PERGAMON

Computers and Structures 72 (1999) 1–16

Computers & Structures

Finite element analysis of fluid flows fully coupled with structural interactions

Klaus-Jürgen Bathe[a,*], Hou Zhang[b], Shanhong Ji[b]

[a]*Massachusetts Institute of Technology, Mechanical Engineering Dept., Cambridge, MA 02139, USA*
[b]*ADINA R&D, Inc., 71 Elton Avenue, Watertown, MA 02472, USA*

Abstract

Some advances in capabilities for analysis of fluid flows fully coupled with structural interactions are presented. Incompressible Navier–Stokes and compressible Navier–Stokes or Euler fluids and the full interaction with structures undergoing large deformations, nonlinear material response and contact conditions can be considered. The analysis capabilities are available in the ADINA System, and are integrated within computer-aided design using the available ADINA modeler and CAD interfaces. Various analysis cases are presented to illustrate the solution capabilities.

1. Introduction

The solution of structural problems and fluid flow problems is now well-established, although, of course, significant further advances in both fields are still much needed. In structural analysis, advances for the solution of highly nonlinear problems, such as encountered when considering mechanical coupling, are still much desired, whereas in fluid flow analysis significant improvements in analysing high Reynolds and Peclet number flows are still sought. Of great importance in both fields is also the establishment of precisely bounding and computable error measures for linear and nonlinear analysis [1].

A relatively new field of analysis is the solution of fully coupled fluid flows with structural interactions. Such analyses are the natural next step in modeling many physical problems more accurately, for example those pertaining to motor car brake systems, disk drives, compressors, the hydroplaning of tires, tall buildings, bridges and airplanes in severe weather conditions, and biological systems such as arterial blood flows through stenoses.

A simple but very restrictive approach to analyse such systems is to perform the fluid flow analysis first, assuming the structure to be rigid, and then, given the fluid forces acting onto the structure, perform the structural analysis. If the structure does not deform significantly and a steady-state analysis is sufficient, the complete fluid flow analysis is performed first, and then the structural analysis is carried out. In a transient analysis, the fluid flow conditions usually change during the time integration (for example, when valves open or close) and such changes would need to be incorporated using frequent restarts in the analysis process. Of course, large deformations in the structure cannot be taken into account using this approach. A key requirement is also that completely different meshes (based on different elements) can be used for the fluid and the structure which renders the force transfer—as to be performed in this simplified analysis

* Corresponding author. Tel.: +1-617-253-6645; fax: +1-617-253-2275.

0045-7949/99/$ - see front matter
PII: S0045-7949(99)00042-5

approach—from the fluid domain to the structure complicated.

The approach reported upon in this paper represents a very general procedure for the analysis of fluid flows with structural interactions. The fully coupled steady-state or transient analysis is performed using the ADINA program, for general linear or nonlinear structural conditions and incompressible or compressible fluid flows. The structure can be subjected to nonlinear material behavior and undergo very large displacements that have a drastic effect on the fluid flow and in turn on the structural conditions. The solution is obtained in a fully coupled manner at any time throughout the complete time domain considered, using an arbitrary Lagrangian–Eulerian formulation for the fluid and a Lagrangian formulation for the structure.

In the following sections of the paper, first presented are views of how, in practice, a fully coupled fluid–structure interaction (fsi) analysis might be performed. The paper then focuses on some capabilities available in ADINA to model the structural and fluid domains. This description includes a mentioning of some key aspects of the finite element solution procedures, regarding the equations solved, the iterative solvers and the mesh updating in the arbitrary Lagrangian–Eulerian formulation. Then the results of some illustrative analyses that demonstrate the solution capabilities are presented, and finally in the last section of the paper, conclusions are given regarding the current state of fsi analysis.

2. fsi analysis in engineering practice

To an increasing extent engineering analysis is being performed using computer-aided design tools to describe the geometry. Typically, the geometry has been generated using a CAD program such as Pro/E, SolidWorks, or I-DEAS, and the analysis is to be performed for the stresses, deflections, heat transfer, fluid flow or pressure distributions in the envisaged design. The complete analysis may involve posing a number of questions in solid mechanics, fluid mechanics and multiphysics. Traditionally, the solid and structural mechanics analyses are performed by a group of engineers using certain analysis programs and the fluid mechanics analyses are carried out by another group of analysts. Few analyses are conducted in which the interactions between the structural components and fluid flows are investigated, and then very simplified models are used. However, the possibilities to perform refined structural, fluid flow and interaction analyses have dramatically increased in recent years.

Assuming that the geometry of a design has been constructed with a CAD program, an important requirement is that the geometry can be modified for *analysis purposes*. The analysis requires in the first instance to construct an appropriate mathematical model [1]. This model should contain all the important ingredients to answer the analysis questions with confidence, but should not involve undue complexity. The preparation of this model frequently involves changing the given CAD geometry to remove details such as holes and chamfers that do not affect the analysis answers sought. Small geometric details require fine finite element meshes in these areas and if the details are not required, result in larger finite element systems to be solved than is necessary.

Considering ADINA, the CAD geometry would be read into ADINA-M (the ADINA System modeler) or be constructed in this modeler. ADINA-M is using as its kernel Parasolid, and hence any geometry built in a Parasolid-based CAD system can be directly loaded into ADINA-M, see Fig. 1. The program also accepts IGES files and Pro/E and AutoCAD geometry. For analysis purposes, the CAD geometry is then changed (that is, simplified) within ADINA-M. These changes surely depend on the complexity of the initial CAD

<u>use ADINA-M</u>: Parasolid-based modeler for ADINA

- Read CAD geometry from Parasolid-based CAD modeler, or
- Read IGES, Pro/E or AutoCAD files, or
- Construct CAD geometry
- Simplify/modify CAD geometry

<u>use AUI</u>: ADINA user interface

- Apply boundary conditions and loads
- Mesh surfaces and volumes
- Specify material data
- Create data input for ADINA

<u>use ADINA</u>:

- Solve the finite element model

<u>use AUI</u>:

- Post-process the computed results
- Visualize results

Fig. 1. Typical solution steps in an fsi problem.

data, on whether a structural, fluid flow or fluid–structure interaction analysis is to be conducted, and of course on the actual mathematical model to be solved.

In practice, the analyst best starts with the simplest possible model and increases the complexity as need arises. ADINA can be used effectively in this modeling process. For example, first a simple to complex structural analysis may be conducted, then a fluid flow analysis, a thermal analysis, and finally a fluid–flow structural interaction analysis corresponding to the multiphysics conditions may be pursued.

To perform these analyses requires the construction of the geometry, the generation of the finite element mesh, the specification of the loading, boundary conditions and material data, the analysis solution using the solver program, and then the post-processing and visualization of the analysis results, all performed in the ADINA System.

For the finite element meshing, the ADINA System offers mapped and free-form meshing capabilities. For mapped meshing, all element types can be employed, but only simple geometries can be meshed. The free-form meshing can be used for almost any geometry, but only triangular elements in two-dimensional and tetrahedral elements in three-dimensional conditions can be employed. An important feature is that a complex geometry can be broken up into simpler geometric domains and the different meshing tools can then be applied to each of these domains. In this way, the mapped meshing can be used in certain areas while free-form meshing is used in the rest of the geometry. The free-form meshing can be performed using an advancing front procedure or a Delaunay scheme with some control on minimizing sliver elements and for mesh grading. For fluid flow analysis, in particular, mesh grading in the boundary layers can be specified.

An interface for the use of I-DEAS and Patran with the ADINA System is also available. In this case, all geometry construction, meshing and post-processing is performed in I-DEAS or Patran, while the solution of the finite element model is carried out using ADINA.

