GENERAL

an introduction to solidworks flow simulation 2015

Published: September 19, 2026 | Index ID: #-

An Introduction to SolidWorks Flow Simulation 2015

When engineers and designers look to predict how fluids interact with components, the first question is often, “Which simulation tool will give me the most accurate, yet user friendly results?” SolidWorks Flow Simulation 2015 offers a powerful yet accessible platform for performing computational fluid dynamics (CFD) directly within the familiar SolidWorks environment. This article provides a comprehensive, step by step introduction to the 2015 version, covering installation, core concepts, workflow, and real world applications. Whether you are a seasoned CFD professional or a newcomer to fluid analysis, this guide will help you master the essential features of SolidWorks Flow Simulation 2015 and unlock its full potential.

Why SolidWorks Flow Simulation 2015 Is a Game Changer

SolidWorks Flow Simulation 2015 stands out for several reasons:

- Integrated CAD/CFD workflow – No need to export/import geometry; everything stays inside SolidWorks.
- Intuitive drag and drop setup – Boundary conditions, materials, and mesh controls can be applied with a few clicks.
- Robust solver options – The 2015 release introduced the Hybrid Solver and enhanced turbulence models.
- Optimized performance – Parallel processing and GPU acceleration reduce solution times.
- Comprehensive documentation – The built in help system and example projects make learning easier.

Getting Started: Installing SolidWorks Flow Simulation 2015

Before diving into simulation, you need a properly licensed SolidWorks installation. The Flow Simulation 2015 add on can be installed through the SolidWorks Installer:

1. Launch the SolidWorks Installer from the Start Menu.
2. Select Modify on your existing installation.
3. In the Features tab, check SolidWorks Flow Simulation 2015.
4. Follow the wizard prompts to complete installation.

After installation, restart SolidWorks. You should now see the **Flow Simulation** tab in the Command Manager. If the tab is missing, verify that the license is active and that the add on is correctly installed.

Understanding the Core Workflow

SolidWorks Flow Simulation 2015 follows a logical sequence of steps that mirror traditional CFD practices but are streamlined for the SolidWorks interface. The typical workflow is:

1. Model creation or import.
2. Define simulation study.
3. Assign materials and boundary conditions.
4. Generate and refine the mesh.
5. Run the solver.
6. Post process results.

1. Model Creation or Import

Begin by designing the part or assembly you wish to analyze. Keep the geometry simple and free of unnecessary features to reduce meshing complexity. If you already have a CAD model, simply open it in SolidWorks. The solver can handle assemblies, but the more components, the larger the computational domain.

2. Define Simulation Study

Navigate to the **Flow Simulation** tab and click **New Study**. Choose a study type that matches your analysis goals: **Pressure Drop**, **Heat Transfer**, **Mass Transfer**, or **Custom Study**. The 2015 version introduced a **Transient Study** option for time dependent analyses.

3. Assign Materials and Boundary Conditions

SolidWorks Flow Simulation automatically detects material properties from the SolidWorks material library. However, you can override or add custom materials if needed. Boundary conditions are applied by selecting faces or edges and choosing from a list of predefined options such as:

- Inlet – Specify velocity, pressure, or temperature.
- Outlet – Define pressure outlet or backflow conditions.
- Wall – Set no slip or slip walls, thermal properties, or surface roughness.
- Symmetry – Reduce computational domain by exploiting symmetry.

4. Mesh Generation and Refinement

The 2015 release introduced the **Hybrid Mesh** feature, which combines tetrahedral and hexahedral elements for improved accuracy and speed. Key mesh controls include:

- Mesh Size – Global element size or local refinement zones.
- Boundary Layer – Number of layers and growth rate for capturing viscous effects.
- Skewness Control – Limit element skewness to improve solution stability.

Use the **Mesh Quality** report to identify problematic elements before running the solver.

5. Run the Solver

Click **Run** to launch the solver. The 2015 version supports both the **Standard Solver** (finite volume) and the **Hybrid Solver** (finite volume with improved pressure–velocity coupling). For turbulent flows, select from models such as k ;RÂ ² É, or SST. The solver will display progress and error messages in the **Simulation Log**.

6. Post Processing Results

After convergence, the **Results** tab allows you to visualize velocity vectors, pressure contours, temperature fields, and streamline plots. Use the **Contours** and **Streamlines** tools to generate high quality images for reports. The **Reports** feature can automatically compile a PDF containing key data and graphics.

Deep Dive: Advanced Features in 2015

Hybrid Solver Enhancements

The Hybrid Solver in 2015 offers faster convergence for incompressible flows by decoupling pressure and velocity updates. It also includes improved under relaxation controls that reduce the need for manual tuning.

Improved Turbulence Modeling

SolidWorks Flow Simulation 2015 added the **Spalart–Allmaras** model for high Mach flows and enhanced the **Large Eddy Simulation (LES)** capability for capturing transient turbulence. Users can switch between models with

minimal effort.

