SANDIA REPORT

SAND2017-12575
Unlimited Release
Printed November 15, 2017

EMPHASIS™/Nevada CABANA User Guide Version 2.1.2

C. David Turner, William J. Bohnhoff, Jennifer L. Powell

Prepared by
Sandia National Laboratories
Albuquerque, New Mexico 87185 and Livermore, California 94550

Sandia National Laboratories is a multimission laboratory managed and operated by National Technology and Engineering Solutions of Sandia, LLC, a wholly owned subsidiary of Honeywell International, Inc., for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA0003525.

Sandia National Laboratories

Issued by Sandia National Laboratories, operated for the United States Department of Energy by National Technology and Engineering Solutions of Sandia, LLC.

Available to DOE and DOE contractors from
U.S. Department of Energy
Office of Scientific and Technical Information
P.O. Box 62
Oak Ridge, TN 37831

Telephone: (865) 576-8401
Facsimile: (865) 576-5728
E-Mail: reports@adonis.osti.gov
Online ordering: http://www.osti.gov/bridge

Available to the public from
U.S. Department of Commerce
National Technical Information Service
5285 Port Royal Rd
Springfield, VA 22161

Telephone: (800) 553-6847
Facsimile: (703) 605-6900
E-Mail: orders@ntis.fedworld.gov
Online ordering: http://www.ntis.gov/help/ordermethods.asp?loc=7-4-0#online

SAND2017-12575
Unlimited Release
Printed November 15, 2017

EMPHASIS^{TM}/Nevada CABANA
User Guide Version 2.1.2

C. David Turner, William J. Bohnhoff, Jennifer L. Powell

Sandia National Laboratories
P.O. Box 5800
Albuquerque, New Mexico 87185

Abstract

The CABle ANAlysis (CABANA) portion of the EMPHASIS^{TM} suite is designed specifically for the simulation of cable SGEMP. The code can be used to evaluate the response of a specific cable design to threat or to compare and minimize the relative response of difference designs. This document provides user-specific information to facilitate the application of the code to cables of interest.

Acknowledgment

The authors would like to thank all of those individuals who have helped to bring CABANA to the point it is today, including Gary Scrivner and Wesley Fan for many useful theory and design discussions.

Contents

Appendix

List of Figures

EMPHASIS CABANA User Guide

1 Introduction

EMPHASIS™/Nevada CABle ANAlysis (CABANA) [12] solves the electrical portion of a cable system-generated electromagnetic pulse (SGEMP) simulation. It takes specific results from the deterministic radiation-transport code SCEPTRE [5] as sources and computes the resulting electrical response to an arbitrary cable load. The cable geometry itself is also arbitrary and is limited only by the patience of the user in meshing and by the available computing resources for the solution. The CABANA simulation involves solution of the quasi-static Maxwell equations using finite-element method (FEM) techniques [12].

The CABANA sources required from SCEPTRE are:

1) CHARGE_OFF_CONDUCTORS, the net charge knocked off each cable internal conductor (supplied implicitly by SCEPTRE),

2) CHARGE deposited into each finite element, and

3) ENERGY_DEPOSITION at each element node

These data are written to the SCEPTRE output in EXODUSII [7] format suitable for reading into CABANA. Since CABANA is written in MKS units, it expects the SCEPTRE results in units of Coulomb(C)/m for both CHARGE_OFF_CONDUCTORS and CHARGE. The units of ENERGY_DEPOSITION are expected to be Rads (dielectric).

The sources produced by SCEPTRE for CABANA are not time dependent. Instead, they represent the total integrated values over the specified radiation pulse. CABANA applies the time dependence to the simulation by assuring, for example, that the total charge in a given element is applied over the specified pulse shape.

2 Cable SGEMP Simulation Process

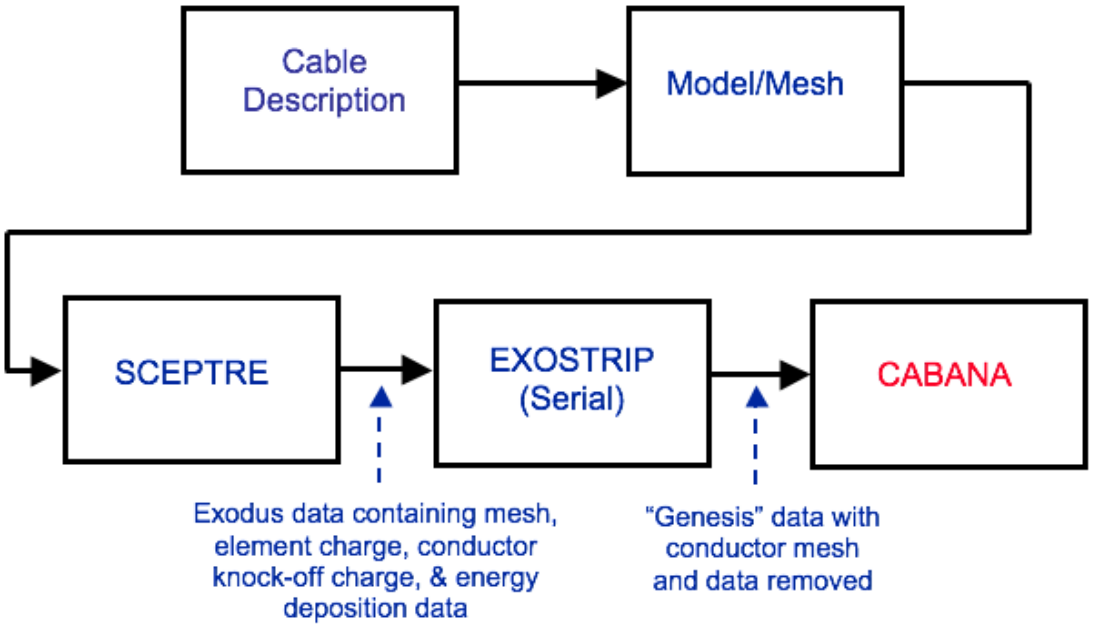


Figure 1. The Cable-SGEMP simulation process.

