

# SPARC

Simulation Package for Ab-initio Real-space Calculations

User guide

Material Physics & Mechanics Group

PI: Phanish Suryanarayana

Main Developers: Qimen Xu, Abhiraj Sharma

Collaborators: J.E. Pask (LLNL), A.J. Medford (GT), E. Chow (GT)

Georgia Institute of Technology

# Introduction

SPARC is an open-source software package for the accurate, efficient, and scalable solution of the Kohn-Sham density functional theory (DFT) problem. The main features of SPARC currently include

- Isolated systems such as molecules and clusters as well as extended systems such as crystals, surfaces, and wires.
- Calculation of ground state energy, atomic forces, and stress tensor.
- Unconstrained collinear magnetization via spin polarized calculations.
- Structural relaxation and quantum molecular dynamics (QMD).
- Local and semilocal exchange correlation functionals.
- ONCV and TM pseudopotentials in psp8 (ABINIT) format.

Additional details regarding the formulation and implementation of SPARC can be found in the accompanying paper. Please direct any questions and report any bugs to Prof. Phanish Suryanarayana.

# Acknowledgments

- U.S. Department of Energy, Office of Science: DE-SC0019410
- U.S. National Science Foundation: 1553212, 1663244, and 1333500

# Citation

If you publish work using/regarding SPARC, please cite some of the following articles, particularly those that are most relevant to your work:

- General: <https://doi.org/10.1016/j.softx.2021.100709>,  
<https://doi.org/10.1016/j.cpc.2016.09.020>,  
<https://doi.org/10.1016/j.cpc.2017.02.019>
- Non-orthogonal systems:  
<https://doi.org/10.1016/j.cplett.2018.04.018>
- Linear solvers: <https://doi.org/10.1016/j.cpc.2018.07.007>,  
<https://doi.org/10.1016/j.jcp.2015.11.018>
- Stress tensor/pressure: <https://doi.org/10.1063/1.5057355>
- Atomic forces: <https://doi.org/10.1016/j.cpc.2016.09.020>,  
<https://doi.org/10.1016/j.cpc.2017.02.019>
- Mixing: <https://doi.org/10.1016/j.cplett.2016.01.033>,  
<https://doi.org/10.1016/j.cplett.2015.06.029>,  
<https://doi.org/10.1016/j.cplett.2019.136983>

# Installation - Option 1

Prerequisite: C compiler, MPI.

There are several options to compile SPARC, depending on the available external libraries.

- Option 1 (default): Compile with BLAS and LAPACK.
  - Step 1: Install/Load OpenBLAS/BLAS and LAPACK.
  - Step 2: Go to `src/` directory, there is an available `makefile`.
  - Step 3 (optional): Edit `makefile`. If the BLAS library path and LAPACK library path are not in the search path, edit the `BLASROOT` and `LAPACKROOT` variables, and add them to `LDFLAGS`. If you are using BLAS instead of OpenBLAS, replace all `-lopenblas` flags with `-lblas`.
  - Step 4 (optional): To turn on `DEBUG` mode, set `DEBUG_MODE` to 1 in the `makefile`.
  - Step 5: Within the `src/` directory, compile the code by  
`$ make clean; make`

# Installation - Option 2

- Option 2 (recommended): Compile with MKL.
  - Step 1: Install/Load MKL.
  - Step 2: Go to `src/` directory, there is an available `makefile`.
  - Step 3: Edit `makefile`. Set `USE_MKL` to 1 to enable compilation with MKL. If the MKL library path is not in the search path, edit the `MKLROOT` variable to manually set the MKL path.
  - Step 4 (optional): For the projection/subspace rotation step, to use SPARC routines for matrix data distribution rather than ScaLAPACK (through MKL), set `USE_DP_SUBEIG` to 1. We found on some machines this option is faster.
  - Step 5 (optional): To turn on `DEBUG` mode, set `DEBUG_MODE` to 1 in the `makefile`.
  - Step 6: Within the `src/` directory, compile the code by  
`$ make clean; make`

# Installation - Option 3

- Option 3: Compile with BLAS, LAPACK, and ScaLAPACK.
  - Step 1: Install/Load OpenBLAS/BLAS, LAPACK, and ScaLAPACK.
  - Step 2: Go to `src/` directory, there is an available makefile.
  - Step 3: Edit makefile. Set `USE_SCALAPACK` to 1 to enable compilation with ScaLAPACK. If the BLAS library path, LAPACK library path, and/or ScaLAPACK library path are not in the search path, edit the `BLASROOT`, `LAPACKROOT`, and/or `SCALAPACKROOT` variables accordingly, and add them to `LDFLAGS`. If you are using BLAS instead of OpenBLAS, replace all `-lopenblas` flags with `-lblas`.
  - Step 4 (optional): For the projection/subspace rotation step, to use SPARC routines for matrix data distribution rather than ScaLAPACK, set `USE_DP_SUBEIG` to 1. We found on some machines this option is faster.
  - Step 5 (optional): To turn on DEBUG mode, set `DEBUG_MODE` to 1 in the makefile.
  - Step 6: Within the `src/` directory, compile the code by  
`$ make clean; make`

## Installation - lib

Once compilation is done, a binary named `sparc` will be created in the `lib/` directory.



# Input files

The required input files to run a simulation with SPARC are

- “.inpt” file – User options and parameters.
- “.ion” file – Atomic information.

It is required that the “.inpt” and “.ion” files are located in the same directory and share the same name. A detailed description of the input options is provided in this document. Examples of input files can be found in the directory SPARC/tests.

In addition, SPARC requires pseudopotential files of psp8 format which can be generated by D. R. Hamann’s open-source pseudopotential code **ONCVSP**. A large number of accurate and efficient pseudopotentials are already provided within the package. For access to more pseudopotentials, the user is referred to the **SG15 ONCV potentials**. Using the **ONCVSP** input files included in the **SG15 ONCV potentials**, one can easily convert the **SG15 ONCV potentials** from upf format to psp8 format. Paths to the pseudopotential files are specified in the “.ion” file.

