Appendix A. Importing Boundary and Volume Meshes

The volume mesh generation scheme of TGrid requires sets of line segments (2D) or triangular and/or quadrilateral elements (3D) defining the boundaries of the computational domain. In addition to the basic capability provided in GAMBIT, data interfaces have been written to popular CAD/CAE software packages for use by TGrid.

A.1 GAMBIT Meshes

GAMBIT can create 2D and 3D surface and volume meshes. See the GAMBIT Modeling Guide for details.

A.2 TetraMesher Volume Mesh

ICEM CFD Engineering writes a RAMPANT file from TetraMesher. This file can be read by TGrid, using the File/Read/Mesh... menu item or the `file/read-mesh` text command. TetraMesher generates tetrahedral volume grids using a recursive subdivision octree scheme that interfaces with ICEM CFD.

A.3 Meshes from Third-Party CAD Packages

A single filter allows you to convert files created by several finite-element packages to the grid file format used by TGrid. This filter, `fe2ram`, allows you to convert surface or volume meshes from ANSYS, I-deas, NASTRAN, PATRAN, or other packages. ARIES files can be converted if they are first saved as ANSYS Prep7 files, as described in Section A.3.5: ARIES Files.

A.3.1 Using the `fe2ram` Filter to Convert Files

When you import one of these types of grid files using the menu items in the File/Import pull-down menu, the changes are transparent and you can follow the instructions provided in Section 6.7: Importing Files. If you choose to convert the file manually before reading it into TGrid, enter the following command:

`utility fe2ram` [*dimension*] *read-format* [*merge*] [*zoning*] [*write-format*] *input-file output-file*

The items enclosed in square brackets are optional.

dimension indicates the dimension of the dataset. Replace *dimension* by `-d2` to indicate that the grid is two-dimensional. For a 3D grid, do not enter anything for *dimension*, because 3D is the default. For a surface mesh, replace *dimension* by `-surface`.

read-format indicates the format of the file you wish to convert. Replace *read-format* by `-tANSYS` for an ANSYS file, `-tIDEAS` for an I-deas file, `-tNASTRAN` for a NASTRAN file, or `-tPATRAN` for a PATRAN file. For a list of conversion capabilities from other CAD packages, type `utility fe2ram -cl -help`.

merge indicates the grid tolerance. The default is 10^{-6} (`1.0e-06`), and you can reset the tolerance to this value by replacing *merge* by `-m`. To set another tolerance value, replace *merge* by `-mTOLERANCE`, where `TOLERANCE` is an appropriate real number value.

zoning indicates how zones were identified in the CAD package. Replace *zoning* by `-zID` for a grid that was zoned by property ID's, or `-zNONE` to ignore all zone groupings. For a grid zoned by group, do not enter anything for *zoning*, because zoning by groups is the default.

write-format indicates the output format for the file you wish to convert. Replace *write-format* by `-oFIDAP7` to write the grid in FIDAP format. To write the grid for use in FLUENT or TGrid, do not enter anything for *write-format*, because it is the default selection.

input-file and *output-file* are the names of the original file and the file to which you want to write the converted grid information,respectively.

For example, to convert the 2D I-deas volume mesh file `sample.unv` to an output file called `sample.grd`, enter the following command:

```
utility fe2ram -d2 -tIDEAS sample.unv sample.grd
```

After the output file has been written, you can read it into TGrid using the File/Read/Mesh... menu item. For volume meshes, the resulting output file can also be read into FLUENT..

i All boundary types are considered as wall zones. You can set the appropriate boundary types in TGrid or in the solver.

A.3.2 I-deas Universal **Files**

For I-deas surface meshes, the filter reads triangular and quadrilateral elements that define the boundaries of the domain and have been grouped within I-deas to create zones. For volume meshes, the filter reads a 2D or 3D mesh that has its boundary nodes *or* 2D boundary elements appropriately grouped to create boundary zones. Do not include nodes and boundary elements in the same group. All boundary zones will be considered wall zones; you can set the appropriate boundary types in TGrid or in the solver. See Section A.3.1: Using the `fe2ram` Filter to Convert Files for information about using the `fe2ram` grid filter.

