15.2. FLUENT – A Software Package for CFD

FLUENT is a state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries. **FLUENT** provides complete mesh flexibility, solving your flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge, and mixed (hybrid) meshes. **FLUENT** also allows you to refine or coarsen your grid based on the flow solution.

This solution-adaptive grid capability is particularly useful for accurately predicting flow fields in regions with large gradients, such as free shear layers and boundary layers. In comparison to solutions on structured or block structured grids, this feature significantly reduces the time required to generate a ``good'' grid. Solution-adaptive refinement makes it easier to perform grid refinement studies and reduces the computational effort required to achieve a desired level of accuracy, since mesh refinement is limited to those regions where greater mesh resolution is needed.

FLUENT is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all made possible. In addition, **FLUENT** uses a client/server architecture, which allows it to run as separate simultaneous processes on client desktop workstations and powerful compute servers, for efficient execution, interactive control, and complete flexibility of machine or operating system type.

All functions required to compute a solution and display the results are accessible in **FLUENT** through an interactive, menu-driven interface. The user interface is written in a language called **Scheme**, a dialect of **LISP**. The advanced user can customize and enhance the interface by writing menu macros and functions.

15.2.1. Structure of the Program

FLUENT package includes the following products:

- **FLUENT**, the solver.
- **prePDF**, the preprocessor for modeling non-premixed combustion in **FLUENT**.
- **GAMBIT**, the preprocessor for geometry modeling and mesh generation.
- **TGrid**, an additional preprocessor that can generate volume meshes from existing boundary meshes.
- filters (translators) for import of surface and volume meshes from CAD/CAE packages such as **ANSYS**, **I-DEAS**, **NASTRAN**, **PATRAN**, and others.

Figure 14.25 shows the organizational structure of these components. Note that a "grid" is the same thing as a "mesh"; the two words are used interchangeably here and throughout this manual.

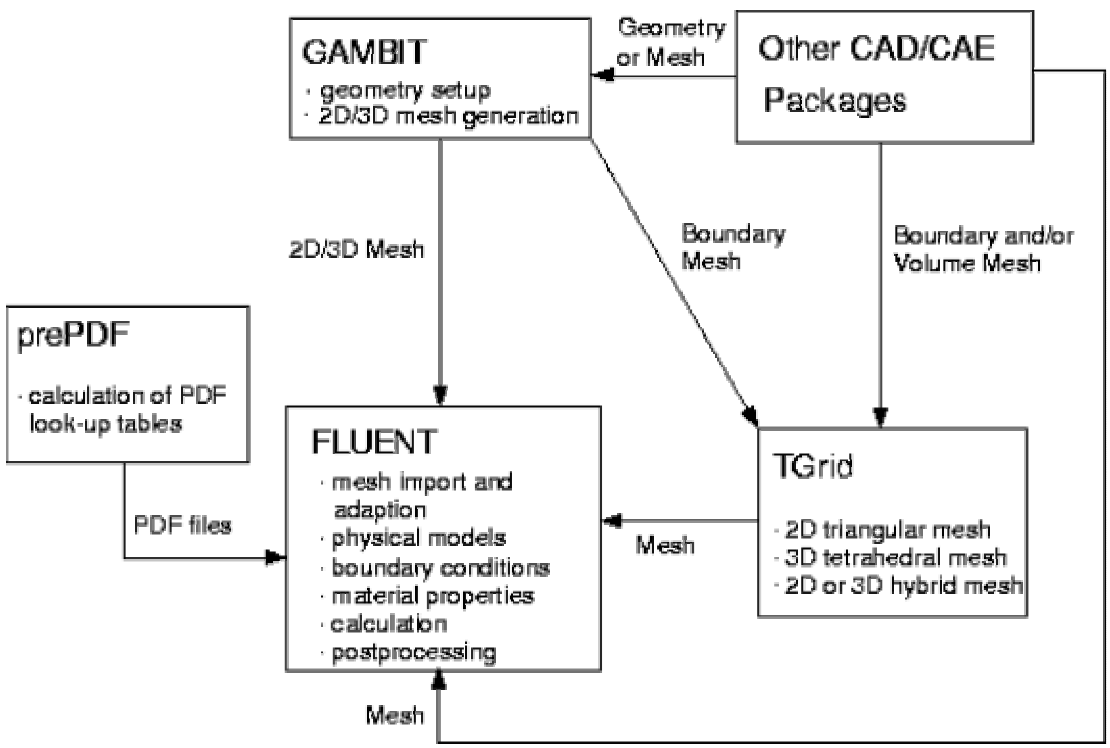

Fig. 15.5. **Basic Program Structure**

One can create required geometry and grid using **GAMBIT**. One can also use **TGrid** to generate a triangular, tetrahedral, or hybrid volume mesh from an existing boundary mesh (created by **GAMBIT** or a third-party CAD/CAE package). It is also possible to create grids for **FLUENT** using **ANSYS** (**Swanson Analysis Systems, Inc.**) or **I-DEAS** (**SDRC**) ; or **MSC/ ARIES**, **MSC/ PATRAN**, or **MSC/ NASTRAN** (all from **MacNeal-Schwendler Corporation**). Interfaces to other CAD/CAE packages may be made available in the future, based on customer requirements, but most CAD/CAE packages can export grids in one of the above formats.

Once a grid has been read into **FLUENT**, all remaining operations are performed within the solver. These include setting boundary conditions, defining fluid properties, executing the solution, refining the grid, and viewing and postprocessing the results.

