Computational Fluid Dynamics (CFD) with General Equation Mesh Solver (GEMS)

by Xuxiao Li

May 3, 2020

Contents

Prologue

In this document, I will cover multiple topics on both the theoretical and practical aspects of computational fluid dynamics (CFD). With such a rich range of contents in CFD, I will only focus on the certain methods that I am specialized in, while general concepts will also be discussed. The major method discussed in this document will be the Finite Volume Method (FVM). FVM is a somewhat "old school" method compared to the more recent and advanced variants of the Finite Element Method (FEM). However, each method has its own advantages and disadvantages when applied to specific problems. FVM served for a long period of time as the workhorse in solving the fluid mechanics problems, while FEM was born to solve solid mechanics problems. This is a big gap between the "fluid" and the "solid" community. Although the trend is to merge the merits from both methods (e.g., discontinuous Galerkin), but that is still an ongoing research topic and indeed, old habits die hard.

Within the "fluid" community, there is a division between those studying compressible flow problems and those studying incompressible flow problems. Accordingly, the CFD methods can be divided into the density-based methods (suitable for compressible flow) and pressure-based methods (suitable for incompressible flow). The major difference here is that the fluid velocity in compressible flow is typically very large (large Mach number), while fluid velocity is relative small for incompressible flows. This gap has already been filled through years of efforts. Density-based methods can also solve incompressible flow problems using the preconditioning technique, which gives a unified framework for solving fluid dynamics problems. This approach will be the focus of this document.

In the 80's, there were several pioneers who contributed significantly to the preconditioning methods, e.g., Eli Turkel, Bram Van Leer, and Charles Merckle. This document will be focused on Merckle's preconditioning system, and in fact, the practical part of the document is made based on one of the in-house codes developed at Merckle's research group. The in-house code is named as the General Equation Mesh Solver (GEMS) whose main creator is Dr. Ding Li. He worked as a research associate with Prof. Merckle, initially at the University of Tennessee, and later at Purdue University. Dr. Li embarked on the development of GEMS about early 1999. In 2002, the version 1.0 is completed. In 2005, he has added the Maxwell equation into GEMS and also refined multiple features to improve the generality of the code. In a paper Dr. Li published in 2006, he demonstrated the capability of GEMS with impressing results.

I feel obliged to mention how I can have the access to the GEMS code. During the time Dr. Ding Li was at Purdue university, there was a PhD student named Shaoyi Wen who worked with Dr. Li. Shaoyi was then advised by Prof. Yung Shin whose research group had some collaborations with Prof. Merckle's group. Shaoyi modified GEMS code for his needs with the help of Dr. Li and published a paper in 2010 using GEMS to solve thermal-fluid problems in direct laser deposition

processes. Thereafter, the GEMS code seemed to be made available to Prof. Shin's group. Before Shaoyi graduated from Shin's group, he passed the GEMS code to another PhD student at Shin's group, Wenda Tan. At this time, Dr. Li has left Purdue (I actually don't know where he went). Wenda has never made acquaintance with Dr. Li. However, he managed to exploit the GEMS code and have three papers published in 2013, 2014 and 2015 with it. In these papers, Wenda simulated the laser keyhole welding processes, and with GEMS, his model incorporated multiple physics and has high fidelity. In 2015, Wenda ceremoniously graduated from Purdue and became an assistant professor at the University of Utah. In the fall of 2015, I was registered as a PhD student at the University of Utah and I was advised by Wenda (Dr. Tan) since then. Therefore, I have the privilege to study the GEMS code and apply it for my PhD research. I have been exploring the GEMS code since 2017 and still learn new things about it today. So far I only touched upon the "basic" functions in GEMS which is to solve the basic equations: mass, momentum, energy, and species conservation equations. The unexplored part is: turbulence modeling and incorporation of Maxwell equations. In the definitive version of the GEMS, all the equations could be solved and can be customized such that multiple sets of equations can be assigned to different partitions of the domain. In this document, I will focus on the "basic" package which I feel the most comfortable with.

The philosophy of this document will be that more focus is put on the theoretical side. By that I do not mean that I will linger on derivations of equations, as "computational" fluid dynamics should emphasize on **data structure** and **algorithm** and not on mathematical reasoning. However, I will not go into details such as "this subroutine does this", or "this variable is used for this", or "how do I output such information". Instead, I will try to **speak out** what the code is trying to do, why it should be doing this, and how certain things are done. With this global picture, you should be able to "code it up" by any language, any way you want. In situations where practical guide is necessary, I will try to separate it by distinct sections.

Chapter 1: Mesh

1.1 Definition of Basic Mesh

The mesh describes the data structure of carrying out CFD simulations. Mesh is a discretization of a calculation domain (a physical domain in which CFD simulations is conducted). For example, we have a two-dimensional rectangle in which we want to carry out CFD simulations, as shown in Fig. 1.1a. An exemplary mesh is shown in Fig. 1.1b. The rectangle has a width of 2 of a height of 1. The boundary conditions are defined as follows. The left and right edge form boundary condition batch 1, the top edge is boundary condition batch 2, and the bottom edge is boundary batch 3.

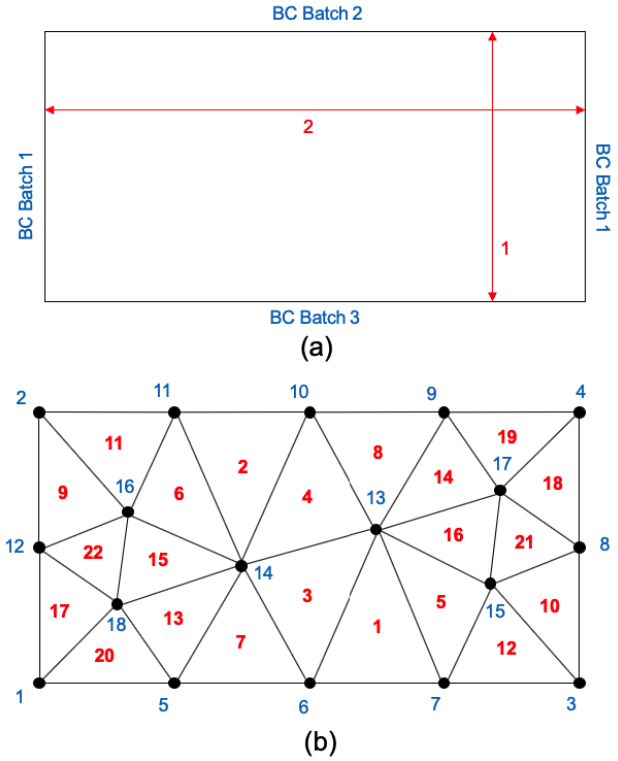

Figure 1.1: Illustration of mesh. (a) Definition of the calculation domain. (b) A discretization of the calculation domain (mesh).

Several notes on defining the calculation domain. First, the dimensions are be scaled up or down when simulating a particular problem. That is, the numbers of 1 and 2 can mean "meter" or "micrometer" depending on the specific problem. No need to specify units here. Second, the boundary conditions must be carefully stated. We need to first think about how many distinct BC's we want to define. Then we assign each distinct BC (BC batch) to a segment of the domain

boundary. Here, we have three unique BC's. Each of them is assigned with a certain segment of the domain boundary.

After the calculation domain is defined. We seed the domain with a set of points (black circles in Fig. 1.1b) in the domain, referred to as the "**nodes**". The nodes are indexed by the blue numbers. We have 18 nodes in the domain. Next, we connect the nodes to form a "tessellation" of the domain. Here we use all triangles. In general, there is a large flexibility of the tessellation. You can use triangles, polygons, or their mixture in two-dimension. In three-dimension, there are the tetrahedral, pyramid, prism, and hexagon, or their mixture (called hybrid mesh). This flexibility is incorporated in the GEMS code, therefore the name "General Equation Mesh Solver". Now with the tessellation, we divide the domain into triangular "elements", from which the name "Finite Element Method" is originated. However, we will only focus on the Finite Volume Method in this document, and the elements will be referred to as the "**cells**". We have 22 cells in the domain and they are indexed with the red numbers. It should be mentioned that the indexing of the nodes and cells does not matter in the definition of the mesh.

Now we can define the "**cell-node connectivity**" by associating cells with its corresponding nodes. For example:

```
cell 1 ---> node 6, 7, 13
cell 2 ---> node 11, 10, 14
...
```

With that, let's design the following data structures `cell` and `node` to better represent this information. For `node`, we have (in Fortran):

```
type node
   real :: xyz(ndim)
end type node
```

where `ndim` is the number of dimensions and `xyz` is the coordinate of the node. We will need to have an array of nodes `nodes(:)` to store all the information about nodes. For cells, we have the following data type:

```
type cell
   real :: centp(ndim)
   integer, pointer :: c2n(:)
end type cell
```

where `centp` is the centroid of the cell and `c2n` is the node indices associated with this cell. Note that we can calculate the centroid of the cell based on the nodal coordinates associated with the cell. We will also need an array of cells `cells(:)` to store all the information about cells. Now we have introduced two important data types `node` and `cell`. Each of these data types can have more attributes which we will build on later.

