CFDYNAMICS

Coupled **F**low **DYNAMICS** Program

User's Manual

Atul Ninu Waghode

Advanced Separation Technologies Group,
Department of Chemical Engineering,
Loughborough University,
Loughborough, Leicestershire,
LE11 3TU, United Kingdom

OVERVIEW – ABOUT CFDYNAMICS

Coupled Flow DYNAMICS (CFDYNAMICS) is a Finite Element Based (FEM) FORTRAN computer code designed to simulate the hydrodynamics of coupled free and porous flow regimes. The numerical procedure has been developed with the aim of predicting fluid dynamical behaviour of combined free and porous flow regimes of high performance pleated cartridge aeronautical filters. It allows the user to enter the physical and rheological properties of fluid, permeation characteristics of porous medium along filtration operating conditions such as inlet volumetric flow rates in order to evaluate fluid trajectories and pressure field distribution.

TECHNICAL SPECIFICATIONS

1. **CFDYNAMICS** is written in FORTRAN 90 on Windows platform.

2. **CFDYNAMICS** is a 2 - dimensional combined Free/Porous flow simulation package. The solution scheme for **CFDYNAMICS** is based on the **U-V-P** finite element method employing **Taylor-Hood** elements with unequal order interpolation functions for velocity and pressure.

3. **CFDYNAMICS** solves the **DARCY** equation for the low permeability porous regime and **STOKES** momentum balance equations for creeping laminar regime along with the mass **CONTINUITY** equation for incompressible flow problems

4. **CFDYNAMICS** solves the equations in both Cartesian as well as axisymmetric coordinate systems.

5. **CFDYNAMICS** can program the solution of non-Newtonian and non-isothermal incompressible transient and steady flow problems.

6. Algebraic equations in **CFDYNAMICS** are solved by a frontal method.

7. Post-processing files generated by **CFDYNAMICS** can directly be exported to **TECPLOT** and **COSMOS** for purposes of visualisation.

8. **CFDYNAMICS** has 17 subroutines that are directly accessible to the user and can be easily modified depending on the modelling requirements.

SYSTEM REQUIREMENTS & INSTALLATION

---

1. System Requirements (Essential)

a. The recommended hardware for the system consists of a **Desktop PC** with a **Pentium** type processor having at least **32MB RAM**. However, we would specifically recommend a **Pentium - 4 (3.2 GHz)** processor with **512 MB RAM** for faster computations.

b. **Microsoft Windows 95, 98, ME, NT, XP or 2000** operating system.

c. **Compaq Visual Fortran 6.6** for **CFDYNAMICS** compilation and execution.

2. System Requirements (Auxiliary)

a. Installed **COSMOS/M 2.6** for mesh generation and visualisation

b. Installed **TECHPLOT 8.0** for visualisation.

c. Installed **GOLDEN SOFTWARE SURFER** for visualisation.

3. Installation Procedure

a. Make a sub-directory for **CFDYNAMICS** in the main **WINDOWS** directory and position the **StokesDarcyUVP.for** program with accompanying **Input data files**.

b. Right click on the **CFDYNAMICS** program and open it with **COMPAQ VISUAL FORTRAN 6.6** compiler**.**

c. Go to the **COMPAQ VISUAL FORTRAN 6.6** main menu**,** click on **Build command Compile CFDYNAMICS Execute CFDYNAMICS**

GETTING STARTED

Working Step 1

Prepare finite element mesh of the pleated cartridge filter geometry in the pre-processor of **Geostar** using 4-noded as well as 9-noded elements. Use the auxiliary finite element utility **FORTRAN** program feut.for to convert the GFM files generated by **Geostar** containing geometrical details of the finite element mesh into a format readable by the program *StokesDarcy UVP.for* of **CFDYNAMICS**.

Working Step 2

For the execution of **CFDYNAMICS** two data files are the requisites,

- Default data file *df.dat* containing fluid and porous material properties, nodal coordinates and elemental connectivity generated from feut.for and boundary constraints;
- Input data file *CORD-4.dat* containing nodal coordinates and elemental connectivity of mesh constructed using 4-noded elements

Once the program *StokesDarcyUVP.for* of **CFDYNAMICS** is executed, it prompts the user to input the name of the default data file and thereafter, the number crunching starts.

Working Step 3

The simulated results are presented in four separate files which can be used for post-processing purposes,

- *aerout.txt* contains the data for finite element simulations in form of velocities and pressures which can be visualised in **COSMOS 2.6/M** or surfer considering 9-noded finite element;

- *out-4.txt* contains values of velocities and pressures which can be visualised in **COSMOS 2.6/M** or **SURFER** considering 4-noded finite element;
- *tp.dat* and *tp-4.dat* are datafiles containing nodal velocities and pressures respectively accompanied by nodal coordinates and connectivity in a format suitable to be imported in **TECPLOT** for visualisations.

INPUT FILE SPECIFICATIONS

The default data file *df.dat* provides physical properties, geometrical characteristics of finite element mesh of the flow solution domain and the values of the imposed boundary constraints as a formatted input. The format specification of the data file has been explained per each line.

Line 1

Variable	Identity	Format
	Title	a

Line 2

Variable	Identity	Format
ncn	Number of nodes per element	i5
ngaus	Number of Gauss integration points	i5
nter	Number of iterations	i5

Line 3

Variable	Identity	Format
nnp	Total number of nodes	i5
nel	Total number of elements	i5
nbc	Total number of boundary constraints	i5
nmat	Total number of fluid mediums	i5

Line 4

Variable	Identity	Format
ntep	Count parameters for printing results of each iteration	i5
icord	Switch for selecting the coordinate system of equations	i5
nter	Number of iterations	i5

Line 5

Variable	Identity	Format
tolv	Tolerance value for discrepancy in velocity values obtained in two successive iterations	f10.5
tolp	Tolerance value for discrepancy in pressure values obtained in two successive iterations	f10.5

Line 6

Variable	Identity	Format
permx	Permeability value in x-direction	d10.5
permy	Permeability value in y-direction	d10.5

Line 7

Variable	Identity	Format
rvisc	Consistency coefficient for viscosity	d10.5
power	Power law index for viscosity	d10.5
tref	Reference temperature	d10.5
pref	Reference pressure	d10.5
roden	Fluid density	d10.5
gamad	Shear rate	d10.5

Line 8………………Line (8 + nnp)

Variable	Identity	Format
jnp	Counter for node number	i5
cord(jnp, 1)	x-coordinate of node jnp	e20.8
cord(jnp, 2)	y-coordinate of node jnp	e20.8

where, nnp: Total number of nodes and jnp = 1,….,nnp

Line (8 + nnp) + 1………………Line (8 + nnp) + nel

Variable	**Identity**	**Format**
jel	Counter for element number	i5
node(iel, 1)….node(iel, ncn)	Node numbers associated with element iel	9i8

where, nel: Total Number of elements and jel = 1, …., nel

ncn: Number of nodes per element

Line (8 + nnp) +(nel + 1)………………Line (8 + nnp) + (nel + nbc)

Variable	**Identity**	**Format**
ibc	Node number having boundary constraints	i5
jbc	Numerical identifier for the boundary constraint	i5
vbc	Value of the boundary constraint jbc	f10.4

where, nbc: Total number of imposed boundary constraints and vbc = 1, ….,nbc

---

Sample input data file of df.dat

---------------------------------------START OF FILE---------------------------------------

```
Coupled free/porous flow dynamics
    9    3    1
22761 5600 1276    1
    1    0
   0.00001   0.00001
  10.0D-12  10.0D-12
   0.80D+2   1.00D00 273.00D00   1.013D5  0.970D+3   2.00D-1
```

Mesh, physical and numerical data

```
    1      0.49999999E-02      0.49999999E-02
    2      0.49999999E-02      0.50625000E-02
    3      0.49999999E-02      0.51249997E-02
    4      0.49999999E-02      0.51874998E-02
    5      0.49999999E-02      0.52500004E-02
    6      0.49999999E-02      0.53124996E-02
    7      0.49999999E-02      0.53750002E-02
    8      0.49999999E-02      0.54374998E-02
    9      0.49999999E-02      0.54999995E-02
   10      0.49999999E-02      0.55625001E-02
   11      0.49999999E-02      0.56250002E-02
   12      0.49999999E-02      0.56875004E-02
   13      0.49999999E-02      0.57500000E-02
   14      0.49999999E-02      0.58125001E-02
   15      0.49999999E-02      0.58749993E-02
   16      0.49999999E-02      0.59374999E-02
   17      0.49999999E-02      0.60000005E-02
   18      0.49999999E-02      0.60624997E-02
   19      0.49999999E-02      0.61249994E-02
   20      0.49999999E-02      0.61875000E-02
```

Nodal coordinates

```
1    1   82  163  164  165   84    3    2   83
2    3   84  165  166  167   86    5    4   85
3    5   86  167  168  169   88    7    6   87
4    7   88  169  170  171   90    9    8   89
5    9   90  171  172  173   92   11   10   91
6   11   92  173  174  175   94   13   12   93
7   13   94  175  176  177   96   15   14   95
8   15   96  177  178  179   98   17   16   97
```

Elemental connectivity

```
     2     1    0.1000
     3     1    0.1000
     4     1    0.1000
     5     1    0.1000
     6     1    0.1000
     7     1    0.1000
     8     1    0.1000
     9     1    0.1000
    10     1    0.1000
    11     1    0.1000
    12     1    0.1000
    13     1    0.1000
    14     1    0.1000
    15     1    0.1000
    16     1    0.1000
    17     1    0.1000
    18     1    0.1000
    19     1    0.1000
    20     1    0.1000
```

Boundary conditions

--------------------------------------END OF FILE---------------------------------------

The data file *cord-4.dat* provideds nodal coordinates and connectivity of finite element mesh generated using 4-noded element which can be used for post-processing of pressure values. The format specification of the data file has been explained per each line.

Line 1

Variable	**Identity**	**Format**
qnp	Total number of nodes in 4-node element mesh	i5
qel	Total number of elements in 4-node element mesh	i5

Line 2………………Line (2 + qnp)

Variable	**Identity**	**Format**
jnp	Counter for node number	i5
gord(jnp, 1)	x-coordinate of node jnp	e20.8
gord(jnp, 2)	y-coordinate of node jnp	e20.8

where, qnp: Total number of nodes and jnp = 1,….,qnp

Line (2 + qnp) + 1………………Line (q + nnp) + qel

Variable	**Identity**	**Format**
jel	Counter for element number	i5
rode(iel, 1)….rode(iel, ncn)	Node numbers associated with element iel	9i8

where, qel: Total Number of elements and jel = 1, …., qel

ncn: Number of nodes per element = 4

Sample input data file of cord-4.dat

--START OF FILE--

Mesh data

```
5781 5600
```

Nodal coordinates

```
    1       0.49999999E-02      0.49999999E-02
    2       0.49999999E-02      0.51249997E-02
    3       0.49999999E-02      0.52500004E-02
    4       0.49999999E-02      0.53750002E-02
    5       0.49999999E-02      0.54999995E-02
    6       0.49999999E-02      0.56249998E-02
    7       0.49999999E-02      0.57500000E-02
    8       0.49999999E-02      0.58749993E-02
    9       0.49999999E-02      0.60000005E-02
   10       0.49999999E-02      0.61249994E-02
   11       0.49999999E-02      0.62499996E-02
   12       0.49999999E-02      0.63749994E-02
```

Elemental connectivity

```
       1       1      42      43       2
       2       2      43      44       3
       3       3      44      45       4
       4       4      45      46       5
       5       5      46      47       6
       6       6      47      48       7
       7       7      48      49       8
       8       8      49      50       9
```

PROGRAM SUBROUTINES - HEIRARCHY

SUBROUTINES, LEVEL 1

1.	CLEAN	:	Called by **CFDYNAMICS**
2.	CONTOL	:	Called by **CFDYNAMICS**
3.	FLOW	:	Called by **CFDYNAMICS**
4.	GETELM	:	Called by **CFDYNAMICS**
5.	GETMAT	:	Called by **CFDYNAMICS**
6.	GETNOD	:	Called by **CFDYNAMICS**
7.	OUTPUT	:	Called by **CFDYNAMICS**
8.	PUTBCV	:	Called by **CFDYNAMICS**
9.	SECINV	:	Called by **CFDYNAMICS**
10.	SETPRM	:	Called by **CFDYNAMICS**
11.	TECPLOT	:	Called by **CFDYNAMICS**

