FLUTAN 2.0
Input Specifications

G. Willerding, W. Baumann

Institut für Neutronenphysik und Reaktortechnik
Projekt Nukleare Sicherheitsforschung

Forschungszentrum Karlsruhe
Technik und Umwelt

Wissenschaftliche Berichte
FZKA 5712

FLUTAN 2.0

INPUT SPECIFICATIONS

G. Willerding, W. Baumann
Institut für Neutronenphysik und Reaktortechnik
Projekt Nukleare Sicherheitsforschung

FLUTAN 2.0 -INPUT SPECIFICATONS

Abstract.

FLUTAN is a highly vectorized computer code for 3D fluiddynamic and thermal-hydraulic analyses in Cartesian or cylinder coordinates. It is related to the family of COMMIX codes originally developed at Argonne National Laboratory, USA, and particularly to COMMIX-1A and COMMIX-1B, which were made available to FZK in the frame of cooperation contracts within the fast reactor safety field.

FLUTAN 2.0 is an improved version of the FLUTAN code released in 1992. It offers some additional innovations, e.g. the QUICK-LECUSSO-FRAM (cf. Ref./7/,/8/,/9/) techniques for reducing numerical diffusion in the k-ε turbulence model equations; a higher sophisticated wall model for specifying a mass flow outside the surface walls together with its flow path and its associated inlet and outlet flow temperatures; and a revised and upgraded pressure boundary condition to fully include the outlet cells in the solution process of the conservation equations. Last but not least, a so-called visualization option based on VISART standards has been provided.

This report contains detailed input instructions, presents formulations of the various model options, and explains how to use the code by means of comprehensive sample input.

Eingabebeschreibung für FLUTAN 2.0

Zusammenfassung.

FLUTAN ist ein hoch-vektorisierter Computercode für 3-dimensionale fluiddynamische und thermohydraulische Analysen sowohl in kartesischen als auch zylindrischen Koordinaten. Er gehört zur Familie der ursprünglich am ANL (USA) entwickelten COMMIX-Programme und basiert insbesondere auf COMMIX-1A und COMMIX-1B, die dem FZK im Rahmen einer Zusammenarbeit auf dem Gebiet der Schnellbrüter-Sicherheit zugänglich waren.

FLUTAN 2.0 ist die weiterentwickelte Version des im Jahre 1992 freigegebenen FLUTAN-Codes. Diese Version bietet einige zusätzliche Neuerungen wie z. B. die nun im Turbulenzmodell implementierte LECUSSO-QUICK-FRAM-Methode (s. Ref./7/,/8/,/9/) zur Reduzierung numerischer Diffuson, ein anspruchsvolleres Wandmodell zur Spezifikation von Massenströmen inklusive Strömungspfad sowie Ein- und Austrittstemperaturen außerhalb der Randflächen des Rechenmodells und eine überarbeitete, verbesserte Druckrandbedingung, die die Austrittszellen voll in die Lösung der Erhaltungsgleichungen miteinbezieht. Außerdem wurde eine sogenannte Visualisierungsoption im VISART-Format eingefügt.

Der vorliegende Bericht gibt eine detaillierte Anleitung zur Benutzung des Codes, zeigt die mathematischen Formulierungen einzelner Optionen und erläutert die Eingabe anhand eines umfassenden Beispiels.

FLUTAN 2.0

A Computer-Code for 3-Dimensional Fluid- and Thermal-Dynamic Analysis in Cartesian or Cylinder Coordinates.

FLUTAN is a highly vectorized program related to the family of COMMIX codes, which were originally developed at the Argonne National Laboratory (ANL), USA. Optimization and vectorization was done in the Institut für Neutronenphysik und Reaktortechnik (INR) of the Forschungszentrum Karlsruhe (FZK), initially on a CYBER-205 vector computer, later on Fujitsu/Siemens VP50 and VP400-EX. Program language is FORTRAN-77.

To a large extent, FLUTAN uses basic concepts and structures imported from the codes COMMIX-1B (cf. Ref./1/, /2/) COMMIX-2 (cf. Ref./3/), which FZK was able to obtain from ANL in the frame of a US-German cooperation on fast reactor safety. Users completely unfamiliar with the general design of the COMMIX codes are advised first to consult the reports just mentioned and, especially, a recent report on COMMIX-1C (cf. Ref./4/, /5/).

Not all features of the original code versions have been implemented in FLUTAN; instead, the previously released version (cf. Ref./18/) was provided with some essential innovations, e.g. the CRESOR algorithm (cf. Ref./6/) and general 3-dimensional rebalancing for solving the pressure equation, as well as the QUICK-LECUSSO-FRAM techniques (cf. Ref./7/, /8/, /9/) to tackle the numerical diffusion problems, both for the enthalpy and momentum equations. In a recent effort, the same techniques have now been introduced into the k-ε turbulence model equations (cf. Ref./19/).

In contrast to the conventional duct wall model with its uniform temperature boundary condition given as a heat sink or source, a higher sophisticated wall model has been developed which enables the user to specify a mass flow outside the surface wall, its flow path, and its associated inlet and outlet flow temperatures (cf. Ref./14/).
The pressure boundary condition has been revised and upgraded in order to fully include the outlet cell volumes in the computational process, i.e. to facilitate the solution of the conservation equations for all nodes within the defined model boundary surfaces.

For visualization or other postprocessing programs a so-called visualization option based on VISART standards has been provided (cf.Ref./15/).

---

[1] Institut für Angewandte Thermo- und Fluiddynamik
[2] Institut für Reaktorsicherheit
[3] Visiting scientist from Ben Gurion University of the Negev, Beer-Sheva, Israel

CONTENTS

General Comments

The units used in FLUTAN are meter, kilogram, second, and degrees Celsius. These and other derived units are indicated after the description of variables requiring them.

Default values are indicated either by an asterisk or a value in parentheses after the variable description.

Arrays are indicated by the use of a subscript following the variable name. The ranges of the subscripts are indicated in the following table.

Index	Range	Current Limit
I	IMAX	IJKMAX
J	JMAX	IJKMAX
K	KMAX	IJKMAX
N	NSURF	IJKMAX
L	NL1	LBOUND
M0	NM1	LCELL
NH	NHEATC	10
NM	NMATER	10
NF	NFORCE	100
NC	NCORR	20

Heat Exchanger Option, cf. Ref./14/ :

NR	NHEX	99

The range limits IJKMAX, LBOUND, LCELL must be set properly at compile time by a PARAMETER statement.

Some Terminology

The computational area is partitioned into a number of computational cells, each bounded by consecutive X, Y, and Z direction grid planes.
Surfaces (portions of a plane or cylinder) may be defined both on the exterior, bounding the computational area, and in the interior. The intersection of a surface and consecutive grid planes outlines a surface element.
Surfaces which coincide with a grid plane are called regular surfaces, otherwise, they are called irregular surfaces.
A regular cell is one with all faces coinciding with grid planes.
Irregular cells have one and only one irregular surface element.

General Input Structure

Input for FLUTAN can be described in one of two ways:

1. Cartesian Geometry : IGEOM = 0
2. Cylinder Geometry : IGEOM = – 1

Both geometry options allow the user to describe the geometry in terms of the cells formed by the X, Y, and Z or by the R, θ, and Z grid planes. A typical input sequence is as follows:

Problem Description Records (Title)	(Optional)
NAMELIST /GEOM/	
Boundary Surface Identification Records	
NAMELIST /DATA	
NAMELIST /TURB/	
Force Structure Specification Records	(Optional)
Thermal Structure Specification Records	(Optional)
Thermal Structure Location Records	(Optional)
Boundary Value Initialization Records	
Internal Cell Initialization Records	
NAMELIST /REBAL/	(Optional)
Rebalancing Region Records	(Optional)

Problem Description Records

Any number of records with user comments can precede NAMELISTs or be interspersed with non-NAMELIST input as long as columns 1-4 are left blank.
Comment text may also follow the data in formatted input records and may especially start in column 5 of a record with END plus trailing blank in columns 1-4.

Reserved Key Words

Input to FLUTAN is a mixture of NAMELISTs, formatted records and comments. When processing formatted input, columns 1-4 of each line of this type of input is compared with a group of key words. When a match is found the line is reread in the appropriate format. (If blanks are found the record is treated as a comment record.) The actual list of key words follows (_ indicates a leading blank).

AL	HL	P	RL	UREB	YFOR
ALX	HLB	PB	RLB	VELBN	ZFOR
ALY	IN	QBN	TL	VL	_&F
ALZ	IREG	QSOU	TLB	WL	_&M
END	OUT	REG	UL	XFOR	_&T

IMPORTANT

The following variables (Cf. General Comments, NAMELIST /GEOM/) are used to allocate space. It is important that they are specified correctly:
They can be approximated by a value larger than actually needed but limited by the 'Current Limit' The required values are printed after being computed. Any of these variables that remain unchanged for a subsequent restart run should not be respecified since they are read from the restart file.

IMAX	JMAX	KMAX
NM1	NL1	IFREB

The amount of data specified explicitly by IFREB (User-specified Rebalancing) and implicitly through the Thermal Structure Prototype Records (for at most 100 thermal structure prototypes) have to share an array, whose length NAVAIL is set at compile time by a PARAMETER statement in the header routine.

IGEOM	0	Regular box geometry option. (*)
	-1	Cylindrical geometry option using box geometry input.

Notes (cf. notes 3 and 4 on page 10) :

1. A surface ('center line') must be dedicated to the origin R=0.0 (if the origin is not excluded as in an annular system).
Set KFLOW (N) = −3 and KTEMP (N) = 400 for that surface.
2. For full 2π radian ranges J = JMAX is automatically linked to J = 1, thus no surfaces need be defined at Y = 0.0 and Y = 2π.

IMAX	The maximum number of cells in the X(R)-direction. (1)
JMAX	The maximum number of cells in the Y(θ)-direction. (1)
KMAX	The maximum number of cells in the Z-direction. (1)
DX (I)	The calculational cell sizes along the X(R)-axis, **m**.
DY (J)	The calculational cell sizes along the Y(θ)-axis, **m** or **rad**.
DZ (K)	The calculational cell sizes along the Z-axis, **m**.
NL1	Total number of surface elements. (0)
NM1	Total number of computational cells. (0)

Notes :

1. NL1 and NM1 can be approximated by values larger than actually required; however, this must not be done when restarting (ISTATE>0).
2. To specify NM1 and/or NL1, makes sense only at the start of a steady-state run (ISTATE=0).

NSURF	The number of unique surfaces enclosing the calculational area. Unique surfaces are determined by a unique combination of the following three characteristics:

       1. Velocity Boundary Condition

       2. Temperature Boundary Condition

       3. The unit normal vector to the surface

       The unit normal vectors (referred to by the following three variables) are those pointing into the calculational area.

XNORML (N)	The X-component of the unit normal vector to surface N
YNORML (N)	The Y-component of the unit normal vector to surface N.
ZNORML (N)	The Z-component of the unit normal vector to surface N.

ITURKE		In this version of FLUTAN four turbulence models are included. For all of the details of input requirements for these options see the Turbulence Models in NAMELIST /TURB/.
	0	Constant turbulent viscosity and conductivity model (*).
	10	Zero-equation turbulence model.
	11	One-equation turbulence model.
	12	Two-equation turbulence model.

NFORCE		Number of force structures. (0)

The input defining the force structures, i.e., the Force Structure section of NAMELIST /DATA/ and the FORCE STRUCTURE SPECIFICATION RECORDs, must be included at the start of a steady-state run (ISTATE=0) when NFORCE > 0.

These two sections can be completely redefined at the beginning of a transient run (ISTATE=2) by setting NEWFOR=1 in NAMELIST /DATA/.

ISTRUC	*0*	No thermal structures are used. (*)

Do not include THERMAL STRUCTURE PROTOTYPE RECORDs or THERMAL STRUCTURE LOCATION RECORDs in the input.

	1	The input defining the thermal structures, i.e. THERMAL STRUCTURE PROTOTYPE RECORDs and THERMAL STRUCTURE LOCATION RECORDs must be included at the start of a steady-state run (ISTATE=0) when ISTRUC=1. These two sections can be completely redefined at the beginning of a transient (ISTATE=2) by setting NEWTS=1 in NAMELIST /DATA/.

IPRES0	I-index of the pressure reference point.
JPRES0	J-index of the pressure reference point.
KPRES0	K-index of the pressure reference point.

Restart Option (Use of Unit 9 & 10)

There are two ways to force the code to write a restart file (IFRES = 1 or 3, see below):

1. The first is to run the job to maximum CPU-time.This is done by specifying large values for NTMAX and TIMAX. The amount of time remaining for the job is checked at the end of each iteration using the system routine TREMAN (see the appendix section entitled MACHINE DEPENDENT ROUTINES). If the amount of time remaining is greater than TREST, an input parameter in NAMELIST /DATA/, another iteration is performed. If not, a restart file is written.

2. The second way to obtain a restart file is to set NTMAX or TIMAX to a time step or time which will be reached before the CPU job time expires. A restart file will be written at this time step or time. Thereafter execution terminates.

When restarting from a previous run, make sure that ISTATE is set to the appropriate value. Also, it is advisable to delete all input for variables that one does not intend to change. In some cases variables will be reset back to their initial values if the input specification remains in the input stream. In short, the minimum input necessary is the correct input for restart cases.

IFRES	*0*	New case with no restart data written. (*)
	1	New case with restart data written to unit 10.
	2	Restart of previous run, restart data read from unit 9 with no new restart file written.
	3	Restart of previous run, restart data read from unit 9 with new restart file written to unit 10.

Note: Cf. the note to the Plotting Option on page 23.

Solution Method Control

To avoid recursitivity problems in vectorizing SOR, FLUTAN establishes a red/black indexing scheme for the computational cells. (The indices follow the order given by cycling through the coordinate directions according to *ascending maximum coodinate indices*, i.e. IMAX, JMAX, KMAX, both for red and black cells, the last 'red index' followed by the first 'black index'.)

ISYMCH *Iterative Solution Methods* :

1 No solution of momentum equation; else same as 2.

2 SOR-Method for all equations.
(Use of relaxation parameters , cf. OMEGA, RELAXE, RELAXK)
Global convergence is accelerated by coarse mesh rebalancing, i.e. direct determination of pressure increment corrections, so as to yield mass balance within the coarse mesh. The initial coarse mesh rebalancing is repeated at intervals of IREBIT iterations.

3 CRESOR-Method for pressure equation only; else same as 2. (*)
CRESOR combines a 2-step (red/black) *adaptive* SOR technique -as a preconditioner- with the method of Conjugate Residuals.
The initial coarse mesh rebalancing is repeated when the *stepwise decrement* of the residual norm is less then 2% (once) or is less then 20% for a (not necessarily continuating) series of IREBIT iterations.
In trying to escape from special stagnation situations CRESOR (being adaptive) still uses OMEGA for an auxiliary series of IREBIT simple SOR iterations.
(Cf. Ref./6/).

