

ME EN 541 – Modeling Project 1

Modeling the Flow-Induced Deformation of a Compliant Beam in a Channel Wall

Due Feb. 9th at 4:30 pm in the ME 541 HW box (CTB 4th floor)

Overview

For this project you will duplicate the results of the flow-induced deformation of a compliant beam in a channel wall from Luo and Pedley (1995).¹ The purposes are to:

1. Mimic best practices in CFD
2. Gain experience using the commercial solver ADINA for studying fluid-solid interactions

Activities

1. Complete, if you haven't done so before, several ADINA tutorials ("primers"). ADINA is available on the CAEDM Linux machines.² The primers are found under **Help > Primer** after launching ADINA. I suggest completing tutorials 1, 2, 4, 6, 7, 10, 15, 18, and 28 before proceeding.
2. Study the model setup and results in the paper. See the attached thesis excerpt for help with model setup & nondimensionalization. Also see the attached tips & suggestions for generating and troubleshooting the model.
3. Duplicate Figure 3b (not 3a) to verify grid-independence.
 - a. In addition, show filled contour plots of vorticity for a coarse grid solution and your grid-independent solution. Refer to the attachment for help with generating vorticity results.
 - b. Report the number of nodes and elements used for each of your results.
 - c. Report the time required to solve each of the cases.
4. Using a plot similar to Fig. 3b, demonstrate convergence criteria independence. There are three convergence criteria that you'll need to alter: solid solver convergence, fluid solver convergence, and coupled FSI convergence. For convenience, you may set them all to the same value and adjust them together. Report the time required to solve each of the cases.
5. Duplicate Fig. 5 (a, b, & c) for values of $\beta = 1, 25, 100$, and 129. Include a copy of Fig. 5a,b,c from the paper. Discuss how well your results compare with theirs.
6. Investigate increasing β beyond 129. How high of a value of β can your model simulate while still maintaining a realistic pressure distribution (i.e., without the spurious fluctuations shown for $\beta = 130$ in Fig. 5b)? Show a contour plot of vorticity for the case with the highest value of β as well as a pressure distribution plot (like Fig. 5b) for this case.
7. Report details of your model: order of fluid solver, order of solid solver, number of elements and nodes in your converged model, etc.

Formatting Figures

Generating professional-quality figures is an important skill and habit. Thus in this assignment, as well as in all others in this class, it is expected that you will generate high-quality, well-formatted figures, like what you would expect to see in a good journal. I will give you a handout early in the semester with principles and tips for generating publication-quality figures.

Possible Timeline

Jan 8 – 17	Read paper & complete ADINA tutorials
Jan 18 – 24	Create low-fidelity working model
Jan 25 – 31	Perform grid & convergence criteria studies
Feb 1 – 9	Run final cases & prepare write-up

¹Luo XY, Pedley TJ. 1995. "A numerical simulation of steady flow in a 2-D collapsible channel," *Journal of Fluids and Structures* 9:149-174. PDF download: <http://www.maths.gla.ac.uk/~xl/JFS95.pdf>. Tim Pedley, used to be the editor for the *Journal of Fluid Mechanics*, the top journal in the field of fluid mechanics.

²ADINA v. 9.3 is available on the CAEDM Linux machines and can be accessed in the CAEDM labs (CTB 450, CB 425, CB 308) or via the HP RGS (Remote Graphics Software) client (see <https://caedm.et.byu.edu/wiki/index.php/RGS>). If you use RGS, select RGS Linux (not RGS Windows). ADINA is under **Applications > Caedm Apps**.

ME EN 541 – Modeling Project 1

Tips & Suggestions

General

1. The output (*.por) files can be opened in ADINA while the solution is progressing.
2. If the solver fails after solving at least one time step, the *.por files can still be opened to inspect the intermediate results and look for reasons as to why it might be failing.
3. Use SI units everywhere. The channel height should be 0.01 m. The inlet centerline velocity should be 0.015 m/s for $Re = 100$. (Re is based on average velocity, which is $2/3$ the maximum velocity for flow through parallel plates.)
4. At some point, generate and work from *.in rather than *.idb files. We can discuss this in class.

