True*Grid*® Output Manual For Fluent®

A Guide and a Reference

by

Robert Rainsberger

Version 2.3.0

XYZ Scientific Applications, Inc.

February 12, 2013

Table of Contents

I. Fluent® Output Guide

Fluent® is a three-dimensional simulation code by ANSYS, Inc. to model fluid dynamics. The focus in this manual will be on those features in **True*Grid*®** that are specific to creating Fluent® input data. The **True*Grid*®** User's Manual covers the creation of a mesh and will not be covered in this manual. This manual is incomplete in another sense because it cannot be used as a substitute for a working knowledge of Fluent® .

Cell Shapes

The typical cell shape is a hexahedron brick. To form a wedge, pyramid, tetrahedron, or other degenerate brick cell, gather some of the nodes to collapse edges and faces. Then be sure to issue a merge command in the merge phase, such as the **stp** command, so that coincident nodes are merged into one node. See the Fluent® Manual for the allowable degenerate bricks.

Commands to create a Fluent® Input File

The following is a list of **True*Grid*®** commands that can be used to produce features for the Fluent® input file.

Fluent® feature	**True*Grid*®** commands
choose Fluent® format	**fluent**
define the cell type within a zone	**fluemats**
assign cells to a zone	**mate**, **mt**, **mti**, **mtv**
assign boundary conditions with zone numbers	**fbc**, **fbci**
assign nodes to a zone	**nset**, **nseti**
form a face set	**fset**, **fseti**
create the input file for Fluent®	**write**
name the input file for Fluent®	**mof**

The **fluent** command must be issued before the **write** command. The **write** command can only be issued in the merge phase. The default file name for the input file to Fluent® is trugrdo. Use the **mof** command before issuing the **write** command to change this file name.

Cell types are set by defining what is typically called a material in **True*Grid*®** using the **fluemats** command. This is done in the control phase, usually at the beginning. One typically develops the model first and then inserts the **fluemats** commands at the beginning of the rerun file and forms the mesh one last time in order to write the file after merging nodes. Every material is associated with a zone number (typically called a material number). Then cells are assigned a zone (material) number using the **mate**, **mt**, **mti** and **mtv** commands. In the merge phase, each material is colored differently. You can also choose which materials are shown using the **dm**, **dms**, **dam**, **dms**, **ams**, and **rms** commands.

Use the **fbc** and **fbci** commands in the part phase to specify facial boundary conditions and zone numbers. Or, use the **fbc** command with face sets to define boundary conditions and zone numbers to faces in the merge phase. Face sets can be formed using the **fset** and **fseti** commands in the part phase or the **fset** command in the merge phase. Interior faces not assigned a boundary condition are automatically set to interior. Any exterior faces not assigned a boundary condition are assigned to be a wall. In the merge phase, use the **condition** (**co**) command to highlight the zone specified using the fbc and fbci commands.

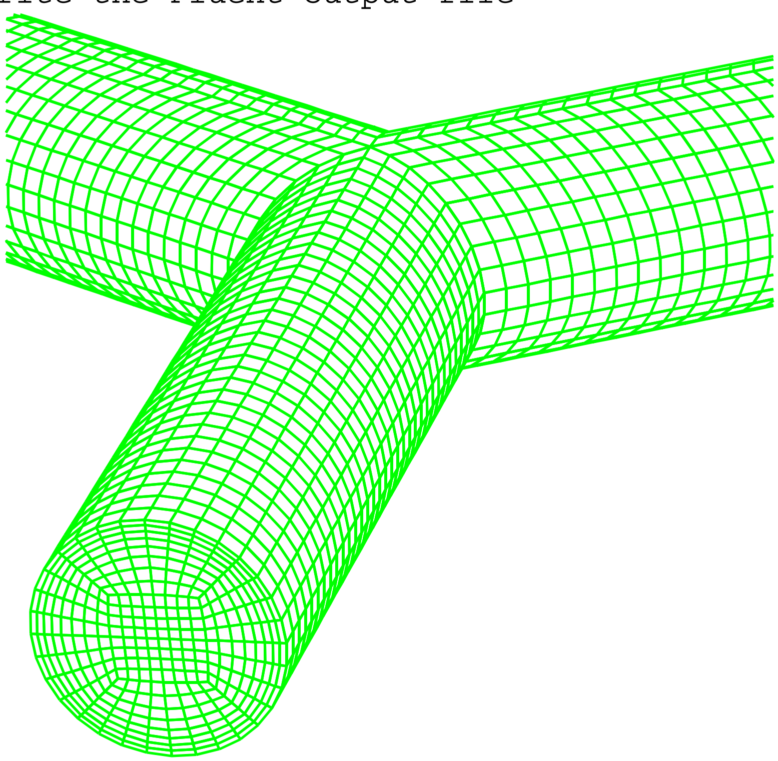
Use the **nset** and the **nseti** commands in the part phase and the **nset** command in the merge phase to assign nodes to a zone. If the name of the set is a number, that number becomes the zone number of the nodes in the set. If the node set has a non-numeric name, then **True*Grid***® chooses a zone number for the nodes in that set. If a node is in several node sets, it will show up in the Fluent® output file as being in the node set that was defined last. In the merge phase, click on the Pick panel and then sets. Choose nodes in the Set Editing window, enter the set name into the Open Set text field, and click on Open to highlight the nodes in a node set.