4 ADI-Method for pressure equation; else same as 2.
(Alternating Direction Implicit; implementation suspended).

Direct Solution Methods
$\Rightarrow$ imposing strong limitation on NM1 (PARAMETER LCELL)
$\Rightarrow$ requiring recompilation for adjustment of FORTRAN PARAMETERs :

5 LU-Decomposition Method for all equations.
(PARAMETERs LBAND=1, LDARY=1 need adjustment)
Note: For full 2π radian systems without boundary surfaces being defined at Y = 0.0 and Y = 2π, the code runs into an error stop.

6 GAUSS-Solver for all equations (Author F. Schmitz, HDI/FZKA).
Adjustment of PARAMETERs :
- LGAUS = NM1
- LIDG = 1 + 2 × min { IMAX × JMAX, IMAX × KMAX, ...}
- LBAND = LGAUS × LIDG

Rebalancing Option

IFREB	0	No user-specified-region rebalancing. (*)
	>0	Rebalancing is performed over user defined rebalancing regions. The value of IFREB is used to allocate dynamic storage and must be at least as large as the total number of cells in the rebalancing regions plus the total number of cells used to specify rebalancing surfaces. A value of $2 \times NM1$ should be adequate space for most cases. The exact value needed will be printed in the Rebalancing Summary.

The input defining the rebalancing regions i.e., NAMELIST /REBAL/ and the REBALANCING REGION RECORDs must be included at the start of a steady state run (ISTATE=0) when IFREB > 0. (Cf. page 49).

These two sections can be completely redefined at the beginning of a transient run (ISTATE=2) by setting NEWREB=1 in NAMELIST /DATA/.

Note: In addition to rebalancing over user specified regions, plane-by-plane rebalancing is available and controlled by the variables IXREB, IYREB, and IZREB in the *Rebalancing Option* section of NAMELIST /DATA/.
In this case , however, IFREB=0 is required. (Cf. page 31).

Linkage of User Provided Heat Exchanger Model Package

NHEX	0	No linkage of special heat exchanger model package (*)
	> 0	Linkage of special heat exchanger model package (cf. KTEMP) At the same time NHEX specifies the number of heat exchangers.
NH1		Total number of surface elements covered by heat exchangers. Observe the notes below.

Notes:
1. The flag (IHXMOD) intended to control inclusion of user provided routines for special heat exchanger model and the predefined slots (i.e. calls to -at the time being- dummy FLUTAN subroutines HXMODA, HXMODB, etc.) are not utilized in the implementation of the Heat Exchanger Model.
2. Only those input parameters of the Model are announced in this report which have entered the conventional FLUTAN namelists, viz./GEOM/ and /DATA/, cf.C0MU, C1MU, C2MU.
3. The user is advised to read Ref./14/ for an understanding of the model and the complete input description.

BOUNDARY SURFACE IDENTIFICATION RECORDS

This set of records must follow NAMELIST /GEOM/ and be present at the start of steady-state runs (ISTATE=0).

The purpose of these records is to specify sets of cells (cf. IB,...., KE below) forming boundary surfaces which completely enclose the calculational region *and* to define any other boundary surfaces inside the calculational region. These interior boundary surfaces must completely surround a cell, a group of cells (i.e. a hole in the calculational region) or a *surface*:

To completely *surround a surface* one must specify two *collapsing* surfaces with normals in *opposite* directions. A single-sided boundary surface is not allowed in the interior of the calculational region. Also be sure that all surfaces specified bound calculational cel's.

Each boundary surface is defined by specifying one or more BOUNDARY SURFACE IDENTIFICATION RECORDs, each of which contains the following variables having the FORMAT (A4,E10.3,7I4):

KEY	AREA	IB	IE	JB	JE	KB	KE	N

KEY
- $R\iota G$ — The surface is a regular surface. Regular surfaces lie on grid planes.
- *IREG* — The surface is an irregular surface. Irregular surfaces do not lie on grid planes.
- *END* — A record with 'END ' in columns 1-4 must terminate the BOUNDARY SURFACE IDENTIFICATION RECORDs.

AREA
- *< 0.0 or blank* — The area of each surface element is set to its actual geometrical value, e.g. for cartesian geometry either $DX \times DY$, $DY \times DZ$, or $DX \times DZ$, whichever is appropriate.
- ≥ 0.0 — The area of each surface element is assigned a value of AREA, m^2.

IB, IE, JB, JE, KB,KE — These six variables are the beginning and ending I-, J-, and K-indices that define a rectangular solid composed of one or more cells. The rectangular solid that defines or *partially (cf. N and note 1) below) defines* a surface is the one adjacent to and on the side pointed to by the surface normal. (Keep in mind that the surface normals XNORML, YNORML, and ZNORML always point into the calculational region.)

A *surface element* is defined by the intersection of a single cell and the surface.

N — The surface number. All surfaces with the same combination of the three characteristics following below can be assigned the same surface number:

1. Velocity boundary condition,
2. Temperature boundary condition,
3. Unit normal vector to the surface.

Notes:

1. It is possible for two surface elements to lie in the same surface and have either the same or different surface numbers as well as for two surface elements to lie in different surfaces and have the same or different surface numbers.
2. The order of the BOUNDARY SURFACE IDENTIFICATION RECORDs must be as follows:
   - All IREG records (irregular surfaces) must precede all REG records (regular surfaces).
   - The surface numbers, N, of all IREG - and REG records must start from 1 and form a series with increment 1.
3. When using cylindrical geometry (IGEOM = −1), a surface must be specified at the origin when calculational cells are bounded by the origin (center line). When an annular region is being modeled, a surface should not be defined at the origin but rather at the boundary of the first calculational cell; this cell has an I-index > 1 (as being counted from the center). Set KFLOW (N) = −3 and KTEMP (N) = 400 for surfaces defined at the origin.
4. When using cylindrical geometry (IGEOM= −1), with 2π radian geometries, J=JMAX and J=1 are automatically linked, thus, no surfaces need be defined at Y=0.0 and Y= 2π.

ALPHA *0.0* Semi-implicit time advancement in the equations for both, *turbulent kinetic energy* and *turbulent kinetic energy dissipation rate*.

1.0 Fully-implicit time advancement in the equations for both, *turbulent kinetic energy* and *turbulent kinetic energy dissipation rate*. (*)

FCTLO See FCTHI

FCTHI To allow the user to spot check property values a small table is printed with nine temperature values ranging from FCTLO to FCTHI at a pressure PRESO. When the sodium package is present, the default values of FCTLO and FCTHI are 300.0 and 700.0 °C. When the water package is present, the default values are 20.0 and 100.0 °C.

IFENER *0* No enthalpy calculation.

1 Enthalpy calculation is performed. (*)
Thermal diffusion D across the interface F of two neighboring cells is computed as $D_{i+1/2} = 2F/(\Delta x_i/d_i + \Delta x_{i+1}/d_{i+1})*(h_i - h_{i+1})$,
h specific enthalpy, Δx cell width normal to F, d thermal diffusivity
($d = \lambda/c_p$; $\lambda =$ thermal conductivity; $c_p =$ specific heat).

2 Enthalpy calculation is performed.
Accounting for the cell volume porosity γ, the thermal diffusion D is computed as $D_{i+1/2} = 2F/(\gamma_i\Delta x_i/d_i + \gamma_{i+1}\Delta x_{i+1}/d_{i+1})*(h_i - h_{i+1})$

IDRODT Control parameter for the treatment of the compressibility term

$$\left(\frac{\partial \rho}{\partial t}\right)_0 \text{ in the pressure equation } (\delta p = p - p^n)$$

$$a_0 \delta p_0^{r+1} - \sum_{\beta=1}^{6} a_\beta \delta p_\beta^{r+1} = -V_{r0}\left(\frac{\partial \rho}{\partial t}\right)_0 + G^r$$

where r is an iteration index, n indicates the last time step number and G^r collects all other terms of the right hand side :

≤ 0 $\cong \dfrac{\rho_0^r - \rho_0^n}{\Delta t}$ (*)

$1 - 9$ $\cong \dfrac{\rho_0^{r+1} - \rho_0^n}{\Delta t} = \dfrac{\rho_0^{r+1} - \rho_0^r}{\Delta t} + \dfrac{\rho_0^r - \rho_0^n}{\Delta t}$

$\cong \dfrac{1}{\Delta t}\left(\dfrac{\partial \rho}{\partial p}\right)_0^r \left(p_0^{r+1} - p_0^r\right) + \dfrac{\rho_0^r - \rho_0^n}{\Delta t}$

$= \dfrac{1}{\Delta t}\left(\dfrac{\partial \rho}{\partial p}\right)_0^r \left(\delta p_0^{r+1} - \delta p_0^r\right) + \dfrac{\rho_0^r - \rho_0^n}{\Delta t}$

Introducing the compressibility into the pressure equation, the term with the unknown δp_0^{r+1} ought to be brought to the left hand side, thus improving diagonal dominance.
The same applies to the next option.

$$\geq 10 \quad = \left(\frac{\partial \rho}{\partial p}\right)_h \frac{\partial p}{\partial t} + \left(\frac{\partial \rho}{\partial h}\right)_p \frac{\partial h}{\partial t}$$

$$\cong \left(\frac{\partial \rho}{\partial p}\right)_h^r \frac{p_0^{r+1} - p_0^n}{\Delta t} + \left(\frac{\partial \rho}{\partial h}\right)_p^n \frac{h_0^{r+1} - h_0^n}{\Delta t}$$

$$= \frac{1}{\Delta t}\left(\frac{\partial \rho}{\partial p}\right)_h^r \delta p_0^{r+1} + \left(\frac{\partial \rho}{\partial h}\right)_p^n \frac{h_0^{r+1} - h_0^n}{\Delta t}$$

Note: For IDRODT = 2, 4, 6, .. the slope of the vapor pressure curve is computed from the Clausius-Clapeyron-Equation. Otherwise the analytical derivative of the semi-empirical vapor pressure is used.

The two variables below give the user some control over the frequency that the momentum and enthalpy calculations are performed. The need for this control might arise in cases when one of the two fields (velocity or enthalpy) varies slowly compared to the other. The intent is to be able to perform one of the calculations (momentum or enthalpy) every time step while performing the other only occasionally resulting in a savings of CPU time. Before the user activates these variables it is highly recommended that s/he understands the full implications of this approximation. The following combinations are allowed:

ISETMO = 1 and ISETEN = N where N is any nonzero integer.

ISETMO = N and ISETEN = 1 where N is any nonzero integer.

ISETMO = M and ISETEN = N where one of the following conditions is satisfied:

- M < 0 and N divides M or
- N < 0 and M divides N

ISETEN	$N<0$	When N is less than zero the enthalpy calculation is turned off every (-N)th timestep.
	$N>0$	When N is greater than zero the enthalpy calculation is turned on only every Nth timestep. (1)
ISETMO	$N<0$	When N is less than zero the momentum calculation is turned off every (-N)th timestep.
	$N>0$	When N is greater than zero the momentum calculation is turned on only every Nth timestep. (1)
		Note: At the commencement of a computation ISETMO must not be set (different from 1).

LCRES	≥ 0	If the CRESOR solution method is used for the pressure equation, LCRES is the orthogonalization level for the residual increments. (2) LCRES is limited to 7 by the code.
NUMDIF	≥ 0	Cf. paragraph on numerical diffusion. (0)
FACFRK	$< .5$	Cf. paragraph on numerical diffusion. (0.1)
FACFRD	$< .5$	Cf. paragraph on numerical diffusion. (0.1)
FACFRE	$< .5$	Cf. paragraph on numerical diffusion. (0.1)
FACFRM	$< .5$	Cf. paragraph on numerical diffusion. (0.1)

ISTATE	0	Start of steady-state run. Geometry, boundary conditions, and initial conditions are specified from the input stream. Other parameters take default values or zero. (*)
	1	Continuation of a steady-state run. Initial conditions are read from the restart file of a previous run in which steady-state has not yet been achieved. Some parameters may be changed in the input stream.
	2	Beginning of a transient run. Initial conditions are read from the restart file of a previous run. It is desirable although not necessary that this previous run has achieved steady-state. Some parameters may be changed in the input stream.
	3	Continuation of a transient run. Initial conditions are read from the restart file of a previous beginning-of-transient run or continuation-of-transient run. Limited changes may be made in the input stream.

The following three parameters are used when ISTATE=2. In other cases these variables are ignored.

NEWTS	0	No new thermal structure input is read.
	1	New thermal structure information is read if ISTRUC=1 and ISTATE=2.
NEWREB	0	No new rebalancing information is read.
	1	New rebalancing information is read if IFREB>0 and ISTATE=2.
NEWFOR	0	No new force structure information is read.
	1	New force structure information is read if NFORCE>0 and ISTATE=2.
MODEL		Two-phase model flag (cf. Ref./3/).
	1	Slip-Model. A simplified Two-Phase Model with *either* • a constant slip ratio with the limiting case of the Homogeneous Equilibrium Model (slip ratio = 1.0) *or* • a constant relative slip velocity normalized by the mixture velocity in each of the coordinate directions. (*)
	2	Separated Phases Model (suspended).
I2PMUL		Controlling the meaning of the slip parameter triple VSLIPX/Y/Z.
	1102	VSLIPX is constant slip ratio, i.e. U_g / U_l. (*)
	1103	VSLIPX is constant normalized relative slip velocity, i.e. $(U_g - U_l) / U_m$. U_m is the mass weighted velocity of the phase mixture. (with similar expressions for the Y & Z-direction)
VSLIPX		Slip parameter for X-direction. (1.0)
VSLIPY		Slip parameter for Y-direction. (1.0)
VSLIPZ		Slip parameter for Z-direction. (1.0)

Note: The default values 1.0 (I2PMUL=1102) refer to the Homogeneous Equilibrium Model. (In the single - liquid or vapor - phase regime VSLIPX/Y/Z are not used).