The cable SGEMP simulation process is shown in Fig. 1. The cable must first be modeled and meshed with using 2D or 3D-periodic unstructured elements suitable for SCEPTRE and CABANA. This is an involved process and is described in the original CEPTRE user guide [4]. These two types of elements are associated with the two CABANA simulation modes: 2.5D-mode for 2D cable cross section structures and 3D-mode for 3D-periodic cable structures. Examples of 3D-periodic structures are twisted-shielded pair or cables with braided cable shields where the shield breaks can be modeled as approximately periodic.

The critical aspects of the mesh for CABANA are that boundaries are described by EXODUSII sidesets (either single- or double-sided) and that the elements are 2^{nd} order, utilizing quadratic shape functions. This is required to obtain sufficient accuracy in the computation of electric field. The elements can be either 6-node triangles or 8-node quadrilaterals in 2D and 20-node hexahedra in 3D. The elements should also be reasonably well shaped. Quadrilaterals with aspect ratios of up to approximately 80 can be handled, but expect run times to increase dramatically as the matrix becomes more ill-conditioned requiring significantly more iterations to converge to the required solution tolerance.

After SCEPTRE solves the radiation-transport problem, the data for CABANA are written into an EXODUSII results file. Since SCEPTRE is required to mesh the cable conductors but CABANA has no need for this mesh, it is removed along with the associated

data using the SCEPTRE post-processing tools while creating a "Genesis" file for CABANA. "Genesis" normally describes a mesh description file in EXODUSII format containing mesh only, no data, thus the quotation marks. This "Genesis" file, along with a CABANA input file is all that is required for a CABANA simulation. During this process, SCEPTRE post-processing also converts the mesh coordinates from cm to m for CABANA.

3 CABANA Results

The desired result from a CABANA simulation is normally the current delivered to the cable load(s). This is provided in HISPLT format under the name "LOAD-CURRENT" for a cable with a single inner conductor in units of Amps. For a cable with multiple inner conductors, the name is "LOAD-CURRENT-X" where "X" runs from 0 to N-1 where N is the number of inner conductors. The order of these load current results in the HISPLT file is determined by the order that the cable conductors are defined in the input file using the **conductor** keyword described in a later section.

Additionally, a running integral of all load currents is computed and dumped at the end of the simulation as "Conductor # Load Charge= *value*", where in this case the "Conductor #" is the sideset id identified with the conductor on the **conductor** keyword line for that conductor.

Also monitored is the maximum global electric field magnitude throughout the simulation and is reported at the end of the simulation as "Global Max Magnitude Electric Field = *value*".

If requested, additional results such as POTENTIAL, ELECTRIC-FIELD, CHARGE-DENSITY, and CONDUCTIVITY are dumped in the EXODUSII results file. A discussion of CABANA output units and their relationship to the SCEPTRE simulation is presented in Appendix A.

4 CABANA Input File and Keywords

```
cable sgemp
  pulse shape, triangle, risetime 1.e-8, end
  initial time step, 5.e-10, end
  spice model, build
  number sections, 3, end
  section params, sec 1, len .1, exp, no
  section params, sec 2, len .45, exp, yes
  section params, sec 3, len .45, exp, yes
  default section length, .1
  load charge density, no
  load transport charge density, yes
  load dose, yes
  conductor, sideset 1, potential 1., rinpt 1.e8, rload 50.
  shield, sideset 21, potential 0.

  block 1
    material 1
  end

  block 2
    material 1
  end

  gradual startup factor 1.0
  maximum time step ratio 1.2
end
```

Figure 2. Typical CABANA physics keywords.

CABANA input uses the standard NEVADA input file format, which includes keywords for debugging, physics type, solver control, output control, and more. Details of all of these except for the specific physics can be found in the ALEGRA/NEVADA users guide [6]. A complete input file for one of the CABANA regression problems is given in Appendix B.

The format for specifying cable System Generated ElectroMagnetic Pulse (SGEMP) physics (i.e., CABANA) and associated keywords is shown in Fig. 2. The keyword `CABLE SGEMP` specifies CABANA physics to the code. Most of the remaining keywords are specific to CABANA and are described below.

For verification purposes, another keyword exists for CABANA physics: `CABLE SGEMP VERIFICATION, VERIFY` *int*. This keyword instantiates a different CABANA object and is NOT used for normal cable simulations. The `VERIFY` parameter specifies a particular analytic solution to be computed for comparison to the verification simulation.

For a single Poisson solution, the keyword `CABANA POISSON` instantiates another type of CABANA object for this purpose only. See the **Poisson solution keywords** section.

4.1 Time and Time History keywords

`PULSE SHAPE, type, [parameter real], [parameter real], END`

Specifies the time history of the radiation pulse.

Options for `type, [parameter real], [parameter real]` are:

`STEADY_STATE`

Unity for all time, useful primarily for verification

`TRIANGLE, RISETIME real, FALLTIME real, END`

If `RISETIME` only is given, rises linearly from zero to 1/`RISETIME` at t = `RISETIME`, in seconds (s), then falls linearly to zero at time 2*`RISETIME`. If `RISETIME` and `FALLTIME` are given, rises linearly from zero to 2/(`RISETIME` + `FALLTIME`) at t = `RISETIME` (s), then falls linearly to zero at time `RISETIME` + `FALLTIME`. Both have unit integrals.