An example demonstrating the input preparation for an fsi analysis is shown in Fig. 2. In this case, the analysis of a flow distributor is considered. The struc-

(a)

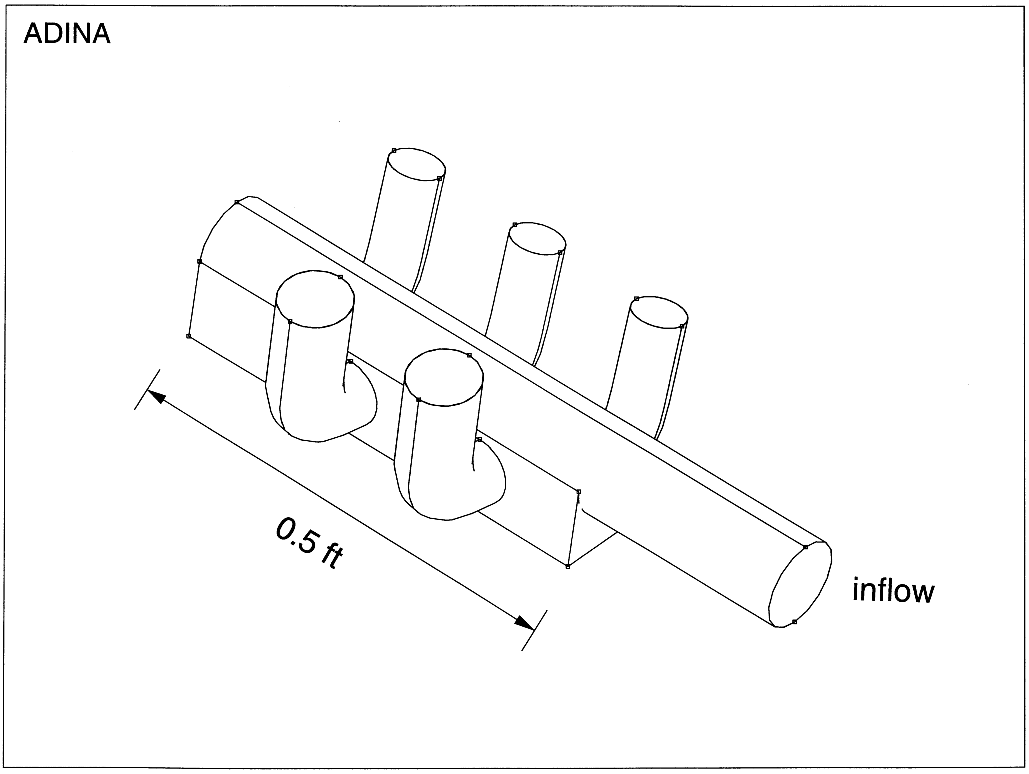


Fig. 2. Analysis of flow distributor. (a) Geometry lines and surfaces. (b) Shaded image of geometry. (c) Shell finite element mesh. (d) Fluid finite element mesh.

(b)

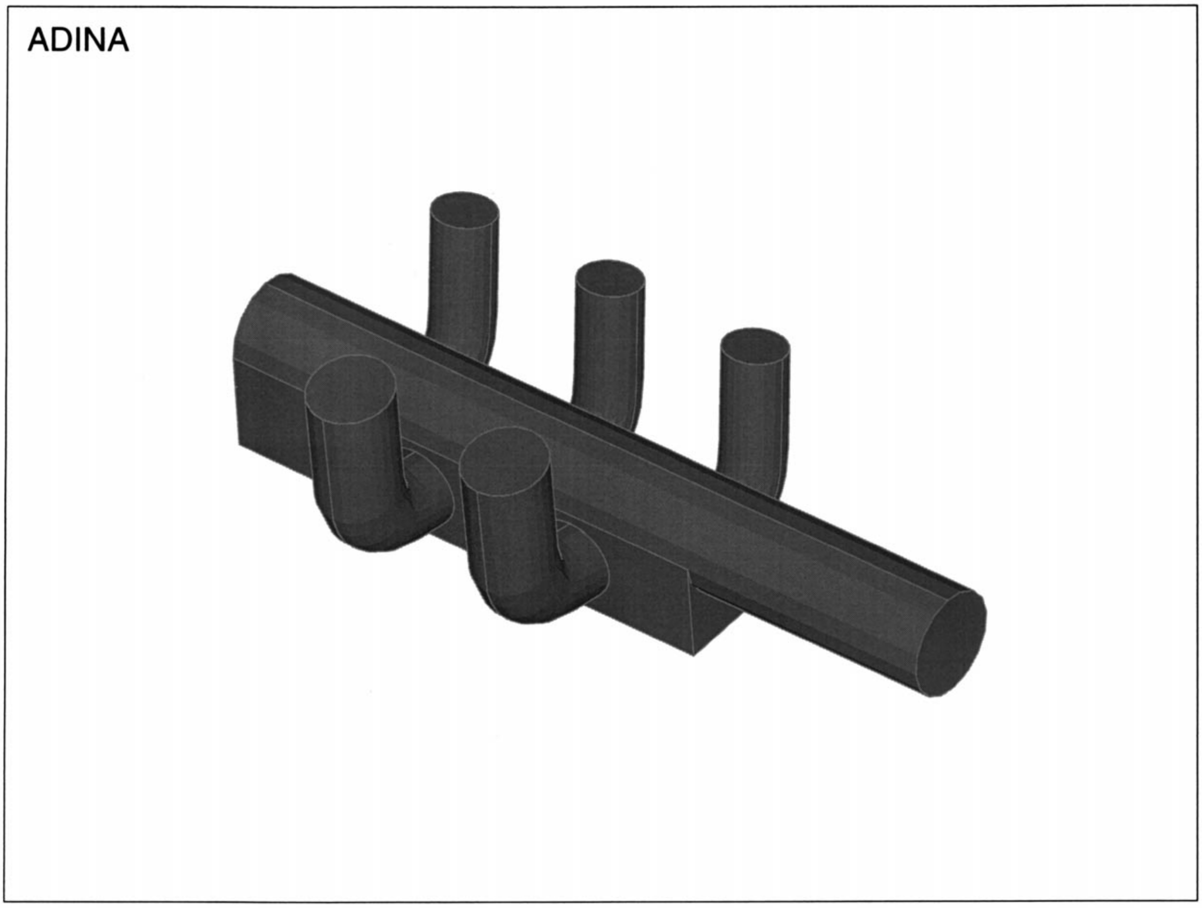


Fig. 2 (*continued*)

ture is a flexible thin shell and the device is used to distribute the flow of a quite viscous fluid. The CAD program SolidWorks was used to construct the geometry, which was then loaded into ADINA-M. There was no need to remove holes or other details. Using the free-form mesher for the complete system, the shell structure was meshed using the MITC 4-node shell element, and the fluid domain was meshed using the tetrahedral 3D fluid flow element [1–3]. Fig. 2 shows these meshes, where it is seen that the structural mesh is considerably coarser than the fluid mesh; that is, a number of fluid elements abut to a single shell element. This is, of course, an important requirement for typical fsi analyses. The results of this analysis are given in Section 4.1.

3. ADINA capabilities for fsi

Consider a generic domain partly fluid and partly solid, as schematically shown in Fig. 3. Note that this domain includes free surface(s) of the fluid and of course the fluid–structure interfaces. Our objective is to identify a mathematical model for the domain and solve this model using finite element procedures.

The solid is mathematically modeled using the classical Lagrangian formulations, whereas the fluid is modeled using an arbitrary Lagrangian–Eulerian (ALE) formulation of the Navier–Stokes equations [1]. The fluid can be a fully incompressible, a slightly compressible or a fully compressible medium. For the fully compressible case, the Euler fluid conditions (no vis-

(c)