# Execution

SPARC can be executed in parallel using the `mpirun` command. Sample PBS script files are available in “SPARC/tests” folder. It is required that the “.inpt” and “.ion” files are located in the same directory and share the same name. For example, to run a simulation with 8 processes with input files as “filename.inpt” and “filename.ion” in the root directory (SPARC/), use the following command:

```
$ mpirun -np 8 ./lib/sparc -name filename
```

As an example, one can run one of the tests located in ‘SPARC/tests/'. First go to ‘SPARC/tests/Example\_tests/’ directory:

```
$ $ cd tests/Example_tests/
```

There are a few input files available. Run a DC silicon system by

```
$ mpirun -np 24 ../../../../lib/sparc -name Si8_kpt
```

# Execution

The result is printed to output file “Si8\_kpt.out”, located in the same directory as the input files. If the file “Si8\_kpt.out” is already present, the result will be printed to “Si8\_kpt.out\_1” instead. The max number of “.out” files allowed with the same name is 100. Once this number is reached, the result will instead overwrite the “Si8\_kpt.out” file. One can compare the result with the reference out file named “Si8\_kpt.refout”.

In the tests/ directory, we also provide a suite of tests which are arranged in a hierarchy of folders. Each test system has its own directory. A python script is also provided which launches the suite of test systems. To run a set of four quick tests locally on the CPU, simply run:

```
$ python test.py quick_run
```

# Execution

The result is stored in the corresponding directory of the tests. A message is also printed in the terminal showing if the tests passed or failed. The tests can also be launched in parallel on a cluster by using the Python script. Detailed information on using the python script can be found in the 'ReadMe' file in the 'tests/' directory.

# Output

Upon successful execution of the `sparc` code, depending on the calculations performed, some output files will be created in the same location as the input files.

## Single point calculations

- “.out” file – General information about the test, including input parameters, SCF convergence progress, ground state properties and timing information.
- “.static” file – Atomic positions and atomic forces if the user chooses to print these information.

## Structural relaxation calculations

- “.out” file – See above.

# Output

- “.geopt” file – Atomic positions and atomic forces for atomic relaxation, cell lengths and stress tensor for volume relaxation, and atomic positions, atomic forces, cell dimensions, and stress tensor for full relaxation.
- “.restart” file – Information necessary to perform a restarted structural relaxation calculation. Only created if atomic relaxation is performed.

## **Quantum molecular dynamics (QMD) calculations**

- “.out” file – See above.
- “.aimd” file – Atomic positions, atomic velocities, atomic forces, electronic temperature, ionic temperature and total energy for each QMD step.
- “.restart” file – Information necessary to perform a restarted QMD calculation.

# Input file options

## System

CELL | LATVEC\_SCALE | LATVEC | FD\_GRID | MESH\_SPACING | ECUT | BC |  
FD\_ORDER | EXCHANGE\_CORRELATION | SPIN\_TYP | KPOINT\_GRID |  
KPOINT\_SHIFT | ELEC\_TEMP\_TYPE | ELEC\_TEMP | SMEARING | NSTATES |  
D3\_FLAG | D3\_RTHR | D3\_CN\_THR  
ATOM\_TYPE | PSEUDO\_POT | N\_TYPE\_ATOM | COORD | COORD\_FRAC | RELAX |  
SPIN

# Input file options

## SCF

CHEB\_DEGREE | CHEFSI\_BOUND\_FLAG | RHO\_TRIGGER | MAXIT\_SCF |  
MINIT\_SCF | TOL\_SCF | SCF\_FORCE\_ACC | SCF\_ENERGY\_ACC | TOL\_LANCZOS  
| MIXING\_VARIABLE | MIXING\_HISTORY | MIXING\_PARAMETER |  
MIXING\_PARAMETER\_SIMPLE | MIXING\_PARAMETER\_MAG |  
MIXING\_PARAMETER\_SIMPLE\_MAG | PULAY\_FREQUENCY | PULAY\_RESTART |  
MIXING\_PRECOND | MIXING\_PRECOND\_MAG | TOL\_PRECOND |  
PRECOND\_KERKER\_KTF | PRECOND\_KERKER\_THRESH |  
PRECOND\_KERKER\_KTF\_MAG | PRECOND\_KERKER\_THRESH\_MAG | FIX\_RAND

## Electrostatics

TOL\_POISSON | MAXIT\_POISSON | TOL\_PSEUDOCARGE |  
REFERENCE\_CUTOFF



# Input file options

## Stress calculation

CALC\_STRESS | CALC\_PRES

## QMD

MD\_FLAG | MD\_METHOD | MD\_NSTEP | MD\_TIMESTEP | ION\_TEMP |  
ION\_TEMP\_END | ION\_VEL\_DSTR | ION\_VEL\_DSTR\_RAND | QMASS  
| NPT\_NH\_QMASS | NPT\_NH\_BMASS | NPT\_NP\_QMASS | NPT\_NP\_BMASS |  
NPT\_SCALE\_VECS | TARGET\_PRESSURE | RESTART\_FLAG | TWTIME

## Structural relaxation

RELAX\_FLAG | RELAX\_METHOD | RELAX\_NITER | TOL\_RELAX |  
TOL\_RELAX\_CELL | RELAX\_MAXDILAT | NLCG\_SIGMA | L\_HISTORY |  
L\_FINIT\_STP | L\_MAXMOV | L\_AUTOSCALE | L\_LINEOPT | L\_ICURV |  
FIRE\_DT | FIRE\_MASS | FIRE\_MAXMOV | RESTART\_FLAG

# Input file options

## Print options

PRINT\_ATOMS | PRINT\_FORCES | PRINT\_MDOUT | PRINT\_RELAXOUT |  
PRINT\_RESTART | PRINT\_RESTART\_FQ | PRINT\_VELS | OUTPUT\_FILE

## Parallelization options

NP\_SPIN\_PARAL | NP\_KPOINT\_PARAL | NP\_BAND\_PARAL | NP\_DOMAIN\_PARAL  
| NP\_DOMAIN\_PHI\_PARAL | EIG\_SERIAL\_MAXNS | EIG\_PARAL\_BLKSZ

**System: .inpt file**

# CELL

Type

Double

Unit

Bohr

Default

None

Example

CELL: 10.20 11.21 7.58

## Description

A set of three whitespace delimited values specifying the cell lengths in the lattice vector ([LATVEC](#)) directions, respectively.

