Guidelines for Solving Navier-Stokes Problems with GLS3D

R.C. Berger and Joseph H. Schmidt [1]

September 21, 2000

[1] USAE Waterways Experiment Station

Contents

List of Tables

List of Figures

Abstract

Guidelines are presented for using the US Army Corps of Engineers (USACE) GLS3D modeling software to model Navier-Stokes problems. GLS3D is can be used in conjunction with the Department of Defense Groundwater Modeling System (GMS) and the examples in this manual did use this system. Other pre- and post-processors are available for grid generation and visualization and can be used with GLS3D with some modification of the files.

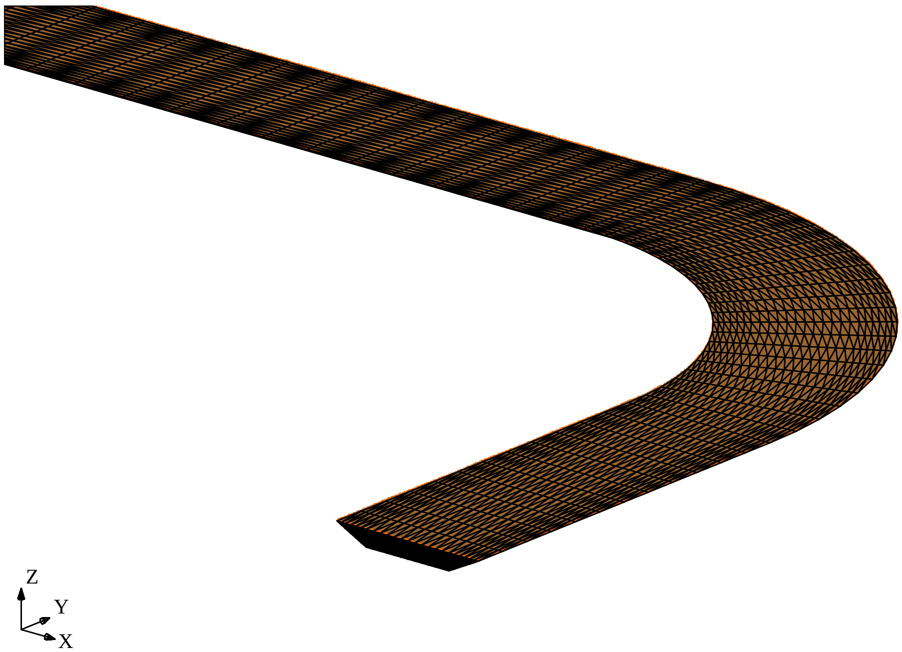
Chapter 1

Introduction

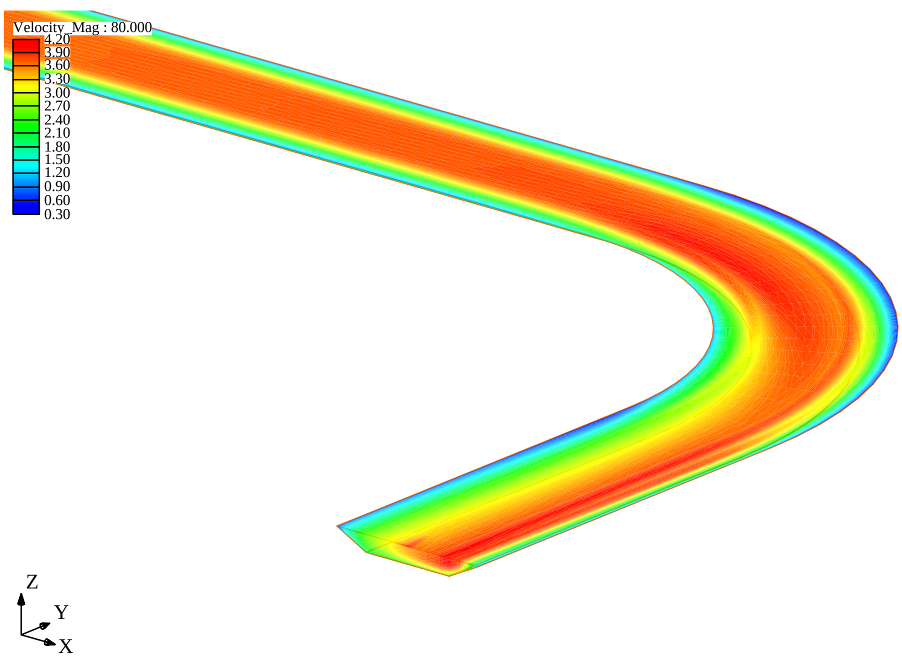
The GLS3D Model (Galerkin Least-Squares) is a software package that is combined with the ADH software ([8]) and so together can describe both saturated and unsaturated groundwater, overland flow, and 3D Navier-Stokes problems. This particular manual contains descriptions for the Navier-Stokes solutions. The model was designed to work in conjunction with the DoD Groundwater Modeling System (GMS). The GMS ([4]) is a modeling package for building models, running simulations, and visualizing results. For further information regarding the GMS, contact the USACE Waterways Experiment Station or visit the website at http://chlnet.wes.army.mil/software/.

A general example of the use of GLS3D for bendway flow is now shown. Flow in a bendway produces an interesting deflection of the water currents that is responsible for the sinuous nature of river systems. The flow approaching a bend has a vertical velocity distribution that is slow near the bed and faster near the surface. This is a result of the bed drag. At the bend the surface surrents will then be thrown to the outside of the bend and the pressure gradients will push the near bed currents more toward the inside of the bend. This helical flow in the lateral plane results in the erosion of the outside and downstream boundary of the bend and buid up of material on the inside of the bend. As an example a trapezoidal channel with a 90° bend is shown in part *a* of Fig. 1.1. Flow is approaching the bend from the left and exiting toward the bottom of the page. Part *b* of Fig. 1.1 shows the resulting velocity magnitudes as contours. This figure demonstrates the helical nature of the flow with surface velocity magnitudes that are high near the inside of the approach to the bend but veer to the outside as flow leaves the bend. This model clearly demonstrates the capabilities of GLS3D.

Only three files are needed to run a model in GLS3D. These files are the mesh file, the boundary conditions file, and the hot start file. The mesh file must be constructed first and can be generated directly with the GMS. Once a mesh file has been constructed, the boundary conditions for the problem and operating parameters for GLS3D must be specified in the boundary conditions file. Many of the operational parameters and boundary conditions can be specified through



(a) Bendway Trapezoidal Channel



(b) Velocity Magnitude

Figure 1.1: Movement of a solute plume without dispersion or diffusion.

the use of the GMS, using it's tools to select boundary nodes and faces as well as entire sides of a domain. The hot start file is then generated to establish the initial conditions of the problem.

Once the three required files have been created, the GLS3D model is run with the command

`adh` *filename*

where *filename* is the root of the model's filenames, i.e. for a model named `layer` the following three files would be required `layer.3dm`, `layer.hot` and `layer.bc`. All three files must have the same *filename* as their root followed by one of three suffixes. After the model is run, the GMS can be used to visualize the results. Visualization capabilities of the GMS include contours of the flow head and chemical concentrations. With the GMS it is also possible to cut cross sections and overlay the mesh, allowing the user to view things such as grid dependencies in the solutions.

1.1 Mesh Files

Mesh files can be generated quickly and efficiently using the GMS. Mesh files, *filename*`.3dm`, follow the GMS 3-D mesh format. GLS3D uses only tetrahedral elements, although the GMS supports tetrahedral, pyramid, wedge and hexahedral elements. Fig. 1.2 shows a tetrahedral mesh generated with the GMS representing trapezoidal bendway channel.

GLS3D uses the mesh files generated with the GMS directly, without any modifications. GMS mesh files are designated with a .3dm extension. Simple rectilinear domains are convenient starting points for more complicated models. Instructions for producing example meshes are given in Sec. 2. For full details on how to produce mesh files using the GMS, refer to the GMS Reference Manual and tutorials.

1.2 Boundary Condition Files

The boundary condition file contains many pieces of information necessary to perform simulations with GLS3D including specified pressure and velocity boundary conditions; Neumann and outflow boundary conditions; time step data; output controls; adaptivity controls; error tolerances; and maximum levels of mesh refinement. GLS3D accepts two types of boundary conditions: specified pressure or velocity (Dirichlet), and flux (Neumann). The code also has the capability of sending and receiving boundary information to and from outside programs.