Time and Time Step Related Parameters

TSTART		Initial time s. (0.0) Should be reset to zero at the beginning of a transient run, ISTATE=2.
IDTIME	0	The time step size is taken from the user specified variable DT.
	1	The time step size is determined internally as the smaller value of DT(1) and the product of the Courant time step size (largest allowable time increment given the conditions) and a user specified variable, RDTIME. (*)
DT(1)		Time step size for time steps 1 through LASTDT if IDTIME=0, resp. upper limit for time step size if IDTIME=1, s. (0.1)
DT(2)		Time step size for time steps after LASTDT, s. (0.1) This value is used only if IDTIME=0.
LASTDT		This variable in combination with DT allows the user to change the time step size during a run. The time step size for all time steps through LASTDT is taken from DT(1). After step number LASTDT, the time step size is taken from DT(2). (99999) This value is used only if IDTIME=0.
RDTIME		The time step size is computed internally as the product RDTIME times the largest allowable time increment following Courant. (0.8) This value is used only if IDTIME=1.
DTENER	≥1.0	The time step size used for integrating the enthalpy equation is the product DTENER times the basic time step as determined internally under control of IDTIME. (1.0) DTENER is meant to be utilized in stationary applications, where the settlement of the enthalpy (temperature) field strongly lags behind that of the velocity field. This occurs with natural convection problems, when the heat capacity is sufficiently high, as e.g. for water.
DTFUEL	>0.0	Time step used for integrating the 1-dimensional thermal structure heat conduction equation, and only until steady state is reached, s. (1.E+40)
	0.0	Input of a zero value results in replacing the default value of DTFUEL by the basic time step due to IDTIME, thus forcing the code to follow the normal integration procedure.
NTMAX		The maximum time step number for this run. Normal termination occurs after completion of this time step. (99999)
TIMAX		The maximum time of this run. Normal termination occurs after this time has been reached, s. (3.6E+7)
	Note:	TIMAX refers to the simulation or problem time and not the computer CPU time needed to run the problem, cf. TREST.
TREST		The amount of time remaining for the job is checked at the end of each iteration. If this amount is greater than TREST another iteration is performed. If not, the job terminates regularly, a restart file may be written (cf. IFRES). When running long jobs or jobs requiring several seconds per iteration, one might wish to choose a larger more conservative value of TREST, s. (5.0)

This implementation depends on the routine TREMAN which returns the time left until the total job time as specified on the JOB card has elapsed (cf. page 53).

Iteration Control Parameters

The general definitions and default values of control parameters are given in this section.

IT(1)   Number of inner iterations for time steps 1 through LASTIT. (1)

IT(2)   Number of inner iterations for time steps after LASTIT. (10)

During a transient (ISTATE=2 or ISTATE=3) *within each time* step the solution is to converge, i.e. IT iterations must suffice to reach the criteria. Failure of this is indicated by a double ## under the heading NTIME in the time step summary thus sparing the user inspecting the single criteria therein. An iteration number equal to IT (printed in the detailed iteration summary under the heading 'IT', cf. LMPRNT) only makes this failure most probable. Before increasing IT beyond 100, however, the user is encouraged to examine the input and results for possible improvements..

LASTIT   This variable in combination with IT allows the user to change the number of iterations per time step during a run. The number of iterations for all time steps through LASTIT is taken from IT(1). After step number LASTIT, the number of iterations is taken from IT(2). (99999)

ITENMX   Reserved for future code extensions.

ITMAXP   Number of iterations in the pressure iteration loop. (100)

ITMAXE   Number of iterations in the enthalpy iteration loop. (100)

ITMOMX   Number of iterations in the 'X/Y/ZMOMI' iteration loop (1)

ITMASX   Number of iterations in the 'PEQN' iteration loop. (1)

RELAXE   Relaxation factor for SOR-solution of the enthalpy equation. (0.95)

OMEGA   Relaxation factor for SOR-solution of the pressure equation (even CRESOR being adaptive, eventually utilizes OMEGA in an auxiliary SOR procedure). (1.5)

OMEGAE   Relaxation factor for the coefficients of the enthalpy equation. (0.8)

OMEGAV   Relaxation factor for the coefficients of the momentum equation. (0.8)

OMEGAM   Factor for pressure increment correction term (for IDRODT $\neq$ 0):
$- \omega_m \times \Delta t \int \delta L \, d\tau / \int \partial \rho / \partial p \, d\tau$, δL = mass residue $(kgs^{-1}m^{-3})$. (0.0)

OMEGAR   Controls density evaluation at boundaries (RLB). With OMEGAR > 0.0, RLB is determined from boundary values of enthalpy, temperature, and pressure (HLB, TLB, and PB). With OMEGAR $\leq$ 0.0, RLB values are copied from the adjacent boundary cells. Though not quite exact, this is more efficient.(0.0)

EPS1   Steady state convergence criterion parameter. (1.0E-5)

EPS2   Steady state convergence criterion parameter. (1.0E-6)

EPS3   Steady state convergence criterion parameter. (5.0E-5)

EPS4	Reserved for future code extensions.
EPS5	Enthalpy convergence criterion parameter. (1.0E-5)
	Note: For EPS1, EPS2, EPS3 cf. page 54.

Boundary Condition Types

All *external* surfaces must have a *velocity* boundary condition type and a *temperature/heat flux* boundary condition type.
Internal surfaces may also be assigned boundary condition types.

KFLOW(N)   Type of *velocity* boundary condition.
(The default for all NSURF surfaces is 1)

-6   Constant outlet velocity set from VELOC (N).

-5   Continuative mass flow outlet.
Allowing for *back flow* a temperature TEMP(N) should be input even in case KTEMP(N) = 400. By means of TEMP(N) < –273.2 °C this temperature is automatically evaluated from the outflowing enthalpy.

-4   Uniform velocity outlet.

-3   Free slip boundary.

-2   Continuative velocity outlet.

-1   Continuative momentum outlet.

1   Constant velocity boundary with normal velocity set from VELOC (N) or from the BOUNDARY VALUE INITIALIZATION RECORDs. The tangential component in effect is zero. The presence of a solid wall (no slip boundary) must be indicated by using this type of boundary condition (KFLOW (N) = 1) with the constant velocity set to 0.0. (*)

100+NF   Uniform transient velocity boundary with normal velocity set from the product of the NFth transient function and VELOC (N).

KTEMP(N)   Type of *temperature/heat flux* boundary condition.
(The default for all NSURF surfaces is 1)

1   Specified constant temperature boundary with temperature set from TEMP (N) or from the BOUNDARY VALUE INITIALIZATION RECORDs. (*)

Note: The surface heat flux is nominally computed considering the fluid conduction but not the presence of a wall. If one wishes to account for both the fluid convection and a wall conduction, the following four variables from the Wall Model section below must be specified: IHTWAL (N), HYDWAL (N), WALLDX (N), and MATWAL (N).

100+NF   Uniform transient temperature boundary with temperature set from the product of the NFth transient function and TEMP (N).

Note: The surface heat flux is computed with the options as specified above for KTEMP (N) = 1.

200	Specified constant heat flux boundary with normal heat flux set from TEMP (N) or from the BOUNDARY VALUE INITIALIZATION RECORDs.
300+NF	Uniform transient heat flux boundary with normal heat flux set from the product of the NFth transient function and TEMP (N).
400	Adiabatic or zero-diffusive heat flux boundary.
500+NF	Duct wall temperature boundary. This boundary condition type accounts for fluid convection, thermal capacity of the wall, and the heat transfer to the surrounding atmosphere or medium. The variables in the Wall Model section below must be specified. The transient function defined by NF is a multiplier of the volumetric heat source in the wall. If a constant volumetric heat source is desired, simply specify a value of 500 for KTEMP (N).
600	If NHEX > 0, KTEMP = 600 refers to the special heat exchanger model package.

KPRES(N)   Type of *pressure* boundary condition. Pressure boundary conditions are applied to the *surface elements* of the boundary surface specified. (The default for all NSURF surfaces is 0)

0	No pressure boundary condition is applied. (*)
1	No more used, leads to error stop.
2	Uniform constant pressure boundary with pressure set from PRES (N).
3	Static pressure (evaluated automatically from the gravity – determined by the gravity vector GRAVX, GRAVY, GRAVZ and the density depending on TEMP0 - and from the value PRES0 at the reference point centered in the cell with indices IPRES0, JPRES0, KPRES0) signifies the boundary condition for *all surface elements* of surface N. (Cf. KPRES = 4).
4	Static pressure signifies the boundary condition for only *one surface element* - belonging to the maximum coordinate indices - of surface N (cf. KPRES = 3). The remaining surface elements are governed by the selected flow condition for surface N.
100+NF	No more used, leads to error stop.
200+NF	Uniform transient pressure boundary with pressure set from the product of the NFth transient function and PRES (N).
300+NF	Transient 'static' pressure condition according to KPRES = 3.
400+NF	Transient 'static' pressure condition according to KPRES = 4.

Boundary and Cell Initialization

The following four input variables allow easy specification of uniform velocity, temperature/heat flux, pressure and volume fraction values at boundaries at the beginning of a run (ISTATE = 0).

By using the BOUNDARY VALUE INITIALIZATION RECORDs, nonuniform distributions can also be specified. Cf. page 45, Note 2.

To change the values of the five corresponding FLUTAN variables (VELBN, TLB/QBN, PB, THLB) on subsequent restarts the BOUNDARY VALUE INITIALIZATION RECORDs must be used (ISTATE = 2, or 3).

VELOC(N)	Initial velocity at surface N in the direction indicated by XNORML(N), YNORML(N), and ZNORML(N), ms^{-1}. (0.0)
TEMP(N)	According to the boundary condition type, cf. KTEMP(N):
	Initial temperature for surface N, °C. (0.0)
	Initial heat flux, Wm^{-2}. (0.0)
PRES(N)	Initial pressure for surface N, Pa . (0.0)
THETA(N)	Initial dimensionless boundary volume fraction. (1.0)
TEMP0	Initial temperature for all internal cells, °C. (0.0)
PRES0	Initial pressure at the pressure reference point, centered at the cell with the coordinate indices IPRES0, JPRES0, KPRES0, Pa . (1.01325E+5)
	The initial static head pressure at any point is computed with respect to the pressure reference point.

Gravity Vector

GRAVX	X-component of gravity vector, ms^{-2}. (0.0)
GRAVY	Y-component of gravity vector, ms^{-2}. (0.0)
GRAVZ	Z-component of gravity vector, ms^{-2}. (0.0)

Wall Model

The variables in this section are used when temperature boundary condition is either 1, or 100 + NF, or 500 + NF.

HYDWAL(N)　　Hydraulic diameter (characteristic length) associated with surface N, m. (0.0)

IHTWAL(N)　　Heat-transfer correlation number for the calculation of heat transfer between coolant and wall. The value of this variable is used as the index NH in the Fluid-Structure Heat Transfer section below. (0)

Note:　　If IHTWAL = 0 (default) is taken, then the coolant to wall heat-transfer coefficient is evaluated simply as the fluid conductivity divided by the fluid conduction length.

MATWAL(N)　　Material type for surface N. The value of this variable is used as the index NM in the Material Properties (Solids) section below. (0)

WALLDX(N)　　Wall thickness, m. (1.0)

Notes:　　1. Regarding the *reduced wall model* (KTEMP=1 or 100+NF) i.e. considering wall heat conduction only without the surrounding medium :
- if MATWAL = 0 (default) is taken, then wall resistance is neglected.

2. Regarding the *full wall model* (KTEMP=500 or 500+NF) i.e. considering both wall heat conduction and the surrounding medium :
- the option MATWAL = 0 from above is not allowed;
- the four following variables are used only with KTEMP=500 or 500+NF.

WALLQS(N)　　Average wall volumetric heat source, Wm^{-3}. (0.0)
Eventually to be multiplied by the transient function NF.

TSINK(N)　　Temperature of surrounding atmosphere or medium, °C. (0.0)

HSINK(N)　　Heat-transfer coefficient from wall to surrounding atmosphere or medium, $Wm^{-2}K^{-1}$. (0.0)

DTWALL　　Time step size used for with temperature boundary condition type 500+NF. This time step size is used only until steady-state is reached, s. (1.E+40)
Input of a zero value results in replacing the default value of DTWALL by the basic time step due to IDTIME, thus forcing the code to follow the normal integration procedure.
Cf. DTFUEL, page 14.

Fluid-Structure Heat Transfer

Heat transfer correlations are defined by specifying coefficients to compute the Nusselt number. These coefficients and thus the heat transfer correlations are indexed by the values of IHTWAL in the Wall Model and IHT in the THERMAL STRUCTURE PROTOTYPE RECORDs. The Nusselt number (NU) is computed from the following equation:

$$NU = HEATC1(NH) + HEATC2\,(NH) \times RE^{HEATC3(NH)} \times PR^{HEATC4(NH)}$$

where
  RE is the Reynolds number, and
  PR is the Prandtl number.

NHEATC Number of heat transfer correlations. (1)

    This value must be at least as large as the largest value of IHT and IHTWAL

HEATC1(NH) Nusselt number coefficient.

    Since the Nusselt number, NU, must always be positive, HEATC1 (NH) should be positive to accommodate a zero-flow situation. (3.66)

HEATC2(NH) Nusselt number coefficient. (0.023)

HEATC3(NH) Nusselt number coefficient. (0.8)

HEATC4(NH) Nusselt number coefficient. (0.4)

    Note: The default values are set for water, must be respecified for sodium or user-provided fluid packages.

The Nusselt number is used to specify the heat transfer coefficient (h) in the following equation

$$h = (k/D) \times NU$$

where
  k is the conductivity and
  D is the reference length.

h is in turn used to compute the Fluid-Structure heat transfer (q) as follows:

$$q = A \times h \times (T_s - T_f)$$

where
  A is the area,
  T_s is the temperature of the structure, and
  T_f is the temperature of the fluid.

QK( ) Thermal Structure heat source multiplier. (1.0)
    As the Thermal Structure is axially aligned, the index of the intervals along that axis is corresponding to the index of QK( ).

Material Properties of Solids and Fluids other then Coolant

The following equations are used to define the thermal conductivity, specific heat, and density of materials applying to the *Wall Model, Thermal Structure* and *Heat Exchanger* (cf.Ref./14/) options :

$$\text{CONDUCTIVITY} = \text{C0K (NM)} + \text{C1K (NM)} \times \text{TC} + \text{C2K (NM)} \times \text{TC}^2 \quad \text{Wm}^{-2}\text{K}^{-1}$$

$$\text{SPECIFIC HEAT} = \text{C0CP (NM)} + \text{C1CP (NM)} \times \text{TC} + \text{C2CP (NM)} \times \text{TC}^2 \quad \text{Jkg}^{-1}\text{K}^{-1}$$

$$\text{DENSITY} = \text{C0RO(NM)} + \text{C1RO (NM)} \times \text{TC} + \text{C2RO(NM)} \times \text{TC}^2 \quad \text{kgm}^{-3}$$

Equation used to define the dynamic viscosity of *heat exchanger* fluids (cf. Ref./14/):

$$\text{VISCOSITY} = \text{C0MU(NM)} + \text{C1MU(NM)} \times \text{TC} + \text{C2MU(NM)} \times \text{TC}^2 \quad \text{kgm}^{-1}\text{s}^{-1}$$

TC is the temperature in °C and NM is the number of the material.

