HORSES3D
User Manual

Andrés Rueda
Many others (future?)

October 17, 2018

Contents

Chapter 1

Running a Simulation

List of all the mandatory keywords for running a simulation and some basic optional keywords. The specific keywords are listed in the other chapters.

Table 1.1: General keywords for running a case.

Keyword	Description	Default value
solution file name	*CHARACTER*: Path and name of the output file. The name of this file is used for naming other output files.	**Mandatory keyword**
simulation type	*CHARACTER*: Specifies if NSLITE3D must perform a 'steady-state' or a 'time-accurate' simulation.	'steady-state'
time integration	*CHARACTER*: Can be 'implicit', 'explicit', or 'FAS'. The latter uses the Full Algebraic Storage (FAS) multigrid scheme, which can have implicit or explicit smoothers.	'explicit'
polynomial order	*INTEGER*: Polynomial order to be assigned uniformly to all the elements of the mesh. If the keyword *polynomial order file* is specified, the value of this keyword is overridden.	–*
polynomial order i polynomial order j polynomial order k	*INTEGER*: Polynomial order in the i, j, or k component for all the elements in the domain. If used, the three directions must be declared explicitly, unless you are using a polynomial order file. If the keyword *polynomial order file* is specified, the value of this keyword is overridden.	–*
polynomial order file	*CHARACTER*: Path to a file containing the polynomial order of each element in the domain.	–*
restart	*LOGICAL*: If .TRUE., initial conditions of simulation will be read from restart file specified using the keyword *restart file name*. **THIS SHOULD NOT BE MANDATORY... DEFAULT FALSE WOULD BE EASIER**.	**Mandatory keyword**
cfl	*REAL*: A constant related with the **convective** Courant-Friedrichs-Lewy (CFL) condition that the program will use to compute the time step size.	–**
dcfl	*REAL*: A constant related with the **diffusive** Courant-Friedrichs-Lewy (DCFL) condition that the program will use to compute the time step size.	–**
dt	*REAL*: Constant time step size.	–**
final time	*REAL*: This keyword is mandatory for time-accurate solvers	–
mesh file name	*CHARACTER*: Name of the mesh file. The currently supported formats are *.mesh* (SpecMesh file format) and *.h5* (HOPR hdf5 file format).	**Mandatory keyword**
mesh inner curves	*LOGICAL*: Specifies if the mesh reader must suppose that the inner surfaces (faces connecting the elements of the mesh) are curved. This input variable only affects the hdf5 mesh reader.	.TRUE.
number of time steps	*INTEGER*: *Maximum* number of time steps that the program will compute.	**Mandatory keyword**

Table 1.1: General keywords for running a case - continued.

Keyword	Description	Default value
output interval	*INTEGER*: In steady-state, this keyword indicates the interval of time steps to display the residuals on screen. In time-accurate simulations, this keyword indicates how often a 3D output file must be stored.	**Mandatory keyword**
convergence tolerance	*real*: **WHY MANDATORY?.. ONLY USED FOR STEADY-STATE**	**Mandatory keyword**
manufactured solution	*CHARACTER*: Must have the value '2D' or '3D'. When this keyword is used, the program will add source terms for the conservative variables taken into account an exact analytic solution for each primitive variable j (ρ, u, v, w, p) of the form: $j = j_C(1) + j_C(2)\sin(\pi j_C(5)x) + j_C(3)\sin(\pi j_C(6)y) + j_C(4)\sin(\pi j_C(7)z)$ Where $j_C(i)$ are constants defined in the file *ManufacturedSolutions.f90*. Proper initial and boundary conditions must be imposed (see the test case). The mesh must be a unit cube.	–
Number of boundaries	*INTEGER*: Specifies the number of boundaries of the geometry. This keyword must be followed by an equal number of lines in the control file that define the boundary conditions as follows: *boundaryName1 boundaryValue1 boundaryType1* *boundaryName2 boundaryValue2 boundaryType2* ...	**Mandatory keyword**

* At least one of these two keywords must be specified

** For Euler simulations, the user must specify either the CFL number or the time-step size. For Navier-Stokes simulations, the user must specify the CFL and DCFL numbers **or** the time-step size.

Chapter 2

Restarting a Case

Table 2.1: Keywords for restarting a case.

Keyword	Description	Default value
restart	*LOGICAL*: If .TRUE., initial conditions of simulation will be read from restart file specified using the keyword *restart file name*.	**Mandatory keyword**
restart file name	*CHARACTER*: Name of the restart file to be written and, if keyword *restart* = .TRUE., also name of the restart file to be read for starting the simulation.	**Mandatory keyword**
restart interval	*INTEGER*: Indicates how often restart files have to be written.	Huge number
restart polorder	*INTEGER*: Uniform polynomial order of the solution to restart from. This keyword is only needed when the restart solution is of a different order than the current case.	same as case's
restart polorder file	*CHARACTER*: File containing the polynomial orders of the solution to restart from. This keyword is only needed when the restart solution is of a different order than the current case.	same as case's
get discretization error of	*CHARACTER*: Path to solution file. This can be used to estimate the discretization error of a solution when restarting from a higher-order solution.	–

Chapter 3

Implicit Solvers

3.1 General Keywords

The keywords for the implicit solvers are listed in table 3.1

Table 3.1: Keywords for implicit solvers.