`SQUARE, WIDTH real, END`

Amplitude 1/`WIDTH` at t = 0, remains constant until t = `WIDTH` (s), jumps back to amplitude zero (integral is unity)

`SIN_SQUARED, WIDTH real, END`

$f(x) = \frac{2}{\text{WIDTH}} \sin^2\left(\frac{\pi t}{\text{WIDTH}}\right), \quad 0 < t < \text{WIDTH}$, zero otherwise (integral is unity)

`EXPONENTIAL_PULSE, ALPHA real, END`

$$f(x) = \frac{1}{\text{ALPHA}} \exp\left(-\frac{1}{\text{ALPHA}}\right) \text{(integral is unity)}$$

`DOUBLE_EXPONENTIAL_PULSE, ALPHA real, BETA real, END`

$$f(x) = \frac{1}{\text{ALPHA-BETA}} \left(\exp\left[-\frac{1}{\text{ALPHA}}\right] - \exp\left[-\frac{1}{\text{BETA}}\right]\right) \text{(integral is unity)}$$

`NEVADA, FUNCTION int, END`

Use a NEVADA generated function, either built-in or tabulated, with the NEVADA `FUNCTION` keyword [11].

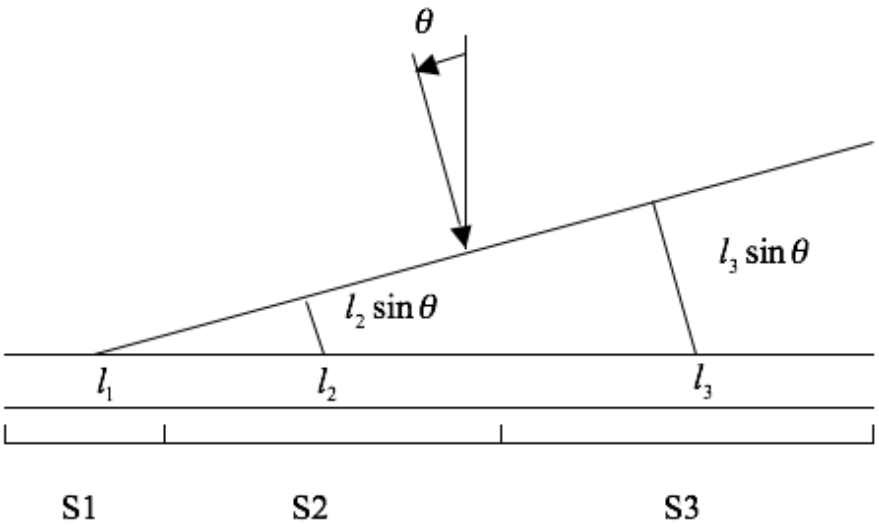
`INITIAL TIME STEP, real, [GAMMA DOT RATIO real], [TIME STEP SCALE real], END`

Specifies the beginning time step (s) for the simulation

`GAMMA DOT RATIO` and `TIME STEP SCALE` are optional parameters which, if included, will cause the time step to change to `INITIAL TIME STEP * TIME STEP SCALE` after the `PULSE SHAPE` function falls below `PULSE_SHAPE_MAX * GAMMA_DOT_RATIO`, where

PULSE_SHAPE_MAX is computed during the simulation. Typical values for GAMMA DOT RATIO and TIME STEP SCALE are 0.1 and 5.0, respectively. If these optional parameters are not included, the time step will remain constant at INITIAL TIME STEP for the entire simulation.

4.2 Radiation-incidence keyword



$$Section1 : f(t - \frac{(l_1 - l_o)\sin\theta}{c})$$

$$Section2 : f(t - \frac{(l_2 - l_o)\sin\theta}{c})$$

$$Section3 : f(t - \frac{(l_3 - l_o)\sin\theta}{c})$$

Figure 3. Non-normal incidence.

By default, CABANA applies the cable excitation as if it were planar, normally incident broadside to the cable sections. In this case, all cable sections are excited simultaneously. The following keyword can be used to change the angle of incidence as shown in Fig. 3 for a three-section cable. Here, the cable-section excitations are staggered by the appropriate retarded time based on the timing at the section *centers*. The user should keep in mind also that the "load" end of the cable is the end on the right in Fig. 3. Therefore, if it is desired to have the excitation plane strike the load-end first, the angle should be negative. Also in Fig. 3, l_0 is the cable origin, either the left end (for θ positive) or the right end (for θ negative).

`NONNORMAL INCIDENCE, ANGLE real`

Specifies non-normal planar incidence with angle angle from normal.

The `ANGLE` is the angle θ in Fig. 3, where $-90 < \theta < 90$.

4.3 SPICE control keywords

`SPICE MODEL, spice_option`

Specifies how SPICE is utilized and the origin of the spice model deck for the simulation.

Options for spice_option are:

`NO` - SPICE not used in simulation (default and NOT typical)

`USE` - Lumped-parameter SPICE deck will be read from file

`BUILD` - Lumped-parameter SPICE deck will be created and written to a file during startup

Generally, the initial simulation for a given cable geometry is accomplished using the `BUILD` option which writes the SPICE deck to a file (see `SPICE FILE` keyword below). If custom changes are desired to this SPICE model for subsequent simulations, the file can be edited and the `USE` option invoked thereafter. See Appendix C. Since the default is `NO`, either `BUILD` or `USE` *must* be specified for cable simulations.

`SPICE FILE, "filename"`

Specifies the filename from which to read the SPICE model deck if `SPICE MODEL, USE` is specified or the file to which to write the SPICE deck if `SPICE MODEL, BUILD` is specified.

"filename" is an ascii string and the quotes are required. If `SPICE FILE` is not specified, the SPICE deck is either read from or written to the default filename *problem_ name*.in. Note that this file, either the default or that specified by `SPICE FILE`, will be OVERWRITTEN for the next simulation if it is left in place on the file system.

`SPICE STEP FRACTION, real`

Specifies the fraction of the initial time step (see `INITIAL TIME STEP` keyword) which SPICE will use as a maximum internal time step.

The default is 0.1. In some cases, it is necessary to increase this value to something like 0.5 to allow SPICE to converge correctly. If such a SPICE error occurs, the error is trapped and the user is notified to modify this value.

`SPICE SOLVER, solver_option`

Specifies the desired circuit solver. Options are SPICE (default) or XYCE [3]. Either option will provide equivalent results.