Now we still need to add the last part of the mesh definition, the boundary conditions. To define the boundary conditions of the mesh, we have to introduce another important concept, "**faces**". Faces are the edges (or facets in 3D) that form the boundary of a cell. There are three faces per one triangle cell, and six faces per one hexagon cell (in 3D), etc. Faces that are shared between two cells are defined as **interior faces**. Faces that are only belong to a single cell are defined as **boundary faces**. Faces can be distinguished by a set of nodes, which brings about the "face-node"

connectivity. We will discuss in further details about faces in the next section. For now, we use face to define the boundary conditions as follows:

```
BC batch 1 ---> 4 faces: (1, 12), (2, 12), (3, 6), (4, 8)
BC batch 2 ---> 4 faces: (2, 11), (11, 10), (10, 9), (9, 4)
BC batch 3 ---> 4 faces: (1, 5), (5, 6), (6, 7), (7, 3)
```

That is, we use the boundary faces (shown as node sets above) to define the segments of boundaries. Each set of faces represents a unique BC. It is convenient to create a data type for the BC's:

```
type bc_type
   integer :: label
   integer :: igrp
   integer :: itype
end type bc_type
```

Here, `label` is a distinct integer to indicate the batch of BC. `igrp` indicates which "group" of BC it is. In GEMS, there are 6 groups of BC's:

- Group 1, Inlet
- Group 2, Outlet
- Group 3, Farfield
- Group 4, Wall
- Group 5, Geometric (e.g., symmetric, periodic)
- Group 6, MHD (for Maxwell equations)

Each group of BC can have sub-categories (types) which is saved in the attribute `itype`. Again, we need an array of BC's `bc(:)` to store all the information. Apparently, the association between BC and faces should be defined based on the face lists of each batch of BC, which will be discussed in the next section.

So far, we have introduced the complete information to define a mesh, summarized as follows:

1. Nodal coordinates.
2. Cell-node connectivity.
3. Face list in each BC batch.

We emphasize this contains the complete information of a mesh. We refer this piece of information as the "basic mesh" or the finite element mesh. The information is complete but is not organized in a way convenient for finite volume method. As we will see, additional data types can be constructed based on the basic mesh and be used to facilitate finite-volume computation.

1.2 Finite Volume Mesh

We establish the data type `face` from the basic mesh to form the "finite volume mesh". The `face` type has much more emphasis in the finite volume method as the fluxes on the faces need to be computed. On the opposite side, the finite element method emphasize on the data type `node`. The `face` type is defined as:

```
type face
```

```
    integer :: itype
    type(cell), pointer :: left_cell
    type(cell), pointer :: right_cell
    integer, pointer :: f2n(:)
    real :: centp(ndim)
end type face
```

Here, `itype` defines the type of the face which we will elaborate later. The pointers `left_cell` and `right_cell` refers to the two cells that share this face. This is referred to as the **face-cell connectivity**. The pointer `f2n(:)`, like `c2n(:)`, is the **face-node connectivity**, which refers to the set of nodes that defines the face. At last, the center of the face `centp` can be calculated based on the nodal coordinates which can be found through `f2n(:)`. The `face` type constructed based on Fig. 1.1b is shown in Fig. 1.2. The faces are indexed with the green number and there are 39 faces in the calculation domain.

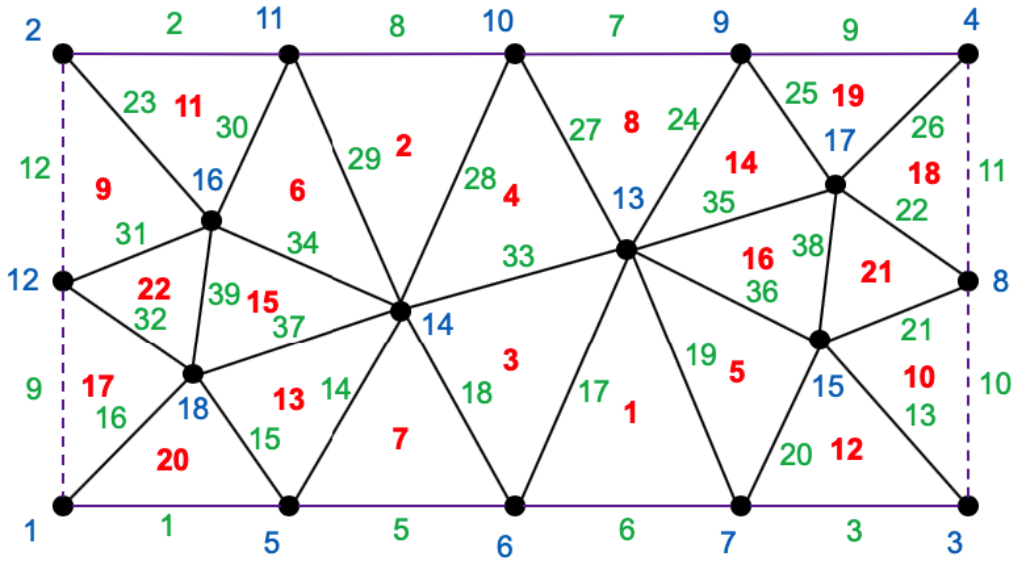

Figure 1.2: Construction of faces to make the finite volume mesh.

How to collect all the face information from the basic mesh? The face information can be found from the cell-node connectivity. For example:

```
cell 1 ---> node 6, 7, 13, we can find:
   face 1 --> node 6, 7
   face 2 --> node 7, 13
   face 3 --> node 13, 6
cell 2 ---> node 11, 10, 14, we can find:
   face 4 --> node 11, 10
   ...
```

By doing so we obtain an array of faces, `faces(:)`. Two points need to be noted. First, the face-node connectivity is contained in the cell-node connectivity, but we still need to know which nodes in a cell can form a face. If all cells are triangles, any combination of the three nodes of the cell will form a face. But for quadrilaterals and 3D cells, a set of rules need to be pre-defined to help extracting the face-node connectivity. The second note is, faces identified this way will be

double-counted. We need to delete the repeating faces. This can be done by checking whether the face-node connectivity `f2n` is repeating. In doing so, every time we find a repeating face, it means two cells share the same face. Therefore, the `left_cell` and the `right_cell` can be found. We note that the "left" and "right" do not mean the literal directions. In practice, we always make the cell with the smaller index (red number in Fig. 1.2) to be the left cell and the other cell to be the right cell.

There are some faces that are only belong to a single cell. These faces are referred to as the "**boundary faces**" (face 1 - 12 in Fig. 1.2). Boundary faces are regarded as special faces and we will make them to be at the front of `faces(:)` by **swapping faces** in the face array. We assign the only cell that a boundary face is associated to be the **left cell** of the boundary face. That is, the boundary faces only have `left_cell` which is the interior cell, for now.

In CFD computation, it is required that every face be associated with two cells, including the boundary faces. Therefore, we need to create a "**ghost cell**" as the right cell for the boundary faces. Ghost is an overused word (but I still sometimes use it). The ghost cell is alternatively referred to as the "**boundary cell**". To create the ghost cells, we can simply mirror the interior cell with respect to the boundary face. However, there is one exception, the periodic boundary condition. For periodic BC, we need to translate the same cell "on the other side" to form the ghost cell. To illustrate, the boundary faces are marked purple in Fig. 1.2. We let BC batch 1 to be the periodic BC, and the periodic boundary faces are marked as dashed purple. To create the boundary cells, we can mirror cell 11, 2, 8, 9, 20, 7, 1, and 12 for face 2, 8, 7, 9, 1, 5, 6, 3, respectively. For periodic boundary cells, we need to translate cell 9, 17, 18, 10 for face 11, 10, 12, 9, respectively. This indicates that the periodic boundary faces must "match", otherwise, the mesh is ill-defined for the periodic BC. The periodic boundary faces are special boundary faces. We will move these faces to the end of the boundary faces. That is, `face(1)` to `face(12)` are the boundary faces, and `face(9)` to `face(12)` are reserved for periodic boundary faces. We will dedicate an entire section to periodic BC later.

One last note about face types. The interior faces (black solid line in Fig. 1.2) has `itype = 0`. The boundary faces has a positive `itype` which equals to the index in `bc(:)` that the boundary face is associated with. For example, we have 3 batches of BC's, and we create an array `bc(:)` with a length of 3. Each element in `bc` corresponds to one specific BC batch. Say, batch 1 corresponds to `bc(3)`. Then the `itype` for face 9, 10, 11, 12 will be equal to 3. There can be other `itypes` which we shall discuss later.

To summarize, the finite volume mesh consists of the following information:

1. Nodal coordinates.
2. Cell-node connectivity.
3. Face types, face-cell connectivity and face-node connectivity.

Notice that the information about BC's can be fully described by the `bc(:)` array and the `itype` in `face`.

1.3 Partitioning of Mesh

Now we have got a finite volume mesh, with three essential data types, `node`, `cell`, and `face`. It is no problem to plug this mesh into some CFD code. However, today's computation lives in

a parallel world. Any practical CFD code must enable parallel computing. GEMS uses MPI for parallel computing. In MPI, the finite volume mesh is decomposed into partitions. Each partition is assigned to one "core" (or one CPU, one "processing element", any way you want to call it). Each core only has information about its own partition of the entire domain and has no idea of what the entire domain is like. Such a partitioning is shown in Fig. 1.3.

