and print parameters are specified in the 'Prob. Description' dialog box, as described below.

##### 4.4.1 The Problem Description dialog box

The problem description dialog box may be obtained by clicking the **Problem Description** button on the Main dialog box.

This dialog is used to collect all input parameters necessary to complete a solution of a meshed model.



## 4.4.2 Description of menu items on the Problem Description dialog box

### 4.4.2.1 Problem type

**No CFD:** No CFD simulation will be done if you select this option. Control will be returned to the calling process. Do not select this option for the sample problem.

**Physical CFD, free convection:** Select this option for Type 1 Problems as described in Section 3.4. This is the input for the sample problem.

**Physical CFD, forced/mixed convection:** Select this option for Type 2 problems as described in Section 3.4. Do not select this option for the sample problem.

**Engineering CFD:** Select this option for Type 3 problems. Do not select this option for the sample problem.

### 4.4.2.2 Flow/Thermal Parameters

**Centrifugal parameter ( $Ra_r$  or  $Ra_{rt}$ ):** This refers to the Rotational Rayleigh number or Rotational Rayleigh number in equations (1) through (4) for problem Type 1 or (5) through (8) for problem Type 2. The parameter is 0.0 for the sample problem.

**Coriolis parameter ( $Re_r$  or  $Re_{rt}$ ):** This refers to the rotational Reynolds number in equations (1) through (4) for problem Type 1 or (5) through (8) for problem Type 2. The parameter is 0.0 for the sample problem.

**Prandtl Number:** This refers to the parameter  $Pr$  in equations (1) through (4) or (5) through (8). This parameter is 0.71 for the sample problem.

**Non-dimensional Viscosity:** This refers to the parameter  $\eta$  in equations (1) through (4) or (5) through (8). The reference viscosity for non-dimensionalization is the dynamic viscosity of the fluid evaluated at the reference temperature. The viscosity is assumed constant in the sample problem. Therefore, the value of 1.0 is used for the dimensionless viscosity.

**Normal gravity parameter ( $Ra$  or  $Ra_f$ ):** This refers to the gravitational Rayleigh number in equations (1) through (4) for problem Type 1 or (5) through (8) for problem Type 2. A value of  $10^4$  is used for the sample problem.

**Regular Reynolds Number:** This refers to the parameter  $Re$  in equations (5) through (8). This is the familiar Reynolds number in isothermal flow situations. This data is not required for problem Types 1 and 3. This parameter is 0.01 for the sample problem.

**Continuation Indicator:** A continuation indicator is used to enhance convergence to steady state when a critical flow parameter has a large value. This parameter actually applies to  $Ra$  (problem Type 1) and  $Re$  (problem Type 2), but  $Ra$  will be used for illustration. In a continuation procedure, calculation is started with a small (user-input)  $Ra$  value to obtain a converged solution. INSTED/CFD (2D) then increases  $Ra$  and uses the converged solutions as initial conditions for the higher  $Ra$  value, to seek convergence for the new  $Ra$  value. This is continued until the desired  $Ra$  is obtained. **The indicator is set (i.e. YES) if a continuation process is implied.** In this case, INSTED/CFD (2D) starts the computations with the input value of  $Ra$  until a convergence criterion (actually an 'intermediate' convergence criterion) is met. The converged  $Ra$  value is then increased (2.5 fold). Convergence for the new  $Ra$  is sought, using again, the intermediate convergence criterion. When convergence is achieved, the new  $Ra$  value is again increased 2.5 fold. The process is continued, convergence being ensured during the intermediate  $Ra$  calculations, until the  $Ra$  value is equal to the desired final value. A more stringent convergence criterion must then be imposed on the final  $Ra$  calculation. A continuation process is used for high  $Ra$  values to forestall problems with poor initial guess solutions. The linearization method used (Newton-Raphson) is particularly sensitive to a poor initial guess. Note: For Type 2 problems, it is important that you replace  $Ra$  with  $Re$  in the discussions presented above. The current version of INSTED/CFD (2D) does not support a continuation process for Type 3 problems. For the latter, use the transient option with the error control if you have a strong flow, and allow the code to choose  $\Delta t$ . No continuation process is used in the sample problem. This input is left at its default state of 'NO'.

**Final Rayleigh Number:** This refers to the final  $Ra$  or  $Re$  in the continuation process discussed above. No continuation process is used for the sample problem therefore this input data is not relevant. The final  $Ra$  is set to 0.

**Axisymmetry Indicator:** Toggle the radio button to ‘YES’ for axisymmetric problems and ‘NO’ otherwise. Remember that  $x$ - is the axial direction,  $y$ , the radial direction in axisymmetric problems. The state of the radio button is ‘NO’ for the sample problem.