Solid Domain Setup

1. Try running the solid solver alone to see if the beam is deforming correctly, and compare the results with Fig. 4A.
2. The external pressure, p_e , should be set in the solid domain using a pressure boundary condition. Make sure it is directed downwards (towards the fluid domain), not upwards.
3. In the element group definition, use Plane Strain as the element sub-type.
4. In the “Define Element Group” window, set the Displacements field to “Large.”
5. In the solid domain, subdivide the line before meshing it. A rule of thumb for flow-structure interfaces is to use the same element spacing in the solid domain as in the fluid domain.
6. Only three degrees of freedom in the solid domain are needed (Control > DOFs): y-translation, z-translation, and x-rotation.
7. Use large deformation & small strain in the solid domain (Control > Kinematics).

Fluid Domain Setup

1. Use quadrilateral (4-node) elements, not triangular 3-node elements.
2. In the fluid domain, make sure the element sub-type is “Planar,” not “Axisymmetric.”
3. For the $Re = 500$ case, you need to use a time function to ramp up the fluid velocity. Otherwise the beam model will “balloon” outwards and the solver will fail. Note that “time function” is different than a “time step.” A time function in the fluid domain should be applied that ramps up the inlet velocity from a starting value of 0 m/s at time 0, then 0.075 m/s at some later time (like 5-10 seconds later). ADINA primer 7 describes how to work with time functions.

FSI Setup

1. In the FSI control box, increase the “Maximum Number of Fluid-Structure Iterations” beyond the default setting of 15. A better number would be around 100.
2. In the FSI control box, set the “FSI Solution Coupling” to “Direct.”
3. Make sure there is an FSI boundary condition applied to both solid and fluid models.

Initial Tension

1. In the “Define Initial Condition” window, check the box indicating that the initial strains should be interpreted as initial stresses.
2. After defining the initial strain, it still needs to be applied to the beam. Failing to do so would be like defining a material property, but then neglecting to tell the model what part of the model is to be composed of that material. Select Model > Initials conditions > Apply, and then apply the initial strain BC to the line. Use Strain-11.
3. For reference, an initial strain value of 200 Pa corresponds to $\beta = 80.5$.

4. Under the “Define Element Group” window, on the “Advanced” tab, change the “Applied Initial Strains” from “None” to “Nodal and Element.” This tells ADINA that the element group will receive the initial strain that has been defined.

Convergence Criteria

1. There are convergence criteria for the solid domain, for the fluid domain, and for the coupled fluid-solid interaction control. The following shows where these are found.
 - a. Solid domain: Solution > Solution Process > Iteration Tolerances. The solid convergence criteria is the “Energy Tolerance” field if the “Convergence Criteria” drop-down box is set to “Energy.”
 - b. Fluid domain: Control > Solution Process > Iteration Tolerances. The fluid convergence criteria is the “Relative Tolerance for Degrees of Freedom” field.
 - c. FSI control: From within the fluid model, click on the “c” with three dots underneath it that is next to the “with Structures” drop-down box. Set the “Convergence Criteria” drop-down box to “Force and Displacement/Velocity,” and the convergence criteria are the “Relative Force Tolerance” and the “Relative Displacement/Velocity Tolerance.”
2. Recall that for convenience you can set all criteria to the same value and change them together to explore convergence criteria-independence.

THESIS EXCERPT

The channel dimensions and flow parameters were selected to match those used by Luo and Pedley (1995, 1996): $H = 1$ cm and $L = 5$ cm for all cases; $L_u = 2$ cm, $L_d = 7$ cm, and $p_e = 0.93$ Pa for the steady cases, and $L_u = 5$ cm, $L_d = 30$ cm, and $p_e = 1.76$ Pa for the unsteady simulations. Luo and Pedley (1995, 1996) used $p_d = 0$ for both steady and unsteady cases. The downstream pressure, p_d , for the present study was also zero, although some cases with $p_d < 0$ were performed for grid-independence and convergence criteria studies. The fluid was water, with density $\rho_f = 10^3$ kg/m³ and viscosity $\mu = 10^{-3}$ kg/(m·s). The structural thickness was $d = 10^{-4}$ m (1% of the channel height), the density was $\rho_s = 1000$ kg/m³, and the Poisson's ratio was $\nu = 0$. The beam modulus of elasticity, E , was varied. The beam was assumed to have unit depth. The small value of the ratio $\rho_s d / \rho_f H = 0.01$ suggested minimal influence of wall inertia (Pedley and Luo, 1998).