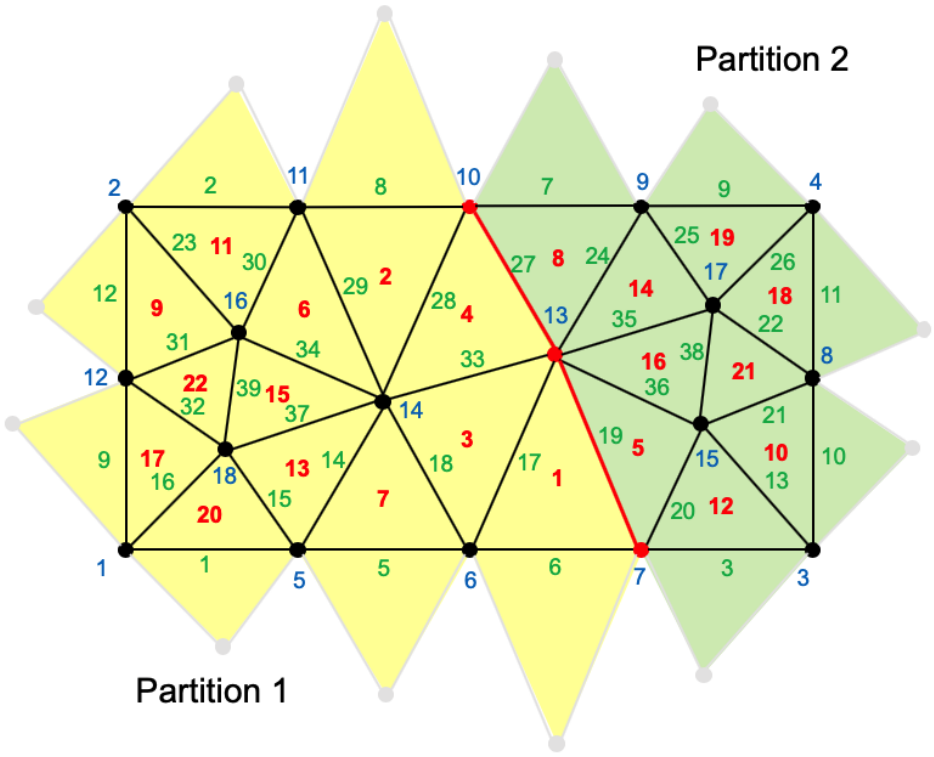

Figure 1.3: Partitioning of calculation domain

Here the contour formed by the solid black line is the calculation domain, including the nodes (indexed by blue), cells (indexed by red) and faces (indexed by green). We also show the ghost cells (boundary cells) marked by the gray lines. It is noted that the ghost cells can be entirely derived from the boundary faces and need not be store prior to CFD simulation. Based on this configuration, we partition the domain into two partitions. First, we identify which **cells** belong to which partition. Then the nodes and faces shared by cells from both partitions can be identified (marked by red). These delineates the **interface** of the partitions. The nodes, cells, and faces belong to partition 1 is shadowed by yellow and those belong to partition 2 is shadowed by green.

Now we split the domain into two partitions as shown in Fig. 1.4. Take partition 1 for example, the nodes, cells, and faces all need to be re-indexed as partition 1 does not have any information about partition 2. The re-indexed numbers are shown by the corresponding blue, red and green numbers. For partition 1, to establish the "communication" with the external world (which means partition 2 in this case), we identify all the cells (including ghost cells) in partition 2 that shared nodes with partition 1. Those are the orange-shaded cells indexed from 1 - 7. This means there are 7 cells we need to create in partition 1 as "containers" to receive the information from partition 2. Same for partition 2, we need to create 6 "container" cells to receive information from partition 1. From a global point of view (Fig. 1.3), the container cells (orange-shaded cells in Fig. 1.4) for both partitions can be identified. Then, we can correspondingly mark in each partitions the cells

that need to be "sent" out to the external world (the sky-blue shaded cells in Fig. 1.4). Note that the "sending cells" and the "receiving cells" must match. That is, the receiving cells in partition 1 (orange) must be match the sending cells in partition 2 (sky-blue), and vice versa.

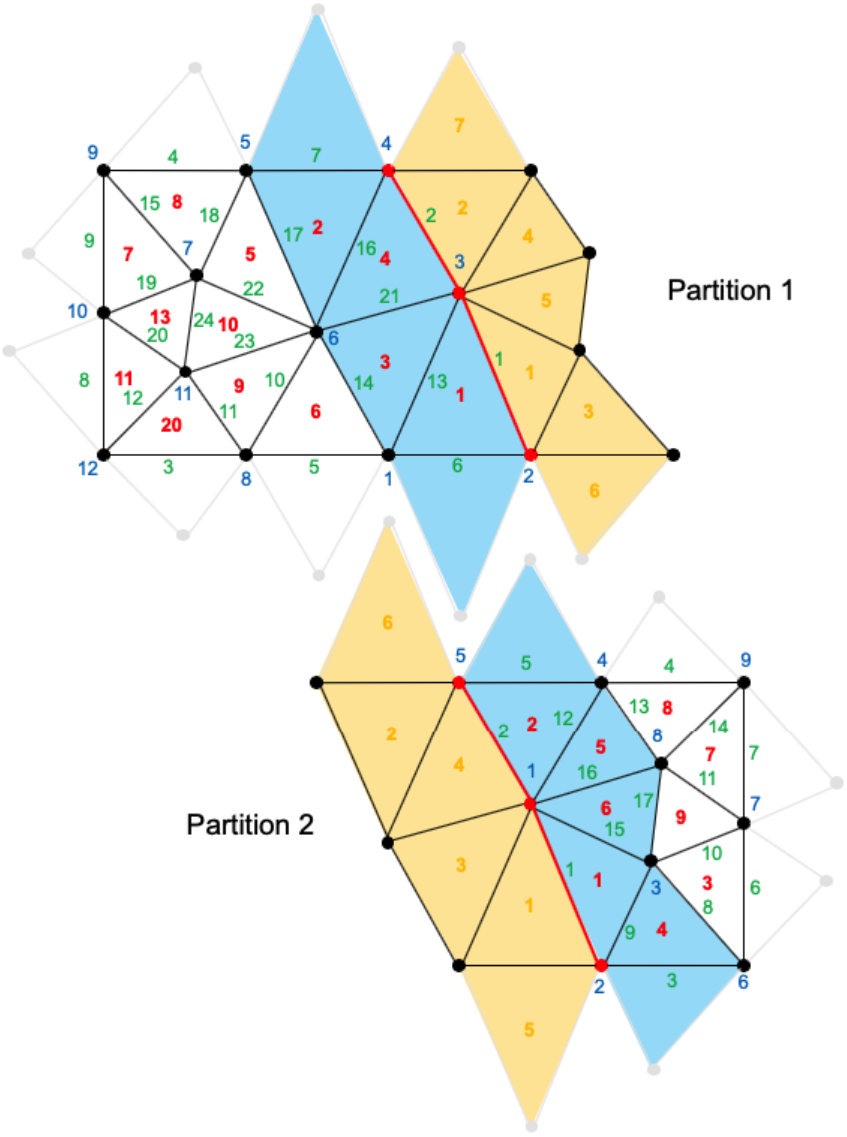

Figure 1.4: Splitting of the domain and re-indexing.

To represent the data structure for communication among partitions, in GEMS, the data type `itf` (standing for "interface") is created:

```
type itf
   integer :: nn
   integer, pointer :: np(:), nnp(:)
   type(cell), pointer :: pcell(:)
end itf
```

where `nn` is the number of partitions from which the current partition receives data; `np` and `nnp` are arrays with a length of `nn`. `np` stores the id of the partitions from which the current partition

receives data, and `nnp` stores the number of cells to receive from each partition. Finally, `pcell` is the array of "container" cells to store the received data. `pcell(:)` should have a length of `sum(nnp)`. It should be noted that the `pcell(:)` array needs to be **allocated** as it is ghost cells that do not exist in the physical domain of partition 1.

So far, the attributes of `itf` are only relative to receiving (orange-shaded cells in Fig. 1.4). We need to add similar attributes for sending:

```
type itf
   ...
   integer :: nc
   integer :: cp(:), ncp(:)
   type(neighbour_cell), pointer :: scell(:)
end type itf
```

Here `nc` is the number to partitions to which the current partition needs to send data; `cp` is the id's of partitions to which the current partition sends data and `ncp` is the number of cells to send to each partition in `cp`. The array `scell(:)` points to the cells in the current partition whose information will be sent to external partitions. Notice that instead of using `type(cell)`, we invented a new data type `neighbour_cell` for `scell(:)`. This data type is a **nested type** for the type `cell`:

```
type neighbour_cell
   type(cell), pointer :: to_cell
end type neighbour_cell
```

There is nothing in the type `neighbour_cell` but a `cell` pointer. This nested structure is adopted in GEMS to represent that the cell pointer **needs not be allocated**. Rather, it is merely a reference to cells already allocated.

