Title, string="Furman-Pivi Model ILC cavity example – Nate Pogue";

Option, ECHO=FALSE;

// If True, command prints echo of the input files on the standard error file

Option, INFO=FALSE;

// If false, it suppresses all information messages. Also effects gnu.out and eb.out files in Opal Cyclotron //simulations

Option, PSDUMPFREQ=1;

// Defines after how many time steps the phase is dumped into H5hut file, Default is 10

Option, STATDUMPFREQ=1;

//Defines how many time steps we dump statistical data, such as RMS beam emittance to the .stat file

// default is 10, only available in Opal T

Option, PPDEBUG=FALSE;

// To run the parallel plate benchmark simulation, set the option PPDEBUG to true. The input file and

// the geometry file needed by the parallel plate benchmark simulation is in the regression test folder.

Option, SURFDUMPFREQ=100;

//If the surface loss data is needed during post processing, we should specify the dump frequency

//use a positive integer for SURFDUMPFREQ in the OPAL input file, otherwise, the default value of -1

// is used and the \* Surface.h5 will not be generated.

MAXPARTSNUM=1000000;

// Set an upper limit of simulation particle number to prevent memory overflow.

DistSurf: DISTRIBUTION, DISTRIBUTION = "SURFACEEMISSION",

NPDARKCUR =0, INWARDMARGIN = 0.0,

FNBETA = 0.1, FNMAXEMI = 2,

SECONDARYFLAG = 1,

NEMISSIONMODE=false,

SURFMATERIAL=0;

//SURFACEEMISSION type of distribution can be used to customize the type and the parameters of //secondary emission within the model

//NPDARKCUR – has to do with dark current – unknown definition and how and why it is used, please //help with parameter space and definition.

//INWARDMARGIN define starting location of the particles off the inward normal w.r.t the boundary

//surface during field emission – 0 equals the surface itself

//FNBETA is the field enhancement factor related to field emission, typical values can range between 3 //and 300. A value of .1 will reduce field emission to a minimum.

//FNMAXEMI defines the maximum number of particles that emitted from a single triangle (or mesh //element) per time step.

// SECONDARYFLAG defines type of model used for evaluation of secondary particles. If 0 – no model used, 1 = Furman-Pivi Model, 2= Vaughn(0) model.

//NEMISSIONMODE – false will renormalize simulation particle approach – but what does that //mean?????? It will Emit real No. secondary’s or not (true). Could someone define this better?

// SURFMATERIAL, 0= surface material is copper, 1 = Stainless steel, used only for Furman-Pivi model

//If using the Vaughn model one must define the following parameters:

// note I cannot verify the numbers stated in manual with paper referenced in manual, link below

// <http://ieeexplore.ieee.org/stamp/stamp.jsp?arnumber=34278>

//VSEYZERO = δ0 in Vaughan’s model (0.5)

//VEZERO = 10 (In eV) Secondary Emission yield (SEY) will be δ0, if energy is less than VEZERO in

//Vaughan’s model (12.5 eV)

//VSEYMAX = δmax in Vaughan’s model (2.22)

//VEMAX = Energy related to δmax in Vaughan’s model (165 eV)

//VKENERGY = The roughness of surface for impact **energy** in Vaughan’s model(1.0)

//VKTHETA = The roughness of surface for impact **angle** in Vaughan’s model(1.0)

//VVTHERMAL = 7.26829821\*1e5; is mean energy of secondary emission, if given in Furman Pivi model

//the command is ignored

DistSurf1: DISTRIBUTION, DISTRIBUTION = "SURFACERANDCREATE",

INWARDMARGIN = 0.0, NPDARKCUR =10000,

EINITHR = 0.2;

//SURFACEANDCREATE specifies the position distribution of initial seed electrons for the study

// INWARDMARGIN: seed electron positions along the inward normal w.r.t the boundary surface.

// NPDARKCUR – once again – not 100% on what is being defined

// EINITHR is never defined in the manual??? Help?

 ge: GEOMETRY, FGEOM="inputgeometryfile.h5",

S=0.0, ZSHIFT=0.631, DISTRS={DistSurf, DistSurf1};

//S – can anyone give a better definition of S??