**Convergence Weighting Function  $\lambda$ :** This refers to the weighting factor associated with a relaxation factor. This input is not used in the present version of the code. Moreover, relaxation factors that depend on the variables (see the ‘Prob. Description’ dialog box) have rendered the present input redundant. Specify relaxation factors via the ‘Prob. Description’ dialog box.

#### **4.4.2.3 Specification of Initial Condition**

**Zero Initial Condition Indicator:** Set this radio button to ‘NO’ if the initial condition is to be set internally in INSTED/CFD (2D), and ‘YES’ if you wish to specify your own initial conditions. For the latter case, follow your operating system’s file loading protocol to load the file containing the (arbitrary) initial conditions. INSTED/CFD (2D) reads the contents of the file as follows:

```

READ (indev, *) NUMBER, TIMENOW
DO 2 I=1, NUMBER
    READ (indev, *) N, XORD(N), YORD(N), (U(J,N), J=1, 3)
    READ (indev, *) N, (SCALAR(J,N), J=1, NUMSCALAR)
    READ (indev, *) N, RKINETIC(N), EPSILON(N)
2 CONTINUE
    
```

Here, NUMBER is the total number of nodes in the mesh including the nodes at the centroid of the nine-node elements. TIMENOW is the total time of integration at which this initial condition was generated. See the next item below. N is a particular node, while U(J,N) are the variables  $u$ ,  $v$ , and  $T$  at node N. SCALAR(J,N) is the value of the  $J^{\text{th}}$  scalar at node N. NUMSCALAR is the number of scalars in the model. RKINETIC(N) and EPSILON(N) are the values of the turbulence kinetic energy and the turbulence dissipation rates respectively. The data for the nodes on the boundaries of all elements must be supplied first, followed by the data for the centroids of the elements. If NUMNP is the number of nodes in the grid, excluding the nodes at the centroids of all elements, the running variable N will then refer to the (NUMBER - NUMNP)th element after the NUMNP data for the boundary nodes have been read. The grid implied here must be consistent with that from the INSTED/CFD (2D) Mesh Generator. The mesh generator

assigns NUMNP to the total number of nodes in the model excluding those at the element centroids. The total number of centroid nodes in the model is equal to the number of elements, NUMEL. Therefore, NUMBER = NUMNP + NUMEL.

The foregoing implies that initial conditions that you specify must have been converted into nodal values for  $u$ ,  $v$ , and  $T$ . Only divergence-free initial conditions can give physically-meaningful transient solutions. Note that INSTED/CFD (2D) generates a file 'arbtr.dat' at every print step during analysis. This file could be used to 'restart' the calculations. That is, it can be used as an initial condition file. The format of this file is compatible with that given above. For the sample problem, INSTED/CFD (2D) has been asked to set the initial conditions, which are zero for  $u$ ,  $v$ , and  $T$  at all nodal points. The state of the radio button is 'OFF' or 'NO' for the sample problem.

**Total Previous Integration Time:** This variable is associated with the user-supplied initial condition discussed above. It represents the total integration time in previous calculations of the problem. Therefore, the variable is useful in cases where you want to resume the simulation of an initial-value problem that was prematurely terminated in a previous run when calculations had not reach a desired steady state condition. You now want to continue the simulation from where it was terminated. Strictly speaking, this variable is needed for terms that depend explicitly on the total time of integration (such as  $RaPr \sin(tRe,Pr/2)$  in equations (2) and (3)). You must be aware of the effect of initial conditions on the solution of a problem. For an initial condition that does not satisfy the steady state Navier-Stokes equations, information on  $\partial u/\partial t$  must be provided at  $t = 0$ , and must ideally be incorporated into INSTED/CFD (2D) (true for any solver) to prevent unphysical solutions (that satisfy Navier-Stokes) at short time. As you can see, this involves some risks and you will be better off using an initial condition that satisfies the steady Navier-Stokes equations. For the present problem this input is irrelevant, because of the use of zero initial conditions. The variable is set to the default value of zero. **For all simulations, disregard this input if you would like to keep the parameter TIMENOW that was loaded in the item above.** This input is irrelevant for the sample problem.

#### 4.4.2.4 Solver/Print Parameters

**Stream Function Indicator:** Set this item to 'YES' to calculate the stream function and 'NO' otherwise. The button is set to 'ON' for the sample problem.

**Nusselt Number Indicator:** Toggle to 'ON' to calculate Nusselt number and 'OFF' otherwise. The Nusselt number is to be calculated on side 2 of the domain for the sample problem, so you must set the button to 'ON'. Tagging of side 2 for Nusselt number calculation was done during the mesh generation stage for the present problem. Note that a request to simultaneously calculate the Nusselt number and prescribe heat flux conditions cannot be accommodated in the current version of this code. See the description on Heat Flux Boundary Condition below.