SUBROUTINES, LEVEL 2

1.	DERIV	:	Called by **FLOW**
2.	FRONT	:	Called by **FLOW**
3.	GAUSSP	:	Called by **FLOW**
4.	SHAPE	:	Called by **FLOW**
5.	VISCA	:	Called by **FLOW**

SUBROUTINES, LEVEL 3

1.	BACSUB	:	Called by **FRONT**

Please note that all the subroutines at a particular level have been listed alphabetically

PROGRAM SUBROUTINES - FUNCTIONS

CLEAN	Cleans the arrays and prepares them for solution
CONTOL	Makes a check for the convergence of variables
DERIV	Calculates the Jacobean matrix, the determinant of the Jacobean matrix and global derivatives of the shape functions used in finite element calculations
FLOW	Calculates the velocities and pressures based on finite element method
FRONT	Employs the frontal method for solving the final set of equations
GAUSSP	Specifies the gauss points and weights for quadrature integration
GETELM	Specifies the nodal connectivity array for every element
GETMAT	Reads the material parameters
GETNOD	Reads the coordinates of the filter mesh from the MESH.FEM file
OUTPUT	Prints the final solution in form of 2 - dimensional velocities (x and y components) and the pressure field at different times
PUTBCV	Imposes the primary boundary conditions for velocity
SECINV	Calculates the second invariant of rate of deformation stress tensor, which depends on the velocity gradients for a particular time step

Please note that all the subroutines at a particular level have been listed alphabetically

PROGRAM SUBROUTINES - FUNCTIONS

SETPRM	Sets the location data for nodal degrees of freedom that are the velocities and the pressures associated with each node in the cartridge mesh
SHAPE	Calculates the shape functions and their derivatives used in finite element calculations
TECPLOT	Generates output files for post-processing in Tecplot
VISCA	Calculates the viscosity according to the power law model valid for shear thickening hydraulic fluids used in aircraft filters

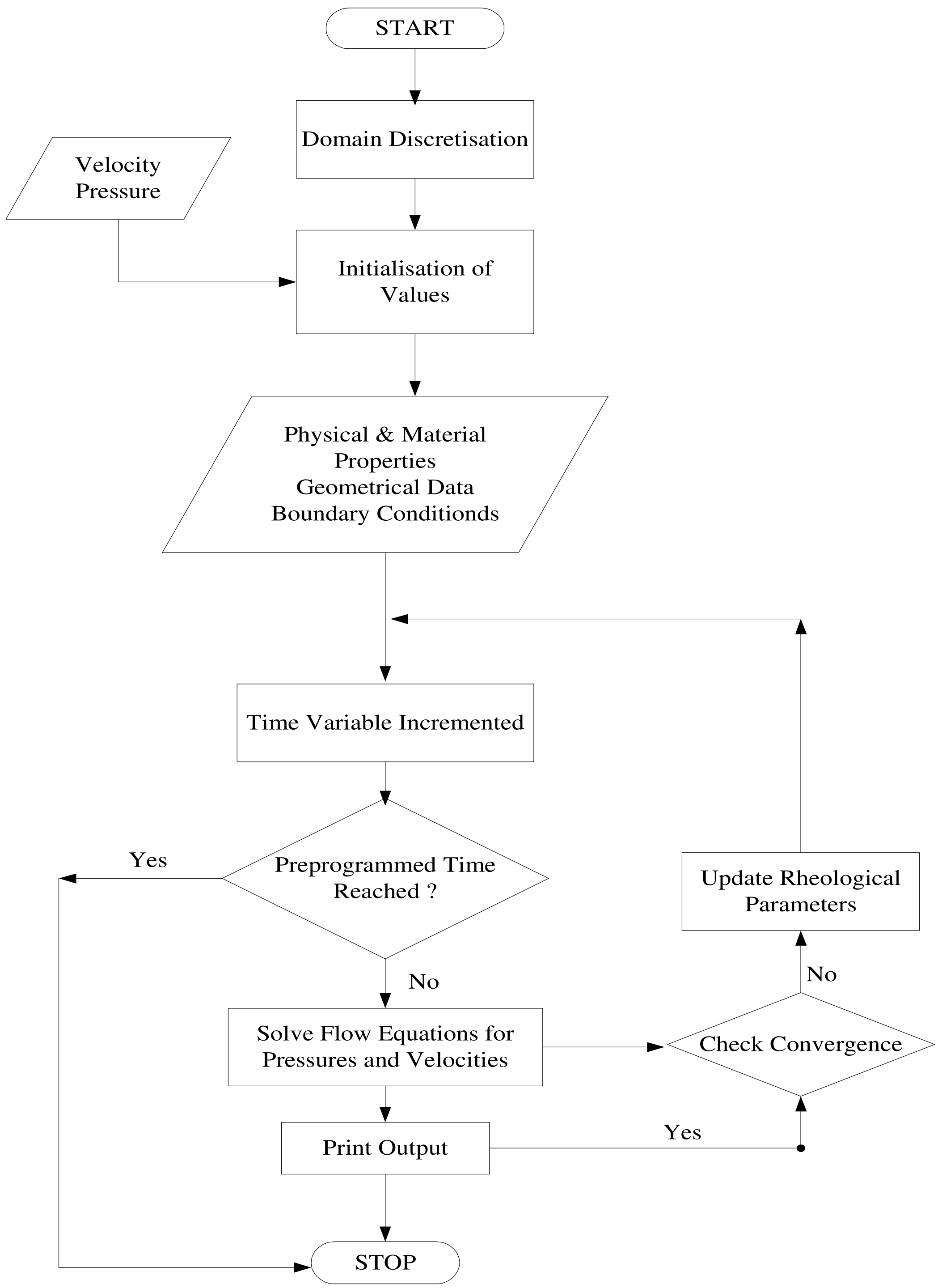
LIST OF PRIME VARIABLES - CFDYNAMICS

aa:	Left hand side finite element stiffness matrix
ak:	Right hand side finite element stiffness matrix
b:	Measure of Gaussian integration
bc:	Array for storing boundary constraint value
cg:	Weighting functions for Gauss integration
cord:	Array storing coordinate values of each nodal point
da:	Global derivative of shape function
del:	Local derivatives of shape function
detj:	Determinant of Jacobian matrix
dispc:	Value of dispersion coefficient
errp:	Square of discrepancy between pressure values calculated in two successive iterations
errv:	Square of discrepancy between velocity values calculated in two successive iterations
eta:	y-coordinate in local coordinate system
gamad:	Rate of deformation of viscous stress tensor
icord:	Switch for changing the coordinate system of equations
maxbc:	Limit for maximum number of boundary constraints that can be specified
maxdf:	Limit for maximum number of nodal degrees of freedom
maxel:	Limit for maximum number of elements in mesh
maxnp:	Limit for maximum number of nodal points in mesh
maxst:	Maximum dimension of finite element stiffness matrix
nbc:	Total number of boundary-node constraints
ncn:	Number of nodes associated with each finite element
ncod:	Array for constraint switch defined for every degree of freedom
ndf:	Total number of nodal degrees of freedom
ndim:	Dimensions of the solution domain
nel:	Total number of elements in pleated cartridge mesh
ngaus:	Number of quadrature points required for Gaussian integration

LIST OF PRIME VARIABLES - CFDYNAMICS

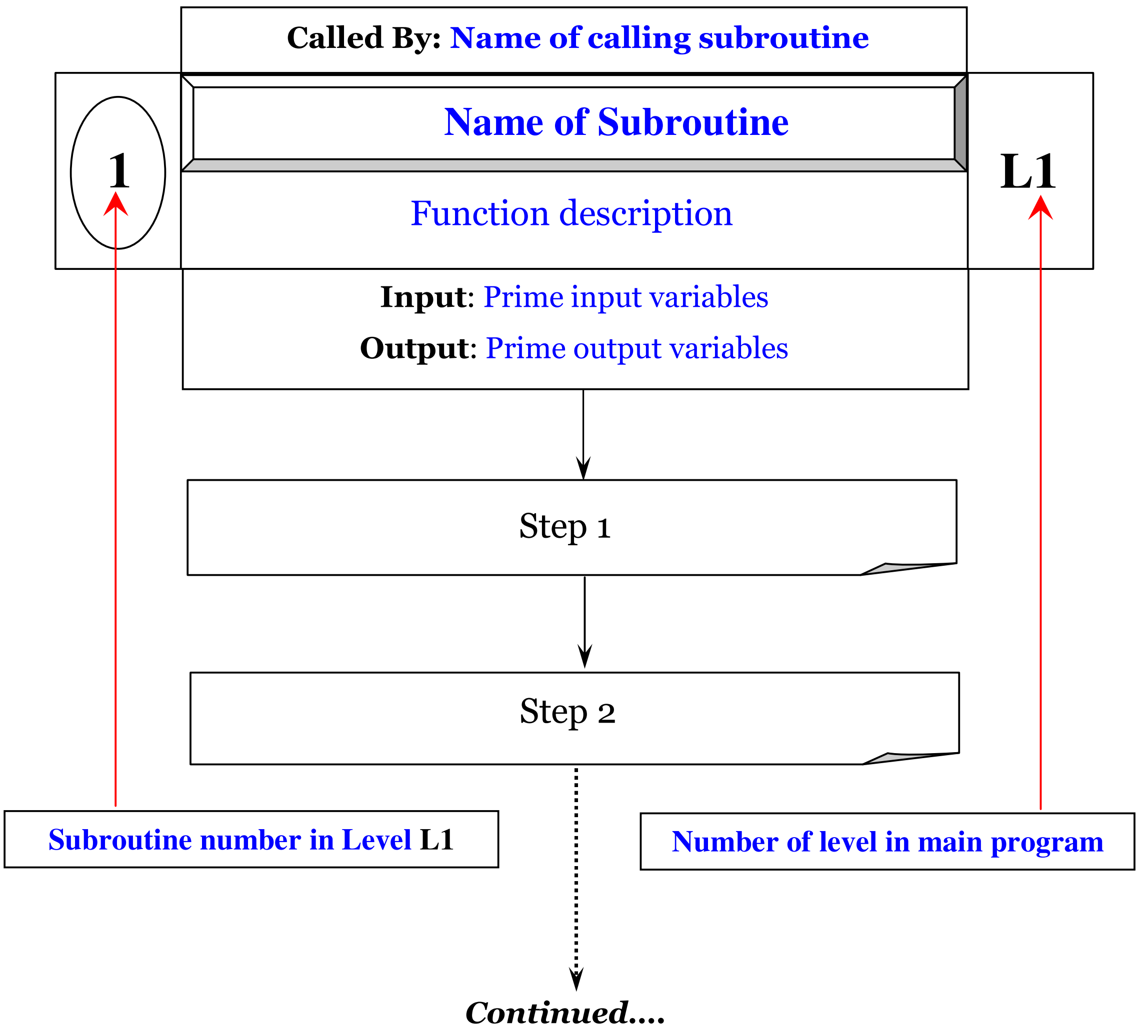
nmat:	Total number of fluids
nnp:	Total number of nodal points in pleated cartridge mesh
node:	Array for nodal connectivity of elements
nter:	Number of solution iterations
num:	Number of integration points per element
p:	Finite element shape or interpolation functions
permx:	Permeability of the filter medium in x-direction
permy:	Permeability of the filter medium in y-direction
pmat:	Array storing values of physical properties of the fluid
power:	Power law index for the fluid viscosity
press:	Array for nodal pressure
rpef:	Reference pressure
rr:	Array for storing the elemental load vector
rtem:	Reference temperature
rvisc:	Consistency coefficient for viscosity of fluid
sinv:	Array storing components of second invariant of rate of deformation of stress tensor
soln:	Array for storing solution values calculated by Frontal Solver
tolv:	Maximum allowable tolerance for fluctuation in velocity value
tolp:	Maximum allowable tolerance for fluctuation in pressure value
vel:	Array for nodal velocities
viisc:	Viscosity of fluid used for each set of experiment
visc:	Updated value of viscosity according to power law model
xg:	Coordinates of Gauss quadrature points for integration
xi:	x-coordinate in local coordinate system

