CFdesign Motion Module

Inside the CFdesign Motion Module

Many engineering applications require an understanding of how liquids and gases interact with solid objects that are moving. The CFdesign Motion Module allows you create a virtual prototyping environment that simulates how components of pumps, fans, blowers, compressors, valves, pistons, and other mechanical devices interact with and respond to flow. All the physical effects of the motion as well as the time-history are output to facilitate reviewing the results and creating animations for visual studies and design comparisons.

License Requirements: The CFdesign Base System and Advanced Module are requisites for the Motion Module.

Motion Simulation Scope

The CFdesign Motion Module allows you to simulate the interaction between flow and moving parts in your CAD models.

Types of Motion

- Linear
- Angular
- Rotating/Turbomachinery
- Combined Linear and Angular
- Combined Orbital and Angular
- Nutation
- Sliding Vane
- Unconstrained Motion

Time History Outputs

- Force
- Torque
- Rotational speed
- Linear velocity
- Angular velocity
- Linear displacement
- Angular displacement
- Pressure rise/drop
- Temperature rise/drop
- Flow rate
- Animation of motion

Because of the broad diversity of engineering applications that incorporate Motion, we'll explore how the CFdesign Motion Module makes simulating motion straightforward and powerful. The basic process consists of two steps:

1. Define ***how*** the object moves.
2. Define ***what*** moves the object.

Defining *How* the object moves

The Motion Module contains three primary ways to define the path that an object in motion will take. Some are quite broad, and are used in numerous applications; others are more specific, and are used for more specialized applications. The three together, however, form a comprehensive, highly flexible modeling solution that spans a wide diversity of industries.

Method 1: Pre-Defined Motion Paths

If the direction of motion is known (and it often is in many manufacturing and industrial applications), use a Pre-Defined Motion Path to define where the object moves. In most of the Paths, the velocity can either be prescribed (as in a mechanically-driven system) or driven by the flow. The Motion Module contains these pre-defined paths:

- **Linear**: Linear motion is the motion of a solid in a straight line. Examples include pistons, linear actuators, valves, and items on a conveyor process
- **Angular**: Angular motion is the rotation of an object about a centerline. Examples include positive displacement pumps, gear pumps, trichodal pumps, and check valves.

- **Combined linear and angular**: In Combined motion, the object translates linearly along a path while rotating about an axis. Examples of Combined motion are found in flow meters that contain a piston that oscillates and rotates about its direction of travel at the same time. Additionally, many manufacturing components exhibit combined motion behavior.

- **Orbita**l: In Orbital motion, the object rotates, while orbiting about an axis parallel to its axis of rotation. A typical example is a pump shaft that rotates with an eccentric orbit (“whirl”) component.

- **Nutating**: An object with a Nutating motion wobbles about its reference axis, while maintaining its angular position relative to the axis. Several types of liquid flow meters use Nutating motion.

- **Sliding vane**: In a Sliding Vane motion, an object rotates, while translating in a radial direction. The direction of travel changes at every angular position due to the rotation. The most common examples are found in sliding-vane positive displacement pumps.

What can you do with Pre-defined Motion Paths?

Pre-defined motion paths are great for moving something in a known direction through a fluid. They provide valuable insight into the interaction between an object and the surrounding fluid. Additional benefits include the ability to:

- Visualize and animate how the flow reacts to moving objects.
- Understand if moving objects create excessive flow disturbance, which could lead to contaminant dispersal.
- Compute the resultant forces on moving objects as they pass through a fluid.
- Extract data points such as local velocity values, part temperatures, and forces directly from the model.
- Use the Decision Center to compare results from multiple designs and scenarios, and make informed design decisions.

Method 2: Turbomachinery

A major challenge faced by pump and turbine designers is predicting device performance across a wide range of operating conditions. Prototype-benchmark testing is a technique that has been used for many years, but is time consuming and expensive. Many companies have discovered the benefits of Virtual prototyping, and have leveraged CFdesign to reduce costs and time to market.

Examples of common turbomachines include:

- Centrifugal pumps
- Axial fans
- Centrifugal compressors and blowers
- Hydraulic turbines

What can you do with turbomachinery?

With the CFdesign Motion Module, users have developed significant insight into the performance of their turbomachinery devices, while reducing costly prototype testing. The Turbomachinery tools provide the ability to:

- Apply a known rotational speed to devices such as pumps and fans. Alternatively, vary the rotational speed with time to simulate cyclical operation.
- Impel flow-driven rotation in turbines, and extract the operating rotational speed.
- Extract performance data such as the head-capacity characteristic curve, hydraulic torque, power, and efficiency.
- Animate rotor or impeller rotation to visualize the blade-passage flow distribution and areas of hydraulic inefficiency.
- Compare designs and make design decisions based on device performance, flow distribution, and other critical parameters.

Method 3: Unconstrained Motion

Unlike the other motion types, Unconstrained Motion allows motion in any direction. This is the most flexible of the motion types, and can be used to simulate unconstrained (or partially constrained) movement of objects within an active flow field.

Unconstrained Motion is always flow driven, and is limited as necessary by enabling or disabling any of the six degrees of freedom. Objects moving with unconstrained motion can collide with other solid parts, walls, and even moving solids. The resulting bounce is computed based on the angle of impact, forces, and torque to determine the altered path of the object.