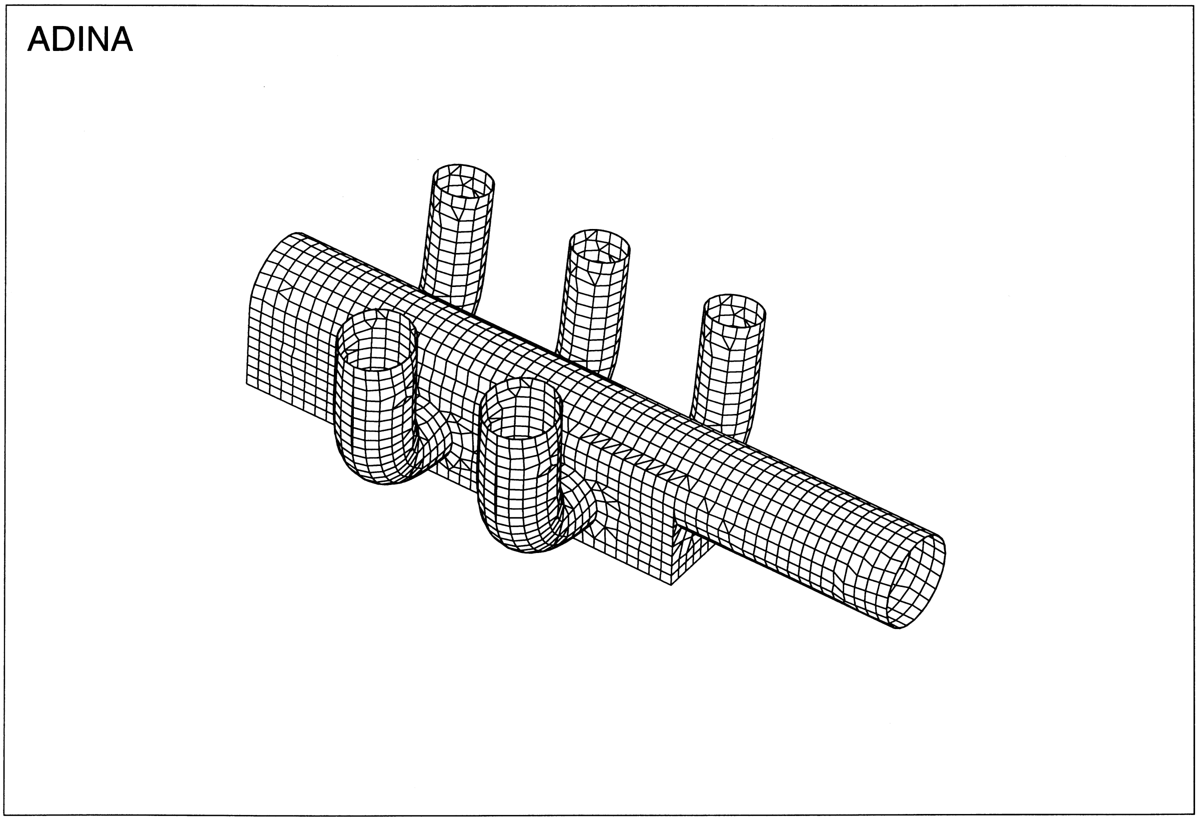


Fig. 2 (*continued*)

cous effects) can also be assumed. The solid can be an actual two- or three-dimensional (2D or 3D) solid, or a beam, plate or shell structure.

The equations governing the solid and fluid response have been detailed in Refs. [1–3], where the mathematical model equations and the finite element discretizations used in ADINA have been summarized. Note that the solid/structural domains can undergo very large motions with elastic or inelastic material conditions and can involve contact conditions [1,4]. The fluid can contain free surfaces and is coupled to the structure by satisfying the kinematic and equilibrium conditions between the fluid and structural parts at the fluid–structure interfaces. The kinematic conditions are the no-slip condition for the Navier–Stokes fluid and the tangential slip condition when the special case of an Euler fluid is assumed.

An important feature for the analysis of fsi problems is the ALE formulation for the fluid domain, in which the total time derivative for all the solution variables is given by [2,3],

$$\frac{\mathrm{d}(\cdot)}{\mathrm{d}t} = \frac{\delta(\cdot)}{\delta t} + ((\mathbf{v} - \mathbf{v}_\mathrm{m}) \cdot \nabla)(\cdot) \tag{1}$$

where $\delta(\cdot)/\delta t$ is the transient term at the mesh position considered. The mesh velocity at that position is given by $\mathbf{v}_\mathrm{m}$ and the actual fluid particle velocity is $\mathbf{v}$. In the solution $\mathbf{v}_\mathrm{m}$ is prescribed by the algorithm and must be chosen to achieve a stable and accurate solution. The primary purpose of using the ALE formulation is to preserve a good mesh quality when a change to the fluid domain is imposed by a free surface or a fluid–structure interface. In ADINA, an algorithm can be employed based on solving the Laplace equation for nodal positions in simple domains [2,3].

Using ADINA, the solid/structure is meshed using element groups, and the fluid is meshed independently in groups, but using of course the same pre-processor.

(d)

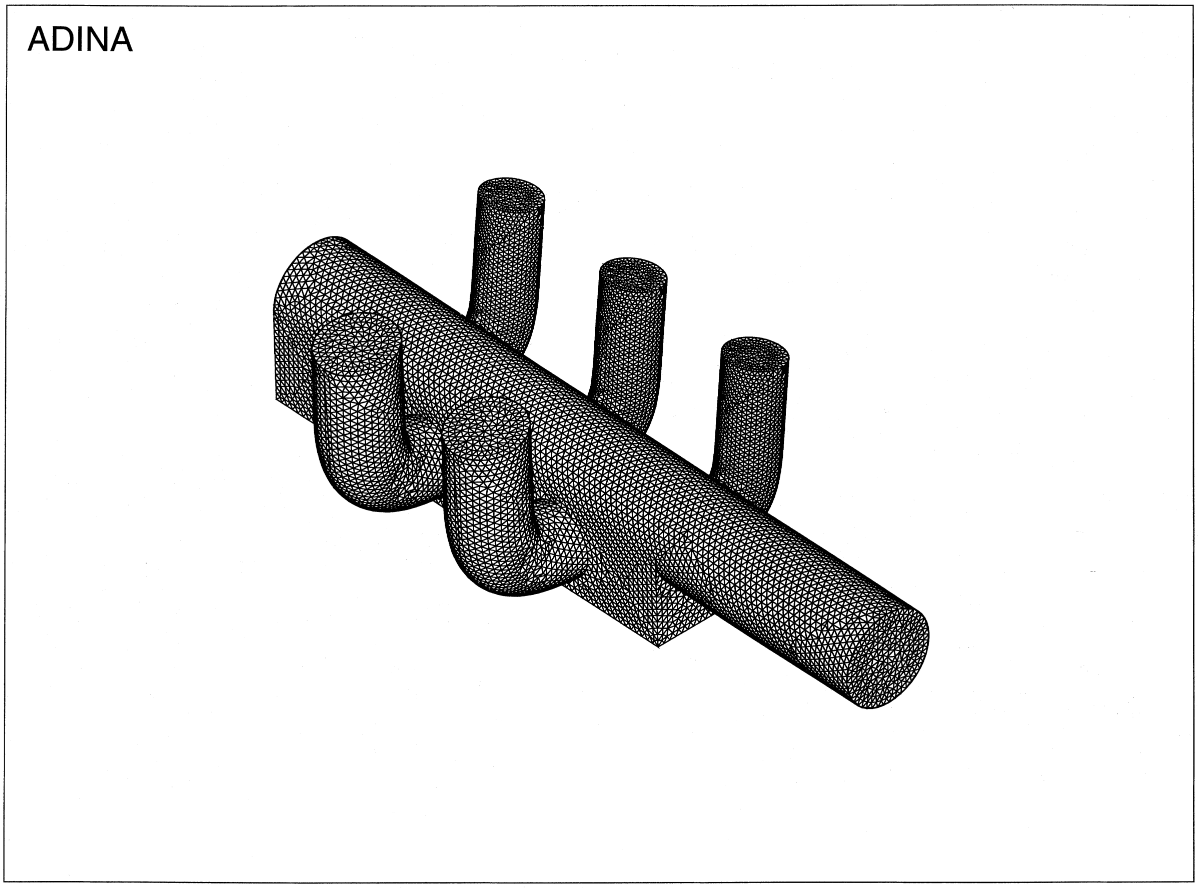


Fig. 2 (*continued*)

The fluid–structure interfaces are defined on the geometry level by lines (in 2D analyses) and surfaces (in 3D analyses).

Assume that the finite element discretization has been performed. The governing finite element equations to be solved are then, for each discrete time t selected in the step-by-step solution,

$$ {}^{t}\mathbf{F} = {}^{t}\mathbf{R} \qquad (2) $$

where

$$ {}^{t}\mathbf{F} = \begin{bmatrix} {}^{t}\mathbf{F}^{\mathrm{F}} \\ {}^{t}\mathbf{F}^{\mathrm{I}} \\ {}^{t}\mathbf{F}^{\mathrm{S}} \end{bmatrix}; \quad {}^{t}\mathbf{R} = \begin{bmatrix} {}^{t}\mathbf{R}^{\mathrm{F}} \\ {}^{t}\mathbf{R}^{\mathrm{I}} \\ {}^{t}\mathbf{R}^{\mathrm{S}} \end{bmatrix} \qquad (3) $$

Here the vector ${}^{t}\mathbf{F}$ lists the element nodal point forces corresponding to the element internal stresses and the

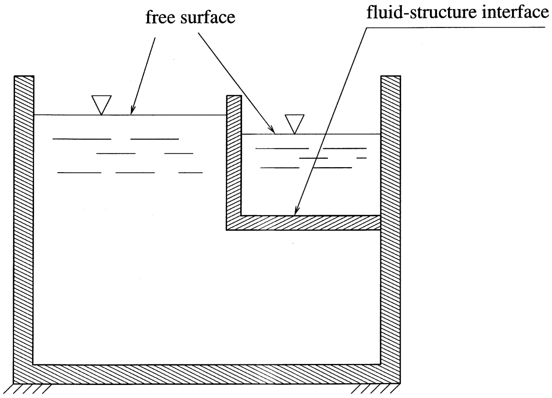


Fig. 3. Schematic of fsi problem.

