Journal of Computational Physics 371 (2018) 801–819

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

Single-mode perturbation growth in an idealized spherical implosion

M. Flaig [a,*], D. Clark [b], C. Weber [b], D.L. Youngs [c], B. Thornber [a]

[a] Aerospace, Mechanical and Mechatronic Engineering, University of Sydney, NSW 2006, Australia
[b] Lawrence Livermore National Laboratory, Livermore, CA 94550, USA
[c] University of Strathclyde, Glasgow, G1 1XJ, UK

ARTICLE INFO

Article history:
Received 30 September 2017
Received in revised form 9 April 2018
Accepted 3 June 2018
Available online 5 June 2018

Keywords:
Code comparison
Fluid instabilities
Implosion

ABSTRACT

This paper considers a dense imploding spherical shell, where perturbations on the inner surface grow due to hydrodynamic instabilities, geometric convergence and compression. A low-convergence implosion with a single spherical harmonic mode perturbation with mode numbers in the range from $\ell = 5$ to $\ell = 100$ and at three different amplitudes is considered. The linear theory of Epstein [1] is extended to incorporate reshock, making it applicable to the full implosion process (while perturbations are linear). Linear theory is employed to choose modes such that quantified geometric convergence and compression effects contribute significantly to perturbation growth at the lower mode number, while at the higher mode number the contribution from Rayleigh–Taylor instability dominates. Simulation results from four independent simulation codes (FLASH, HYDRA, Miranda and Flamenco) are presented. The simulation predictions are validated against linear theory pre-reshock and employed to validate the extended theory across the reshock. The simulations continue to substantially non-linear perturbation amplitudes, beyond the limits of the analytical approach, and the presented perturbation amplitudes can inform future non-linear modeling. The simulated perturbation amplitudes agree to within approximately 10% for most cases, with isolated cases having differences of greater than 50% from the simulated ensemble-mean. This occurs in the case of a low mode number perturbation with a very small amplitude, where the growth of secondary small-scale instabilities leads to substantial differences between the codes after re-shock.

1. Introduction

Inertial confinement fusion (ICF) holds the promise of providing a nearly limitless energy source. The basic physical process consists of compressing a mm-sized shell of thermonuclear fuel to very high densities via (direct or indirect) laser-driven ablation of an outer layer [2–7]. The aim is to achieve a “hot spot” with high enough densities and temperatures in the center of the compressed shell, such that the self-heating due to alpha particles generated by the nuclear reactions leads to self-sustained burning of the thermonuclear fuel. Ideally, the energy output will then be much higher than the amount of energy that has been put into the system.

* Corresponding author.
E-mail address: my.flaig@gmail.com (M. Flaig).

https://doi.org/10.1016/j.jcp.2018.06.014
0021-9991/

Despite many decades of intensive research, ICF is still a long way from becoming a practical energy source. Amplification of perturbations on unstable interfaces due to Richtmyer–Meshkov (RM) instability [8–10] and Rayleigh–Taylor (RT) instability [11,12] has been identified as a major performance-decreasing factor in ICF implosions [4]. Even very small initial perturbations may grow to large amplitudes at late times and lead to hot spot deformation and reduced overall compression. Understanding the processes that drive the perturbation growth on the unstable interfaces is therefore crucially important in order to overcome the challenges faced by current ICF designs.

Two fluids of different densities become RT unstable when the lighter fluid pushes against the heavier fluid, which happens, for example, in a situation where a heavy fluid sits atop a light fluid in a gravitational field. The RM instability, on the other hand, occurs due to an impulsive, rather than continuous, acceleration, such as when a shock wave passes through the interface between two fluids of different densities. The compression of an ICF capsule typically involves multiple shocks, each of which may contribute to the seeding of RM instabilities. The perturbations will be further amplified via RT instabilities as the implosion decelerates. Since the capsules are compressed to a small fraction of their initial radius, geometric convergence and compression effects also play a prominent role.