SOLUTION ALGORITHM - CFDYNAMICS



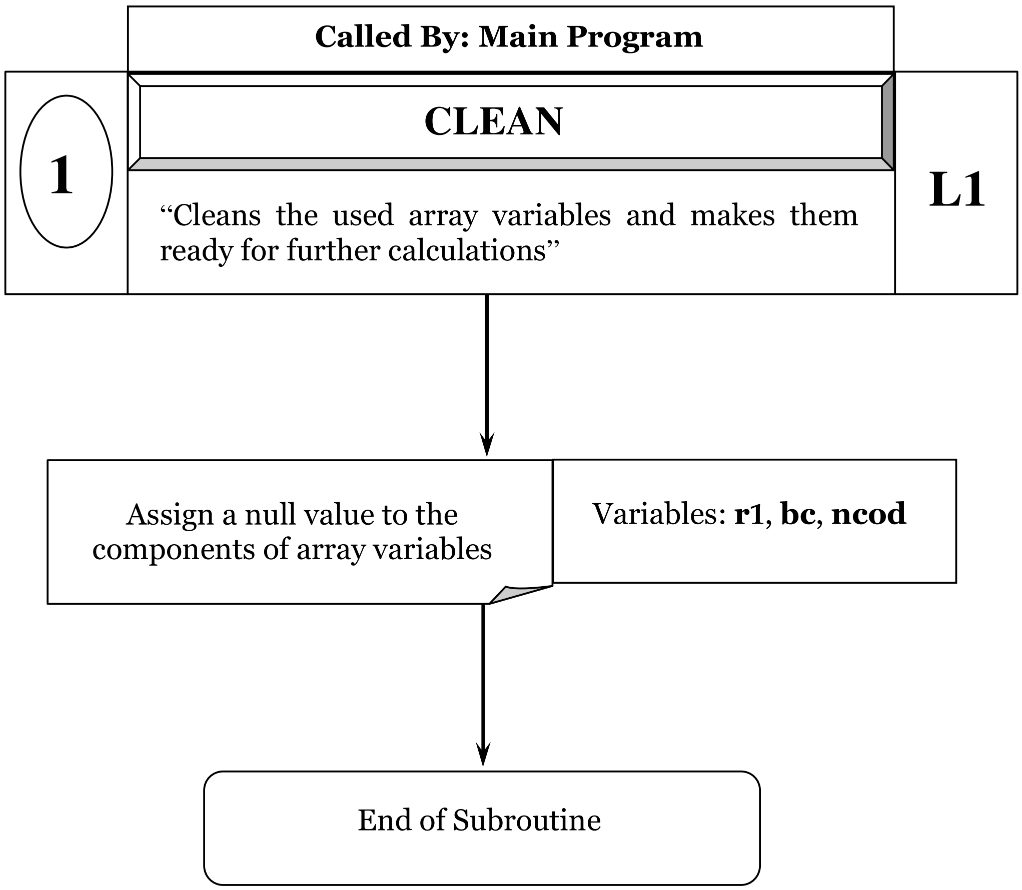
PROGRAM SUBROUTINES - INTERACTIONS

A PROTOTYPE CONVENTION



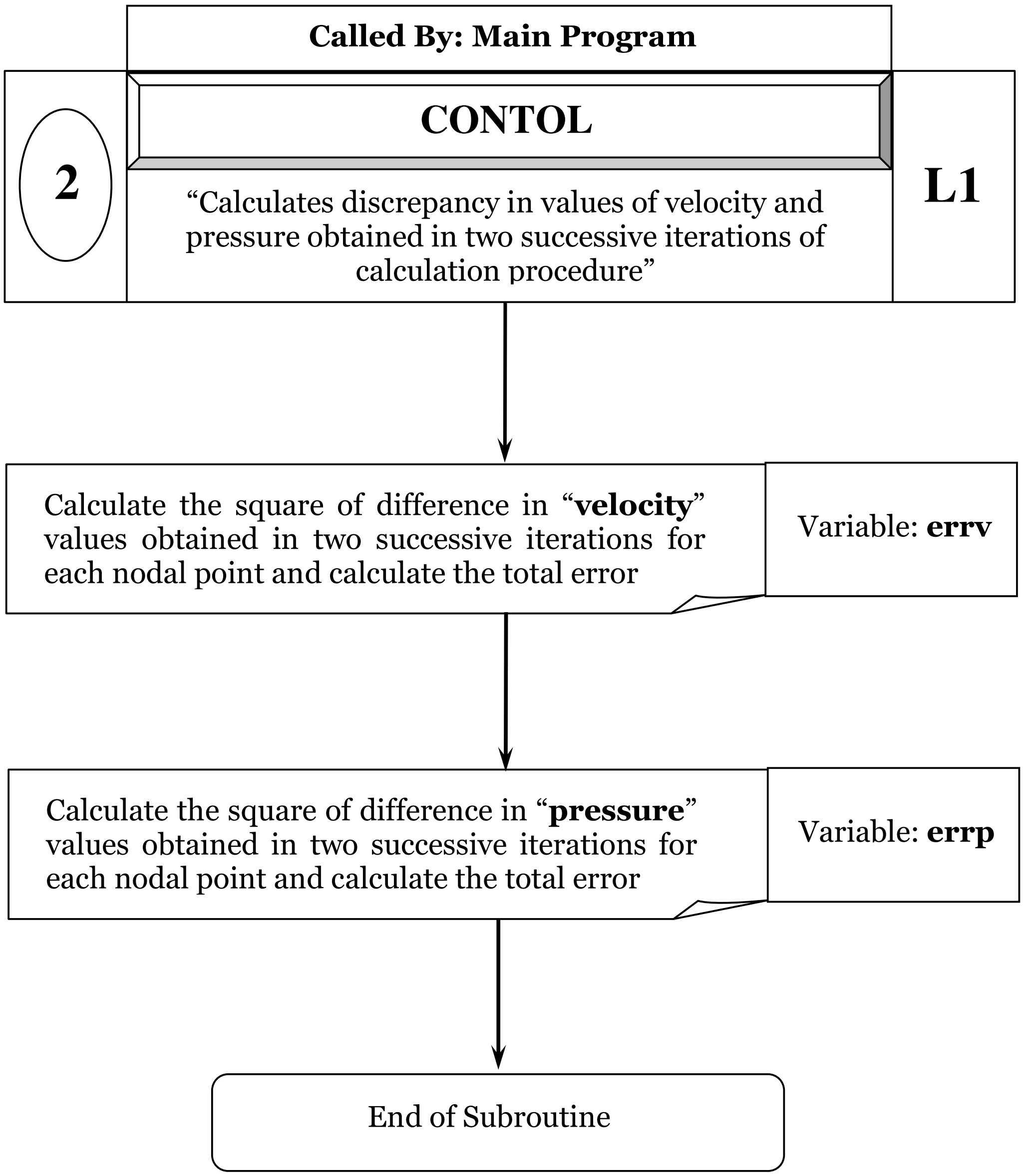
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE CLEAN, LEVEL 1



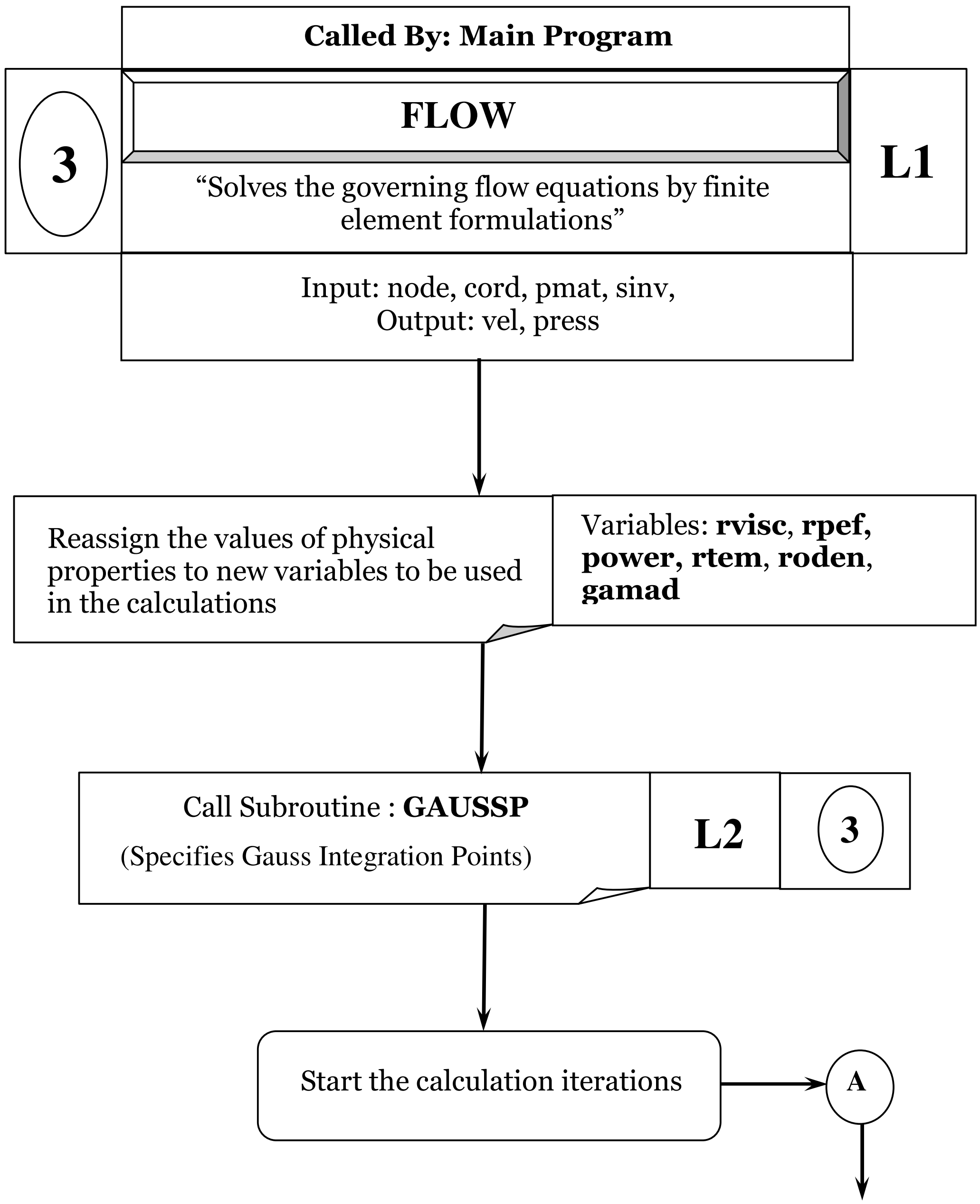
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE CONTOL, LEVEL 1

