

## A Novel Design Code for Volutes in Incompressible and Compressible Flow Based On a 2D Inverse Design Approach

### TURBOdesign Volute Versus Conventional Volute Design

Most conventional volute designs are based on 1D methods and on the assumption of conservation of angular momentum and mass flow through the volute section. In many cases the design is based on assuming linear or other variation of area of the volute between throat area and the exit plane. Because of the 1D nature of the design method the volute inlet velocity and flow angle is assumed uniform circumferentially. In reality there is substantial variation in inlet flow angle especially at off-design conditions. In a 1D based design approach there is no way to account for this circumferential flow variation at the inlet of volute.

### Main Features of TURBOdesign Volute

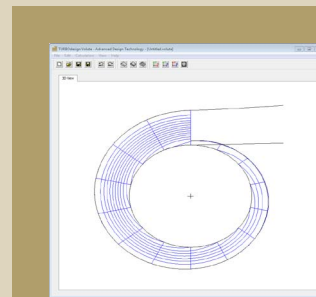
TURBOdesign Volute uses a 2D inverse design approach to compute the shape of the outer wall for a user specified volute cross section.

An initial mesh generated based on 1D design information in  $r$ - $\theta$  plane is computed automatically based on user specified flow field information (see Fig.1).

Different cross section shapes or arbitrary shapes can be specified by modification of control points.

Uniform or circumferentially varying inlet flow angle and velocity are specified together with inlet radius and axial width.

The outer boundary is then computed by solving the streamlines curvature method together with the condition of mass conservation.



The outer boundary is found as the limiting streamline.

Geometry can be output as IGES surface file.

Fig 1. Streamline/Quasi-orthogonal mesh in  $r, \theta$  plane



Fig 2. Volute geometry output.

### Basic Steps

- Input the inlet radius, axial width, transition angle to cover the cut water to exit pipe intersection.
- Number of quasi-orthogonals in transition region and main volute region.
- Select cross section shape for transition region using a pre-set selection of half-circle, half-rectangle or use B-splines to define arbitrary sections.
- Select main volute cross section area from set selection of circle, rectangular, ellipse, trapezoid with straight or rounded edges, half circle or arbitrary by B-spline.
- Select the main diffuser pipe cross section shape as per previous item.
- Specify the diffuser exit angle and tongue (cut-water) radius.
- Specify the distribution of inlet velocity and flow angle.
- Diffuser pipe area and shape can be modified.

### Outputs

- View and compare volute height, shape, velocity and flow rate.
- Streamlines and velocity vectors.
- IGES surface geometry.

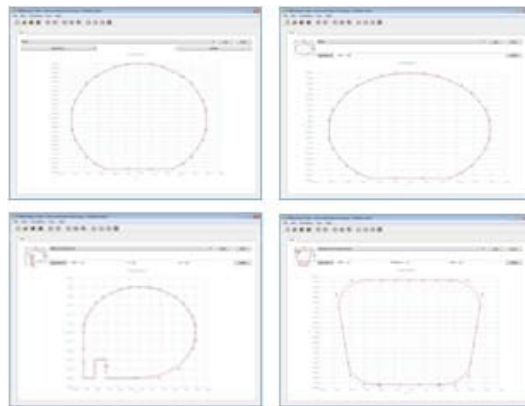
TURBOdesign Volute combines traditional 1D volute sizing with unique 2D inverse design capabilities and robust 3D export features providing designers with a superior platform for the design of high performance volutes for all turbomachinery applications.

### Symmetric and Asymmetric Volute

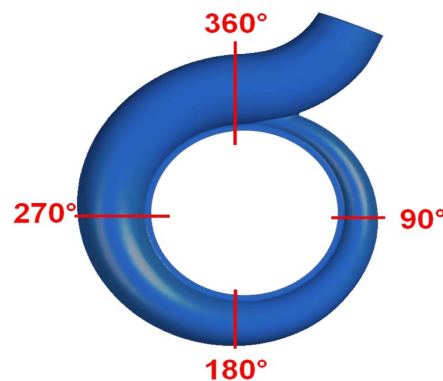
TURBOdesign Volute supports all most common volute configurations such as Symmetric and internal or external Asymmetric volutes. Designers can switch from one configuration to another with ease and estimate the relative difference in packaging sizes and 2D flow calculation associated with their choice.

### Flexible Selection of Volute Cross-Sections

The selection of the volute cross-sections has a major impact on the volute overall sizing and performances, TURBOdesign Volute features a wide range of pre-defined cross sections for immediate volute design. All cross-sections can easily be adapted to fit specific packaging or manufacturing constraints. The code will automatically loft different cross-sections providing a smooth and clean 3D surface file.



3D CAD model for an asymmetric volute as output from TURBOdesign Volute



Comparison of streamlines and velocity contours from the original and redesigned volute shows clear performance improvements

### Volute Area Variation

TURBOdesign Volute was developed specifically to provide users with optimal area variation for specified inlet velocities. Traditionally volutes are designed with a linear variation from inlet to outlet and assuming a constant velocity at the inlet of the volute (1D Method), some designers may also manually experiment with more complex area variations but this process remains iterative and computationally very expensive.

TURBOdesign Volute instead allows designers to input the volute inlet velocity components from the upstream components and automatically calculates optimal area variation from inlet to outlet bringing new answers to complex flow-related environments.

### Tongue and Outlet Pipe Configuration

TURBOdesign Volute provides advanced controls for detailed design of the tongue area and also allows users to model the shape length and typology of the outlet pipe.

### Robust CAE Modelling

The solid and robust export functionality to surface files ensures minimal time is spent in manual CAD work, the surface files output from TURBOdesign Volute can directly be read and meshed for CFD analysis, FEA modelling or solid construction in all major CAE software.

### Higher Performances, Lower Development Times and Costs

TURBOdesign Volute has been successfully applied to all types of turbomachinery applications providing designers with new answers to complex flow phenomena.