The experimental diagnostics available in ICF experiments provide only very limited information about the physical processes taking place during the implosion, therefore, the study of ICF physics relies to a large degree on numerical modeling. Unfortunately, the results of numerical simulations are usually dependent on the details of the numerical method involved, as well as on other details such as the grid geometry, numerical resolution and the presence of numerical instabilities. However, a report summarizing a 2016 ICF workshop held at Los Alamos National Laboratory stated that it *'is generally believed that current numerical algorithms are simply incapable of accurately simulating the growth of high-mode asymmetries and the ensuing mix states they generate'* [13]. It is therefore important to compare the results of different simulation codes for the same given problem, in order to quantify numerical uncertainty, and to build carefully the understanding from low convergence problems through to higher convergence [14–18].

The present study employs four independent algorithms implemented in independent codes to study an idealized two-layered spherical implosion problem, alongside an extension of small-perturbation theory. The codes under consideration are the Eulerian finite-volume codes FLASH [19] and Flamenco [20,21], HYDRA [22], which uses an arbitrary Lagrangian–Eulerian (ALE) method, and Miranda [23], which is based on a high-order finite difference scheme. All codes have been extensively applied in the context of high-energy density physics/ICF and/or turbulent mixing problems.

A small-amplitude analysis along the lines of the works of Bell and Plesset [24,25] (whose results were partly contained in the earlier, but less widely known work of Penney and Price [26,27]) is employed for comparison with the simulation results and to indentify the processes driving the perturbation growth. Here it is shown that the model of Epstein [1] can be solved analytically for the current problem, allowing quantification of the relative importance of RM and RT instabilities, as well as compressibility and geometric convergence effects, during the different stages of the implosion. The model is extended here to include reshock, and cross-validated against the simulation results.

The layout of the paper is as follows: In Sec. 2, the physical model under consideration is detailed and the numerical codes as well as code-specific elements of the setup are described. The results of the numerical simulations are presented in Sec. 3. Sec. 4 is devoted to the analytical solution to the small-amplitude analysis, extension to reshock and a cross-validation of the analytical model with the numerical results. In Sec. 5 we summarize the key conclusions of the study.

2. Model

The physical model that is adopted in the present paper is similar to the one studied in Refs. [14,15]. The computational domain of interest consists of two layers: 1. An inner low-density ($\rho = 0.05\,\mathrm{g\,cm^{-3}}$) "gas" region that initially extends from $r = 0$ to $r = r_{\mathrm{gas}} + \delta R$, where $r_{\mathrm{gas}} = 1\,\mathrm{mm}$ and δR is the interface perturbation, to be described below. 2. An outer high-density ($\rho = 1\,\mathrm{g\,cm^{-3}}$) "shell" region that initially extends from $r = r_{\mathrm{gas}} + \delta R$ to $r_{\mathrm{shell}} = 1.2\,\mathrm{mm}$. Both regions are initialized with zero velocity and initial pressure $p = 10\,\mathrm{Mbar}$. The equation of state is assumed to be that of an ideal gas with adiabatic index $\gamma = 5/3$.

The shell region is surrounded by a high-pressure "source" region with prescribed, time-varying physical conditions that drive the implosion. The inner radius $r_{\mathrm{src}}(t)$ of the source region moves inwards with time according to the following prescription:

$$r_{\mathrm{src}}(t) = r_{\mathrm{shell}} - v_0 t, \tag{1}$$

where $v_0 = 0.24$ mm/ns. The radial velocity in the source region is assumed to vary as a function of time and space according to

$$v_{\mathrm{src}}(r,t) = -v_0 r / r_{\mathrm{src}}(t). \tag{2}$$

The density in the source region is held constant at $\rho_{\mathrm{src}} = 0.1\,\mathrm{g\,cm^{-3}}$ throughout the simulation. The pressure is initially set to $p = 1$ Gbar. This value is kept until $t = 0.5\,\mathrm{ns}$, from then on the pressure decreases linearly such that it would reach a value of $p = 10$ Mbar at $t = 3\,\mathrm{ns}$, however, simulations are run only until $t = 2.5\,\mathrm{ns}$, as this is the time where the interface reaches stagnation.