## Remark

Note that [CELL](#) ignores the lengths of the lattice vectors specified in the `.inpt` file and only treats them as unit vectors. [LATVEC\\_SCALE](#) and [CELL](#) cannot be specified simultaneously.

# LATVEC\_SCALE

Type

Double

Unit

Bohr

Default

None

Example

LATVEC\_SCALE: 10.20 11.21 7.58

## Description

A set of three whitespace delimited values specifying the scaling factors in the lattice vectors ([LATVEC](#)), respectively.

## Remark

The difference between [LATVEC\\_SCALE](#) and [CELL](#) is that [CELL](#) treats the lattice vectors as unit vectors, whereas [LATVEC\\_SCALE](#) scales the lattice vectors directly as specified by the user. [LATVEC\\_SCALE](#) and [CELL](#) cannot be specified simultaneously.

# LATVEC

## Type

Double array

## Unit

No unit

## Default

```
1.0 0.0 0.0  
0.0 1.0 0.0  
0.0 0.0 1.0
```

## Example

```
LATVEC:  
0.5 0.5 0.0  
0.0 0.5 0.5  
0.5 0.0 0.5
```

## Description

A set of three vectors in row major order specifying the lattice vectors of the simulation domain ([CELL](#)).

## FD\_GRID

### Type

Integer

### Unit

No unit

### Default

None

### Example

FD\_GRID: 26 26 30

### Description

A set of three whitespace delimited values specifying the number of finite-difference intervals in the lattice vector ([LATVEC](#)) directions, respectively.

### Remark

The convergence of results with respect to spatial discretization needs to be verified. `ECUT`, `MESH_SPACING`, `FD_GRID` cannot be specified simultaneously.



# MESH\_SPACING

Type

Double

Unit

Bohr

Default

None

Example

MESH\_SPACING: 0.4

Description

Mesh spacing of the finite-difference grid.

Remark

The exact mesh-size will be determined by the size of the domain.  
[MESH\\_SPACING](#), [FD\\_GRID](#), [ECUT](#) cannot be specified simultaneously.

# ECUT

Type

Double

Unit

Ha

Default

None

Example

ECUT: 30

## Description

Equivalent plane-wave energy cutoff, based on which [MESH\\_SPACING](#) will be automatically calculated.

## Remark

This is not exact, but rather an estimate. [ECUT](#), [MESH\\_SPACING](#), [FD\\_GRID](#) cannot be specified simultaneously.

**Type**

Character

**Unit**

No unit

**Default**

None

**Example**

BC: P D D

**Description**

A set of three whitespace delimited characters specifying the boundary conditions in the lattice vector directions, respectively. P represents periodic boundary conditions and D represents Dirichlet boundary conditions.

## FD\_ORDER

Type

Integer

Unit

No unit

Default

12

Example

FD\_ORDER: 12

Description

Order of the finite-difference approximation.

Remark

Restricted to even integers since central finite-differences are employed. The default value of 12 has been found to be an efficient choice for most systems.

# EXCHANGE\_CORRELATION

Type

String

Unit

No unit

Default

No Default

Example

EXCHANGE\_CORRELATION: LDA\_PW

Description

Choice of exchange-correlation functional. Options are LDA\_PW (Perdew-Wang LDA), LDA\_PZ (Purdew-Zunger LDA), GGA\_PBE (PBE GGA), GGA\_RPBE (revised PBE GGA), and GGA\_PBEso1 (PBE GGA revised for solids).

Remark

For spin-polarized calculation ([SPIN\\_TYP](#) = 1), LDA\_PZ is not available.

# SPIN\_TYP

Type

Integer

Unit

No unit

Default

0

Example

SPIN\_TYP: 1

Description

SPIN\_TYP: 0 performs spin unpolarized calculation.

SPIN\_TYP: 1 performs unconstrained collinear spin-polarized calculation.

Remark

SPIN\_TYP can only take values 0 and 1.

# KPOINT\_GRID

## Type

Integer array

## Unit

No unit

## Default

1 1 1

## Example

KPOINT\_GRID: 2 3 4

## Description

Number of k-points in each direction of the Monkhorst-Pack grid for Brillouin zone integration.

## Remark

Time-reversal symmetry is assumed to hold.

# KPOINT\_SHIFT

## Type

Double array

## Unit

No unit

## Default

0.0 for odd k-point mesh  
0.5 for even k-point mesh

## Example

KPOINT\_SHIFT: 0.5 0.5 0.5

## Description

Shift of k-points with respect to k-point grid containing  $\Gamma$ -point.

## Remark

The shift is in reduced coordinates.



## ELEC\_TEMP\_TYPE

Type

String

Unit

No unit

Default

gaussian

Example

ELEC\_TEMP\_TYPE: fd

Description

Function used for the smearing (electronic temperature). Options are: `fermi-dirac` (or `fd`), `gaussian`.

Remark

Use `ELEC_TEMP` or `SMEARING` to set smearing value.

# ELEC\_TEMP

Type

Double

Unit

Kelvin

Default

2320.904 for gaussian

1160.452 for fermi-dirac

Example

ELEC\_TEMP: 315.775

Description

Electronic temperature.

Remark

This is equivalent to setting **SMEARING** ( $0.001 \text{ Ha} = 315.775 \text{ Kelvin}$ ).

# SMEARING

Type

Double

Unit

Ha

Default

0.007350 for gaussian

0.003675 for fermi-dirac

Example

SMEARING: 0.001

Description

Value of smearing.

Remark

Equivalent to setting [ELEC\\_TEMP](#) ( $0.001 \text{ Ha} = 315.775 \text{ Kelvin}$ ).

# NSTATES

Type

Integer

Unit

No unit

Default

$N_e/2 \times 1.2 + 5$

Example

NSTATES: 24

Description

The number of Kohn-Sham states/orbitals.

Remark

This number should not be smaller than half of the total number of valence electrons ( $N_e$ ) in the system. Note that the number of additional states required increases with increasing values of [ELEC\\_TEMP](#)/[SMEARING](#).

