Structural & Fluid

(Coupled Multiphysics FSI)

Practical Simulation Methods

August 6th, 2025

Agenda

1 Intro to Epsilon?

2 Intro to SimuTech Group?

3 Fluid Structure Interaction (FSI)

- Applications in Industry
- Implementation / FSI Simulation Challenges
- Troubleshooting / Convergence
- Dynamic Meshing
- Prestressing Structure

4 Q&A

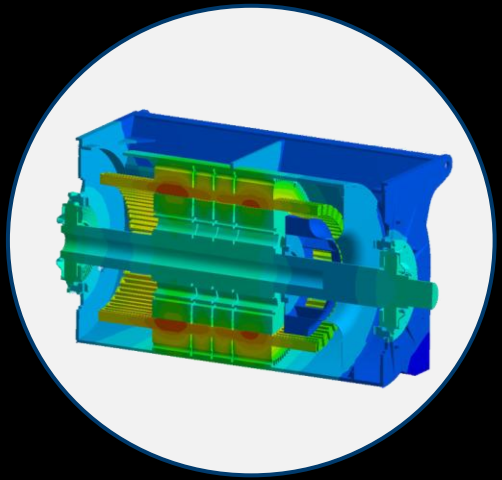
About Epsilon

Epsilon FEA was formed in 2008 to provide a new class of Engineering Service utilizing the Finite Element method and related CAE tools.

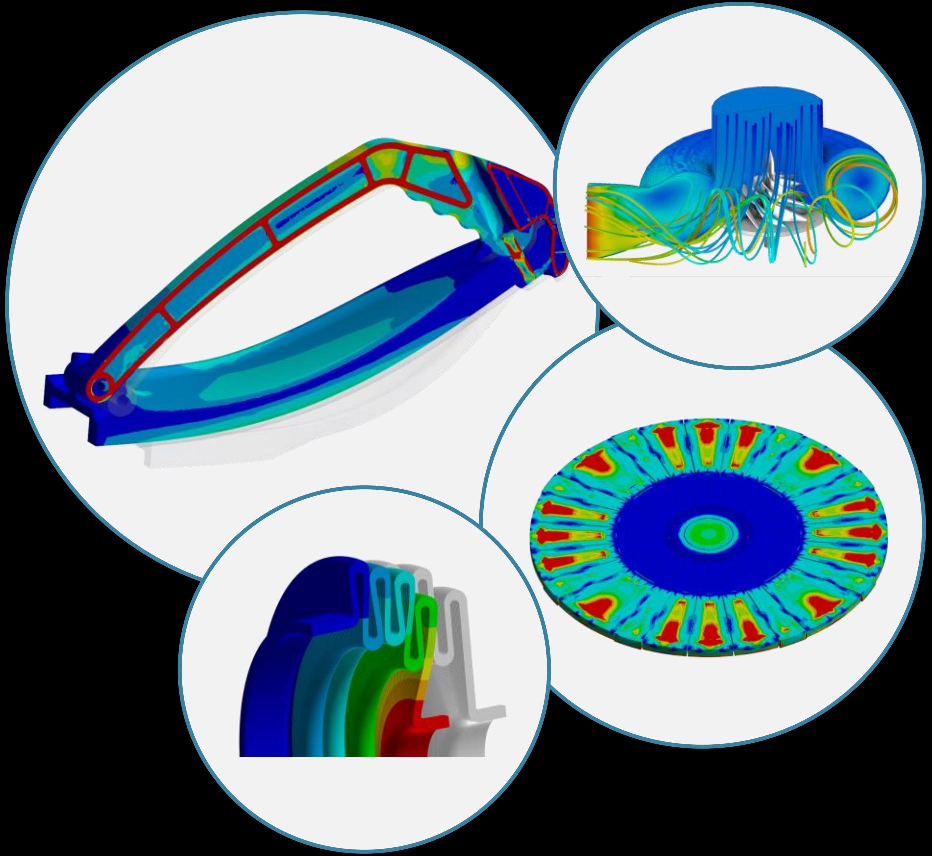
Our Core Values

Epsilon focuses on three cumulative characteristics of our services:

- ✓ **Exhibit Excellence with the Simulation Tools**
- ✓ **Infuse Technology into Customer Design System**
- ✓ **Communicate Thoroughly and Clearly**



What We Do



Ansys

Using ANSYS tools (Workbench, Fluent, ICEPAK, LS-DYNA, etc.) and High Performance Computing (HPC), Epsilon's analysts have made FEA / CFD a career-long focus. Including up-to-date simulation methods this expertise results in superior accuracy of predictions as well as task efficiency for lower total project cost.

Our strategic partners rely on Epsilon FEA low-cost simulation and support of advanced engineering challenges.

Why Epsilon?

Superior Engineering Analysis

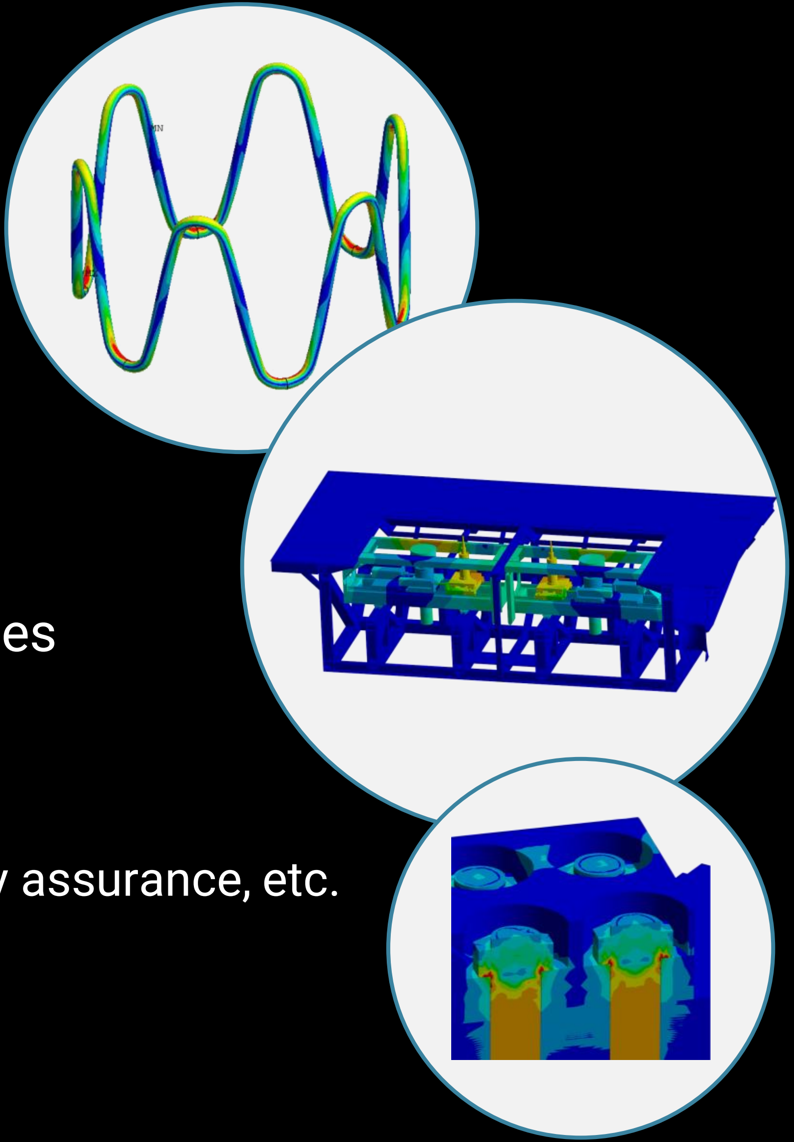
- Simulation experts with diverse industry experience
- Subject matter mastery

Affordable Solutions

- Rates often lower than internal fully burdened labor rates
  - Includes access to high-cost simulation software
- Big Business Interface
  - Detailed invoicing/SOWs, updated toolset, insured, quality assurance, etc.

Big Business Process / Small Business Service

- Rod's Cell Phone: 612-819-5288



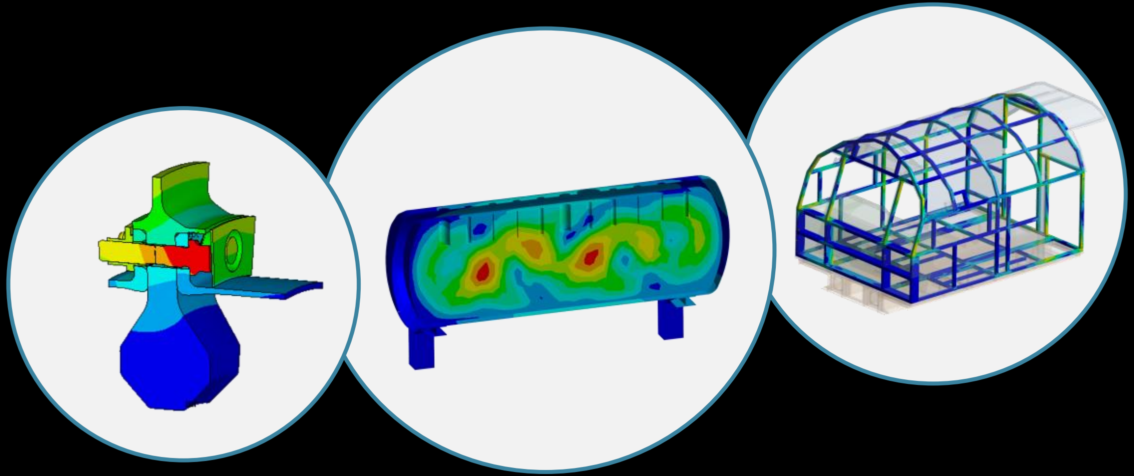
Who Do We Serve?

Load-Leveling

- Analyst is a team member, not a black box
  - Interface with same Epsilon analyst to leverage past experience
- Open and frequent communication
- Any new FEA methods/lessons learned are **well** communicated
- Schedule/budget fidelity with frequent status updates
  - Achieved by using the right person, tools, and technical approach

External Expertise

- We infuse up-to-date FEA methods/tools
  - Leverage other industries' FEA innovations
- We are not a software reseller
  - Unbiased tool selection, infrastructure advice
- We share our knowledge, files, and lessons learned