Keyword	Description	Default value
time integration	*CHARACTER*: This is the main keyword for activating the implicit solvers. The value of it should be set to 'implicit'.	'explicit'
jacobian flag	*INTEGER*: Specifies the type of Jacobian matrix to be computed. Options are: 1. Numerical Jacobian: Uses coloring algorithm for computing Jacobian. 2. Analytical Jacobian: Being implemented. To date only available for the compressible Euler equations and 'matrix-free smooth' linear solver to compute the blocks of the Jacobian.	1
jacobian by convergence	*LOGICAL*: When .TRUE., the Jacobian is only computed when the convergence falls beneath some threshold (see keyfords: blah and blah blah). This improves performance but can introduce big numerical errors for time-accurate simulations.	.FALSE.
linear solver	*CHARACTER*: Specifies the linear solver that has to be used. Options are: • 'petsc': PETSc library Krylov-Subspace methods. • 'pardiso': Intel MKL PARDISO. • 'matrix-free gmres': A matrix-free version of the GMRES algorithm. Can be used without preconditioner or with a recursive GMRES preconditioner using 'preconditioner=GMRES'. • 'smooth': Traditional iterative methods. One can select either 'smoother=WeightedJacobi' or 'smoother=BlockJacobi'. • 'matrix-free smooth': A matrix-free version of the previous solver. Only available eith 'smoother=BlockJacobi' • 'multigrid': .	'petsc'
print newton info	*LOGICAL*: If .TRUE., the information of the Newton iterations will be displayed.	'.FALSE.'

Table 3.1: Keywords for implicit solvers - continued.

Keyword	Description	Default value
implicit adaptive dt	*LOGICAL*: Specifies if the time-step should be computed according to the convergence behavior of the Newton iterative method and the linear solver.	.FALSE.
newton tolerance	*REAL*: Specifies the tolerance for the Newton method.	10^{-6} or for time-accurate simulations and $MaxResidual \times 10^{-3}$ for steady-state simulations

3.2 Multigrid

Table 3.2: Keywords for the multigrid solver.

Keyword	Description	Default value
multigrid levels	*INTEGER*: Number of multigrid levels for the computations.	**Mandatory keyword**
delta n	*INTEGER*: Interval of reduction of polynomial order for creating coarser multigrid levels.	1
multigrid output	*LOGICAL*: If .TRUE., the residuals at the different multigrid levels will be displayed.	.FALSE.

Chapter 4

p-Adaptation Methods

The p-adaptation methods are used when the p-adaptation region is specified in the control file:

```
#define p-adaptation
    Truncation error type = isolated
    truncation error      = 1.d-2
    Nmax                  = [10,10,10]
    Nmin                  = [2 ,2 ,2 ]
    Conforming boundaries = [InnerCylinder,sphere]
    order across faces    = N*2/3
    increasing            = .FALSE.
    regression files      = .FALSE.
    adjust nz             = .FALSE.
    mode                  = time
    interval              = 1.d0
    restart files         = .TRUE.
    padapted mg sweeps pre      = 10
    padapted mg sweeps post     = 12
    padapted mg sweeps coarsest = 20
#end
```

Table 4.1: Keywords for the p-adaptation algorithms.

Keyword	Description	Default value
truncation error type	*CHARACTER*: Can be either "isolated" or "non-isolated".	isolated
truncation error	*REAL*: Target truncation error for the p-adaptation algorithm.	**Mandatory keyword**
Nmax	*INTEGER*(3): Maximum polynomial order in each direction for the p-adaptation algorithm.	**Mandatory keyword**
Nmin	*INTEGER*(3): Minimum polynomial order in each direction for the p-adaptation algorithm.	[1,1,1]
conforming boundaries	*CHARACTER*(*): Specifies the boundaries of the geometry that must be forced to be conforming after the p-adaptation process.	–
order across faces	*CHARACTER*: Mathematical expression to specify the maximum polynomial order jump across faces. Currently, only $N*2/3$ and $N-1$ are supported.	$N-1$
increasing	*LOGICAL*: If .TRUE. the multi-stage FMG adaptation algorithm is used.	.FALSE.
regression files	*LOGICAL*: If .TRUE., the program writes files containing the information about the lin-log regression procedure that is performed for the τ-estimation extrapolation. The files are stored in the folder *./RegressionFiles/Stage_XX/*, where XX is the adaptation stage number.	.FALSE.

Table 4.1: Keywords for the p-adaptation algorithms - continued.

Keyword	Description	Default value
adjust nz	*LOGICAL*: If .TRUE., the order accross faces is adjusted i n the directions xi, eta, and zeta of the face (being zeta the normal direction). If .FALSE., the order is only adjusted in the xi and eta directions. The adjustment currently consists (hard-cod ed) in allowing jumps in the polynomial order of at most 1.	.FALSE.
mode	*CHARACTER*: p-Adaptation mode. Can be *steady*, *time* or *iteration*. Steady p-adaptation is performed once at the beginning of a simulation. Unsteady adaptation can be by *time* or by *iteration*.	*steady*
interval	*INTEGER/REAL*: In unsteady p-adaptation cases, this keyword specifies the iteration (integer) or time (real) interval for p-adaptation.	*huge number*
restart files	*LOGICAL*: If .TRUE., the program writes restart files before and after the p-adaptation.	.FALSE.
padapted $\ll$ *keyword* $\gg$	*MULTIPLE*: Specifies control file keywords that should be replaced after the adaptation procedure. Currently, only 'mg sweeps ', 'mg sweeps pre', 'mg sweeps post', and 'mg sweeps coarsest' are supported.	–

Chapter 5

Input files

- Control file
- Mesh file
- Polynomial order file