**Lumped Parameter Indicator:** The consistent numerical approach to calculate the Nusselt number (heat flux) is used in INSTED/CFD (2D). You have the option of using either a lumped method or to invert a matrix. The lumped approach is more approximate, but it is cheaper and it could actually give better results at corners. Ordinarily, the direct method should be more accurate, but this has not been found to be the case. This is also the observation of numerical analysts from Lawrence Livermore National Labs. Toggle the button to 'ON' if the Nusselt number is to be calculated with the lumped mass approach and 'OFF' otherwise. The lumped mass approach will be used for the sample problem so the button should be set to 'ON' or 'YES'.

**Print Frequency for  $u$ ,  $v$ ,  $T$ ,  $\psi$ ,  $k$ ,  $\varepsilon$ .** The data for viewing the results for  $u$ ,  $v$ ,  $T$ ,  $\psi$ , and  $\phi$ , will be generated/stored based on the frequency of print in this input. **Important:** Be aware of your hard disk memory situation before specifying print frequency and number of restarts. Note also that no data will be saved for post-processing if the print frequency is greater than the number of time steps per restart for the current restart. The minimum value for the present data is 1. For the sample problem post-processing data for  $u$ ,  $v$ ,  $T$ ,  $\psi$ , and  $\phi$  is calculated and stored every 8 time steps. For steady state calculations results are printed at the end of the simulation.

**Print Frequency for Nusselt number:** Post-processing data for the Nusselt number will be generated/stored based on the frequency of print in this input. Unlike the other variables (item above), the Nusselt number consists is displayed in line graphs for its variation along the specified surface. The post-processing of the other variables consists of contour maps and vector plots. Print frequency for the Nusselt number can be set equal to that for the other variables, but this is strictly not necessary. **Important:** Be aware of your hard disk memory situation before specifying print frequency and number of restarts. Note also that no data will be saved for post-processing if print frequency is greater than the number of time steps per restart. The minimum value for the present data

is 1. For the sample problem the Nusselt number data is to be calculated and stored every 8 time steps.

**Initial Data Print Indicator:** Set this field to 'YES' if the initial data is to be printed in an ASCII file, and 'NO' otherwise. Printing out initial conditions allows you to restart the analysis using the data in this file if, for any reasons, the analysis is halted. The ASCII data generated by the solver was discussed earlier. In the sample problem, a print-out of initial data is desired, so keep the default value of 'YES' for this parameter.

**Sample points indicator:** Set this field to 'ON' or 'OFF', depending on whether or not you would like to monitor the values of the dependent variables as a function of time at specified locations in the model. A maximum of 30 monitoring (sampling) points may be selected.

**Vorticity/Pressure Calculation Indicator:** Set this field to 'ON' or 'OFF', to indicate whether or not the vorticity and pressure are to be calculated. These are secondary variables that are obtained by processing the solution for the velocities. These variables are calculated for the sample problem. Therefore, this field is set to the 'ON' state.

**Vorticity/Pressure Print Frequency:** Post-processing data for pressure and vorticity will be generated/stored based on the frequency of print in this input. The minimum value is 1. Pressure will be calculated and stored every 8 time steps for the sample problem.

Important Note: Because vorticity is of little direct use in engineering design, its post-processing has been blocked-out in the current version of INSTED/CFD (2D). However, vorticity results are available in ASCII data, as are pressure data.

**Indicator for Rotating Frame Solution:** Set this field to 'ON' to view solution  $u$ ,  $v$ , and  $\psi$  in the inertial frame after a rotating flow problem has been calculated. Set to 'OFF' otherwise. This input is relevant only to rotating flows. This capability has been deactivated in the current version of INSTED/CFD (2D), meaning that the rotating flow solution is always given in the rotating frame. No input is needed for the sample problem.

**Iteration Tolerance  $\epsilon$ :** This is the tolerance that you give to assess attainment of iteration convergence. This applies only to calculations for which multiple iterations are used. Convergence is achieved when

$$\left\| \frac{u_m - u_{m-1}}{u_m} \right\| \leq \varepsilon,$$

where  $u$  is any of the variables  $u$ ,  $v$ , or  $T$ , and  $\|\cdot\|$  denotes the Euclidean norm, with summation over all variables and all nodes in the domain.  $m$  is the iteration number. A one step Newton-Raphson iteration is used in many of the calculations. However, there is no reason (other than cost) why multiple iterations cannot be used. The default  $\varepsilon$  value of 0.01 is retained for the sample problem, but this is not really used because the sample problem is solved as a time-dependent one and the number of iterations per time step is 1.

**Continuation Process Tolerance:** Intermediate iteration tolerance for continuation process purposes only (see item on Continuation Indicator above). This parameter is not needed for the sample problem since no continuation process is used.

**Maximum Number of Iterations Per Time Step:** This is the maximum number of iterations allowed per time step in a time-dependent calculation or the maximum number of iterations in a steady state calculation. A value of 32 is used for the sample problem.