4.4 Section keywords

`NUMBER SECTIONS, int, [SEC int, LEN real],[SEC int ,LEN real],[...], END`

Specifies the number of sections to divide the cable length into for a 2.5D simulation and optionally specifies the lengths of each section. However, for a large number of sections, the line length may exceed the maximum. In this case, use multiple of the below `SECTION PARAMS` keywords.

Each `[SEC` int, `LEN` real] specifies the section number and length, in meters (m), of one of the sections. If a section is not specified, its' length is defined by the `DEFAULT SECTION LENGTH` keyword below. In addition, it will be exposed to radiation by default (see `SECTION PARAMS` below).

`SECTION PARAMS, SEC[TION] int, LEN[GTH] real, EXP[OSED], bool [, LOAD real]`

Optionally specifies the lengths of each section and whether the section is exposed to radiation. The length parameter can be specified with the `NUMBER SECTIONS` keyword above if desired.

Each keyword of this type specifies the section number and length (m) of one section. Options for `EXP` are YES (or TRUE) and NO (or FALSE). Note that the comma after `EXP` must be present. If a section is not specified, its' length is defined by the `DEFAULT SECTION LENGTH` keyword below and `EXP` is YES.

The optional keyword `LOAD` specifies a resistive load (ohms) at the *midpoint* of each section. Loads need not be specified on every section but if one section has a load all will have placeholder loads inserted into the SPICE deck at the default value of 1.e8, or essentially open. The user may later modify these loads in the SPICE deck followed by using the `SPICE MODEL, USE` keyword to force re-use of the deck.

`DEFAULT SECTION LENGTH, real`

Specifies the default length (m) of each cable section

`MULTISECTION MAP, RADTRANS_FILE "filename", SECTION_INDEX int, [...],END`

Optionally specifies the mapping of auxiliary radiation-transport results to cable sections for multi-section excitation.

Lines of the above form are repeated for each auxillary SCEPTRE simulation results file. Note that the use of this keyword is OPTIONAL. If no multisection map is

specified, the default map is used: SCEPTRE data stored in the simulation Genesis file is applied to ALL SECTIONS in the cable. The required parameters are:

`RADTRANS_FILE:` "filename" is an ASCII string for the name of a SCEPTRE results (.exo) file

`SECTION_INDEX` (or shortened to `SECTION`): an integer value in the range 1 thru `NUMBER SECTIONS` specifying each section the specified SCEPTRE results will be applied to. Additional `SECTION,` int combinations can be added before the `END` specifying other sections to receive data from this `RADTRANS_FILE`.

4.5 SCEPTRE data keywords

`LOAD CHARGE DENSITY, option`

Specifies whether the SCEPTRE data for CHARGE density contained in the simulation Genesis file is loaded before the first time cycle

Options are:

`NO` - CHARGE not loaded up front, typical for actual cable simulation (default)

`YES` - CHARGE loaded up front, typical for verification simulation

`LOAD TRANSPORT CHARGE DENSITY, option`

Specifies whether SCEPTRE data for CHARGE density contained in the simulation Genesis file is prepared for loading into the simulation over the radiation pulse defined by `PULSE SHAPE`.

Options are:

`YES` - CHARGE prepared for loading, typical for actual cable simulation (default)

`NO` - CHARGE not prepared for loading, typical for verification simulation

`LOAD DOSE, option`

Specifies whether SCEPTRE data for ENERGY_DEPOSITION contained in the simulation Genesis file is prepared for loading into the simulation over the radiation pulse defined by `PULSE SHAPE`

Options are:

`YES` - ENERGY_DEPOSITION prepared for loading, typical for actual cable simulation (default)

`NO` - ENERGY_DEPOSITION not prepared for loading, typical for verification simulation or for turning off radiation induced conductivity (RIC)

`LOAD CONDUCTOR CURRENT, option`

Specifies whether the SCEPTRE data for knock-off charge from each cable `CONDUCTOR`, contained in the simulation Genesis file, is prepared for loading into the simulation over the radiation pulse defined by `PULSE SHAPE`

Options are:

`YES` - CONDUCTOR CHARGE prepared for loading, typical for actual cable simulation (default)

`NO` - CONDUCTOR CHARGE not prepared for loading, typical for verification simulation

4.6 Conductor knock-off charge keyword

`ENFORCE NEGATIVE KNOCK OFF, option`

This keyword specifies whether an error is issued and the simulation halted in the case that any conductor has a positive knock-off charge. In the case of a conductor defined by multiple side sets, a positive knock-off charge on any of those side sets will trigger the error. Normally this indicates a poorly resolved SCEPTRE mesh but for high (> 1 MeV) photon energy it may be feasible to get a valid positive knock-off charge. This keyword removes enforcement of this condition and allows the simulation to continue.

Options are:

`YES` - Fatal error is issued and the simulation halted (default)

`NO` - Simulation continues with warning

4.7 Cable topology keywords

`CONDUCTOR, SIDESET int, [SIDESET int,...], POTENTIAL real, RINPT real, RLOAD real`

Each internal cable conductor is specified with one of these keyword lines.

Required parameters are:

`SIDESET`: Specifies one (or more) sideset id(s) in the Genesis file defining this conductor

`POTENTIAL`: Specifies the initial potential, in volts (V), of the conductor (applies only to verification simulations, ignored for actual cable simulations)

Optional parameters are:

RINPT: Specifies the resistance (ohms) to be applied to the "input" (often floating) end of the SPICE model.

RLOAD: Specifies the resistance (ohms) to be applied to the "load" end of the SPICE model. It is through RLOAD that the main CABANA output, LOAD-CURRENT in the HISPLT file, flows.

If the optional cable loads are not specified, the defaults are RINPT=1.e8 ohms and RLOAD=50 ohms. The user should keep in mind that although RINPT can be anything, it is only the current through RLOAD which is monitored and output to the HISPLT file.