How can we construct such data type `itf`? It must start from the global domain (Fig. 1.3). First we identify the container cells for each partition (1 & 2). Then we can identify the sending cells in partition 1 based on the container cell in partition 2, and we record the cell id's and boundary face id's (for boundary cells to send). Same for partition 2. For container cells, they don't yet exist in each partition. Therefore, (for example, in partition 1), we simply allocate 7 "empty" cells. The empty cells will be fulfilled once the data is received from partition 2.

Although the container cells are empty, the cell-node and cell-face connectivity between the container cells and the nodes and faces needs to be identified. Take partition 1 for example, for cell-node connectivity, we need to record in each container cell which node index in partition 1 it is associated with. Note, only record those nodes in partition 1, so we have 2 nodes for `pcell(1:2)` and only 1 node for `pcell(3:7)`. For face-cell connectivity, we need to associate the `right_cell` of `faces(1:2)` with `pcell(1:2)` for partition 1. For partition 2, we associate the `right_cell` of `faces(1:2)` with `pcell(1)` and `pcell(4)`. The `left_cell` will be assigned to the other cell which is interior cell. It should be mentioned that the `left_cell` of a face always belong to the interior of the domain.

The indexing convention for the partitioning is as follows. We first index interior cells for `pcell` and then the boundary cells. Same for the `scell(:)`. Note, the indexing of `scell(:)` and `pcell(:)` must match across partitions. The faces that has a `right_cell` as a container cell are referred to as the "**partitioning faces**" and are considered as another type of special faces. A special (large) number is reserved for the `itype` of the partitioning faces, 19621011. The partitioning faces are

indexed from very beginning of the `faces` array (by proper face swapping). Therefore, we have a rather complex rule for face indexing (take partition 1 as example):

1. Index partitioning faces (face 1 & 2), `itype = 19621011`
2. Index boundary faces that are not periodic boundaries (face 3 - 7), `itype =` positive number (from one to the number of BC's)
3. Index periodic boundary faces (face 8 & 9), `itype =` the number for periodic BC
4. Index interior faces (face 10 - 23) , `itype = 0`

Also, we record the number of partitioning faces in the type `itf` by adding the variable `nitf`:

```
type itf
   ...
   integer :: nitf
end type itf
```

With the mesh partitioning, we will generate separate mesh files for each partition. Let us now summarize the information contained in each mesh file:

1. Nodal coordinates for current partition
2. Cell-node connectivity for current partition
3. Face-cell connectivity, face-node connectivity as well as face type for current partition
4. Sending information: No. of sending-to partitions, No. of total sending cells. For each sending-to partition,
   (a) sending-to partition id, No. of to-be-sent interior cells, No. of to-be-sent boundary cells
   (b) for to-be-sent interior cells, store those cell id
   (c) for to-be-sent boundary cells, store those boundary face id
5. Receiving information: No. of receive-from partitions, No. of total receiving (container) cells. For each receive-from partition,
   (a) receive-from partition id, No. of container cells to receive from this partition
   (b) for each container cell, record the node shared between container cell and the current partition

1.4 Periodic Boundary Condition

The treatment of periodic BC deserves a separate section to describe. The difficulty in treating periodic BC arises when information in the boundary cells for periodic BC cannot be directly obtained from the left cell of the boundary face (i.e., mirroring). Rather, we need to find the left cell of the matching boundary face "on the other side". Moreover, when the domain is partitioned, the matching face may not even exist in the current partition, and communication must be established specially for treating periodic BC.

To better illustrate the treatment of periodic BC, we modified the original definition of BC's in Fig. 1.1. As shown in Fig. 1.5a, we define only two batches of BC's and both of them are assigned to be periodic BC's. Each batch consists of a pair of walls. The corresponding mesh (before partition)

is shown in Fig. 1.5b. We emphasize again that boundary faces in the mesh for periodic BC must "match", i.e., sharing the exact same nodal coordinates in all directions excluding the periodic direction. Otherwise, the face meshing is ill-defined for the periodic BC.

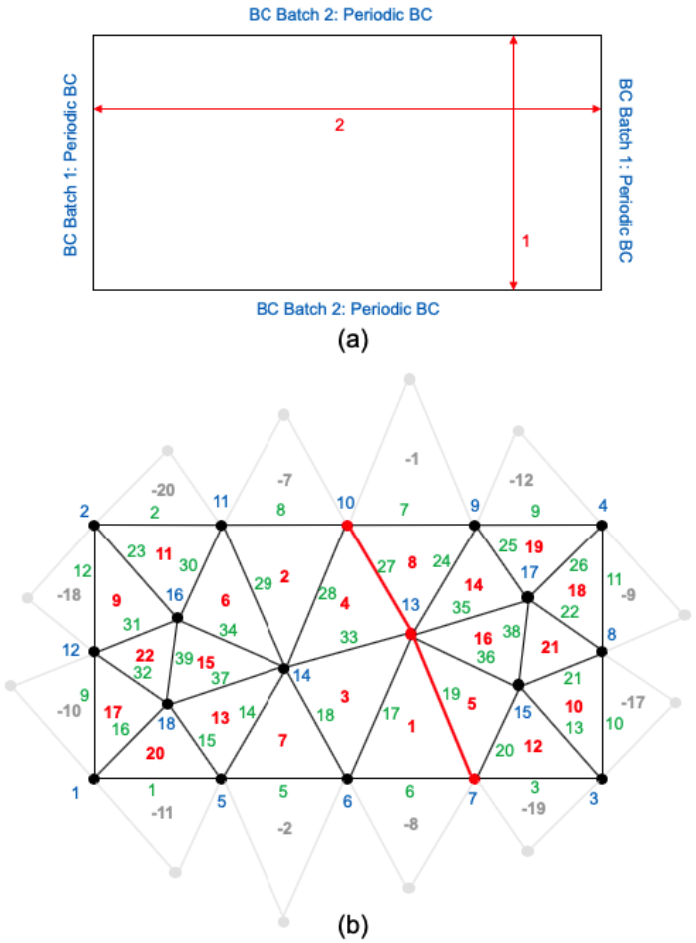

Figure 1.5: Re-define the boundary conditions. (a) Set two pairs of walls as 2 batches of periodic BC. (b) The mesh corresponding to the setup in (a).

In Fig. 1.5b, again, the node, cells, and faces of the global domain are indexed. The global domain will be partitioned into two parts as separated by the red line like before. The boundary cells are marked by the gray edges. The first step to construct periodic BC is to find the "matching cells" for the boundary cells. This step is shown with the global indices in Fig. 1.5b. We index the periodic boundary cells by a negative number whose absolute value matches the global indices of the matching interior cells, as shown in Fig. 1.5b. After the matching cells are found (globally), we merge the two (or more) batches of periodic BC as one batch. This is due to the special treatment of periodic BC in GEMS. As long as matching cells are found, the complete information regarding to all periodic BC's is known. Therefore, they are viewed as a single package (of BC) in GEMS.

Now, let's illustrate the specific data structure of periodic BC by considering the partitioning layout shown in Fig. 1.6. Take partition 1 for example, the yellow region is the physical domain of partition 1 and the orange region is the container cells to receive information from partition 2, like before. There are 7 container cells, as indexed by the orange numbers. Now, the periodic boundary

cells are considered as a second type of container cells, as indexed by the purple numbers. There are 8 **periodic container cells** and they are indexed as negative number to differentiate with the "partitioning container cells". It is noted that some of these periodic container cells needs to be received from partition 2 (-4, -6 and -7), but some of them are actually just the interior cells in partition 1 but "on the other side" of the boundary faces of -2, -5, -1, -3. GEMS will treat all the periodic container/boundary cells by communications between partitions, in a unified manner. For cell -2, -5, -1 and -3, they will be communicated from partition 1 and to partition 1. This is the difference between communicating partitioning cells and periodic boundary cells. The latter involves communicating with the partition itself (totally OK in MPI).