## D3\_FLAG

Type

0 or 1

Unit

No unit

Default

0

Example

D3\_FLAG: 1

Description

Flag for adding Grimme's DFT-D3 correction on the result

Remark

Only active when using GGA-PBE, GGA-RPBE and GGA-PBEsol.

## D3\_RTHR

Type

Double

Unit

Bohr<sup>2</sup>

Default

1600

Example

D3\_RTHR: 9000

Description

Square of cut-off radius for calculating DFT-D3 correction between two atoms

Remark

Only applicable when DFT-D3 correction [D3\\_FLAG](#) is used.

## D3\_CN\_THR

Type

Double

Unit

Bohr<sup>2</sup>

Default

625

Example

D3\_CN\_THR: 1600

Description

Square of cut-off radius for calculating CN value of every atom and DFT-D3 correction between three atoms

Remark

Only applicable when DFT-D3 correction [D3\\_FLAG](#) is used.

**System: .ion file**



# ATOM\_TYPE

Type

String

Unit

No unit

Default

None

Example

ATOM\_TYPE: Fe

Description

Atomic type symbol.

Remark

The atomic type symbol can be attached with a number, e.g., Fe1 and Fe2. This feature is useful if one needs to provide two different pseudopotential files ([PSEUDO\\_POT](#)) for the same element.

# PSEUDO\_POT

Type

String

Unit

No unit

Default

None

Example

PSEUDO\_POT: ../psp/Fe.psp8

Description

Path to the pseudopotential file.

Remark

The default directory for the pseudopotential files is the same as the input files. For example, if a pseudopotential Fe.psp8 is put in the same directory as the input files, one can simply specify PSEUDO\_POT: Fe.psp8.

## N\_TYPE\_ATOM

Type

Integer

Unit

No unit

Default

None

Example

N\_TYPE\_ATOM: 2

Description

The number of atoms of a [ATOM\\_TYPE](#) specified immediately before this variable.

Remark

For a system with different types of atoms, one has to specify the number of atoms for every type.

# COORD

Type

Double

Unit

Bohr

Default

None

Example

COORD:

0.0 0.0 0.0

2.5 2.5 2.5

## Description

The Cartesian coordinates of atoms of a [ATOM\\_TYPE](#) specified before this variable. If the coordinates are outside the fundamental domain (see [CELL](#) and [LATVEC](#)) in the periodic directions (see [BC](#)), it will be automatically mapped back to the domain.

## Remark

For a system with different types of atoms, one has to specify the coordinates for every `ATOM_TYPE`. One can also specify the coordinates of the atoms using `COORD_FRAC`.

## COORD\_FRAC

Type

Double

Unit

None

Default

None

Example

COORD\_FRAC:

0.5 0.5 0.0

0.0 0.5 0.5

### Description

The fractional coordinates of atoms of a [ATOM\\_TYPE](#) specified before this variable.  $\text{COORD\_FRAC}(i,j) \times \text{CELL}(j)$ , ( $j = 1, 2, 3$ ) gives the coordinate of the  $i^{\text{th}}$  atom along the  $j^{\text{th}}$  [LATVEC](#) direction. If the coordinates are outside the fundamental domain (see [CELL](#) and [LATVEC](#)) in the periodic directions (see [BC](#)), it will be automatically mapped back to the domain.

### Remark

For a system with different types of atoms, one has to specify the coordinates for every `ATOM_TYPE`. One can also specify the coordinates of the atoms using `COORD`.

# RELAX

Type

Integer

Unit

No unit

Default

1 1 1

Example

RELAX:

1 0 1

0 1 0

Description

Atomic coordinate with the corresponding RELAX value 0 is held fixed during relaxation/QMD.



# SPIN

Type

Double

Unit

No unit

Default

0.0

Example

SPIN:

1.0

-1.0

Description

Specifies the net initial spin on each atom for a spin-polarized calculation.

# SCF

## CHEB\_DEGREE

### Type

Integer

### Unit

No unit

### Default

Automatically set.

### Example

CHEB\_DEGREE: 25

### Description

Degree of polynomial used for Chebyshev filtering.

### Remark

For larger mesh-sizes, smaller values of CHEB\_DEGREE are generally more efficient, and vice-versa.

## CHEFSI\_BOUND\_FLAG

Type

Integer

Unit

No unit

Default

0

Example

CHEFSI\_BOUND\_FLAG: 1

### Description

Flag to recalculate the bounds for Chebyshev filtering. If set to 0, then only for the very first SCF will the upper bound be evaluated based on the maximum eigenvalue using Lanczos algorithm, and the upper bound will be the same for the rest steps. If set to 1, the upper bound will be reevaluated for every SCF.

# RHO\_TRIGGER

Type

Integer

Unit

No unit

Default

4

Example

RHO\_TRIGGER: 5

## Description

The number of times Chebyshev filtering is repeated before updating the electron density in the very first SCF iteration.

## Remark

Values smaller than the default value of 4 can result in a significant increase in the number of SCF iterations. Larger values can sometimes reduce the number of SCF iterations.

# MAXIT\_SCF

Type

Integer

Unit

No unit

Default

100

Example

MAXIT\_SCF: 50

Description

Maximum number of SCF iterations.

Remark

Larger values than the default of 100 may be required for highly inhomogeneous systems, particularly when small values of [SMEARING](#)/[ELEC\\_TEMP](#) are chosen.

# MINIT\_SCF

Type

Integer

Unit

No unit

Default

3

Example

MAXIT\_SCF: 5

Description

Minimum number of SCF iterations.

# TOL\_SCF

Type

Double

Unit

No unit

Default

Automatically set for  $10^{-3}$   
Ha/Bohr accuracy in force

Example

TOL\_SCF: 1e-5

Description

The tolerance on the normalized residual of the effective potential or the electron density for convergence of the SCF iteration.

Remark

Only one of `TOL_SCF`, `SCF_ENERGY_ACC`, or `SCF_FORCE_ACC` can be specified.