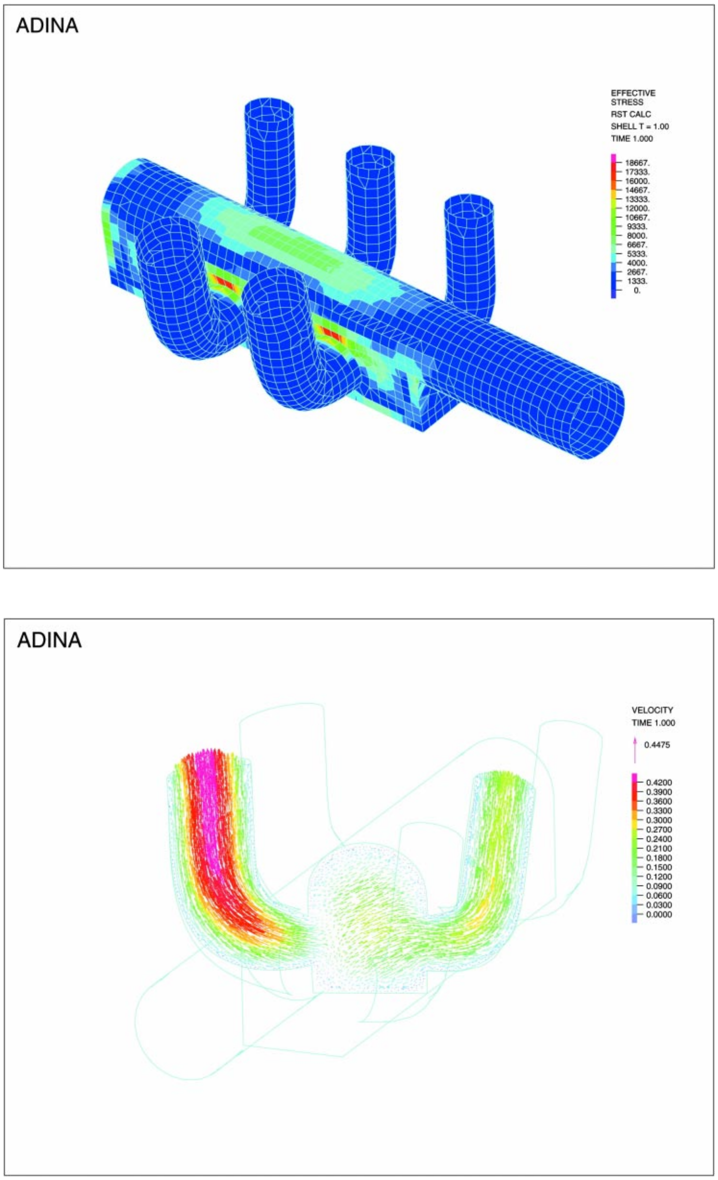


Fig. 4. Some solution results for analysis of flow distributor (for units see Table 1). (Top) Effective stress in shell. (Bottom) Velocity and pressure distributions at a section.

vector ${}^{t}\mathbf{R}$ lists the externally applied nodal point forces including the inertia forces. In each vector, the forces corresponding to the fluid domains (superscript F), fluid–structure interfaces (superscript I) and solid/structural domains (superscript S) are listed. In Eq. (2), the interface equations involve the fluid and solid element meshes and describe the compatibility and force transfer conditions along the interfaces for different element types and meshes in the solid and fluid domains.

It should be noted that Eq.(2) contains *all* the ingredients and conditions for a *fully* coupled, steady-state or transient analysis of the fluid–solid system. There are no additional conditions to be satisfied for a fully coupled analysis of the system.

In general, Eq. (2) is highly nonlinear in the fluid

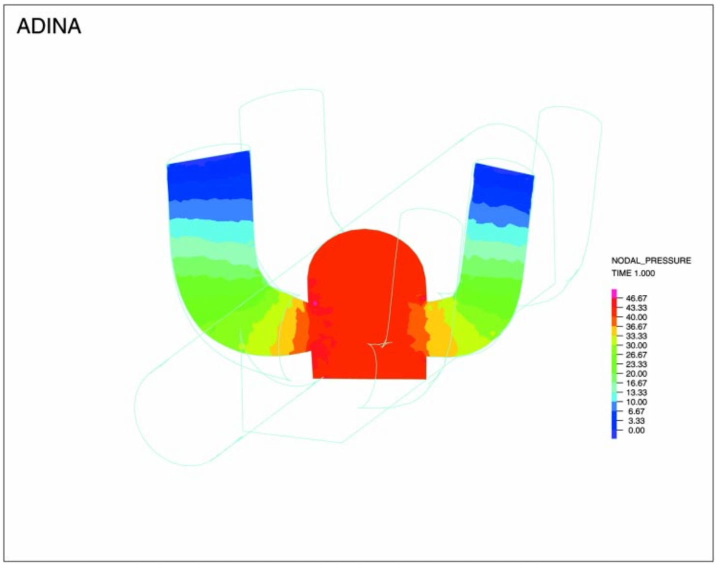


Fig. 4 (*continued*)

velocities and structural displacements. In addition, the number of equations can be very large. Of course, various solution strategies can be pursued. In structural analysis, Newton–Raphson iteration is frequently most effective, in which the resulting matrix equations are solved using a sparse or an iterative solver [1]. In fluid–flow analysis, successive substitution and Gauss–Seidel type iterative schemes are widely employed, but Newton–Raphson iteration can also be effective. The convergence in the iterations is frequently improved by nondimensionalizing the fluid equations. An option is available in ADINA to have the program carry out this nondimensionalization automatically based on user-specified characteristic values of length, velocity, etc. Using the Newton–Raphson method, the resulting matrix equations are solved with an iterative scheme such as the biconjugate gradient technique when the number of fluid equations is very large. A sparse solver is, however, effective if the number of equations considered is not too large (say less than one-quarter million equations).

In ADINA, the user can select how to solve Eq. (2). For the nonlinearities, Newton–Raphson iterations can be used for the solid and the fluid, and simple successive substitution can be employed for the fluid. For the interface conditions, successive substitution is used with an acceleration scheme. To solve the matrix equations of the fluid and structural domains, sparse solvers or iterative solvers with pre-conditioners (conjugate gradient and multigrid methods for the structure, and biconjugate gradient, GMRES and multigrid methods for the fluid) can be used.

4. ADINA sample solutions

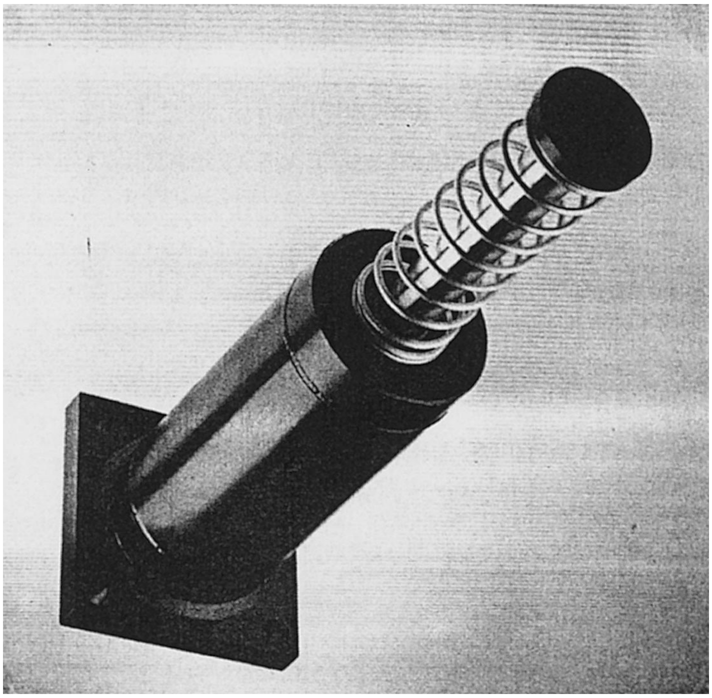
The objective in this section is to present some fsi solutions obtained with ADINA. These solutions illustrate the current capabilities available. Some additional solutions using ADINA have been given, for example, in Refs. [2,5–10].