Error tolerances are specified in the boundary condition file. Mesh refinement is governed by the specified error tolerance in the boundary condition file.

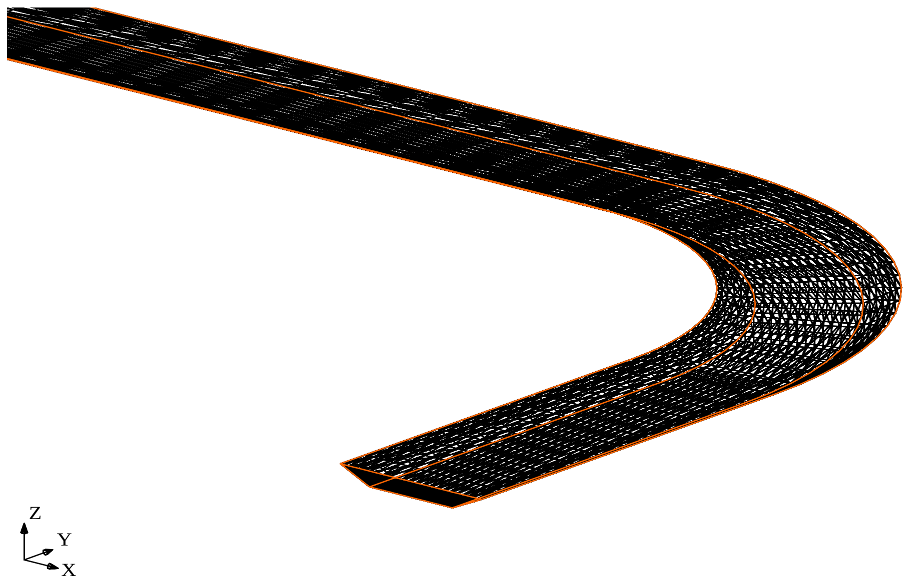


Figure 1.2: Channel tetrahedral mesh.

In addition, the maximum levels of refinement must be specified by material type, giving the user additional control over the mesh adaptation.

A boundary condition file, *filename*`.bc`, for the GLS3D code contains a series of one line control cards. Cards are single line entries and cannot be wrapped across lines. The cards fall into eight basic categories: operational parameters, iteration parameters, material properties, boundary strings, solution controls/boundary conditions, time controls, output controls, and series. Operation Parameters control the operation of the code, the reserved memory space, type of problem being modeled, and the solver preconditioning arrangement. Iteration parameters control the iterative methods employed by the model. Material Properties define the flow and transport constants for each material in the model. Boundary conditions are set using string array cards defining the interior and surface boundaries of the problem, including node and face boundaries. Solution Controls specify the initial/boundary conditions. Time controls specify the time steps used to run the model. Output controls define the times at which the output is printed and Series cards are used to define various parameters. The different cards and their categories are shown in Table 1.1.

Each card consists of at least one character string identifying the type of card. It may then contain further character fields and/or numeric data fields. There are two important points to note about this file. First, the leading 6 columns are reserved for character field keywords ONLY. All numeric data MUST start in column 7 or later. As an example, consider the line of input

Operation Parameters (Sec. 1.2.1)	`OP NS`	Navier-Stokes
	`OP INC`	Incremental Memory
	`OP TRN`	Transport Quantities
	`OP BLK`	Blocks per processor
	`OP MV`	Moving Grid Option
	`OP PRE`	Preconditioner
	`OP THT`	θ time weighting
Iteration Parameters (Sec. 1.2.2)	`IP NIT`	Non-Linear Iterations
	`IP NTL`	Non-Linear Tolerance
	`IP MIT`	Maximum Linear Iterations
	`IP FNI`	Forced Non-Linear Iterations
	`IP FLI`	Forced Linear Iterations
Materials (Sec. 1.2.3)	`MP EV`	Eddy Viscosity
	`MP MU`	Viscosity
	`MP G`	Gravitational Acceleration
	`MP RHO`	Density
	`MP NCE`	Materials to make non-conservative calculations
	`MP TMN`	Momentum Stabilization Coefficient
	`MP TCN`	Continuity Stabilization Coefficient
	`MP U0`	Reference Velocity
	`MP L0`	Reference Length
	`MP R0`	Reference Density
	`MP MU0`	Reference Viscosity

Boundary Strings (Sec. 1.2.4)	`NDS`	Node String
	`FCS`	Face String
	`BNS`	Border Node String
	`GNS`	Ghost Node String
Solution Controls (Sec. 1.2.5)	`DB VEL`	Dirichlet - Velocity
	`DB PRS`	Dirichlet - Pressure
	`NB VEL`	Neumann - Flow
	`OB VEL`	Outflow Boundary
	`DB EXT`	Dirichlet - Pressure (externally supplied)
	`OB EXT`	Outflow Boundary (externally supplied)
	`DB FRS`	Free-surface pressure boundary
	`DB MVS`	Displacement boundary
Time Control (Sec. 1.2.6)	`TC T0`	Start Time
	`TC IDT`	Time Series
	`TC TF`	Final Time
	`TC ADP`	Adaptive Time Control
	`TC NDP`	Non-Adaptive Time Control
Series	`XY1`	X-Y Series Cards
	`XYT`	Curve Fit Tolerance
Output Controls (Sec. 1.2.7)	`OC`	Output Control Series
	`END`	Signifies End of BC file

Table 1.1: Cards

```
MP␣EV␣␣1␣2.6␣2.6␣2.6␣0.0␣0.0␣0.0
```

This input would result in two character fields being read. One would have the value "MP" and the other, the value "EV" as intended. The following incorrect line

```
MP␣EV1␣2.6␣2.6␣2.6␣0.0␣0.0␣0.0
```

would result in fields containing the values "MP" and "EV1". It is important to note that the parser cannot handle lines more than 150 characters wide. An input file template is provided in Table A.8.

1.2.1 Operation Parameters

Each Operation Parameter card consists of two character fields and may contain one numeric field. Operational parameter cards are identified by an "OP" in the first field. `OP` cards control the type of system is being modeled. An `OP NS` card is used to specify Navier-Stokes flow modeling.

The code allocates memory as needed during the run to store the additional elements and nodes created during the refinement process. The memory is allocated in blocks. The size of the block is specified by the user on the `OP INC` card. If the specified number is too small, the program will continually seek additional memory, slowing the run time of the program. Alternately, if the number is too large, the program will require excess memory not needed to run the code.

The preconditioner for the linear solver and the manner in which it is implemented are specified by the `OP PRE` and the `OP BLK` cards. The first card specifies the preconditioner used. The integer can be 0,1,2,or 3 for various preconditioning schemes. The second card defines how many blocks per processor are to be used in the preprocessing. These are subdividing the problem to perform a direct solve on each block and the total group of all blocks can be used to perform a coarse grid solve. Which of these options is used is specified by the `OP PRE` choice. In this case, the 2 indicates two-level Additive Schwarz preconditioning using 10 blocks per processor. The
tt OP MV allows the code to make moving grid calculations. Boundary conditions will then need be specified to dictate how nodes are moved or if the nodes are free-surface. At the completion of the Navier-Stokes calculation these boundary nodes will have moved to their new locations and a subsequent calculation moves the interior nodes.

```
OP NS
OP INC 2000
OP PRE 2
OP BLK 10
OP MV
```

After finding a flow solution, an associated transport problem can be solved. The number of transported quantities (*#transport_equations*) is given on an `OP TRN` card. The `OP TRN` card is a required input card. If the problem does not involve transport, zero (0) quantities are specified on the `OP TRN` card. In addition, if transport equations are not being modeled, no transport properties or boundary conditions may be specified. An error message will be displayed if transport properties are included in the input file but no transport quantities have been specified. The format of the `OP` cards is found in Table A.1. The following card specifies one transported quantity. Presently, no transported items are supported in the Navier-Stokes solver.

```
OP TRN 0
```

1.2.2 Iteration Parameters

There are three iteration parameter cards that must be specified by the user. Iteration parameter cards are identified by an "IP" in the first field. An `IP NIT` card specifies the maximum number of non-linear iterations. An `IP NTL` card specifies the convergence tolerance for the non-linear iterations. An `IP MIT` card specifies the maximum number of linear iterations for each non-linear iteration. At the maximum number of iterations specified on the `IP NIT` or `IP MIT` cards, if the convergence is not sufficient the calculations will fail. Another option is available for each of these cards. These function like these two cards but if the maximum iteration count is reached the calculations are accepted and GLS3D proceeds. The `IP FNI` card then is for the non-linear iteration maximum and `IP FLI` is for the linear iteration maximum. The format of these cards is specified in Table A.2.

```
IP NIT 5
IP NTL 1.0E-3
IP MIT 20000
```

1.2.3 Materials

Material Property cards are identified by the designation "MP". There will be a set of cards for each material type in the model. Each group contains a set of refinement control cards. Refinement may be adjusted to independently follow the error in the flowfield and transport equations. For two of these cards, the first two fields contain character strings, specifying the type of card (MP) and the specific parameter (EV or NCE as defined below). The third field is an integer field containing the material number to which the values apply, (*mat#*). The remainder of are applied throughout the problem and do not include a material number. These are MU, G, RHO, TMN, TCN, U0, L0, R0, MU0. The formats for these cards are listed in Tables A.3 and A.4.