**Indicator for Heat Flux in a Closed Region:** Set this field to 'ON' if heat flux (Nusselt number) is to be determined along a 'closed' boundary region (e.g., the entire circular boundary of a horizontal cylinder). Set to 'OFF' otherwise. Steady state ASCII data (solutions) for  $u$ ,  $v$ , and  $T$  in cylindrical coordinates (in addition to the solutions in the Cartesian system) are given when this parameter is on. The Nusselt number in a closed region (cylinder surface) is not desired, therefore this parameter is set 'OFF' for the sample problem.

**Indicator for Time-Dependent Boundary Conditions:** Set this field to 'ON' if boundary conditions are time-dependent, and 'OFF' otherwise. When this parameter is on, you provide (fill-in) a routine for the time dependence. This capability has been deactivated in the current version of INSTED/CFD (2D). User-input is not needed for the sample problem. Customization of this capability is available from Thaerocomp.

**Indicator for Transient Problems:** Set this field to 'ON' if a time-dependent calculation is to be done, and 'OFF' otherwise. The sample problem involves a time-dependent calculation. The default value of 'YES' is used for this parameter.

**Indicator for Predictor/Corrector (JPRED):** Set this field to ‘ON’ if a predictor-corrector time integration scheme is to be used, and ‘OFF’ otherwise. The integer variable *JPRED* will be referred to in this manual as the indicator for predictor/corrector. Be aware that INSTED/CFD (2D) gives many options for time integration, as follows:

1. Backward Euler (BE); constant  $\Delta t$
2. Forward Euler (FE); constant  $\Delta t$
3. FE predictor with BE corrector; error-controlled  $\Delta t$
4. Trapezoid rule (TR) with BE start; constant  $\Delta t$
5. Adams-Bashforth (AB) predictor, TR corrector; and constant  $\Delta t$ . This scheme is started with FE predictor with BE corrector and constant  $\Delta t$
6. AB predictor with TR corrector; error-controlled  $\Delta t$ . Scheme is started with FE-BE and constant  $\Delta t$
7. A ‘restart’ scheme with any of the schemes above. The restart scheme is intended primarily for (6). Moreover, nothing is gained by using a restart scheme for a constant step-size calculation. Note: The various options allowed for time integration can be explored by a judicious combination of the input states or values represented by the labels *NBE*, *NTIME*, *NOERC*, *JRSTAT*, *NOCDDT*, and *JPRED*. The input value for this parameter is set to ‘ON’ for the sample problem.

**Number of Backward Euler Steps (NBE):** Because of its superior stability, start the time integration with at least a few (3 or 4) backward Euler steps. This is crucial for the predictor/corrector scheme. After the initial (backward Euler) steps, you can switch to the Trapezoid rule. For the predictor/corrector scheme the minimum number of backward Euler steps is 2. A value of 3 is recommended and used in the present sample calculation. The integer variable *NBE* will be referred to in this manual as the number of backward Euler steps.

**Maximum Number of Time Steps Per Restart (NTIME):** A restart capability with the time integration error-control scheme is provided in INSTED/CFD (2D). With this scheme, time integration is started afresh, after a pre-assigned number of time steps. The number of time steps per restart refers to the time step after which integration is restarted. INSTED/CFD (2D) can change your input internally in order to optimize the error-control scheme. This depends on the parameter ‘*nbf4*’ discussed below. A value of 32 is recommended for the maximum number of time steps per restart for most problems, although some experimentation might be needed for some tough problems. A value of 32 is used for the sample problem. More discussions on the maximum number of time steps

per restart will be provided later. Note: If the number of backward Euler steps is equal to or greater than the maximum number of time steps per restart, backward Euler will be used for all time steps. This input is not necessarily the total number of time steps required for a simulation. This will become apparent after more discussions below of the parameter 'nbf4'. **Important:** Do not confuse the restart scheme discussed here with the process of resuming a calculation from a previous state. In the present use, a more complicated process of error 'zeroing' is involved. In INSTED/CFD (2D), resuming a calculation from a previous run is effected by using the appropriate initial condition and the total time of previous integration. The integer variable *NTIME* will be referred to in this manual as maximum number of time steps per restarts.

**Maximum Number of Newton-Raphson Per Time Step:** In earlier versions of INSTED/CFD (2D), options for both the Newton-Raphson and the Quasi-Newton methods for the iterations within a time step were allowed. In some cases the two schemes were also combined in one simulation. However, in the current version of INSTED/CFD (2D) the Quasi-Newton option has been dropped. So the maximum number of Newton-Raphson now equals the maximum number of iterations per time step (item discussed earlier). Further, a one-step Newton-Raphson iteration is used for most of these calculations, although multiple iterations might be necessary for very tough problems. For the sample problem the default value of 1 is used for this parameter.

**Maximum Time Integration Error:** The time integration error control in INSTED/CFD (2D) allows you to set an upper limit on this error. The code picks a time step size  $\Delta t$  that ensures the specified error limit is not exceeded. Only the local error is controlled, however. The recommended (default) value of .002 is used for the sample problem. This corresponds to a maximum time integration error of .2%.