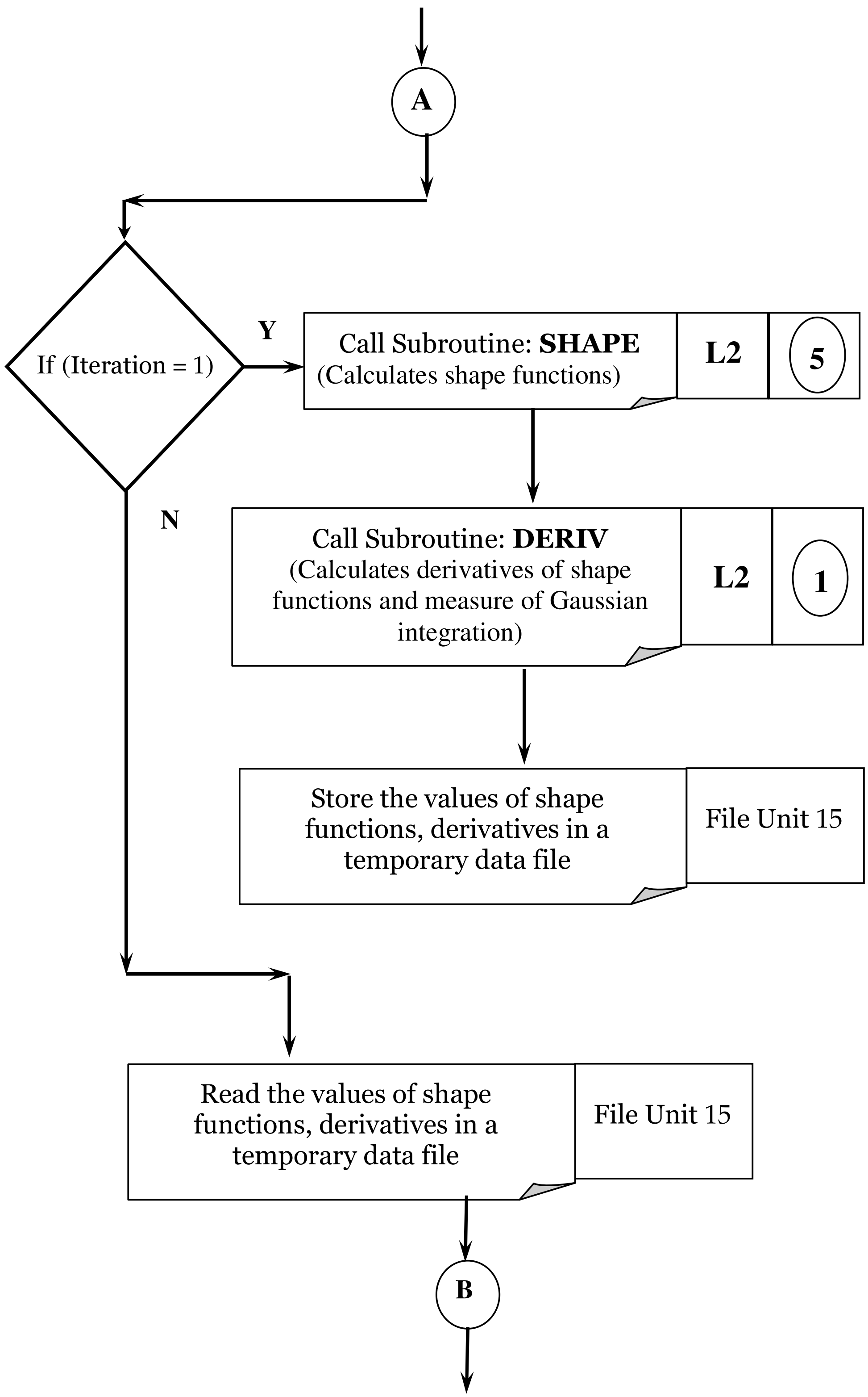


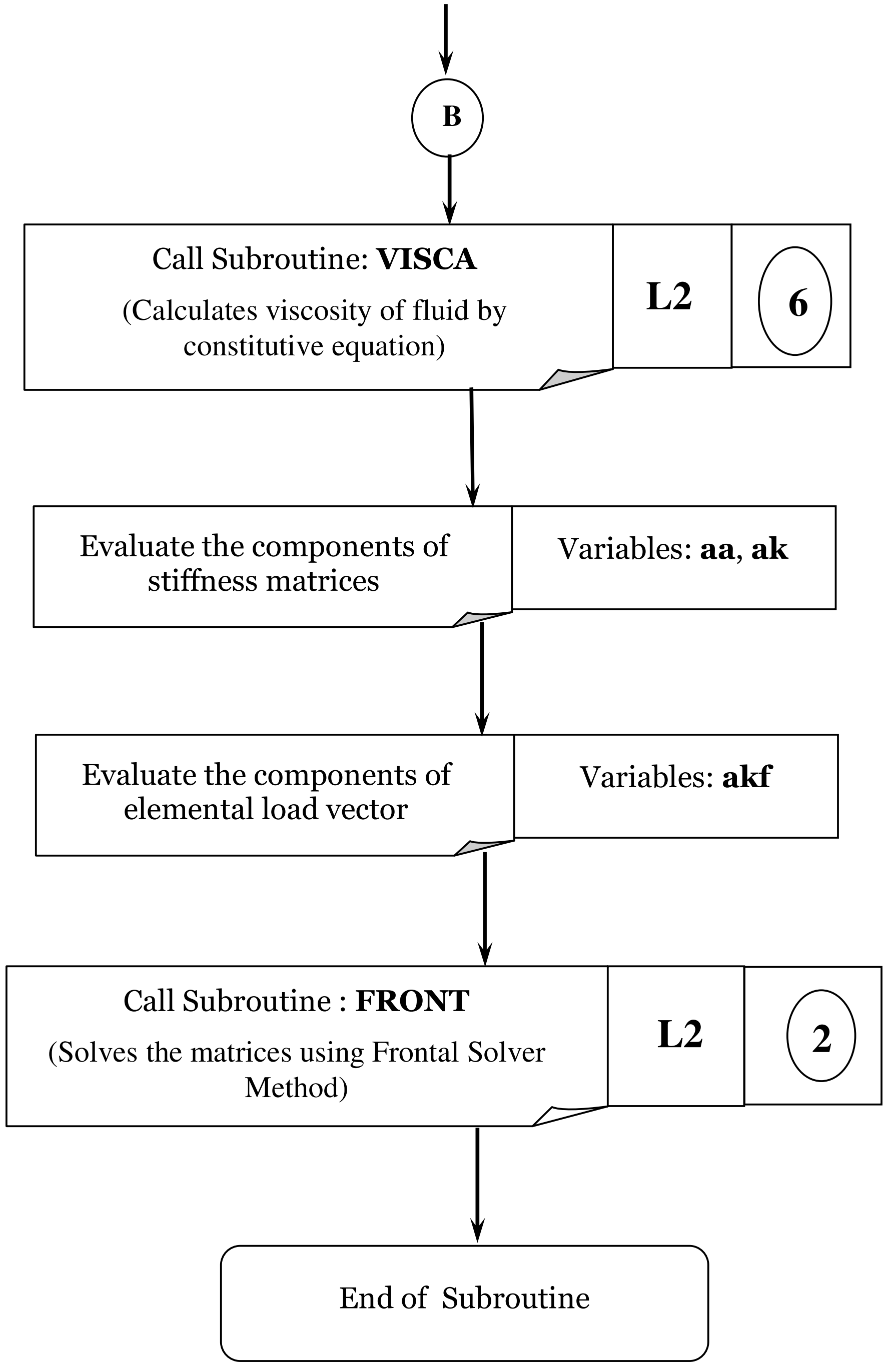
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE FLOW, LEVEL 1



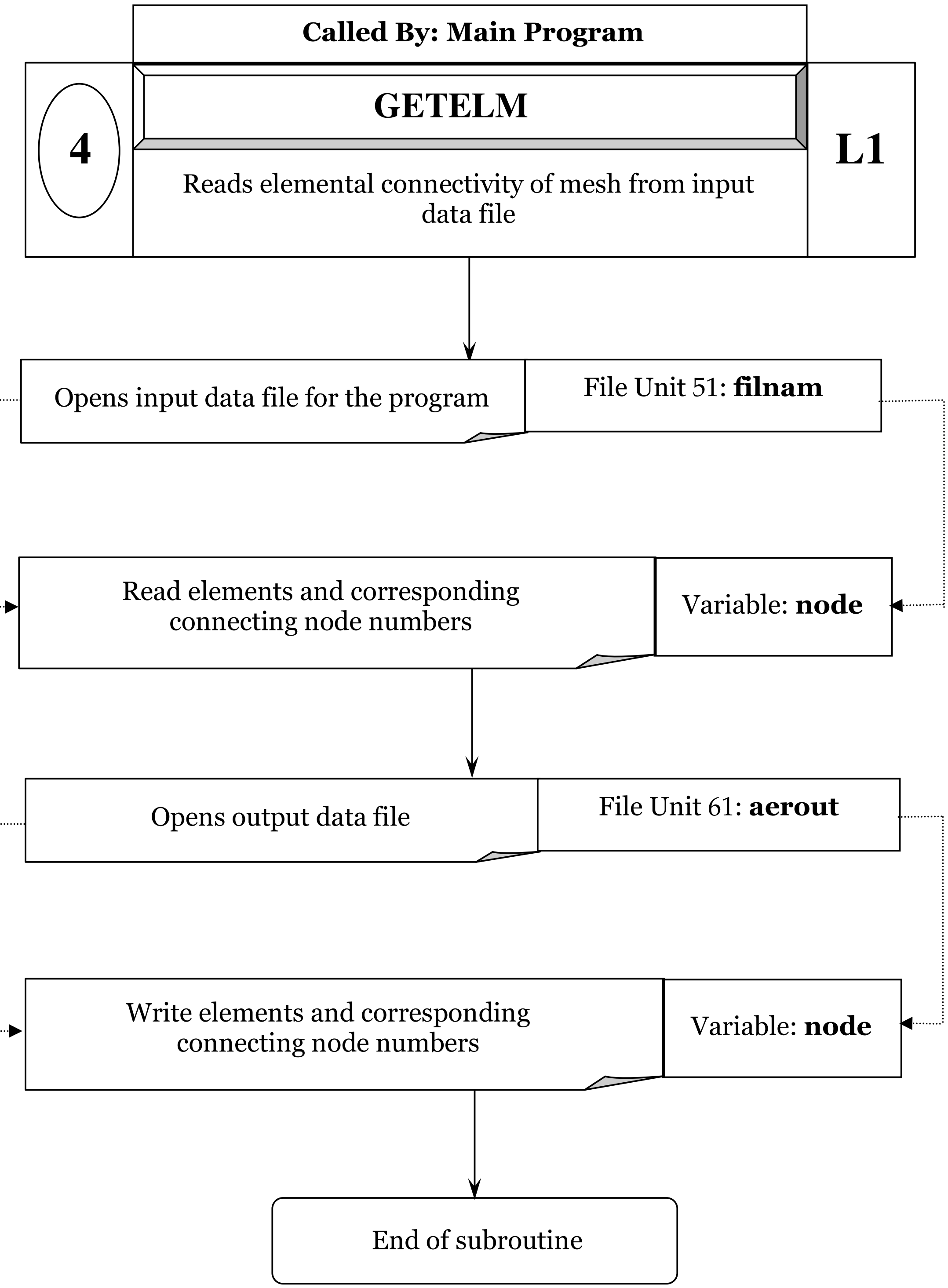
PROGRAM SUBROUTINES - INTERACTIONS




B
Call Subroutine: **VISCA**
(Calculates viscosity of fluid by constitutive equation)
L2
6
Evaluate the components of stiffness matrices
Variables: **aa**, **ak**
Evaluate the components of elemental load vector
Variables: **akf**
Call Subroutine : **FRONT**
(Solves the matrices using Frontal Solver Method)
L2
2
End of Subroutine