Flow Parameters

Three cards can be used to specify flow parameters: kinematic eddy viscosity (EV), kinematic viscosity (MU), and density (RHO). The acceleration due to gravity is defined (G). Two parameters associated with the numerical model stabilization are defined in two cards: momentum equation related stabliliza-tion (TMN) and continuity related stabilization (TCN). Other parameters are used to non-dimensionalize the equation set: viscosity (MU0), density (R0), length (L0), and velocity (U0). Most of these parameters are obvious but some explanation of the kinematic eddy viscosity is warranted.

Kinematic Eddy Viscosity, EV The eddy viscosity is representative of the turbulence generated spreading of momentum that is smaller than can be represented by the grid resolution. Kinematic eddy viscosity has units of L^2/T and is related to the flow itself. The kinematic viscosity on the other hand is a fluid property.

The kinematic eddy viscosity is expressed as a tensor in the following form:

$$\begin{matrix} EV_{xx} & EV_{xy} & EV_{xz} \\ EV_{xy} & EV_{yy} & EV_{yz} \\ EV_{xz} & EV_{yz} & EV_{zz} \end{matrix}$$

The six values in the upper right quadrant of the tensor are entered on the `MP EV` card in the following order: EV_{xx}, EV_{yy}, EV_{zz}, EV_{xy}, EV_{xz}, EV_{yz}. If the hydraulic conductivity is independent of the direction of measurement, the formation is termed *isotropic*. In the isotropic case, $EV_{xx} = EV_{yy} = EV_{zz}$ and $EV_{xy} = EV_{xz} = EV_{yz} = 0$. The `MP EV` card is required. Or all terms in the tensor can be set equal to 0 and declare the total viscosity through the `MP MU` card.

Mesh Refinement

An `MP ML` card is used to specify the maximum levels of mesh refinement, or the total number of times that an original element may be split within a material type. Refinement can be turned off in a material by specifying zero (0) as the maximum levels of refinement. When refinement is on, the solution error tolerance is given on the "FRT" card. If the solution error on an element exceeds the refine error tolerance given on the "FRT" card, the element is split. Similarly the unrefine tolerance is given on the "FUT" card. When the grid solution error improves the elements are recombined.

```
MP ML 1 5
MP FRT 1 1.0e10
MP FUT 1 1.0e0
```

Different material types can have different levels of refinement. Some experimentation with the error tolerance is usually necessary to gain the desired level of refinement. These tolerances and the maximum number of refinement levels may also be changed as the model advances.

1.2.4 Boundary Strings

For most problems, boundary data includes Dirichlet data on or interior to the domain and flux data (Neumann) through a region of the domain surface. Each of these boundary conditions is applied to a "string" of element nodes or faces. Each component of a string is input on a card that specifies the string type and node or element face it contains. There are three types of boundary strings: node, face, and edge. Complete strings are input on multiple cards with one node, face or edge per card. Cards may be input in any order and cards for different node strings may be interspersed. See Table A.5 for details on the format of these cards.

Node Strings

Dirichlet data are specified on node strings. These can be made up of boundary and/or interior nodes as the problem requires. The identifier for this card is `NDS`. On each card, the node number is followed by a string number (*string#*).

Face Strings

Neumann data are specified across face strings. The identifier for this card is `FCS`. The card lists the identifier, element number, element face number (*face#*),and then the string number. Currently, mixed or Robin boundary conditions are not supported.

```
NDS 1037 2
FCS 1008 1 1
```

1.2.5 Solution Controls

Solution development is controlled through the specification of the initial and boundary conditions and the time step parameters. Dirichlet boundary conditions are specified on a `DB` card, and Neumann data on a `NB` card. These cards contain either a `VEL` or `PRS` specifier in the second field to signify if they apply to the flow or pressure declarations.

The third field of the `DB` and `NB` cards specify the node or face string, respectively. The forth, fifth and sixth fields of the `DB VEL` card specifies the x-, y-, and z-component velocity time- boundary series. The forth field specifies the time-boundary series for the `DB PRS` card.

The forth field of the `NB` card contains the time-boundary series for the velocity flow through a boundary. The fifth field contains the drag coefficient. This coefficient is used in the surface drag relationship.

If the `OP MV` moving grid option is specified then there are additional cards that should be included. These include `DB FRS` and `DB MVS` cards. The first indicates a nodes string that moves to satisfy a stated pressure. So the third and forth fields will contain the string number followed by the series number for the pressure time history. The `DB MVS` card allows the user to actually move certain nodes in time. The third and forth fields indicate the string number and the time-boundary series, respectively.

$$\tau = \rho C_D \left|\left|\mathbf{u}\right|\right| \mathbf{u} \tag{1.1}$$

Values for the Dirichlet or Neumann boundary conditions are linearly interpolated by GLS3D; therefore, it is important to define the series with a linear interpolation in mind.

For outflow boundaries, a `OB` card is used. The second field of an `OB` card references flow (VEL) or transport (TRN). The third field references a boundary string, representing the outflow face, and the fourth field contains the ID number of the transport quanitity. See Table A.6 for a complete description of these cards.

```
DB VEL 1 1 2 3
DB PRS 2 1
NB VEL 3 2 0.06
OB VEL 4
DB FRS 5 1
DB MVS 6 2
```

1.2.6 Time Controls

Evolution of the solution is determined by a group of five cards with the Time Control specifier `TC`. The start time is specified on a `TC TO` card. The `TC IDT` references a time series in the third field that will control the time steps. The final time, at which the run will terminate, is specified on a `TC TF` card. The final time does not have to correspond with the largest value in the time series.

For adaptive time control, a `TC ADP` card is used. Adaptive time control allows the model to refine the time steps from those specified by the user in the time series. The model may choose finer time steps than those specified but will not adapt the time steps to a larger time step value.

A `TC NDP` card is used for non-adaptive time steps. In this case, the code will only refine the time steps for stability purposes, not for accuracy. Time steps will only be reduced if the model fails to converge for the current time step.

```
TC T0 0.0
TC IDT 5
TC TF
TC ADP
TC NDP
```

1.2.7 Output Control

An `OC` card causes the solution to be printed at startup and at each specified time step. The `OC` card references a series that controls the output data. These are output as data set files.

An END statement is used at the end of the boundary conditions file. The code will read the boundary conditions file through to the `END` statement. Any information in the boundary conditions file after the `END` statement will not be read as input to the run. Reference Table A.7 for a full description of the above cards.

```
OC 3
END
```

1.3 Hot Start File

The hot start file, *filename*`.hot` is used to specify two types of model data: initial conditions and scale factors. Initial conditions consist of pressures and velocity components. If the moving grid option is used then the intial displacement of the grid and the grid speed will need to be specified at all the nodes.

Field data is often available and used as a starting point for many problems. The field data can be specified as the initial conditions used in the flow and transport equations for a specific problem. These data are specified in the "hot-start" file. The GMS provides a simple interface for entering field data. This data is entered into a scatter point data file and interpolated to the problem mesh.

A hot start file is a required part of a model; however, it may be a zero-length file (empty). For simple problems a cold start simulation (initial conditions of `0.0`) may be sufficient. Cold start problems in GLS3D are handled by leaving the hot-start file blank. If data types are not specified default values will be supplied. A set of predefined dataset names is used to declare data types, shown below.

```
p or P        Initial Pressure
iv or IV      Initial Velocity
id or ID      Initial Displacement
igs or IGS    Initial Grid Speed
```

The datasets used for the hot-start file can be generated with the GMS. A standard GMS 3-D mesh data set format is used in the hot start file. Multiple data sets are exported from the GMS and copied to the hot start file in any order. If a dataset is not supplied for one or more of the parameters, GLS3D will assign default values to all the cells for that parameter. Default initial conditions on the pressure and velocity components assume a value of `0.0`.