The coefficients listed below are indexed by values of MATWAL from the *Wall Model* section of NAMELIST /DATA/, MI from the THERMAL STRUCTURE DESCRIPTION RECORDs and MCHE from NAMELIST /HEXD/ (cf.Ref./14/) :

NMATER Number of materials. (0) This value must be at least as large as the largest value of MATWAL and MI.

C0K(NM) Conductivity coefficient. (0.0)

C1K(NM) Conductivity coefficient. (0.0)

C2K(NM) Conductivity coefficient. (0.0)

C0CP(NM) Specific heat coefficient. (0.0)

C1CP(NM) Specific heat coefficient. (0.0)

C2CP(NM) Specific heat coefficient. (0.0)

C0RO(NM) Density coefficient. (0.0)

C1RO(NM) Density coefficient. (0.0)

C2RO(NM) Density coefficient. (0.0)

The following coefficients are indexed by values of MCHE from NAMELIST /HEXD/ only:

C0MU(NM) Viscosity coefficient. (0)

C1MU(NM) Viscosity coefficient. (0)

C2MU(NM) Viscosity coefficient. (0)

Transient Functions

All transient driving functions are input into the following three variables. They must be input at the beginning of the transient (ISTATE=2) even if they have been input previously. Each function is defined by a user specified set of points. Cubic spline fit coefficients are then generated in SUBROUTINE FITIT.

50 equally spaced values are printed to allow the user to check the adequacy of the input distribution. About 10-15 values with points concentrated at rapidly changing Y-values should be adequate.

Currently the the specification of transient functions is limited in two respects

• the maximum number of transient functions is 25
• the total number of points allowed for the specification of transient functions is 100.

TVAL(NP)	The independent variable, usually time, for the transient functions.
FVAL(NP)	The dependent variable for the transient functions. The first value of the second function immediately follows the last value of the first function. The same pattern must be followed for all subsequent functions. Make sure that the entire range of the function used lies within the range input as the fitting routine does not extrapolate. Discontinuities are indicated by specifying the same X-coordinate twice with the same or different Y-coordinate values.
NEND(NF)	The number of points in the NFth transient function.
NTOTS	In order to simplify thermal structure input in certain cases, the heat source transient function numbers can be overridden in NAMELIST /DATA/. These values are input into the variable NTOTS in the order in which the thermal structure prototypes were defined. Any values specified in NTOTS will override all other input and previous values. If no values of NTOTS are defined, no changes to the heat source transient function numbers are made.
NOFQT	Number of the transient function which is used as a multiplier of total heat source where no thermal structures act as heat sources. (0) Cf. QSOU, Q

Plotting Option (Use of Unit 20)

Several FLUTAN arrays with physical quantities may be written to unit 20 (sequential file, binary data, 4 byte integer, 8 byte real) for off-line plotting purposes, i.e. isoline and vector plots for coordinate planes . (Cf. Special Tools; cf. VISUALIZATION OPTION)

Note: If (for IFRES=1,3) *restart information* is written to unit 10, the *plotting information* due to IFPLOT and pertaining to the restart time step number is *appended* to unit 10.

The following list is showing the physical quantities and the corresponding FORTRAN arrays which may be selectively written to unit 20:

1.	Velocity	UL, VL, WL, VELBN
2.	Temperature	TL, TLB
3.	Pressure	P, PB
4.	Enthalpy	HL, HLB
5.	Density	RL, RLB
6.	Turbulent Kinetic Energy	TURK, TURKB
7.	Turbulent Kinetic Energy Dissipation	TKED, TKEDB
8.	Turbulent Viscosity	TURVIS
9.	Turbulent Conductivity	TURCON
10.	Volume Fraction	THL, THLB

Two input arrays have to be used to specify the plotting information:

IFPLOT   An array of 10 values, 1 for each of the numbered quantities listed above, is used to specify whether the corresponding FLUTAN arrays are to be written to unit 20 for plotting purposes, or not:

    1   The corresponding arrays are written.
         (*) for IFPLOT(1) and IFPLOT(2).
         (I.e. temperature and velocity arrays are written by default).

    0   The corresponding arrays are not written.
         (*) for IFPLOT(3) to IFPLOT(10).

NTPLOT   Up to 25 values to specify when plotting information is to be written to unit 20. The following are acceptable values of NTPLOT :

    0   No (more) plotting information is written

    N > 0   Time step number for which plotting information is written to unit 20. After the Nth positive time step in NTPLOT has been processed, the (N+1)th value of NTPLOT is used to determine subsequent writes.

    N < 0   A negative N indicates that information is written to unit 20 every INIth time step. No subsequent values of NTPLOT are considered.

Examples:
NTPLOT =   -5 indicates that every 5th step is to be processed (starting with 5).
NTPLOT =   6, 11, -20 indicates that steps 6, 11, 20, 40, 60, etc., are to be processed.
NTPLOT =   5, 20, 0 indicates that only steps 5 and 20 are to be processed.

Printing Option

The variables NTPRNT and TPRNT specify the time points (i.e. end points of time steps, cf. IDTIME), at which the subroutine OUTPUT is to be called. They can be used individually or together. The information printed at each call to subroutine OUTPUT is determined by the variables ISTPR and NTHPR described below.

NTPRNT   Up to 50 time step numbers at which subroutine OUTPUT is to be called. The acceptable values for NTPRNT are:

0   No more calls to OUTPUT.
When *restarting*, previous specification of NTPRNT values may be *overridden* by specifying the desired new values followed by a zero in NTPRNT.

N > 0   Time step number for which subroutine OUTPUT is to be called.
After the Nth positive time step in NTPRNT has been processed the N+1th value of NTPRNT is used to determine subsequent calls to OUTPUT.

N < 0   A negative N indicates that subroutine OUTPUT is called every |N|th time step. No subsequent values of NTPRNT are considered.

-9999   Subroutine OUTPUT is called just before the run is terminated. (*)

Examples:
NTPRNT = 0 indicates that after initialization, OUTPUT is never called.
NTPRNT = 5, 10, -9999 indicates that subroutine OUTPUT is called at steps 5, 10, and just before termination.

TPRNT   Up to 50 problem time values specifying time steps at which subroutine OUTPUT is to be called. The acceptable values for TPRNT are (s) :

0.0   No more calls to OUTPUT. (*)
When *restarting*, previous specification of TPRNT values may be *overridden* by specifying the desired new values followed by a zero in TPRNT.

T > 0.0   The subroutine OUTPUT is to be called at the end point of the time step *selected* by T in that the *associated* time step *comprises* T: associated time steps arise from the original ones by shifting each time point to the right by 1% of the preceeding (original) time step length.
After the Nth positive time in TPRNT has been processed, the N+1th value of TPRNT is used to determine subsequent calls to OUTPUT.

T < 0.0   A negative T indicates a series of values at |T|-second intervals:
If the Nth value is negative, then the (N+1)th value stores the next time value at or after which OUTPUT is to be called. This is nominally set to zero but can be specified by the user. No subsequent values of TPRNT are considered.

Examples:
TPRNT = 1.0, 6.0, -10.0, indicates that OUTPUT is to be called at steps selected by 1.0, 6.0, 10.0, 20.0, .. etc..
TPRNT = 1.0, 5.0, -10.0, 6.0, indicates that OUTPUT is to be called at steps selected by 1.0, 5.0, 6.0, 16.0, 26.0, .. etc.

ISTPR   Up to 50 coded values which specify the arrays to be printed in the first call to subroutine OUTPUT. (0)

NTHPR   Up to 50 coded values which specify the arrays to be printed in all calls after the first call to OUTPUT.

For 'internal arrays', each value of ISTPR and NTHPR is a signed integer of the form 'SVVPLL' (5 or more digits; cf.LL) which is coded according to the following rules:

S	+	Only the plane specified by 'VVPLL' is to be printed. (*) (A plus sign is assumed and need not be specified.)
	–	The planes with indices from some range of values are to be printed. Cf. LL below.

VV	01	UL	U-component of velocity
	02	VL	V-component of velocity
	03	WL	W-component of velocity
	04	HL	Enthalpy
	05	TL	Temperature
	06	AL	Volume porosity
	07	RL	Density
	08	P	Static pressure
	09	DL	Residual mass

(01 - 09 may be specified without the leading zero)

	10	ALX	X-direction surface permeability
	11	ALY	Y-direction surface permeability
	12	ALZ	Z-direction surface permeability
	13		not used
	14	TURK	Turbulent kinetic energy
	15	QSOUR	Volumetric heat source
	16	PSTAT0	Initial static pressure
	17		P-PSTAT0
	18	THL	Dimensionless volume fraction
	19		not used
	20	TURCON	Turbulent conductivity
	21	TURVIS	Turbulent viscosity
	22	TKED	Dissipation of turbulent kinetic energy

P	1	An I - plane is specified (LL below is I index)
	2	A J - plane is specified (LL below is J index)
	3	A K - plane is specified (LL below is K index)

LL   I-, J- or K-index of specific plane to be printed:
LL has to be specified with the number of digits (leading zeros eventually), necessary to represent the maximum of IMAX, JMAX, KMAX and NSURF, but with two digits at least.
If S is '+', only one plane is indicated.
If S is '–', the couple of 'LL' values from the current and from the next item of ISTPR or NTHPR is indicating the range of planes to be printed.
(The SVVP portion of the "next" item is not significant.)

For heat exchanger respectively thermal structure information, both quoted as 'structure' below, each value of ISTPR/NTHPR is a signed integer of the form 'S7NNNN' respectively 'S8NNNN' (5 or more digits; cf. NNNN) which is coded according to the following rules:

S	+	Only structure number 'NNNN' is to be printed. (*)
		(A plus sign is assumed and need not be specified.)
	−	The structures with numbers from some range of values are to be printed. (Cf. NNNN below).

NNNN Number of specific structure to be printed.

NNNN has to be specified with the number of digits of LL plus 2, (cf. LL).

If S is '+', only one structure is to be printed.

If S is '−', the 'NNNN' values in the current and next items of ISTPR/NTHPR indicate the range of structures to be printed.

For 'surface arrays', each value of ISTPR/NTHPR is a signed integer of the form 'S9VVLL' (5 or more digits; cf.LL) which is coded according to the following rules:

S	+	Only the surface with number 'LL' is to be printed. (*)
		(A plus sign is assumed and need not be specified.)
	−	All surfaces with numbers between the values of 'LL' in the current and the next items of ISTPR/NTHPR are to be printed.

VV	01	VELBN	Normal surface velocity.
	02	QBN	Normal surface heat flux.
	03	MB	Adjacent internal cell number.
	04	HLB	Surface enthalpy.
	05	TLB	Surface temperature.
	06	AREA	Surface element area.
	07	RLB	Surface density.
	08	PB	Surface pressure.
	09		Adjacent internal cell indices. Each value is of the form 'IIJJKK' where II is the I index, JJ is the J index, and KK is the K index in LL-format, cf. LL below.
	10		Overall heat transfer coefficient from coolant to wall as used in the transient duct wall model (KTEMP(LL)=500).
	11	THLB	Dimensionless Boundary Volume Fraction.

LL Number of specific surface to be printed.

LL has to be specified with the number of digits (leading zeros eventually), necessary to represent the maximum of IMAX, JMAX, KMAX and NSURF, but with two digits at least.

If S is '+', only one surface is to be printed.

If S is '−', the couple of 'LL' values from the current and the next item of ISTPR/NTHPR is indicating the range of surfaces to be printed.

(The S9VV portion of the "next" item is not significant.)

Example.

ISTPR = 6105, −10301, −10305,

(−10305 may be replaced by 99905 or −05 or 5 e.g.)

NTHPR = 1105, −2302, 5, 90101, −90501, 4,

The first call to OUTPUT will print the

- plane for I=5 of volume porosity and
- planes for K=1 through 5 of the X-direction surface permeability.

All subsequent calls to OUTPUT will print the

- plane for I=5 of the U-component of velocity,
- planes for K=2 through 5 of the V-component of velocity,
- the boundary velocity for surface 1 and
- surface temperature for surfaces 1 through 4.

LMPRNT	*0*	No detailed information of iterative processes is printed.
	1	Detailed iteration information is printed for each time step. (*)
	N< 0	Detailed iteration information is printed for the first, last end every INI-th time step, while the output of convergence criteria is *reduced* to the same time steps (LMPRNT = 1, and −1 have the same meaning).
	2	Cell number and surface number arrays are printed (useful only for debugging purposes).

Force Structures

The Force Structure parameters are required only when NFORCE of NAMELIST /GEOM/ is greater than zero.

The Force Structure is a mechanism whereby a force can be applied to the fluid across a cell face between two computational cells. To this end the user has to establish some generic force correlation(s), as follows:

For the laminar regime:

$$FCORR = ACORRL(NC) \times RE^{BCORRL(NC)} + CCORRL(NC)$$

For the turbulent regime:

$$FCORR = ACORRT(NC) \times RE^{BCORRT(NC)} + CCORRT(NC)$$

Regime transition is accomplished automatically by calculating both values and taking the larger one.

$$RE = RL \times \sqrt{(UL^2 + VL^2 + WL^2)} \times REYLEN(NF) / VIS$$

> RL is the local density,
>
> UL, VL, and WL are local velocities, and
>
> VIS is the local viscosity.

Then the drag or resistance forces (**Pa m^{-1}**) will have the following components:

$$DPDX = - FORCEF(NF) \times RL \times ABS(UL) \times UL \times FCORR / CLENTH(NF)$$

$$DPDY = - FORCEF(NF) \times RL \times ABS(VL) \times VL \times FCORR / CLENTH(NF)$$

$$DPDZ = - FORCEF(NF) \times RL \times ABS(WL) \times WL \times FCORR / CLENTH(NF)$$

Note: The FORCE STRUCTURE SPECIFICATION RECORDs (cf. page 39) are used to specify the *locations* of the Force Structures.

FORCEF(NF)		Force coefficient for force structure NF.
REYLEN(NF)		Length used to compute the Reynolds number for force structure NF, **m.**
CLENTH(NF)	*>0.0*	The value input is used as the characteristic length in the above equation, **m.**
	<0.0	A characteristic length computed from either DX, DY, or DZ, whichever is appropriate, is used for CLENTH(NF) in the above equation.
ICORR(NF)		The correlation type of force structure NF. The values of ICORR must not exeed NCORR and are used as indices NC of the user specified correlation coefficients below, viz. NC = ICORR(NF).
NCORR		The number of correlation types available for force structures. This value must equal or exceed the maximum value specified in ICORR but be 20 at most, i.e. $1 \leq NC \leq NCORR \leq 20$.
ACORRL(NC)		Correlation coefficients *when* the Reynolds number above,
BCORRL(NC)		RE,
CCORRL(NC)		is in the *laminar* regime.

