10. MIXED-FLOW PUMP IMPELLER

This tutorial employs the configuration of a mixed-flow pump impeller to demonstrate the use of hybrid hexahedral/tetrahedral meshing capabilities in conjunction with GAMBIT turbo modeling. Such capabilities are particularly useful for meshing turbo models that involve highly twisted blades.

10.1 Prerequisites

Prior to reading and performing the steps outlined in this tutorial, you should familiarize yourself with the steps, principles, and procedures described in Tutorials 1, 2, 3, 4, and 8.

10.2 Problem Description

Figure 10-1 shows the turbomachinery configuration to be modeled and meshed in this tutorial. The configuration consists of an impeller rotor on which are affixed five identical blades, each of which is spaced equidistant from the others on the rotor hub.

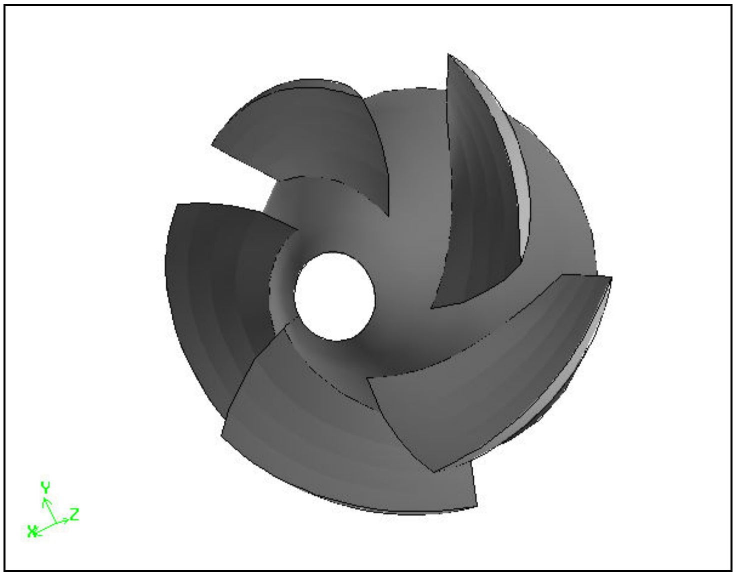


Figure 10-1: Mixed-flow impeller rotor

The overall goal of this tutorial is to create a geometric model of the flow region immediately surrounding one of the impeller blades and to mesh the model using hybrid hexahedral/tetrahedral mesh.

10.3 Strategy

In general, the GAMBIT turbo modeling procedure includes seven basic steps:

Creating or importing edge data that describes the turbo profile

Creating the turbo profile

Creating the turbo volume

Assigning zone types to regions of the turbo volume

Decomposing the turbo volume

Meshing the turbo volume

Viewing the turbo volume

This tutorial illustrates six of the seven steps listed above. The tutorial excludes the turbo decomposition step, because the bulk of the turbo volume is to be meshed using unstructured tetrahedral mesh elements. Turbo volume decomposition is primarily used to facilitate the creation of structured meshes (see Tutorial 9 in this guide).

NOTE: In this tutorial, the turbo-volume viewing operation (Step 7, above) is illustrated in conjunction with the mesh examination step (see "Step 10:Examine the Mesh," below).

10.4 Procedure

1. Copy the file

   `path/Fluent.Inc/gambit2.x/help/tutfiles/rotor-cyl-mod.tur`

   (where `2.x` is the GAMBIT version number) from the GAMBIT installation area in the directory `path` to your working directory.

2. Start GAMBIT using the session identifier "`Pump_Impeller`".

Step 1: Select a Solver

1. Choose the solver from the main menu bar:

   Solver → FLUENT 5/6

   The choice of solver affects the types of options available in the **Specify Boundary Types** *form (see below). For some systems,* **FLUENT 5/6** *is the default solver. The currently selected solver is shown at the top of the* GAMBIT *GUI.*

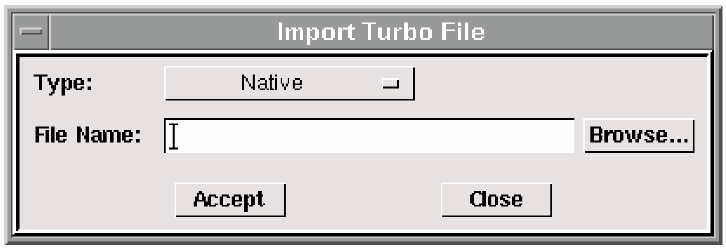
Step 2: Import a Turbo Data File

Turbo data files contain information that GAMBIT *uses to define the turbo profile (see "Step 3: Create the Turbo Profile," below). Such information includes: point data that describes the shapes of the profile edges, edge-continuity data, and specification of the rotational axis for the turbo volume.*

1. Select the **Import Turbo File** option from the main menu bar.

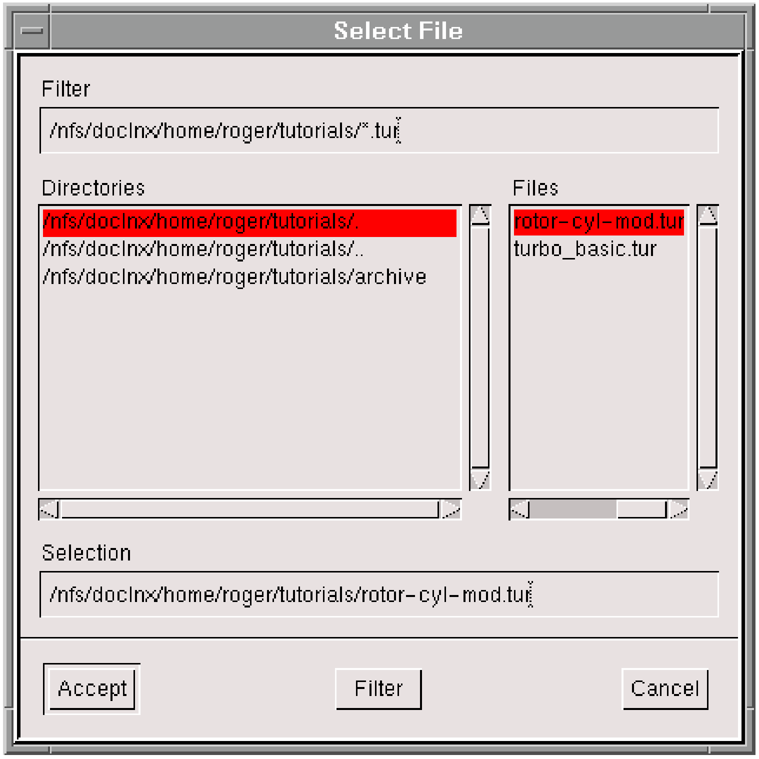
   File → Import → Turbo...

   This command sequence opens the **Import Turbo File** *form.*



2. Click the **Browse...** button.

   This action opens the **Select File** *form.*



a) In the Files list, select `rotor-cyl-mod.tur`.

b) On the **Select File** form, click Accept.

3. On the **Import Turbo File** form, click **Accept**.

GAMBIT *reads the information contained in the data file and constructs the set of edges shown in Figure 10-2. The two straight edges shown in the figure describe the hub and casing for the turbo volume. The five sets of curved edges constitute cross sections of a single impeller blade.*