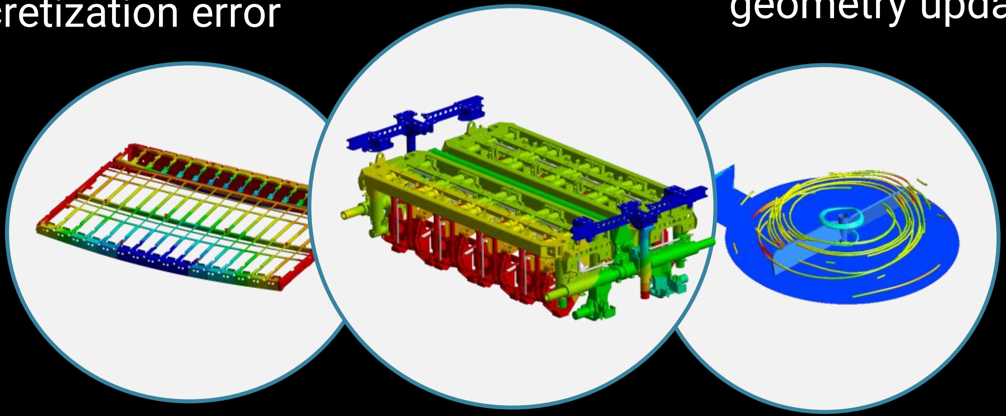
Epsilon's Customers

Accurate Simulation

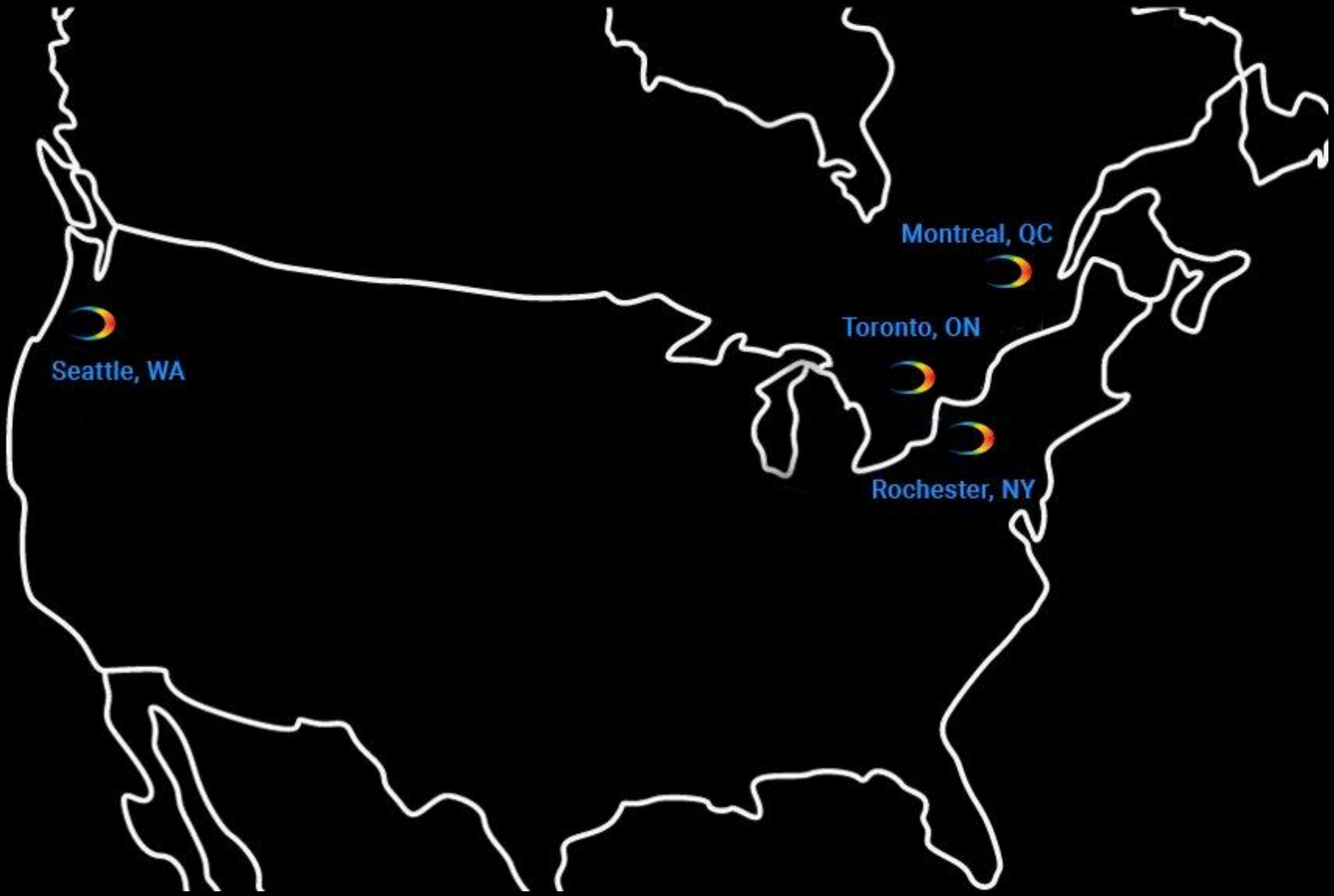
- In-depth knowledge of tools
  - ANSYS® Suite of Multi-Physics software
- Experience with industry successes/failures
  - Aerospace, Rotating Machinery, Electronics, Manufacturing, Packaging, etc.
- Validation with calibration runs and hand-calcs
  - Experience assessing discretization error

Affordable Simulation

- Low hourly rates and/or fixed-price estimates
- Specialized, experienced engineers
- Detailed statements of work, scope, and budget tracking
- Automation (APDL, CAD-associativity)
  - Accommodates shifting inputs, materials, minor geometry updates, etc.



SimuTech Group Overview



25+ Year Ansys Partnership

#1 Ansys Partner in North America

150+ Employees Across US & Canada

50+ On-Staff Engineers With a Combined 900+ Years of Engineering Experience

SimuTech Group Offerings

Your Trusted Ansys Experts

- CONSULTING
- TRAINING
- 1:1 MENTORING
- ANSYS SOFTWARE LICENSING
- DEDICATED SUPPORT

Physics Types We Are Knowledgeable In

Computational Fluid Dynamics (CFD)

Electromagnetics (EMAG)

Optics & Photonics

Finite Element Analysis (FEA)

Industries We Support

Aerospace

Energy

Space

Automotive

Healthcare

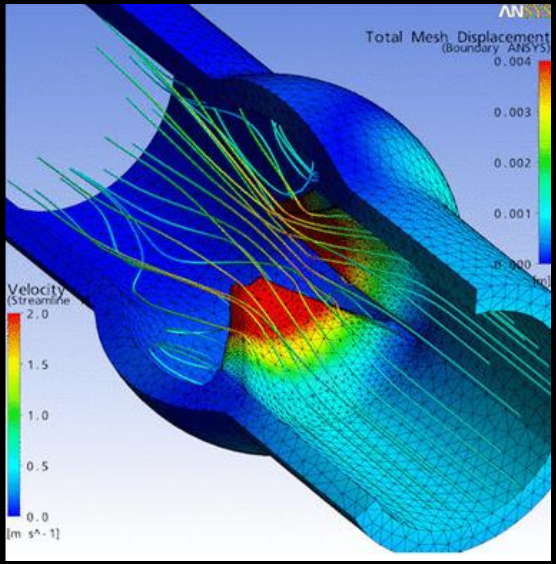
AND MORE

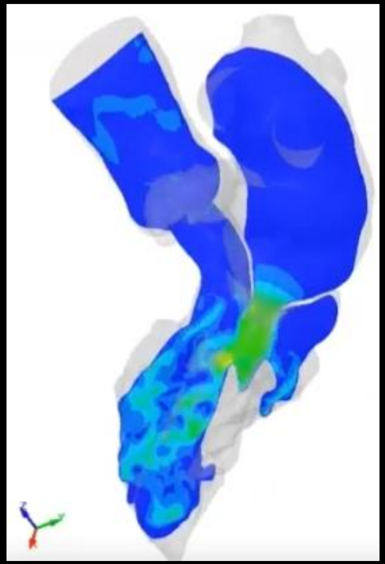
Multiphysics Fluid Structure Interaction (FSI) Modeling

Multiphysics in Industry

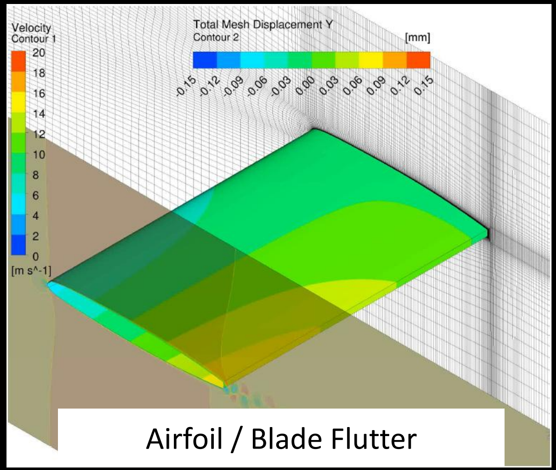
- Health care
  - Intrabody devices such as implants, stents, pacemakers
- Aerospace
  - Satellites / airplane wing flutter
- Automotive
  - Exhaust systems, shock absorbers and hydromounts

Cardiac / Arterial Flow

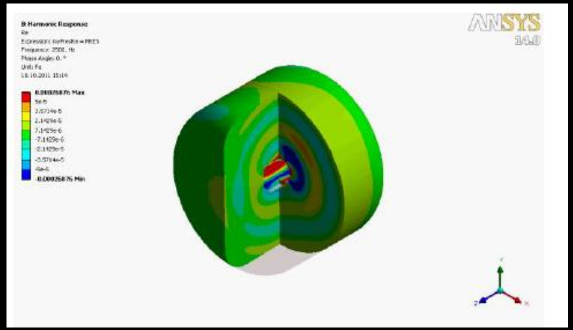




Chimakurthi, S.K., Reuss, S., Tooley, M. *et al.* ANSYS Workbench System Coupling: a state-of-the-art computational framework for analyzing multiphysics problems. *Engineering with Computers* **34**, 385–411 (2018).

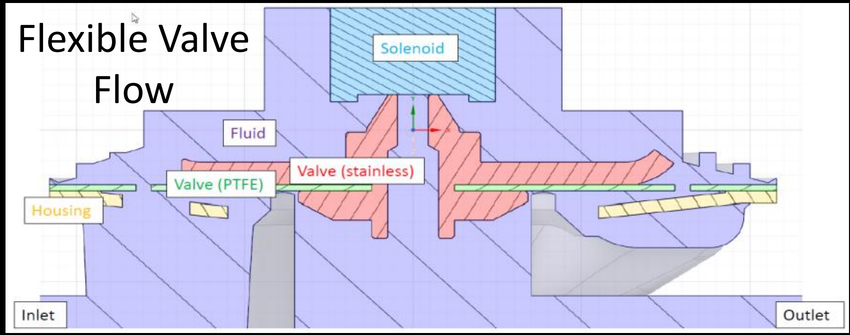


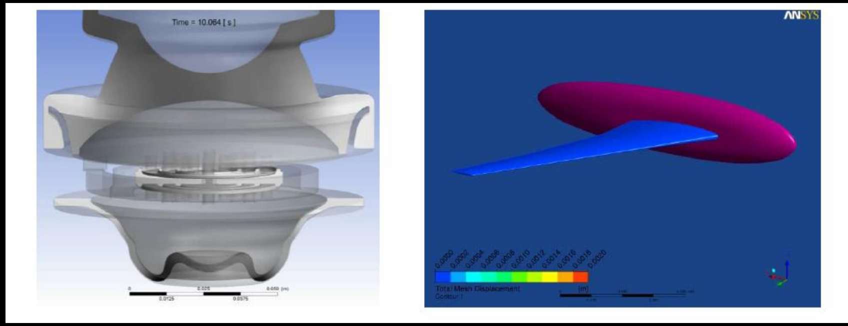
Airfoil / Blade Flutter

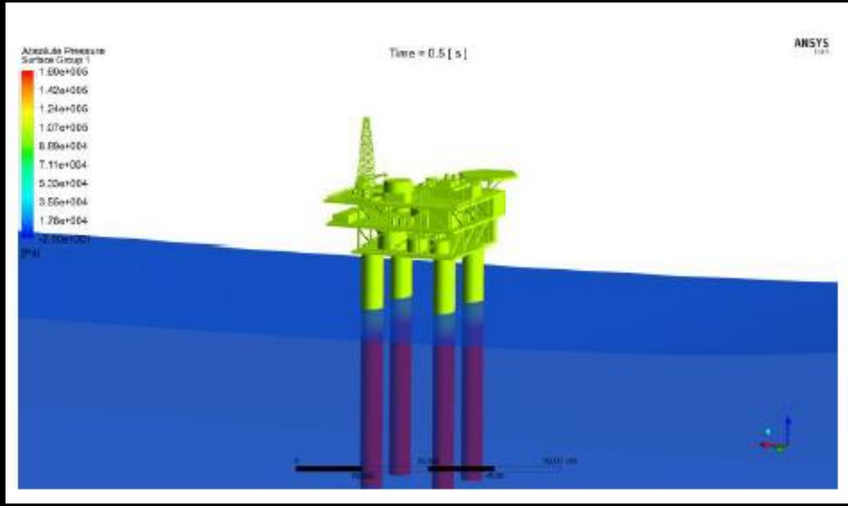


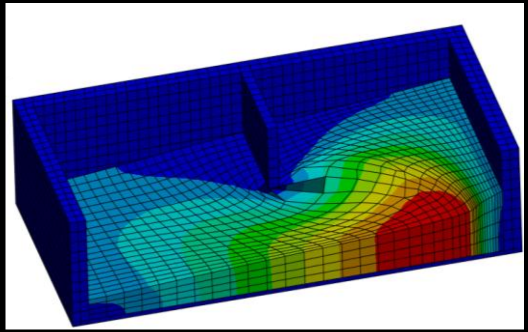
Multiphysics in Industry

- Consumer products
  - Heat, noise, and vibration in appliances
- Electronics
  - Thermal management of graphics cards, welds, handheld devices, PCBs
- Energy Process equipment
  - Thermal cycling in power plants and forces on oil platforms
- Transportation
  - Tank Sloshing / Baffles