4.1. Analysis of flow distributor

The geometry and meshing of this flow device were already presented in Section 2, see Fig. 2. Table 1 lists the material properties used. In this analysis, the deformations of the structure are not very large, but the fluid pressure exerts considerable forces on the structure. The coupled analysis gives the flow rates in the

Table 1
Material properties in flow distributor problem

Fluid	Structure
$\mu = 0.1413$ lbm/ft s	$E = 2.0 \times 10^6$ lbf/ft^2
$\rho = 54.69$ lbm/ft^3	$\nu = 0.45$



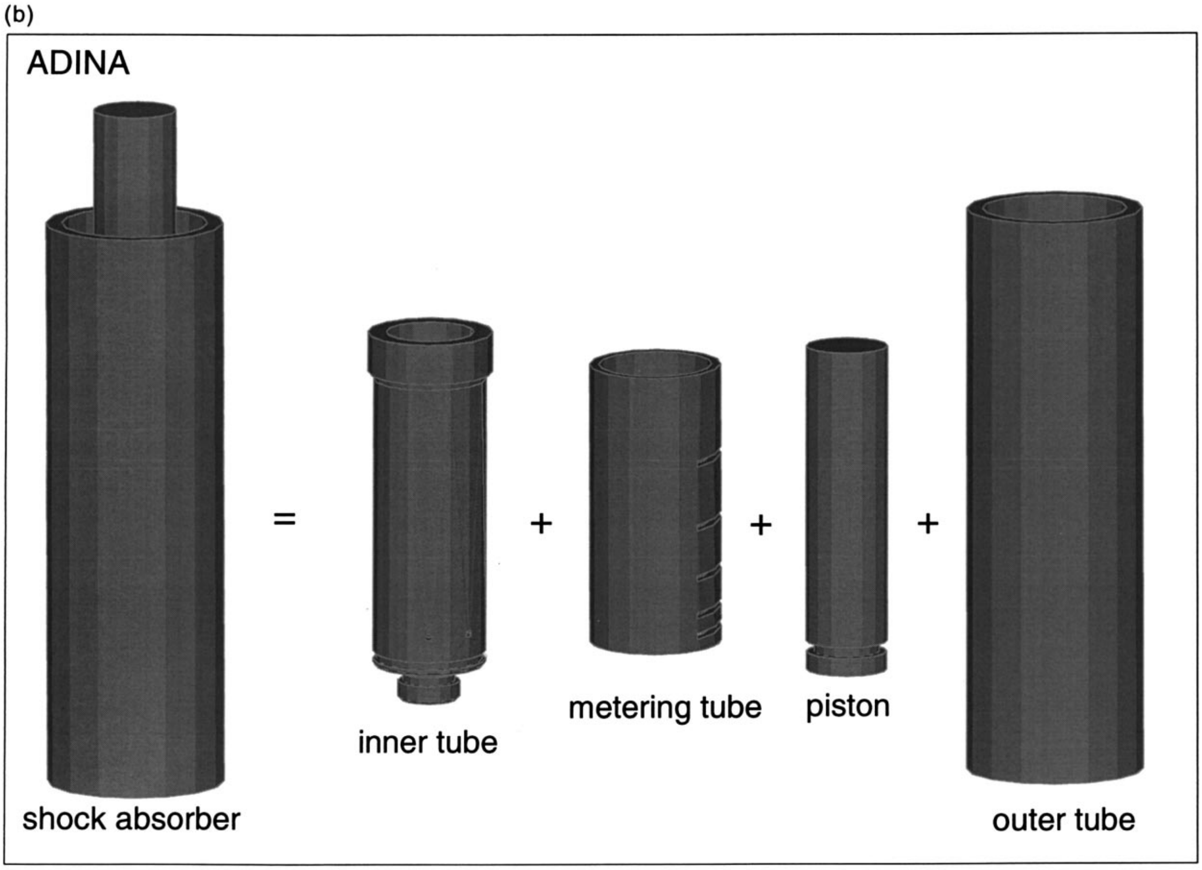
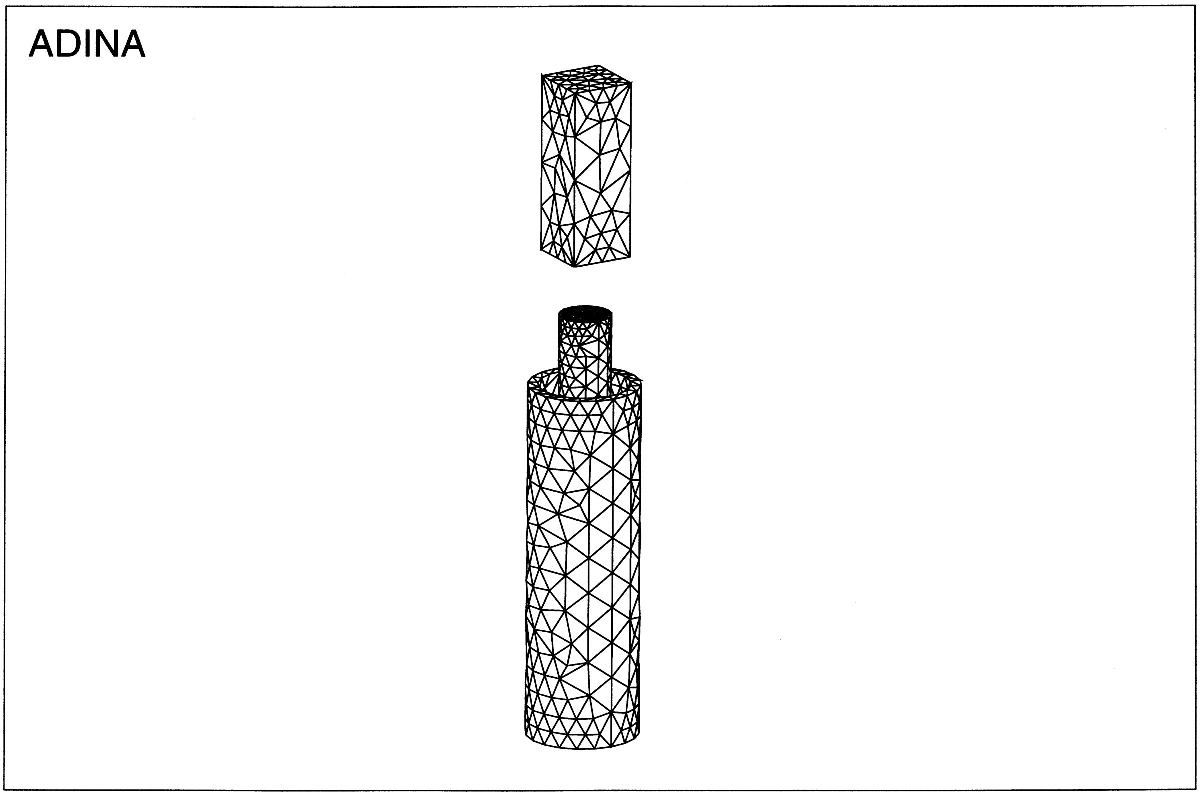


Fig. 5. Analysis of shock absorber. (a) Typical device. (b) Geometric entities. (c) Mesh of solid domain. (d) Mesh of fluid domain. (e) Solution results, reaction force versus stroke.