Recognized I-deas Datasets

The following Universal file datasets are recognized by the grid filter:

Node Coordinates dataset number 15, 781, 2411

Elements dataset number 780 or 2412

Permanent Groups dataset number 752, 2417, 2429, 2430, 2432, 2435

Since TGrid uses linear elements, you should use linear elements to generate the grid inside the mesh areas. If parabolic elements exist in the dataset, the filter ignores the mid side nodes. This assumption is valid if the edges of the element are near linear. However, if this is not the case, an incorrect topology may result from this assumption. For example, in regions of high curvature the parabolic element may look much different than the linear element.

For volume meshes, note that mesh area/mesh volume datasets are *not* recognized. This implies that writing multiple mesh areas/mesh volumes to a single Universal file may confuse `fe2ram`, TGrid, or FLUENT. For 2D volume meshes, an additional constraint is placed on the elements: existence in the Z=0 plane.

Grouping Elements to Create Zones for a Surface Mesh

The `Group` command in I-deas is used to create the boundary zones needed by TGrid. All faces grouped together are listed together in the output as a single zone. In FLUENT, boundary conditions are set on a per-zone basis.

One technique is to generate groups automatically based on mesh areas— i.e., every mesh area will be a different zone in TGrid. Although this method may generate a large number of zones, the zones can be merged in TGrid or in FLUENT. Another technique is to create a group of elements related to a given mesh area manually. This allows you to select multiple mesh areas for one group.

Grouping Nodes to Create Zones for a Volume Mesh

The `Group` command is used in I-deas to create the boundary zones needed by TGrid or FLUENT. All nodes grouped together are listed together in the output as a single zone. It is important *not* to group nodes of internal faces with nodes of boundary faces.

One technique is to generate groups automatically based on curves or mesh areas—i.e., every curve or mesh area will be a different zone in TGrid or FLUENT. Another technique is to create the groups manually, generating groups consisting of all nodes related to a given curve (2D) or mesh area (3D).

Periodic Boundaries

In general, it is difficult to generate a valid grid with periodic boundaries in I-deas. However, a special feature exists in TGrid that allows you to generate a grid in a domain with periodic boundaries. See Section 7.14.5: Creating Periodic Boundaries for further details.

Deleting Duplicate Nodes

I-deas often generates duplicate nodes in the process of creating triangular elements. These must be removed by using either the `remove coincident node` command in I-deas or the Merge button in the Boundary Nodes panel (or the `boundary/merge-duplicates` text command) in TGrid. The node merging process is usually faster in TGrid, but more visual in I-deas.

A.3.3 PATRAN Neutral Files

For PATRAN surface meshes, the filter reads triangular and quadrilateral linear elements that define the boundaries of the domain and have been grouped by named component or identified by property ID's within PATRAN to create zones. For volume meshes, the filter reads a 2D or 3D mesh that has its boundary nodes grouped by named component to create boundary zones. All boundary zones will be considered wall zones; you can set the appropriate boundary types in TGrid or in the solver. See Section A.3.1: Using the `fe2ram` Filter to Convert Files for information about using the `fe2ram` grid filter.

Recognized PATRAN Datasets

The following Neutral file datasets are recognized by the grid filter:

Node Data Packet Type 01

Element Data Packet Type 02

Distributed Load Data Packet Type 06

Node Temperature Data Packet Type 10

Name Components Packet Type 21

File Header Packet Type 25

Grouping Elements to Create Zones

In PATRAN, named components are applied to the nodes to create groups of faces called zones. In FLUENT, boundary conditions are applied to each zone. For instance, all nodes on a curve or patch may be put in a Name Component.

For 2D volume meshes, an additional constraint is placed on the elements: existence in the Z=0 plane.