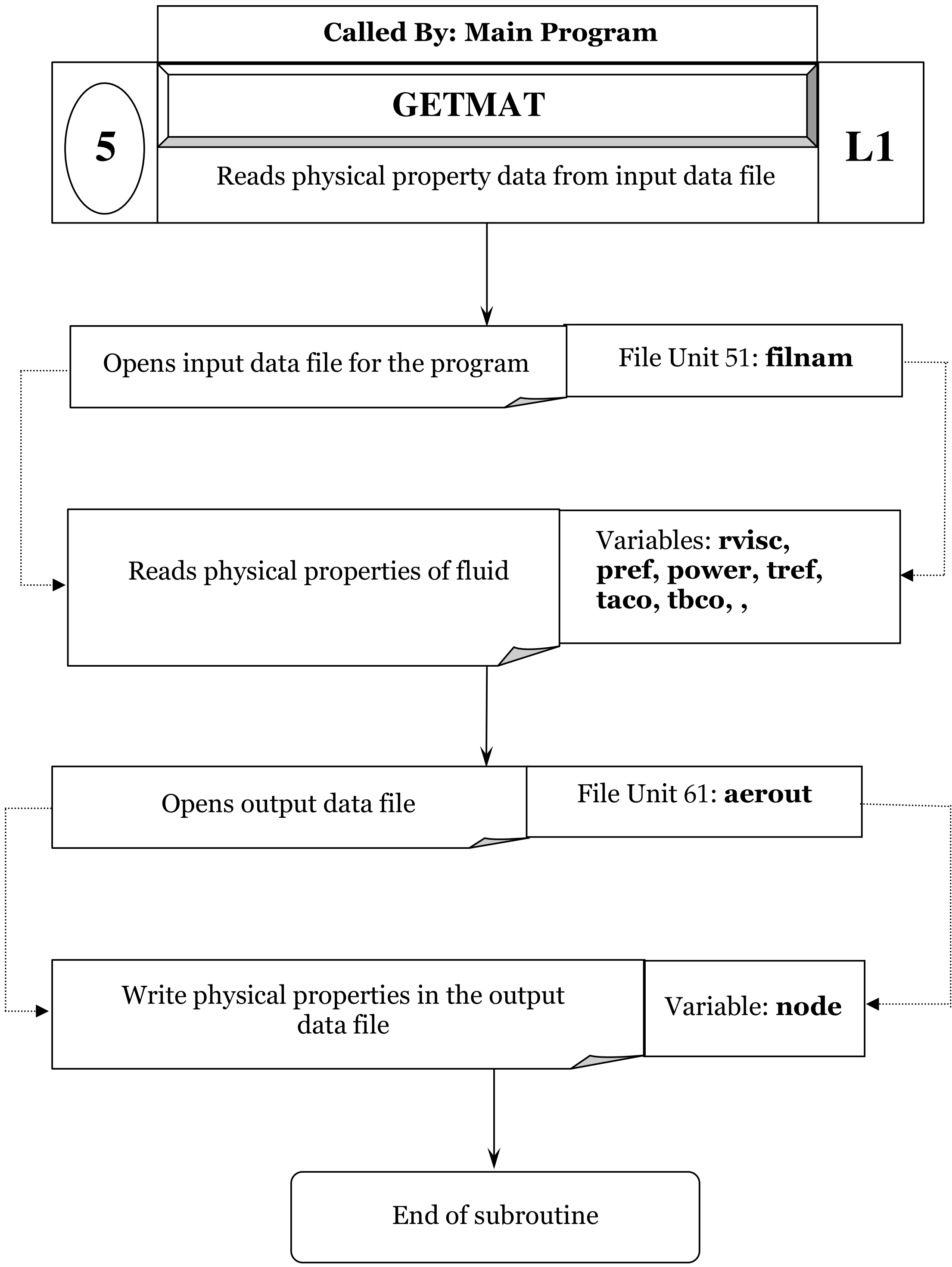
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE GETELM, LEVEL 1



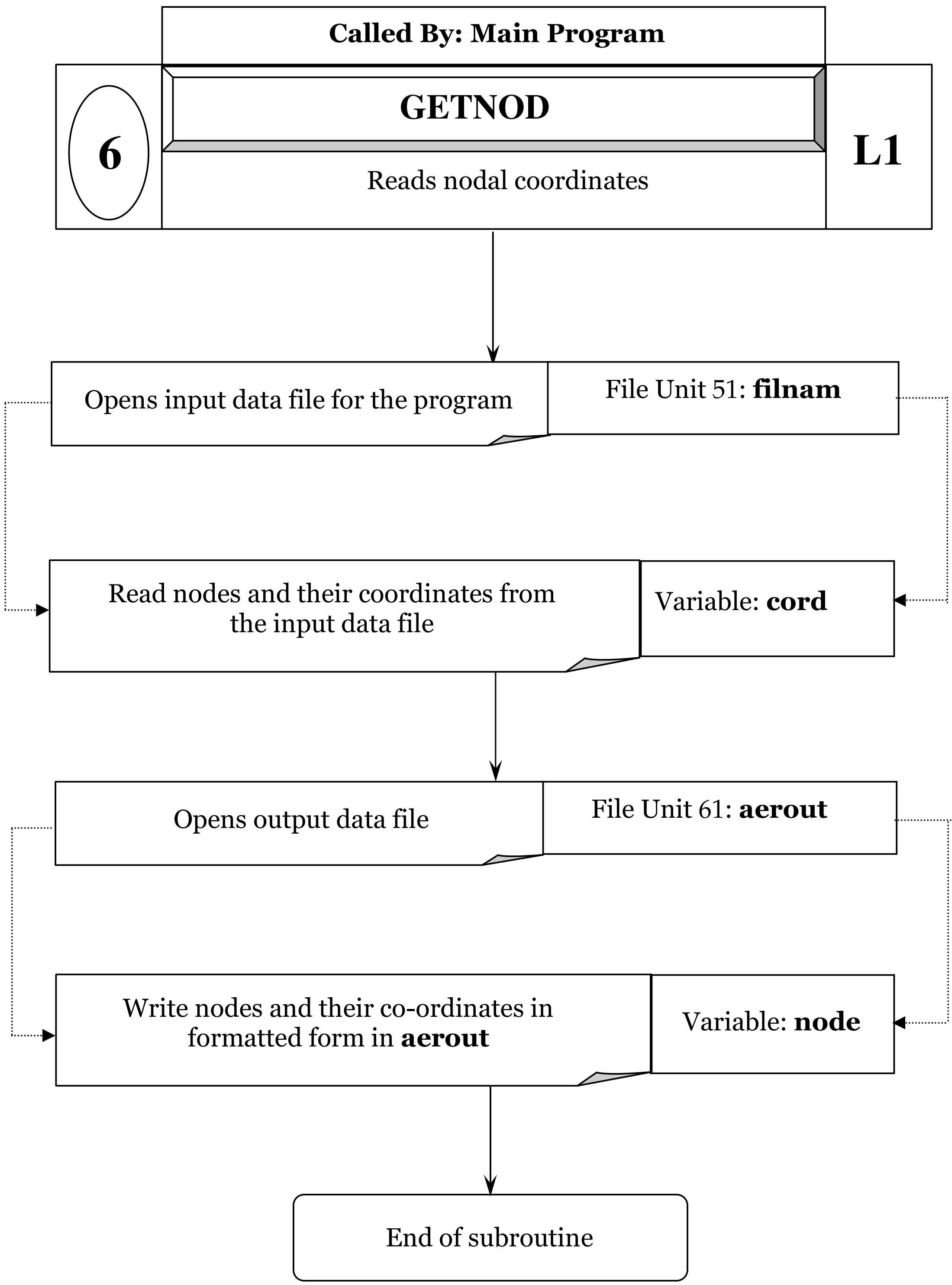
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE GETMAT, LEVEL 1



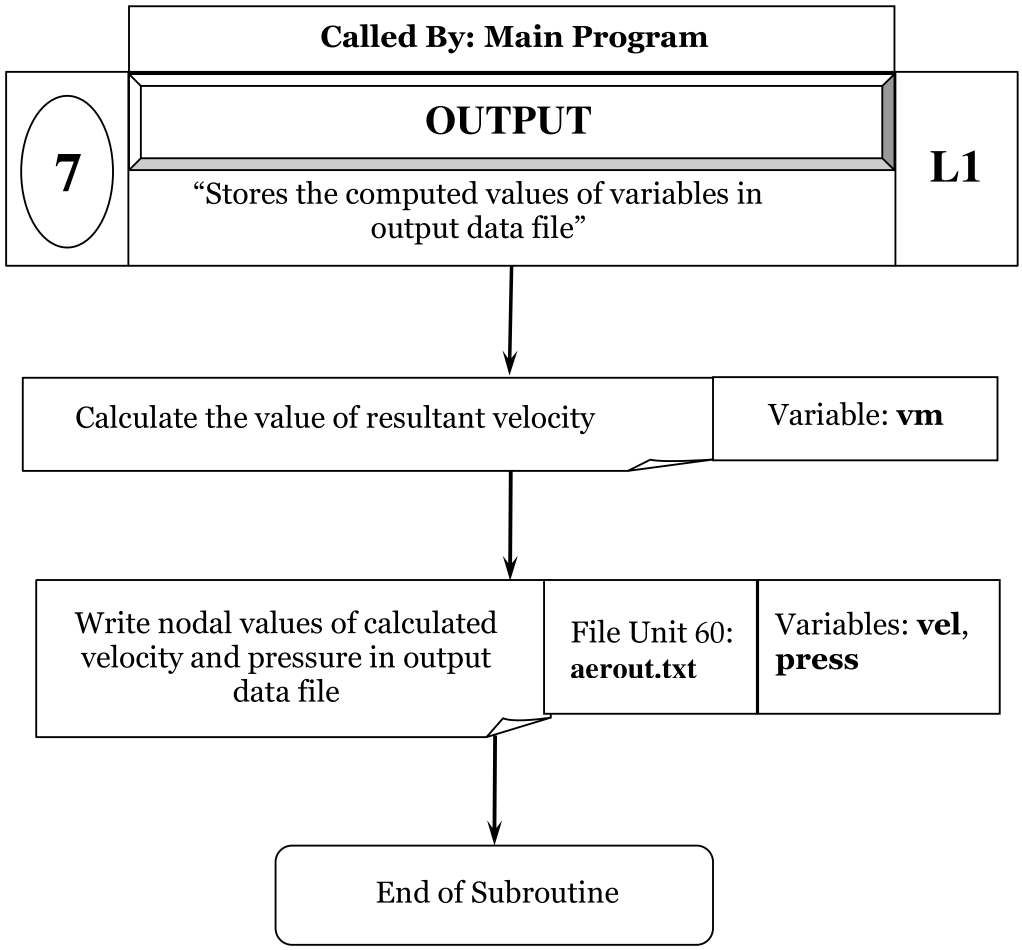
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE GETNOD, LEVEL 1



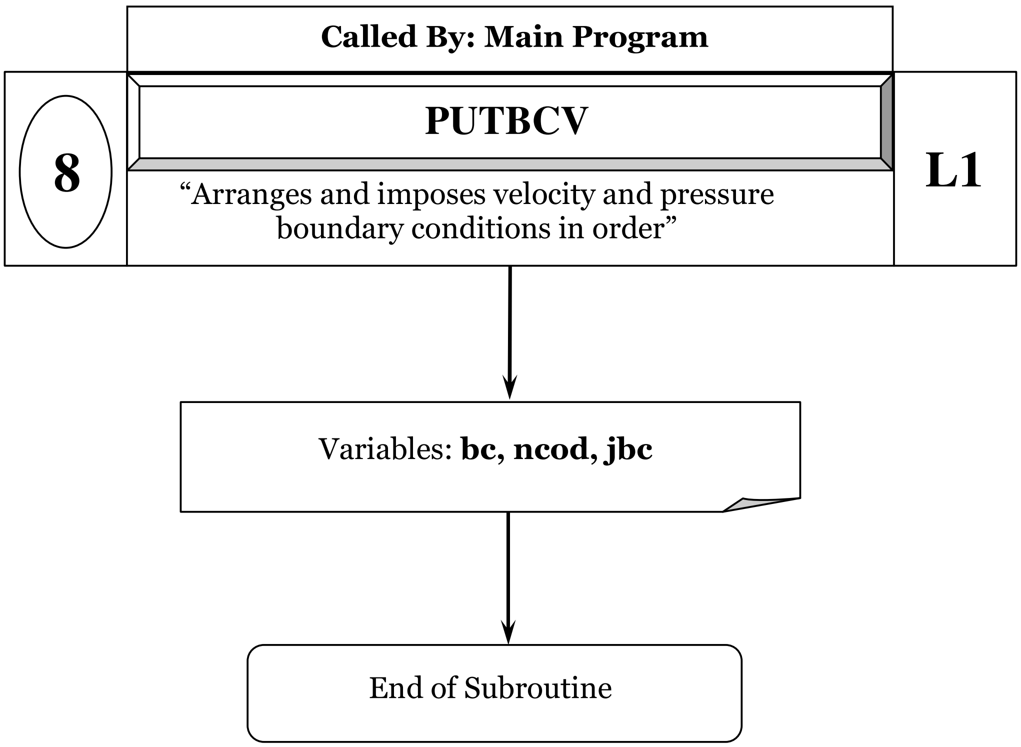
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE OUTPUT, LEVEL 1