(c)



(d)

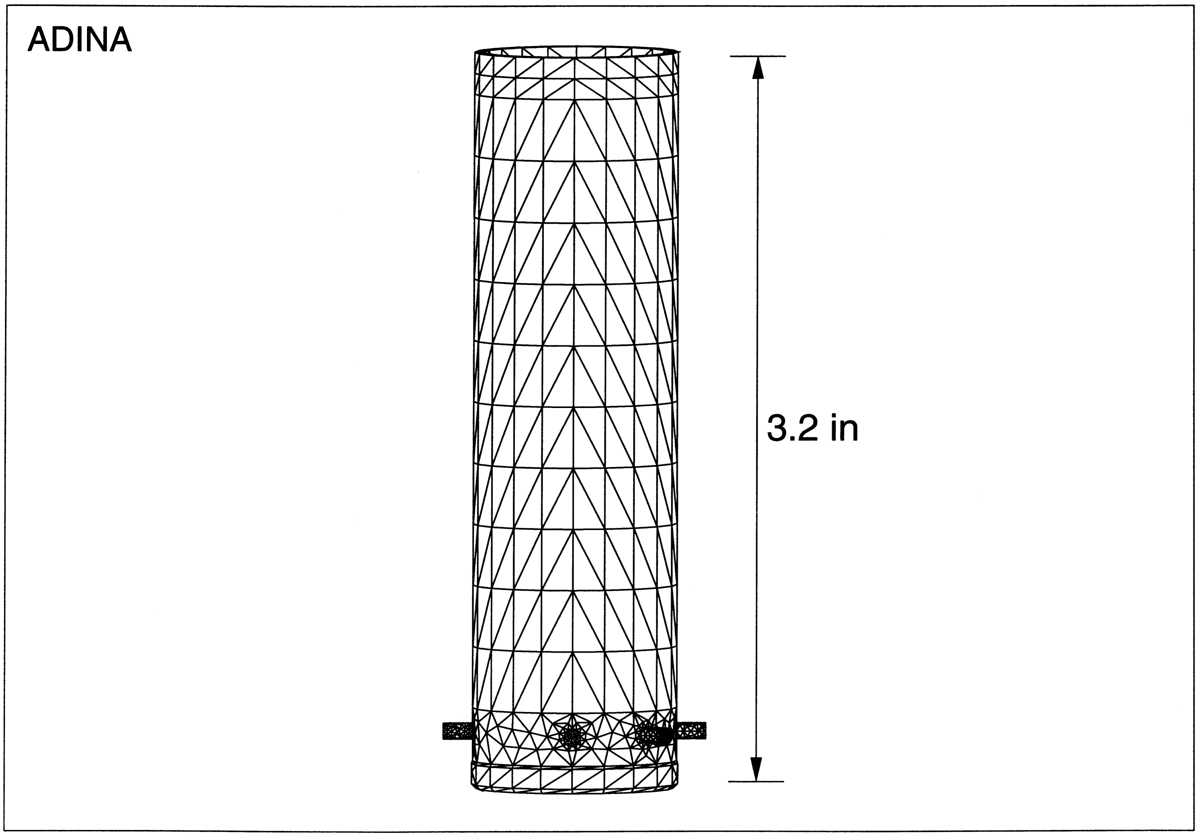


Fig. 5 (*continued*)

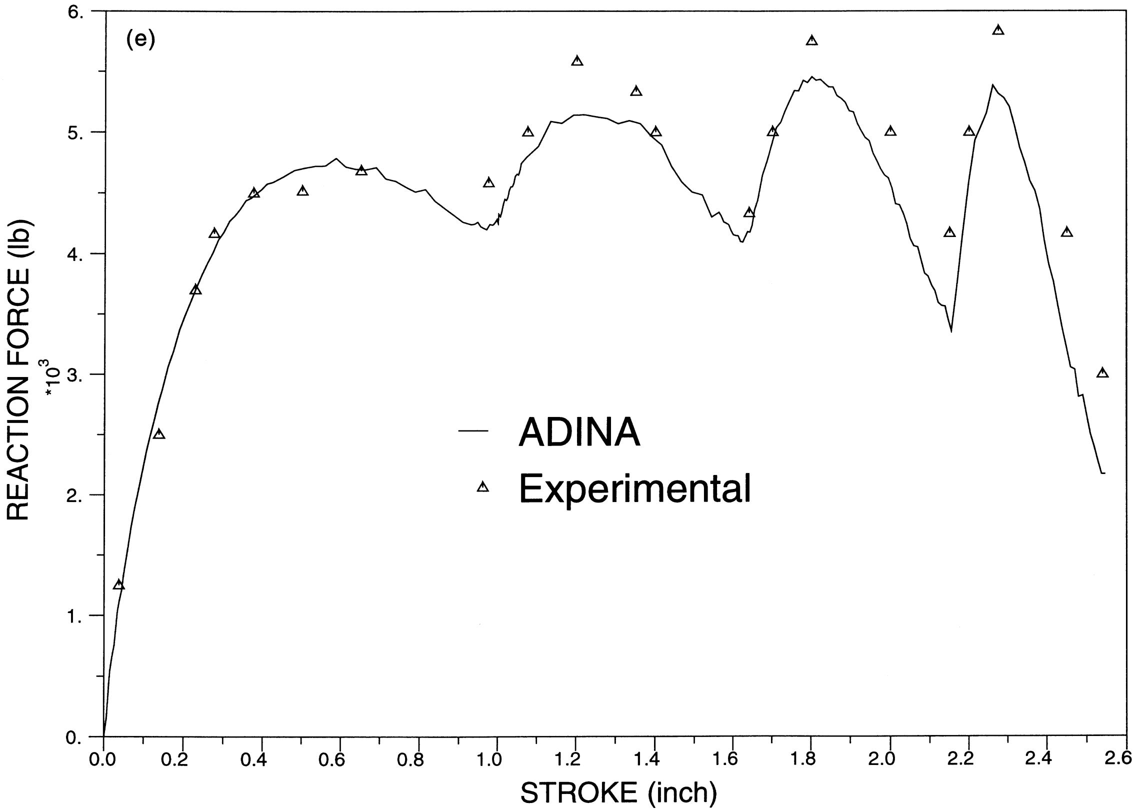


Fig. 5 (*continued*)

various sections of the device, the pressure and viscous stresses in the fluid and the stress distributions in the structure, all in one analysis run.

Some calculated quantities are shown in Fig. 4.

4.2. Analysis of shock absorber

A shock absorber, see Fig. 5, is subjected to a weight dropping on it. A laboratory experiment was conducted to measure the reaction force as a function of the stroke. ADINA was used to analyse the problem, with the aim to obtain detailed stress distributions in the structure. In the analysis, the fluid was assumed to be an almost incompressible Navier–Stokes fluid, and the structure was a solid with a part of it undergoing large displacements. Table 2 lists the material properties.

For the analysis, ADINA-M was used to construct the geometry, shown in Fig. 5(b). Note that the shock is absorbed by a piston pressing the fluid out of the opening at the bottom of the structure. The finite element meshes used are shown in Fig. 5(c) for the structure and Fig. 5(d) for the fluid. These meshes are quite coarse and yet, as seen in Fig. 5(e), the calculated force-stroke relationship is reasonably close to the experimental results. Only one single analysis of the problem was conducted without any tuning of the model.

A key point is that the overall length of the shock absorber is about 3.2 in, and the maximum stroke of

Table 2
Material properties in shock absorber problem[a]

Fluid	Structure
$\mu = 0.8058$ g/in s	$E = 5.2578 \times 10^{12}$ g/in s^2
$\rho = 14.4008956$ g/in^3	$\nu = 0.3$
$\kappa = 4 \times 10^{10}$ g/in s^2	$\rho = 127.8186$ g/in^3

[a] Mass of the dropping weight = 800 lb, initial velocity of the weight: 8 ft/s.