//ZSHIFT – state to change axis from z to y but unknown how to define change, assistance?

Box: RFCavity, PLENGTH = 1.262, VOLT = 1,

GEOMETRY = ge, FMAPFN = "../CyciaeEM.h5",

ELEMEDGE =0, FAST=true, FREQ =44.6, LAG = 0.0, DX = 0, DY = 0;

//PLENGTH – is the physical element length of ?????

//VOLT – is in MV, and scales field to a peak RF voltage, The effect of the cavity is δE = VOLT ⋅ sin(LAG − //2π ⋅ FREQ ⋅ t) but does it scale the input Efield map to a maximum of 1 MV or does it just multiply all

//the fields within input file by “value\*e6”?

//FMAPFN – calls in the field map for the RF structure here.

//ELEMEDGE – Physical start of the element on the floor – no idea what that means???

//FAST – if true uses interpolation of the points in the field map, if false uses FFT based algorithm for each particle and each step

//FREQ – frequency of the cavity in MHz

//LAG- phase advance or lag

//DX – No clue

//DY – No clue

//Below inserts a background magnetic field on the RF cavity

Mag: DCBfield, FMAPFN = "BackgroundField.h5", ELEMEDGE =0, DX = 0, DY = 0;

//ELEMEDGE, DX, DY same above??

//Assume DZ is simliar

Benchmark: Line = (Box, Mag);

Fs1:FIELDSOLVER, FSTYPE = NONE, MX = 32, MY = 32, MT = 256,

PARFFTX = true, PARFFTY = true, PARFFTT = false,

BCFFTX = open, BCFFTY = open, BCFFTT = open,

BBOXINCR = 0.1, GREENSF = INTEGRATED;

//FSTYPE – Type of field solver such as FFT, FFT Periodic, MG, AMR or none – None here because we are

//using a field map.

//MX - Number of grid points in x specifying rectangular grid

//MY - Number of grid points in y specifying rectangular grid

//MZ - Number of grid points in z specifying rectangular grid

//MT – is it the third dimension or is it time? – assuming time but clarification would be nice.

//What is the rectangular grid defining here? Is it dividing the geometry to be distributed to the different

//processors?

//PARFFTX - If TRUE, the dimension x is distributed among the processors

//PARFFTY - If TRUE, the dimension y is distributed among the processors

//PARFFTZ - If TRUE, the dimension z is distributed among the processors

//BCFFTX - Boundary condition in x [OPEN]

//BCFFTY - Boundary condition in y [OPEN]

//BCFFTZ - Boundary condition in z [OPEN,PERIODIC] – unused?

//BCFFTT = Boundary condition is open, but what is it defining, Z??

//GREENSF - Defines the Greens function for the FFT Solver

//BBOXINCR - Enlargement of the bounding box in % - What is the bounding box???

//qb = total charge

bfreq=300;

//bfreq is bunch frequency

bcurrent=qb\*bfreq;

//bcurrent is bunch current

beam1: BEAM, PARTICLE = ELECTRON, pc = P0,

NPART = 2000, BFREQ = bfreq ,

BCURRENT = bcurrent, CHARGE =-1;

//P0 is gamma\*beta\*EMASS; where gamma= (Edes+EMASS)/EMASS; beta=sqrt(1-(1/gammaˆ2)) EMASS is defined within OPAL,

//Edes is never defined in the example problem, where should it be defined?? Does it need to be

//defined?

Select, Line=Benchmark;

track, line= Benchmark, beam=beam1,

MAXSTEPS=23000, DT=4e-12, ZSTOP=3;

// MAXSTEPS - The maximal number of time steps

//DT - Initial time step for tracking

// ZSTOP - Defines a z-location [m], after which the simulation stops when SPOS > ZSTOP, and SPOS is the position along the trajectory.

run, method = "PARALLEL-T", beam = beam1,

fieldsolver = Fs1, MULTIPACTING=true;

endtrack;

Quit;