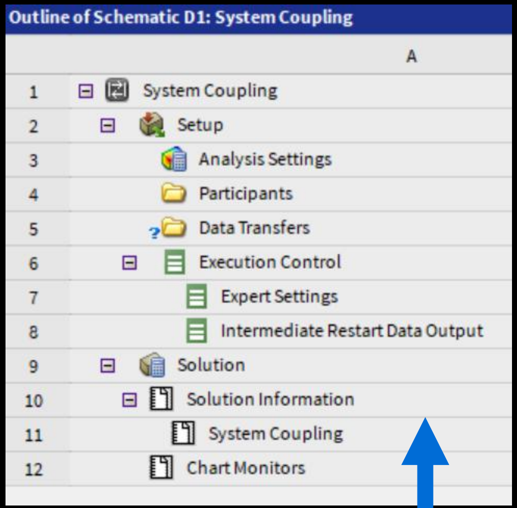


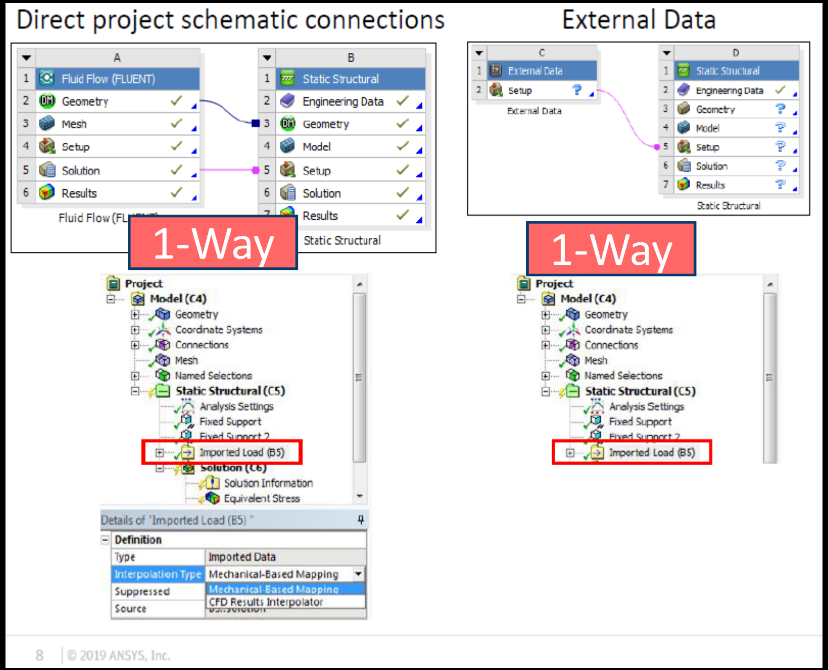




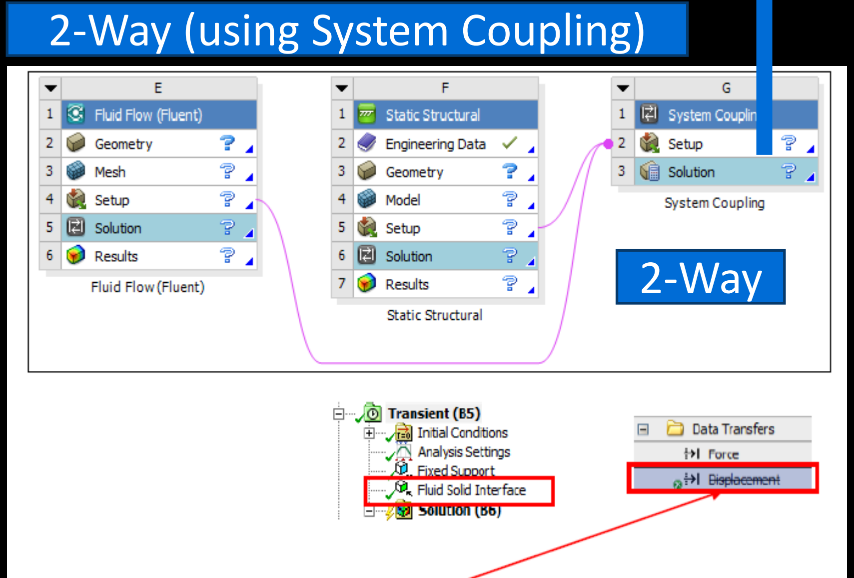
Implementation: Project Page

- Select 1-Way or 2-Way coupling.
  - Does fluid flow affect structure? Does structural deformation affect fluid?
- 1-Way Supports Direct Links on the project page
  - Data from other sources linkable via "External Data" system
- 2-Way uses "System Coupling" with controlling settings





Pro Tip:
Using Workbench Project Page for Coupling is good start – but can eventually switch to running the Coupling tool outside of Workbench for command line/script control of data flow/solutions.

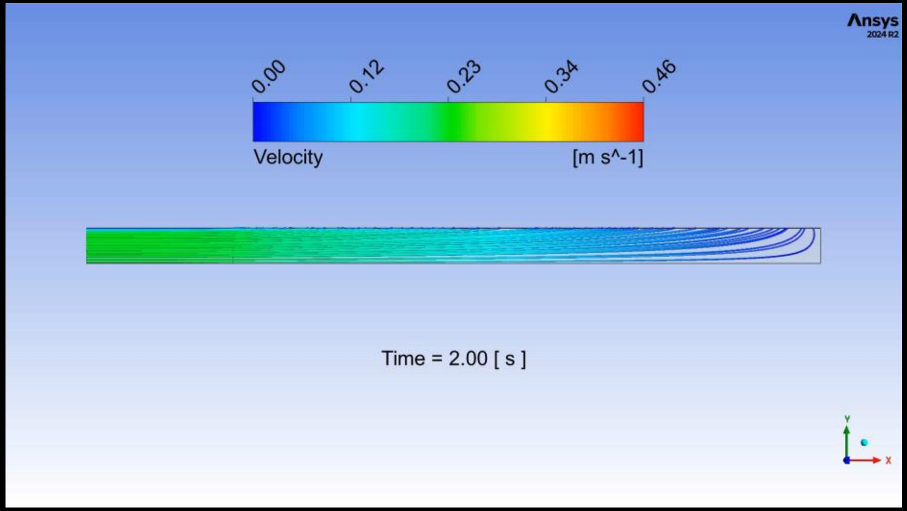


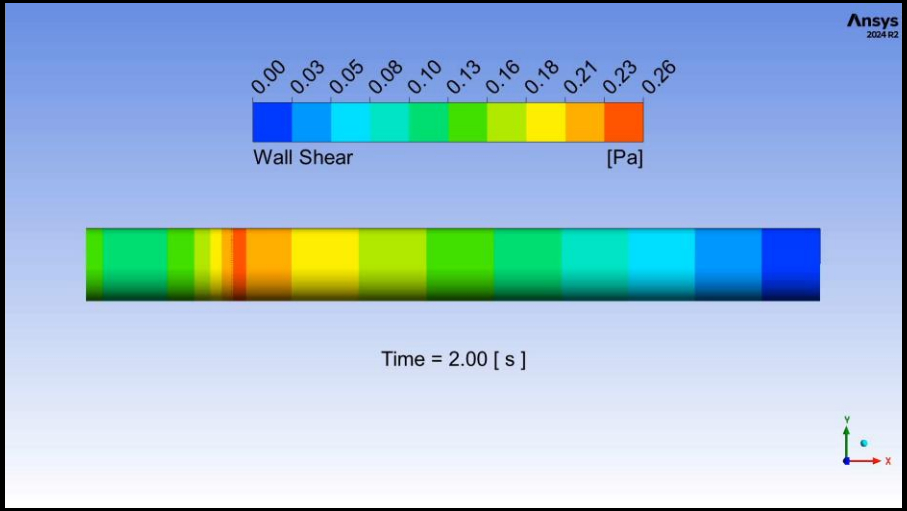
Displacement data transfer has been suppressed, creating a 1-way transfer. *Note we can also send Displacement but not Force with this method!*

Challenges in FSI

Pro Tip:
Post-Process using Ensight

- What makes 2-way FSI numerically challenging?
  - Material nonlinearities often present (e.g. hyperelastic)
  - Relative stiffness of fluid **much** lower than that of structure
    - Causes residuals / iteration equilibrium instability
  - Nonlinear fluid response near flow stoppage (e.g. valve closure)





Case Study – Fluid Film Bearings

Challenge

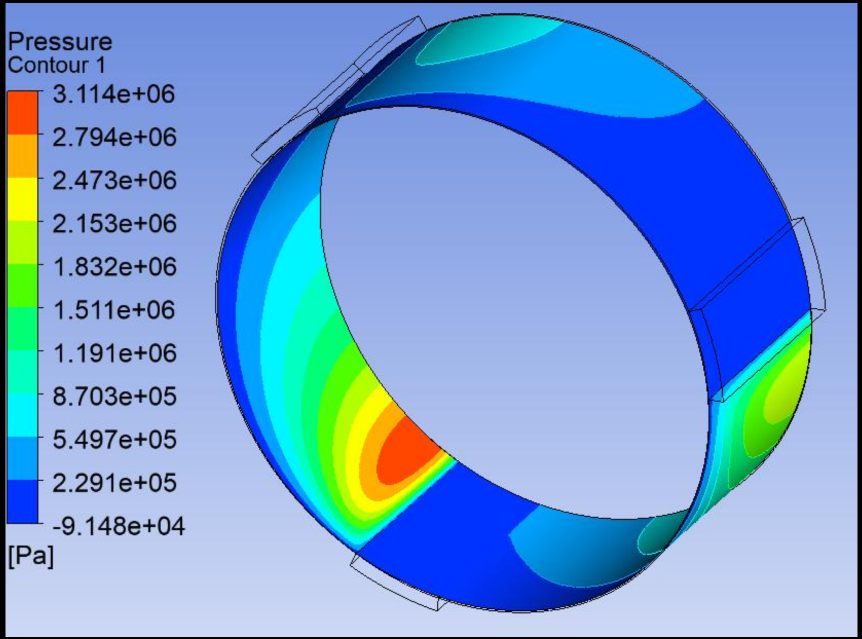
- Fluid film bearings are best modeled using Fluid-Structure Interaction (FSI) simulations. However, convergence can be challenging due to the unconstrained nature of the shaft position

Solution

- Simulation issues with fluid film bearing calculation of the shaft position can be resolved by increasing the density of the shaft to introduce additional inertia to the Mechanical solution. Further improvements are seen by ramping the applied force between the initial fluid force and the target load.

Benefits

- The approach herein unlocks the full potential of fluid film bearing FSI simulations to understand the complex physics taking place. Ultimately, this can lead to better bearings and better turbomachinery

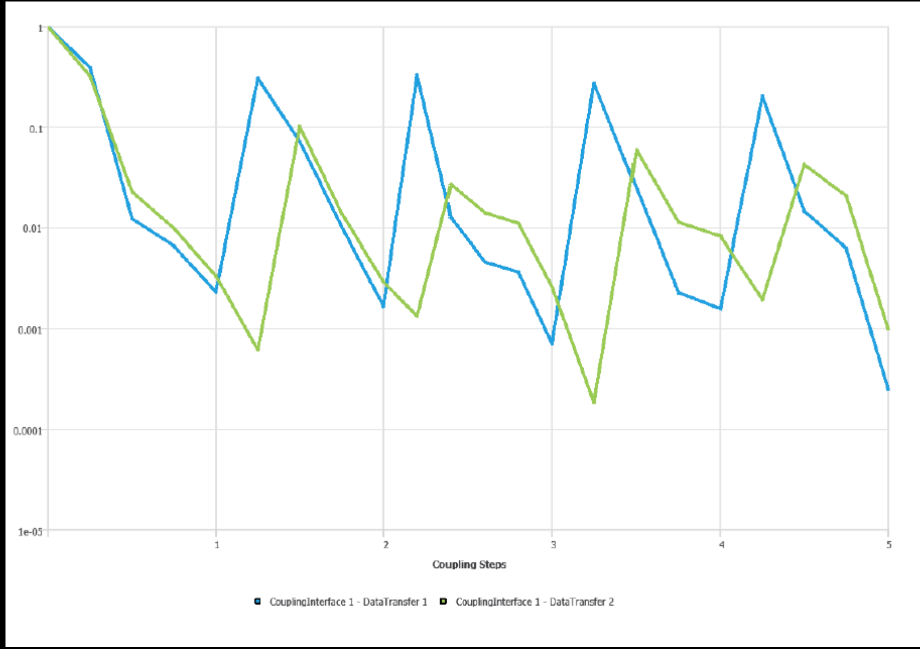


Convergence

- There are several controls that play a role in convergence, including:
  - Initialization
  - Time Step Size
  - Stability Controls (structural)
  - Number of Fluent iterations per step
  - Number of Coupling Iterations
  - Interface Under Relaxation Factors
  - Choice of data to transfer
- It is important to understand *why* a case has convergence problems before adjusting these controls

Monitoring Convergence

- The default Data Transfer convergence chart provides a good overview of the force/displacement convergence at the FSI interface(s)
- The saw-tooth pattern is desirable, showing convergence within each time step, then typically a jump at the start of the next time step
- By default, the normalized change in a data transfer must be less than 1% (i.e. 0.01) to be converged
- Default # of Coupling Iterations: Min: 1, Max 5 (for transients)
- Default Data Transfer Under Relaxation Factor: 1
- Perhaps 5 Fluent iterations per Coupling Iteration (for transients), but this is case dependent
- A Time Step Size based on the physics you need to resolve
  - 1/20th of the oscillation period corresponding to the highest structural frequency of interest
  - Flow field Courant number



Troubleshooting / Convergence

- Troubleshooting simple issues on a complex/expensive simulation can be very time consuming. Try to troubleshoot the simple issues on simple(r) models.
- Incrementally add complexities.
  - 2.5D (one element thick) instead of full 3D - True 2D analyses are not possible with System Coupling.
  - Mechanical-only with representative fluid loading, Fluent-only with representative structural motion (especially when rigid body motion can be employed).
  - Fluent equations initially off to troubleshoot dynamic meshing issues.
  - Linear-elastic solid before hyperelastic solid.
  - Begin with a Gap Model turned on.