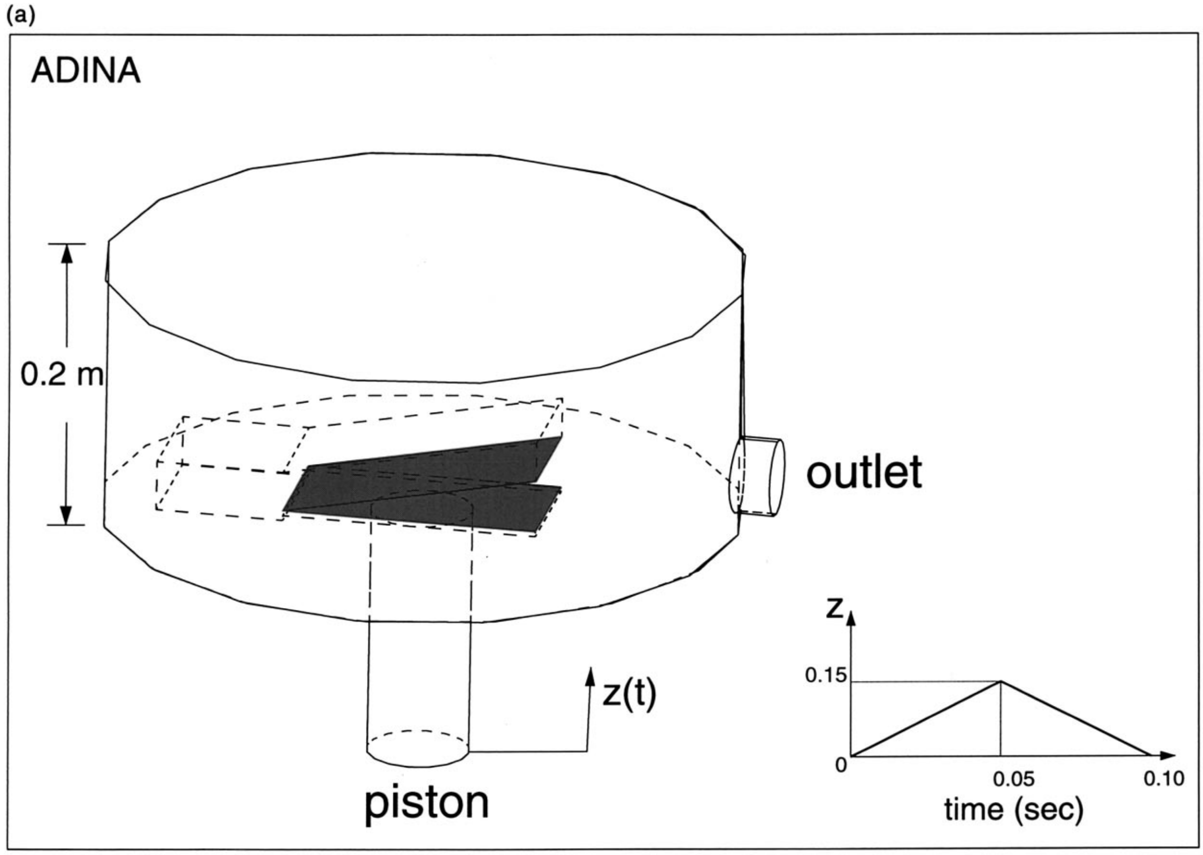


Fig. 6. Analysis of air compressor (for units see Table 3). (a) Geometry of compressor and displacement of piston. (b) Mesh of fluid. (c) Flow and pressure results at various times, t = 1, 47, 65 and 99 (millisecs). (d) Valve opening as a function of time.

the piston is about 2.5 in. Hence, the fluid domain is compressed by 2.5 in for an overall length of about 3.2 in. This compression of the fluid domain requires a 'mesh compression' of that magnitude, which is performed effectively using the arbitrary Lagrangian–Eulerian formulation used in ADINA.

4.3. Analysis of air compressor

The air compressor shown in Fig. 6 was analysed for the flow and structural response. In this analysis, the outer structure was assumed to be rigid, and only the valve was modeled as a flexible structure. The fluid (air) was assumed to be a fully compressible fluid governed by the corresponding Navier–Stokes equations. Table 3 lists the material properties used for the structure and the fluid. The valve is initially closed, opens as the piston moves up, and then closes again as the piston returns to its original position. The imposed motion of the piston is given in Fig. 6(a).

Fig. 6(b) shows the mesh used for the fluid domain and Fig. 6(c) shows some calculated flow and pressure results in the compressor. Fig. 6(d) shows the calculated opening of the valve. It is seen that the valve

Table 3
Material properties in air compressor problem

Air	Structure
$c_p = 1004.5$ m^2/s^2 K	$E = 2 \times 10^7$ N/m^2
$c_v = 717.5$ m^2/s^2 K	$\nu = 0.3$
$\mu = 1.5 \times 10^{-5}$ kg/m s	$\rho = 3900$ kg/m^3
$\rho = 1$ kg/m^3	
$k = 0.01$ N/s K	

(b)

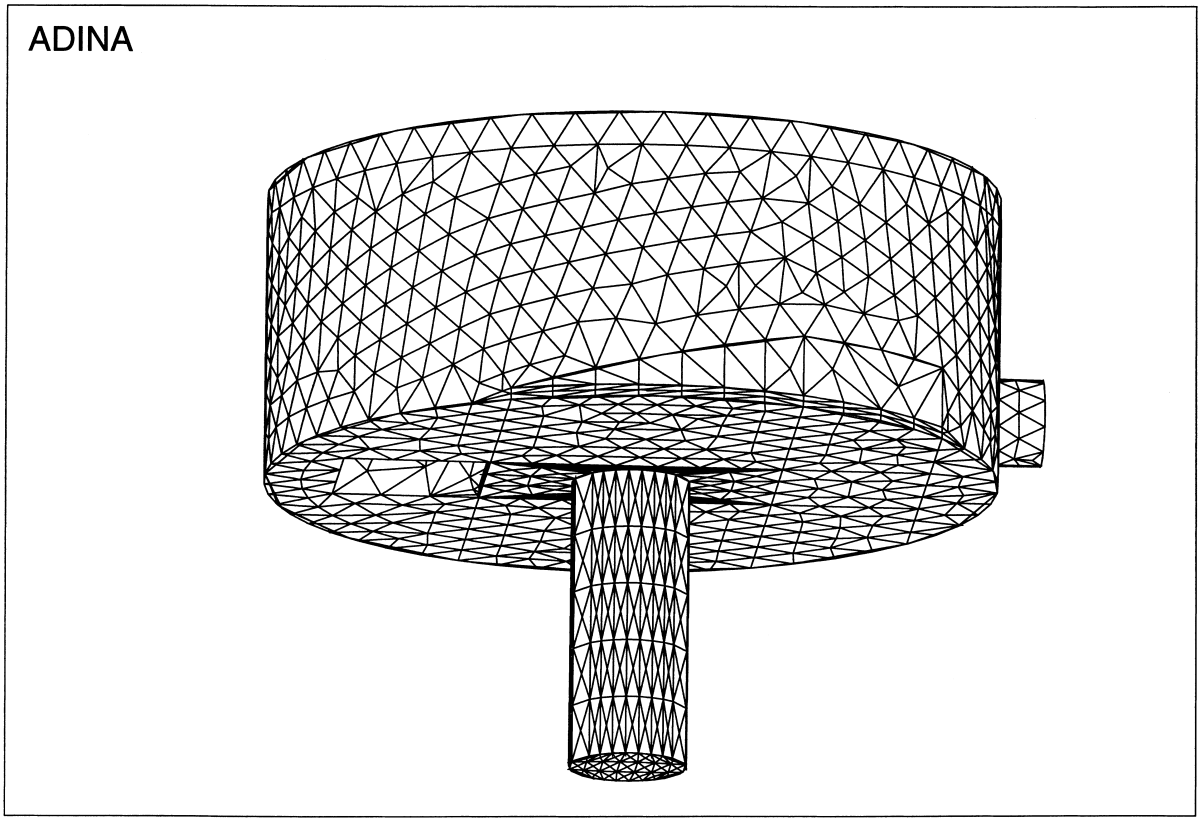


Fig. 6 (*continued*)

opening has some delay to reach the maximum opening, measured on the piston movement, and that because of the time stepping used the valve closure is only detected after, in fact, the valve has already overshot the closed-condition.

5. Concluding remarks

The objective in this paper was to present some advances in capabilities for the analysis of fluid flows with structural interactions. The structural and fluid domains can be of a very general nature, that is, of complex geometries, with the structure undergoing large deformations and the fluid governed by the incompressible or compressible Navier–Stokes equations. An arbitrary Lagrangian–Eulerian formulation is used to solve for the fluid response with structural interface and free surface conditions.

The key to successful solutions in engineering practice is that the capabilities can be employed in the CAD environment. This usage is achieved with some important solution ingredients: widely-employed CAD packages can be used to define the original geometry; the finite element system can be employed to modify the geometry for analysis purposes and to define the analysis parameters; the fluid and structural domains can be meshed automatically; and the arbitrary Lagrangian–Eulerian formulation is sufficiently versatile to accommodate the possibly large motions of the fluid boundaries.

The ADINA System has been developed to offer these capabilities, but of course further advances in these areas will be pursued. The current state of the analysis capabilities and the continuous further advances should lead to many exciting applications in the field of fluid flows with structural interactions.