The structure was a plane strain beam, modeled based on the Timoshenko beam model, which allows for large displacements with small strains.

The fluid domain boundary conditions were as follows:

$$u_1 = 600Uy(1-100y) \quad \text{Inlet (AB)} \quad (4.4a)$$

$$u_2 = 0 \quad \text{Inlet (AB), outlet (EF)} \quad (4.4b)$$

$$u_1 = u_2 = 0 \quad \text{Rigid walls (BC, DE, AF)} \quad (4.4c)$$

$$p = p_d \quad \text{Outlet (EF)} \quad (4.4d)$$

where u_1 and u_2 denote the x - and y -components of velocity, respectively. For all simulations, the mean velocity at the inlet was set to $U = 0.03$ m/s, for a Reynolds number $Re = \rho UH / \mu = 300$.

The beam was fixed and free to rotate ("pinned") at its endpoints (C and D), thus constituting a pinned-pinned structure. The difference between pinned-pinned and fixed-fixed beam boundary conditions is discussed in Section 4.1.7.

The flow domain was meshed using 9-node (second-order) quadrilateral elements. Time integration was performed using a second-order modified trapezoidal rule scheme. The structure

was meshed using 2-node isoparametric beam elements, which were formulated using the Total Lagrangian approach (Bathe, 1996).

The mesh for the flow domain was updated at each load or time step to preserve mesh quality. “Leader-follower” constraints were applied to guide the mesh movement by constraining the displacement of certain “follower” points to be equivalent to the displacement of specified “leader” points, as illustrated in Fig. 4.2. Points at locations *a* and *i* denote the beam endpoints. The following were leader-follower combinations: *c-b*, *c-d*, *e-f*, *g-h*, and *g-j*. These constraints did not affect the beam boundary position, but resulted in more regular and predictable mesh movement, with less element distortion.

The solution for the coupled fluid-structure system was obtained using either an iterative or a direct approach. For the iterative method, the fluid and structural domains were treated separately, using the most recent solution for one domain to obtain the solution for the other domain. These iterations were carried out until the coupled equations were satisfied. For the direct method, the fluid and structural equations were combined into a matrix system of equations and solved simultaneously. The direct method requires more memory, but is computationally faster than the iterative method. However, in cases where the beam deformation was large, when the upstream section of beam began to bulge outwards, the direct method failed and the iterative approach was required. Solutions of the matrix equations were obtained using a sparse matrix direct solver based on Gaussian elimination (Bathe, 1996).

Results for steady simulations were obtained starting with an initially undeformed channel and a straight beam aligned with the channel upper wall. Because of difficulties in obtaining converged solutions due to nonlinearities associated with very large beam deformations, it was necessary to incrementally apply the fluid inlet velocity and the external beam load. This minimized element distortion associated with rapid mesh deformation. A relaxation factor of 0.5 was typically applied to the solid displacement and the fluid stress terms in the matrix equations to facilitate convergence (ADINA 2002b).

Steady solutions supplied the initial conditions for the time-marching simulations. A small difference in selected parameters was enforced to perturb the system. This triggered the initiation of the dynamic response, usually leading eventually to a steady-state oscillation. This was typically accomplished by decreasing the downstream pressure, but occasionally the beam

modulus of elasticity was varied or the external pressure was temporarily increased. A relaxation factor of unity (no relaxation) was typically used for transient solutions.

The meshes considered in the unsteady simulation grid-independence study, meshes E, F, and G, are also listed in Table 4.1. The variable of interest was the wall position; the metric was the time-dependent vertical displacement of one beam node. Because the flow domain considered in the unsteady calculations was longer than the domain for the steady calculations, additional elements were added upstream and downstream of the deformable structure to accommodate the increased duct length. The mesh densities in the region of the beam were the same for meshes E, F, and G, as for meshes A, B, and C, respectively. The simulation parameters were $E = 13.4$ kPa, $p_d = -3$ Pa, $\varepsilon = 10^{-4}$, and the nondimensionalized time step was $\Delta t = 0.03$. Figure 4.4 shows the y -component of the wall position of one node on the beam, located near the position of minimum wall height in the corresponding steady solution. All of the results are very similar, and thus all meshes were deemed acceptable. Mesh G was used for efficiency.

