

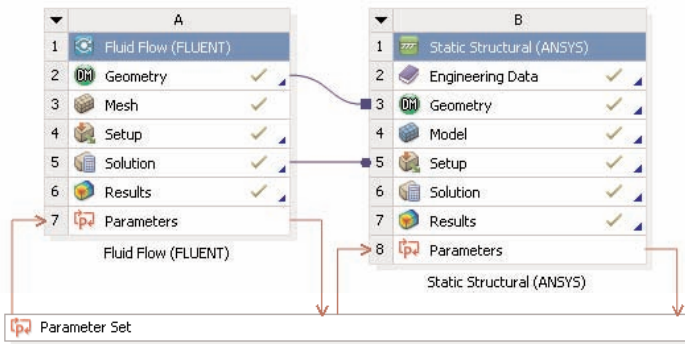
Optimizing Options

Technologies converge in ANSYS Workbench for parametric fluid structure interaction analysis.

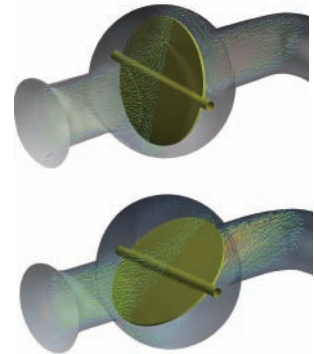
By Aashish Watave, Technology Specialist, ANSYS, Inc.

The ANSYS Workbench environment provides a convenient way to conduct parametric studies for geometry shape and size variations as well as boundary conditions, which can be extended to cover multiphysics simulations within a single working environment. In this example, the effect of a butterfly valve position on the fluid flow rate and pressure drop across the valve is calculated using ANSYS FLUENT software, and the effect of fluid pressure on the valve deformation is determined using ANSYS Structural capabilities.

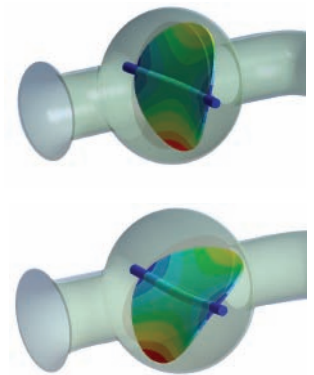
The ANSYS Workbench project schematic shows the two analysis systems used in this project and how they are connected to share data with each other: a fluid flow analysis system using ANSYS FLUENT and an ANSYS static structural system. The fluid and structural systems share the same geometry. Results from the fluid flow analysis are used to assign boundary conditions for the structural analysis, and the blue lines in the project schematic indicate information sharing between different analysis systems.



ANSYS Workbench project schematic

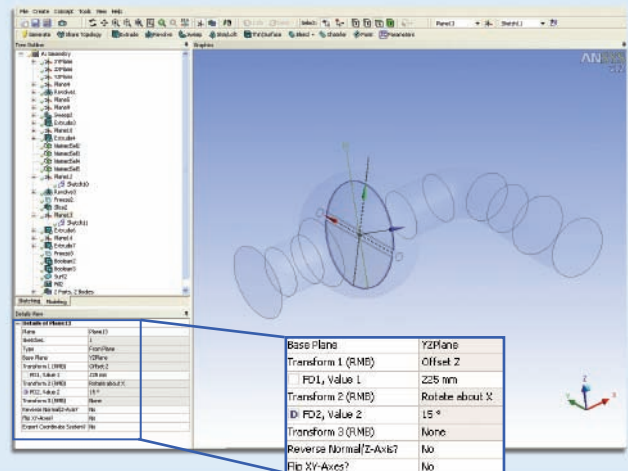


Fluid flow simulation of a butterfly valve with disk angle of 15 degrees (top) and 50 degrees (bottom)



Deformation of a butterfly valve (magnified) as a result of fluid pressure with disk angle of 15 degrees (top) and 50 degrees (bottom)

- The geometry can be defined in two ways: It can be imported from a CAD system and, if necessary, simplified, repaired or prepared for simulation using the ANSYS DesignModeler tool within ANSYS Workbench; alternately, the geometry can be created in ANSYS DesignModeler. Certain geometric inputs can be set as parameters in the tree outline view. For the butterfly valve, the disc angle is defined as an input parameter and a 15-degree initial value was assigned.