`INTERFACE, SIDESET int, [SIDESET int,...], INITIAL CHARGE DENSITY real`

Each cable dielectric interface of differing conductivity properties is specified with one of these keyword lines.

Required parameters are:

SIDESET: Specifies one (or more) sideset id(s) in the Genesis file defining this interface

INITIAL CHARGE DENSITY: Specifies the initial charge density (C/m) on the interface before the simulation begins (normally zero for cable simulations)

`SHIELD, SIDESET int, [SIDESET int,...], POTENTIAL real`

Specifies the required surrounding cable shield.

Required parameters are:

SIDESET: Specifies one (or more) sideset id(s) in the Genesis file defining the shield

POTENTIAL: Specifies the shield potential (V) for the simulation, normally zero for cable simulations

The user is responsible for properly defining a conductor, interface, or shield with the correct number of sidesets. CABANA has no way of knowing that a portion of the boundary for say, a conductor, has been left out.

CABANA requires these sidesets be single-sided. SCEPTRE [5] uses either single- or double-sided sidesets. The SCEPTRE post processor, when it generates stripped CABANA data, will guarantee that the sidesets going into CABANA are single-sided.

`SYMMETRY, type`

Specifies quarter or half mesh symmetry. A correction is applied to compute the correct response for the full problem.

Options are:

NONE	No symmetry (default)
QUARTER	Two symmetry planes
HALF	One symmetry plane

4.8 Shockley method keyword

`SHOCKLEY METHOD, option`

Specifies the use of the simple Shockley integral method [1]. This method removes the requirement for μm-scale meshing for SCEPTRE near the interfaces and provides faster convergence as well. However, RIC must be ignored since it cannot be modeled with this formulation. SCEPTRE must now provide a current density rather than a charge density. This current density is integrated against the computed Green's function solutions using the following:

$$R = -\int_V \bar{E} \bullet \bar{J} \mathrm{d}v$$

If this method is specified the other SCEPTRE data keywords are overridden to NO. Plot data is not available for CHARGE_DENSITY, POTENTIAL, or ELECTRIC_FIELD and should not be specified. Requesting TRANSPORT_CURRENT for plot data will provide the time history and spatial distribution of the source current density.

Options are:

`NO`	Use standard full-simulation method (default)
`YES`	Use Shockley method and override everything else

4.9 Charge relaxation keyword

`ENFORCE RELAXATION LIMIT, option1 [, PRINT WARNING, option2]`

If conductivity is present, this keyword specifies whether an error is issued and the simulation halted in the case that an element relaxation time is too short relative to the time step chosen. Here the element relaxation time is defined as *permittivity/conductivity.* Based on convergence verification tests, it has been determined that good results are obtained as long as the element relaxation time is less than twice the time step. In other words, as long as there are at lease two time samples per relaxation time.

Options for `ENFORCE RELAXATION LIMIT` are:

`YES` - Fatal error is issued and the simulation halted with information about the shortest elemental relaxation time (default)

`NO` - Simulation continues with or without warning

If `ENFORCE RELAXATION LIMIT` is `YES`, `PRINT WARNING` is irrelevant. Otherwise, the options for `PRINT WARNING` are:

YES - Simulation continues with warning printed containing the shortest elemental relaxation time

NO - Simulation continues without warning (default)

4.10 Box or Cavity SGEMP Drive inclusion keywords

Current sources can be included in the simulation to accommodate the forward coupling of box and cavity SGEMP drives during the cable simulation for a cable with a single inner conductor. These sources (one for box drive and/or one for cavity drive) are automatically inserted into the SPICE deck across the RINPT end of the conductor, as described by the CONDUCTOR keyword. Typically, the drive is provided through a tabular FUNCTION of samples from a box or cavity SGEMP simulation but it can be provided by one of the functional forms of the FUNCTION keyword as well.

```
BOX DRIVE, FUNCTION int [,SCALE real SHIFT real]
```

Specifies that a box-drive current source be created in the SPICE deck and that it be driven with the supplied function.

The tabular function can be scaled or shifted in time as necessary.

```
CAV DRIVE, FUNCTION int [,SCALE real SHIFT real]
```

Specifies that a cavity-drive current source be created in the SPICE deck and that it be driven with the supplied function.

The tabular function can be scaled or shifted in time as necessary.

BOX DRIVE is deprecated by the BOX CABLE coupling keyword below.

4.11 Box-Cable SGEMP Coupling keyword

```
BOX CABLE
```

Specifies a full coupled Box-Cable SGEMP simulation.

A detailed use description is found in the UTDEM User Manual [11].

4.12 Poisson solution keywords

```
POISSON SOLUTION, CHARGE DENSITY real, RESULTS FILE "filename"
```

Specifies a single Poisson solution and exit

Required parameters are:

`CHARGE DENSITY`: Specifies a uniform charge density (C/m^2) in the elements

`RESULTS FILE`: ASCII file to which the resulting element electric fields are written

The boundary conditions for the solution are specified using the `CONDUCTOR` keyword above.

```
EXPORT RESULTS, GENESIS FILE "filename", SIDESET int
```

Specifies that the resulting element electric fields are to be exported for a subsequent 3D **UTDEM** simulation

Required parameters are:

`GENESIS FILE`: 3D Genesis file for results export

`SIDESET`: Sideset in Genesis file to store results with

This Genesis file and sideset must be the same used to generate the 2D mesh for the Poisson solution in the first place using the `UTDEM SIDESET EXTRACTOR` physics option in EMPHASIS^{TM}.

The Poisson solution requires the framework keyword `TERMINATION CYCLE = 1` to be specified. This ends the simulation after a single solve as desired.