**Tolerance for Steady State:** This is the tolerance that you give for establishing the attainment of steady state in a transient calculation. The same Euclidean norm as iteration convergence (item previously discussed) is used but with *m* referring to the time step. The default value of .005 is used in the sample problem.

**Input Time Step Size:** This time step size is used for all time steps if a fixed time step size integration scheme is used. In the error-control scheme the input step size will be used only for the number of time steps in which the step size  $\Delta t$  is constant (item discussed later). Error control is not active during the constant step size period. Therefore, the error limit can be exceeded if the input time step size is large. For a problem you

know little about, the number of time steps in which step size is fixed should be few (about 3), and the input step size should be small. For the sample problem, the default value of .005 is used for the input step size.

**Indicator for upwinding:** For convection-dominated flows, it is advisable to turn on upwinding in order to significantly enhance the nonlinear stability of the equations. Wildly oscillatory and unphysical solutions may be the result of the nonlinear instability of the numerical procedure if the strength of the flow is large. In the current problem, upwinding will be turned off. Note that upwinding may affect your ability to obtain steady state solutions in a transient calculation. This is because of the nonlinear effects of the procedure.

**Indicator for Error Control (NOERC):** Set this field to 'ON' if the error-control scheme should be used and 'OFF' otherwise. Note that  $\Delta t$  must be suitable for all time steps if time integration error is not controlled. This could be risky, so you might as well take advantage of the error-control scheme in this code. For the sample problem, the time integration error is controlled so this parameter should be set 'ON'. The integer variable *NOERC* will be referred to in this manual as the indicator for no error control.

**Indicator for Restart Scheme (JRSTAT):** Set this field to 'ON' if a restart scheme should be used in the error-control procedure and 'OFF' otherwise. The option to use a restart scheme is suggested when error-control is in effect and *Ra*, *Re* or  $1/\mu$  (as the case may be) is large. The restart scheme is used for all error-control calculations. The default state of 'ON' is used for this parameter in the sample problem. The integer variable *JRSTAT* will be referred to in this manual as the indicator for the restart scheme.

**Number of Restarts (LRSTLM):** The number of restarts depends on an estimate of the number of time steps required for the simulation and the value specified for the parameter *nbf4* discussed below. Since this is not known for most problems, you should be generous in specifying the number of restarts. For many problems, the simulation will be terminated when a steady state is reached (if one exists). However, in difficult problems, all the restarts could be used up.

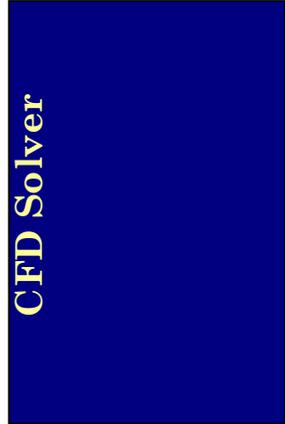
**Important:** You should be aware of your disk drive memory situation before specifying the print frequency, and the number of time steps per restart. Note that no data will be saved for post-processing if the value of the print frequency is greater than the number of time steps per restart. **Also Important:** Do not confuse the restart scheme discussed here with the process of resuming a calculation from a previous run. In the present use, a more

complicated process of error ‘zeroing’ is involved. In INSTED/CFD (2D), resuming a calculation from a previous state is accomplished by using the appropriate initial conditions and the total time of the previous integration (items previously discussed). Eight restarts are specified for the sample problem. The integer variable *LRSTLM* will be referred to in this manual as the number of restarts **plus one**. It is this variable that is actually displayed for this field.

**NBF4:** This variable is associated with the number of time steps per restart in a restart scheme. The variables *JRSTAT* and *LRSTLM* (items discussed previously) are needed to define ‘*nbf4*’. A restart scheme is used if *JRSTAT* is ‘ON’. The restart scheme is not activated if *JRSTAT* is ‘OFF’. In any case, INSTED/CFD (2D) first goes through *NTIME* steps for the ‘zeroth’ restart. If a restart scheme is used, the accelerations needed for the calculation of the solution are reinitialized after the zeroth restart and integration is continued beyond the input *NTIME* time steps (i.e., the beginning of the first restart). The calculations are terminated after the *NTIME* steps if no restart scheme is used. The purpose of reinitializing the acceleration in the manner discussed above is to ‘throw away’ the accumulated round-off errors which otherwise pollute the calculation of the predicted solution, giving unreliably small time step sizes. This is particularly true in strong (high *Ra*, *Re*, or  $1/\mu$ ) flows at which the regular error-control scheme becomes unreliable. For example, you will find the error-control scheme to be far more efficient than constant time step size schemes for  $Ra < 10^5$  provided a restart scheme is used when  $Ra > 10^4$ .