# SCF\_FORCE\_ACC

Type

Double

Unit

Ha/Bohr

Default

None

Example

SCF\_FORCE\_ACC: 1e-4

Description

The tolerance on the atomic forces for convergence of the SCF iteration.

Remark

Only one of `SCF_FORCE_ACC`, `TOL_SCF` or `SCF_ENERGY_ACC` can be specified.

## SCF\_ENERGY\_ACC

Type

Double

Unit

Ha/atom

Default

None

Example

SCF\_ENERGY\_ACC: 1e-5

Description

The tolerance on the free energy for the convergence of the SCF iteration.

Remark

Only one of `SCF_ENERGY_ACC`, `SCF_FORCE_ACC`, or `TOL_SCF` can be specified.

# TOL\_LANCZOS

Type

Double

Unit

No unit

Default

1e-2

Example

TOL\_LANCZOS: 1e-3

Description

The tolerance within the Lanczos algorithm for calculating the extremal eigenvalues of the Hamiltonian, required as part of the CheFSI method.

Remark

Typically, the Lanczos tolerance does not need to be very strict.

# MIXING\_VARIABLE

Type

String

Unit

No unit

Default

density

Example

```
MIXING_VARIABLE:  
potential
```

Description

This specifies whether potential or density mixing is performed in the SCF iteration. Available options are: `potential` and `density`.

# MIXING\_HISTORY

Type

Integer

Unit

No unit

Default

7

Example

MIXING\_HISTORY: 40

Description

The mixing history used in Pulay mixing.

Remark

Too small values of **MIXING\_HISTORY** can result in poor SCF convergence.

# MIXING\_PARAMETER

Type

Double

Unit

No unit

Default

0.3

Example

MIXING\_PARAMETER: 0.1

Description

The value of the relaxation parameter used in Pulay/simple mixing.

Remark

Values larger than the default value of 0.3 can be used for insulating systems, whereas smaller values are generally required for metallic systems, particularly at small values of [SMEARING](#) or [ELEC\\_TEMP](#).

# MIXING\_PARAMETER\_SIMPLE

## Type

Double

## Unit

No unit

## Default

Automatically set to the same as [MIXING\\_PARAMETER](#)

## Example

MIXING\_PARAMETER\_SIMPLE:  
0.1

## Description

The value of the relaxation parameter used in the simple mixing step in the periodic Pulay scheme.

# MIXING\_PARAMETER\_MAG

## Type

Double

## Unit

No unit

## Default

Automatically set to the same as [MIXING\\_PARAMETER](#).

## Example

MIXING\_PARAMETER\_MAG: 4.0

## Description

The mixing parameter for the magnetization density in Pulay mixing for spin-polarized calculations.



### Remark

For spin-polarized calculations, when SCF has difficulty to converge, increasing the mixing parameter to magnetization density might help. For example, setting it to 4.0, while turning off the preconditioner applied to the magnetization density (by setting `MIXING_PRECOND_MAG` to 'none') is a good choice.

## MIXING\_PARAMETER\_SIMPLE\_MAG

### Type

Double

### Unit

No unit

### Default

Automatically set to the same as [MIXING\\_PARAMETER\\_MAG](#)

### Example

MIXING\_PARAMETER\_SIMPLE\_MAG: 4.0

### Description

The value of the relaxation parameter for the magnetization density used in the simple mixing step in the periodic Pulay scheme for spin-polarized calculations.

# PULAY\_FREQUENCY

Type

Integer

Unit

No unit

Default

1

Example

PULAY\_FREQUENCY: 4

Description

The frequency of Pulay mixing in Periodic Pulay.

Remark

The default value of 1 corresponds to Pulay mixing.

## PULAY\_RESTART

Type

Integer

Unit

No unit

Default

0

Example

PULAY\_RESTART: 1

Description

The flag for restarting the 'Periodic Pulay' mixing. If set to 0, the restarted Pulay method is turned off.

## MIXING\_PRECOND

Type

String

Unit

No unit

Default

kerker

Example

MIXING\_PRECOND: none

Description

This specifies the preconditioner used in the SCF iteration. Available options are: none, kerker.

## MIXING\_PRECOND\_MAG

Type

String

Unit

No unit

Default

none

Example

MIXING\_PRECOND\_MAG: kerker

Description

This specifies the preconditioner used for the magnetization density in the SCF iteration for spin-polarized calculations. Available options are: none, kerker.

# TOL\_PRECOND

Type

Double

Unit

No unit

Default

$h^2 \times 0.001$

Example

TOL\_PRECOND: 1e-4

## Description

The tolerance on the relative residual for the linear systems arising during the real-space preconditioning of the SCF.

## Remark

The linear systems do not need to be solved very accurately.  $h$  is the mesh spacing.

## PRECOND\_KERKER\_KTF

Type

Double

Unit

Bohr<sup>-1</sup>

Default

1.0

Example

PRECOND\_KERKER\_KTF: 0.8

Description

The Thomas-Fermi screening length appearing in the kerker preconditioner ([MIXING\\_PRECOND](#)).



## PRECOND\_KERKER\_THRESH

Type

Double

Unit

No unit

Default

0.1

Example

PRECOND\_KERKER\_THRESH: 0.0

Description

The threshold for the kerker preconditioner ([MIXING\\_PRECOND](#)).

Remark

This threshold will be scaled by the [MIXING\\_PARAMETER](#). If the threshold is set to 0, the original kerker preconditioner is recovered.

## PRECOND\_KERKER\_KTF\_MAG

Type

Double

Unit

Bohr<sup>-1</sup>

Default

1.0

Example

PRECOND\_KERKER\_KTF\_MAG: 0.8

Description

The Thomas-Fermi screening length appearing in the kerker preconditioner for the magnetization density ([MIXING\\_PRECOND\\_MAG](#)).

## PRECOND\_KERKER\_THRESH\_MAG

Type

Double

Unit

No unit

Default

0.1

Example

PRECOND\_KERKER\_THRESH\_MAG: 0.0

Description

The threshold for the kerker preconditioner the magnetization density ([MIXING\\_PRECOND\\_MAG](#)).

Remark

This threshold will be scaled by the [MIXING\\_PARAMETER\\_MAG](#). If the threshold is set to 0, the original kerker preconditioner is recovered.