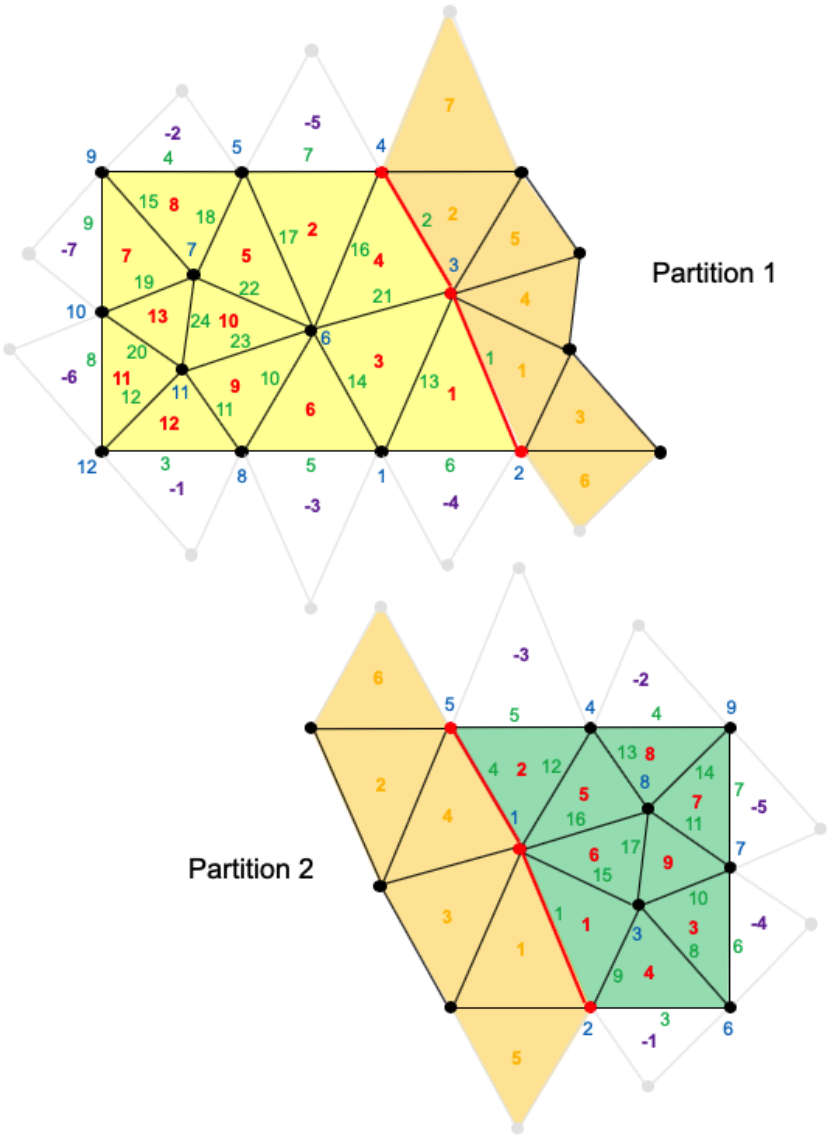

Figure 1.6: Demonstration of the treatment of periodic BC.

If we examine on the interior cells in Fig. 1.5b, each interior cell belongs to a unique partition (separated by the red line). Also, with the matching cell known for each periodic boundary face, we can know from where the periodic container cells should receive. The exact same data structure

`itf` as in partitioning communication is used to represent the communication for periodic BC. `pinterf` (of the type `itf`) can be constructed in both partitions in the following way:

```
pinterf of Partition 1:
   2 receiving batches:
   receive-from id ---> 1 (itself) and 2
   No. of container cells ---> 4 (from id = 1) and 3 (from id = 2)
   periodic boundary faces to bear the container cells:
      face 4, 7, 3, 5 for from id = 1
      face 8, 9, 6 for from id = 2

   2 sending batches:
   sending-to id ---> 1 (itself) and 2
   No. of to-be-sent cells ---> 4 (to id = 1) and 3 (to id = 2)
   To-be-sent iterior cell indices:
      cell 8, 2, 6, 11 for to id = 1
      cell 7, 11, 1 for to id = 2
end pinterf

pinterf of Partition 2:
   2 receiving batches:
   receive-from id ---> 1 and 2 (itself)
   No. of container cells ---> 3 (from id = 1) and 2 (from id = 2)
   periodic boundary faces to bear the container cells:
      face 4, 5, 3 for from id = 1
      face 6, 7 for from id = 2

   2 sending batches:
   sending-to-id ---> 1 and 2 (itself)
   No. of to-be-sent cells ---> 3 (to id = 1) and 2 (to id = 2)
   To-be-sent iterior cell indices:
      cell 2, 7, 3 for to id = 1
      cell 8, 4 for to id = 2
end pinterf
```

In constructing the above data structure, we first identify the container cells in both partitions. These are "ghost cells" to be allocated and are not interior cells in the domain. Then, the sending cells (interior cells) are determined based on the container cells in each partition. It is noted that for periodic boundary cells, we do not need the cell-node connectivity as in partitioning cells. We only need the cell-face connectivity. We associate periodic boundary faces with their corresponding container cells. Then, the cell-node connectivity of the container cells is equivalent to the face-node connectivity of the periodic boundary faces. This can be applied to other type of boundary cells. The cell-node connectivity of boundary cells will always be equivalent to the face-node connectivity of the corresponding boundary faces.

As a final note, the communication of periodic BC must precede the communication of the partitioning. This is because we periodic BC communication will first fill in all the boundary cells, some of which can then be sent out via the partitioning communication.

1.5 Practical Guide

We shall give some practical instructions in the section regarding the generation and use of mesh files. The action items are summarized by the flow chart shown in Fig. 1.7.

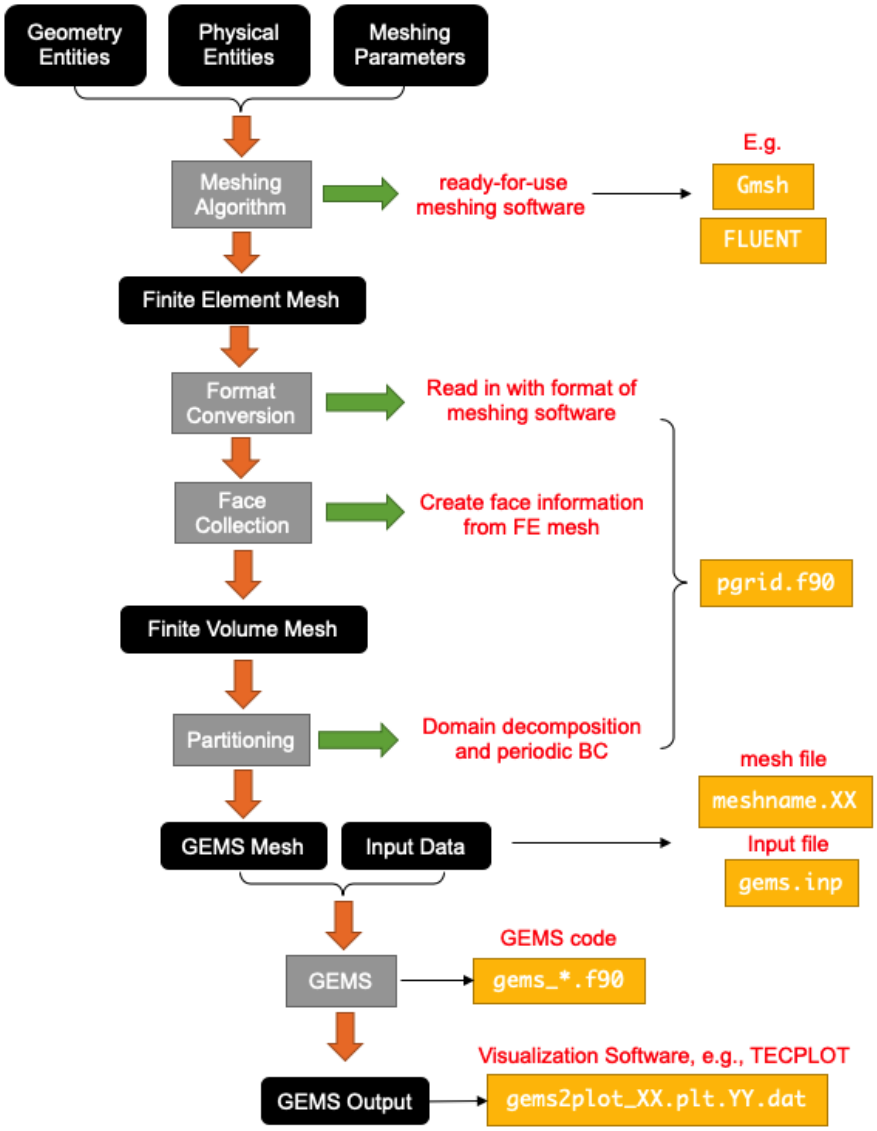

Figure 1.7: Flow chart of generating and using mesh files.

The first step is to generate a finite element mesh as described in section 1.1. We note that the mesh file is defined by three components: geometry entities, physical entities, and meshing parameters. The geometry entities refer to the point, lines, curves, etc. that define the geometry of the calculation domain. This is analogous to the CAD drawing you use for a design. Second, you associate the geometry entities with physical entities. For example, the geometry entities in Fig. 1.1(a) is 4 points, 4 lines and a rectangle. The lines that form the rectangle are associated with physical entities BC batch 1 (e.g., periodic boundaries), BC batch 2 (e.g., solid wall) and BC batch 3 (e.g., another solid wall). Finally, the meshing parameters that specify the way to "seed" the domain with nodes are defined. That can be how many nodes should be seeded along the lines,

etc. The geometry entities, physical entities, and meshing parameters are then fed into the meshing algorithm to generate a finite element mesh like that in Fig. 1.1(b).

The meshing algorithm itself is a totally different science. We simply rely on ready-for-use toolboxes to generate the finite element mesh, rather than implement the mesh algorithm ourselves. `Gmsh` seems to be a good option for doing this, as it gives flexible automation with Gmsh scripts. However, you still need to learn how to draw points and lines, associated them with physical entities, as well as control the meshing schemes with meshing parameters. For `Gmsh`, those are specified in the `.geo` files, which are inputted into `Gmsh` to obtain the mesh (`.msh`) files.