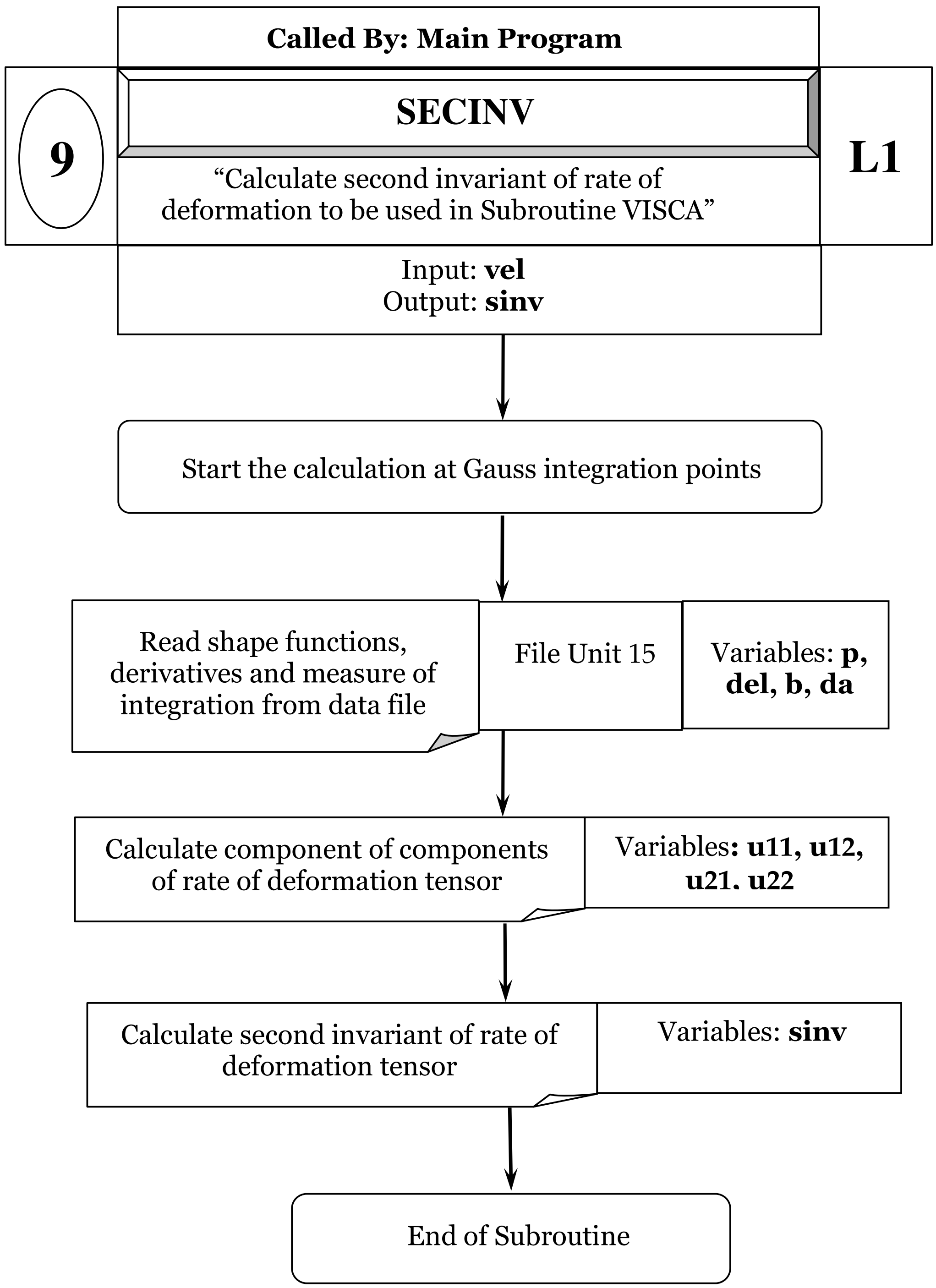
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE PUTBCV, LEVEL 1



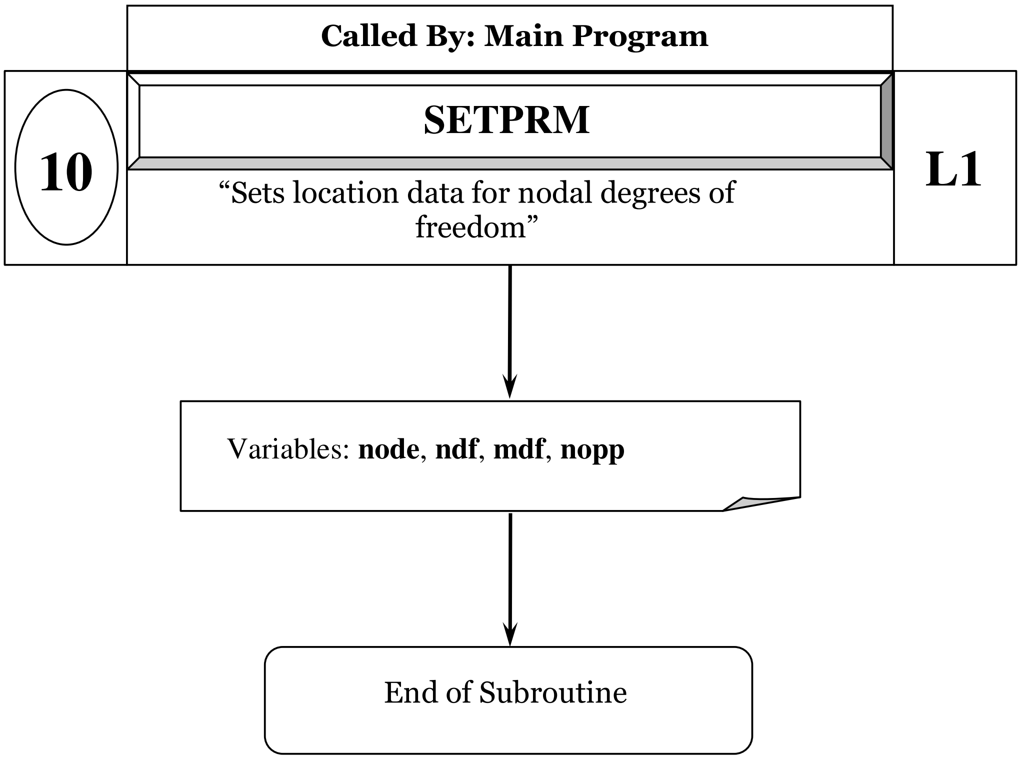
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE SECINV, LEVEL 1



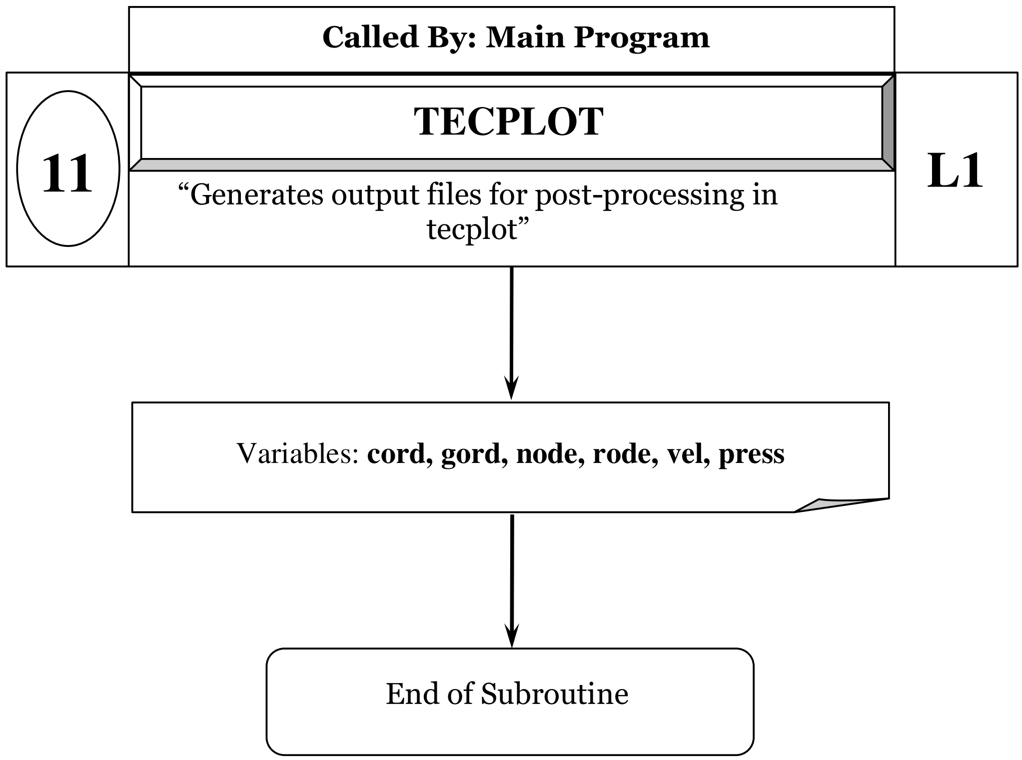
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE SETPRM, LEVEL 1