Chapter 2

Sample Problem

This chapter develops setting up a problem using GLS3D by demonstrating the development of the boundary condion file for a simple cube, Fig. 2.1, composed of tetrahedral elements. The problem solved is for flow imposed on the upstream face with a velocity of 1.0 feet per second in the positive x direction. Flow exits directly downstream. The sidewalls have an imposed drag.

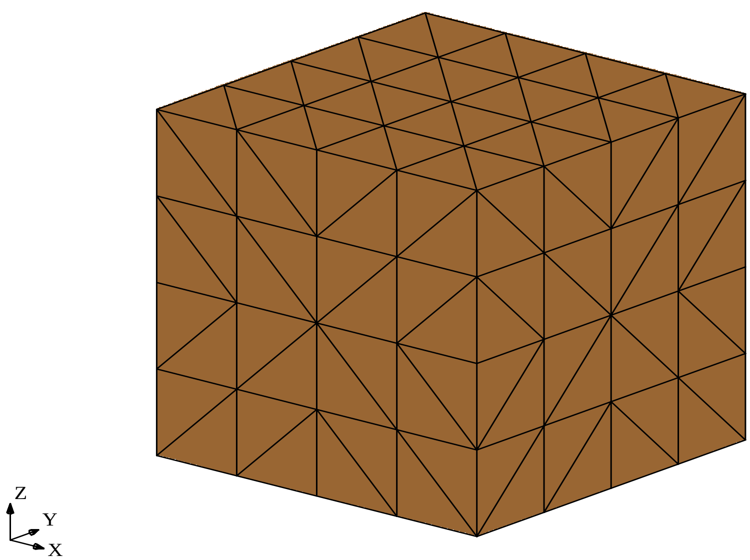


Figure 2.1: Cube Flow Domain.

2.1 Geometry File

Fig. 2.2 shows a tetrahedral mesh generated with the GMS representing a rectangular box domain. Portions of the geometry file are shown here.

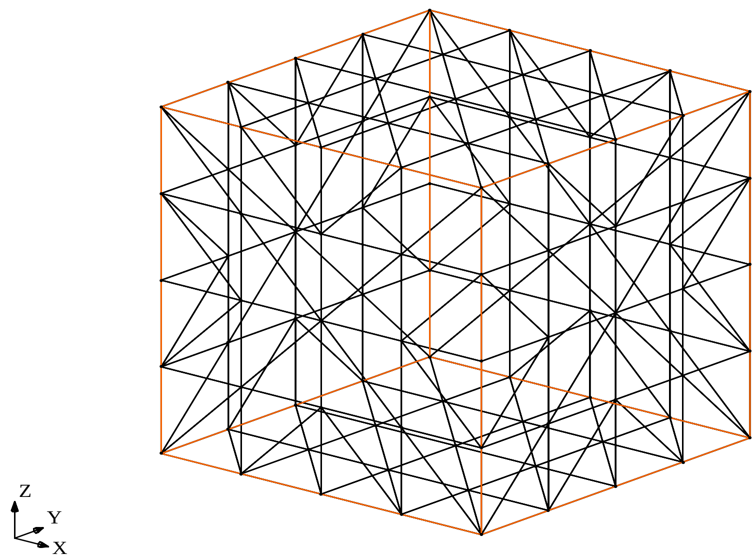


Figure 2.2: Flow Domain Showing the Boundary Grid.

```
MESH3D
E4T     1    26     2    27     1     1
E4T     2    27     2     7     1     1
E4T     3     1    27    26    28     1
E4T     4     1    28    26    29     1
E4T     5     1     7    27     6     1
E4T     6     1     6    27    28     1
E4T     7     3    30    31     2     1
E4T     8    31    30    32     2     1
E4T     9    30    32     2    27     1
E4T    10    31     2    32    26     1
E4T    11    32     2    27    26     1
 .      .     .     .     .     .     .
 .      .     .     .     .     .     .
 .      .     .     .     .     .     .
E4T   377   123   108    96   122     1
E4T   378   123   122    96   102     1
E4T   379   113   109   124    98     1
E4T   380   124   109   107    98     1
E4T   381   124    98   107   122     1
E4T   382   125   107   108   122     1
E4T   383   125   124   107   122     1
E4T   384   125   108   123   122     1
ND      1 1.00000000e+00 0.00000000e+00 0.00000000e+00
ND      2 1.00000000e+00 0.00000000e+00 -2.50000000e-01
ND      3 1.00000000e+00 0.00000000e+00 -5.00000000e-01
ND      4 1.00000000e+00 0.00000000e+00 -7.50000000e-01
ND      5 1.00000000e+00 0.00000000e+00 -1.00000000e+00
ND      6 1.00000000e+00 2.50000000e-01 0.00000000e+00
ND      7 1.00000000e+00 2.50000000e-01 -2.50000000e-01
ND      8 1.00000000e+00 2.50000000e-01 -5.00000000e-01
 .     .       .              .              .
 .     .       .              .              .
 .     .       .              .              .
ND    120 0.00000000e+00 7.50000000e-01 0.00000000e+00
ND    121 0.00000000e+00 1.00000000e+00 -5.00000000e-01
ND    122 0.00000000e+00 5.00000000e-01 -7.50000000e-01
ND    123 0.00000000e+00 7.50000000e-01 -1.00000000e+00
ND    124 0.00000000e+00 2.50000000e-01 -1.00000000e+00
ND    125 0.00000000e+00 5.00000000e-01 -1.00000000e+00
```

The term “MESH3D” starts the file. Each element is “E4T Element # Four Node # Material ID”. This particular example has 384 tetrahedral elements. This list is followed by the list of nodal coordinates. These are “ND Node # X, Y, and Z coordinates".

2.2 Hotstart File

A complete hotstart file is given below. The first part is the initial estimate of pressures. In this case they are all set to $0.00000000e + 00$. Actually, if the pressures were not included in the file they would be assumed to be $0.00000000e + 00$ but in the interest of showing a complete hotstart file these pressures were included in the file. If either pressures and/or velocities are not in the hotstart file their values are assumed to be zero. The initial velocities do have some slightly different values near the upstream face of the cube. These hotstart files can easily be developed from output from previous runs by simply copying the timestep that you want to use from the output file. The "NAME" row then is modified to match the hotstart designation. This is the pressure part of the hotstart file.

```
DATASET
OBJTYPE "mesh3d"
BEGSCL
ND 125
NC 384
NAME p
TS 0  0.00000000e+00
 0.00000000e+00
 0.00000000e+00
 0.00000000e+00
 0.00000000e+00
       .
       .
       .
 0.00000000e+00
 0.00000000e+00
 0.00000000e+00
ENDDS
```

The velocity portion of the hotstart file then is:

```
DATASET
OBJTYPE "mesh3d"
BEGVEC
ND 125
NC 384
NAME iv
TS 0  0.00000000e+00
 9.80371189e-01  7.24992496e-04 -8.75434988e-04
 9.92464260e-01  8.20541110e-04 -4.82678151e-04
 9.92951932e-01  5.95874789e-04  8.02983747e-05
 9.91987646e-01  8.94939353e-04  6.70094774e-04
 9.79720716e-01  9.27319722e-04  3.99363696e-04
 9.92919590e-01  6.46211220e-04 -1.05409554e-03
 1.00099678e+00  5.44002814e-04 -5.90068403e-04
 1.00543751e+00  5.59672376e-04  5.06365478e-05
 1.00254039e+00  5.60845242e-04  5.35764013e-04
 9.90657771e-01  6.64147343e-04  5.23329804e-04
 9.94600043e-01  2.87396643e-05 -8.07358816e-04
 1.00560027e+00 -2.38268306e-04 -5.17897391e-04
 1.01175457e+00 -1.38953678e-04 -3.72662666e-05
 1.00668167e+00 -1.84153423e-04  4.41567443e-04
 9.94517141e-01  1.31039860e-04  6.06299730e-04
 9.93058859e-01 -6.19611483e-04 -6.84240593e-04
 1.00235204e+00 -7.95659981e-04 -3.15630633e-04
 1.00846665e+00 -6.87403248e-04 -8.63308832e-05
 1.00281911e+00 -7.41354099e-04  2.17580243e-04
 9.93085811e-01 -4.34111576e-04  6.10838052e-04
 9.80008380e-01 -4.08766769e-04 -6.22617469e-04
 9.91207547e-01 -5.16831013e-04 -4.83040845e-04
 9.94243780e-01 -4.64964194e-04 -3.76198470e-06
 9.90284042e-01 -5.50017997e-04  3.87572489e-04
 9.79544993e-01 -3.12483789e-04  5.03047908e-04
 0.00000000e+00  0.00000000e+00  0.00000000e+00
 0.00000000e+00  0.00000000e+00  0.00000000e+00
 0.00000000e+00  0.00000000e+00  0.00000000e+00
       .                .                .
       .                .                .
       .                .                .
 0.00000000e+00  0.00000000e+00  0.00000000e+00
 0.00000000e+00  0.00000000e+00  0.00000000e+00
ENDDS
```

