

## 1.1 Version 5.0

### The Evolving Software Products

Simerics CFD software products **Simerics MP** and **PumpLinX** were originally developed with a focus on pumps, valves, compressors and other fluid machinery simulations. Now the software capabilities and algorithms have been extended for much broader applications such as marine hull, vehicle aerodynamics, electronic cooling, and fluid systems. In line with this expanded capability, under the **Version 5.0** Release, the CFD software of Simerics Inc. has been packaged into two products:

- **Simerics-MP** - It is a collection of core 3-D CFD capabilities and physical modules that enables accurate virtual testing for multi-purpose (MP) applications involving single phase or multiphase flow, turbulence, cavitation/aeration, heat and mass transfer, particle and fluid-structure interaction. It includes modeling comprehensive physics, accurate predictions, fast model creation, faster simulation speed and ability to model complex details down to the micro scale.
- **Simerics-MP+** - It includes all the capabilities of Simerics-MP with additional features such as streamlined setup procedures, automated mesh/re-mesh for key components especially moving components and customized data reduction.

The CFD solutions currently offered for vertical industrial applications:

- **Simerics-MP+ for Marine**
- **Simerics-MP+ for Vehicle**
- **Simerics-MP+ for Turbo**
- **Simerics-MP+ for PD**
- **Simerics-MP+ for Valves**
- **Simerics-MP+ for Systems**

The Version 5.0 Products provides major new capabilities as compared with the previous releases. One of the major new features of version 5.0 release is the development of two new parallel algorithms. The MPI parallel running on multiple computers in background, is super fast and super scalable. It has been tested on both Linux and Windows systems. The hybrid DME parallel demonstrates the same performance but runs on a single computer with/without GUI. The original TPP parallel capability will also be available to the customers for the features currently not supported by the new parallel versions. When a user runs DME version with unsupported features, solver will warn the user and prompt to switch to TPP parallel.

In addition to the new parallel capability, Version 5.0 has added four new templates: Marine template, Vehicle template, Piston cooling template, and Fuel tank template. These templates widely extend to applications ranging from marine planing hull porpoising to car gas tank filling.

### New Features & Enhancements

A brief summary of the new features and enhancements are listed below, with more information in the updated manuals and tutorials.

#### Help File

- New help file is created, which is easier to access and read, with additional features like drop-down menus, pictures references, image pop-up etc.
- New tutorials have been added for different physical modules.

#### Templates

This release has the following new templates for different applications:

- **Marine Template** for ship hull simulation with/without propulsion.
- **Vehicle Template** for simulation of automotive vehicle external, under-hood flow and thermal analysis.
- **Piston Cooling Template** for simulation of two phase piston cooling using oil injection.
- **Fuel Tank Template** for simulation of multiphase flow in automotive fuel tank.

The new enhancements in the existing templates are described as follows:

- **Crescent Template**: New mesh is smoother, along with the ability to use CAD surfaces as basis for remesh.
- **Swash Plate Piston Template**: Refinement zone for swash plate piston is added.
- **Dynamics Template**: Moved *Dynamics* module integrals plot access from boundaries to *Dynamics* module.
- **Valve Template**: Valve Gap **Full Closure Model** using user specified (radial) location is added for Valve gap, and a warning will appear, if closure is not found.
- **Scroll Template**: Rotor Mesh is improved.

#### User Interface

- Added new functions to map the values of data points in a table to distributions in boundary/volume, based on the distance from the each grid point to the points in the table.
- Improved reading time of grid with big surfaces.
- Added new button to access global *Expression Editor* in **Properties Panel**.
- Users can split CAD surfaces into two groups using the mouse pointer, with user specified parameters.
- A pop-up window will appear to ask if the user wants to save the results, if user force stops a steady or transient simulation in GUI.
- For steady state simulation, intermediate saving of results is enabled.
- Switch to template mode/extended mode by a single click, instead of multiple clicks, for cases with more than one templates.
- Added stop and save feature while terminating a job using "Ctrl-C", if a file named "project\_name.spro.save\_and\_stop" exists when running in batch mode.
- Users can define text labels (through expression with/without variables) to be viewed in the **Display Panel**.
- Users can specify the background color for the **Display Panel**.