# FIX\_RAND

Type

Integer

Unit

No unit

Default

0

Example

FIX\_RAND: 1

## Description

Flag to fix the random seeds for setting initial guesses. Once set to 1, the random seeds will be fixed for different runs and for different numbers of processors. This option will make sure the answers will be exactly the same (up to machine precision) when SPARC is executed with different numbers of processors.

# Electrostatics

# TOL\_POISSON

Type

Integer

Unit

No unit

Default

`TOL_SCF` × 0.01

Example

TOL\_POISSON: 1e-6

Description

The tolerance on the norm of the relative residual for the Poisson equation.

Remark

The tolerance for poisson solver should not be worse than `TOL_SCF`, otherwise it might seriously affect the convergence of the SCF iteration.

# MAXIT\_POISSON

Type

Integer

Unit

No unit

Default

3000

Example

MAXIT\_POISSON: 1000

Description

The maximum number of iterations for solving the Poisson equation using an iterative linear solver.

# TOL\_PSEUDOCHARGE

Type

Double

Unit

No unit

Default

`TOL_SCF` × 0.01

Example

TOL\_PSEUDOCHARGE: 1e-6

Description

The error in the net enclosed charge for the pseudocharge density of each atom.



## REFERENCE\_CUTOFF

Type

Double

Unit

Bohr

Default

0.5

Example

REFERENCE\_CUTOFF: 1.0

Description

The cutoff radius of the reference potential used for evaluating the electrostatic correction arising from overlapping pseudocharge densities.

Remark

This number should be smaller than half the smallest interatomic distance.

# Stress calculation

# CALC\_STRESS

Type

Integer

Unit

No unit

Default

0

Example

CALC\_STRESS: 1

Description

Flag for calculation of the Hellmann-Feynman stress tensor (in cartesian coordinates).

## CALC\_PRES

Type

Integer

Unit

No unit

Default

0

Example

CALC\_PRES: 1

Description

Flag for calculation of the pressure.

Remark

Pressure is directly calculated, without calculation of the stress tensor.

**QMD**

## MD\_FLAG

Type

Integer

Unit

No unit

Default

0

Example

MD\_FLAG: 1

Description

QMD simulations are performed if the flag is set to 1.

Remark

[MD\\_FLAG](#) and [RELAX\\_FLAG](#) both cannot be set to a value greater than 0.

## MD\_METHOD

Type

String

Unit

No unit

Default

NVT\_NH

Example

MD\_METHOD: NVE

### Description

Type of QMD to be performed. Currently, NVE (microcanonical ensemble), NVT\_NH (canonical ensemble with Nose-Hoover thermostat), NVK\_G (isokinetic ensemble with Gaussian thermostat), NPT\_NH (isothermal-isobaric ensemble with Nose-Hoover thermostat) and NPT\_NP (isothermal-isobaric ensemble with Nose-Poincare thermostat) are supported

## MD\_NSTEP

Type

Integer

Unit

No unit

Default

1e7

Example

MD\_NSTEP: 100

Description

Specifies the number of QMD steps.

Remark

If  $\text{MD\_NSTEP} = N$ , the QMD runs from 0 to  $(N - 1) \times \text{MD\_TIMESTEP}$  fs.



# MD\_TIMESTEP

Type

Double

Unit

Femtosecond

Default

1

Example

MD\_TIMESTEP: 0.1

Description

QMD time step.

Remark

Total QMD time is given by:  $\text{MD\_TIMESTEP} \times (\text{MD\_NSTEP} - 1)$ .

# ION\_TEMP

Type

Double

Unit

Kelvin

Default

No Default

Example

ION\_TEMP: 315

Description

Starting ionic temperature in QMD, used to generate initial velocity distribution.

Remark

Must be specified if [MD\\_FLAG](#) is set to 1. It is also the target temperature in [MD\\_METHOD](#) NPT\_NH and NPT\_NP.

## ION\_TEMP\_END

Type

Double

Unit

Kelvin

Default

ION\_TEMP

Example

ION\_TEMP\_END: 100

Description

Specifies the final temperature of the thermostat. Thermostat temperature is varied linearly from `ION_TEMP` to `ION_TEMP_END` with respect to time.

Remark

Available for NVT\_NH quantum molecular dynamics only. Not supported in NPT\_NH and NPT\_NP.

## ION\_VEL\_DSTR

Type

Integer

Unit

No unit

Default

2

Example

ION\_VEL\_DSTR: 1

Description

Specifies the type of distribution for the initial velocity of atoms based on their initial temperature. Choose 1 for uniform velocity distribution and 2 for Maxwell-Boltzmann distribution.

Remark

Currently, the code supports only two options for the variable.

## ION\_VEL\_DSTR\_RAND

Type

Integer

Unit

No unit

Default

0

Example

ION\_VEL\_DSTR\_RAND: 1

Description

Flag to reseed the initial velocities of atoms in a QMD simulation. Set this flag to 1 to shuffle (change the random seed for) the initial velocities for different runs. Set this flag to 0 to maintain the same initial velocities.

Remark

This option is convenient for parallel statistics calculations.

# QMASS

Type

Double

Unit

atomic unit

Default

1653.654933459720

Example

QMASS: 100000

Description

Gives the inertia factor for Nose Hoover thermostat.

Remark

Applicable to NVT\_NH [MD\\_METHOD](#) only.

# NPT\_NH\_QMASS

## Type

1st number int; others double

## Unit

atomic unit

## Default

No default value

## Example

NPT\_NH\_QMASS: 2 700.0 700.0

## Description

Gives the amount (first number) and inertia masses (others) of thermostats in NPT\_NH. The maximum amount of thermostat variables of the Nose-Hoover chain is 60

## Remark

Applicable to NPT\_NH [MD\\_METHOD](#) only. Program will exit if NPT\_NH is selected but NPT\_NH\_QMASS is not input

# NPT\_NH\_BMASS

Type

Double

Unit

atomic unit

Default

No default value

Example

NPT\_NH\_BMASS: 5000

Description

Gives the inertia mass for the barostat variable in NPT\_NH.