2.3 Boundary Condition File

Next is shown the boundary condition file. The first portion of the file contains the "OP", "IP", and "MP" cards. The "OP" cards designate that this is a Navier-Stokes run, that memory is added in blocks of 10, no transport constituents, there are to be 2 blocks per processor, and the preconditioner uses the two-level hybrid approach. The "IP" cards indicate that the maximum number of linear iterations allowed are 2000, the maximum non-linear iterations is 5, and the tolerance on the non-linear iterations is 0.0001. The "MP" cards designate the other properties of the fluid or this particular problem.

```
OP NS
OP INC 10
OP TRN 0
OP THT 1.
OP BLK 2
OP PRE 3
IP MIT 2000
IP NIT 5
IP NTL 0.0001
MP EV  1 0.00  0.00  0.00  0.0  0.0  0.0
MP ML 1 0
MP MU  0.1
MP G   32.2
MP RHO  1.94
MP TMN  0.3
MP TCN 0.3
MP U0  1.0
MP R0  1.94
MP L0  1.0
MP MU0 0.01
```

The "strings" are designated by "NDS" for node strings and "FCS" for face strings. Strings 1,2,3 and 4 are face strings indicating the sidewalls of the cube. String 5 is a face string that includes both the upstream and downstream boundaries. These could have been designated separately.

FCS	384	4	1
FCS	383	4	1
FCS	382	4	1
FCS	380	4	1
FCS	379	4	1
FCS	377	4	1
FCS	376	4	1
FCS	365	2	2
FCS	362	2	3
FCS	361	2	3
FCS	352	2	2
FCS	351	2	2
FCS	350	1	2
FCS	349	1	2
FCS	348	3	2
FCS	345	2	2
FCS	336	3	4
FCS	335	1	4
FCS	334	3	4
FCS	333	3	4
FCS	331	3	4
FCS	330	3	4
FCS	329	3	4
FCS	328	2	3
FCS	326	2	3
FCS	324	4	1
FCS	324	2	3
FCS	323	3	2
FCS	323	2	3
FCS	321	3	3
FCS	319	4	1
FCS	318	2	2
FCS	316	4	1
FCS	315	4	1
FCS	314	4	1
FCS	313	3	4
FCS	311	2	4
FCS	308	4	1
FCS	307	4	1
FCS	306	4	1
FCS	301	3	3
FCS	298	3	3
.	.	.	.
.	.	.	.
.	.	.	.

FCS	94	2	5
FCS	92	2	5
FCS	89	2	5
FCS	88	2	5
FCS	87	2	5
FCS	85	2	5
FCS	84	2	5
FCS	75	2	5
FCS	71	2	5
FCS	69	3	5
FCS	64	3	5
FCS	63	3	5
FCS	62	2	5
FCS	59	3	5
FCS	58	3	5
FCS	52	3	5
FCS	48	2	5
FCS	44	3	5
.	.	.	.
.	.	.	.
.	.	.	.
FCS	384	2	5
FCS	383	3	5
FCS	381	3	5
FCS	379	2	5
FCS	378	3	5
FCS	376	3	5
FCS	375	3	5
FCS	371	3	5
FCS	366	3	5
FCS	365	3	5
FCS	363	3	5
FCS	359	3	5
FCS	358	3	5
FCS	357	3	5
FCS	355	3	5
FCS	354	3	5
FCS	352	3	5
FCS	350	3	5
FCS	348	2	5
.	.	.	.
.	.	.	.
.	.	.	.

String 6 is a node string that includes all the nodes on the upstream face. String 7 is a single node on the downstream face.

```
NDS 110  6
NDS  120  6
NDS  119  6
NDS 118  6
NDS 116  6
NDS 106  6
NDS 104  6
NDS 117  6
NDS 100  6
NDS 101  6
NDS 121  6
NDS 103  6
NDS 97  6
NDS 99  6
NDS 115  6
NDS 112  6
NDS 102  6
NDS 122  6
NDS 98 6
NDS 114 6
NDS 111  6
NDS 123 6
NDS 125  6
NDS 124  6
NDS 113  6

NDS  13  7
```

The XY-series are shown next. These can most easily be explained through the boundary condition cards.

```
XY1  1  2  0  0  0  0
0.0    0.0
200.0  0.0

XY1  2  2  0  0  0  0
0.0    1.0
200.0  1.0

XY1  3  10  0  0  0  0
1.0   0.0
2.0   0.0
3.0   0.0
4.0   0.0
5.0   0.0
6.0   0.0
7.0   0.0
8.0   0.0
9.0   0.0
10.0  0.0

XY1  4  2  0  0  0  0
0.0   1.
200.0 1.
```

The first "DB" card designates that the upstream nodes (string 6) has Dirichlet boundary condition on velocity. The x-component is represented by series 3 which is a velcoity of 1.0 . The y- and z-components are both designated by string 1, which has a value of 0.0 . A pressure Dirichlet boundary condition is enforced at the single node on the downstream face (string 7). "Outflow" boundary conditions are enforced on string 5. String 5 is both the upstream and downstream faces. Even though the upstream face has no outflow we use this boundary condition to weakly enforce the pressure boundary condition on each face. The "NB VEL" boundaries are on each sidewall face. The series number indicates that no flow is allowed out these boundaries. The drag coefficient is 0.06 .

```
DB VEL  6  2  1  1


DB PRS  7  1


OB VEL  5

NB VEL  1  1 0.06
NB VEL  2  1 0.06
NB VEL  3  1 0.06
NB VEL  4  1 0.06

OC  3
```

The "OC" card designates series 3 to contain the list of timesteps to be output. Series 3 requests that every second is to be output through time 10.0. "TC T0" and "TC TF" designate the starting and ending times to be 0.0 and 3.0, respectively. The timestep size is setup by the "TC IDT" card. The timestep size is set by series 4, in this case. Series 4 then indicates that a time step of 1.0 is to be used between times 0.0 and 200.0. "TC NDP" precludes time adaptation.

```
TC T0  0.0
TC IDT  4
TC TF 3.
TC NDP
END
```

2.4 Running GLS3D

GLS3D is run after a setup program called "PRE_ADH". PRE_ADH reads in the input files and checks that they are somewhat consistent. It also checks the geometry to make sure that the elements are properly formed. The actual command to run PRE_ADH on this sample problem is:

```
pre_adh sample
```

At this point the program itself can be run. Depending upon the flags set during compilation the program can be run with or without "MPI". If the compiler options do not include "-D_MESSG -D_MPI" then running GLS3D is:

```
adh sample
```

This will create files called "sample_vel.dat" and "sample_prs.dat", for velocities and pressures, respectively. These can be directly imported into GMS.

If instead the compiler flags are set to run MPI is as follows:

```
mpirun -np 2 adh sample
```

In this case, we have requested two processors. The output from this particular example command is files:

```
sample_prs.pdat0
sample_vel.pdat0
sample_prs.pdat1
sample_vel.pdat1
```

If we had run the moving grid option,then there would be additional files for the displacement and grid speed:

```
sample_dpl.pdat0
sample_gsp.pdat0
sample_dpl.pdat1
sample_gsp.pdat1
```

There are two files produced per processor. In this case, we requested two processors (processors 0 and 1). Our output files contain the portions of the results contained by that processor. All of the "sample_prs.pdat*" files must be combined into a single file "sample_prs.pdat". Similarly, the "sample_vel.pdat*" files must be combined into "sample_vel.pdat". These two files can then be turned into the "sample_prs.dat" and "sample_vel.dat" files by running "ADH_GMS". A similar operation can be conducted for displacement and grid speed when using the moving grid option.

```
adh_gms sample_prs
adh_gms sample_vel
```

These results are shown in the next two figures. Fig. 2.3 shows velocity results and Fig. 2.4 shows pressures at time 3.0 seconds.