ACORRT(NC)   Correlation coefficients *when* the Reynolds number above,

BCORRT(NC)   RE,

CCORRT(NC)   is in the *turbulent* regime.

Note: At restart (ISTATE=2, NEWFOR=1, cf. page 13) redefinition of the force structure NF is possible through a change of FCORR by means of the correlation coefficients above (neither FORCEF, REYLEN, CLENTH nor ICORR can get changed).
Force structure NF corresponds to force correlation NC through NC = ICORR(NF)

Reducing Numerical Diffusion

When the direction of the flow is highly oblique to the grid lines, numerical diffusion may be significant. Several options to reduce this numerical diffusion are currently under assessment. The default for computing the convective flux terms of the enthalpy and momentum equation is the pure-upwind differencing scheme (NUMDIF=0).

If, however, the user feels that reducing numerical diffusion is necessary for a specific problem then he may override the default value of NUMDIF (0) by specifying NUMDIF as a number with 7 digits [abcdefg] having the following meaning (leading zeros need not be specified):

Digits a,b,c and e		signify either five options for the treatment of convective terms in the
Digit a		- equation for the turbulent kinetic energy
Digit b		- equation for the turbulent kinetic energy dissipation rate
Digit c		- enthalpy equation
Digit e		- momentum equations
	0	1st-order upwind technique
	1	(not used)
	2	QUICK technique
	3	QUICK assisted by the FRAM technique
	4	LECUSSO technique
	5	LECUSSO assisted by the FRAM technique.
Digit d		(reserved for future use).
Digit f		signifies handling of boundary conditions for the convective terms in the momentum equations:
	0	1st-order upwind technique
	1	interpolation using the arithmetic average

	2	• for X-X, Y-Y, Z-Z terms: the same as 1
		• for cross terms X-Y; X-Z; Y-X; etc.:
		  − linear interpolation (symmetry condition only)
		  − 2nd-order interpolation (else)

Digit g		signifies the way to estimate the transport velocity at the center plane of a momentum control volume:
	0	linear interpolation between front and back surface
	1	hard-limited 2nd-order interpolation between front and back surface,
	2	LECUSSO interpolation (implementation pending).

Notes:

1. Cf. Ref./7/, /8/, /9/, /19/ about the LECUSSO technique.
2. The FRAM technique utilizes the discrimination variables FACFRK, FACFRD, FACFRE and FACFRM respectivly in the equations for Turb. Kinetic Energy, Turb. Kinetic Energy Dissipation Rate, Enthalpy and Momentum. The default value 0.1 for all variables may be overridden in NAMELIST /DATA/.

Rebalancing Option

Large scale pressure distributions such as those which exist in an initial static state or which occur during overall velocity transients are most effectively addressed with the coarse mesh rebalancing scheme. This rebalancing is effective in reducing the number of iterations required to achieve convergence of the pressure equation.

Rebalancing has been implemented in two different modes which can be only applied separately. Plane-by-plane rebalancing in the X-, Y-, or Z-direction can be applied simply by specifying the appropriate values for IXREB, IYREB, and IZREB. Only one plane-by-plane rebalancing option can be specified.

IXREB	*0*	No X-direction plane-by-plane rebalancing. (*)
	1	Plane-by-plane rebalancing in the X-direction is performed.
IYREB	*0*	No Y-direction plane-by-plane rebalancing. (*)
	1	Plane-by-plane rebalancing in the Y-direction is performed.
IZREB	*0*	No Z-direction plane-by-plane rebalancing. (*)
	1	Plane-by-plane rebalancing in the Z-direction is performed.

In each rebalancing region the pressure is adjusted uniformly in such a way as to force the net mass conservation.

The frequency at which rebalancing occurs is specified by the following variable.

IREBIT	*N>0*	Rebalancing is performed before every Nth iteration. (10)

If user-defined-region rebalancing is desired, IFREB in NAMELIST /GEOM/ must be assigned an appropriate positive value and NAMELIST /REBAL/ must be supplied plus -optional- REBALANCING REGION RECORDs.

One approach to choosing rebalancing regions is to exclude all cells adjacent to exits and then group the remaining cells into as many rebalancing regions as possible. Another guideline is to put rebalancing surfaces between regions of grossly different pressures.

TURBULENCE MODELING

For Turbulence Modeling in FLUTAN cf. Ref./1/, /2/, /4/, /5/, /10/ !

In all of the following turbulence models an effective viscosity is used in the diffusion term of the momentum equation. This effective viscosity is the sum of the turbulent viscosity and the molecular viscosity. Similarly an effective thermal conductivity is used in the diffusion term of the enthalpy equation which is likewise the sum of the turbulent thermal conductivity and the molecular thermal conductivity.

Constant Turbulent Diffusivity Model (ITURKE=0)

The turbulent viscosity and turbulent conductivity are assumed to have some constant value (≥ 0.0) everywhere.

For this option the following input must be specified:

TURBC   Turbulent conductivity, $\mathbf{Wm^{-1}K^{-1}}$. (0.0)

TURBV   Turbulent viscosity, **Pa s**. (0.0)

Zero-Equation Turbulence Model (ITURKE=10)

This option does not solve any governing equations involving turbulent quantities.

The turbulent viscosity (TURVIS) is computed from the following equation:

$$\mu_t = \rho \, l_m^2 \, \sqrt{\sum \left(\partial U_i / \partial x_j \left(\partial U_i / \partial x_j + \partial U_j / \partial x_i \right) \right)}$$

U_i average velocity in the direction of the i'th coordinate

l_m Prandtl mixing length scale (χy_w)

y_w distance to the nearest wall determined by the code
(a cutoff value of $0.175 \times$ HYDIN is used)

χ von Karman constant (AKAPPA)

According to the definition of the turbulent Prandtl Number (PRNDLH) for thermal energy transfer

$$Pr_t = v_t / \Gamma_t \quad (v_t = \mu_t / \rho; \quad \Gamma_t = \lambda_t / (\rho \, c_p) ; \text{ turbulent thermal diffusivity})$$

the turbulent conductivity (TURCON) is computed from the following equation:

$$\lambda_t = c_p \, \mu_t / Pr_t$$

Accounting for buoyancy effects, both v_t and Γ_t are amplified by a factor introducing the Richardson number Ri determined by the code

$$v_t = v_t^0 (1 + \beta_v \times R_i)^{\alpha_v}$$

$$\Gamma_t = \Gamma_t^0 (1 + \beta_\gamma \times R_i)^{\alpha_\gamma}$$

<u>For this option the following input must be specified:</u>

AKAPPA   von Karman constant. (0.4)

HYDIN   Hydraulic diameter, **m**. (X(IMAX))

OMEGAT   Relaxation factor for the turbulent viscosity. (0.7)

PRNDLH   Turbulent Prandtl number for thermal energy transfer,
viz. for $Ri=0 : Pr_t = v_t^0/\Gamma_t^0$. (0.9)

ALFAN   Coefficient for turbulent kinematic viscosity, viz. α_v. (-0.5)

BETAN   Coefficient for turbulent kinematic viscosity, viz. β_v. (10.0)

ALFAG   Coefficient for turbulent thermal diffusivity, viz. α_γ. (-1.5)

BETAG   Coefficient for turbulent thermal diffusivity, viz. β_γ. (3.33)

Note: The user is advised to override the default value for HYDIN.

One-Equation Turbulence Model (ITURKE=11)

⇒ The equation for turbulent kinetic energy (TURK) is solved:

$$\frac{\partial(\rho k)}{\partial t} + \sum \frac{\partial(\rho k U_i)}{\partial x_i} = \sum \frac{\partial(\mu^k \partial k/\partial x_i)}{\partial x_i} + S_k$$

k   turbulent kinetic energy, $m^2 s^{-2}$

U_i average velocity in the direction of the i'th coordinate

μ^k $\mu_l + \mu_t/\sigma_g$ effective (laminar plus turbulent) diffusivity

σ_g Prandtl number for turbulent kinetic energy (PRNDLK)

S_k $P_k + S_g + \rho\varepsilon$ source of turbulent kinetic energy made up of
        − production due to the main stream or alternately due to wall effects
        − production or dissipation due to buoyancy
        − dissipation through the fluid viscosity (sink)

⇒ The dissipation rate of the turbulent kinetic energy, ε, is computed from the following equation (using the solution of the equation for the kinetic turbulent energy k):

$$\varepsilon = \frac{c_\mu^{3/4} k^{3/4}}{\chi y_w}$$

c_μ coefficient for computation of turbulent viscosity (CDTURB)

χ von Karman constant (AKAPPA)

y_w distance to the nearest wall
        (a cutoff value of $0.175 \times HYDIN$ is used in the code).

⇒ The turbulent viscosity (TURVIS) is computed using the following equation:

$$\mu_t = \frac{c_\mu \rho k^2}{\varepsilon}$$

c_μ coefficient for computation of turbulent viscosity (CDTURB)

ρ local density

⇒ Wall function corrections are applied to cells adjacent to solid walls for both the turbulent kinetic energy equation and the momentum equations.

⇒ The turbulent shear stress in the turbulent zone next to the viscous sublayer is computed from the following equation:

$$\tau^t = \frac{\chi \rho c_\mu^{1/4} U_p k^{1/2}}{\ln\left(\dfrac{E \rho c_\mu^{1/4} y_w k^{1/2}}{\mu_t}\right)}$$

ρ density

U_p velocity component parallel to the wall

y_w wall distance

μ_t turbulent viscosity

χ von Karman constant (AKAPPA)

c_μ coefficient for computation of turbulent viscosity (CDTURB)

E wall roughness (EE)

<u>For this option the following input must be specified:</u>

AKAPPA   von Karman constant. (0.4)

CDTURB   Coefficient for computation of turbulent viscosity. (0.09)

EE   Wall roughness coefficient. (9.0)

EPS6   Convergence criterion parameter for the equation of the turbulent kinetic energy (cf. *Two Equation Turbulence Model*, note 1.a)). (1.0E-5)

HYDIN   Hydraulic diameter, m. (X(IMAX))

ITMAXK   Maximum number of iterations in the equation for the turbulent kinetic energy (cf. *Two Equation Turbulence Model*, note 1.a)). (50)

IFITUP   Initialization index for the inlet values of the turbulent kinetic energy. (Cf. *Two Equation Turbulence Model*, note 1.b)).

   0   Inlet values are computed by means of TKIN (and TDIN), cf. below. (*)

   ≠ 0   Inlet values are set equal to TKBD (and TDBD), cf. below.

ITURMX   Reserved for future code extensions. (1)

OMEGAK	Relaxation factor for the coefficients of the equation for the turbulent kinetic energy. (0.7)
OMEGAT	Relaxation factor for the turbulent viscosity. (0.7)
PRNDLH	Turbulent Prandtl number for thermal energy transfer. (0.9)
PRNDLK	Turbulent Prandtl number for turbulent kinetic energy. (1.0)
RELAXK	Relaxation factor for the SOR-solution of the equation for the turbulent kinetic energy (cf. *Two Equation Turbulence Model*, note 1.a)). (0.8)

Note : For the next two parameters cf. IFITUP above.

| **TKIN** | Coefficient to initialize the inlet turbulent kinetic energy. (0.001) $\quad$ TURKB = TKIN × VELBN^2. (VELBN = inlet velocity) |
| **TKBD(N)** | Initial inlet values of the turb. kinetic energy on surface N. (0.0) |

Note: The user is advised to override the default value for HYDIN and TKIN.

Two-Equation Turbulence Model (ITURKE=12)

This is the most rigorous turbulence model:

⇒ The equation for the turbulent kinetic energy is solved (TURK, cf. One-Equation Model)

⇒ The equation for the dissipation rate of turbulent kinetic energy (TKED) is solved:

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \sum \frac{\partial(\rho\varepsilon U_i)}{\partial x_i} = \sum \frac{\partial(\mu^\varepsilon \partial\varepsilon/\partial x_i)}{\partial x_i} + S_\varepsilon$$

ε	turbulent kinetic energy dissipation rate, $m^2 s^{-3}$
U_i	average velocity in the direction of the i'th coordinate
S_ε	$c_{1\varepsilon}(\varepsilon/k)(P_k + S_g)(1 + c_{3\varepsilon}R_f) - c_{2\varepsilon}\rho\,\varepsilon^2/k$ source of turbulent kinetic energy dissipation with
	− P_k und S_g being the terms from the turbulent kinetic energy equation
	− R_f the Richardson number, viz. − S_g/P_k
μ^ε	$\mu_l + \mu_t/\sigma_\varepsilon$ effective (laminar plus turbulent) diffusivity
σ_ε	Prandtl number for turbulent kinetic energy dissipation (PRNDLD)

⇒ The turbulent viscosity (TURVIS) is computed

⇒ Wall function corrections are applied to cells adjacent to solid walls

⇒ The turbulent shear stress in the turbulent zone next to the viscous sublayer is computed following the same procedure as in the *One Equation Turbulence Model* above.

For this option, in *addition* to the input for the *One Equation Turbulence Model* above, the following input parameters must be specified:

Three empirical constants used to compute the source term in the equation for the turbulent kinetic energy dissipation rate:

CT1 $c_{1\varepsilon}$ (1.47)

CT2 $c_{2\varepsilon}$ (1.92)

CT3 $c_{3\varepsilon}$ (0.8)

OMEGAD   Relaxation factor for the coefficients of the equation for the turbulent kinetic energy dissipation rate. (0.7)

PRNDLD   Turbulent Prandtl number for the turbulent kinetic energy dissipation rate. (1.3)

        Note : For the next two parameters cf. IFITUP above.

TDIN   Coefficient to initialize the inlet values of the turbulent kinetic energy dissipation rate, m^{-1}. (2000.)

               $TKEDB = TDIN \times TURKB^{1.5}$. (TURKB = inlet turbulent kinetic energy)

               TDIN   can be either determined empirically or by using the equation: $TDIN = CDTURB/(0.04 \times HYDIN)$

TDBD(N)   Initial inlet values of the turb. kinetic energy dissipation rate on surface N. (0.0)

Notes:

    1. For this option the input parameters
       a)   EPS6, ITMAXK and RELAXK (to be specified for the turbulent kinetic energy solution procedure, cf. *One Equation Turbulence Model*)
       b)   IFITUP (to be specified for the turbulent kinetic energy inlet value initialization procedure, cf. above)
       are used for the turbulent kinetic energy dissipation rate accordingly.
    2. The user is advised to override the default values HYDIN, TKIN and TDIN.