The number of time steps used for the first restart depends on the value assigned to *NBF4*. If *NBF4* is less than 3 the program uses *NTIME* steps for the first restart and for all subsequent restarts. *NTIME/2* time steps is used for the first and subsequent restarts if *NBF4* is 3. *NTIME/2* time steps is used for the first restart but *NTIME/4* time steps for the second restart if *NBF4* is 4, and so on. Suppose you set *NTIME* = 40 and *LRSTLM* = 5 (i.e., 4 restarts) then the number of time steps used at every restart for 4 values of *NBF4* will be given as in the table below. The desired number of restarts is (*LRSTLM* - 1) and the maximum number of restarts allowed by the program is 17. Note that the specified maximum time integration error is assured irrespective of whether the calculated time step size becomes very small or quite large. Time steps could easily cover the range  $10^{-3}$  to  $10^2$ , or wider, in the simulation of a weak flow. The restart scheme is strongly recommended for transient simulations with low or moderate parameter values.

Table 1. Number of time steps versus restart values for various *NBF4* values



Restart	Number of time steps			
	NBF4			
	≤ 2	3	4	5
0	40	40	40	40
1	40	20	20	20
2	40	20	10	10
3	40	20	10	5
4	40	20	10	5

The optimum values of *LRSTLM*, *NBF4*, and *NTIME* depend on the physics of the flow in the problem being solved. You will often need to experiment with different values of these parameters (using coarse grids) to determine the combination that gives a cost-effective simulation. You need not be too fuzzy about the selection of these parameters. Keep *NTIME* = 64 (simpler problems) or 32 (tougher problems), *NBF4* at 3, and use a value of *LRSTLM* sufficient to give the desired number of time steps. The constant time step size schemes are at your disposal in case you are not satisfied with the error-control schemes. For the sample problem, *NTIME* = 32, *LRSTLM* = 4, and *NBF4* = 3 are used (i.e., a maximum of 80 or 32+16+16+16 time steps is allowed for the simulation). **If you do not understand this variable, set it to 3 and the frequency of print, to *NTIME*/2.**

**Number of Constant Time Step Size (NOCDT):** This refers to the number of time steps in which the input (or default) time step size is used before a switch is made to the error-control scheme that determines time step size (automatically). The step size determined by the error-control scheme is designed to meet the time integration error that you specify. The integer variable NOCDT has been referred to in this manual as the number of constant time step sizes. The minimum number of constant time step size is 2 because of the multilevel nature of the error-control scheme. The maximum value is of course the number of time steps per restart. A value of 3 is used for the sample problem.

**Indicator to Print Predicted/Corrected Solution:** Set this field to ‘ON’ in order to print the predicted and corrected solution or to ‘OFF’ otherwise. This capability is intended for diagnosis only and should be turned on only when it is absolutely necessary. Further, printing predicted/corrected solutions should be attempted only for a trivially coarse grid, because of the memory demand. This value should be set to ‘OFF’ for the sample problem.

**Print Frequency for Predicted/Corrected Solution:** This is the number that tells the code when to print the predicted/corrected solution. The minimum frequency of print is set at 8, because of the large amount of data that is printed with this option. This entry is not needed if the indicator to print predicted/corrected solution described above is set to 'OFF', as is the case for the sample problem.

**Periodicity Argument Cut-Off:** The argument  $tRe,Pr/2$  for sine and cosine in equations (2) and (3) are set to zero if the entry for periodicity argument cut-off is less than 9.0. The actual value of the argument is used otherwise. The normal gravity field is fixed in the present problem and the argument is zero.  $Re_r = 0$ . The default value (.1) is used for the sample calculations.

**Indicator to Specify Heat Flux Boundary Condition:** Set this field to 'ON' to specify a heat flux boundary condition on a portion of the surface or to 'OFF' otherwise. For the selection of the surface on which this kind of boundary condition is to be specified, follow the same procedures as that for selecting a surface for Nusslet number calculation, which is discussed under mesh generation. In the present version of INSTED/CFD (2D), you cannot combine the specification of a heat flux boundary condition and (at the same time) require that a heat flux be specified in the same simulation. Heat flux parameters are needed if the present indicator is on. For the sample problem, no heat flux boundary condition is specified. Therefore, this field is set to the 'OFF' state.

#### 4.4.2.5 Fixing Flow Variables

**Fixing Values of the Dependent Variables:** The buttons - **Fix u**, **Fix v**, **Don't solve for Temp.**, and **Don't solve for Scalars** - are used to fix variables during solution. 'Don't Solve for Scalar' is automatically set 'ON' if there are no scalars defined. If either  $u$ ,  $v$ , or  $T$  is fixed, a value for the fixed variable is required. The input for this value appears automatically on the 'Prob. Description' dialog box.