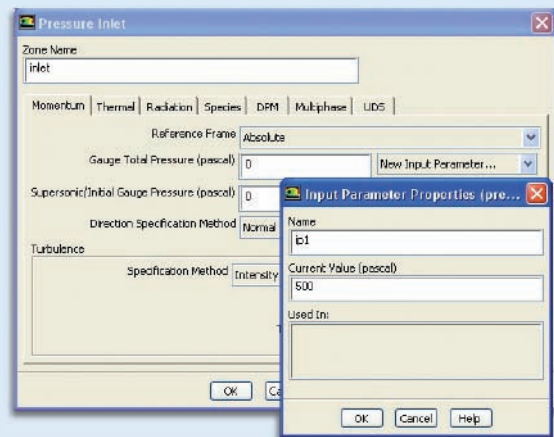
2 All the design point parameters are defined in this manner using the design point table, which is created as soon as the first parameter is defined. This table is represented by a bar called Parameter Set on the project page. It may be viewed by double clicking on the parameter set bar. Arrows in the ANSYS Workbench project schematic pointing toward the analysis system indicate input

Table of Design Points													
	A	B	C	D	E	F	G	H	I	J	K	L	M
1	Name	P1 - valveage	P2 - valvearea2D	P6 - ip1	P8 - ip1a	P9 - impu(pressure)	P4 - avgpsurf	P5 - deltaP	P5 - massflow	P10 - Total Deform...	P11 - Total	Exported	Note
2				Pa	Pa	Pa	Pa	Pa	kg m ⁻¹ s ⁻¹	mm	mm		
3	Current	15	150	500	400	201.4	365.82	446.98	0.13129	0.007402	5.0669E-05		
4	DP 1	20	150	500	400	202.71	365.13	448.06	0.12984	0.007859	5.0666E-05		
5	DP 2	30	150	500	400	206.1	356.74	450.50	0.12905	0.008015	5.0104E-05		
6	DP 3	40	150	500	400	205.92	355.29	452.6	0.12977	0.008206	5.0194E-05		
7	DP 4	50	150	500	400	205.1	356.32	454.58	0.12148	0.008686	5.1111E-05		
8	DP 5	60	150	500	400	203.39	348.01	453.99	0.1224	0.007945	4.9428E-05		
9	DP 6	70	150	500	400	202.00	345.76	447.71	0.13008	0.004725	4.1917E-05		
10	DP 7	20	150	1000	800	412.46	733.99	899.31	0.18883	0.003324	9.8104E-05		
*													

parameters, and those pointing toward the parameter set bar indicate output parameters from the given system.

3 For the fluid flow simulation, the geometry is first meshed using the ANSYS Meshing tool. Zones that are unnecessary for fluid flow simulation, such as the valve body thickness, are suppressed when meshing the fluid zones. However, the valve body and valve disc thickness are maintained as part of the actual geometry since they are required for the ANSYS structural analysis. In ANSYS Meshing, the boundary zones (such as inlet and outlet) required for fluid flow analysis are identified using named selections. These named selections are persistent throughout the project and appear in other tools, such as ANSYS FLUENT. Setup of the problem in ANSYS FLUENT proceeds as normal with input parameters defined as part of the problem setup. For our example, inlet pressure is defined as a named selection in ANSYS

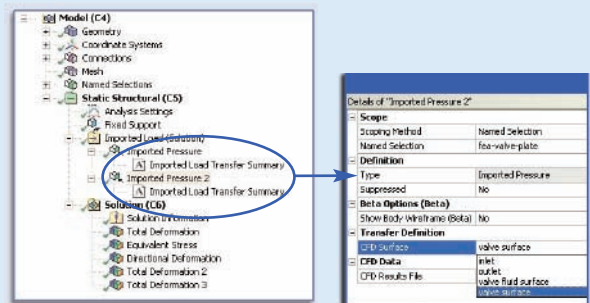
Meshing, an input parameter in ANSYS FLUENT, and is shown in the design table on the project schematic as P8-ip1.



4 The results obtained from the ANSYS FLUENT solution can be post-processed using the CFD-Post tool by clicking on the Results cell in the project schematic. Expressions can be used to define output parameters such as pressure drop (ΔP) or average

surface pressure (on a given surface; for example, the valve disc). The output parameters defined in CFD-Post are available in the design table as output parameters. Alternatively, output parameters can be defined in ANSYS FLUENT.

5 The static structural analysis shares the same geometry used by the fluid flow system. Geometry parts that are not required for structural simulations, such as fluid flow volumes, are suppressed in the ANSYS Meshing tool. For each design point, the pressure distribution on the valve body and valve disc obtained from the ANSYS FLUENT results is used as a boundary condition for the ANSYS Structural simulation, with the data for the corresponding zones being mapped and interpolated between the two systems accordingly. Information such as maximum deformation and maximum/minimum stress levels from the static structural analysis are defined as output parameters. After completing the analysis for a base point (Design Point No. 0), additional design points can be directly defined in the design point table. Subsequent simulation results are obtained by updating all or selected design points from the table. ANSYS Workbench



updates the geometry and mesh and then performs both simulations, including automatically updating the results in the design point table. The project instances for each design point can be saved separately using the Export check box in the design table. ■