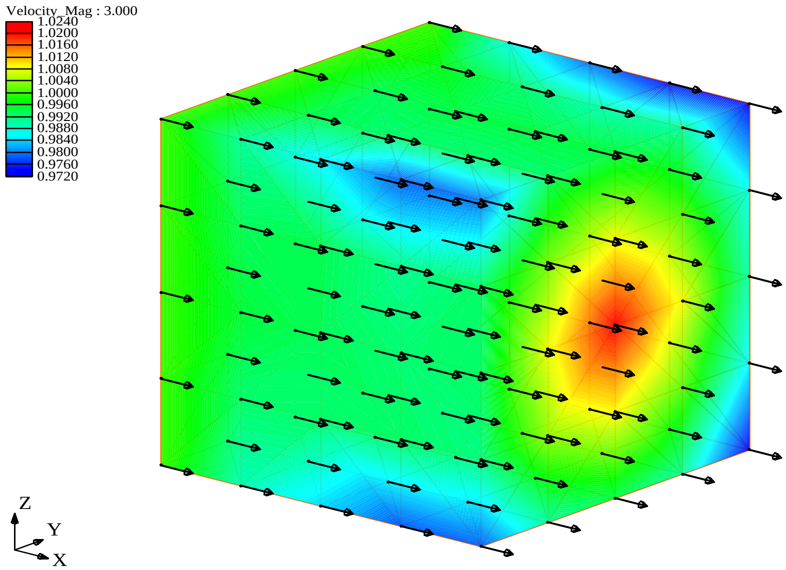


Figure 2.3: Velocity Results.

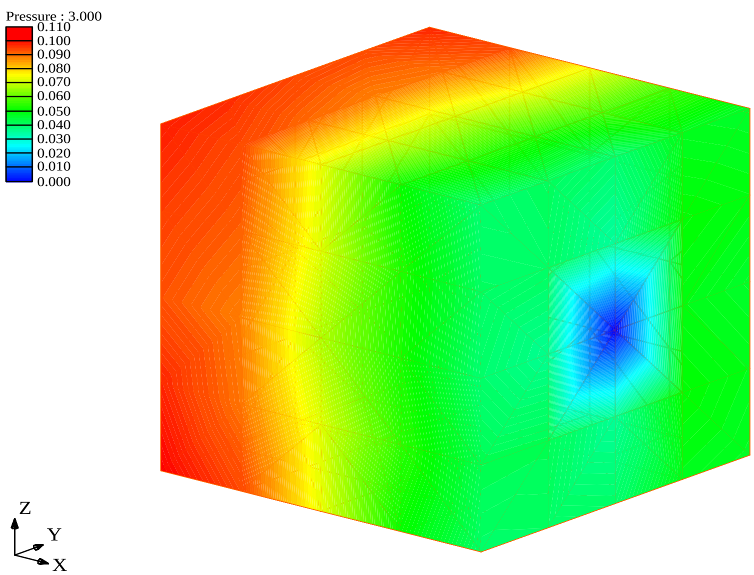


Figure 2.4: Pressure Results.

Chapter 3

Galerkin Least-Squares Method, Navier–Stokes Equations

The Navier–Stokes equations in terms of velocity $u\,(x,t)$ and pressure $p\,(x,t)$, can be written as

$$\rho\left(\frac{\partial u}{\partial t}+u\cdot\nabla u\right)-\nabla\cdot\sigma = 0 \tag{3.1}$$

$$\nabla\cdot u = 0 \tag{3.2}$$

where

$$\sigma = -pI+\tau, \tag{3.3}$$

$$\tau = 2\mu D, \tag{3.4}$$

$$D_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right),\ \text{and} \tag{3.5}$$

$$\mu = \text{kinematic velocity.} \tag{3.6}$$

These equations are applicable within the domain $\Omega\subset\mathcal{R}^3$ with the following boundary conditions:

$$u = \hat{u}\ \text{on}\ \Gamma_g \tag{3.7}$$

$$\sigma = \hat{\sigma}\ \text{on}\ \Gamma_h \tag{3.8}$$

$$u\cdot n = h\ \text{on}\ \Gamma_h. \tag{3.9}$$

$$u=\hat{u}\ \text{on}\ \Gamma_g \tag{3.10}$$

$$\left.\begin{array}{rcl}\sigma & = & \hat{\sigma} \\ u \cdot n & = & h\end{array}\right\} \text{ on } \Gamma_h. \tag{3.11}$$

The boundary of Ω is denoted as Γ, and n is the outward normal. Γ_g and Γ_h represent non–overlapping subregions of Γ such that

$$\Gamma = \overline{\Gamma_g \cup \Gamma_h}.$$

The structure of the three–dimensional elements in ADH is for tetrahedras with linear interpolation of the dependent variables. In this case, this would yield linear velocities and linear pressures. This is, in fact, the consistency or LBB condition ([7], [1], [3]) and result in an oscillatory pressure field for a Galerkin approximation. An approach that is not restricted by the LBB condition for the Stokes problem was introduced by Hughes, et. al. [6] Tezduyar, et. al. [9] developed a generalization for Navier–Stokes problems. The Galerkin least squares (GLS) used here is patterned after that of Behr and Tezduyar [2]. This results in a weighting function that has the form of the variation of the momentum equation and also the τ_{cont}– term proposed by Franca and Hughes [5] that provides stability at high Reynolds numbers.

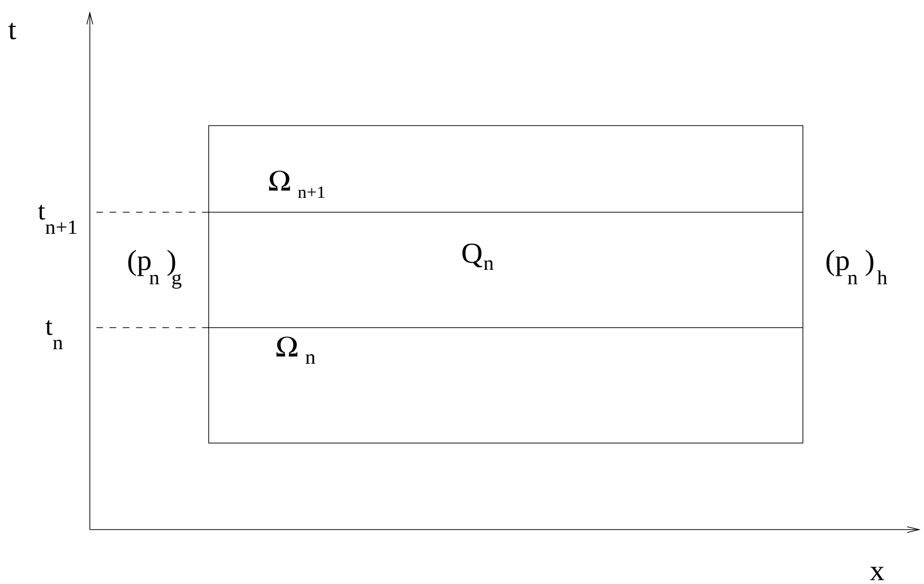


Figure 3.1: Channel tetrahedral mesh.

This formulation is applied by dividing the overall time interval $(0, T)$ into subintervals $I_n = (t_n, t_{n+1})$, where t_0 is the initial time and t_N is the final time (see Figure 3.1). Ω_n is the spatial domain and Γ_n is the spatial boundary at time t_n. The space–time slab Q_n is the domain enclosed by Ω_n, Ω_{n+1}, $(p_n)_g$, and $(p_n)_h$. For each space–time slab, the following finite element interpolation function spaces are defined for the velocity and pressure [2]:

$$\left(\xi_u^h\right)_n = \left\{u^h \mid u^h \in \left[H^{1,h}\left(Q_n\right)\right]^{n_{sd}}, u^h = \hat{u}^h \text{ on } \left(p_n\right)_g\right\}, \qquad (3.12)$$

$$\left(\nu_u^h\right)_n = \left\{u^h \mid u^h \in \left[H^{1,h}\left(Q_n\right)\right]^{n_{sd}}, u^h = 0 \text{ on } \left(p_n\right)_g\right\}, \qquad (3.13)$$

$$\left(\xi_p^h\right)_n = \left(\nu_p^h\right)_n = \left\{p^h \mid p^h \in H^{1,h}\left(Q_n\right)\right\}. \qquad (3.14)$$

where n_{sd} is the number of spatial dimensions. We are interested in steady–state problems at this point, and use 0^{th} order polynomials in time. The interpolation functions are continuous in space but discontinuous in time.