#### Mesher/Pre-Processing

- Added advanced option for directional grid compression.
- Users can create template for a problem and reuse the settings for meshing, modules, boundary/volume/initial conditions, etc. They can automatically build the mesh and setup the model by using only the parameters specified in the project file from scratch or from an existing model. This feature also enables a design/optimization software to do automatic geometry optimization without user interference.
- Users can build a part of the volume by adding symmetry side during the interior/exterior mesh build setup.
- Added options to keep biggest volume or combine all the volumes, while multiple volumes are generated during meshing.
- Added options to automatically combine sub-features to corresponding regular boundary.
- Automatically detect sheet metal CAD surface and recue to single layer surface.
- Allows user to set mesh name in project file for automatic building of mesh, when project file is loaded.
- Added mouse selection to Geometrical Operations.
- For complicated refinement case change mesh refinement zone function (defined using expression function of  $x, y, z$ ) is added.

- For complicated refinement zone shapes, mesh refinement zone function (defined using expression function of  $x, y, z$ ) is added.
- Changed grid initial splitting, so that the binary tree mesh (general mesher) size can be adjusted almost continuously.
- General mesher is added with relative sizing to global model size (all CAD surfaces) option in addition to relative sizing to selected CAD surfaces.
- Automatically sets (default) grid/volume name from common prefix of selected CAD surfaces.

#### ▾ Solver

- Added two new parallel algorithms:
  - MPI parallel running on multiple computers in background, with Linux or Windows systems.
  - Direct Memory Exchange (DME) parallel running on single computer supporting unlimited number of cores with/without GUI.

The original TPP parallel will also be provided to customer as a temporary solution for the features currently not supported by the new parallel versions.
- Major improvement of heat transfer simulation in *Multiphase* module.
- Added heat exchanger modelling feature.
- Added diameter as a function in particle filter model.
- Added viscosity as a polynomial function of temperature.
- Added gauge pressure system.
- Added two enthalpy models: 1) Cp as a function of temperature 2) Melting/solidifying model.
- Added conductivity (k) as a function of temperature.
- Added mass fraction display variable for phase changes.
- Added six particle erosion models.
- Added particle release on MGI and Interface and more particle output quantities.
- Allows gravity as a function for particle module.
- Added mass fraction averaged viscosity for multiphase module.
- For *Multiphase* module,
  - Added adaptive courant number to speed up explicit VOF simulation.
  - Added courant number distribution output for display.
  - Improved implicit VOF high resolution scheme to enhance sharpness of the phase interface.
- For *Radiation* module,
  - Added transparent boundary condition.
  - Changed the default boundary condition from environment to radiation.
  - Linked open boundary condition to flow inlet/outlet boundary condition.
  - Linked opaque to flow blanked volume.
- Added parcel for particle with mass, and particles can be released in volume.
- Added  $dp/dt$  output for user defined damage model.
- Added directional resistance models, with one or two specified flow directions.
- Added vapor mass rate change model for *Cavitation* module.
- In turbulence model, changed wall turbulent dissipation calculation for  $y^+ < y_{critical}$
- Added pressure jump and fan models at interfaces.

#### ▾ Post Processing

- User can control the average/maximum of distributed values of output in a user defined time cycle.
- User can add user defined (in expression) plot variables.
- Moved access of module specific integrals (ie. Valve displacement) to the corresponding modules.
- Animation frames can be saved in multiple PNG files.
- More output variables for particles on boundaries and volumes are created.
- User can specify the number of colors for color map, display value for each color in the legends, and can also add more color schemes for color map.
- Added more output variables for VOF model.