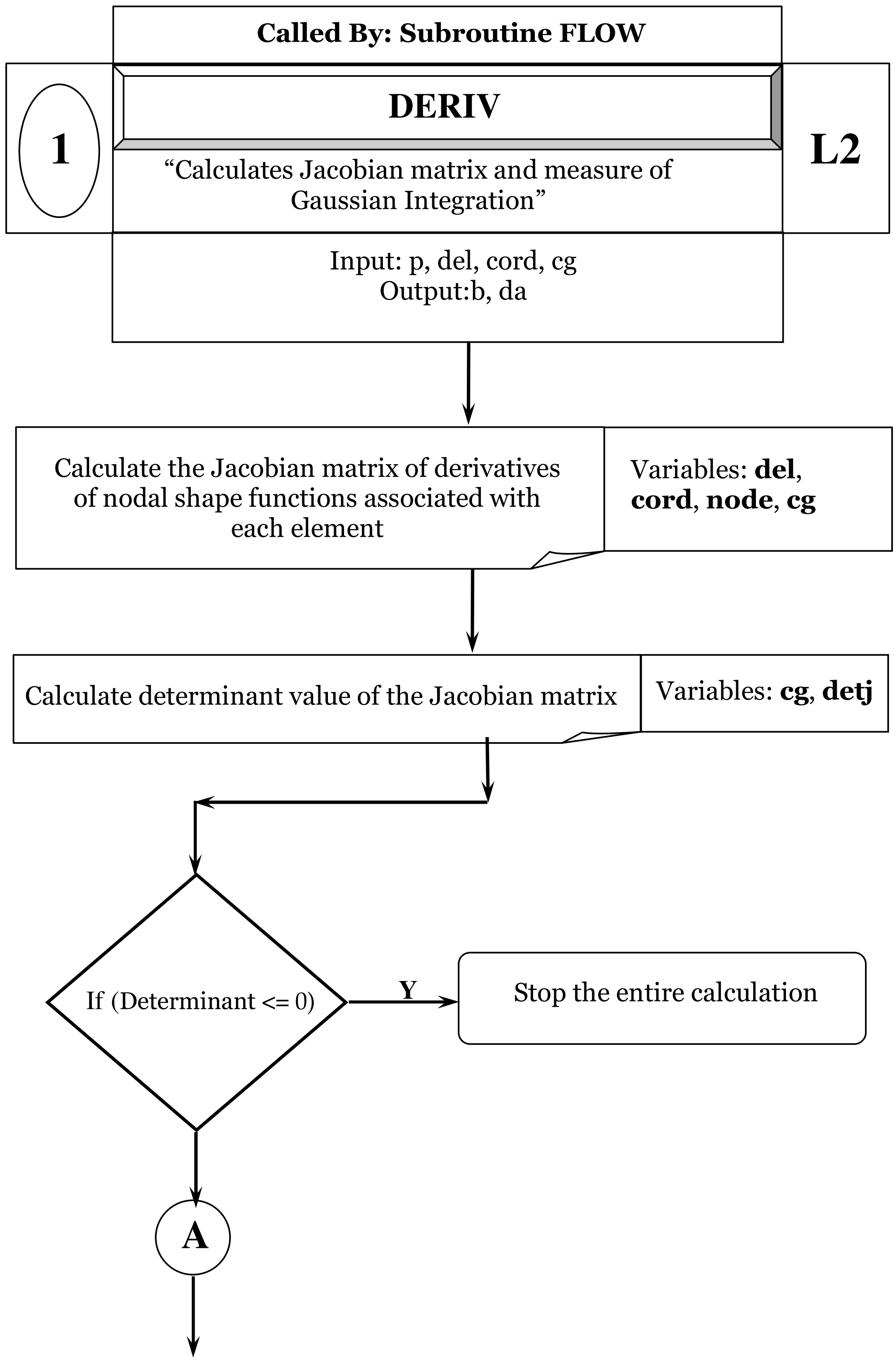
PROGRAM SUBROUTINES - INTERACTIONS

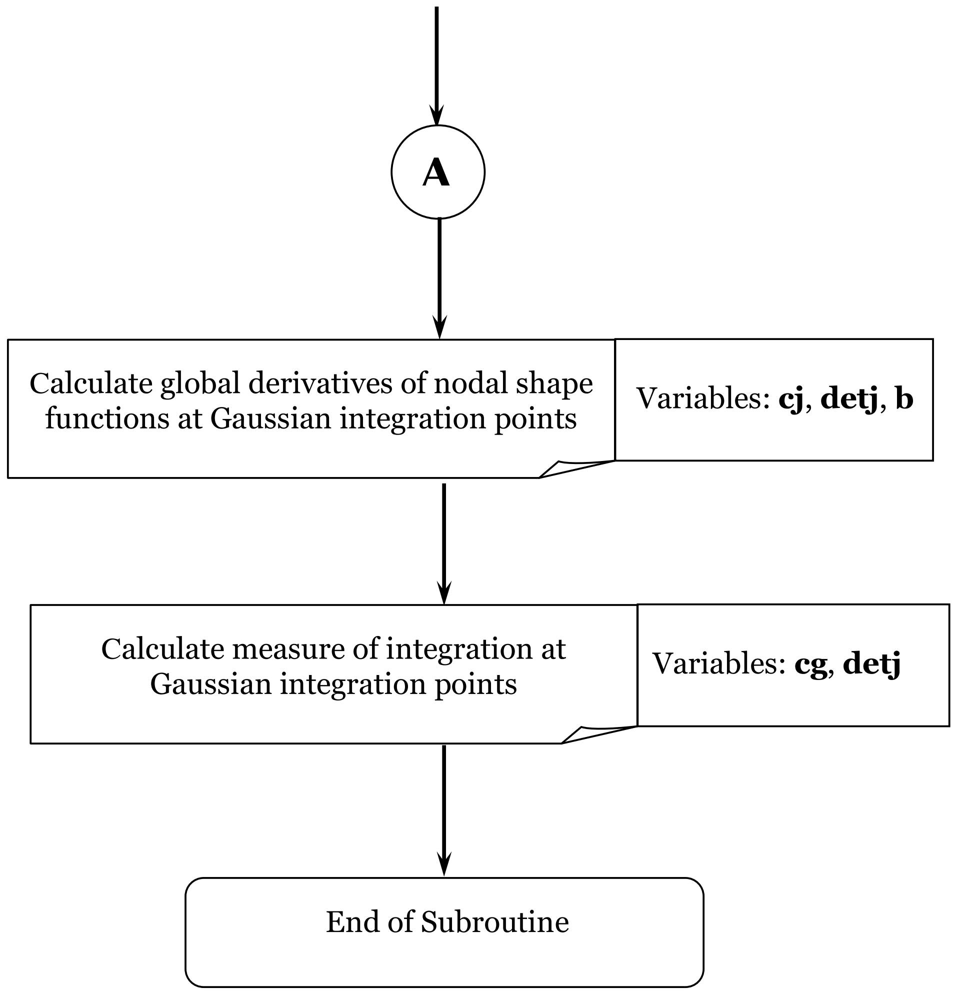
SUBROUTINE TECPLOT, LEVEL 1



PROGRAM SUBROUTINES - INTERACTIONS

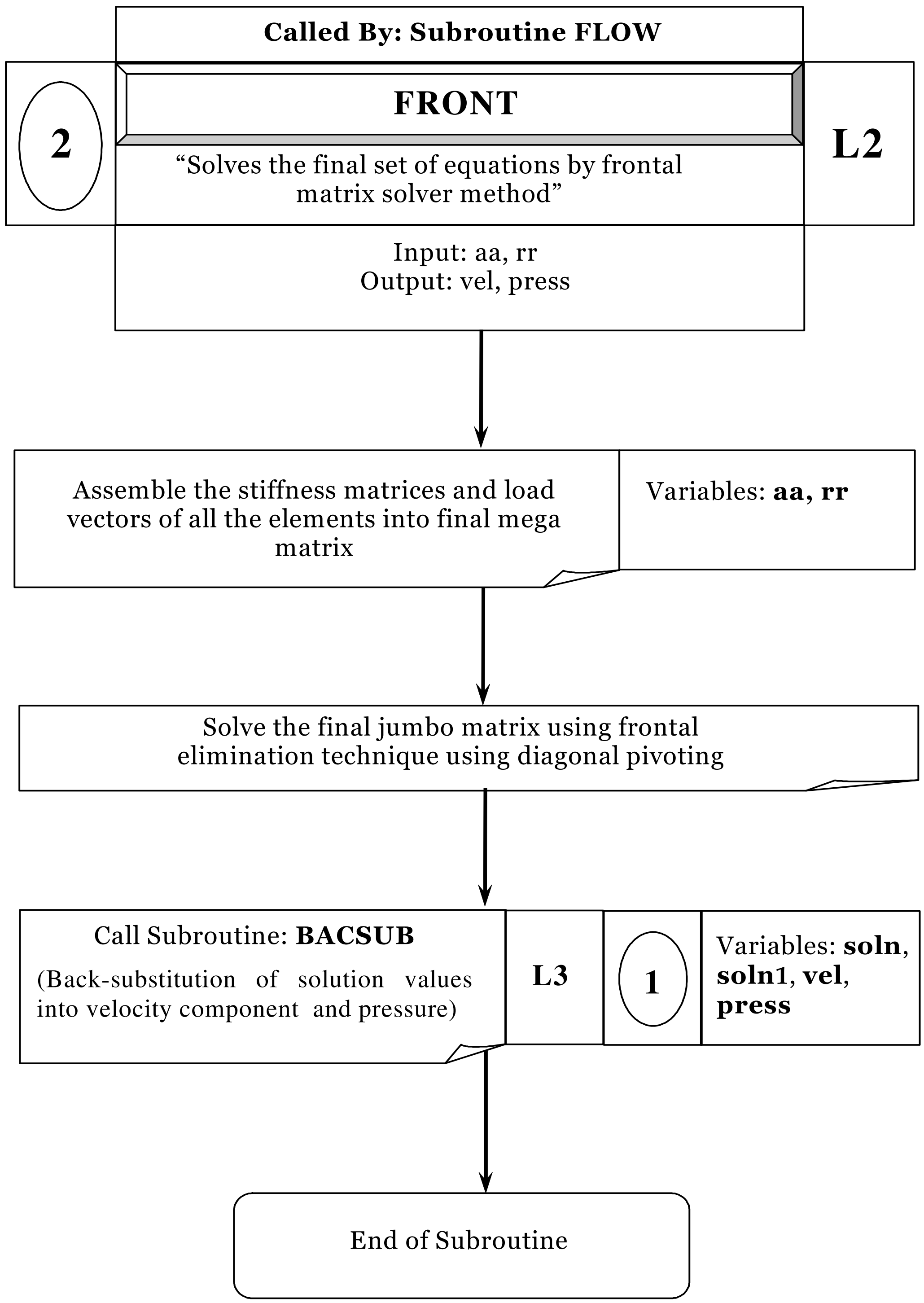
SUBROUTINE DERIV, LEVEL 2




A
Calculate global derivatives of nodal shape functions at Gaussian integration points
Variables: **cj**, **detj**, **b**
Calculate measure of integration at Gaussian integration points
Variables: **cg**, **detj**
End of Subroutine

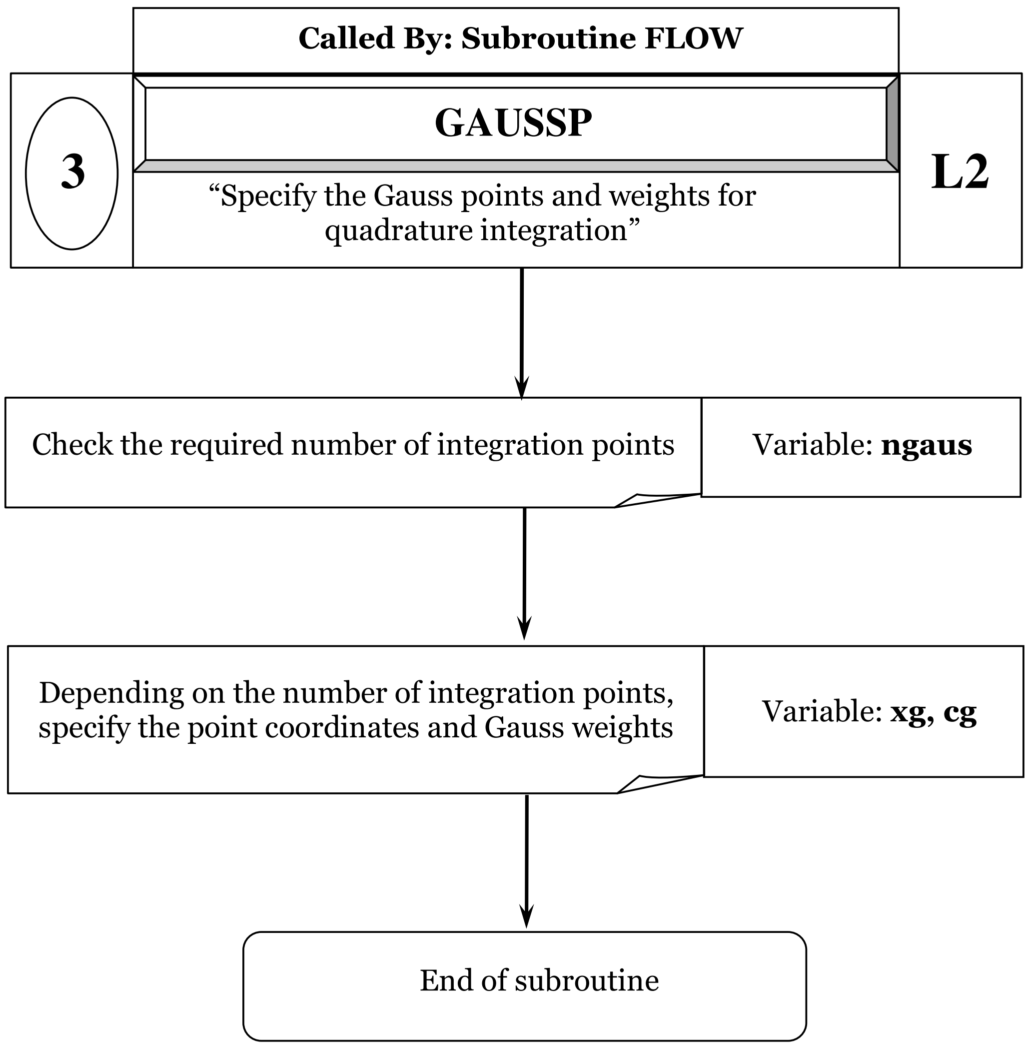
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE FRONT, LEVEL 2