Troubleshooting / Convergence

- Try Mixed u-P formulation in structural elements
  - Can dramatically help convergence
    - But only about 25% success rate
  - Implementation is easy – Single line in snippet
  - Adds a pressure DOF – prevents element collapse / negative volume
  - Added DOF is usually small impact on matrix size
    - 13 DOF instead of 12 (i.e. assuming 4 integration-point elements)
  - Enable the Mixed u-P formulation by doing one of the following:
    - Setting keyopt(6) = 1 as shown by the comment snippet on the right. It must be inserted under the hyperelastic solid body.

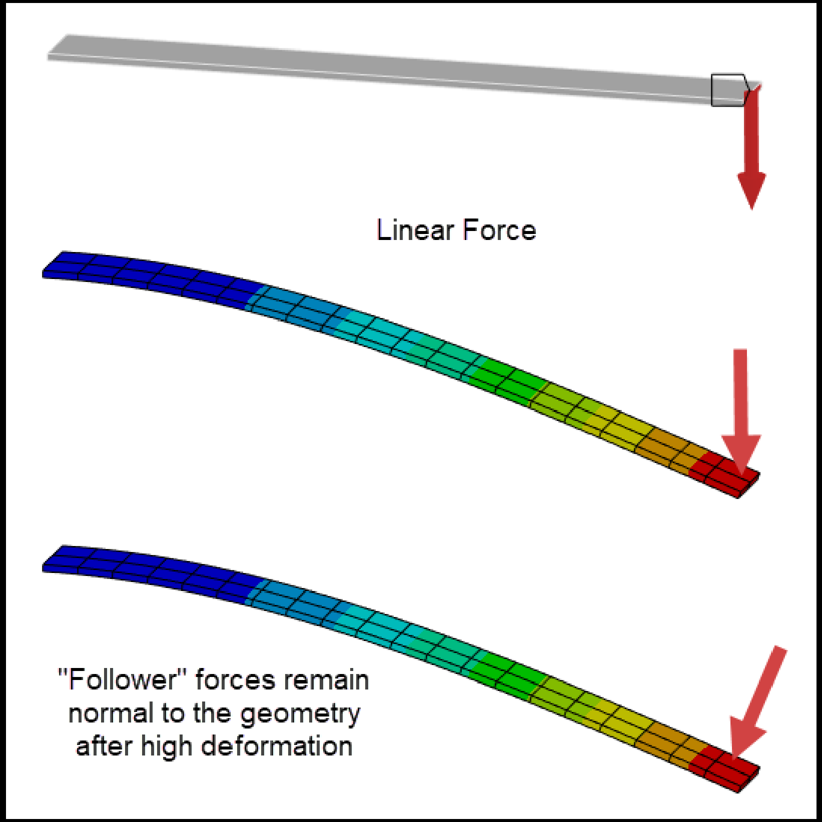
```
!    Commands inserted
!    The material numbe
!    The element type n

!    Active UNIT system
!    NOTE:  Any data th
!                 See S

keyopt,typeids(1),6,1

```

Troubleshooting / Convergence

- Flexible Components Often use Hyperleastic Materials
- Engineering Data allows for utilization of all major hyperelastic models, such as Neo-Hookean, Mooney-Rivlin, Ogden, Yeoh, Hencky, and many more.
- Linear-elastic solid before hyperelastic solid
- For challenging problems such as flexible valves, it is often beneficial to troubleshoot small quick simulations involving a subset of the required settings before attempting to solve the complete problem
- Test the FEA model in standalone with representative forces

Troubleshooting / Convergence

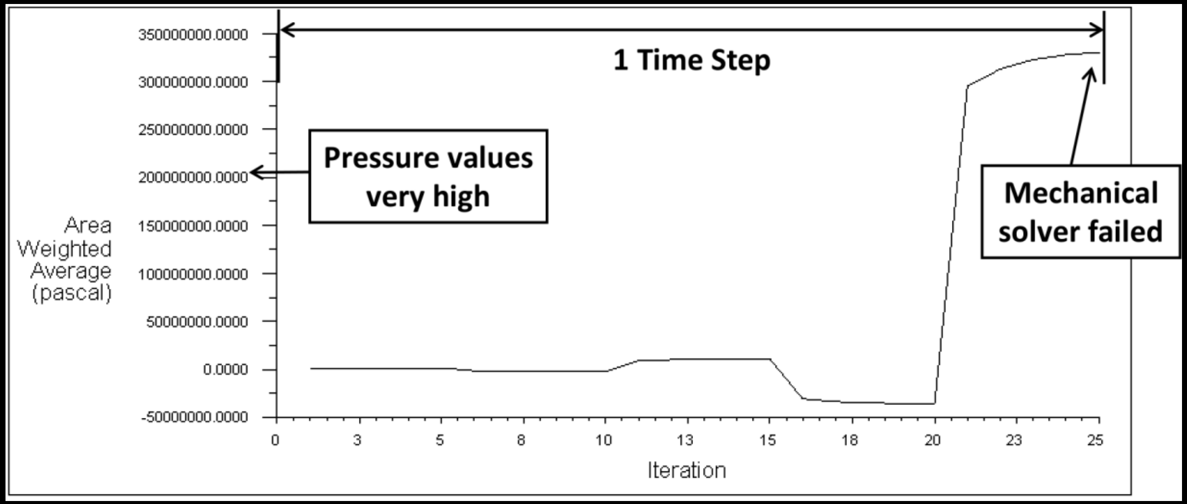
- For Mechanical simulations involving hyperelastic materials, follower effects are often important.
- Follower effects allow the force loads to rotate (i.e. follow) the deforming structure as it deforms within a time step.
- Load following is used by default in Mechanical when a pressure load is applied
- LS-DYNA Considerations
  - SyC force loads are not following deformation.
  - Unlike force loads, force density loads leverage follower effects. Using force density also activates special SyC mapping behavior to preserve the force density magnitude and direction across nonmatching CFD and structural grids, which can be important when there are very large pressure forces and soft materials.
  - Force density should not be used if you have two-sided shells, and you want to transfer force to both sides. In such cases, use force.
  - For converged cases, results using force density are expect to match those using force.



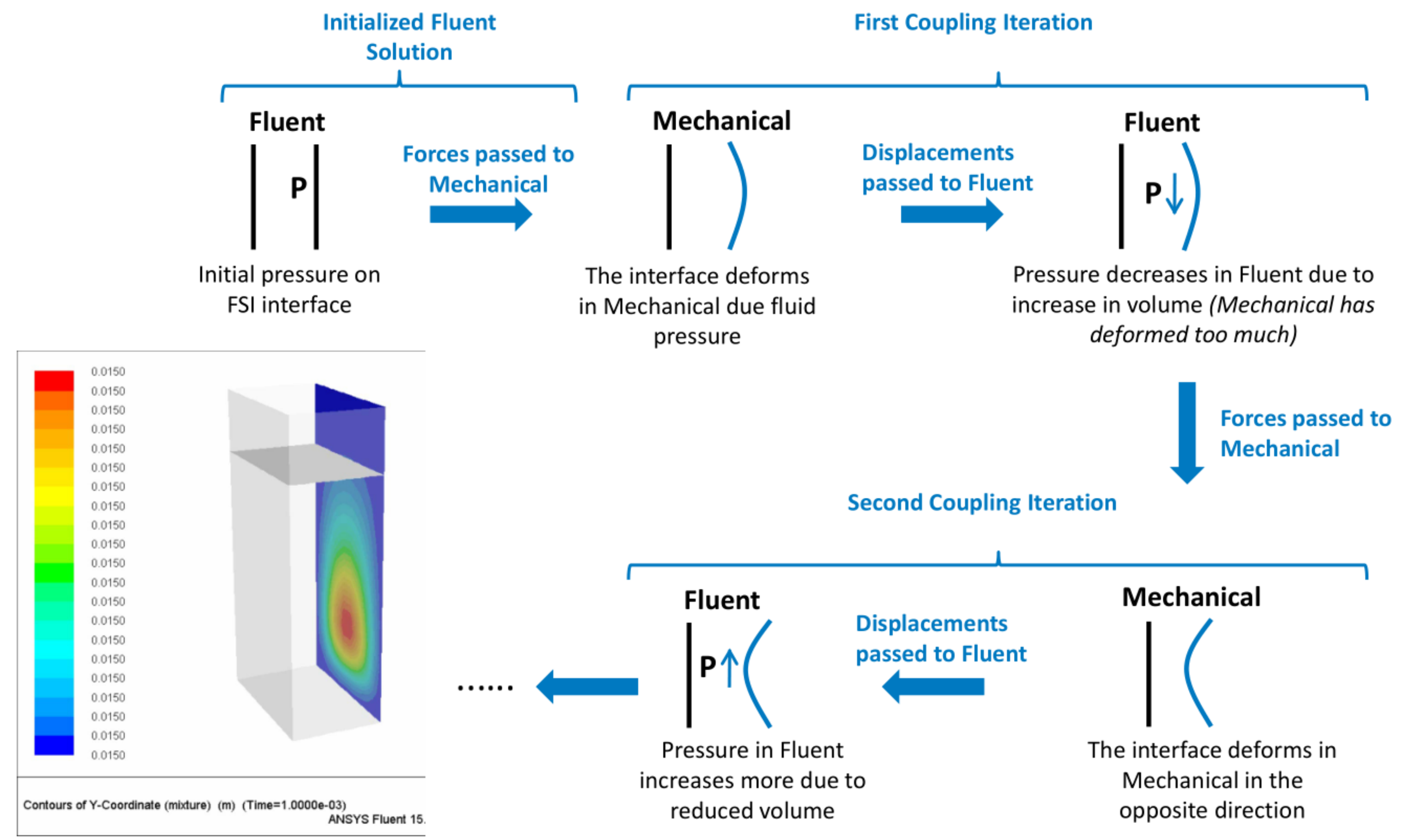
Convergence Stabilization

- Reaching convergence can be challenging if a small change in the fluid force leads to a significant change in deformation or vice versa.
  - This is particularly common when the solid body is slender or if the solid density is comparable to or smaller than the fluid density, leading to significant added mass effect (both are common for flexible valves).

- Oscillation within a timestep, from one Coupling Iteration to the next, is indicative of FSI interface instability.