Care should be taken in assigning zone numbers (i.e. the zone numbers in the **fbc** and **fbci** commands, the material numbers, and the numeric name of node sets) such that they are unique. For example, if there are two different types of boundary conditions with the same zone number or if there is a node set and a material with the same number, **True*Grid***® will terminate the writing of the Fluent® output with an error.

II. Fluent® EXAMPLE

```
fluent c output format
fluemats 2 1 zname water ; c associate material 2 with zone 2
parameters r .3 c  outer radius
           d  4 c  number of elements in smallest block
           l  2 c  length of one leg
           q [.25*%pi*%r/%d]  c  mesh density
           m [%l/%q] c  number of elements in the length
           a .01 ; c  boundary layer thickness
sd 1 cy 0 0 0 1 0 0 %r  c outer cylinder surface
sd 2 plan 0 0 0 [-sin(-60)] [cos(-60)] 0 c symmetry plane
sd 3 plan 0 0 0 [-sin( 60)] [cos( 60)] 0 c symmetry plane
sd 4 plan 0 0 0 0 1 1 c diagonal plane for the miter
sd 5 plan 0 0 0 0 1 -1 c diagonal plane for the other miter
sd 6 cy 0 0 0 1 0 0 [%r/2] c inner cylinder surface
intyp 2  c  use transfinite interpolation as default
block 1 [1+%m];  c  create a 12 block part
      1 [1+2*%d] [1+3*%d] [1+4*%d] [1+6*%d] ;
      1 [1+2*%d] [1+4*%d] [1+6*%d] ;
      0 %l
      [-%r/3] [-%r/3] 0 [%r/3] [%r/3]
      [-%r/3] [-%r/3] [%r/3] [%r/3]
dei ;1 2 0 4 5;1 2 0 3 4;  c  remove corner blocks
as 1 4 3 2 5 3 j 1 %a  c  boundary layer constraints
as 1 4 3 2 4 4 k 1 %a
as 1 2 3 2 2 4 k 1 %a
as 1 1 3 2 2 3 j 0 %a
as 1 2 1 2 2 2 k 0 %a
as 1 4 1 2 4 2 k 0 %a
as 1 4 2 2 5 2 j 1 %a
as 1 1 2 2 2 2 j 0 %a
as 1 3 1 2 3 2 k 0 %a
as 1 3 3 2 3 4 k 1 %a
sfi ;-1 -5;-1 -4;sd 1  c  project (constrain) faces to surfaces
sfi -1;1 3;1 4;sd 2
sfi -1;3 5;1 4;sd 3
sfi 1 2;-2;3 4;sd 4
sfi 1 2;1 2;-3;sd 4
sfi 1 2;4 5;-2;sd 4
sfi 1 2;-4;1 2;sd 4
sfi 1 2;-2;1 2;sd 5
sfi 1 2;1 2;-2;sd 5
sfi 1 2;-4;3 4;sd 5
```

```
sfi 1 2;4 5;-3;sd 5
sfi 1 2;-2 0 -4;-2 0 -3;sd 6
bb 1 4 3 2 4 4 1;bb 1 4 3 2 5 3 1; c glue the miters together
bb 1 4 2 2 5 2 2;bb 1 4 1 2 4 2 2;
bb 1 2 1 2 2 2 3;bb 1 1 2 2 2 2 3;
bb 1 1 3 2 2 3 4;bb 1 2 3 2 2 4 4;
unifm 2 1 2 2 5 3 & 2 2 3 2 4 4 & 2 2 1 2 4 2 4 0 1 ; C smooth
unifm 1 2 1 1 4 4 & 1 1 2 1 2 3 & 1 4 2 1 5 3 4 0 1 ;
fbci ;-1 -5;-1 -4;wall 1      c  Fluent wall boundary condition
nset 1 2 2 2 4 3 = 6 c nodal zone through the center
mate 2 c cell zone (material) assignment
lct 2 rz 120;rz -120;lrep 0 1 2;  c  replicate 3 times
merge c final phase required to write the Fluent input file
stp .0001  c  specify the tolerance in the node merging
sd 7 plan 2 0 0 1 0 0 c face set at each end
fset face1 = surface 7 .001 3
sd 8 plan [2*cos(120)] [2*sin(120)] 0 [cos(120)] [sin(120)] 0
fset face2 = surface 8 .001 3
sd 9 plan [2*cos(-120)] [2*sin(-120)] 0 [cos(-120)] [sin(-120)] 0
fset face3 = surface 9 .001 3
fbc fset face1 pr_inlet 3 c different end boundary conditions
fbc fset face2 outflow 4
fbc fset face3 outflow 5
write   c  write the Fluent output file
```