What can you do with Unconstrained Motion?

Because unconstrained motion is the most flexible way to define how an object moves, it is developed with a full complement of tools to fine-tune the motion:

- Limit motion in the six degrees of freedom with constraints
- Control the movement with externally-applied forces. Forces can either be constant or vary by orientation.
- Study the influence of gravity (or other body force) on the moving objects.
- Control the initial path and/or velocity by assigning linear and angular velocities at the onset of the simulation
- Simulate collisions with other solid objects (moving and stationary) and with walls.
- Animate motion results and share with others with Dynamic Images, AVI, and MPEG files
- Determine the resultant velocity, displacement, force, and torque for each moving object.

Defining *What* moves the object

In the Motion Module, an object moves either because of forces imparted by the flow or along a constrained path because of an externally-applied mechanism. Both causes of motion are found in engineering applications across many industries.

Method 1: Flow-Driven Motion

In flow-driven motion, the object moves (or stops) in response to fluid impingement or resistance. The object can move along a defined path, or in the case of fully unconstrained motion, freely. A comprehensive set of physical attributes are available to define flow-driven motion:

a. Range and extent of the motion
b. Driving and resistive forces and torque
c. Initial linear and/or angular velocity

In many devices, two or more objects that are driven by the flow are physically connected so that their motions are related. Examples include:

- Hydraulic rams that slide linearly together through multiple cylinders
- Gears in a gear pump rotate in opposite directions at the same rotational speed

There are a huge number of real-world devices that are subject to flow-driven motion. Two common examples are **check valves** and **turbines**.

Flow-Driven Example 1: Check Valves

In a **check valve**, a spring typically holds the valve closed until the flow rate reaches a level that can overcome the force or torque of the spring, and pushes the valve open. If the flow reverses direction, the valve closes, thus ensuring the flow moves only in one direction. To model this with the CFdesign Motion Module, define the valve with a spring force that pushes in the direction opposite to the flow. Define the amount of force that causes it to open (usually known as a spring constant), and define the limits of the valve movement to simulate the physical stops in the device.

By varying parameters such as the flow rate and critical dimensions of the valve, a check valve design study provides a great deal of insight into the performance of different design parameters and conditions:

- The amount of flow required to open the valve for a range of spring rates and valve sizes
- The speed at which the valve opens for a range of flow rates
- How quickly the valve responds to perturbations in the flow rate

Flow-Driven Example 2: Turbines

In a **turbine**, hydraulic or pneumatic energy is converted into rotational mechanical energy. To model a turbine with the CFdesign Motion Module, define a flow across the turbine, an axis of revolution, and either a resistive torque or inertia to simulate a load (such as a generator).

A turbine design study delivers performance data and important information, including…

- Steady-state rotational speed for a known load
- Relationship between rotational speed and load
- Resultant torque for a given rotational speed
- Torque, power, and efficiency
- Effect of blade angles on the turbine performance and the flow distribution within the blade passage

---

Method 2: Mechanically-Driven Motion

In many devices, objects in motion do not react to the flow, but instead move in a completely specified direction over a defined distance. In other words, a motor or other mechanism causes the object to move (slide, rotate, oscillate, etc.), and the flow reacts to it.

The CFdesign Motion Module provides a comprehensive set of tools for defining this type of motion, including…

a. Constant velocity motion
b. Definition of reciprocating or oscillating motion
c. Tabular specification of position as a function of time.
d. Constant or time-varying rotational speed

There are a huge number of real-world devices that are subject to flow-driven motion. Two examples often found in engineering applications are **hydraulic cylinders** and **pumps**.

Mechanically-Driven Example 1: Hydraulic Rams

In a **hydraulic cylinder**, hydraulic fluid forces a piston within a cylinder to move in a linear direction at a known velocity and distance. While possible to simulate such motion as flow-driven, a more convenient way is to define the motion with a **Linear** motion using a prescribed distance and specified stroke time. The cylinder moves only in the specified direction at the defined velocity.

By varying such parameters as piston diameter, ram velocity, and the fluid properties, a design study of a hydraulic ram provides valuable insight into the performance of different design parameters and conditions including…

- The effect of the piston and other moving parts on the fluid
- The amount of force needed to move the piston along its defined path
- The effect of the piston and cylinder dimensions on the piston force and the flow of hydraulic oil

In many manufacturing processes, simulating mechanically-defined motion provides valuable design insight into the effects of the motion on the environment air flow, which is especially critical when contaminate dispersal is a factor.

Mechanically-Driven Example 2: Pumps

In a **pump**, mechanical energy is converted into rotational energy, which in turn elevates the pressure of the working fluid. Liquid enters axially, is discharged radially, passes through a volute, and is discharged. To simulate a pump the CFdesign Motion Module, assign the pressure head across the inlet and outlet, and specify the rotational velocity of the impeller. The rotational speed can be varied with time to simulate a physical (non-impulsive) start-up.

A pump design study delivers performance data and important information, including…

- Performance characteristic such as the run-out flow rate, the flow rate at a given head, or the head at a given flow rate
- Hydraulic torque
- Blade-to-blade flow distribution to identify jet-wake phenomena, reversed flow, and other potential flow inefficiencies
- The effect of the impeller-volute interaction on the flow