#### 4.4.2.6 Relaxation Factors

**Relaxation Parameters:** Relaxation parameters are introduced to help with convergence in the simulation of moderate to very difficult flow and heat transfer problems. Rather than using the current value of a variable to compute the solution for the next iteration,

we use only a fraction of the current value. The relaxation parameter is the fraction of the most recent value used.

For weak flows no relaxation is needed. That is, the parameter is set to 1. For very difficult flow problems, very low values (e.g. 0.333) may be required for convergence. For intermediate strength flows, a value of 0.667 may be acceptable.

Each of the variables, namely the velocities ( $u$ ,  $v$ ), temperature ( $T$ ), kinetic energy of turbulence  $k$ , dissipation rate of turbulence  $\varepsilon$ , and the scalars, has its own governing equation that has to be solved to convergence. (Note, however, that for all practical purposes, these equations are coupled.) The various equations have different levels of difficulty (nonlinearity) and hence, require different relaxation values. For example, the turbulence equations are very stiff, mathematically speaking, and may require lower values for the relaxation parameters when compared to the equations for  $u$ ,  $v$ , and  $T$ . The equations for  $u$ ,  $v$ , and  $T$  have similar levels of difficulty for the most part. Without chemical reactions, the scalar equations may be slightly easier to solve, when compared to the equations for  $u$ ,  $v$ , and  $T$ .

**Important:** If any of the relaxation factors is not set to 1, you must specify multiple Newton Raphson iterations per time step, to obtain a convergence of the iterative procedure you initiate. Otherwise, the results will be useless.

If the Apply Relaxation Factor button is not set to the 'ON' position, INSTED/CFD (2D) will attempt to use built-in values of the relaxation parameters, which may vary with iterations and time steps.

#### **4.4.2.7 Turbulence Parameters**

**Turbulence Model Options:** Select the turbulence model of interest from the listbox that appears if the 'Solve as Turbulent Flow' is set on. The supported turbulence models are listed in the dialog box.

Other parameters are required if a turbulence calculation is requested. These are listed under the 'Flow/Thermal Parameters' section of the 'Prob. Description' dialog box. These additional data are described below:

**Inlet Mean Flow Length:** Characteristic length of the opening (i.e. diameter, hydraulic diameter, etc.)

**Turbulence Length Factor:** The fractional factor by which the inlet mean flow length is multiplied by to obtain an approximate value for the turbulence integral length scale.

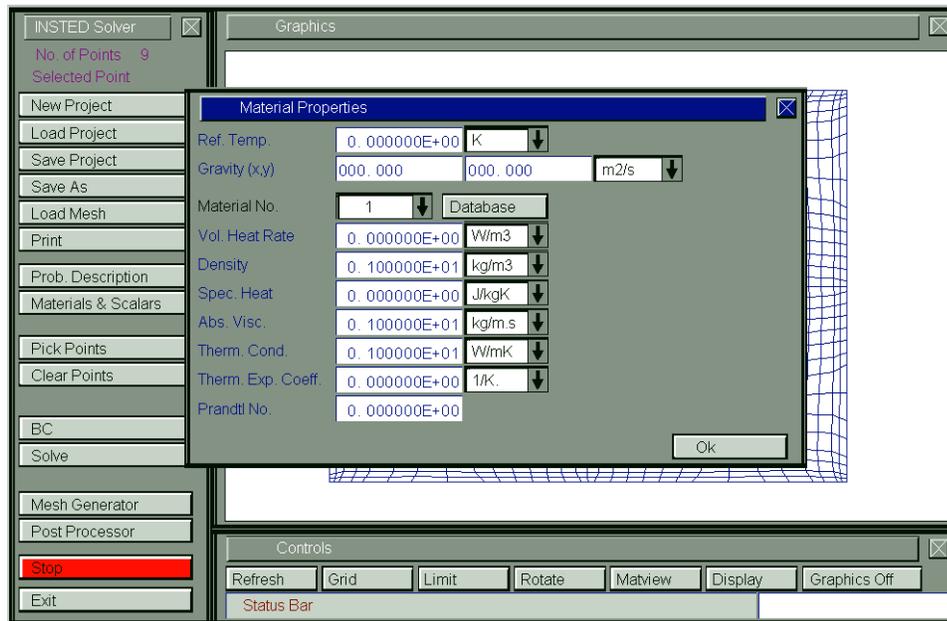
**Turbulence Velocity Factor:** The fractional factor by which the inlet mean flow length scale is multiplied to obtain an approximate value for the rms turbulence velocity.

## 4.5. Specifying Material Properties

All material properties are specified in the 'Materials Properties' dialog box. This dialog box is described below.

### 4.5.1 The Materials Properties dialog box

The problem description dialog box may be obtained by clicking the **Materials Properties** button on the Main dialog box.



The thermophysical properties for the different materials present in the model are assigned values here. Information on the number of materials is obtained from the mesh data.