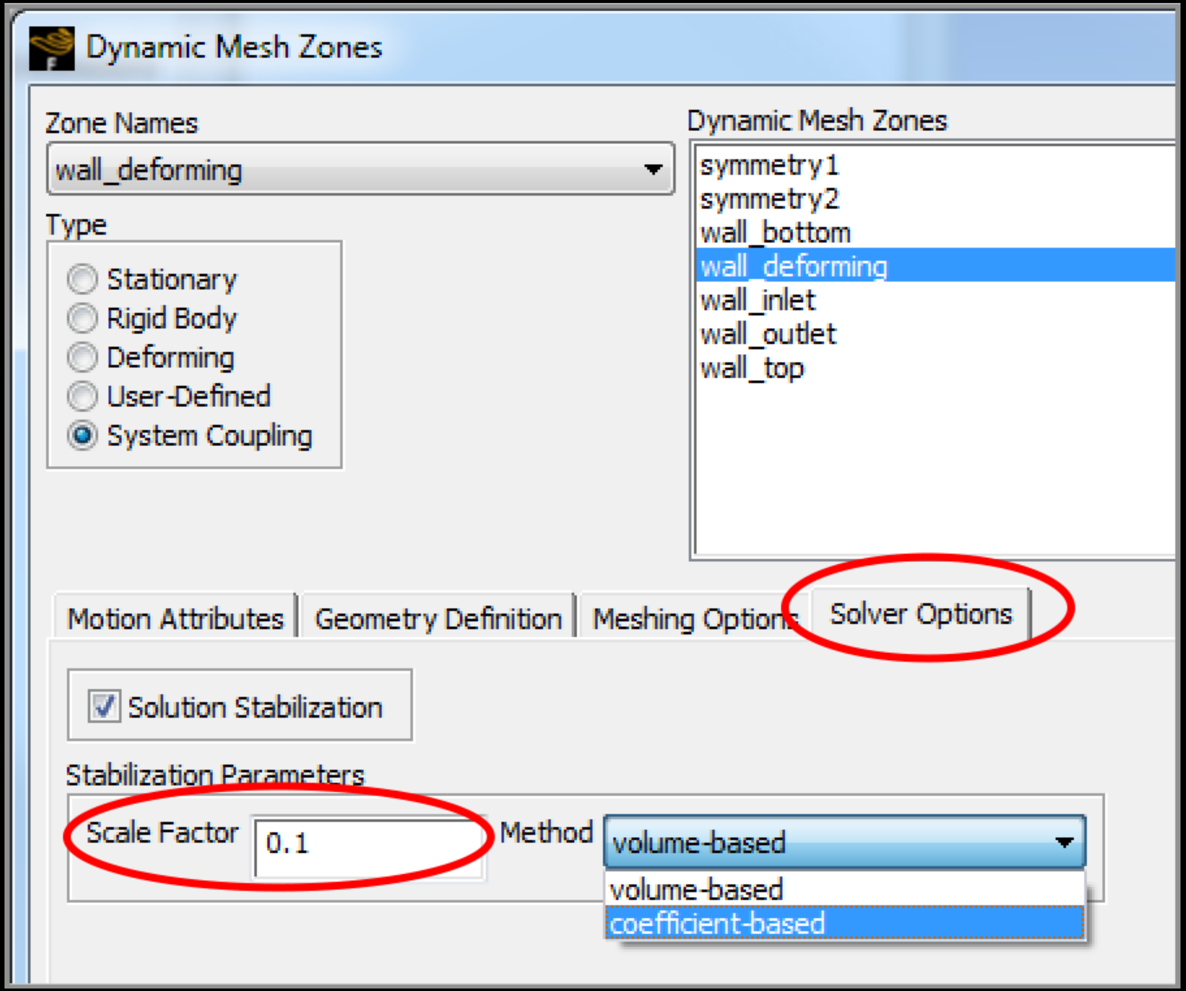


Convergence Stabilization



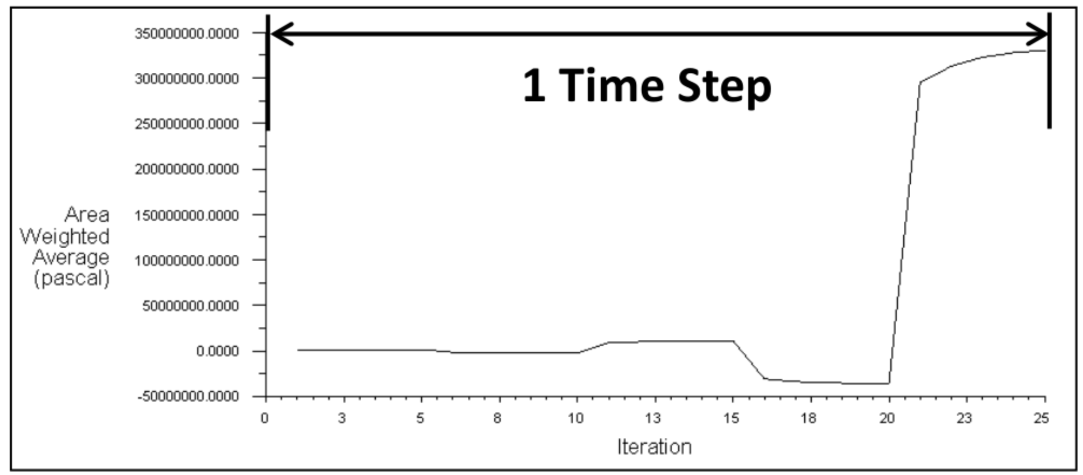
Convergence Stabilization

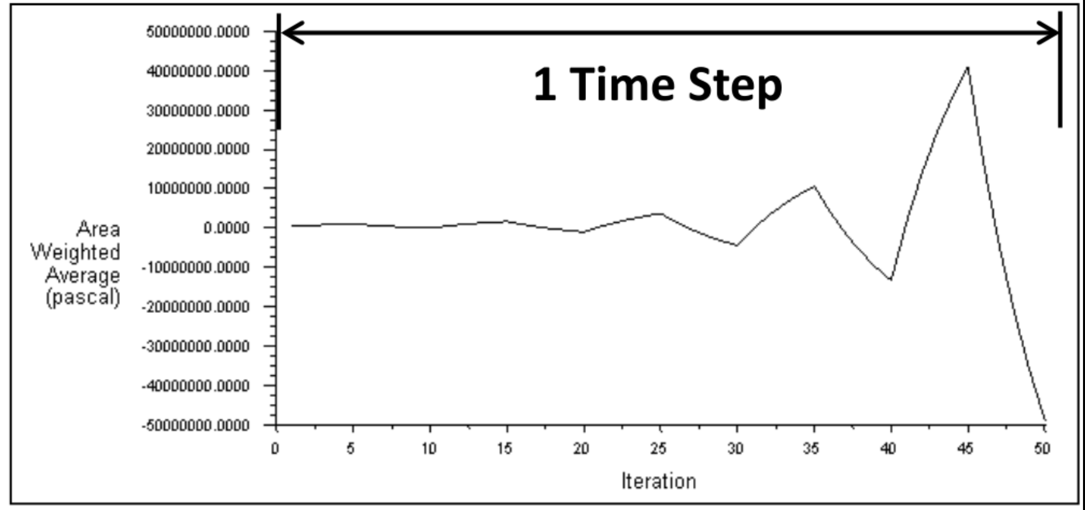
- A higher Scale Factor results in more stabilization (i.e. a slower pressure response).
  - The coefficient-based method has been shown to be more robust and have a more predictable range of required scale factor - independent of boundary cell size - than the volume-based method so it is recommended.
  - For the coefficient-based method Scale Factors between 1e- 5 and 1e-3 are typical.
  - The appropriate value is case specific.
- How do we find the correct Scale Factor?



Convergence Stabilization

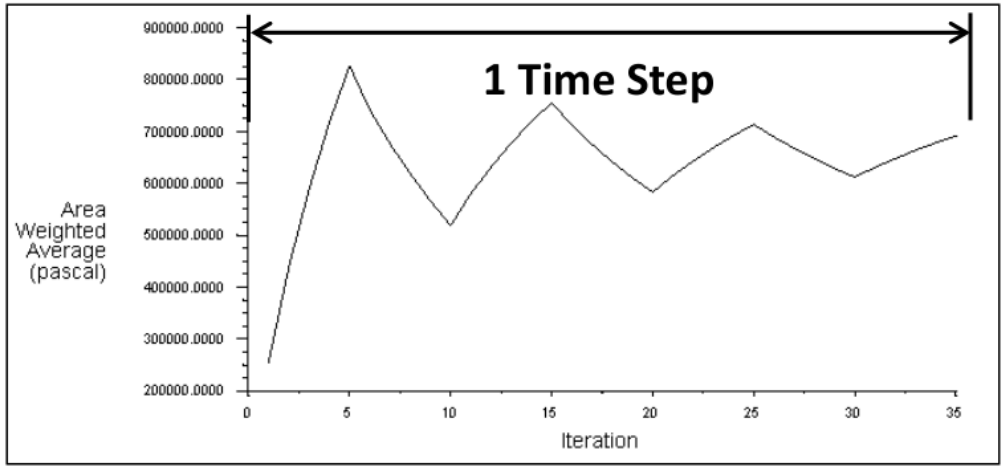
- Scale Factor = 0 with coefficient-based method
- Baseline divergent case
- Scale Factor = 1e-5
- Still divergent, but completes more
- Coupling Iterations
- Notice the pressure response is no longer a step change



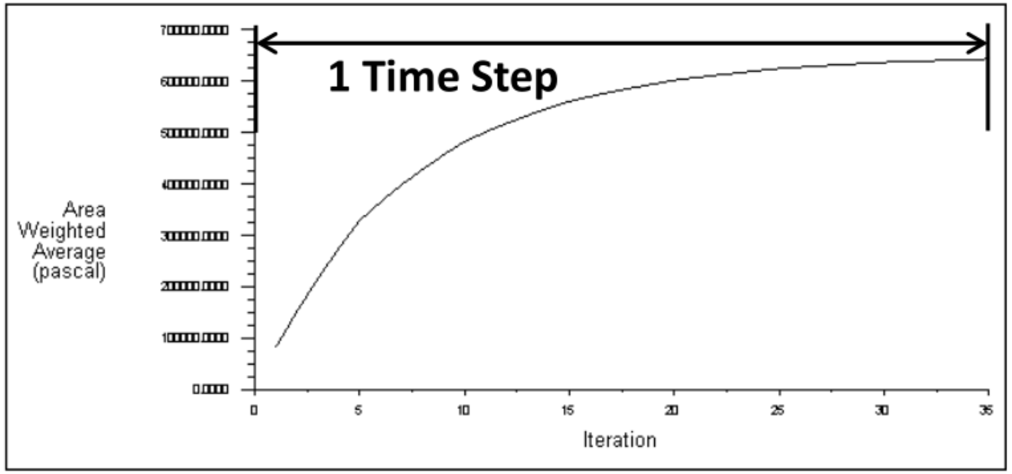


Convergence Stabilization

- Scale Factor = 1e-4
- No longer diverges, but still under-damped. Did not fully converge after 7 Coupling Iterations

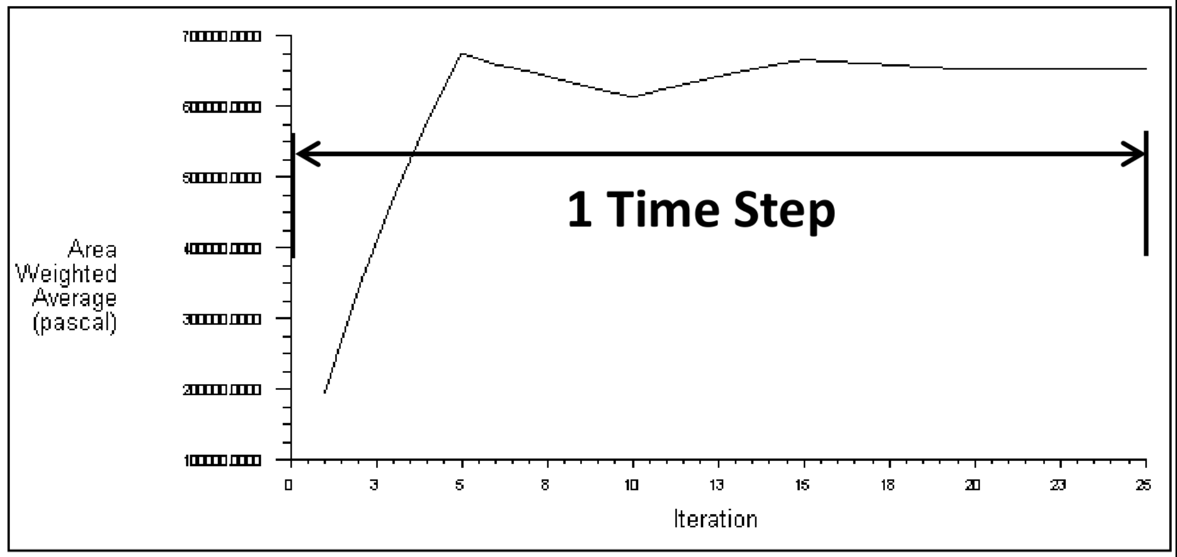


- Scale Factor = 1e-3
- Now stable, but over-damped. Not quite fully converged within the time step after 7 Coupling Iterations



Convergence Stabilization

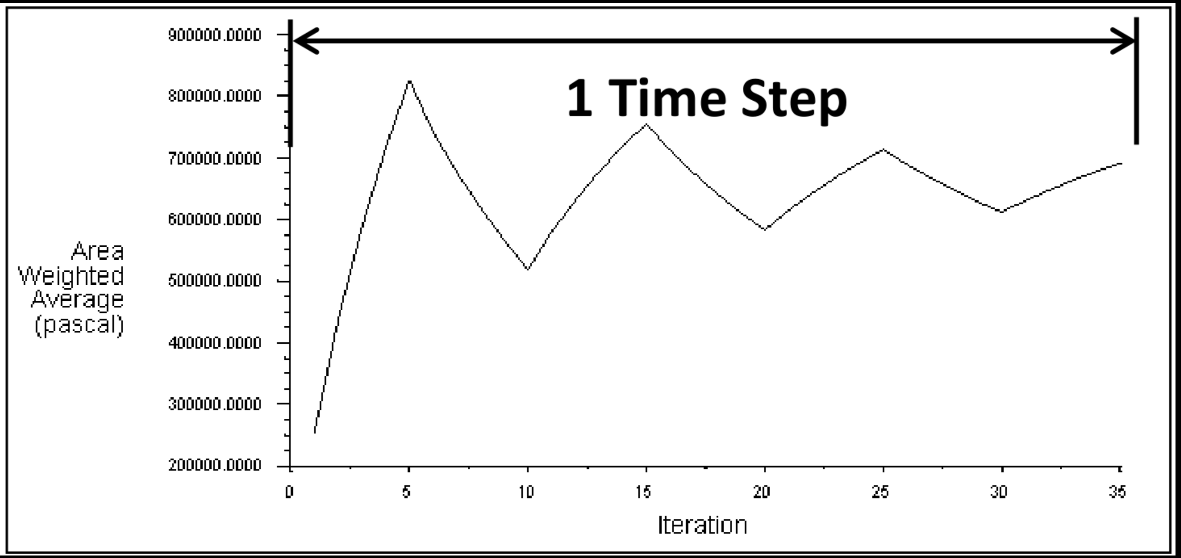
- Scale Factor = 3e-4
- Good response. Well converged after 5 Coupling Iterations



- Recommended Process:
  1. Start with Large Value to obtain convergence (also verifying overall model setup)
  2. Incrementally decrease Scale Factor until model diverges… and then increase it slightly
  – Avoids scenario where over-damped is mistaken as flat/converged!