FORCE STRUCTURE SPECIFICATION RECORDS

This set of records - *to be completed with an END record* - must be included *only* when NFORCE > 0 in NAMELIST /GEOM/ (cf. ISTATE, NEWFOR in NAMELIST /DATA/).

These records are used to locate the force structures described in the Force Structure section of NAMELIST /DATA/. These forces can be applied at cell faces between two computational cells. The locations therefore correspond to portions of grid planes. Each record in this section contains the following variables in the FORMAT (A4, 7I4).

KEY	NF IB IE JB JE KB KE

KEY
    XFOR   X-direction force.

    YFOR   Y-direction force.

    ZFOR   Z-direction force.

    END   This record terminates the FORCE STRUCTURE SPECIFICATION RECORDs. It is only necessary when NFORCE > 0.

NF   Force structure number

IB, IE, JB, JE, KB,KE   These six variables are the beginning and ending I-, J-, and K-indices that define a rectangular solid composed of one or more cells defining cell faces at which the force with force structure number NF is to be applied:

The cell face defined by cell (I,J,K) for an X-direction force is that one between cells (I,J,K) and (I+1,J,K). For a Y-direction force, it is the one between cells (I,J,K) and (I,J+1,K), and for a Z-direction force, it is the one between cells (I,J,K) and (I,J,K+1).

THERMAL STRUCTURE SPECIFICATION RECORDS

This set of records - *to be completed with an END record* - must be included *only* when ISTRUC=1 in NAMELIST /GEOM/ (cf. ISTATE, NEWTS in NAMELIST /DATA/).

A thermal structure is a collection of thermal structure elements each of which has the same characteristics as specified by a thermal structure prototype. Thermal structure prototypes are defined using TYPE, FLUID, and MATERIAL namelists with the names /T/, /F/, and /M/ respectively. The order in which these namelists are input indicates the construction of the thermal structures and must conform to the following rules:

1. A TYPE namelist must commence the definition of each thermal structure prototype.

2. If fluid interacts with *surface 1*, a FLUID namelist must be present after the TYPE namelist, *before the first* MATERIAL namelist. If, in addition, fluid interacts with *surface 2*, a FLUID namelist must also be present *after the last* MATERIAL namelist.

3. A gap exists after each material except the last. The gap parameters are specified in the MATERIAL namelist.

4. The initial default for all namelist variables is zero. Subsequent defaults are the values in effect after reading the previous namelist. If, for example, the geometrical type is the same for all thermal structure prototypes, IXYZ need be specified only on the first TYPE namelist.

5. An END record must follow the last prototype definition.

6. Blank records or comment records may be interspersed as desired.

Notes:

1. The number of thermal structure prototypes is limited to 100.

2. Thermal structure heat sources Q invalidate uniform heat sources QSOU occurring in the same cell eventually.

3. See notes and warning in the THERMAL STRUCTURE LOCATION RECORDs section.

TYPE NAMELIST /T/

N		Thermal structure prototype number. This number does not need to correspond to its index or ordinal number.
IXYZ		Geometrical type or characteristic.
	1	Rods (cylinders) with axis aligned in the I-direction.
	2	Rods (cylinders) with axis aligned in the J-direction.
	3	Rods (cylinders) with axis aligned in the K-direction.
	11	Slab with the normal aligned in the I-direction.
	12	Slab with the normal aligned in the J-direction.
	13	Slab with the normal aligned in the K-direction.
	101	Spheres aligned in the I-direction
	102	Spheres aligned in the J-direction.
	103	Spheres aligned in the K-direction.

Note : The alignment specification is included in the spherical option to allow the normalized axial power distribution multiplier, QK, to be operative.

NT The number of the transient function to be used as a multiplier for the heat source.

RODFR **Rods or cylindrical thermal structures:**

>0 Number or fraction of actual rods interacting with each associated coolant cell.

<0 The absolute value is the number or fraction of rods per unit area interacting with each associated coolant cell, m^{-2}.
The rods are perpendicular to the cell area.

Slab thermal structures:

>0 Slab area in each associated coolant cell, m^2.

<0 The absolute value is the slab area divided by the cell area. In the case of two-sided thermal structures this value is equivalent to a solid permeability for the structure.

Spherical thermal structures:

>0 Number or fraction of spheres interacting with each associated coolant cell.

<0 The absolute value is the number or fraction of spheres per unit volume interacting with each associated coolant cell, m^{-3}.

OUTR Thermal structure outer radius, m.
This is *not used* for *slab type* thermal structures.

FLUID NAMELIST /F/

IHT Heat transfer correlation index. This value is used as the index, NH, of the variables HEATC1, HEATC2, HEATC3 and HEATC4 described in the Fluid-Structure Heat Transfer section of NAMELIST /DATA/.

HYD Hydraulic diameter or reference length. This value is used as D, the reference length, as described in the Fluid-Structure Heat Transfer Section of NAMELIST /DATA/.

MATERIAL NAMELIST /M/

MI Material type index. This value is used as the index NM described in the Material Properties (Solids) Section of NAMELIST /DATA/.

NP Number of partitions in the material. A thermal structure temperature will be computed for each material partition. NP, summed up for all materials of a single thermal structure prototype, must not exeed 50.

DR Partition size, m.

Q Volumetric heat source for the material region, Wm^{-3}.
Cf. page 43, note 4.

The following gap properties must be correctly specified or defaulted only when another material follows. If a fluid follows, the gap properties are ignored.

SGAP   Gap size, m.

HGAP   Gap heat transfer coefficient, $Wm^{-2}K^{-1}$.

THERMAL STRUCTURE LOCATION RECORDS

This set of records - *to be completed with an END record* - must be included *only* when ISTRUC = 1 in NAMELIST /GEOM/ (cf. ISTATE, NEWTS in NAMELIST /DATA/).

Once the thermal structure prototypes have been defined the location of the thermal structure elements are specified by the THERMAL STRUCTURE LOCATION RECORDs. These records contain the following variables in FORMAT (A4,7I4)

LOC NUM IB IE JB JE KB KE

LOC	*OUT*	The cells specified interact with the outside or surface 1.
	IN	The cells specified interact with the inside or surface 2.
	END	A record containing 'END ' in columns 1-4 is needed to terminate the THERMAL STRUCTURE LOCATION RECORDs.
NUM		Thermal structure prototype number.
IB, IE, **JB, JE,** **KB, KE**		These six variables are the beginning and ending I-, J-, and K-indices that define a rectangular (cylindrical) solid composed of one or more cells which are to interact with thermal structure NUM.

Notes:

1. A cell should not be specified twice by the indices unless the true intention is to have two occurrences of the thermal structure prototype NUM.

2. Many THERMAL STRUCTURE LOCATION RECORDs may be needed to define all the cells interacting with a given thermal structure prototype.

3. The order in which cells are specified is arbitrary except when the thermal structure prototype has fluid cells interacting with both surfaces. In this case cells are paired off in the order in which they are specified, especially the number of cells interacting with one surface must equal the number of cells interacting with the other surface.

4. For a thermal structure (prototype number NUM) having some material with volumetric heat source Q specified (different from 0.0), all specified cells are forced to have zero values of QSOU, cf. page 47, note 2.

WARNING!

While all thermal structure variables can be redefined upon restart, changes in the order in which the thermal structures are defined, changes in NP, or changes in the order of or the values on the THERMAL STRUCTURE LOCATION RECORDs will scramble the internally stored thermal structure temperatures.

BOUNDARY VALUE INITIALIZATION RECORDS

The purpose of this set of input records is to permit initialization of boundary values of any of the magnitudes listed below. Each record in this section contains the following variables in the FORMAT (A4,E10.3,7I4).

KEY	RVAL	IB	IE	JB	JE	KB	KE	N

KEY   *HLB*   Enthalpy, Jkg^{-1}

         PB   Pressure, **Pa**, *should not be specified, cf.note 2 below.*

         QBN   Heat flux, normal to the surface in the direction indicated by XNORML(N), YNORML(N), and ZNORML(N), Wm^{-2}.

         RLB   Density, kgm^{-3}.

         TLB   Temperature, °C.

         VELB   Velocity, normal to the surface in the direction indicated by XNORML(N), YNORML(N), and ZNORML(N).

                The user may enter/modify the values of surface element areas and adjacent cell volume fractions :

         AREA   Surface element area, m^2.

         THLB   Adjacent cell volume fraction.

         END   Terminating the BOUNDARY VALUE INITIALIZATION RECORDs. *This record must always be included.*

RVAL   The value to be assigned to the variable named.

IB, IE,
JB, JE,
KB, KE   These six variables are the beginning and ending I-, J-, and K-indices that define a rectangular solid composed of one or more cells. The rectangular solid that defines or partially defines a surface is the one which is totally interior and adjacent to, or partially interior to and intersecting that surface.
Cf. BOUNDARY SURFACE IDENTIFICATION RECORDs.

N   The surface number of the boundary being set.

Notes:

1. Initial (ISTATE=0) uniform velocity, temperature/heat flux and pressure boundary conditions and boundary volume fractions can be more easily specified using the variables VELOC, TEMP, PRES, THETA in NAMELIST /DATA/, cf. note 2.
2. With ISTATE=0, values VELB, TLB/QBN, PB, THLB and AREA from BOUNDARY VALUE INITIALIZATION RECORDs take precedence over *Boundary and Cell Initialization* values VELOC, TEMP, PRES, THETA from NAMELIST /DATA/, and over area values from BOUNDARY SURFACE IDENTIFICATON RECORDs.
3. The END - record must be included even if actually there is no boundary value initialization.

INTERNAL CELL INITIALIZATION RECORDS

The purpose of this set of input records is to permit initialization of internal cell values of any of the arrays listed below. Each record of this section contains the following variables in the FORMAT (A4,E10.3,6I4):

KEY RVAL IB IE JB JE KB KE

KEY		
	AL	Volume porosity, the dimensionless ratio of fluid volume in a cell to total cell volume. (1.0)
	ALX	Surface permeability, the dimensionless ratio of free flow area to the total surface element area, between cell (I,J,K) and cell (I+1,J,K). (1.0)
	ALY	Surface permeability, the dimensionless ratio of free flow area to the total surface element area, between cell (I,J,K) and cell (I,J+1,K). (1.0)
	ALZ	Surface permeability, the dimensionless ratio of free flow area to the total surface element area, between cell (I,J,K) and cell (I,J,K+1). (1.0)
	HL	Enthalpy, Jkg^{-1}. (0.0)
	QSOU	Volumetric heat source per computational cell volume $DX(I) \times DY(J) \times DZ(K)$, Wm^{-3}. (0.0)
	TL	Temperature, °C. (0.0)
	UL	U-component of velocity at the surface element between cell (I,J,K) and cell (I+1,J,K), ms^{-1}. (0.0)
	VL	V-component of velocity at the surface element between cell (I,J,K) and cell (I,J+1,K), ms^{-1}. (0.0)
	WL	W-component of velocity at the surface element between cell (I,J,K) and cell (I,J,K+1), ms^{-1}. (0.0)
	END	Terminating the INTERNAL CELL INITIALIZATION RECORDs. *This record must always be included.*

RVAL   The value to be assigned to the variable named.

IB, IE,   These six variables are the beginning and ending I-, J-, and K-indices
JB, JE,   that define a rectangular solid composed of one or more cells to become
KB, KE   initialized (cf. Note 1.).

Notes:

1. The values of ALX, ALY, ALZ resp. UL, VL, WL apply to the cell faces of the rectangular solid which are *not* elements of boundary surfaces. Surfaces lying on boundaries must be initialized using

   - BOUNDARY SURFACE IDENTIFICATION RECORDs
     as to the definition of non-standard surface areas
   - BOUNDARY VALUE INITIALIZATION RECORDs
     as to the definition of normal velocities.

2. The values of QSOU are forced to be zero by the code for cells where thermal structures act as heat sources, cf. page 43 note 4.

3. The END - record must be included even if actually there is no internal cell initialization.

GENERAL 3-DIMENSIONAL REBALANCING OPTION

This set of records must be included only when IFREB > 0 , cf. NAMELIST /GEOM/.
(Cf. ISTATE, NEWREB (NAMELIST /DATA/).

NAMELIST /REBAL/

This namelist input is required if IFREB > 0.

NUSREB	Number of rebalancing regions specified by the user. (0) Specification is done through REBALANCING REGION RECORDs (see below).
NXSEGM	Number of groups, into which the X-grid is segmented. (1) This involves only cells not allocated by REBALANCING REGION RECORDs.
NYSEGM	same for the Y-grid.
NZSEGM	same for the Z-grid.

LXSEGM (I) I = 1,...,(NXSEGM-1)
Numbers of increments for segmentation of the X-grid. Must be positive. The missing value (for I = NXSEGM) is not input but evaluated from the sum (up to NXSEGM-1) and the required total IMAX.

LYSEGM(I) I = 1,...,(NYSEGM-1)
corresponding input array for the Y-direction.

LZSEGM(I) I = 1,...,(NZSEGM-1)
corresponding input array for the Z-direction.

REBALANCING REGION RECORDS

This set of records is required if NUSREB > 0. Each record contains the following variables and specifications in the FORMAT (A4,7I4).

KEY	NUM	IB	IE	JB	JE	KB	KE

KEY	*UREB*	marks records with valid geometry specifications.
	END	marks end of specifications.
	blank	used to insert comments (starting in column 5).
NUM		(sequential) index of user-specified rebalancing region (NUM = 1,...,NUSREB)
IB, IE, **JB, JE,** **KB, KE**		specify a box of cells which is part of rebalancing region NUM. Standard conventions are followed here.

Note:

This input is checked for completeness (cf. NUSREB) and sequencing of region index NUM. The code checks that the regions specified for general, 3-dimensional rebalancing (in detail, through rebalancing region records and more globally, through NXSEGM, LXSEGM etc.) are, in fact, assemblies of cells, which are geometrically and physically connected.

VISUALIZATION OPTION

A sequential file (binary data, 4 byte integer, 4 byte real) is written to unit 21 serving as an interface between the FLUTAN code and Visualization (or other Postprocessing) Programs. The format of the data set, *VISART* for *visualization standard*, is specified in Ref./15/.

NAMELIST /XVIS/

This namelist input is required only if VISART output is wanted; however it may be added without creating VISART output, cf. IOUTVI = *0 or 2*.

IOUTVI	*0*	Neither VISART nor FLUTAN postprocessor output is written.
	1	VISART postprocessor output is written on unit 21.
	2	FLUTAN postprocessor output is written on unit 20. (*)
	3	VISART and FLUTAN postprocessor output is written.

PQUANT Any name under the heading *VISART name* from both tables below.
The quantity specified by PQUANT is to be written to the VISART file
  - at problem time steps specified by NTPLOT,
    (cf. *Plotting Option*, page 23).
  - for *all system points*, cf. TQUANT below.