Remark

Applicable to NPT\_NH [MD\\_METHOD](#) only. Program will exit if NPT\_NH is selected but NPT\_NH\_BMASS is not input



## NPT\_NP\_QMASS

Type

Double

Unit

atomic unit

Default

No default value

Example

NPT\_NP\_QMASS: 100

Description

Gives the inertia mass for the thermostat variable in NPT\_NP.

Remark

Applicable to NPT\_NP [MD\\_METHOD](#) only. Program will exit if NPT\_NP is selected but NPT\_NP\_BMASS is not input

## NPT\_NP\_BMASS

Type

Double

Unit

atomic unit

Default

No default value

Example

NPT\_NP\_BMASS: 20

Description

Gives the inertia mass for the barostat variable in NPT\_NP.

Remark

Applicable to NPT\_NP [MD\\_METHOD](#) only. Program will exit if NPT\_NP is selected but NPT\_NP\_BMASS is not input

## NPT\_SCALE\_VECS

### Type

Permutation of 1, 2, 3

### Unit

No unit

### Default

1 2 3

### Example

NPT\_SCALE\_VECS: 2

### Description

Specify which lattice vectors can be rescaled in NPT-NH

### Remark

Only numbers 1, 2 and 3 can be accepted. For example, if “2 3” is the input, the cell will only expand or shrink in the directions of lattice vector 2 and lattice vector 3. Only NPT-NH can specify the rescaled vector

# TARGET\_PRESSURE

Type

Double

Unit

GPa

Default

0.0

Example

TARGET\_PRESSURE: 40.9611

Description

Gives the outer pressure in NPT\_NH and NPT\_NP.

Remark

Applicable to NPT\_NH and NPT\_NP [MD\\_METHOD](#) only.

# RESTART\_FLAG

Type

Integer

Unit

No unit

Default

0

Example

RESTART\_FLAG: 0

Description

Flag for restarting quantum molecular dynamics and structural relaxation. Stores last three histories for quantum molecular dynamics simulations in .restart, .restart-0 and .restart-1 files, respectively.

## RESTART\_FLAG

### Remark

Restarts from the previous configuration which is stored in a .restart file. Currently, code provides restart feature for atomic relaxation and QMD only.

# TWTIME

Type

Double

Unit

min

Default

1e9

Example

TWTIME: 1000

Description

Gives the upper bound on the wall time for quantum molecular dynamics.

# Structural relaxation



# RELAX\_FLAG

Type

Integer

Unit

No unit

Default

0

Example

RELAX\_FLAG: 1

## Description

Flag for performing structural relaxation. 0 means no structural relaxation. 1 represents relaxation of atom positions. 2 represents optimization of volume with the fractional coordinates of the atoms fixed. 3 represents full optimization of the cell i.e., both atoms and cell volume are relaxed

## Remark

This flag should not be specified if [MD\\_FLAG](#) is set to 1.

## RELAX\_METHOD

Type

String

Unit

No unit

Default

LBFGS

Example

RELAX\_METHOD: NLCG

Description

Specifies the algorithm for structural relaxation. The choices are 'LBFGS' (limited-memory BFGS), 'NLCG' (Non-linear conjugate gradient), and 'FIRE' (Fast inertial relaxation engine).

Remark

LBFGS is typically the best choice.

# RELAX\_NITER

Type

Integer

Unit

No unit

Default

300

Example

RELAX\_NITER: 25

Description

Specifies the maximum number of iterations for the structural relaxation ([RELAX\\_FLAG](#)).

Remark

If [RESTART\\_FLAG](#) is set to 1, then relaxation will restart from the last atomic configuration and run for maximum of [RELAX\\_NITER](#) iterations.

# TOL\_RELAX

Type

Double

Unit

Ha/Bohr

Default

5e-4

Example

TOL\_RELAX: 1e-3

Description

Specifies the tolerance for termination of the structural relaxation. The tolerance is defined on the maximum force component (in absolute sense) over all atoms.

## TOL\_RELAX\_CELL

Type

Double

Unit

GPa

Default

1e-2

Example

TOL\_RELAX: 1e-3

Description

Specifies the tolerance for termination of the cell relaxation. The tolerance is defined on the maximum principle stress component.

# RELAX\_MAXDILAT

Type

Double

Unit

No unit

Default

1.06

Example

RELAX\_MAXDILAT: 1.4

## Description

The maximum scaling of the volume allowed with respect to the initial volume defined by [CELL](#) and [LATVEC](#). This will determine the upper-bound and lower-bound in the bisection method (Brent's method) for the volume optimization.

## NLCG\_SIGMA

Type

Double

Unit

No unit

Default

0.5

Example

NLCG\_SIGMA: 1

Description

Parameter in the secant method used to control the step length in NLCG ([RELAX\\_METHOD](#)).

Remark

Default value works well in most cases.

# L\_HISTORY

Type

Integer

Unit

No unit

Default

20

Example

L\_HISTORY: 15

Description

Size of history in LBFGS ([RELAX\\_METHOD](#)).

Remark

Default value works well in most cases.



# L\_FINISH\_STP

Type

Double

Unit

Bohr

Default

5e-3

Example

L\_FINISH\_STP: 0.01

Description

Step length for line optimizer in LBFGS ([RELAX\\_METHOD](#)).

Remark

Default value works well in most cases.

## L\_MAXMOV

Type

Double

Unit

Bohr

Default

0.2

Example

L\_MAXMOV: 1.0

Description

The maximum allowed step size in LBFGS ([RELAX\\_METHOD](#)).

Remark

Default value works well in most cases.

# L\_AUTOSCALE

Type

Integer

Unit

No unit

Default

1

Example

L\_AUTOSCALE: 0

Description

Flag for automatically determining the inverse curvature that is used to determine the direction for next iteration in LBFGS ([RELAX\\_METHOD](#)).

Remark

Default works well in most cases.

# L\_LINEOPT

Type

Integer

Unit

No unit

Default

1

Example

L\_LINEOPT: 0

Description

Flag for atomic force based line minimization in LBFGS ([RELAX\\_METHOD](#)).