The space–time formulation then is [2]:

Given $\left(u^h\right)_n^-$, find $u^h \in \left(\xi_u^h\right)_n$, and $p^h \in \left(\xi_p^h\right)_n$ such that $\forall w^h \in \left(\nu_u^h\right)_n$, $\forall q^h \in \left(\nu_p^h\right)_n$:

$$\begin{aligned}
\int_{Q_n} w^h &\cdot \rho\left(\frac{\partial u^h}{\partial t} + u^h \cdot \nabla u^h\right) dQ + \int_{Q_n} \nabla w^h : \sigma\left(p^h, u^h\right) dQ - \int_{Q_n} \nabla q^h \cdot u^h \, dQ \\
&+ \int_{\Omega_n} \left(w^h\right)_n^+ \cdot \rho\left(\left(u^h\right)_n^+ - \left(u^h\right)_n^-\right) d\Omega \\
&+ \sum_{e=1}^{(n_{el})_n} \int_{Q_n^e} \tau_{mom} \frac{1}{\rho}\left[\rho\left(\frac{\partial w^h}{\partial t} + u^h \cdot \nabla w^h\right) - \nabla \cdot \sigma\left(q^h, w^h\right)\right] \\
&\cdot \left[\rho\left(\frac{\partial u^n}{\partial t} + u^h \cdot \nabla u^n\right) - \nabla \cdot \sigma\left(p^h, u^h\right)\right] dQ \\
&+ \sum_{e=1}^{(n_{el})_n} \int_{Q_n}^{e} \tau_{cont} \nabla \cdot w^h \rho \nabla \cdot u^h \, dQ \\
&= \int_{(p_n)_h} \left(w^h \cdot \hat{\sigma}\right) \cdot n \, dp - \int_{(p_n)_h} qh \, dp
\end{aligned}$$

where

$$\begin{aligned}
\left(u^h\right)_n^{\pm} &\equiv \lim_{\epsilon \to 0} u\left(t_n \pm \epsilon\right) \\
\int_{Q_n} \ldots \, dQ &\equiv \int_{I_n} \int_{\Omega_t^h} \ldots \, d\Omega \, dt \\
\int_{p_n} \ldots \, dp &\equiv \int_{I_n} \int_{\Gamma_t^h} \ldots \, d\Gamma \, dt.
\end{aligned}$$

The first three terms and the right–hand–side are the Galerkin formulation. The fourth term weakly enforces continuity of velocity over Ω_n. The remaining terms are the stabilization, which are enforced only within each element, that penalizes oscillatory modes.

The particular form of τ_{mom} and τ_{cont} here are:

$$\begin{aligned} \tau_{mom} &= \alpha \frac{h^e}{2\left|u^h\right|_2} \\ \tau_{cont} &= \nu \left|u^h\right|_2 h_e \end{aligned}$$

where α and ν are numerical coefficients from 0 to 1, and h_e is an element length.

Appendix A

Constructing Model Files

This appendix gives an overview of the three files needed to run GLS3D. Details and explanations of each of the input parameters is given in the body of the report and in the sample problems. Three files are required for running GLS3D: the 3-D Mesh File, the Boundary Conditions File, and the Hot Start File. The generation of each of these files and the components of the files are described in the following sections.

A.1 3-D Mesh Files

The three dimensional mesh files needed for GLS3D are generated completely within the GMS. Once the mesh has been generated in the GMS, the file will be used in GLS3D without modification. The filename given to the mesh file, having an extension *.3dm*, will serve as the root name for all GLS3D input files. Details on mesh generation can be found in the example problems contained within this text or in the GMS reference manual.

A.2 Boundary Conditions

The boundary conditions file contains a series of cards that represent the operation controls, iteration parameters, material properties, boundary strings, solution controls, time controls, and output controls. The following tables contains all of the possible boundary condition file cards and a description of their input.

Navier-Stokes Problems			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	NS	Specifies Navier-Stokes Problem

Moving Grid Option			
Field	Type	Value	Description
1	char	OP	Card type.
3	char	MV	Indicates the grid will be allowed to move

Incremental Memory			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	INC	Parameter.
3	int	> 0	Incremental Memory Allocation

Transport Equations			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	TRN	Parameter.
3	int	≥ 0	Total number of transported materials (unsupported in Navier-Stokes, set to 0).

Time Step Weighting			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	THT	Parameter.
3	real	$1 \geq \# \geq 0$	θ Time step weighting.

Block Specification for Pre-conditioner			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	BLK	Parameter.
3	int	> 0	Number of blocks per processor, used to perform pre-conditioning

Pre-conditioner Selection			
Field	Type	Value	Description
1	char	OP	Card type.
2	char	PRE	Parameter.
3	int	$3 \geq \# \geq 0$	Prec_value 0 No pre-conditioning 1 one level Additive Schwarz pre-conditioning 2 two level Additive Schwarz pre-conditioning 3 two level Hybrid pre-conditioning

Table A.1: Operation Parameter Cards

Non-linear Iterations (option 1)			
Field	Type	Value	Description
1	char	IP	Card type.
2	char	NIT	Parameter
3	int	≥ 1	Number of non-linear iterations per time step, if at NIT the tolerance is not satisfied GLS3D will reduce the time step and recalculate.

Non-linear Iterations (option 2)			
Field	Type	Value	Description
1	char	IP	Card type.
2	char	FNI	Parameter.
3	int	≥ 1	Number of non-linear iterations per time step, if at FNI the tolerance is not satisfied then GLS3D will accept the solution and proceed to the next time step.

Non-linear Tolerance			
Field	Type	Value	Description
1	char	IP	Card type.
2	char	NTL	Parameter.
3	real	$\# \geq 0$	Tolerance for Non-Linear Equations

Linear Iterations (option 1)			
Field	Type	Value	Description
1	char	IP	Card type.
2	char	MIT	Parameter.
3	int	≥ 1	Maximum number of linear iterations per non-linear iteration by the interative solver. If the internal linear tolerance (0.01 * NTL) is not met at MIT the solution stops.

Linear Iterations (option 2)			
Field	Type	Value	Description
1	char	IP	Card type.
2	char	FLI	Parameter.
3	int	≥ 1	Maximum number of linear iterations by the iterative solver. If the internal tolerance (0.01 * NTL) is not met at FLI the solution will proceed to the next nonlinear iteration.

Table A.2: Iteration Parameter Cards

Table A.3: Material Property Cards

Eddy Viscosity			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	EV	Parameter.
3	int	≥ 1	Material type ID number.
4	real	$\# > 0$	E_{xx}
5	real	$\# > 0$	E_{yy}
6	real	$\# > 0$	E_{zz}
7	real	$\# > 0$	E_{xy}
8	real	$\# > 0$	E_{xz}
9	real	$\# > 0$	E_{yz}

Non-conservative Calculation Option			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	NCE	Parameter. If present calculations are non-conservative, otherwise, the calculations are conservative.
3	int	≥ 1	Material type ID number.

Viscosity			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	MU	Parameter.
3	real	$\# \geq 0$	Uniform background viscosity.

Gravitational Acceleration			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	G	Parameter.
3	real	$\# \geq 0$	Value of gravity induced acceleration.

Density			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	RHO	Parameter.
3	real	$\# \geq 0$	Density.

Table A.3: Material Property Cards (cont.)

Momentum Stabilization Coefficient			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	TMN	Parameter.
3	real	$1.0 \geq \# \geq 0.0$	Galerkin Least Squares stabilization coefficient associated with the momentum equations.

Continuity Stabilization Coefficient			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	TCN	Parameter.
3	real	$1.0 \geq \# \geq 0.0$	Galerkin Least Squares stabilization coefficient associated with the continuity equation.

Reference Velocity			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	U0	Parameter.
3	real	$\# > 0$	Reference Velocity.

Reference Length			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	L0	Parameter.
3	real	$\# > 0$	Reference Length.