The next *four items* are associated as *quadruples* having the meaning

TQUANT Any name under the heading *VISART name* from *both* tables below.
The quantity specified by TQUANT is to be written to the VISART file
  - at all problem time steps,
  - for the system point specified by the following coordinate indices.

ITQUA Coordinate index I (X/R-direction) associated with TQUANT.

JTQUA Coordinate index J (Y/Θ-direction) associated with TQUANT.

KTQUA Coordinate index K (Z - direction) associated with TQUANT.

or being an *orientation quadruple*

    ORIENT indicating that the associated triple I/J/KTQUA is specifying one cell face
        (of six) by its inward directed normal vector (its *orientation*)
    0
    0 (bottom face for example)
    1

The single quadruple *immediately following* an orientation quadruple again comprises:

TQUANT Any name under the heading *VISART name* from *only the second* table
below (specifying a *cell face centered quantity*).
The quantity specified by TQUANT is to be written to the VISART file
  - at all problem time steps,
  - for the system point specified by the following coordinate indices:

I/J/KTQUA Three coordinate indices of the associated TQUANT specifying a cell and
-together with the *orientation* specified before- the face (bottom face for
example) of this cell comprising the *cell face centered point* associated
with TQUANT. The cell face must be an element of a boundary surface,
as is checked by the code.

Table 1: Internal Cell Scalar/Vector Quantities
(VISART Groups No.17 or 21, Dimension 0/3)

These quantities represent a subset of *cell centered* quantities in FLUTAN

VISART-name	System quantity	FLUTAN name
SITL	Temperature	TL
SIP	Pressure	P
SIHL	Enthalpy	HL
SIRL	Density	RL
SITURK	Turbulent Kinetic Energy	TURK
SITKED	Turbulent Kinetic Energy Dissipation	TKED
SITURVIS	Turbulent Viscosity	TURVIS
SITURCON	Turbulent Conductivity	TURCON

The components of the Velocity Vector represent a set of *cell centered* values being the arithmetic mean each of a pair of values of *cell face centered* quantities in FLUTAN :

SIVEL	Velocity Vector	UL, VL, WL

Table2: Surface Element Scalar/Vector Quantities
(VISART Groups No.17 or 21, Dimension 0/3)

These quantities represent a subset of quantities *centered* at *surface elements* in FLUTAN

VISART-name	System quantity	FLUTAN name
SSTL	Temperature	TLB
SSP	Pressure	PB
SSHL	Enthalpy	HLB
SSRL	Density	RLB
SSTURK	Turbulent Kinetic Energy	TURKB
SSTKED	Turbulent Kinetic Energy Dissipation	TKEDB

The *surface element centered* Velocity Vector is the product of the vector normal to the surf. element with the single *surface element centered* quantity VELBN in FLUTAN :

SSVEL	Velocity Vector	VELBN

Note: A surface element is a cell face coplanar with a Boundary Surface.

Note:
To transfer the space- and time dependent data in the VISART file to existing postprocessing programs they must be transformed into the required format.
A tool for this task is the program VISAPTER, specially providing an interface for the Visualization System AVS, utilizing Field Data and Unstructured Cell Data formats.
Cf. Ref./16/ and Ref./17/.

MACHINE DEPENDENT ROUTINES

Only one machine dependent function is required in FLUTAN, **TREMAN.**

This function returns the CPU time left for the current run in units of seconds. This time starts at the TIME value specified on the JOB card and ends at zero when the job is terminated by the system. It is used for timing and deciding when to terminate and write a restart file.
Cf. pages 6, 14.

The routine is available in ASSEMBLER for IBM computers under MVS, and in FORTRAN for the FZK-VP400-EX using system routine RTIME; for the FZK-CRAY-J916 under UNIX the system routine TREMAIN is to be referenced.
For HP- and IBM systems under UNIX the "remaining time" method of JOB termination is not applicable (i.e. the RTIME reference should be removed from TREMAN). Instead the user may utilize NTMAX or TIMAX while providing for sufficient CPU time and ensuring that TREST < TLEFT (local variable in TREMAN) e.g. by setting "TLEFT = 1.e30".

Another useful (but not required) IBM-ASSEMBLER routine CALSEQ is referenced only once in the error handling routine TRACBK. On IBM-computers and compatibles (e.g. the VP- and S-series of Fujitsu/Siemens) it is used to obtain the calling sequence of routines to the location of the detected error. On UNIX systems the CALSEQ reference should be removed from TRACBK, which thus is only to return an error code.

STORAGE ALLOCATION

Space for a few of the geometry dependent variables is allocated dynamically in the array S (INTS), which is defined in the header routine FLUTAN (Parameter NAVAIL). The address of such a variable is computed at the beginning of each run. These addresses are then passed into called subroutines where the variables have individual names and are adequately dimensioned.

Generally, data are held in named COMMON blocks, dimensions of arrays given by the PARAMETER constants LCELL, LBOUND, IJKMAX. These parameters are held in the "Common Deck" PARAM, which is adequately handled by the source management code HISTORIAN (corresponding to CDC UPDATE and similar products).

An alternative to the use of HISTORIAN would be setting up the HISTORIAN decks as members of an IBM-Partitioned Data Set (PDS) and applying (instead of the HISTORIAN CALL-function) the FORTRAN INCLUDE-directive.

STEADY-STATE DEFINITION

Steady-state is reached when the following conditions are met:

1. DL < 1.0 where DL=(maximum cell residue)/DCONV,
   DCONV=EPS1 × (UVWMAX + EPS2), and UVWMAX is computed in
   SUBROUTINE GDCONV

2. The change of the U -velocity component divided by the maximum velocity magnitude in the entire field is less than EPS3.

3. The change of the V -velocity component divided by the maximum velocity magnitude in the entire field is less than EPS3.

4. The change of the W-velocity component divided by the maximum velocity magnitude in the entire field is less than EPS3.

5. Maximum (DH/H) < EPS3 where H is the current enthalpy and DH is the change in enthalpy over two consecutive time steps.

ERROR HANDLING

Much effort has been invested to instrument all routines with self-explaining error messages, giving name of routine, error type and information usable to locate the error in source text. After any error is detected, the routine TRACBK is called with a non-zero argument to terminate the job.

SPECIAL TOOLS

Preparing input for FLUTAN may become a cumbersome procedure, especially for large systems. In order to improve upon this circumstance a couple of generators for geometrical input have been developed:

- FUNGEO (for cartesian and cylindrical coordinates, cf. Ref./11/)
- HEXPIN (for Rod Bundle Geometry, cf. Ref./12/)

Physical quantities evaluated with FLUTAN may get represented through the generation of isoline or vector plots on coordinate planes by the FORTRAN postprocessing program

- FLUPLO (for cartesian and cylindrical coordinates, cf. Ref./13/)

FLUTAN NAMELIST SUMMARY

Namelist-groups of FLUTAN code (alphabetical order by columns):

(1) Namelist **&GEOM** :

DX	IFRES	ISTRUC	JPRES0	NL1	NH1
DY	IGEOM	ISYMCH	KMAX	NM1	XNORML
DZ	IMAX	ITURKE	KPRES0	NSURF	YNORML
IFREB	IPRES0	JMAX	NFORCE	NHEX	ZNORML

(2) Namelist **&DATA** :

ACORRL	C2MU	FORCEF	IREBIT	KTEMP	NTOTS	THETA
ACORRT	C2K	FVAL	ISETEN	LASTDT	NTPLOT	TIMAX
ALPHA	C2RO	GRAVX	ISETMO	LASTIT	NTPRNT	TPRNT
BCORRL	DTENER	GRAVY	ISTATE	LCRES	NUMDIF	TREST
BCORRT	DTFUEL	GRAVZ	ISTPR	LMPRNT	OMEGA	TSINK
CCORRL	DTWALL	HEATC1	IT	MATWAL	OMEGAE	TSTART
CCORRT	EPS1	HEATC2	ITENMX	MODEL	OMEGAM	TVAL
CLENTH	EPS2	HEATC3	ITMASX	NCORR	OMEGAR	VELOC
C0CP	EPS3	HEATC4	ITMAXE	NEND	OMEGAV	VSLIPX
C0KO	EPS4	HSINK	ITMAXP	NEWFOR	PRES	VSLIPY
C0MU	EPS5	HYDWAL	ITMOMX	NEWREB	PRES0	VSLIPZ
C0RO	FACFRD	ICORR	IXREB	NEWTS	QK	WALLDX
C1CP	FACFRE	IDRODT	IYREB	NHEATC	RDTIME	WALLQS
C1KO	FACFRK	IDTIME	IZREB	NMATER	RELAXE	
C1MU	FACFRM	IFENER	I2PMUL	NOFQT	REYLEN	
C1RO	FCTHI	IFPLOT	KFLOW	NTHPR	TEMP	
C2CP	FCTLO	IHTWAL	KPRES	NTMAX	TEMP0	

(3) Namelist **&TURB** :

AKAPPA	BETAN	CT3	IFITUP	OMEGAK	PRNDLK	TKIN
ALFAG	CDTURB	EE	ITMAXK	OMEGAT	RELAXK	TKBD
ALFAN	CT1	EPS6	ITURMX	PRNDLD	TDIN	TURBC
BETAG	CT2	HYDIN	OMEGAD	PRNDLH	TDBD	TURBV

(4) Namelist **&HEXD** (For heat exchangers, cf. Ref./14/):

HEA	HEQW	HEU	HEW	IPHEX	MCHE	NFHE
HEM	HETB	HEUQ	IHTHE	KHEX	MWHE	NPHE

(5) For thermal structures:

(5a) Namelist **&T** :

IXYZ   N   NT   OUTR   RODFR

(5b) Namelist **&F**:

HYD   IHT

(5c) Namelist **&M** :

DR   HGAP   MI   NP   Q   SGAP

(6) Namelist **&REBAL** :

LXSEGM   LYSEGM   LZSEGM   NUSREB   NXSEGM   NYSEGM   NZSEGM

(7) Namelist **&XVIS** :

IOUTVI   PQUANT   TQUANT   ITQUA   JTQUA   KTQUA

INPUT EXAMPLE

```
         +------------------------------------------------------+          Comm. Records
         +     SAMPLE INPUT FOR STEADY-STATE CALCULATION    +
         +------------------------------------------------------+
&GEOM                                                                      Namelist GEOM
   IGEOM=0, IMAX=52, JMAX=10, KMAX=29,
   NM1=2773, NL1=2084, NSURF=19,
   ISTRUC=1, NFORCE=7, ITURKE=12,
                                                                          Comm. Record
   DX=.035, 2*.03, .025, .02, .01, .03, .04, .08, ...
   DY= 2*.02, .0263, .0237, .06, .10, .15, 2*.20, .30,
   DZ=.20, .30, 2*.40, .30, .20, .35, ...
                                                                          Comm. Record
   XNORML=  3*-0.707107, 0.8660, -0.8660, 1.0, -1.0, ...
   YNORML=  0.0, -0.707107, 3*0.0, 2*0.0, 1.0, -1.0, ...
   ZNORML=  -0.707107, 0.0, 0.707107, -0.5, 0.5, ...
                                                                          Comm. Record
   IPRES0=1, JPRES0=1, KPRES0=1,
                                                                          Comm. Record
   &END
IREG 7.0711E-4    7    7    1    2   24   24    1   NOZZLE TOP             Boundary
IREG 5.5791E-4    7    7    3    3   23   23    1                         Surface
IREG 1.0055E-3    7    7    4    4   22   22    1                       Identification
IREG 1.2728E-3    7    7    5    5   21   21    2   NOZZLE SIDE            Records
             .
             .
             .
REG              1    1    1   10    1   29    6  +X DC C.B. SIDE
REG              6    6    1   10    1   15    7  -X DC P.V. SIDE
REG              6    6    1   10   25   29    7
             .
             .
REG 2.08400E-4   35   35    1    1   25   25   19  -Z INJ INLET
REG              36   37    1    1   25   25   19
REG 3.25230E-4   38   38    1    1   25   25   19
END    BOUNDARY SURFACE IDENTIFICATION
                                                                          Comm. Record
 &DATA                                                                    Namelist DATA
   ISTATE=0, IFENER=0,
   IDTIME=0, DT=1.0,   IXREB= 1, IREBIT= 10,
                                                                          Comm. Record
   KFLOW= 7*1, 2*-3, 1, -5, 3*1, -3, 2*1, -3, 1,
   KTEMP=5*400, 1, 2*400, 2*1, 5*400, 1, 2*400, 1,
   VELOC= 18*0.0,  0.010,
   TEMP =19*300.0,
   PRES0=110.0E5, TEMP0=300.0,  GRAVZ=-9.8,
                                                                          Comm. Record
     FORCEF=6*0.5, 5.0E4,
     REYLEN=0.05, 0.19, 2*0.3, 2*1.0, 0.3,
     CLENTH=0.05, 0.19, 2*0.3, 3*      ,
        ICORR=1,1,2,2,3,3,1,
        NCORR=3,
     ACORRL=  64.0,  96.0,   0.0,      ACORRT= ...,
     BCORRL=  -1.0,  -1.0,  -1.0,      BCORRT= ...,
     CCORRL=   0.0,   0.0,   1.0,      CCORRT= ...,
                                                                          Comm. Record
NHEATC=1,
     HEATC1=3.66, HEATC2=0.0240, HEATC3=0.8, HEATC4=0.4,
          NMATER=2,
```

```
      COK =    38.0,    50.0,    C1K = ...,
      COCP=   453.0,   440.0,    C1CP= ...,
      CORO= 7856.0,  7850.0,
    NTPRNT= 1,-9999,
    NTHPR= 1201, 3201, 5201, 8201,
    &END


  &TURB
    HYDIN=0.19, TDIN=12.355, ITMAXK= 299,
    &END

ZFOR    1  33  36   1   1  23  23    INJ        )
XFOR    2   8  51   1   2  17  23    CL         )   FRICTION
  .         .
  .         .
END    FORCE STRUCTURE LOCATION

       THERMAL STRUCTURE PROTOTYPE RECORDs (PLUS END)
  &T   N= 1, IXYZ=3, RODFR=0.01421, OUTR=0.224, &END
  &F         IHT=1, HYD=0.3,                     &END
  &M         MI=1, NP=2, DR=0.112, Q=0.0,   &END
  &T   N= 2, IXYZ=3, RODFR=0.01421, OUTR=0.224, &END
  &F         IHT=1, HYD=0.3,                     &END
  &M         MI=1, NP=2, DR=0.112, Q=0.0,   &END
  .         .         .
  .         .         .
  .         .         .
END    THERMAL STRUCTURE PROTOTYPE

OUT    1   6   6   1   1   2  29     DC PRESS. VESS. SIDE.
OUT    2   6   6   2   2   2  29
OUT    3   6   6   3   3   2  29
OUT    4   6   6   4   4   2  29
  .         .         .
  .         .         .
OUT   19   7   7   1   1  16  16
END    THERMAL STRUCTURE LOCATION

       THE FOLLOWING END-RECORD IS MANDATORY
END    BOUNDARY VALUE INITIALIZATION

AL     0.41670    7   7   1   2  24  24    IRREG. CELLS
AL     0.75000    7   7   3   3  23  23    JUNCTION BETW.
AL     0.50000    7   7   4   4  22  22    CL AND DC
  .       .           .
  .       .           .
  .       .           .
ALZ    0.9622    38  38   1   1  25  25
ALX    0.0        1   5   1  10   1   1    DC OUTLET
ALY    0.0        1   6   1  10   1   1
END    INTERNAL CELL INITIALIZATION
```