Remark

Required only if [L\\_AUTOSCALE](#) is 0.

## L\_ICURV

Type

Double

Unit

No unit

Default

1.0

Example

L\_ICURV: 0.1

Description

Initial inverse curvature, used to construct the inverse Hessian matrix in LBFGS ([RELAX\\_METHOD](#)).

Remark

Needed only if [L\\_AUTOSCALE](#) is 0. Default value works well in most cases.

# FIRE\_DT

Type

Double

Unit

Femto second

Default

1

Example

FIRE\_DT: 0.1

Description

Time step used in FIRE ([RELAX\\_METHOD](#)).

Remark

Default value works well in most cases.

# FIRE\_MASS

Type

Double

Unit

Atomic mass unit

Default

1.0

Example

FIRE\_MASS: 2.5

Description

Pseudomass used in FIRE ([RELAX\\_METHOD](#)).

Remark

Default value works well in most cases.

# FIRE\_MAXMOV

Type

Double

Unit

Bohr

Default

0.2

Example

FIRE\_MAXMOV: 1.0

Description

Maximum movement for any atom in FIRE ([RELAX\\_METHOD](#)).

Remark

Default value works well in most cases.



# Print options

# PRINT\_ATOMS

Type

0 or 1

Unit

No unit

Default

0

Example

PRINT\_ATOMS: 1

## Description

Flag for writing the atomic positions. For ground-state calculations, atom positions are printed to a '.static' output file. For structural relaxation calculations, atom positions are printed to a '.geopt' file. For QMD calculations, atom positions are printed to a '.aimd' file.

# PRINT\_FORCES

Type

0 or 1

Unit

No unit

Default

0

Example

PRINT\_FORCES: 1

## Description

Flag for writing the atomic forces. For ground-state calculations, forces are printed to a '.static' output file. For structural relaxation calculations, forces are printed to a '.geopt' file. For QMD calculations, forces are printed to a '.aimd' file.

## PRINT\_MDOUT

Type

0 or 1

Unit

No unit

Default

1

Example

PRINT\_MDOUT: 0

Description

Flag for printing the the QMD output into the .aimd file.

# PRINT\_RELAXOUT

Type

0 or 1

Unit

No unit

Default

1

Example

PRINT\_RELAXOUT: 0

Description

Flag for printing the structural relaxation data in a .geopt file.

Remark

Required only if [RELAX\\_FLAG](#) is greater than 0.

# PRINT\_RESTART

Type

0 or 1

Unit

No unit

Default

1

Example

PRINT\_RESTART: 0

Description

Flag for writing the .restart file, used to restart QMD and structural relaxation simulations.

Remark

Relevant only if either `MD_FLAG` is 1 or `RELAX_FLAG` is 1.

## PRINT\_RESTART\_FQ

Type

Integer

Unit

No unit

Default

1

Example

PRINT\_RESTART\_FQ: 10

Description

Frequency at which .restart file is written in QMD and structural optimization simulations.

Remark

Relevant only if either `MD_FLAG` is 1 or `RELAX_FLAG` is 1.

# PRINT\_VELS

Type

0 or 1

Unit

No unit

Default

1

Example

PRINT\_VELS: 0

Description

Flag for printing the ion velocities in an QMD simulation into the .aimd file.

Remark

Relevant only if [MD\\_FLAG](#) is set to 1.



## OUTPUT\_FILE

Type

String

Unit

No unit

Default

Same as the input file name

Example

OUTPUT\_FILE: myfname

Description

The name of the output files. The output files are attached with a suffix ('.out', '.static', '.geopt' and '.aimd').

### Remark

If an output file with the same name already exist, the results will be written to a file with a number attached, e.g., 'myfname.out\_1'. The maximum number of output files with the same name allowed is 100. After that the output files will be overwritten in succession.

# Parallelization options

## NP\_SPIN\_PARAL

Type

Integer

Unit

No unit

Default

Automatically optimized

Example

NP\_SPIN\_PARAL: 2

Description

Number of spin groups.

Remark

This option is for development purpose. It's better to let SPARC choose the parallization parameters in practice.

# NP\_KPOINT\_PARAL

Type

Integer

Unit

No unit

Default

Automatically optimized

Example

NP\_KPOINT\_PARAL: 5

Description

Number of k-point groups.

Remark

This option is for development purpose. It's better to let SPARC choose the parallization parameters in practice.

## NP\_BAND\_PARAL

Type

Integer

Unit

No unit

Default

Automatically optimized

Example

NP\_BAND\_PARAL: 5

Description

Number of band groups.

Remark

This option is for development purpose. It's better to let SPARC choose the parallization parameters in practice.

# NP\_DOMAIN\_PARAL

Type

Integer

Unit

No unit

Default

Automatically optimized

Example

NP\_DOMAIN\_PARAL: 3 3 2

Description

Dimensions of the 3D Cartesian topology embedded in each band group.

Remark

This option is for development purpose. It's better to let SPARC choose the parallization parameters in practice.

## NP\_DOMAIN\_PHI\_PARAL

### Type

Integer

### Unit

No unit

### Default

Automatically optimized

### Example

NP\_DOMAIN\_PHI\_PARAL: 1 1 2

### Description

Dimensions of the 3D Cartesian topology embedded in the global communicator.

### Remark

This option is for development purpose. It's better to let SPARC choose the parallization parameters in practice.



# EIG\_SERIAL\_MAXNS

Type

Integer

Unit

No unit

Default

2000

Example

EIG\_SERIAL\_MAXNS: 1000

## Description

Maximum **NSTATES** value up to which a serial algorithm will be used to solve the subspace eigenproblem.

## Remark

If one wants to use a parallel algorithm to solve the subspace eigenproblem for all cases, simply set EIG\_SERIAL\_MAXNS to 0. Alternatively, set EIG\_SERIAL\_MAXNS to a very large value to always use serial algorithm.

## EIG\_PARAL\_BLKSZ

Type

Integer

Unit

No unit

Default

128

Example

EIG\_PARAL\_BLKSZ: 64

Description

Block size for the distribution of matrix in block-cyclic format in a parallel algorithm for solving the subspace eigenproblem.