Convergence Stabilization

- Note that the response is a function of the Scale Factor AND the number of Fluent iterations.
  - Consider the case shown – if you perform more Fluent iterations per Coupling Iteration each oscillation would extend further (more unstable).
  - In practice set the num. of Fluent iterations to a small number and then adjust the Scale Factor to get the optimal response.
    - There's no point in using lots of Fluent iterations.



Methods for Convergence / Stability

- Try reducing the force-related Under Relaxation Factors within System Coupling along with increasing the minimum and maximum number of iterations to slow down the solution
  - A typical URF is around 0.5-0.75 for steady state

- Keep the following formula in mind if you adjust the Under-Relaxation Factor:

$$\% \, \Delta \, \text{Load Received} = (1 - (1 - \text{URF})^{\wedge \text{Max. Coupling Iters}}) * 100\%$$

- For example, given a URF of 0.25 and a maximum of 5 Coupling Iterations, we have:

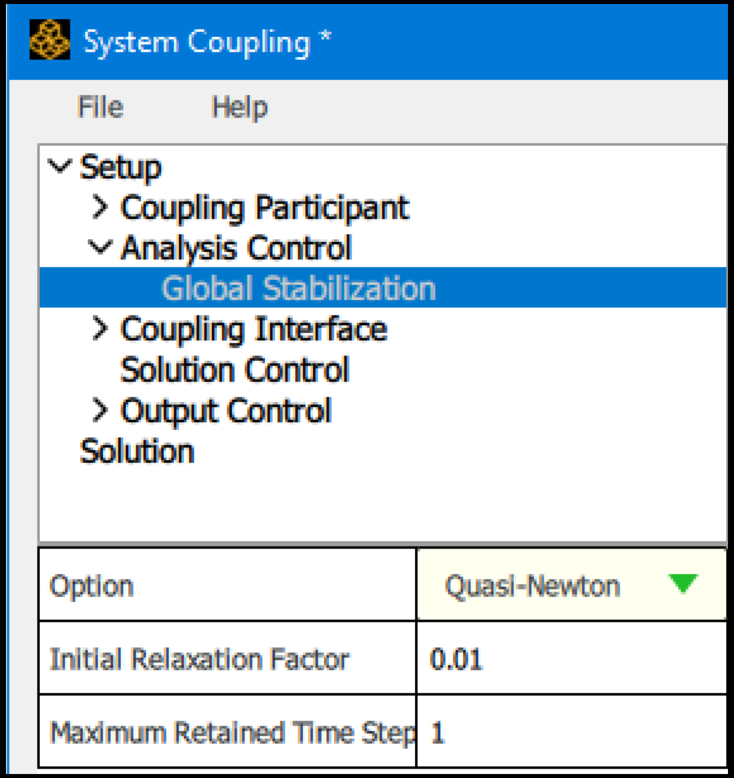
$$\% \, \Delta \, \text{Load Received} = (1 - (1 - 0.25)^{\wedge 5}) * 100\% = 76\%$$

Methods for Convergence / Stability

- Ramp fluid pressure or loads to gradually approach desired final conditions
- Have sufficient Fluent iterations for stability (i.e. force is stabilizing), rather than relying on coupling iterations. But do not need to fully converge on force – better to be done through coupling iterations.
- Include a small amount of compressibility in the fluid by using a bulk modulus, where the fluid density varies slightly depending on the local static pressures.
- Changing the density of the solid components to increase their inertia and slow down their movement

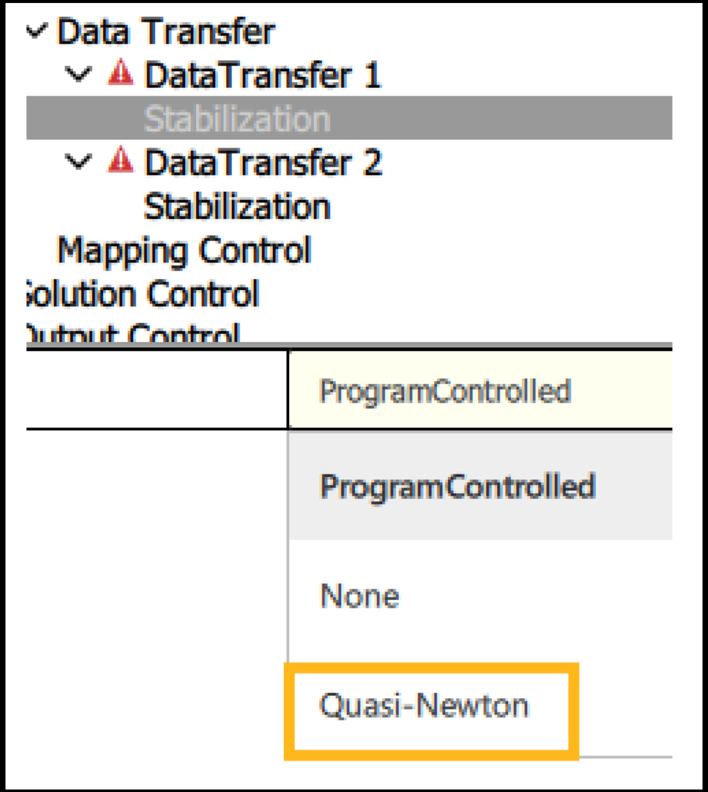
Convergence Stabilization – Quasi-Newton (iQN)

- Instead of calibrating the solution stabilization in Fluent, System Coupling can automatically stabilize the data transfer using iQN stabilization.
- Activate IQN stabilization under Solution Control->Global Stabilization in the GUI. This will stabilize the most appropriate data transfer for the problem.
- Works for force-displacement FSI as well as thermal FSI problems.
- For strongly coupled problems, make sure you set enough coupling iterations that the coupled step converges.
- Advantages over solution stabilization:
  - Targets the total force, not just pressure.
  - Good for viscous liquids, and flows with high shear where solution stabilization will not be effective.
  - It is adaptive, so no need for manual tuning.
  - Can target thermal transfers.
- Disadvantages:
  - Noticeably slower to converge
  - Convergence not always guaranteed



Convergence Stabilization – Quasi-Newton (iQN)

- Things to look out for with IQN stabilization:
  - More Fluent iterations per coupling iteration are recommended than a traditional FSI case.
- Use 10-20 or whatever is sufficient for good convergence in the coupling iteration.
  - More coupling iterations per coupling step are recommended than a traditional FSI case.
- Can be 10-20 or more, even for transient FSI



Stabilization with Pressure Coefficient

- Stabilize with Pressure Coefficient

  See Support portal for these demo/sample files

  Problem/Description:
  How do you Stabilize strongly coupled 2-way FSI simulations between FLUENT or CFX and Mechanical?
  Solution:
  See the attached demo to learn how to stabilize two-way FSI simulations that are unstable due to strong coupling between the fluid and structure. This is a common problem when dealing with highly flexible structures such as biomedical applications, diaphragms and flexible valves. This material applies to both CFX and FLUENT.
  The following commands will be explained and used in Fluent to help stabilize a system:
  (rpsetvar ‘dynamesh/sc-bc-compressibility-type 2)
  (rpsetvar ‘dynamesh/sc/bc-compressibility x)
  In CFX a Mass Flux Pressure Coefficient will be used to stabilize the system:
  Boundary > Sources > Continuity > Fluid Mass Flux > Pressure Coeff. = x
  In both cases “x” represents the stabilization factor, which must be found through trial and error.
  A presentation and project archive are included which outline best practices for solution stabilization in Fluent.
  Attachments:
  1. StabilizationDemo.wmv
  2. Solution_stabilization_For_FSI_&_6DOF.pptx
  3. Solution_stabilization_For_FSI_&_6DOF_Files.zip

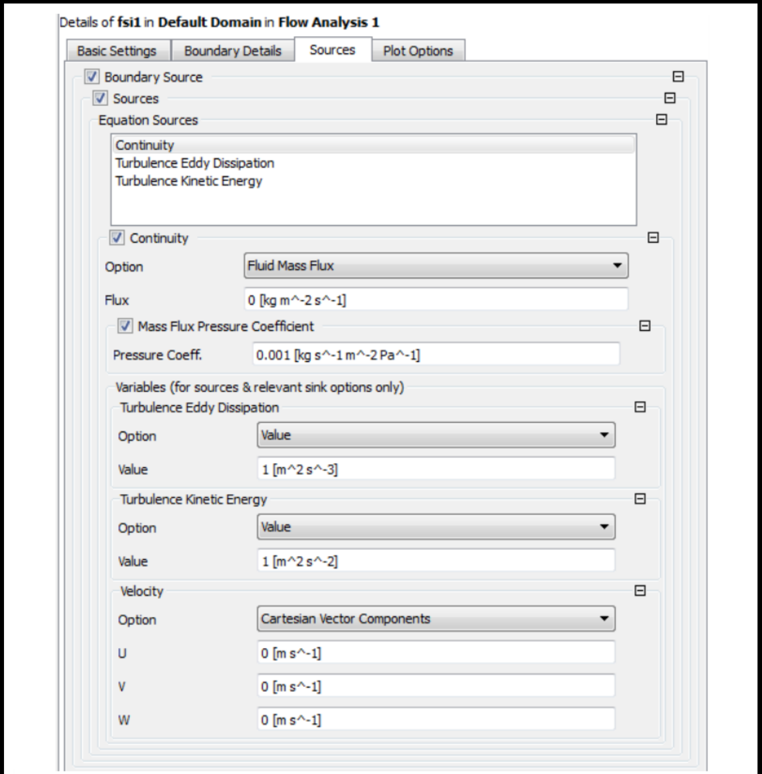


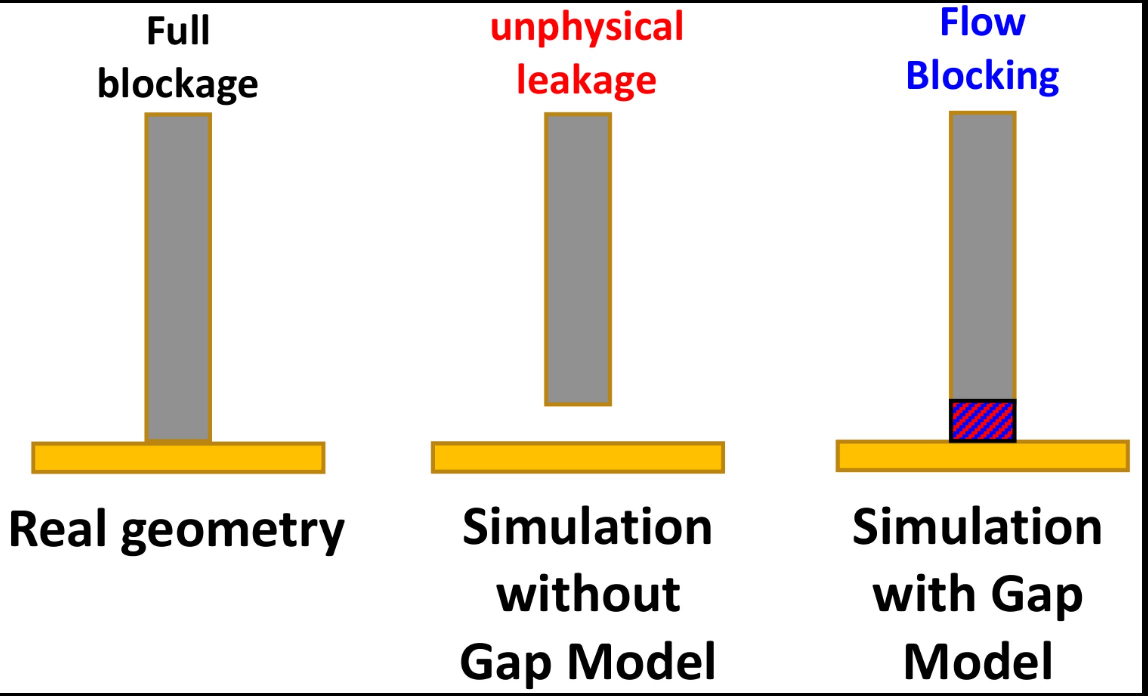
Figure 1: Mass Flux Pressure Coefficient settings in CFX

Modelling Valve Closure

- Given that valves regulate flow by opening, closing, or partially obstructing the flow path, valve closure is often of interest to the simulation analyst.
- Valve closure introduces added complexity on both the fluid and structural sides of the simulation.
- In Fluent, since walls cannot come into contact, a Gap Model must be employed to block the flow once neighboring walls come within a specified proximity.
- In Mechanical, contact must be defined to prevent the neighboring walls from approaching each other any closer than the specified proximity.