4.13 Single-cycle solution

If RIC is not important and the time history of the response is not required, it is possible to compute the total charge transferred to the load with a single computation cycle. This is accomplished by setting the following keywords:

`SPICE MODEL`,	no
`NUMBER SECTIONS`,	1, end
`DEFAULT SECTION LENGTH`,	length
`LOAD CHARGE DENSITY`,	yes
`LOAD CONDUCTOR CURRENT`,	yes
`LOAD TRANSPORT CHARGE DENSITY`,	no
`LOAD DOSE`,	no
`TERMINATION CYCLE`	1

After a single cycle, the response is computed from the conductor knock-off charge and the rho-induced charge on the conductor as follows:

$$R = q_{ko} - q_i$$

4.14 Framework-related keywords within the physics block

These keywords are really framework keywords [6] and must be placed within the physics definition keyword block.

`BLOCK int MATERIAL int`

Defines a finite-element block.

Relates the mesh block id `BLOCK` to material definition `MATERIAL`.

`FUNCTION int`

Define a user-defined function for use by the box/cavity drive keywords.

```
FUNCTION int
 time0   value0
 time1   value1
 time2   value2
 .       .
 .       .
 .       .
END
```

`GRADUAL STARTUP FACTOR real`

Factor by which the initial time step is multiplied, default is 0.01.

Gradually increases the initial time step. For CABANA, the value is normally set to 1.0.

`MAXIMUM TIME STEP RATIO real`

Maximum ratio by which a time step may grow in a given cycle.

Gradually increases the time step from the old to the new value. This effects CABANA when `TIME STEP SCALE` > 1.0 on the `INITIAL TIME STEP` line.

5 Framework Keywords

These keywords are framework keywords [6] which are required for a successful CABANA simulation.

5.1 Material-related keywords

`MATERIAL int`

Define a material model for a material.

```
MATERIAL int
 MODEL  int
 MODEL  int
 .      .
END
```

Relates the material `MATERIAL` to material model(s). For CABANA only one model applies to each material.

`MODEL int string`

Define a material model.

```
MODEL int string
 parameter  real
 parameter  real
 .          .
END
```

Relates the material model `MODEL` to a specific model name defined by the name string.

Options for string and parameter(s) are:

```
RIC ELECTRICAL
 EPS          real
 MU           real
 SIGMA0       real
 COEFFICIENT  real
 EXPONENT     real
END
```

or

```
HP GAS ELECTRICAL
```

EPS	real
MU	real
SIGMA0	real
DENSITY	real
WATER_FRACTION	real

END

where EPS is the relative permittivity of the medium, MU is the relative permeability of the medium, and SIGMA0 is the initial or dark conductivity (Mho/m) of the material.

The radiation-induced conductivity (RIC) model [2] takes the following form:

$$\sigma = \sigma_0 + \varepsilon K \dot{\gamma}^e$$

where σ is the conductivity in Mho/m, σ_0 is the dark conductivity in Mho/m, ε is the permittivity in Farad(F)/m, K (COEFFICIENT) is in ((Mho/F)/(Rad/s), $\dot{\gamma}$ is the dose rate in Rad/s, and e is the EXPONENT parameter. Typical values for Kapton are $\epsilon = 3.5\epsilon_0$, $K = 3.23 \cdot 10^{-6}$, and $e = 0.95$.

The high-pressure (HP) gas model is described in [8][10]. The independent variables in the model are DENSITY (kg/m^3) and WATER_FRACTION, nominally 1.23 and 0.02, respectively.

Example input file fragments for these model specifications are shown in Fig. 4.

```
Model 1 RIC Electrical
  eps 2.
  mu 1.
  sigma0 1.e-3
  coefficient 3.23e-6
  exponent 0.95
end

Model 2 HP Gas Electrical
  eps 2.
  mu 1.
  sigma0 0.
  density 1.23
  water_fraction 0.02
end
```

Figure 4. Typical material model descriptions.

5.2 Simulation time and output control keywords

Typical simulation time and output control keywords are shown in Fig. 5.

```
termination time = 1.e-8

emit screen, cycle interval = 1
emit plot, cycle interval = 10
emit hisplt, cycle interval = 1
plot variable
  potential
  electric_field
  charge_density
  vis_face_charge_density
  econ
  electron_concentration
end
```

Figure 5. Typical simulation and output control keywords.

`TERMINATION TIME = real`
Total time (s) for which to run the simulation

`TERMINATION CYCLE = int`
Total cycles for which to run the simulation

`EMIT SCREEN, CYCLE INTERVAL = int`
Print status line to standard out every `CYCLE INTERVAL` cycles

`EMIT PLOT, CYCLE INTERVAL = int`
Write `PLOT VARIABLE`s to exodus file every `CYCLE INTERVAL` cycles

`EMIT HISPLOT, CYCLE INTERVAL = int`
Write global variables to hisplt file every `CYCLE INTERVAL` cycles

`PLOT VARIABLE`

`PLOT VARIABLE`
registered-variable name
registered-variable name
...
`END`

Valid plot variables for CABANA are

potential
electric_field
charge_density
vis_face_charge_density (interface charge density)
econ (conductivity) (RIC and HP Gas model)
electron_concentration (HP Gas model)
negative_ion_concentration (HP Gas model)
avalanche_rate (HP Gas model)
attachment_rate (HP Gas model)

If plot variables are not desired the "emit plot" line can be omitted. If plot variables are desired but not at frequent intervals, the cycle interval should be set to large values to avoid exceedingly large exodus files.

5.3 Linear solver keywords

Typical linear solver keywords, in this case for AZTEC [9], are shown in Fig. 6. In this case, conjugate gradient (cg) is specified with no preconditioning but with symmetric diagonal scaling. No output is requested from AZTEC after each solve to a tolerance level of 1.e-9 with a maximum number of cg iterations set to 1000 (default is 500). The "polynomial order" should always be set to "1" for efficiency.

```
aztec
  solver,    cg
  precond,   none
  scaling,   sym_diag
  output,    none
  tol       = 1.e-9
  polynomial order, 1
  max iterations, 1000
end
```

Figure 6. Typical solver control keywords.

5.4 Debug mode keyword

Specific code debug information at run time can be requested using the following syntax:

`DEBUG MODE: debug-opt`

Some relevant values for `debug-opt` are:

`CABANA`

Produces verbose intermediate Cabana results useful for understanding and following the simulation as it progresses.