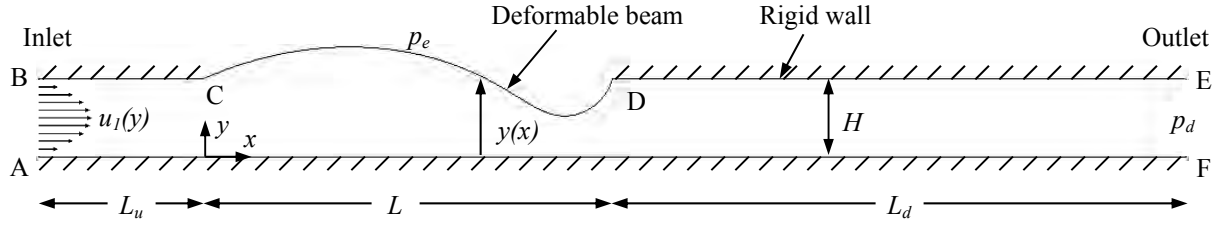
Time step size. The time step size had a more significant impact on computational cost than the convergence criteria or the density of the mesh. Figure 4.5 shows the wall position vs. time for nondimensional time step sizes, Δt , ranging from $\Delta t = 0.003$ to $\Delta t = 0.15$, with the same flow and structural parameters used in the unsteady grid and convergence studies. A time step of $\Delta t = 0.15$ was too large for the given parameters; the solution failed during the second oscillation cycle. The solution yielded a gradual increase in period as the time step size decreased. The phase error gradually increased over time, but the waveform was preserved. The differences between the $\Delta t = 0.015$ and 0.003 results showed discernable but acceptable phase differences. A value of $\Delta t = 0.015$ was chosen for the non-pre-stressed beam studies; a value $\Delta t = 0.03$ was similarly verified and used for the pre-stressed beam studies, for which the oscillation frequency was lower.

The model was verified by comparing the numerical model results with those of Luo and Pedley (1995, 1996). The Luo and Pedley cases used a pre-stressed membrane in which the tension was uniform and constant. To compare the beam results of the present numerical model with the membrane results of Luo and Pedley, the beam modulus was set to $E = 1$ Pa; this value is sufficiently small for the stiffness of the beam to be negligible in comparison to the pre-stress. This resulted in a case similar to a membrane with minimal bending stiffness, but longitudinal tension variation.

Figure 4.6 shows the beam position along the duct for initial tension values, T_i , ranging from $T_i = 2.22$ ($\beta_i = 80.5$) to $T_i = 33.3$ ($\beta_i = 5.37$). The final values of β , denoted β_f , were smaller than β_i because of the increased stretching due to beam deformation. The tension parameter, β_f , was calculated as $\beta_f = T^* / \bar{T}_f$, where \bar{T}_f is the averaged tension along the deformed beam:

Table 4.1. Number of elements and nodes used in the different meshes.

Mesh	Elements	Nodes	Simulation
A	4796	19665	Steady
B	2608	10791	Steady
C	1199	5037	Steady
D	324	1417	Steady
E	9790	40095	Unsteady
F	5328	22011	Unsteady
G	2442	10235	Unsteady



The equations solved were cast in dimensional form. However, for the presentation of results, the nondimensional form of Luo and Pedley (1995, 1996) and Cai and Luo (2002, 2003) was adopted for convenience. The nondimensional variables are

$$x = \bar{x} / \bar{H}, \quad y = \bar{y} / \bar{H}, \quad t = \bar{t} \bar{U} / \bar{H}, \quad p = \bar{p} / \rho \bar{U}^2, \quad T = \bar{T} / \rho \bar{U}^2 \bar{H}, \quad \omega = \bar{\omega} \bar{H} / \bar{U}, \quad (4.12)$$