Modelling Valve Closure - The Gap Model Approach

- For flows with moving face zones, narrow gaps may open or close during every time step. You can use the Gap Model to simulate the blockage of flow when a pair of face zones moves within a specified proximity threshold distance of the other.
- The two Gap Model types are as follows:
  - Flow-blocking - The cells within the gap no longer participate in the solution
  - Flow-modeling – The flow is decelerated within the gap

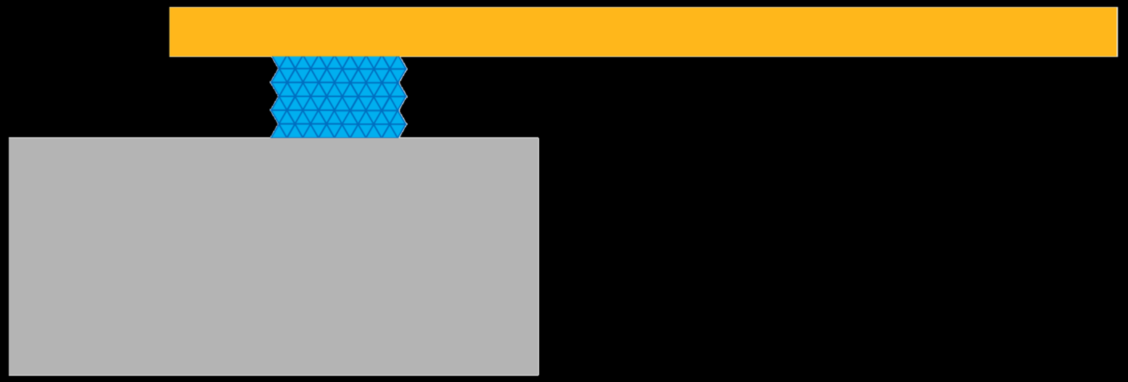


Modelling Valve Closure

- When using flow-modeling, you'll need to calibrate the settings such that the flow is sufficiently slowed down in the gap without destabilizing the solution. Typically, ideal settings minimize, but do not fully prevent, leakage.
- Typically for valves, the flow-blocking type is preferred since it will ensure that there is no flow leakage. However, there are two reasons to consider using flow-modeling instead:
  - Flow-blocking is less stable than flow-modeling since the valve closure is more abrupt. When flow-blocking does not produce a stable solution, and if turning on Gap Model solution stabilization doesn't resolve the issue, instead use flow-modeling to decelerate the flow inside the gap regions and minimize leakage.
  - When cells are deactivated from the solution during flow-blocking, they will not be accounted for in the load transfer to Mechanical which can adversely affect the accuracy of the solution.

Dynamic Meshing

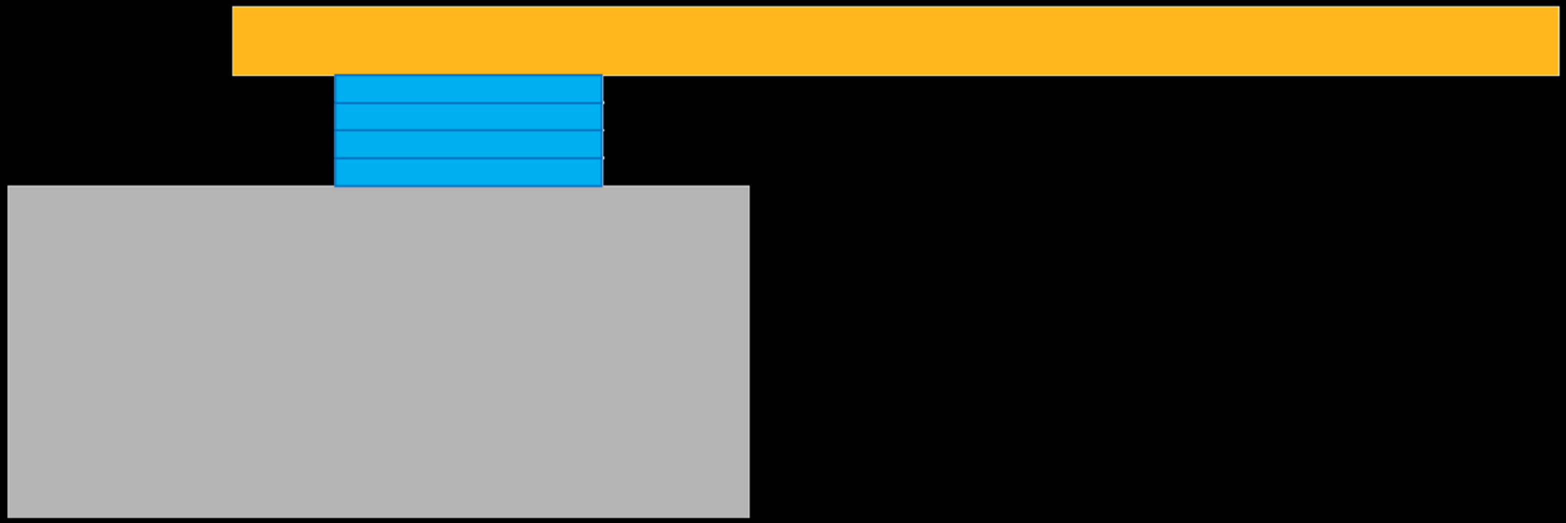
- Valve modeling often presents a dynamic meshing challenge as the gap width can range from very small (to represent a fully closed valve) to orders of magnitude larger (in the case of a fully open valve). In the gap region, a mesh suitable for one extreme is likely to be unsuitable for the other extreme.



- Option 1: Very Fine Mesh
  - A straightforward option is to use a very fine mesh in the gap region. As the gap widens, remeshing can be used to add additional small elements in the gap. In this case, the dynamic mesh settings are simple but given that the motion of the wall within a timestep should ideally be no larger than the local element size (to avoid negative element volumes), a small timestep is required. A small timestep in combination with a fine mesh would have this easy-to-setup approach being long-to-solve.

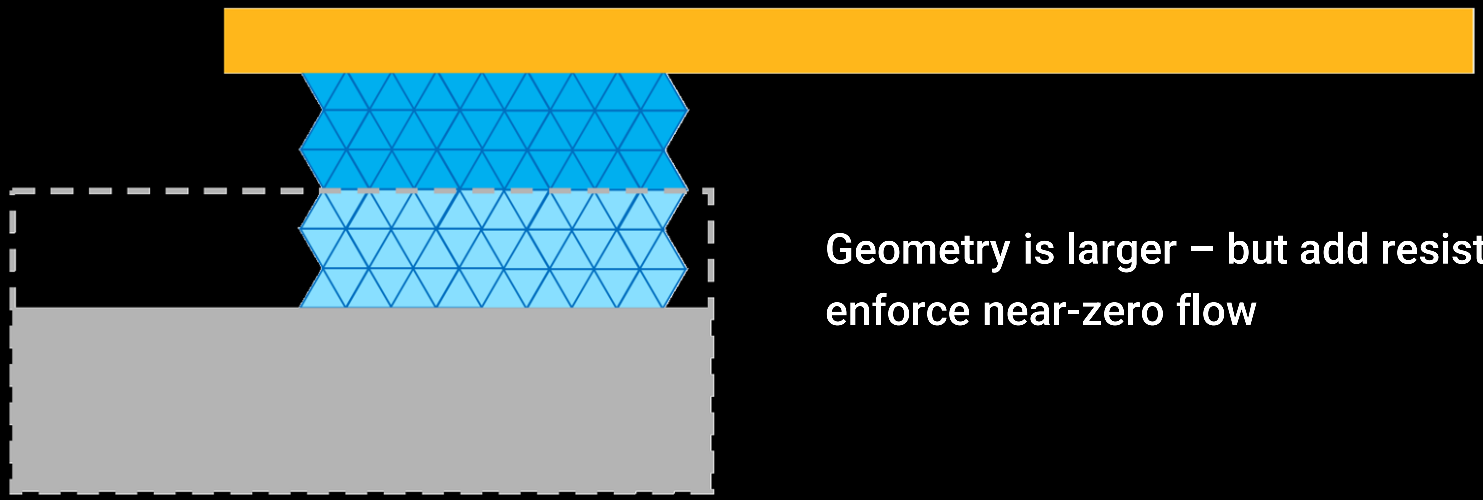
Dynamic Meshing

- Option 2: High Aspect Ratio Elements
  - Given that the small length scales are only in one direction (normal to the wall), one option is to use high aspect ratio elements being thin in the normal to wall direction and long in the flow direction. This approach will require careful meshing (utilizing the Sweep or Multizone method in this region) and since remeshing does not apply to quad/hex elements, you are restricted to using smoothing. In cases where the gap significantly widens, the (deformed) original mesh may no longer be adequate.



Dynamic Meshing

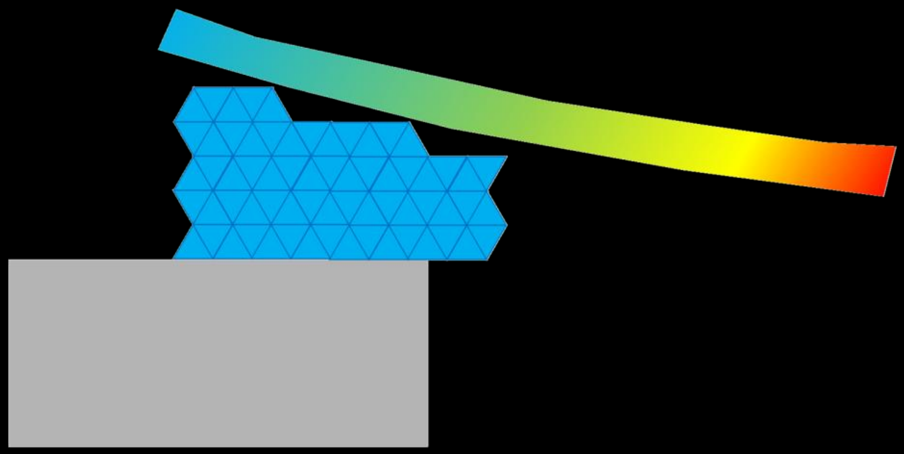
- Option 3: Artificially Increase Gap Width
  - To avoid the small gap, part of the valve seat geometry can be removed. In Mechanical, increase the contact offset so that the valve cannot penetrate the original seat geometry (dashed line). In Fluent, increase the Gap Model proximity threshold to block the flow at this wider gap. This approach should work well in the closed state. However, in the open state (when the Gap Model is no longer in effect), unphysical flow will occur in the artificial gap (light blue elements). Given this, it is best to keep the artificial gap width to a minimum.



Geometry is larger – but add resistance to enforce near-zero flow

Dynamic Meshing

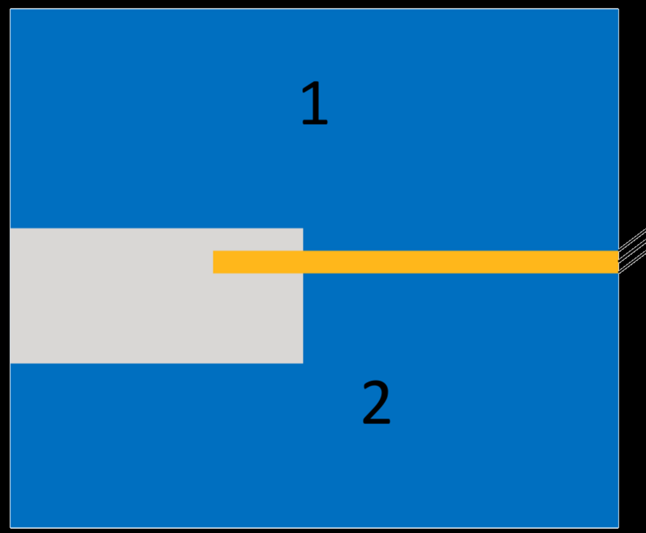
- Option 4: Utilize Prestressing Workflows
  - If you're only interested in the steady-state flow through the open valve, one option is to prestress your solid (using one of the prestressing workflows detailed in this document) and build your fluid mesh on the open state. In this state, there are no small gaps, and therefore large elements can be utilized.