III. Fluent® OUTPUT REFERENCE

The syntax for commands are described below were literals are highlighted in **bold**. Symbols to be substituted are *italicized*. Each command is described by an entry like the following:

Command Syntax Conventions

When an arbitrarily long list of arguments are required, a semi-colon terminates the list. When a semi-colon is found in the description of an option or command, this indicates such a list. It is common to have a list inside another list. Each list must have a terminating semi-colon. This is analogous to parenthesis in algebraic expressions where the opening parenthesis must be balanced with a closing parenthesis. In this case, the keyword that initiates a list of items must be balanced with a closing semi-colon. Sometimes a short list of arguments and options can be repeated indefinitely, forming a list. The set of arguments and options that can be repeated are placed in square brackets. Sometimes the abbreviation *#_things* is used to mean "number of things". Each command is described by an entry like the following:

command summary description

command *arguments* brief description of functionality
with brief descriptions of what the *arguments* should be.
indentation is used to indicate a list of options to the *arguments*

Some commands in the part phase require a region specification. The region selects a face of the mesh, among other things. Others may require a progression specification. The progression selects multiple faces, among other things. In the merge phase, such commands require an option. In all of these cases, a portion of the mesh is identified.

Remarks

When present, the Remarks section describes the command in even greater detail. It may describe the context in which the command is normally used, and other commands used in association with this command. It may describe side effects. It may describe other, similar commands. In many cases, it includes a description of where to find the command in the menus.

Examples

When present, this shows the exact use of the command. If you use the dialogues, this command will be generated by simple selection options with the mouse and entering data where indicated. The command, as shown here, will appear in the session file for later reuse and possible modification.

You can also enter the command into the text window or insert it into a command file to be run in batch mode.

fbc FLUENT boundary conditions - part phase

fbc *region type [name] zone_#*

where *type* can be

interior	
wall	
pr_inlet	pressure-inlet
inlet_ve	inlet-vent
intake_f	intake-fan
pr_outle	pressure-outlet
exhaust_	exhaust-fan
outlet_v	outlet-vent
symmetry	
per_shad	periodic-shadow
pr_far_f	pressure-far-field
velocity	velocity-inlet
periodic	
fan	
porous_j	porous-jump
radiator	
mass_flo	mass-flow-inlet
interfac	interface
outflow	
axis	

Remarks

The option to name the zone can be done by placing the *name* before the zone number.

fbci FLUENT boundary conditions by progression - part phase

fbci *progression type [name] zone_#*

where *type* can be

interior	
wall	
pr_inlet	pressure-inlet
inlet_ve	inlet-vent

intake_f	intake-fan
pr_outle	pressure-outlet
exhaust_	exhaust-fan
outlet_v	outlet-vent
symmetry	
per_shad	periodic-shadow
pr_far_f	pressure-far-field
velocity	velocity-inlet
periodic	
fan	
porous_j	porous-jump
radiator	
mass_flo	mass-flow-inlet
interfac	interface
outflow	
axis	

Remarks

The option to name the zone can be done by placing the *name* before the zone number.

fbc FLUENT boundary conditions - merge phase

fbc fset *face_set type [name] zone_#*

where *type* can be

interior	
wall	
pr_inlet	pressure-inlet
inlet_ve	inlet-vent
intake_f	intake-fan
pr_outle	pressure-outlet
exhaust_	exhaust-fan
outlet_v	outlet-vent
symmetry	
per_shad	periodic-shadow
pr_far_f	pressure-far-field
velocity	velocity-inlet
periodic	
fan	
porous_j	porous-jump

radiator	
mass_flo	mass-flow-inlet
interfac	interface
outflow	
axis	

Remarks

Use the **fset** and the **fseti** commands in the part phase and the **fset** command in the merge phase to create a face set first. Then apply the boundary condition to the **fset** using the **fbc** command. You can also use the Set Editing window under the Pick Panel to select faces for a set using the mouse. The option to name the zone can be done by placing the *name* before the zone number.

fluent select the FLUENT output format

fluent (no argument)

fluemats define the fluent cell type

fluemats *material_# type* ;
where *type* can be

1	for fluid
2	for solid

Remarks

This command is only available in the control phase.

condition specify type of condition/constraint to be displayed

condition fbc *zone_#*
or
co fbc *zone_#*

Remarks

This command is only available in the merge phase.