Right-margin annotations:

Namelist DATA
continued

END DATA
Comm. Record
Namelist TURB

END TURB
Comm. Record
Force Locat.
Records
(END incl.)
(Cf. Fig.2)
Optional END
Comm. Records

Namelist T
Namelist F
Namelist M

Namelist M

Optional END
Comm. Record
Ther. Struc
Location
Records
(END incl.)
(Cf. Fig.2,3)

Optional END
Comm. Records

Mandatory END

Internal Cell
Initialization
Records

Mandatory END

```
    +----------------------------------------------+          Comm. Records
    +    SAMPLE INPUT FOR TRANSIENT CALCULATION    +
    +----------------------------------------------+

&GEOM                                                          Mandatory
 IFRES=3, ( or: IFRES=2, )                                     Mandatory
 &END                                                          Mandatory
                                                               Comm. Record
&DATA                                                          Mandatory
 ISTATE=2,                                                     See note
           IFENER=1, TSTART=0.0,                               Optional
 IDTIME=0, DT=0.020, 0.050,   LASTDT=5,
 NTMAX=20000, IT=99,
 KFLOW(19)=101, VELOC(19)=0.4348,
 KTEMP(19)=102,
 NEND=4,4,
    TVAL=       0.0,     0.10,     0.10,   600.00,
                0.0,     0.10,     0.10,   600.00,
    FVAL=      0.23,     1.00,     1.00,     1.00,
               1.0,   0.06667,   0.06667,  0.06667,
 NTPRNT= -9999, 1
 NTHPR=1201, 3201, 5201, 8201, 17201,
 &END                                                          Mandatory
                                                               Comm. Record
&TURB                                                          Mandatory
 HYDIN=0.19, TDIN=12.355, ITMAXK= 299,                         Optional
 &END                                                          Mandatory
                                                               Comm. Record
END                                                            Mandatory
END                                                            Mandatory
```

Note:

On *restart*, ISTATE must be set at the

- first continuation of a steady state calculation
- commencement of a transient calculation
- first continuation of a transient calculation
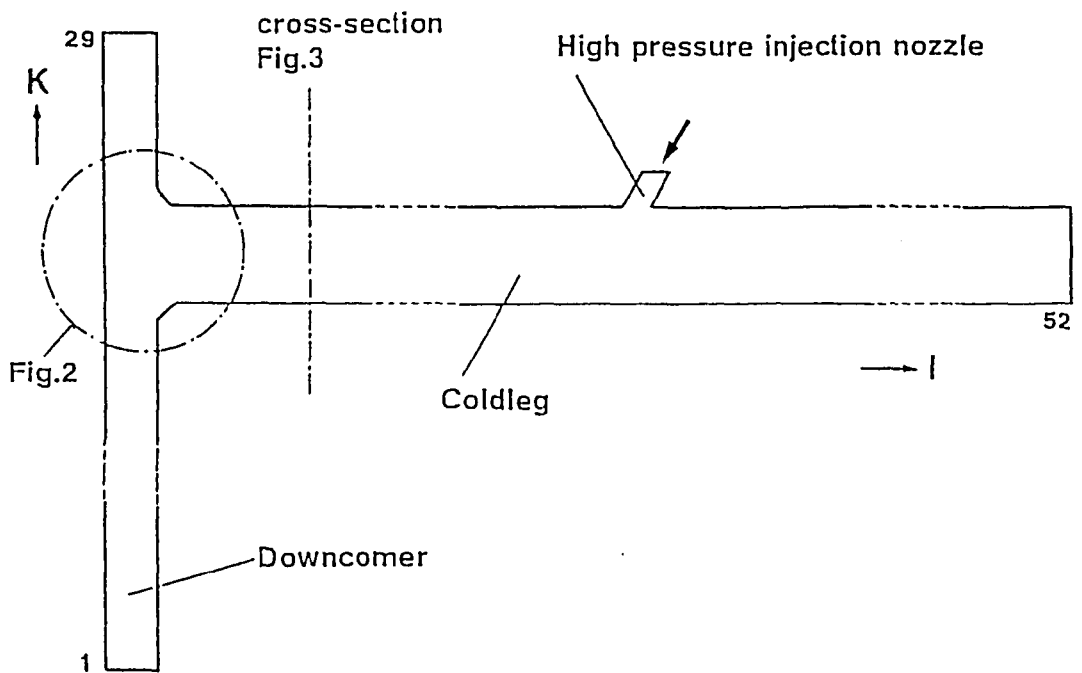
Figure 1. Input Sample Model. Overall View.
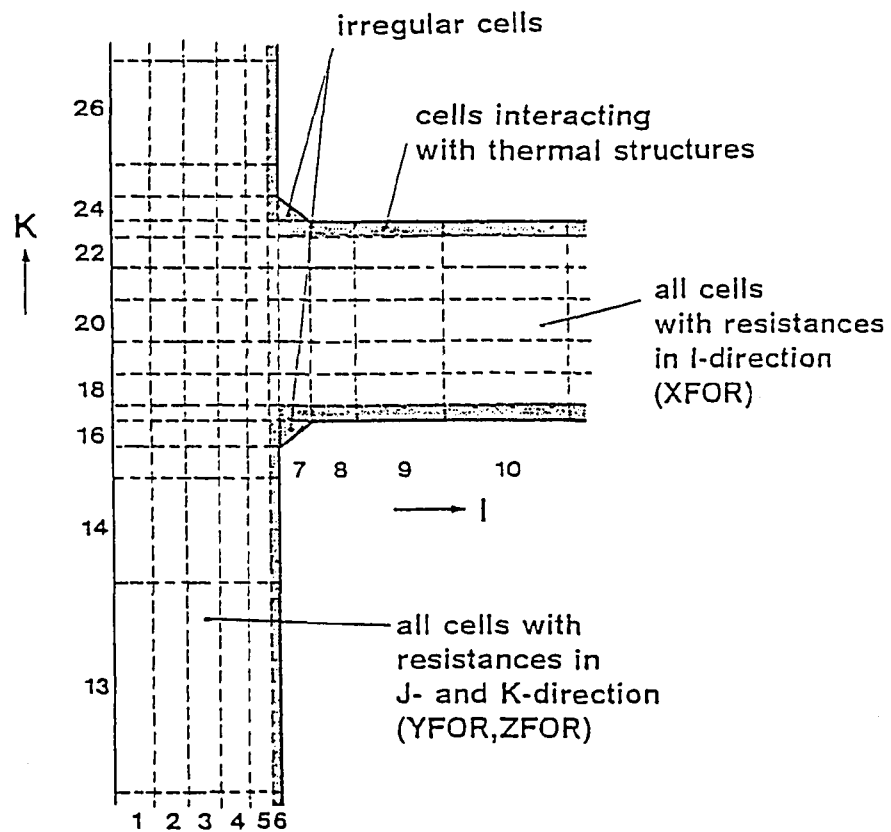
Figure 2. Input Sample Detail. Junction of Cold Leg and Downcomer.
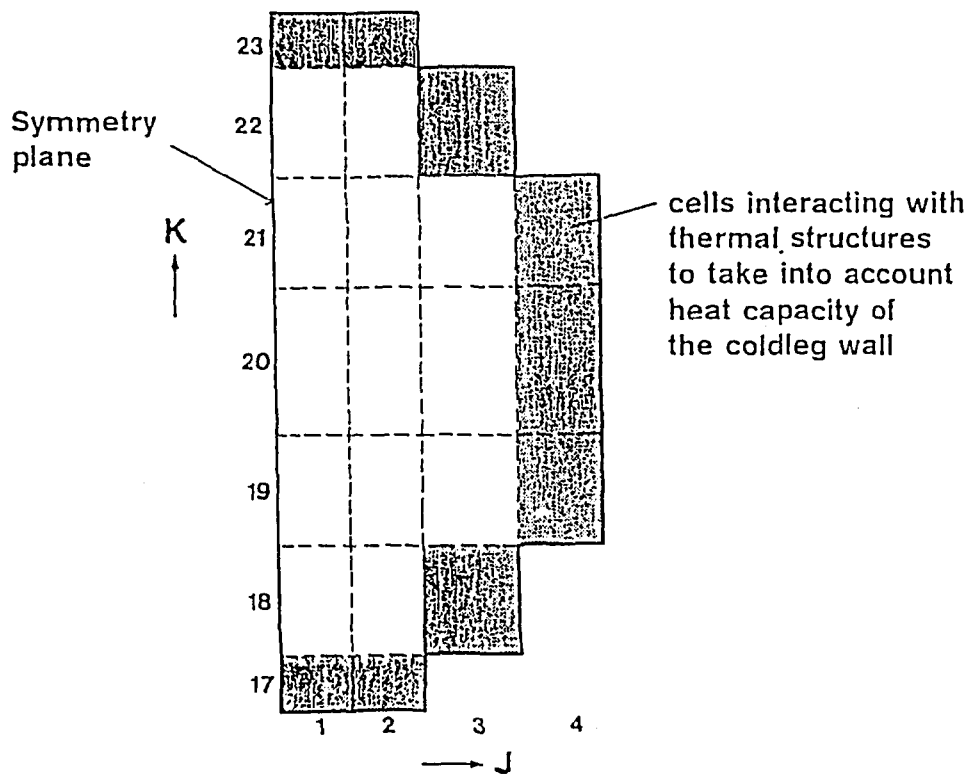
Figure 3. Input Sample Detail. Section across the Cold Leg.

REFERENCES

1. H. M. Domanus, W. T. Sha, R. C. Schmitt, V. L. Shah, F. F. Chen, C. C. Miao
   "COMMIX-1B: A Three-Dimensional Transient Single-Phase Computer Program for Thermal Hydraulic Analysis of Single and Multicomponent Systems,
   Volume I: Equations and Numerics",
   NUREG/CR-4348 Vol. I, ANL-85-42 Vol. I, September 1985

2. H. M. Domanus, W. T. Sha, R. C. Schmitt, V. L. Shah, F. F. Chen, C. C. Miao
   "COMMIX-1B: A Three-Dimensional Transient Single-Phase Computer Program for Thermal Hydraulic Analysis of Single and Multicomponent Systems,
   Volume II: Users Manual",
   NUREG/CR-4348 Vol. II, ANL-85-42 Vol. II, September 1985

3. M. Bottoni, W. L. Baumann, H. N. Chi, T. H. Chien, H. M. Domanus, R. C. Schmitt, W. T. Sha, V. L. Shah
   "COMMIX-2: A Three-Dimensional Transient Computer Program for Thermal-Hydraulic Analysis of Two-Phase Flows",
   NUREG/CR-4371, ANL-85/47, September 1985

4. H. M. Domanus, Y. S. Cha, T. H. Chien, R. C. Schmitt, W. T. Sha
   "COMMIX-1C: A Three-Dimensional Transient Single-Phase Computer Program for Thermal Hydraulic Analysis of Single and Multicomponent Engineering Systems", Volume I: Equations and Numerics", NUREG/CR-5649 Vol. I, ANL-90/33 Vol. I, November 1990

5. H. M. Domanus, Y. S. Cha, T. H. Chien, R. C. Schmitt, W. T. Sha
   "COMMIX-1C: A Three-Dimensional Transient Single-Phase Computer Program for Thermal Hydraulic Analysis of Single and Multicomponent Engineering Systems, Volume II: Users Guide and Manual",
   NUREG/CR-5649 Vol. II, ANL-90/33 Vol. II, November 1990

6. H. Borgwaldt
   "CRESOR, a Robust Vectorized Poisson-Solver Implemented in the COMMIX-2(V) Thermal-Hydraulics Code",
   Paper AC064, Int. Conf. on Supercomputing in Nuclear Applications (SNA '90), Mito City, Japan, March 1990

7. Cl. Günther
   "A Consistent Upwind Method of Second Order for the Convection-Diffusion Equation",
   Proceedings of the 1987 International Conference on Computational Techniques and Applications (CTAC-87), p. 249   Sydney, Australia, August 1987

8. Cl. Günther
   "Vergleich verschiedener Differenzenverfahren zur numerischen Lösung der 2-d-Konvektions-Diffusionsgleichung anhand eines Beispiels mit bekannter exakter Lösung",
   KfK 4439, Kernforschungszentrum Karlsruhe, August 1988

9. K. Sakai, D. Weinberg
   "Investigation on the LECUSSO (Local Exact Consistent Upwind Scheme of Second Order) Scheme - Extension to Nonuniform Mesh Size Grids and Numerical Stability Analysis - and Application for Natural Circulation Test Analyses using COMMIX-2(V)",
   Paper AS081, Int. Conf. on Supercomputing in Nuclear Applications (SNA '90), Mito City, Japan March 1990

10. M. Bottoni, G. Willerding, W. Baumann
    unpublished report, Kernforschungszentrum Karlsruhe (1986)

11. G. Willerding, W. Baumann
    unpublished report, Kernforschungszentrum Karlsruhe (1992)

12. G. Willerding, W. Baumann
    unpublished report, Kernforschungszentrum Karlsruhe (1990)

13. G. Willerding
    unpublished report, Kernforschungszentrum Karlsruhe (1990)

14. A. Class
    unpublished report, Kernforschungszentrum Karlsruhe (1990)

15. S. Kleinheins
    unpublished report, Kernforschungszentrum Karlsruhe (1991)

16. S. Kleinheins
    unpublished report, Kernforschungszentrum Karlsruhe (1994)

17. S. Kleinheins
    unpublished report, Kernforschungszentrum Karlsruhe (1994)

18. H. Borgwaldt, W. Baumann, G. Willerding
    "FLUTAN Input Specifications",
    KfK 5010, Kernforschungszentrum Karlsruhe, Mai 1992

19. Y. Kimhi, G. Grötzbach
    unpublished report, Kernforschungszentrum Karlsruhe (1994)