Reference Density			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	R0	Parameter.
3	real	$\# > 0$	Reference Density.

Reference Viscosity			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	MU0	Parameter.
3	real	$\# > 0$	Reference Viscosity.

Refinement Levels			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	ML	Parameter.
3	int	≥ 1	Material type ID number.
4	int	$\# \geq 0$	Maximum number of refinement levels.

Flow Refinement Tolerances			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	FRT	Parameter.
3	int	≥ 1	Material type ID number.
4	real	$\# \geq 0$	Error tolerance for the refinement terms.

Flow Unrefinement Tolerances			
Field	Type	Value	Description
1	char	MP	Card type.
2	char	FUT	Parameter.
3	int	≥ 1	Material type ID number.
4	real	$\# \geq 0$	Error tolerance for unrefining elements.

Table A.4: Material Meshing Control Cards

Node Strings			
Field	Type	Value	Description
1	char	NDS	Card type.
3	int	≥ 1	ID number of a node with a Dirichlet condition.
2	int	≥ 1	String ID number.

Face Strings			
Field	Type	Value	Description
1	char	FCS	Card type.
2	int	≥ 1	Element number.
3	int	≥ 1	ID number of a face with a Neumann condition.
4	int	≥ 1	String ID number.

Border Node String			
Field	Type	Value	Description
1	char	BNS	Card type.
2	int	≥ 1	Node number list, in order, of nodes which have data that are to be supplied to an external program.

Ghost Node String			
Field	Type	Value	Description
1	char	GNS	Card type.
2	int	≥ 1	Node number list, in order, of nodes which have data that are supplied by an external program.

Table A.5: String Structures

Table A.6: Initial and Boundary Conditions

Dirichlet Velocity Boundary Condition			
Field	Type	Value	Description
1	char	DB	Card type.
2	char	VEL	Parameter.
3	int	≥ 1	String ID number
4	int	≥ 1	Series ID number for x-velocity component.
5	int	≥ 1	Series ID number for y-velocity component.
6	int	≥ 1	Series ID number for z-velocity component.

Table A.6: Initial and Boundary Conditions (cont.)

Dirichlet Pressure Boundary Condition			
Field	Type	Value	Description
1	char	DB	Card type.
2	char	PRS	Parameter.
3	int	≥ 1	String ID number.
4	int	≥ 1	Series ID number.

Dirichlet External Pressure Boundary Condition			
Field	Type	Value	Description
1	char	DB	Card type.
2	char	EXT	Parameter.
3	int	≥ 1	String ID number, pressure defined by external program.

Neumann Flow Boundary Condition			
Field	Type	Value	Description
1	char	NB	Card type.
2	char	VEL	Parameter.
3	int	≥ 1	String ID number.
4	int	≥ 1	Series ID number containing the flow (in velocity) out of this boundary.
5	real	$\# \geq 0$	The Darcy-Weisbach friction boundary parameter for this boundary.

Outflow Boundary Condition			
Field	Type	Value	Description
1	char	OB	Card type.
2	char	VEL	Parameter.
3	int	≥ 1	String ID number identifying the boundary face for which flow is allowed to pass out of the domain.

Outflow External Boundary Condition			
Field	Type	Value	Description
1	char	OB	Card type.
2	char	EXT	Parameter.
3	int	≥ 1	String ID number identifying the boundary face for which flow information is supplied by an external program. If this flow is into the model this flow is enforced, if the externally supplied flow is out then the face is treated as an outflow boundary.

Table A.6: Initial and Boundary Conditions (cont.)

Dirichlet Free-Surface Pressure			
Field	Type	Value	Description
1	char	DB	Card type.
2	char	FRS	Parameter.
3	int	≥ 1	String ID number, indentifies the free surface nodes.
4	int	≥ 1	Series ID number containing the pressure.

Dirichlet Displacement Condition			
Field	Type	Value	Description
1	char	DB	Card type.
2	char	MVS	Parameter.
3	int	≥ 1	String ID number.
4	int	≥ 1	Series ID number containing the displacement history.

Starting Time			
Field	Type	Value	Description
1	char	TC	Card type.
2	char	T0	Parameter.
3	real	#	Starting time of the model.

Number of Time Steps			
Field	Type	Value	Description
1	char	TC	Card type.
2	char	TF	Parameter.
3	real	#	Ending time of the model.

Time Step Size			
Field	Type	Value	Description
1	char	TC	Card type.
2	char	IDT	Parameter.
3	int	> 0	Series ID number containing the length of timestep (Δt).

Output			
Field	Type	Value	Description
1	char	OC	Parameter
2	int	> 0	Series ID number that contains the time steps to be output.

Stopping the Model			
Field	Type	Value	Description
1	char	END	Close the model.

Table A.7: Solution Control Cards

OP NS *Navier-Stokes*
OP INC *#memory_allocation_increment*
OP TRN *#transport_equations*
OP BLK *#blocks_per_processor*
OP MV *Moving grid option*
OP PRE *pre-conditioner_selection*
OP THT *time-weighting,* θ
IP MIT *max_linear_iterations_per_non-linear_iteration*
IP NIT *max_non-linear_iterations_per_time_step*
IP NTL *non-linear_tolerance*
MP EV *mat#* EV_{xx} EV_{yy} EV_{zz} EV_{xy} EV_{xz} EV_{yz}
MP ML *mat# max_levels_refinement*
MP FRT *mat# error_tolerance_for_refinement*
MP FUT *mat# error_tolerance_for_unrefinement*
MP MU *background_viscosity*
MP G *gravity*
MP RHO *fluid_density*
MP TMN *coefficient_gls_momentum*
MP TCN *coefficient_gls_continuity*
MP U0 *reference_velocity*
MP R0 *reference_density*
MP L0 *reference_length*
MP MU0 *reference_viscosity*
NDS *node# string#*
FCS *element# face# string#*
DB VEL *string# series# series# series#* Dirichlet_bc velocity
DB PRS *string# series#* Dirichlet_bc pressure
NB VEL *string#* Neumann_bc
DB FRS *string# series#* Free-surface Dirichlet_bc pressure
DB MVS *string# series#* Dirichlet_bc displacement
OB VEL *string#* Outflow_bc
TC T0 *initial_time*
TC TF *final_time*
TC IDT *series#* timestep_size
TC NDP no time adaption
OC *series#* output
END

Table A.8: GLS3D input file template.

Bibliography

[1] L. Babuška and A. K. Aziz. Survey lectures on mathematical foundations of the finite element method. In A. K. Aziz, editor, *The Mathematical Foundations of the Finite Element Method with Applications to Partial Differential Equations*. Academic Press, New York, 1972.

[2] M. Behr and T. E. Tezduyar. Finite element solution strategies for large–scale flow simulations. *Computer Methods in Applied Mechanics and Engineering*, 112:3–24, 1994.

[3] F. Brezzi. On the existence, uniqueness and approximation of saddle– point problems arising from lagrangian multipliers. *Rev. Fr. Autom. Inf. Rech. Oper. Numer. Anal., R2*, pages 129–151, 1974.

[4] Brigham Young University. *GMS – The Department of Defense Groundwater Modeling System Reference Manual*. Brigham Young University Engineering Computer Graphics Laboratory, Provo, Utah, 1994.

[5] L. P. Franca and T. J. R. Hughes. Two classes of mixed finite element methods. *Computer Methods in Applied Mechanics and Engineering*, 69:88–129, 1988.

[6] T. J. R. Hughes, L. P. Franca, and M. Balestra. A new finite element formulation for computational fluid dynamics: V. circumventing the Babuška – Brezzi condition: A stable Petrov–Galerkin formulation of the Stokes problem accomodating equal–order interpolations. *Computer Methods in Applied Mechanics and Engineering*, 59:85–99, 1986.

[7] O. A. Ladyzhenskaya. *The Mathematical Theory of Viscous Incompressible Flows*. Gordon and Breach, New York, 2nd edition, 1969. English translation.

[8] Kimberlie Staheli, Joseph H. Schmidt, and Spencer Swift. *Guidelines for Solving Groundwater Problems with ADH*, January 1998.

[9] T. E. Tezdoyar, S. Mittal, S. R. Ray, and R. Shih. Incompressible flow computations with stabilized bilinear and linear equal–order–interpolation velocity–pressure elements. *Computational Methods in Applied Mechanics and Engineering*, 95:221–242, 1992.