Periodic Boundaries

In general, it is difficult to generate a valid grid with periodic boundaries in PATRAN. However, a special feature exists in TGrid that allows you to generate a grid in a domain with periodic boundaries. See Section 7.14.5: Creating Periodic Boundaries for further details.

A.3.4 ANSYS Files

For ANSYS surface meshes, the filter reads triangular and quadrilateral linear elements that define the boundaries of the domain and have been grouped within ANSYS using node and element selection. For volume meshes, the filter reads a 2D or 3D mesh that has its boundary nodes grouped within ANSYS using node and element selection. All boundary zones will be considered wall zones; you can set the appropriate boundary types in TGrid or in the solver. See Section A.3.1: Using the `fe2ram` Filter to Convert Files for information about using the `fe2ram` grid filter.

Recognized Datasets

The following datasets are recognized by the grid filter:

NBLOCK node block data

EBLOCK element block data

CMBLOCK element/node grouping

The elements must be STIF63 linear shell elements. In addition, if element data without an explicit element ID is used, the filter assumes sequential numbering of the elements when creating the zones.

Periodic Boundaries

In general, it is difficult to generate a valid grid with periodic boundaries in ANSYS. However, a special feature exists in TGrid that allows you to generate a grid in a domain with periodic boundaries. See Section 7.14.5: Creating Periodic Boundaries for further details.

A.3.5 ARIES Files

ARIES provides a filter to TGrid or you may write a Prep7 file from ARIES and use the fe2ram filter with arguments for an ANSYS file. For more information on importing ANSYS files, see Section A.3.4: ANSYS Files.

In general, to write a Prep7 file within ARIES the following criteria must be met:

- Name the part in the Geom module.
- Create a material or read one from the mat_lib in the Material module. To create a material, you must supply density, Poisson's ratio, and elastic modulus.
- Generate face pressures for the surface in the Environment module. Later, when you write the Prep7 file, these will be transferred to the individual elements.
- Generate at least one restraint in the Environment module.
- Set the element type to be STIF63 (triangular shell elements) and specify some finite thickness.
- Write the Prep7 file, making sure you let it automatically assign the pressure to the elements.

The Prep7 file can be filtered using the ARIES or Fluent Inc. filter, whichever you find most convenient.

A.3.6 NASTRAN Files

For NASTRAN surface meshes, the filter reads triangular and quadrilateral linear elements that define the boundaries of the solution domain. For volume meshes, the filter reads a 2D or 3D mesh. All boundary zones will be considered wall zones; you can set the appropriate boundary types in TGrid or in the solver. See Section A.3.1: Using the fe2ram Filter to Convert Files for information about using the fe2ram grid filter.

Recognized NASTRAN Bulk Data Entries

The following NASTRAN bulk entries are recognized by the grid filter:

GRID single-precision node coordinates

GRID* double-precision node coordinates

CBAR line elements

CTETRA, CTRIA3 tetrahedral and triangular elements

CHEXA, CQUAD4, CPENTA hexahedral, quadrilateral, and wedge elements

Since TGrid uses linear elements, you should use linear elements in the mesh generation process. If parabolic elements exist in the dataset, the filter ignores the mid side nodes. This assumption is valid if the edges of the element are near linear. However, if this is not the case, an incorrect topology may result from this assumption. For example, in regions of high curvature the parabolic element may look much different than the linear element.

For 2D volume meshes, an additional constraint is placed on the elements: existence in the Z=0 plane.

Periodic Boundaries

In general, it is difficult to generate a valid grid with periodic boundaries in NASTRAN. However, a special feature exists in TGrid that allows you to generate a grid in a domain with periodic boundaries. See Section 7.14.5: Creating Periodic Boundaries for further details.

Deleting Duplicate Nodes

NASTRAN often generates duplicate nodes in the process of creating triangular elements. These must be removed by using the Merge button in the Boundary Nodes panel (or the `boundary/merge-duplicates` text command) in TGrid.