Once we have the FE mesh file, we feed it into the code `pgrid.f90` which is a part of GEMS code and functions as a pre-processor to convert a FE mesh to the mesh file GEMS recognizes (referred to as the GEMS mesh). In `pgrid.f90`, the first task is to read in the FE mesh generated by other software (e.g. `Gmsh`). Unfortunately, the software usually writes a mesh file in complicated ways, and different software writes the mesh file differently. `pgrid.f90` has different interfaces for reading FE mesh files with different format, but mostly of them are obsolete. For example, the `GAMBIT` format was used in the past which is generated by `FLUENT`. But since `FLUENT` was acquired by `ANSYS`, we have lost the way to generate `GAMBIT` files. As of today, two formats are actively used: (1) Cartesian mesh format, which is generated without using any software, and (2) SU2 format, which is generated by `Gmsh` (choose `SU2` format when exporting).

The Cartesian mesh format is a simple format and it can be written in the following way (in FORTRAN, and use a 2D mesh as example):

```
do j=1, Nj
   do i=1, Ni
      write xn(i)
   end do
end do

do j=1, Nj
   do i=1, Ni
      write yn(j)
   end do
end do
```

Here `Ni` and `Nj` are the number of **nodes** (not cells) along each dimension. `xn(:)` and `yn(:)` are 1D arrays that contains the nodal coordinates along X and Y dimension, respectively. Besides this, the boundary condition information must be written in a separate file in the following way:

```
write ndim*2
write 1,Ni,1,1,0,0,label1
write 1,Ni,Nj,Nj,0,0,label2
write 1,1,1,Nj,0,0,label3
write Ni,Ni,1,Nj,0,0,label4
```

where `ndim` is the number of dimensions, and `label1 - label4` are the labels for BC's on the four edges of the rectangle. Note that the edge is specified by the starting and ending indices of the 1D arrays, and the zeros indicates that the third dimension is turned off in the 2D mesh. GEMS will then convert the Cartesian mesh into an essentially an unstructured mesh (in the code `pgrid.f90` by creating nodal coordinates and cell-node connectivity, etc.). This is because GEMS treats all

the meshes as unstructured mesh to achieve its generality.

After the FE mesh is read in, the data type `face` will be used to generate the finite volume mesh, as described in section 1.2. After that, the mesh is partitioned to enable parallel computing. We note that this partitioning action includes both domain decomposition (described in section 1.3) and creating the ghost cells for the periodic BC's if there is any (described in section 1.4). These are all done in `pgrid.f90` and separate files will be generated for each partition. The GEMS mesh files are named as `meshname.XX` where `XX` stands for the indices of the partition (starting from 1, and then 2, 3, etc.).

A list of parameters that needs to be given in `pgrid.f90` are given in Table 1.1.

Table 1.1: List of key input parameters in pgrid.f90.

variable name	**meaning**
`gridfile`	the address of FE mesh files
`iftm`	the format of FE mesh file (structured, SU2, etc.)
`ndim`	dimension of the mesh
`nparts`	number of desired partitions
`im`	which partitioning algorithm
`npartsX, npartsY, npartsZ`	number of partitions along each dimension, if `im=7` is selected
`boundfile`	address of boundary condition file, if `iftm=1` is selected
`id`	specify which type of periodic BC is desired (0 means no periodic BC)
`npbc`	number of periodic BC labels (later will merge into one label for all periodic BC)
`lab_period,iax`	pair of periodic BC label and along which direction the periodicity is
`ngeom`	which mesh geometry to specify in TECPLOT file (1=triangle, etc.)

Couple of notes, (1) the ideal partitioning algorithm should the `METIS` algorithm which distributes the number of cells evenly among partitions. However, the FORTRAN interface of `METIS` has been lost and a poor man's partitioning algorithm `im = 7` is commonly used now. The poor man's partitioning algorithm divides the domain evenly according to the length along the X, Y and Z dimensions. Uneven distribution of cells will be caused if, for example, a non-uniform mesh size is applied. (2) When specifying multiple periodic BC's, a loop will be used to scan all the periodic BC's labels. Each label corresponds to a type of periodicity (cylindrical, 2D planar and 3D planar), and with each type of periodicity, the direction along which the periodicity occurs must also be specified. (3) a tecplot file will be generated for the GEMS mesh after the main programs of `pgrid.f90` are finished. In tecplot, the mesh geometry needs to be explicitly specified (by `ngeom`), whether it be triangle, quadrilateral, tetrahedron, or brick. If the mesh is of other geometries, specify `ngeom` as 5 which basically treats all 2D meshes as quadrilaterals and all 3D meshes as bricks. This sometimes causes messy visualizations but this is the best that tecplot can do.

Now let's move on to the tasks after the GEMS mesh is generated (Fig. 1.7). The mesh files and the input data are fed into the GEMS code, after which the GEMS ouptut, i.e., the results of the CFD simulation can be obtained. The input data is specified in the file `gems.inp`. This is a manually written file in which a group of FORTRAN namelists are specified. These namelists record the material properties, numerical parameters, etc. used for the GEMS computation. The GEMS code consists of about 20 `f90` files with the suffix `gems_`. These files are different modules of GEMS and form the main programs of the CFD simulation. The output files are TECPLOT

files with the name `gems2plot_XX.plt.YY.dat` where `XX` stands for the time strand and `YY` stands for the id of the partition. Note that each partition outputs independently, and therefore, if there are 4 partitions, 4 output files will be written (`YY` from 0 to 3) for one time strand `XX`. The data become difficult to manage when a large number of partition is used (e.g., 80). To deal with this, a python script is used to concatenate the 80 files for each time strand. Also, the tecplot script `preplot` is used convert all the `.dat` ASCII files into binary files to reduce the file size as well as the time it takes to load data into tecplot.

Chapter 2: Fundamentals of Fluid Mechanics

For a topic having such a variety of different facets like fluid mechanics, even the fundamentals are different in different contexts. Here we will introduce the fundamentals from the "computational" perspective. The essence is a set of partial differential equations that we are going to numerically solving. We shall properly define these PDE's and prepare them in a format ready for numerical computation.

2.1 Viewpoints of Describing Fluid Mechanics

The PDE's we are going to solve are the so-called conservation laws. That is, some **conservative variable** such as mass, momentum and energy should be conserved. Let's take the mass conservation for example. Here we focus our attention on a **material region**. That is a selection of particles (atoms, molecules) of material, and we track the same particles over time. The conservation of mass states that the mass of the tracked material region shall not be changed over time, as shown in 2.1.

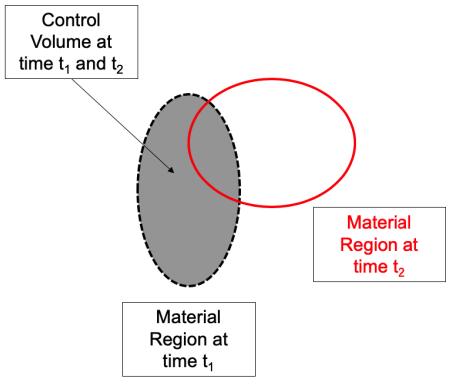

Figure 2.1: Material region and control volume. Black-dashed and red-circled region are the two material region at two time moments. The gray area overlaps with the material region at t_1 and represents the control volume which is fixed regardless of time.

Let's express the mass conservation in the format of **integral equation**:

$$\frac{\partial}{\partial t}\int_{\Omega_M(t)} \rho d\Omega = 0 \tag{2.1}$$

Here Ω_M stands the volume of the material region which can be changing with time t. ρ is the density and a function of time and space, and $d\Omega$ is a infinitesimal volume in 3D. Next, we want to re-write Eqn. 2.1 in the context of **control volume**, instead of the material region. A control volume is a fixed volume of space and do not change with time. Mass can flow into or out of the control volume. Typically, we only care a specific type of control volume. That is the cells we introduced in chapter 1. We will be dealing with tons of control volumes as each cell represents a

fixed volume of space and is viewed as one control volume. The difference of control volume and material region should be emphasized. The material region is the volume of space that encompasses a certain set of particles being tracked, so the material region changes with time depending on the velocities of the particles. The control volume is a fixed volume of space, and particles can freely flow into or out of the control volume. Technically, the control volume can also be set to be moving, but let's just set it to be stationary all the time. These two concepts represent two perspectives of describing the mechanics of particles. The material region is called the **Lagrangian point of view** and the control volume is referred to as the **Eulerian point of view**.