#### 4.5.2 Some parameters in the Material Properties dialog box

**Parameters for Heat Flux Boundary Conditions:** If this option is turned on in the 'Prob. Description' dialog box, you must provide information on the values of the heat flux parameters (defined below). INSTED assumes that a heat flux boundary condition (in dimensional or dimensionless units) is specified as power divided by area. For all problem types, the heat flux  $q''$  is written as

$$q'' = P_1 P_2 (P_4 T - P_3),$$

where  $P_1$ ,  $P_2$ ,  $P_3$ , and  $P_4$  are called the four heat flux parameters. For Types 1 and 2 CFD problems, this expression assumes the dimensionless form:

$$q'' = Nu h'(T - T'_\infty),$$

where  $Nu$  is the Nusselt number  $hL/k$ ,  $h'$  is dimensionless film coefficient  $h/h_{ref}$ ,  $T$  is surface temperature and  $T'_\infty$  is ambient temperature.

For this case,  $P_1 = Nu$ ,  $P_2 = h'$ ,  $P_3 = T'_\infty$ , and  $P_4 = 1$ . For type 3 problems,  $P_1 = h$  (W/m<sup>2</sup>K),  $P_2 = 1$ ,  $P_3 = T_\infty$  (K), and  $P_4 = 1$ .

**Important:** You should consult the literature if you have difficulties with these discussions on boundary conditions.

Convention for Surface Heat Transfer

The convention used in INSTED/CFD (2D) for heat transfer on the surface is:

$$-k \frac{\partial T}{\partial n} = h(T - T_\infty).$$

This expression implies that  $h(T - T_\infty)$  is the heat conducted to the surface. The ‘natural’ way in which surface heat flux occurs in finite element equations is:  $k\partial T/\partial n$ , on the right-hand side of the energy equation. You should specify the heat flux parameters for  $k\partial T/\partial n$  (not  $-k\partial T/\partial n$ ), as shown in the table below.

Flux Type	Expression	$P_1$	$P_2$	$P_3$	$P_4$
Film coeff.	$-h(T - T_\infty)$	$-h$	1	$T_\infty$	1
Film coeff.	$-Nu h'(T - T'_\infty)$	$-Nu$	$h'$	$T'_\infty$	1
Specified flux	$-q_s''$	1	1	$-q_s''$	0

Once you turn on the indicator to specify heat flux boundary conditions in the ‘Prob. Description’ dialog box, you will need to enter the above parameters in the ‘Materials Description’ dialog box.

### 4.5.3 Using the Matview command to view material properties

For a tabular description of all the materials in the current project and the values of the material properties on the graphics screen, click the **Matview** button located on the 'Controls' dialog box.

## 4.6 Sample Points in INSTED/CFD

Sample points for history data may be selected in INSTED/CFD. It may be necessary to focus (or zoom) on an area of the screen to aid in point selection. The 'Limits' dialog box which has a similar function as that in the INSTED Mesh Generator is useful for this purpose. The latter dialog box is described below.

### 4.6.1 The Limits dialog box

This dialog box is similar to the 'Limits' dialog box of the INSTED Mesh Generator. The latter was described in Section 2.4. Information for the graphics screen is obtained from the meshed model. Varying the values in the 'Limits' dialog will modify the graphic screen. This can be used to translate the model or zoom in on specific areas (e.g. for selecting points for time history data).

### 4.6.2 Zooming in INSTED/CFD (2D)

Zooming in the INSTED/CFD (2D) Solver environment follows the same procedure as in INSTED/CFD (2D) Mesh Generator. Refer to section 2.13.2 for a detailed description of the procedure. You might also wish to turn on the nodal point labeling described in Section 2.13.1 to help you with the selection of nodal points for time history data.

### 4.6.3 Selecting Sample Points for Time-History Data

For transient problems, INSTED/CFD (2D) allows the data for specific points to be stored for later time-history analysis. To select the desired points, you need to click the **Pick Points** button on the Main dialog box. The program then prompts you for the desired points. Click the nodes on the graphics screen to select the points. To end the process, you will need to click the **Stop** button, also on the Main dialog box.

### Clear Points

To clear all previously selected points (to reselect points), click the **Clear Points** button on the Main dialog box.

## **4.7 Procedure for Obtaining a Solution for a Meshed Model**

In summary, the steps for executing the Solver are as follows:

- Click the 'New Project' button to remove material and solver data for the current project from the memory
- Load the meshed model by clicking the **Load Mesh** button on the Main dialog box. Locate the mesh files and select the main file (usually called mesh.dat from the mesh generator). Other secondary model files such as bound.dat, xnuslt.dat, etc., will also reside in this mesh directory.
- Enter the solution parameters in the 'Prob. Description' dialog box (or Load from a Project)
- Enter the material properties in the 'Material Properties' dialog box (this will also be loaded from a project)
- Select sample points (if any) for which you wish to store time history data
- Click the **Solve** button, which resides on the Main dialog box.