Note that **preBFC** and **GeoMesh** are the names of Fluent preprocessors that were used before the introduction of **GAMBIT**.

15.2.2. Capabilities of FLUENT

The **FLUENT** solver has the following modeling capabilities:

- flows in 2D or 3D geometries using unstructured solution-adaptive triangular/tetrahedral, quadrilateral/hexahedral, or mixed (hybrid) grids that include prisms (wedges) or pyramids. (Both conformal and hanging-node meshes are acceptable.)
- incompressible or compressible flows
- steady-state or transient analysis
- inviscid, laminar, and turbulent flows
- Newtonian or non-Newtonian flow
- convective heat transfer, including natural or forced convection
- coupled conduction/convective heat transfer

- radiation heat transfer
- inertial (stationary) or non-inertial (rotating) reference frame models
- multiple moving reference frames, including sliding mesh interfaces and mixing planes for rotor/stator interaction modeling
- chemical species mixing and reaction, including combustion submodels and surface deposition reaction models
- arbitrary volumetric sources of heat, mass, momentum, turbulence, and chemical species
- Lagrangian trajectory calculations for a dispersed phase of particles/droplets/bubbles, including coupling with the continuous phase
- phase-change models
- flow through porous media
- lumped-parameter models for fans, pumps, radiators, and heat exchangers
- multiphase flows, including cavitation
- free-surface flows with complex surface shapes

These capabilities allow **FLUENT** to be used for a wide variety of applications, including the following:

- Process and process equipment applications
- Power generation and oil/gas and environmental applications
- Aerospace and turbomachinery applications
- Automobile applications
- Heat exchanger applications
- Electronics/HVAC/appliances
- Materials processing applications
- Architectural design and fire research

In summary, **FLUENT** is ideally suited for incompressible and compressible fluid flow simulations in complex geometries. Fluent Inc. also offers other solvers that address different flow regimes and incorporate alternative physical models. Additional CFD programs from Fluent Inc. include **Airpak**, **FIDAP**, **Icepak**, **MixSim**, and **POLYFLOW**.

15.2.3. Using FLUENT – An Overview

FLUENT uses unstructured meshes in order to reduce the amount of time in generating meshes, simplify the geometry modeling and mesh generation process, model more-complex geometries than one can handle with conventional, multi-block structured meshes, and lets the adaptation of the mesh to resolve the flow-field features. **FLUENT** can also use body-fitted, block-structured meshes (e.g., those used by **FLUENT 4** and many other CFD solvers). **FLUENT** is capable of handling triangular and quadrilateral elements (or a combination of the two) in 2D, and tetrahedral, hexahedral, pyramid, and wedge elements (or a combination of these) in 3D. This flexibility allows to pick mesh topologies that are best suited for a particular application.

One can adapt all types of meshes in **FLUENT** in order to resolve large gradients in the flow field, but one must always generate the initial mesh (whatever the element types used) outside of the solver, using **GAMBIT**, **TGrid**, or one of the CAD systems for which mesh import filters exist.

15.2.4. Physical Models in FLUENT

FLUENT provides comprehensive modeling capabilities for a wide range of incompressible and compressible, laminar and turbulent fluid flow problems. Steady-state or transient analyses can be performed. In **FLUENT**, a broad range of mathematical models for transport phenomena (like heat transfer and chemical reactions) is combined with the ability to model complex geometries. Examples of **FLUENT** applications include laminar non-Newtonian flows in process equipment; conjugate heat transfer in turbomachinery and automotive engine components; pulverized coal combustion in utility boilers; external aerodynamics; flow through compressors, pumps, and fans; and multiphase flows in bubble columns and fluidized beds.

To permit modeling of fluid flow and related transport phenomena in industrial equipment and processes, various useful features are provided. These include porous media, lumped parameter (fan and heat exchanger), streamwise-periodic flow and heat transfer, swirl, and moving reference frame models. The moving reference frame family of models includes the ability to model single or multiple reference frames. A time-accurate sliding mesh method, useful for modeling multiple stages in turbomachinery applications, for example, is also provided, along with the mixing plane model for computing time-averaged flow fields.

Another very useful group of models in **FLUENT** is the set of free surface and multiphase flow models. These can be used for analysis of gas-liquid, gas-solid, liquid-solid, and gas-liquid-solid flows. For these types of problems, **FLUENT** provides the volume-of-fluid (VOF), mixture, and Eulerian models, as well as the discrete phase model (DPM). The DPM performs Lagrangian trajectory calculations for dispersed phases (particles, droplets, or bubbles), including coupling with the continuous phase. Examples of multiphase flows include channel flows, sprays, sedimentation, separation, and cavitation.

Robust and accurate turbulence models are a vital component of the **FLUENT** suite of models. The turbulence models provided have a broad range of applicability, and they include the effects of other physical phenomena, such as buoyancy and compressibility. Particular care has been devoted to addressing issues of near-wall accuracy via the use of extended wall functions and zonal models.

Various modes of heat transfer can be modeled, including natural, forced, and mixed convection with or without conjugate heat transfer, porous media, etc. The set of radiation models and related submodels for modeling participating media are general and can take into account the complications of combustion. A particular strength of **FLUENT** is its ability to model combustion phenomena using a variety of models, including eddy dissipation and probability density function models. A host of other models that are very useful for reacting flow applications are also available, including coal and droplet combustion, surface reaction, and pollutant formation models.