How to bridge the two point of views of describing mechanics? Let's find a time moment where the material region and control volume are overlaid on one another, as shown in Fig. 2.1. If we take the Eulerian perspective, the mass within the **control volume** should be changing and the rate should be equal to the net mass rate flowing into/out of the control volume. Therefore, writing in the format of integral equation, we have:

$$\int_{\Omega} \frac{\partial \rho}{\partial t} d\Omega + \int_{\partial\Omega} \rho (V \cdot n) d\sigma = 0 \tag{2.2}$$

Here Ω is the control volume (we dropped the subscript M to differentiate control volume and material region). Note that we can move the partial derivative inside the integral since Ω does not change with time. ρ is still density as a function of time and space. V is the velocity (a 3D vector) and n is the normal unit vector (also a 3D vector) on the 2D boundary of the 3D control volume (written as $\partial\Omega$. The dot product $V \cdot n$ represents the normal component of velocity on the boundary $\partial\Omega$. Finally, $d\sigma$ stands for the infinitesimal surface area on $\partial\Omega$. One note regarding notation, we do not use the arrow or bold font to differentiate vector, tensor and scalar. This clarity of the notation is sacrificed to trade for conciseness in our equations.

The two equations Eqn. 2.1 and 2.2 are equivalent to each other. They describe the same conservation law of mass but with two point of views. Eqn. 2.1 is the Lagrangian view and Eqn. 2.2 is the Eulerian view. We will only use the Eulerian point of view of describe fluid mechanics. The equivalence of the two point of views can be proved mathematically based on the Leibnitz's theorem [1]. The Leibnitz's theorem states that for any quantity Q:

$$\frac{\partial}{\partial t} \int_{\Omega_M(t)} Q d\Omega = \int_{\Omega} \frac{\partial Q}{\partial t} d\Omega + \int_{\partial\Omega} Q (V \cdot n) d\sigma \tag{2.3}$$

In the context of fluid mechanics, Q can be mass, momentum and energy.

2.2 Governing equations

For now on, we will be focusing on using control volume to describe the conservation equations (Eulerian viewpoint). We have already introduced the mass conservation equation as Eqn. 2.2. Next, momentum conservation. In 3D, the momentum along the three dimensions should all be "conserved" which gives 3 momentum conservation equations. Unlike the mass conservation, the momentum of the control volume can be changed by forces. This is Newton's second law. We write the integral form of momentum conservation equation as:

$$\int_{\Omega} \frac{\partial (\rho V_i)}{\partial t} d\Omega + \int_{\partial\Omega} (\rho V_i)(V \cdot n) d\sigma = \int_{\Omega} \rho f_i d\Omega + \int_{\partial\Omega} R_i d\sigma \tag{2.4}$$

On the left-hand-side (LHS), the first term is the temporal rate of the momentum along the i dimension ($i = x, y, z$), integrated over the control volume. The second term is the momentum

flowing into or out of the control volume. Note that the entire left-hand-side can be interpreted in the Lagrangian viewpoint as the temporal rate of momentum of the **material region**. Use Leibnitz's theorem, Eqn. 2.3. The right-hand-side stands for the force. The first term on the right-hand-side (RHS) is the **body force** (the component along the i direction), while the second term is the **surface force**. We note that the body force has the unit of N/kg or the unit of acceleration m^2/s. The surface force has the unit of N/m^2. The most common body force is the gravitational force $g = [0, -9.81, 0]^T$

The surface force needs our special attention. To describe a surface force, we need to use **stress tensor**. Surface force can be alternatively referred to as the surface stress. Stress tensor is a matrix T, and the surface force is given by $R = Tn$. This is a matrix multiplication where n and R are column vectors. We will assume throughout this document a vector is a column vector, unless otherwise specified. This can also be written as $R = (n \cdot T)^T$. I will avoid using matrix multiplications and instead use dot products (inner product) or outer product in this document. Note the dot (inner) product of matrices A and B are defined as $A \cdot B = A^T B$. The multiplication of two matrices can be expressed as $AB = A^T \cdot B$. The outer product of two matrices are defined as: $A \otimes B = AB^T$.

The stress tensor has two components, pressure and viscous stress, expressed as:

$$T = -pI + \tau$$

where p is the **thermodynamic pressure**, I is a 3 by 3 unit matrix (in 3D), and τ is the viscous stress tensor which is a matrix as a function of the gradient of velocities. This relationship is referred to as **constitutive relations** which will be discussed later.

Next, let's consider the conservation of energy. Again, writing as integral equation:

$$\int_\Omega \frac{\partial(\rho e_{tot})}{\partial t} d\Omega + \int_{\partial\Omega} (\rho e_{tot})(V \cdot n) d\sigma = \int_\Omega \rho f \cdot V d\Omega + \int_{\partial\Omega} (n \cdot T)^T \cdot V d\sigma - \int_{\partial\Omega} (n \cdot q) d\sigma \qquad (2.5)$$

Here e_{tot} is the **total energy**, $e_{tot} = e + \frac{1}{2}V^2$, where e is the **specific** internal energy, or energy per unit mass. The two terms combined on the LHS stands for the temporal rate of total energy of the mass region, again, Leibnitz's theorem (Eqn .2.3). The first two terms on the RHS stand for the **work** done by the body force and surface force per unit time, or the **power**. The last term of the RHS stands for the heat transfer, or heat flux flowing into or out of the control volume, where q is the heat flux vector (with the unit of W/m^2. The heat flux is a function of temperature gradient. Such a relationship is also part of the constitutive relations. The long equation (Eqn. 2.5) is nothing other than the **first law of thermodynamics**, written as "$dU = \delta Q - \delta W$" in typical thermodynamic textbooks.

Eqn. 2.2, 2.4, and 2.5 constitutes the **governing equations** of fluid mechanics, written in the form of integral equations. This set of equations is the basic set while more governing equations need to be included when considering additional physics, e.g., conservation of chemical species. That being said, we limit our discussion to this set of basic equations in this chapter.

2.3 Constitutive Relations

There are essentially five equations from Eqn. 2.2, 2.4, and 2.5. Therefore, we need to pick five independent variables, referred to as the **primitive variables**, and express all the variables in Eqn. 2.2, 2.4, and 2.5 as functions of the independent. Such relations are generally referred to as

the **constitutive relations**. We will be exclusively using (p, V, T) as the primitive variables in this document. Note that the choice of primitive variables can be arbitrary, but different choice may lead to entirely different constitutive relations. The choice of (p, V, T) is because that it is a convention to express other properties of a material as functions of p, V, T in, for example, material handbooks and databases.

The **thermodynamic relations** include $e = e(p, T)$ (internal energy), $\rho = \rho(p, T)$, and sometimes the **enthalpy**, $h = e + p/\rho$. The **total enthalpy** can be expressed as $h_{tot} = h + \frac{1}{2}V^2$. These relations are for the "state functions" and are often referred to as the **equation of state** (EOS).

The **kinetic relations** describe the "transfer" of some quantity. For example, the transfer of heat is described by the **Fourier's law**:

$$q = -k\nabla T$$

where k is the thermal conductivity. Note that k is referred to as (one of) the **transport properties** and can be a function of primitive variables, $k = k(p, T)$. Also, Fourier's law only governs the **conduction** of heat transfer. Convection and radiation can be introduced by including additional kinetic relations. But this is beyond the scope of the current discussion. The transfer of momentum is described by the **Newton's viscosity law**:

$$\tau = \left(-\frac{2}{3}\mu\nabla \cdot V\right) I + 2\mu S$$

where μ is the viscosity and can be a function of primitive variables, $\mu = \mu(p, T)$. $\nabla \cdot V$ is the divergence of velocity. Note that the nabla operator can be also treated as a column vector: $\nabla = [\partial/\partial x, \partial/\partial y, \partial/\partial z]^T$. Again, I is a 3 by 3 unit matrix, and S is the **strain rate tensor**:

$$S = \frac{1}{2}\begin{bmatrix} 2\frac{\partial u}{\partial x} & \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} & \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & 2\frac{\partial v}{\partial y} & \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \\ \frac{\partial u}{\partial z} + \frac{\partial x}{\partial w} & \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} & 2\frac{\partial w}{\partial z} \end{bmatrix}$$

where (u, v, w) are the three components of the velocity vector V. One final note regarding the thermodynamic and kinetic relations. The term "thermodynamic" and "kinetic" are from material science. Basically, "thermodynamic" is associated with the states of equilibrium while "kinetic" is associated with the states in between equilibrium, and therefore, with the gradient of thermodynamic properties. This is a hand-wavy distinction of these two categories of constitutive relations, based on concepts in material science.

Up to now, we can express all the quantities in Eqn. 2.2, 2.4, and 2.5 as functions of the primitive variables (p, V, T). The body force f may be the exception, but for now we can simply view it as a constant as for the gravitational force.

2.4 Differential Governing Equations

In this section we brief discuss the differential form of governing equations. Although this form is not our focus in numerical implementations, it can be convenient to use this form in analysis. To derive the differential form, we need to use Gauss's theorem

$$\int_\Omega \nabla\phi \, d\Omega = \int_{\partial\Omega} n\phi \, d\sigma, \quad \phi \text{ is scalar} \tag{2.6a}$$

$$\int_\Omega \nabla \cdot v \, d\Omega = \int_{\partial\Omega} n \cdot v \, d\sigma, \quad v \text{ is vector} \tag{2.6b}$$

$$\int_\Omega \nabla \cdot T \, d\Omega = \int_{\partial\Omega} n \cdot T \, d\sigma, \quad T \text{ is matrix (tensor)} \tag{2.6c}$$

Again, the nabla operator is treated as a column vector $\nabla = [\partial/\partial x, \partial/\partial y, \partial/\partial z]^T$, and note the LHS and RHS of Eqn. 2.6c are both row vectors. Eqn. 2.6c can be transposed to give the column vector form.