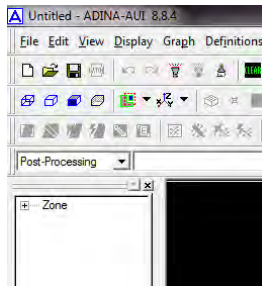
where \bar{H} is the channel height, \bar{U} is the mean inlet velocity, $\bar{\omega}$ is the vorticity, \bar{A} is the beam cross sectional area (thickness \times unit depth), and an overbar here denotes a dimensional quantity. A tension parameter β , used by Luo and Pedley (1995, 1996), was also used:

$$\beta = \frac{T^*}{\bar{T}}, \quad (4.13)$$

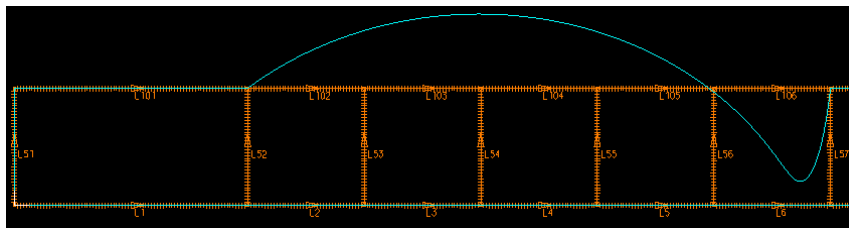
where $T^* = 1.61 \text{ N}$; thus a large value of β indicates a small tension.

Plotting Vorticity Results in ADINA

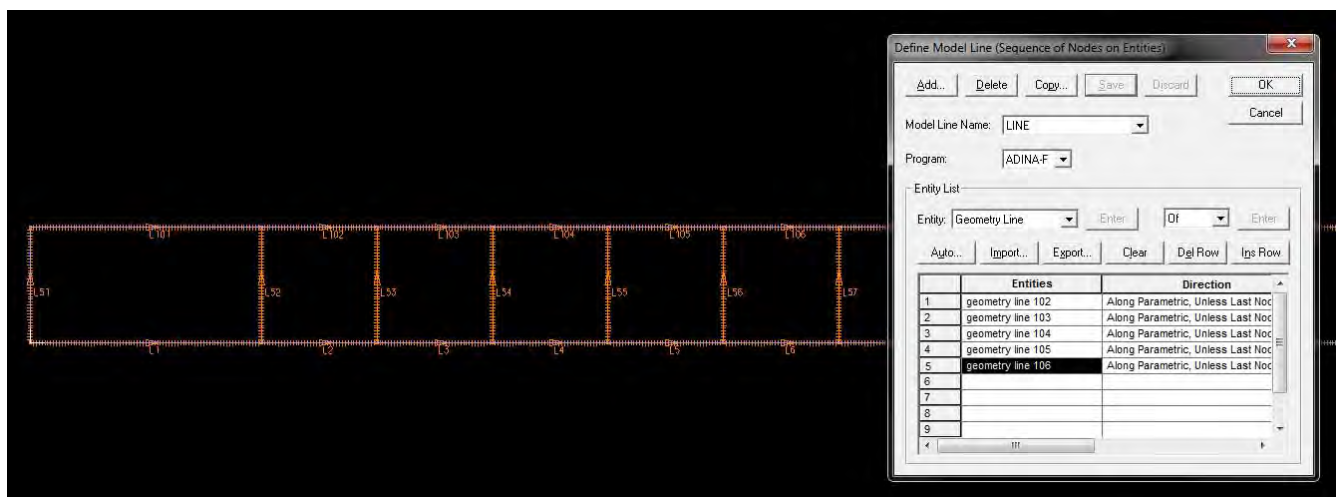
1. Select Post-Processing mode





2. Open fluids .idb file while in post-processing mode (File > Open, and change type to *.idb)
3. Without closing the .idb file, open the fluids .por file. You should see your results superimposed on your original geometry, such as:

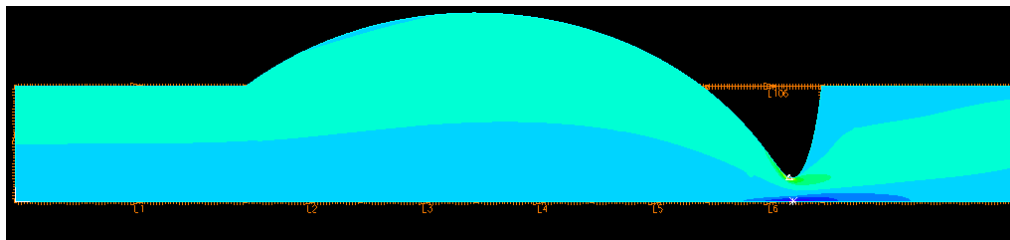
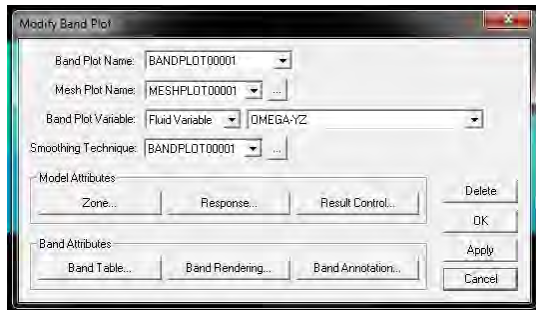