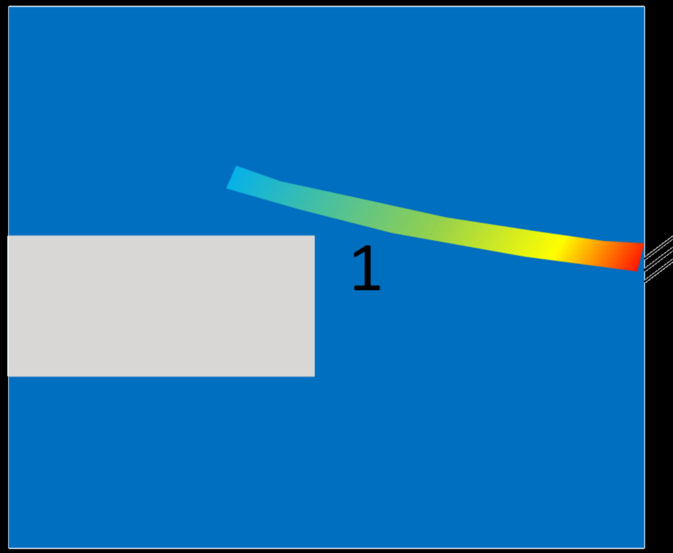
Pre-Stressing Structure

- In the case of valves, the unstressed geometry typically would interfere with the rigid geometry.
  - fluid topology before prestressing is different from the fluid topology after prestressing.
  - the fluid topology cannot change during a simulation (generally), the typical approach of prestress the solid, interrupt it, add the fluid loading, then continue is not possible.
- Different approaches are required for linear-elastic and hyperelastic materials.

Unstressed solid – two separate volumes

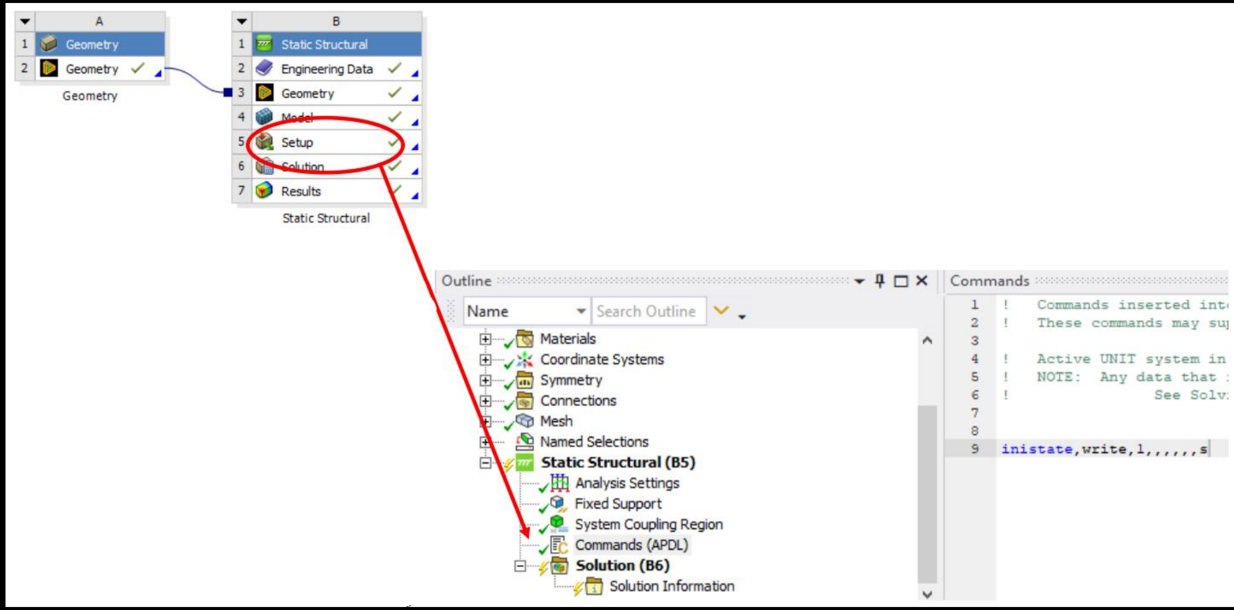


Stressed solid – single fluid volume



Pre-Stressing Linear-Elastic Solids

- Static Structural (Mechanical-only) simulation
- Insert inistate command to save the stress state to file.ist
- Note: inistate will not read the stress history required for hyperelastic materials.

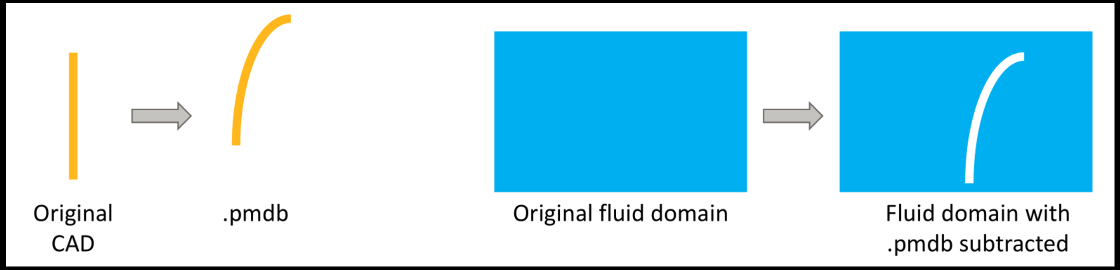


Implement Prestress State (.ist file)

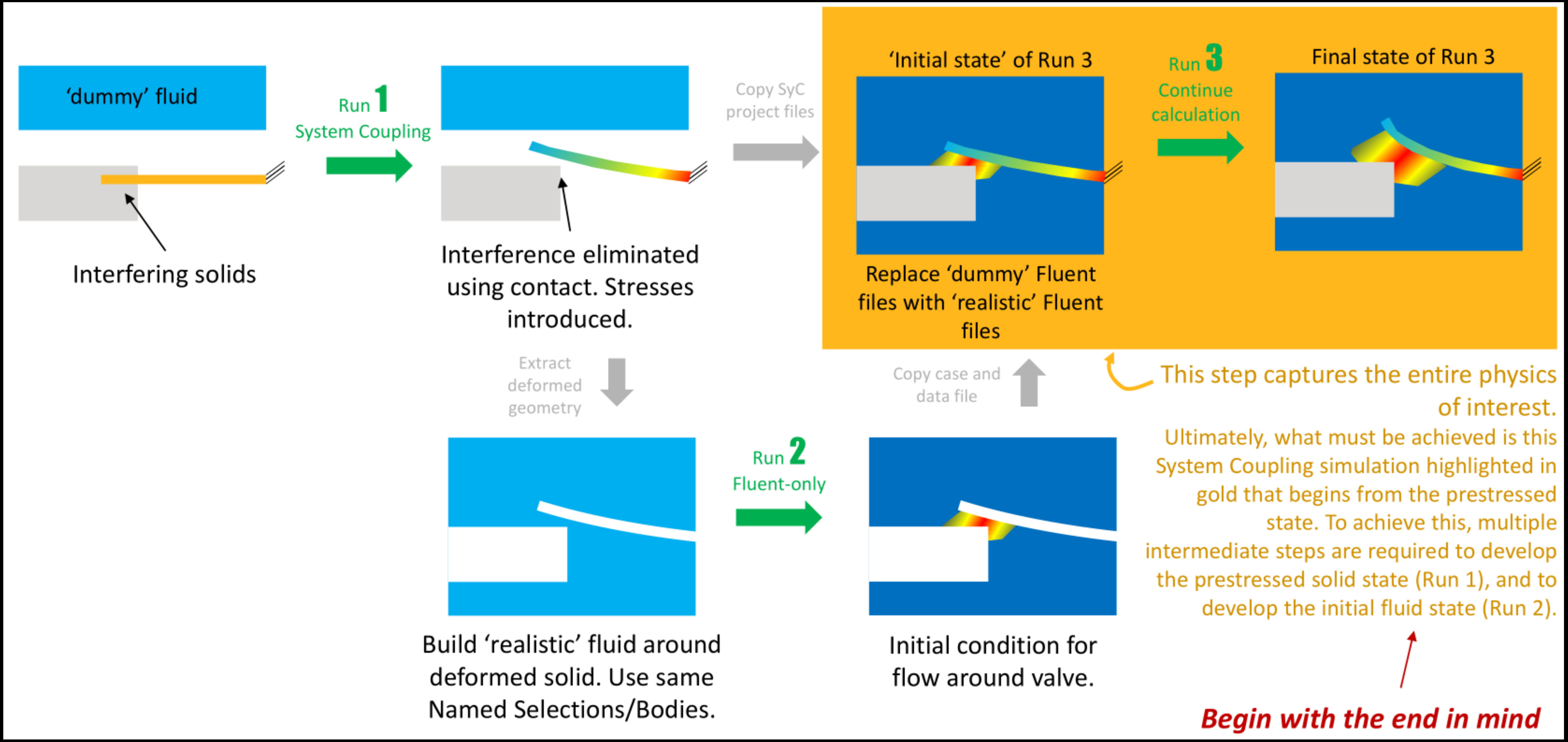
- Solve Transient FSI Problem
- Whether using .pmdb files or .stl files for the deformed geometry/mesh, the remaining steps are the same.
  - Setup transient Fluent with new mesh
  - Connect Fluent and Transient Structural to System Coupling Setup
- Export System Coupling Setup
- Copy file.ist to the Mechanical folder
  - If running the System Coupling case from within Workbench, right-click Solution > Open Solver Files Directory. Place file.ist file here.
  - If running the System Coupling case outside of Workbench, place in the Mechanical subdirectory created in the previous step.
- Solve

Building Deformed Geometry

- Import .pmdb into Discovery and build the fluid domain based on the deformed geometry.
  - Generate mesh as usual.
  - The .pmdb file procedure is not always robust, and it may sometimes be necessary to use an alternate procedure involving .stl files and SpaceClaim or Fluent Meshing
- Read the STL file into Discovery.
  - Convert the STL to solids without auto merging the faces.
  - Merge each surface individually by double clicking on a STL face on the specific surface and use “Merge Faces” (Repair tab).

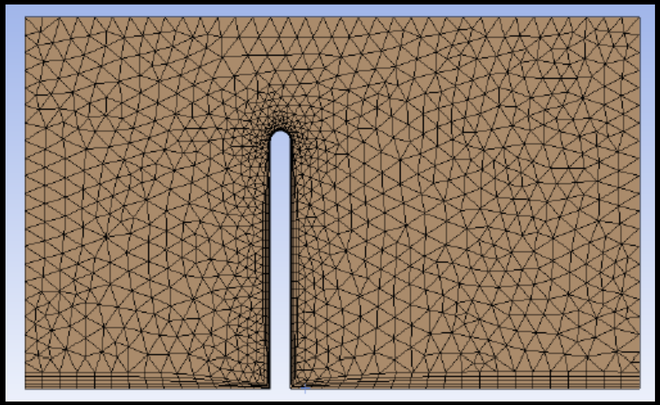


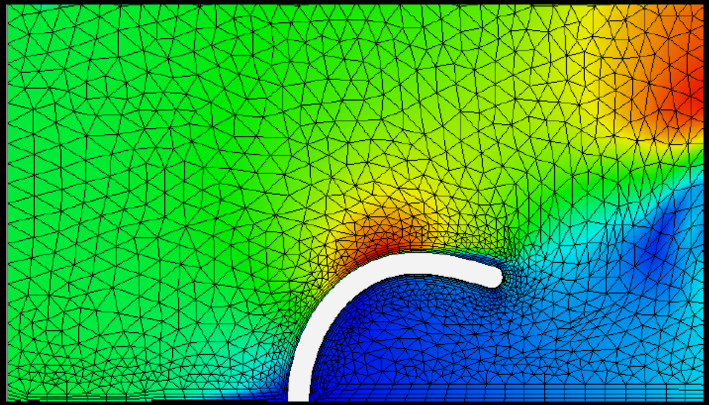
Building Deformed Geometry

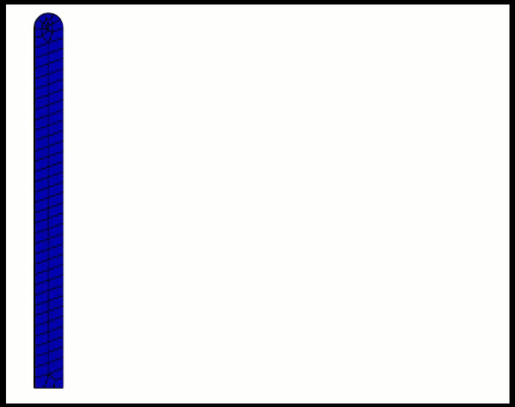


Recap

1. Convergence / Stability of FSI coupled systems
2. Gaps / Closure challenges
3. Dynamic Meshing
4. Prestressing Solids







Input/Questions