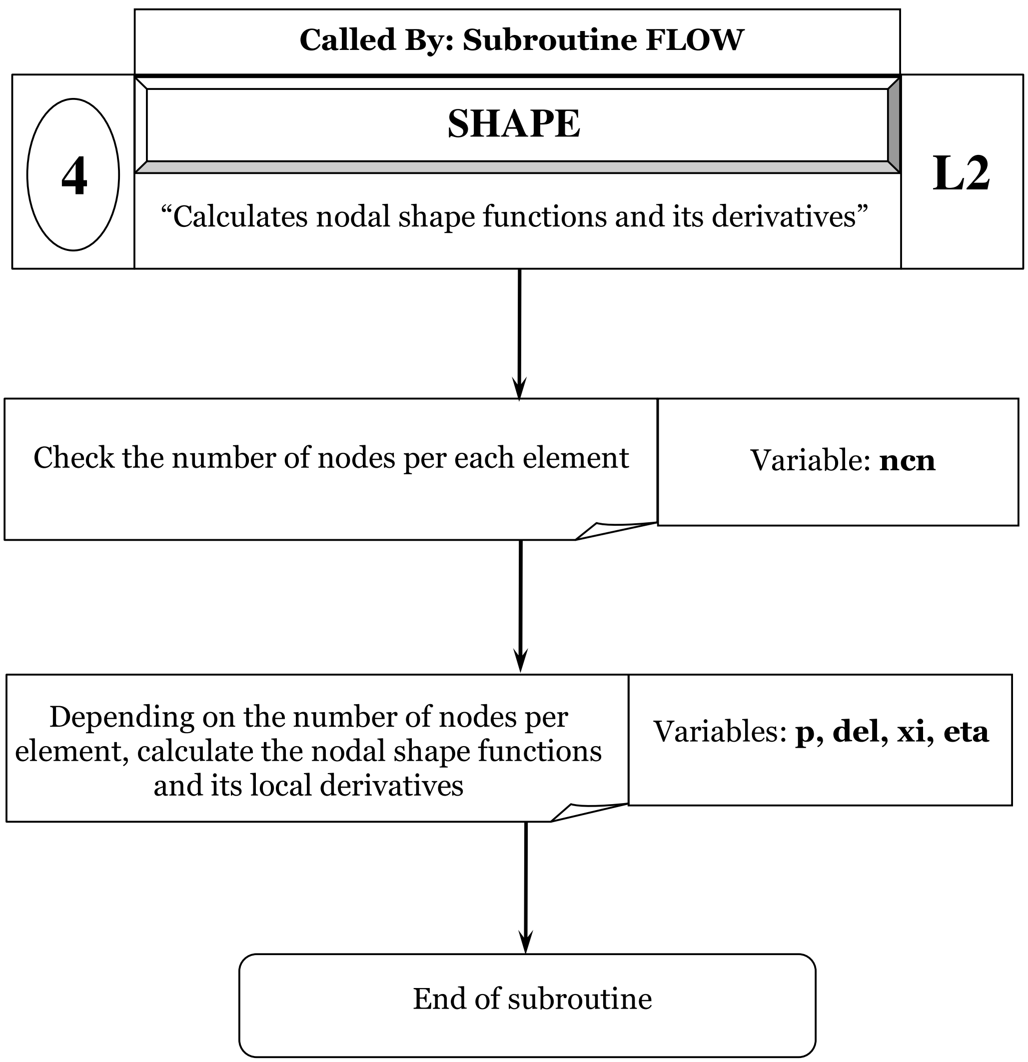
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE GAUSSP, LEVEL 2



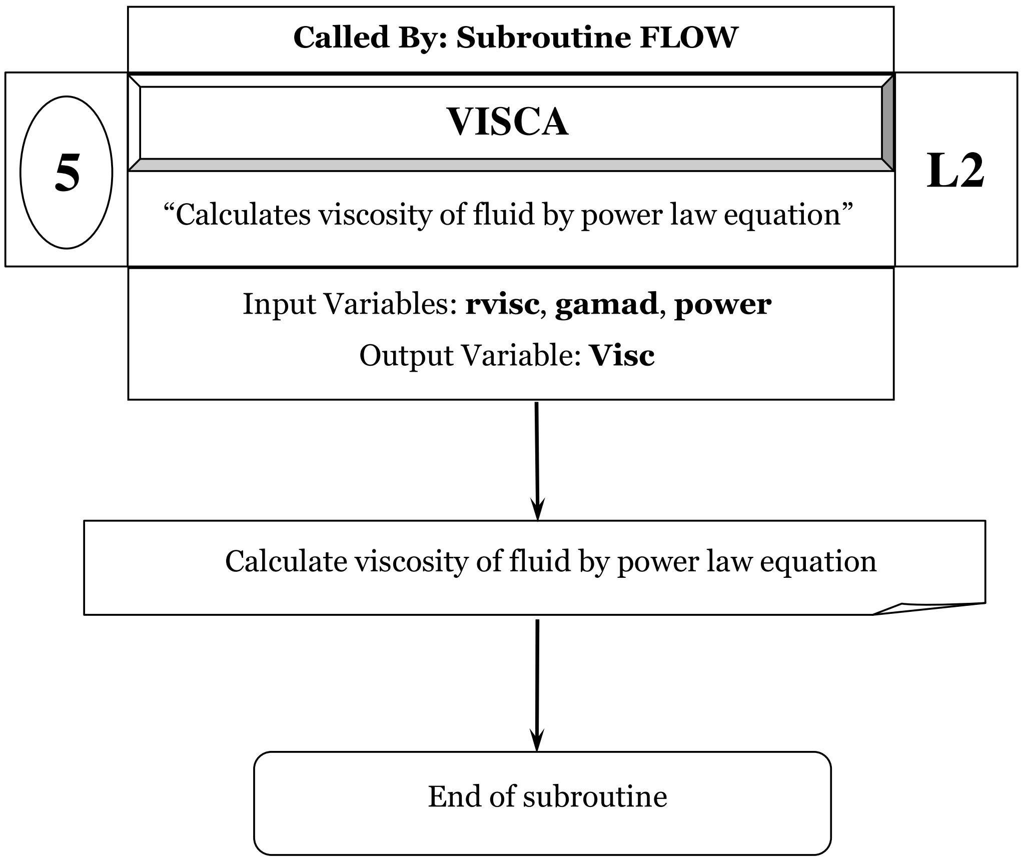
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE SHAPE, LEVEL 2



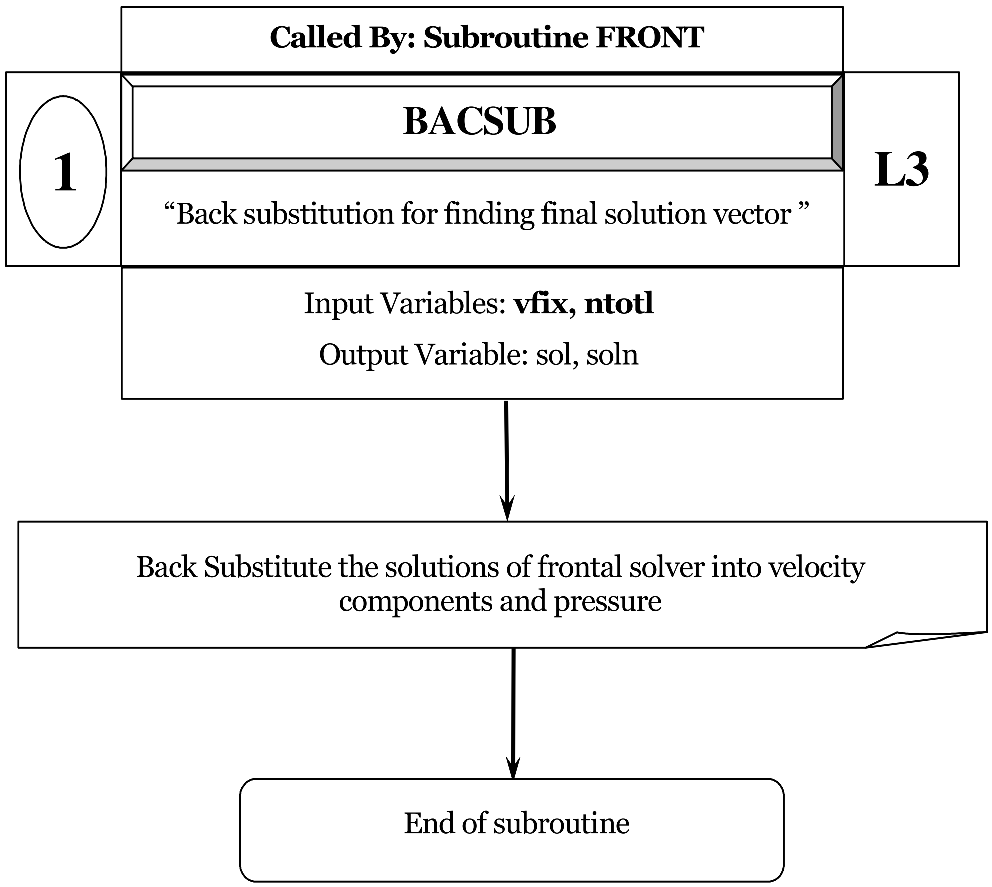
PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE VISCA, LEVEL 2



PROGRAM SUBROUTINES - INTERACTIONS

SUBROUTINE BACSUB, LEVEL 3



LINKING OF FREE AND POROUS FLOW EQUATIONS

The essence of the computational algorithm **CFDYNAMICS** lies in successful linking of the Stokes and the Darcy equations at the interface. In the stiffness matrix of the free flow elements present on the free/porous interface, the Stokes terms corresponding to the interfacial nodes are replaced by the appropriate form of the discretised Darcy components and vice-versa at the porous/free interface. This can be illustrated in Figure 1.

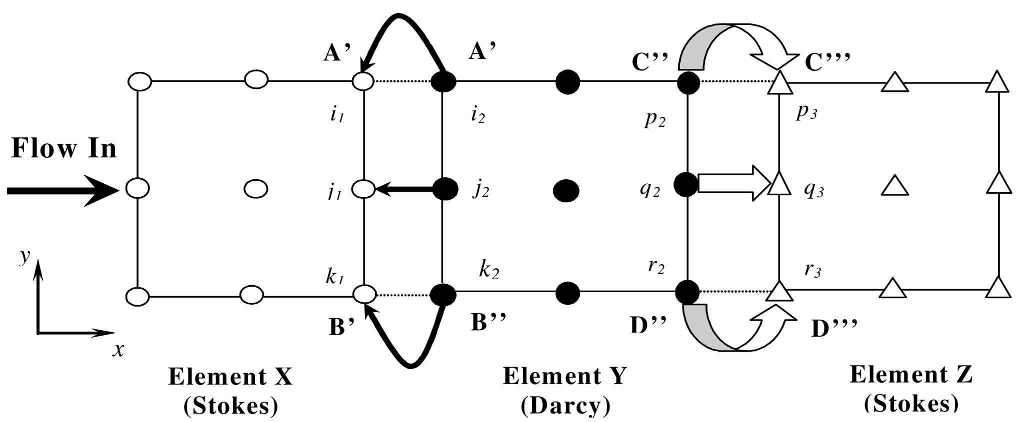


Figure 1: Schematic representation of linking of Stokes and Darcy equations

Figure 1 shows three different elements X, Y and Z connected in series. Element X is in Stokes flow region connected to element Y in Darcy region. Element Y is further connected to element Z in Stokes flow region on the right hand side. This system is a typical representation of coupled free-porous-free flow system, which are the actual flow phenomenon in pleated cartridge filters operated in dead-end filtration mode. The boundary A'B' of the element X coincides with boundary A''B'' of element B at the interface. Similarly, the boundary A''B'' of the element Y coincides with boundary A'''B''' of element Z at the interface.

i_1, j_1, k_1 are the nodes of Stokes flow element X on the interface A'B'-A''B'' whereas i_2, j_2 and k_2 are the nodes of the Darcy element Y on the interface A'B'-A''B''. In the assembly of stiffness matrices, the terms of nodes i_1, j_1, and k_1 in the stiffness matrix of Stokes element X are replaced by the terms of corresponding nodes i_2, j_2 and k_2 from the stiffness matrix of the neighbouring element Y which is in Darcy region.