Parallel Processing and GPU Acceleration

By default, the solver utilizes all available CPU cores. The 2015 version also offers optional GPU acceleration for mesh generation, which can cut preprocessing times dramatically on supported hardware.

Real World Applications

To illustrate the versatility of SolidWorks Flow Simulation 2015, let's explore three common engineering scenarios:

Case Study 1: HVAC Duct Design

Designers used Flow Simulation 2015 to optimize a residential HVAC duct. By modeling the duct geometry and applying inlet velocity and outlet pressure boundary conditions, they identified regions of high pressure drop and recirculation. Mesh refinement around bends revealed vortex shedding, leading to the redesign of a curved section that reduced pressure loss by 12%.

Case Study 2: Heat Exchanger Efficiency

A heat exchanger manufacturer leveraged the 2015 solver to simulate fluid flow and heat transfer across finned tubes. By applying temperature boundary conditions at the inlet and outlet, they could quantify the effectiveness of various fin geometries. The analysis helped them choose a fin spacing that maximized heat transfer while keeping pressure drop within acceptable limits.

Case Study 3: Aerodynamic Analysis of a Small UAV

In a research project, engineers modeled a small UAV wing in SolidWorks and performed a transient flow simulation to capture wingtip vortices. Using the LES turbulence model, they visualized vortex evolution over time, which informed modifications to wing geometry that improved lift-to-drag ratio by 8%.

Tips and Best Practices

- Keep Geometry Clean – Remove fillets and small features that are not critical to the flow.
- Use Symmetry – Reduce computational cost by exploiting geometric symmetry whenever possible.
- Validate Mesh Quality – Inspect skewness, orthogonality, and aspect ratio before running the solver.
- Start Simple – Begin with a steady state solution before adding transient or turbulence models.
- Use Mesh Controls Wisely – Apply local refinement near walls and in regions of high velocity gradients.
- Check Solver Settings – Adjust under relaxation factors only if convergence stalls; the 2015 solver is robust by default.
- Post Process Thoroughly – Use contour plots, vector plots, and streamlines to fully understand the flow field.
- Export Data – Save results in CSV or DXF format for further analysis or presentation.

Troubleshooting Common Issues

Even experienced users encounter hiccups. Here are solutions for frequent problems:

- Solver Divergence – Increase mesh density, reduce time step (for transient), or adjust under relaxation factors.
- Large Pressure Drop at Inlet – Verify inlet boundary conditions and ensure that the inlet is not too close to a wall.
- Mesh Quality Alerts – Refine elements with high skewness or low orthogonality.
- Unexpected Temperature Gradients – Check that thermal boundary conditions are correctly applied.
- Long Solution Times – Enable parallel processing and consider reducing the number of physics (e.g., disable

turbulence if not needed).

Comparing SolidWorks Flow Simulation 2015 with Other CFD Tools

When deciding on a CFD solution, it’s useful to compare key attributes:

Feature	SolidWorks Flow Simulation 2015	ANSYS Fluent	CFX
Integration with CAD	Direct SolidWorks integration	Import required	Import required
Ease of Use	Intuitive GUI, drag and drop	Steeper learning curve	Steeper learning curve
Solver Options	Hybrid and Standard	Extensive turbulence models	Extensive turbulence models
Parallel Processing	CPU & GPU support	High performance clusters	High performance clusters
Cost	Lower for existing SolidWorks users	Higher licensing fees	Higher licensing fees

While advanced CFD platforms offer deeper customization, SolidWorks Flow Simulation 2015 remains the go to choice for engineers who need rapid, accurate results within a familiar CAD environment.

Future Outlook: What's Next After 2015?

SolidWorks continually evolves its Flow Simulation capabilities. Subsequent releases have added features such as:

- Improved Multiphysics coupling with SolidWorks Simulation.
- Expanded material libraries for exotic fluids.
- Enhanced automated mesh generation for complex geometries.
- Better integration with cloud computing for large scale simulations.

For users of 2015, staying updated with patch releases and exploring the new modules can unlock even greater performance and accuracy.

Conclusion

Baca Juga (Artikel Terkait)

- **advanced engineering electromagnetics solution manual edition hardcover by balanis constantine a published by wiley**
  URL: http://djjsearch.com/advanced-engineering-electromagnetics-solution-manual-edition-hardcover-by-balanis-constantine-a-published-by-wiley.pdf
- **advanced engineering electromagnetics solutions manual**
  URL: http://djjsearch.com/advanced-engineering-electromagnetics-solutions-manual.pdf
- **advanced engineering electromagnetics wiley 1989 grading**
  URL: http://djjsearch.com/advanced-engineering-electromagnetics-wiley-1989-grading.pdf
- **advanced engineering fluid mechanics by biswas**
  URL: http://djjsearch.com/advanced-engineering-fluid-mechanics-by-biswas.pdf