`LOCATION`
: Provides a quick way to observe in the standard out stream progress through the code execution, and in particular to see where the most time is being spent.

`SIGNALS | FPE`
: These will catch and report events such as floating-point exceptions.

6 Follow-up Simulations with Stored Charge

It may be desirable to do a series of simulations where the stored charge remaining in the dielectric from previous simulations is used as an initial charge distribution for subsequent simulations. This can be accomplished with CABANA by utilizing the restart capability of the framework. The procedure is as follows:

Add the following restart keywords to the initial input file:

`RESTART DUMPS, 1`

`EMIT RESTART, CYCLE INTERVAL =` int OR `EMIT RESTART, TIME INTERVAL =` real

`READ RESTART DUMP = -1`

The first indicates that only one restart dump is saved during the simulation. The second tells the framework to emit a restart dump either every `CYCLE INTERVAL` cycles or every `TIME INTERVAL` seconds. The third says that upon restart, the framework should read and load data from the latest restart dump available. In the case of the initial simulation of the series, the user should verify that no restart dumps exist in the directory and the simulation will then start at time=0. as desired. Restart dumps are written to files in the format: *problem_ name.*dmp*.restart_ number.*

For subsequent simulations, the user *must* change the `TERMINATION TIME` to have a later termination time for the next simulation. With this change, the follow-on simulation will begin at the time of the final restart dump from the previous simulation after loading the residual dielectric charge. In addition, the original *problem_ name.*gen file can be optionally replaced at this time with a new one having the identical mesh description but containing a different set of radiation-transport results. The user is also free to change other simulation parameters such as pulse shape, time step, etc.

A key assumption in this process is that the SPICE circuit model is completely discharged after each simulation in the series, since no SPICE state is saved. This requires that each simulation be run until such time as this is essentially the case, i.e., all cable reflections have subsided and a reasonably steady-state condition has been reached.

7 Conclusion

This document, along with the user guides for a modeling and meshing tool such as CUBIT, the radiation-transport code SCEPTRE, and the SCEPTRE post-processing tools should allow the user to successfully utilize CABANA to simulate the cable SGEMP response of an arbitrary cable geometry. To gain experience, the CABANA regression suite contains several realistic cable simulations, both 2.5D and 3D. Although these do not have realistic radiation-transport data in their Genesis files, the data does adequately exercise the algorithms in the code and fully demonstrates the cable simulation.

References

[1] C. R. Drumm, G. J. Scrivner, and J. J. Hohlfelder. Compton Diode and Coaxial Cable Response to Gamma Radiation. Technical report, Sandia National Laboratories.

[2] Steven Face. Cable Response Parameter Determination. Technical Report K-87-32U(R), Kaman Sciences Corporation, Colorado Springs, CO, April 1987.

[3] E. R. Keiter et al. Xyce^{TM} parallel electronic simulator reference guide, version 6.2. Technical Report SAND2014-18057, Sandia National Laboratories, September 2014.

[4] J. L. Liscum-Powell et al. CEPTRE Version 1.0 Users Manual. Technical Report SAND2005-7798, Sandia National Laboratories, January 2006.

[5] S. Pautz, B. Bohnhoff, C. Drumm, and W. Fan. Parallel Discrete Ordinates Methods in the SCEPTRE Project. Proc. Int. Conf. on Mathematics, Computational Methods and Reactor Physics, May 3-7 2009.

[6] A. C. Robinson et al. ALEGRA user manual. Technical Report SAND2014-16031, Sandia National Laboratories, 2014.

[7] L. A. Schoof and V. R. Yarberry. EXODUS II: A Finite Element Data Model. Technical report SAND92-2137, Sandia National Laboratories, Albuquerque, NM, November 1995.

[8] Tom Stringer. ITT Corp., private communication.

[9] R. S. Tuminaro, M. Herous, S. A. Hutchinson, and J. N. Shadid. Official Aztec User's Guide Version 2.1. Technical Report SAND99-8801J, Sandia National Laboratories, November 1999.

[10] Tumolillo and Wondra. PRES 3D: A Computer Code for the Self Consistent Solution of the Maxwell Lorentz Three Species Air Chemistry Equations in Three D. *IEEE Trans. Nucl. Sci.*, NS-24(6):2456–2460, 1977.

[11] C. D. Turner, D. B. Pasik, M. F. Seidel, T. D. Pointon, and K. L. Cartwright. EMPHASIS^{TM}/Nevada UTDEM user guide version 2.1.1. Technical Report SAND2014-16735, Sandia National Laboratories, August 2014.

[12] C. D. Turner and G. D. Scrivner. CABANA Serial/Development Version Description, Verification, and Validation. Technical Report SAND2001-3567, Sandia National Laboratories, November 2001.

A CABANA output units

The units associated with the CABANA output depend on the source normalization applied during the SCEPTRE simulation. Assume that the source intensity is F cal/cm^2 with an average spectral energy of E MeV/photon. SCEPTRE source normalization multiplies the source intensity F by

$$\frac{2.61 \cdot 10^{13}(MeV/cal)}{E(MeV/photon)}$$

yielding the fluence in photon/cm^2. With this normalization, the load currents from CABANA are in units of Amps (A). Without this normalization, the load currents are in units of A/(photon/cm^2). Centimeters appear here because SCEPTRE is written in CGS units.

The units of other possible CABANA outputs are V for potential, V/m for electric field, C/m^3 for charge density, and mho/m for conductivity.