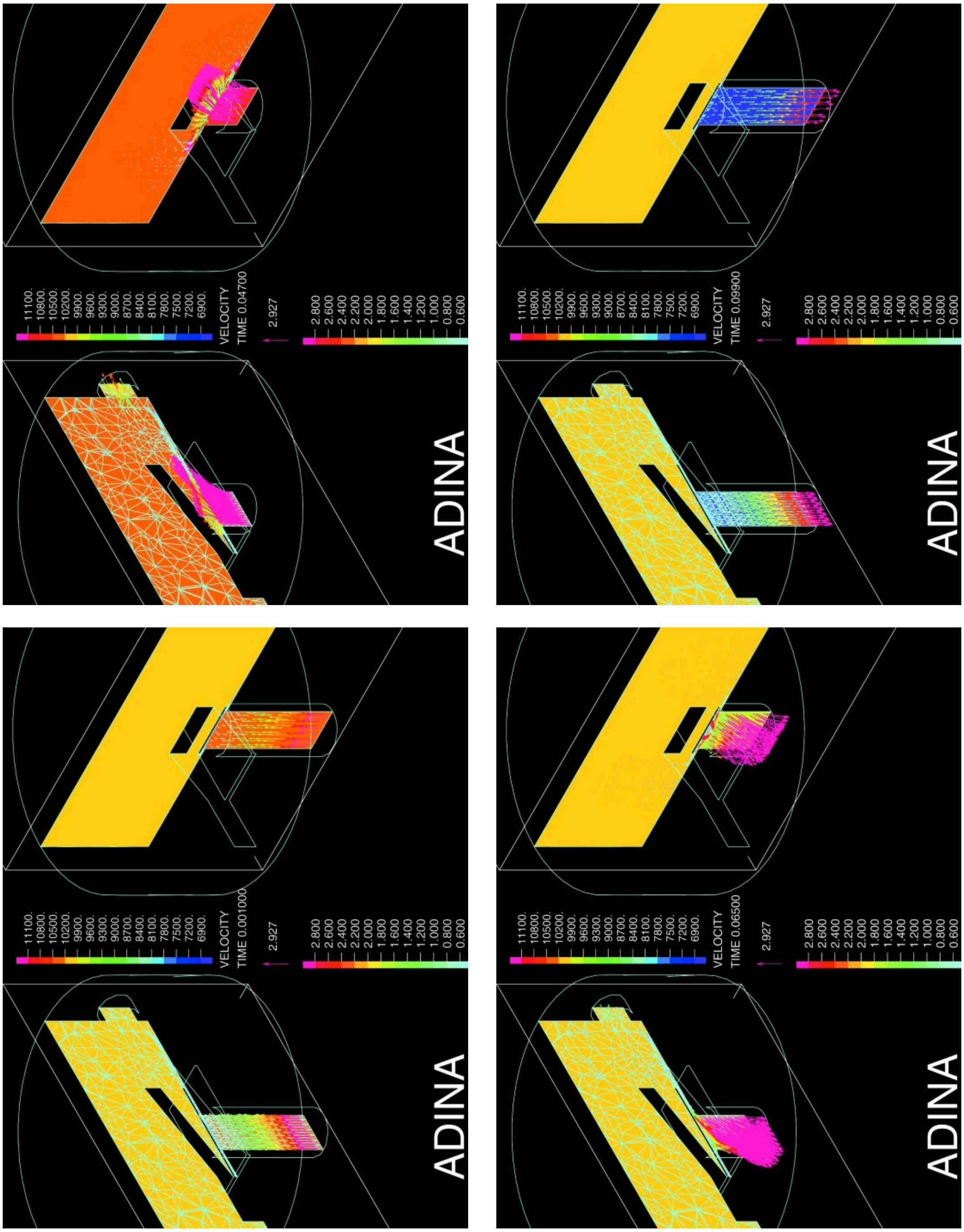


Fig. 6 (continued)

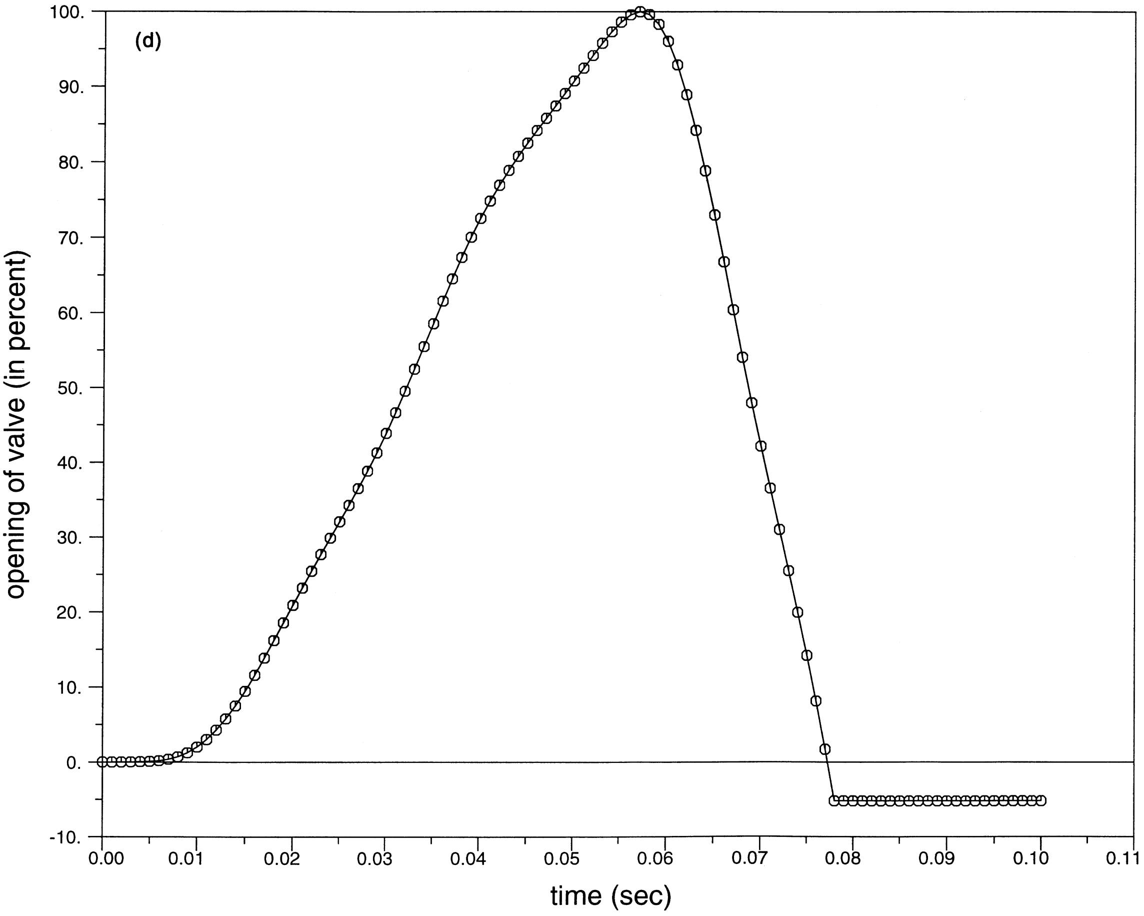


Fig. 6 (*continued*)

References

[1] Bathe KJ. Finite element procedures. Prentice Hall, 1996.
[2] Bathe KJ, Zhang H, Zhang X. Some advances in the analysis of fluid flows. Computers & Structures 1997;64(5/6):909–30.
[3] ADINA R&D Inc. ADINA Theory and Modeling Guides, Reports ARD 97-7 and 97-8.
[4] Bathe KJ, Bouzinov PA. On the constraint function method for contact problems. Computers & Structures 1997;64(5/6):1069–85.
[5] Bathe M, Kamm RD. A fluid–structure interaction finite element analysis of pulsatile blood flow through a compliant stenotic artery, ASME, J Biomechanical Eng (in press).
[6] Wang X. Finite element analysis of air-sheet interactions and flutter suppression devices. Computers & Structures 1997;64:983–94.
[7] Moore WI, Donovan ES, Powers CR. Thermal analysis of automotive lamps using the ADINA-F coupled specular radiation and natural convection model. Computers & Structures 1999;72(1–3):19–32.
[8] Wang X. Analytical and computational approaches for some fluid–structure interaction analyses. Computers & Structures 1999;72(1–3):429–39.
[9] Tang D, Yang C, Huang Y. Wall stress and strain analysis using a 3-D thick-wall model with fluid–structure interactions for blood flow in carotid arteries with stenoses. Computers & Structures 1999;72(1–3):347–62.
[10] Tang D, Yang C. A 3-D thin-wall model with fluid–structure interactions for blood flow in carotid arteries with symmetric and asymmetric stenoses. Computers & Structures 1999;72(1–3):363–83.