Now, we can use Eqn. 2.6 to transform the surface integrals to volume integrals in Eqn. 2.2, 2.4, and 2.5. Then, we can drop the integral and keep the intergrand to obtain the differential equations. For example, the mass conservation can be written as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho V) = 0 \tag{2.7}$$

by applying Eqn. 2.6b and let $v = \rho V$. To derive momentum conservation equation, we need to group the terms inside the surface integral to form tensors. First, we write Eqn. 2.4 as:

$$\int_\Omega \frac{\partial(\rho V)}{\partial t} d\Omega + \int_{\partial\Omega} (n \cdot (\rho V \otimes V))^T \, d\sigma = \int_\Omega \rho f d\Omega + \int_{\partial\Omega} (n \cdot T)^T d\sigma \tag{2.8}$$

where $T = -pI + \tau$, and note that $\rho V \otimes V$ is a matrix. Then, apply Eqn. 2.6c to the surface integrals to get:

$$\frac{\partial(\rho V)}{\partial t} + (\nabla \cdot (\rho V \otimes V))^T = \rho f + (-\nabla p) + (\nabla \cdot \tau)^T \tag{2.9}$$

Note that we applied the transpose operation to keep very vector as a column vector. Finally, to derive the energy conservation equation, we observe that the work done by the surface force can be rewritten as $(n \cdot T)^T \cdot V = n \cdot (T^T \cdot V)$. That is, the dot product of n and some matrix. Therefore, we can apply Eqn. 2.6c and obtain:

$$\frac{\partial(\rho e_{tot})}{\partial t} + \nabla \cdot (\rho e_{tot} V) = \rho f \cdot V + (-\nabla \cdot (pV)) + \nabla \cdot (\tau^T \cdot V) - \nabla \cdot q \tag{2.10}$$

Now we have the set of differential equations, Eqn. 2.7, 2.9, and 2.10, that governs the conservation of mass, momentum and energy.

2.5 Matrix Representation

In this section, we will prepare the set of governing equations in integral form, Eqn. 2.2, 2.8, and 2.5 in a format that is more suitable for numerical computation. Let's first seperate the pressure and the viscous stress from the surface stress tensor T, and rewrite the equations as follows:

$$\int_\Omega \frac{\partial \rho}{\partial t} d\Omega + \int_{\partial\Omega} n \cdot (\rho V) d\sigma = 0 \tag{2.11a}$$

$$\int_\Omega \frac{\partial(\rho V)}{\partial t} d\Omega + \int_{\partial\Omega} (n \cdot (\rho V \otimes V))^T \, d\sigma = \int_{\partial\Omega} (n \cdot (-pI))^T d\sigma + \int_{\partial\Omega} (n \cdot \tau)^T + \int_\Omega \rho f d\Omega \tag{2.11b}$$

$$\begin{aligned} \int_\Omega \frac{\partial(\rho e_{tot})}{\partial t} d\Omega + \int_{\partial\Omega} n \cdot (\rho e_{tot} V) d\sigma = & \int_{\partial\Omega} n \cdot (-pV) d\sigma + \int_{\partial\Omega} n \cdot (\tau^T \cdot V) d\sigma - \int_{\partial\Omega} (n \cdot q) d\sigma \\ & + \int_\Omega \rho f \cdot V d\Omega \end{aligned} \tag{2.11c}$$

Notice that for the energy equation Eqn. 2.5, we used:

$$(n \cdot T)^T = n \cdot (T^T \cdot V) = n \cdot ((-pI + \tau^T) \cdot V) = n \cdot (-pV) + n \cdot (\tau^T \cdot V)$$

We observe a similarity in Eqns 2.11 that all the surface integrals has the form of the norm vector n, dot product with some vector or tensor. Also, we want to distinguish the terms related to gradients (e.g., τ and q) from those which does not. For that, we can move the pressure-related terms from the RHS to the LHS to give:

$$\int_\Omega \frac{\partial \rho}{\partial t} d\Omega + \int_{\partial\Omega} n \cdot (\rho V) d\sigma = 0 \tag{2.12a}$$

$$\int_\Omega \frac{\partial (\rho V)}{\partial t} d\Omega + \int_{\partial\Omega} \left(n \cdot (\rho V \otimes V + pI)\right)^T d\sigma = \int_{\partial\Omega} (n \cdot \tau)^T + \int_\Omega \rho f d\Omega \tag{2.12b}$$

$$\int_\Omega \frac{\partial (\rho e_{tot})}{\partial t} d\Omega + \int_{\partial\Omega} n \cdot (\rho h_{tot} V) d\sigma = \int_{\partial\Omega} n \cdot (\tau^T \cdot V) d\sigma - \int_{\partial\Omega} (n \cdot q) d\sigma + \int_\Omega \rho f \cdot V d\Omega \tag{2.12c}$$

Note that for Eqn. 2.12c, we used $\rho e_{tot} + p = \rho h_{tot}$. Now, with the form of Eqn. 2.12, we can group terms together to give the **matrix representation** of the governing equations:

$$\int_\Omega \frac{\partial Q_c}{\partial t} d\Omega + \int_{\partial\Omega} (n \cdot F_c)^T d\sigma = \int_{\partial\Omega} (n \cdot F_v)^T d\sigma + \int_\Omega S d\Omega \tag{2.13}$$

where the terms in Eqn. 2.13 are as follows:

$$Q_c = \begin{bmatrix} \rho \\ \rho V \\ \rho e_{tot} \end{bmatrix}_{5\times 1} \qquad F_c = \begin{bmatrix} \rho V & \rho V \otimes V + pI & \rho h_{tot} V \end{bmatrix}_{3\times 5}$$

$$F_v = \begin{bmatrix} 0 & \tau & \tau^T \cdot V - q \end{bmatrix}_{3\times 5} \qquad S = \begin{bmatrix} 0 \\ \rho f \\ \rho f \cdot V \end{bmatrix}_{5\times 1}$$

Again, all the vectors are expressed as column vectors. Eqn. 2.13 can also be succinctly written as:

$$\int_\Omega \frac{\partial Q_c}{\partial t} d\Omega + \int_{\partial\Omega} (n \cdot (F_c - F_v))^T d\sigma = \int_\Omega S d\Omega \tag{2.14}$$

The corresponding differential form of the matrix representation can be derived by simply applying Eqn. 2.6c:

$$\frac{\partial Q_c}{\partial t} + (\nabla \cdot (F_c - F_v))^T = S \tag{2.15}$$

Eqn. 2.15 is the one used in [2], but the integral form will be the foundation for numerical computation.

Now let's briefly discuss the importance of such matrix representation Eqn. 2.14. Here, Q_c is referred to as the **conservative variables**, F_c the **convective flux**, F_v the **viscous flux** and S the source term. It is a succinct representation of a general form of conservation law. It states that the temporal rate of a vector variable Qc is due to the fluxes due to translation of such variable (F_c), the gradient of such variable (F_v) as well as the external source addition/destruction of such variable (S). With this form, we can conveniently add more conservative variables to the vector Q_c and modify accordingly the convective flux, viscous flux and the source term.

The control volume Ω in Eqn. 2.14 will be millions of cells in the mesh. For each cell, we write down Eqn. 2.14, with 5 unknowns in Q_c. Then, we will have millions of unknowns and millions of equations for the entire calculation domain. Notice that the boundary of the cell $\partial\Omega$ will be composed of faces. This foreshadows the surface integrals on the boundary of a cell will be approximated by

summations over the faces of the cell.

It is more convenient to deal with the primitive variables $Q_p = [p, V, T]^T$, rather than the conservative variables Q_c. Therefore, we can express Eqn. 2.14 as:

$$\int_\Omega \Gamma \frac{\partial Q_p}{\partial t} d\Omega + \int_{\partial\Omega} (n \cdot (F_c(Q_p) - F_v(Q_p)))^T = \int_\Omega S(Q_p) d\Omega, \text{ where } \Gamma = \frac{\partial Q_c}{\partial Q_p}$$

That is, expressing every term as functions of the primitive variable Q_p. Here, Γ is referred to as the **conservative jacobian** which connects primitive variable and conservative variable.

2.6 Preconditioning: Motivation

Chapter 3: Algorithm

Chapter 4: Benchmark Tests

4.1 One-Dimensional Euler Equation

4.2 Propagation of Vortex

4.3 Flow Around a Cylinder

4.4 Decay of Isotropic Turbulence*

Chapter 5: Awkward Level-Set GEMS

5.1 The Level-Set Function

5.2 Hamilton-Jacobi Equations

5.3 Ghost Fluid Method

5.4 Lagrangian Particle Tracking*

Reference

[1] R. L. Panton, *Incompressible flow.* John Wiley & Sons, 2013.

[2] D. Li and C. L. Merkle, "A unified framework for incompressible and compressible fluid flows," *Journal of Hydrodynamics, Ser. B*, vol. 18, no. 3, pp. 113–119, 2006.