B Complete CABANA input file

```
$----------------------------------BEGIN_QA---------------------------
$ Tags:       cabana physics library
$ CVS: $Id: coax_quad8.inp,v 1.6 2003/01/22 05:32:12 wjbohnh Exp $
$-----------------------------------END_QA----------------------------

$debug mode: CABANA
$debug mode: LOCATION

title
  CABANA: Coax w/coarse quad8 mesh

$$$$$$$$$$$$$$$$$$$ physics options $$$$$$$$$$$$$$$$$$$

$ The following two lines should be specified instead of CABLE SGEMP
$ if a verification test is desired rather than a normal simulation
$CABLE SGEMP VERIFICATION
  $VERIFY, 1

CABLE SGEMP
  pulse shape, triangle, risetime 1.e-9, end
  initial time step, 1.e-10, end
```

```
  spice model, build
  number sections, 1, end
  default section length, .1
  load charge density, no
  load transport charge density, yes
  load dose, yes
  conductor, sideset 1, potential 1., rinpt 1.e8, rload 50.
  shield, sideset 21, potential 0.

block 1
  material 1
end

block 2
  material 1
end

  gradual startup factor 1.0
  maximum time step ratio 1.2

end

aztec
  solver,   cg
  precond,  none
  scaling,  sym_diag
  output,   none
  tol      = 1.e-9
  polynomial order, 1
end

units, si

$$$$$$$$$$$$$$$$$$ execution control $$$$$$$$$$$$$$$$$$

termination time = 1.e-8

$$$$$$$$$$$$$$$$$$ output control    $$$$$$$$$$$$$$$$$$

emit screen, cycle interval = 1
emit plot, cycle interval = 1
emit hisplt, cycle interval = 1
plot variable
  potential
  electric_field
```

```
  charge_density
end

$$$$$$$$$$$$$$$$$$ material models $$$$$$$$$$$$$$$$$$

Material 1
  Model 1
end

Model 1 RIC Electrical
  eps 2.
  mu 1.
  sigma0 1.e-3
  coefficient 3.23e-6
  exponent 0.95
end

exit
```

C Modifying the SPICE Deck

As noted in the **SPICE control keywords** section, the `SPICE MODEL, BUILD` keyword is typically used in an initial simulation to generate the SPICE deck for a cable. For a subsequent simulation, if the user wishes to modify the cable load to something more complicated than the simple resistive load added by the `CONDUCTOR` keyword then the SPICE deck must be edited manually. For subsequent simulations, the user *must* use the `SPICE MODEL, USE` keyword to have the code read the modified deck. In addition, if the user desires to change the name of the SPICE deck from the default name of *problem_ name*.in, the `SPICE FILE,` "filename" keyword must be used to specify the new name.

Modifications to the SPICE deck follow standard SPICE conventions. For example, if a complicated load is to be connected to the cable then the load SPICE circuit description can either be inserted directly into the file, using unique node numbers and device names of course, or a SPICE **.SUBCKT** can be used.

The first step in modifying the file for new load(s) is to determine what nodes the load(s) are connected to. The SPICE deck generation algorithm places resistive loads on each end of each cable conductor. Referring to the `CONDUCTOR` keyword, the `RINPT` value is placed on the "input" end of the cable and the `RLOAD` value is placed on the "output" end. The "output" end is the "LOAD-CURRENT" which is monitored and output by the code in the *problem_ name*.his file. The SPICE solver simply returns the node voltage across the output "load", assumed to be a simple resistor. The return value is then divided by the load resistance to determine the load current.

The algorithm used for numbering the loads is as follows. Assume the cable has N conductors divided into M sections. The "input" load on the first conductor (the first `CONDUCTOR` keyword line in the *problem_ name*.inp file) is R[1]. The "output" load on the first conductor is R[2]. The input load on the second conductor is R[3] and the ouput load is R[4], etc, down to the final conductor N which has input load R[2N-1] and output load R[2N]. Therefore, for the nth conductor, the input load is R[2n-1] and the output load is R[2n]. Note that the square brackets are used here only for clarity and are not in the actual SPICE deck, *problem_ name*.in.

To determine how to modify the return voltages to provide the desired value, the nodes to which the loads connect must be determined. The automated resistive loads always connect between some floating node and ground, which is always node 0. The input load on the first conductor connects to nodes 1 and 0 and the output between nodes M+2 and 0. On the second conductor, the input load is on node (M+2)+1 and the output load node 2(M+2) down to the final conductor N where the input load is on node (N-1)(M+2)+1 and the output load on node N(M+2). Therefore for the nth conductor, the input load connects to node (n-1)(M+2)+1 and the output load to node n(M+2).

To aid in the location of the load nodes, CABANA now places notations into the SPICE deck indicating where the loads are, including section loads if applicable.

At the end of the SPICE deck, there is a series of **.PRINT TRAN** V(i), V(j), V(k), ... lines. Presently, only the output load voltage, V(n(M+2)) for each conductor n, is monitored by the code. This voltage is divided by the `RLOAD` value on the appropriate `CONDUCTOR` keyword line to determine the load current. If this happens to be the desired result across the new load or subcircuit, then nothing need be done in the **.PRINT TRAN** section. To have the code only report the voltage, then change the `RLOAD` value to 1 and remember that the LOAD-CURRENT in the *problem_ name*.his file is actually load voltage.

If it is desired to have the voltage between any two nodes instead, then replace each of the V(n(M+2))'s, for each conductor n, with something like V(7,5) for example which specifies the voltage between nodes 7 and 5. Again, if the result desired is simply this voltage difference then change the `RLOAD` value to 1 for that conductor.

A SPICE **.SUBCKT** can be used if the voltage or voltage difference desired is on or between the external nodes of the subcircuit. If results are required from inside the subcircuit, then the circuit must be placed into the SPICE deck inline so that the necessary nodes are global. The desired nodes can then be referred to in the **.PRINT TRAN** line(s).

DISTRIBUTION:

1	MS 1152	K. L. Cartwright, 1352
1	MS 1152	C. D. Turner, 1352
1	MS 1167	E. F. Hartman, 1343
1	MS 1179	W. J. Bohnhoff, 1341
1	MS 1179	C. R. Drumm, 1341
1	MS 1179	W. C. Fan, 1341
1	MS 0899	Technical Library (electronic copy), 9536


Sandia National Laboratories