4. Define membrane line (or set of lines) by:
 - a. Selecting Definitions > Model Line > General
 - b. Add (enter name for line)
 - c. Selecting "Geometry Line" under "Entity," and enter the line numbers corresponding to the FSI interface. Keep the "Direction" as the default (Along Parametric) if lines are defined from left to right (as in figure below, in which the small arrows are pointing from left to right for the FSI interface lines 102 through 106).

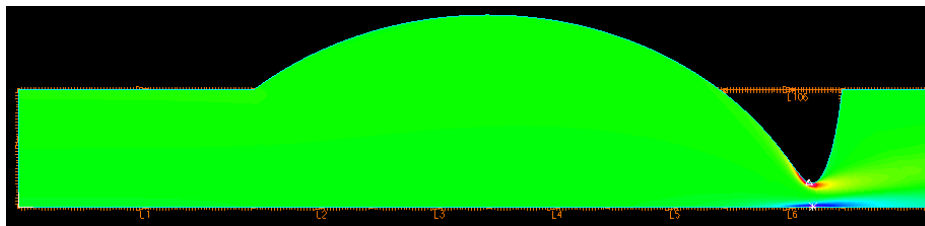


5. To plot vorticity contours:

- Click the band plot icon ()
- Click the modify band plot icon ()
- Set Band Plot Variable to Fluid Variable & OMEGA-YZ. Note that OMEGA is vorticity, and YZ is the out-of-plane component of vorticity.

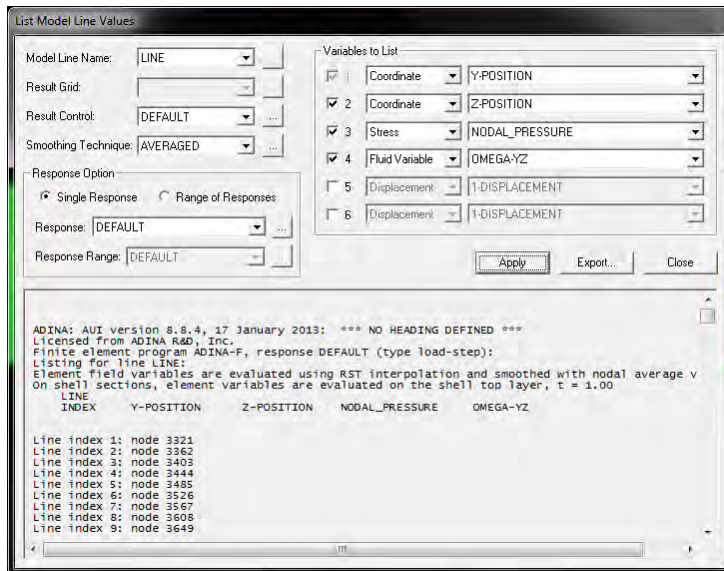


6. To make the plot look better, in the Modify Band Plot window, click Band Table and reduce the Value Range (-100 to 100 for the example shown here), and increase the number of colors to 64. For this problem, I suggest selecting a value range that is symmetric about 0.



7. To export the vorticity along the line,

- Click List > Value List > Model Line
- In Model Line Name, select the line you defined in step 4 above.
- In Smoothing Technique, choose Averaged.
- In Response Option, select the Single Response button.
- In Variables to List, choose Coordinate & Y-POSITION as the first variable.
- Check the 2nd Variables to List box to make it active. Choose Fluid Variable & OMEGA-YZ as the second variable.
- Optional: You could also output more variables, including Coordinate & Z-POSITION for the height profile, and Stress & NODAL PRESSURE for the pressure profile. Here is how I often like to export the results:



- h. Click Apply then Export, give it a filename, and then open it with Excel. For your plots, make sure you include the Y-POSITION data on the horizontal axis since the beam nodes will have moved downstream. Here is a plot of vorticity vs. position for this example model:

